STRUCTURAL ANALYSIS AND STEEL DESIGN USING SODA 3.2.5

by

Ronald S. Harichandran
Professor and Chairperson

Department of Civil and Environmental Engineering
Michigan State University
East Lansing, MI 48824-1226

Phone: (517) 355-5107
Fax: (517) 432-1827
E-Mail: harichan@cee.msu.edu
http://www.egr.msu.edu/~harichan

August 2001

SODA is an acronym for Structural Optimization Design and Analysis Software for Structural Engineering, a program written by Acronym Software Inc., 22 King Street South, Suite 302, Waterloo, Ontario N2J 1N8, Canada [Phone: (519) 885-2454]. It is an extremely user-friendly program for IBM-compatible PC's and uses the Microsoft Windows 3.1 interface for mouse and menu oriented interaction. SODA is capable of performing static analysis of framed structures made of any material, and of designing framed structures made of structural steel satisfying the U.S. or Canadian specifications.

1. Mouse Usage in MS-Windows Applications

To use a drop-down menu, point to the menu bar, press the mouse button and drag the mouse down to the required option before releasing the button. (An alternate technique is to click once on the menu bar option, and click again on the required drop-down menu selection.) Options that are followed by ellipses (…) will bring up data entry forms for you to fill in. Menu options or data input fields that are not applicable for the current selections are shown in gray, and cannot be specified. Menu items sometimes show alternate shortcut keystrokes that may be used for specific items (e.g., *Ctrl+N* implies that this option may also be invoked by pressing the CTRL and N keys together).

2. Using SODA in the PC Laboratories

SODA is available on the PC's in Rooms 110 EB, 254 EB and 220 EB. The PC's in Room 126 EB presently do not have enough memory to run the SODA engine (analysis or design), and the PC's in Room 220 EB are available for public use only during weekends. The labs are open 24 hours a day. The PC's are configured with a 3.5 in., 1.44 MB floppy disk drive as A:, and a 5.25 in., 1.2 MB floppy disk drive as B:. To use a PC you need to have a userid and password. The userid is the same as your UNIX userid (usually the first eight characters of your last name). If you have never logged into a PC before, type **newuser** as your userid and follow the instructions to create your account. If you have difficulty logging in, please see someone at the Case Center Consulting Office. When you are finished using the PC please type **logout** to prevent others from corrupting your files.

Files created by you or SODA can be saved temporarily on the D: drive, or permanently on the M: drive or on floppy diskette drives A: or B:. The M:\ directory is identical to your UNIX home directory. In order not to clutter your UNIX home directory with PC files, it is recommended that you create a subdirectory off the M:\ directory by typing `mkdir M:\soda` and save files you wish to retain to this directory. You may run SODA with the D:\, A:\, B:\ or M:\soda directory open. However, about 5 MB of space will always be available in the D:\ directory, while the space available in the A:\, B:\ or M:\soda directory will depend on how much space you have already used up. Files saved to the D: drive are deleted when you logout.

To use SODA you must run Windows 3.1. After logging in, exit from the menu system by pressing **9** and then type **win31** to run Windows. After Windows is loaded, double click on the *SODA* icon at the bottom of the *Program Manager* window, and double click again on the *SODA 3.2.5* icon in the *SODA* window.

3. Mouse and Keyboard Usage and Data Input

On-line help on the overall use of SODA can be obtained by using the *Help* menu. SODA 3.2.5 uses the standard Windows format for on-line help messages. Clicking on the under-

lined text displayed in green on any help screen presents information related to that item. To obtain help on any of the data entry forms, press the F1 key when the form is displayed.

Data entry forms may have radio buttons, check boxes, alphanumeric fields, and/or drop-down lists. To select between various options using radio buttons, simply point and click at the desired one and all other radio buttons will be inactivated. To activate a check box, point and click. To fill-in or change alphanumeric data on a data entry form, point and click the mouse to place the cursor within the data field and then erase characters to the left with the BACKSPACE key or characters to the right with the DEL key. You can also highlight the text to be changed by clicking and dragging the mouse over the text and then typing in the replacement text. To display all available options in a drop-down list, click on the ↓ symbol at the right end of the field, or step through each available option using the ↑ or ↓ arrow keys.

In some data entry forms that display lists (e.g., nodes, members, etc.), multiple lines can be selected for modification. Clicking and dragging the mouse downwards or upwards selects consecutive lines. The SHIFT or CTRL keys can also be used in conjunction with the mouse. Pressing the SHIFT key and then clicking on a line selects all lines between the previous and current selections. Pressing the CTRL key and then clicking on a line adds it to the selection. When multiple lines are selected, only fields with identical values are shown in the corresponding data fields at the top of the form.

4. File Menu

4.1 New

This option is available when a problem is already being input or run, and indicates that you want to start a new problem. You will be asked whether you want to save the current data to the open file.

4.2 Open (Ctrl+O)

This option allows you to access an existing data file. SODA will display a list of directories and file names. To select the file to be opened, double click on the directory name (e.g., [-m-]); then double click on the subdirectory name if any (e.g., [soda]); then double click on the file name. As an alternative to double clicking, you may single click on a name and then single click on the *Open* button. Clicking on the symbol [..] selects the parent directory of the current subdirectory (e.g., if the current directory is D:\windows, then double clicking on [..] selects the D:\ directory).

4.3 Save (Ctrl+S)

This option overwrites data in the open file with the current data. If there is no open file, then this is the same as the *Save As* option.

4.4 Save As

This option prompts you for a file name to which the current data should be saved, and then creates the file in the directory displayed on the top right corner of the pop-up window. To write the file to a different location, specify the full path name including the drive, directory and subdirectories (if any) along with the file name (e.g., M:\soda\hw1). You will be warned if the specified file already exists, in which case you may choose to replace it with the current data. The *Save As* option also opens the specified directory, subdirectory and file.

TABLE 1 FILES PRODUCED BY SODA

File name Extension	Description
.POP	Input data file used for a SODA run, stored in binary form. Do not try to print.
.WNG	Warning messages produced during a run.
.ERR	Error messages produced during a run.
.ECH	Produced by the *Input Echo* option. Useful as a record of the input data.
.ANL	Analysis results.
.DSP	Deformed geometry graphics stored in binary form. Do not try to print.
.FNL	Final design results and verification.
.NML, .MDM, .DTL	Normal, medium and detailed reports produced during design.
.ELF	Effective length factors for members used in the final design.
.HTY	Design history and the cross sections used during each stage.

4.5 Erase

This allows you to erase individual DOS or SODA files, or all SODA files having selected extensions. You can make more room on your disk by erasing unwanted files. The various file name extensions, and the contents of files having these extensions are shown in Table 1.

- To erase SODA files, select the *SODA Worksheet* file type, select all extensions that you want to delete by using the check boxes, select the associated .POP file from the file list, and click on *Erase*.
- To erase DOS (non-SODA) files, select the *MS-DOS* file type, select the file to be erased from the file list, and click on *Erase*.

4.6 Print Topology

This option sends the graphical display of the structure to be printed on the currently active printer. See Section 4.8 on how to activate a printer. The currently active graphics text options will control the annotation in the printout (see Section 9.3 on page 16).

4.7 Print Output

This option is used to send output files to be printed on the active printer. Select the type of output that you would like to print from the cascading menu. The various output files are described in Section 8.4 on page 12.

You may also print any previously saved files having the extensions listed in Table 1 (except for those having the .POP extension) from DOS, by using the print command.

4.8 Printer Setup

This option is used to select between the different printers that are available for printing. Two printers are available in the PC labs: a high resolution HP LaserJet III, and a dot-matrix printer. You may activate either of these printers by double-clicking on its name (or clicking on its name and then clicking on *Ok*).

4.9 Exit

This option allows you to exit from SODA to the MS-Windows environment.

5. General Menu

5.1 Title

This option is used to enter a description for the overall problem. The *Project Title* is a one line description, and an elaborate description can be entered in the *Description* box.

5.2 Definition

This option is used to define the problem type. The following selections are required for all problems:

- **Dimension –** Select *2-D* for planar structures, or *3-D* for space structures. Planar structures are oriented in the *X-Y* plane. The global *Y*-axis is vertical for all structures.
- **Structure –** Select *Frame* or *Truss*.
- **Problem Type –** Select *Design* for a problem in which at least some of the members are to be chosen, *Verify* for a problem in which the structure is to be verified against a design specification, or *Analysis* for an analysis problem.
- **Behavior Type –** If second-order effects can be neglected, then select *First-order*. If second-order effects are significant select *P-Delta*. SODA is capable of accurately accounting for $P\Delta$ effects on the overall structure, arising from geometric non-linearity under large displacements. However, $P\delta$ effects arising due to combined axial load and flexure in a member are not accounted for in an analysis problem, and are only approximately accounted for in design problems using an appropriate magnification factor.

The following information is required only for **design** or **verify** type problems:

- **Sidesway –** Select *Prevented* if the frame is braced, or *Permitted* if it is unbraced.
- **Design Code –** Select *AISC-LRFD* for the LRFD specifications.

5.3 Section Database

This option allows selection of the cross section database. The AISC.SDA database is the default and should not be changed, unless no database is needed, in which case selecting *None* enhances efficiency.

5.4 Units

The system of units (imperial or metric), and the specific force and length units within each system are chosen in this option.

5.5 Fabrication

This option is used to specify connection details for tension members, so that the effective area will be used in design. *All* tension strengths are computed using the same properties. The following may be specified:

1. Bolt hole diameter for use with *all* tension members (including bending members with tension). SODA assumes that the bolt are in the webs. The default bolt hole diameter is

0.8125 in. (3/4 in. bolt + 1/16 in. oversize). For fracture paths having more than one bolt hole, specify the bolt hole diameter as a multiple of the number of bolts (e.g., for two 3/4 in. bolts, specify $0.8125 \times 2 = 1.625$ in.). Fracture along complicated paths involving diagonal lines is not checked.

2. Gusset plate thicknesses for use with *all* double angle sections. The default value is 0.375 in., and alternative options are 0.0 in. or 0.75 in.
3. The ratio A_e/A_n (U factor) for *all* tensile members.

6. Structure Menu

The data entry forms selected in this menu option remain on the screen until another option is selected from the menu.

6.1 Nodes (Ctrl+N)

This option brings up a data entry form for inputting/editing the nodes on the structure. The following information must be specified for each node:

4. The node name. This may be any unique alphanumeric string of up to 9 characters (e.g., 1, N1, Node1, etc.). Do not use the apostrophe (') in node names as this causes a run-time error. The nodal coordinates (*X*, *Y*, and *Z* for 3-D, or *X* and *Y* for 2-D). The *X* axis is horizontal left to right, the *Z* axis is horizontal out of the plane, and the *Y* axis is vertical upwards.
5. The type of support at the node. Click on the ↓ symbol at the right of the *Support* drop-down list to reveal all the available options. The support type may be *None* (free), *Fixed* (against rotation and translation in all directions), *Pinned* (allows rotations but preventing translation in all directions), *Pinned* [*except X*] (allows rotations about the *Y* and *Z* axes, and does not allow rotation about the *X* axis or translations), *Pinned* [*except Y*], *Pinned* [*except Z*], *Roller* [*along X*] (allowing all rotations, but only *X* translations), *Roller* [*along Y*], *Roller* [*along Z*], or *Other*. The *Other* support type may be selected to specify all other types of supports, including spring supports on the accompanying data entry form.

Nodes may be added, modified or deleted as follows:

- To add a node, enter the node name and coordinates in the data entry fields, select a support type if required, and click on *Add*.
- To modify existing nodes, select one or more nodes in the list, edit the name and/or coordinates in the data entry fields, change the support type if required, and click on *Modify*.
- To delete existing nodes, select one or more nodes in the list, and click on *Delete*.

6.2 Groups (Ctrl+G)

This option is used to define member groups. All members that have the same cross sectional and material properties should be grouped into a single group. The number of groups is often substantially smaller than the number of members. Each group is identified by a name, and may be associated with a section shape and/or section designation. For example, for a W14×90, the section shape is W and the section designation is 14X90 (with an upper case X and no blanks). The section shapes allowable in the AISC.SDA database, example section designations, and the description of each shape are given in Table 2. Note that all thicknesses are given in decimal form, and must be specified exactly as given in Table 3.

TABLE 2 SECTION SHAPES IN THE AISC.SDA DATABASE

Name	Example designation	Description
W	16X31	Wide flange sections
M	10X9	Miscellaneous beams and columns
S	12X35	Standard beams
C	9X20	Standard channels
MC	13X50	Miscellaneous channels
EL1L	3.5X3.5X.4375	Equal leg angles
UL1L	3.5X2.5X.4375	Unequal leg angles
EL2L	3.5X3.5X.4375	Two equal leg angles back to back
LL2L	3.5X2.5X.4375	Two unequal leg angles with long legs back to back
SL2L	3.5X2.5X.4375	Two unequal leg angles with short legs back to back
RHS	8X4X.3125	Rectangular hollow sections (tubing)
SHS	8X8X.3125	Square hollow sections (tubing)
CHS	3.5X.226	Circular hollow sections (pipe)

TABLE 3 DECIMAL EQUIVALENTS FOR THICKNESSES

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

The remaining input data depends on whether a design/verify or analysis is being performed. The following steps are applicable for a **design** or **verify** problem:

1. If the section size is to be chosen, leave out the section designation.
2. If necessary, change the Young's and shear moduli, and yield and ultimate stresses (the default values are for A36 steel).
3. If necessary, change the maximum and/or minimum nominal section depths. These may be used to limit the section selection. For example, if the section shape is W, and the maximum and minimum depths are specified as 8 in. and 14 in., then the selected shape will be restricted between W8's and W14's. If the minimum and maximum depths are made the same, then the section is limited to a single nominal depth.
4. If necessary, change the maximum allowable slenderness ratio, *KL/r*, for compression and tension. The default values for these are set based on the active design code.

   In framed structures, SODA designs beams as beam-columns, even if the axial loads are very small, and hence uses the applicable slenderness limit. If the slenderness ratio is the controlling factor for a beam, then a redesign may be appropriate, with increased limits on the slenderness ratios.

The following steps are applicable for an **analysis** problem:

1. If standard rolled steel sections are used then specify their shapes and designations, and their properties will be obtained from the active section database. For other sections leave

the shape and designation fields blank, and instead specify A, I_x, I_y and J_z (polar moment of inertia) for the cross section.

2. For materials other than steel, modify the Young's and shear moduli.

After data is entered for each group, add it to the group list by clicking the *Add* box. Modifications and deletions of groups in the list may be made as for nodes.

6.3 Members (Ctrl+M)

This option is used to describe the members. The following information is required for each member:

3. The member name. This may be any unique alphanumeric string of up to 9 characters (e.g., 1, C1, Column1, etc.). Do not use the apostrophe (') in node names as this causes a run-time error.The start and end nodes that the member connects. Existing node names may be typed in directly or selected from the drop-down list.
4. The group name that the member belongs to. An existing group name may be typed in directly or selected from the drop-down list. The cross sectional properties of the member are taken from those specified for the group.
5. The Beta Angle. This angle defines the orientation of the cross section with respect to the longitudinal (local z) axis of the member.
   - For a horizontal beam, the default Beta Angle of zero means that the strong (local x) axis is horizontal, and therefore strong axis bending occurs. For weak axis bending of a horizontal beam the Beta Angle should be 90°.
   - For a vertical column, the default Beta Angle of zero means that the local 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 is strong axis bending).
6. The effective length factors K_x, K_y, B_t and B_b. K_xL and K_yL, in which L is the length between the nodes, are taken as the effective lengths for strong and weak axis buckling, while B_tL and B_bL are taken as the unbraced length of the top and bottom flange for lateral torsional buckling. The top flange is that face of the section from which the local y-axis emerges.

   Use the drop-down list to display the available choices for K_x and K_y: *Calculate, 1.0* or *0.0,* or type the appropriate effective length directly. If *Calculate* is selected, SODA automatically computes K_x and/or K_y. For unusual situations, such as when intermediate nodes are used in a member, it is not advisable to select the automatic procedure. For a member with intermediate nodes, K_x and K_y should be greater than one (e.g., $K_y \approx 2$ for a member with a mid-span node) so that the effective length of each segment of the member is equal to the effective length of the full member.

   For the unbraced length, SODA uses B_tL and B_bL for the top and bottom flanges, respectively.
7. The type of end joints. Use the drop-down list to display the available choices. For rigid joints, the +—+ type should be selected. If the right end, left end, or both ends of the member are pinned, then the +—o, o—+, or o—o types should be selected, respectively. For trusses, the o—o joint type is the only available one. SODA cannot account for axial and shear releases.

6.4 Regular Framework (Ctrl+R)

This option is used to automatically generate nodes, groups and members in a regular rectangular frame. For a large framed structure, it is quicker to automatically generate the frame and then selectively delete and/or modify individual nodes, groups and/or members, than to specify each node, group and member individually.

6.5 Diagonal Bracing

This option is used to automatically generate diagonal bracing elements in regular rectangular frames.

6.6 Move Nodes

This option enables a group of nodes in a regular frame to be moved to new locations.

7. Loads Menu

7.1 Nodal Loads

This option is used to create or edit loads applied to the nodes of the structure. Loads are grouped into load sets (e.g., Wind-1, Wind-2, EQ, etc.), and in each set nodal loads are applied at specific nodes.

Occasionally, loads in a structure may be applied along restrained degrees-of-freedom (i.e., into supports). These loads do not deform the structure, but simply influence the reactions at the support. It should be noted that while SODA allows such loads to be input, they are not used when computing the reactions. For equilibrium to be satisfied, the reactions must be corrected to include the effect of these loads.

To **create** individual nodal loads, perform the following:

1. Enter a load name (e.g., Wind-1) in the *Specified Nodal Load* data field for a specific load set. Add this to the *Specified Nodal Load* list by clicking on the *Add* button at the left edge of the screen.
2. Enter the node name or select one from the drop-down list, and then enter the forces/moments along/about the *X, Y* and/or *Z* axes. Enter these loads into the list by clicking the *Add* button at the bottom of the screen. This adds the node to the *Node Name* list.
3. Repeat step 2 for other nodal loads belonging to the same load set.
4. If necessary, graphically view all the nodes that are loaded by clicking on *View*. The nodes that are loaded are displayed as large red dots.
5. Repeat steps 1 through 5 for each additional load set.

The following steps describe how to **modify** nodal loads:

1. To modify the name of a service load, select the service load from the load list, change its name in the *Specified Nodal Load* data field, and click on the *Mod* button at the left end of the screen.
2. To modify the node name and/or magnitudes of the loads applied at the node, first select the appropriate service load from the load list, then select the node from the node list, edit

TABLE 4 ORIENTATIONS FOR MEMBER LOADS (ALSO SEE FIG. 1)

Orientation	Dimension	Explanation
P	2-D	Load acts perpendicular to member. Positive magnitude indicates upward load (i.e., having a positive component in the global *Y* direction).
A	2-D or 3-D	Load acts along the longitudinal axis of the member. Positive magnitude indicates load directed from start to end node of member.
X or Y	2-D or 3-D	Load acts along the global *X* direction. Distributed loads act on the projected length.
Z	3-D	Load acts along the global *Z* direction. Distributed loads act on the projected length.
x or y	3-D	Load acts perpendicular to member along the local *x* or *y* direction.

the node name and/or magnitudes of the forces and/or moments, and click on the *Mod* button at the bottom of the screen.

The following steps describe how to **delete** nodal loads:

1. To delete an entire load set, select the load from the *Specified Nodal Load* list, and click on the *Del* button at the left end of the screen.
2. To delete loads at a specific node in a particular load set, first select the load set from the *Specified Nodal Load* list, then select the node from the *Node Name* list, and click on the *Del* button at the bottom of the screen.

7.2 Member Loads

This option is used to create or edit loads applied to the members of the structure. SODA does not allow member loads on trusses (i.e., you must convert all member loads to equivalent nodal loads). As with nodal loads, member loads are grouped into load sets (e.g., Dead, Live-1, Live-2, etc.).

Loads may be directed along the *X*, *Y* or *Z* global axes, perpendicular to the member, or along the longitudinal axis of the member as described in Table 4 and illustrated in Fig. 1. Take careful note of the sign conventions, especially the fact that downward loads perpendicular to members should have negative magnitudes.

The steps required to **create** member loads are similar to that described for nodal loads, and are described below.

1. Enter a load name (e.g., Live-1) in the *Specified Member Load* data field for a specific load set. Add this to the *Specified Member Load* list by clicking on the *Add* button at the left edge of the screen.
2. Enter the member name or select one from the drop-down list, select the load type, and enter the load information as described below:
   (a) For a uniform load over the entire length of the member: select the *Full UDL* load type; enter the magnitude of the load in the *w @ a* data field; and enter *X*, *Y*, *Z* or *A* in the *Orientation* data field.

(b) For a point load: select the *Point Load* type; enter its magnitude in the *w @ a* data field; enter its position along the longitudinal axis of the member from the start node of the member, as a fraction of the member length, at the *Distance a/L* data field; and *X*, *Y*, *Z* or A in the *Orientation* data field.

(c) For a uniform load distributed over a part of the member, or for a linearly varying distributed load: select the *Other DL* load type; enter the magnitude at the beginning of the load in the *w @ a* data field; enter the position to the beginning of the load along the longitudinal axis of the member from the start node of the member, as a fraction of the member length, at the *Distance a/L* data field; enter the magnitude at the end of the load in the *w @ b* data field; enter the position to the end of the load along the longitudinal axis of the member from the start node of the member, as a fraction of the member length, at the *Distance b/L* data field; and select an orientation from the drop-down list in the *Orientation* data field.

After completing the load information, click the *Add* button at the bottom of the screen to enter the member into the *Member Name* list. Members having different load types may appear more than once in the *Member Name* list.

3. Repeat step 3 for other member loads belonging to the same load set.
4. If necessary, graphically view all the members that are loaded by clicking on *View*. The members that are loaded are displayed in a dashed red line.
5. Repeat steps 1 through 5 for each additional load set.

Member loads may be modified or deleted in the same manner as nodal loads.

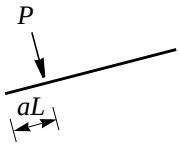
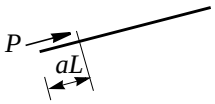
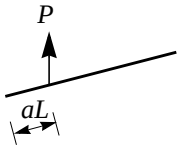
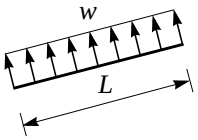
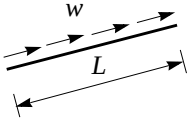
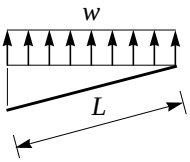
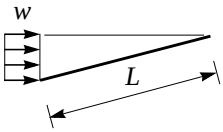
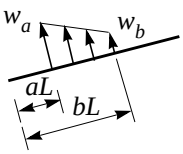
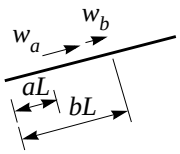
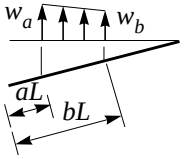
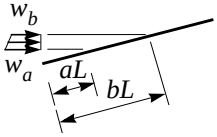
Orientation / Load Type	Perpendicular	Axial	Y	X or Z
Point Load				
Full UDL				
Other DL				

Figure 1 Member loads

7.3 Temperature Loads

The influence of temperature variations may be accounted for using this option. The technique for entering the information is very similar to that used for member loads, the only difference being in the load description. The temperature change and coefficient of thermal expansion may be entered for selected members.

7.4 Support Settlement

The influence of support settlements may be accounted for using this option. The technique for entering the information is very similar to that used for nodal loads. The displacement/rotations along/about the global *X*, *Y* and/or *Z* axes may be entered at support nodes.

7.5 Area Loads

For 3-D structures, this menu option allows the quick specification of a load to be applied normal to surfaces.

7.6 Load Combinations

The combinations of the various individual load sets to be used for the analysis or design must be specified using this option. **Even if only one load is being used, a combined load consisting of only this load must be specified**.

The steps required to **create** load combinations are described below:

1. Enter a name for the load combination (e.g., Dead + Live, Dead + Live + Wind, etc.) in the *Load Combination Name* data field. Add this to the *Load Combination Name* list by clicking on *Add*.
2. Select a load that is required for the load combination from the *Load Name* list by clicking anywhere on the line containing it. If necessary, edit the factor displayed in the *Load Factor* data field (this should be zero for loads that are not to be included in the combination). Click on the *Mod* button next to the *Load Factor* data field.
3. Repeat step 2 for all the loads required for the load combination.
4. Repeat steps 1 to 3 for each additional load combination.

If lateral drifts are to be checked under service loads, then an appropriate load combination with unit load factors may be specified.

7.7 Displacement Limits

If the design should satisfy displacement limits under certain load combinations, then this option allows the maximum displacements at selected nodes to be specified. Displacement limits are normally considered with service loads. Rather than creating separate load combinations with unit load factors to signify service loads and specifying displacement limits for these load combinations, it is usually more convenient to specify displacement limits that are factored values of the service load limits and apply these to factored load combinations used in strength design. For example, if the lateral drift under service earthquake load (E) is to be limited to 0.5 in. at a particular node, and the strength design included a load combination with 1.5 E, then a X-translation limit of $1.5 \times 0.5 = 0.75$ in. can be specified for the factored load combination.

To limit mid-span deflections of beams, beams must be divided into two members such that a node exists at the mid-span locations, and displacement limits must be specified at the mid-span nodes. However, as described in Section 6.3, the effective length factors K_x and K_y must be doubled to reflect the actual length of the member so that the effective length for each half of the member will equal the effective length of the full member.

8. View Menu

8.1 View Topology (^D)

This option is used to draw the structure. To display node, member and/or group names on the drawing, see Section 9.3 on page 16. When the structure is displayed, the mouse pointer turns into a magnifying glass symbol when it is moved into the graphics area. The drawing can be enlarged about any point by placing the mouse pointer at that point and double-clicking. To restore the drawing back to its normal size, select the View Topology option from the View menu (or press CTRL-D).

8.2 View Deflections

This option is only available after a run, and is used to display the deflected shape of the structure under any load combination. When this option is selected, the *Load Combinations* list is presented, and any one may be selected by clicking on it. The scale factor (default 100) may be increased to magnify the displayed deflections. Only the joint translations are used to draw the deflected shape, and members are drawn as straight lines. Joint rotations and member curvatures are not displayed.

To obtain a printout of the deflected shape, first view the deflected shape on the screen and then select the Print Topology option from the File menu.

8.3 View Moment and Shear

This option is used to view the moment and shear diagrams for any member. Select the member and load combination using the appropriate drop-down lists, and for 3-D structures select the axis (i.e., strong or weak) for which the moment and shear diagrams are to be displayed. The maximum positive and negative moment and shear values are shown, along with the positions at which these maxima occur. The total length of the member and the axial force are also shown. Regardless of the actual orientation of the member, the moment and shear diagrams are plotted with the member laid horizontally with its start node at the left and its end node at the right. Positive moments cause tension on the “top” face of the member (this is opposite to the usual convention). This convention results in the moment diagram being drawn on the tension face of the member. Moment and shear diagrams for different members and load combinations can be viewed repeatedly by selecting these from the drop-down lists.

8.4 View Output Files

This option is used to view the various output files on the screen. A cascading sub-menu is displayed from which the specific type of file can be selected. If a file is very large you will only be able to view a part of the file. You may view the entire file using a general editor or by printing the file.

Parts of some typical output files are reproduced in Figs. 2, 4and 5, and are annotated in bold print explaining the various output quantities. Each type of output is described below:

1. **Analysis results:** Selecting this output displays the results from the latest analysis. Fig. 2 shows excerpts of a typical analysis output file. Note that member shear and axial forces are given in the local coordinate system shown in Fig 3, while reactions and joint displacements are given in the global coordinate system. Bending moments are positive if they cause tension on the "top" face of the member. Results are tabulated for each load combination, and an equilibrium check is provided at the end by summing the *x* and *y* loads and reactions. If loads are applied into supports, then these are not reflected in the support reactions, and the equilibrium check will show a mismatch between the loads and reactions.
2. **Final design report:** Selecting this output displays a summary of the design (this is displayed automatically at the end of a *Design* or *Verify* type problem). Fig. 4 shows a typical final design report. The governing load combination and code clause is shown only for the most critical member. For a different member, another load combinations may be more critical. Occasionally, the slenderness ratio may be the critical factor for the most critical member, in which case zero is printed for the governing load combination. This is only a summary report, and either the normal, medium or detailed reports should be examined.

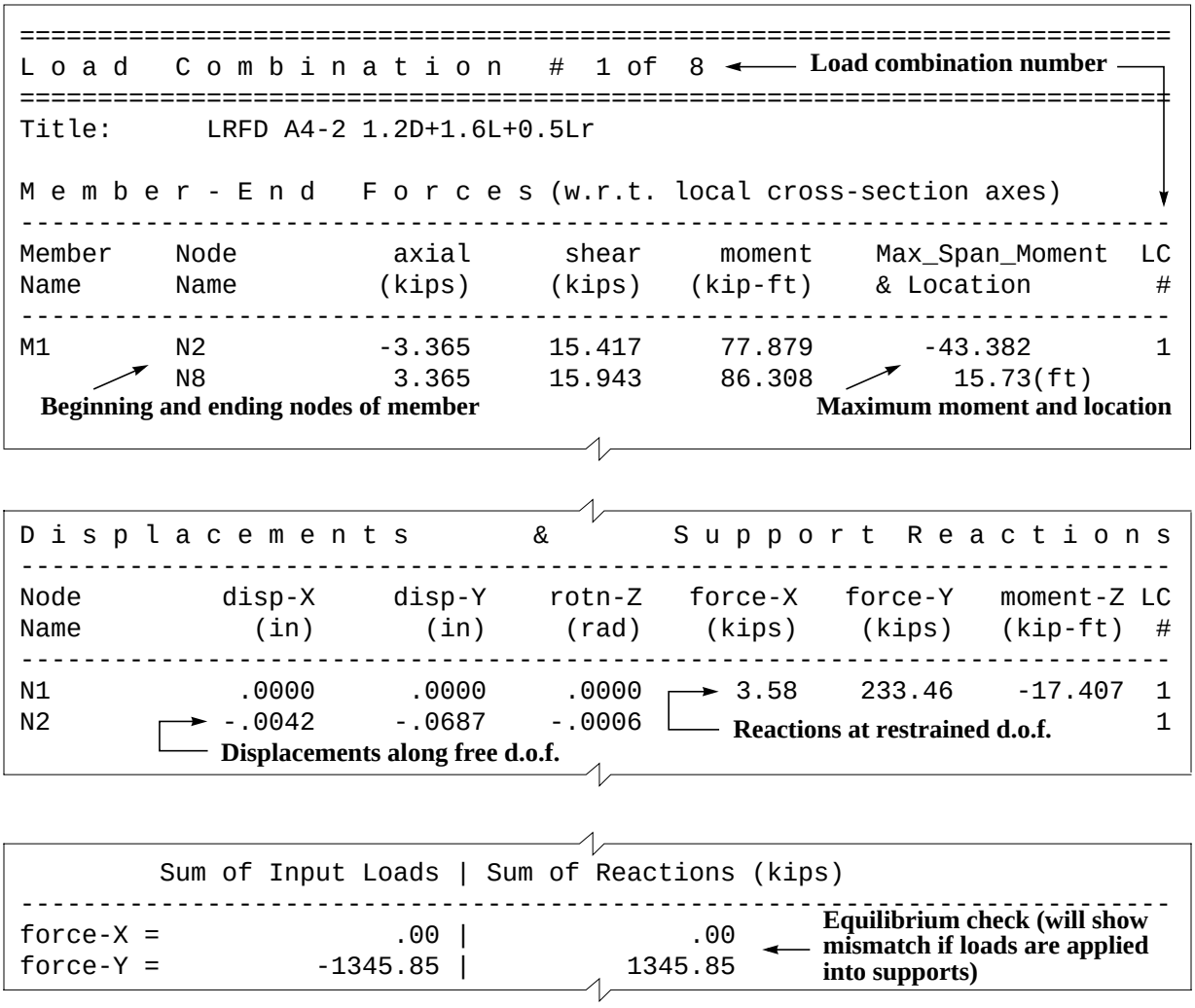
```
**************************************************************************
          F I N A L   D E S I G N / V E R I F I C A T I O N
**************************************************************************


F i n a l  D e s i g n
==========================================================================
    Group     X-Section                Length       Volume       Weight
    Name      Shape+Designation          (ft)        (ft)3        (lbs)
    ----------------------------------------------------------------------
    B1        W    12X53                288.0         31.2      15311.5
    B2        W    10X49                192.0         19.2       9422.4
    B4        W    16X67                 76.0         10.4       5102.5
    B5        W    12X30                 48.0          2.9       1437.9
    B6        W    18X119                76.0         18.5       9091.2
    B7        W    12X40                 48.0          3.9       1930.3
    ----------------------------------------------------------------------
                                     TOTAL=          86.2      42295.7

    Governing Clause           : eq.H1-1a
    Governing Group            : B5
    Governing Member           : M31
    Governing Load_Combination :   2

    Maximum constraint response  =    .984
    Specified allowable response =   1.000

    All specified design constraints have been satisfied.
```

Cross sections chosen for each group

Number of the LRFD code clause

This information applies only to the member found to be most critical

Figure 4 Typical final design report




```
================================================================================
L o a d   C o m b i n a t i o n   #  1 of  8
================================================================================
Title:      LRFD A4-2 1.2D+1.6L+0.5Lr

M e m b e r - E n d   F o r c e s (w.r.t. local cross-section axes)
--------------------------------------------------------------------------------
Member    Node          axial      shear     moment    Max_Span_Moment  LC
Name      Name         (kips)     (kips)   (kip-ft)    & Location        #
--------------------------------------------------------------------------------
M1        N2           -3.365     15.417     77.879       -43.382        1
          N8            3.365     15.943     86.308         15.73(ft)

D i s p l a c e m e n t s       &       S u p p o r t  R e a c t i o n s
--------------------------------------------------------------------------------
Node         disp-X     disp-Y    rotn-Z   force-X   force-Y   moment-Z LC
Name           (in)       (in)     (rad)    (kips)    (kips)   (kip-ft)  #
--------------------------------------------------------------------------------
N1            .0000      .0000     .0000      3.58    233.46    -17.407  1
N2           -.0042     -.0687    -.0006                                 1

        Sum of Input Loads | Sum of Reactions (kips)
--------------------------------------------------------------------------------
force-X =              .00 |              .00
force-Y =         -1345.85 |          1345.85
```

Figure 2 Excerpts from typical analysis results output

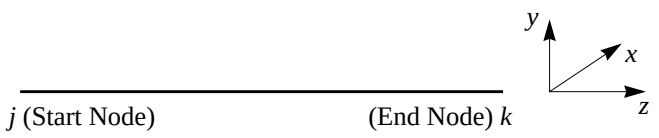


Figure 3 Local coordinate system

3. **Normal design report:** Selecting this output displays a normal report, excerpts of which are shown in Fig. 5. The largest slenderness ratios for each group, and the width/thickness ratios for the corresponding member are shown. For each load combination, information about the strength ratios for axial, shear, bending, and combined load effects are given for the most critical member in each group (P_r, V_r and M_r are used in the output for ϕP_n, ϕV_n and ϕM_n, respectively). In a separate table at the end, the chosen section shape, and governing load combination, code clause and response ratio (a measure of how close the design is to the specifications) based on all load combinations is given for the most critical member in each group.

```
*****************************************************************************
                        N O R M A L   R E P O R T
*****************************************************************************
Pdelta Analysis = Off   Sway = On    Cb, Cm = calculated

Summary of Governing Slenderness Ratios for each Group
-----------------------------------------------------------------------------
Group     Governing       Kl/r       KL/r       KL/r        b/t        h/w
Name      Member          (x)        (y)      (allw)
-----------------------------------------------------------------------------
B1        M1            75.481    154.839    200.000      8.696     27.826
B2        M10           96.219    151.181    200.000      8.929     22.353
```

```
L o a d   C o m b i n a t i o n   #   1 of  8
=============================================================================
     LRFD A4-2 1.2D+1.6L+0.5Lr   ← Load combination name
-----------------------------------------------------------------------------

Code Clause Status for Groups Governed by COMPRESSION        Code clauses
-----------------------------------------------------------------------------
Group     Governing         E2       F2.2         F1   eq.H1-1a   eq.H1-1b
Name      Member          Pu/Pr     Vu/Vr      Mu/Mr     Combined_Stresses
-----------------------------------------------------------------------------
B1        M8              .019       .139       .529       .000       .540
B4        M16             .424       .021       .067       .485       .000
```

```
F o r   A l l   L o a d   C o m b i n a t i o n s   Most critical member, load comb.
                                                   and code clause for each group
=============================================================================

Governing Load Combination and Code Clause for each Group
-----------------------------------------------------------------------------
Group      Member          Section       Load    Code            Response
Name       Name       Shape Designation  Case    Clause             Ratio
-----------------------------------------------------------------------------
B1         M7         W     12X53           2    eq.H1-1a            .980
B2         M10        W     10X49           2    eq.H1-1b            .841
```

Figure 5 Excerpts from typical normal design report

4. **Medium and detailed reports:** Selecting one of these outputs displays medium or detailed reports. The medium report is similar to the normal report, and in addition contains information about the code check for each member (not just the most critical member in a group). The detailed report contains substantial information that enables the design procedure to be traced, one step at a time.
5. **Design history:** Selecting this output displays the design history that was displayed on the screen during the design process, showing the cross sections shapes selected during each design stage.
6. **Effective lengths:** Selecting this output lists a table showing the effective lengths used for each member.
7. **Warnings and errors:** Selecting one of these outputs displays the warning or error messages issued during the run.

8.5 Zoom In

This option zooms the drawing of the structure about the point at the center of the screen. Zooming may also be performed using the mouse. A rectangular zoom border appears when the mouse is clicked and dragged in the graphics region, and the region enclosed by the border when the mouse button is released is magnified.

8.6 Zoom Out

This option may be selected after the *Zoom In* option, to redraw the structure at the scale prior to zooming in.

8.7 Zoom Reset

This option redraws the structure at the original scale prior to zooming.

9. Options Menu

9.1 Graphics

This option enables the *X, Y* and *Z* axes to be displayed/suppressed on the graphics screen, and/or a small *X-Y-Z* axis diagram to be displayed/suppressed at the lower left of the graphics screen.

9.2 Graphics View

This option is used to select which view of the structure is to be drawn. For 3-D structures, isometric, projection or plan views may be displayed by selecting the appropriate radio buttons. The view angle may be specified in the *Horizontal angle* and *Vertical angle* data fields.

9.3 Graphics Text

This option is used to select the format of the annotation that is displayed on the graphics screen. The font used for the annotation from the *Font* drop-down list. Member, group and/or node names and/or the worksheet title may be annotated or suppressed. To maintain clarity, it is often necessary to suppress some of the annotation. The text is normally written horizontally, but may be slanted along the member by selecting the *Align Text with Members* radio button.

10. Run Menu

10.1 Run Engine

This option is used to perform the design, verification or analysis. New files are created automatically in the open directory, and results are stored in these files. The following two fields must be selected for all runs from the pop-up form that is displayed when *Run Engine* is selected:

- *Output Format* – This controls the printing of decimal numbers. The free format prints numbers as 1.234, 12.34, etc., while an exponential format prints them as .1234E+01, .1234E+02, etc. If the output does not contain enough significant figures when the free format is used, rerun the problem using the exponential output format.
- *Behavior Type* – If second-order effects can be neglected, then select *First-order*. If second-order effects are significant select *P-Delta*. This selection overrides the selection made in the general definition (see Section 5.2 on page 4). For the design of large structures, considerable computational time may be saved by first performing the design neglecting second-order effects, and then performing a subsequent design including second-order effects.

The remaining fields are applicable only to **Design** or **Verify** type problems:

- **Displacement Constrained –** Check this box if the displacement limits specified earlier (see Section 7.7 on page 11) are to be applied to the current design or verification.
- **Constraint Factor –** If left blank, this is taken to be 1.0. This factor is sometimes useful for scaling all aspects of the problem. For example, if structure is to be designed for 10% greater loads (i.e., all loads are to be increased by 10%), then a constraint factor of 0.9 can be used without having to change all the loads.
- **Report Detail –** You may select among the three levels of detail in the final report.
  (a) A *Normal* report contains the analysis results, and for *Design* or *Verify* type problems the code check details are provided for the critical member in each group. For *Design* problems, the chosen cross sections are listed for each group.
  (b) A *Medium* report gives all the information in the *Normal* report, and in addition, for *Design* or *Verify* type problems, the performance of every member is assessed against the governing design code.
  (c) A *Detailed* report provides a great amount of secondary calculated data so that the run may be fully traced.
- **Optimization –** If this is turned *On* then the entire structure is optimized to produce the least weight. If this is turned *Off*, then the run is made faster, but only a member-by-member selection routine is used. For large structures it may be desirable to do an initial design with optimization disabled and then do a subsequent design with optimization enabled.
- **Bending Coefficients –** The *Calculate* option requests SODA to compute the C_b and C_m bending coefficients based on the moment diagram, the presence of span loads and the sidesway condition. The *Unity* option conservatively sets $C_m = C_b = 1.0$.
- **Design Process –** The *Continuous* option specifies that the iterative design process is to be conducted continuously without user intervention until the final design of the structure is found. The *Stage-by-Stage* option specifies that the iterative design process is to be interrupted after each design stage to allow the user to change the run time options available in this data entry form, if required.

When the run begins, a new *SODAENG* window appears within which windows entitled *Design History* and *Unit ** are displayed. **Do not resize these windows!** The design history is displayed in the *Design History* window. Run time messages and user input options are displayed in the *Unit ** window. The status of the run (*Running, Input Pending* or *Finished*) is displayed at the left end of the status bar located at the bottom of the *SODAENG* window.

For a **design** problem in which the *Stage-by-Stage* option was selected for the *Design Process*, you will be prompted with a menu such as that shown below (the status may vary depending on the chosen options):

```
Options Menu
-------------------------Status
O - Optimizer              off
S - Stop at each stage     off
P - P-Delta analysis       off
A - Analysis output        floating pt.
R - Reporting level        normal
B - Bending coefficients   calculate

Q - Quit (halts execution of Engine)
X - eXit this menu
Enter a letter ->
```

At this stage, you may respond with:

1. O, S, P, A, R or B to toggle any one of these options to the next available choice (e.g., if you press O, the optimizer will be turned on). The menu will be re-displayed showing the new settings. This menu allows the refinement of a design.
2. Q to quit and return to the data entry form for modifying these options and restarting the design procedure.
3. X to exit the menu and continue the run with the displayed run time options.

For all **design** problems, the following *Action Menu* window will be displayed on top of the *Unit ** window when the design has been completed (or after every design stage if the *Stage-by-Stage* option is in effect):

```
To proceed, choose a LETTER from the following...
 S = Select best (# n) and Exit.
 U = User selects from Design History & Exits.
 M = Menu of options (Optimizer, P-Delta...)
 C = Continue for another design stage.
 Q = Quit (no results will be saved).
 Enter a letter -->
```

Typically you would respond with an S, but you may choose any of the other options to control the final design.

When the run is complete, a dialog box entitled *Engine Task Completed* will appear with the prompt “Exit Engine and return to SODA?”. If you respond with *Yes* the *SODAENG* window disappears. If you respond with *No* the *SODAENG* window remains open and the status message *Finished* is displayed. You may manually close the *SODAENG* window using the *File/Exit* command.

10.2 Review Input

This option produces an echo of the input data, which is useful for verifying that the input data is correct. It creates a file with a .ECH extension.