
Celal N. Kostem
A GUIDE TO SODA

by

Celal N. Kostem

Department of Civil Engineering
Lehigh University
Bethlehem, Pennsylvania

July, 1994

Fritz Engineering Laboratory Report No. 400.52
A GUIDE TO SODA¹

(STRUCTURAL, OPTIMIZATION, DESIGN, and ANALYSIS)²

by

Celal N. Kostem³,⁴

1. INTRODUCTION

SODA (Structural, Optimization, Design, and Analysis) is a WINDOWS-based software which runs on 386, 486, and higher, PC-platforms. SODA can analyze two or three dimensional trusses and frames. It can also design these types of structures so long as structural steel, designed in accordance with the US or Canadian Steel Specifications is used. The software is also capable of designing the aforementioned structures for "least structural weight," i.e., weight optimization. It can be used to check the validity of a completed design using either US or Canadian Steel Specifications.

Both the developers of SODA and the author of this guide have tested the software using suites of problem sets. In using the software, however, the user accepts and understands that no warranty is expressed or implied by the developers or distributors of SODA, Lehigh University, Lehigh University Computing Center, or Celal N. Kostem on the accuracy or the reliability of the software. The users must clearly understand the basic assumptions of the software and must verify their own results.

2 Copyright by Celal N. Kostem, 1994.

3 Professor of Civil Engineering; Department of Civil and Environmental Engineering; Fritz Engineering Laboratory; 13 E. Packer Avenue; Lehigh University; Bethlehem, PA, 18015-3176 USA

Phone: 610-758-3529; FAX: 610-758-4522; Voice Mail: 610-758-6666, Box No.: 83529; Answering Machine: 610-865-1162; E-Mail: cnk0@Lehigh.edu

4 The author has taken the liberty of "copying" many definitions, as provided by Acronym Software, Inc., of the SODA terminology. They are short and descriptive. There was no need to re-invent the wheel.
The current version of SODA, Version 3.2.5 (April 20, 1994 Release) can handle a maximum of 1,000 members, 1,000 nodes, 200 groups, and 100 load combinations. SODA contains extensive context-sensitive Help facilities. Thus, a detailed manual for the software is redundant, so long as the use of a "pull down menu" structure is understood. The purpose of this Guide is to provide basic information on the use of SODA. Some of the sophisticated options of the software are not described. In order to use these options, the user first must develop a mastery of the other features. After that through the use of the context-sensitive help screens, these additional features can be mastered.

In the structural analysis mode, either 2D or 3D, a working knowledge of structural analysis concepts may suffice. However, in the design or design verification mode the user must have full mastery of structural steel design, with either US or Canadian Specifications. Inappropriate "educated guesses" may lead to serious code vs. construction violations.

In the use of SODA it is essential that the user employ a mouse. Any other means of interaction with the software is too tedious to consider.

2. MASTER MENU
When the software is running, initially a title page appears. When this title is okayed, the master menu of the program appears. At the top there will be list of options, each of which contains sub-and sub-sub-menus. The top line, i.e., the SODA Menu Bar, will contain the following:

FILE GENERAL STRUCTURE LOAD VIEW OPTIONS RUN HELP

Each of these entries is subject-oriented. A brief description of each item follows:

FILE  Commands for loading and saving worksheets.
GENERAL Commands for describing the project worksheet.
STRUCTURE Commands for creating and modifying structures.
LOADS Commands for creating and modifying loads.
VIEW Commands for viewing structure.
OPTIONS Commands for setting up options for viewing.
RUN Commands for running the engine, i.e., SODA-engine.
HELP Command for accessing SODA help.

3. FILE COMMANDS
If you move the cursor to FILE, and click the mouse the following menu will appear. A brief explanation of each of these sub-menu commands is as follows:

A Guide to SODA (Version 3.2.5), by C. N. Kostem 2
NEW
Starts a new SODA worksheet.

OPEN
Opens an existing SODA worksheet.

SAVE
Saves the current worksheet under the name that appears in the title bar.

SAVE AS...
Saves the current worksheet under a new name.

ERASE
Erases any or all output associated with a SODA worksheet.

PRINT TOPOLOGY
Prints the current displayed structure.

PRINT OUTPUT
Prints a SODA output files.

PRINTER SETUP
Changes printer selection or printer options.

EXIT
Exits from SODA to WINDOWS.

3.1 Print Output
When the PRINT OUTPUT command is issued, the following sub-sub-menu appears:

INPUT ECHO
ANALYSIS RESULTS
FINAL SUMMARY
NORMAL REPORT
MEDIUM REPORT
DETAILED REPORT
EFFECTIVE LENGTHS
DESIGN HISTORY
WARNINGS
ERRORS

After experimentation with a few simple problems, one can get a better grasp of the contents of each of these files. The most important one is the "Input Echo." Inspection of this file reveals whether any major errors were committed!

4. GENERAL COMMANDS
If you move the cursor to General, and click the mouse the following menu appears. A brief explanation of each of these sub-menu commands is as follows:

TITLE
Name and brief description of the project (optional). The name will appear before all
major printed files. The description will be printed only in the "input echo" file.

**DEFINITION**

Define the major characteristics of the project. This entry is provided for a brief description of the project. Any salient data to refresh the user's memory should be entered in this block.

**SECTION DATABASE**

Define the database, if any, which will be used to define the structural steel cross-sections. US, i.e., AISC, and Canadian structural sections are already included in the databases. The user can generate another database, if needed.

**UNITS**

Identification of the units that will be used in the definition of the project. Built-in options are provided for "Customary Old English," i.e., Imperial Units, or the SI units.

**FABRICATION**

Fabrication/erection-related systematic assumptions.

4.1 **Definitions**

Each of the above sub-menus of the DEFINITION contain sub-sub-menus of its own, as shown below:

**DIMENSION**

Project definition identifies whether the problem is two-dimensional or three-dimensional.

**STRUCTURE**

Project definition identifies whether the problem is a frame or a truss. If the project contains a few or numerous "end-releases," i.e., shear connections, it should be defined as a frame, and the "moment end-releases" must be applied.

**BEHAVIOR**

Identify whether the project is to be analyzed as a "first-order," i.e., equilibrium and compatibility equations will be based on undeformed geometry, or whether the "P-Delta" effects are to be approximated.

**SIDESWAY**

It is either PERMITTED or PREVENTED. Depending upon the structure and loading in question, this decision must be reached by the user. In "high-rise buildings with substantial lateral loads, the use of P-Delta effects for the final design or
design verification" will be a judicious approach.

PROBLEM TYPE There are three possible options:

Design
Verify
Analysis

The third option, i.e., the analysis, is independent of any CODE or SPECIFICATION. However, both the design and the verification options must be in accordance with some code. The existing specifications are the U.S. AISC AISC-78, AISC-89, and AISC-LRFD codes, or the Canadian CAN3-S16.1-M84 or CSA-S16.1-M89 codes. It should be noted that U.S. AISC AISC-78, and AISC-89 are based on the "working stress design approach."

FABRICATION Requires definitions in the case of bolted connections (e.g., What is the bolt hole diameter?), use of double angle gusset plate (i.e., $A_e/An=?$), and whether Class-H sections are used.

5. STRUCTURE
If you move the cursor to Structure, and click the mouse the following menu appears. A brief explanation of each of these sub-menu commands follows:

NODES Define the nodes, or more colloquially "joints," in the structure.

GROUPS Define the groups of members in the structure. Here, the "group" refers to different cross-section.

MEMBERS Define the members in the structure.

REGULAR FRAMEWORK Generate a regular framework.

DIAGONAL BRACING Generate a diagonal bracing for a regular framework.

MOVE NODES... Move the nodes in the structure.

Each of the above sub-menus of STRUCTURE contains sub-sub-menus of its own, as discussed in the following sub-sections.

A Guide to SODA (Version 3.2.5), by C. N. Kostem
5.1 Nodes
If you move the cursor to Nodes, and click the mouse the following menu appears:

<table>
<thead>
<tr>
<th>Node Name</th>
<th>X-Coord.(ft)</th>
<th>Y-Coord.(ft)</th>
<th>Z-Coord.(ft)</th>
<th>Support Type</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>None</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Fixed</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Pinned</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Pinned(except X)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Pinned(except Y)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Pinned(except Z)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Roller(along X)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Roller(along Y)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Roller(along Z)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Other (sub menus</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>for springs.)</td>
</tr>
</tbody>
</table>

5.2 Groups
If you move the cursor to Groups, and click the mouse the following menu appears:

<table>
<thead>
<tr>
<th>Group</th>
<th>Shape</th>
<th>Designation</th>
<th>Modulus(ksi)</th>
<th>Depth(in)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Young's Min</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Shear Max</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Stress(ksi)</td>
<td>Max. Allow. KL/r</td>
<td>Compression</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Yield</td>
<td>Ultimate</td>
<td>Tension</td>
</tr>
</tbody>
</table>

5.3 Members
If you move the cursor to Members, and click the mouse the following menu appears:

<table>
<thead>
<tr>
<th>Member</th>
<th>Start Node</th>
<th>End Node</th>
<th>Group</th>
<th>Beta Angle</th>
<th>Joint Type</th>
<th>Effective Length Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>+---------+---------+-------+------------</td>
<td>------------</td>
<td>-------------</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>o--------+o--------+</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>+--------+o--------+</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>o--------+o--------+</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Regular Framework
(Extensive pull-down sub menus; not discussed here.)

Diagonal Bracing
(Extensive pull-down sub menus; not discussed here.)
6. LOADS
If you move the cursor to Loads, and click the mouse the following menu appears. A brief explanation of each of these sub-menu commands follows:

**NODAL LOADS**
Define the nodal loads on the structure, which could be forces or moments.

**MEMBER LOADS**
Define member loads on the structure, which could be uniformly distributed load, concentrated force, or linearly varying distributed load from one point on the beam to another.

**TEMPERATURE LOADS**
Only uniform temperature change for the full member can be considered.

**SUPPORT SETTLEMENTS**
Node point settlements can be translational or rotational.

**AREA LOADS**
(This is used for Regular Framework. Not discussed here.)

**LOAD COMBINATIONS**
Define the load combinations for the structure.

**DISPLACEMENT LIMITS**
Define the displacement limits for selected nodes for selected load combinations. (Not discussed here.)

6.1 Nodal Loads
If you move the cursor to Nodal Loads, and click the mouse the following sub-menu appears:

- Specified Nodal Load (name of the node)
- Loaded Node (name of the node)
- Forces and Couples
  - Force X (kip)
  - Force Y (kip)
  - Force Z (kip)
  - Couple X (kip-ft)
  - Couple Y (kip-ft)
  - Couple Z (kip-ft)
6.2 Member Loads
If you move the cursor on Member Loads, and click the mouse the following sub-menu will appear:

Specified Member Load (name of the loading)

Loaded Member (name of the member)

Load Type
- Full Uniformly Distributed Load
- Point Load (i.e., concentrated force)
- Other Dead Load

Load Information
- \( w @ a: \) (kip/ft)
- Distance \( a/L: \)
- \( w @ b: \) (kip/ft)
- Distance \( b/L: \)
- Orientation
  - \( X \)
  - \( Y \)
  - \( Z \)
  - (axial)
  - etc.

Temperature Load
- Specified Load Name
- Loaded Member (i.e., member's assigned name)
- Temperature Change (Degree F)
- Thermal Expansion Coefficient (/Degree F)

Support Settlement
- Support Settlement Load Name (NOTE: The support settlement should be treated as a load case.)
- Displacement
  - \( X \) (ft)
  - \( Y \) (ft)
  - \( Z \) (ft)
- Rotation
  - \( X \) (radians)
  - \( Y \) (radians)
  - \( Z \) (radians)

Area Loads (Not discussed here.)

Load Combinations
- Name of the combination
- Loads
  - Load Name
  - Load Type
Load Factor
Load Factor (Numerical value should be entered.)

Displacement Limits  (Not discussed here.)

7. VIEW
If you move the cursor to View, and click the mouse the following menu appears. A brief explanation of each of these sub-menu commands is as follow:

**VIEW TOPOLOGY**
View the topology of the structure.

**VIEW DEFLECTIONS**
View the deflection of the structure.
(Note: Cubic-spline is not yet available. Thus, all "deformed" members are depicted in straight lines.)

**VIEW MOMENT AND SHEAR**
Moment and shear diagram of members for any given load case.

**VIEW OUTPUT FILES**
All ASCII input and output files can be viewed. (NOTE: This command is similar to FILE/PRINT OUTPUT COMMAND.) After viewing the files, they can be ported to the printer.

**ZOOM IN**
Doubles the size of the current view (NOTE: This command can be repeatedly issued in order to zoom into a small area.)

**ZOOM OUT**
Halves the size of the current view. (NOTE: This command is opposite the ZOOM IN command.)

**ZOOM RESET**
Resets the zooming to 100%.

7.1 View Deflections
If you move the cursor to View Deflections, and click the mouse the following query appears:

Load Combination
Scale Factor

If you move the cursor on View Moment and Shear, and click the mouse the following query will appear:

Member Name

A Guide to SODA (Version 3.2.5), by C. N. Kostem
Load Combination Name
Strong or Weak Axis? (Moment and shear diagrams of the member for the given load combination will be graphically displayed. Numerical values of Min/Max values of moment, shear and axial forces will be displayed. There is no provision to print the content of these screens!)

7.2 View Output Files
If you move the cursor to View Output Files, and click the mouse the following menu appears:

Input Echo
Analysis Results
Final Summary
Normal Report
Medium Report
Detailed Report
Effective Lengths
Design History
Warnings
Errors

8. OPTIONS
If you move the cursor to Options, and click the mouse the following menu appears. A brief explanation of each of these sub-menu commands is as follows:

GRAPHICS Options for turning axes on and off.
GRAPHICS VIEW Options for setting 3D viewing options.
GRAPHICS TEXT Options for setting text on graphics displays.

8.1 Graphics View
If you move the cursor to Graphics View, and click the mouse the following menu appears:

Isometric View
Projection View
Plan-X
Plan-Y
Plan-Z
Horizontal Angle
Vertical Angle

A Guide to SODA (Version 3.2.5), by C. N. Kostem 10
8.2 Graphics Text
If you move the cursor to Graphics Text, and click the mouse the following menu appears:

- Font (Has a pull-down menu.)
- Size (This is the size of the "lettering." Has a pull-down menu.)

Display
- Member Names (Yes/No)
- Group Names (Yes/No)
- Node Names (Yes/No)
- Worksheet Title (Yes/No)
- Align Text with Members (Yes/No)

9. Run
If you move the cursor to Run, and click the mouse the following menu appears. A brief explanation of each of these sub-menu commands are as follow:

Run Engine   Runs the SODA Engine.
Review Input Generates an input echo file and displays it.

9.1 Run Engine
If you move the cursor to Run Engine, and click the mouse the following menu appears:

- Displacement Constrained (Yes/No)
- Constraint Factor (Enter a numerical value, otherwise the value of 1.00 will be assumed.)
- Run Time Options (Defines output format to be used, i.e., FM.N or EM.N.)
  - Free
  - Exponential

Optimization
  On Both the members and the "groups," i.e., the cross sections, will be selected in such a manner that the total structural weight of the project will be minimum.
F.E.L. Report No. 400.52

Off Members will be designed one-by-one in accordance with the declared "steel design specification."

Bending Coefficients (Refer to a structural steel design manual for the meaning of the term!)
Calculate
Unity

Report Detail
Normal
Medium
Detailed

Behavior Type
First Order
P-Delta (Meaning and ramifications of P-Delta analysis; refer to an advanced structural analysis text.)

Design Process
Continuous
Stage-by-Stage

10. HELP
If you move the cursor to Help, and click the mouse the following menu appears. A brief explanation of each of these sub-menu commands follows:

CONTEXT SENSITIVE Recommends the use of <F1> key!
CONTENTS Provides a list of major help topics.
HOW TO USE HELP Help on help!
ABOUT SODA Information about SODA.