1994

A guide to soda, July 1994.

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

F.E.L. Report No. 400.52

A GUIDE TO SODA[1]

(STRUCTURAL, OPTIMIZATION, DESIGN, and ANALYSIS)[2]

by

Celal N. Kostem[3],[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.

---

[1] 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.

[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:

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 compatibilty 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., Ae/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.

5.1 Nodes
If you move the cursor to Nodes, and click the mouse the following menu appears:

Node Name X-Coord.(ft) Y-Coord.(ft) Z-Coord.(ft) Support Type
None
Fixed
Pinned
Pinned(except X)
Pinned(except Y)
Pinned(except Z)
Roller(along X)
Roller(along Y)
Roller(along Z)
Other (sub menus for springs.)

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

Group Shape Designation Modulus(ksi) Depth(in)
Young's Min
Shear Max

Stress(ksi) Max.Allow. KL/r
Yield Compression
Ultimate Tension

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

Member Start Node End Node Group Beta Angle Joint Type
+-----+
o-----o
+-----o
o-----+

Effective Length Factor
Kx
Ky
Bt
Bb

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

Diagonal Bracing
(Extensive pull-down sub menus; not discussed here.)

Move Nodes
(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

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

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.**

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.