SASSI - Structural Analysis Software System Interface

Lee Abbuhl
Viswa Padma_nabhan

Boeing Computer Services
Wichita, Ks.

Abstract

SASSI contains a set of tools for structural engineering analysis in a distributed workstation environment. These tools provide 1) access to a host of engineering applications software through a single command 'SASSI' entered in a workstation window. Analysis data post-processing, network tasks, plotters and printers, and additional system and analysis tools are available and accessed by a set of pop-up menus and dialog boxes in the menu. 2) Data transfers between the various applications are very efficient in SASSI. 3) An interactive menu interface provides transparent access to engineering computing tasks and applications that enhances the productivity of stress engineers.

Some of these will be illustrated by examples using MSC/NASTRAN.

Introduction

SASSI (Structures Analysis Software System Interface) was credited by Boeing Engineering Management for providing a 10 to 1 productivity improvement for the 777 program over the methods used for the 767 program. The economic consequences of the software measure in the millions.

The nucleus of the first generation set of programs is a post-processor, which was designed by one of our most experienced stress engineers and a veteran PATRAN user, for displaying MSC/NASTRAN and ELFINI outputs in a consistent format. This program incorporates many equations and styles of presentation that represent the designer's analysis style. The program has all of the usual buttons and switches for controlling the information build-up, and rotating the model, etc. It supports postscript, HP 7550 color plot, and HP draftmaster printing for documentation. The software replaced a manual process used on the 767 program of hand drawing freebody plots by transcribing data from MSC/NASTRAN listings. Since it replaced a manual process, its most significant contribution was in reduction of engineering errors and in improving the quality of the airplane.

The software includes code that identifies the critical load case envelop and it has a seperate graphics screen for displaying the model where the elements are displayed with the load case numbers in which the maxima occured.

SASSI is a client/server program. A portion of SASSI runs on the CRAY Y-MP, VAX or IBM Mainframe which hosts the MSC/NASTRAN software. MSC/NASTRAN output data is processed on the host and the results are sent to UNIX workstations for display.

The program was written at a time when the company had just acquired a large number of UNIX workstations and most of the engineering community had little exposure to the UNIX operating system. So SASSI was written to provide a graphical user interface to all of the company's internal and commercial engineering software and to provide as many other convenience features as possible. For example, the main screen provides menus to print, get the status of, and kill print jobs. Similarly, it automates many network tasks including job submission to the Cray Y-MP, VAX, and the IBM mainframe hosts. When the host jobs complete, the results are sent back to the submitting workstation, and the user is notified of the arrival.

SASSI was one of the first generation of graphical user interface programs which dramatically improved engineer productivity. The second genration programs have more promise than the first because they help simplify the end-to-end engineering process.

For instance, second generation extensions of the SASSI program tie together several commercial and Boeing developed programs to provide a first time ever integrated set of tools for fatigue and ultimate analysis.

Access to many commercial and Boeing developed programs is provided. The network of data paths between programs is better developed.

Various data entry spreadsheets are used in the different Motif screens. SASSI has a save and restore capability so that all the variables and graphics in effect in one segment of the program reappear the next time the engineer runs it.

The following figures provide a quick look at the SASSI menu structure.

SASSI

SASSI - Structural Analysis Software System Interface is a transparent computing environment that contains two sets of softwares: One set that are used for general purpose data management, and the other for engineering applications.

General purpose data mangement software

+ can do Network Tasks (NEAT),
+ produces prints & plots from various devices in the system (PLOTS),
+ prepares documents (AUTODOC),
+ helps in Data Exchange (MOEDMS),
+ gives access to database (SADS),
+ generates JCL packages (JCLGEN).

Engineering applications software

+ post-processes Finite Element Analysis results (FERMI)
+ gives access to
  . various engineering applications (STRAPPS)
  . PC Software (PCAPPS).

For illustrating SASSI, this paper describes the codes used and the necessary steps on the CRAY, VAX or IBM MSC/NASTRAN host in the post-processing of a MSC/NASTRAN run.

FERMI

FERMI is the Finite Element Results and Modeling Interface, which can pop-up the Nastran Post-process Menu. Some of the options in this menu are NASFREE, NASPLOT, FREEBODY, MAXMIN, Maxminplot, Freebodyplot.

NASFREE

Nasfree performs two steps required to post-process MSC/NASTRAN analysis data. The first reformats the Nastran bulk data file into geometry data files. The second step processes the Nastran OUTPUT2 file and creates a set of load case results files.

NASPLOT

NASPLOT provides the facility to select a portion out of a large Nastran model for downstream analysis processing. It displays on screen the geometry of a Nastran model. It also allows user to

a) manipulate the viewing of the model by rotation along the x, y, z-axes;
b) inquire the geometric coordinates of the grid points and the vertices of the elements by either a keyboard input of the id or a mouse picking of a grid point or an element.
c) turn grids and elements and their ids on or off.
d) trim away elements by either a keyboard input of the element id, by a mouse picking of an element or by defining a trim window;
e) scale down the model by defining a window;
f) create an output file of the extracted elements in the scaled down model; and
g) create output hardcopy plotter files for the appropriate plotter/printer.

FREEBODY

FREEBODY creates Freebodyplot data files from NASFREE results and models geometry data. For large models, it is recommended to execute this program on either CRAY-

YMP or IBM mainframe where Nastran and Nasfree were executed. For small models it is available in the workstations.

The input:

a) Nasfree results file.
b) Model geometry files.
c) NASPLOT elements extract file.

The output:

Freebodyplot file for the submodel.
DISPLOT file for the submodel.
MAXMIN input file for the submodel.

Freebodyplot

Freebodyplot graphically displays a structural analysis freebody of finite element results and shows the model geometry, model elements, the applied and resultant forces acting on grids and elements, and element stresses. Additional information, such as grid ids, element axis, and force values, is also displayed. Display features which include windowing, zoom and 3D-rotations are also available. Data files used by Freebodyplot are created by finite element results processor FREEBODY.

Post-processing capabilities include

internal loads,
grid forces, and
element and coordinate properties.

Display and hardcopy capabilities include

color graphics display,
element query,
selected element display,
menu-driven user interface,
windowing and rotation, and
hardcopy output.

MAXMIN

MAXMIN finds the maximum and minimum loads in each element (listed in the extract element file) from the freebodyplot files.

Input:

a) Freebodyplot file for the submodel.
b) MAXMIN file for the submodel.

output:

Maxminplot files for the submodel.

Maxminplot

Maxminplot graphically displays a structural analysis freebody of finite element results from a set of different load cases and shows

model geometry,
model elements, and
either the maximum or minimum loads and stresses in the model elements.

Additional information, such as grid ids, element axis, and force values, is also displayed. Display features, which include windowing, zoom and 3D rotations, are also available in Maxminpplot. Data files used by Maxminplot are created by the filite element results processor FREEBODY and MAXMIN.

Steps for post-processing a MSC/NASTRAN run

Step 1) Set up base directory:

+ create a seperate directory for each Nastran model.
+ copy the MSC/NASTRAN input file to this directory.
+ run MSC/NASTRAN after reviewing "nastran restrictions" and "nastran setup".
+ place the resulting binary OUTPUT2 file in the directory created.

Step 2) Run SASSI on Nastran host to reformat the bulk data file:

+ run SASSI and Select "NASFREE" - "BDF".

. This reformats the bulk data file and performs coordinate transformations to put all GRIDs in terms of default coordinate system. Spherical and cylindrical coordinates are transformed to rectangular.

. The user is prompted for a six-character model name and this becomes the first six characters of the output files. In general the user does not need to remember which files are produced by SASSI - only the model name.

The following files are produced by the reformat step.

ASCII files

<model>abdf.els - images of the element BDF cards.
<model>abdf.grd - grid cards.
<model>abdf.geo - concatenation of the above files.

Binary files

<model>bbdf.els
<model>bbdf.grd
<model>bbdf.geo

The binary files are used as input to other portions of the SASSI program. The ASCII <model>.abdf.geo is to be sent to an RS6000 workstation as input to the NASPLOT program, which plots the geometry and allows electronic scissoring of a subsection of the model of interest to an individual engineer.

Step 3) Run SASSI on the MSC/NASTRAN host to split the output binary file into data type and subcase files.

Eight files are produced for each subcase or load case. The user need not remember these files but they are

<model>xxxx.dis - displacement results files for each subcase.
<model>xxxx.elf - Element force results files.
<model>xxxx.bem- Beam force results files.
<model>xxxx.els - element stress results files.
<model>xxxx.rxn - grid point reaction results files.
<model>xxxx.app - grid point force balance applied loads files.
<model>xxxx.spc - grid point force balance reaction files.
<model>xxxx.elm -grid point force balance element forces.

where xxxx = integer subcase number in the range 1 - 9999.

a) These files constitute the "database" for the model and should never be copied to the workstation. They are binary incompatible with the workstation and can't be used for anything.

b) During the execution of Nasfree, the file 'nasfree.dat' is written to record the number of subcases run, the subcase numbers and their subtitles.

c) database now created.

1. go to workstation to create an EXTRACT file which defines a subsection of the model.
2. send it back Cray,Vax or IBM Nastran host.
3. Log back into the Nastran host and run FREEBODY and MAXMIN.

Step 4) The BDF reformat and MSC/NASTRAN output2 file split up steps above on the Nastran host create a SASSI database for this model. The files created are now ready for input to the FREEBODY and MAXMIN steps. But before these steps can occur the user must log into the RS 6000 workstation and bring up the SASSI GUI (Graphical User Interface).

That done, the user clicks on NASPLOT , plots the geometry and selects a subsection (bulkhead, wing etc) of the model by a rubber banding operation. This creates an ASCII EXTRACT file . This file is then copied back to the Nastran host via ftp.

Step 5) Final steps on the Nastran host.

Once the EXTRACT file is copied to the MSC/NASTRAN host, two final steps are made before downloading the corresponding SASSI output Freebody and Maxmin files to the workstation for graphical display. FREEBODY is run against all load cases. MAXMIN is then run to decide which of the loadcases are critical and then the maxmin ouput files and the critical loadcase files are copied to the workstation for graphical display via Freebodyplot and Maxminplot.

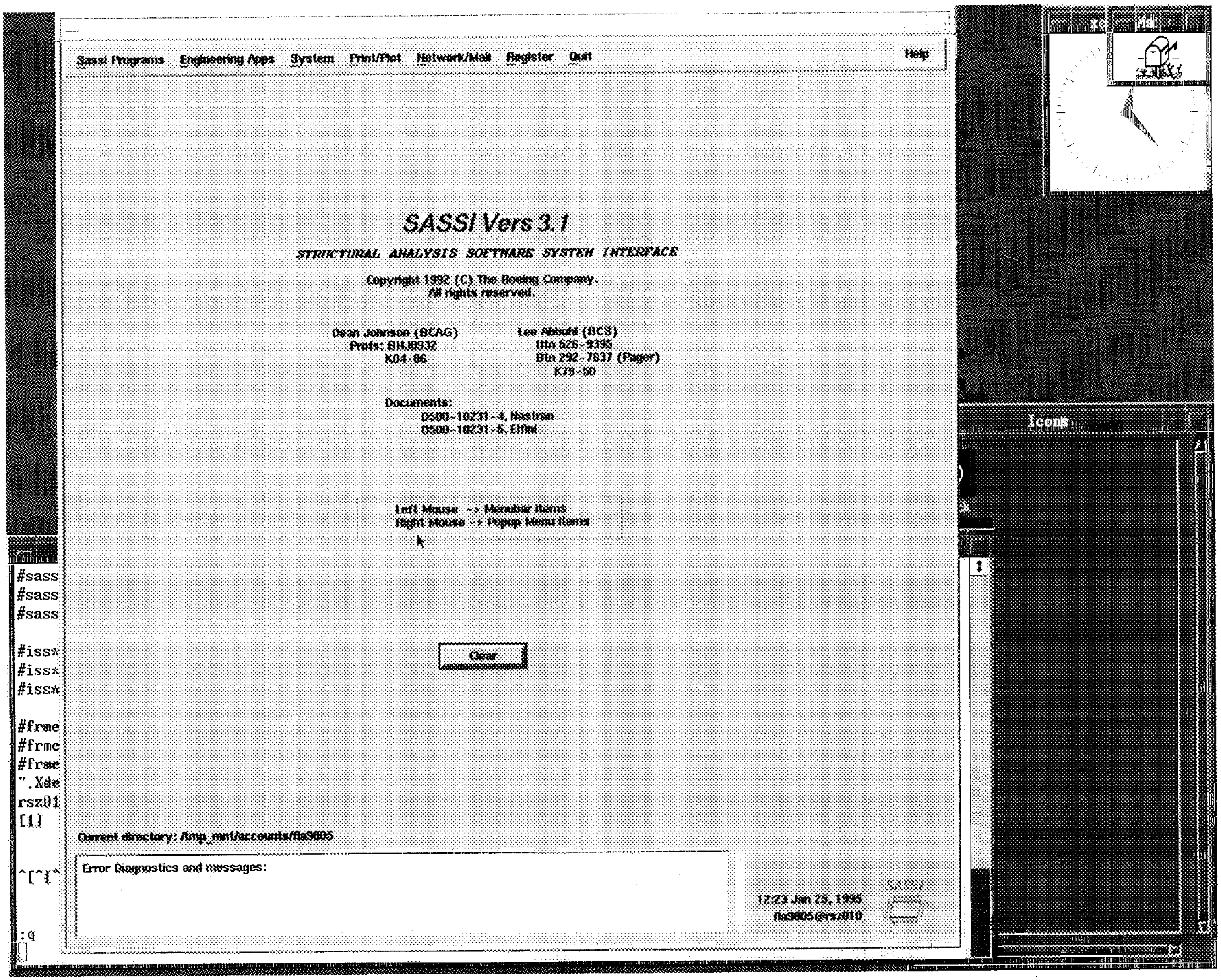

Figure 1 - Main Menu

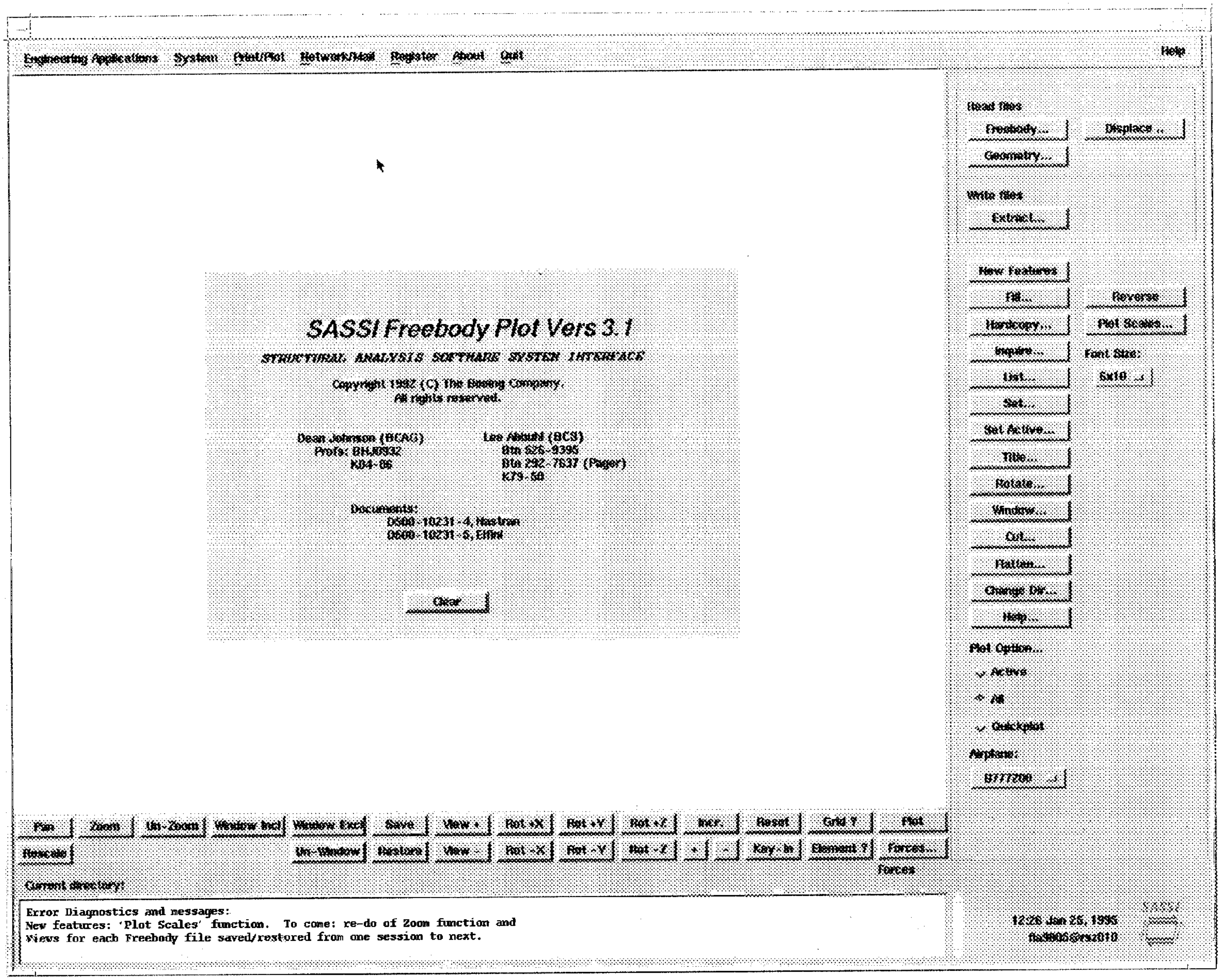

Figure 2 - Freebody Plot Main Menu

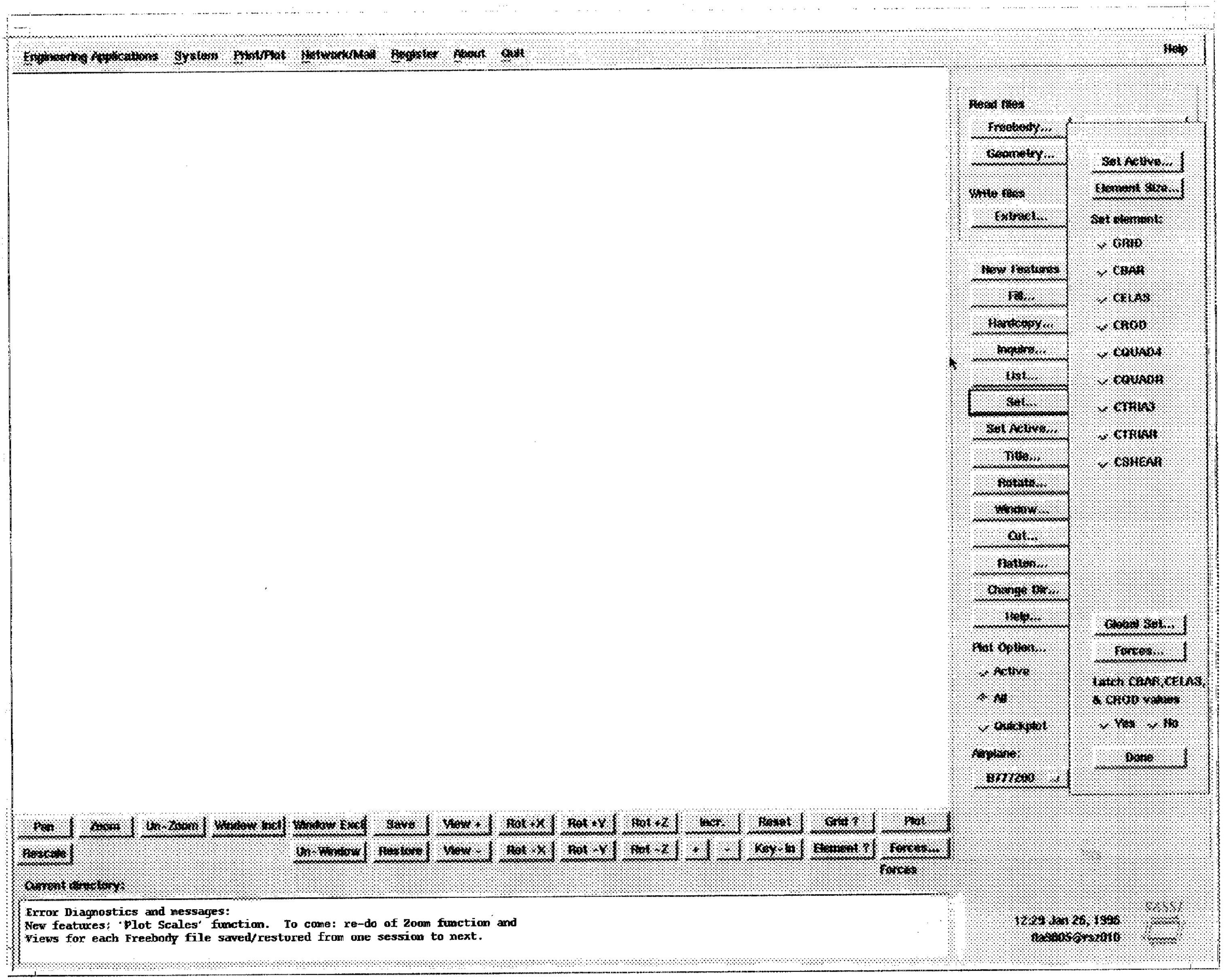

Figure 3 - Form that pops up after "Set" button presses

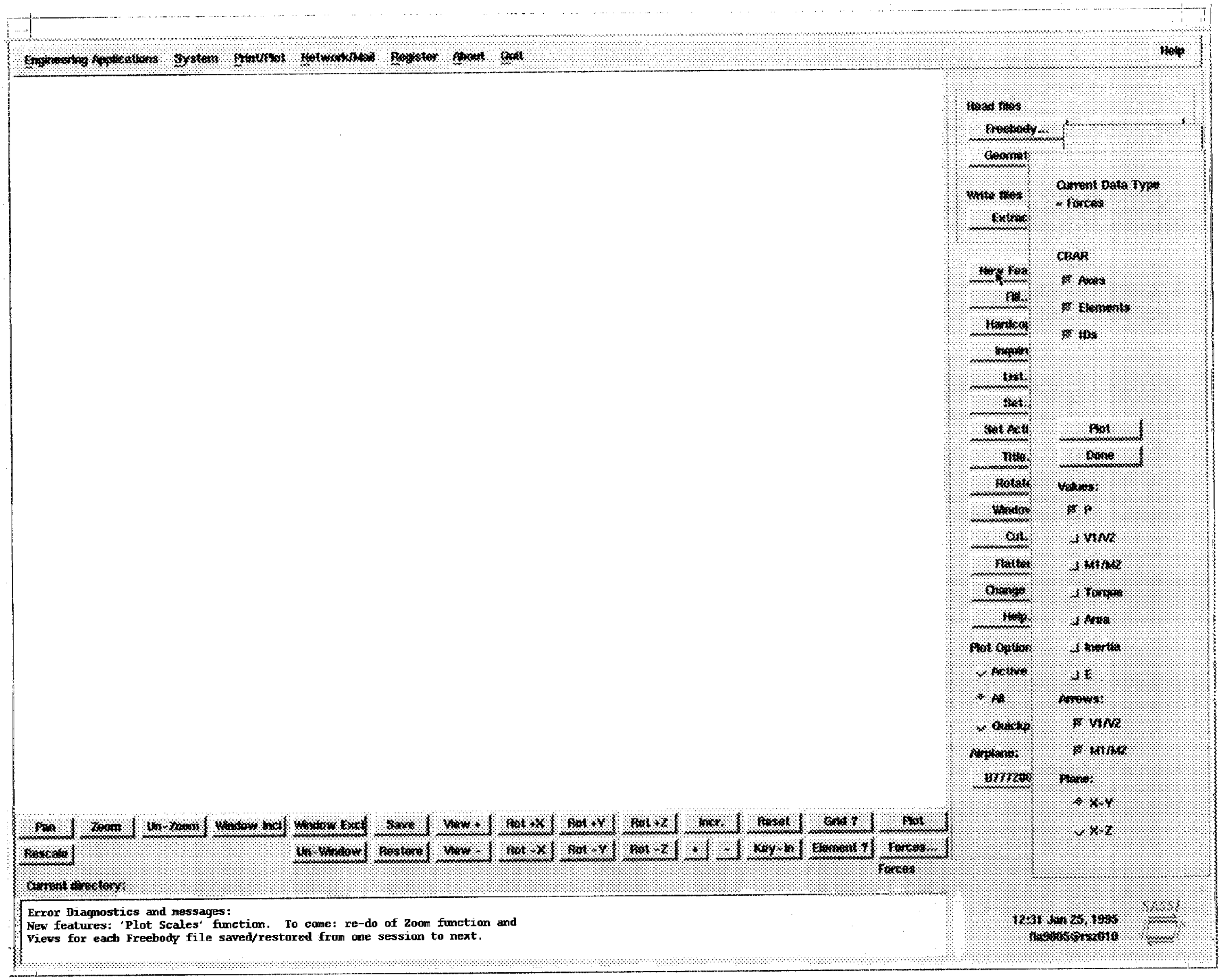

Figure 4 - Form that pops up after "CBAR" clicked.

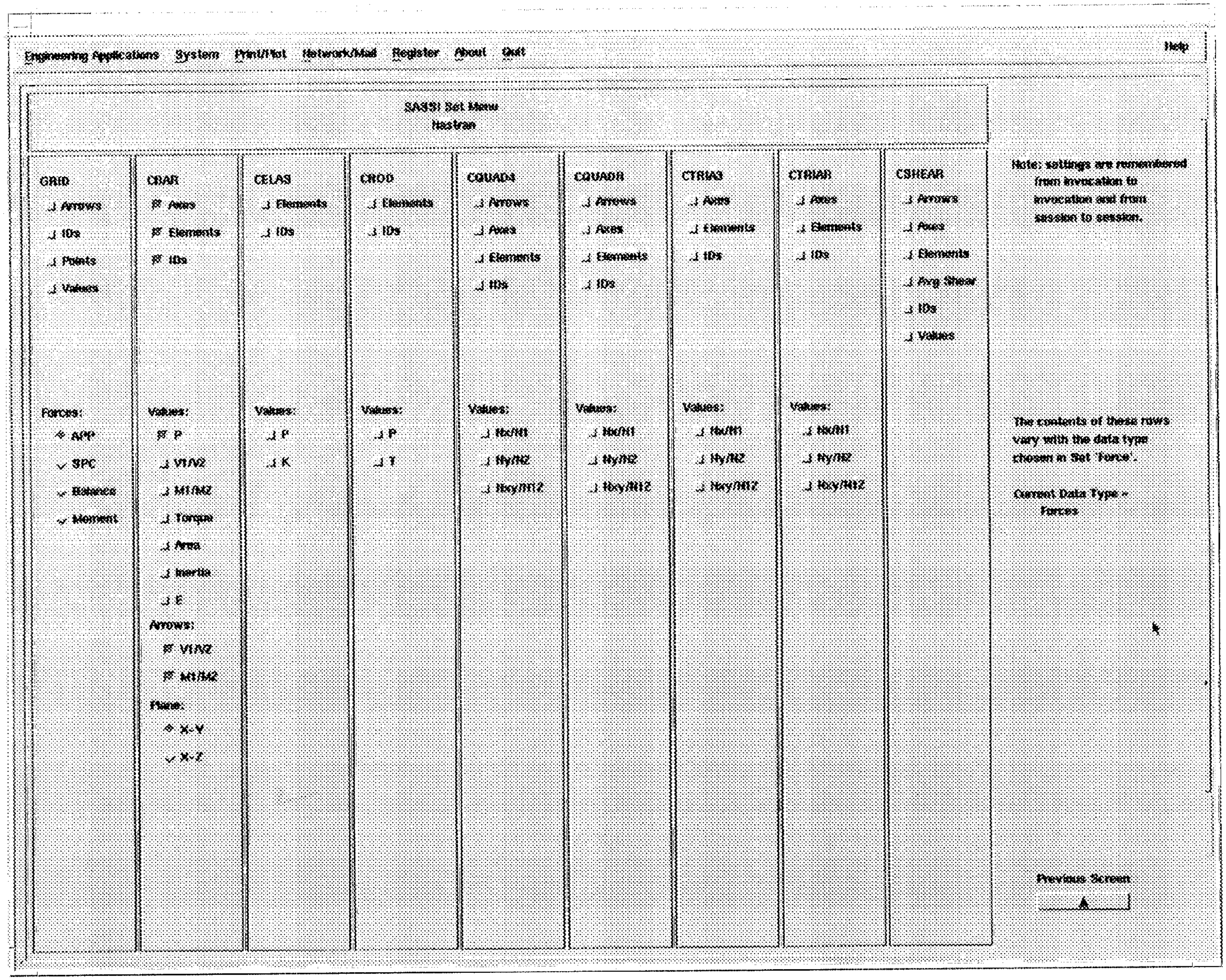

Figure 5 - Form that pops up after "Global Set" button presses

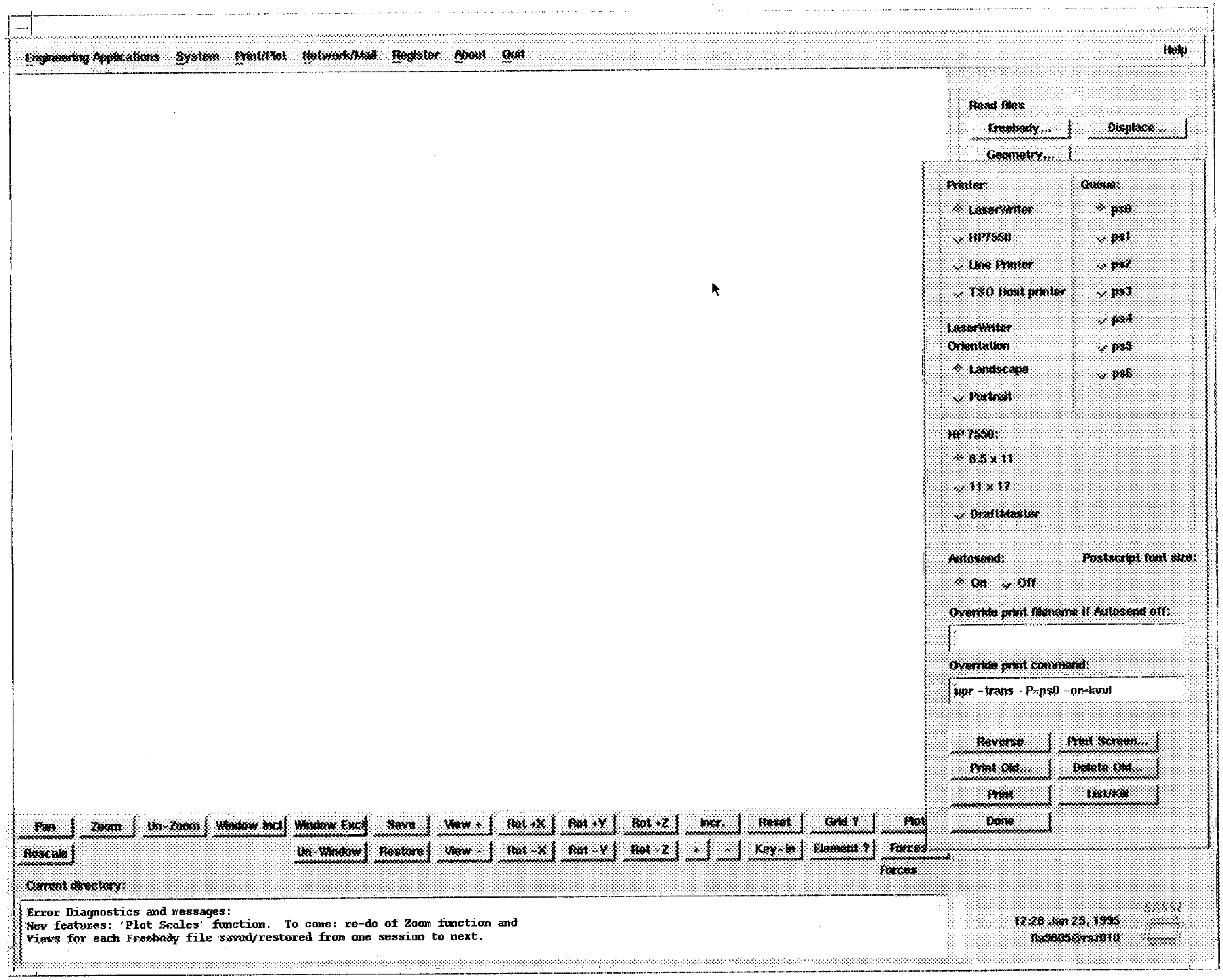

Figure 6 - Form that pops up when "Hard copy" button presses on Freebody Plot Main Menu