

PFC/RR-91-3

**SOLDESIGN USER'S MANUAL<sup>©</sup>**

**Robert D. Pillsbury, Jr**

Plasma Fusion Center  
Massachusetts Institute of Technology  
Cambridge, MA

**FEBRUARY, 1991**

**PURPOSE**

**SOLDESIGN** is a general purpose program for calculating and plotting magnetic fields and Lorentz forces, and for calculating resistances and inductances for a system of coaxial, uniform current density solenoids. **SOLDESIGN** is available on both the CRAYs of the MFE network at Livermore, CA, and on the VAX cluster system at the Plasma Fusion Center at MIT. There is also a version available for PC's with reduced dimensions and no graphics.

This work has been supported in part by DOE contracts, EX-76-A-01-2295, DE-AT01-76ET10813, DE-AC02-78ET51013, DE-AC22-84PC70512, and DOE Grant DE-FG02-91ER054110.

## TABLE OF CONTENTS

| <u>TITLE</u>                          | <u>PAGE NO.</u> |
|---------------------------------------|-----------------|
| DISCLAIMER                            | iii             |
| LOCATION                              | iii             |
| INTRODUCTION                          | 1               |
| HOW TO USE SOLDESIGN                  | 2               |
| AVAILABLE COMMANDS                    | 3               |
| DESCRIPTION OF SOLDESIGN COMMANDS     | 6               |
| ENDLIST                               | 6               |
| FIELD                                 | 7               |
| FILE                                  | 8               |
| FORCE                                 | 9               |
| HELP                                  | 10              |
| INDUCT                                | 10              |
| LOOPSW                                | 11              |
| NOLIST                                | 11              |
| PEAKFIELD                             | 12              |
| PRINT                                 | 12              |
| READ                                  | 12              |
| READCURR                              | 13              |
| READFIELD                             | 14              |
| READFORCE                             | 14              |
| RESIS                                 | 14              |
| SAVE                                  | 15              |
| SAVEFORCE                             | 15              |
| SETUP                                 | 16              |
| START                                 | 19              |
| STOP                                  | 19              |
| TERMINAL                              | 19              |
| UNITS                                 | 19              |
| SOLDESIGN EXAMPLES                    | 20              |
| PLOTTING WITH SOLDESIGN               | 23              |
| COIL PLOTTING                         | 27              |
| CONTOUR PLOTTING                      | 29              |
| FUNCTION PLOTTING                     | 33              |
| VECTOR PLOTTING                       | 36              |
| HINTS AND FURTHER EXPLANATIONS        | 37              |
| ERROR MESSAGES                        | 40              |
| REFERENCES                            | 42              |
| HANDY DANDY POCKET GUIDE TO SOLDESIGN | 43              |

## DISCLAIMER

This program and documentation is provided as is. No warranty is implied or stated by the author, the Plasma Fusion Center, or MIT with respect to the accuracy of the program, the documentation, or any results obtained from using the program.

Additions to this document usually lag additions to the program. The author makes a real attempt to make changes that are compatible with the previous generations of the code. If, however, something that worked or something the document says should work does not, contact the author.

## LOCATION

The definitive VAX version of the executable and the document can always be copied from:

XANADU::DKB100:[RDPJ.SOL]SOLDESIGN.EXE

and

XANADU::DKB100:[RDPJ.SOL]SOLDESIGN.DOC

For the convenience of users at PFC, a copy of both reside in:

USER5:[RDPJ.SOL]SOLDESIGN.EXE

and

USER5:[RDPJ.SOL]SOLDESIGN.DOC

These files are directly addressable from ALCVAX and NERUS using DFS (Disk File Server).

The CRAY versions reside in a library SOLLIB that can be obtained from CFS (Common File Storage) by issuing the command:

```
CFS GET 11032/PUBLIC/SOLLIB END
```

The library contains a file README describing the library and any updates, and an example of input and output files as well as the document and source. It also contains an executable XSOLx where x is the machine that the executable was generated on. There may be more than one of these executables, each for a different machine.

## INTRODUCTION

**SOLDESIGN** is a general purpose program for calculating and plotting magnetic fields, Lorentz body forces, resistances and inductances for a system of coaxial uniform current density solenoidal elements. The program was originally written in 1980 and has been evolving ever since. **SOLDESIGN** can be used with either interactive (terminal) or file input. Output can be to the terminal or to a file. All input is free-field with comma or space separators. **SOLDESIGN** contains an interactive help feature that allows the user to examine documentation while executing the program.

Input to the program consists of a sequence of word commands and numeric data. Initially, the geometry of the elements or coils is defined by specifying either the coordinates of one corner of the coil or the coil centroid, a symmetry parameter to allow certain reflections of the coil (*e.g.*, a split pair), the radial and axial builds, and either the overall current density or the total ampere-turns (NI). A more general quadrilateral element is also available. If inductances or resistances are desired, the number of turns must be specified. Field, force, and inductance calculations also require the number of radial current sheets (or integration points). Work is underway to extend the field, force, and, possibly, inductances to non-coaxial solenoidal elements.

Printed output can consist of the coordinates of the field point(s), the flux, and the components of the magnetic flux density. If forces are desired, they can be output as a total force per unit length or per radian acting on a coil due to the coil itself and all other coils. In addition, if the user desires, the body force densities at a number of points can also be printed. Similarly, if desired, the force produced on a coil by each additional coil in the problem may be printed separately (*i.e.*, an influence coefficient). There is also a feature associated with the plot package that will allow the coordinates of a contour of constant flux or flux density to be printed instead of plotted - *i.e.*, a contour follower.

Plotted output can be generated within **SOLDESIGN**. Available plots are coils, flux lines, contours of constant flux density components, or contours of constant field homogeneity, field or force vectors and function plots of flux or flux density components versus position (along constant coordinate lines (*e.g.*,  $B_z$  versus  $r$  for fixed  $z$ )). The coils can be drawn on the contour plot. The underlying plot package used by **SOLDESIGN** is **GRAFLIB** developed on and for the MFE network at Livermore and also available on the VAXs.

For the purposes of this manual, alphabetic commands are printed in upper case letters to differentiate them from numeric variables (written in lower case). **However, in actual use, all commands must be entered in lower case.** Program prompts are also printed in upper case letters.

In addition, the terms "coils" and "elements" will be used interchangeably in this manual. For example, passive conducting elements are modeled as uniform current density solenoids or coils.

## HOW TO USE SOLDESIGN

**VAX:** RUN USER5:[RDPJ.SOL]SOLDESIGN

**CRAY:** XSOLx

On the VAX the program prompts for the input file name with the statement:

WHAT IS THE INPUT FILE (terminal OR t IS ACCEPTABLE)?

The user should type in the name of the file containing the input data. If the user wishes, the program can be run in the interactive mode by specifying the input file to be the "terminal" ("t" for short). The quotes are not typed. As will be discussed later, the user may switch input from a file to the terminal or from one file to another during a session.

The program then prompts for the name of the output file with the statement:

WHAT IS THE OUTPUT FILE (terminal OR t IS ACCEPTABLE)?

The user should type the name of the file into which the output will be written. If the user wishes, the output can be written to the terminal. If the output is to a file, the user will not see any of the output until after the run is over. The file must then be printed, viewed within an editor or otherwise listed.

If the input is from a file, the program can echo print the input file into the output file at either the start or the end of the output file. If output is to the terminal, the echo print is to the terminal.

After the echo print, the program will execute the data file. If file input and file output are used, the system prompt will appear. If the input is from a file and the output to the terminal, all output will appear at the terminal followed by the system ready prompt. If the user misspells a command, the system prints the misspelled command at the terminal and into the output file, and stops the program.

If input is from the terminal, the program will prompt the user for a command with the statement:

COMMAND

The user issues the desired command. If more input is required, the program will prompt for it. Once the data is read, the program processes the data and outputs the results and returns to the COMMAND mode. If the user misspells a command in the terminal input mode, the program prints a warning and requests the command be retyped - it does not stop execution.

On the CRAYs, the input is assumed to be in file SOLIN and the output is written to the file SOLOUT. The line

XSOL SOLIN=INFILE,SOLOUT=OUTFILE

can be used to change the input and output file names. It is not recommended that the CRAY version be run in interactive mode.

## AVAILABLE COMMANDS ARE

- ENDLIST** - echo list the input file at the end of the output file. Suppress header and timing prints.
- FIELD** - calculate the flux and flux density components at the specified point(s).
- FILE** - define new input file.
- FORCE** - calculate the Lorentz force components.
- HELP** - on-line explanation of commands and data.
- LOOPSW** - turn on loop calculation mode or reduced integration for inductances.
- NOLIST** - do not echo list the input file. Suppress header and timing prints.
- INDUCT** - calculate the inductance matrix for the coils.
- PEAKFIELD** - search for the peak field on a coil boundary.
- PLOT** - invoke the system plot package (has a separate HELP).
- PRINT** - print the coil data in a formatted form.
- READ** - read the coil set and field data SAVED from a previous SOLDESIGN run.
- READCURR** - read an input current file and use those currents at the specified time.
- READFIELD** - read an ASCII data file for plotting.
- READFORCE** - read the coil set and force data from a previous SOLDESIGN run.
- RESIS** - calculate coil resistances.
- SAVE** - save the current set of coils and field data.
- SAVEFORCE** - save the current set of coils and force vectors.
- SETUP** - define coil geometry (has a separate HELP).
- START** - input istart and jstart other than (1,1).
- STOP** - terminate session.
- TERMINAL** - when input is from file, switch input to terminal.
- UNITS** - define scale factor for units of length for input and output.

All input to SOLDESIGN is lower case and free field with either comma or space separators. Leading and imbedded blanks are ignored. However, an entry of ,, permits

the use of the program defaults. Blanks are ignored, so care must be taken when mixing blanks and commas. Data may have up to 20 characters. Input can be either from the terminal or from a file. In the terminal input mode, most inputs consist of two lines - the first is the command, the second is for any additional input required. In the file input mode, the command and all additional data are on one line. Reference sheets are given at the end of this document that list the commands and required input.

## ON-LINE HELP

**SOLDESIGN** has an on-line help command. If the user desires an explanation of one or more of the commands, he should type **HELP** in response to the **COMMAND** prompt, or place it at any point in the input file. A general help message is printed along with the list of possible commands. If the user wishes an explanation of the individual commands, the program prompts for the command name with the statement:

COMMAND TO BE EXPLAINED (end TO EXIT HELP)

Typing any of the possible commands will cause a brief description of that command to be printed (either at the terminal or into the output file). The response **ALL** will cause the program to print an explanation of all commands. Type **END** to exit the help subroutine.

## COMMANDS

All input is lower case and free field (comma or space separated) with any spaces ignored.

**SOLDESIGN** is built around a command language. Commands are alphabetic and are mnemonic. If input is from the terminal, the program will prompt for the next command with the statement:

COMMAND

If input is from a file, no prompts are issued, and the command and all data for that command must be on one line. Furthermore, as an option, all input from a file can be echo printed at the start of the program - this is recommended as an aid in debugging misspelled lines.

Variables enclosed in parentheses in the prompts are optional. To accept the program defined defaults for any optional variable, type ',' or if last nondefault entry has been entered a carriage return picks program defaults. For example, if magnetic fields are sought, the prompt:

r0, z0 (,dr, dz, nr, nz, ipolar, iprint, istart, jstart)

is issued. If the field at a single point is desired - r=1.5, z=0.54 - then the input is:

1.5, 0.54 < return >

the default values for the remaining entries will be used. If the same field is desired at the

same single point is desired with `iprint=2` then the input is:

1.5, 0.54,,,,,2 < return >

The command `HELP` causes the above information and a list of available commands to be printed. For information on the individual commands type the command name in response to the prompt. To get a complete set of explanations type `ALL` in response to the prompt. To exit `HELP`, type `END`. Certain commands have a separate `HELP` option. For example, to get on-line help for the coil setup portion of the program, type `SETUP` in response to the prompt `COMMAND`. The program will respond with another prompt:

`SETUP COMMAND (group, symmgroup, endgroup, solenoid, rho, help OR end)`

Typing `HELP` at this point will cause the `SETUP` help to be printed.



## DESCRIPTION OF SOLDESIGN COMMANDS

A detailed description of each SOLDESIGN command, the required input, and the output is described in this section. The commands and descriptions are listed in alphabetical order. Each input variable is described. Program-defined default values for variables are given in square brackets.

The required input line is given and followed by a detailed description of each variable. Whenever possible, the command line and description is kept on a single page for the sake of clarity. Therefore, some pages may be partially empty in order to accommodate this requirement.

The program output is also described. The terms field and flux density are used interchangeably in this manual, although the latter is more precise. Similarly, force and force per unit length or force per radian are also used interchangeably.

### ENDLIST

The ENDLIST command is used to change the default location of the echo list of the input file from the start of the output file to the end of the output file. The use of this option still allows a record of the input, but without cluttering up the beginning of the output. This must be the first command in the input file. Header and timing prints are suppressed.

## FIELD

The **FIELD** command invokes the field calculation phase. Fields can be calculated at a single point, along a line, or at the intersection or nodes of a rectangular or polar mesh. **SOLDESIGN** employs a Gaussian quadrature to integrate the flux, and flux density components azimuthally unless the loop switch is on<sup>[1]</sup>. If the loop switch is on, then a loop calculation is used if the distance from the coil center to the field point is greater than the input loop tolerance times the coil diagonal. The input is:

**FIELD**, r0, z0 (, dr, dz, nr, nz, ipolar, iprint, istart, jstart)

where

r0 [0] - the radial coordinate of the initial field point.

z0 [0] - the axial coordinate of the initial field point.

dr [0] - the increment in the radial coordinate.

dz [0] - the increment in the axial coordinate.

nr [1] - the number of radial coordinate increments.

nz[1] - the number of axial coordinate increments.

ipolar [0] = 0 a regular rectilinear mesh is generated  
= 1 a special flag to generate a polar grid about (0,0) with radius  $r0+i*dr$  ( $i=1,nr$ ) and angle  $z0+j*dz$  ( $j=1,nz$ ) The axial inputs become azimuthal inputs in degrees measured counterclockwise from the R-axis.

iprint [1] = 0 do not print field components (recommended for large meshes used in plotting).  
= 1 print flux and flux density components at the point(s).  
= 2 print flux and flux density components produced by each group of coils (*i.e.* an influence coefficient). The total is not printed.  
= 4 very special form for calculating flux, flux density, and five derivatives of B at each point due to each coil. Probably useful to the author only.

istart [1]

jstart [1] - starting values of the counters for i and j. This input is the same as for the **START** command and can be used for multiple field calculations. See contour plotting.

Normally, the field calculator uses a 10 point quadrature for points "away" from the coil(s). The user may override this integration with the use of the `ngauss` parameter on the solenoid definition line. If the parameter, `ngauss`, is greater than or equal to 20, the field integrator will use `ngauss` points. In the neighborhood of the coil - that is, field points with coordinates ( $R_p/R_i$ ,  $Z_p/R_i$ ) between 80% and 120% of the normalized coil parameters  $R_o/R_i$  and  $Z_o/R_i$  - the integrator switches to a 50 point quadrature. These integrations produce errors of less than 0.5% ("exact" was assumed to be a 100 point quadrature). If

the loop switch is on, the field is calculated from a loop, if the tolerance parameter on the LOOPSW command line times the coil diagonal is less than the distance from the coil center to the field point.

## FILE

The FILE command is used to switch input from the present file or the terminal to a different file. This command can be used to run several data sets in the same SOLDE-SIGN session. The program expects a new filename. If input is from the terminal, the program prompts for a filename. If file input is used, the new filename appears as the second entry on the line. The form is:

FILE  
filename

or

FILE, filename

## FORCE

The FORCE command allows the user to calculate Lorentz forces per unit length or per radian acting on one coil or one coil group due to itself and any others that are defined. A Gaussian quadrature is used to integrate the  $\vec{j} \times \vec{B}$  body force density over the cross section. The program expects the input of:

FORCE, group no. (,coil no., intr, intz, iprint, igstart, igstop, nrs, nzs)

or

FORCE, LOOP, grp1, grp2, grpinc (,coil no., intr, intz, iprint, igstart, igstop, nrs, nzs )

where

- group no. - group number for the group of coils on which the force acts.
- grp1,grp2,grpinc - loop limits. Calculates force on coil no. in groups grp1 through grp2 with increment of grpinc.
- coil no. [0] - coil no. within the group. *E.g.* coil 1 in group 10. If =0 then all coils in the group are included.
- intr,intz [3,3] - the number of points in the r and z directions at which the fields and body force densities are calculated. These points are Gauss points and are not evenly spaced.
- iprint [1] - print flag.  
= 1 print the total integrated force per unit length (N/m).  
= 2 print coordinates and force densities at each integration point (N/m\*\*3).  
= 3 print winding pack average hoop and axial stress (N/m\*\*2).  
=11 print the total integrated force per radian (N/rad).  
if iprint < 0 then print force (stress) due to each group and total.
- igstart,igstop - allows only selected coil groups producing the field to be considered. Default is all groups.
- nrs,nzs - a special form of the SUBDIVIDE option which will automatically integrate  $\vec{j} \times \vec{B}$  over each of the nrs  $\times$  nzs subdivisions of the coil. This option can be used to generate input for stress analysis codes or to plot distributions of forces in a coil.

## HELP

The HELP command is used to obtain on-line help.

## INDUCT

The INDUCT command allows the user to calculate the inductance matrix for the coil groups defined. The elements of the upper triangular matrix, as well as the total system inductance, are printed. The turn and ngauss entries on the SOLENOID line in SETUP must be defined. The program uses ngauss current sheets in the radial direction to integrate numerically the expressions for sheet inductances [2]. In addition to the values of the elements of the inductance matrix, the number of current sheets used in the evaluation is also included – useful in assessing accuracy versus CPU time trade-offs. If the loop switch option is on, the program checks the distance from coil center to coil center against the coil diagonals to determine if ngauss or ngmin current sheets are to be used in the mutual calculations. The input is:

```
INDUCT (,igstart, igstop, jgstart, jgstop)
```

where

igstart [1] - starting group number (*i.e.*, starting row number)

igstop [ngroups]- ending group number (*i.e.*, ending row number)

jgstart [1] - starting group number (*i.e.*, starting column number)

jgstop [ngroups]- ending group number (*i.e.*, ending column number)

For example, if there are 50 groups of coils and the self inductances of only groups 45 through 50 and the mutual inductances between groups 45 through 50 and 1 through 45 are desired – *e.g.*, groups 45 through 50 are added groups after a full calculation of the 44 by 44 matrix. The input would be:

```
INDUCT,45,50,1
```

The final 1 is required to pick up the mutuals between groups 45 through 50 and groups 1 through 44.

## LOOPSW

The LOOPSW command sets the loop switch to enable a speedier calculation of fields, forces and/or inductances [3]. The form of the command is:

LOOPSW (,tol, ngmin)

where

tol [10] - tolerance for switching between loop and uniform current density approximations for field calculations, or between SETUP defined ngauss and ngmin current sheets for inductance.

ngmin [2] - number of radial current sheets for inductance calculations if loop test is satisfied.

For field calculations, the loop approximation is used if  $dis > tol * diag$  where  $dis$  is the distance from the coil center to the field point, and  $diag$  is the coil diagonal. The latter is chosen to measure the coil size, since a single loop may not adequately capture the field solution if the coil has a very high (or low) aspect ratio.

For inductance calculations, the ngmin parameter specifies the number of radial sheets to be used in coil  $j$ . If the mutual inductance between coil  $j$  and coil  $k$  is sought, SOLDESIGN calculates the distance between the coil centers,  $dis$ , as well as the two coil diagonals,  $diaj$  and  $diak$ , respectively. If  $dis > tol * diaj$  then ngmin sheets are used in coil  $j$  and if  $dis > tol * diak$  then ngmin sheets are used in coil  $k$ . If the test fails, then ngauss sheets are used (*i.e.*, the entry used in the SOLENOID definition).

## NOLIST

The NOLIST command can be used to suppress not only the echo listing of the input file, but also all the header and timing information usually printed into the output information. This must be the first command in the input file. This is useful when the output of SOLDESIGN is to be used as input to other programs.

## PEAKFIELD

The PEAKFIELD command invokes a peak field search routine. The input is:

PEAKFIELD, coil no. (,nr, nz, iprint)

or

PEAKFIELD,LOOP,ncl1,ncl2 (,nr, nz, iprint)

where

coil no. - coil number for the peak field search.

ncl1,ncl2 - loop limits for peak field search.

nr,nz [5,5] - number of search points in the r- and z-directions.

iprint [0] - print flag.

= 0 peak field value and coordinates are printed.

= 1 all search point coordinates and fields printed.

The coil boundaries are divided into nr (or nz) points including the corners. The routine starts at the lower left corner of the coil crosssection (*i.e.* the minimum r and z coordinates) and searches in a counterclockwise direction. For example, a  $3 \times 3$  search will calculate fields at the four corners and 4 midside points - *i.e.*, 3 points along each side (corners included). A  $4 \times 4$  search will calculate at the 4 corners and 2 third points along each side.

PEAKFIELD will work for SOLENOIDS, but not of the arbitrary QUADs described in the SETUP section.

## PRINT

The PRINT command causes the coil parameters defined in the setup phase to be printed in a easily read format. It is recommended that this command be used after a READCURR command to verify that the proper current assignments have been made.

## READ

The READ command allows the reading of a previously SAVE'd set of coil, flux, and field data for further processing. This feature may be used with a batch generation of the field data and an interactive plot. Data may be generated and saved via the SAVE command in a batch run and then READ in during an interactive session. If the input mode is terminal, the program prompts for a file name. In file input mode, the form is:

READ, filename

## READCURR

The READCURR command allows the coil currents to be read from a file after the coil geometries have been defined. The file format is assumed to be that of the eigenexpansion programs in use at PFC such as NEWEIGEN, SUCIRC, *etc.* These programs output the current in all the elements at each time point in the form:

```
****   ncirc
time1  I(1)  I(2)  I(3)  ...  I(ncirc)
time2  I(1)  I(2)  I(3)  ...  I(ncirc)
.      .      .      .      .      .
```

The program expects (prompts for in terminal input mode) the filename of the currents, the number of currents in the file, the time at which the currents are to be used and a tolerance for the time testing. The form of the input is:

READCURR

WHAT IS THE FILE WITH THE CURRENTS?

filename

NUMBER OF CURRENTS IN THE FILE, TIME, AND TOLERANCE

ncirc, time, tol

This feature was added to SOLDESIGN to allow a more casual user access to a post-processor for output from a circuit code. It is not particularly efficient if fields or forces at a number of time points are desired. In that case, a user-written postprocessor that uses SOLDESIGN-generated influence coefficients is recommended.

READCURR assumes that the model defined in the SETUP section is compatible with the current file. It is up to the user to ensure this. A PRINT command is recommended after the READCURR command so that the current and coil definitions may be checked.

READCURR also multiplies the turns defined on the SOLENOID line by the file read currents. **THIS IS AN EXCEPTION to the usual manner of treating turns and ampere-turns!**



## READFIELD

The READFIELD command can be used to read a set of data generated by other programs in order to use the SOLDESIGN plot package for contour and function plotting. The program prompts for terminal input of a file name with the data. Data must be on a rectilinear mesh and have the form:

i j r(i,j) z(i,j) (data(k) k = 1, ncol)

where the program prompts for the variable ncol. Only four columns can actually be accessed for plotting. If  $ncol > 4$  the program asks for the 4 column numbers of data to be saved for plotting; the others are read but not saved. To access this data for either contour or function plotting, a special type=MISC has been added to the plotting options as described later in this manual. At the time the plot is requested, the program will prompt for the column (of the four) that is to be plotted. The function plotting routine also asks for a Y-axis label. This feature was included primarily to serve as a quick access to a contour and function plotter for users without those packages.

## READFORCE

The READFORCE command allows the reading of a previously SAVED set of coils and force data for further plotting. This feature can be used with a batch generation of the force data and an interactive plot. Data can be generated and saved via the SAVEFORCE command in a batch run and then the READFORCE command is invoked during an interactive session. If the input mode is terminal the program prompts for a file name. In file input mode, the form is:

READFORCE, filename

## RESIS

The RESIS command allows the resistance of the groups of coils or elements to be calculated. The group resistivity is defined in the RHO, GROUP or SYMMGROUP command lines in SETUP. If each coil is to be a different group and the resistivity is the same, then it is not necessary to use the GROUP-ENDGROUP combination around each. Instead, it is necessary to use it around the first group only. SOLDESIGN will use the last resistivity defined.

## SAVE

The SAVE command allows a set of coil, flux, and flux density data to be saved for future processing. With this option, the field data can be generated and SAVED in a batch run and later READ and plotted interactively. In the VAX-version, the file is opened with a status of UNKNOWN which implies that if the file exists, then it is opened and new data will overwrite any old data – this effectively destroys the old information in that file. If the file does not exist, it is created. If the input is from a file, the second entry is the filename. If the input is from the terminal, the program prompts for a file name. The form is:

SAVE, filename

## SAVEFORCE

The SAVEFORCE command allows the user the option of saving the force data generated with a time-consuming run. Force vector plotting requires as input a scale factor for the display of the forces. If an incorrect choice is made in a batch run, the data would have to be regenerated in order to replot. Using the SAVEFORCE and READFORCE commands, the data is generated, saved and plotted in a batch run. The results can be examined and if additional plots are desired, then the data may be read using the READFORCE command and additional plots made. If the input mode is terminal the program prompts for a file name. In file input mode the form is:

SAVEFORCE, filename

## SETUP

The SETUP command invokes the subroutine that is used to define the geometries and groupings of the coils or elements. The setup command line is:

SETUP (, NI, CENTROID, CORNERS)

where the optional parameters NI, CENTROID and CORNERS allow the user to input total ampere-turns (rather than overall current density) and/or the coil centroidal coordinates (rather than the lower corner) or the lower inside and upper outside corners, respectively.

SETUP has a separate help feature that is reproduced here. If terminal input mode is used, the program prompts for each SETUP command with the line:

SETUP COMMAND (solenoid, quad, rho, group, symmgroup, endgroup, help OR end)

A setup command is expected at this point. An incorrect command in terminal input mode will cause a warning message to be printed to the terminal and a new prompt for input issued. In batch mode, an error in a SETUP command will cause the error message to be printed, and the program to terminate execution.

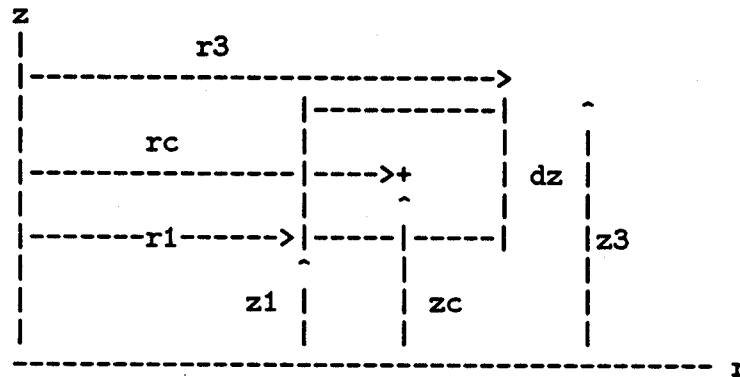
Available SETUP commands are:

- SOLENOID - uniform current density solenoid.
- QUAD - uniform current density quadrilateral.
- RHO - define or redefine the resistivity.
- GROUP - turn on grouping flag. All coils from the next coil to the next GROUP or ENDGROUP or END command are to be treated as a single field/force source. The group resistivity is the second entry on this line if the resistance is required.
- SYMMGROUP - special grouping flag. All coils defined with isym=  $\pm 2$  will be grouped - until the next GROUP or ENDGROUP. The group resistivity is the second entry on this line if the resistance is required.
- ENDGROUP - turn off grouping flag. From this point until next GROUP or SYMMGROUP command, treat each coil as a separate source.
- SUBDIVIDE - automatically subdivides coils into nnr by nnz subcoils where nnr and nnz are the second and third entries, respectively. *E.g.*, SUBDIVIDE, 3, 4.
- HELP - displays this description of SETUP commands.
- END - terminate coil definitions and return to the main program.

## UNIFORM CURRENT DENSITY SOLENOID COMMANDS

```

SOLENOID, r1, z1, isym, dr, dz, cdens  (,turn,ngauss)
SOLENOID, r1, z1, isym, dr, dz, ni    (,turn,ngauss)
SOLENOID, rc, zc, isym, dr, dz, cdens  (,turn,ngauss)
SOLENOID, rc, zc, isym, dr, dz, ni    (,turn,ngauss)
SOLENOID, r1, z1, isym, r3, z3, cdens  (,turn,ngauss)
SOLENOID, r1, z1, isym, r3, z3, ni    (,turn,ngauss)
    
```



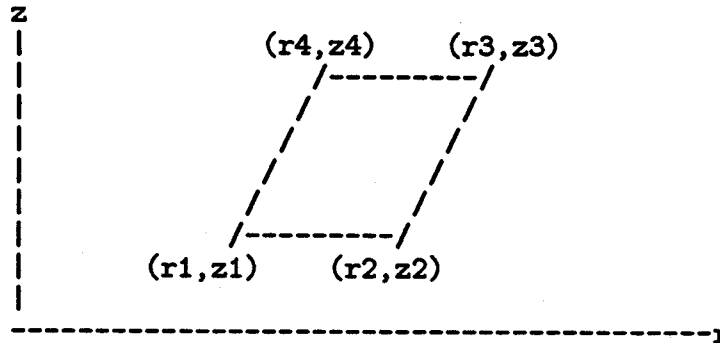
where

- r1,z1 - coordinates of the lower left corner.
- (rc,zc) - coordinates of centroid if any word on SETUP line is CENTROID.
- isym = 0 - do not plot, calculate fields, forces, etc. due to this element.  
 = 1 - no reflections.  
 = 2 - reflect about r axis - split pair with same current.  
 = -2 - reflect about r axis - split pair with opposing current.
- dr,dz - radial and vertical builds of the solenoid.
- r3,z3 - coordinates of the upper right corner if any word on SETUP line is CORNERS.
- cdens(ni) - overall current density (A/m\*\*2) or total ni (ampere-turns) if any word on SETUP line is NI .
- turn - number of turns (for inductance and resistance only).
- ngauss [10] - number of radial integration current sheets for inductance or number of azimuthal quadrature points for field calculations.

## UNIFORM CURRENT DENSITY QUAD COMMANDS

```

QUAD,  r1,z1,r2,z2,r3,z3,r4,z4,  isym,  cdens  (,turn,ngauss,nrads)
QUAD,  r1,z1,r2,z2,r3,z3,r4,z4,  isym,  ni    (,turn,ngauss,nrads)
  
```



where

- r1,z1 -
- r2,z2 - coordinate pairs of the four corners in counterclockwise order
- r3,z3 -
- r4,z4 -
  
- isym = 0 - do not calculate fields, etc due to this element
- = 1 - no reflections
- = 2 - reflect about r axis - split pair with same current
- = -2 - reflect about r axis - split pair with opposing current
  
- cdens(ni) - overall current density (A/m\*\*2) or total ni (ampere-turns) if any word on SETUP line is NI
  
- turn - number of turns (for inductance and resistance only)
  
- ngauss [10] - number of radial integration current sheets for inductance or number of azimuthal quadrature points for field calculations
  
- nrads [3] - number of radial subdivisions of the quadrilateral

The QUAD is treated by breaking the element into nrads (number of radial) rectangular regions. The user is responsible for picking a reasonable number. The COILS plot with the OVERLAY option can be used to view both the QUAD and the approximation. See Example 3.

There is a problem in using GROUP-ENDGROUP commands with an isym=0. The code counts groups before it checks for isym. Therefore, if there is a data file with a set of GROUP-ENDGROUP commands, an isym=0 will not suppress the group even if all elements of the group have isym=0. This will cause numbering problems for group-calculated quantities such as inductance. To cure the problem, delete the GROUP-ENDGROUP commands around coil with an isym=0 or delete the set of coils to be ignored.

## START

The START command is used to start the (i,j) ordering of field output at a pair other than (1,1). This ordering is used only for plotting. The program expects the input of istart,jstart. The r-z plane is mapped into a logical i-j plane with (i,j)=(1,1) at the first (r,z) point and i increasing with r and j with z. If a single mesh is used to generate a data set for plotting, this is all transparent to the user. If, however, more than one FIELD command is used to generate the data, the START command must be used before the second FIELD command to get the (istart,jstart) of the subsequent mesh to be consistent with the first. See FIELD command and the I (J) option. The form is:

START, istart, jstart

## STOP

The STOP command is used to terminate the program. All files including the PLOT (if opened) are closed.

## TERMINAL

The TERMINAL command allows the user to switch input devices from a file to the terminal. This is particularly useful when many coils need to be defined. An input file is used to invoke setup and to define the coils. After the END command for setup, the file contains the TERMINAL command. The program will echo list the input, set up the coils, and then revert to the interactive mode of input, with the terminal prompts.

## UNITS

The UNITS command allows the units of length for input and output to be set by the user. The quantity to be input is the multiplier that converts the input to meters. The form of the command is:

UNITS, units

For example, if inches are to be used for input and output then the command would be UNITS, 0.0254.

## SOLDESIGN EXAMPLES

### TERMINAL INPUT AND OUTPUT:

The first example is for a terminal input/output session. A single split pair set of solenoids is defined. The central field is calculated, and then the force between the two and system inductance are computed. The input and output shown below are the same as would appear at the terminal.

```
$ R USER5:[RDPJ.SOL]SOLDESIGN
```

```
INPUT FILE NAME (terminal OR t IS ACCEPTABLE)
```

```
t
```

```
OUTPUT FILE NAME (terminal OR t IS ACCEPTABLE)
```

```
t
```

```
COMMAND
```

```
setup,ni,centroid
```

```
SETUP COMMAND (group, symmgroup, endgroup, solenoid, rho, help OR end)  
solenoid
```

```
rc,zc,isym,dr,dz,NI ,(turn,ngauss)  
1.0, 1.0, 1, 0.1, 0.1, 1.0e6, 1, 5
```

```
SETUP COMMAND (group, symmgroup, endgroup, solenoid, rho, help OR end)  
solenoid
```

```
rc,zc,isym,dr,dz,NI ,(turn,ngauss)  
3.0, 3.0, 1, 0.5, 0.5, 0.5e6, 1, 5
```

```
SETUP COMMAND (group, symmgroup, endgroup, solenoid, rho, help OR end)  
end
```

```
No. Coils = 2 in 2 groups  
Time in setup = 7.000E-02 seconds, 0 loop calculations used
```

```
COMMAND
```

```
print
```

| GRP # | COIL # | Rc (m) | Zc (m) | Delr (m) | Delz (m) | NI (A-t)  | turns     | npts | rho (Ohm-m) |
|-------|--------|--------|--------|----------|----------|-----------|-----------|------|-------------|
| 1     | 1      | 1.000  | 1.000  | 0.100    | 0.100    | 1.000E+06 | 1.000E+00 | 5    | 1.000E+09   |
| 2     | 2      | 3.000  | 3.000  | 0.500    | 0.500    | 5.000E+05 | 1.000E+00 | 5    | 1.000E+09   |

```
Time in print = 2.000E-02 seconds, 0 loop calculations used
```

```
COMMAND
```

```
field
```

r0,z0,(dr,dz,nr,nz,ipolar,istart,jstart)  
2,2

| I | J | R( m)     | Z( m)     | Br(T)       | Bz(T)      | B(T)       | Flux(V-s)  |
|---|---|-----------|-----------|-------------|------------|------------|------------|
| 1 | 1 | 2.000E+00 | 2.000E+00 | -7.3901E-03 | 8.9262E-02 | 8.9568E-02 | 1.8833E+00 |

Time in field = 4.400E-01 seconds, 0 loop calculations used

COMMAND  
peakfield

Coil no.,(nr,nz,iprint)  
1

| SIDE | PT | R( m)     | Z( m)     | Br(T)       | Bz(T)      | B(T)       | Flux(V-s)  |
|------|----|-----------|-----------|-------------|------------|------------|------------|
| 4    | 3  | 9.500E-01 | 1.000E+00 | -1.3647E-02 | 4.0093E+00 | 4.0093E+00 | 3.8436E+00 |

Time in peakfi = 3.580E+00 seconds, 0 loop calculations used

COMMAND  
induct

INDUCTANCE MATRIX FOR SYSTEM

COIL COIL L or M (Henries)

|   |   |               |    |
|---|---|---------------|----|
| 1 | 1 | 4.0425827D-06 | 25 |
| 1 | 2 | 3.7273611D-07 | 25 |
| 2 | 2 | 1.0214448D-05 | 25 |

TOTAL SYSTEM INDUCTANCE IS = 1.5002503D-05 Henries

Time in induct = 1.400E-01 seconds, 0 loop calculations used

COMMAND  
force

group no.,(coil no.,intr,intz,iprint) or  
loop,ng1,ng2,ninc,(coil no.,intr,intz,iprint)  
<return>

| Group | Fr(N/m )   | Fz(N/m )    | Rave(m)   | Zave(m)   | dr(m)     | dz(m)     |
|-------|------------|-------------|-----------|-----------|-----------|-----------|
| 1     | 4.7668E+05 | 1.4420E+04  | 1.000E+00 | 1.000E+00 | 1.000E-01 | 1.000E-01 |
| 2     | 3.0647E+04 | -4.8070E+03 | 3.000E+00 | 3.000E+00 | 5.000E-01 | 5.000E-01 |

Time in force = 1.160E+00 seconds, 0 loop calculations used

COMMAND  
stop

Time in stop = 5.420E+00 seconds  
FORTRAN STOP



## FILE INPUT - TERMINAL OUTPUT

The second example demonstrates an interactive session with a file input of coil geometry and a TERMINAL command. A split pair and a single solenoid are defined using the input file EX2INPUT.DAT. As can be seen, the program echo prints the input file before proceeding. The exclamation point is the SOLDESIGN comment character and the comments are also echo printed, but not processed. Note also, that the commands are followed on the same line by any additional input required. The last command in the file is TERMINAL which switches input back from the file to the terminal. In this example, the next command is a new field command, but it could be anything. The final STOP is required.

```
$ R USER5:[RDPJ.SOL]SOLDESIGN
```

```
INPUT FILE NAME (terminal OR t IS ACCEPTABLE)
ex2input
```

```
OUTPUT FILE NAME (terminal OR t IS ACCEPTABLE)
t
```

```
SOLDESIGN V3.0 11/26/90      10:11
```

```
ECHO LIST OF INPUT DATA
```

```
1 setup
2 solenoid,0.383,-0.358, 1,0.0166,0.716, 9.954e7 !Didn't need inductance
3 solenoid,0.397, 0.4278,2,0.0363,0.2382,1.002e8 !so needn't define rest
4 end
5 field,0,0                                !Only one point, use defaults
6 terminal
```

```
No. Coils = 3 in 3 groups
Time in setup = 1.200E-01 seconds,      0 loop calculations used
```

| I | J | R( m)     | Z( m)     | Br(T)      | Bz(T)      | B(T)       | Flux(V-s)  |
|---|---|-----------|-----------|------------|------------|------------|------------|
| 1 | 1 | 0.000E+00 | 0.000E+00 | 0.0000E+00 | 2.0001E+00 | 2.0001E+00 | 0.0000E+00 |

```
Time in field = 4.300E-01 seconds,      0 loop calculations used
Time in termin = 0.000E+00 seconds,      0 loop calculations used
```

```
COMMAND
field
```

```
r0,z0,(dr,dz,nr,nz,ipolar,istart,jstart)
0.10,0.10
```

| I | J | R( m)     | Z( m)     | Br(T)       | Bz(T)      | B(T)       | Flux(V-s)  |
|---|---|-----------|-----------|-------------|------------|------------|------------|
| 1 | 1 | 1.000E-01 | 1.000E-01 | -1.0390E-05 | 2.0001E+00 | 2.0001E+00 | 6.2835E-02 |

```
Time in field = 2.900E-01 seconds,      0 loop calculations used
```

```
COMMAND
```

```
stop
Time in stop = 8.800E-01 seconds
```

```
FORTRAN STOP
```

## PLOTTING WITH SOLDESIGN

The plotting portion of SOLDESIGN can be used with either batch or interactive (terminal) input. GRAFLIB is the underlying plot package used by SOLDESIGN. Plotting with SOLDESIGN may be used to generate plots of coil locations, vector plots of fields or forces, contours of constant flux, total flux density, or flux density components. In addition, contours of constant field homogeneity may be plotted. These contours in % are given by  $(bz - b_0)/b_0 * 100$ , where  $b_0$  is the value of  $b_z$  at  $(i,j)=(1,1)$  which should be  $(r,z)=(0,0)$  for most plots. It may also be used to generate plots of flux, total flux density, or flux density components versus position ( $r$  or  $z$ ) for fixed  $z$  or  $r$ . The contour and vector plots may also include the outlines of the coils.

The contour plotting package can be used to print the coordinates of the contours of constant (whatever) instead of plotting. This function can serve as a crude contour follower.

The plot package is used to plot the data generated by the FIELD or FORCE commands. These data may have been just generated or they may have been saved with the SAVE or SAVEFORCE commands, in which case, the data must be read with the READ or READFORCE commands, respectively. The  $(i,j)$  pairs – the first two columns of output – are essential for the plotting.

In general, coil boundaries will not coincide with the rectangular mesh generated by a single FIELD command. If nothing were done to get data along the coil boundaries, the contour and function plots would appear to be incorrect. These routines use linear interpolation between data points to draw contours or curves. Therefore, if mesh points do not coincide with coil boundaries, the appropriate components of the flux density will not appear to experience a jump across the boundaries. SOLDESIGN recognizes this fact and fills in this data along all coil boundaries. The user need not define meshes that have lines along coil boundaries. The first time PLOT is called, SOLDESIGN will generate the required field data so that future plots will be valid in and around coils. If the SAVE command is used after the PLOT command, all the data including that along coil boundaries is saved, and future plots using this data will not require the additional computation.

This filling process is very time consuming and, hence, not very satisfactory. Occasionally it leads to some mysterious effects when the testing fails for some reason or other. Future versions of SOLDESIGN will incorporate enhancements that make the filling process more transparent to the user, but for now – what you see is what you get.

**NOTE:** once the program has filled in data, the original user-defined  $i-j$  to  $r-z$  mapping has changed. Care must be taken in the definition of the data to be used in function plots.

The FIELD command mesh for contours has to be purely rectilinear with no missing data or holes. If other data are to be plotted, minor changes can be made to the package to make it accept such data.

After the first PLOT command in the interactive mode, SOLDESIGN prompts with:

## GRAPHICS TERMINAL (y OR n)?

If a supported graphics terminal is being used and plotted output is to be displayed, then the answer is y. If the answer is n, the program then asks:

## SAVE PLOT FILE (y OR n)?

If the plot is to be displayed as well as saved in a file (TEKTRONIX.PLT) the answer is y. If answer to the first question is n (not a graphics terminal), then plot file is automatically saved. Similarly, if the graphics input is from a file, the program automatically saves the file and does not display on the screen.

SOLDESIGN then prompts for a title for all plots. This title must have less than 60 characters. The title will be reproduced at the top of each plot, and is helpful in defining the run for future reference. The prompt is:

## INPUT PLOT TITLE

23456789012345678901234567890123456789012345678901234567890

where the numbers are merely an aid in keeping the title to fewer than 60 characters. Depending on the default font size, 60 characters may still overflow the line or lines and be cut off on the plot. A change in font size or in the header is required to eliminate this truncation – the former requires a program change.

The title can be any combination of letters and numbers. A semicolon (;) within the title serves as a line feed, and, hence, two-line titles are possible. However, the total number of characters must be fewer than 60. In general, the second line of a two-line title will over-write the SOLDESIGN advertisement and not be very aesthetic.

The plotting section uses the same free-field reader as does the main computing section. That is, numeric and alphabetic variables may be mixed on an input line. Numeric values or alphabetic words are separated by commas or spaces. The order of entries is important. Those inputs not specified either by ,, or by < return > are set to the default values (0 unless otherwise specified).

The program then prompts for the plot type and other plot information with:

PLOTTYPE, TYPE, rmin, rmax, zmin, zmax (,scale, iout, iprint, DRAW, NOFILL, P, OVERLAY)

where available PLOTTYPEs are:

COILS, CONTOURS, FUNCTION, VECTORS, HEADER, or END

and where

TYPE - BR, BZ, B, BX, BY, FLUX, HOMOGENEOUS ( $= (bz - b_0) / b_0 * 100$ ), or MISC. (see the READFIELD command for use of MISC option).

rmin,rmax - the r or x limits of the region to be plotted

zmin,zmax - the z or y limits of the region to be plotted. If PLOTTYPE=FUNCTION, zmin and zmax are interpreted as the function minimum and maximum, respectively. If both are entered as zero, the program will calculate the min and max).

scale - scale factor for labels [1.0]. If contour labels are desired in gauss scale=1.e+4.  
- height of coil (group) numbers on coil plots [65.0].  
- length scale for vectors in vector plot [0.12].

iout[0] - flag for drawing coil outlines on vector and contour plots.  
= 0 - omit coils  
= 1 - draw coils

if coil plot (TYPE=COILS) then:

= 2 (-2) number groups without (with) coils drawn  
= 1 (-1) number coils without (with) coils drawn  
= 0 (default) - coil boundaries only

if function plot (TYPE=FUNCTION) then:

= 1 suppress background grid

iprint[0] - flag for printed contour output.  
= 0 - normal plotting  
= 1 - no plotting, but print coordinates of contours.

DRAW - see description under CONTOUR plotting. This invocation of DRAW affects only coil geometry plots. CONTOURS, and VECTORS require the DRAW option to be specified on the second command line.

NOFILL - if NOFILL then the automatic filling of data along the coil boundaries will not be done. LET THE USER BEWARE!

OVERLAY - a special flag to overlay the nrad's radially subdivided rectangular coils used to model a QUAD element.

The major plot types are discussed below in individual sections. The `HEADER` command is used to change the plot header (title) without exiting the plot package.

The `END` command is used to terminate the plot section and return control to the main program. Any subsequent plot command will yield the header prompt, but not the graphics terminal prompt. New plots are appended to any saved plot. An `END, SAVE` command, however, will close the plot file and a subsequent plot command will open a new plot file if the original request was for a file.

## COIL PLOTTING

Any coil plots (either outlines in contouring or vector plots or geometry plots) will draw an X in coils with positive currents – that is, the coil diagonals will be drawn. No diagonals will be drawn for coils carrying a negative current. A coil with zero current will be drawn in a dashed line mode. An `iout=-1` option will cause the coil diagonals to be suppressed for positive currents in order to make the vectors (especially force vectors) more apparent. For coil geometry plotting with coil or group numbers drawn at coil centroids, the diagonals are automatically omitted.

Figure 1 illustrates four sample coil plots. For these samples, the generating command is shown in the position that usually has the title or header. The coils are defined by the following sequence of setup commands:

```
setup,ni,centroid
symmgroup
solenoid, 1.3, 0.4, 2, 0.3, 0.4, 1.e5,
solenoid, 0.5, 0.6, 2, 0.4, 0.2, -1.e5,
quad, 0.20, 0.10, 0.45, 0.20, 0.55, 0.45, 0.16, 0.35, 2, 1.e4, 1, 1, 5
end
```

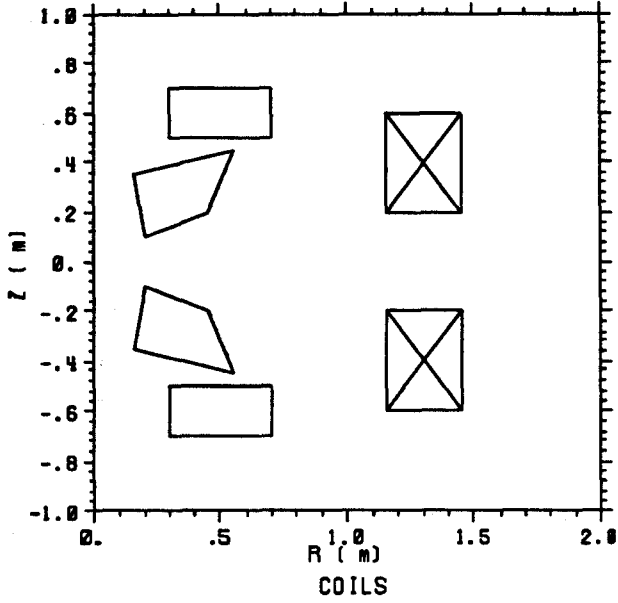
The figure in the upper left represents a basic coil plot with no additional options. The open boxes represent coils with a negative current. The diagonals in a coil imply that the coil is carrying a positive current.

The figure in the upper right shows the coil numbers at the center of each coil cross section. The size of the numeric characters can be adjusted using the scale parameter on the plot command line.

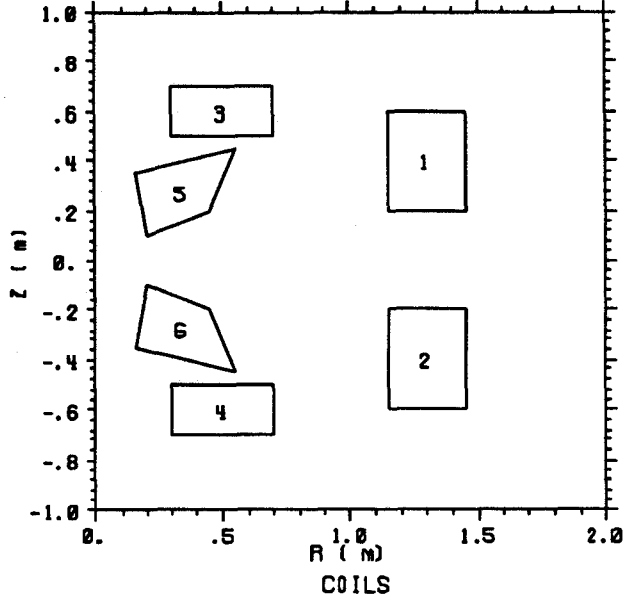
The figure at the lower left shows the coils with the group numbers at the centers. The negative `iout` parameter in both this and the previous plot causes the coil outline to be drawn. A positive `iout` causes the coil outline to be suppressed.

The final figure shows the `OVERLAY` option associated with the `QUAD` option. The pair of irregularly shaped coils are shown with the five radial “coils” used to model the `QUAD` in the field and inductance routines. The use of five subdivisions is arbitrary.

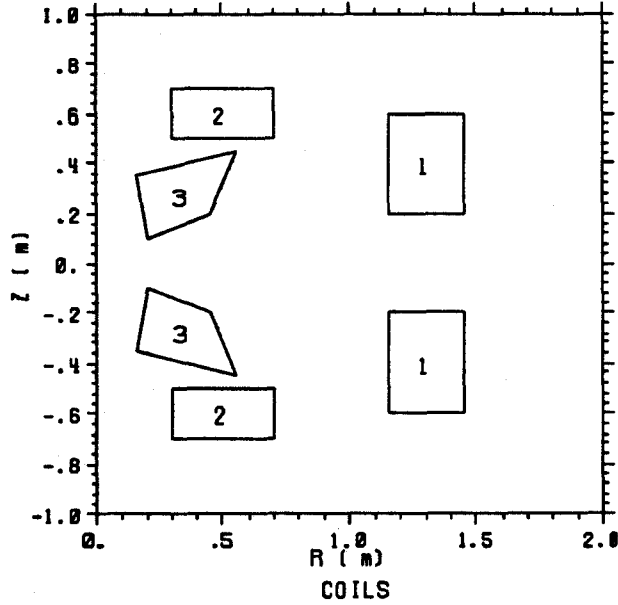
COILS. 0. 2. -1. 1  
 SOLDESIGN V3.0 1/ 2/91 17:43



COILS. 0. 2. -1. 1. 55 . -1  
 SOLDESIGN V3.0 1/ 2/91 17:43



COILS. 0. 2. -1. 1. 45 . -2  
 SOLDESIGN V3.0 1/ 2/91 17:43



COILS. 0. 2. -1. 1. . OVERLAY  
 SOLDESIGN V3.0 1/ 2/91 17:43

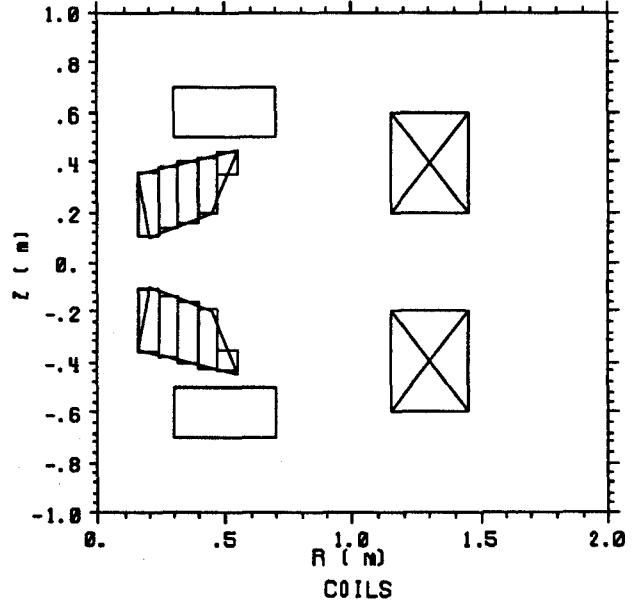


Fig. 1. SOLDESIGN COIL PLOTTING EXAMPLES

## CONTOUR PLOTTING

If PLOTTYPE=CONTOURS, SOLDESIGN prints the maximum and minimum values of the function within the plot region, and prompts for the necessary data for drawing and labeling the contours, with the statement:

ncon, (pmin, dp, iplt, htd, lblfrq, lbl, ndec, lntyp, LOG, STYLE, MORE, DRAW) ?

where

ncon - number of requested contours ( $0 < |ncon| < 100$ ). If  $ncon < 0$ , the routine prompts for input of contours values and linestyles.

pmin - minimum value (first contour) to be plotted. The default is the program determined minimum value in the region.

dp - delta between contours. Calculated default is  $(pmax-pmin)/ncon$

iplt [1000] - measure of number of labels on a contour. Default gives no labels. Iplt represents a distance between labels expressed as a percentage of the plot width. *E.g.*  $iplt=20$  implies the distance between labels is approximately 20% of the plot width.

htd [55] - height of label numbers.

lblfrq [2] - label every lblfrq contour.

lbl [0] - =0 label with contour value; =1 label with contour number.

ndec [1] - number of decimals in label.

lntyp [1] - line style (see the GRAFLIB manual).

LOG - contour increment is logarithmic;  $p(i)=p(i-1)*10.0$  for  $i=2,ncon$ .

STYLE - negative, zero and positive values of the function being contoured are drawn with different line types - light dash, heavy dash, and solid, respectively.

MORE - used with MISC to plot more contour data on same axis. Asks for the next column number to be contoured. Col=0 terminates.

DRAW - draw outline to be overlayed on contours. Prompts for file(s)

FORMAT 1

FORMAT 2

npts

npts

- Number of points

n,nx,ny,xx,yy

or

xx,yy

npts

npts

- Number of points second set

n,nx,ny,xx,yy

or

xx,yy

SOLDESIGN accepts multiple sets of data within a file. It also accepts multiple files. A filename = NONE terminates the outline section.



For each file, SOLDESIGN prompts for:

linetype,linesize,xo,yo,xscle,yscle,ifmt

where

|              |  |
|--------------|--|
| linetype[1.] | - GRAFLIB line style   |
| linesize[.4] | - GRAFLIB thickness  |
| xo,yo        | - origin shifts to account for possible shift in coordinate system between outline and SOLDESIGN               |
| xscle,yscle  | - allow stretching of coordinates of the outline. <i>i.e.</i><br>xplt = xo + xx xscle and yplt = yo + yy yscle |
| ifmt         | - format flag =1 n,nx,ny,xx,yy; =2 xx,yy   |

The scale entry on the plot command can be used to scale contour values. If field contours in gauss are desired, then scale = 1.e4. If TYPE is MISC, the program will prompt for a plot title, X-axis title, and Y-axis title, and column number (1 to 4).

Figure 2 illustrates some of the contouring options of SOLDESIGN. The definition of the coils and the generating field command are:

```
setup,ni,centroid
symmgroup
solenoid, 1.3, 0.4, 2, 0.3, 0.4, 1.e5,
solenoid, 0.5, 0.6, 2, 0.4, 0.2, -1.e5,
end
field,0.0,0.0,0.05,0.05,41,41
```

The plot commands are written at the top of each of the four plots in the space where the plot title is usually placed. As can be seen, the coil boundaries are coincident with the field mesh. That is, fields are automatically calculated along coil boundaries. Therefore, the NOFILL option can be safely used in all four of the examples.

The figure in the upper left corner is generated by invoking the simplest of the contour commands. Fifteen contours of constant flux are requested. SOLDESIGN computes the minimum and maximum values of flux in the region and calculates the increment. The first contour (minimum) and increment values are written in the line of text just above the plot. The iout=1 option on the plot command line is used to superpose the coils on the contours. The figure in the upper left corner is generated with a similar command, but fifteen contours of constant B-magnitude are requested.

The figure in the lower left corner is the result of a request for 15 labelled contours of constant B-magnitude. The starting contour value is required to be zero and the contour increment is 0.01 T. The frequency of labeling is every 5% of the width of the plot. The height of the characters is set at 65 (smaller than the default 55). Every other contour is to be labelled with two decimal places in the label.

**NOTE:** Specifying contour decimal places must be done with care. Rounding

occurs and if a sufficient number of places is not specified, the labels will appear to be incorrect. For example, a single decimal place for the figure in the lower left corner would label contours 0.0, 0.0, 0.1, 0.1, 0.1, 0.1, .... corresponding to true values of 0.02, 0.04, 0.06, 0.08, 0.10, 0.12, *etc.*

The figure in the lower right corner is the same plot, but with the magnitude of B expressed in gauss. The scaling is accomplished with the 1.e4 scale entry on the CONTOURS command line. **NOTE:** the change in the dp from 0.01 to 100. Also, the number of decimal places is set to -1 for an integer form. In addition, the label spacing is set to 10% of the plot width.

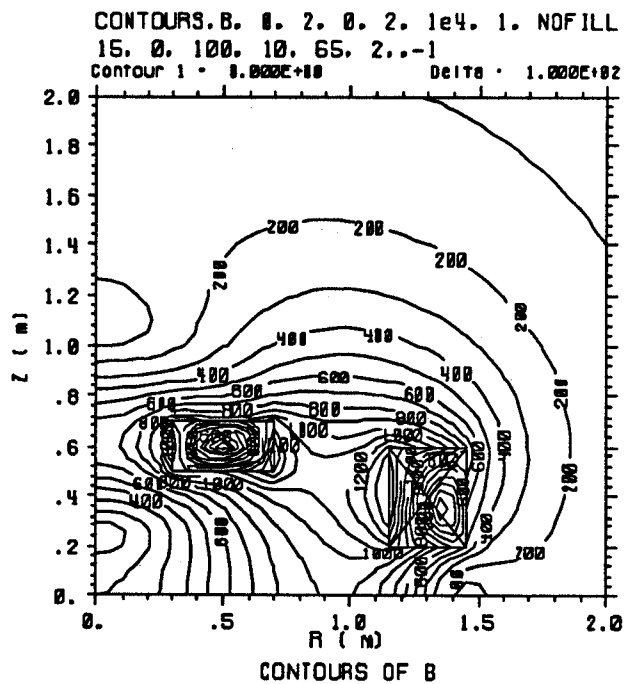
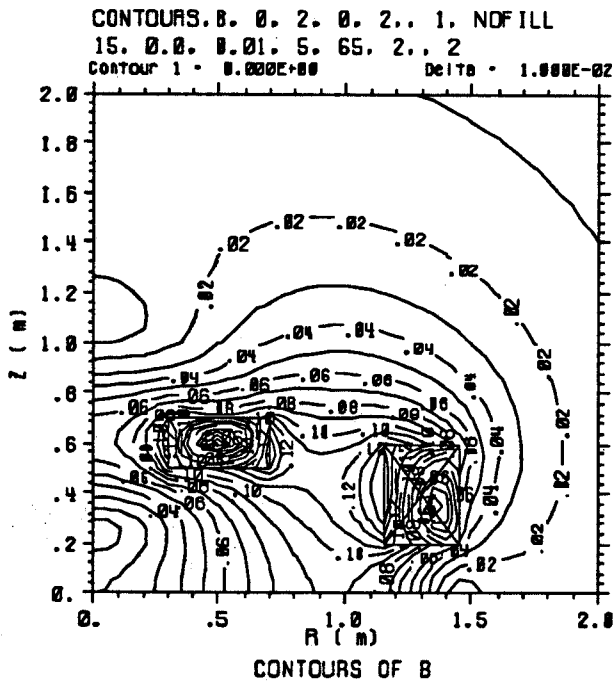
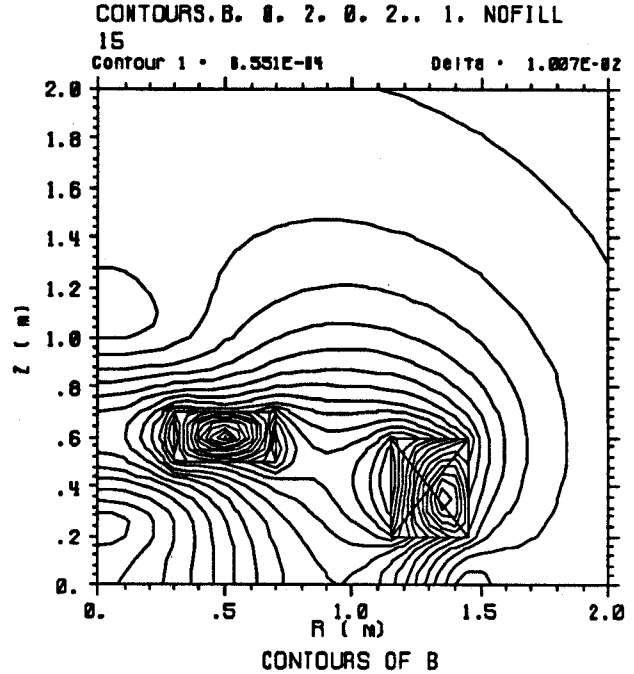
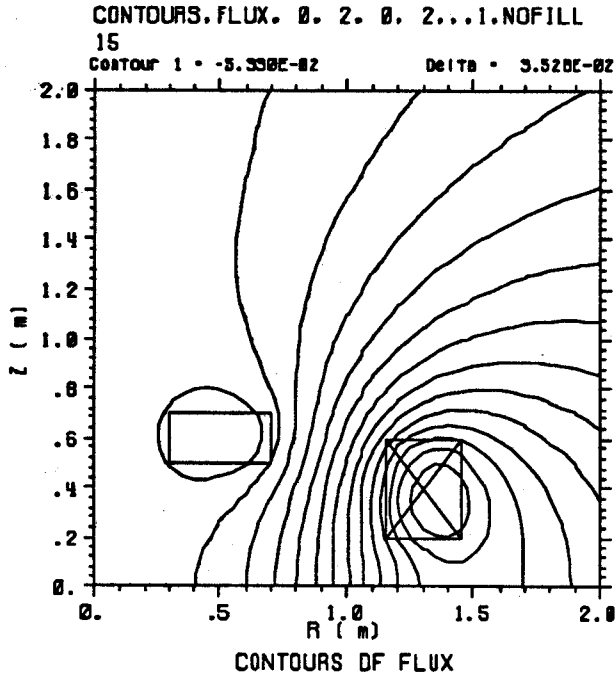


Fig. 2. SOLDESIGN CONTOUR PLOTTING EXAMPLES

## FUNCTION PLOTTING

**SOLDESIGN** allows the user to plot flux, total flux density, or flux density components versus position - *e.g.*, a profile on axis. In addition, function plots of miscellaneous data (read in using the **READFIELD** command) can be generated. If **PLOTTYPE** is **FUNCTION**, the system prompts with the statement:

i or j (for r or z), ijstart, ijfinal, ijnc

where

i (j) - implies the independent variable is r (z).

ijstart - the first j (i) value to be plotted.

ijfinal - the last j (i) value to be plotted.

ijnc - the increment in j (i).

If the **PLOTTYPE** is **FUNCTION** and the **TYPE** is **MISC** the program will prompt for the column number of the dependent variable. It will also prompt for the title for the Y-axis. This routine assumes that the independent variable is stored in the first or second column (corresponding to r or z).

In this plot routine, i corresponds to r and j to z. For example, if field data are generated by the statement:

```
field, 0.0, 0.0, 0.01, 0.01, 21, 21, 0
```

then, there are data from i=1 to 21 and j=1 to 21 corresponding to r between 0.0 and 0.20 and z between 0.0 and 0.20 in even increments of 0.01. To plot Bz versus z for r=0.0, 0.02, 0.04, 0.06, the response to the prompt would be:

```
j, 1, 7, 2
```

That is plot versus z for i=1 (r=0), i=3 (r=0.02), i=5 (r=0.04) and i=7 (r=0.06). If Bz versus r for z=0 is sought then the input would be:

```
i, 1, 1, 1
```

**NOTE:** If data **FILLED** by the program, the (i,j) pairs will have shifted by the filled coordinates. For example, if a single coil is used to generate fields on a rectangular grid, and the coil radial position falls between i=3 and i=4 and the vertical position between j=2 and j=3, then a **FILL** will insert an i=4 at the inner radius and an i=5 at the outer radius, shifting the old i's by 2. Similarly, the program will inset a j=3 and a j=4 at the coil lower and upper vertical extremes and shift larger j's by 2. The user must request the correct reordered (i,j).

**SOLDESIGN** function plots have a dashed line background grid through each major (labeled) tick mark to aid in reading numeric values. If the user wants to suppress this

grid, then an entry of IOUT=1 can be used.

Figure 3 illustrates some of the function and vector plot capabilities of SOLDESIGN. The coils and field generating command are:

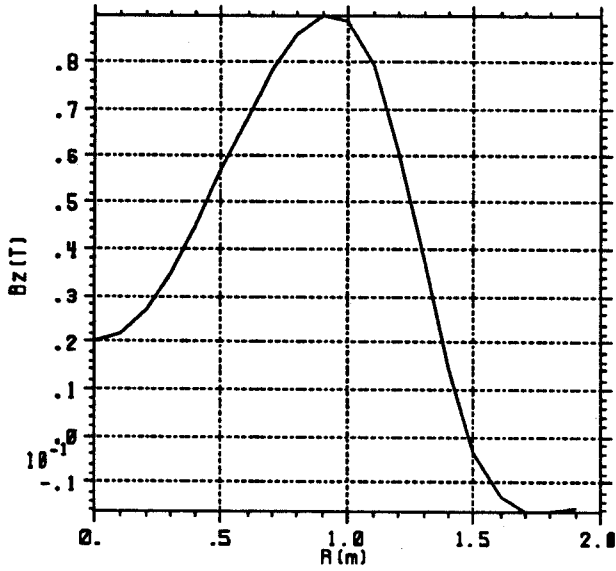
```
setup,ni,centroid
symmgroup
solenoid, 1.3, 0.4, 2, 0.4, 0.4, 1.e5,
solenoid, 0.5, 0.6, 2, 0.4, 0.2, -1.e5,
end
field,0.0,0.0,0.10,0.10,21,21
```

The plot commands are written at the top of each of the four plots in the space where the plot title is placed. The NOFILL option is used in all four of the examples. NOTE: It must be repeated each time.

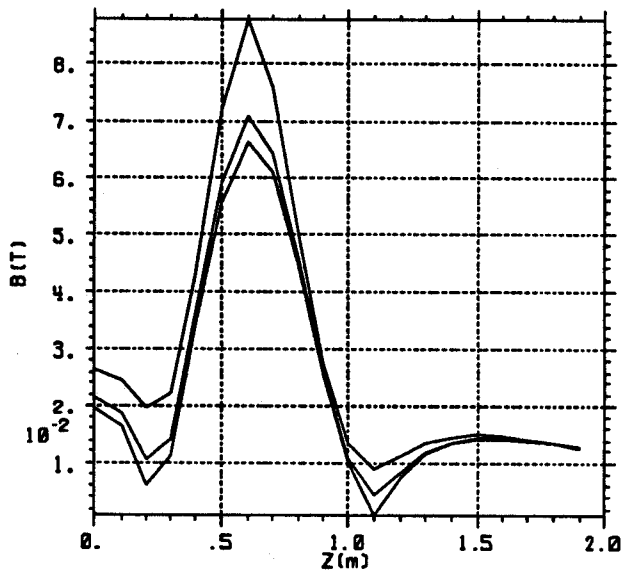
The figure in the upper left corner is obtained by requesting a function plot of Bz versus r or i for j = 1 to 1 with increment 1. That is, the Bz along z = 0 is plotted. SOLDESIGN will accept the characters "r" or "z" as readily as "i" or "j".

The figure in the upper right corner represents a plot of B-magnitude versus j or z for i = 1, 2 and 3 or for r = 0.0, 0.1, and 0.2, respectively. The present version of SOLDESIGN does not label curves or change linestyles. However, such a change is relatively easy and will be made in future versions.

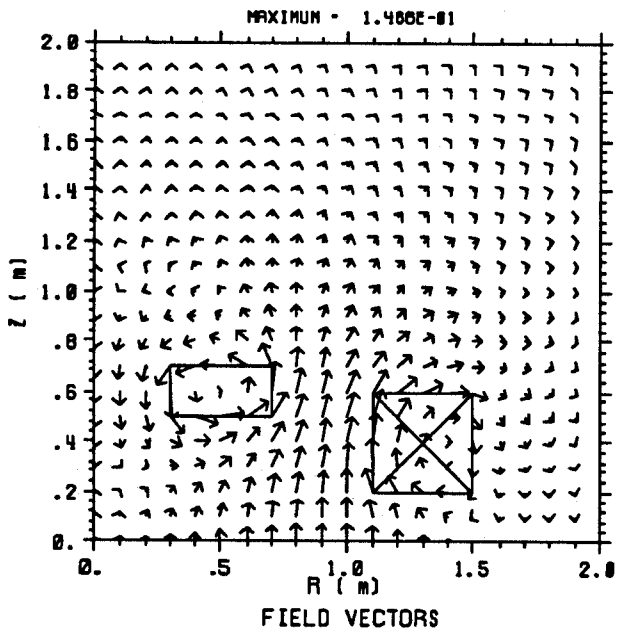
FUNCTION, BZ, 0, 2, NOFILL  
I.1.1.1



FUNCTION, B, 0, 2, NOFILL  
J.1.3.1



VECTORS, B, 0, 2, 0, 2.1, NOFILL



VECTORS, B, 0, 2, 0, 2, 0.2, 1.P, NOFILL

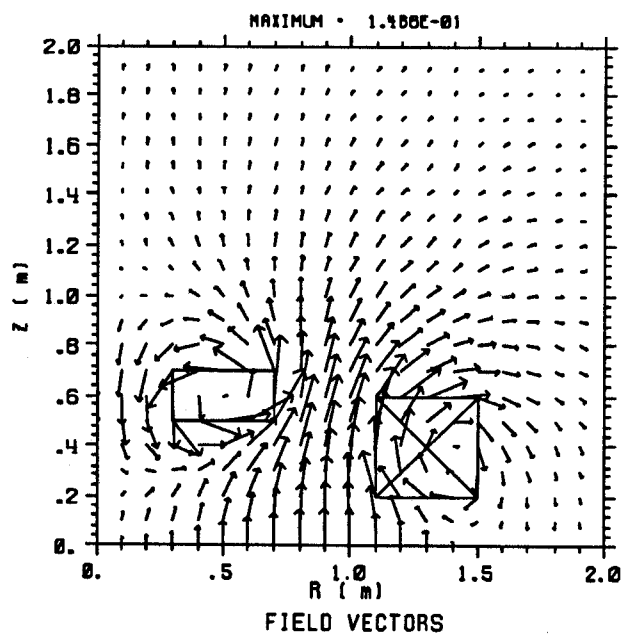


Fig. 3. SOLDESIGN FUNCTION AND VECTOR PLOTTING EXAMPLES

## VECTOR PLOTTING

Vector plots do not require a second command line. For vector plots, the second entry (TYPE) can be FIELD or FORCE. A third alphabetic input of 'P' may be invoked to draw the arrowhead proportional to the length of the shank. Otherwise, the arrowhead is constant in size. The scale option changes the length of the vectors.

The two lower plots in Figure 3 illustrate the use of the VECTORS option. The plot in the lower left corner is of the magnetic flux density or field vectors with only the coil outlines as an option. The plot in the lower right corner shows the same information but with a scale factor of 0.24 instead of the default 0.12 and a arrowhead proportional ("P" option) to the shank.

## HINTS AND FURTHER EXPLANATIONS

### URNS, AMPERE-TURNS or CURRENT PER TURN?

The actual number of turns in a coil may not be known explicitly even though the winding pack envelope and overall current density (or total ampere-turns) are known. SOLDESIGN was originally written to accept only the overall current density and did not calculate inductances or resistances. Over the centuries, the NI option and inductance and resistance calculations were added. For the latter two options, the number of turns in the coil is needed. Hence, the addition of the turns entry in SETUP. The turns are not used in the field or force calculations except with the READCURR option.

If SOLDESIGN is used to generate input to other codes such as circuit codes, then **CARE MUST BE TAKEN TO USE THE ENTRIES TURNS AND AMPERE-TURNS CORRECTLY**. Unfortunately, this requires some input file changes depending on the desired output.

If the total coil inductance and resistance are desired, then the turns entry must be used. Otherwise, the resulting output is on a per-turn-squared basis - *i.e.* the loop voltage equation is on a per-turn basis and the currents in the equation are ampere-turns.

If the field or force influence coefficients (*i.e.*, field components at a point per unit current in a coil) are desired for use in other postprocessing routines, the ampere-turns entry is important and the turns are ignored.

Now, here's the problem. If the inductance and resistance matrices have been calculated using the correct number of turns, then circuit code output is on a current-per-turn basis. If these currents are to then be used with field or force influence coefficients, then the SOLDESIGN coefficient run must be made with the NI equal to the number of turns. That is, the desired output is the field components at a point per unit (turn) current, not the field per unit ampere-turn. NOTE: if READCURR is used, the currents read from the file ARE multiplied by the turns on the SOLENOID definition line.

### READFIELD and the MISCELLANEOUS OPTION for PLOTTING

The READFIELD and MISC plot options were incorporated into SOLDESIGN to allow quick access to contour and function plotting routines. SOLDESIGN requires the format:

```
i j r(i,j) z(i,j) data(k),k=1,ncol
```

where ncol is the total number of data columns in the file. If ncol is greater than 4, then the program prompts for four columns to be used. This requirement is mandated by the storage scheme in SOLDESIGN. The scheme is: r(i,j), z(i,j), br(i,j), bz(i,j), bt(i,j), flux(i,j). The READFIELD routine stores the four columns in the br, bz, bt (mod b),



and flux arrays. Therefore, to shortcut some of the MISC plot input such as titles and column numbers, the regular command such as CONTOURS,FLUX would contour the values stored in the fourth array. At this time, there is no MISC option on the VECTOR command. However, if the desired vector data are read into columns one and two ( $B_r$  and  $B_z$ ), the VECTOR, FIELD option will work.

## MULTIPLE FIELD COMMANDS, COIL FILLING, ETC

**SOLDESIGN** uses two sets of coordinates for calculating, storing, reading and plotting field quantities. It basically assumes calculations are performed on a rectilinear mesh. The indices  $(i,j)$  are used along with the coordinates  $(r,z)$ . The contouring, vector, and function plotting routines make use of these two sets of coordinates to make the plotting more efficient. The various plotting functions may be thought of as piecewise linear interpolations – or connect the dots.

Since **SOLDESIGN** uses a uniform current density coil, there are no jump changes in field components across coil boundaries. However, the derivative of the tangential field component is discontinuous across the boundary. If a field calculation is done with field points not on the coil boundary, a contour or function plot across the boundary will “connect the dots” and not show the correct break in slope. The values at both points are correct, but to get the correct break, the fields need to be calculated on the coil boundary.

An attempt is made in **SOLDESIGN** to include the coil boundaries correctly, if the user has not done so. The mesh is expanded to include the boundaries with the appropriate shifting of the other data. This feature is not particularly efficient and can sometimes cause mysterious effects. The **NOFILL** option in the plotting package overrides this option, but can lead to funny looking plots if a coil does lie within the region of the plot.

One solution is to mesh the original **FIELD** command(s) so that coil boundaries are automatically included in the mesh(es). The original inclusion of the **START** option was meant to aid the user in using multiple **FIELD** commands. The recent inclusion of the **istart** and **jstart** options on the **FIELD** command line is an alternate (and I think better) way to handle this. A point with the same  $(i,j)$  and  $(r,z)$  can be defined more than once. Only the last calculation is used in plotting. If this option or the **START** option are used, then **NOFILL** can be used. Indeed, if it is not used, the program tolerances may be such that **SOLDESIGN** will attempt to fill anyway.

## THE USE OF THE LOOP SWITCH OPTION

The **LOOPSW** option was added to **SOLDESIGN** in order to speed up the calculation of fields, forces, and inductances for a system with a large number of coils. For field calculations, when the field point is sufficiently far from the coil, a loop calculation is used rather than the uniform current density formulation. “Sufficiently far” is defined in **SOLDESIGN** to mean when the distance from the coil center to the field point,  $dis$ , is

greater than a user specified tolerance times the coil diagonal, *i.e.*, when  $dis > tol * dia$ . This test is rather arbitrary, but seems to work. The diagonal measure is used to account for the possibility of a thin, but tall, coil. Typical tolerances of 2 to 5 may be used.

For the calculation of the mutual inductance between coils, the **LOOPSW** option allows a smaller number of radial current sheets to be used than that specified on the **SOLENOID** definition line. If the distance between coil centers, *dis*, is greater than *tol* times the coil diagonal, the number of sheets specified in the **LOOPSW** command is used. Testing on generic problems indicates that two radial sheets with tolerance of 5 give very good agreement with the mutual from a 5 or 10 sheet integration, with a significant savings in time.

Errors in field and inductance calculations can be significant if the tolerance or number of sheets is too small. As always, it is up to the user to make the right choice – this program is not artificially (or otherwise) intelligent.

#### GRAFLIB AXIS LABELING ON THE VAX

**GRAFLIB** was chosen as the underlying graphics package primarily because it was supported on both the **VAXs** and the **CRAYs**. It is a pretty good package. However, the **VAX** version has a bug that sometimes rears its ugly head. **GRAFLIB** chooses the number of major tick marks on each axis that will be have numeric values printed. Occasionally, a set is chosen with minimum and maximum values and a label increment that does not get properly rounded and the result is either a string of 9's or 0's or an asterisk plotted as the tick label. Once in a while, the entire axis line and all tick labels are omitted. The only cure for this is to replot with a forced scale different from that which lead to the funny result.

#### SUBDIVIDING COIL FOR DETAILED FORCE CALCULATIONS

**SOLDESIGN** has the capability of automatically subdividing coils into a number of subregions. This addition was meant to allow the user the capability of calculating inductance, resistance, *etc.*, for a rectangular region with multiple elements. It should not be used to calculate the force distribution over a coil by actually calculating the forces on a number of subcoils. It is not an efficient process, since the force calculator treats each subregion as a distinct and independent coil and calculates the field produced by all the subregions (rather than simply calculating the fields from the overall coil and integrating only over the subregion).

## ERROR MESSAGES

SOLDESIGN has a number of built-in error traps. If a program trapped error occurs, an error number and message is printed to the terminal (or log file in a batch mode) and to the output file. Depending on the input stream and severity of error, the program may try to recover or it may close files and terminate execution. A typical error message has the form:

```
*****  
Error Number iiii encountered in subroutine SUB  
BRIEF EXPLANATION OF ERROR COMMAND i1 i2  
*****
```

where

- iiii - numeric error code - codes are listed below.
- SUB - subroutine name where error occurred.
- BRIEF... - a brief explanation of the error.
- COMMAND - some more information (10 characters) about the command causing the error - for example, a line EMD rather than END would have EMD in the error message.
- i1, i2 - two integers containing more information. For example, if the number of coils defined exceeds the dimension, i1 would contain the present coil number and i2 the allowed maximum dimension.

If the input is from the terminal, most errors will cause the error message to be printed and control to revert to the previous prompt. For example, a misspelled command in SETUP would print an error message and loop back for a new prompt. In batch mode, all errors cause the message to be printed and the program closes files (including the plot file) and terminates the run. In the table of errors, a return code of 0 implies that the error is fatal in all modes. A return code of 1 is fatal in batch mode, but not in terminal mode.

Of course, nontrapped errors (such as division by zero) will cause a system error to be printed and the program terminates. In this case, any plots will be unrecoverable since the file was not closed properly. The error codes are:

SOLDESIGN Error Codes

| error | return | i1    | i2     | sub       | command | EXPLANATION                     |
|-------|--------|-------|--------|-----------|---------|---------------------------------|
|       | code   |       |        |           |         |                                 |
| 1001  | 1      |       |        | MAIN      | command | ILLEGAL OR UNRECOGNIZED COMMAND |
| 1002  | 1      |       |        | FILES     | file    | FILE NOT FOUND                  |
|       |        |       |        | READCURR  |         |                                 |
|       |        |       |        | READFIELD |         |                                 |
|       |        |       |        | USERDRAW  |         |                                 |
| 1003  | 0      | 0     | ntime  | READCURR  | file    | EOF ON CURRENT DATA FILE        |
| 1004  | 1      |       | 0      | SETUP     | command | ILLEGAL SETUP COMMAND           |
| 1005  | 0      | ngrp  | maxcls | SETUP     | command | MAX. NUMBER OF COILS EXCEEDED   |
| 1006  | 0      | ncls  | ngrp   | SETUP     |         | ZERO DR or DZ                   |
| 1007  | 0      | narea | 0      | SETUP     |         | ZERO QUAD AREA                  |
| 1008  | 0      |       |        | GETB      |         | CAN'T GET DERIVATIVES AT RP=0   |
| 1009  | 0      | imax  | maxi   | GETB      |         | MAX. NUMBER OF Is EXCEEDED      |
|       |        |       |        | COILFILL  |         |                                 |
| 1010  | 0      | jmax  | maxj   | GETB      |         | MAX. NUMBER OF Js EXCEEDED      |
|       |        |       |        | COILFILL  |         |                                 |
| 1011  | 1      |       |        | SOLPLOT   | command | ILLEGAL PLOT COMMAND            |
| 1012  | 1      |       |        | CONTOUR   | command | ILLEGAL CONTOUR TYPE            |
| 1013  | 1      |       |        | CONTOUR   |         | NO DATA AVAILABLE               |
| 1014  | 1      | ncrv  | maxcrv | FUNCPLT   |         | MAX. NUMBER OF CURVES EXCEEDED  |
| 1015  | 1      | ncrv  | maxcrv | FUNCPLT   |         | ZERO CURVES REQUESTED           |
| 1016  | 1      | ncrv  | 0      | FUNCPLT   |         | ZERO POINTS REQUESTED           |

## REFERENCES

- [1] G. Brown, L. Flax, E. Itean, and J. Laurence, "Axial and Radial Magnetic Fields of Thick, Finite-Length Solenoids," NASA TR-R-170, Dec. 1963.
- [2] M. Garrett, "Calculation of Fields, Forces, and Mutual Inductances of Current Systems by Elliptic Integrals," Journal of Applied Physics, Vol. 34, No. 9, September, 1963.
- [3] W. Smythe, Static and Dynamic Electricity, Third Edition, McGraw-Hill, 1968.

## HANDY DANDY POCKET GUIDE TO SOLDESIGN COMMANDS

All input is lower case and free-field with comma or space separators. Upper case words imply alphabetic input; lower case imply numeric.

ENDLIST

FIELD,r0,z0,(dr,dz,nr,nz,ipolar,iprint,istart,jstart)

FILE,FILENAME

FORCE,group #,(coil #,intr,intz,iprint)

FORCE,LOOP,grp1,grp2,grpinc,(coil #,intr,intz,iprint)

HELP

INDUCT,igstart,igstop,jgstart,jgstop

LOOPSW,tol,ngmin

NOLIST

PEAKFIELD,coil #,(nr,nz,iprint)

PEAKFIELD,LOOP,nc1,nc2,(nr,nz,iprint)

PRINT

PLOT

GRAPHICS TERMINAL (y OR n)?

if y SAVE FILE (y OR n)?

INPUT PLOT TITLE

PLOTTYPE,TYPE,rmin,rmax,zmin,zmax,(scale,iout,iprint,DRAW,NOFILL)

PLOTTYPE=COILS TYPE is omitted and no additional input needed

PLOTTYPE=CONTOURS TYPE=BR, BZ, B, FLUX, HOMOGENEOUS, or MISC  
ncon,(pmin,dp,iplt,htd,lblfrq,ilbl,ndec,lntyp,LOG,STYLE,DRAW,MORE) ?

PLOTTYPE=FUNCTION TYPE=BR, BZ, B, FLUX, HOMOGENEOUS, or MISC

I(or J),ijstart,ijfinal,inc

PLOTTYPE=HEADER TYPE not needed.

PLOTTYPE=VECTORS TYPE=FIELD or FORCE

PLOTTYPE=END control returns to SOLDESIGN

READ,FILENAME

READCURR FILENAME

ncirc,time,tol

READFIELD

FILENAME (terminal input)

ncol if ncol > 4 i1,i2,i3,i4 (column numbers to be saved)

READFORCE,FILENAME

SAVE,FILENAME

SAVEFORCE,FILENAME

SETUP,(NI,CENTROID,CORNERS)

SOLENOID,r1,z1,ism,dr,dz,cdens,(turn,ngauss)

SOLENOID,r1,z1,ism,dr,dz,NI ,(turn,ngauss) if NI

SOLENOID,rc,zc,ism,dr,dz,cdens,(turn,ngauss) if CENTROID

SOLENOID,rc,zc,ism,dr,dz,NI ,(turn,ngauss) if NI & CENTROID

SOLENOID,r1,z1,ism,r3,z3,cdens,(turn,ngauss) if CORNERS

SOLENOID,r1,z1,ism,r3,z3,NI ,(turn,ngauss) if NI & CORNERS

.

.

.

OR

QUAD, r1, z1, r2, z2, r3, z3, r4, z4, isym, cdens, (turn, ngauss, nrads)

QUAD, r1, z1, r2, z2, r3, z3, r4, z4, isym, ni , (turn, ngauss, nrads)

.

.

.

SUBDIVIDE,nnr,nnz

GROUP,rho

ENDGROUP

RHO,rho

SYMMGROUP,rho

...

END

START,istart,jstart

STOP

TERMINAL

UNITS,units