

UNITED STATES DEPARTMENT OF THE INTERIOR
GEOLOGICAL SURVEY

Guide for the Calculation of Coal Resources
Using the Computer Program GARNET

By Sharon S. Crowley, Antoinette L. Medlin, and Ricke J. Smith

Open-File Report 81-563

This report is preliminary and has not been reviewed for conformity with U.S. Geological Survey editorial standards and stratigraphic nomenclature. Any use of trade names is for descriptive purposes only and does not imply endorsement by the USGS.

TABLE OF CONTENTS

I.	INTRODUCTION.....	1
II.	HOW TO GET FROM PACER TO GARNET.....	2
III.	GENERAL RULES FOR GARNET.....	4
IV.	CREATING PLOT TAPES.....	5
V.	HOW TO ENTER GARNET.....	6
VI.	COMMANDS:	
	1. grid.....	7
	2. contour.....	8
	3. trend.....	10
	4. edit.....	12
	5. locate.....	14
	6. addpt.....	15
	7. window.....	16
	8. unwindow.....	18
	9. opgrid.....	19
	10. boundary.....	20
	A. display.....	21
	B. combine.....	22
	C. reverse.....	25
	D. cursor.....	26
	E. window.....	28
	F. unwindow.....	29
	G. valueset.....	30
	H. quit.....	31
	11. resource.....	32
	12. quit.....	34

INTRODUCTION

GARNET (Graphic Analysis of Resources using Numerical Evaluation Techniques) is a system developed by the National Coal Resources Data System (NCRDS). Through the invocation of approximately 20 commands, the user can create contour plots at specified map scales, calculate coal resource tonnages for a given area, edit data and add interpretive point data.

GARNET is used in conjunction with PACER, the retrieval system which accesses NCRDS data bases. The user retrieves relevant data from the data base of interest (USTRAT, USCHEM, etc.), then runs the data through the auxiliary program called VLATLONG. Once these steps are completed, the user may utilize the GARNET system.

GARNET is currently installed on the Honeywell/Multics 68/80 computer at the U.S. Geological Survey (USGS) in Reston, Virginia. It is designed primarily for "interactive usage", i.e., the user is prompted for commands and parameters while the program is running. However, many of the commands and operations can be accomplished in a batch mode by an experienced GARNET user to reduce costs. This manual is intended for the user who has had at least the "Introduction to Multics" course or equivalent experience.

The program VLATLONG is the interface between PACER and GARNET. It is used to create an "observed point" file, which is later read in GARNET to obtain a "gridded" file. It is assumed in VLATLONG that a PACER "search" has been made on the MCRDS data base of your choice - USTRAT, USCHEM, or USALYT - using the conditions that meet your requirements.

Sample run of VLATLONG:

(Prompts in upper case, user response and explanations in lower case.)

vlatlong

MAXIMUM NO. OF PTS OUTPUT = 1500
ALL ZERO LATITUDES OR LONGITUDES WILL BE SKIPPED

ENTER DATA BASE NAME: ustrat
ENTER FILE NAME: stratfil

(Enter name of output file you retrieved in PACER using the "search" command.)

ENTER VARIABLE NAME: thk

(Enter the variable you wish to plot.)

DO YOU WANT TO SKIP ZERO VALUES? yes
DO YOU WANT TO SUM THE THICKNESS IN EACH DRILL HOLE? yes

(You receive this prompt only if you chose "ustrat" as the data base name. All thickness values-rock, coal, no record - are added for each point or drill hole and the summed value is output.)

The screen clears and you receive a table similar the following:)

PLOT BOUNDARIES:
NORTH 39 20 00
SOUTH 39 16 0
EAST 92 31 0
WEST 92 36 30

CHANGE BOUNDARIES? (YES OR NO): yes

(You may wish to change corner points to those of the quadrangle in which the point data are located.)

KEY IN BOUNDARY VALUES OR
USE CROSS HAIR CURSOR? (K OR C): k

(Key in boundary values as explained below.)

TO CHANGE A BOUNDARY, KEY IN
DEGREES, MINUTES, AND SECONDS;

OTHERWISE, KEY IN '0 0 0'.

(Enter latitudes and longitudes for the corner points as follows.
Latitude determines north/south and longitude determines east/west.)

Example:

NORTH: 39 22 30
39 22 30
SOUTH: 39 15 00
39 15 0
EAST: 92 30 00
92 30 0
WEST: 92 37 30
92 37 30

TO CLEAR THE SCREEN AND CONTINUE,
HIT RETURN KEY.

(Press "carriage return".

The data points are displayed within the corner
points just entered. In case of error, you have
the option of changing boundaries once more:)

CHANGE BOUNDARIES? (YES OR NO): no

A NEW OBSERVED POINT FILE IS BEING CREATED, PROVIDE
NAME OF OBSERVED POINT FILE: testo1

(Enter an output file name of your choice.
File names cannot exceed 8 characters.)

III. GENERAL RULES FOR GARNET

1. Only the first 4 characters of each command need to be entered to invoke the command. For example, to use the "display" command you may enter "disp".

The screen clears before each prompt "ENTER COMMAND: ".

2. When GARNET prompts you for a gridded, observed point, or boundary file, you may press "carriage return" if you wish to use a file which is already loaded.

If you do not wish to use a boundary file, you must enter "none".

3. FILE NAMES CANNOT EXCEED 8 CHARACTERS.

4. Whenever GARNET makes a "hard copy" of the screen or shades an area, you must press "carriage return" to continue.

5. Manual Information:

GARNET prompts are in upper case and possible user response and explanations are in lower case.

When manual specifies user to enter a letter or command, a "carriage return" must follow. For example, to enter "x", type "x" and press "carriage return".

Commands are presented in order of possible usage, however, they may actually be used in any number of combinations. If you are using a PACER data file for the first time, you must invoke either the "grid" or "trend" command to create a gridded file. If you wish to display data points without contours, you may use the "locate" command. All other GARNET commands require a gridded file.

6. Digitized boundary files which are retrieved from the tape library are named according to the following convention. Fire Creek coal bed and Greene County are used in this example.

Type of Boundary	Suffix	Name
outcrop (bed)	-b	fireb
county	-co	greenco
burned	-x	firex
strip mined	-s	fires
deep mined	-d	fired
mined	-m	firem
federal		fed
urban		urb

How to write calcomp plot files to tape:

At the Multics Command Level, write calcomp plot files to tape using the following ec's:

1. ec cal1 xxxxxx (xxxxxx is the number of the tape which you have supplied to the Multics Computer Center or had assigned through the Multics TPMS system.)
2. ec cal2 plot1 (plot1 is the plot file made in "contour" or "resource".)

Repeat step 2 for each file that you wish to write to the same tape. When you have finished writing plots to tape, do a "new_proc". Plot tape on an offline Calcomp plotter, or request plot through the U.S.G.S. Reston Computer Center.

V. HOW TO ENTER GARNET

1. To use GARNET and peripheral programs, add these search rules to each section of your start_up.ec:

```
asr >udd>NCRDS>publib>g
asp ec >udd>NCRDS>publib>e
```

Do this once only.

2. To enter GARNET type:

garnet

(You receive the following prompts:)

ENTER TRANSMISSION RATE (BAUD): 9600

(Enter the appropriate number-
300, 1200 or 9600.)

ENTER '1' IF USING A TEKTRONIX 4010, '2' FOR 4014/4015: 2

(Enter "2" if you are using a Tektronix 4014, 4015, 4016, or 4054.)

TO CONTINUE, PRESS RETURN KEY: 23

(If you press "carriage return", you receive a list of GARNET commands and instructions.

If you enter "23" the list of commands is suppressed.)

FOR COMMAND LIST, PRESS RETURN KEY: 23

(Same as above.)

You may now use the following commands.)

1. ENTER COMMAND: grid

(The grid command is used to create a gridded file which is needed to produce contour plots.)

DO YOU WISH TO WEIGHT THE NEW SET OF VALUES WITH THE VALUES FROM AN OLD GRID?no

(This option allows you to vary a contour plot by weighting the grid you are creating with values from an old grid. If response is "yes", you are prompted for the name of the old gridded file.)

NAME OF OBSERVED POINT FILE: test01

(Enter the name of the output file created by the program VLATLONG.)

ENTER THE FRACTION OF MAP TO BE PLOTTED OUTSIDE OF THE RECTANGULAR BOUNDARY: 0.

(Enter "0." if you do not want plotting outside of the boundaries entered in VLATLONG.)

Enter a fraction (decimal form) if you would like contours to extend beyond quadrangle border. For example, enter ".1" if you would like contours to extend 10% past the border of the quadrangle.)

GRID SIZE IN MILES: .3

(The grid cell size is dependent on the radius of the smallest circle used in resource calculations. If the radius is 1/4 mile, ".3" (miles) is entered for the grid size.)

IS THE DATA DENSE AND DISTRIBUTED UNIFORMLY? yes

(If answer is yes, a "quadratic fit" is used to create the grid. If answer is no, a "planar fit" is used. You may want to create different grids by using quadratic and planar fits, and then compare contours to determine which gridding method is preferable for your data.)

If response is "no" and the observed point file contains less than 7 points, the following warning is printed:)

YOU HAVE LESS THAN 7 OBSERVATION POINTS.
THE RESULTS OF THIS GRID WILL BE UNRELIABLE.

(At this point, the program will pause while it is creating the gridded file. Running time increases with smaller scale maps (e.g. 1:100000) and a large number of data points.)

ENTER NAME FOR GRIDDED FILE: testg1

(File name cannot exceed 8 characters.)

2. ENTER COMMAND: contour

NAME OF OBSERVED POINT FILE: test01

(Enter file created from the program VLATLONG.)

NAME OF GRIDDED INPUT FILE: testj1

(Enter file created from the "grid" command.)

CONTOUR INTERVAL: 14

(Enter contour interval using the same units in which data was retrieved from the PACER data base- inches, feet or meters. In this example, the unit of measure is inches.)

FREQUENCY OF BOLD LINES: 1

(Enter a number that corresponds to how frequently contours are to be labeled and marked with a heavy line. For example, enter "2" to label every other line, "1" for maximum labeling of contours.)

FOR CONTOUR SMOOTHING, ENTER A "1": 1

(If you do not want contours smoothed, press "carriage return". Enter "1" if you want contours smoothed. The smoothing function generates more points, and therefore costs more to run in time and money.)

TO SPECIFY GRAPHICS DEVICE ENTER "0" FOR TEKTRONIX ONLY,
ENTER "1" FOR TEKTRONIX AND CALCOMP,
ENTER "2" FOR CALCOMP ONLY.

GRAPHICS DEVICE (0,1,2): 0

(If "1" or "2" is entered, GARNET automatically sets up a calcomp plot file. You will be prompted to name this file later.)

BOUNDARY FILE NAME: none

(Examples of boundary files are outcrop, overburden, and county lines. Refer to "General Rules for GARNET" for an explanation of boundary line naming convention.

Enter "none" if you do not wish to use a boundary.)

(If you have chosen graphics device "1" or "2", you receive the next 3 prompts:)

NAME OF TITLE/TEXT FILE: none

(Enter "none". This is a future option.)

NAME OF PLOT FILE: plot1

(Enter name for calcomp file. File name cannot exceed 8 characters.

This file must be written to tape, as described in Section IV, before it can be plotted offline.)

ENTER MAP SCALE: 24000

(Enter scale of the map you want. If scale is 1:24000, enter "24000", if 1:100000 enter "100000".

At this point, the screen clears and the contour plot is displayed. Information including names of files used, date, contour interval, and corner points of plot is displayed. When the plot is completed, a hard copy is made automatically. Press "carriage return" to return to "enter command" mode.

See "Creating Plot Tapes", Section IV, for instructions on writing plot files to tape.)

3. ENTER COMMAND: trend

NAME OF OBSERVED POINT FILE: testo1

DO YOU WANT TO USE NONSTANDARD SURFACE PARAMETERS? YES OR NO: yes

(If response is "no", the default parameters used are:
degree of polynomial: 2
order of trig series: 2)

ENTER DEGREE OF POLYNOMIAL: 3

(It is suggested to use a degree of 2 or 3. Do not use "3"
unless you have evenly distributed points everywhere, otherwise
contours will run to infinity.)

ENTER ORDER OF TRIGONOMETRIC SERIES: 0

(At this point you receive statistics similar to the following:)

EXAMPLE: THE FUNDAMENTAL FREQUENCY OF THE TRIGONOMETRIC EXPANSION IS
12.219.
(GIVEN IN THE X - Y UNITS)
THE COMPUTED SCALE FACTOR IS 0.282 E =01

FLS = .5277830d+06 CNORM = .1036185d+01 DNORM = .3129731d+06
FLS = .1618663d+00 CNORM = .1036187d+01 DNORM = .1750634d-02

THE TREND SURFACE HAS CONVERGED TO A SOLUTION...
PRESS THE CARRIAGE RETURN WHEN READY TO PROCEED.

(Press "carriage return". The screen clears and the following
is printed:)

THE SURFACE HAS BEEN FITTED TO THE DATA. THE ROOT MEAN SQUARE ERROR IS 2.414.

THE DATA VALUES WHICH EXCEED 3 TIMES THE ROOT MEAN SQUARE ERROR ARE:
XOB YOB ZOB ERROR

(There are no values in this sample which exceed 3 times the root mean square
error)

THE DATA VALUES WHICH EXCEED 2 TIMES THE ROOT MEAN SQUARE ERROR ARE:

XOB	YOB	ZOB	ERROR
3.214	3.288	51.0	5.4
3.672	2.991	53.0	6.2

(The errors 5.4 and 6.2 are greater than 2 times 2.414.
Press "carriage return" to continue.)

DO YOU WISH TO HAVE THIS TREND SURFACE GRIDDED? (YES OR NO): yes

(A gridded trend file is necessary to use the "edit" command.

If response is "no", the program returns to the "enter command" mode.
If response is "yes", you receive the following prompts.
The next 2 prompts are explained in "grid".)

ENTER FRACTION OF MAP TO BE PLOTTED OUTSIDE OF RECTANGULAR BOUNDARY: 0.

GRID SIZE IN MILES: .3

A NEW OBSERVED POINT FILE IS BEING CREATED, PROVIDE NAME OF OBSERVED POINT
FILE: ttesto1

FILE NAME TO STORE GRID VALUES: ttestg1

(You now have the gridded and observed point files needed to go into
"contour."

File names cannot exceed 8 characters.)

4. ENTER COMMAND: edit

HAS A TREND SURFACE BEEN DONE FOR THESE POINTS? (YES OR NO): yes

(In order to use edit, a trend surface must have been created and gridded.)

If you answer "no", the program returns to the "enter command" mode.

The next 7 prompts are explained in "contour".)

NAME OF OBSERVED POINT FILE: ttesto1

NAME OF GRIDDED INPUT FILE: ttestg1

CONTOUR INTERVAL: 14

FREQUENCY OF BOLD LINES: 1

FOR CONTOUR SMOOTHING, ENTER A "1": 1

TO SPECIFY GRAPHICS DEVICE, ENTER "0" FOR TEKTRONIX ONLY,
ENTER "1" FOR TEKTRONIX AND CALCOMP,
ENTER "2" FOR CALCOMP ONLY.

GRAPHICS DEVICE? (0, 1, OR 2): 0

(The trend surface contours are plotted on the screen.)

VALUES WHICH EXCEED 3 TIMES THE "RMS" ERROR.

(Values which exceed 3 times the "RMS" error are plotted and the cross hairs are displayed. You may adjust the values by positioning the cross hairs over a value, and entering "d", "c", or "n", as explained below.

To delete a value from the data file, position cross hairs over the point and enter "d".

To change a data value, position cross hairs over the point and enter "c". The new data value is then to be entered following the prompt for "value."

To proceed to the next step in the editing process enter "n".)

VALUES WHICH EXCEED 2 TIMES THE "RMS" ERROR.

(Values which exceed 2 times the "RMS" error are plotted. You may delete and change values. Enter "n" to continue.

After the "n" key is entered, a hard copy is made automatically. Press "carriage return" to continue.)

DO YOU WANT TO CREATE A FILE FOR THE EDITED OBSERVATION DATA? (YES OR NO): ye

(If you wish to save edited file, response is "yes".)

NEW OBSERVED POINT FILE IS BEING CREATED, PROVIDE NAME OF OBSERVED POINT FILE
ttesto1c

(File names cannot exceed 8 characters.)

To obtain a contour plot of the edited observation data, "grid" the observed point file, and then use the "contour" command.)

5. ENTER COMMAND: locate

(Refer to "contour" command for an explanation of the following prompts.)

NAME OF OBSERVED POINT FILE: test01

ENTER FRACTION OF MAP TO BE PLOTTED OUTSIDE OF RECTANGULAR BOUNDARY: 0.

TO SPECIFY GRAPHICS DEVICE, ENTER "0" FOR TEKTRONIX ONLY.
ENTER "1" FOR TEKTRONIX AND CALCOMP,
OR ENTER "2" FOR CALCOMP ONLY.

GRAPHICS DEVICE? (0, 1, OR 2): 0

(Points in the observed point file are plotted. A hard copy is made automatically. Press "carriage return" to continue.)

6. ENTER COMMAND: addpt

NAME OF OBSERVED POINT FILE: test01

NAME OF GRIDDED INPUT FILE: testg1

DO YOU WANT A CONTOUR PLOT? yes

(Refer to "contour" instructions for the next 3 prompts:)

CONTOUR INTERVAL: 14.

FREQUENCY OF BOLD LINES: 1

FOR CONTOUR SMOOTHING ENTER A "1": 1

BOUNDARY FILE NAME: none

(The points, contours, and cross hairs are displayed.

Position cross hairs over desired location. Enter "a" to add a point. You are prompted for values for each of the points that you add. enter values with a decimal.

Enter "n" when you are finished.)

A NEW OBSERVED POINT FILE IS BEING CREATED, PROVIDE NAME OF OBSERVED POINT FILE: test01a

(File name cannot exceed 8 characters.

To obtain a contour plot with the added points, "grid" this observed point file and then use the "contour" command.)

7. ENTER COMMAND: window

(The purpose of this command is to allow you to limit the gridded area you are working with- the smaller the gridded area, or frame, the less computer time it takes to work with boundary files. This command is especially useful prior to using the "combine" command.)

DEFINE BOUNDARY FRAME: testg1

(Enter name of gridded file.)

CURSOR INPUT? (YES OR NO): yes

(If response is "no", see below.

If response is "yes", you receive the following prompts:)

BOUNDARY FILE NAME: fireb

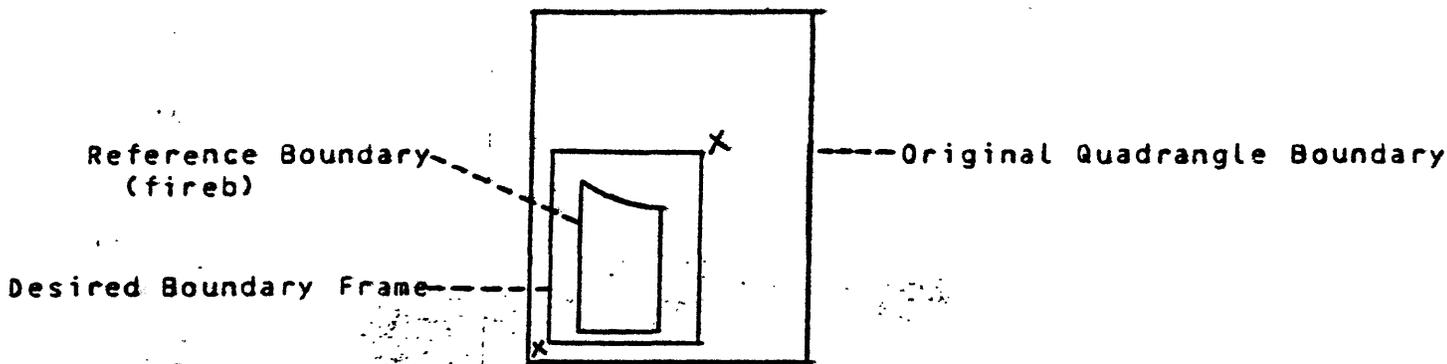
(The boundary file you enter is displayed as a reference file.)

NAME OF OBSERVED POINT FILE: none

(The observed point file is optional. You may enter "none".

The boundary file is displayed. To mark off area for your new grid, or "boundary frame", position cross hairs and enter "x" at one corner of the desired "frame", and another "x" at the desired corner diagonally opposite the first corner.)

EXAMPLE:



(A rectangle determined by the "x"'s you entered is displayed. Press "carriage return" to continue.)

(If response was "no" to "cursor input?" you receive the following prompts:)

Example:

NORTH: 82 20 00

SOUTH: 82 15 00

EAST : 36 52 30

WEST : 36 55 30

(Enter the values for minimum and maximum coordinates, using latitudes and longitudes, as in the example above. These points form the corner points for the area in which you wish to work.)

UNTIL you use the command "unwindow", or do a new_proc, the frame you just created will be your "boundary frame". When you are asked to "define boundary frame" in the future, press "carriage return".

The only way to return to the original grid or a different boundary frame is by using the "unwindow" command.)

8. ENTER COMMAND: unwindow

(You do not receive a prompt in this command. The total gridded file you used prior to the "window" command becomes your boundary frame.)

9. ENTER COMMAND: opgrid

(This command may be used in several applications.
In the following example, it is used to determine
the overburden of a quadrangle by subtracting the grid
for elevation of the top of a coal bed from the grid for
ground surface elevation.)

ENTER THE OPERATION SYMBOL FOR 'SURFACE ONE' (+,-,*,/) 'SURFACE TWO': -

(You may add, subtract, multiply or divide two surfaces (grids).
In this example, surface two is subtracted from surface one.)

ENTER FILE NAME FOR SURFACE ONE: topog1

ENTER FILE NAME FOR SURFACE TWO: coalg1

(Enter a gridded file for surface one and surface two. The
grids must have identical corner points and grid intervals.)

ARE NEGATIVE Z-VALUES ACCEPTABLE? no

(The z-value refers to the type of isoline map that is being created -
in this case, overburden.)

A NEW GRIDDED DATA FILE HAS BEEN CREATED. DO YOU WISH TO SAVE IT? yes

FILE NAME TO STORE GRID VALUES: overg1

(If you save and name the new gridded file, you may use it in the
"contour" command to obtain an isoline map.)

File names cannot exceed 8 characters.)

BOUNDARY COMMANDS

10. ENTER COMMAND: boundary

(Once you have entered the "boundary" section of GARNET, you may use any of the following boundary sub-commands:

display
combine
reverse
cursor
window
unwindow
valueset
quit

Only the first 4 characters need to be typed in for each sub-command.)

PRESS CARRIAGE RETURN TO CONTINUE: 23

(If you press "carriage return", you receive a page of printed instructions. Enter "23" to suppress printing of instructions.)

A. ENTER BOUNDARY COMMAND: display

DEFINE BOUNDARY FRAME: testg1

(The boundary frame is either a gridded file or a window. (See "window" command.) If you have just used the window command, press "carriage return".)

BOUNDARY FILE NAME: outcrop

(If the boundary file has been used before, the boundary file is displayed and shaded. Press "carriage return" to continue.

If the boundary file has never been used before, there is a prompt for you to place cross hairs over the area you want shaded and to enter "i". Once the area is shaded, you receive the following:)

THE RAW BOUNDARY FILE JUST DISPLAYED WAS NOT LOWER COMPLETE. IT HAS BEEN COMPLETED.

NAME OF OUTPUT BOUNDARY FILE: fireb

(When a file is "lower complete", the program knows which area (inside or outside of the polygon) to shade. You may call this file the same name as the "boundary file name".)

B. ENTER BOUNDARY COMMAND: combine

DEFINE BOUNDARY FRAME: testg1

(Enter name of a gridded file, or press "carriage return" if you are using the window command.)

BOUNDARY FILE 1: fireb

BOUNDARY FILE 2: greenco

(Enter the names of boundaries you wish to combine.)

The program pauses while reading in the boundary files. Boundaries are displayed with solid lines for file 1, dashed lines for file 2, and heavy solid lines for areas of intersection. Information including names of boundary files and corner points is displayed in the left margin.)

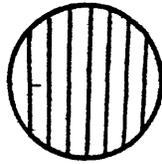
TO CREATE NEW BOUNDARY SET, ENTER:

1. IF YOU WANT ALL THAT IS IN fireb OR greenco
2. IF YOU WANT ALL THAT IS IN fireb AND greenco
3. IF YOU WANT ALL THAT IS IN fireb BUT NOT IN greenco
4. IF YOU WANT ALL THAT IS IN greenco BUT NOT IN fireb

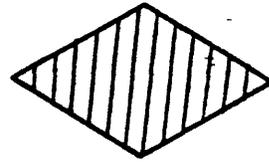
ENTER COMBINATION CODE: 2

(Choose logic according to the type of intersection you want. It is essential to know how the original boundaries were shaded. The table below shows the areas obtained by using different logic. The shaded area indicates the intersection.)

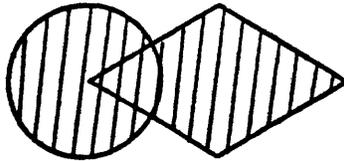
A



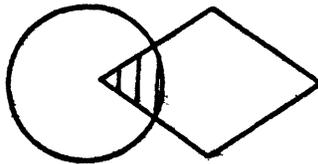
B



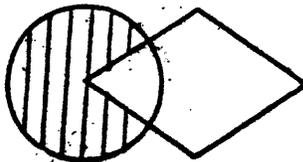
1. All that is in A or B



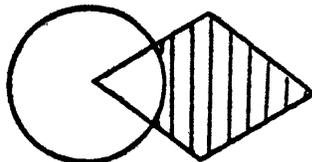
2. All that is in both A and B



3. All that is in A but not in B



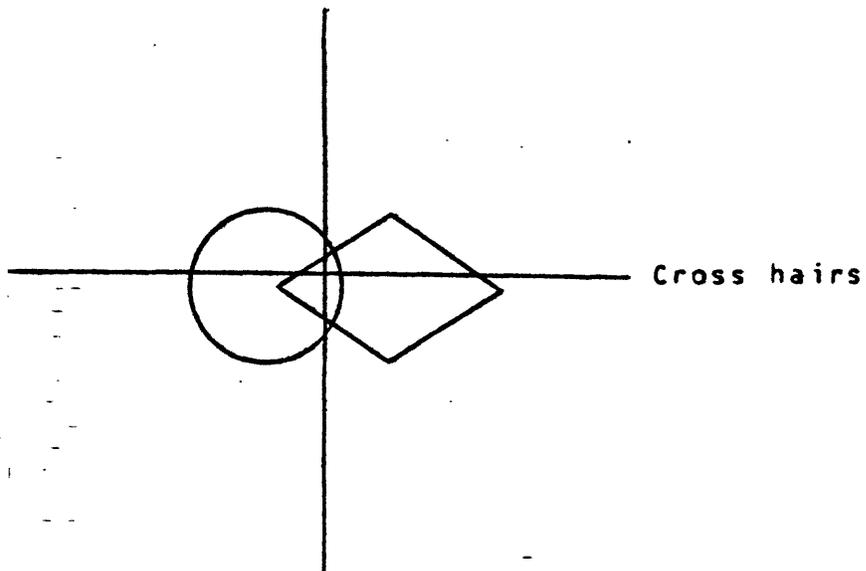
4. All that is in B but not in A



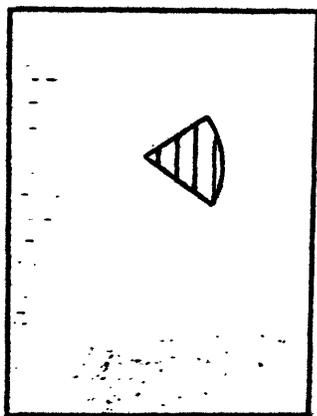
IF YOU WISH TO SAVE THIS FILE, ENTER NAME: firebgco

(File names cannot exceed 8 characters.)

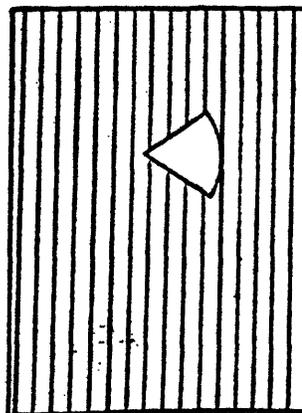
POSITION CROSS HAIRS IN THE DESIRED REGION. ENTER AN "i" TO DESIGNATE THAT REGION FOR INCLUSION IN THE RESOURCE CALCULATIONS.



(You could shade the inside or outside of the polygon, obtaining:



or



Once the area is shaded, press "carriage return" to continue.)

C. ENTER BOUNDARY COMMAND: reverse

DEFINE BOUNDARY FRAME: testq1

BOUNDARY FILE NAME: greenco

(The boundary file is displayed with the shading reversed.
The shading remains reversed until you invoke the "reverse"
command once more or do a "new_proc".

Press "carriage return" to continue.)

D. ENTER BOUNDARY COMMAND: cursor

DEFINE BOUNDARY FRAME: testg1

DO YOU WISH TO DISPLAY A BOUNDARY SET FOR REFERENCE WHEN CREATING THE NEW BOUNDARY SET? (YES OR NO): yes

(If response is "no", the boundary frame is displayed and you may define the boundary file you wish to create by using the cross hairs as described below.

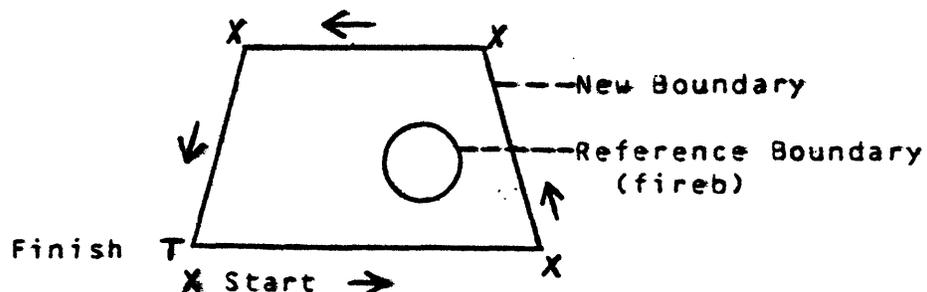
If response is "yes", you receive the following prompts:)

BOUNDARY FILE NAME: fireo

(The boundary file is displayed for reference.

To create a new boundary, position cross hairs and enter "x" to mark off line nodes, which are connected by straight lines. Enter "t" to terminate the line. Boundaries must be closed polygons unless both ends of the line intersect the map border, in which case lines must start and terminate outside of the quadrangle boundary.

EXAMPLE:



After you have entered "t" and "carriage return", you receive the following prompts:)

DO YOU WISH TO CREATE ANY MORE BOUNDARIES?(YES OR NO): no

(Additional polygons you create are saved under the same name.

Once you have finished creating polygons, all polygons are redisplayed.)

PLACE THE CURSOR IN THE DESIRED REGION. ENTER AN "i" TO DESIGNATE THAT REGION FOR INCLUSION IN THE RESOURCE CALCULATIONS.

(Position cross hairs and enter "i" in the desired region. The boundary is shaded. Press "carriage return" to continue.)

DO YOU WISH TO SAVE THIS BOUNDARY FILE? (YES OR NO): yes

(If the answer is "yes",)

NAME OF OUTPUT BOUNDARY FILE: polygon

(File-name cannot exceed 8 characters.)

E. ENTER BOUNDARY COMMAND: window

(See instructions for "window" command, Part 7.)

F. ENTER BOUNDARY COMMAND: unwindow

(See instructions for "unwindow" command, Part 8.)

G. ENTER BOUNDARY COMMAND: valueset

NAME OF GRIDDED INPUT FILE: testg1

ENTER Z-VALUE TO DEFINE BOUNDARY: 28.

(The z-value is the contour that you wish to pull off an isopach or isoline map. Use the same units in which data was retrieved from the NCRDS data base - inches, feet or meters. In this example, the unit of measure is inches.

The contour line(s) selected is displayed.)

PLACE THE CURSOR IN THE DESIRED REGION. ENTER AN "i" TO DESIGNATE THAT REGION FOR INCLUSION IN THE RESOURCE CALCULATIONS.

(The boundary is shaded according to where you position the cross hairs and enter "i".

Press "carriage return" to continue.)

DO YOU WISH TO SAVE THIS BOUNDARY FILE? (YES OR NO): yes

(If response is "yes",)

NAME OF OUTPUT BOUNDARY FILE: black28g

(Names cannot exceed 8 characters.

NOTE: It is not possible to use a "window" during valueset. If the "window" command is used prior to invocation of "valueset", the previous grid is used as a boundary frame.)

H. ENTER BOUNDARY COMMAND: quit

(Enter "quit" to return to the "ENTER COMMAND:" level.)

11. ENTER COMMAND: resource

AT EACH PROMPT, ENTER THE RELIABILITY RADIUS IN MILES.
ENTER THE SMALLEST RADIUS AT THE PROMPT AND CONTINUE TO ENTER THE RADII
IN ASCENDING ORDER. AFTER THE LAST RADIUS HAS BEEN ENTERED, ENTER
A "0" AT THE NEXT PROMPT TO TERMINATE THE INPUT PROCEDURE.

IF YOU ENTER A "0" FOR THE FIRST RADIUS, THE U.S.G.S. STANDARD
RADII OF 0.25, 0.75, AND 3.0 MILES WILL BE USED.

ENTER RADIUS FOR RELIABILITY CATEGORY 1.: 0

(In the example above, a "0" was entered for the first radius
and U.S.G.S. standard reliability resource circles will be used.
However, you may enter radii of your choice. Example: ".30"
for radius 1, ".60" for radius 2, and ".90" for radius 3.)

NAME OF OBSERVED POINT FILE: test01

NAME OF GRIDDED INPUT FILE: testg1

DO YOU WISH TO PLOT A RESOURCE MAP? yes

(If response is "no", the resource map is not plotted. You
receive only tonnage figures.)

DO YOU WISH TO PLOT THE RESOURCE MAP WITH A CONTOUR SURFACE? (YES OR NO): no

(The rest of the commands in "resource", up to the plot are identical to
those in "contour". Refer to "contour" instructions.)

During the plot, there is a pause after each set of resource
circles has been drawn. At this time the program is computing the
volume of that particular set of circles.

After the plot is finished, and you have pressed "carriage return", the
screen clears and you receive this prompt:)

ENTER TITLE (LESS THAN 32 CH): Demonstration Data Set.

ENTER TONNAGE PER ACRE FOOT: 1800.

(Refer to the table below for tonnage per acre-foot.)

Rank of Coal	Tonnage per Acre-Foot
Anthracite and semi-anthracite	2000.
Bituminous	1800.
Subbituminous	1770.
Lignite	1750.

Different tonnages per acre-foot may be entered, if you have additional information.

Once you have entered the tonnage per acre-foot, you receive a table similar to the following.)

TABLE OF RESOURCE ESTIMATES FOR Demonstration Data Set

	RADIUS -----	ACRE/FEET -----	TONNAGE (short tons) -----
MEASURED:	0.00-0.25 MILES	469.82	845681.35
INDICATED:	0.25-0.75 MILES	2994.45	5390018.56
INFERRED:	0.75-3.00 MILES	23918.55	43053388.50

(A hard copy is made automatically. Press "carriage return" to return to the "enter command" mode.)

12. ENTER COMMAND: quit

(This command takes you out of GARNET and back to the Multics Command Level.)