MCTYNM	county check between files

PURPOSE

To compare county names between a frame unit file (type 50) and a segment catalog file (type 49).

INPUT

1) A segment catalog file (type 49) must be supplied by the user, as well as a
2) A frame unit file (type 50).

OUTPUT

To the terminal, a list of counties not contained in both files will be printed.

HOW THE PROGRAM WORKS

Upon entering a frame unit file and a segment catalog file, the program compares the counties in both files.If a county is contained in the two files, the county will not be listed. If,on the other hand, a county is contained in one file but not the other, it will be listed under a heading of the type of file it was in.The other possibility is when all the counties are in both files.At this time, a message will state the similarity.

COMMAND SUMMARY

None.

HOW TO USE IT

This program is very simple for the user.The only input the user is to enter are the file names for the two files.If either of these are entered incorrectly or a question mark (?) is entered, a message to correct the situation will be displayed.

NOTES/LIMITATIONS

None.

MEDIT:	Mask file EDITor

PURPOSE

To create a submask file from an input mask file given the part number and submask coordinates that are within the mask file boundaries.Medit may also be used to change the field numbers in a file.

INPUT

1) A mask file (type 17) must be entered.
2) An ouput filename to be written to for the new mask file.
3) User entered data (ie coordinates, part number) in repsonse to the prompts.

OUTPUT

1) A type 17 mask file will be created.

HOW THE PROGRAM WORKS

This program is designed to create a submask or change the field numbers in a mask file. Once through the initial file prompts, directories and filename prompts,the user is to enter which part(s) are to be manipulated.Each part is separately run, therefore, header data is printed for each prior to manipulation.Once the data appears, the user has a menu which a listing of tract, field, and area for a part, a submask can be created and/or editing of a field can be an option for a part.

Note:
Even though a part is not manipulated (no submask is wanted), a message states that that part was written in its original form to the output file.

COMMAND SUMMARY

Create : Create a new mask file with the user given coordinates, that is, the submask of the input file.
List : List the tract, field and area data of the input file for the particular part that is being manipulated.This does not mean that a submask is being created.
Edit : Editing the field numbers for user entered fields.
Quit : Exits this manipulation of a part ONLY. It does not exit the program until all parts, manipulated or not, have been written to the output file.Upon then completion of the last part, only then will it exit the program.

HOW TO USE IT

Medit is a combination menu and prompt oriented.
Upon the start of the program the user is required to enter the directory(ies) which the input file(s) can be found, the input file name, the output file name, and the part(s) which require

further manipulation. Carriage return <cr> to the part number prompt means that there are no more part entries to be made.
At this time, data for the first part will appear. Either a prompt "#:" or a message stating that the part was written to the ouput file will follow.
If the prompt appears, four options are possible in response:

a) CREATE a submask which is as stated above in the command summary,

b) EDIT field numbers asked for the field which is needed to be changed and the new field number it is to be replaced with.

Note:Even after using the list option, the user can create a submask or quit for this part.

c) LIST tract and field information as stated above.

Note:Even after using the list option, the user can create a submask or quit for this part.

d) QUIT do no manipulation but write it in its original form to the output file.

At the end of the part, a message stating the manipulation is written.It will now proceed processing parts until all parts are completed.

NOTES/LIMITATIONS

The part manipulation in Medit is in numerical order. It will not manipulate part two prior to part one since writing to an output file in this order would require a great amount of storage or overwriting data.Therefore, the user may enter the part numbers in any sequence, but the program will sort these into numerical order.

MSKGEN	segment MaSK GENeration

PURPOSE

To generate a mask file from a segment network file

INPUT

1) Calibration File (No MARS-PED file type)
2) A type 47 Segment Network File
3) File of Segment Shifts (if available)
4) Segment Region File (if available)
5) Possible user entered data
 a) state name and year
 b) window border for output mask window file
 c) Landsat frame date
 d) boundary width for boundary generation

OUTPUT

1) A type 17 Segment Mask File
2) File containing mask window coordinates

HOW THE PROGRAM WORKS

With a calibration file as input,the program converts all vertices of a segment network file to row and column coordinates. These coordinates for all of the segment edges are used to create a rasterized version of the segment network file - the mask file. The mask file contains several entries describing the number of pixels of each field that exist in varying locations on each raster line.
If desired, pixels that are within a specified distance from any segment edges can be flagged as boundary pixels.

COMMAND SUMMARY

Mskgen does not have a menu-driven command structure.

HOW TO USE IT

Upon entering the program, the user should have the following:

1) A list of segments for which mask files will be generated.
This list may be numbers
 a) on a piece of paper,
 b) in a 'segment region' text file which is of the form:
  SEG(a,b,....)# where a,b, etc are segment numbers,
 c) in a text 'file of segment shifts' which is of the form:

seg1, rshift1, cshift1
seg2, rshift2, cshift2

.where seg1,seg2, etc. are segment numbers and rshift1 and cshift1, etc. are the corresponding row and column shifts.

2) **Segment Network Files** on the user's directory for each input segment which have file name of the form of the default for the system being used.

3) **A global calibration file** applicable to all of the segments to be processed during this execution. The program extracts the Landsat frame name from the first line of the file for output mask file name generation.

4) **The date of the Landsat scene** applicable to this execution.

EXAMPLE

```
Below is presented a sample script of a typical session. User input
is terminated with a <CR>. Each user prompt is preceded with a
number or letter followed with a ')'.
Comments follow each prompt with a '-'.

Peditor Version

1) Enter State and Year Identifier: KS83<cr>
- enter a 2 character state abbreviation followed by a 2 digit year
designation. This designation is later placed in the generated mask
file name and in the mask file itself.

 2) Enter "SE" for Segment or "ST" for Strata or <CR> for other:
SE<cr>
- if segment network file names are in the default form, then enter
SE.
- if the names have a .STRATA extension(i.e., XXXX.STRATA, where
XXXX is the segment number), enter ST.
- if the names are to be entered from the terminal, enter <cr>.

3) Save mask window coordinates in a file? (Y or N): Y<cr>
- a text file is written with the north, west, east, and south row
or column coordinates augmented by the border.
If yes-
a) Output File: MSK.WIN<cr>
- specify a text file name which will hold the above data.
b) Window border (Default = 1): 2<cr>
- the extent of the window as determined by the program will be
enlarged by two times the border selected in both directions. (i.e.
north becomes north-border, east becomes east+border, etc.).

4) Global calibration input file name: 3134.CAL<cr>
- enter a global calibration file name normally of the form:
xxxx-xxxxx.PCAL - #/aabb or
xxxxx-xxxxx.CAL - #/aabb.

5) Use other directories? (Y or N): N<cr>
- enter names of directories (other than the current) that are to
be searched when segment network file names must be found.

6) Landsat frame date: 11/8/74<cr>
- enter Landsat scene date in the form:
month/day/year where month and day may be one or two digits while
year is two digits.(e.g. 1/30/83).

7) Generate boundary info? (Y or N): Y<cr>
- enter Y if boundary delineations are to be inserted into the
masked data
```

```
if yes-
a) Boundary width (Default = 1.0): 3.0<cr>
- enter a number denoting the number of pixels wide that the
boundaries between fields will be.

8) Use a file of segment shifts? (Y or N): N<cr>
- enter Y if the list of segments is in a text file along with
their corresponding segment shifts.
if yes (the following is presented for completeness)
a) Segment shift file name: AD31-SEG.SHIFTS<cr>
- specify an existing segment shift file.

9) Use a segment region file? (Y or N): N<cr>
- enter Y if the list of segments is in a segment region file.
if yes (the following is presented for completeness)
a) Segment region file name: AD31.REG<cr>
- specify an existing segment region file.

If the segment numbers are not in any file, the following is
queried:

10) Enter segment numbers, one per line. End with CR only.
3134<cr>
3430<cr>
- enter the segment numbers, each one followed by a CR.
a) Segments selected OK? (Y or N): Y<cr>
- double check all segment numbers for accuracy. If not OK,then
restate all segment numbers.

Segment Network data processing is begun at this point with the
announcement of the segment being processed. (e.g. Processing
segment number 3134).
If neither SE not ST were specified near the beginning of the
program,the following prompt will appear (this is presented for
completeness):

11) Enter segment network file name for segment x: NEWFILE.SEG<cr>
- x is the segment number for which a file name is to be entered.

After receiving valid segment network file data, the mask output
file name is generated and checked for duplication in the user's
directory. If it already exists, the user may overwrite the file or
type in the name of a file which is to receive the masked data.
At this point, the following information is displayed for each
segment:

13 fields, 55 edges, 43 vertices
Total segment acreage is 5237642.00
Latitude 38.00 Longitude 98.50
Segment Scale is 1 : 502058
One inch = 7.92 miles
Mask Window: -300 -43 2184 3634
Mask output file is 3134.MASK/1234-56789/KS83

At the conclusion of the mask processing for the last segment in
the list, program execution is terminated.
```

NOTES/LIMITATIONS

It may not work if the input segment network file has digitization errors.

MSPLIT	SPLIT Mask by frame

PURPOSE

Several mask files are often created for one file of segment or strata data due to the overlap of Landsat frames. Each of these mask files may contain data common to other mask files as well as filler (blank) data. Module MSPLIT allows the extraction of selected parts of each mask (submasks) so as to eliminate frame overlapping and filler data.

INPUT

1) User entered data
   a) county name
   b) Landsat frame name for each input mask
   c) strata mask type
   d) state name and year
   e) directories
   f) left and right frame margin to avoid filler
   g) limits of usable frame area
   h) column filler in the four corners of the usable frame area
2) Calibration File (no MARS-PED file type) for each input Landsat frame
3) A type 17 Segment or Strata Mask File for each input Landsat frame

OUTPUT

1) A type 17 Segment or Strata Submask File(s)

HOW THE PROGRAM WORKS

After all calibration file and mask file data for all frames are received for a county, each frame is processed in order of user specified priority. All of the area in the first frame (mask) that is within the county mask is retained for output to a submask file. Each subsequent frame is processed in terms of the previous frame's calibration file data and only those areas not already output for a previous frame will be written to other submask files created for each county and frame combination.

COMMAND SUMMARY

MSPLIT does not have a menu-driven command structure.

HOW TO USE IT:

Upon entering the program, the user should have the following:
1) The names of each county to be processed.
2) Calibration files on disk for each mask from which split masks will be generated for each county.
3) Mask files on disk for each frame from which split masks will be generated for each county.

4) The extent of usable data within the Landsat frame.

EXAMPLE

Below is presented a sample script of a typical session.User input is terminated with a <cr>.
Each user prompt is preceded with a number or letter followed with a ')'.
Comments follow each prompt with a '-'.

```
Mask splitter by frames [ Version 1.0 2/24/84 ]
1) Enter State and Year Identifier: KS83<cr>
- enter a 2 character state abbreviation followed by a 2 digit year
designation. This designation is later placed in the generated mask
file name and in the mask file itself.

2) Enter directories to use, one per line; CR only when done.
: <cr>
- enter other directories to be searched for input calibration
and/or mask files.

3) Extra margin on the left and right to avoid filler
(default = 5): 0<cr>
- enter number between 0 and 20 to avoid the possible inclusion of
filler in the output submask.

** Here begins the frame acquisition loop **

Enter within this loop all Landsat frames that will be used for all
counties to be processed later.

4) Frame: 1234-67890<cr>
- enter a valid Landsat scene identification number. A <cr> only
causes the frame acquisition loop to be exited.
At this time, a calibration file name is automatically generated,
and a search for a file with that name is initiated with the
message:
Searching for calibration file with generated name of 0.m7890KS83.
(See NOTES below regarding file name generation)
If a file with the generated calibration file name cannot be found,
the user has the option of either stating the actual file name or
listing other directories through with the program can search:
a) Select a course of action: e<cr>
Enter File Name
b) Calibration file name for frame 1234-67890: 3631.PCAL<cr>

5) Specify the limits of usable area in the frame.
Upper row (Default = 1): <cr>
Lower row (Default = 2983): <cr>
Left column (Default = 1): <cr>
Right column (Default = 3548): <cr>
- enter the portion of the frame to be used for processing

6) Filler in upper left corner (Default = 0): <cr>
 Filler in upper right corner (Default = 0): <cr>
 Filler in lower left corner (Default = 0): <cr>
 Filler in lower right corner (Default = 0): <cr>
- enter number of pixels of filler data in the corners of the above
specified usable area in the frame. At this time, a summary of the
frame is printed. The capital letter is the frame symbol for this
frame, which is used in the later selection of frames for each
county.
Frame= 1234-67890 (A)
Four corners are (row,col,lat,lon) in order NW,NE,SW,SE
        1,          1,          44.008,       105.121
        1,          3548,       43.619,       102.667
        2983,       1,          42.508,       105.539
        2983,       3548,       42.124,       103.146

** Here ends the frame acquisition loop **
```

This loop repeats until a <cr> only is entered at the "Frame" prompt.
(In this example, one other frame was also entered)
At that time, a summary of all frames entered is displayed:

Total Number of frames = 2
Symbol Frame
A 1234-67890
B 6543-56721

** Here begins the county processing loop **

7) County: 2414<cr>
- enter county name (used in submask file name generation)

8) Enter Mask Type (Default is "seg"): <cr>
- if the mask file names on disk are in strata mask format, then enter "str", else enter <cr> only or "seg".

9) Enter list of frames (by symbol) for county 2414 in order of priority
: b,a<cr>
- enter frame symbols(letters) corresponding to frames in the frame summary that are to be used for mask splitting for this county. The list of entered symbols is checked syntactically and for the existence of frames for each input symbol. Once a valid symbol list is entered, file names are generated for each input mask file corresponding to each frame symbol in the list. A search for each file is initiated with the statement:
Searching for mask file for frame 6543-56721 with name of 2414.m6721KS83
(See NOTES below regarding file name generation)
-If the file is invalid for any reason, the user must respond to:
a) Select a course of action: e<cr>
Enter file name
b) Input Mask File: 3631.MSK<cr>
-The four possible responses to the "Select" prompt are:
a- ignore this frame = remove this frame from further processing for this county only.
b- re-enter entire frame list = go back to "Enter list" prompt.
c- enter new directory names = search new directory list for file name.
d- enter file name= manually type in the desired file name.
-When a set of valid mask files is obtained, the procedure of calculating a submask for each frame in the county is initiated with a statement such as:
Begin processing county 2414 in frame 6543-56721 (B)
-If the processing for a frame is successful, the submask file is then created with a generated file name and a message is printed:
File 2414.m6721KS83 Created.
-The submasks are then calculated, and output submask files are created for all subsequent frames that were listed in the frame symbol list for this county.

** Here ends the county processing loop **

This loop repeats until a <cr> only and a verifying response to the query "Done processing and wish to exit" are entered.

NOTES/LIMITATIONS

Only single part submasks are output.
File name generation of calibration, mask and submask files is not yet implemented as required. At present, file names generated do not correspond to the necessary format. Therefore, the resultant generated names may not appear as expected.

PACK	PACKing

INTRODUCTION

The PACK program is used to create packed files, descriptively packed files, and table files. It is also used to unpack files, display table files, and create lists of segments.
A packed file contains only pixels thet meet criteria specified by a SELECT OPTIONS statement and for segments specified by a SELECT REGION statement.
A descriptively packed file is similar to a packed file, but it contains additional information for each pixel. This additional information consists of the segment number, tract, field number, cover, and Landsat row and column. The descriptively packed file is intended for external use.
A table file is a tabulation by cover and category of the number of pixels packed, a tabulation of a packed file. A table file created from a raw data packed file is assumed to have one category. A table file may be by segment with a separate tabulation for each segment, or for all segments with a single tabulation containing the totals for all segments.
An unpacked file has the pixels packed restored to their original spatial locations (Landsat row and column) with special filler pixels in the areas from which no pixels were packed.

INPUT

1) Segment catalog file (type 14)
   name:{ssyy}.CATLG
2) Ground truth files (type 52)
   name:{segment number}.GTRUTH._{ssyy}
3) One pixel border multi-window file (type 51)
   name:{anything}.MWN
4) Segment mask file (type F17)
   name:{segment number}.MASK_{first five of scene-id}${second five of scene- id}_{ssyy}

OUTPUT

1) Packed file including border pixels (type 57)
   name: NB.PACK
2) Packed file excluding border pixels (type 57)
   name:$NB.PACK.
3) Table file including border pixels (type 56)
   name:NB.TABLE
4) Table file excluding border pixels (type 56)
   name:$NB.TABLE
5) Individual crop packed file (type 57)
   name: COVER.PACK

MAIN LEVEL COMMANDS

PACK A FILE
Creates a packed file using SELECT OPTIONSand SELECT REGION.
This command is described more fully in the section on packing.

DESCRIPTIVELY PACK A FILE
Similar to PACK A FILE except that a descriptively packed file is created.

AUTOMATIC PACKING
A standardized methodof creating several packed files for various covers using a user-input SELECT REGION statement and program-generated SELECT OPTIONS statements. This command is described more fully in the section on automatic packing.

TABULATE
Generates a table file from an input packed file. The table file may be by segment or for all segments.

PRINT A TABLE FROM A FILE
Prints a table file onto the user terminal or sends it to a file for later printing. If the table file was created by segment, it is possible to print the tables for only certain segments.

UNPACK
Generates an unpacked file from an input packed file.

LIST OF SEGMENTS FILE CREATION
Creates a list of segments in the specified outputASCII file.
This list of segments is suitable for use in a SELECT REGION statement or in most places where a list of segments is requested.

SELECT REGION and SELECT OPTIONS
The SELECT REGION statement is used to select segments by either explicitly specifying those segments or using attributes of the segments to find them in a segment catalog file. The SELECT OPTIONS statements is used to select fields, and hence segments in those fields, based on attributes of the fields in the ground truth files.
Both SELECT statements are Boolean expressions using the operators AND, OR, and NOT. Some operands have parameters and others do not.If an operand has parameters, it may have a single parameter separated from the operand by one or more spaces or a group of one or more parameters enclosed in paretheses and separated by commas. The effect of more than one parameter is OR, that is to select segments or fields satisfying at least one of the parameters. All operands and non-numeric parameters may be abbreviated. Non-numeric parameters may contain underscore ("_")anywhere except the first character to improve readability, but no embedded spaces are allowed.
A SELECT statement may extend over more than one line and is terminated by a "#". No identifiers or numbers may be split at line boundaries.A line may be a maximum of 128 characters long. Parentheses may be used to any level for grouping.
When entering SELECT statements, "?" may be used at any place to get help. In particular, if a parameter is expected, one may obtain, where pertinent, a list of all allowed parameters.

SELECT REGION
SELECT REGION has the operands SEGMENT, COUNTY, FRAME, STRATA, ANALYSIS_DISTRICT, and ALL. All except ALL take parameters. All except SEGMENT require the segment catalog file. ALL indicates all segments in the segment catalog file. SEGMENT has segment numbers as parameters. The other operands have parameters of the appropriate type. Some examples of SELECT REGION statements are:
SEG(1023,2048,5192)#
(COUNTY CLAY OR STRATA(11,12)) AND FRAME(50055-16274)#

SELECT OPTIONS

SELECT OPTIONS has the operands COVER, NOTE, IRRIGATION, INTENDED_USE, FIELD, and BACKGROUND. All except ALL and BACKGROUNDtake parameters.

ALL indicates the entire rectangular masked area containing thesegment part.

BACKGROUND indicates that part of the masked area not assigned to any field.

FIELD takes fully qualified field names as parameters. A fully qualified field name contains the segment, tract, and field number. Other operands are optionally followed by a "/" and the use number and then by appropriate parameters. The use number selects the use from a ground truth file. If the use number is omitted, the global use is selected, as will be described. The parameters may only be those appearing in the CROPS file for that operand.

SELECTOPTIONS has an additional operator, "-". When applied to an operand, "-" indicates that boundary pixels are to be excluded. Boundary pixels are pixels on the boundary between two fields. By default, boundary pixels are included.

Some examples of SELECT OPTIONS statements are:

COVER/1 CORN#

-(COVER/2(CORN,SOYBEANS) AND NOT NOTE BAD_FIELD)#

-(COVER/1 WINTER_WHEAT AND COVER/2 OATS)#

Some PARAMETERS, GLOBAL USE and OPERATING MODE

Global Use

Ground truth files may have up to four uses. Each use indicates the statusof the segment at a particular time. However, it is often necessary to get a single value representing some attribute of a field, such as cover. To get this value, a use must be selected. In the SELECT OPTIONS statement, the operands COVER, NOTE, IRRIGATION, and INTENDED_USE require a use number. If the use number is not supplied following the operand, the global use is selected. During tabulation, a use number is necessary to obtain the cover associated with a field and thus with a particular pixel. The global use is always selected during tabulation. The user is always required to input the global use.

Operating Mode

The PACK program, particularly when doing automatic packing, may run for quite some time but require only minimal user interaction. It is, therefore, a likely candidate for batch operation. The user is asked for the operating mode and may reply BATCH or INTERACTIVE. The difference in the modes is in error handling. In BATCH mode, the program will be exited when there is no obvious recovery. Currently, the only obvious recovery is to skip a segment when some files pertinent to that segment are missing. In INTERACTIVE mode, the user is queried for an action to be taken when an error occurs.

PACKING

General Packing

The creation of a packed file requires that the mask, ground truth, and window files be present for all segments to be processed.

SELECT REGION is used to select the segments.

SELECT OPTIONS is used to select the fields and thus the pixels to be packed. The packed file contains only the pixels thus selected. The header of the packed file contains the list of segments and an internal representation of the SELECT OPTIONS statement. The list of segments contains only those segments for which some pixels were packed and thus is a subset of the list of segments generated by the SELECT REGION statement. The header of the packed file also contains the names of the frames used in creating the packed file.

The input window files are contained in one or more multi-window files. A multi-window file contains one or more rectangular windows of Landsat data. Each window is large enough to

contain a segment part. More than one multi-window file may be used in a packing session. Before entering the names of the multi-window files, the user is asked for a list of frames. If some frame names are entered, each multi-window is checked to see if it belongs to one of those frames and is assigned to that frame. If no frame names are entered, the frame name is obtained from the header of the multi-window file and added to the frame list if necessary. In both cases, if the frame name cannot be found in the header of the multi-window file, the user is asked to assign the file to some frame. When a window is needed for a segment, only those multi-window files belonging to the same frame as the mask file are searched. The search is not for the segment number, but for a window containing the segment part.
When a packed (or descriptively packed) file has been created, the user is asked if another SELECT OPTIONS statement is to be entered. A YES reply assumes the same SELECT REGION (list of segments) and the same multi-window file.

Automatic Packing

Automatic packing is a partial automation of the packing process
suited to the analysis generally done at USDA. A SELECT REGION statement and some parameters are entered by the user. The program then generates SELECT OPTIONS statements to create several packed files. There are two varieties of automatic packing, ALL CROPS and MAJOR CROPS. First, for both kinds of automatic packing, two packed files, and corresponding raw data table files for all segments, are created. The first is with the SELECT OPTIONS statement:
NOT BACKGROUND#

and the second with the SELECT OPTIONS statement:
-(NOT BACKGROUND AND NOT NOTE BAD_FIELD)#

When doing ALL CROPS, the user supplies a minimum number of pixels for creating a packed file. Then, using the second table file, a packed file is created for each cover with at least that many pixels. When doing MAJOR CROPS, the user selects the covers to be used and a packed file is created for each cover selected. The only check is that there are some pixels for that cover in the second table file.
In both cases, a packed file is created for all remaining covers. Next, the user is asked for special SELECT OPTIONS statements to handle unusual or special cases. The user must assign each of these to one of the covers for which an individual packed file was created. Finally, an ASCII file is created listing the names of all packed files created, except the first two, the cover assignment (OTHER for the packed file of all covers for which individual packed filew were not created), and the percentage for that cover as taken from the second table file (for OTHER this is the percentage for the sum of all the covers used).
All files created during automatic packing have specific names generated by the program. These names are, of course, dependent on the operating system in use.

PARTS	network and mask file PARTS manipulation program

PURPOSE

To manipulate multi-part segment network files, strata network files or segment mask files. May be used to combine several one part files into a multi-part file or to break out a part of a multi-part file into a single part file.

INPUT

A file type 17 (mask files) or 47 (network files)
User-entered commands

OUTPUT

A file type 17 (mask files) or 47 (network files)

HOW IT WORKS

There are four user-entered commands. Briefly, they are:

INPUT FILE OPEN
close any previously opened input files and open a file for input. User is prompted for the file name.If a previous file has been opened for input, program verifies that the file type (mask or network) and segment number are identical with those of the previous input file.

OUTPUT FILE OPEN
close any previously opened output files and open a file for output. User is prompted for the file name.

COPY A PART
copy one part of the input file onto the end of the output file. User is requested to supply the input part number unless the input file has only one part, in which case that part is copied.If a previous part has been copied for output, program verifies that the segment number is identical with that of the previous part output.

QUIT
close all files and end execution.

PCDIGIT	DIGITize boundaries or strata with a digitizer on the PC

PURPOSE

PCDIGIT allows the digitization of strata and segments and the updating of previous digitizations; it produces a file in .VBO format containing the geographic coordinates of the objects, the extension in m^2 and the code of digitized strata or field. Moreover it's possible to make a file containing ground control point in geographic coordinates to use later to generate global calibration file.

ACCESS

On PC: MS-DOS prompt>PCDIGIT.

INPUT

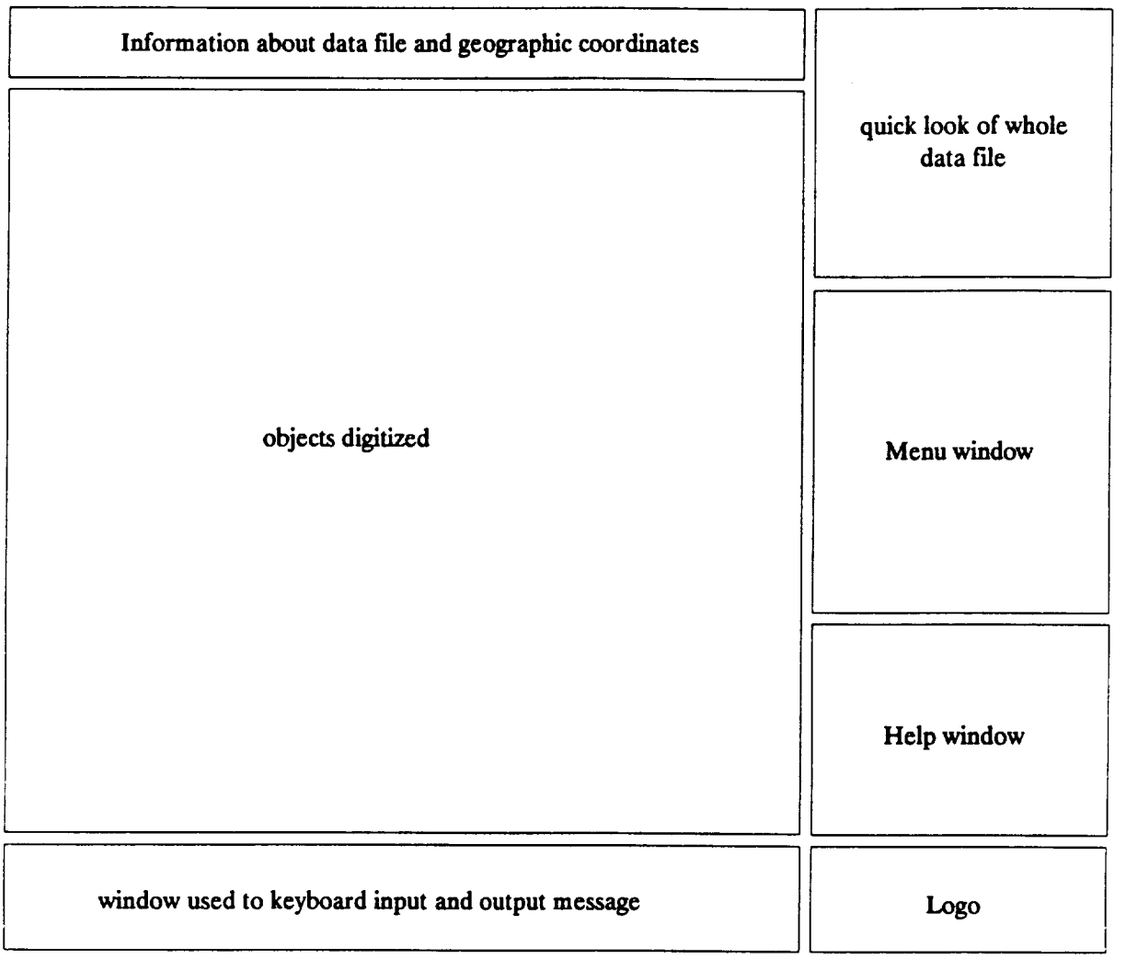
A set of files containing previous recorded data (.BDB, .PAV, .ATT, .REL, .GCP)
A file containing coordinates to calibrate the map on tablet (.COR)
All these files are not MARS PED file type but in owner PCDIGIT format.

OUTPUT

A file .VBO for each segment or strata digitized.
A file .VBO for each set of Ground Control Points.
Both are ASCII file (not MARS PED file type).

HOW IT WORKS

PCDIGIT is an interactive program.
To run this program type PCDIGIT and then use the puck to select the various commands.
The puck keys used are 0,1,2 and 3.
Usually the key 0 is used to escape or to return on previous menu.
In a menu, the keys 1 and 2 are used to move the cursor to point an option (1 goes down, 2 goes up) and key 3 is used to select the pointed option.
The option selected will be showed in bold.
The windows that appears on the screen are described in following figure:



data window: this window is used to display the portion of work area in which the user is working.
information window: to display data file name and the coordinates of portion of work area in which the user is working
menu window: to display the options of menu.
help window: to display the functions associated to puck.
quick-look window : to display the initial existing data contained in the file opened .
input/output window : to display messages and to input data from keyboard
logo window : to display the logos

COMMAND SUMMARY

The items for the main menu are:

CREATE DATA FILE: used to define the coordinates of the area that will be digitized or to modify an existing area.

OPEN DATA FILE: open an existing file to insert or update the data.

EXPORT: to generate the .VBO file containing segment or strata, or the ground control points file.

EXIT: to exit the program.

Each item have subcommands or submenus.

CREATE DATA FILE:
Selecting this item a full screen mask will appear. The first input required is the Data File Name.
If the name inserted is the name of an existing data file the program ask if user wants change the limits of work area or the name in order to create a new data file.
If the name has not already been used or if the user has choosen to update an existing file the program will ask the COORDINATES of the lower left and upper right points of the work area.
To Confirm the actual selections press the key "y" ,to reinsert the data press the key "n"; to leave this section without updating press "ESC".
Reducing the area of an existing data file containing some objects, user must pay attention that the objects out of new area will be not processed. To process these objects a redefinition of work area is necessary.
The data file are recorded in DIGIT/DATA directory.

OPEN DATA FILE: data file must be created before run this section (if doesn't exist any data file, a message will appear on the bottom of the screen).
Selecting this item in the window, in place of the previous menu will appear the data file names. Using the puck keys user may select a data file.

After this selection in the bottom line is requested the INPUT SCALE: that is the scale of the map to digitize. This input parameter defines the precision of digitization. If user inserts the real scale of map that must be digitized, there will be a great precision (this is the case when a very good map has to be digitized); this means that for example when user wants snap an existing point, he must be very accurate to point with the puck the selected point. Inserting a scale greater then the real scale of the map, the precision is reduced (this is the case of not very good map in which the row are drowed with a dense stroke); in this case a good precision is not necessary and, for example, the snap is made to a greater distance then the previous case.

After this step the menu for the CALIBRATION of the map is displayed.
To calibrate a map on the digitizer it is necessary to define at least four points on the map and to digitize the corrispondent on the digitizer (or tablet). If the error between the two systems is too high the program cannot continue.

The CALIBRATION menu shows the following items (that can be selected using the puck keys):

COORDINATES FILE
POINTS FROM KEYBOARD
POINTS FROM DIGITIZER
UPDATE POINT
DELETE POINT
DISPLAY CALIBRATION
CHECK COORDINATES
DELETE ALL POINTS
END CALIBRATION
RETURN TO MAIN MENU

COORDINATES FILE: allows to select an existing file containing the coordinates of calibration points previously defined with POINTS FROM KEYBOARD option. After this step it is necessary to select the item POINTS FROM DIGITIZER for a correct calibration. If

it is not done the program will not allow the digitization. The coordinates files are recorded in DIGIT\COORD directory.

POINTS FROM KEYBOARD: in this section you have to insert at least four points: the Map Calibration Points. If some points have been previously defined you can add new points (to delete points see the DELETE POINTS item).
To insert points user must enter the x and y coordinates separated with a comma and followed by a <Return>. To leave this option press <ESC>. Before leaving this section the user is asked if he wants save this set of point in a file. There are three possible ways to answer:
Type a name followed by <Return> : the points are saved in a file called "name.COR".
Press only <Return>, the points are saved in a file with the same name of the data file and extension ".COR".
Press <ESC> the points are not saved.
After this section it is necessary to select the item POINTS FROM DIGITIZER for a correct calibration. If it is not done the program will not allow the digitization.

POINTS FROM DIGITIZER: user must digititze the Map Calibration Points on the tablet in the same order used in the POINTS FROM KEYBOARD. This order must be respected. To digitize use the puck (it does not matter which keys are used). After this operation, a little square for every point appears on the screen at a distance from the point in according to the calculated error.

UPDATE POINT: allows to update the coordinates of a map calibration point using the keyboard. It is requested the point number and then the new x,y coordinates (the new coordinate must be separated with a comma and confirmed with a <Return>).
Before leaving this section the user is asked if he wants save the new set of point in a file. There are three possible ways to answer:
Type a name followed by <Return> : the points are saved in a file called "name.COR".
Press only <Return>, the points are saved in a file with the same name of the data file and extension ".COR".
Press <ESC> the points are not saved.
After this section it is necessary to select the item POINTS FROM DIGITIZER for a correct calibration. If it is not done the program will not allow the digitization.

DELETE POINT: allows to delete a map calibration point. It is requested the point number and the confirm of the selection.
Before leaving this section the user is asked if he want save the new set of point in a file. There are three possible ways to answer:
Type a name followed by <Return> : the points are saved in a file called "name.COR".
Press only <Return>, the points are saved in a file with the same name of the data file and extension ".COR".
Press <ESC> the points are not saved.
After this section it is necessary to select the item POINTS FROM DIGITIZER for a correct calibration. If it is not done the program will not allow the digitization.

DISPLAY CALIBRATION: for each point the following items are displayed on the screen:

- x,y coordinates in digitizer unit
- x,y coordinates in UTM unit
- the deviations between the original and transformed coordinates (in UTM unit).

To leave press <ESC>.

CHECK COORDINATES: Displays the coordinates values of the point digitized. To check the coordinates of a generic point of the map, it's necessary to point with the puck the point and press one of puck key: the system will show the coordinate values into the input/output window on the bottom of the screen. To leave this option press <ESC>.

DELETE ALL POINTS: in this case it is necessary to remake the calibration (redefine the set of points with POINT FROM KEYBOARD or COORDINATE FILES and then digitize them with POINT FROM DIGITIZER).

END CALIBRATION: selecting this the program exit only if the calibration has been correctly done.

RETURN TO MAIN MENU:

If the CALIBRATION phase is correctly finished in the input/output window the system requires two points from digitizer to define the area to be digitized. To define these two points, it's necessary to point with the puck the points and to confirm them pressing a puck key (any key can be used).

After this step the DIGITIZATION menu will appear. It has the following items:

OBJECT TYPE
DIGITIZE
OBJECT OPERATION
EXIT

The second and third item assign different actions to puck keys.

OBJECT TYPE: to select the object type to digitize (ARCS or Ground Control Points). At the selection of this option the following menu will be showed:

Return
Arc
Ground Control Point

After the selection of object type to digitize, the system will return to DIGITIZATION menu.

DIGITIZE: selecting this option, the puck keys are set in the following way:

0 -> EXIT	Return to the digitization menu
1 -> DIGITIZE POINT	Digitize a point of the selected object type. *ARC*: If the first point is on an existing arc, a BEEP will be produced to signal that a snap has been made. If a successive point is digitized on an arc or on the first point of arc in working, a BEEP will be produced (second snap) and the arc will be closed and recorded. To force the end of an arc press puck key <3> (CLOSE OBJECT). It's no possible snap a point of the object in working different to the first. *GCP*: when press key <1> the GCP number will be asked; to insert the GCP number, type it from keyboard. The GCP number must be unique.
2 -> DELETE POINT	Delete the last point digitized.
3 -> CLOSE OBJECT	Force the end of the ARC

When user digitizes arcs must pay attention not to cross arcs, otherwise the algorithm to build polygons will be finished in a wrong way. It's always necessary that in every cross point there must be the starting or finishing point of one or more arcs. At the begin and at the end of an arc the system adds a node that is rappresented graphically with a little circle with a cross on the center. Nodes are generated automatically when you force the end of an arc or make a snap.

OBJECT OPERATION:

0 -> EXIT	Return to the digitization menu
1 -> DELETE OBJECT	Delete procedure. Selecting this option, the message "Select object" will appear in input/output window. To select an object, press key puck <1> or <2> on it. If the object has been found, it will be redraw in red color and the system will ask the confirm to delete (press puck key <0> to delete or puck key <1> to skip), otherwise the message "No object found" will appear on the input/output window. To exit from delete procedure, press puck key <0> or <3> when the message "Select object" is showed.
2 -> DELETE LAST POINT	Delete the points of the arc in working.
3 -> ZOOM	Start the zoom procedure. This procedure allows to make a zoom. The point in which user press the puck key <3> to call zoom represents the first point of window to zoom. The second point can be defined pressing again the puck key <3> (this represents the second point of the zoomed window). Instead to press againg puck key <3> user can press the other puck keys, in this case the following operation are performed: <0> - escape: return to DIGITIZATION menu without permorming anything. <1> - All space: perform zoom showing the whole work area <2> - Redraw: redraw the same window displayed

EXIT: close the data file and return to MAIN MENU.

EXPORT:this option performs the export of data in .VBO format.
There are two format in output:
the first format for ground control point (an ASCII file containing for every line a GCP in the following form

x,y,n

where x and y are the GCP coordinates and n the GCP number)
the second format (VBO format) for strata or field.

The files are recorded in DIGIT\EXPVBO directory.
At the begin, the system requires the data file to export; the data file set is showed in window menu. Use puck key to select data file.
After selection the system requires the output file name (max.8 characters without extension). Typing a name followed by a <Return> the file will be called "name.VBO".

Pressing only <Return>, the file will be called with the same name of the data file and extension ".VBO".
Pressing <ESC> the system return to MAIN MENU.
If a file with the same name already exists, the system requires to confirm to overwrite the file.
Then the type of object to export is required. The type can be one of following:

STRATA
FIELDS
GROUND CONTROL POINTS

In the case of STRATA or FIELDS and if the data file contains arcs, a procedure to build polygons defined by arcs starts.
When the procedure has built all polygons (this may be time consuming depending to the number of polygons to be found) they will be showed one at a time in red color and it'll be required to user to insert in FIELD case the field number and the cropcode, in STRATA case only the strata code. It's possible discart a polygone, answering with a "." to the field number or to strata code question; it's possible also to abort the operation answering with "/" to the field number or strata code question. During the polygons building a message with built polygone number is showed in input/output window.

EXIT: selecting this option, the program return control to DOS system.

HARDWARE REQUIREMENT

The hardware required is:

- a PC IBM compatible with numeric coprocessor 80387 and hard disk
- a standard VGA graphic card
- a tablet or a digitizer CALCOMP with a 4 or 16 keys puck

PCSHIFT	Segment SHIFTing on PC

INTRODUCTION

PCSHIFT is used to display raw data using grey-levels, display categorized data using colors, and do segment shifting using the raw data display or the categorized data display. The function to be used must be selected after starting PCSHIFT; to select a different function, PCSHIFT must be exited and then started again.

PCSHIFT will run on the IBM PC or clones. As like all remote sensing programs on the PC, you should have at least 640K of memory installed. The VGA graphic card and monitor is required for PCSHIFT; it will not work with other graphic systems. It is better, but not absolutely necessary, to have an hard disk and a math coprocessor. Without these, PCSHIFT may run rather slowly. Finally, PCSHIFT may run uncomfortably slowly on a PC/XT.

PCSHIFT assumes the remote sensed imagery input is a standard MARS-PED multi-window file. One window at a time is displayed, as selected. PCSHIFT determines the screen pixel size for each window. This value is made as large as possible to still fit on the screen, but not exceeding 5. Also, the number of screen pixels is kept the same in the horizontal and vertical directions. The value is not allowed to get below 1. If the window is too large to fit on the screen, only the part that will fit starting with the upper left is displayed. Standard VGA graphic card allows a resolution of 640 by 480 screen pixels. Note that the number of screen pixels per image may vary among the windows in a multiwindow file.

GREY-LEVELS DISPLAY

For the grey-levels display, the user selects a channel (the value must be between 1 and the number of the channels in the data). The user then selects the window to be shown (the value must be between 1 and the number of windows). Each window is displayed in the channel selected; the channel may not be changed without exiting the program. Ten grey levels are shown, although all may not necessarily be clearly distinguishable. PCSHIFT is exited by entering a carriage return when asked for the window to display.

CATEGORIZED DISPLAY

Before a categorized file is displayed, the user is shown a menu of colors and additional choices, SHOW and QUIT. The '#' symbol is displayed and may be moved using the up-arrow and down-arrow keys on the numeric keypad. The movement is circular, that is if moved off the bottom, it starts at the top and vice-versa. When '#' is positioned at a color, a list of categories may be entered. This list is a list of integers separated by commas with each integer being between 1 and the number of categories in the input file. In addition, a range of categories may be entered separated by a "-".

Thus, "1-10,17" are the categories 1 through 10 and also 17. A list is terminated by carriage return. To delete a list of categories at some color, simply position the '#' at that color and enter a carriage return. All colors are initially assumed to have no categories assigned. If a list is to be changed, position the '#' and enter the new list of categories. The old categories will be automatically deleted when a new list is entered. It is not possible to edit a list once entered; a complete new list must be entered for any color to be changed.

Once the first window has been displayed, the same assignments remain for other windows. Before each window is displayed, the color menu is displayed allowing changes to be made if required. The SHOW position is used to sort the category lists and re-display them. Also, duplicate and unassigned categories are shown. Carriage return at the SHOW position activates the show functions. Unassigned categories are displayed as black (the same for background of the display). Duplicate categories are given the last color, in the order from top to bottom, for which they are assigned. The QUIT position is used to go to the display be entering a carriage return. The QUIT function will not be accepted if all categories are assigned the same color.

SEGMENT SHIFTING

The type of display used for shifting is determined by the type of the window file supplied. If the file is categorized, the categorized display will be used otherwise the raw data grey-levels display will be used. However, before segment shifting can begin, the user is asked to enter a scene identifier, which is used to select a global calibration file. If the global calibration file cannot be found using the standard name, the user is asked to enter the name of the global calibration file.

Then, if shifting is taking place on categorized data, the color menu is shown and the user is asked to enter the category assignments just as for the categorized display. However, the segment color may also be entered by using "S" instead of a category assignment for some color. It is not possible to have the same color assigned to some categories and to the segment. If a segment color is not selected, black is used. While black will show up fairly well against most of the colors, it has the disadvantage of causing the segment to disappear if moved out of the window since the background for a categorized display is also black. Users will have to determine by trial and error which color they find most useful for the segment.

Next, the useris asked for the source of the list of segments. There are two possible choices, to enter a list of segments or to get the list of segments from the multi-window file. The list of segments is entered in the "standard" MARS-PED manner, as "SEG(" followed by the segments separated by commas followed by ")#". This list of segment input may be on several lines. The option from the multiwindow file will only be allowed if there appear to be some segment numbers in the multiwindow file. In either case, the segments are sorted. The normal sequence of processing is to do the segments in numeric order and the parts within each segment in order. As will be described, this order may be changed, but it is never possible to skip a part in a segment, although it is possible to skip to another segment before all parts of the current segment have been completed.

Finally, the user is asked to enter the name of the output file of segment shifts. If the file already exists, the user is asked if the new shifts will be appended to that file. If so, the file named is read into the table of shifts. If not, the table of shifts is set to empty and nothing is done immediately to the file. However, the file is completely re-written with all shifts in the table after each successful shift so that after the first shift, the old file disappears and is replaced by the file containing the current table of shifts.

Before each segment part is shifted, the user is allowed to make changes in what will be shifted next. To take the default, a carriage return only is entered. The default is displayed at the top of the screen. When all segments in the list have been processed, the default will be to exit PCSHIFT. The segment and part may always be changed. If shifting is taking place using grey-levels of raw data, the channel may be changed. If shifting is taking place using categorized data, the color selection may be changed.

When changing the segment, there are two possibilities, either some segment may be processed and then a return made to the normal sequencing or else a jump may be made forwards or backwards in the sequence.

The SEGMENT SELECTION command is used to change the segment to be processed.The NEW SEGMENT SEQUENCE command is used to change the sequence. Thus, if segment 5006 is next to be processed and the SEGMENT SELECTION command is used to choose segment 1040, after the one part of segment 1040 has been shifted, segment 5006 will be next to be shifted. However,if the NEW SEGMENT SEQUENCE command is used, after segment 1040 is shifted, the next segment after segment 1040, say 1152, will be shifted. Note that if the new sequence is to go to a higher segment than the next default segment, some segments in the list may be skipped if the SEGMENT SELECTION or NEW SEGMENT SEQUENCE commands are not used to later return to those segments. When a new segment is called for, the part is always set to 1. This may be changed with the PART SELECTION command.

The PART SELECTION command is used to change which part will be shifted. The part selected may only be some part which has already been shifted or the default part to be shifted next. It is not possible to shift part 1 of a segment and then skip and do part 3. The reason for this is that the shift file as written does not contain part numbers and thus skipping of parts would cause the file to get mixed up. After the part selected has been shifted, the sequencing reverts back to normal sequence.

The CHANNEL SELECTION command is used to select a new channel to be displayed and is used only if shifting is taking place using grey scales of raw data. This channel selection, once made, is kept until changed by another CHANNEL SELECTION command. The use of the CHANNEL SELECTION command does not affect the segment sequencing.

The COLOR SELECTION command is used to change color assignments. Use of this command calls up the color menu allowing changes as described before. Once color changes are made, they stay in effect until changed again.

When using the SEGMENT SELECTION or NEW SEGMENT SEQUENCE command, it is possible to enter a more complicated descriptor which describes the segment, part, and channel. The form of this descriptor is /<PART>;<CHANNEL>. Thus, 1040/2;3 specifies segment 1040, part 2, channel 3. The selection of segment, part, and channel follow the rules described. In addition, however, if used with the NEW SEGMENT SEQUENCE command, the part will also set the sequence as per the part.

Various of the fields may be left out in which case the defaults will be used. Note that if the segment is used and the part is left out, the default part is 1 and not the next part shown at the top of the screen.

When one of the above commands has been entered, the screen iscleared andthe display at the top of the screen shows the updated next segment, part, and, for grey-scale shifting only, channel.

Three additional commands are available. DISPLAY SHIFTS shows all shifts thus far performed, including any read in from a file of shifts. PROCEED WITH SHIFTING is the equivalent of entering a carriage return only. QUIT SHIFTING AND EXIT PCSHIFT exits PCSHIFT immediately.

Then, for each segment, PCSHIFT searches the multi-window file for a window containing that segment. That window is displayed for the channel selected and the segment is shown, in an amber color for the grey-scale display or the selected color for the categorized display, at its initial calculated position based on the segment calibration and the global calibration file. If the segment has been shifted, whether in this session or the shift has been read from the file of shifts, the shifts will be accounted for before the segment is shown.

The segment may be moved using the numeric keypad:

The arrows move in the directionof the arrow.
The 7(HOME) key moves diagonally up and left.
The 9 (PAGE UP) keymoves diagonally upand right.
The 1 (END)key moves diagonallydown and left.
The 3 (PAGE DOWN) keymoves diagonally down and right.

For large shifts,a numeric modifier is entered before the key for the shift in the desired direction. This number will be displayed to the right of the message showing the window and segment numbers. This number is the number of satellite pixels (not screen pixels) to be shifted in the direction specified by the key entered. This number is reset after any shift. If a numeric motifier is not entered, each key entered is a shift of one screen pixel in the direction specified. Holding down one of the direction keys will cause faster movement, but the numeric modifier applies only to the first input in that direction. Each additional movement is one screen pixel only. The translation of screen pixels to satellite pixels varies depending on the size of the satellite image displayed. There may be flicker during movement. This will be more noticeable for complicated segments or slower PCs.

The 0 (INS) key displays the current shifts, with the display being erased the next time a key is pressed. If a numeric modifier has been entered but no shift direction key pressed, use of the 0 INS) key will reset the numeric modifier. A carriage return is used to accept and enter the curretn shift values. The DEL key is used if the shift is not to be saved. If the DEL key is used, the normal sequencing is to do again the same segment and part allowing a change of channel if needed.

PRINCIPL and DISPRNCP	principal components modules

INTRODUCTION

PRINCIPL is a program that calculates principal components, eigan values, and the rotation matrix from a statistics file and outputs a new type 66 Principal Components file.
DISPRNCP is a program that reads and displays a type 66 Principal Components file.

To use PRINCIPL place desired data into a Statistics file (type 54) by using WINSTAT or one of the clustering programs. You may wish to use STATED to trim undesired data from the statistics file. The output file (type 66) has no standard name so the user must supply the output file name. The file will go to the current directory unless otherwise specified. The program searches for the input statistics file directories as specified by the Directories command. The program will ask for a Range of categories to be calculated. Syntax for the response can be
[N<CR>, N-N<CR>, N-M<CR>, <CR>]

Where M>N>0 and M<= number of categories in the Statistics file. The range question is repeated until a <CR> response is received. The program then writes the output file and deletes the temporary file. If the program does not terminate normally this temporary file may not be deleted.

The Temporary file is named
<user supplied name before extension>.TMP.

DISPRNCP displays a type 66 File one category at a time. Inputs: Principal component file, category. Ranges are not accepted and one category at a time is displayed. A <CR> will terminate the program. The Rows of the rotation matrix are weights that will convert principal components to Channel values. The columns are weights that will convert channel values to principal component values. Channel order is preserved and this means that the principal components may appear in any order. The first principal component corresponds to the largest eigan value, The second corresponds to the next largest and so on.

PRINCPWN	WiNdow file with PRINCipal component values

PURPOSE

The MARS-PED module "PRINCPWN" creates a "window" file with principal component values replacing raw data values. SUBWINDOW module then can be used to reduce the dimensionality of the data.

INPUT

Principal component file type 66.
Window file type 51.

OUTPUT

Window file type 51.

HOW IT WORK

If the input window file is a "Multi-Window" file the output is also a multi-window file and can be used like a raw data window file. If the input window file is a "packed" file the output is also a packed file and can be operated on the same as any packed file.
Shifting, moving the output data in the range 0-255, and scaling, limiting the range to a length of 255 is necessary to retain window file attributes. These procedures, shifting and scaling, may alter the orthogonality of the principal components, however, the off diagonal elements remain small relative to the raw data values.
Principal components may be used to reduce the dimensionality of the data, examine the information content of the scene (segment), evaluate individual channels of raw data, simulate raw data from given statistics files and perform a multi-variate edit of raw data files.

PRXSEG	SEGment Plotter program (not used in MARS-PED)

PURPOSE

Generate plot files in Landsat or Digitized image from input segment network files. These plot files contain the necessary code to be able to be printed, in plot mode, on a Printronix line printer.

INPUT

1) Segment Network File(s)
2) Global Calibration File
3) User entered data in response to prompts

OUTPUT

1) A binary file - no MARS-PED file type

HOW THE PROGRAM WORKS

After all prompts have been answered by the user, PRXSEG will begin to process one segment network file after another, until all have beenprocessed. While processing each segment network file the coordinates and other necessary information is output to the user specified output file in binary form. Upon termination of the program, the output file can then be routed to the proper destination or printing (plotting).

COMMAND SUMMARY

Below are the descriptions of the prompts that the user will need to respond to while getting ready to process segments.Following each description is an (O)ptional or (R)equired letter, showing the user what is needed to process properly.

1. If the user wishes to use a directory, other thanthe current one, then input the needed directory or directories. This prompt is library routine GETDIR. (O)

2. If file names are to be program generated then the user enter the state abbreviation and last two digits of the year or which the segment network files pertain. Ex:MO85, IA86. This input is used by library routine ENAMEGEN. (O)

3. The user will now enter a list of segments to be processed in the manor specified by the prompt. If the user wishes to enter segment network files individually then only a "#" is needed. The library routine SEGLIST is used for inputting and parsing the list.(R)

4.Now the user will enter the global calibration file which is needed to process landsat images into the binary plot files. The user will need to enter the fully qualified name of the file if prompt 1 was not answered, otherwise, enter file name.In either case, the library routine GLBCAL is used, which also performs an initalization step needed for later use within the program. (O,R)

5.The user specifies the name of the output file which will contain the binary plot image data. (R)

6.Select the desired type of plot. (R)

7.This prompt will only be shown to the user if the"#" was the reply to prompt 3.The user will be prompted for an individual segment network file which will be processed immediately. The user will then be prompted, upon completion, if another is to be processed. If yes, then the input prompt is again displayed, otherwise, the program terminates. (R = > If shown).

REGCP	REGistrationControl Point

PURPOSE

Generates and stores either image-to-map or image-to-image calibration information useful in the subsequent location of data on selected remotely sensed images.

INPUT

1) Altek or Talos Digitizing tablet (including location and type)
2) Tablet and terminal hooked up so that the tablet and user can both talk to the CPU via one or two communication lines
3) The port(s) on which the tablet and terminal are hooked to:
   a) On MIDAS, both may be hooked up to a single (as on BBN) or two separate user-selected ports
   b) On BBN, only one line (the terminal port) is used to connect to both the terminal and the tablet with BBN.
4) Maps and images containing marked and numbered control points
5) Control point file name (if desired)
6) User-entered input
   a) Choose between image-to-map(MAPIMAGE) and image-to-image (IMAGEIMAGE) registration modes
   b) Other prompted inputs

OUTPUT

Possible outputs:
1) Control point file (text file- no file type) MAPIMAGE mode:
2) Image calibration file (text file- no file type) IMAGEIMAGE mode:
3) Overlay parameter file (text file - no file type)
4) Primary List of Blocks File (file type 14)
5) Secondary List of Blocks File (file type 14)

HOW IT WORKS

After the selection of the registration mode is made, the control point data (latitude, longitude, image row, image column, etc.) for each control point, obtained via digitization or from an input file, is stored in a buffer that is accessed by the least squared routines to create another buffer of transformation information. These two buffers can be edited by invoking editing functions or additional digitizing functions. The contents of the buffers can be written to disk as control point files, or, depending on the registration mode selected, image calibration files, overlay parameter files and list of blocks files.

COMMAND SUMMARY

The commands available in REGCP for the two registration modes differ slightly. In the interest of easy reference, complete descriptions of the commands will be given for both registration codes despite much repetition.

The commands available in the **MAPIMAGE** registration mode are(preceded with the "MI)" prompt):

Add CP pairs: digitize or enter both map and image control points and add to the list of control points in the buffer.Will first prompt for source of map and image coordinates,if not selected as yet.

Change registration mode: switches to image-to-image (IMAGEIMAGE) registration mode. All data in the control point and transformation buffers are initialized if given permission by the user.

Degree of LSQ polynomial: specify the order of the polynomial (must be between 0 and 4).

Edit CPs: view and adjust characteristics of the control points (and thus the least squares transformation) using various subcommands which are described near the end of the

The Least Squares analysis is performed on the control points prior to entering the submenu.

Image source: specify the source of image coordinates- either via digitization or by entering at the terminal.

Initialize: purge the control point and transformation buffers of all data. Before clearing out all data, the user is prompted to verify the request.

LSQ analysis of CP pairs: perform a least squares analysis between the map and image control point coordinates and place the transformation values in the data buffers. A degree of 1 is assumed unless reset by the "Degree" command.

Map Source: specify the source of map coordinates: either via digitization or by entering at the terminal.

Output image calibration file: specify the name of an output precision calibration file to which the data in the image-to-map transformation data buffer is to be written.The file name of an input calibration file name must be specified if a degree of more than 1 is specified.

Quit: confirm that the program is to be exited even though there is data in the data buffers.

Read old CP pairs file: specify the name of a text file that contains the latitude, longitude, row, column and other identifying information (as specified in a MARS-PED control point file) and append it to the control point list data buffer. The registration mode will be changed automatically to the mode specified in the control point file.

Redigitize control points: select a range of control points to be re-digitized, obtain new tablet transformations (if requested), redigitize or retype the control points selected and locate in the data buffer in place of the old points.

Transform CP pairs using Overlay Parameter file: Specify the name of a text file that contains the latitude, longitude, row, column and other identifying information (as specified in a MARS-PED control point file). Specify also the name of an input overlay parameter file. All buffers will be initialized before the files are read.The calibration file will be used on the original row, column part of the pairs to create a transformed control point file (latitude, longitude, new row, new column). The transformed pairs are left in the control point list data buffer.

Write new CP pairs file: specify the name of a file to which the data in the control point list data buffer is to be written.

The commands available in the **IMAGEIMAGE** registration mode are (preceded with the "II)" prompt):

Add CP pairs: digitize or enter both primary and secondary image control points and add to the list of control points in the buffer. Will first prompt for source of primary and secondary image coordinates, if not selected as yet.

Change registration mode:switches to image-to-map (MAPIMAGE) registration mode. All data in the control point and transformation buffers are initialized if given permission by the user.

Degree of LSQ polynomial: specify the order of the polynomial (must be between 0 and 4).

Edit CPs: view and adjust characteristics of the control points (and thus the least squares transformation) using various subcommands which are described near the end of the

The Least Squares analysis is performed on the control points prior to entering the submenu.

Generate List of Blocks: specify the number of rows of blocks and the number of blocks per row and an overlay parameter file. A Primary List of Blocks File and a Secondary List of Blocks File (both are type 14) will be created with a name specified by the user.

Initialize: purge the control point and transformation buffers of all data. Before clearing out all data, the user is prompted to verify the request.

LSQ analysis of CP pairs: perform a least squares analysis between the map and image control point coordinates and place the transformation values in the data buffers. A degree of 1 is assumed unless reset by the "Degree" command.

Output overlay parameter file: specify the coordinates of the window to be used and the name of an output overlay parameter file to which generated window coordinates and the data in the image-to-image transformation data buffer is to be written.

Primary Image Source: specify the source of the primary image coordinates- either via digitization or by entering at the terminal.

Quit: confirm that the program is to be exited even though there is data in the data buffers.

Read old CP pairs file: specify the name of a text file that contains the row, column for both the primary and secondary image coordinates and other identifying information (as specified in a MARS-PED control point file) and append it to the control point list data buffer. The registration mode will be changed automatically to the mode specified in the control point file.

Redigitize control points: select a range of control points to be re-digitized, obtain new tablet transformations (if requested), redigitize or retype the control points selected and locate in the data buffer in place of the old points.

Secondary Image Source: specify the source of the secondary image coordinates- either via digitization or by entering at the terminal.

Write new CP pairs file: specify the name of a file to which the data in the control point list data buffer is to be written.

The following commands are in the submenu structure accessed by entering "E" in either the MI or II mode.

Auto Delete Until LSQ Error <= a Value: assigns the control point type, DELETE, to the control point with the highest row or column error exceeding a user-specified maximum,recalculates the least squares transformation, and checks to see if the row or column error of any control point still exceeds the maximum. In short, the deletion and re-calculation is performed until no points have a row or column error higher than the maximum.

Change CP type: change the control point type of a selected set of control points to a specified control point type which may be either CONTROL, CHECK or DELETED.

Containing Windows for CPs: calculates the coordinates of the largest window containing all of the control point pairs for both map and image (mode MI) and primary and secondary image (mode II)

Delete CPs with LSQ Error > a Value: simply assigns the control point type, DELETE, to any control point which exhibits a row or column error higher than a user-specified maximum.

List CP pairs: list to the terminal the latitude, longitude, row and column (mode MI) or primary and secondary row and column (mode II) coordinates and control point type (CONTROL, DELETED, or CHECK) of a selected set of control points and/or control point types.

Retitle CP pairs: the user may type in new titles for the control points prior to writing to a control point file. The names assigned automatically by the program are CP 1, CP 2, CP 3, etc.

Sort CP pairs by maximum LSQ error: list to the terminal a selected range of control points in the order of maximum least squares error in meters.

HOW TO USE IT

The user is immediately placed into MAPIMAGE mode (image-to-map calibration) at the outset of the program. A "Change" command will quickly move the processing to IMAGEIMAGE mode for scene-to-scene registration. The control point pairs may be obtained via digitizing or by entering into the terminal, or, if the pairs exist in a control point file, by reading in the data from disk.

If digitization is chosen for either the primary or secondary data, the user mounts maps (mode MI) and/or images containing the same point on the ground onto the digitizing tablet. After digitizing map (mode MI) and image corners, tablet-to-map (mode MI) and tablet-to-image least squared transformations are generated and verified for accuracy. Control point coordinates are then digitized, transformed and stored in a buffer alo*ng with ancillary information, such as titles and row and column least squared errors. The user must enter enough control points data into the program in order to generate a least squares analysis for the order selected.

Once all control points have been obtained, the control point editing submenu may be entered. The data collected may be examined by listing, or altered by executing the "change" redigitize", "auto delete" or "delete" commands. When the control point information is deemed to be accurate for the data in the buffer, the least squares analysis may be invoked, and the results for each point may be viewed by using the "sort" command.

After the user decides that the least squares analysis is returning satisfactory results for the control points selected, the control point and image calibration file (mode MI) or overlay parameter file (mode II) can be written. If the user is in "II" mode, a primary and a secondary list of block file may be generated in preparation for scene-to-scene registration.

The user is prompted for verification that the data has been written to disk before being allowed to exit the program.

NOTES/LIMITATIONS

Complete checkout of the digitizing interface has not been tested on all system and port combinations. As of this writing (4/8/85) using the console port of MIDAS as a terminal and also as a means of receiving date from the tablet does not work.

SEGED	SEGments Digitization commands (Not used in MARS-PED)

COMMAND SUMMARY

SEGED, the digitization program, allows commands to be entered in two ways, from the keyboard and from the digitizing cursor. The commands from the keyboard are entered as one or two letters, as required by the command. Some commands have parameters, but prompts are issued for the user to enter those prompts. Commands from the digitizer cursor are entered by pressing one of the buttons. The assignment of buttons to commands is governed by the file \PEDITOR\DIG\BUTTON.COM. The BUTTON program allows one to assign commands to buttons.

SEGED operates in three modes: **SEGMENT, COUNTY,** and **JES.**

SEGMENT mode is used to digitize segments, specifying the tract and fieldfor each polybon digitized.

COUNTY mode is used to digitize strata boundaries, similar to SEGMENT mode, but using strata and count unit (PSU) notation for each polygon. In addition, due to the larger size of county strata files, a special fast digitization mode is available. Finally, to accomodate users at the NASS Fairfax facility in using existing plotting software, the location of labels may be entered into the digitized file.

JES mode is used to digitize the outline of segments in a quick way, for area only and not to save the digitized segment.

Various of the commands are available in various modes. For each command, the modes in which the command is available will be mentioned. This paper assumes some familiarity with digitization using SEGED and is ntended as a quick reference for the various commands.

KEYBOARD COMMANDS

A
Align segment or county, that is digitize the four corners if the picture or map. Normally, this command is not entered, but rather the program requests alignment before digitization or calibration may be performed. Available in **SEGMENT and COUNTY** modes.

C
Calibrate the segment or county. Calibration refers to obtaining corresponding points on a map and the image containing the outline of the segment or county. These may be the same image. Calibration serves to locate the segment or county as well as provide a scale. Available in **SEGMENT** and **COUNTY** modes.

CH
Check digitization.Checks for various digitization errors.
The CH command should be used before writing a digitized file to disk. Available in **SEGMENT** and **COUNTY** modes. See also the LH command for addition checking.

D
Digitize. Start digitizing boundaries. Once this command is entered, all inputs come from the cursor button until a command is given from thecursor, such as QUIT, to return to the keyboard for commands. Available in all modes.

DA
Delete all. Delete all digitized polygons to start over.Note that the calibration, if any, is retained. Available in **SEGMENT** and **COUNTY** modes.

DE
Delete a field or count unit. The name of the field or count unit to be deleted will be requested. Available in **SEGMENT** and **COUNTY** modes.

DL
Delete labels. Used to delete labels entered for the plotting program in the Fairfax office. The user is prompted for the labels to delete. Available in **COUNTY** mode.

FD
Fast digitize. Enter a special fast digitization mode in which checking is suspended until the polygon has been completed.
This is still a point mode of digitization, but goes faster since the points are stored up and checked later. Available in **SEGMENT** and **COUNTY** modes, but generally only used in the larger county files.

I
Initialize. Clears out any data in memory and leaves the program in the same state as if it had just been started. Available in **SEGMENT** and **COUNTY** modes.

LC
Label count units.For a count unit, the name is requested and then the location is digitized with the cursor. Only useful in conjunction with the Fairfax office plot program. Available in **COUNTY** mode.

LH
List holes. A hole occurs when one side of a polygon edge does not border a digitized field or count unit. Thus there is a "hole" around the entire boundary of the segment or county. Additional "holes" indicate a digitization error.
These holes are often referred to as "slivers".The LH check should be done before the segment or county file is written to disk. See the CH command for additional checking. Available in **SEGMENT** and **COUNTY** modes.

NS
Next state. Start with another state. The useris prompted for the state identifier, a two-letter postal code.Available in **JES** mode.

PA
Plot ASCII. Makes an ASCII (text) file of vertices and labels, suitable for the Fairfax office plot program. Available in **COUNTY** mode.

PS
Plot on the screen. Plots the outline of the segment or county on the PC screen. The plot may be exited by entering a carriage return. Available in all modes.

Q
Quit, exit the program. Available in all modes.

R
Read the (next) file part, or open the file and read the first part. Used always to read other than the first part. Used to initially open the file and read the first part if the file has a standard name. The standard names are <segment number>.SEG (e.g. 1024.SEG) in **SEGMENT** mode or <county fips code>.STR (e.g. 101.STR) in **COUNTY** mode. Use the RN command for a file with a nonstandard name. Available in **SEGMENT** and **COUNTY** modes.

RC
Recompute field or count unit sizes. Recomputes the sizes of all fields or count units using the current scale, from calibration if calibrated or from the default scale. The RC command should be used if a segment or county is recalibrated or if the scale is changed for an uncalibrated file. Note that the recompute is not automatic when there is a scale change. Available in **SEGMENT** and **COUNTY** modes.

RE
Rename field or count unit. The user is prompted for the old and new names. Available in **SEGMENT** and **COUNTY** modes.

RN
Read a non-standard named file. See the R command for reading standard named files. Available in **SEGMENT** and **COUNTY** modes.

SA
Save a digitized file. See also the SI command for automatic saving. Available in **SEGMENT** and **COUNTY** modes.

SE
Set minimum vertex separation. Sets the minimum distance in a fraction of an inch for which two digitized points will be considered to be separate vertices. If closer, they will be considered the same vertex. Available in **SEGMENT** and **COUNTY** modes.

SI
Set save increment.
In **SEGMENT** mode, specify the number of fields to be digitized between automatic saves.
In **COUNTY** mode, specify the number of count units to be digitized between automatic saves.
In **JES** mode, specify the number of segment outlines to be digitized between automatic saves.
The default is to do no automatic saves. The user is prompted for the value. Available in all modes.

SM
Sample points. Point sampling generates within a digitized area a random point. Then using the digitizing cursor, the user keeps entering points until one sufficiently close is reached. Generally used for rangeland segments. Available in **COUNTY** and **JES** mode

SP
Specify number of parts. A part of a segment or county is that portion which is shown on a single image. Also, separate parts may be generated for very large counties with too many edges or vertices for the capacity of the program. Available in **SEGMENT** and **COUNTY** modes.

SS
Set default scale. Sets the default scale to be used in computing field or count unit sizes. If the segment or county has been calibrated,the scale from calibration is used and the default scale is ignored. Available in **SEGMENT** and **COUNTY** modes.

SY
Specify the state and year to be used in the header of the file. The state and year is the two letter postal identifier for the state followed by the last two digits of the year. Available in **SEGMENT** and **COUNTY** modes.

V
View information about the file.
In **SEGMENT** mode, all fields digitized are shown along with their sizes and, optionally, bordering fields.
In **COUNTY** modethe display is the same but for count units.
In **JES** mode, all segments digitized are shown. Available in all modes.

VL
View label information. Shows the labels entered for the Fairfax office plot program. Available in **COUNTY** mode.

W
Write output file.
For **SEGMENT** and **COUNTY** modes, a standard named output file is written. See the R command for a description of a standard named file. Use the WN command to write a non-standard named file.
In **JES** mode, an output file is written with a name specified by the user. Available in all modes.

WN
Write an output file with a non-standard name. Use the W command to write a file with a standard name. See the R command for a description of a standard name. Available in **SEGMENT** and **COUNTY** modes.

DIGITIZER CURSOR COMMANDS

AUTODIGITIZE
Allows automatic digitization when a lengthy boundary must be redigitized. Commonly used when two fields or count units have a lengthy common boundary. Available in **SEGMENT** and **COUNTY** mode.

CLOSE FIELD OR COUNT UNIT
Close the polygon being digitized by generating an edge from the last vertex digitized to the first in the polygon. Available in all modes.

DELETE FIELD OR COUNT UNIT
Delete the polygon being digitized and start over. Available in all modes.

END STREAM
Quit digitizing in stream mode. Available in **JES** mode.

ERROR
Indicates that an error of some sort has been made. Action varies with the mode and also whether digitizing polygons or doing calibration. Available in all modes.

LABEL
The point digitized is to be the location of a label for the current count unit. Available in **COUNTY** mode.

MARK
Treat the digitized point as a point depending on the context, ot a command. Available in all modes.

NEW FIELD OR COUNT UNIT
Specify a new field number in the same tract in **SEGMENT** mode or a new count unit in the same strata in **COUNTY** mode. Available in **SEGMENT** and **COUNTY** modes.

NEXT COUNTY
Go to the next county in **JES** mode. This can only be used if the current polygon has been completed. The user is prompted for the name of the next county. Available in **JES** mode.

NEXT FIELD or COUNT UNIT
Go to the next field in SEGMENT mode or the next count unit in **COUNTY** mode.This may only be used if the polygon for the current field or count unit has been completed. The next field is in the same tract. If a partial field, the number after the "." is incremented by one else the field number is incremented. The next count unit has the same strata withthe count unit incremented by one. Available in **SEGMENT** and **COUNTY** modes.

NEXT PART
In **SEGMENT** of **COUNTY** mode, begin digitizing a polygon which is another part of the samefield or count unit. Commonly used if a field or count unit is contained entirely within another. In **JES** mode, begin digitizing another part of the current segment. Can only be used if the current polygon has been completed. Available in all modes.

NEXT SEGMENT
Go to the next segment in **JES** mode. This can only be used if the current polygon has been completed. The user is prompted for the segment number. Available in **JES** mode.

NEXT STATE
Start a new statein **JES** mode. The user is prompted for the two letter postal abbreviation for the state. Available in **JES** mode.

NEXT TRACT or STRATA
Go to thenext tract in **SEGMENT** mode or the next STRATA in **COUNTY** mode. This may only be used if the polygon for the current field or count unit has been completed. The next tract has the next higher letter with the field number set to 1. The next strata is the current strata incremented by one with the count unit set to 1.
Available in **SEGMENT** and **COUNTY** modes.

QUIT
Quit digitization and take commands from the keyboard. Available in all modes.

SET TRACT AND FIELD or STRATA AND COUNT UNIT
Enter the tract and field in **SEGMENT** mode or the strata and count unit in **COUNTY** mode. Can only be used when the last polygon has been completed or before any vertices have been entered since starting digitization. Available in **SEGMENT** and **COUNTY** modes.

START STREAM
Start digitizing in stream mode. Available in JES mode.

STATED	STAtistics file EDiting

PURPOSE

Display, editing, and creation of statistics files.

METHOD

The statistics file editor uses a command menu structure to allow the user to read statistics files into the input buffer, to inspect and display the statistics values of input categories, and to selectively copy the statistics records of categories to the output buffer. Most of the main level commands operate on categories in the input buffer (An exception is RENAME which names categories in the output buffer). Three commands (WRITE, CREATE, and POOL) add categories to the output buffer. A new statistics file may be created from the contents of the output buffer with the GENERATE command.

INPUT

Statistics File (type 54)

OUTPUT

Statistics File (type 54)
Backup listing file (optional)

COMMAND SUMMARY

The main level commands are as follows:

APRIORI PROBABILITIES
Enters the apriori probability subcommand mode.
Subcommands are:

CALCULATE APRIORI PROBABILITIES
Calculates the apriori probabilities for covers or groups of categories. The cover names of categories in the input buffer are used to group the input categories into groups with the same crop name. Only categories having one name which is not "UNKNOWN" are used for this grouping.Apriori probabilities are assigned to categories in each group accordng to the ratio of the number of points in each category to the total number of points in that group and a user supplied factor between 0.0 and 1.0. Remaining categories which have not been grouped by crop name may be grouped arbitrarily by user supplied lists of categories.

DELETE APRIORI PROBABILITIES
Sets the apriori probability of all categories in the input buffer to a null value.

ENTER APRIORI PROBABILITIES
Allows the user to enter the apriori probability for each category within a range of categories.

LIST APRIORI PROBABILITIES
Displays on the terminal the current apriori probability values for all categories in the input buffer.

QUIT
Returns to the main command level.

AUTODELETE
Deletes categories in the input buffer with less than a minimum number of points supplied by the user. The current default minimum number of points is 75.

AUTONAME
Assigns a user supplied cover name to all categories in the input buffer.Only a single valid cover name is allowed.

BACKUP (USE PRINT DISK FILE FOR LISTINGS)
Redirects certain output display information to a file on disk for later offline listing.The backup disk file name is supplied by the user. If a backup file is already in use, the user may elect to close it in order to return to terminal output.

CHECK CATEGORIES FOR ILL-CONDITIONED MATRICES
Tests all categories in the input buffer for ill-conditioned matrices by inverting the covariance matrix. Categories for which the inversion fails are noted on the terminal and deleted from the input buffer.

CLEAR OUTPUT BUFFER
Removes all categories from the output buffer.

CORRELATIONS (DISPLAY CORRELATION MATRIX)
Displays the correlation matrix for all categories in the input bufferon the terminal (or backup listing).

COVARIANCES (DISPLAY COVARIANCE MATRIX)
Displays the covariance matrix for all categories in the input buffer on the terminal (or backup listing).

CREATE NEW CATEGORY
Enters the create subcommand mode.The new category is initialized with zero means and the identity matrix.
Subcommands are:

USE VALUES FROM AN EXISTING CATEGORY
Copies the means, variances, and covariances of an input category specified by the user.

IDENTITY VARIANCE-COVARIANCE MATRIX
Sets the variances to 1.0 and the covariances to 0.0.The current means remain unchanged.

MEANS (ENTER MEANS)
Prompts the user to enter the mean for each channel. A <CR> only leaves te current value unchanged.

VARIANCES (ENTER VARIANCES)
Prompts the user to enter the variance for each channel.A <CR> only leaves the current value unchanged.

COVARIANCES (ENTER COVARIANCES)
Prompts the user to enter the covariance for each pair of channels. A <CR> only leaves the current value uncanged.

SHOW CURRENT VALUES
Displays the current means, variances, and covariances on the terminal.

WRITE NEW CATEGORY TO OUTPUT BUFFER
Creates a new category in the output buffer from the current values and returns to the main command level.If the current variance-covariance matrix is ill-conditioned, the write does not occur and control remains in the create mode.

QUIT WITHOUT WRITING
Returns to the main command level. No new category is created.

DELETE CATEGORIES
Flags a category or range of categories in the input buffer as deleted. Deleted categories may not be written or pooled to the output buffer.

ENTER INPUT CATEGORY NAMES
Assigns one or more crop names to a category or range or categories in the input buffer.Crop names may be entered on one or more lines with the names on each line separated by a comma and/or spaces.Names withspaces should be entered without spaces or with an underscore (_) to improve readability (e.g., WINTER_WHEAT).Crop names entered must be valid, i.e.,must be listed in the Crops File.The entry of crop names is ended with a <CR> only.

GENERATE NEW STATISTICS FILE FROM OUTPUT BUFFER
Writes a new statistics file to disk with the categories in the output buffer. The output file name is supplied by the user.After the write the output buffer is cleared.

MEANS (DISPLAY MEANS)
Displays the means for all categories in the input buffer on the terminal (or backup listing).

NAMES (DISPLAY NAMES FOR CATEGORIES)
Dislays the crop names for all categories in the input or output buffer on the terminal (or backup listing).

NUMBER (DISPLAY NUMBER OF POINTS FOR CATEGORIES)
Displays the number of points for each category in the input buffer on the terminal (or backup listing).

OPEN NEW INPUT FILE
Reads a statistics file from disk into the input buffer using the file name supplied by the user.The previous contents of the input buffer are lost.

POOL CATEGORIES AND WRITE TO OUTPUT BUFFER
Writes a new category to the output buffer formed from a list of 2 or more input categories supplied by the user.Pooling uses the LARS algorithm. Input categories to be pooled must have at least 2 points and must not have been deleted.

PRIORS (DISPLAY APRIORI PROBABILITIES)
Displays the apriori probabilities for all input categories on the terminal (or backup listing).

QUIT
Exits the program.If the output buffer is not empty, the user isasked to confirm the quit.

RENAME OUTPUT CATEGORIES

Assigns one or more crop names to a category or range of categories in the output buffer.Crop names may be entered on one or more lines with the names on each line separated by a comma and/or spaces.Names with spaces should be entered without spaces or with an underscore (_) to improve readability.Crop names entered must be valid, i.e., must be listed in the Crops File. The entry of crop names is ended with a <CR> only. This command is similar to the ENTER command but operates on the output buffer.The command is useful for naming categories which have been created or pooled.

SEPARABILITY

Enters the separability subcommand mode.
Subcommands are:

SWAIN-FU

Computes the Swain-Fu separability for categories in the input buffer. The separability matrix is displayed on the terminal (or backup listing).

TRANSFORMED DIVERGENCE

Computes the divergence, statistical separability, for categories in the input buffer.The divergence matrix is displayed on the terminal (or backup listing).The user can request to see the average divergence for each and allcategories, which are also displayed on the terminal (or backup listing).

SOURCE:

Remote Sensing: The Quantitative Approach - Swain/Davis.

GENERALIZED VARIENCE

Computes the determinant for each category in the input buffer.The values are displayed on the terminal (or backup listing).
NOTE: Transformed Divergence and Generalized Varience were added to the Stated Program on 2/12/86.

UNDELETE CATEGORIES

Removes the delete flag of a category or range of categories in the input buffer.

VARIANCES (DISPLAY VARIANCES)

Displays the variances for all categories in the input buffer on theterminal (or backup listing).

WRITE CATEGORIES TO OUTPUT BUFFER

Copies a category or range of categories from the input buffer to theoutput buffer.Only undeleted categories are copied.Categories written to the output buffer are added to the existing contents of the output buffer.

USAGE

The program initially prompts for an input statistics file. If the name of a valid statistics file is given, that file is read into the input buffer; if a <CR> only is given, the program goes directly to the main command level.
A new statistics file may be read into the input buffer at any time with the OPEN command. The statistics file editor is typically used in the following ways.

1.) Display the contents of a statistics file.Use the OPEN command to read the file. Then use the commands MEANS, VARIANCES, COVARIACES, CORRELATIONS,NUMBER, NAMES, PRIORS, SEPARABILITY as appropriate to display its contents.
2.)Selectively copy and/or reorder statistics records. Use the OPEN command to read the file. Then use a series of WRITE commands to control the ordering of statistics records in the output buffer followed by the GENERATE command to create a new file.
3.) Concatenate several statistics files. Use the OPEN and WRITE command for each input file to join the statistics records in the output buffer followed by the GENERATE command to create the new file.

CATEGORY SPECIFICATION

A category may be specified by its numeric value or its one or two character equivalent.Categories 1 through 61 may be specifed by the characters 1-9,A-Z,a-z. Categories 62 through 91 may be specified by the characters |$#%&'()*+,-/:;< = >@[\]~_\{\}\\ Categories 92 and above are specified by a two character string.
For example, B specifies category 11 and | specifies category 62.
A category range is specified by 2 categories representing the low and high values of the range separated by a space, comma(,), or dash(-).If a single category is entered, then that value represents both the low and high value,i.e., a range of one.As a special case, the keyword ALL in upper or lower case may be used to specify all valid categories.For example, for 20 categories:

Category range	Low High
1 10	1 10
1-10	1 10
1-A	1 10
A-D	10 13
B	11 11
all	1 20

A category list is specified by a list of values separated by a comma and/or spaces.

NOTES/LIMITATIONS

1.) The maximum number of channels is 8.
2.) The maximum number of categories is 150.
3.) The maximum number of crop names per category is 5.
4.) The maximum number of values in a category list is 20.
5.) The maximum number of covers for prior probabilities is 30.

STOT	TOTals file editor

INTRODUCTION

The totals file editor is used to create, update, and list the totals file. The totals file contains sizes by segment for various SELECT OPTIONS statements and will be used to input segment size values to the new estimator program. Until the new estimator is ready, the totals file will create the ASCII file of totals to be used as the ground data input to the old estimator, effectively replacing the GTOT program.

The Totals File

The totals file contains one list of segments.The actual data is placed into separate data blocks which will be referred to as option blocks. Each option block is specified by a SELECT OPTIONS statement, a default use, and a size type. The SELECT OPTIONS statement is any statement acceptable to the SELECT OPTIONS parser and is used to select the fields which will be included in the size computation for each segment. The default use is applied to any SELECT OPTIONS parameters requiring a use but for which no use is explicitly supplied. The default use is also used to build the list o covers for the option block by looking up the cover for each field selected. This list of covers is of limited capacity, currently ten covers, and is associated with the entire option block and not with individual segments. The size type is field, planted, harvested, or abandoned. Abandoned is the difference between harvested and planted. Thus, it is possible to have several option blocks with the same SELECT OPTIONS statement, but with different default use or different size type.

File Commands

When running the totals file editor, the user is first requested to enter a file commnad to create a new file or read an existing file. During processing, the entire contents of the file are kept in memory.

INITIALIZE TO A NEW FILE

Clears memory so that the currently open file is empty.

OPEN INPUT FILE

Opens and reads into memory the totals file as specified by the user. No default name is, as yet, assigned to the totals file.

Main Level Commands

Once the totals file has been initialized to empty or read into memory, various commands are available to change or display the contents of the file.

ESTIMATOR INPUT TOTALS FILE CREATION

Creates the ASCII totals file to be used as the ground data input to the old estimator. A SELECT REGION statement is used to select the segments to place in the output file. Any segments not in the totals file will have their sizes set to zero but will still be entered in the output file. A warning message is printed if any such segments are found. An option block is selected by the user and the size for each segment selected is copied into the output file. For compatibility, the cover list is placed on each line following the size, but the cover list is the same for all segments, being the cover list associated with the option block.

LIST

Lists the contents of the currently open totals file. The listing may be of the entire file or of a selected option block. The listing shows the SELECT OPTIONS statment used, the default use, the size type, the list of covers, and the size for each segment. The listing may be on the user's terminal or directed to a file for later printing.

OPTIONS DELETE

Deletes an entire option block as selected by the user.

RECOMPUTE

Recomputes the sizes and crops for a specified option block for segments selected. The option block selected may be one already existing or a new option block may be created. The segments selected may be all segments currently in the list or may be specified using a SELECT REGION statement, allowing partial recomputation and/or the addition of new segments to the list.

SEGMENT DELETE

Deletes the specified segment. The size value for that segment is deleted from each option block. However, the cover list in each option block is not changed.

UPDATE

Updates the list of covers or the size values of specific segments in the option block selected.

QUIT

Exits the program. If the totals file has been changed, it is written with a name specified by the user. No default name is as yet assigned to the totals file.

Selecting the Option Block

Several commands require selection of an option block. When a selection is required, all option blocks are listed and each is assigned an integer value, starting at one. For each option block, the SELECT OPTIONS statement used, the default use, and the size type are displayed. The user enters the value corresponding to the option block desired to select that block.

RECOMPUTE

The RECOMPUTE command is used to recompute the size values and drops list for an existing option block or to create a new option block and compute the size values and crops list for the new block.

The user is first asked to select segments to use. The allowed responses are ALL SEGMENTS CURRENTLY IN LIST or SELECT REGION. If the file is empty, the user must use SELECT REGION. When SELECT REGION is used, any segments not currently in the list are added to the list and the size values for those segments are set to zero for all option blocks.

Next the user is asked for the options to use. Allowed responses are NEW SELECT OPTIONS and OLD SELECT OPTIONS. NEW SELECT OPTIONS indicates that a new option block is to be created. The user enters the default use, size type, and the SELECT OPTIONS statment to be associated with the new option block. The cover list is set to empty and the size for each segment is initially set to zero. OLD SELECT OPTIONS indicates that an existing option block is to be used. The size values for all segments selected are recomputed. The cover list is totally recomputed only if all segments in the list are to be recomputed or if a new option block is being created. Otherwise, any covers not currently in the list are added. In either case, if the capacity of the cover list is exceeded, an overflow indicator is set,but the additional covers are lost. This does not affect the sizes put out for the various segments which will still include sizes for fields with excluded covers.

UPDATE

COVER UPDATE

Allows the user to add or delete covers from the cover list for a selected option block. Covers may only be added if the list is below capacity. When a cover is deleted, the overflow indicator is reset, losing any indication that more covers were found than could be stored during the last recompute operation on that option block.

SIZE UPDATE

Allows the user to change the size for selected segments in the option block selected. The segments are selected one at a time. For each segment, the current size is shown and the user enters a new size or carriage return only to retain the old size.

SUBWIN	WINdow file manipulation

PURPOSE

To transfer subwindows or complete copies of individual windows from one or more P-EDITOR window files to a new window file.

INPUT

1) One or more P-EDITOR window files (type 51).
2) User supplied commands.
3) One or more text files containing segment numbers and their corresponding subwindow coordinates.

OUTPUT

1) A newly created P-EDITOR window file (type 51).
2) Listing of input file headers, list of windows containing subwindow coordinates, and output window tables.

HOW IT WORKS

A output file window table is generated by the user with appropriate editing commands. This window table describes the input file, window number, and subwindow coordinates for each new window in the output window file. When the output table is complete, the command WRITE is used to generate the new output window file according to the details spelled out in the output window table. Additional commands are available to clear out the window table, and to list out information regarding the input file and the output window table.

COMMAND SUMMARY

OPEN (NEW) INPUT WINDOW FILE
Specifies a input window file to use for all subsequent LIST, COPY and SUBWINDOW commands. This file will continue to be used until CLEARed or replaced by another OPEN. Supply the name of the input file when prompted.

LIST INPUT WINDOW HEADER
Lists out information on the currently open input window file.

COPY INPUT WINDOWS TO OUTPUT TABLE
Adds a new entry to the output window table defining a complete copy of a specified window from the currently open input window file. Supply the window number when prompted. Enter <CR> to exit.

SUBWINDOW INPUT WINDOWS TO OUTPUT TABLE
Adds a new entry to the output window table defining a subwindow of a specified window from the currently open input window file. Supply the window number and the coordinates of the desired subwindow when prompted. Enter <CR> to exit.

WINDOWS CONTAINING SUBWINDOW
Lists all windows from the input file that would contain user entered coordinates regardless of their segment number. Supply subwindow coordinates when prompted.

DELETE WINDOW FROM OUTPUT WINDOW TABLE
Deletes a selected entry from the output window table. The table is re-numbered after each deletion to format continuous table: entries below the deleted entry in the table are moved up to fill in the vacated spot. Supply the output table entry to delete when prompted.

SELECT CHANNELS TO WRITE TO OUTPUT
Selects the number of channels to write to the output file, together with which input channel to use for each output channel. The current settings are printed, and then you are asked if you wish to change them. Reply "yes", supply the number of channels, and the channel assignments, when prompted.

WRITE OUTPUT WINDOW FILE FROM OUTPUT TABLE
Creates a new P-EDITOR window file, based upon the details contained in the output window table. Supply the name of the new file when prompted and also enter identifying information at the "header info" prompt.

CLEAR INPUT AND OUTPUT TABLES
Initializes the current input file, the output window table, and the channel select assignments back to their original state:no input file assigned, the output window table empty, and the channel select defaulting to using the same values as in the input files. If any values have been set, you will be asked if you really want to clear them. Reply "yes".

DISPLAY OUTPUT WINDOW TABLE
Displays the contents of the window table, describing the window number, operation (copy or subwindow), and input file to use for each entry in the output window table.

QUIT
Exit the program. If any values have been set, you will be asked if you really want to quit. Reply "yes".

HOW TO USE IT

Begin by using OPEN to open a input window file. The use COPY and SUBWINDOW, as desired to transfer over the desired windows from the input window file to the output table. If you wish to include windows from other files, repeat the above OPEN/COPY/SUBWINDOW operations described above. If you need to know more about your currently opened input file, use LIST or WINDOW. Use DISPLAY to see what your output window table looks like. If there are any entries in the table you wish to remove (due to error or change of mind), use DELETE to remove them (but remember that the output table is renumbered afterwards|). When you satisfied with the output window table, use WRITE to actually create and generate the new output window file. If you wish to generate another output window file, or if things really get screwed up, use CLEAR to re-initialize everything back to the beginning. QUIT when finished.

NOTES/LIMITATIONS

The current program works on Pixel Interleaved files only.

SVCAL .	SaVe segment CALibrations (NOt used in MARS-PED)

PURPOSE

Save and restore the calibration portion of Segment Network Files.

METHOD

This program is used to create or update a Calibration Save File with the calibration information from Segment Network Files, typically for all segments in a State. This calibration information includes the State and Year, calibration coefficients, central latitude and longitude, diagonal values, digitizing time, and date last written.
This program is also used to insert this information into existing Segment Network Files or to create the shell of new Segment Network Files.

INPUT

Calibration Save File (type 55)
Segment Network File (type 47)
User-entered commands and values

OUTPUT

Calibration Save File (type 55)
Segment Network File (type 47)
Listing File (optional)

COMMANDS

CLEAR FOR NEW FILE
Removes all segment calibration information from the program.

DELETE SEGMENTS
Deletes an entry for a segment or list of segments from the program.

DIRECTORIES (ENTER DIRECTORIES TO USE)
Sets the search path for Segment Network Files to other directories.

FILE USAGE FOR LISTING
Sets a switch to indicate a listing file may be wanted later.

INSERT CALIBRATION INTO SEGMENT NETWORK FILES
Creates new Segment Network Files with updated calibrations.

LIST SEGMENTS
Displays the segment information for individual segments, a list of segments, all segments, or just the segment numbers in use.

READ AN INPUT SVCAL FILE
Reads into the program entries from an existing Calibration Save File.

SEGMENT NETWORK FILE CREATION WITH CALIBRATIONS ONLY
Creates new Segment Network Files with calibrations only.

UPDATE CALIBRATION SAVE FILE
Enters calibration information from a new segment or list of segments into the program.

WRITE AN OUTPUT SVCAL FILE
Writes an updated Calibration Save File to disk.

QUIT
Exits the program. An output SVCAL file is written if the program has been updated since the last READ or WRITE.

HOW TO USE IT

The program prompts for a State and Year identifier which is a four character key composed of the two-letter postal State abbreviation followed by two digits indicating the year. The State abbreviation portion of the identifier is used to generate the Calibration Save File name used by a subsequent READ or WRITE command (e.g., KS.SVCAL). If a carriage return only is entered, the State and Year is not used but the user is prompted for a valid Calibration Save file name on a subsequent READ or WRITE.

If a Calibration Save File does not exist initially, the user should use the UPDATE command to enter calibration information from various Segment Network Files into the program. If a Calibration Save File does exist (from a previous year), the user may use the INSERT command to update the calibration portion of the Segment Network File for a newly digitized segment, thus exactly duplicating the calibration from a previous year. Alternatively, the user may use the SEGMENT command to generate the shell of a Segment Network File for later digitizing.

NOTES/LIMITATIONS

a.) The maximum number of segment parts in an SVCAL file is 500.
b.) The maximum number of segments input through use of a region file is 200.

TAPLIP	TAPe to LIP file

PURPOSE

Reads a Thematic Mapper tape from ESA and produces an ERDAS 7.2 image file on disk.

INPUT

A full scene (3 tapes) or quarter (1 tape) scene Thematic Mapper image.

OUTPUT

An ERDAS image file with the bands 1,2,3,4,5, and 7.

HOW IT WORKS

The program creates first an image on disk and then reads consequently the bands from tape and writes them into this image file. All bands are taken except band 6 (thermal channel).
When processing a full scene TM the image is divided over three volumes. The operator has to mount first the tape with channels 1,2,3 then 4,5,6 and the last tape mounted should contain channel 7.

COMMAND SUMMARY

@... = subcommand to be entered directly after beginning of the program.
- @TMF will read all necessary parameters from the file TMF.DAT in the executable directory and create a TM full scene.
- @TMQ will read all necessary parameters from the file TMQ.DAT in the executable directory and create a TM quarter scene.

HOW TO USE IT

TAPLIP has two ways of functioning. When the program has started the user has simply to choose between processing a full scene or a quarter scene by answering:

@TMF	for a full scene
or	
@TMQ	to process a quarter scene

He will only be prompted for the name of the output file.
When the 'L' key is pressed one can see at which line the program is.

TAPWIN	TAPe reading to create WINdow files

PURPOSE

Allows copying of selected portions of image data from tape to disk window file

INPUT

1) Magnetic tape containing image data in a form recognized by TAPWIN
2) User entered data

OUTPUT

1) A window file (type 51); the name of the file must be supplied by the user.
2) Depending upon the user commands entered, a file which contains a list of window coordinates may also be created. The name of this file can be supplied by the user.

HOW THE PROGRAM WORKS

The user is asked to enter the type of input tape to be used. TAPWIN obtains various tape parameters (e.g.,tape format, tape record length,image size) either from the tape header, or in the case of headerless tapes, directly from the user. Some tape parameters, such as the number of channels to be used in the windows or the row and column sampling increments, can be altered via user commands.
TAPWIN orders the user-specified window list by the northwest corner so it can make just one pass over the tape. When the windows are to be written to the disk window file, TAPWIN uses a separate procedure for each tape data format. For writing band-interleaved by line windows to disk, TAPWIN reads each row on the input tape, and if it is contained in one or more of the user-specified windows, it is written to the appropriate places in the window file.
TAPWIN finishes by rewinding the input tape.

COMMAND SUMMARY

(1) DESTROY COORDINATE LIST
Removes all the image windows selected up to this point from the window list, allowing the user to restart the window selection. This command does not reset row or column sampling rates, nor does it affect the choice of channels to be copied.

(2) DELETE WINDOW COORDINATES
Removes one window, specified by number, from the current list of window coordinates. The user should enter "PRINT" to examine the current list of coordinates before deleting any. As a safeguard, the coordinates of the window to be deleted are displayed,and the user is asked for confirmation before the window is deleted. If no windows have been selected, a message to that effect is displayed.

(3) IDENTIFY TAPE FILE

Displays data concerning the input tape and current window sampling parameters on the user's terminal. The data displayed are:

- tape format (i.e.,pixel interleaved,band interleaved or band sequential)
- tape record size
- number of images on tape
- number of rows, columns in the image
- number of channels on tape
- channels selected for window file
- row and column sampling rates
- number of windows selected
- identifying information either read from tape header or supplied by user

(4) INSERT WINDOW COORDINATES

Asks for north, west, south and east coordinates of a window within the tape image. TAPWIN will continue to ask for coordinates until a <carriage return> is entered.

(5) MOVE TAPE TO SPECIFIED FILE

For multi-image tapes, moves tape to a specified image. Not yet implemented.

(6) PRINT ALL COORDINATES

Displays the north, west, south and east coordinates of all the selected windows on the user's terminal.

(7) RENAME WINDOW COORDINATES

Allows the user to specify new values for the northernmost and westernmost boundaries of the tape image.The new southern and eastern boundaries are calculated so that the tape image remains the same size. The coordinates of any windows that have been selected are not adjusted. If any windows fall outside the new limits,the are deleted.

The coordinates of the deleted windows are displayed on the user's terminal. The values of the new coordinates must be > 0.

(8) SAMPLE TAPE DATA

Allows the user to set row and column sampling increments.

A row sampling increment of 3 means that rows 1, 4, 7... of the tape image will be read. The default sampling rates are 1, which indicates that all tape image rows and columns will be read.

(9) SAVE WINDOW COORDINATES

Writes the current list of window coordinates to a disk file. If the user does not specify a filename in response to the prompt, the coordinate data are written to a file called TAPWIN.COR.

(10) REDUCE NUMBER OF CHANNELS

Allows the user to select any or all of the channels on the input tape for inclusion in the disk window file. The channels present on the tape are displayed, and the user is prompted for new channels. If the user enters incorrect channels, the old selections are not altered. The default channel selection is all channels.

(11) WRITE TO WINDOW FILE ON DISK FROM TAPE

For each window selected, image data are transferred from the input tape to the window file. If no windows have been selected, TAPWIN displays a message on the user's terminal, and prompts for a new command. If the window copy is successful, TAPWIN exits.

(12) QUIT

Exit from TAPWIN.

HOW TO USE IT

The user begins by entering the tape type in response to the prompt from TAPWIN; IDENTIFY may be used to verify tape characteristics.
To choose windows, the user enters INSERT; prompts will appear for window coordinates. The user can continue entering window coordinates whenever the prompts appear; <return> will take TAPWIN back to the main command level. PRINT can be used to display the current window list at any time. Row increment, column increment, and channel selection can be done at any time prior to writing the window file.
The last command should be WRITE, as TAPWIN exits after a successful write.

NOTES/LIMITATIONS

(1) TAPWIN currently can read BIL (band interlieaved by line), BSQ (band sequential) and PIL (pixel interleaved) images.

(2) EDIPS, BARE (headerless) and BBN Editor tapes care the only formats presently supported.

(3) REFO must be called to reformat the window file; TAPWIN does not change the image format.

(4) TAPWIN is limited to 300 windows.

(5) TAPWIN can not reread the window coordinates it has written via SAVE.

(6) Row and column increments, and channels selected, apply to all windows in the window file.

(7) When reading PIL data, if only certain channels are to be copied from the input tape, TAPWIN may run very slowly.Copying speed decreases with increasing record size.

TTS	Television Tracking System (PC)

PURPOSE

To digitize boundaries with a video camera on the PC.

INPUT

Segment overlay from field survey.

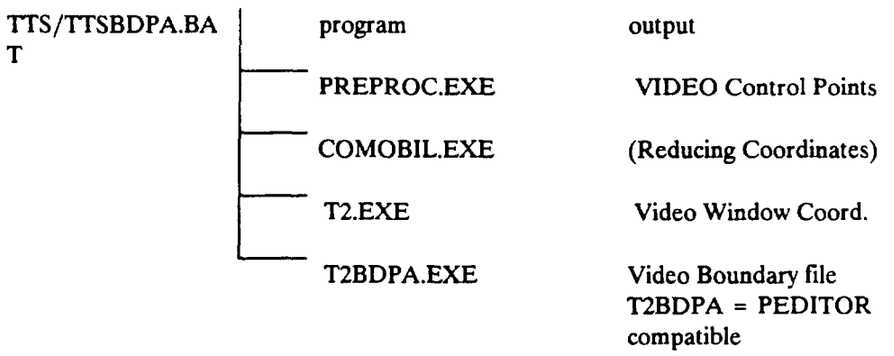
OUTPUT

Video boundary file
Video control point file
Video window file

HOW TO USE THE PROGRAM

The complete chain is started from batch files. Below a flow chart is given for the chain of processes.

FLOW CHART



STARTING THE PROGRAM

The program is started with the TTS command followed by a number < = 9999 on the same line.

Example, TTS 233 <cr>

This invokes the processing chain and will consequently create three data files in the C:\BOU directory,
i..e. 233.VBO, 233.VCO, and 233.VCP, which are respectively the video polygons, the video bounding rectangle, and the video control points file.

Another way of labelling the polygons is provided by the

TTSBDPA

command instead of TTS. This procedure will ask for the field number and the crop code separately. Then both entries are checked. If a field number is already entered it is refused. Moreover, when a crop code is not given in the C:\BOU\CROPCODE.DAT text file it is also refused. This extra checking is build in to prevent mistakes. The C:\BOU\CROPCODE.DAT file can be created using a real ascii texteditor(i.e.EDLIN). Every line may contain only one integer as a code.

DOCUMENT PREPARATION

The polygons on the document should be distinct and closed. Any other text like numbers or crosses that indicate ground control points are allowed as long as they do not form any closed figures as they will be regarded being a polygon. Every not connected line will be removed in the pre-processing chain. Best results can be obtained by black lines on acetate with red felt tip codes when using the red filter.

Polygons that are islands should be connected with the polygons that surround them by drawing a line between them. In this way you can use the V2P program to create MARS-PED segment network files.

USING THE MOUSE

The MS-mouse has only two buttons and in the program they are labelled.

1 Left button
2 Right button.

The program is designed such that one can not accidentally rush through a session because of holding too long a button down.

COMMAND BLOCKS

The session starts with grabbing an image.
On the screen the next message is displayed,

1. Frame grabbing. To stop: Press left button

After adjusting the drawing under the camera so that all the control points are visible and focussing with the smallest possible diaphragm any key stops the real time grabbing to proceed to the next step.
The next processes are executed,

2. Contrast stretch
3. Edge enhancement

which do not need any intervention from the operator.
After this the program enters the ground control point selection program. These control points serve the later registration of the boundaries to an satellite image.

On the screen the next message is displayed,

4. GCP points
   1) Zoom in
   2) End

which calls the mouse buttons.

Only in zoom mode one can select GCP's. By doing so the next messages is displayed,

5. GCP points
1) Zoom out
2) Store GCP point

Number of GCP's selected = 0
By moving the mouse one can center the cross over a point and when touching the right mouse button this point is stored and the same menu(5) comes up with the number of GCP's selected increased with one. One can zoom in and out at any time.
Only in through menu no. 4 one can exit the GCP selection module by selecting End. The next message is displayed,

6. GCP points
1) Exit
2) Continue/Reselect

Selecting Exit will finish the GCP selection module and will step to 7 to create an inverse image.
Selecting Continue/Reselect will display the options,

6.i GCP points
1) Continue
2) Reselect

Both options will bring the operator back to message 4 with the difference that option 1 will not delete any GCP's where option 2 will clear all GCP's to be able to restart.

7. Inverse image

which needs no operator intervention.
Next the threshold selection module is entered with the message,

8.
1) Select threshold
2) Apply threshold

A default threshold is used to start with. By keeping the left mouse button pressed and moving the mouse horizontally over the screen one can interactively select another threshold. A thumb rule is to try to apply the highest threshold without disconnecting any polygon lines. However with a good document preparation one can directly apply the default threshold without selecting it manually.
After selection the following message backs up an accidentally wrong threshold selection by touching button 2 to early,

9. Are you sure ?
1) YES
2) NO

Answering No will bring you back at 8.
Yes continues with the following processes that do not need any operator intervention:

10. Thresholding input

11. Line thinning

12. Isolated pixel removal

and concludes with the question:

13. Is this what you meant ?
1) YES
2) NO

In blue the pixels are colored that will be removed. In light blue or green the lines are displayed that are kept. Answering No displays the following message:

14.
1) Go back to unthresholded image
2) Add some lines

Option 1 will bring you back to menu 8 for a new selection of the threshold.
Option 2 enables the operator to ADD or DELETE lines by cutting existing lines, so they will be removed. With these options the operator does not have to go back, and restart completely from the beginning when some lines on the drawing are not closed.
Add some lines will ask the operator:

15. "Do you want to zoom in"
1) YES
2) NO

Zooming in can be usefull when small fields have to be added or deleted. The following message will be

16. "Move the cursor to the first point and press the left button"

Take the first point of the line to be drawn or the line to cut other lines with.
Then the second point can be selected with the options

17.
1) Draw line
2) Erase line (cut existing line)

When selected the message

18. "Do you want to add more lines"
1) YES
2) NO

Answering YES will bring you back to 16, NO will bring you back to 9.
The pre-processing chain is finished with conferming 13 and processing takes place on a higher level (PTL part 2).

POLYGON LABELLING AND DUMP (T2,T2BDPA)

The polygon tracking and labelling program is started automatically after step 13 of the command block has been passed. Every polygon on the screen is closed as a result of the pre-processing. Sequentially all the fields are made red and the class names are asked. After labelling the fields will become green.
This module (PTL) starts with displaying the HELP page, ENTERING

Any string	=	Label and continue
.	=	Skip
/	=	Abort
?	=	This message

- A string up to 40 characters can be entered as a label but it is advised to use integers here because ERDAS can only proccess integers as polygon labels.
- TTSBDPA asks field number and crop code and writes these separated with a komma to the label field. It checks if a field number (must be < 256) has already been entered and if the cropcode exists in the C:\BOU\CROPCODE.DAT file.
- Use "." when a field is erroneously tracked because of a not well prepared document. The polygon will be tracked but not dumped to the output data file and labelling will continue.
- Use "/" to abort the program and to save the labelled fields. The current green field will not be saved.
- Use "?" to display the help page again.

DATA FILES

Three DOS text files are written to the C:\BOU directory on disk:

1. Boundary file where each polygon has the following format in screen coordinates,

y1,x1, (Fortran I5,X,I4,X)
y2,x2,
.
.
yn,xn,
/ AREA (Fortran F12.2)
/ LABEL (Fortran A40)

and is clockwise reported.
The file extension is .VBO.
In case of the TTSBDPA program the label consists of FIELD NUMBER, CROPCODE.

2. Video control point file where each point is sequentially stored as it was entered in the session and has the following format,

y1 x1 (Fortran 2I4)
y2 x2
.
.
yn xn

3. Window coordinate file where the screen coordinates of the total bounding rectangle is reported in the format,

North West South East (Fortran 4I4)

one per segment.

VIDEO BOUNDARY TRANSFORMATION MODULE (VBT)

A separate module for ERDAS users has been developed.
This **VBT** module does a geometrical transformation using a polynomial function of the first degree of the output from the TTS system to any other coordinate system. Only metric coordinate systems are supported in this module as no other calculations during transformation are applied.
Output is an ERDAS .DIG file in for example a map coordinate system. This .DIG file can then be used for further processing.
The module is able to process 5 video boundary files at one time.
When segments (a collection of connected polygons) have an increasing numbering the VBT module can process them one after the other without operator intervention. Also single segments can be transformed. The following will explain which files the program needs and which it creates.

INPUT

.VBO
.VCP
.VCO
.GCP

- .VBO, .VCP, and .VCO are output from the TTS system.
- .GCP is a Ground Control Point file from ERDAS or a Point file from ILWIS where the coordinates of the MAP control points are reported.
  + With ERDAS the modules GCP or GCPX creates this file.
  + With ILWIS the .PNT file creation module can be used.

However, any program can be used to create this file but it has to be named segmentnumber.GCP and must have the same format as the ERDAS .GCP files where the Image Coordinates can have any value because they will be neglected.

This format equals:

Line 1	first line of the file will be skipped,
Line 2	x1 y1 ! map coordinates of point one in the .VBO file
Line 3	x2 y2
Line 4	x3 y3
Line n	xn yn etc.

OUTPUT

.DIG
.PCP
.MCO
.VCA

-.DIG is an ERDAS .DIG polygon file in the new coordinate system with the origin in the lower left corner.
-.MCO contains the window coordinates in the new projection system. This file can be used to check the result of the transformation or for modules to be developed in the future.
-.PCP is a file that contains both video and map control points pair wise. It can be save to check the RMS errors obtained after the transformation with the PEDITOR module REGCP, and in the future will be used to create higher order polynomial functions. Currently there is no application for this file.

-.VCA is a file that contains the coefficients of transformation. It can be deleted or saved for future applications or be used with the PEDITOR program REGCP.

USAGE

The program is started with the VBT command. It asks for the range of segments to be processed and will abort when they are not available on the current directory.
During execution the operator will be informed about the RMS errors per segment and the segment number that is processed.

Area Summary module (AREA)

To be able to create a list of area occupation for each tracked field a program AREA is provided. The only input it requires are the total area of all the segments present in the .VBO file and the .VBO file itself. It calculates then the area for each field separately. With a total area of 100 one gets a report of the percentages for each field. Ouput is an DOS texfile in the format,

Segment number, Crop code, Area.

Polygon Relabelling module (RELABEL)

During labelling errors can occur. The program RELABEL is used to correct or update the labels in the polygon file. It displays the polygons on the color monitor and asks which polygon to be updated. The program is straight forward and therefore the description is omitted.

Display Polygon module (DISPOL)

The program DISPOL displays the polygons of a video boundary file (.VBO) on a high resolution color monitor.

Video boundary to ARC/Info conversion (VBO2ARC)

The program VBO2ARC converts the contents of a .VBO file into a file readable in ARC/Info with the GENERATE command. The program asks what format the label is put:

'Enter VBO LABEL format, i.e. I = Integer only'
' A = Alpha numeric'
' B = Field,cropcode'

When you digitized the pictures with the TTS command then choose 'I' or 'A' when you have also alpha numeric characters in the labels.
When you digitized the pictures with the TTSBDPA command then choose 'B'. Only the crop code will be reported in the new ARC/Info textfile.
Please note that ALL polygons in this file are reported clockwise.

Video Boundary Transformation program including conversion to ARC/Info (VBT2ARC)

The VBT2ARC program converts a video polygon file (.VBO) with the help of a .GCP file to an ARC/Info file with already transformed coordinates. See the description of the VBO2ARC program and the VBT program description.

NOTES/LIMITATIONS

The horizontal correction factor for non-square pixels is set to approx. 1.45 in the ADDRESS.DIS file. In order to check if your own system has the same factor please use the ASPECT program. If necessary lease update the ADDRESS.DIS file at position nr. 8 for this factor.
Note that whenever you change this parameter in the ADDRESS.DIS file you MUST change it too in the GCP.CMD file on the first line, i.e. xfac=1.45 to xfac=new value.
Update the ADDRESS.DIS file WITH A DOS TEXT EDITOR. **The ADDRESS.DIS file must be terminated with a Carriage Return.**

XGCP	X-window Ground Control Point

PURPOSE

To define Ground Control Points into a multiwindow or ERDAS image format.

INPUT

An image file in one of the two following formats:
1) multiwindow (file type 51).
2) ERDAS.

OUTPUT

1) An ASCII file containing the coordinates of the control points and their labels(number).

COMMAND SUMMARY

XGCP has an on screen menu with items selection done using the mouse.
The first level items are here described with letters, second level items with numbers.
The items are as follows:

a) Images — load images and define how they will appear on the screen using the following sub commands.

1) Image type:	1) multiwindow 2) ERDAS (only one of the two type).
2) Image name:	name of the file containing the image.In case of files in another directory the path has to be specified.
3) Number of windows:	in case of multi-windows images the number of the window wanted.
4) Band number for each colours:	the band to be associated to each colour (Red, Green, Blue).
5) Strecth type for each colour:	1.MIN-MAX 2.Histogram Equalization

6) Row: Column:	number of rows of the image to display. number of columns of the image to display. (In both the cases the range is displayed).
7) Row to skip: Column to skip:	number of rows of the image to skip. number of columns of the image to skip. (In both the cases the range of the value is the maximum of the displayed range in precedent step range minus the input in the precedent step).

When all the option have been selected the image is loaded and then displayed.
A state box that report data regarding the image displayed will appear on the right of the screen.

b) Select Subset:	defines which part of the image has to be displayed. The selection is done by row and column as in point a.6. It is possible display the whole image.
c) Clear Image:	deletes the image displayed.
d) Clear Zoom:	deletes the portion of the image displayed in the right corner of the screen.
e) Image on/off:	switches the image ON or OFF the screen.The Ground Control Points selected still remain on the screen.
f) Open GCP file:	opens the GCP file related to the image on screen. This step is necessary for selecting new GCP because if the file is not open the selection of the points is impossible.
g) Close GCP file:	close the GCP file. This step is necessary to save the Control Points selected in the session or change the image to analyze. If it is not performed the data are lost.
h) Exit:	leaves the program. The data in the GCP file are saved also if it is not closed.

HOW TO USE IT

Call XGCP. A window will appear on the screen with a menu on the right. Select the IMAGES item with the mouse and open the window following the commands that will replace the box of IMAGES. The commands are in the same order as reported in the point a) of .the command summary.
When the selection is complete the image will be loaded and then displayed.
The GCP file must be open clicking the item with the mouse and input the name. If you give a name
of an existing file this will be simply updated.
Now select the zone in which you want fix the Control Points.
This zone will appear in the corner down on the right of the screen (below thestate box). Move in this box with the mouse and choose the point clicking the mouse. After the program will ask you what number assign to the Ground Control Points.If you input a value already used a

message will appear asking you if you want update or not the point with that value. If you don't want update the point insert a value not yet used.
When you have finished the selection of the control points for the image close the GCP file clicking the item.
If do you want use another image repeat the command sequence restarting with the command IMAGES.
If you have finished leave the program clicking the EXIT item.

NOTES/LIMITATIONS

Hardware requirements:X-Term.
For add points open an old GCP file as above described; then select the points on the screen as before but give a different number to each point.

XSFT	X-window Segment shiFT

PURPOSE

To perform the segment shifting on a image in multiwindow format displayed on the screen, using the mouse.

INPUT

1) A Multiwindow File (type 51) containing the image.
2) Segment Network Files (file type 47): one for each segment to shift.
3) Image Calibration File (ASCII text file, no MARS-PED type).

OUTPUT

1) Segment shift file, ASCII file and not MARS-PED type.

COMMAND SUMMARY

XSFT has an on screen menu. The items selection is done using the mouse.
The first level items are here described with numbers, second level items with letters.
The items are as follows:

1) Images	choice the image to display using (in order) the following commands:
a) Enter Segment File Directory:	the directory where the segment files are logged.
b) SSYY:	state and year in analisys.
c) Enter Image File Directory:	the directory where the multiwindow file is logged.
d) Enter Image File Name:	the name of the multiwindow file.
e) Enter Calibration File Name:	the name of the Calibration file.
f) Enter Shift File Name:	the name of the Shift file
g) Window Number:	the number of the window in which the segment is A range is reported
h) Red (1..4):	band number to be associated to red
i) Green (1..4):	band number to be associated to green.
j) Blue (1..4):	band number to be associated to blue.
k) Stretch type:	1) MIN-MAX 2) Histogram Equalization

Using the inputs defined above the program search, load and display the window and the segments.

1) Images	choice the image to display using (in order) the following commands:
2) Select Window:	allows change the window to use. A range of possible values is displayed.
3) Next Window:	allows to select the window following the one in use. When the last window of the multiwindow file is in use this command doesn't work.
4) Clear Image:	deletes the image displayed and close the multiwindow file. A new multiwindow file have to be load if you want continue the segment shifting.
5) Image On / Off:	switches the image On or Off, leaving displayed the segment mask.
6) Step + :	increases the step of the shift.
7) Step - :	decreases the step of the shift.
8) Change Segment Color:	changes the color of the segment mask among eight colors.
9) Change Channel:	changes the relation among bands and colors. A submenu will appear on the up right corner for the selection..
10) Exit :	leaves the program.

HOW TO USE IT

Type XSFT. On the screen will appear a window with a menu on the right. Select the IMAGE item with the mouse and in the IMAGE window the commands will appear following the order above described. When the selection is complete the image will be loaded and then displayed with the edges of the segment . A box with the following data about the image and segment in use will appear below the menu of the commands. The data reported in this box are:

IMAGE FILE:	the image file name.
WINDOW SELECT:	which window is in use.
SEGMENT NUMBER:	the segment number.
CHANNELS SELECTED:	association among colors and bands.
SHIFT:	
STEP	the step value in video pixels.
ROW SHIFT	shift to be applied to the rows to better fit data.
COLUMN SHIFT	shift to be applied to the columns to better fit data.

Now you can set the step to the wished value and move the mouse over the image on the side of the segment where do you want shift the segment. Then clicking the mouse the segment mask

will be shifted of the step value in that direction. You can continue until the wanted position is not reached.
Then you can change image or segment using the menu.
If do you want use another segment select one of the two items: SELECT WINDOW or NEXT WINDOW. If you have finished leave the program clicking the EXIT item.
If do you want use another image repeat the command sequence restarting with the command IMAGES.

NOTES/LIMITATIONS

Hardware requirements: X-term.

FILES
FILES LIST

There are files with a related type:

Name of File	Type	Standard Name	Notes
Button to Menu Mapping	F61		Not used
Calibration Save	F55		Not used
Control Point	No type	{any}.PCP	
Crops	F53	PEDITOR.CRP	
Estimator Parameter	F59	{cover}.ESTS	
Estimator Results	F60	{cover}.ESTL	
Filler	F62	{scene1-id}${scene2-id}.FILLER	
Frame Units	F50	{ssyy}.UNIT	
Ground Truth	F52	{segnum}.GTRUTH_{ssyy}	
Image Calibration	No type	{any}.CA1	
List of Blocks	F14		
Multi-window Categorized	F51	{any}.MWN	
Packed	F57	{any}.PACK	
Pc			
Pcal			
Principal Components	F66		
Scan Mask	F48		Not used
Segment Aggregation	F24		
Segment Catalog	F49	{ssyy}.CATLG	
Segment Masks	F17	{segnum}.MASK_{scene1-id}${scene2-id}_{ssyy}	
Segment Network	F47	{segnum}.SEG_{ssyy}	
Segment Shift	No type	{any}.SFT	
Segment Shifts			
Statistics	F54	{cover}.CFS	
Strata Aggregation	F65		See F24
Strata Mask	F64		See F17
Strata Network	F63	{segnum}.STRATA_{ssyy}	See F47
Table	F56	{any}.TABLE	
Totals	F58	{any}.STOT	
Video Bmask			
Window Coordinates	No type	{any}.WINCOORD	

where:

{any}	anything
{ssyy}	ss = State yy = Year
{cover}	Cover name for the individual crop
{segnum}	Segment Number
{scene1-id}${scene2-id}	{scene1-id} = First 5 digits for scene identification {scene2-id} = Second 5 digits for scene identification

BUTTON TO MENU MAPPING FILE (type 61) (Not used in MARS PED)

The button to menu mapping file contains information concerning the digitizing configuration and is used by the digitizing interface library "diglib".
The file is initially created by the program BUTTON and may be modified during a digitizing session.

FILE CONTENTS

The file consists of 24 words (minimum) and contains
information on the digitizer type, port requirements, the number of cursor buttons in use, the function of each button, and the x and y values of the offset of the menu origin from the digitizer origin.

FILE FORMAT

Word #	Type	Value	Description
1	word	(0)	Zero
2	word	(61)	File type
3	word	(-1..)	Digitizer type The ordinal value of digtype as listed in "diglib.typ" or -1 for unknown
4	word	(-1,..)	Port requirements -1 Unknown 0 No port requirements Positive values indicate that an external port is required. If the value is >100, it is the baud rate and further port information follows. Otherwise, the user specifies the port information.

If word 4 is greater than 100, the following information is seen in the file.

5	word	(1,..)	Length in characters of the portname, call it n
6..5+(n+3)/4	character		The port name as it is pessed to the system used for digitization

For the remainder of the file, if word 4 is >100 set p to 6+(n+3)/4 else set p to 5.

p	word	(..)	X offset of menu in inches * 1000
p+1	word	(..)	Y offset of menu in inches * 1000
p+2	word	(0)	Unused
p+3	word	(0..15)	Maximum button number in use
p+4..p+19	word	(0..)	Function of buttons 0 to 15 for digitization of segments and strata
p+20...p+34	word	(0..)	Function of buttons 0 to 15 forregistration. The functions are as listed in"diglib.con" i.e., MARK = 0 etc.

CALIBRATION SAVE FILE (type 55) (Not used in MARS PED)

The calibration save file is used to save calibration information from Segment Network files. Typically, the file is used to save the calibration information for all segments in a State. The file is created by the program SVCAL.
Segment calibrations are needed for all included segments in the project. It is however, only necessary to calibrate segments one time since segment location does not change from year to year.

Manual Digitization States

The segment calibration coefficients are part of the segment network file that is used to create segment masks during analysis.Segments new to the project are calibrated as they are digitized and the calibration coefficients stored with the segment network files. The calibration coefficients for segments which have been part of the project for one or more years are stored in a calibration save file until they are needed during digitization and they are read into the new (current year) segment network file. At the conclusion of each years processing, the calibration save file must be created to pull off the calibration information for all non-rotating segments to be stored for next years processing by running the SVCAL option.

Video Digitization States

For video digitization states, it is necessary to calibrate all rotating segments and all segments with replacement photography.The calibration information from segments for all non-rotating segments are saved on the PDP-11 from year to year. These files containing the calibration information are merged with the video mask to create segment masks which will be used in analysis on MMDS.

FILE CONTENTS

The file consists of 32-bit words(and reals) with an 8 word header, one word for each segment containing the segment number, one word for each segment containing the number of parts, followed by a 16 word entry for each segment part.

Manual Digitization States (Calibration Save File)

Header

File type
State and year
Number of segments

Body

Segment number
Part number
State and year
Central Latitude
Central Longitude
Calibration coefficients

Video Digitization States (Calibration file)

In video digitization states, the calibration information as described before is saved as an individual file for each segment.

FILE FORMAT

Name	Type	Value [length]	Description
	word	(0)	
	word	(55)	File type
	word	(..)	State and Year Identifier (32 bit ASCII equivalent of a 4 byte field).Two letter postal State abbreviation followed by a two digit year
	word	(..)	Date and Time written, standard PEDITOR Datetime value
	word	(16)	Length of entry in words
	word		Not used
	word		Not used
nsegs:	word	(1..500)	Number of segments
snum:	word	[nsegs] (0..9999)	Segment number of each segment
np:	word	[nsegs] (1..)	Number of parts for each segment
entry:	record		For each segment part
	word	(0..9999)	Segment number
	word	(1..)	Part number
	word	(..)	State and Year Identifier
	word	(..)	Digitizing time
	word	(..)	Date and Time entry created
	word	(..)	Date and time file created
	word	(1..)	Diagonal value 1 (upper left to lower right)
	word	(1..)	Diagonal value 2 (upper right to lower left)
	real	(-90..90)	Central latitude in degrees
	real	(-180..180)	Central longitude in degrees
	real	[6] (..)	Calibration coefficients.If cal[1]>1.0e9 then segment not calibrated
	end	[sum of np[1]..np[nsegs]]	

STANDARD NAMING CONVENTIONS

Manual Digitization States (Save cal file)

Video Digitization States (Calibration file)
SSXXXX.#CM where
SS is the two letter state abbreviation
XXXX is segment number
is the number of the part of the segment.

CONTROL POINT FILE

A control point file is a text file (does not have a file type) that contains information on the control point pairs as digitized from maps and images. Within PEDITOR module, CPEDIT, this file is used as input to the least squares routines to generate a transformation that can be written to disk as an image calibration file or an overlay parameter file.

The file is stored as a text file in ASCII format. The basic unit of information in the file is the line, a collection of ASCII characters terminating in a carriage return. There is a file header of five lines followed by the control point data which contains seven lines of text for each control point pair.

Where more than one entry exists on one line, the comma(,) is used as a separator. Each line in the format represents one line in the file (e.g. a two control point pair file will have 19 lines).

NOTE

Since it is a text file, it can be edited using any text editor.

FILE FORMAT

Header:

Name	Type	Description
title :	str	Title for the entire file
npairs :	integer	Number of Control Point Pairs in the file
regmode :	modetype	Mode of registration (map-image or image-image)
prmtype :	map or imtype	Type of Primary Media (type of map or image)
sectype :	imagetype	Type of Secondary Media (type of image)

Body

The "from" specification refers to the coordinate system from which the data will be transformed, whereas "to" refers to the coordinate system receiving the transformed data.

Name	Type	Description
blankline :		There is a blank line at the beginning of the data for each control point pair.
cpidinfo :	str	Identification information for the control point pair.
lat,long or row columnn	real	Latitude and logitude or row and column for the point in the primary coordinate system (map or image)
row,column :	real	Row and column of the point in the secondary (image) coordinate system.
rowerr,colerr:	real	Row and column root mean squared errors.
meterror :	real	Total error in meters.
flag :	cptype	Type of control point (CONTROL,DELETED,or CHECK).
[1..npairs]		number of control point pairs

STANDARD NAMING CONVENTIONS

name:{anything}.PCP

EXAMPLE

```
 16
IMAGEIMAGE
EDIPS
GODDARD
CP 1
 2607.640,  784.738
 2603.533,  807.164
    0.000,    0.000
    0.000
CONTROL
CP 2
 2866.385, 2298.819
 2861.742, 2319.565
    0.000,    0.000
    0.000
CONTROL
CP 3
 1939.717,  965.3627
 1937.819,  987.6879
    0.000,    0.000
    0.000
CONTROL
..............
..............
{ etc }
```

CROPS FILE (type 53)

The crops file is used to store the names for cover,intended use, note, and irrigation. The numeric value for each name is its position in the list. The first item in each list is number zero and should be named UNKNOWN. Although this is not required by the structure of the file, certain programs expect it.

FILE CONTENTS

The header consists of just the file type.
Each list starts with the number of items in a word, then has the items.Each item has a length in the first byte and then the name of the item at one character per byte. The next item starts where the previous one left off, without regard to word boundaries.

FILE FORMAT

Header:

Type	Value [length]	Description
word	[1] (0)	
word	[2] (52)	file type = 52

Body

Type	Description
array [cover, intended use, note, irrigation] of record	
word	number of elements
array [1..number of elements] of	
array [1..NN+1] of byte	N is the length of the name and is stored in the first byte
end	

STANDARD NAMING CONVENTIONS

name:PEDITOR.CRP

DESCRIPTVELY PACKED FILE (type 57)

A packed file contains pixels from one or more multiwindow files which meet criteria specified by a SELECT OPTIONS statement and for segments specified by a SELECT REGION statement. In the SELECT OPTIONS statement, the analyst may specify which cover(s) to pack. By using the automatic packing option, the user may pack either ALL CROPS or MAJOR CROPS for the segments specified in the SELECT REGION STATEMENT. The SELECT REGION statement will be a list of segments included in the analysis district or a statement allowing the program to select a list of segments from the segment catalog file. In general, a specific packed file contains pixel level data for all fields in segments within an analysis district. That, based on ground truth data have been identified as that specific cover. For example, a packed corn file will contain Landsat pixel values for all fields identified in the ground truth file as corn fields within the analysis district of interest.
The descriptively packed file contains complete information about each pixel as created by the Pack program. This complete information includes the segment number, cover, whether or not the pixel falls on a boundary between two fields, the tract, the field number, the row and column for the pertinent satellite image, and the actual channel values of the pixel.

FILE CONTENTS

The file has a header of 1600 32-bit words. This header is the same as for the window file, type 51.

Header

List of segments with packed pixels in file
Internal representation of SELECT OPTIONS statement
Names of frames used in packing

Body

Pixel data is stored similarly to multiwindow files
Descriptively packed files also contain segment #, tract,
field #, cover,and Landsat row and column information
for each pixel

FILE FORMAT

Header

The header is the same of File Type 51.

Body

The data has 12+NCHAN bytes per pixel,where NCHAN is the number of channels in the satellite data. This data, in bytes, is as follows:

Type	Value [length]	Description
byte	[2]	Segment number
byte	[1]	Cover number, an index into the standard PED file of covers.
byte	[1]	Boundary, non-zero if the pixel falls on a boundary between two fields. The pixel is assigned to one of them anyway.

byte	[2]	Tract. Each byte is an upper-case letter. If the tract is one letter only, the second byte is a space.
byte	[2]	Field number, which is the field number multiplied by 100 with the subfield number added.
byte	[2]	Row. The row number in the pertinent satellite image.
byte	[2]	Column.The column number in the pertinent satellite image.
byte	[NCHAN]	the actual raw data from the satellite, one channel per byte.

STANDARD NAMING CONVENTIONS

name: COVER.PACK

ESTIMATOR PARAMETER FILE (type 59)

The estimator parameter file contains information used in and parameters computed during small scale estimation (sample estimation) to be used as input to large scale estimation. The parameters are the slope and intercept of the regression line. Some of the other information is the list of counties cover and whether strata were combined.
The file consists of a 20-word header followed by the body of the file.

FILE CONTENTS

Header

file type
state and year identifier
units (acres)
number of independent variables
number of strata
group indicator
augmentation indicator

Body

segment numbers
variable values
strata numbers
number of segments in each strata
list of counties
type of variable
type of size
number of covers
list of covers
sum of squares of the dependent variable
variance of the dependent variable
variance of the independent variable
covariance
regression coefficients

FILE FORMAT

Header

The header is as follows:

Type	Value	Description
word	(0)	file type, first word.
word	(59)	file type, second word.
array [1..4] of byte		state and year identifier, set to spaces if no state and year
word	(0..1)	units, 0 = hectares, 1 = acres.
word	(1..5)	number of independent variables, NVAR.
word	(1..450)	number of segments, NSEG.
word	(1..20)	number of strata, NSTRAT.

word	(-20..0)	group indicator, NGROUP. If less than zero, grouping of strata (combined estimate)is used and ABS(NGROUP) is the number of groups. If NGROUP=0, combined estimate is not used.
word	(1..500)	number of counties, NCOUNTY.
word	(0..1)	augmentation indicator, AUGMENT. Set to 1 to indicate file contains variable values by segment and regression coefficients. Set to zero if the file does not contain this data.
array [1..10] of word	(0)	reserved for expansion, explicitly set to zero.

Body

The body of the file is a follows:

Name	Type	Value	Description
	array [1..NSEG] of word	(1..9999)	segment numbers.

IF AUGMENT = 1, the following appears:

array[0..NVAR] of array [1..NSEG] of real		variable values for each segment and variable, with 0 being the dependent variable (y) and 1..NVAR being the independent variables (x[i]).

In all cases, the body of the file continues:

array [1..NSTRAT] of word	(1..100)	strata numbers.
array [1..NSTRAT] of word	(1..)	number of segments in each strata.

If NGROUP is non-zero, indicating use of the combined estimate, the following appears:

array [1..NSTRAT] of word	(-20..0)	group number for each strata, set to zero if the strata is not a memeber of any group.

In all cases, the body of the file continues:

array [1..n] of byte		the list of counties, n is not stored elsewhere. Each county is preceded by one byte giving the length of the county name, not including the length byte. Each county immediately follows the one before it and so does not necessarily start on a word boundary.
array [0..NVAR] of record		0 is dependent variable, and 1..NVAR are independent variables.
word	(0..1)	type of variable, 0=pixels and 1=from totals file.
word	(0..4)	type of size if type of variable is 1 (from totals file),

		0=field, 1=planted, 2=harvested, 3=abandoned, 4=digitized. If type of variable is pixels(0), this value is set to zero.
word	(1..10)	number of covers, NCOV.
array [1..NCOV] of word	(0.255)	list of covers. Each cover is an index into the standard PEDITOR crops file.

end of record.

The remainder of the file contains various regression coefficients. These are different for single-variable regression and multi-variable regression.

For single-variable regression, let SSX be the sum of the squares of the independent variable, SSY be the sum of the squares of the dependent variable, and SSCP be the sum of the products of the independent and dependent variables. Sums are for all segments in a strata. Let XM be the mean value of the independent variable and YM be the mean value of the dependent variable. Means are totals for each strata divided by the number of segments in each strata. Let NSEG be the number of segments in each strata. Then, the remainder of the file is:

array [1..NSTRAT] of real	SSX.
array [1..NSTRAT] of real	S2X=(SSX-XM*XM*NSEG)/(NSEG-1).
array [1..NSTRAT] of real	S2Y=(SSY-YM*YM*NSEG)/(NSEG-1).
array [1..NSTRAT] of real	S2XY=(SSCP-XM*YM*NSEG)/(NSEG-1).

If AUGMENT =1 ,the following appears

array[1..NSTRAT] of array [0..1] of real	B[0] and B[1], the regression coefficients.

For multi-variable regression, let I, J, K be variable indices from 0 to NVAR with 0 indicating the dependent variable. Then let V[I] be the value of the variable, NSEG be the number of segments in the strata, and VM[I]=V[I]/NSEG be the mean value. The matrix SXX[I,J] is sum(V[I]*V[J])-VM[I]*VJ[J]*NSEG/(NSEG-1), where sum is over all segments in a strata. SIGXX is the inverse of SXX. Both SXX and SIGXX are symmetric matrices. Then, the remainder of the file for multi-variable regression is:

array[1..NSTRAT] of record	
array [1..(NVAR+1)*(NVAR+2)/2] of real	SXX, upper triangular portion.
array [0..NVAR] of real	B[I], the regression coefficients.

end record.

STANDARD NAMING CONVENTIONS

name:{anything}.ESTS

ESTIMATOR RESULTS FILE (type 60)

The estimator results file contains the results of large-scale estimation, the estimates and variances as well as other information.
The estimator results file is used as an input to the accumulate estimates program.

FILE CONTENTS

The estimator results file consists of a ten-word header followed by the body of the file.

Header

file type
units (acres)
type of estimator
number of independent variables
type of size
augmentation indicator

Body

state and year indicator
number of analysis districts
analysis district list
number of counties
list of counties
number of strata
strata numbers
group indicator
independent variable description
number of covers
cover numbers
number of categories
list of categories
estimator parameter file name
first aggregation file name
strata
estimate
variance
large scale independent variable values
regression coefficients
segment numbers

FILE FORMAT

Header

The format of the header is:

Type	Value	Description
word	(0)	file type, first word.
word	(60)	file type, second word.
word	(0..1)	units, 0 = hectares and 1 = acres.
word	(0..1)	type of estimator, 0 = multi-variable regression and 1 = single-variable regression.
word	(1..10)	number of independent variables, NVAR.
word	(0..4)	type of size,0 = field size,1 = planted size, 2 = harvested size, 3 = digitized size.
word	(0..1)	augmentation indicator, AUGMENT, = 1 if file augmented with regression coefficients and large-scale and small-scale variable values. Set to zero if this data is not present in the file.
array [1..3] of word	(0)	reserved for expansion, explicitly set to zero.

Body

The body of the file is as follows:

Name	Type	Value [length]	Description
	array [1..4] of byte		state and year identifier, set to spaces if no state and year.
	word	(1..)	number of analysis districts.
	word	(1..)	number of bytes in analysis district list, NBA.
	array [1..NBA] of byte		analysis district list. Each analysis district name is preceded by one byte containing the length of the analysis district name in bytes, exclusive of the length byte. Each analysis district name immediately follows the one before it and thus may not start on a word boundary.
	word	(1..)	number of counties.
	word	(1..)	total number of bytes in county list, NBC.
	array [1..NBC] of byte		county list. Each county name is preceded by one bytecontaining the length in bytes of the county name,exclusive of the length byte. Each county immediately follows the one before it and thus does not necessarily start on a word boundary.
	word	(1..20)	number of strata, NSTRAT.
	array [1..NSTRAT] of word	(1..100)	strata numbers.
	word	(-20..0)	group indictor, NGROUP. If the combined estimate is used, ABS(NGROUP) is the number of groups. Otherwise, NGROUP is set to zero.

		If NGROUP is non-zero, indicating use of the combined estimate, the following appears next in the file:
array [1..NSTRATA] of word	(-20..0)	group for each strata, zero if the strata is not amember of any group.

In all cases, the body of the file continues:

array [1..NVAR] of record		independent variable description.
word	(1..10)	number of covers, NCOV.
array [1..NCOV] of word	(0..255)	cover numbers, each an index into the PEDITOR standard crops file.
word	(1..254)	number of categories, NCAT.
array [1..NCAT] of word		list of the categories used from the aggregation file for this variable.
end record.		
word	(1..128)	length in bytes of estimator parameter file name,NLE.
array [1..NLE] of byte		estimator parameter file name.
word	(1..128)	length in bytes of (first)aggregation file name, NLA.
array [1..NLA] of byte		aggregation file name.
word	(1..)	number of estimator results, NRES.
array [1..NRES] of record		estimator results.
word	(-20.100)	strata or group number, always non-zero. Negative value is group number as from combined estimate, and positive value is strata number.
real		estimate.
real		variance.

If AUGMENT = 1, the record contains the following:

array [1..NVAR] of real	large-scale independent variable values (X[I]) from aggregation.
array [0..NVAR] of real	regression coefficients, B[I].
end record.	

If AUGMENT = 1, the file contains the following:

word	(1..500)	NSEG, number of segments.
array [1..NSEG] of word		segment numbers.
array [0..NVAR] of array [1..NSEG] of real		small-scale variable values for each segment and variable, with 0 being the dependent variable (y) and 1..NVAR begin the independent variables (x[i]).

STANDARD NAMING CONVENTIONS

name:{anything}.ESTS

FILLER FILE (type 62)

The FILLER FILE contains the left and right filler for each row ofa scene. Its primary use is for multitemporal scenes in which the filler determination is not straightforward.

FILE CONTENTS

word#	contents
0	=0
1	=62 file type
2-5	window coordinates, N,W,S,E, each as a 32 bit word.
6-11	reserved for expansion, explicitly set to zero.
12-19	header information, the scene identifiers, stored as ASCII characters, each character stored in one 8-bit byte.

The remainder of the file has the filler data for each row in one 32-bit word. The leftmost 16 bits are the left filler and the rightmost 16 bits are the right filler.

FRAME UNIT FILE (type 50)

The frame unit file is a file listing the number of frame units by county and strata and also the strata factor for each stratum. The frame units are the total number of possible segments; the actual segments chosen for more thorough analysis are selected from among the frame units. The strata factor is used in estimation to adjust for segment sizes in the various strata.

FILE CONTENTS

At least one record exists for each county containing the frame units for each strata within the county. When a Landsat scene splits a county, a record will also exit for each part of a county covered by a different scene.

Header

File type
Number of counties

Body

County name
Analysis district name
Number of strata
Strata name
Number of frame units

FILE FORMAT

The file consists of 32-bit words with an eleven word header followed by the data.

Header

The header has the following format:

Name	Type	Value [length]	Description
zero :	word	(0)	
file type :	word	(50)	frame unit file type
ncty :	word	(1..)	number of counties
nsf :	word	(1..)	number of strata factors
zero :	word	[1..7]	7 words reserved for expansion

Each strata factor entry occupies two words.

sfe :	record		
sfnum :	word	(0..99)	strata number
sfact :	real	(0..1000.0)	strata factor
	end	[1..nsf]	strata factor entries

GROUND TRUTH FILE (type 52)

The ground truth file contains the ground data information for a single segment. The data was obtained as part of the Ground Survey. Following a field level edit on ground truth files are created for each segment containing the field level data.

FORMAT CONTENTS

Header

File type
Segment number
State/year
Number of fields/uses
Units used (acres)

Body

For each Tract/field/use the following information is available.
Cover
Field size
Planted size
Harvested size
Other (waste) size
Intended use
Note
Irrigation

FILE FORMAT

Header

Type	Value [length]	Description
word	[1] (0)	
word	[2] (52)	52 = file type
word	[3]	segment number
array [1..4] of byte		state and year, the two character postal state abbreviation followed by the last two digits of the year.
word	[5]	number of elements, currently set to 8.
word	[6]	number of fields
word	[7]	number of uses
word	[8]	units, 1 = acres and 0 = hectares.
array [1..7] of word		reserved for expansion, currently set to zero.

Body

The body of the file consists of the information about each field as follows:

Type	Value [length]	Description
array [1..number of fields] of record		
array[1..4] of byte		tract
word		field number
array [1..number of uses] of record		
word	[1]	cover
word	[2]	field size
word	[3]	planted size
word	[4]	harvested size
word	[5]	other (waste) size
word	[6]	intended use
word	[7]	note
word	[8]	irrigation
end		
end		

The cover, intended use, note, and irrigation codes are integer values which are indices to the names of the items in a special file containing all the names. The sizes are in tenths of a unit, acres or hectares. That is, they are integer vaklues representing the true size multiplied by 10.

STANDARD NAMING CONVENTIONS

name:{segment number}.GTRUTH._{ssyy}

IMAGE CALIBRATION FILE

The image calibration file is created from digitized control points that have been processed by least squared analysis. It contains the information necessary to transform latitude and longitude to row and column and viceversa. The bulk transform coefficients are also included (or set to zero if the image has not been skew corrected).
The 30-40 digitized control points are created during registration of Landsat data to maps. The control points themselves are stored in a PCP file.

FILE CONTENTS

This is an ASCII file which contains the following information:
- Skew coefficients
- Frame #
- Calibration information
- Coefficients of the four polynomials to determine latitude, longitude, or row, column.

The basic unit of information in the file is the line, a collection of ASCII characters terminating ina carriage return. Each line consists of four numbers with each ofthe first three numbers followed by a comma. The numbers may or maynot have decimal points. Comment lines are not allowed.

FILE FORMAT

Name	Type	Description
a,b,c,d :	integer	a and b are skew coefficients,c and d are the frame number with c having 4 or 5 digits, and d having 5 digits
rt,rn,rk,0 :	real	initial image calibration for row, row = rt*latitude + rn*longitude + rk, where row is rounded to an integer
ct,cn,ck,0 :	real	initial image calibration for column, column=ct*latitude + cn*longitude + ck, where column is rounded to an integer

If no more accurate calibration exists, the file will terminate at this point. Otherwise, there will be additional lines as follows:

n,0,0,0 :	integer	n is the number of terms in the polynomials for more accurate determination of row, column,latitude and longitude
T1,T2,N1,N2 :	real	scaling for latitude and longitude in the polynomials, which use st and sn, st = T1 + T2 * latitude and sn = N1 + N2 * longitude
R1,R2,C1,C2 :	real	scaling for row and column in the polynomials, which use sr and sc, sr = R1 + R2 * row and sc = C1 + C2 * column

The remaining lines consist of the coefficients of the polynomials. There are four polynomials, each with n terms. Two other polynomials determine the row and column from the latitude and longitude and two determine the latitude and from the row and column.

NOTES

The polynomials to determine row and column are in terms of thescaled latitude and longitude, st and sn, and are:

row = a1 + a2*st + a3*sn^2 + a4*st^2 + a5*st*sn + a6*sn^3 +a7*st+ ...

column = b1 + b2*st + b3*sn+ b4*st^2 + b5*st*sn + b6*sn^2+b7*st^3+ ...

The polynomials to determine latitude and longitude are in terms ofthe scaled row and column, sr and sc, and are:

latitude = d1 + d2*sr + d3*sc + d4*sr^2 + d5*sr*sc + d6*sc ^2 +d7*sr ^3+...

longitude = e1 + e2*sr + e3*sc + e4*sr^2 + e5*sr*sc + e6*sc ^2+e7*sr^3+

The coefficients of each polynomial begin a new line and occupy an integral number of lines, padded with zeroes at the end of the last line if the number of coefficients, n, is not divisible by 4.
Let p = (n+3)/4 be the number of lines occupied by the coefficients of each polynomial, then in terms of lines, the contents of theremainder of the file are:

a1,a2,a3,etc : real the "a" coefficients of the row polynomial

b1,b2,b3,etc : real the "b" coefficients of the column polynomial
d1,d2,d3,etc : real the "d" coefficients of the latitude polynomial
e1,e2,e3,etc : real the "e" coefficients of the longitude polynomial

STANDARD NAMING CONVENTIONS

name:{anything}.CAL

LIST OF BLOCKS FILE (type 14)

The list of blocks file is used to give the window coordinates of a list of square blocks which are to be read from tape. The blocks are assumed to be in reasonably straight rows. The use of the blocks is for machine correlation when overlaying images and visual correlation of an image to a map or another image for precision geographical registration.

FILE FORMAT

The file has a six (32-bit, of course) word header followed by a record for each row. All items are of type "word".

FILE CONTENTS

Header

Name	Type	Value	Description
	word	(0)	Zero
	word	(14)	File type
blksize :	word	(32 or 64)	Number of rows or columns in the square blocks
maxblks :	word	(1..340)	Maximum number of blocks per row
maxrows :	word	(32 or 64)	Maximum number of rows per block
nblkrows :	word	(1..340)	Number of rows of blocks

The data describes the blocks containing the bounding image windows.
Starting at the upper left corner of the image, general information on each row of blocks is followed by the bounding windows for each block as the image is scanned from left to right.

Body

Name	Type	Value [length]	Description
rowofblks :	record		
numblks :	word	(1..340)	Number of blocks in this row
maxcol :	word	(1..)	Maximum image column for this row
maxrow :	word	(1..)	Maximum image row for this row
mincol :	word	(1..)	Minimum image column for this row
minrow :	word	(1..)	Minimum image row for this row
eachblk :	record		
toprow :	word		Top row of this block in this row
leftcol :	word		Left column of this block in this row
botrow :	word		Bottom row of this block in this row
rightcol :	word		Right column of this block in this row
	end	[1..numblks]	number of blocks in this row
	end	[1..nblkrows]	number of blocks of rows

PRINCIPAL COMPONENTS FILE (type 66)

Principal Components files contain the mean values of The Principal Components, eigen values, Rotation matrices, class names, crop indices, and number of points for multi-channel data.

FILE CONTENTS

The file consists of 3 parts: a 120 byte header, an array of Principal Components records with one record for each of the classes represented, and an array of words which are the crop indices for the crop names specified by the Principal Components records.
The record size of Principal Components records is variable and is dependent on the number of channels of data. The record size is n**2+2n+7 words, where n is the number of channels. Real values (means, eigen values, rotation matrix elements) are in IEEE floating point format.

FILE FORMAT

Name	Type	Value [length]	Description
	word	(0)	Zero
	word	(66)	File type 66
	word	(0..)	Number of crop name indices in file (0 indicates crop indices not present)
n:	word	(1..)	Number of channels
nc:	word	(1..)	Number of classes
nb:	word	(40..)	Record size of Principal Components records (4(n**2+2n+7) bytes)
id:	byte	[96] (0..127)	ASCII id information (a zero byte terminates the string, actual info is limited to 95 characters)
class:	record		For each class
	byte	[8] (0..127)	Class name, 8 characters ASCII (if less than 8 characters, then left adjusted blank filled)
	word	(0..)	Sequence number from original clustering or 0 if "created" or "pooled"
np:	word	(1..)	Number of points in class
ncr:	word	(0..)	Number of crop names in class
	word	(0..)	Pointer to first crop name index in the crop name index list.
	word		Not used
	real	[n] (0.0..)	Principal component Mean values for n channels in channel order.
	real	[n] (0.0..)	Eigen Values for n channels in channel order.
	real	[nxn] (..)	Rotation matrix see note.
	end	[1..nc]	
names:	word	[sum of ncr for all classes] (0..)	Index to list of crop names stored in the crops file. One word for each crop name.

NOTE

The Rotation matrix is stored such that the columns contain the eigen vector for the corresponding eigen value (covar to P.C>) and the rows contain the Rotation Vector (P.C. to Covar) For example, the following matrix

C11 C21 C31 . . .
C12 C22 C32 . . .
C13 C23 C33 . . .
. . .

post multiplied by

R11 R21 R31 . . .
R12 R22 R32 . . .
R13 R23 R33 . . .
. . .

Gives

E1 0 0 . . .
0 E2 0 . . .
0 0 E3 . . .
. . .

Were E[i] is the eigen value for the i'th channel which is stored as

E1 E2 E3 . . .

Post Multiplying a vector of Raw data Values by the Rotation Matrix results in a set of Principal components Corresponding to The Raw Values. Conversely Pre Multiplying a Vector of Principal components by the Rotation Matrix in a vector of Raw data Values.
Note that the Principal components are in Channel Order. Rows and Columns must be exchanged (Row[i] to Row[1] then Col[i] to col[1]....Row[j] to Row[n] then Col[j] to Col[n];where E[i] is the 1st Principal Component and E[j] is then Last) to get the usual order.

SCAN MASK FILE (type 48) (Not used in MARS PED)

The scan mask file is a file created by the automatic digitizing system . It is NOT the same as the Peditor mask file. The scan mask file is in 'half-pixel' format to allow segment shifting by the ASMA program. The half-pixel format means that each original Landsat pixel is represented by four cells in the mask, so if the segment was 65 by 78 pixels the mask will contain 130 by 156 cells.
The scan mask file contains field boundaries and field numbers, tract information and acreage, and may have several parts. Multi-part scan masks are simple concatenation of several single part files.
The standard name for the file is <segment number>.MSK, e.g. 1002.MSK for segment 1002. As created the file is written in blocks of 32-bit words. Depending on the target machine some formatting may be necessary to get the file into the correct format.

FILE CONTENTS

File type
Segment number
identification of coordinate system
number of fields in the segment
processing information

Each pixel in the mask is identified as
Background pixel
Border pixel
Field pixel with field identification

FILE FORMAT

The format of each part is one 32-word header block, followed by the field information blocks, and finally the mask data blocks, with the data stored in run length encoded form.

Name	Type	Value [length]	Description
	word	(0)	1st word is zero
	word	(48)	file type code
	word	(0..9999)	segment number
ir:	word		1st row number, relative to coordinate system being used, usually Landsat row number
	word		1st column number, as above
lr:	word		last row
	word		last column
nflds:	word	(>0)	number of fields in the segment
	word	(>0)	current part
	word	(>0)	number of parts
	word	(0..99)	year less 1900, e.g. 78 for 1978
	word		right-most 16 bits is time in minutes used for scanning, next 16 bits is julien day of scan
	word		right-most 16 bits is time in minutes used for thinning, next 16 bits is julien day of thinning
	word		right most 16 bits is time to label, next 16 bits is day of labeling

	word		right most 16 bits is time to create mask, next 16 bits is day of creation
	word	(1..9999)	frame name, 1st part
	word	(-99999..-1)	frame name, 2nd part, always negative
	word	[15]	not used
field:	record		
	word	(>0)	size of field in acres, to nearest acre
	byte	[2]	tract name, 2 ascii bytes
	half	(>99)	field # times 100, with subfield (if any) added in
	end	[nflds]	
	word	[some]	not used - fill to reach end of 32-word block
mskdta:	record		
	byte		field length. If length is >128 then the associated cells are boundary and the true length is 128 less than the length stored.
	byte		field number. Both length and number of zero flags the end of a row.
	end	[until lr-ir+1 'end-of-rows' have been seen]	
	word	[some]	not used - fill to reach end of 32-word block

STANDARD NAMING CONVENTIONS

PDP-11 XXXX.MSK
MMDS SSYY.VMASK.FXXXXXX.FYYYYY.DATA(A####) where
XXXX is segment number
SSYY is state/year
FXXXXX.FYYYYY is frame id
is segment number

SEGMENT / STRATA AGGREGATION FILE (type 24 / 65)

Segment Aggregation File is File Type 24.
Strata Aggregation File is File Type 65.

The strata aggregation file contains the number of pixels in each category for each field in an image. The values are obtained by aggregating a categorized image window file with a mask file.
The file has two sections: a 640-word header, and a data section whose size is a function of the number of categories and the number of fields in the image. Most of the header data are transferred intact from the mask file that was used to create the aggregation file.

FILE CONTENTS

Header

file type
aggregation coordinates
number of categories
Landsat scene date
sampling indicator
frame id
segment number
state and year identifier
number of fields
area
tract number
field number
ascii header from the input window file

Body

Pixels in each category in each field

FILE FORMAT

Header

The header data are arranged as follows:

Name	Type	Value [length]	Description
	word	(0)	unused
fileid.	word	(24 OR 65)	indicates type of file
n,w,s,e	word	[4]	aggregation coordinates
numcats	word	(1..255)	# of categories
landate	word		landsat scene date;from mask file
sample	word		sampling indicator if 0, no sampling; if not 0, then bits 0-11: start row of sample bits 12-23:start column bits 24-27:row sample increment

			bits 28-31:column sample increment
frame	byte	[12]	ASCII frame id; from mask file
segment	word	(0..)	segment number;0 if no segment number
STYR	byte	[4]	ASCII state and year; from mask file
nfields	word	(1..255)	number of fields used

Body

for each field: area, tract names, and field and subfield descriptions from mask file record

Name	Type	Value [length]	Description
area	word	(1..)	area
IF fileid is 24			
tract	byte	[2]	tract; ASCII charcters, inrange 'A ' to 'ZZ'.The tract names are in the first 14 bits of these 2 bytes; the first character is in the leftmost 7 bits, and the second in the next 7. The remaining bits are zeroes.
field	half	(100..9999)	field number; this value will be decoded as a word value by the program which reads thefile
ELSE			
strata number	byte	[2]	Strata number, in binary within the two bytes.
count unit	byte	[2] (1..)	Count unit, which is NOT adjusted by a factor.
ENDIF			
end		nfields	
	word	[595 - 2*nfields]	undefined
header	byte	[120]	ascii header information; from input window file. The header text is left justified, and blank-padded if necessary

The aggregation data are arranged as follows:

record			
	record		
	word	[numcats]	pixels in each category for a paricular field
	end	nfields	
end			

STANDARD NAMING CONVENTIONS

No standard names.

SEGMENT / STRATA MASK FILE (type 17 / 64)

Segment Mask File is File Type 17.
Strata Mask File is File Type 64.

A mask file is a rasterized version of a network file. It contains the number of pixels for each field in each raster scan line and their corresponding location.
Editor segment masks can be created from either segment network files in manual digitization states or video masks in video digitization states. In both cases, the file of segment shifts is also used to adjust the coordinate system to identify each pixel within the mask. The segment mask file is a shifted, single pixel format, raster representation of the field and boundary information for a segment. That is, the file contains field or boundary information for each pixel within the segments and the coordinate information needed to match it with the Landsat pixel data.
Editor county masks are the same as editor segment masks except that the area represented is actually a county and the tracts and fields represent strata and substrata areas. The file is in raster format containing border, field and coordinate information for each pixel within the county to allow it to be matched with Landsat pixel data.

FILE CONTENTS

Header

File type
Window coordinates
Segment number
Landsat frame name
State year
Number of fields

Body

For each field:
Field number
Tract letter
Acres
Cover identifier
Each pixel in the mask is identified as:
Background pixel
Border pixel
Field pixel with field identification

FILE FORMAT

Header

Name	Type	Value [length]	Description
	word	(0)	
	word	(17 OR 64)	File type
n,w,s,e :	word	[4]	Window coordinates
segnum :	word	(0..9999)	Segment number