

CAD MEMO 5

**NATIONAL RADIO
ASTRONOMY OBSERVATORY
Green Bank, WV**

**Using AutoCAD for
PC Board Layouts
April 29, 1986**

**PCBOARD.MNU
Version 1.3**

Richard F. Bradley

PROPERTY OF THE U. S. GOVERNMENT
RADIO ASTRONOMY OBSERVATORY
CHARLOTTESVILLE, VA.

JUN 16 1986

TABLE OF CONTENTS

PRINTED CIRCUIT BOARD LAYOUT MENU FOR AutoCAD	2
PURPOSE OF A PC MENU FOR AutoCAD	2
INSTRUCTIONS FOR BEGINNERS	3
LAYER COLORS	4
PC MENU FEATURES	5
ROOT MENU AND SUB-MENU DESCRIPTION	5
PC BOARD INTRODUCTION MENU	5
ROOT MENU	6
LAYER SELECTION MENU	6
COMPONENT OUTLINE MENU	7
TRACE SELECTION MENU	8
SPECIAL PADS MENU	8
DONUT SELECTION MENU	9
CHIP PAD SELECTION MENU	9
SOLID PAD MENU	10
SOLID CIRCLE PAD MENU	10
SOLID SQUARE PAD MENU	10
SQUARE PAD MENU	11
RECTANGULAR PAD MENU	11
HEX/OVAL PAD MENU	12
TEXT STYLE MENU	12
GENERATING NEW PATTERNS	13
PATTERN DEFINITION	13
INSERTION	13
PERMANENT PATTERN STORAGE	13
PLOTTER SETUP NOTES	14
PLOTTER MECHANICAL CONFIGURATION	14
PEN TYPE	14
AUTOCAD LAYER COLOR AND PEN COLOR	14
PLOTTER SETUP CHECKLIST	15
TECHNICAL DISCUSSION OF THE ADDITIONS TO CADADV3.MNU	16
ACKNOWLEDGEMENTS	17
REFERENCE	17

PRINTED CIRCUIT BOARD LAYOUT MENU FOR AutoCAD

A prototype menu configuration is currently being developed for AutoCAD. Using Cadillac Advanced menu features, the proposed scheme uses a hierarchical arrangement of menu options to enable the user to quickly and efficiently access stored block drawings. The menu format permits easy expansion without unnecessary screen clutter. This scheme proves useful for easy access to a large quantity of blocks encountered in such drawings as printed circuit (PC) board layouts and schematic diagrams. A PC board layout menu using the proposed features is currently being developed for the AutoCAD system in Green Bank. This document serves as a users guide for the PC Board menu as well as a discussion of the important features in developing a menu driven block access scheme for AutoCAD.

PURPOSE OF A PC MENU FOR AutoCAD

Using the AutoCAD and Cadillac Advanced programs as a stand alone PC board layout tool is quite cumbersome to use and is not practical for anything but the most simplistic PC drawings. Drawing PC component pads and lands is quite a chore even to an avid AutoCAD user. The obvious answer to the complexity problem is the generation of blocks for all pads, traces and special items such as targets and datums. The resulting library of blocks becomes alarming and trying to name each block with a simple yet descriptive character set is just short of impossible. In addition, a PC layout is on many layers and there is a risk of accidental placement of an object on the wrong layer. The solution is a menu driven scheme which allows easy access to ALL blocks, permits easy future expansion, and encourages non-AutoCAD experts to make use of the system.

INSTRUCTIONS FOR BEGINNERS

The first step in learning to use the PC Layout menu is to learn some of the very basic features of AutoCAD. Also learn how to use the Digitizer Pad, mouse, and menus.

Initial start-up of the AutoCAD program is as follows:

1. Turn Computer ON
2. Wait for prompt C:\>
3. Type AUTOCAD and press RETURN
4. Select 1 or 2 from the list
5. Type name of drawing (8 characters maximum)

The following is a list of AutoCAD commands that should be understood before attempting to use the PC Board menu features.

LINE: how to draw a simple line.
TRACE: how to change the width of a trace.
GRIDSET: how to setup a grid of any desired spacing.
GRID: turn the grid ON or OFF
SNAPSET: how to set the resolution of the mouse.
SNAP: turn the SNAP feature ON or OFF
ORTHO: set up AUTOCAD for orthogonal lines only.
ZOOM: all aspects of the ZOOM command.
REDRAW: removes unwanted pick marks.
ERASE: all aspects of the ERASE command.
UN-ERASE: erase and erasure.
LAYERS: new, set, freeze, thaw, off, and on.
COORDS: toggle the coordinates OFF and ON.
CANCEL: a way to get out of an unwanted command
TEXT: mostly block text will be needed for a PC board.
SOLID: for making large lands
PAN: moving around on the drawing
FILL: toggle the FILL mode OFF and ON
MOVE: change the position of an object
MIRROR: for a mirror image of an object or drawing.
PLOT: setting up the plotter (see section on Plotting notes)

The following procedure is suggested for setup and layout of a PC board using AutoCAD.

1. Run the AutoCAD program.
2. Using the yellow (PICK) button on the mouse, choose PC BOARD from the Cadillac Advance Menu or the MERGE1 menu.
3. Enter the PCB ROOT menu from the INTRODUCTION MENU by selecting PCB ROOT with the mouse.
4. Select SETUP from the PCB ROOT menu. This action will initialize all the layers used by the PC Board Menu system.
5. Select OUTLINE from the PCB ROOT menu. Note that the board layout is in 1:1 scale. Draw a LINE around the perimeter of the board. If it turns out that more board space is needed, this border can be easily changed.
6. Lay out the chips by picking the outlines off the OUTLINE MENU. Insertion point for a chip outline is at the upper left corner of the outline (exactly at pin 1). Next lay out the smaller components.
7. Return to the PCB ROOT menu and select the FRONT or BACK of the board to begin placing pads. To increase the speed at which the computer can redraw a pattern, turn the FILL OFF and/or use DIP patterns for the IC chips instead of the OVAL patterns. Select desired pads from the PCB ROOT menu. REMEMBER TO SELECT WHICH SIDE OF THE BOARD YOU ARE CURRENTLY WORKING BEFORE SELECTING PADS AND TRACES.
8. After all pads are in place, select TRACES from the PCB ROOT menu. Begin connecting the pads. Remember FRONT and BACK.
9. After the traces are in place, the next item is text. Choose FRONT or BACK and TEXT from the PCB ROOT menu. Note that text of the back side of the board must be in reverse. Solid letters are recommended for PC boards.
10. Finally, solid the areas where ground plane is to exist. First PICK the FRONT or BACK of the board from the PCB ROOT menu. Then PICK SOLIDS from the PCB ROOT menu and AREA from the SOLID PAD menu. The AREA command works exactly like the AutoCAD SOLID command.
11. The drawing is complete and ready to PLOT.

LAYER COLORS

The following layers are generated when initializing the PC board layout drawing using SETUP from the PCB ROOT menu.

TRACE-F	: YELLOW	Front side traces
TRACE-B	: CYAN	Back side traces
PADS-F	: RED	Front side pads
PADS-B	: GREEN	Back side pads
OUT	: WHITE	Board component outline
TEXT-F	: MAGENTA	Front side text
TEXT-B	: BLUE	Back side text
DRILL	: WHITE	Drill location notes
NOTES	: WHITE	General notes

PC MENU FEATURES

The following list of features is used in the PC layout menu. Although tailored to the needs of PC layout, the ideas can be used for schematic diagrams and other drawings.

1. A root menu allows the user to pick the type of pad, trace, etc. and the current working side of the PC board.
2. Sub-menus are used to list the style of pads available. For example, a donut pad sub-menu contains a list of available donuts in the range of OD = .050" to .150".
3. When choosing a particular pad, trace, or text, the drawing will change to the corresponding layer before block placement.
4. The BLUE mouse button is programmed to permit quick generation of additional pads upon initial selection from the menu.
5. The LAYERS menu for setting a layer quickly. This feature is for over-riding the automatic layer selection and to access the layer 0 for new block generation.
6. A space on the digitizer pad is programmed for returning to the PC board menu.
7. The new menu configuration is in ADDITION to the cadillac Advanced menu features and does not alter them in any way.

ROOT MENU AND SUB-MENU DESCRIPTION

The following is a description of all the PC board layout menus incorporated into CADADV3.MNU. For a discussion of the menu coding see section on Technical Description of PC Board Cadillac Advance Menu Features.

PC BOARD INTRODUCTION MENU

Access: by choosing PC BOARD from the CADILLAC ADVANCED menu.

Menu Options:

PCB ROOT: advance to PC Board Root Menu

The Introduction menu is used for the version number and important update information. Always glance over this menu area and note any changes that have occurred.

ROOT MENU

Access: by choosing PCB ROOT from the INTRODUCTION MENU.

Use: Pick menu item from list using PICK button on the mouse.

Menu Options:

SET UP: generates a complete set of special layers for a new drawing. This includes setting layer colors.

FRONT: routes pads, traces, and text for working on the Front of a PC board.

BACK: routes pads, traces, and text for working on the Back of a PC board.

LAYERS: advance to LAYER SELECTION MENU.

OUTLINE: advance to COMPONENT OUTLINE MENU.

TRACE: advance to TRACE SELECTION MENU

SPECIAL: advance to SPECIAL PADS MENU (TARGET and DATUM)

DONUTS: advance to DONUT SELECTION MENU

CHIPS: advance to CHIP PAD SELECTION MENU

SOLIDS: advance to SOLID PAD MENU (CIRCLES and SQUARES)

SQUARES: advance to SQUARE PAD MENU

RECTS: advance to RECTANGULAR PAD MENU

HEX/OVAL: advance to HEX/OVAL PADS MENU

TEXT: advance to TEXT STYLE MENU

NEW PAD: routes pads, traces, and text for working on layer 0.

LAYER SELECTION MENU

Access: by choosing LAYERS from PCB ROOT menu

Menu Options:

TRACE F: set Layer to TRACE-F (Front trace)

TRACE B: set Layer to TRACE-B (Back trace)

PADS F: set Layer to PADS-F (Front pads)

PADS B: set Layer to PADS-B (Back pads)

OUTLINES: set Layer to OUT (component Outlines)

TEXT F: set Layer to TEXT-F (Front Text)

TEXT B: set Layer to TEXT-B (Back Text)

DRILL: set Layer to DRILL (hole drilling notes)

NOTES: set Layer to NOTES (general notes)

NEW PAD: set Layer to LAYER 0 (for making new blocks)

PCB ROOT: return to PCB ROOT menu

COMPONENT OUTLINE MENU

Access: by choosing OUTLINE from PCB ROOT menu

Auto-Placement Layer: OUT

Features: DRAG for component placement and rotation.

- Use:
1. Pick desired component from menu.
 2. Place cross-hairs at desired location (Note DRAG feature)
 3. Press PICK Button.
 4. Rotate component around reference point (note DRAG feature).
 5. Press PICK Button for placement.
 6. For additional copies of the same component simply use cross-hairs to locate position, then press the BLUE Button on the mouse.

Menu Options:

- RES 1/8W: rectangular outline x=.210" y=.060" Approximate size of a 1/8 W resistor. Reference point at left wire location.
- RES 1/4W: rectangular outline x=.340" y=.092" Approximate size of a 1/4 W resistor. Reference point at left wire location.
- RES 1/2W: rectangular outline x=.490" y=.138" Approximate size of a 1/2 W resistor. Reference point at left wire location.
- RES 1 W : rectangular outline x=.723" y=.226" Approximate size of a 1 W resistor. Reference point at left wire location.
- CAP L : rectangular outline x=.830" y=.272" Considered a large capacitor with reference at left wire location.
- CAP M : rectangular outline x=.244" y=.103" Considered a medium capacitor with reference to left wire location.
- CAP S : rectangular outline x=.120" y=.063" Considered a small capacitor with reference to left wire location.
- CHIP 8 : square outline of hole perimeter for an 8 pin DIP x=.3" y=.3". Reference at pin 1.
- CHIP 14 : rectangular outline of hole perimeter for a 14 pin DIP x=.3" y=.6". Reference at pin 1.
- CHIP 16 : rectangular outline of hole perimeter for a 16 pin DIP x=.3" y=.7" Reference at pin 1.
- CHIP 20 : rectangular outline of hole perimeter for a 20 pin DIP x=.3" y=.9" Reference at pin 1.
- CHIP 24 : rectangular outline of hole perimeter for a 24 pin DIP x=.6" y=1.1" Reference at pin 1.
- CHIP 28 : rectangular outline of hole perimeter for a 28 pin DIP x=.6" y=1.3" Reference at pin 1.
- CHIP 40 : rectangular outline of hole perimeter for a 40 pin DIP x=.6" y=1.9" Reference at pin 1.
- PCB ROOT : return to PCB ROOT menu

TRACE SELECTION MENU

Access: by choosing TRACE from PCB ROOT menu

Auto-Placement Layer: TRACE-F or TRACE-B depending on current working side of PC board.

- Use:
1. Pick desired trace width from menu.
 2. Pick starting point of trace by using cross-hairs and then pressing the PICK button on the mouse.
 3. Pick end point of first segment by using the cross-hairs and the PICK button.
 4. Additional segments can be chained end to end by picking additional end points using the cross-hairs and the PICK button. Note that segment drawings lag behind the current segment placement.
 5. When all chained segments are positioned, press RETURN or the WHITE button on the mouse.

Menu Options:

50 OHM: width = 0.108" for a 50 OHM Microstrip on G-10 Glass Epoxy Board using 1 oz. copper and Dielectric thickness is 0.061" tol. ± 0.002 "

.016": width = 0.016"

.031": width = 0.031"

.040": width = 0.041"

.050": width = 0.050"

.075": width = 0.075"

.100": width = 0.100"

.125": width = 0.125"

.150": width = 0.150"

PCB ROOT: return to PCB ROOT menu

SPECIAL PADS MENU

Access: by choosing SPECIAL from the PCB ROOT menu

Auto-Placement Layer: PADS-F or PADS-B depending on current working side of the PC Board.

- Use:
1. Pick desired pattern from menu.
 2. Position cross-hairs over desired location.
 3. Press PICK button. The pattern is now on the drawing
 4. for additional copies of the same pattern, simply position cross-hairs over desired location and press the BLUE button on the mouse.

Menu Options:

DATUM: Used as a reference point for tooling holes.
This Datum has O.D. = 0.2" and I.D. = 0.040".
Reference point is at Datum center.

TARGET: Moire pattern and cross-hairs. Reference point is at center of the pattern.

PCB ROOT: return to PCB ROOT menu

DONUT SELECTION MENU

Access: by choosing DONUTS from the PCB ROOT menu
Auto-Placement Layer: PAD-F or PAD-B depending on current working side of the PC board.

- Use:
1. Pick desired donut size from the menu
 2. Position cross-hairs over desired location and press PICK button. Donut now appears on the drawing.
 3. For additional copies of the same donut, simply position the cross-hairs over the desired location and press the BLUE button on the mouse.

Menu Options:

- 0.050" : donut having OD = 0.050" and ID = 0.015".
Reference to donut center.
 - 0.075" : donut having OD = 0.075" and ID = 0.025".
Reference to donut center.
 - 0.100" : donut having OD = 0.100" and ID = 0.040".
Reference to donut center.
 - 0.125" : donut having OD = 0.125" and ID = 0.040".
Reference to donut center.
 - 0.150" : donut having OD = 0.150" and ID = 0.040".
Reference to donut center.
- PCB ROOT : return to PCB ROOT menu

CHIP PAD SELECTION MENU

Access: by choosing CHIPS from the PCB ROOT menu
Auto-Placement Layer: PADS-F or PADS-B depending on current working side of the PC board.

- Use:
1. Pick desired chip from menu
 2. Position cross-hairs over desired location of pin 1.
 3. Press PICK button.
 4. rotate the chip pattern around pin 1 to desired position by moving the mouse.
 5. press PICK button. The pattern now appears on the drawing.
 6. for additional copies of the same chip, use cross-hairs to locate the position and then press the BLUE button on the mouse.

Menu Options:

- DIP X: X is the number of pins on the DIP (8, 14, 16, 20, 24, 28, 40). These pads have square holes. Hole 1 is a rectangular pattern .050" x .125". The remaining pads are stretched hexagonal 0.050 x 0.125". This chip pattern permits a trace width of 0.016" between adjacent pads on the DIP.
- OVAL X: X is the number of pins on the DIP (8, 14, 16). These pads have circular holes. Hole number 1 is a rectangular pattern 0.070" x 0.125". The remaining pads are oval 0.050" x 0.125". This chip pattern permits a trace width of 0.016" between adjacent pads on the DIP.
- PCB ROOT: Return to ROOT MENU

SOLID PAD MENU

Access: by choosing SOLIDS from the PCB ROOT menu

Use: Pick menu item from list by using the PICK button

Menu Options:

AREA: For making large land regions. Uses SOLID command

CIRCLE: advance to SOLID CIRCLE PAD MENU

SQUARE: advance to SOLID SQUARE PAD MENU

PCB ROOT: Return to PCB ROOT menu

SOLID CIRCLE PAD MENU

Access: by choosing CIRCLE from the SOLID PAD MENU

Auto-Placement Layer: PADS-F or PADS-B depending on current working side of the PC board.

- Use:**
1. Pick desired pattern from menu
 2. Position cross-hairs over desired location
 3. Press PICK button. Pattern appears on drawing.
 4. For additional copies of the same pattern, locate cross-hairs over desired position and press the BLUE button on the mouse.

Menu Options:

0.050" : solid disk of diameter 0.050"

0.075" : solid disk of diameter 0.075"

0.100" : solid disk of diameter 0.100"

0.125" : solid disk of diameter 0.125"

0.150" : solid disk of diameter 0.150"

PCB ROOT: return to the PCB ROOT menu

SOLID SQUARE PAD MENU

Access: by choosing SQUARE from the SOLID PAD MENU

Auto-Placement Layer: PADS-F or PADS-B depending on current working side of the PC board.

- Use:**
1. Pick desired pattern from menu
 2. Position cross-hairs over desired location
 3. Press PICK button. Pattern appears on drawing.
 4. For additional copies of the same pattern, locate cross-hairs over desired position and press the BLUE button on the mouse.

Menu Options:

0.100" : solid square pattern of 0.1" Center Reference

0.150" : solid square pattern of 0.15" Center Reference

0.200" : solid square pattern of 0.2" Center Reference

0.500" : solid square pattern of 0.5" Center Reference

PCB ROOT: Return to PCB ROOT menu

SQUARE PAD MENU

Access: by choosing SQUARES from the PCB ROOT menu
Auto-Placement Layer: PADS-F or PADS-B depending on the current working side of the PC board.

Use: 1. Pick desired pattern from menu
2. Position cross-hairs over desired location
3. Press PICK button. Pattern appears on drawing.
4. For additional copies of the same pattern, locate cross-hairs over desired position and press the BLUE button on the mouse.

Menu Options:

0.100" : square pad with square hole. Reference at center of pattern.

0.150" : 0.150" square pad with square hole. Reference at center of pattern.

0.200" : 0.2" square pad with a square hole. Reference at center of pattern.

PCB ROOT: Return to PCB ROOT menu

RECTANGULAR PAD MENU

Access: by choosing RECTS from the PCB ROOT menu
Auto-Placement Layer: PADS-F or PADS-B depending on the current working side of the PC board.

Use: 1. Pick desired pattern from menu
2. Position cross-hairs over desired location
3. Press PICK button. Pattern appears on drawing.
4. For additional copies of the same pattern, locate cross-hairs over desired position and press the BLUE button on the mouse.

Menu Options:

RECT CIR: Rectangular pad 0.070" x 0.125" with a circular hole 0.015" dia. Reference at center of pattern.

RECT SQR: Rectangular pad 0.050" x 0.125" with a square hole. Reference at center of pattern.

PCB ROOT: Return to the PCB ROOT menu

HEX/OVAL PAD MENU

Access: by choosing HEX/OVAL from the PCB ROOT menu
Auto-Placement Layer: PADS-F or PADS-B depending on the current working side of the PC board.

Use: 1. Pick desired pattern from menu
2. Position cross-hairs over desired location
3. Press PICK button. Pattern appears on drawing.
4. For additional copies of the same pattern, locate cross-hairs over desired position and press the BLUE button on the mouse.

Menu Options:

OVAL: oval pad L=0.125" W=0.070". Circular center hole.
Reference to center of pattern

HEX: stretched hexagonal pattern L=0.125" W=0.070"
Square hole in the center. Reference to center of pattern.

PCB ROOT: return to the PCB ROOT menu

TEXT STYLE MENU

Access: by choosing TEXT from the PCB ROOT menu
Auto-Placement Layer: TEXT-F or TEXT-B depending on the current working side of the PC board.

Use: 1. Pick desired text font from the menu
2. Pick spot on drawing by using the mouse (reference point is the first character).
3. Enter the text height then press RETURN
4. Enter desired rotation angle then press RETURN
5. Type the desired text then press RETURN
6. If necessary, use the MOVE and DRAG commands to move text to exact location.

Menu Options:

STANDARD: For standard block type lettering

ITALICS: For italicized text

SOLID F: Normal type filled-in lettering

SOLID B: Backward type filled-in lettering

PCB ROOT: Return to the PCB ROOT menu

GENERATING NEW PATTERNS

Although an attempt has been made to enable easy access to a wide variety of common printed circuit board design patterns, the user may wish to define a hybrid pattern for use on a particular drawing. If such a pattern is useful for many PC board drawings, the user can store the pattern definition on disk for future use. If the new pattern would make a helpful addition to the menu system please discuss the idea with R. Bradley. The following is a step by step procedure for defining a new pattern, inserting the new pattern on the drawing, and storing the new pattern on the hard disk.

PATTERN DEFINITION

1. Pick NEW PAD from the PCB ROOT menu.
2. Draw the new pattern using any existing pad, trace, and text.
3. Using the mouse, pick BLOCK from the Digitizer pad.
4. Type a name for the new pattern (8 characters maximum).
5. Pick a reference point for the pattern (this point need NOT be on the pattern).
6. Type W then RETURN, then pick a point just outside the lower left hand corner of the new pattern (be sure cross-hair clears the pattern).
7. Make the window around the pattern.
8. Press the pick button and press RETURN. (The pattern is now defined and will disappear from view).

INSERTION

1. Using the mouse, pick INSERT from the Digitizer pad.
2. Type the pattern name and press RETURN.
3. Using the mouse, pick the insertion reference point.
4. DO NOT change the scale so press RETURN.
5. DO NOT change the y factor so press RETURN.
6. Rotate the pattern if needed by entering the angle then press RETURN. Reference is zero degrees in positive x direction. Pattern now appears on the drawing.
7. For additional insertions of the same pattern, locate the insertion point using the cross-hair and then press the blue button on the mouse (Note that rotation is NOT retained).

PERMANENT PATTERN STORAGE

1. Define the pattern as described above.
2. Using the mouse, pick WBLOCK from the Digitizer pad.
3. For file name use block\your pattern name then press RETURN.
4. Type the pattern name and press RETURN.
5. The pattern is now stored on the hard disk.

PLOTTER SETUP NOTES

In an attempt to minimize the confusion encountered upon initial confrontation with the HP plotter, the following is a list of suggestions for plotting PC board drawings.

PLOTTER MECHANICAL CONFIGURATION

The HP plotter is set up with its x axis along the paper travel and its y axis along the pen travel. The maximum plotting area is: x-max = 31 1/2" , y-max = 20 1/2.". Note that the drawing is in 1:1 scale and should be plotted as large as possible.

Maximum board size for 2:1 scale is x = 15.5" , y = 10".

Maximum board size for 4:1 scale is x = 7.5" , y = 5"

PEN TYPE

USE ONLY RED OR BLACK INK FOR PC BOARD DRAWINGS.

Use a "P" type pen (Felt Tip) for drawings on "Plotter Paper". Actual pen width is 0.7mm = 0.0275". Set AutoCAD pen width setting to 0.026". Plotting speed = 20 and pen force = 2. Place the same size pen in positions 1 and 2 of the pen carousel.

Use a "V" type pen (Disposable ink) for Vellum Paper. Two pen sizes are available: 0.3mm = 0.012" and 0.6mm = 0.024". For the 0.3mm pen, set AutoCAD pen width setting to 0.010" and speed = 15. For the 0.6mm pen, AutoCAD pen width = 0.023" and speed = 20. Use the 0.3mm pen for everything except the text. To do this place the 0.3mm pen in position 1 and the 0.6mm pen in position 2 of the pen carousel.

AUTOCAD LAYER COLOR AND PEN COLOR

Be sure the autoCAD layer scheme described on the PLOT command sequence is as follows:

LAYER COLOR	PLOTTER PEN #	LINE TYPE	PEN SPEED	
			P	V
1 RED	1	0	20	15
2 YELLOW	1	0	20	15
3 GREEN	1	0	20	15
4 CYAN	1	0	20	15
5 BLUE	2	0	20	20
6 MAGENTA	2	0	20	20
7 WHITE	2	0	20	20

PLOTTER SETUP CHECKLIST

The following procedure is suggested for the beginner to help get the drawing out of AutoCAD and onto the plotter.

1. Choose the size and type of paper desired for the particular PC board drawing. Plotter paper may be used with success however Vellum is preferred because it will not absorb as much ink as will the plotter paper. Place paper into plotter with shortest side parallel to the pen movement. Position paper so that it drapes both sides of the plotter evenly. Be sure the paper touches both stops located on the left side of the plotter.
2. Position "Top Of Paper" pointer, located on the pen travel arm, to point to the edge of the paper.
3. Press the "Chart Hold" button on the plotter and wait until plotter initialization is complete.
4. Then press the "Remote" button on the plotter. The plotter will size the paper and open the port to the computer.
5. Place the proper pens in locations 1 and 2 of the pen holding carousel as described in PLOTTER SETUP NOTES.
6. Go to the computer and type "PLOT" then press RETURN.
7. Type "W" then RETURN and make a window around the drawing using the mouse.
8. For CHANGE ANYTHING? enter "Y". The computer displays a list of pen information. Be sure that layers 1 through 8 are set to the values listed in PLOTTER SETUP NOTES. If information is ok, enter "N". If something needs changed, type "Y" and then "C x" where x = layer number, then change the desired parameter.
9. Size units must be in inches.
10. plot origin should be 0.0,0.0.
11. Be sure the paper size corresponds to the paper that was loaded into the plotter.
12. Note that the x-axis is along the paper travel and the y-axis is along the pen travel. Does this coordinate frame need to be rotated in order to fit the drawing on the plotter?
13. Set pen width to correspond to the type pen used.
14. Must fill boundaries for the pen width used.
15. It is not necessary to remove hidden lines.
16. Set scale to 1=1, 2=1, or 4=1.
17. If plotter is ready, press RETURN at this point.
18. Ignore the warning and enter "Y".
19. At this point, the plotter should start drawing.
20. When plot is finished, go to the plotter and press "VIEW" and then press "CHART UNLOAD".

TECHNICAL DISCUSSION OF THE ADDITIONS TO CADADV3.MNU

The follow information is a collection of notes that may be useful to anyone wishing to setup a menu system using Cadillac advance as a base. Beginning AutoCAD users should skip this section.

The cadadv3.mnu file can be loaded into a text editor such as EMACS or PEDIT. Note the following:

- ";" is used as a RETURN
- "\" indicates a pause for user input
- "^C" CANCEL command

SETUP OF SCREEN MENU

A screen menu can be set up as follows:

```
**MENU_NAME
***screen
[option 1]command for option 1;
[option 2]command for option 2;
```

There is enough room on the screen for 20 such options. The characters appearing inside the braces will be shown on the screen. Directly to the right of the braces is the command. The following commands illustrate the types of commands added to Cadillac Advance for aiding in PC board design.

BLOCK INSERTION: `^CINSERT DN_50;\;;;`

This command will insert a block named "DN_50". It does not allow the user to scale or rotate the block.

SET CURRENT LAYER: `^CLAYER SET PADS-F;;`

This command will change the current layer to "PADS-F"

GO TO SUB-MENU: `^C$$=PCBOARD`

This command will display the "PCBOARD" root menu.

SET TRACE WIDTH: `^CTRACE 0.040;`

This command is used to pick a trace of 0.040" width.

BLOCK INSERTION WITH DRAG: `^CINSERT C_16;DRAG;\;;DRAG;\`

This command will insert a block named "C_16" and permit the user to DRAG it to the insertion point and then DRAG the rotation. Note that the scale is not changed.

FRONT versus BACK of the board

When changing from one side of the board to another, Three variables are used: PADS, TRS and TXT. For example, to set the PADS variable to PADS-F the following command is issued from the menu:

```
(setq PADS "PADS-F");
```

Then, when a DONUT is picked from the menu, the following command is issued:

```
^CLAYER SET !PADS;;^C$S=DONUT
```

Here, the layer is set the variable PADS = PADS-F and the menu "DONUT" is displayed.

OTHER CHANGES

The other changes to the Cadillac Advance menu is to the mouse button definition. Note that ***BUTTONS appears at the very top of the cadadv3.mnu file. The ";" denotes RETURN for the definition of the white button and ;;\;;; defines the blue or green button as two RETURNS a PICK and three RETURNS. This button is used for inserting the same block again and again once it is defined from the menu.

Under ***TABLET4 the SM-21 area is defined as:

```
^C$S=B $S=PCBOARD
```

This permits the user to get back to the PC board layout root menu from anywhere within AutoCAD.

ACKNOWLEDGEMENTS

I would like to thank Don Stone and Roger Norrod for their helpful suggestions and comments.

REFERENCE

1. *The AutoCAD Drafting Package User Reference*, Autodesk Inc. 1985.
2. *AUTOLISP Programmer's Reference*, Pub. No. 111.001, Autodesk Inc. Dec. 16, 1985.
3. *Bishop Graphics Printed Circuit Drafting Technical Manual and Catalog 107A*, Bishop Graphics, 1985.
4. *Cadillac Advanced User's Manual, Version 2.1x of AutoCAD*, CAD Technologies, 1985.