

USER TUTORIALS
FOR
EDA-3000 & 3100 REV 4.0 SOFTWARE

- o How to Use the Insight Router
- o Designing with Shapes
- o How to Create Imbedded Split Planes

Customer Release

February 16, 1987

DESIGNING A P.C. BOARD - INSIGHT ROUTER

The following is designed to teach the operator how to use the Insight Router, as well as familiarize the operator with the Router controls.

INTRODUCTION

You will enter a project and call up a board that has been placed and assigned. From there, you will prepare the board for routing, and then use the command "Insight Router". Once Insight Router is invoked, it immediately starts router set-up. Please note that to use the Insight Program, there must be an editing terminal logged on to the system.

1. To prepare the P.C. board for routing the following steps must be done:
 - o Router keep-in rectangle on layer 146
 - o Router keep-out rectangles on layers 70-84
 - o Via keep-out rectangles on layer 44
 - o Reverse routing rectangles on layers 221-234
 - o Create physical design rule check file "PDRG-CON"

Once you have prepped your P.C board you are ready to start the Insight Router. Follow these commands:

Select: Router from the last page of the "Loop of 4"

At the terminal keyboard type:

@ Insight

Press: Return or Enter;

```
* * * * *                               * * * * *
*                                                                 *
*   Note: The operator is prohibited from making menu selections *
*           on the Telesis system until the INSIGHT program is   *
*           completed.                                           *
*                                                                 *
* * * * *                               * * * * *
```

The following questions are displayed, one at a time, on the terminal screen. You must answer each question as it is displayed, or type quit to exit, restart to start over.

1. Which router will you be using (RSX or CoRouter)?
2. What are the board dimensions?
3. Would you like to specify all the routing layer directions? (Y or N)
4. Which are your VERTICAL routing layers? (Use 1-14 only) (Please use a space to separate each layer number.)

5. Which are your HORIZONTAL routing layers? (Use 1-14 only) (Please use a space to separate each layer number.)
6. What are your routing layers (use 1-14 only) (Please use a space to separate each layer number.)
7. Which layer is the component side?
8. Which direction is the etch going on the component side (H, V, or N)?
9. Which layer is the solder side?
10. Which direction is the etch going on the solder side (H, V, or N)?
11. Are there finger connectors on the edge of the board (Y or N)?
12. What is the primary component orientation (H, V, or N)?
13. Are the components mostly IC types or discrettes (I or D)?
14. Do any of the components have pins on 4 sides (Y or N)?
15. Do most components have pins on 4 sides (Y or N)?
16. What is the default line width (in MILS)?
17. What is the typical IC pad Y and X value?

Please enter the pad Y value (in MILS).

Please enter the pad X value (in MILS).

18. What is the distance between PIN 1 and PIN 2 (in MILS) of a typical IC (if there are no ICs, enter 100)?
19. What is the line-to-line spacing requirement (in MILS)?
20. Do you wish to SAVE_DWG_NEW_REV_AUTO between multiple executions of PASS 5 (Y or N)?

After a long pause, the system will display the following message:

Analyzing your input data, Please standby!

Once again there is a pause, then the following message will be displayed:

The system will now replace the old ROUTER-CON or
CO-ROUTER-CON text file (if any) with a new one.

Do you want to continue [Y/N]?

When all questions have been answered, and the system has created the new Router-Con text file, the system will display the message:

Command completed!

You may now make selections from the menu as well as edit the new Router-Con file.

Listed below are some considerations when editing the Router-Con file:

- GRID SIZE -

You may specify different grids in the X and Y directions. If you do not specify, it is assumed the X and Y are alike.

```
Modified Format: X_GRID_SIZE N
                  Y_GRID_SIZE N
Standard Format:  _GRID_SIZE N
```

- A Special Signal Line Width -

Are there ground, voltage, or other connections you wish to route at a larger width than the default line width? If so, you must go into the "ROUTER" pages - "Set Net Line Width" with required nets, then edit the ROUTER_CON file.* Add under the "DEF_LINE_WIDTH N" statement:

```
Format: MAX_LINE_WIDTH N.
```

* NOTE: Any menu pick made on the "Set Line Width" menu page, affects the NET-DATA-BASE until deleted. (Refer to "Guide to the Router Menus".)

- Control Via Placement -

You may specify where the vias should be placed, possibly allowing only on the same axis as IC's. If none is specified, vias will be placed anywhere on the Routing grid.

```
Format: P5_VIA_GRID_SIZES XGRID1 YGRID1**
```

- Control Number of Vias -

In addition to controlling where vias are placed, you may control the number of vias in a connection.

```
Format: P5_VIA_LIMITS N1**
```

- Control JOGS in a Line -

The Router will inevitably put more JOGS than the operator desires, therefore to control the JOGS:

```
Format: P1_JOG_LIMIT N
         P2_JOG_LIMIT N
         P5_JOG_LIMIT N1**
```

Keep in mind, any connection with more than the specified number of JOGS is a fail and will not be Routed.

** Account for each pass in Pass 5

If you are now ready to edit the ROUTER-CON file, at the editor type:
@ TEXT (C/R)

A prompt will appear "Enter Name of Text File to be Edited:"
Please type: ROUTER-CON (C/R)

A prompt will follow: you will be editing an existing test file named ROUTER-CON
Rev 1 - Enter Return to continue (C/R)

The file will appear. Items within () are comments that the user has input. Items not in () are the actual ROUTER-CON parameters the system has generated from your input. To move through the file use the arrows on the keyboard. If you wish to edit the file, type as you would on a typewriter. (For more information on text editing refer to users manual, Volume 1, Basics - page 91 through page 104.)

To exit the file, type at the keypad:

PF1 PF7

At the bottom of the screen "command:" will appear. At the keypad type:
EXIT

At the keypad press ENTER

You are now ready to start the Insight Router. At the TELESIS select:
INSIGHT ROUTER

GUIDE TO THE ROUTER MENUS

DONE	WORLD	ROAM	<- MENU	MENU ->	STATUS	DISPLAY	CANCEL
MENU ->							
ROUTER	INPUT DRC RULE	INTERACTIVE PLACEMENT	NET COMPARE				
RATSNEST	DESIGN RULE CHECK	AUTOMATIC PLACEMENT	CREATE BOM REPORT				
CREATE NC DRILL	DELETE LAYER	SHAPES	CREATE NETLIST-REPORT				
CREATE PHOTOPLLOT	CREATE/MERGE DRAWING	RELOCATE DRAWING ORIGIN	CREATE COMPONENT-REPORT				
CURRENT INDEX							

DONE	WORLD	ROAM	<- MENU	MENU ->	STATUS	DISPLAY	CANCEL
ROUTER							
AUTO-ROUTER				INSIGHT-ROUTER			
CURRENT INDEX				PROJECT INDEX			

DONE	WORLD	ROAM	<- MENU	MENU ->	STATUS	DISPLAY	CANCEL
AUTO-ROUTER							
SET ROUTER PARAMS	EDIT CONNECTIONS			EDL TOOLBOX			
RATSNEST	ROUTER SETUP						
LIST ROUTER-LOG				SAVE SAME & CONT		SAVE NEW & CONT	
PRINT ROUTER-LOG						LIST ELEMENT	
ROUTE BOARD	ROUTE BY WINDOW	ROUTE NET	ROUTE PIN PAIR				

DONE	WORLD	ROAM	<- MENU	MENU ->	STATUS	DISPLAY	CANCEL
MENU ->							
SET NO-ROUTE NET	SET NET LINE WIDTH	SET NO VIA ELIM	SET FLOATING CON				
DEL NO-ROUTE NET	DEL NET LINE WIDTH	DEL NO VIA ELIM	DEL FLOATING CON				
LST NO-ROUTE NET	LST NET LINE WIDTH	LST NO VIA ELIM	LST FLOATING CON				
		VIA ELIMINATE					

DONE	WORLD	ROAM	<- MENU	MENU ->	STATUS	DISPLAY	CANCEL
SET ROUTER PARAMS							
ROUTER GRID SIZE	CHANNEL PREFERENCE	SKIP PASS 1	SKIP PASS 2				
ROUTING LAYERS	DEF LINE WIDTH	P1 WINDOW EXPAN	P2 WINDOW EXPAN				
COMP ORIENTATION	MAX LINE WIDTH	P1 PIN KEEP AWAY	P2 PIN KEEP AWAY				
DIAGONAL ALLOWED	TS ALLOWED	P1 TOLERANCE	P2 TOLERANCE				
STATUS MESS FREQ		P1 JOG LIMIT	P2 JOG LIMIT				

DONE	WORLD	ROAM	<- MENU	MENU ->	STATUS	DISPLAY	CANCEL
MENU ->							
NUM P5 EXECUTES	P5 WINDOW EXPAN	P5 ROUTER TYPES					
	P5 LAYER PAIRS	P5 JOG LIMITS					
	P5 PIN KEEP AWAY	P5 VIA LIMITS					
	P5 JOG SIZES			RE-RATSNEST			
	P5 VIA ALLOWED	P5 VIA GRIDS		FAILURE LIST			

DONE	WORLD	ROAM	<- MENU	MENU ->	STATUS	DISPLAY	CANCEL
EDIT CONNECTIONS							
SET LAYER PAIR	HIGHLIGHT NET	DE-HIGHLIGHT NET	ADD CONNECTION				
DELETE ELEMENT	HIGHLIGHT NET NUM	SET TRAP SIZE	SET LINE WIDTH				
DELETE VERTEX	MOVE VERTEX	INSERT VERTEX	SET NECK WIDTH				
DELETE SEGMENT	MOVE SEGMENT	ZOOM RATIO	CHANGE				
DEL NET SECTION	MOVE SYMBOL	ROTATE SYMBOL	LIST ELEMENT				

DONE	WORLD	ROAM	<- MENU	MENU ->	STATUS	DISPLAY	CANCEL
CHANGE							
CHANGE LAYER	CHANGE WIDTH	CHANGE LAYR & WIDTH	ADD CONNECTION				
CHANGE SEG LAYER	DELETE ELEMENT	SET LAYER PAIR	SET LINE WIDTH				
MOVE SECTION	MOVE VERTEX	INSERT VERTEX	SET NECK WIDTH				
DELETE SEGMENT	MOVE SEGMENT						
DEL NET SECTION	MOVE SYMBOL			LIST ELEMENT			

TELESIS
PRINTED CIRCUIT BOARD DESIGN

SHAPES

1. Once you have finished routing your board, and you have run netcompare and PDRC to check your design, you can add shapes to your drawing. The SHAPES program gives you the capability to add any shape etch area to your drawing. For the sake of clarity, it is recommended that you add a shape to a different DB Layer than the one that the signal etch is on. Set up a standard on a series of unreserved layers. For example, if you want to add a shape to the component side etch (layer 1), put the shape on DB Layer 11. If you want to add a shape to the solder side (layer 2), put it on DB Layer 12, and so on. The corresponding layers can be merged together at artwork time. Before using the SHAPES program, you must have an APERTURE-TAB file in your current project or in the SYSTEM-LIBRARY. This is so the system knows which apertures are available to fill in the shape.

Referencing Figure 1, follow these commands:

OLD PROJECT PROJECT NAME INPUT NAME [CR]

DRAWINGS/SYMBOLS

BOARD DRAWING

OLD DRAWING DRAWING NAME INPUT NAME [CR]

DESIGN BOARD

<- MENU

SHAPES

SET ACTIVE LAYER 11 [CR]

LINE LOCK OFF

ADD SHAPE

Now add a shape of your choice. The shape must end at the same X,Y location that it was started at.

2. You can now add voids to your shape. (See Figure 2.)

ADD VOID SHAPE P1 P2 P3 P4 P5 [F7] (or ENTER)

ADD VOID CIRCLE P6 [F4] (or KEYPAD) INC X 100 ENTER

You now have a choice on how to fill the shape. You may fill solid or crosshatch. By choosing crosshatch there are some parameters that must be set.

XHATCH PARMS

The parameters are of your choice, however if you choose a crosshatch line width that is not in the aperture table, an error message will appear: "CROSSHATCH LINE WIDTH NOT FOUND IN APERTURE-TAB".

<u>XHATCH LIN WIDTH</u>	<u>INPUT VALUE</u>	[C/R]
<u>XHATCH LIN SPACE</u>	<u>INPUT VALUE</u>	[C/R]

Pick one of the following:

<u>HORZ XHATCH</u>	<u>VERT XHATCH</u>
<u>/// XHATCH</u>	<u>XHATCH</u>
<u>HRZ & VERT XHATCH</u>	<u>XXX XHATCH</u>

DONE

FILL CROSSHATCH

The shape will start to fill. If the shape or the void shape is too small to accommodate your smallest aperture, an error message will appear, and a butterfly appears on your drawing notating the area of the error. To correct an error hit:

EDIT SHAPE P1 (pick the shape) [F7] (or ENTER)

Now edit the shape or void shape using the commands on that menu page.

3. Now, set your active layer to 12, and add a shape for the solder side of your drawing.

When you are finished,

DONE

DONE

SAVE DRW SAME REV

DONE	WORLD	ROAM			STATUS	DISPLAY	
SHAPES				PROCESSING APERTURE-TAB FILE			
	SET ACTIVE LAYER		SET TRAP SIZE				
ADD SHAPE				EDIT SHAPE			
				ADD SHAPE TO NET			
LINE LOCK OFF			DELETE ELEMENT				
LINE LOCK 45/ON	LINE LOCK 90/ON			LIST ELEMENT			

	WORLD	ROAM			STATUS	DISPLAY	CANCEL
ADD VOID SHAPE	ADD VOID CIRCLE				SET TRAP SIZE		
DELETE ELEMENT			XHATCH PARAMS				
DELETE VERTEX	MOVE VERTEX	INSERT VERTEX		FILL CROSSHATCH			
DELETE SEGMENT	MOVE SEGMENT				FILL SOLID		
	MOVE SECTION				LIST ELEMENT		

DONE	WORLD	ROAM		LIST	STATUS	DISPLAY	
XHATCH PARAMS							
XHATCH LIN WIDTH				XHATCH LIN SPACE			
HORZ XHATCH		VERT XHATCH		HRZ & VRT XHATCH			
/// XHATCH		\\ \\ XHATCH		XXX XHATCH			

SPLIT PLANES

OBJECTIVE

The following lab is designed to teach the operator how to create split imbedded planes.

INTRODUCTION

Generally when it is determined that a P.C. board must have a split plane, a great deal of forethought was given to placement. The reason being, to allow an easily divided split plane.

In this lab you will enter a project and call up a P.C. board that has been fully placed and assigned. At that point an imbedded/split plane layer should be determined and the lines of demarcation added. (Demarcation is determined by highlighting individual nets.) Once complete, the parameter files should be created and photoplot run. The last consideration: PENPLOT ARTWORK, the penplots represent the eventual artwork, this step should not be overlooked. Finally, the art files go to tape and are shipped to the vendor.

LAB SESSION

The following session is a step by step process on how to create a split plane.

```
OLD PROJECT      PROJECT NAME   XYZ   [CR]
DRAWINGS/SYMBOLS
BOARD DRAWINGS
OLD DRAWINGS    DRAWING NAME   PCB   [CR]
DESIGN BOARD
MENU ->
MENU ->
SET LINE WIDTH  50 [CR]
<- MENU
SET ACTIVE LAYER 5 [CR]
SET GRID SIZE   GRID SIZE X 50 [CR] Y 50 [CR]
```

Referring to Figure 1, add a continuous line on top of the board outline, overlapping the start point. The purpose of this line is to allow clearance of the plane from the edge of the P.C. board. By adding the line, it will prevent potential shorts with other layers as well as assembly in a card cage.

```

MENU ->
MENU ->
CR NETLIST RPT
<- MENU
NETLOCK
HIGHLIGHT NET NUMBER NET NUMBER 1 [CR] CANCEL (OR ESC)
DONE
MENU <-

```

Now you are ready to add the dividing line of individual nets. Once again, a reminder to overlap lines in order to prevent shorts. Refer to Figure 2.

```

ADD LINE P1 - Pn ENTER
MENU ->
NETLOCK
DEHIGHLIGHT NET
HIGHLIGHT NET NUMBER NET NUMBER 2 [CR]
CANCEL (OR ESC)
DONE
<- MENU
ADD LINE P1 - Pn ENTER

```

Now you must go back and check separations, insuring the correct nets are in the appropriate sections. To accomplish this you will select "HIGHLIGHT NET".

Once verified, check all the photoplot text files for correct information. Refer to the examples given. Notice the photoplot-par file has the entry:

```

Split-plane      5      1      2      3

```

This indicates a split plan on DBlayer 5, net 1, 2 and 3. The net numbers are determined by selecting "CREATE NETLIST REPORT".** This is how the software will know which layer and nets to place thermal reliefs on.

If all files are intact, select the command CREATE PHOTOPLOT. When complete a listing of the PHOTOPLOT - LOG will appear.

PRINT (F5)

** CAUTION! Anytime the net-data-base changes the net numbers change also. You must re-create the netlist report.

To complete the photoplot process:

PENPLOT ARTWORK

Penplot each film, checking all layers and once more verifying the split plane has the appropriate spacings and the thermal reliefs appear in the correct sections. Refer to Figure 3 for an example of the "PENPLOT ARTWORK" split plane.

You may now transfer the "-Art Files" to tape and send to your photoplotting vendor. Figure 4 depicts an example of a split plane film.

Let's recap the process, referencing Figure 5.

1. P.C. board with net-data-base
2. All photoplot related files: pin files, layerstd, photoplot-par, photoplot-con and aperture-tab
3. Add lines of demarcation
4. Create photoplot
5. Penplot artwork and verify
6. Art file to tape

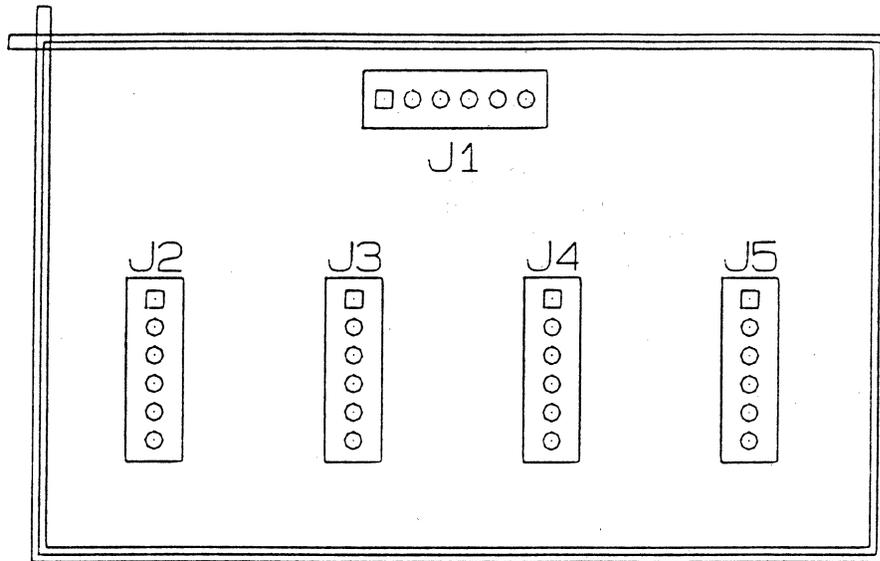


FIGURE 1

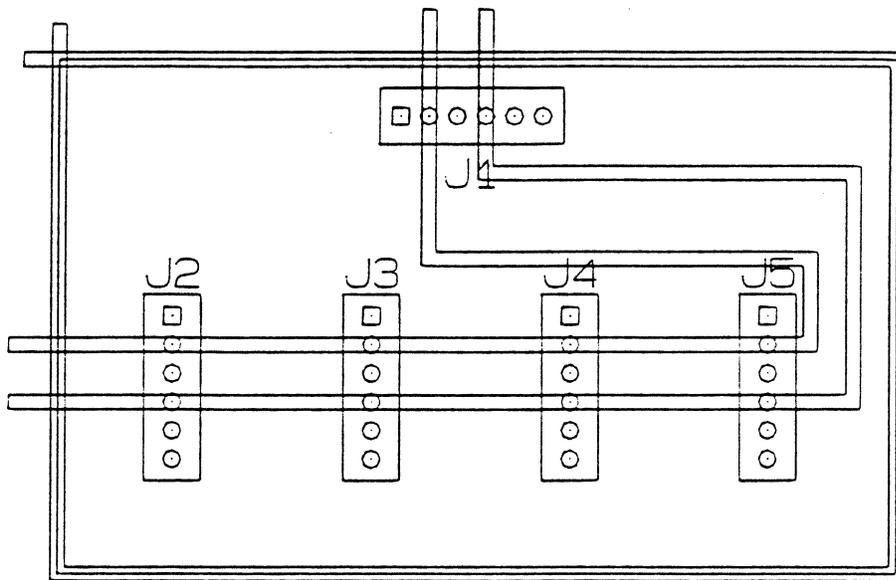


FIGURE 2

PHOTOPLOT PROCESS

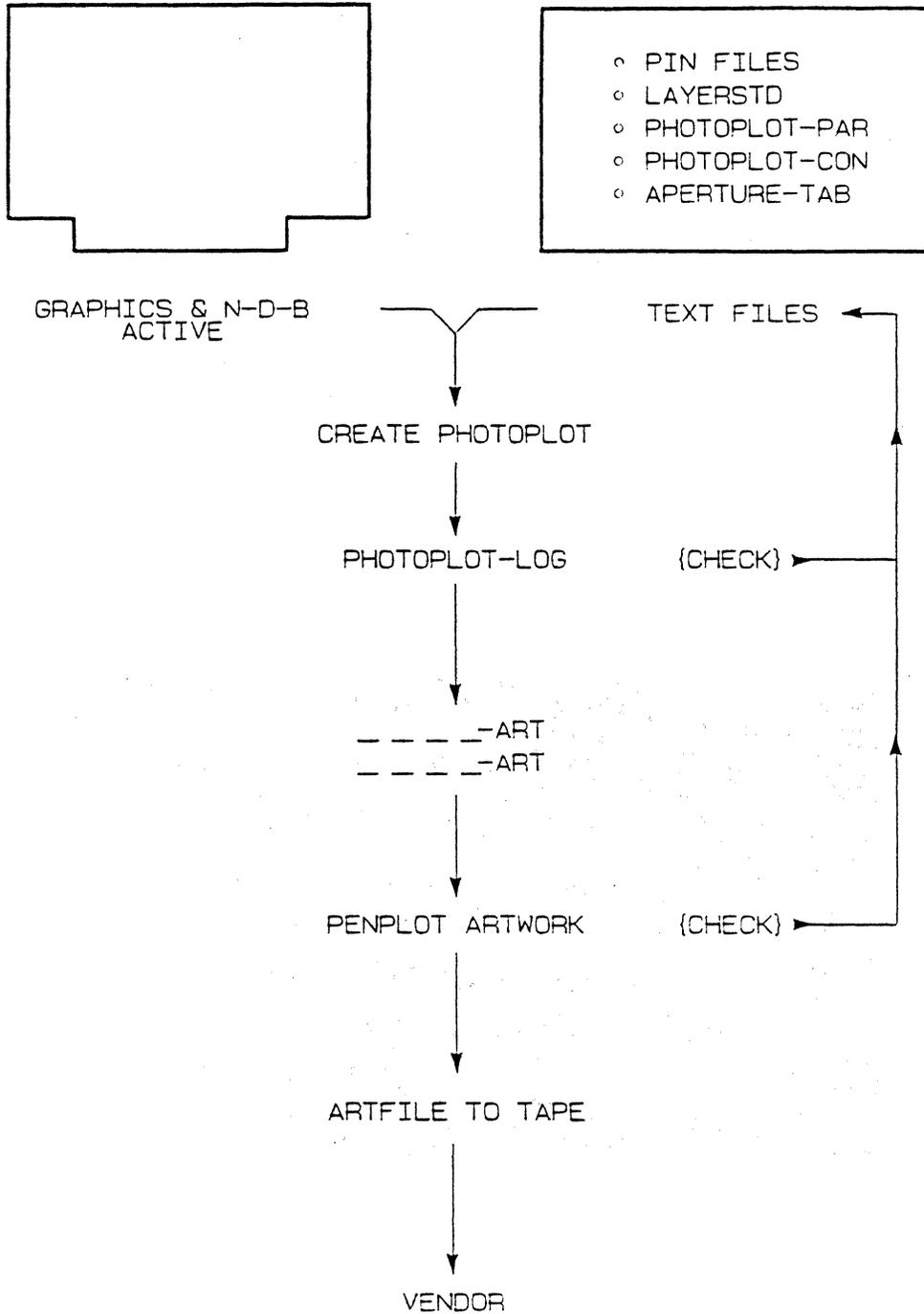


FIGURE 5

```

(FILE NAME: LAYERSTD)
(TELESIS STANDARD LIBRARY)
(PHYSICAL LAYER COMPONENT SIDE)
DBLAYER 1 COMPONENT-SIDE
(PHYSICAL LAYER SOLDER SIDE)
DBLAYER 2 SOLDER-SIDE
(PHYSICAL LAYER INTERNAL SIGNAL)
DBLAYER 3 INTERNAL-SIGNAL
(PHYSICAL LAYER INTERNAL SIGNAL)
DBLAYER 4 INTERNAL-SIGNAL
(PHYSICAL LAYER IMBEDDED SPLIT PLANE +5V,+12V, +15V)
DBLAYER 5 IMBEDDED-PLANE +5V
(PHYSICAL LAYER IMBEDDED GND)
DBLAYER 6 IMBEDDED-PLANE GND
DBLAYER 7 CLEAR-BORDER
DBLAYER 15 COMPONENT-SOLDER-MASK
DBLAYER 16 SOLDER-SOLDER-MASK
DBLAYER 33 CARD-OUTLINE
DBLAYER 34 PLATING-BAR
DBLAYER 36 DRAWING-FORMAT
DBLAYER 51 COMPONENT-OUTLINE
DBLAYER 54 REFERENCE-DESIGNATOR
DBLAYER 88 BOARD-DIMENSIONS
DBLAYER 89 SILKSCREEN
DBLAYER 90 TOOLING-CORNERS
DBLAYER 99 DRILL-CODE
END

```

```

(PIN FILE FOR PACKAGE SYMBOL CON6)
(NAME OF FILE CON6-PIN)
(TELESIS STANDARD PACKAGE LIBRARY)
PINTYPE A
DRILL .039-P
PAD SQUARE=.062 COMPONENT-SIDE
PAD SQUARE=.062 SOLDER-SIDE
PAD CIRCLE=.050 INTERNAL-SIGNAL
THERMAL-RELIEF FLASH=AB00 IMBEDDED-PLANE
ANTI-PAD CIRCLE=.062 IMBEDDED-PLANE
PAD SQUARE=.080 COMPONENT-SOLDER-MASK
PAD SQUARE=.080 SOLDER-SOLDER-MASK
PINTYPE B
DRILL .039-P
PAD CIRCLE=.062 COMPONENT-SIDE
PAD CIRCLE=.062 SOLDER-SIDE
PAD CIRCLE=.050 INTERNAL-SIGNAL
THERMAL-RELIEF FLASH=AB00 IMBEDDED-PLANE
ANTI-PAD CIRCLE=.062 IMBEDDED-PLANE
PAD CIRCLE=.080 COMPONENT-SOLDER-MASK
PAD CIRCLE=.080 SOLDER-SOLDER-MASK
PIN 1 A
PIN 2-6 B
END

```

```
(PHOTOPLOT PARAMETER FILE: FILENAME: PHOTOPLOT-PAR)
DEVICE-TYPE GERBER6240
CODE ASCII
NUMBER-OF-WHEELS 1
END-BLOCK-CHAR *
COORDINATES INCREMENTAL
ENGLISH
FORMAT 5.3
LEADING-ZEROS NOT-SUPPRESS
TRAILING-ZEROS NOT-SUPPRESS
TEXT-THICKNESS .012
UNDEF-LINE-WIDTH .012
MACHINE-OFFSET 0.0.0.0
SPLIT-PLANE 5 1 2 3
END
```

```
(APERTURE-TAB)
WHEEL 1
(STATION) (SIZE) (CODE)
(1) LINE=.004 D10
(2) LINE=.012 D11
(3) LINE=.050 D12
(4) LINE=.100 D13
(5) CIRCLE=.050 D14
(6) CIRCLE=.062 D15
(7) CIRCLE=.080 D16
(8) SQUARE=.062 D17
(9) SQUARE=.080 D18
(10) FLASH=AB00 D28
END
```

```
(PHOTOPLOT CONTROL FILE: FILENAME: PHOTOPLOT-CON)
ARTWORK FILM-2
(OBLAYER 5)
IMBEDDED-PLANE
CLEAR-BORDER
END
```