

# PRISMA

## TABLE OF CONTENTS

### A. TUTORIAL

Introduction . . . . .	1
Realtime Integration (RTI) between Schematic and PCB . . . . .	2
General information . . . . .	3
About this manual . . . . .	4
Starting the system and entering CIS . . . . .	5
Reading in a file or starting a new job . . . . .	6
Entering schematics . . . . .	7
Main menu for schematics . . . . .	8
Modes including grids and panning . . . . .	9
Graphic setup . . . . .	10
Selecting grid sizes (Routing, Display, Digitizing) . . . . .	11
Modifications (work menu) . . . . .	12
Creating a new symbol . . . . .	13
Adding terminals to a symbol . . . . .	14
Changing symbol graphics . . . . .	15
Creation of some other new symbols . . . . .	16
Saving a file . . . . .	17
Symbols for supply signals . . . . .	18
Symbols out of columbine library server . . . . .	19
Submenu for symbols . . . . .	20
Creating a bus . . . . .	21
Submenu for bus . . . . .	22
Connecting signals from the bus to the symbol . . . . .	23
Signals and connections . . . . .	24
Submenu for signals . . . . .	25
Creating some new signals . . . . .	26
Supply signals . . . . .	28
Adding supply signals to the schematics . . . . .	29
Moving down one level in the hierarchy . . . . .	30
Placing symbols from the columbine library server . . . . .	31
Moving back to the top level of hierarchy . . . . .	34
Other useful functions of the schematics . . . . .	35
Symbol MOVE, COPY, RENAME, DELETE, ROTATE . . . . .	35
MODIFY GRAPHIC WITH OFFSET, LINE, CIRCLE, ROTATION POINT . . . . .	37
Multi sheet schematics . . . . .	39
Entering the pcb layout . . . . .	41
Main menu . . . . .	42

# PRISMA

## TABLE OF CONTENTS

### A. TUTORIAL (cont.)

Modes . . . . .	43
Technology . . . . .	46
Placement and routing . . . . .	46
Sizes . . . . .	47
Track technology . . . . .	49
Via technology . . . . .	50
Modifications . . . . .	51
Component placement . . . . .	52
Manual placement . . . . .	52
Semi automatic placement . . . . .	56
Creating new shapes . . . . .	60
Building a relation between symbol and shape . . . . .	62
Guided placement . . . . .	63
Submenu for tracks . . . . .	64
Digitizing tracks . . . . .	65
Automatic routing using ldROUT . . . . .	67
Working with a routed file . . . . .	69
Copper areas . . . . .	71

### CAM FEATURES

Center . . . . .	73
Centers connections to pads, vias and SMD pins . . . . .	73
Teardrops . . . . .	77

### B. CIS REFERENCE MANUAL

Introduction . . . . .	2
Menu-driven handling . . . . .	2
Term definitions . . . . .	3
Function keys . . . . .	5
1. Modification . . . . .	6
Full view . . . . .	8
Window loc . . . . .	8
Refresh . . . . .	8
Center . . . . .	8
Grid ON/OFF . . . . .	8
Restore . . . . .	8
Zoom . . . . .	8
Grids . . . . .	9
Digi-grid . . . . .	10

# PRISMA

## TABLE OF CONTENTS

### B. CIS REFERENCE MANUAL (cont.)

Disp-grid . . . . .	10
Place-grid . . . . .	10
Route-grid . . . . .	10
Color on . . . . .	11
Color off . . . . .	11
Color clear . . . . .	11
Set color . . . . .	11
Std. Color . . . . .	12
Scale . . . . .	12
Optic . . . . .	12
Pan ON/OFF . . . . .	13
Window . . . . .	13
Full view . . . . .	13
Window loc . . . . .	13
Get symbol . . . . .	15
Status window . . . . .	16
Create symbol . . . . .	16
Insert symbol . . . . .	17
Move symbol . . . . .	18
Rename symbol . . . . .	18
Modify symbol . . . . .	19
Copy symbol . . . . .	19
Delete symbol . . . . .	19
Swap symbol . . . . .	19
Replace symbol . . . . .	20
Scale symbol . . . . .	20
List symbol . . . . .	21
Digitize symbol . . . . .	21
Change symbol logic . . . . .	22
Open symbol . . . . .	22
Close symbol . . . . .	22
Popup . . . . .	22
Get terminal . . . . .	23
Create terminal . . . . .	23
Insert terminal . . . . .	24
Move terminal . . . . .	24
Rename terminal . . . . .	24
Modify terminal . . . . .	24
Copy terminal . . . . .	24
Delete terminal . . . . .	24
List terminal . . . . .	25
Digitize terminal . . . . .	25
Change terminal logic . . . . .	25
Get signal . . . . .	26
Create signal . . . . .	26
Insert signal . . . . .	26
Rename signal . . . . .	27

# PRISMA

## TABLE OF CONTENTS

### B. CIS REFERENCE MANUAL (cont.)

Modify signal . . . . .	27
Auto-snap . . . . .	28
X,Y VALID . . . . .	28
45 DEG. VALID . . . . .	28
Nonorthogonal . . . . .	28
Tap connect mode . . . . .	28
Ignore symbol-coll . . . . .	28
Ignore term.-Collis . . . . .	28
Complete signal . . . . .	28
Immediate kill . . . . .	28
Implicit sig -> trm . . . . .	29
Implicit trm -> sig . . . . .	29
Impl. Sig -> bus . . . . .	29
Impl. Siga -> sigb . . . . .	29
List signal . . . . .	30
Digitize signal . . . . .	30
Status window at signal/mod. Parameter . . . . .	30
Next signal . . . . .	30
Get bus . . . . .	31
Create bus . . . . .	31
Insert bus . . . . .	31
Delete bus . . . . .	32
Tap bus . . . . .	32
List bus . . . . .	32
Status window . . . . .	33
Format popup . . . . .	36
Modify graphic . . . . .	38
Modify parameters . . . . .	41
Route mode . . . . .	45
Route mode off . . . . .	45
Route mode on . . . . .	45
Continue . . . . .	45
Multi . . . . .	45
2. Graphic setup . . . . .	46
3. Final check . . . . .	51
4. Miscellaneous . . . . .	52
5. Modes . . . . .	57
6. Technology . . . . .	60
7. Change file . . . . .	63
8. Save file . . . . .	64
9. Exit . . . . .	65
10. Status . . . . .	66
11. Modification . . . . .	67

# PRISMA

## TABLE OF CONTENTS

### B. CIS REFERENCE MANUAL (cont.)

Colors . . . . .	74
I. Definition of the colors . . . . .	74
II. Assignment of the colors and the kind of display to the board elements . . . . .	75
Specifying the element representation . . . . .	76
Color specifications . . . . .	77
Graphic setup . . . . .	113
Final check . . . . .	118
Miscellaneous . . . . .	119
Report . . . . .	120
Plot . . . . .	121
Modes . . . . .	122
Technology . . . . .	125
Change file . . . . .	145
Save file . . . . .	146
Exit . . . . .	147
Status . . . . .	148

### C. ROUTER

ROUTER LDROUT . . . . .	1
I. Introduction . . . . .	1
II. Handling . . . . .	2
a. Starting the program . . . . .	2
b. Exit and restart . . . . .	4
c. Options . . . . .	5
1. Router options . . . . .	6
2. Selection of the control file . . . . .	8
d. Standard control file ldroun.Ctl . . . . .	10
Busrouting . . . . .	10
SMD connections . . . . .	10
Glossing . . . . .	11
e. Optimizing router . . . . .	11
III. Output format . . . . .	13
a. Basic information . . . . .	13
b. Additional information . . . . .	14
IV. Router technology . . . . .	16
a. General . . . . .	16
Router symbols . . . . .	16
Display of routable tracks . . . . .	17

# PRISMA

## TABLE OF CONTENTS

### C. ROUTER (cont.)

	Display of vias and non-routable tracks . . . . .	17
b.	User defined technologies . . . . .	18
	General . . . . .	18
	General definition instructions . . . . .	18
	Placement & routing . . . . .	20
	Sizes . . . . .	25
	Layer plan . . . . .	26
	Change track technology . . . . .	30
	Track width overall . . . . .	31
	Via technology . . . . .	34
	Drills . . . . .	37
	Reference component . . . . .	38
c.	Standard technologies . . . . .	39
d.	Ldr-file . . . . .	46
VV	Multilayer routing . . . . .	48

### D. COLUMBINE

1.	Introduction . . . . .	1
2.	Starting the program . . . . .	6
3.	Library sections . . . . .	7
3.1	Data transfer . . . . .	8
3.1.1	Conditions for the data transfer . . . . .	8
3.1.2	Execution of the transfer . . . . .	10
3.2	Body . . . . .	13
3.3	Logic . . . . .	17
3.4	Function types . . . . .	20
3.5	Devices . . . . .	30
3.5.1	Device parameters for plotter . . . . .	31
3.5.2	Parameters for assembly machines . . . . .	35
3.5.3	Variants of assembly machine . . . . .	37
3.5.4	Parameters for tester . . . . .	39
3.5.5	Parameters for drilling machines . . . . .	41
3.5.6	Parameters for nc-routing machines . . . . .	43
3.6	Graphic macros (for plotter) . . . . .	45
REPORT	. . . . .	48
1.	Introduction . . . . .	48
2.	User specification . . . . .	48
3.	Area selection . . . . .	48
3.1	Function types . . . . .	50
3.2	Body . . . . .	50

PRISMA

TABLE OF CONTENTS

D. COLUMBINE (cont.)

3.3 Devices . . . . . 51  
3.4 Macros . . . . . 52  
3.5 Stock ids . . . . . 52  
3.6 Logic . . . . . 53  
BTOOLS . . . . . 54  
1. Introduction . . . . . 54  
2. Handling . . . . . 54

E. SCSCOM

GENERAL . . . . . 1  
A.0 INTRODUCTION . . . . . 2  
1. Title Specification . . . . . 2  
2. Units . . . . . 2  
3. Technology Definitions . . . . . 2  
4. Sheet Definition . . . . . 2  
5. Symbol Definitions . . . . . 2  
6. Symbol Placement . . . . . 2  
A.1 NAMES/COORDINATES . . . . . 3  
1. Characters and names . . . . . 3  
2. Coordinates . . . . . 4  
A.2 Title . . . . . 4  
A.3 Units . . . . . 5  
A.4 Technology . . . . . 6  
Technology options . . . . . 6  
A.5 Schematics sheet . . . . . 7  
A.6 Bodies . . . . . 7  
A.7 Terminal definition . . . . . 9  
A.8 Placement . . . . . 10  
1. Component and shape types . . . . . 10  
2. Placement specification . . . . . 10  
3. Rotation . . . . . 11  
4. Graphic elements . . . . . 11

# PRISMA

## TABLE OF CONTENTS

### E. SCSCOM (cont.)

A.9 Graphic elements . . . . .	12
1. Line segments . . . . .	12
2. Text fields . . . . .	12
A.10 Syntax description . . . . .	14
1. General . . . . .	14
2. Basic elements of the syntax . . . . .	14
3. Global structure of a pcb-file . . . . .	15
4. Syntax elements of the global structure . . . . .	15
5. Keywords . . . . .	17
6. Syntax diagrams . . . . .	17

### F. POSTPROCESSING

Introduction . . . . .	1
General . . . . .	1
Function selection . . . . .	1
Notes to the documentation . . . . .	2
Output structure . . . . .	2
Starting the procedure generator . . . . .	6
Procedures and Output . . . . .	7
Define/change procedures . . . . .	7
1.1 Menu 0 . . . . .	7
1.2 Menu description . . . . .	7
For the further proceeding . . . . .	10
Execute procedures . . . . .	11
Output of the plot data on the plotter (online) or in a file (offline) . . . . .	13
Processing the procedure library output.Sys . . . . .	15
Selection output data . . . . .	18
Introduction . . . . .	18
Menu 2 . . . . .	18
Menu description . . . . .	18
Procedure definition plot . . . . .	20
Introduction . . . . .	20
'Sheet' definition . . . . .	21
General . . . . .	21
Menu 3 . . . . .	21
Menu description . . . . .	21
Functions . . . . .	23
Handling instructions . . . . .	24
'Picture' definition . . . . .	25



# PRISMA

## TABLE OF CONTENTS

### F. POSTPROCESSING (cont.)

General . . . . .	25
Menu 4 . . . . .	25
Menu description . . . . .	25
Functions . . . . .	28
Handling instructions . . . . .	29
Display of the picture . . . . .	30
Menu 5 . . . . .	30
Menu description . . . . .	30
General remarks for the further proceeding . . . . .	34
Layer plot . . . . .	35
Layer selection . . . . .	35
Handling instructions . . . . .	35
Selection of the layer elements of signal layers . . . . .	36
Menu 24 . . . . .	36
Menu description . . . . .	36
For the further proceeding . . . . .	38
Solder resist plan . . . . .	39
Layer selection . . . . .	39
Selection of the signal layer elements that are Not to be covered . . . . .	40
Silk screen . . . . .	42
Layer selection . . . . .	42
Component selection of component layers . . . . .	43
Menu 21 . . . . .	43
Menu description . . . . .	43
Component display . . . . .	45
Drill plan . . . . .	46
Layer selection . . . . .	46
Selection of the kind of via for the drill plan . . . . .	47
Menu 30 . . . . .	47
Menu description . . . . .	47
Display of the drills . . . . .	48
Production plot . . . . .	48
Control plot . . . . .	49
For the further proceeding . . . . .	50
Negative plan . . . . .	51
Layer selection . . . . .	51
Enlargement of the vias that do not belong to the Plane signal . . . . .	52
Display of the vias and heat traps for control Plots . . . . .	53
Display of vias/heat traps for production plots . . . . .	54
For the further proceeding . . . . .	55
Element selection for documentation layers . . . . .	56
Mask selection . . . . .	57
Mask selection for control plots (manual) . . . . .	57

# PRISMA

## TABLE OF CONTENTS

### F. POSTPROCESSING (cont.)

Mask selection for the via display in production	
Plots . . . . .	59
Segment display in production plots . . . . .	66
Procedure definition drill . . . . .	71
Introduction . . . . .	71
Drill tape definition . . . . .	72
Menu 66 . . . . .	72
Functions . . . . .	72
Selection of the elements to be drilled . . . . .	74
Menu 67 . . . . .	74
Menu description . . . . .	74
Specifications to offset, fix-file, etc. . . . .	76
Menu 69 . . . . .	76
Handling instructions . . . . .	76
Changing the standard values for drill tools . . . . .	78
For vias . . . . .	78
Menu 70 . . . . .	78
Menu description . . . . .	78
For test points . . . . .	80
Menu 71 . . . . .	80
Procedure definition nc-routing . . . . .	81
Introduction . . . . .	81
Tape definition . . . . .	82
Menu 72 . . . . .	82
Functions . . . . .	82
Specifications to offset and fix-file . . . . .	84
Menu 73 . . . . .	84
Handling instructions . . . . .	84
Specifications for nc-routing tools . . . . .	85
Menu 74 . . . . .	85
Menu description . . . . .	85
Introduction . . . . .	91
List selection . . . . .	91
List output in a file . . . . .	93
List output on the terminal . . . . .	94
Output form of the lists . . . . .	95
Netlist . . . . .	95
Component list . . . . .	97
Non connected terminals . . . . .	99
Cross-reference-list (= component connections) . . . . .	100
Component collisions . . . . .	100
Technology . . . . .	100
Statistics . . . . .	101

**G. ERROR MESSAGES**

**H. INDEX**

# TABLE OF CONTENTS

## A. TUTORIAL

Introduction . . . . .	1
Realtime Integration (RTI) between Schematic and PCB . . . . .	2
General information . . . . .	3
About this manual . . . . .	4
Starting the system and entering CIS . . . . .	5
Reading in a file or starting a new job . . . . .	6
Entering schematics . . . . .	7
Main menu for schematics . . . . .	8
Modes including grids and panning . . . . .	9
Graphic setup . . . . .	10
Selecting grid sizes (Routing, Display, Digitizing) . . . . .	11
Modifications (work menu) . . . . .	12
Creating a new symbol . . . . .	13
Adding terminals to a symbol . . . . .	14
Changing symbol graphics . . . . .	15
Creation of some other new symbols . . . . .	16
Saving a file . . . . .	17
Symbols for supply signals . . . . .	18
Symbols out of columbine library server . . . . .	19
Submenu for symbols . . . . .	20
Creating a bus . . . . .	21
Submenu for bus . . . . .	22
Connecting signals from the bus to the symbol . . . . .	23
Signals and connections . . . . .	24
Submenu for signals . . . . .	25
Creating some new signals . . . . .	26
Supply signals . . . . .	28
Adding supply signals to the schematics . . . . .	29
Moving down one level in the hierarchy . . . . .	30
Placing symbols from the columbine library server . . . . .	31
Moving back to the top level of hierarchy . . . . .	34
Other useful functions of the schematics . . . . .	35
Symbol MOVE, COPY, RENAME, DELETE, ROTATE . . . . .	35
MODIFY GRAPHIC WITH OFFSET, LINE, CIRCLE, ROTATION POINT . . . . .	37
Multi sheet schematics . . . . .	39
Entering the pcb layout . . . . .	41
Main menu . . . . .	42
Modes . . . . .	43

Technology . . . . .	46
Placement and routing . . . . .	46
Sizes . . . . .	47
Track technology . . . . .	49
Via technology . . . . .	50
Modifications . . . . .	51
Component placement . . . . .	52
Manual placement . . . . .	52
Semi automatic placement . . . . .	56
Creating new shapes . . . . .	60
Building a relation between symbol and shape . . . . .	62
Guided placement . . . . .	63
Submenu for tracks . . . . .	64
Digitizing tracks . . . . .	65
Automatic routing using ldROUT . . . . .	67
Working with a routed file . . . . .	69
Copper areas . . . . .	71
 CAM FEATURES	
Center . . . . .	73
Centers connections to pads, vias and SMD pins . . . . .	73
Teardrops . . . . .	77

---

## INTRODUCTION

The new CALAY PRISMA SYSTEM includes the following main programs:

**CIS**            CALAY Interactive System. This is the interactive program for schematic capture and PCB - layout. Both parts of CIS are using Real - Time - Integration (RTI). CIS reads and writes .adb-files.

**LDROUT**      Automatic - router with INITIAL-, LEE- and OPTIMIZING-ROUTER. LDROUT reads and writes .gph-files.

**OUTPUT**      Postprocessing program used for creating

        Pen - and photoplot files  
        Drilling and milling files  
        Lists and documentation  
        Automatic insertion  
        Automatic testing

**OUTPUT** reads and writes .gph-files.

## COLUMBINE LIBRARY SERVER

        Interactive library editor with symbol and component library.

**SCSCOM**      Interface program between the different PRISMA programs as well as between Prisma's .adb-file structure and the .gph-file structure of CALAY V04-Rev.5.1. SCSCOM converts between :

**.adb into .rai**    The program RAIGPH converts this .rai-file into an .gph file, which then can be used by LDROUT.

**.rai into .adb**    The program GPHRAI converts the routed .gph file into an .rai file which then can be converted back to an .adb file by running SCSCOM.

**.adb into .gph**    Converts an .adb file into .gph file which can be used for postprocessing only. The .gph files for LDROUT and OUTPUT are different !!

**.adb into .pcb**    Interface between PRISMA and V04-Rev 5.1

**.pcb into .adb**    Use only this way to convert files between V04 and PRISMA or the way back.

**Realtime Integration (RTI) between Schematic and PCB**

**REALTIME INTEGRATION** makes it possible to handle both schematic and PCB-design changes simultaneously. Any modification done in the schematic or layout is realized automatically throughout the entire design, thus guaranteeing 100 % consistency between the schematic and PCB layout. There is no longer a need for forward and back-annotation. Any PIN or GATE SWAP within the PCB-layout is simultaneously done in the schematic too.

The basis for the **REALTIME INTEGRATION** is a fixed relationship between the schematic symbol and the physical shape. This fixed relation is realized within the **COLUMBINE LIBRARY SERVER**. For example, a component such as a 7400 copied out of the **COLUMBINE LIBRARY SERVER** includes 4 gates (NAND) and a physical shape (DIP14). By placing only one gate there will be 3 spare gates. These gates are not displayed on the actual page, they are reserved for later use. Using the function **COPY** one of this reserved gates will now be used.

**REALTIME INTEGRATION** works also with signals. Any signal created or modified in the schematic will automatically be created or modified in the PCB layout and vice-versa.

Components which are not yet in the **COLUMBINE LIBRARY SERVER** may be defined locally. For that you have to create one (or more) schematic symbol and a physical shape and build the relation between them.

Working with **PRISMA** now means, working on a schematic and PCB-layout at the same time. Some parameters of the PCB layout (track width, layers, critical signal etc.) now may be defined already during the work in the schematic.

## GENERAL INFORMATION

PRISMA displays schematics and PCB-layout using on screen graphics. While running CIS, the system displays different menus like:

- Function window (on the right)
- Status window (on top)
- Popup menu (at cursor position)
- Working window (rest of screen)

To enter these menus you can use:

- Optical mouse with cursor on screen
- Popup-menu activated by right mouse button
- Keyboard and function keyboard with special functions on keys L1 - L10, F1 - F9 and R1 - R15. You can see the actual assignments of the key functions in:

~/etc/doc/fkeys.eng

**CLB** = click left cursor button  
**CMB** = click middle cursor button  
**CRB** = click right cursor button



### ABOUT THIS MANUAL

This manual is structured around the actual creation of a schematic and PCB-layout. Within CIS we will see how to create new symbols and components, how to take them from the **COLUMBINE LIBRARY SERVER**, place them on schematic and board; how new connections are added, routed and the results are plotted.

Refer to the table of contents for specific functions you may be interested in learning. If you are a first time user, you will find it helpful to follow the steps given in this manual to guide you in the creation of a schematic and board.

The board we will be using for the example consists of a clock, register, counter, digital/analog converter (DAC) and a connection for an input voltage.

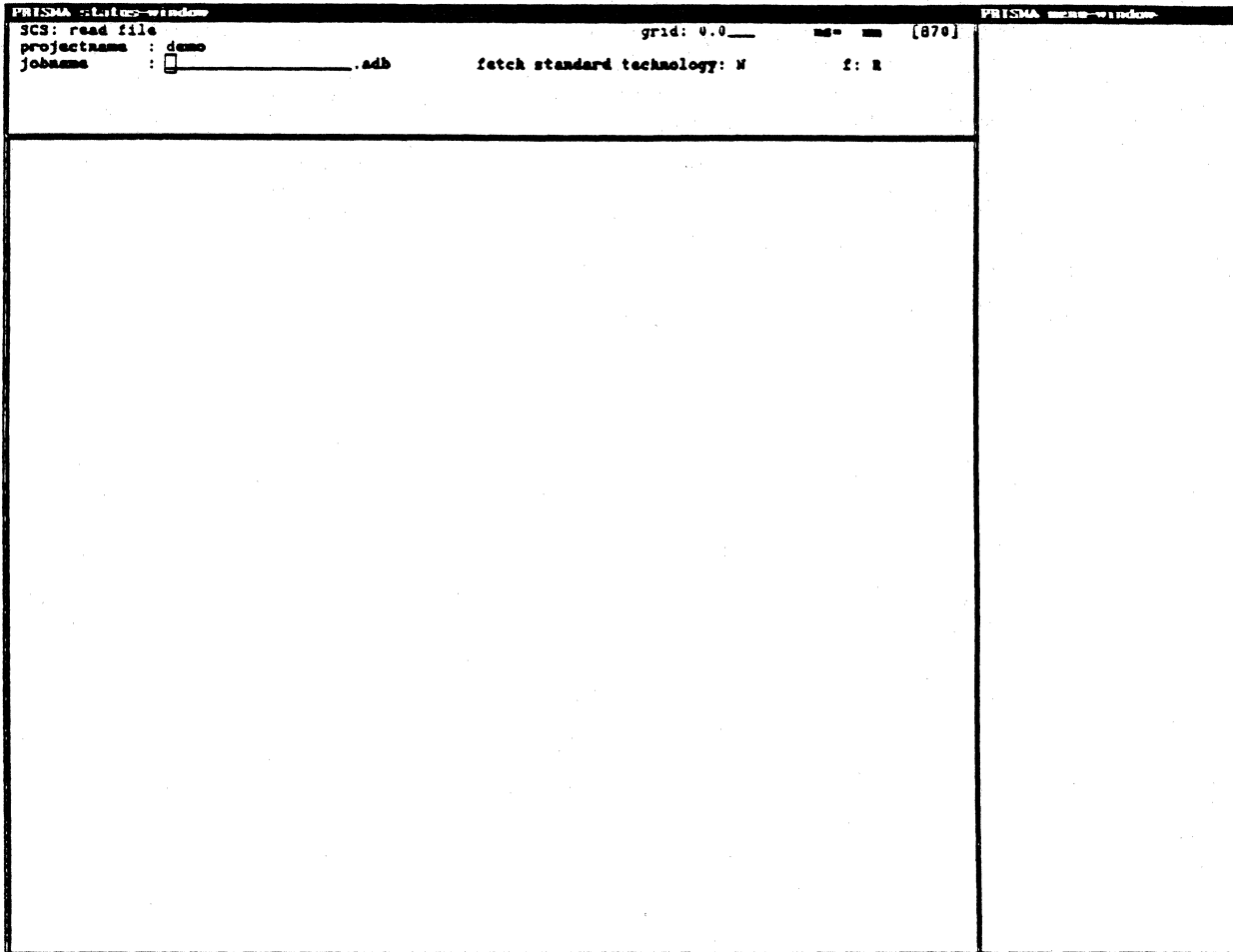
## **STARTING THE SYSTEM AND ENTERING CIS**

After "LOGIN" and specifying your ".PROJECT" (i.e.:demo) you will see the CALAY logo and all icons lined up on the graphic screen. To select CIS move your cursor to the icon "CIS" and press the left mouse button. You have now started CIS and a new menu will appear on the screen.

The system is now ready to read in an .adb file. Each .adb file includes both the schematic and the PCB part. After reading in an existing file, you can choose to work on the schematic or the layout, and at any time switch back and forth.

If you don't have an existing file to work on, you have to read in a start-up file which includes a default technology.  
(i.e.: tec.adb).

READING IN A FILE OR STARTING A NEW JOB



To select an existing file you have two possibilities:

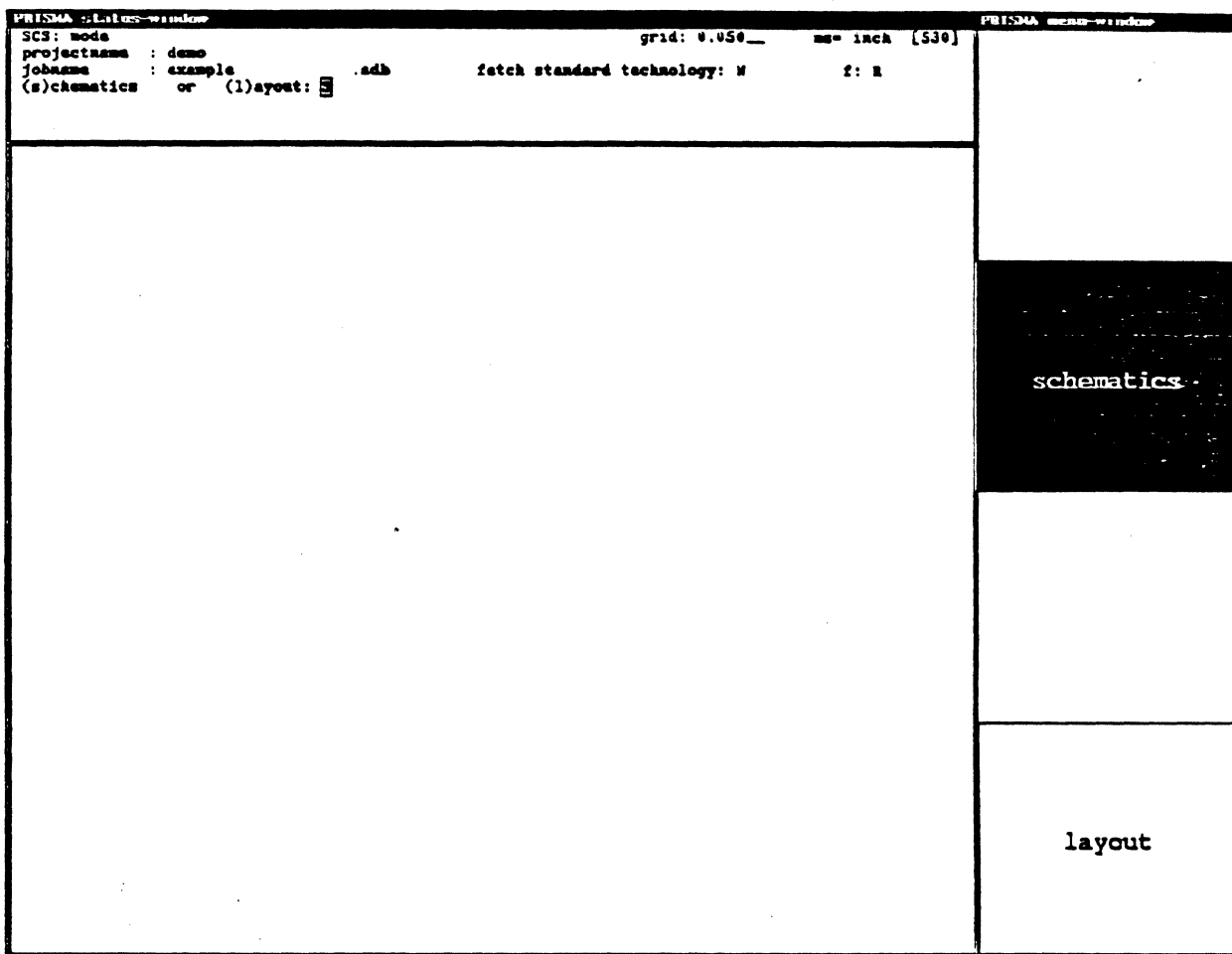
1. Type in the filename using the keyboard.
2. Use the mouse. You will get a list of existing .adb files by pressing the right button. Use the arrow keys to move the cursor inside that list. With <Return>, you move downwards one page.

The right button now marks the selected file or removes this selection by pressing the right button a second time. The left button now writes the file name into the position **JOBNAME** of the **READ FILE** menu.

Pressing the left button a second time (or <CR> ) reads in that file.

To start a new job, you have to read in a technology file called "tec.adb" or similar. This file only consists of a default technology and frames.

## ENTERING SCHEMATICS



## Selection between SCHEMATICS and LAYOUT:

- move the mouse into SCHEMATICS or LAYOUT and click the left mouse button;  
or
- type in "S" for SCHEMATICS
- "L" for LAYOUT

The selected drawing then will be displayed on the screen. For this example please select SCHEMATICS.

MAIN MENU FOR SCHEMATICS

PRISMA: main-window

SCS: functions grid: 0.050 mm inch [540]

(B) modification (1) (A) misc. (C) change file  
 (G) graphic (M) modes (U) update file  
 (F) final check (T) technology (X) exit  
 (L) layout (S) status

modification  
 graphic setup  
 final check  
 layout  
 misc.  
 modes  
 technology  
 change file  
 save file  
 exit  
 status

		CALAY

This is the top level of the menu hierarchy. Selecting one of the functions of this menu with the left mouse button brings you into a new menu. The status menu on the top of the screen is related to the function menu at the right of the screen and changes at the same time. Selecting of functions in this menu also can be done by the displayed key letters.

- MODIFICATION** Interactive working mode
- GRAPHIC SETUP** Def. of screen display, grids, colors, pages etc.
- FINAL CHECK** Not yet implemented
- LAYOUT** Display of PCB (jumps to the PCB main menu).
- MISC** Submenu with more global functions
- MODES** Definition of global schematic parameters
- TECHNOLOGY** Setup for technology (line width, text height, etc)
- CHANGE FILE** Not yet implemented, please use EXIT and restart
- STATUS** Shows which software version you are using

MODES INCLUDING GRIDS AND PANNING

The screenshot displays the PRISMA configuration window. The top section contains a list of system parameters with their current values and options to toggle them on or off. The bottom section shows a graphical workspace with a grid and a small table containing the word 'CALAY'.

**PRISMA - : : edit - window**

```

SCS: global parameter
system - units (I) inch,(M) mm: I real-time-integration: X grid: 0.450_ ms= inch [420]
symbol-frames: _ pop-ups: _ relation-help-graph: X lowercase char.: _
crosshair: X automatic-rotating: X textfield by number: X
airlines: X rotation: X automatic-reroute: X textfield-size automatic: X
search range: 0.0250_ terminal-help: X size: 0.0750_

autoclearance to symbols: 0.10_ to signal: 0.03750_ auto-rotating grid: 0.050_
display grid: 0.10_ interactive grid: 0.050_ level: 1_ A: 0.28475_

auto - enter terminal into signal: X auto - rename if signal - collision: _
auto - enter signal into bus: X auto - delete in layout: X

auto - create textfield for SCS name: X textsize Y: min. 0.10_ max. 0.40_
textfield position ( 0=center 1-up 2-down 3-right 4-left ): 0 minimal length: 1

rate of valid climb orthog.: X 45 degrees: X all angles: _ bus: conn. 15 deg: X 0.10_

standard-graphic-mode (P,W): W auto-panning-size: _ or fullsize: X
panning-area (pixels) in X: 1652 (min=1152 max=3456) in Y: 1300 (min=900 max=2700)
preference-value: 900_ pixels be adequate to 12.0_ system - units
    
```

**PRISMA - : : edit - window**

- modification
- graphic setup
- final check
- layout
- misc.
- ← modes**
- technology
- change file
- save file
- exit
- status

The graphical workspace shows a grid with a small table containing the word 'CALAY'.

		CALAY

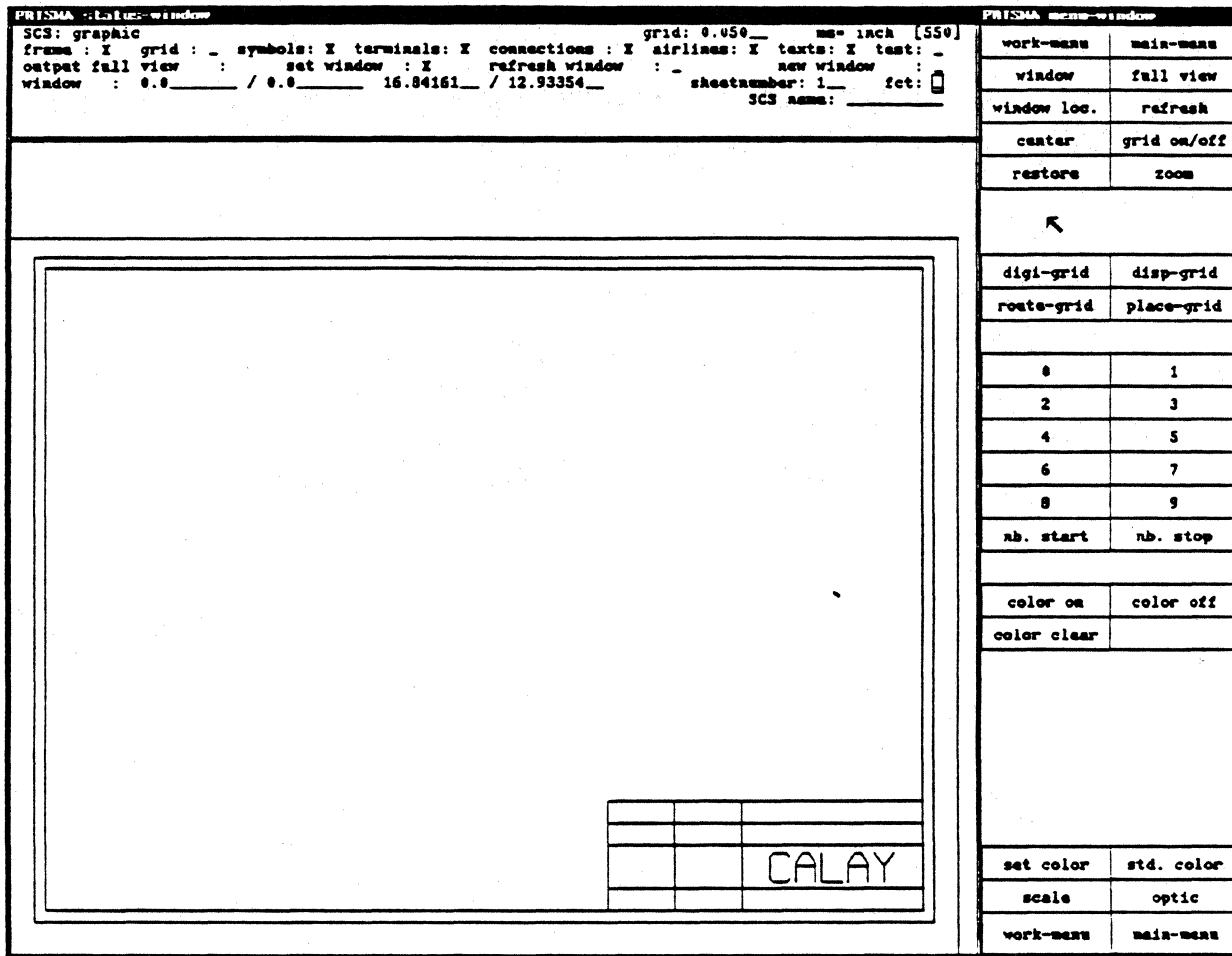
To change the values, move the cursor to your selected position and press the left mouse button. Now you can type in the correct value, i.e.: place the cursor at SYSTEM UNITS press left button and type in "I". The units will now be in inches.

**PANNING** Is activated at **STANDARD-GRAPHIC-MODE** P = panning  
W = window

The **PANNING AREA** in X and Y should be changed to values within the limits (depending on main memory X=2000 Y=1600).

Please check the highlighted positions and change them if necessary.

GRAPHIC SETUP



Enter **GRAPHIC SETUP** from **MAIN MENU**

You may enter the **GRAPHIC SETUP** from the **MAIN MENU** or from the **WORK MENU**.

You can define what you want to display by marking the positions on your status menu (#550). Other functions call up a different status menu.

**Color ON/OFF** Click **Color ON/OFF** followed by selected number of color (displayed in status menu #412)

**Color CLEAR** Click **Color CLEAR** followed by a number. Clears this color from screen.

**SET Color** Definition of color assignment.

**STD. Colors** Calls back the default **Color SET**.

**SCALE** Def. of sizes of a help scale.

**OPTIC** Terminal representation and selection of text.

SELECTING GRID SIZES (ROUTING, DISPLAY, DIGITIZING)

The screenshot shows the PRISMA software interface. At the top, there are two windows: 'PRISMA edit-window' and 'PRISMA menu-window'. The 'edit-window' contains a list of grid adjustment values (0-9) and their corresponding grid sizes (e.g., 0 = 0.00625, 1 = 0.01250, etc.). It also shows the current grid size as 0.050 and the unit as inches. The 'menu-window' contains a grid of menu options: work-menu, main-menu, window, full view, window loc., refresh, center, grid on/off, restore, zoom, a cursor arrow, a grid selection table, color on, color off, color clear, set color, std. color, scale, optic, and another work-menu/main-menu pair. The grid selection table is as follows:

0	1
2	3
4	5
6	7
8	9
nb. start	nb. stop

The main workspace shows a schematic diagram with a grid overlay. A small table in the bottom right corner of the workspace contains the text 'CALAY'.

You may work with three different grids, which you can select out of a list of ten default (but user definable) values.

Call up **GRAPHIC SETUP** from **MAIN MENU** or **WORK MENU**.

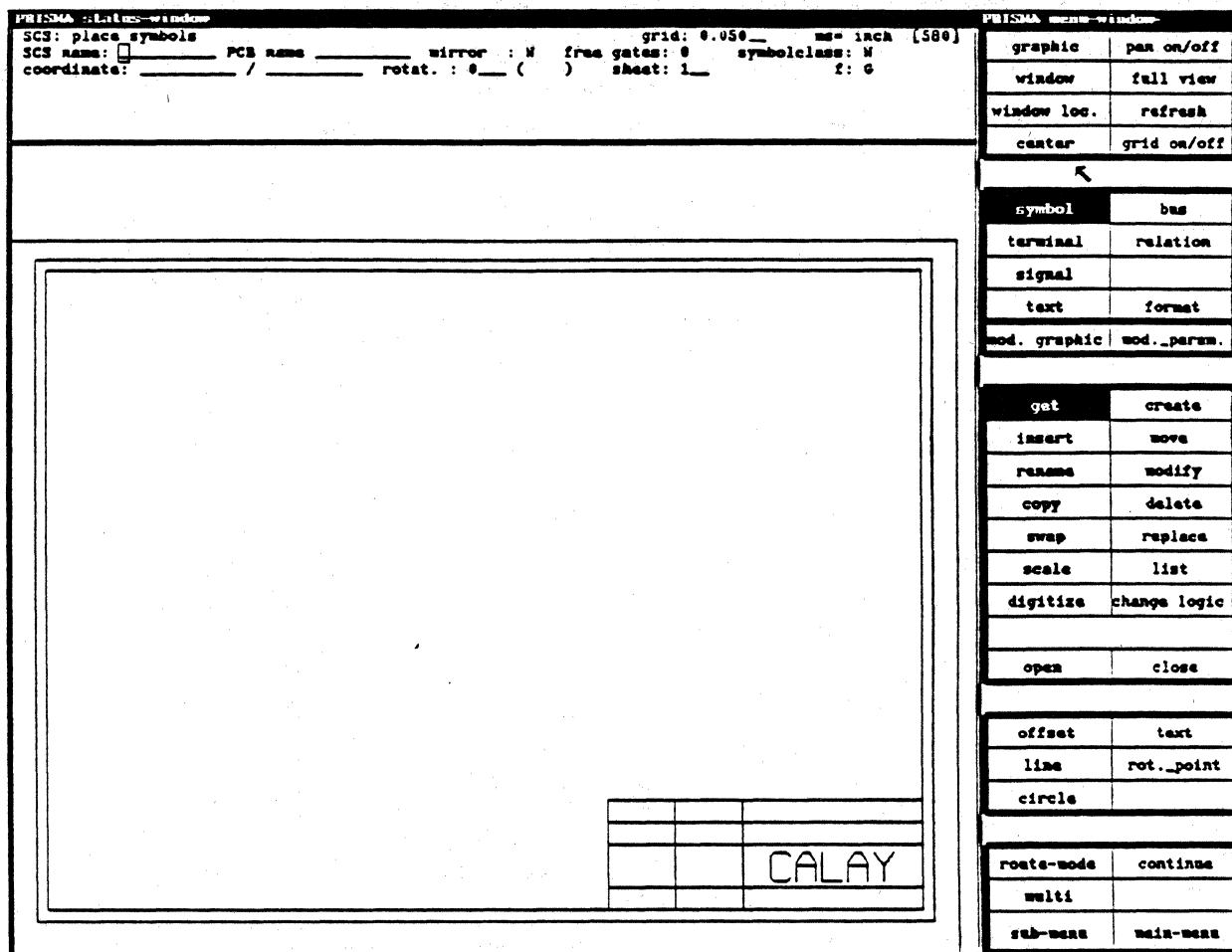
- DISP-GRID**    Displayed grid when **GRID ON/OFF** is clicked on.
- DIGI-GRID**    Interactive grid for all manual work.
- ROUTE-GRID**   Grid for automatic routing.
- PLACE-GRID**   Not yet implemented.

- i.e. Click **DISP-GRID** followed by 4    changes **DISP-GRID** to .1"
- Click **DIGI-GRID** followed by 2    changes **DIGI-GRID** to .05"
- Click **ROUTE-GRID** followed by 4    changes **ROUTE-GRID** to .1"

The active values are displayed on the right part of **STATUS MENU**.



MODIFICATIONS (WORK MENU)

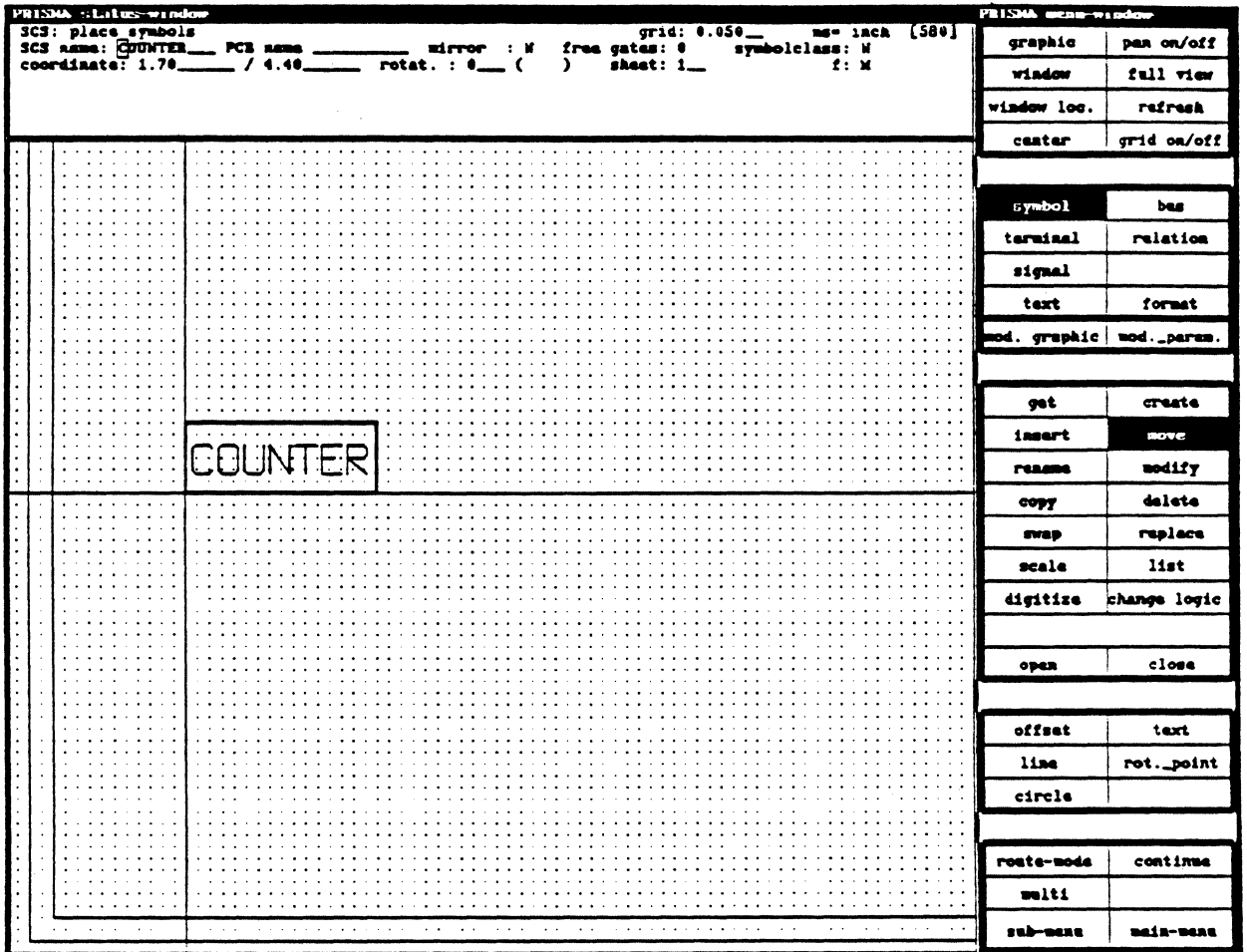


Now call up **MODIFICATION** from the **MAIN MENU**. This is the menu where you will do most of your work. This menu is divided into different parts:

- graphical adjustment like window, refresh, grid, panning.
- Elements like symbols, signals, bus, text, relations.
- What to do with these elements, like create, insert, move.
- Line modes like circle, offset, line.

According to the selected element (symbol, terminal, signal, bus) the status menu will change. The **POPUP MENU** only displays the possible operations for the selected element.

## CREATING A NEW SYMBOL



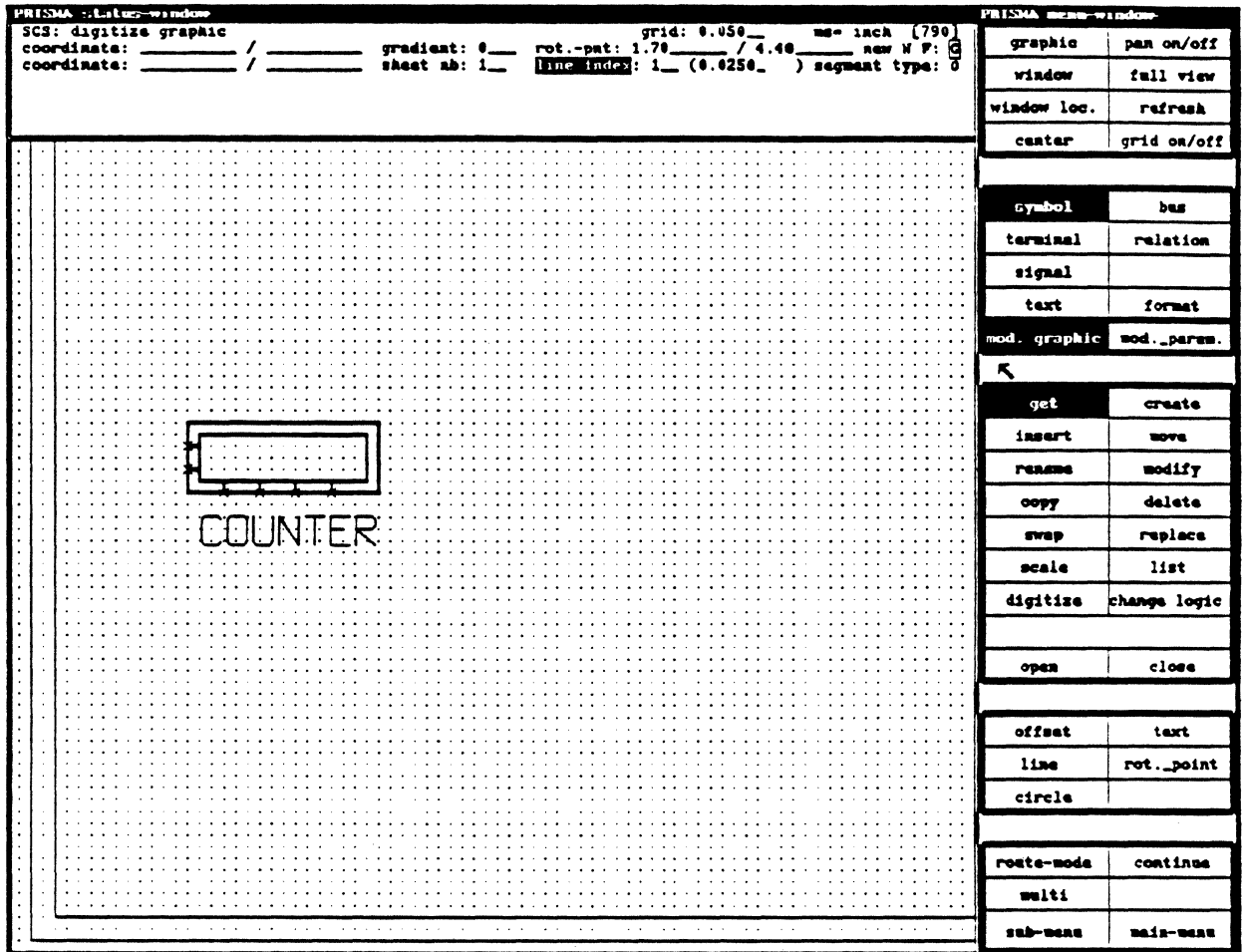
1. Call up **WINDOW**, mark the window at the lower left corner and then in the middle of your screen (always <CLB>). The screen will refresh
2. Call up **GRID ON/OFF** and <CLB>; the grid will be displayed
3. Place the mouse cursor in **CREATE** and <CLB>. Now we can create a new symbol.
4. Type in the symbol name "COUNTER" in the symbol name field in the upper menu.
5. Place the mouse cursor at a grid dot (1) inside the sheet and press the left button.
6. Move the cursor right 16 grid dots and up six grid dots (2), <CLB>  
The frame of a new symbol is created.

ADDING TERMINALS TO A SYMBOL

The screenshot shows the PRISMA software interface. The main window is titled "PRISMA --status--window" and contains a grid with a symbol labeled "COUNTER". The status bar at the top displays various parameters: "SCS: place terminals", "term-name: ( signal:", "SCS name:", "grid: 0.050", "ms= inch (620)", "coord.: /", "rotat.: 0", "mirror: N", "PCB name:", "ref.-no.:", and "f: C". On the right side, there is a "PRISMA menu--window" with a list of menu items: "graphic", "pan on/off", "window", "full view", "window loc.", "refresh", "center", "grid on/off", "symbol", "bas", "terminal", "relation", "signal", "format", "text", "mod. graphic", "mod..param.", "get", "create", "insert", "move", "rename", "modify", "copy", "delete", "swap", "replace", "scale", "list", "digitize", "change logic", "open", "close", "offset", "text", "line", "ret..point", "circle", "route-mode", "continue", "multi", "sub-menu", and "main-menu".

1. Select **TERMINAL**, system defaults to create.
2. Place cursor at 1 (see diagram) and <CLB>. The terminal is created and displayed as a cross.
3. The function has changed to **MOVE**; select **CREATE** again and place the second terminal.
4. Select **CONTINUE**, place your third terminal, the function is still **CREATE** and you can continue to place new terminals. Using **CONTINUE** please place a total of 6 terminals.
5. To make a reservation for the phys. text field (later to become the physical pin number) select **GET**, pick up one terminal, select **MOD.GRAPHIC** and **TEXT** (beside **OFFSET**), with <CRB> you can select the text field shown as **TEXT TYPE** in the **STATUS MENU**. Select **TEXT TYPE 4** "phys. terminal" and <CLB> at the position where this text should start. Move the cursor, define the text size and <CLB> again. A white box is now displayed.

## CHANGING SYMBOL GRAPHICS



The symbol now is shown as a rectangle only. To change the symbol graphic, please follow these steps:

1. Select **SYMBOL** (**MOVE** will now be active, if not, select **GET** and move the cursor to the symbol "counter" and <CLB>).
2. Select **MOD. GRAPHICS**. **GET** is active now. <CLB> in the middle of the symbol. You picked up the symbol name text field "COUNTER" and you can move this text now to any other position. Place this text below the symbol (<CLB>). Select **DIGITIZE**, you now can create new graphic inside the symbol. Please create a new smaller rectangle and lines from the terminals to the rectangle. To start a new line select **OFFSET**.
3. Select **GET**. Move the cursor to a place inside the symbol where no graphics are and <CLB>. Select **DELETE**, and the original graphic is deleted.

CREATION OF SOME OTHER NEW SYMBOLS

PRISMA -L:1:top-window

SCS: create symbols      grid: 0.050      mm= inch [580]

SCS name: DAC      PCB name      mirror : N      free gates: 0      symbolclass: N

coordinate:      /      rotat.: 0      (      )      sheet: 1      f: C

size :      /      symbolclass -> (N) norm. (G) gad (P) pwr : N

PRISMA menu-window

graphic	pan on/off
window	full view
window loc.	refresh
center	grid on/off

symbol	bus
terminal	relation
signal	
text	format
mod. graphic	mod. param.

↩

get	create
insert	move
rename	modify
copy	delete
swap	replace
scale	list
digitize	change logic

open	close
------	-------

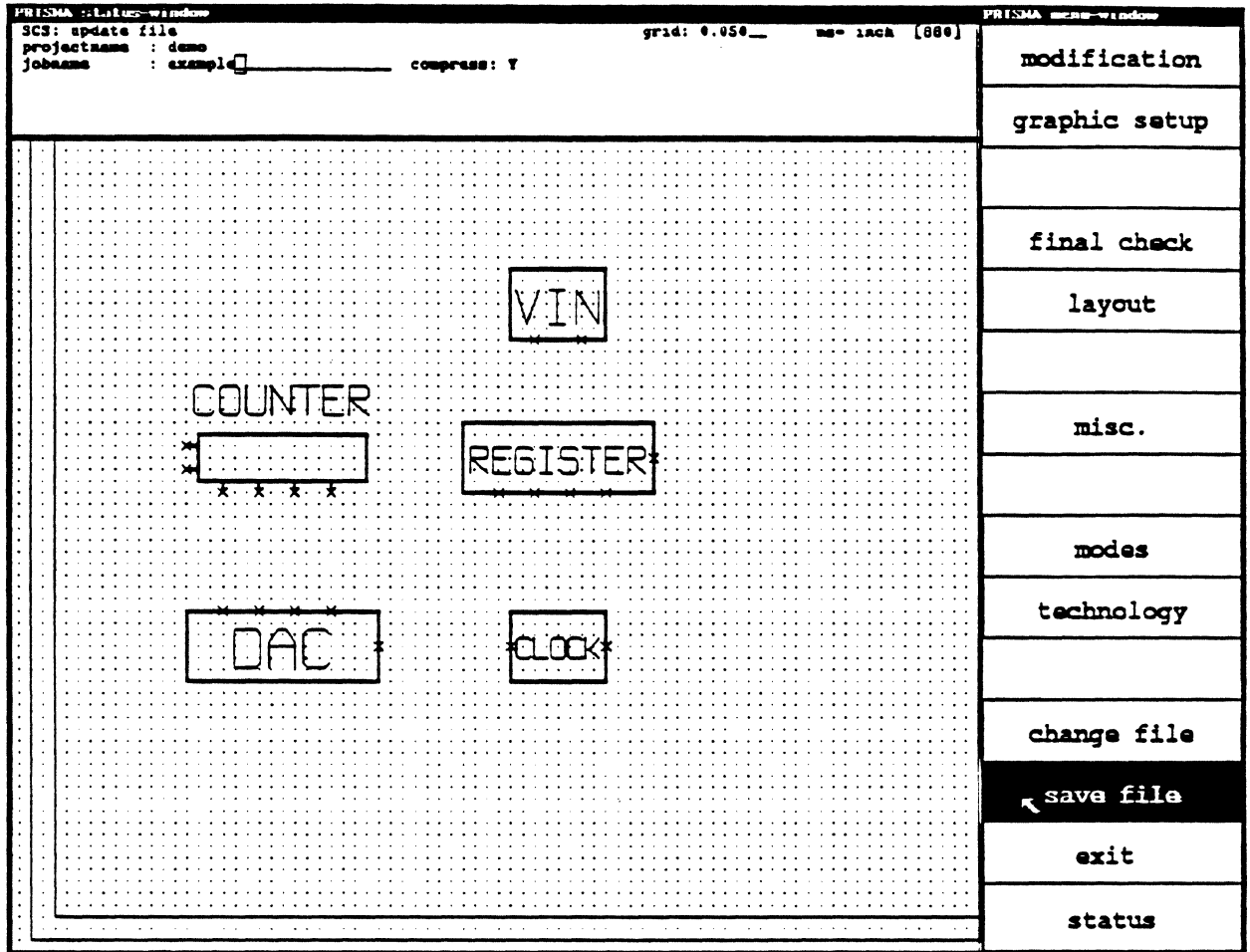
offset	text
line	rot. point
circle	

route-mode	continue
multi	
sub-menu	main-menu

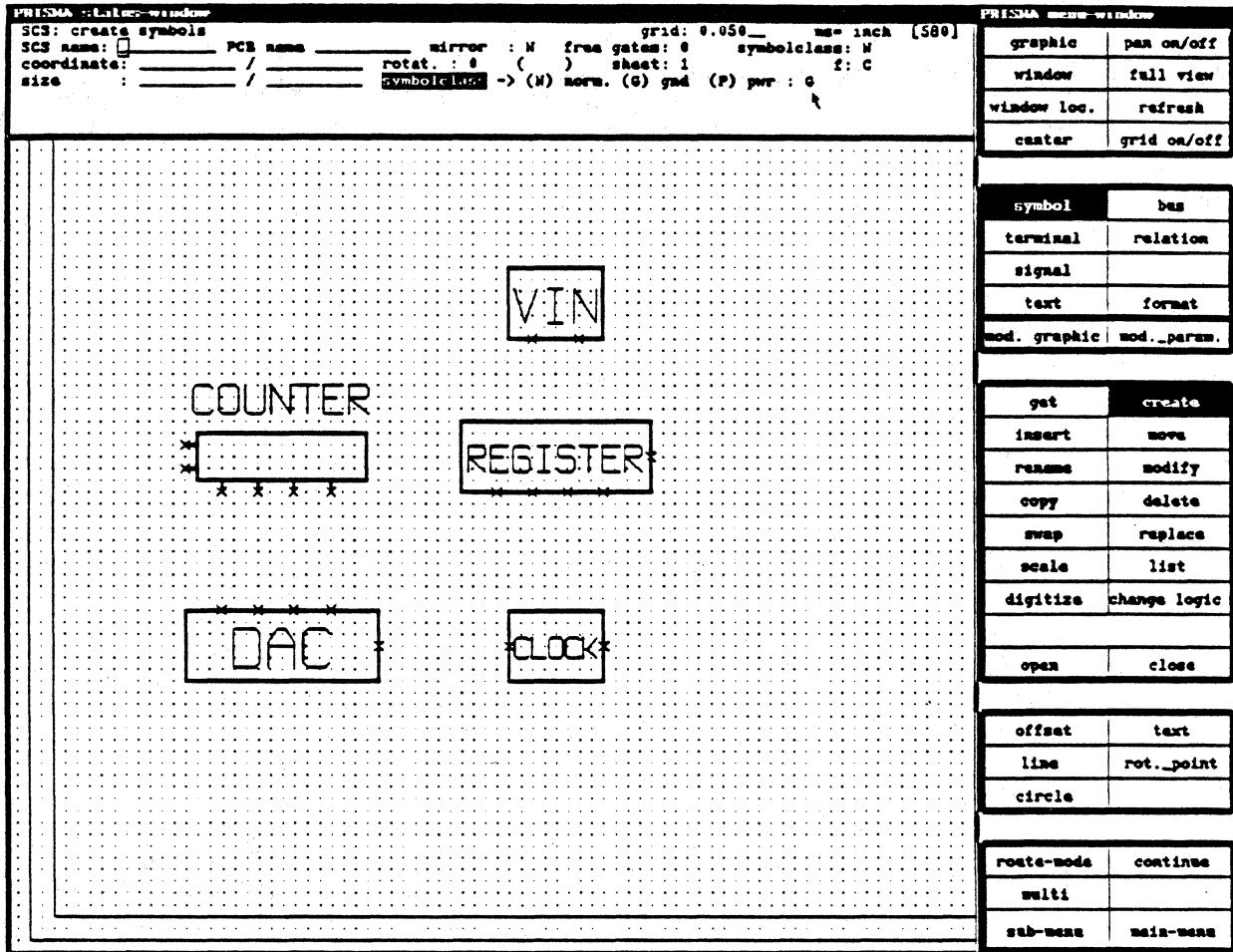
1. Select **SYMBOL CREATE**
2. Type "DAC" in upper menu in the **SYMBOL NAME** field.
3. Place cursor on a grid dot in sheet format, <CLB>
4. Move cursor right 16 grid dots and up 6 grid dots, <CLB>
5. Select **TERMINAL** and create one terminal, add a textfield for the phys. terminal (see some pages before).
6. Select **TERMINAL** and pick up that terminal, select **COPY** and **CONTINUE** and copy this terminal now five times.
7. Repeat the entire process and create a symbol called "REGISTER" using a dimension of 16 grid dots wide and 6 grid dots high. Create 5 terminals.
8. Create a symbol called "CLOCK", 8 grid dots wide and 6 grid dots high. Create two terminals.
9. Create a symbol called "VIN", 8 grid dots wide and 6 grid dots high. Create two terminals.

SAVING A FILE



1. Select MAIN MENU and leave the WORK MENU.
2. Select SAVE FILE with <CLB>.
3. Overwrite the JOBNAME with a new name i.e.: "example" without an extension and press <CR>.
4. You may compress the .adb file by selecting COMPRESS "Y" and then <CR>. The stored file uses the extension .adb.Z. To READ in a compressed file there is an automatic function to uncompress the file.

SYMBOLS FOR SUPPLY SIGNALS



There are also special symbols for power connections:

1. Select **SYMBOL** and **CREATE**.
2. Type in at **SYMBOLCLASS** in the upper menu "G" for GND or "P" for PWR without <CR> and then define the symbol frame by marking the diagonal corners with the cursor. Within the menu **PLACE SYMBOLS** the **SYMBOLCLASS** has changed to "P" or "G" and there is no symbol name visible. You can change the symbol graphic now and create your own GND symbol.
3. Create one terminal.
4. Select **SIGNAL**, type in GND at **SIGNALNAME** and <CR>. The function changes from **GET** to **INSERT**. <CLB> at your terminal.
5. The symbol is ready to use as a connection to GND.

## SYMBOLS OUT OF COLUMBINE LIBRARY SERVER

PRISMA : status-window

SCS: place symbols grid: 0.050\_\_ us= inch [500]  
 SCS name: U10 PCB name U10 mirror : M free gates: 0 symbolclass: M  
 coordinate: / rotat. : 0 ( ) sheet: 1 f: I  
 search-string: LM\* realised on comp.:  
 search-sequence (LPSB) : 1=1 2= 3= 4= (L) Shape: Lib: columbine

PRISMA menu-window

graphic	pan on/off
window	full view
window loc.	refresh
center	grid on/off

symbol	bas
terminal	relation
signal	
text	format
mod. graphic	mod..param.

get	create
insert	move
rename	modify
copy	delete
swap	replace
scale	list
digitize	change logic

open	close
------	-------

offset	text
line	rot..point
circle	

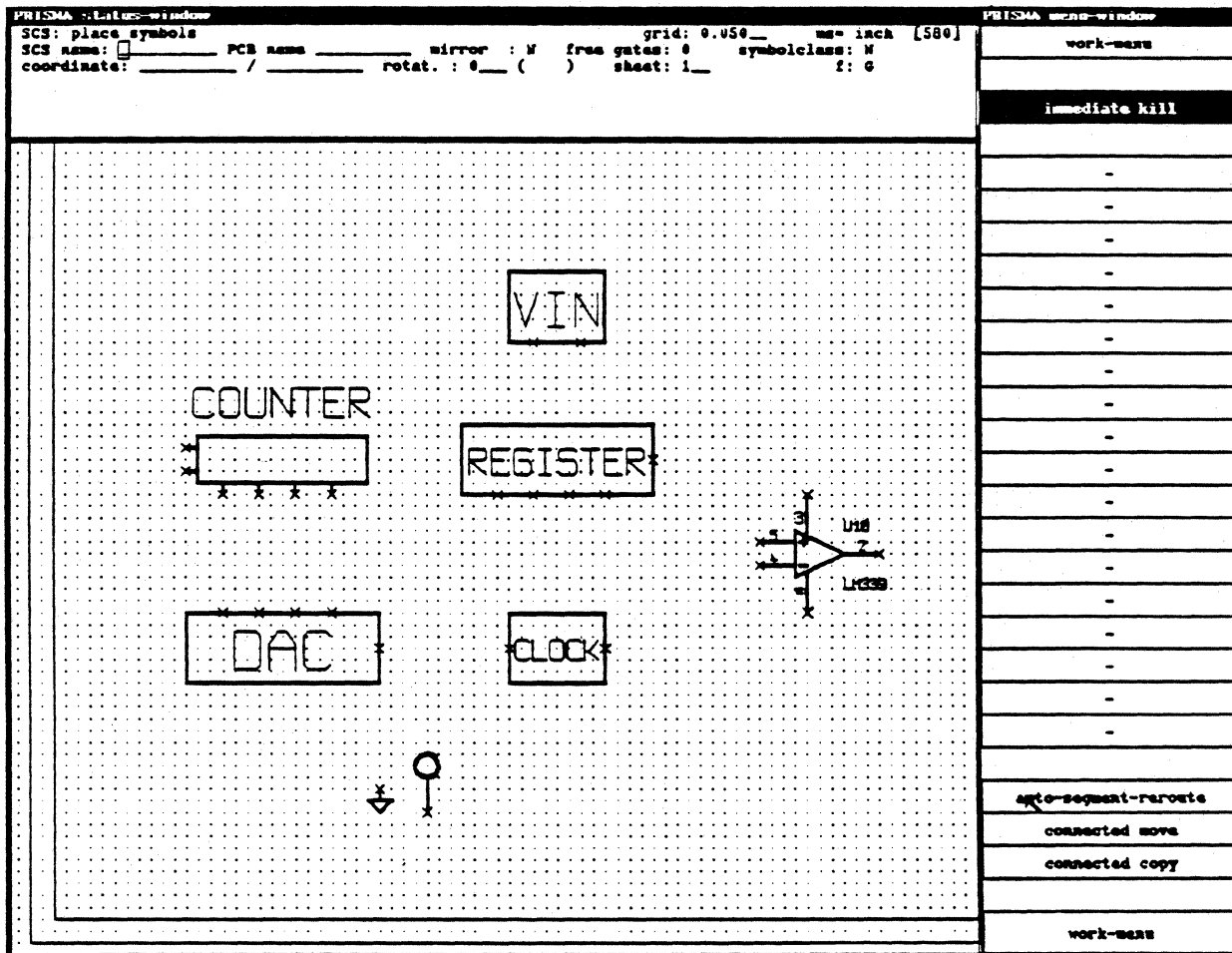
route-mode	continue
wait	
sub-menu	main-menu

1. Select SYMBOL and INSERT; LIST now opens a new window and lists the library. To end the listing press QUIT inside that window.
2. Type in at position SCS NAME "U10" ,at position PCB NAME "U10" and at position SEARCH STRING "LM\*" and press <CR>. You will get a list of all component names starting with LM... Move the cursor inside that window and <CLB> at LM339. The window will close and the name LM339 is written in SEARCH STRING. Move the cursor to a free position in your drawing and <CLB>. The symbol will be placed at this position.
3. To select a specific symbol you also may type in the full name of the symbol in the SEARCH STRING and continue like above.

Please note: When using library components you always get a schematics symbol and a physical shape for the PCB.



**SUBMENU FOR SYMBOLS**



Within this submenu, you may define some default values for the work with symbols.

- |                           |   |
|---------------------------|---|
| <b>IMMEDIATE KILL</b>     | Deletes symbols without a second <CR>   |
| <b>AUTO SEG. RE-ROUTE</b> | Automatical segment reroute, when moving a symbol on top of a segment.  |
| <b>CONNECTED MOVE</b>     | Not yet implemented   |
| <b>CONNECTED COPY</b>     | Using <b>MULTI COPY</b> the symbols and signals inside the selected area will be copied. The copied symbols and signals will get new names. |

CREATING A BUS

The screenshot shows the PRISMA software interface. The top-left window displays the following text:

```

PRISMA : Latus-window
SCS: modify bus          grid: 0.050_  ms= inch [280]
bus-name : DATA        fix bus : Y    kill bus : M
coordinate: /           sheet: 1      f: C
signal(s) : D0         signal(s) : □
    
```

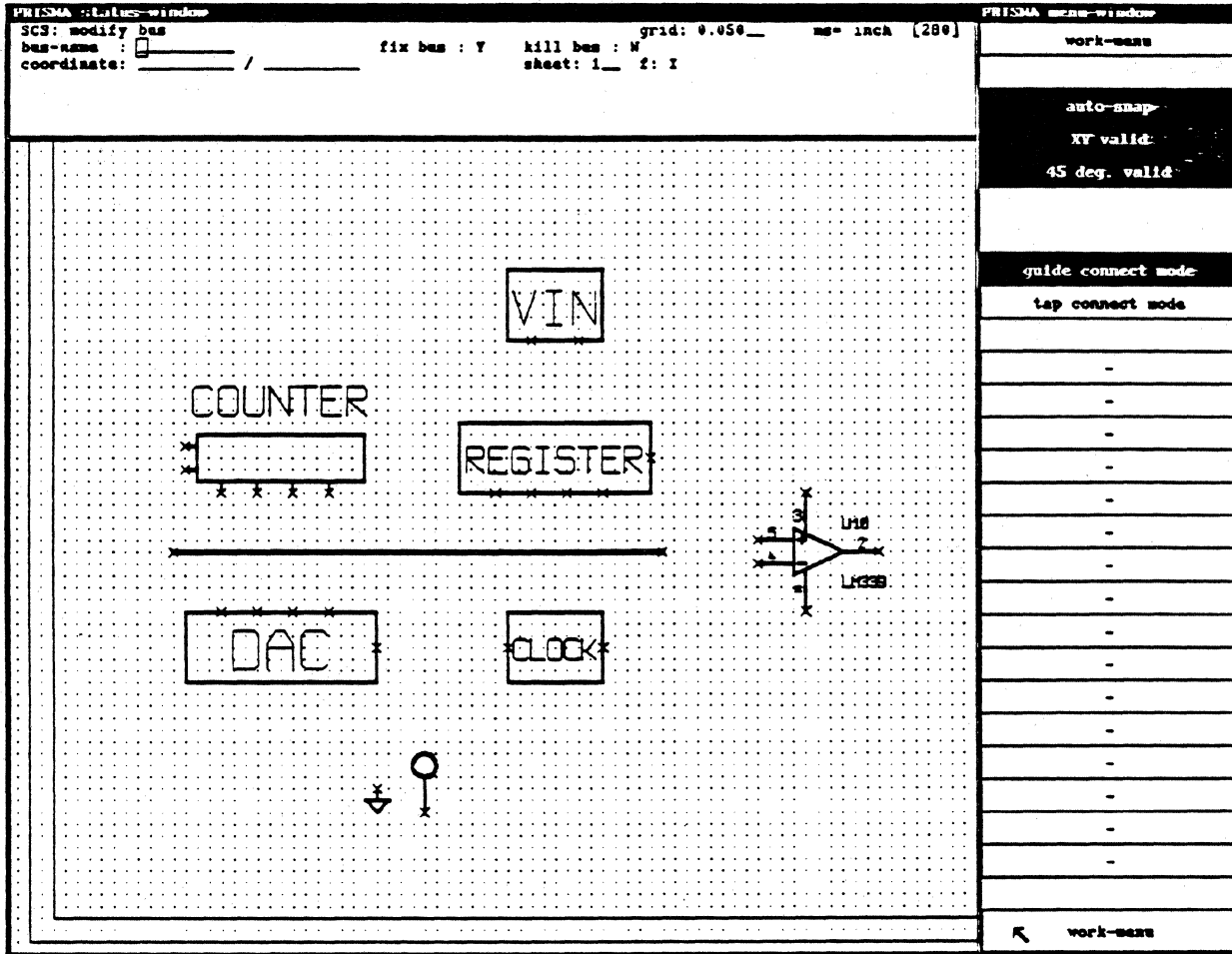
The main drawing area shows a schematic diagram on a grid. It includes several components: a box labeled 'VIN' at the top; a box labeled 'COUNTER' with four asterisks below it; a box labeled 'REGISTER' with four asterisks below it; a box labeled 'DAC' with four asterisks below it; and a box labeled 'CLOCK' with four asterisks below it. A horizontal line with arrows at both ends represents a bus, with the number '3' written above it. To the right of the bus is a triangular symbol with three lines extending from its vertices, labeled 'L123' and 'L1233'. At the bottom center, there are two small symbols: a circle with a downward arrow and a circle with an upward arrow.

The right-hand window shows a menu with the following items:

graphic	pan on/off
window	full view
window loc.	refresh
center	grid on/off
symbol	bus
terminal	relation
signal	
text	format
mod. graphic	mod. param.
get	create
insert	move
rename	modify
copy	delete
swap	replace
scale	list
digitize	change logic
open	close
offset	text
line	rot. point
circle	
route-mode	continue
multi	
sub-menu	main-menu

1. Select **BUS, CREATE**.
2. Type in the **BUS NAME "DATA"** in the upper menu.
3. Move the cursor to **SIGNAL(S)** and type in "D0", move the cursor one field to the right and type in "3".  
(To move the cursor you can use the arrows R8, R10, R12 or R14 on your keyboard; or move the arrow of the cursor into the selected field and <CLB>).
4. Move the cursor to the position 1 on the drawing screen and draw a line to position 2 (two times <CLB>).
5. You have just created a bus called **DATA** with four signals D0, D1, D2 and D3. Each **BUS** always needs its own name and own signals. A signal can only belong to one **BUS**.

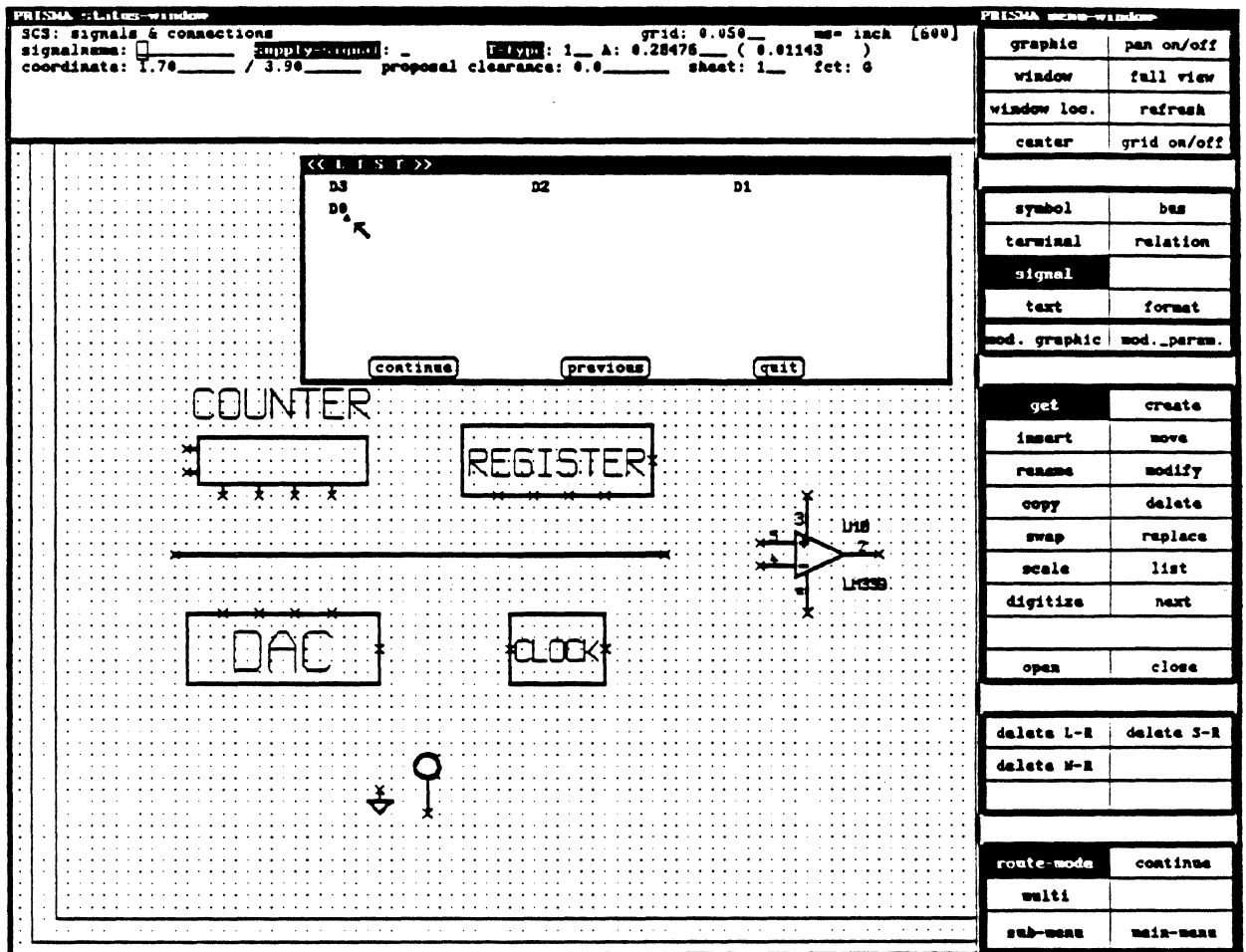
SUBMENU FOR BUS



Within the Submenu for Bus you may set some default values for the work with a Bus:

- |                           |   |
|---------------------------|---|
| <b>AUTO - SNAP</b>        | Automatical SNAP to the closest grid point.                             |
| <b>XY VALID</b>           | BUS may be drawn only horizontal or vertical                            |
| <b>45 DEG. VALID</b>      | BUS may be drawn only with 45 deg.                                      |
| <b>GUIDE CONNECT MODE</b> | No "T" connection possible.   |
| <b>TAP CONNECT MODE</b>   | Creates a "T" connection exactly at the selected position of the cursor |

## CONNECTING SIGNALS FROM THE BUS TO THE SYMBOL



1. Select **SIGNAL**, **GET** and **ROUTE MODE**, move the cursor to a position on top of the bus line and <CLB>. A **LIST** window will be displayed on the screen listing the bus signals D0 - D3.
2. Place the cursor inside this window at D0 and <CLB>. The **LIST** window will close and the selected signal name will be added in the upper menu at **SIGNAL NAME**, the function will change to **INSERT**. You are now ready to insert this signal onto pins.
3. Place the cursor on terminal 1 of symbol **COUNTER** and <CLB>. Because of the active **ROUTE MODE**, the system automatically draws a line between the terminal and the bus. Using **ROUTE MODE** off, only an airline appears.
4. The **SIGNAL NAME** will increment one step and you may now add signal D1 to terminal 2. Add the signals D2 and D3 to the terminals 3 and 4.
5. Repeat this procedure for the symbols **DAC** and **REGISTER**.

SIGNALS AND CONNECTIONS

The screenshot shows the PRISMA software interface. The top status bar displays: "PRISMA: signals & connections", "grid: 0.050", "ms= inch [600]", "signalname: [ ]", "supply-signal: [ ]", "I-type: 1", "A: 0.28476 ( 0.01143 )", "coordinate: [ ] / [ ]", "proposal clearance: 0.0", "sheet: 1", "fet: G". Below the status bar, it says "no signal selected". The main drawing area contains a schematic with components: VIN, COUNTER, REGISTER, DAC, and CLOCK. A bus is shown connecting the COUNTER and REGISTER. A cursor is positioned near the DAC. The right-hand menu has the following sections:

graphic	pan on/off
window	full view
window loc.	refresh
center	grid on/off

symbol	bus
terminal	relation
signal	
text	format
mod. graphic	mod. param.

get	create
insert	move
rename	modify
copy	delete
swap	replace
scale	list
digitize	next
open	close

delete L-R	delete S-R
delete W-R	

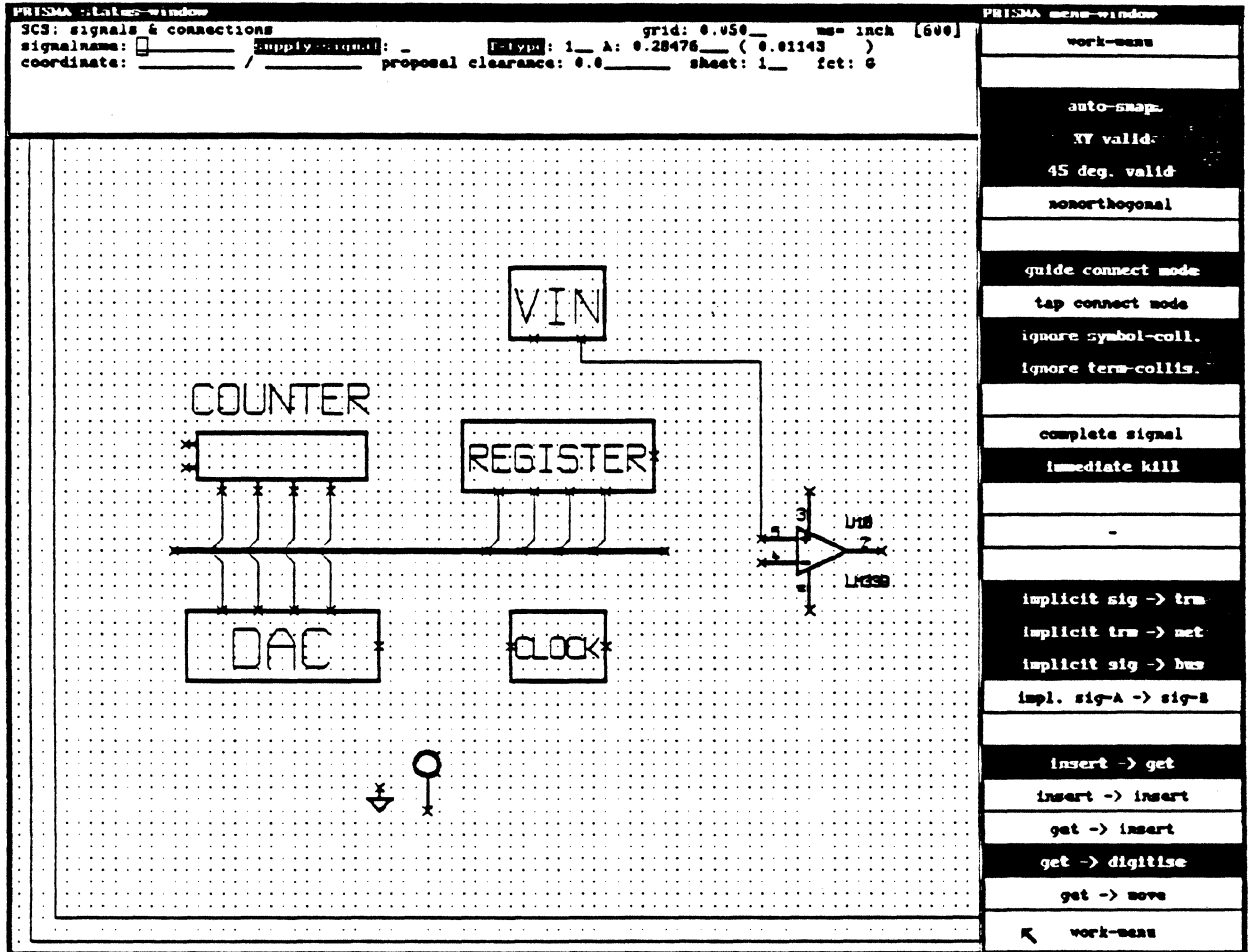
  

route-mode	continue
multi	
sub-menu	main-menu

Now you will see, how to add normal signals (not belonging to a bus) on the drawing.

1. Move the cursor to **ROUTE MODE** and <CLB>. **ROUTE MODE** should no longer be highlighted on the menu. **SIGNAL** and **GET** are still highlighted.
2. Place the cursor on VIN (1) and <CLB>. A new signal (#0) is automatically created, the name displayed in the status menu and an airlines starts at VIN (1) .
3. Move the cursor to U10 (5) and <CLB>. You created a signal #0 including VIN (1) and U10 (5) displayed as airline.
4. Select **DIGITIZE**, move the cursor to any position on top of the airline and <CLB>. Move the cursor to the right and <CLB>. You will see a yellow signal line and a shorter airline. <CLB> now you can digitize this signal manually to completion.

SUBMENU FOR SIGNALS



Within this submenu you may define some values for the interactive work with signals and connections.

**AUTO - SNAP**

automatical snap to the closest grid point.

**XY VALID**

Only horizontal and vertical lines valid.

**45 DEG. VALID**

Only 45 Deg., no hor. or vertical lines valid

**NONORTHOGONAL**

Any angle valid.

**GUIDE CONN. MODE**

A connection between two pins may be digitized

**TAP CONNECT MODE**

T-connections are possible.

**IGNORE ... COLL.**

Ignores collisions between digitized line and symbol body or terminal.

**COMPLETE SIGNAL**

not yet implemented.

**IMMEDIATE KILL**

Deletes a pin out of a signal or a complete signal without 2nd <CR>.

**IMPL. SIG -> TRM**

Creates a new signal onto the empty terminal.

**IMPL. TRM -> NET**

Inserts an empty terminal into a net.

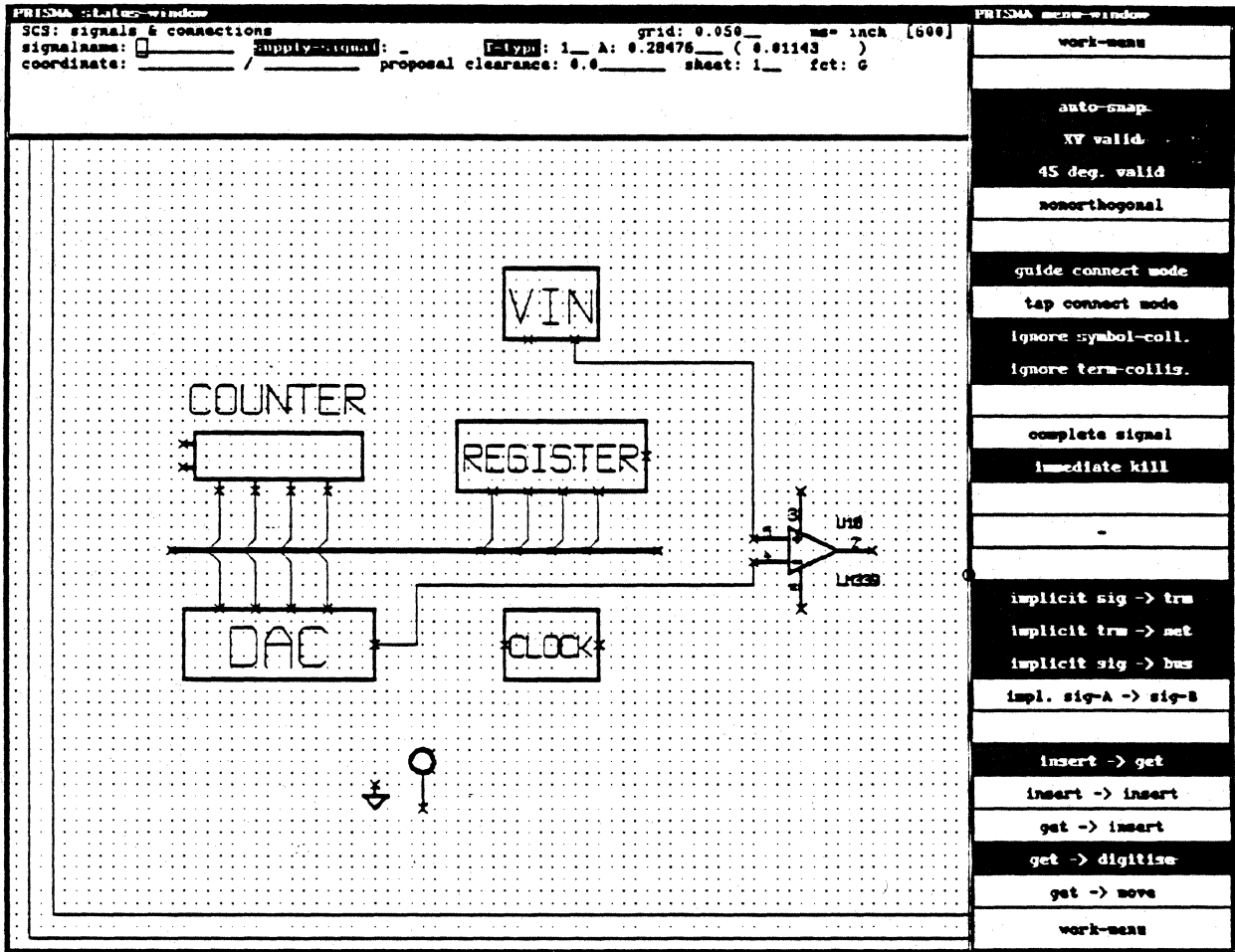
**IMPL. SIG -> BUS**

Insert a signal into a bus.

**IMPL. SIGA-> SIGB**

Adds signal A to signal B.

CREATING SOME NEW SIGNALS



1. Enter SUBMENU and change GET -> INSERT into GET -> DIGITIZE and go back to the WORK MENU.
2. Move the cursor to the right terminal of the DAC and <CLB>. A new signal will be created and a yellow line will be shown between the terminal and the cursor.
3. Move the cursor to the right and <CLB>, move the cursor up and <CLB>, connect the signal to symbol U10 (4).
4. Digitize a new signal between the output of U10 and the free terminal of REGISTER.

PRISMA :: Later-window  
 JCS: digitize connections  
 signalname: #1 coordinate: 3.650 / 3.10 grid: 0.050 ms= inch [810]  
 coordinate: 3.650 / 3.10 sheetnumber: 1 gradient: 0 f: D segment-type: A

PRISMA menu-window

graphic	pan on/off
window	full view
window loc.	refresh
center	grid on/off

symbol	bus
terminal	relation
signal	
text	format
mod. graphic	mod...param.

get	create
insert	move
rename	modify
copy	delete
swap	replace
scale	list
digitize	next

open	close
------	-------

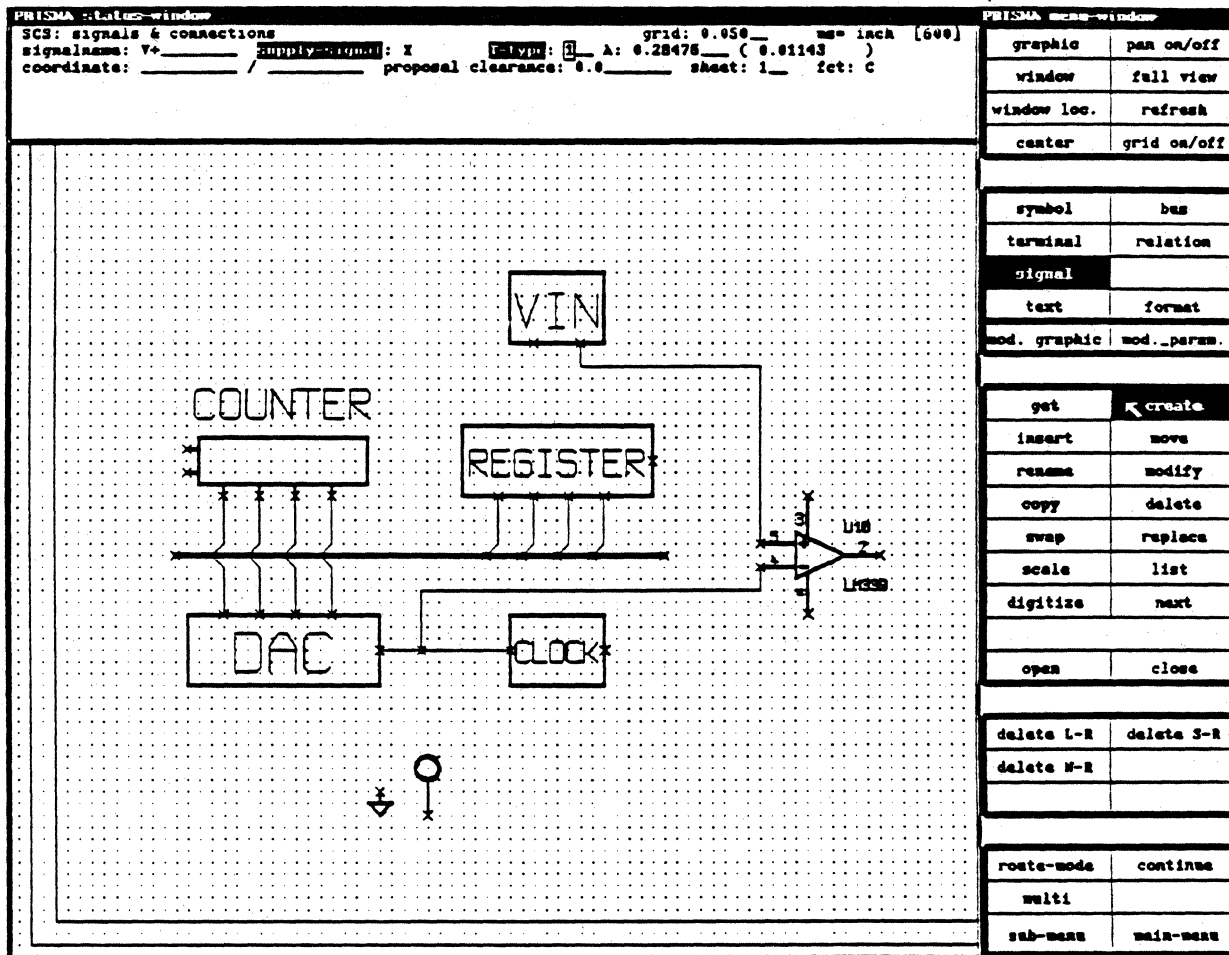
delete L-R	delete S-R
delete W-R	

route-mode	continue
multi	
sub-menu	main-menu

1. Place the cursor on top of the last signal at position 1 (shown in the diagram) and click the **MIDDLE** button of the mouse.
2. At the position of the cursor there will now be a terminal and a yellow line to the cursor.
3. Move the cursor up (position 2) and <CLB>.
4. Move the cursor to the free terminal of CLOCK and <CLB>. You have created a signal with three connected terminals.



SUPPLY SIGNALS

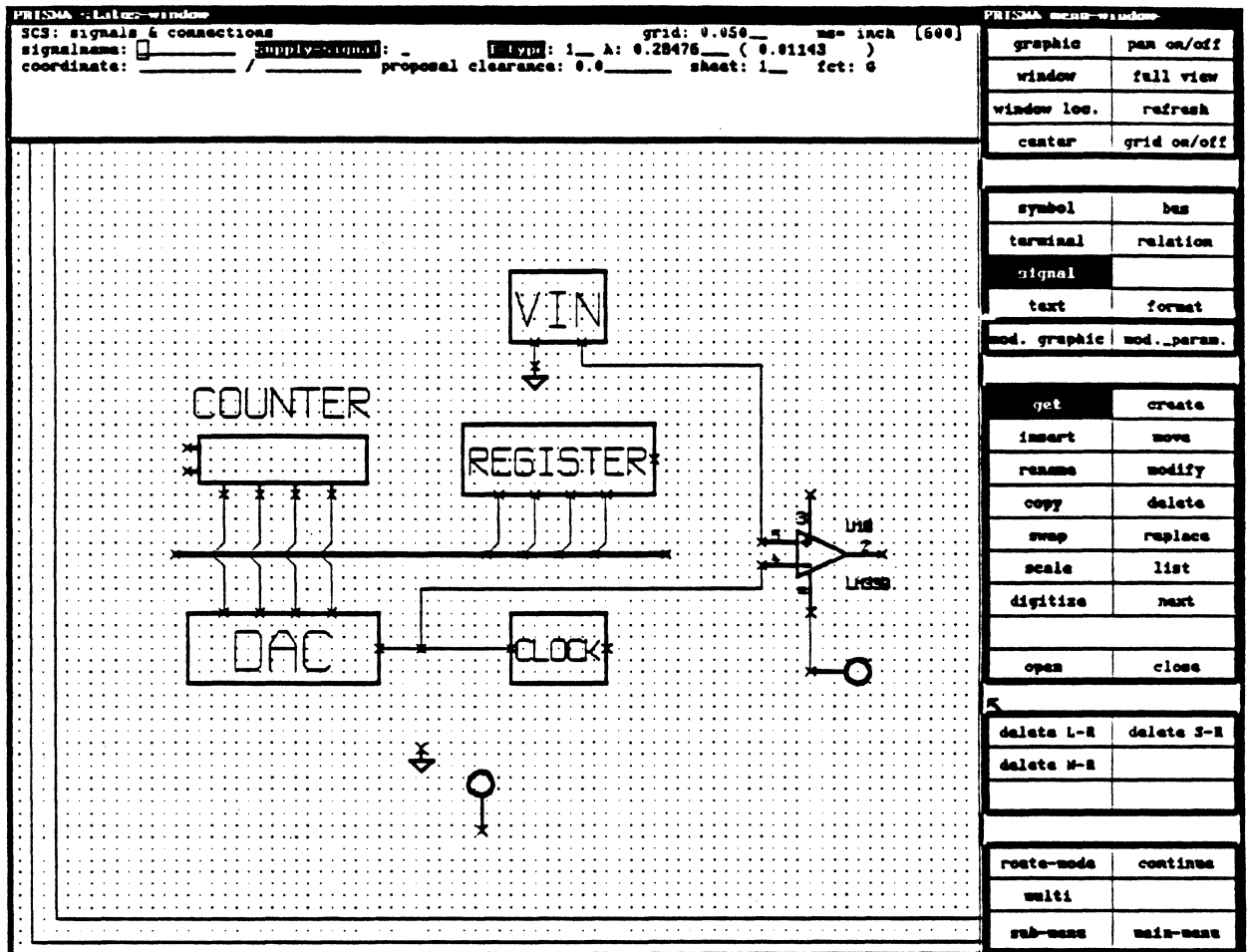


SUPPLY SIGNALS are signals, which are not always totally displayed on the screen. Terminals are automatically connected to SUPPLY SIGNALS, if they are already connected to pwr or gnd in the COLUMBINE LIBRARY SERVER. SUPPLY SIGNALS may also be displayed as signals between terminals and special SUPPLY SYMBOLS. To create SUPPLY SIGNALS, follow the steps:

1. Select SIGNAL and CREATE.
2. Type in the signal name "V+" in the status menu. Move the cursor to SUPPLY SIGNAL and type in "X" and <CR>. V+ now is created as SUPPLY SIGNAL.
3. Create the SUPPLY SIGNAL "V-" in the same way.

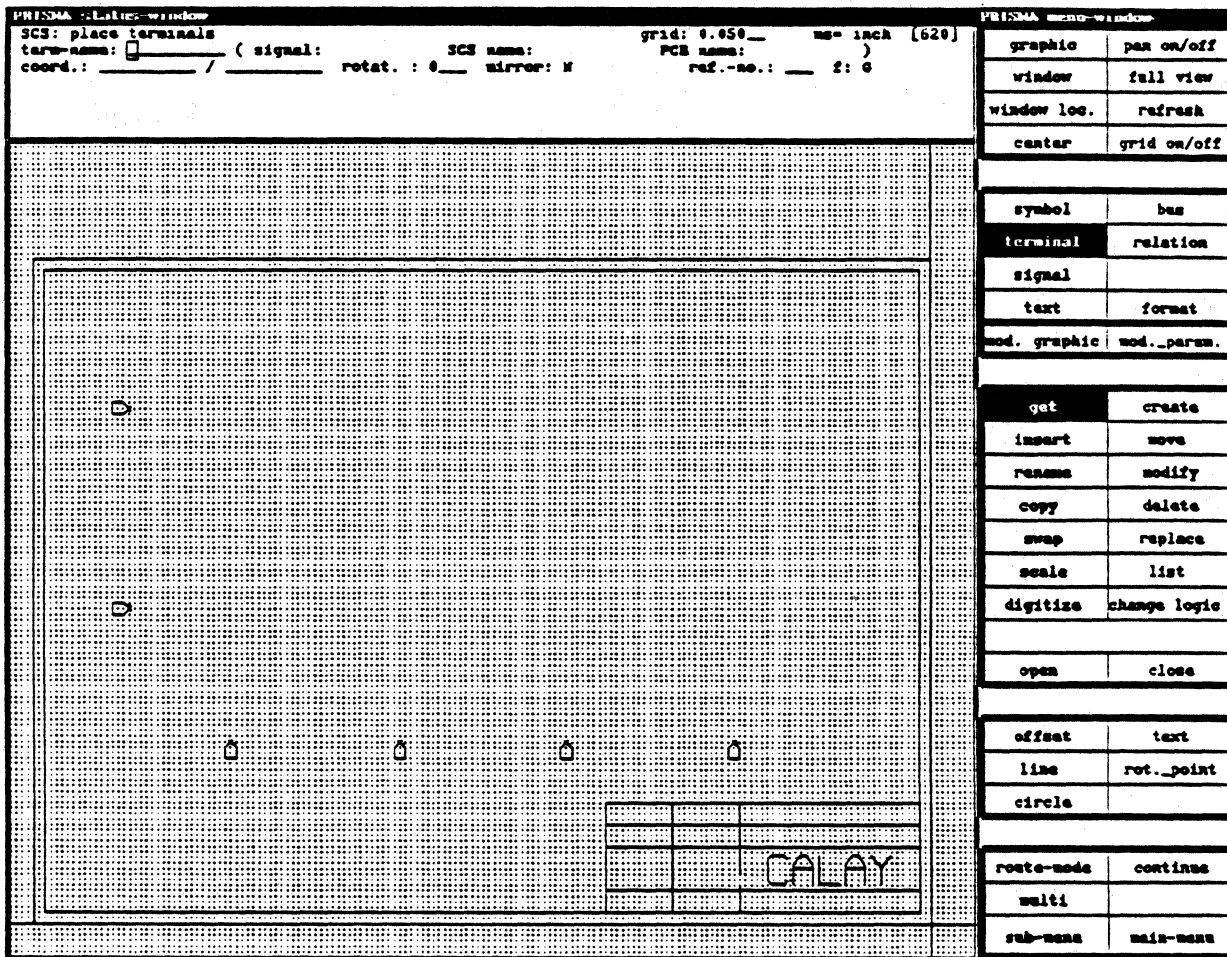
The signals "VCC" and "GND" are already created as SUPPLY SIGNALS in the technology file tec.adb.

## ADDING SUPPLY SIGNALS TO THE SCHEMATICS



1. Select **SYMBOL** and **GET**. Pick up your **SUPPLY SYMBOL**, select copy, move the cursor to a free position on your working sheet (close to the free terminal of VIN) and <CLB>.
2. Select **SIGNAL**. Move the cursor to the terminal of the **SUPPLY SYMBOL** and <CLB>. The **SIGNAL NAME** in the status menu is now **GND**.
3. You can digitize (select **DIGITIZE** if necessary) this signal now to the unconnected terminal of VIN.
4. Move the cursor to the terminal of the other **SUPPLY SYMBOL** and <CLB>. Digitize the signal to U10 pin 12.
5. Select **SIGNAL**. Move the cursor to any position of the **GND** signals and <CLB>. The complete **GND** signal will be **HIGHLIGHTED**.
6. Select **INSERT** and **LIST**. Inside a new **LIST WINDOW** there will be a list of those pins, which are connected to **GND**. Use **QUIT** to exit this **LIST WINDOW**.

MOVING DOWN ONE LEVEL IN THE HIERARCHY

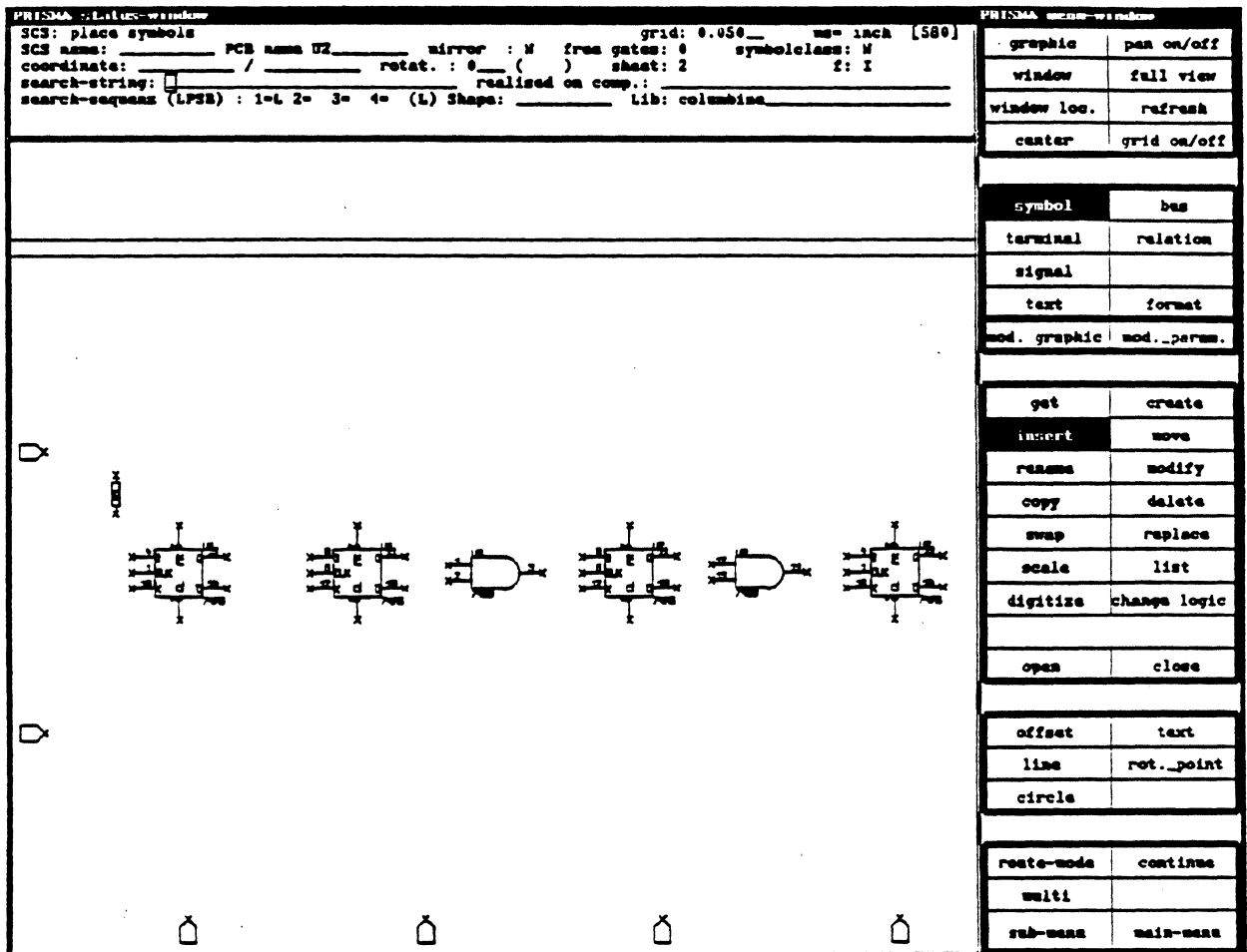


PRISMA allows to work in up to 255 HIERARCHY LEVELS or on up to 255 PAGES. Each level of HIERARCHY may include a complete drawing with its own symbols and signals. How to enter the next level of hierarchy:

1. Select **SYMBOL** and place the cursor within the symbol COUNTER and <CLB>.
2. Select **OPEN** from the menu. The screen will refresh, and you will see the six terminals associated with the COUNTER. You are now inside the COUNTER and down one level in the hierarchy.

The next step will be to design the COUNTER circuit using flipflops and gates.

## PLACING SYMBOLS FROM THE COLUMBINE LIBRARY SERVER



1. Select **SYMBOL** and **INSERT**.
2. In the **STATUS MENU** at **SEARCH STRING** type in "7476".
3. Place the cursor at **SCS NAME** and type in "U1" and move the cursor to **PCB NAME** and type in again "U1" and <CR>.
4. A 7476 flip-flop is now attached to the cursor. Place it at position 1 using <CLB>.
5. Select **COPY** and place the next flip-flop at position 2.
6. Repeat the **COPY** function for position 3 and 4 as well.
7. Select **INSERT** and type in at **SEARCH STRING** "7408". Type in at **SCS NAME** and at **PCB NAME** "U3". Place one gate at position 5, select **COPY** and place another gate at position 6.
8. Select **SYMBOL** and **INSERT**. Type in at **SEARCH STRING** "RES", at **SCS NAME** and at **PCB NAME** "R1" and <CR>
9. A resistor is now attached to the cursor. Press the middle mouse button to rotate the resistor 90 degrees.
10. Move the resistor to position 1 and <CLB>.

DIGITIZING SIGNALS

PRISMA --Laluz-window

SCS: signals & connections grid: 0.050 mm inch (600)

signalname:  supply-signal: - type: 1 A: 0.28476 ( 0.01143 )

coordinate:  /  proposal clearance: 0.0 sheet: 2 fet: 0

PRISMA menu-window

graphic	pan on/off
window	full view
window loc.	refresh
center	grid on/off
symbol	bus
terminal	relation
signal	
text	format
mod. graphic	mod. param.
get	create
insert	wave
rename	modify
copy	delete
swap	replace
scale	list
digitize	next
open	close
delete L-R	delete S-R
delete M-R	
route-mode	continue
multi	
sub-menu	main-menu

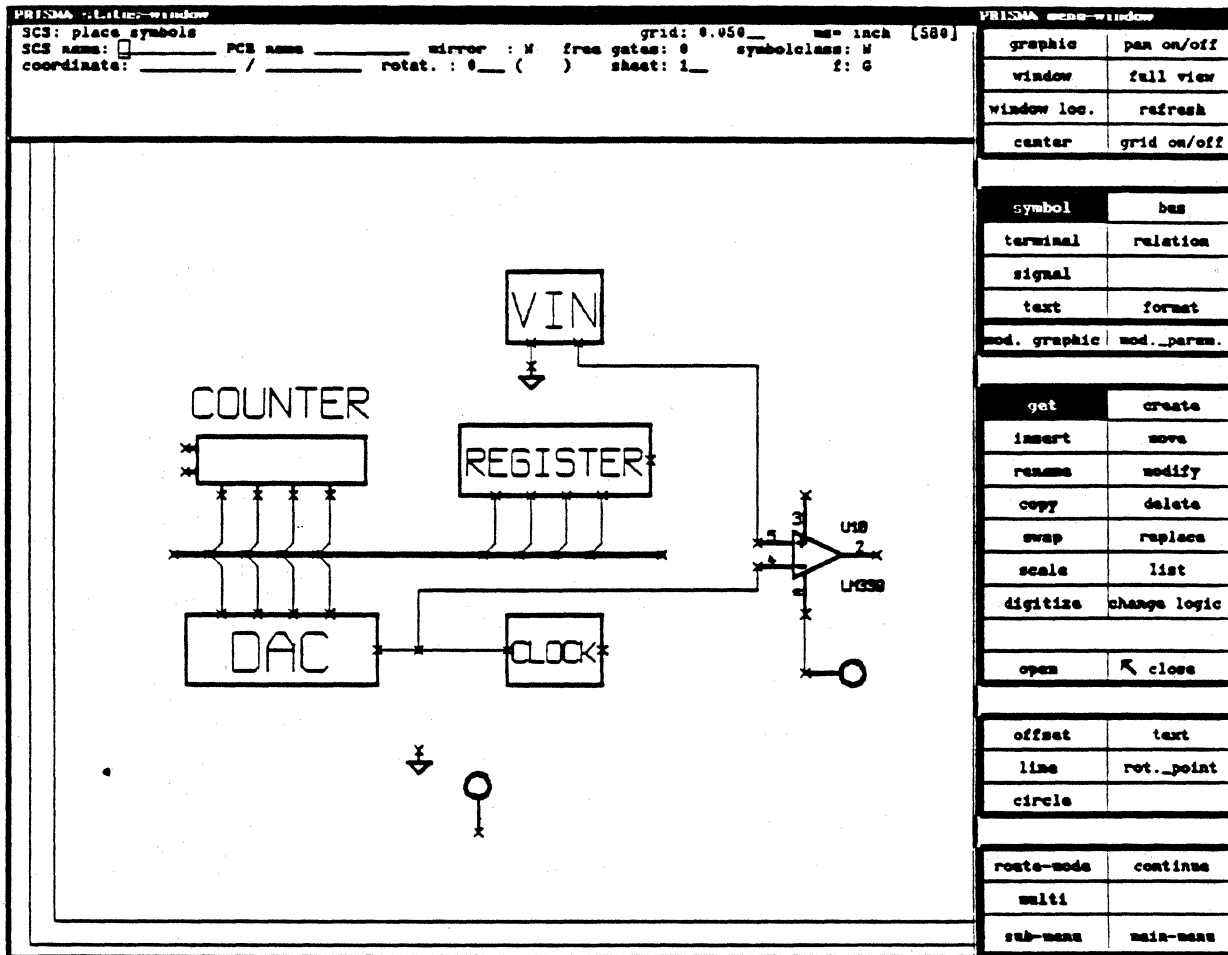
- We will now create signals for the internal construction of the COUNTER.
1. Select SIGNAL and GET, enter the SUBMENU, select GET DIGITIZE and go back to the WORK MENU.
  2. <CLB> at the lower terminal of the resistor, move the cursor downwards and digitize a connection to the "J" input of the first flip-flop.
  3. Start at the "K" input and digitize to the "J" input as shown.
  4. Create a new signal between the "Q" output of the first flip-flop to the "J" input of the second flip-flop. Using the middle button connect this signal to the "K" input. Move the cursor to the input of the first AND gate and <CLB>. Digitize the signal to the tie point of the last signal.
  5. Connect the "Q" output to the second input of the AND gate.

PRISMA : List: window  
 SCS: signals & connections grid: 0.050 mm inch [500]  
 signalname: [ ] supply: [ ] type: 1 A: 0.28476 ( 0.01143 )  
 coordinate: [ ] / [ ] proposal clearance: 0.0 sheet: 2 fct: 0

graphic	pan on/off
window	full view
window loc.	refresh
center	grid on/off
symbol	bus
terminal	relation
signal	
text	format
mod. graphic	mod. param.
get	create
insert	move
rename	modify
copy	delete
swap	replace
scale	list
digitize	next
open	close
delete L-R	delete S-R
delete M-R	
route-mode	continue
multi	
sub-menu	main-menu

6. Connect the output of the first AND gate to the "J" and "K" inputs of the 3rd JK flip-flop. Digitize this signal also to the input of the second AND gate.
7. Connect the "Q" output of the 3rd flip-flop to the second input of the AND gate.
8. Digitize a signal from the output of the AND gate to the "J" input of the 4th flip-flop. Use the middle button to connect the "K" input to the same signal.
9. Place the cursor at the upper left terminal and <CLB>. Digitize a signal to the "CL" inputs of all four flip-flops.
10. In the same way digitize a connection from the lower left terminal to all "CLK" inputs of the flip-flops.
11. Connect the bottom terminals each to an existing "Q" output signal of the flip-flops. (Start at the terminal or use the middle mouse button.)

MOVING BACK TO THE TOP LEVEL OF HIERARCHY



Now the internal description of the COUNTER is ready. To enter the top level of the hierarchy select **CLOSE** and <CR>. The screen will refresh and displays again the original drawing. Now you can select another symbol, **OPEN** this symbol and create the internal drawing of this symbol in the same way we have done it with the symbol COUNTER.

---

**OTHER USEFUL FUNCTIONS OF THE SCHEMATICS****Symbol MOVE, COPY, RENAME, DELETE, ROTATE**

Select **SYMBOL** and **GET** before you use the following functions.

**MOVE**

- Left button:** Pick up the symbol inside the symbol frame, the function defaults to **MOVE**. The symbol is now attached to the cursor. Move the cursor to the new position and place the symbol, the function defaults back to **GET**.
- Middle button:** If a symbol is selected, you may **ROTATE** the symbol in 90 deg. steps ccw.  
If no symbol is selected, you will get a new window including a **LIST OF UNUSED SYMBOLS**.
- Right button:** Same function as the left button, but after placing the symbol the function stays at **MOVE**.

**COPY**

- Left button:** Pick up the symbol inside the symbol frame, the function defaults to **MOVE**. Now change the function to **COPY** (on the **FUNCTION MENU**). Move the cursor to the new position and place the copied symbol. The function defaults to **MOVE** then. The symbol name has incremented one step.
- Middle button:** Same function as the middle button for **MOVE**.
- Right button:** Same function as the left button, but after placing the copied symbol, the function stays at **COPY**. You may continue to have more copies of the selected symbol.

**RENAME**

The function **RENAME** allows to change the **SYMBOL NAME** of the schematic and/or the **COMPONENT NAME** of the PCB layout at the same time. The functions of the left and right button are the same.

After the selection of a symbol the **SCS NAME** and **PCB NAME** are displayed in the **STATUS MENU**. Select **RENAME** now, type in the new names for **SCS NAME** and/or **PCB NAME** and **<CR>**. The new names will be displayed on the screen.

**DELETE**

Select the symbol you want to delete and then the function **DELETE** on the work menu. Depending on the function **IMMEDIATE KILL** (on/off, to be changed within the **SUBMENU**) the symbol is deleted or you must add a **<CR>**. Use only the left button.



## **ROTATE**

Each symbol may be rotated in steps of 1/10 degree. Using the middle mouse button, you may rotate symbols in steps of 90 degree counter-clockwise (ccw). The actual rotation of a symbol is displayed in the **STATUS MENU** (in 1/10 degree units).

To rotate a symbol, select it with the left or right button, move the cursor into the **ROTATE** field in the **STATUS MENU** and type in the angle you want to rotate ccw and place the symbol with the mouse button or <CR>. Select the symbol again and the **ROTATE** field will display the actual rotation of the symbol.

## **MIRROR**

To **MIRROR** a symbol, select it with the left or right button, type in "Y" in the **MIRROR** field in the **STATUS MENU** and place the symbol. It will be mirrored around the X-axis. The function is different from rotating a symbol twice.

## **SCALE**

If a symbol appears too small or too big, **SCALE** allows to change the size of the symbol. Select the symbol and <CLB> at **SCALE** on the **FUNCTION MENU**. A frame will be created on the screen. You may re-size that frame by moving the cursor. After you have determined the correct new size, click one of the mouse buttons and the symbol will be displayed with its new size inside this frame.

---

**MODIFY GRAPHIC WITH OFFSET, LINE, CIRCLE, ROTATION POINT**

**MODIFY GRAPHIC** is a global function for the manipulation of existing symbols graphics. To use this function, select a symbol and **MODIFY GRAPHIC** on the **FUNCTION MENU**. Now you can modify any element of the symbol using the following functions of the **FUNCTION MENU**:

**GET, MOVE, COPY, DELETE, DIGITIZE**

Using **DIGITIZE** you may add graphic elements to the symbol (to be selected on the **FUNCTION MENU**):

- OFFSET** Default function. The system expects the first coordinate to be a **LINE** and changes the function to **LINE** then.
- LINE** Allows to add graphic lines to the symbol according to the allowed angles defined in the **SUBMENU**.
- CIRCLE** Creates circles or circular elements using the mouse or keyboard inputs in the **STATUS MENU**.
1. Place the left button at the coordinate you want to start the circle. Move the cursor, there will be a circle attached to the cursor, touching the start coordinate. Click the middle button in the center of the circle and the circle is drawn. Using the right button you can select the angle of the circle in steps of 90 degrees before placing it by using the middle button.
  2. Type in the angle of the circle in the **STATUS MENU**; you now may place this circle element with the left button (start coordinate) and the middle button (center).
- TEXT** You may select one of the following **TEXT TYPES**:
- |                      |   |
|----------------------|---|
| <b>FREE TEXT</b>     | May be entered in the <b>TEXT</b> field on the <b>STATUS MENU</b> . |
| <b>SCS NAME</b>      | Name of the schematic symbol.                                       |
| <b>PCB NAME</b>      | Name of the PCB component.  |
| <b>SYMBOL</b>        | Logical symbol type (nand).   |
| <b>FUNCTION TYPE</b> | Function type (7400).   |

To select a **TEXT TYPE** click the right button and step through the different types. Place the text by pressing the left or middle button at the diagonal coordinates of a rectangle on the screen. If the information exists it will be displayed immediately, otherwise there will be a frame for the text field reservation.

**ROTATION POINT**

The rotation point is the placement coordinate of each symbol. The cross hair of the cursor always is the rotation point of the symbol. A new symbol has its **ROTATION POINT** in the lower left corner of the symbol frame. To re-locate the rotation point select

**ROTATION POINT** on the **FUNCTION MENU**. The existing rotation point will be displayed as a big cross on the screen. Move the cursor to the new coordinate and press any of the mouse buttons. The **ROTATION POINT** has moved to this new coordinate.

## **REPORT**

Enter **MAIN MENU** and select **MISC**. Select **REPORT** on the new submenu. You may get statistic information about:

<b>COMPONENTS</b>	List of all components including shapes, used and free gates
<b>TECHNOLOGY</b>	List of track technologies (PCB).
<b>SIGNALS</b>	List of signals including page, number of terminals, length of connection.
<b>CONNECTORS</b>	not yet implemented.
<b>PCB SCS</b>	Relation between PCB and SCS.
<b>STATISTICS</b>	List of used elements.

To select one of these functions, move the cursor to the function on the **FUNCTION MENU** and <CLB>. CIS displays a **LIST WINDOW** on the screen including the requested information. To continue use **QUIT**.

## **MULTI SHEET SCHEMATICS**

Each schematic may have up to 255 pages. These pages may be within one or more levels of hierarchy. Page 0 is used for the spare gates. You may have a list of these gates by clicking the middle button.

The schematic drawing starts on page 1. If you want to call up a new page, please follow the next steps:

1. Select **MISC.** from the **MAIN MENU**. Call up **NEW SHEET** next or select **FORMAT** from the **WORK MENU**.
2. Press the right button. A **POPUP MENU** should be displayed at the cursor's position. (If not, press **R11** to activate the **POPUP MENU** and press the right button again).
3. Hold the button down and move the light bar inside the **POPUP MENU** onto the function **NEW SHEET**.
4. Release the button now and type in the new page number at the position **SHEET-NO** on the **STATUS MENU** and **<CLB>**.

When entering a new level of hierarchy, you will start on a new page automatically.

## **CHANGE FROM ONE PAGE TO ANOTHER**

Select **GRAPHIC SETUP** from the **MAIN MENU** or **FORMAT** from the **WORK MENU**. Place the cursor at **SHEET NUMBER** and type in the page you want to display. This page must have been created already, otherwise you can't display it.

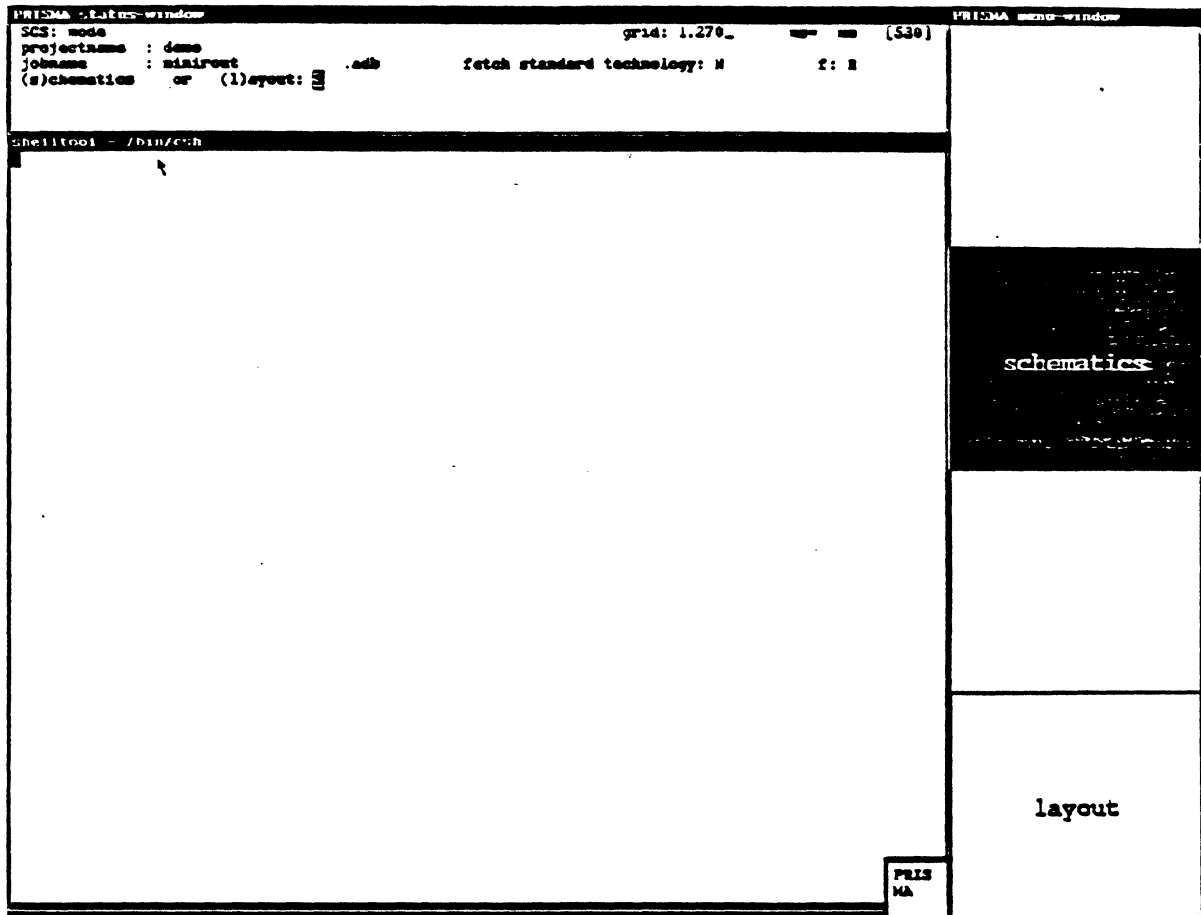
## **SHEET TO SHEET TAGS**

Select **TERMINAL** and **CREATE**. Place the cursor at the position you want to have the sheet to sheet tag and **<CLB>**. At the same position on the new page there will also be a sheet to sheet tag.

Select the **SIGNAL** you want to continue on another page and digitize it onto this sheet to sheet tag. The signal name will be displayed in a special textfield at the sheet to sheet tag. Change the page now, select **TERMINAL**, pick up the sheet to sheet tag and move it to the position you want to have it. Select **SIGNAL** and **<CLB>** at the sheet to sheet tag and continue to digitize this signal.

The following section will take you through the PCB Layout side of Prisma.

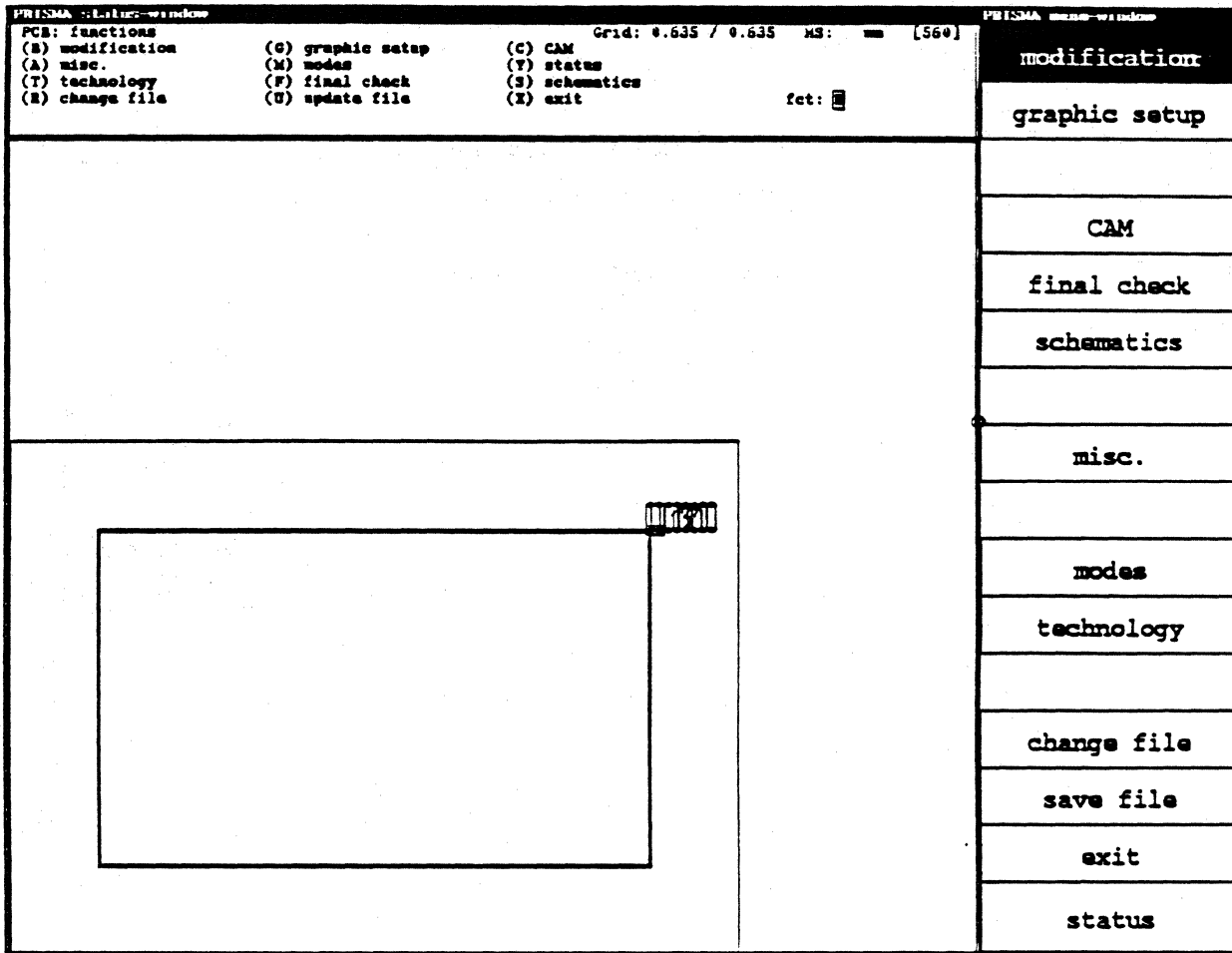
## ENTERING THE PCB LAYOUT



1. Start the system and enter CIS.
2. Read in a file (i.e. example.adb created during work in the schematics).
3. Select LAYOUT with the left or middle button or type in „L” on the STATUS MENU and <CR>.

The layout will be displayed on the screen. We will see the standard board outline. All components are placed in the upper right corner, one on top of the other.

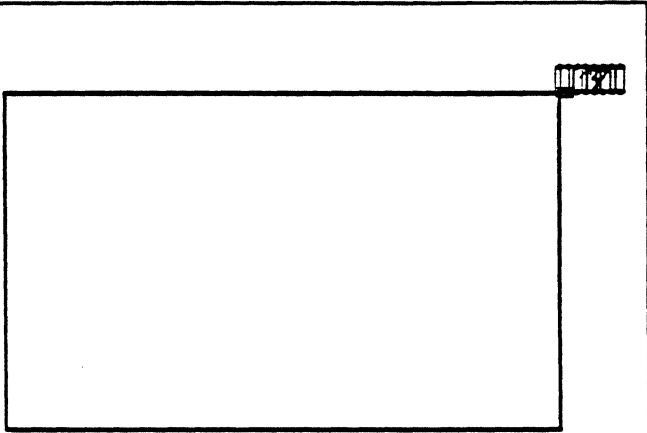
**MAIN MENU**



The **MAIN MENU** of the **PCB LAYOUT** looks similar to the **MAIN MENU** of the **SCHEMATICS**. Using the mouse or key letters you may select one of the following functions:

- |                      |  |
|----------------------|--|
| <b>MODIFICATION</b>  | Interactive working mode   |
| <b>GRAPHIC SETUP</b> | Sel. of screen display, layers, colors, grids, etc.                    |
| <b>CAM</b>           | Features for changing the form of connections.                         |
| <b>FINAL CHECK</b>   | Layout check for shorts and opens.                                     |
| <b>SCHEMATICS</b>    | Display of schematics (jumps to the schematics)                        |
| <b>MISC.</b>         | Submenu for more global functions (measurement, plots, relation, etc.) |
| <b>MODES</b>         | Def. of display modes (panning, system units, etc.)                    |
| <b>TECHNOLOGY</b>    | Setup for technology (tracks, vias, text, layers)                      |
| <b>CHANGE FILE</b>   | Not yet implemented, please use <b>EXIT</b> and restart.               |
| <b>SAVE FILE</b>     | Updating the file.   |
| <b>EXIT</b>          | Leaving the program with or without updating.                          |

MODES

<pre> PRISMA - Later - window PCB: global parameters          Grid: 0.025 / 0.025  MS: 1ach [430]  units : (I)ach (M)m (C)=1/20"  searchrange+- :  x: 0.00929_  y: 0.00929_ cursor: elements plus crossh.: X  plus terminals: X  digit.-cursor plus crosshair: X text  : default height: 0.0_____ automat. width: _ areas : auto - close : X  digit.: auto - rooting: X      gradients : 1 (0=X/Y only; 1=X/Y+1:1; 2=no restr.) misc. : RTI : X      popups : _ lower case: -  standard-panic-mode (P,W): W  auto-panning-size: _ or fullsize: X panning-area (pixels) in X: 1600 (min=1152 max=3456) in Y: 1200 (min=900 max=2700) preference-value: 900_ pixels be adequate to 12.0_____ inch                 </pre>	<p>modification</p> <p>graphic setup</p> <p>CAM</p> <p>final check</p> <p>schematics</p> <p>misc.</p> <p><b>modes ←</b></p> <p>technology</p> <p>change file</p> <p>save file</p> <p>exit</p> <p>status</p>
	

Select MODES from the MAIN MENU and <CLB>.

Within this menu you may setup how to display the PCB layout on the screen. The cursor can be displayed with/without crosshair or pins during placement and digitizing.

**Search range** Swap area of the cursor when sel. an element  
**Text:Default height** defines a standard text height for user defined text.  
**Automat. width** assigns the length of a text field according to the number of characters.

**Basic units:** Place the cursor at SYSTEM UNITS and type in:

I = Inch or M = Metric or C = 1/20 Inch

**Panning:** Same function as in MODES of SCHEMATICS.

P = Panning W = window



GRAPHIC SETUP

PRISMA - Left window		Grid: 0.025 / 0.025		MS: inch [570]		PRISMA - main window	
PCB: graphic setup							
set: V	components: <input checked="" type="checkbox"/> (FY)	nets: X	pads: X	X-all	Y-free	work-menu	main-menu
	terminals: X (ACQM)	grid: -	frame: X (ZOR)	C-comp.	J-sign.	window	full view
	segments: X (VFPRT)	airlines: -	restr.: X (YCB)	P-pads	T-term.	window loc.	refresh
	vias: X (VFPV)	copper: X	texts: CPT (CPTD)	P-fix	V-var.	center	grid on/off
layer: COMP	X	remaining	X	U-proposals		restore	zoom
	SOLD	X		H-hattrap			
				A-adjustments			
				G-plus spots		temp-grid	disp-grid
				Q-test spots		digit-grid	place-grid
				M-milling sp.			
				B-via block		0	1
				Z-sheet		2	3
				O-outline		4	5
window: x: -1.0	y: -1.0	// x: 13.02568	y: 9.77101			6	7
comp.:						8	9
						nb.-start	nb.-stop
						colour on	colour off
						colour_clear	
						layer on	layer off
						display mode	via mode
						0/0	coos on/off
						colours	def. colours
						work-menu	main-menu

Within this GRAPHIC MENU you will find the definitions for the complete board display. You may select what is to be displayed and how to display these elements. Within the GRAPHIC SETUP MENU, you will find the elements and layers to be displayed. On the right hand side there are codes which should be entered at the text field behind the elements and layers, i.e:

- "X" behind COMPONENTS displays all existing components.
- "V" behind SEGMENTS displays variable track segments.
- "X" behind LAYER displays the selected layer.

WINDOW displays the size of the displayed working window. The definition may be done with the cursor (lower left corner and upper right corner) or manual by typing in the absolute coordinates of the window.

A SEARCH FUNCTION refreshes the screen and displays the entered COMPONENT and PIN (last row) in the middle of the screen.

GRAPHIC SETUP (continued)

PRISMA -- status -- window										PRISMA -- status -- window													
PCB: element representation & color										Grid: 0.025 / 0.025 MS: inch [575]													
set: v	variable:	trk. vias	CR (FC,1-4,AD)	comp.	: R_ (RM)	grid: 2 (1,2)	C-cont. F-fill			work-menu	main-menu												
fix :	C3R	CR (FC,1-4,AD)		pads :	R_ (RM)	top layers: 2	Normal D-dist.			window	full view												
pads :	F1R	FR (FC,1-4,AD)		vias :	R_ (RM)					window loc.	refresh												
proposal:	C2R	CR (FC,1-4,AD)		copper :	C_ (RPCM)	highl. techn.	M-plus mark			center	grid on/off												
terminal:	C1R	CR (FC,1-4,AD)		restr. :	R_ (RM)	wrong: N				restore	zoom												
heattrap:	F	(FC)																					
colour:	airlines	17	highlight	16	frame	19_	remaining	19_	1-solid														
	COMP	19_							2-dotted	temp-grid	disp-grid												
	SOLD	19_							3-short dashed	digit-grid	place-grid												
									4-long dashed														
												0		1									
												2		3									
												4		5									
												6		7									
												8		9									
												nb.-start		nb.-stop									
												colour on		colour off									
												colour_clear											
												layer on		layer off									
												display mode		via mode									
												0/0		coos on/off									
												colours		def. colours									
												work-menu		main-menu									

1. Select GRAPHIC SETUP and Colors.

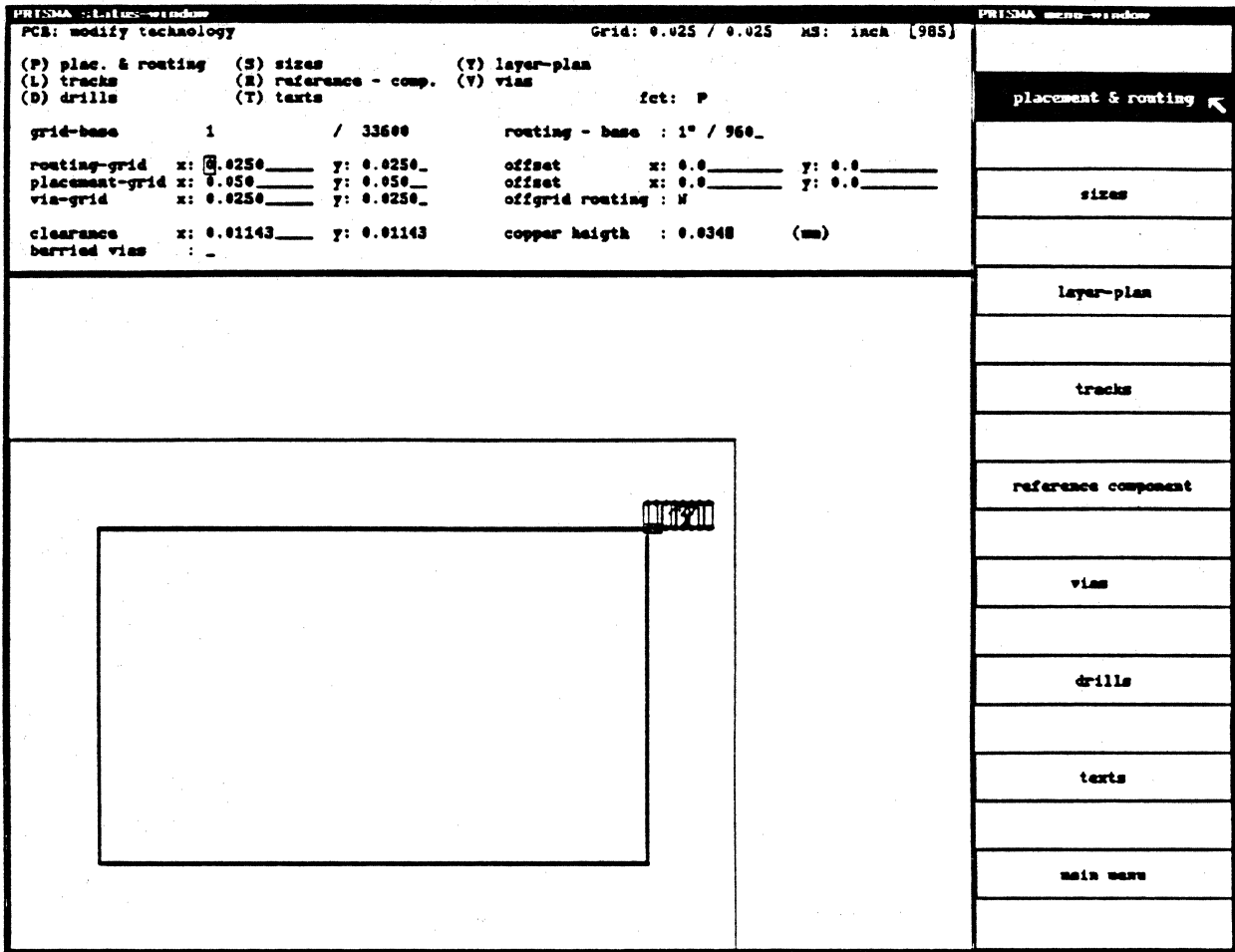
This menu allows to define how to display the selected elements. The definitions must be done with short keys which are described in the right part of this menu. The possible inputs are shown in brackets behind the elements.

The Color TABLE defines up to 4 layers with different colors and the remaining layers are displayed in the background. All layers can be viewed pairwise using key "F5".

Using the TOP LAYER you may enter 1, 2 or 4 to define the number of layers which will be displayed with their defined colors and all other layers will be displayed in the background. The key "F5" allows to toggle through the existing layers.

COOS ON/OFF displays the coordinates of the cursor position in the STATUS MENU. The function 0/0 defines a new reference point. All displayed values are based on this coordinate then.

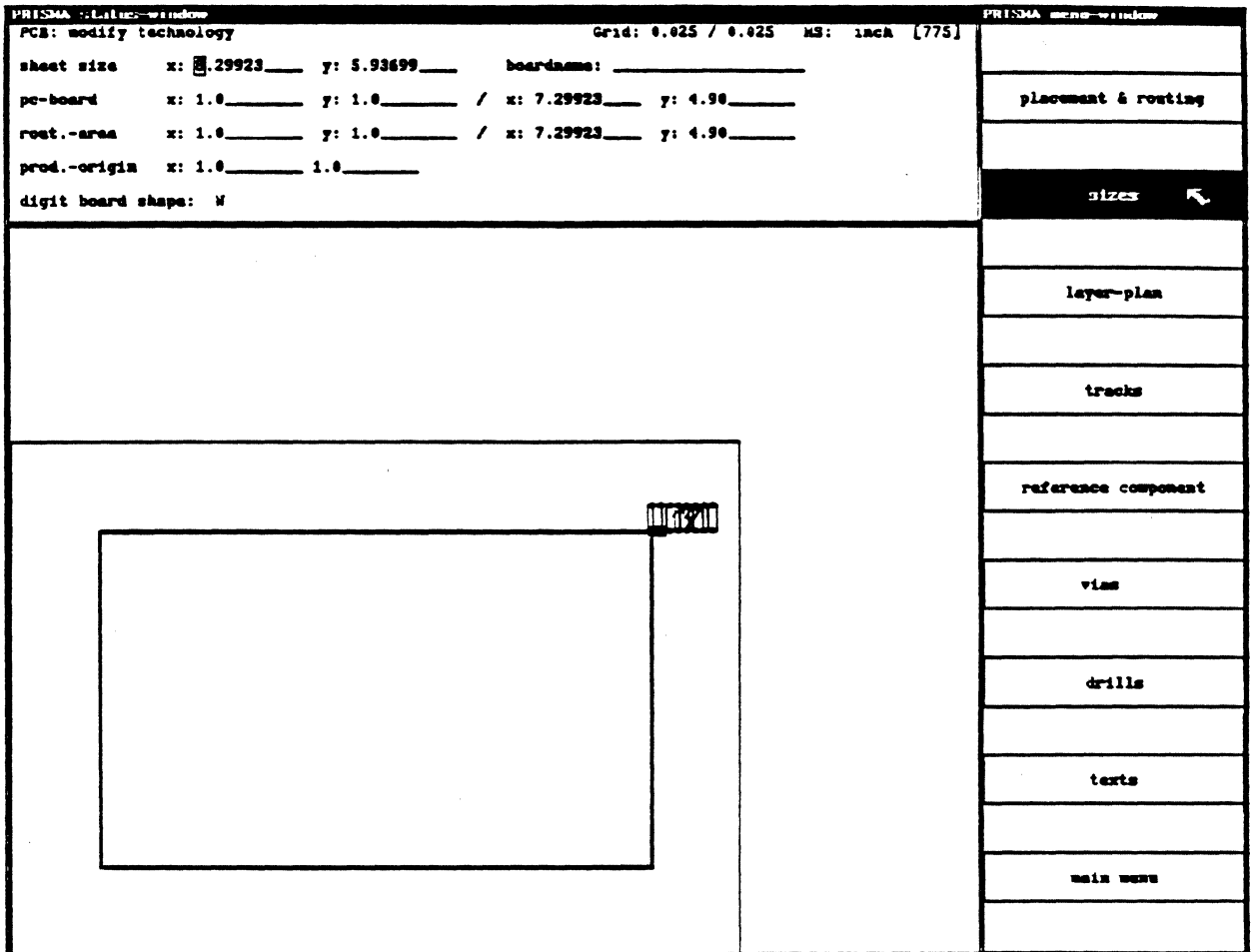
TECHNOLOGY      PLACEMENT AND ROUTING



Select TECHNOLOGY and PLACEMENT & ROUTING from the MAIN MENU

- Grid Base      Fixed system grid base for interactive work
- Routing Base      Grid Base for automatic routing (default 1/960")
- Routing Grid      Grid for autom. and interactive routing (for autom. routing it must be a multiple of the Routing Base).
- Placement Grid      Grid for manual and autom. component placement
- Via Grid      Grid for automatical via placement for Ldrou.
- Clearance      Global value for the minimum distance between tracks, vias, areas of different signals.
- Buried Via      Allows the use of buried vias.
- Shifted Rout.      Used for autom. routing with shifted tracks and vias in 1/60 inch.

SIZES



- Sheet Size** Total working area including PCB area, measurements and all other elements of the file.
- Boardname** Input field for user text.
- PC - Board** Size of a rectangle PC - board area. The **PC-BOARD** must be smaller or equal to the sheet size.
- Rout. Area.** Max. routing area for **LDROUT** , smaller or equal to the **PC BOARD**.
- Prod. Origin** Definition of a point within the **SHEET SIZE**. All coordinates of **CIS** are based on this point.
- Digit. Board** To digitize a non rectangle board outline, use the cursor or coordinates input and create a closed frame.

All coordinates of this page are **ABSOLUTE COORDINATES** based on the lower left corner.

LAYER PLAN

PRISMA --Layer-- window      PRISMA menu window

PCB: change layer plan      Grid: 0.025 / 0.025    MS: inch [770]

act: Y    modify no.: 0    (C) check, (W) write: \_

0 key	name	0 key	name
2	COM COMP	17	SHP SHAPE 0
1	SOL SOLD	18	SHP SHAPE 1
0		19	RES RESIST 0
0		20	RES RESIST 1
0		21	DRL DRILL
0		22	DOC DOCU 5
0		23	DOC DOCU 6
0		24	DOC DOCU 7

placement & routing

---

sizes

---

layer-plan

---

tracks

---

reference component

---

vias

---

drills

---

texts

---

via wires

Definition for the layer assignment of the PC board. This includes Signal layers, Planes and Documentation layer.

KEY	NAME	MEANING
COM	COMP	Signal layer component side
SOL	SOLD	Signal layer solder side
PLN	GND	Plane layer
FRE	EMPTY	Empty layer
SHP	SHAPE 1	Silk screen of component side
RES	RESISTO	Solder resist layer 0
DRL	DRILL	Drill map
DOC	DOCU 1	Free documentation layers

The Key Words are fixed, the Names are user definable. Any change must be updated (use "W" for WRITE, "C" for CHECK is not running).

TRACK TECHNOLOGY

PRISMA -- Edit --> window		PRISMA -- Edit --> window	
PCB: modify track-technology		Grid: 0.025 / 0.025 MS: inch [986]	
fct: L modify no.: 1 (C) check, (W) write: _			
no.	trackwidth overall core	resist overall core	add-clear. attribute via
1	<input checked="" type="checkbox"/> 0.01143_ 0.0_ 0.01143_ 0.0_	0.0_	_RAD_ 1_
2	0.01655_ 0.0_ 0.01655_ 0.0_	0.0_	_RA_ 1_
3	0.03857_ 0.0_ 0.03857_ 0.0_	0.0_	_RA_ 1_
4	0.05512_ 0.0_ 0.05512_ 0.0_	0.0_	_RA_ 2_

attributes: S-shifted R-routable A-active D-default

placement & routing
sizes
layer-plan
tracks ←
reference component
vias
drills
texts
main menu

- Track width (see also LDROUT Manual for the Routing Technology)
- Overall Total width of the track incl. the rectangular core
- Core The rectangular core of a track. (ex. a rectangular track is created by using the same size for core and overall; a rounded track is created with a core size 0 and a free value for overall.)
- Resist Size for solder resist.
- Add. clear. Additional clearance for that TRACK TYPE only.
- Via Number of VIA TYPE assigned to the track type used for manual or automatic routing.

Tracks may be used for routing and the creation of SMD lands. Attributes sets up the use of track types. All routable tracks (up to 16) need to have "RA". One track type needs an additional "D" as a default track type used when creating a new signal. "S" for a half grid shifting is only working with automatic routing.

VIA TECHNOLOGY

PRISMA --Lattice-window
Grid: 0.025 / 0.025 MS: inch [987]

PCB: modify via-technology

fet: V      modify no.: 0      (C) check, (W) write: \_

no.	via-size		resist		add-clear.	attribute	drl
	overall	core	overall	core			
1	0.03857	0.0	0.05639	0.0	0.0	RD	1
2	0.05512	0.0	0.06693	0.0	0.0	RD	1
3	0.0	0.0	0.0	0.0	0.0		0
4	0.0	0.0	0.0	0.0	0.0		0

attributes: S=shifted R=rotatable D=on-every-layer      all sizes in inch

PRISMA modify-via-technology

---

placement & routing

---

sizes

---

layer-plan

---

tracks

---

reference component

---

vias ←

---

drills

---

texts

---

main menu

The via definitions are similar to the track definition. In addition a **DRILL TYPE** at **DRL** may be defined. The **ATTRIBUTES** for automatic routing must be set to "RD", if shifted vias are used add "S".

The menu displays the first 4 out of 255 VIA types. If you want to display another 4 via types, type in at **MODIFY NO.** the first number of the via type and <CR>. Any change of values must be updated by using "W" at **WRITE**.

MODIFICATIONS

PRISMA - Edit: window		PRISMA - menu-window			
PCB: place components	Grid: 0.050 / 0.050	MS: inch (590)	graphic   pan on/off		
set: G component: <input type="checkbox"/>	layer : C00P		window   full view		
coord. : 0.0 / 0.0	rotat. : 0 ( )		window loc.   refresh		
			center   grid on/off		
		component	track		
		terminal	copper area		
		signal	restr. area		
		mod. graphic	mod. param.		
		get	create		
		insert	move		
rename	delete				
copy	remove				
exch. pos.	exch. shape				
digitize	list				
guided pl.	place groups				
transform	swap				
offset	text				
line	rot. point				
circle	area				
via					
route-mode	continue				
multi-action	multi-select				
sub-menu	main-menu				

After more theoretical information about technology we will come back to the example, which was used already during the schematic work. Please read in this file now and display the LAYOUT on the screen. CIS displays the whole SHEET including PC-BOARD. All shapes are placed in the upper right corner. These shapes are related to those symbols of the schematic, which were used out of the COLUMBINE LIBRARY SERVER.

Select now MODIFICATION and the WORK MENU will be displayed. Many functions are very similar to the SCHEMATICS. The differences are:

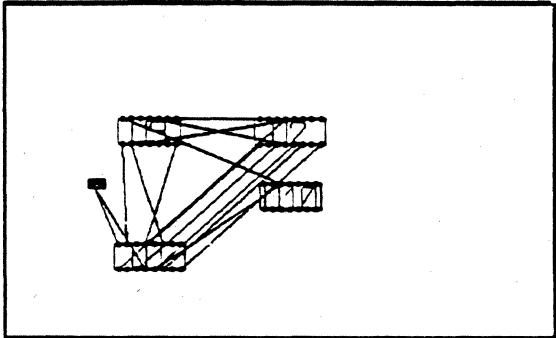
- COPPER AREAS** To build COPPER AREAS on the board, explained later.
- MULTI SELECT** A global select command for COMPONENTS, SIGNALS and TRACKS. After the selection, it defaults to:
- MULTI ACTION** The now following function (i.e. MOVE, DELETE or MOD. PARAM.) is active for the whole sel. elements.



COMPONENT PLACEMENT      MANUAL PLACEMENT

PRISMA: status window  
 PCB: place components  
 fet: 6 component: RI layer: Grid: 0.050 / 0.050 MS: inch [590]  
 coord. : 1.0 / 1.80 rotat. : 0 ( )

graphic	pan on/off
window	full view
window loc.	refresh
center	grid on/off
component	track
terminal	copper area
signal	restr. area
mod. graphic	mod. param.
get	create
insert	move
rename	delete
copy	remove
exch. pos.	exch. shape
digitize	list
guided pl.	place groups
transform	swap
offset	text
line	rot. point
circle	area
via	
route-mode	continue
multi-action	multi-select
sub-menu	main-menu



CIS supports manual and semi automatical placement tools. The default functions after entering the WORK MENU are COMPONENT and GET. For manual placement please follow the steps:

1. Move the cursor to the upper right corner (over the group of components) and <CLB>. One of the components is attached to the cursor.
2. Now the cursor displays a cross hair with the outline of the selected shape and small crosses at the coordinates of the component pins (this setup may be changed within MODES).
3. The signals connected to this shape are shown as blue AIRLINES. They will follow any movement of the cursor.
4. Move the cursor to the place shown on the diagram and <CLB>.
5. The component will be placed with its name, outline and pins at the cursor position. AIRLINES will be displayed only for signals inside the PC-BOARD.
6. Use this function to place all components on the board.

---

**MANUAL PLACEMENT (continued)**
**FUNCTIONS OF THE CURSOR**

- Left button** Pick up the component (1st click) and place it at the new position (2nd click). The function then changes back to **GET**.
- Middle button** Rotates the component in steps of 90 deg. counter clock wise. May also be used to **GET** a component.
- Right button** Picks up the component and places it at the new position. Function does not default back to **GET**, but remains in **MOVE**.

**SHORT DESCRIPTION OF OTHER PLACEMENT FUNCTIONS**

- MOVE** Moves a selected component to a new position. The function defaults to **MOVE** after selecting (**GET**) the component.
- INSERT** **INSERT** is used to copy components out of the **COLUMBINE LIBRARY SERVER** onto the actual board. The component includes the related schematic symbol. Spare gates are placed on page 0 in the schematics.
1. Select **INSERT** on the **FUNCTION MENU**.
  2. Type in at **SEARCH STRING** the component name, you want to copy out of the **COLUMBINE LIBRARY SERVER** and <CR>. The selected component will be attached to the cursor and you may place the component on the board. Using the wildcard function "\*", there will be a subwindow with a list of components, which exist in the **COLUMBINE LIBRARY SERVER**. Move the cursor to one of these components and <CLB>. This component will now be attached to the cursor, the subwindow will automatically be closed.

Before using one of the following functions, a component has to be selected (**GET**). Call up the desired function with the cursor then.

- RENAME** Type in the new component name at the text field **COMPONENT** in the **STATUS MENU** and <CR>. The component will be renamed, the function changes to **MOVE** and the component has to be placed with the left or right button or release with **GET**, or "R9".
- DELETE** Deletes a selected component immediately or after an additional <CR> (depending on the definition within the **SUBMENU** of **COMPONENTS**). The related symbol (symbols) of the schematic including connections will also be deleted.
- COPY** This function copies components which are already on the board, it has no access to the **COLUMBINE LIBRARY SERVER**.
- It copies the selected component without its signals and increments its name. If the function **CONNECTED M+C+D** within the **SUBMENU** of **COMPONENTS** is active, the component is copied including connections. The connected pins of the new components will be added to the existing signals. The related symbols will be placed in the schematic on page 0.

**EXCH. POS.** Exchanges the placement coordinates of two components. Component collisions will be checked and a warning will be displayed if existing.

1. Move the cursor to the first component and <CLB>.
2. Select **EXCH. POS.** on the **FUNCTION MENU**
3. Move the cursor to the second component and <CLB>. The positions of the components will be exchanged, if possible.

**EXCH. SHAPE** Exchanges the shapes of two components, i.e: technology change from "DIL" shapes to "SMD" shapes.

1. Select the new shape you want to use for the component to change.
2. Select **EXCH. SHAPE** on the **WORK MENU**.
3. <CLB> inside the component outline of that component you want to change the shape.
4. This component now will change its shape into the new shape, which was selected in step 1.

**Manual Placement (continued)**

**CONTINUE**

Allows to continue the work of the following functions:

<b>COPY</b>	<b>RENAME</b>
<b>MOVE</b>	<b>DELETE</b>

1. Select **COMPONENT** and **GET**.
2. <CLB> on the component you want to use.
3. Select **CONTINUE** on the **FUNCTION MENU**
4. Select one of the above functions.

Select **CONTINUE** again or **GET** to stop this function.

**ROTATE**

Use the middle button to **ROTATE** a selected component in steps of 90 deg. ccw.

Any other angle must be entered at the field **ROTATION** in the **STATUS MENU**. The units are in 1/10 degree i.e the input for an angle of 60 degree must be "600".

**SUBMENU FOR COMPONENTS**

The **SUBMENU** for components must be entered from the **WORK MENU**.

The **SUBMENU** allows to have a default setup for the work with **COMPONENTS**.

**IMMEDIATE KILL**

Deletes a component without an additional <CR>.

**REORG. AIRLINES**

When moving components, the **AIRLINES** may not be displayed with its shortest distance. **REORGANIZE AIRLINES** calculates the length of the **AIRLINES** and displays the shortest connections on the whole board.

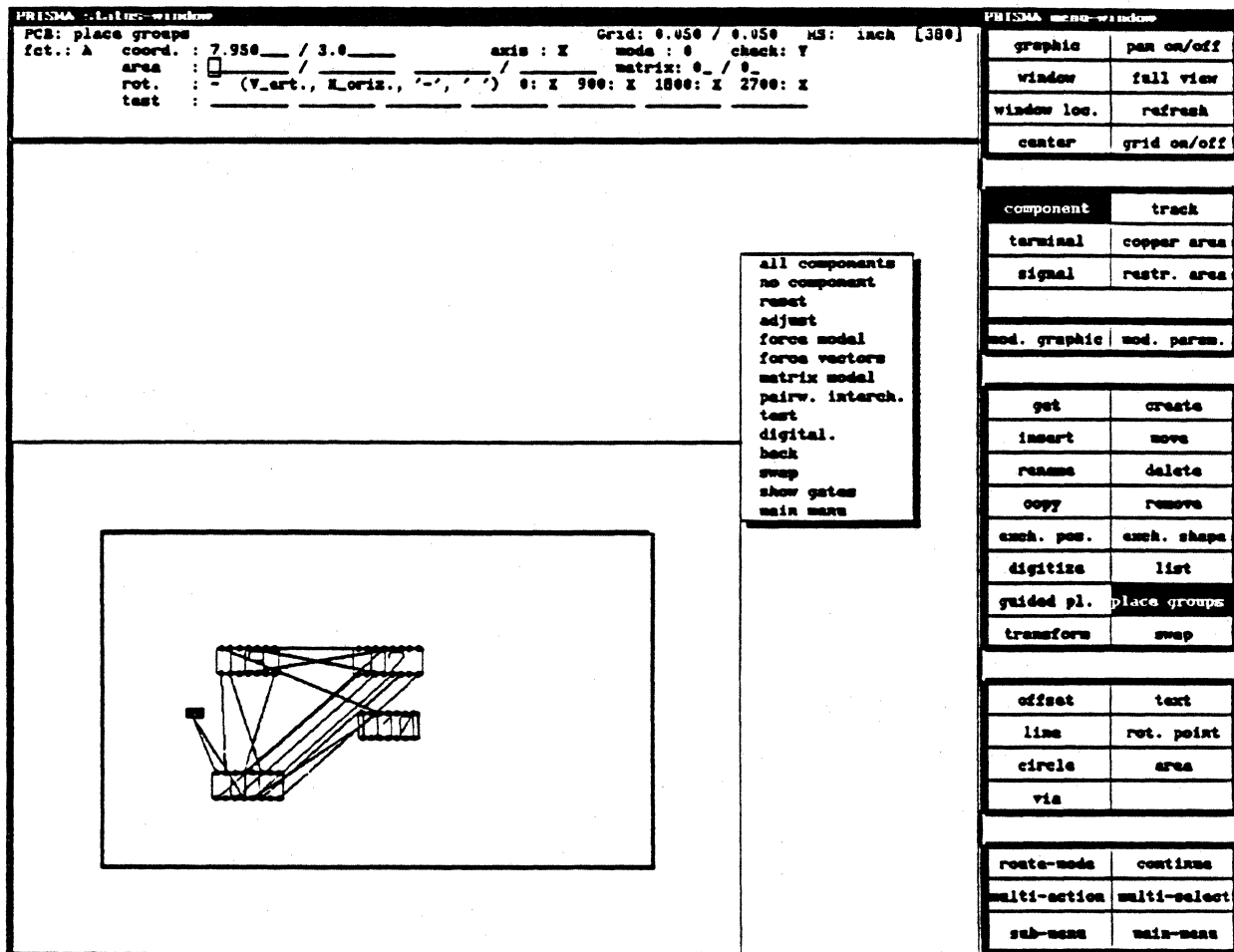
**REORG. ON/OFF**

The **REORGANIZATION** of the airlines may be active during the whole placement work. After placing each component, the airlines of this component will be reorganized.

**CONNECTED MCD**

If **CONNECTED MCD** is active, components are copied including connections. The connected pins of the new components will be added to the existing signals. If this function is **OFF**, only the pins holding **SUPPLY SIGNALS** will be added to the existing signals.

## SEMI AUTOMATIC PLACEMENT



The SEMI AUTOMATIC PLACEMENT supports the user with semi automatical placement tools. These tools can be used at any time during the placement with all components on board or with a selected number of components. They may be selected using the MULTI SELECT COMMAND for components or with a function on the POPUP MENU. The AUTOMATIC PLACEMENT is divided into different single tools, please follow the steps to run through the possibilities:

### RESET

1. Select COMPONENT and PLACE GROUPS.
2. Hold down the right mouse button and move the highlighted bar to ALL COMPONENTS (to select all components) and release the button then.
3. Hold down the right button again and move the highlighted bar to RESET and release it. All not fixed components will be placed off the board now.

**Semi Automatic Placement (continued)****MATRIX PLACEMENT**

The **MATRIX PLACEMENT** is used to place a group of components within a matrix on the board. The components should have nearly the same size, otherwise components may overlap. We can continue with:

1. Define a matrix field by pressing the middle button in the lower left corner the first time and in the upper right corner the second time.
2. Press the right button and move the highlighted bar to **MATRIX MODEL** and release it.

The matrix field will be displayed and the selected components will be placed inside these fields. The number of component fields will be calculated automatically. Within the **STATUS MENU**, you may also type in the size of the **MATRIX FIELD** and the number of rows and columns for the matrix.

**FORCE VECTORS**

As a result of the **MATRIX PLACEMENT**, the components will be displayed with all of its airlines. The function **FORCE VECTORS** now allows to display all airlines of the selected components as a summary vector. The direction and the length of these summary vectors are a helpful information to tell the user in which direction the components should be placed to have the shortest connections.

1. Hold down the right button and move the cursor to **FORCE VECTORS** and release it.

You will see now the **FORCE VECTORS** as red lines starting in the middle of the components and showing the direction of the best placement.

**PAIRWISE INTERCHANGE**

Exchanges the positions of the components in order to minimize the total length of connections. A random number generator selects the first component and compares the placement coordinates of this one with all the other components and then changes the positions.

## Automatic Placement (continued)

To run the **PAIRWISE INTERCHANGE**:

1. Hold the right button down and move the cursor to the function **PAIRWISE INTERCHANGE**.
2. Release the button.

The **PAIRWISE INTERCHANGE** is running. There will be a small "clock" displayed on the screen to indicate that this function is still working.

## GUIDED PLACEMENT

The **GUIDED PLACEMENT** is often used to place analog components on an already pre-placed board. A group of selected components will be placed one after the other. For each of these components, the system displays a red placement area and the best placement position. The component is still attached to the cursor and the user may decide where to place the component. Rotate it with <CLB>. After the placement of this component (CLB), the next one comes up.

**SHOW GATES** Displays all changeable gates of the selected components. All pins of a gate are shown as red lines connected together in the center line of the component. The gates are already pre-defined in the **COLUMBINE LIBRARY SERVER**. For new components, this definition must be done within **SWAPCLASS** in **MODIFY TERMINALS** in the **SCHEMATICS !!** **SWAPCLASS** definitions are:

0 = no swap possible  
1 or higher = swap possible with terminals of the same swapclass within the same gate.

**SWAP GATES** Allows to swap **PINS** and **GATES** of the same function type. This function works within **PLACE GROUPS**.

1. Enter **PLACE GROUPS** and select the components you want to swap gates.
2. Hold down the right button and move the bar to **SHOW GATES** and release it. The gates will be displayed.
3. Hold down the right button again and move the bar to **SWAP** and release it or select **SWAP** on the **FUNCTION MENU**.
4. Move the cursor to one pin of a gate and <CLB>. Move the cursor then to a pin of the other gate and <CLB>.

**Semi Automatic Placement (continued)**

**ADJUST**

Lines up a selected group of components. To use this function, type in at **AXIS "X"** or **"Y"** to enter the axis for the adjustment. There are 3 different **MODES**:

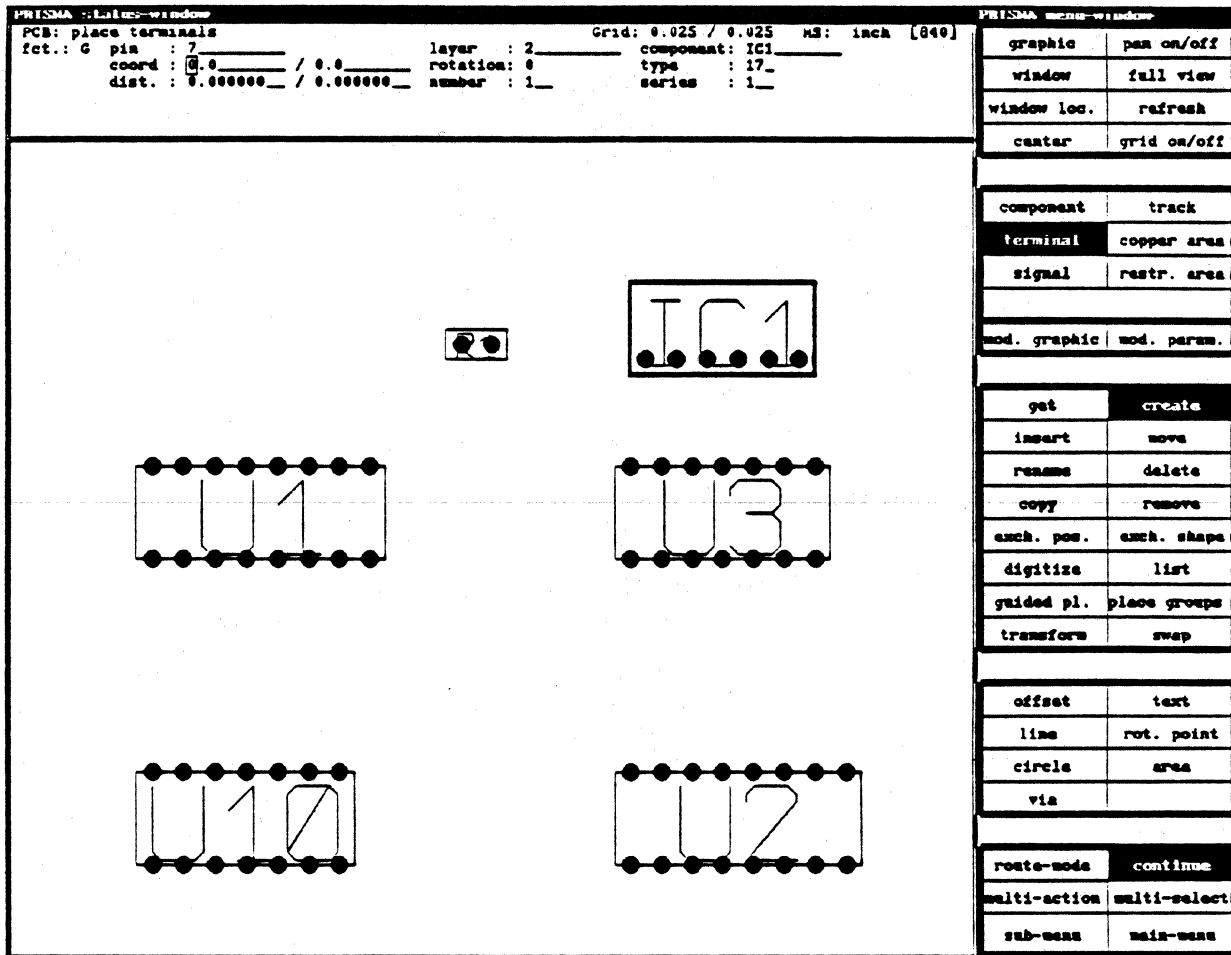
- |               |  |
|---------------|--|
| <b>MODE 0</b> | Lines up the <b>PLACEMENT</b> coordinates.   |
| <b>MODE 1</b> | Lines up the <b>LEFT</b> component outline for adjustment on the <b>Y-AXIS</b> .<br>Lines up the <b>LOWER</b> component outline for adjustment on the <b>X-AXIS</b> .  |
| <b>MODE 2</b> | Lines up the <b>RIGHT</b> component outline for adjustment on the <b>Y-AXIS</b> .<br>Lines up the <b>UPPER</b> component outline for adjustment on the <b>X-AXIS</b> . |

How to use this function:

1. Select **COMPONENTS** and **PLACE GROUPS**.
2. Hold down the right button and move the bar to **NO COMPONENT** and release it.
3. Select **MULTI SELECT** and **<CLB>** on some components you want to line up.
4. Select **PLACE GROUPS** again.
5. Type in at **AXIS "Y"** and at **MODE "0"** for an adjustment in **Y-AXIS**.
6. Place the cursor at the coordinate you want to line up the components and **<CLB>**.
7. Press the right button and move the highlighted bar to the field **ADJUST** and release it.
8. The shapes will be lined up.



CREATING NEW SHAPES



A new shape has to be defined **LOCALLY** and may be copied into the **COLUMBINE LIBRARY SERVER** later on.

1. Select **COMPONENT** and **CREATE**.
2. At **COMPONENT NAME** in the **STATUS MENU**, you will find a default name for this new component starting with "\$". You may overwrite it with "IC1".
3. Type in the size of a rectangle (0.6" x 0.3") at **SIZE** in the status menu and place the cursor at a free position on the board and <CLB>. A component outline of the defined size appears on the screen. The **FUNCTION** menu has changed to **TERMINAL**, **CREATE** and **CONTINUE**.
4. Place the cursor at **TYPE** in the **STATUS MENU** and <CLB>. Type in "17" for the desired **VIA TYPE**.

---

**CREATING NEW SHAPES (continued)**

5. Move the cursor back to the working window and <CLB> at the lower left corner of the new component to place pin 1.
6. Move the cursor to **GRID ON/OFF** and <CLB>.
7. Create 6 other pins in the X-axis with a distance of 0.1".
8. Move the cursor to **DIST.** <CLB> and type in - 0.1 (units in inch) for the X-value. Move the cursor to the field **NUMBER** and type in "7".
9. Move the cursor back to the working window and <CLB> at the upper right corner (0.3" above pin 7) of the new component. You will get 7 pins lined up to the left with a distance of 0.1 inch.
10. You have created a new component with 14 pins.

**CREATING A SHAPE WITH SMD PINS.**

Following the first 3 steps of the description above, please create a new component "IC2" with a size of 0.4" x 0.25".

1. Move the cursor to **TYPE** in the **STATUS MENU** and type in "-1" as a code for SMD-terminals with a user definable SMD-pin.
2. Move the cursor back inside the shape and <CLB> (close to the lower left corner). You will see a small cross.
3. Select **MODIFY GRAPHIC** and type in "2" at **TRK-TYPE**.
4. Place the cursor 1 grid above the cross and <CLB>.
5. Move the cursor down 2 grids and <CLB> again. Now there will be an SMD-pin on layer 2 (component side) with the length of 2 grids. Use this way to create your own SMD-pins.
6. Select **TERMINAL** again. The function defaults to **MOVE** and the cursor displays the outline of the selected terminal. To rotate the terminal, use the middle button, place it with the left button.
7. Select the terminal again and change the function to **COPY** and **CONTINUE**.
8. Use this function to copy 5 terminals.

You may use all track types and all via types to digitize any outline for the terminal. The functions **MOVE**, **RENAME**, **DELETE** and **COPY** can be used during the creation or modification of terminals.

**BUILDING A RELATION BETWEEN SYMBOL AND SHAPE**

PRISMA - Layer: window

SCS: enter SCS-PCB relations

PCB name : IC1    phy. terminal-pos: 0.0 / 0.0    grid: 0.050    net= lach [39]

signalname:    loc. terminal-pos: 0.0 / 0.0    PCB-terminal name:   

SCS-terminal name:   

graphic support (Y/N): Y    show initial relations: X    function (CRSDP): P

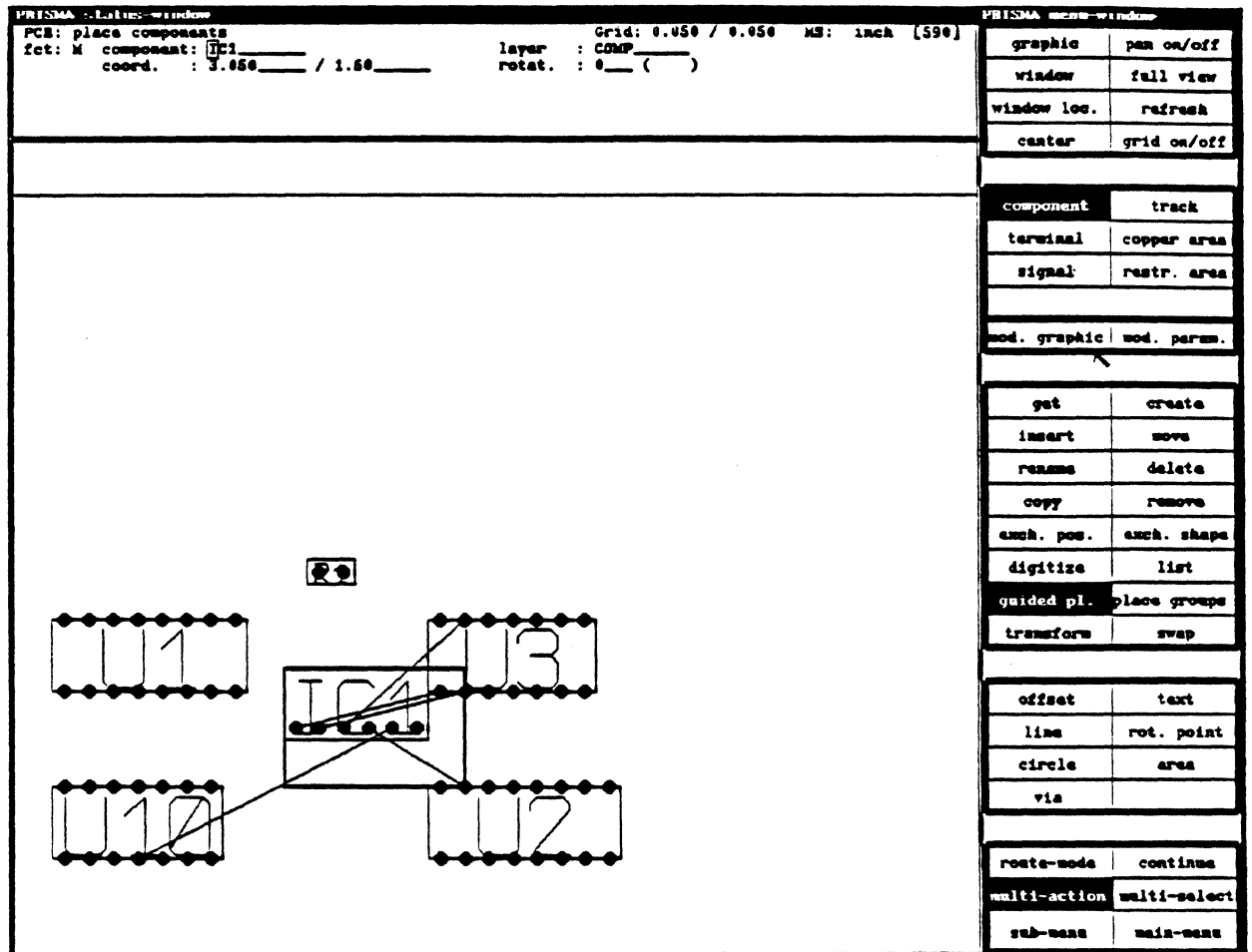
PRISMA menu-window

graphic	pan on/off
window	full view
window loc.	refresh
center	grid on/off
symbol	bus
terminal	relation
signal	
text	format
mod. graphic	mod. param.
get	create
insert	move
rename	modify
copy	delete
swap	replace
scale	list
digitize	change logic
open	close
offset	text
line	rot. point
circle	
route-mode	continue
multi	
sub-menu	main-menu

1. Enter the SCHEMATICS part of CIS from the MAIN MENU or press "F2" and select the MODIFICATION.
2. Select RELATION on the FUNCTION MENU.
3. Type in "IC1" at PCB NAME of the STATUS MENU and <CR>.
4. The cursor displays the physical shape of IC1. Move the cursor close to the DAC and <CLB>. Display a window of these elements.
5. Move the cursor onto a terminal of the symbol and <CLB>. Move the cursor now to a terminal of the physical shape and <CLB> again. There will be a line between the two terminals. This line indicates the relation between the logical terminal and the physical terminal.
6. Continue to relate all symbol terminals to the physical terminals.
7. Create relations between IC2 and the REGISTER, too.

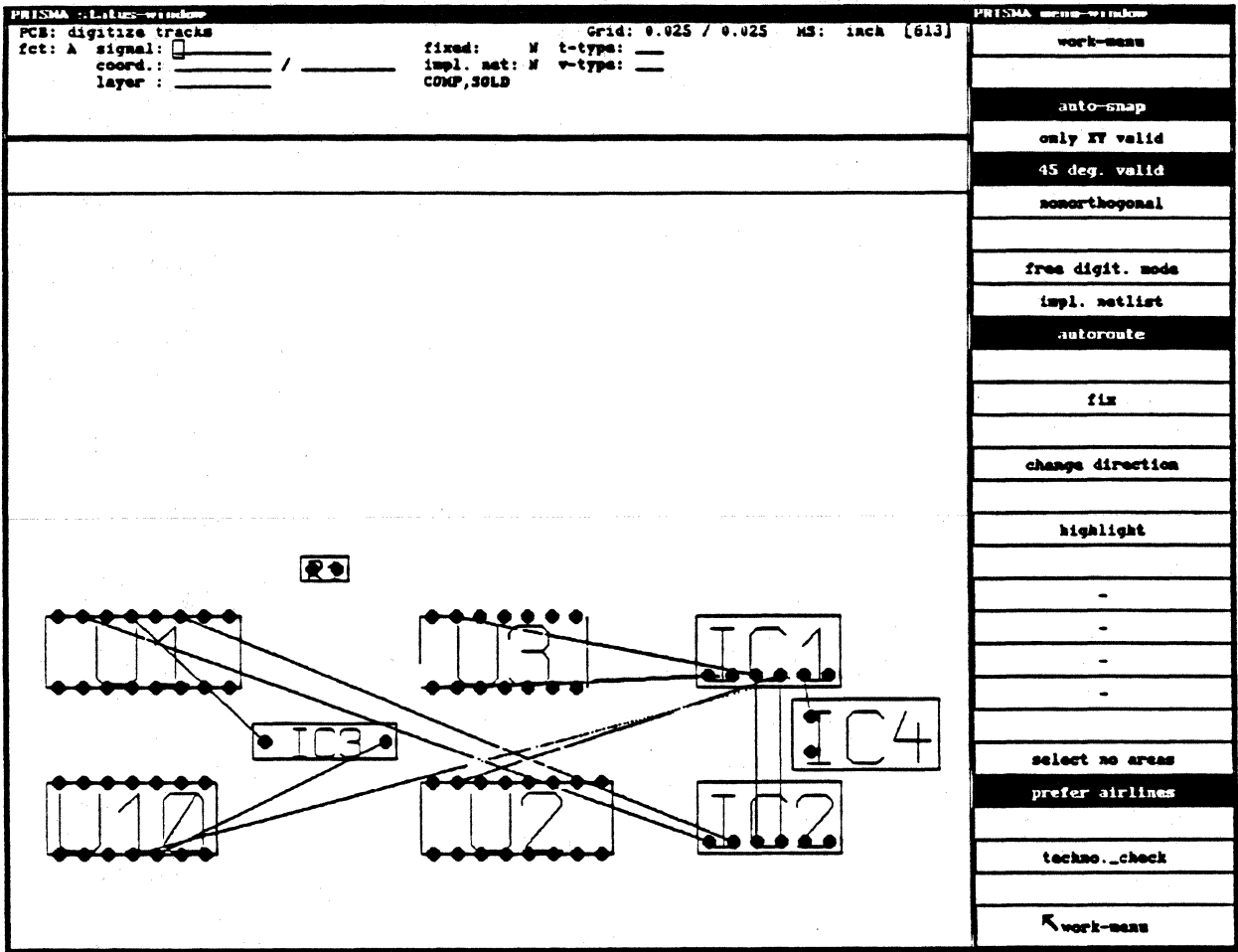
There are some other functions to be selected from the POPUP MENU.

## GUIDED PLACEMENT



1. Jump back to the **WORK MENU** of the **PCB LAYOUT** (use key "F2")
2. Select **PLACE GROUPS** and **MULTI SELECT**.
3. Type in "IC\*" at **ELEMENT** and <CR>. The function will change to **PLACE GROUPS**. Select **RESET** on the **POPUP MENU**. The two IC's will be placed outside the board.
4. Select **MOD. PARAM.** to enter the function **GUIDED PLACEMENT**. The function **MULTI ACTION** is highlighted, too.
5. The cursor displays one of the selected components. A red frame defines the best placement area. Inside this area, the component will be pre-placed at its best placement position.
6. Move the cursor to a place inside that frame and place the component using <CLB>. If this is not possible, you may place it at any other position on the board.
7. The screen now displays a new red frame for the next selected component to be placed.

**SUBMENU FOR TRACKS**



**AUTO - SNAP**  
**ONLY XY VALID**  
**45 DEG. VALID**  
**NONORTHOGONAL**  
**FREE DIG. MODE**  
**IMPL. NETLIST**

**AUTOROUTE**  
**FIX**

**CHANGE DIRECTION**  
**HIGHLIGHT**  
**SELECT NO AREA**  
**PREFER AIRLINES**  
**TECHNO. CHECK**

Automatic snap to the closest grid point.  
 Only horizontal and vertical tracks valid.  
 45 deg. and horiz. or vertical tracks valid  
 Any angle valid for digitizing.  
 Free dig. of signals incl. "T"-connections.  
 Adds non-connected pins to a signal when touched during the digitizing.  
 Rerouting of tracks when moving a via.  
 Digitizing of fixed tracks. These tracks will not be moved by LDROUT.  
 Changes the starting point of the track.  
 Highlights the selected signal during dig.  
 The selection of signals instead of areas.  
 Airlines are preferred when selecting signals.  
 Check for shorts and opens of the sel. signal.



## Digitize Tracks (continued)

6. Digitize one segment, press the right button to place a via and change the layer of digitizing.

Tracks in **RED** are digitized on the **SOLDER SIDE (LAYER 1)**.

Tracks in **GREEN** are digitized on **COMPONENT SIDE (LAYER 2)**.

7. Digitize some other track elements using the left button, place a via and connect this track to the existing GND track (like in the diagram above).

In this way you may continue to digitize some other signals. The default **TRACK TYPE** is "1" (0.1143 inch, defined in **TRACK TECHNOLOGY**). The digitized **FIXED** tracks will not be removed by the automatic router **LDROUT** and are shown therefore as dashed lines.

## THE FUNCTIONS OF THE MOUSE BUTTONS

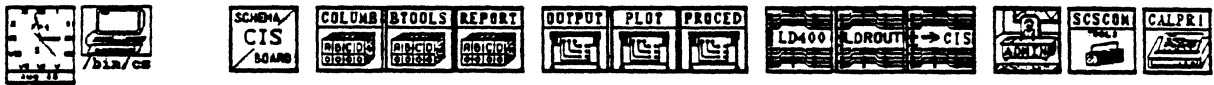
- |               |   |  |
|---------------|---|--|
| Left button   | - | Selects an <b>AIRLINE</b> if no airline has been selected.   |
|               | - | Enters the coordinates of a digitized track.   |
| Middle button | - | Selects an <b>AIRLINE</b> if no airline has been selected.   |
|               | - | Deletes the last digitized track or via segment and continues to delete elements by pressing the middle button again. Changes the function from <b>DIGITIZE</b> to <b>GET</b> after the last element has been deleted. |
| Right button  | - | Selects an <b>AIRLINE</b> if no airline has been selected.   |
|               | - | Places a via at the end of a track element and changes the digitizing layer.   |
|               | - | Changes the digitizing layer only if the cursor is not placed at the end of a track element.   |

## PREPARING A BOARD FOR AUTOROUTING

After finishing the placement and digitizing some fixed tracks (mainly for supply signals), the board is ready for automatic routing. Therefore we have to update the file, exit this program and start **LDROUT**.

1. Enter the **MAIN MENU** and select **EXIT**. The **STATUS MENU** changes to **UPDATE FILE**. The old file name appears at **JOB NAME**.
2. Press **<CR>** to update the file and to exit the program **CIS** in one step.

AUTOMATIC ROUTING USING LDROUT



```
<< MESSAGE >>
[?] can only restrict planes to <= cmm_size-1 in pw_petattributes
[?] calaytool 1017 exited with return code: 0
```

1. Place the cursor inside the **LDROUT** icon and <CLB>.
2. A new window appears on the screen. Move the cursor inside this new window.
3. Type in the file name (i.e **example**) without extension and <CR>.
4. Type in the extension (i.e.**adb**) and <CR>.
5. Next you are asked to enter the parameters for **LDPRE**. Type in the following option:
 

-n <CR>
6. The menu changes to the options of **LDROUT**. Enter the following options without blanks:
 

-f-l-s-y <CR>
7. The router extracts the routing data and then displays the **PC-BOARD** including component pins and fixed tracks on the screen. You will see how **LDROUT** works. After completing the routing, the system will return to the initial **PRISMA SCREEN**.



### Automatic Routing (continued)

Some information about the command procedure started with the **LDROUT** icon. After entering the options for automatic routing, **PRISMA** starts to extract the routing data. The program **SCSCOM** reads the file **example.adb** and creates a file called **example.rai**. The file name **example.adb** is protected during the following routing procedure and cannot be overwritten by **CIS**. Next the program **RAIGPH** creates a file called **example.gph**, which will then be used for automatic routing. The automatic routing is divided into three parts:

<b>LDPRE</b>	Layering, netlist sorting and checks for routing technology.
<b>LDROUT</b>	Main router to place tracks and vias according to the netlist
<b>LDOPT</b>	Optimization for production needs.

The result of the automatic routing is the file **example.gph** including the routed tracks and vias. To display the results of the automatic routing within **CIS**, the **example.gph** file has to be converted back. Therefore, the program **GPHRAI** creates a new file called **example.rai** and **SCSCOM** converts to **example.adb**.

To run the conversion back continue with:

1. Select the icon **LDROUT to CIS** next to the **LDROUT** icon.
2. A new window appears on the screen and asks you to enter the file name without extension. Please enter: **example**
3. The conversion back starts now. The message **program ended normally** appears at the end of the conversion. Type in **<CR>** to exit this program.

The automatic routing is now completed. **PRISMA** is ready to continue the interactive work with the routed file.

## WORKING WITH A ROUTED FILE

PRISMA -> status -> window  
 PCB: digitize tracks Grid: 0.025 / 0.025 MS: inch [613]  
 fet: A signal:  fixed: N t-type:   
 coord.:  /  impl. act: N v-type:   
 layer:  COMP, S0LD

PRISMA menu -> window

graphic	pan on/off
window	full view
window loc.	refresh
center	grid on/off

component	track
terminal	copper area
signal	restr. area

mod. graphic	mod. param.
--------------	-------------

get	create
insert	move
rename	delete
copy	remove
exch. pos.	exch. shape
digitize	list
guided pl.	place groups
transform	swap

offset	text
line	rot. point
circle	area
via	

route-mode	continue
multi-action	multi-select
sub-menu	main-menu

Read in the routed board and display the **PCB LAYOUT**. In the following, we will describe some other important interactive functions. The system's **DESIGN RULE CHECK** will apply to all.

### Move Tracks

1. Enter the **WORK MENU** and select **TRACK**.
2. Place the cursor on top of a track and **<CLB>**. The values of the initialized signal are displayed in the **STATUS MENU**.
3. Select **MOVE** on the **FUNCTION MENU**. Move the cursor back into the working window.

## WORKING WITH A ROUTED BOARD (continued)

4. You will see a small line between the cross hair and the selected track. Move the cursor to the location you wish to move the track to, and press one of the mouse buttons. (There should be an angle of 90 deg. between the track and the cross hair line). The track element will be moved to the new position. CIS automatically creates corners for the track according to the design rules. Track corners which are no longer necessary will be cleaned up.
5. Pick up a track and move it onto another track. Both tracks will be moved into the wanted direction. This allows the user to move complete groups of tracks with only one command.

## MOVE VIAS

The via move may be done with all three mouse buttons.

1. Select **TRACK** and **VIA** on the **FUNCTION MENU**.
2. Move the cursor in the middle of a via and **<CLB>**.
3. Move the cursor now horizontal or vertical to the new and free position for the via and **<CLB>**. The via will be moved to this position. The connected tracks will be rerouted to the new via position. If this rerouting is not possible (not enough space for the tracks), an airline will appear between the new via position and the end of the track.

## DELETE TRACKS

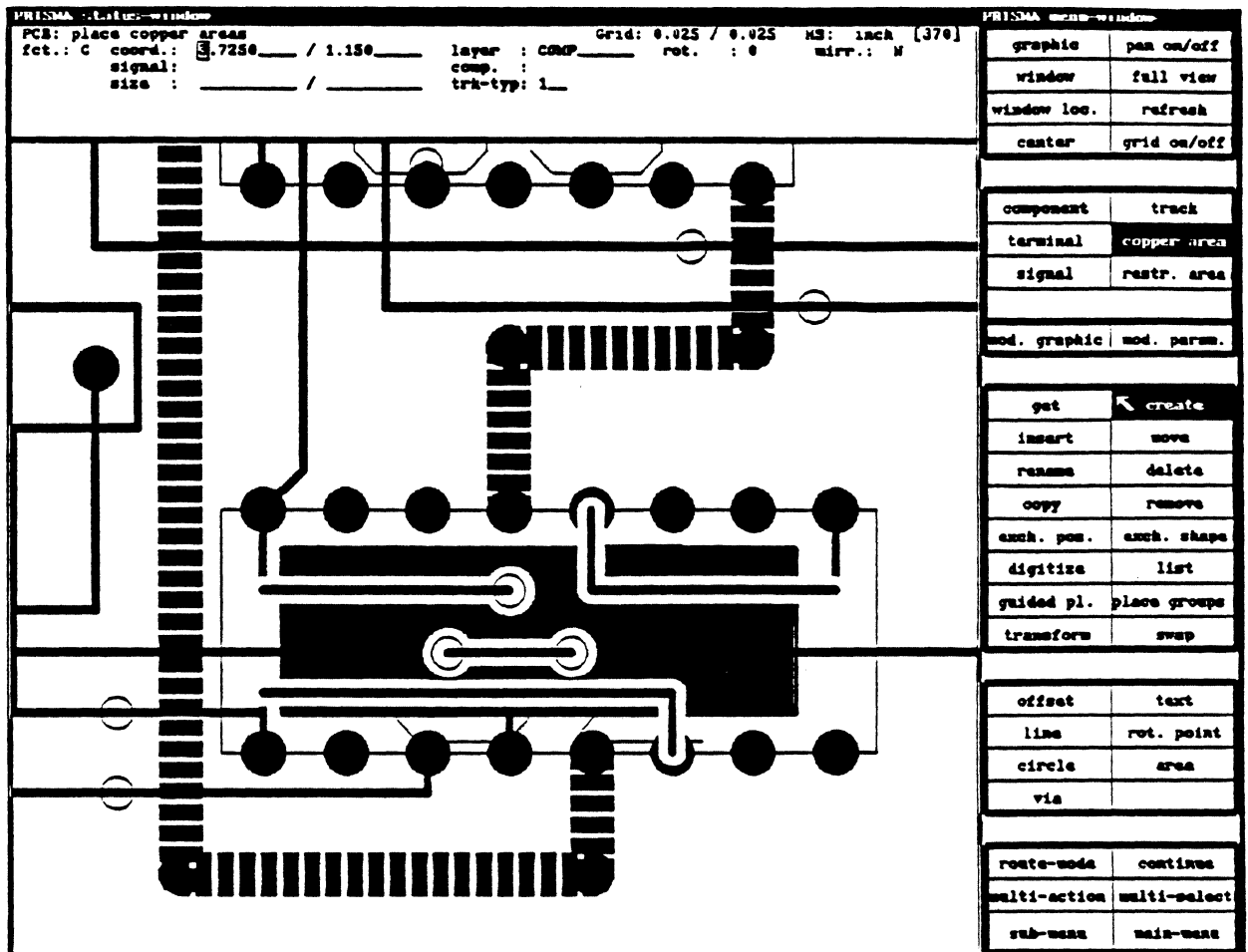
A signal on the board is always represented with an airline, a track or a combination of both. During the digitizing of an airline tracks are created, when deleting tracks, airlines are recreated. In both cases, there has been no change of the signal. Tracks can be deleted segment by segment or by desired sections.

1. Move the cursor onto the track and **<CLB>** to select the track segment you want to delete.
2. Press the middle button to delete this segment. An airline appears between the two ends of the deleted track segment. The segment will also be deleted after selecting **DELETE** on the **FUNCTION MENU**.

## DELETE VIAS

1. Select **TRACK** and **VIA** on the **FUNCTION MENU**.
2. Place the cursor onto the via and **<CLB>**. Move the cursor to **DELETE** and **<CLB>**. The via will be deleted.

## COPPER AREAS



**COPPER AREAS** are free definable areas on the board, which will be filled with copper but will be isolated from existing elements such as tracks, vias, etc. The complete **COPPER AREA** will be displayed on screen in its realistic form. It is possible to create filled, cross hatched or "individual" **COPPER AREAS**.

1. Select **COPPER AREA** and **CREATE** on the **WORK MENU**.
2. <CLB> at the lower left corner and at the upper right corner to create a rectangular **COPPER AREA** on the component layer. The **COPPER AREA** is displayed as a filled area, all other elements existing on this layer and inside the area are isolated. The sizes of the isolation depend on the definition of the **MINIMAL CLEARANCES** (global plus special defined clearance for different track types or via types).

## Copper Areas (continued)

3. The function defaults to **MOVE**. The **COPPER AREA** may be moved and placed at its final position (mirrored or rotated).
4. To change the rectangular form of the **COPPER AREA**, select **MOD. GRAPHIC**. The **COPPER AREA** will be displayed with its frame only. Move the cursor on a segment of the frame and <CLB> to pick up the frame element. Select **DELETE** to delete this element. Select **DIGITIZE** next and digitize a free contour to close the frame again. Use **STOP** to fill the closed **COPPER AREA**.
5. Select **GRAPHIC** from the **WORK MENU** and type in "X" and <CR> at **COPPER PLANES REAL**. Select **WORK MENU** again.
6. Move the cursor into the **COPPER AREA** and <CLB>. Select **MOD. PARAMETER** and <CR>. The **COPPER AREA** will be displayed as a cross hatched area. <CR> displays the filled area again. The used **TRACK TYPE**, **GRID**, and **DISTANCE** may now be changed in the **STATUS MENU**. Another <CR> displays these changes of the cross hatched area.
7. All cross hatched elements of the **COPPER AREA** are **TRACKS !!** and can be handled as tracks. Therefore select **MOD. GRAPHIC**, the function defaults to **GET**. Move the cursor into the **COPPER AREA** and <CLB>. The cursor displays the selected track segment which can be moved or deleted. Track elements can be added after selecting **DIGITIZE**. To finish the graphical modification of the **COPPER AREA**, select **STOP**. The new layout of the **COPPER AREA** may be updated and will be stored in the .adb-file. To change it back into the original form of the filled **COPPER AREA**, select the **COPPER AREA** and **MOD. PARAM.** and press <CR>, but

**CAUTION:** All interactive modifications of the **COPPER AREA** are lost.

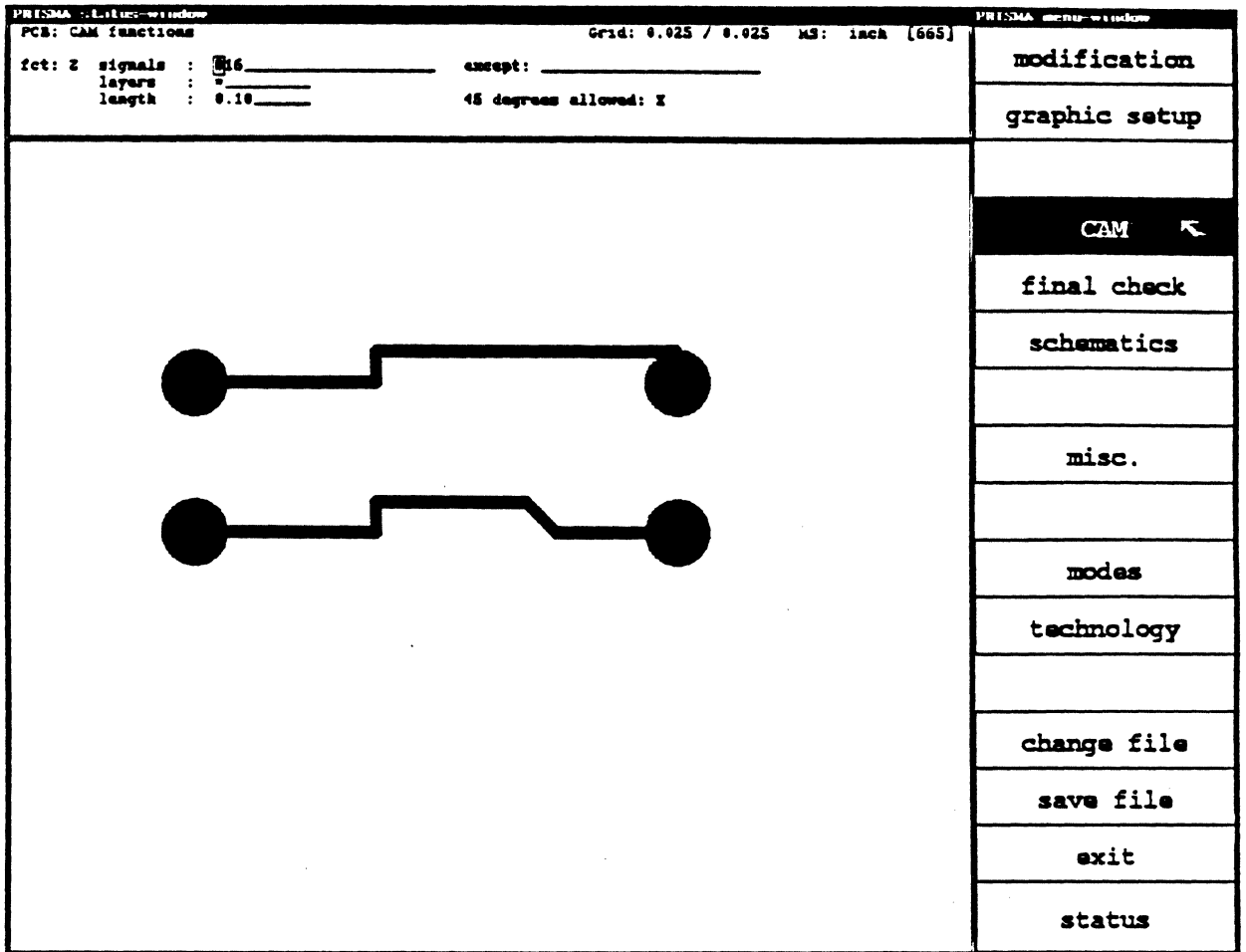
## RESTRICTED AREAS

1. Select **RESTR. AREA** and **CREATE** on the **FUNCTION MENU**.
2. <CLB> at the lower left corner and <CLB> at the upper right corner to create the **RESTRICTED AREA**.

The outline of the **RESTRICTED AREA** may now be modified like a copper area after selecting **MOD. GRAPHIC**.

Restrictions for **TRACKS**, **VIAS** and/or **COMPONENTS** will be setup after selecting **MOD. PARAM.** The restriction is valid inside or outside the defined area.

CAM FEATURES CENTER



CENTER - Centers connections to pads, vias and SMD pins.

The "Center" function along with all other functions in the CAM application are designed to be used just prior to OUTPUT. The offgrid segments and vectors generated during these proceedings make subsequent editing very difficult.

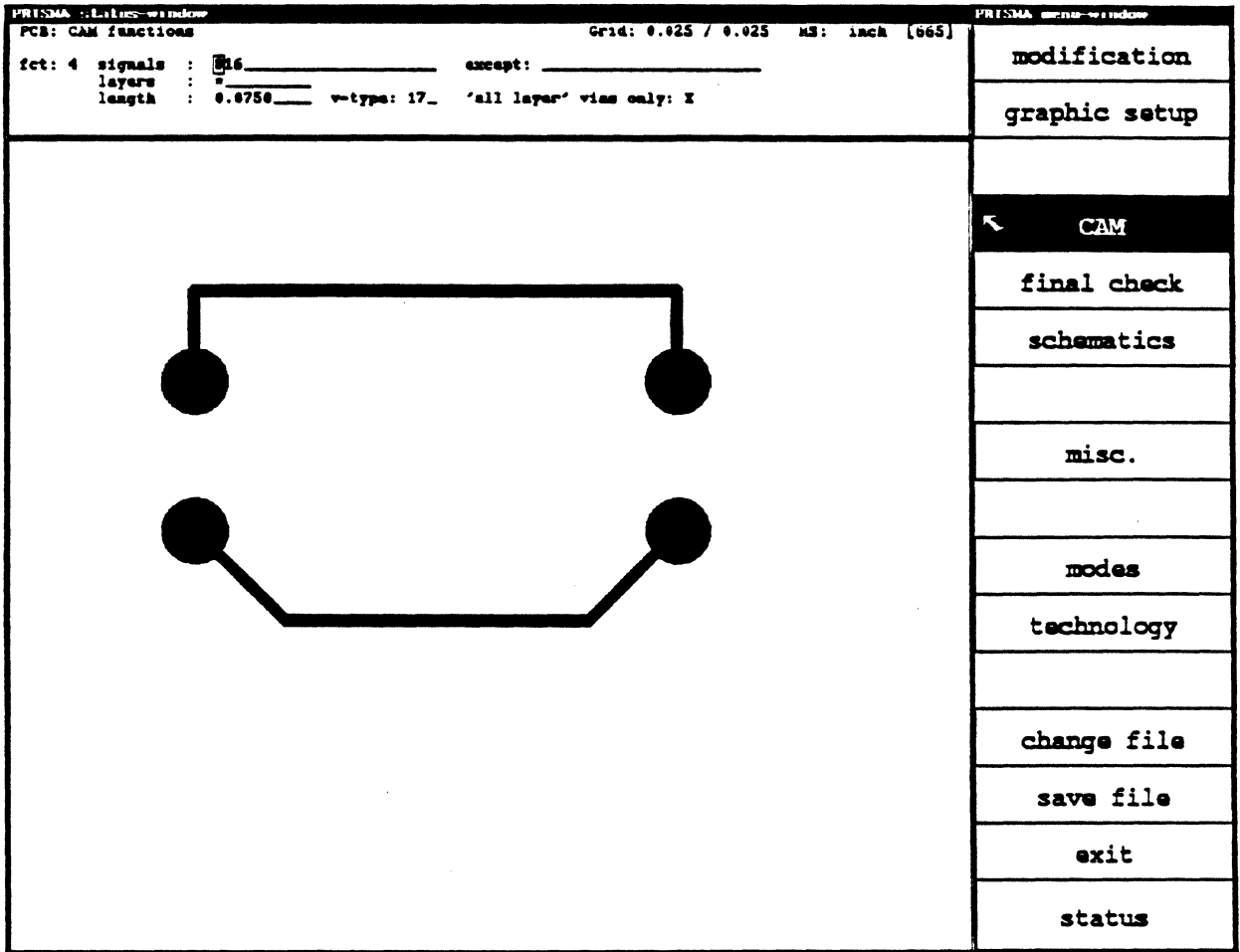
The center function allows the user to either globally or selectively center off-center routed elements. This is of great importance in manufacturing and should save you many hours in cleanup editing.

To use the center function, we must first select **CAM** from the main menu. Then activate the "pop-up" menu and select **CENTER**. A new menu is displayed and the following parameters should be set:

1. **SIGNALS:** Specifies which signal or signals will be evaluated. (\* is used as a wildcard).
2. **EXCEPT:** Used when a wildcard has been specified in the signals menu. Signals entered here will be ignored.
3. **LAYERS:** Specifies which layers are to be evaluated.
4. **LENGTH:** Specifies the length of the segment to be centered.
5. **45° Allowed:** If this field is marked, any segments that are no longer than the length specified will be shifted using a 45° angle where possible.

After all data is entered, a <CR> will update all specified elements.

CAM FEATURES 45° CONNECTIONS



45 DEGREE - Converts track which enter pads or vias at 0 or 90 degrees to 45 degrees.

This function allows the user to globally or selectively change tracks that enter or exit pads or vias at 0 or 90 degrees to 45, provided certain requirements are met.

To use the 45° function, we must first select **CAM** from the main menu. Then activate the "pop-up" menu and select **45 Degree**. A new menu is displayed and the following parameters should be set.

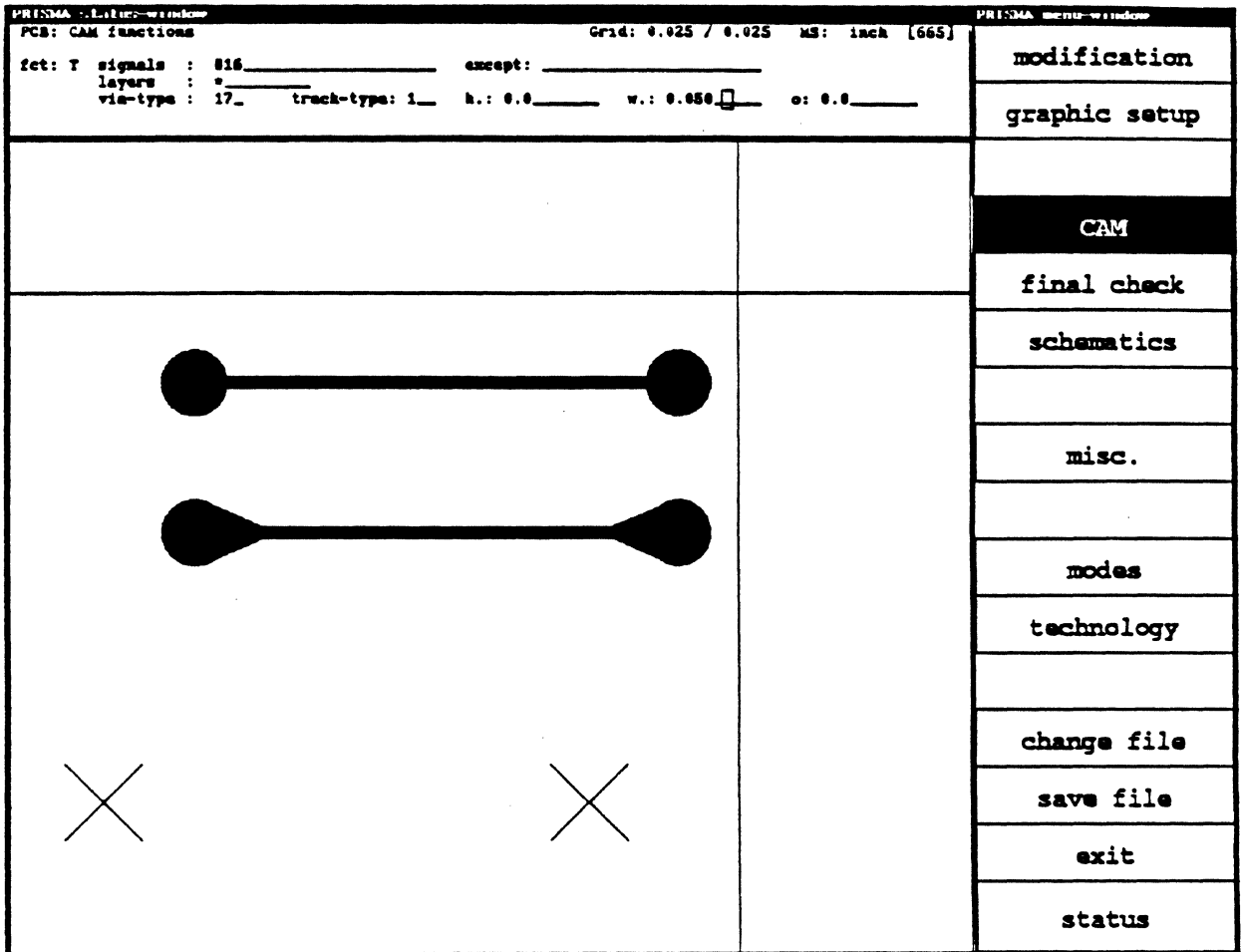
1. SIGNAL: Specifies which signal or signals will be evaluated (\* is used as a wildcard)



2. **EXCEPT:** Used when a wildcard has been used in the signals menu. Signal names entered will be ignored.
3. **LAYERS:** Specifies which layers are to be evaluated.
4. **LENGTH:** Specifies the search range from the center of the pad or via. The search range must encompass the entire segment you wish to modify or it will not be changed.
5. **V-TYPE:** Specifies which via type will be evaluated (no wildcards).
6. **"ALL LAYER":** If this field is marked, all via types that  
**VIAS ONLY** have the letter D in the technology menu will be evaluated.

After all data is entered, a <CR> will update all specified elements.

CAM FEATURES TEARDROPS



- TEARDROPS - Creates teardrops for specified pads and vias.
- REMOVE TEARDROPS - Removes teardrops from specified pads and vias.

The teardrop command allows the user to globally or selectively "teardrop" both pads and vias. NOTE: that SMD's are not supported.

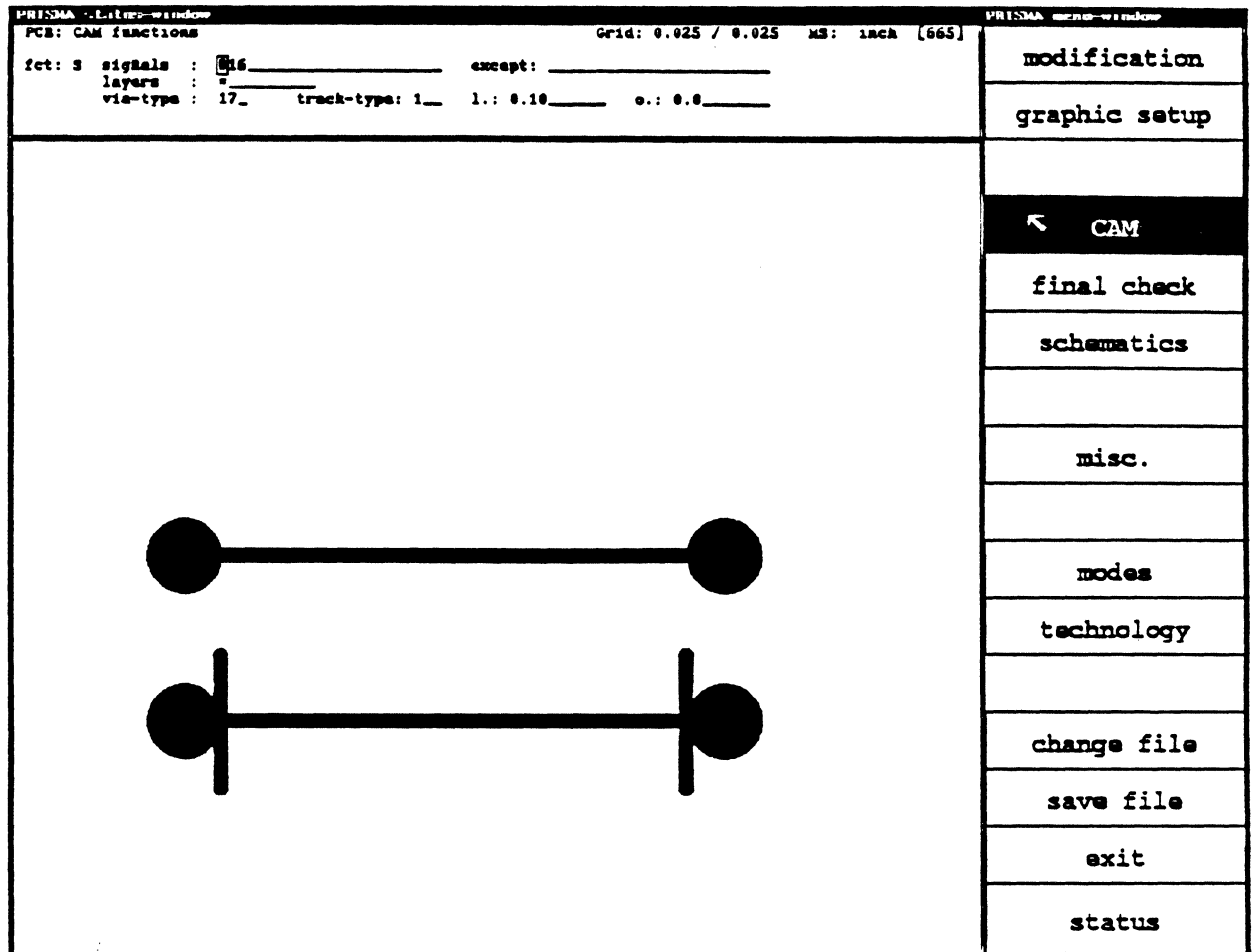
To use the teardrop function, we must first select **CAM** from the main menu. Then activate the "pop-up" menu and select **TEARDROP**. A new menu is displayed and the following parameters should be set:

1. **SIGNAL:** Specifies which signal or signals will be evaluated (\* is used as a wildcard for all).
2. **EXCEPT:** Used when a wildcard has been specified in the signal menu. Signals entered here will be ignored.
3. **LAYERS:** Specifies which layers are to be affected.
4. **VIA TYPE:** Specifies which via type is to be evaluated (no wildcards).
5. **TRACK TYPE:** Specifies which track type is to be used for teardrop graphics.
6. **H:** Specifies from the center of the pad or via how far the teardrop graphics will draw.
7. **W:** Refers to the width desired for the via type. Normally, this is the width of the via - the track diameter used to draw the graphics.
8. **O:**

After all data is entered, a <CR> will update all specified elements.

**NOTE:** To remove teardrops, activate the pop-up menu and enter the "Remove Teardrops" command. It will require that all fields are set except for the H, W, and O: fields.

CAM FEATURES SUBLANDS



**SUBLANDS -** Adds additional copper to specified pads or vias at the point where the track enters the pad or via.

The Sublands function allows the user to either globally or selectively add additional copper at the point of intersection between tracks and pad or tracks and vias.

To use the Subland application, we must first select **CAM** from the main menu. Then activate the "pop-up" menu and select **SUBLANDS**. A new menu is displayed and the following parameters should be set:

1. **SIGNALS:** Specifies which signal or signals will be evaluated (\* is used as wildcard).
2. **EXCEPT:** Used when a wildcard was used in the signals field. Signal names entered here will be ignored.
3. **LAYERS:** Specifies which layers are to be evaluated.

- 4. VIA TYPE: Specifies which via type is to be evaluated (no wildcards).
- 5. TRACK TYPE: Specifies which track code is to be used for the added graphics.
- 6. L (LENGTH): Specifies the length of the track to be added.
- 7. O (OFFSET): Specifies the offset desired in reference to the point of intersection.

After all data is entered, a <CR> will update all specified elements.

NOTE: Sublands are removed with the Remove Teardrops function.

SEE : "Teardrops" earlier in this section.

**CAM FEATURES    NECKING**

**NECKING** -Used to correct technology errors by automatically shifting tracks or by necking down to a specified track code in the areas in error.

The necking function allows the user to correct Design Rule Errors in two ways. The first will, if possible, shift only the portion of the track that is in error; the second will allow the specification of a smaller track to be "spliced" into the area in error, thus creating a neck down to correct the Design Rule Error.

To use the necking function, we must first select CAM from the main menu. Then activate the "pop-up" menu and select NECKING. A new menu is displayed and the following parameters should be set:

1. **SIGNAL:** Specifies which signal or signals will be evaluated (\* is used as a wildcard).
2. **EXCEPT:** Used when a wildcard has been specified in the signals menu. Signal names entered here will be ignored.
3. **LAYERS:** Specifies which layers are to be evaluated.
4. **TRK-TYPE:** Specifies which track type is to be evaluated.  
  
Specifies the track code you wish to use for the areas in error. If you wish to shift the areas in error, this will be the same as trk-type.
5. **SHIFT:** If this field is marked, your tracks will be shifted, if possible to avoid the Design Rule Error.

---

After all data is entered, a <CR> will update all specified elements.

**CAM FEATURES VIA DISTANCE**

**VIA DISTANCE -** Allows the user to increase his clearance around routed vias by up to .012 . Note: This feature only works with specific technologies.

The via distance function allows the user to either globally or selectively create greater clearance for routed vias. This function will only affect tracks which are within one grid point of the specified via type. So realistically, this function will only be useful when routing fineline micro-via boards (.015 - .020).



To use the via distance function, we must first select **CAM** from the main menu. Then activate the "pop-up" menu and select **VIA DIST**. A new menu is displayed and the following parameters should be set.

1. **SIGNALS:** Specifies which signal or signals will be evaluated (\* is used as a wildcard).
2. **EXCEPT:** Used when a wildcard has been specified in the signals menu. Signals entered here will be ignored.
3. **LAYERS:** Specifies which layers are to be evaluated.
4. **DISPLACE:** Specifies the distance to move track from via. (NOTE: The max. distance allowed is the routing grid -2).
5. **V.TYPE:** Specifies which via type is to be evaluated.
6. **"ALL LAYERS":**  
**VIAS ONLY** Specifies that only those vias with the attribute "D" in the technology list will be evaluated. No buried vias.

After all data is entered, a <CR> will update all specified elements.

# TABLE OF CONTENTS

## B. CIS REFERENCE MANUAL

Introduction . . . . .	2
Menu-driven handling . . . . .	2
Term definitions . . . . .	3
Function keys . . . . .	5
1. Modification . . . . .	6
Full view . . . . .	8
Window loc . . . . .	8
Refresh . . . . .	8
Center . . . . .	8
Grid ON/OFF . . . . .	8
Restore . . . . .	8
Zoom . . . . .	8
Grids . . . . .	9
Digi-grid . . . . .	10
Disp-grid . . . . .	10
Place-grid . . . . .	10
Route-grid . . . . .	10
Color on . . . . .	11
Color off . . . . .	11
Color clear . . . . .	11
Set color . . . . .	11
Std. Color . . . . .	12
Scale . . . . .	12
Optic . . . . .	12
Pan ON/OFF . . . . .	13
Window . . . . .	13
Full view . . . . .	13
Window loc . . . . .	13
Get symbol . . . . .	15
Status window . . . . .	16
Create symbol . . . . .	16
Insert symbol . . . . .	17
Move symbol . . . . .	18
Rename symbol . . . . .	18
Modify symbol . . . . .	19
Copy symbol . . . . .	19
Delete symbol . . . . .	19
Swap symbol . . . . .	19
Replace symbol . . . . .	20
Scale symbol . . . . .	20
List symbol . . . . .	21
Digitize symbol . . . . .	21
Change symbol logic . . . . .	22
Open symbol . . . . .	22
Close symbol . . . . .	22
Popup . . . . .	22
Get terminal . . . . .	23
Create terminal . . . . .	23

	Insert terminal . . . . .	24
	Move terminal . . . . .	24
	Rename terminal . . . . .	24
	Modify terminal . . . . .	24
	Copy terminal . . . . .	24
	Delete terminal . . . . .	24
	List terminal . . . . .	25
	Digitize terminal . . . . .	25
	Change terminal logic . . . . .	25
	Get signal . . . . .	26
	Create signal . . . . .	26
	Insert signal . . . . .	26
	Rename signal . . . . .	27
	Modify signal . . . . .	27
	Auto-snap . . . . .	28
	X,Y VALID . . . . .	28
	45 DEG. VALID . . . . .	28
	Nonorthogonal . . . . .	28
	Tap connect mode . . . . .	28
	Ignore symbol-coll . . . . .	28
	Ignore term.-Collis . . . . .	28
	Complete signal . . . . .	28
	Immediate kill . . . . .	28
	Implicit sig -> trm . . . . .	29
	Implicit trm -> sig . . . . .	29
	Impl. Sig -> bus . . . . .	29
	Impl. Siga -> sigb . . . . .	29
	List signal . . . . .	30
	Digitize signal . . . . .	30
	Status window at signal/mod. Parameter . . . . .	30
	Next signal . . . . .	30
	Get bus . . . . .	31
	Create bus . . . . .	31
	Insert bus . . . . .	31
	Delete bus . . . . .	32
	Tap bus . . . . .	32
	List bus . . . . .	32
	Status window . . . . .	33
	Format popup . . . . .	36
	Modify graphic . . . . .	38
	Modify parameters . . . . .	41
	Route mode . . . . .	45
	Route mode off . . . . .	45
	Route mode on . . . . .	45
	Continue . . . . .	45
	Multi . . . . .	45
2.	Graphic setup . . . . .	46
3.	Final check . . . . .	51
4.	Miscellaneous . . . . .	52
5.	Modes . . . . .	57
6.	Technology . . . . .	60
7.	Change file . . . . .	63
8.	Save file . . . . .	64
9.	Exit . . . . .	65

10. Status . . . . .	66
11. Modification . . . . .	67
Colors . . . . .	74
I. Definition of the colors . . . . .	74
II. Assignment of the colors and the kind of display	
to the board elements . . . . .	75
Specifying the element representation . . . . .	76
Color specifications . . . . .	77
Graphic setup . . . . .	113
Final check . . . . .	118
Miscellaneous . . . . .	119
Report . . . . .	120
Plot . . . . .	121
Modes . . . . .	122
Technology . . . . .	125
Change file . . . . .	145
Save file . . . . .	146
Exit . . . . .	147
Status . . . . .	148

**C I S**  
**R E F E R E N C E M A N U A L**

The CIS Reference Manual is broken into two separate sections:

Section one deals with the Schematic

Section two deals with the Layout

## INTRODUCTION

The handling of the CAD system "CALAY PRISMA" is menu-driven and independent from a program language.

To guarantee a more flexible handling, there is no hierarchy in the menus.

In this manual, the menus will therefore be described in the sequence of their appearance.

"CALAY PRISMA" offers different handling possibilities:

## MENU-DRIVEN HANDLING

In most cases, the functions are selected in the menu window on the right side of the screen.

The selection of a function or a work menu is made by positioning the cursor arrow on the respective field on the screen and pressing the left mouse button ("mouse-click"). This input possibility will especially be described here. The three mouse buttons will be called <LMB> (left mouse button), <MMB> (middle mouse button) and <RMB> (right mouse button) in the following.

## "STATUS WINDOW"

The mouse menus are not the only handling possibility of the system. The entries can also be made alphanumerical and with the help of the arrow keys in the "Status window" (in the upper left corner of the screen).

Some functions of the system are only available in the status windows, e.g. if you want to enter numerical standard values. In these cases, the handling and the function of the respective menu in the status window is described.

## POPUP MENUS

In the menu window, all functions are available any time.

An advanced user however might prefer the much faster function selection with the popup menus. They are called by pressing the right mouse button (<RMB>) after the popup mode has been switched on with the function key <R11>.

At the current cursor position, a small "popup" window appears. It disappears when releasing the right mouse button. By moving the mouse, you can activate the function of your choice. The activated function is highlighted in red.

By releasing <RMB>, the window disappears and the selected function is executed. Popup menus will only be described in the manual if they offer functions that are not available in the standard menu window.

## TERM DEFINITIONS:

The following terms whose meaning is explained here will be used in the PRISMA programs:

### 1. COMPONENT

Stands for the element existing on the PC board (not relevant to COLUMBINE).

### 2. PCB NAME

The placement name of a component on the PC board (e.g. IC1 - not relevant to COLUMBINE).

### 3. SHAPE

The physical expression of a component that consists of the following elements:

#### a. Shape body

Consists of the shape terminals and the shape extension (area that the shape needs on the PC board). The following information belongs to a shape terminal:

- terminal position (referring to the lower left corner of the shape);
- graphical display of the terminal on the respective signal layers in the layout.

Note:

The terminal graphics is determined by assigning a via type and/or a track type (with segment length) to the terminal in CIS; in addition, it may contain graphical elements (lines and texts) that will be digitized to the terminal by the user in CIS with the function "Modify graphic".

#### b. Shape graphics 1

Consists of the graphical display of the shape frame, the assigned text and the additional graphical elements (lines and texts - no terminals).

Note:

At present, the graphical display of the frame is automatically created during the generation of shapes in CIS - it corresponds to the extension of the shape.

With the function "Modify graphic", the shape graphics that consists only of the frame can be supplied or changed.

#### c. Shape graphics 2

This is an alternative graphical display that may be used instead of graphics 1 (e.g. DIP14F graphics as alternative to the DIP14 graphics).

## 4. SYMBOL

Stands for the logical side of a component and represents an electrical function element (e.g. 7400 or NAND). A symbol is defined with the following elements:

### a. Symbol body

Consists of the symbol terminals and the symbol extension (area that the symbol needs in the schematics).

Note:

When inserting standard symbols (e.g. the symbol 7400) from the library, the extension of the symbol is not visible in CIS!

### b. Symbol graphics 1

Consists of the graphical display of the symbol (lines and text - no terminals) generated in CIS.

Note:

When generating symbols in CIS, the extension of the symbol is automatically displayed as 'frame' (marking of the extension). With the function "Modify graphic", the symbol graphics, consisting only of the 'frame' at first, can be supplied or changed.

### c. Symbol graphics 2

Specifies the 'inverse' graphics to graphics 1.

## 5. SCS NAME

This is the placement name of a symbol in the schematics (not relevant to COLUMBINE).

## 6. FUNCTION TYPE

Specifies a function specific standard component (e.g. the 4 NAND component 74LS00) that may exist in one or several 'variants'. According to the number of variants, a function type contains one or several relations between shape(s) (e.g. 4 NAND symbols 7400) and their terminals as well as the assignment of a stock ID to each relation. If several relations exist for one function type, the definite selection of a 'variant' (represented by a concrete component) can only be achieved with the stock ID.

## 7. STOCK ID

Definite identification criterion for a function type 'variant', i.e. for a concrete component (see item 6). With the stock ID, the assignment of device parameters for insertion machines and testers is effected.



## FUNCTION KEYS

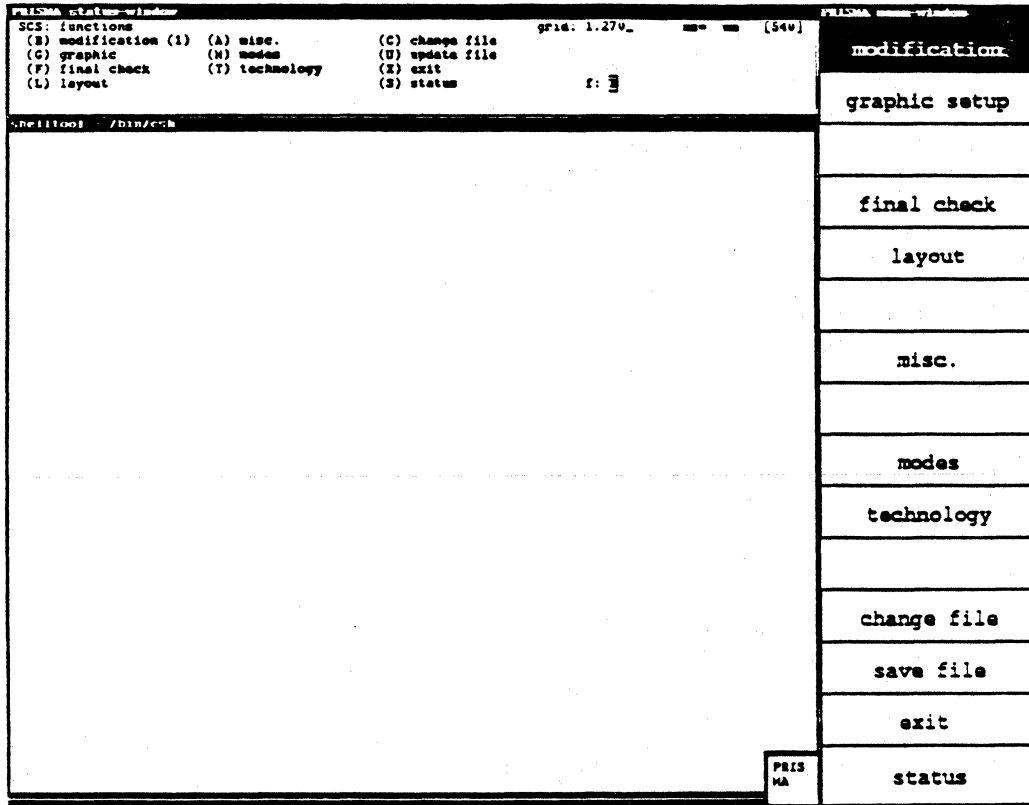
In CALAY PRISMA, the function keys (F1-F9, L1-L10, R1-R15) have the following meaning:

F1 = modify graphics  
F2 = switch SCS <--> PCB  
F3 = modify parameters  
F4 = free  
F5 = layer pair in front/behind (only PCB)  
F6 = mixed color change vias (only PCB)  
F7 = SHELL-ESCAPE  
F8 = Graphics in front/behind  
F9 = Subwindow in front/behind (only PCB)

L1 = occupied by SUN  
L2 = SUBMENU on/off  
L3 = get symbol/component  
L4 = get signal  
L5 = digitize  
L6 = delete  
L7 = shift  
L8 = insert  
L9 = get  
L10 = copy

R1 = status window REPRINT  
R2 = HELP key (not implemented)  
R3 = TTY-HOME  
R4 = PRINT status window  
R5 = TTY window in front/behind  
R6 = MENU window in front/behind  
R7 = fix menu field  
R8 = menu-IO-field (arrow up)  
R9 = panning  
R10 = menu-IO-field (arrow left)  
R11 = POPUP on/off  
R12 = menu-IO-field (arrow right)  
R13 = list  
R14 = menu-IO-field (arrow down)  
R15 = center

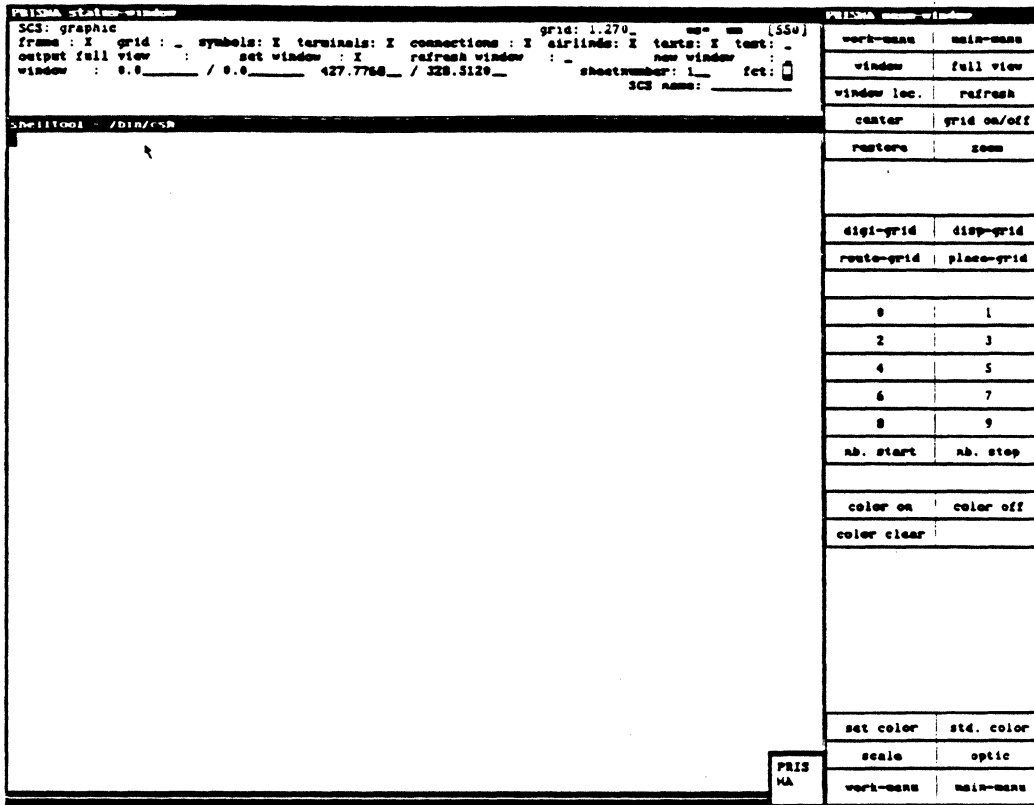
## 1. MODIFICATION



The menu "Modification" which supports the graphics is the largest menu in the system. Standard values like line width, grid size and colors, can be determined in the menus "Graphic setup", "Technology" and "Modes".

The functions do not have a hierarchy so that the user can select all executable functions at any time. These functions can be combined in five units:

- (1) Help for the graphic display (graphic parameters, grids, windows, etc).
- (2) Objects (switch symbols, connection points, signals)
- (3) Actions that can be executed with these objects (edit)
- (4) Graphic elements
- (5) Functions that allow the menu selection or the connection of the subsidiary items (e.g. a repeat function)



The menu "Graphic setup" offers the following functions:

**WORK MENU**

You return to the superior menu "Modification".

**MAIN MENU**

You return to the main menu.

**WINDOW**

Here, you can define a new window.

Procedure:

Move the cursor to the place where you want to define the lower left corner of the window. With the left mouse button (called <LMB> in this description), you fix this corner. You can then open the window with the mouse until it has the desired size. The final size of the window is determined by pressing <LMB>. With <RMB>, this procedure can be interrupted. If you press <MMB> in the middle of the screen, the current window will be zoomed. At the edges of the graphic screen, <MMB> may be used as shifting function, i.e. the current window moves in the direction of the crosshair.

## FULL VIEW

Deletes the current window and shows the total work sheet again.

## WINDOW LOC.

For very large windows, there is a useful orientation help:

By selecting "Window loc.", the current window is represented by a symbol in relation to the total size of the work sheet. Here, you may choose a new window directly: move the crosshair to the place where the lower left edge of the window is to be defined. Fix this edge with <LMB>. You can then open the window with the mouse until it has the desired size. The final size of the window is determined by pressing <LMB>.

## REFRESH

With this function, the screen is cleared and you get a new artwork. It contains the current data.

## CENTER

Sometimes, it is necessary to keep the current scale but to represent a slightly different window.

When selecting this function, the crosshair position becomes the new center of the picture. Note that it is not possible to move the window over the left edge of the sheet (the lower left corner is the coordinate origin). If you enter a name in the status window in "SCS name", the system tries to move the respective element to the center.

## GRID ON/OFF

With this function, the grid may be switched on/off. The kind of presentation of the grid can be defined in "Color" in the status window.

If the grid is not needed for the placement, it should be switched off since it has to be drawn anew for each picture; this delays the artwork.

## RESTORE

Works like "Refresh" - but the contents of the picture is not calculated anew, it will be newly written by means of the graphic memory (this is faster).

## ZOOM

As known from the photo optics, you can determine a part of the picture here that shall be zoomed. With the crosshair, you define the section to be enlarged (with <LMB>) or reduced (with <MMB>) by a constant factor. With <RMB>, you can interrupt the procedure.

GRIDS

According to the user's requirements, different grids are available. The resolution can be changed for the display, the digitizing or the routing. The user can choose between 9 standard settings and one optional setting. The standard settings must have been defined manually in the status window before. The crosshair is moved with the arrow keys or the tab key. Please do not forget to store your entries with <RETURN>!

In the menu, you can select the desired grid with the mouse or with a respective entry in the status window.

Ex.: Setting the "Disp-grid":

- (1)Click "Disp-grid" with <LMB>.
- (2)Click the desired standard setting, e.g. "4" in the menu window with <LMB>.

In the status window, the current values appear for the different grids (digitizing, auxiliary, and routing grid). The measurements (either inches or mm) to be assigned to the numbers can be defined in the menu "Modes".

The current measurement system is shown in the field "ms". (When changing the units, the system automatically changes the entry).

The presentation of the grid - lines or points - can only be defined in the function "Color" at present.

The screenshot displays two windows from the PRISMA software. The top-left window is titled 'PRISMA - status window' and contains the following text:

```

SCS: grid adjustment      grid: 1.270_   ms= mm [410]
0 = 0.1500_   4 = 2.540_   8 = 0.4997_   interactive grid: 0.270_   (1.270_ )
1 = 0.3175_   5 = 3.810_   9 = 1.0001_   display grid : 2.540_   (2.540_ )
2 = 0.6350_   6 = 5.080_   autoroute grid : 2.540_   (2.540_ )
3 = 1.270_   7 = 10.160_   hit <RETURN>-key after entering the values via form !
    
```

The top-right window is titled 'PRISMA menu window' and contains a menu structure:

work-mans	main-mans
window	full view
window loc.	refresh
center	grid on/off
restora	zoom
digit-grid	disp-grid
route-grid	place-grid
0	1
2	3
4	5
6	7
8	9
nb. start	nb. stop
color on	color off
color clear	
set color	std. color
scale	optic
PRIS MA	work-mans main-mans

The bottom-left window is titled 'Shelltool - /bin/csh' and is currently empty.

### **DIGI-GRID**

This is the resolution used for the digitizing. The crosshair is moved according to this resolution.

### **DISP-GRID**

Specifies the resolution of the auxiliary grid that can be activated with "Grid on/off".

This value cannot be smaller than the value of the "Digi-grid" (must be a multiple of it).

### **PLACE-GRID**

This is only available in the model "Layout".

### **ROUTE-GRID**

This grid may be chosen freely. In connection with the option "Route mode" (ref. to the menu "Modification"), the resolution with which signals are routed in the SCS-model is defined with the route grid.

### **NUMBER START**

This function is not yet available.

**NUMBER STOP**

This function is not yet available.

**COLOR ON**

This function is activated from the status window. According to the kind of display, colors that have been faded out can be faded in again one by one. Therefore, you click the respective code number in the menu window (0 = all colors).

**COLOR OFF**

This function is also activated in the status window. For better display, it may be useful to fade out some colors.

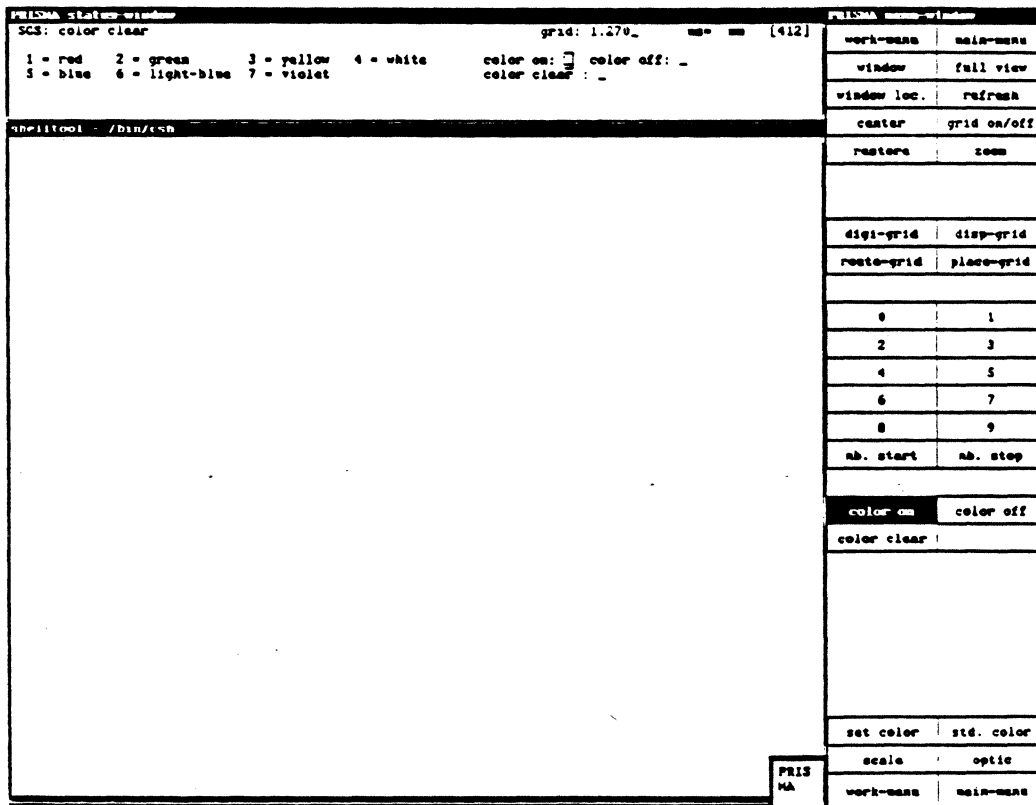
**COLOR CLEAR**

By clicking the respective code number, all objects of this color will be deleted from the screen. When selecting "Refresh", these objects appear again.

**SET COLOR**

Seven colors are available for the creation of the schematics. With this function, the colors are assigned to the objects (symbols, terminals, crosshair, etc). The assignment of color codes (number between 1 and 7) to the objects is made with a respective entry in the status window.

There are two different kinds of display for the auxiliary grid (display grid): as a net with solid lines (grid type L) or as marking of the nodes (grid type P).



### STD. COLOR

With this function, the objects will be displayed in the standard color again (symbols red, text and grid white, terminals green, auxiliary lines violet, crosshair blue, frame light-blue).

### SCALE

For a placement true to the scale, a yard-stick is sometimes necessary. Therefore, you can use this function. You determine the starting point of the scale with the crosshair.

The origin of the scale lies exactly in the crosshair. The scale becomes visible with <LMB>. The display unit is shown in the status window.

#### Status window:

The length and the axial division of the scale can be defined here.

### OPTIC

This function allows the use of graphic figures as optical support:

point (P), cross (X), square (Q), circle (K), rhomb (R).

The assignment of an object to a graphic figure is made with a respective entry in the status window. By pressing <RETURN>, the selected figures are displayed.

In "Text-selection", you can choose the text types to be shown on the screen. After selecting "Refresh", the changes are executed.

For comfort reasons, the "Work menu" and the "Main menu" (ref. to page 2) are offered again.



**PAN ON/OFF**

Here, the pan (or scrolling) function can be switched on or off. But first you have to set "window" or "pan mode" in the menu "Modes".

The functions "Window", "Full view", "Window loc.", "Refresh", "Center", and "Grid on/off" are all needed for a better display. They are also available in the submenu "Modification graphic".

**WINDOW**

Here, you can define a new window.

Procedure:

Move the cursor to the place where you want to define the lower left corner of the window. With <LMB>, you fix this corner. You can then open the window with the mouse until it has the desired size. The final size of the window is determined by pressing <LMB>. With <RMB>, this procedure can be interrupted. If you press <MMB> in the middle of the screen, the current window will be zoomed. At the edges of the graphic screen, <MMB> may be used as shifting function, i.e. the current window moves in the direction of the crosshair.

**FULL VIEW**

Deletes the current window and shows the total work sheet again.

**WINDOW LOC.**

For very large windows (function "WINDOW"), there is a useful orientation help:

By selecting "Window loc.", the current window is represented by a symbol in relation to the total size of the work sheet. Here, you may choose a new window directly: move the crosshair to the place where the lower left edge of the window is to be defined. Fix this edge with <LMB>. You can then open the window with the mouse until it has the desired size. The final size of the window is determined by pressing <LMB>.

**REFRESH**

With this function, the screen is cleared and you get a new artwork. It contains the current data (see also "Restore").

**CENTER**

Sometimes, it is necessary to keep the current scale but to represent a slightly different window.

When selecting this function, the crosshair position becomes the new center of the picture. Note that it is not possible to move the window over the left edge of the sheet (the lower left corner is the coordinate origin). If you enter a name in the status window in "SCS name", the system tries to move the respective element to the center.

### **GRID ON/OFF**

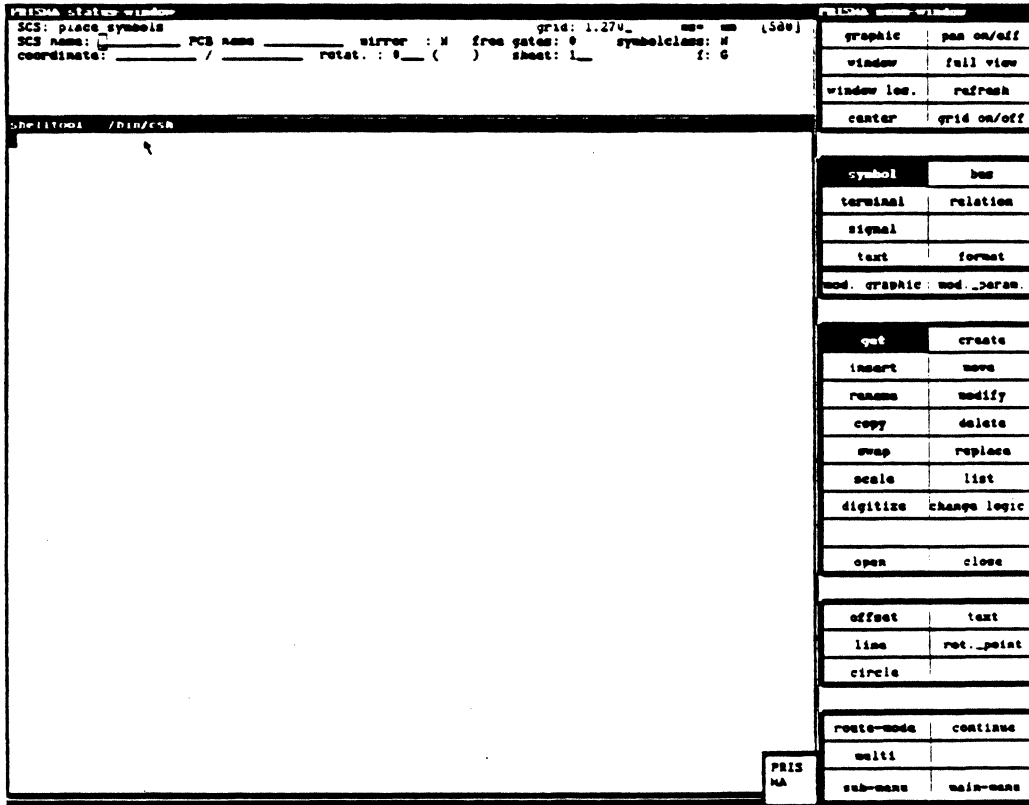
With this function, the grid may be switched on/off. The kind of presentation of the grid can be defined in "Color" in the status window.

If the grid is not needed for the placement, it should be switched off since it has to be drawn anew for each picture; this delays the artwork.

A symbol can be connected with several single operations that will be described here:

**GET SYMBOL**

This function always precedes each operation because all operations are executed for single elements (exception: global changes).



"Get" means activating an already existing symbol for further processing. The activation is achieved by clicking the symbol with <LMB>. Available free gates can be taken up with "Insert".

If you press <MMB> first, a list of all available free gates of the schematics that have not yet been used appears. The specifications in the list have the following meaning:

Example: &320 = {-> C19 [.]}  
 SCS name PCB name already connected on PCB side

Example: &164 = {7474 -> U14}  
 SCS name function type PCB name

By clicking a name with <LMB>, the respective free gate of an already existing symbol will be inserted in the schematics.

The free gates exist on the internal page 0. For further information ref. to "Insert symbol".

With "list", you get a listing of all symbols existing in the schematics. By clicking a name, the respective symbol will be taken up.

The activated object is highlighted (by standard, it will be displayed in blue).

After "Get", the cursor automatically jumps to "Move symbol". With <MMB>, you can rotate the symbol in 90° steps relative to its starting position.

## STATUS WINDOW:

**SCS name:** You can also select a symbol with its SCS name. The respective PCB name appears behind it if the symbol has been related.

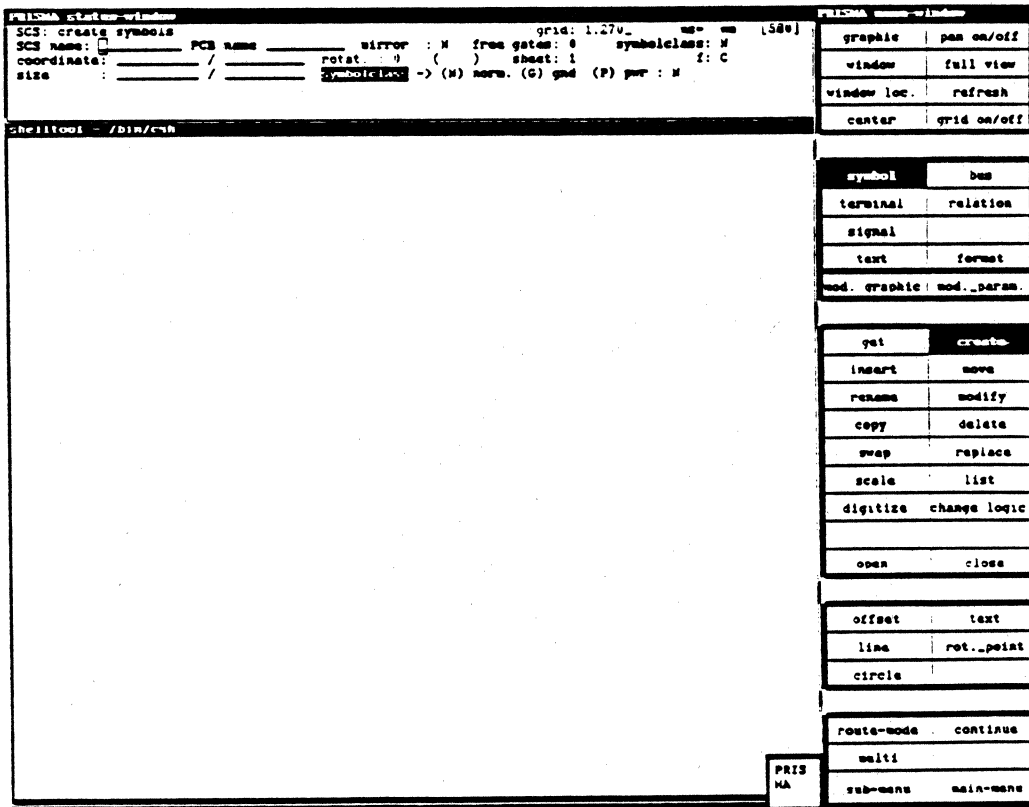
**Coordinate:** You can get a symbol by specifying a coordinate within the desired symbol.

## CREATE SYMBOL

With this function, new symbols that do not yet exist in the library can be taken into the system.

These new symbols are only placeholders at first, represented by a rectangle.

With <LMB>, this contour can be opened from the lower left corner to the upper right side. A name is assigned to each created symbol.

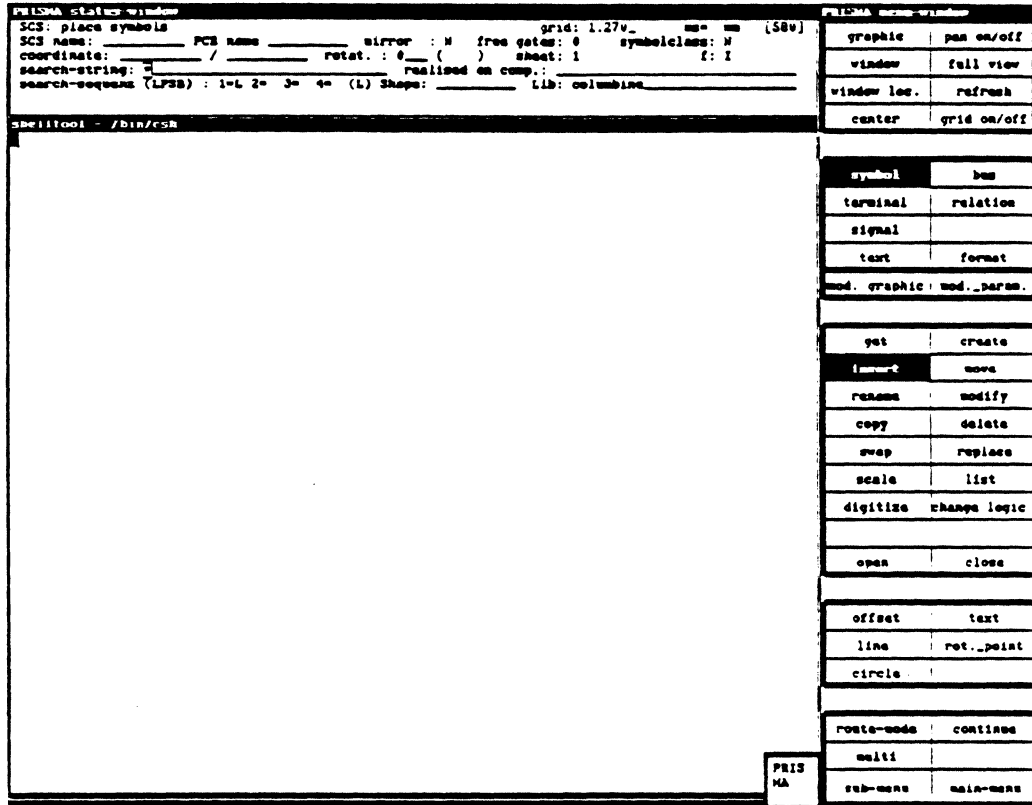


Newly created symbols must be specified in detail in the status window. The symbol name is proposed by the system, but may be changed by the user.

**INSERT SYMBOL**

Allows the access to the library. With "list", the existing symbols will be shown in a window.

The symbol is selected by clicking the respective name with <LMB>. It will be entered in the status window and is attached to the crosshair. The access is also possible by entering the respective name in "Search-string" in the status window.



**STATUS WINDOW**

Allows the access to the library. The selection is made by clicking the respective name in the list or by entering the name in "Search-string".

Search-sequenz:

- L (logical): Demanding a logical function (e.g. NAND)
- P (physical): Function type specific selection (e.g. 74HC10)
- S (stock ID): Selection with internal item number
- B (shape): Demanding a shape type (e.g. DIP14)

The fields have the following meaning:

- Search-string:** e.g. name of logical function (NAND, NOR...)
- realized on:** function type that realizes the function

The coordinates of the symbol origin are displayed after the taking up. The activated symbol can be rotated. The angle is specified in  $1/10^\circ$  (e.g. "90°" corresponds to "900"). The direction of the rotation is mathematically positive, i.e. counter-clockwise. The rotation angle appears in the field "rotation", the angle of the starting point appears in brackets. By clicking <MMB>, 90° rotations are possible.

The symbol class specifies if the symbol represents a conventional component or supply or multiple connections. There are also fields that are relevant for an access to the library COLUMBINE from CIS.

The number of free gates (i.e. non-used gates of existing symbols) is also shown.

If you demand a logical gate function, a function type is taken from the library that contains this gate function (or several, as a rule). If only one gate is used, the others remain free. These free gates are still available.

They can be called in two different ways:

1. In "Get symbol", you can demand a list of all free gates with <MMB>. No symbol must have been selected.
2. You can also use the free gates of a function type that corresponds to a component on the PCB side.

A LS7400 contains 4 NAND gates, for example. The system administers them and informs you about the number of available connections.

Free gates can be taken into the schematics with "copy". Therefore, you have to take up a symbol with free gates and copy it. The number of remaining gates is immediately shown in the status window.

### MOVE SYMBOL

The element to be shifted must be activated with <LMB>.

When moving the mouse, the contour of the selected symbol is also shifted.

Move to the desired position and fix it with <LMB>. "Collisions" with other symbols cause error messages in the status window. "Move symbol" remains activated until the respective symbol has been moved to its new position. Rectangular arrangements may also be entered in the text window: rotation in  $1/10^\circ$  steps.

### RENAME SYMBOL

You can assign a name to the activated symbol if the numbering during the generation becomes tangled. In the status window, you can enter the desired name with the keyboard.

The name may consist of up to 10 characters.

**MODIFY SYMBOL**

This operation must always be selected if you want to manipulate (delete, move) the terminals belonging to a symbol.

The operations can only be executed for activated elements.

Procedure:

- (1) Click the symbol with <LMB>.
- (2) Select "Modify symbol". (The symbol remains activated, the cursor moves to "Get terminal").
- (3) Modify the terminals.

**COPY SYMBOL**

An activated symbol is copied to the crosshair position with <LMB>.

The cursor jumps directly to "Move symbol".

When copying symbols, the following has to be considered:

Gates of function types that already exist on the schematics will be duplicated. The next free gate of the respective function type will be taken. In this way, the number of free gates is always reduced by one.

If no free gate is available any more, a new gate of the same function type is taken. On the PCB side, this means the use of a new component element; further copying of the gate effects the repeated "use up" of the free gates of the new PCB element.

**DELETE SYMBOL**

An activated symbol is deleted by clicking <LMB> in the field "delete". In the submenu, you may have to confirm this function with <RETURN> or <LMB>.

**SWAP SYMBOL**

With this function, a pair of already drawn symbols can be exchanged. In the ROUTE MODE, the connections are moved together with the symbols.

Procedure:

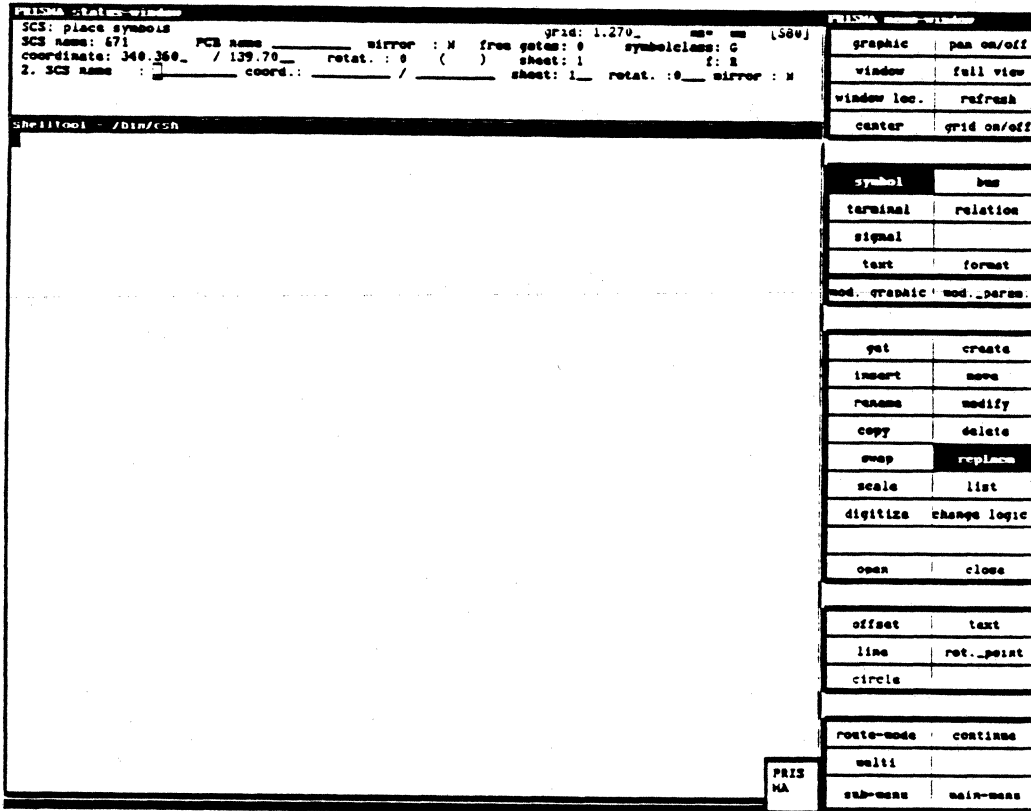
1. Get the first symbol.
2. Select "Swap symbol".
3. Click the second symbol.

## REPLACE SYMBOL

With this function, you can replace one symbol by another.

Procedure:

1. Click symbol on the schematics.
2. Select "Replace symbol".
3. Get the symbol to be replaced.



## SCALE SYMBOL

With this function, already drawn symbols can be displayed with a new scale.

Enlargements or reductions in x or y direction are also possible. After selecting the function "Scale", a window appears in the lower left corner whose size can be changed by moving the mouse. The window shows the new "scaled" symbol size that can be fixed with <LMB>.

Then the cursor automatically jumps to "Move symbol". The connections remain and will be digitized in an active route mode, if possible.



**LIST SYMBOL**

This function can only be used in connection with the operations "get", "create" or "insert".

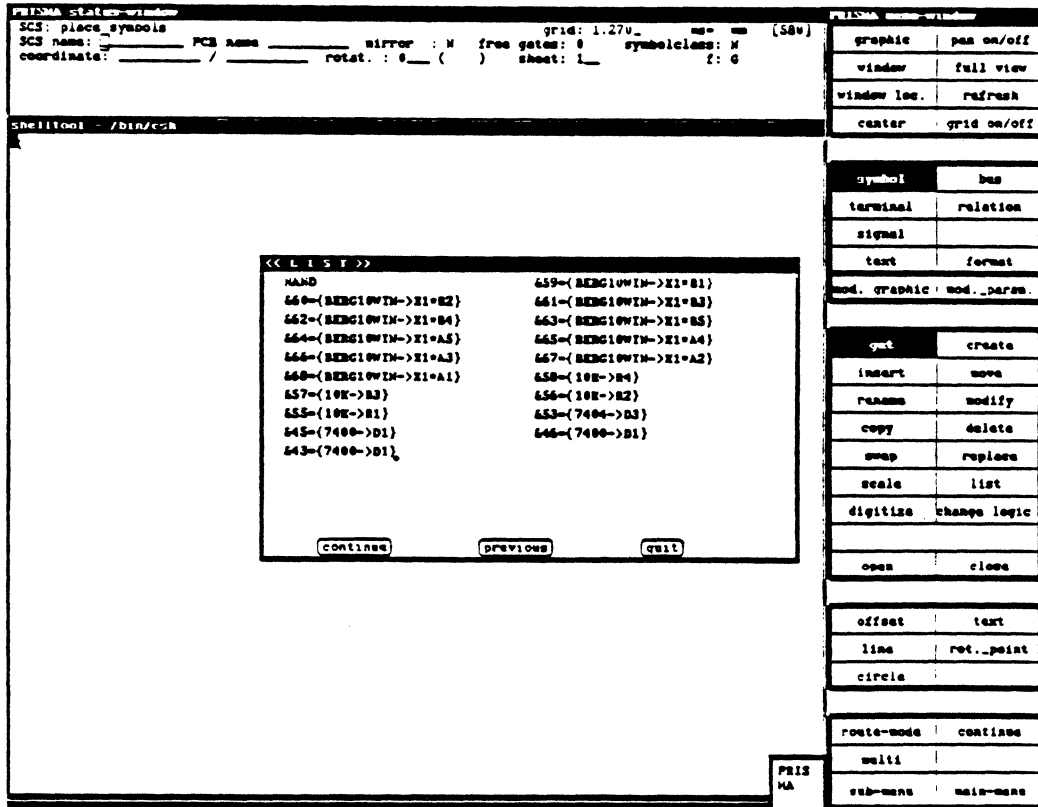
A list of symbol names will be displayed in a text window that appears on the screen. By clicking the fields "continue" or "previous", paging is possible. The symbol is selected by clicking the respective name.

You can only continue your work in the schematics if a symbol name has been selected or the window has been closed by clicking "quit".

In connection with the operations "Get symbol" or "Create symbol", a listing of the symbols that already exist on the schematics is output.

If you click "Insert symbol" and "list", a list of all symbols available in the library is shown.

All other combinations require the activation of a symbol with "Get symbol". For this reason, the list of the already used symbols is output each time.



**DIGITIZE SYMBOL**

This function can only be used in connection with "Modify graphic".

### **CHANGE SYMBOL LOGIC**

Allows the switch to the complement logic (i.e. switch from negative to positive logic and vice versa) if the symbol has two display modes (e.g. US-norm and ISO-display).

Generating a second display mode:

Get symbol and click "Change symbol logic" twice with <LMB>. In this way, the 2. display mode is created and exchanged with the 1. display mode. But the second mode is now displayed like the first one. In order to change the 2. display mode, use the function "Mod. graphic".

### **OPEN SYMBOL**

When creating block diagrams, this function allows the internal "wiring" of block symbols. An activated symbol that was opened in this way appears enlarged on the screen. Existing terminals are also shown. (This function is only possible for logical function blocks.)

### **CLOSE SYMBOL**

Finishes the internal "wiring" and switches to the "normal" display.

### **POPUP:**

Besides the operations in the modification menu, the popup menu also offers the functions "copy incl. hierarchy" and "def. superior group".

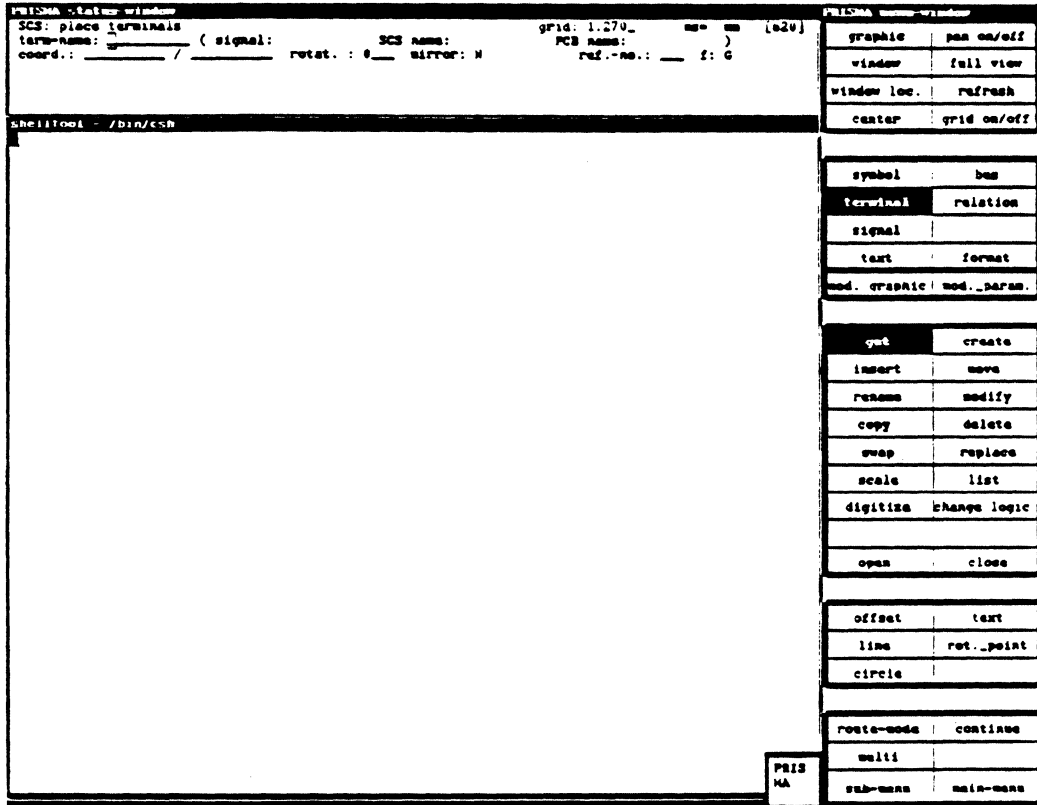
With "copy incl. hierarchy", a group can be copied together with all its subgroups.

With "def. superior group", symbols that have been selected with MULTI can be combined in a superior group.

At present, the symbols must be within the open window. After the release, a new superior group that contains the selected symbols will be created.

**GET TERMINAL**

In the option "get", terminals (connection points) are activated with <LMB> for further processing. The activated terminal is highlighted. For comfort reasons, the cursor automatically jumps to "Move terminal".



**CREATE TERMINAL**

Terminals can be created at all positions that are not occupied by other objects. If you want to assign terminals to a symbol, the symbol must be preselected with the option "Modify symbol".

The terminal belonging to a symbol may also lie outside the symbol frame, but can only be manipulated in the preselection of the element. A name is automatically assigned to each terminal. This name is shown in the status window.

You are also informed about the signal to which the terminal is connected, the symbol to which it belongs and the respective element on the PCB (layout) side if the terminal does not belong to a subordinate group (box).

Terminals can be created in groups:

A repeat function with <RMB> is available in the option "create" (popup switched off).

### **INSERT TERMINAL**

Not yet available.

Please use the function "Create terminal".

### **MOVE TERMINAL**

This function is used for the shifting of connections. Terminals belonging to a symbol (ref. to status window) can only be shifted if the respective element has been preselected (ref. to "Modify symbol").

"Free" terminals that do not belong to a symbol can be shifted freely.

### **RENAME TERMINAL**

Activated terminals can be renamed with a respective entry in the status window.

This operation is possible for all terminals.

### **MODIFY TERMINAL**

This function only makes sense in connection with the option "Modify parameters".

In the status window, the terminal attributes can be changed with the keyboard.

### **COPY TERMINAL**

"Free terminals", i.e. terminals that are not assigned to a symbol can always be copied.

The copy contains the specification of the original (@, followed by a number).

Terminals belonging to symbols can only be copied after the preselection (ref. to "Modify symbol").

### **DELETE TERMINAL**

According to the terminal attributes (ref. to "Modify terminal", "Modify parameters"), "free" terminals that do not belong to a symbol can be deleted.

Terminals belonging to the symbol can only be deleted after the preselection of the respective symbol (ref. to "Modify symbol").

### **SWAP TERMINAL**

Invalid operation

### **REPLACE TERMINAL**

Invalid operation

### **SCALE TERMINAL**

Invalid operation

**LIST TERMINAL**

The lists created with this option are shown in an additional window.

By clicking the fields "continue" or "previous", paging is possible. With <LMB>, the desired terminal can be chosen directly from the list.

If you do not select a terminal and leave the mask with "quit", an error message appears that disappears when selecting a new operation.

Without preselection of a respective symbol, all "free" terminals will be listed.

When preselecting a special symbol and clicking "modify" and "list", all assigned terminals will be listed. The corresponding signal is specified in parentheses.

**DIGITIZE TERMINAL**

With this function, the terminal graphic can be completed/changed (like "Modify graphic").

**CHANGE TERMINAL LOGIC**

Invalid operation

The cursor automatically moves back to "Get terminal".

**OPEN TERMINAL**

Invalid operation

The cursor automatically moves back to "Get terminal".

**CLOSE TERMINAL**

Invalid operation

The cursor automatically moves back to "Get terminal".

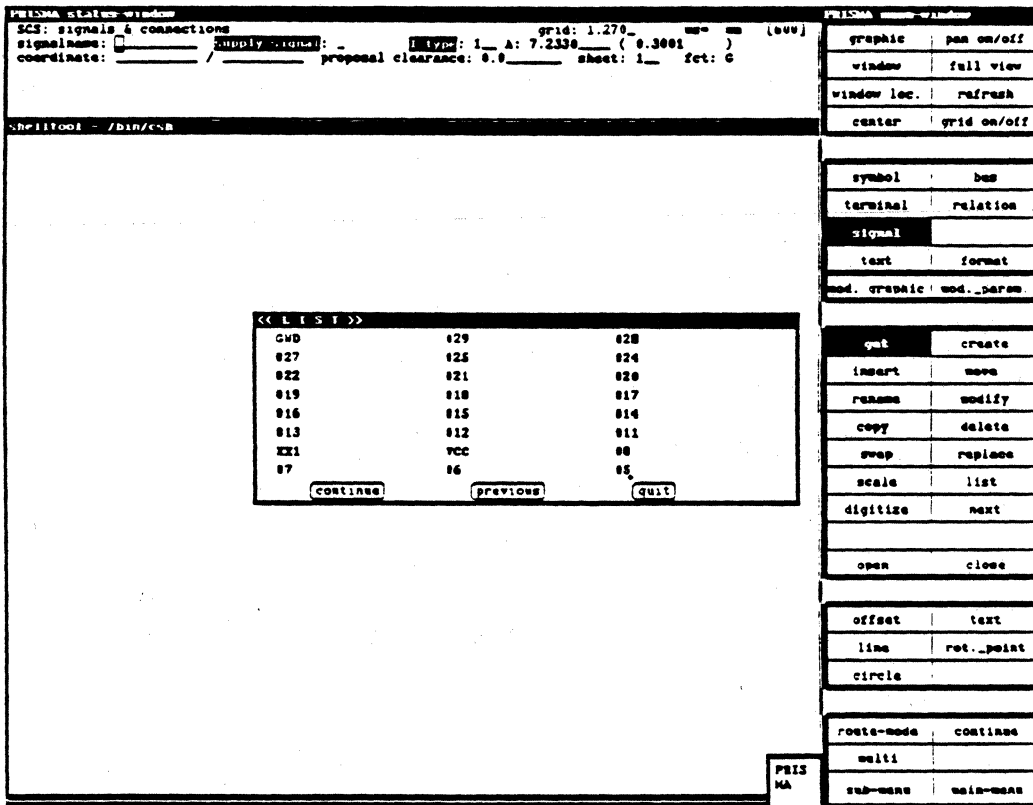
## GET SIGNAL

Signal lines in the crosshair can directly be taken up with <LMB>. In the "submenu", you can specify if the cursor shall move to "digitize" or "insert".

In both cases, the signal is highlighted. With "guided connection" in the submenu, the two ends of the selected signal part will be connected to the crosshair with airlines during the digitizing; one airline is violet, the other is yellow (by default).

A bus is handled like a signal.

As always several signals are connected to the bus, a window is shown after the selection of a bus wire in which all signals are listed. The signal can then directly be selected in the list window.



## CREATE SIGNAL

Here, you can create a new signal by entering a name in the status window.

## INSERT SIGNAL

With this function, further terminals can be connected to a previously activated signal line by clicking the terminals to be connected.

"Packing" of signals is also possible (ref. to the submenu "Impl. sigA -> sigB"). The respective presetting in the submenu ("impl. sig -> bus") effects the connection of a signal to a bus.

**MOVE SIGNAL**

Invalid operation

**RENAME SIGNAL**

By clicking the symbol with <LMB> and entering a new name in the status window, it can be renamed.

**MODIFY SIGNAL**

Only "mod. parameters" possible.

**COPY SIGNAL**

Invalid operation

**DELETE SIGNAL**

After selecting "delete", the activated signal is deleted with <LMB>.

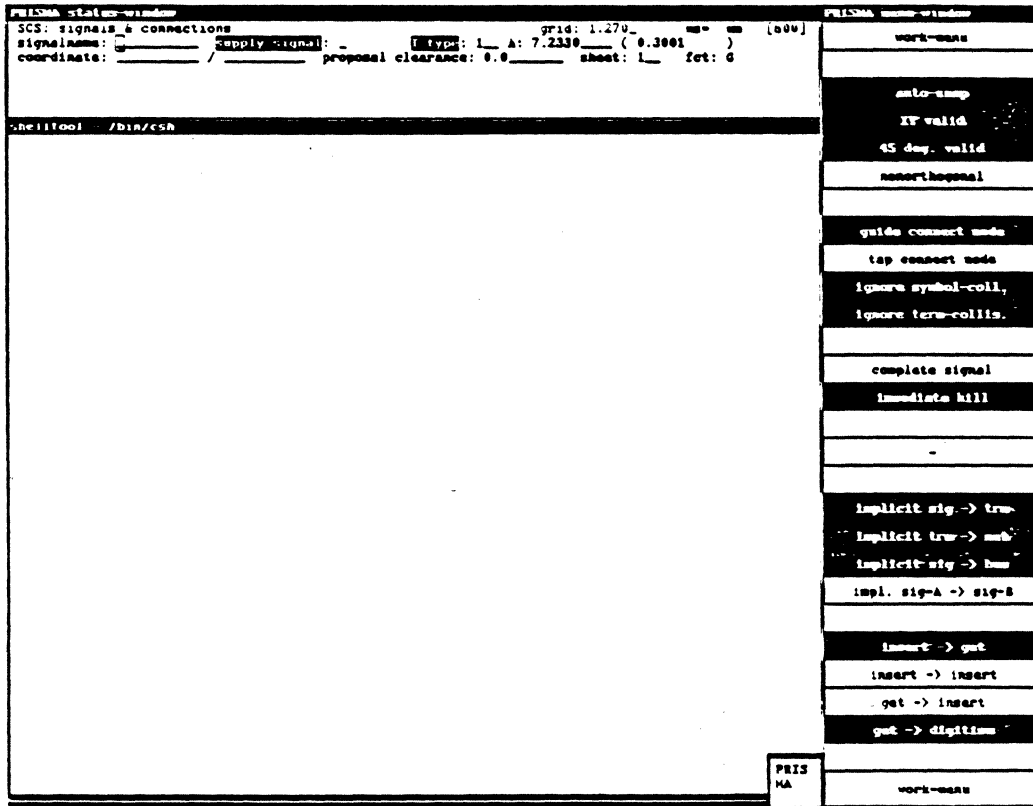
**SWAP SIGNAL**

Invalid operation

**REPLACE SIGNAL**

Invalid operation

**SUBMENU "SIGNALS & CONNECTIONS"**



## **AUTO-SNAP**

Special gradients can be defined for the digitizing of signals (45°, X,Y). Other gradients are not allowed then.

When the auto-snap mode is switched on, the signals are only drawn with the defined gradients.

The work will be facilitated if the user need not adhere exactly to the gradient.

When pressing <LMB>, the system draws the vertical component. If the auto-snap mode is switched off, the error message "Gradient not allowed" appears if the work was inaccurate.

## **X,Y VALID**

As gradient for the signals, only multiples of 90° are allowed, i.e. 90°, 180°, etc.

## **45 DEG. VALID**

As gradient for the signals, only multiples of 45° are allowed, i.e. 45°, 90°, 135°, 180°, etc.

## **NONORTHOGONAL**

All gradients are allowed, but the option "auto-snap" must not be activated.

## **GUIDE CONNECT MODE**

When getting a segment, it is partly displayed as airline, partly as signal line.

## **TAP CONNECT MODE**

When getting a segment, a terminal for the T-connection is created.

## **IGNORE SYMBOL-COLL.**

Error messages concerning symbol collisions may be ignored. By pressing <LMB> twice, the appearing error message becomes ineffectual.

## **IGNORE TERM.-COLLIS.**

See above: the appearing error message will be ignored when pressing <LMB> and the placement is executed.

## **COMPLETE SIGNAL**

Deletes signal on all pages

## **IMMEDIATE KILL**

Allows the immediate deletion by clicking the respective object; you need not press <RETURN> or <LMB>.



### **IMPLICIT SIG -> TRM**

The terminal will be taken into the signal automatically.

### **IMPLICIT TRM -> SIG**

Get signal; click terminal; the terminal will be taken into the signal and connected with the airline.

### **IMPL. SIG -> BUS**

The signals will be taken into the bus (ref. to "Insert bus").

### **IMPL. SIGA -> SIGB**

Combining signals ("packing"). The combined signal gets the name of signal B.

**INSERT -> GET**

**INSERT -> INSERT**

**GET -> INSERT**

**GET -> DIGITIZE**

Here, you can define that the cursor automatically moves to the next function.

### **SCALE SIGNAL**

Invalid operation

### **LIST SIGNAL**

In an additional text window, the names of all signals used in the drawing are listed.

With "continue" or "previous", paging is possible.

You leave the window with "quit" or by selecting a signal name with <LMB>.

### **DIGITIZE SIGNAL**

Even if the route mode is switched on, it is not always possible to route all signals in the schematics automatically (ref. to the submenu "guide connect mode" or "tap connect mode").

In the submenu, the kind of digitizing is defined.

This function is necessary for the manual digitizing. Activated signals are moved with the crosshair as "airlines", by pressing <LMB>, a new part of the line is determined each time. A terminal is placed at the end of each part. Signal collisions or collisions with other objects are shown as error messages in the status window (ref. to the submenu "Ignore signal/terminal collision").

### **STATUS WINDOW AT SIGNAL/MOD. PARAMETER**

All signal parameters that can be modified apply to the connection of the SCS and PCB models. Here is agreed:

- Names on the respective side
- Proposal clearance that must be adhered to during the routing and the placement
- Track type and cross-section
- Route mode (not to be routed, critical, preferred)
- Shielded
- Status signal (complete, technological wrong, global available) etc.

### **NEXT SIGNAL**

Consecutively numbered signal lines (e.g. address or data lines) are automatically incremented.

A new signal is created. The system expects an entry: the cursor moves to "Insert signal".

### **OPEN SIGNAL**

Invalid operation

### **CLOSE SIGNAL**

Invalid operation

**GET BUS**

Like all other objects, bus wires are also taken up with <LMB> for further processing (routing, connection).

The activated bus is highlighted and the cursor automatically moves to "Insert".

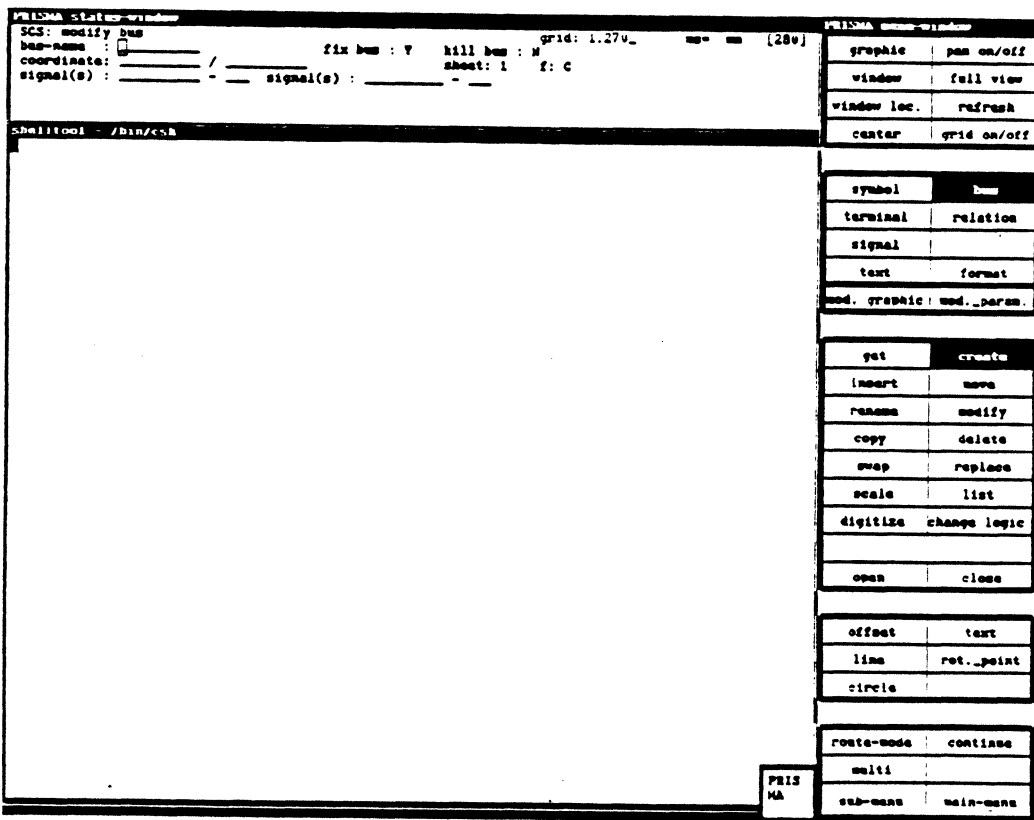
**CREATE BUS**

This function allows the introduction of a bus in the schematics.

Procedure:

- (1) Click the menu function "Create bus" with <LMB>.
- (2) Enter bus name.  
No entry --> system names bus automatically
- (3) Enter signal names that shall belong to bus  
e.g.: signal(s) DB0 - 7
- (4) Digitize bus course with <LMB>.

At present, it is not yet possible to change names with the function "rename".



**INSERT BUS**

With this function, several wires are assigned or combined to a bus.

The cursor automatically moves from "Create bus" to this function. With "Insert bus", you can see which signal lines go to the respective bus.

As in most cases several groups of signal lines (e.g. all address or data wires of a processor) go to a bus, you can enter the symbolic specification of the respective signal lines (e.g. "A0 -A7").

Procedure for inserting signals with the mouse:

- (1) Get bus
- (2) The cursor automatically moves to "Insert": Get the signals to be inserted with <LMB>.
- (3) Select "signal", "get", "digitize" and connect the signal to the bus.

Therefore, "impl. sig -> bus" must be active in the submenu for signals.

**MOVE BUS**  
**RENAME BUS**  
**MODIFY BUS**  
**COPY BUS**

Invalid functions

### **DELETE BUS**

Procedure:

- (1) Get bus with <LMB>.
- (2) Select menu "delete".
- (3) Enter "Y" in "Delete bus".
- (4) Press <RETURN> or <LMB>.

All signals existing in the bus are moved outward.

If "N" is entered in "Delete bus", single signals may be deleted from the bus. Therefore, you have to enter the signal names in "Signal(s)".

### **TAP BUS**

- (1) Activate menu function "Get signal".
- (2) Click bus.  
A window appears in which all signals of the bus are listed.
- (3) Select one of these signals with <LMB>.
- (4) Digitize to the target with <LMB>.

**SWAP BUS**  
**REPLACE BUS**  
**SCALE BUS**

Invalid operations

### **LIST BUS**

In an additional text window, all busses created in the current schematics are listed.

By activating the window fields "continue" or "previous", paging is possible.

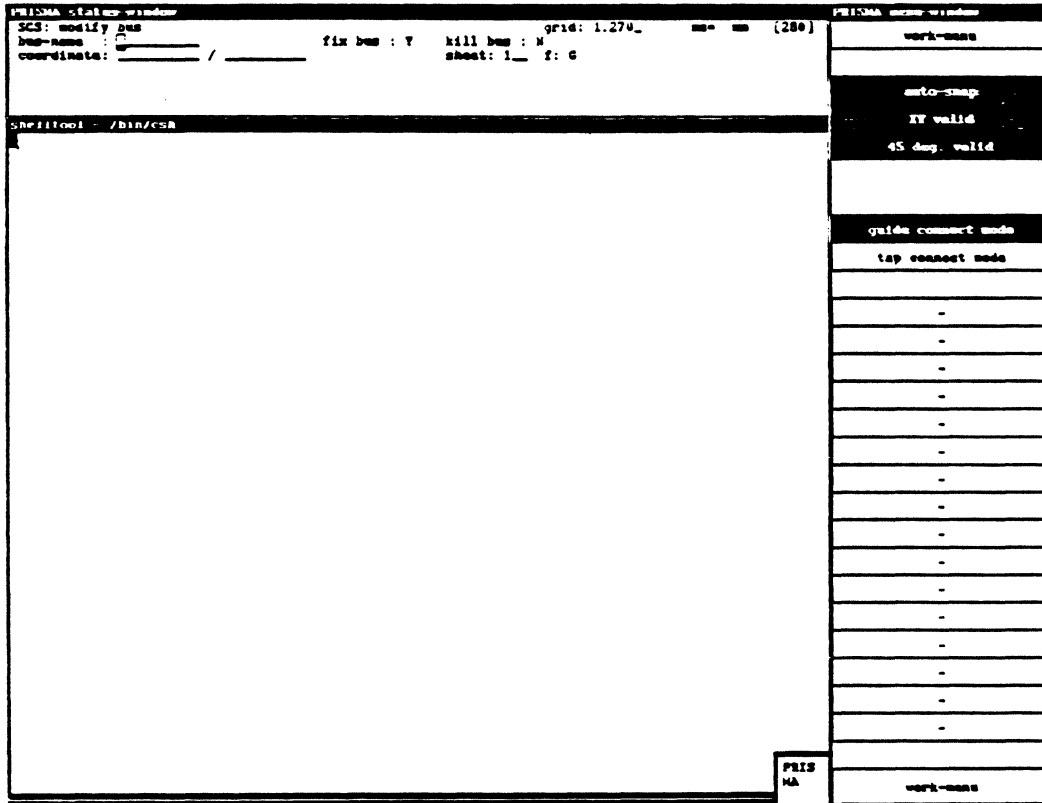
The desired bus can be selected directly in the window (by clicking the name). If no selection has been made, the text window can be left with "quit".

**STATUS WINDOW**

All functions of the menu window are also available here with a popup menu to which you get by pressing <RMB> (possibly you have to switch on the popup menu with <R11> first).

When generating a bus wire, you have to specify in the status window which signals are to be combined (e.g. "A1-A23") if you do not want to insert these signals one by one.

**SUBMENU BUS**



**AUTO-SNAP**

Special gradients can be defined for the digitizing of signals (45°, X,Y). Other gradients are not allowed then.

When the auto-snap mode is switched on, the signals are only drawn with the defined gradients.

The work will be facilitated if the user need not adhere exactly to the gradient.

When pressing <LMB>, the system draws the vertical component. If the auto-snap mode is switched off, the error message "Gradient not allowed" appears if the work was inaccurate.

### **X,Y VALID**

As gradient for the signals, only multiples of 90° are allowed, i.e. 90°, 180°, etc.

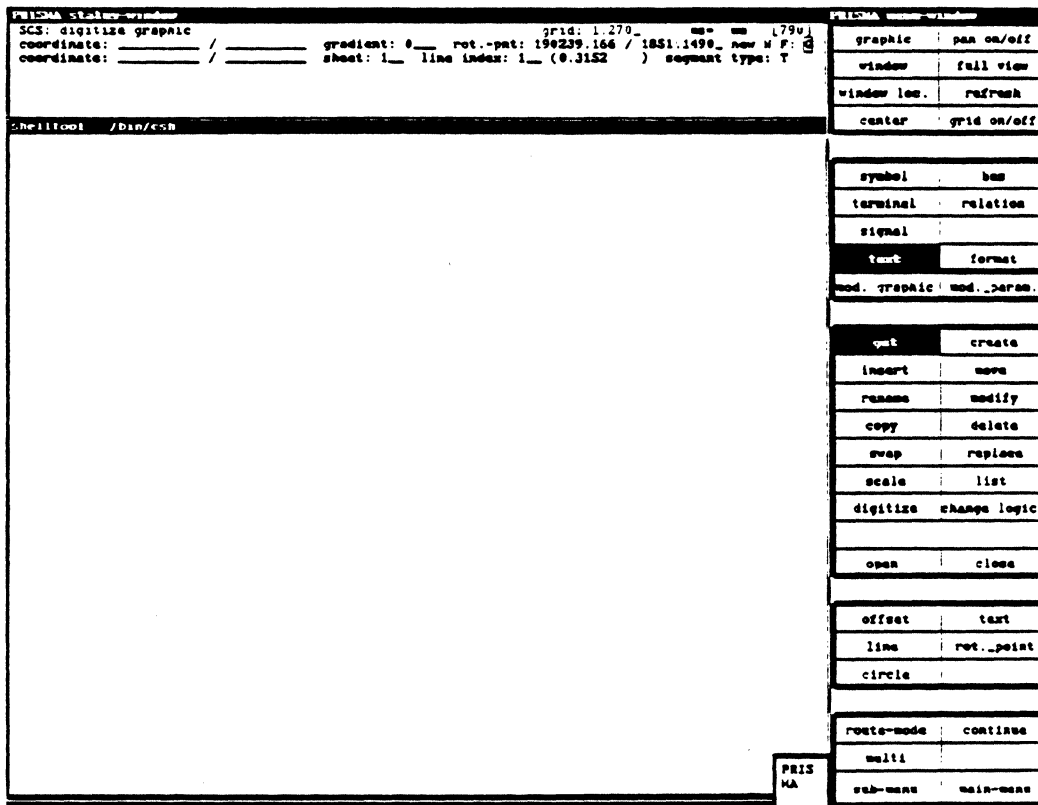
### **45 DEG. VALID**

As gradient for the signals, only multiples of 45° are allowed, i.e. 45°, 90°, 135°, 180°, etc.

### **NONORTHOGONAL**

All gradients are allowed, but the option "auto-snap" must not be activated.

## TEXT



With the function "Text", drawings can be lettered fast and easily. Existing text fields can directly be taken up with <LMB> and can then be shifted, modified, deleted or rotated with <MMB>. New text fields must either be assigned to a symbol or a terminal. Therefore, you have to take up the respective element and select the desired text field type in the status window. The following types are available:

For symbols:

- SCS name
- PCB name
- symbol
- function type
- free text

For terminals:

- terminal name
- signal name
- equivalent phys. terminal
- phys. or logical terminal
- free text

# SCHEMATICS/Modification

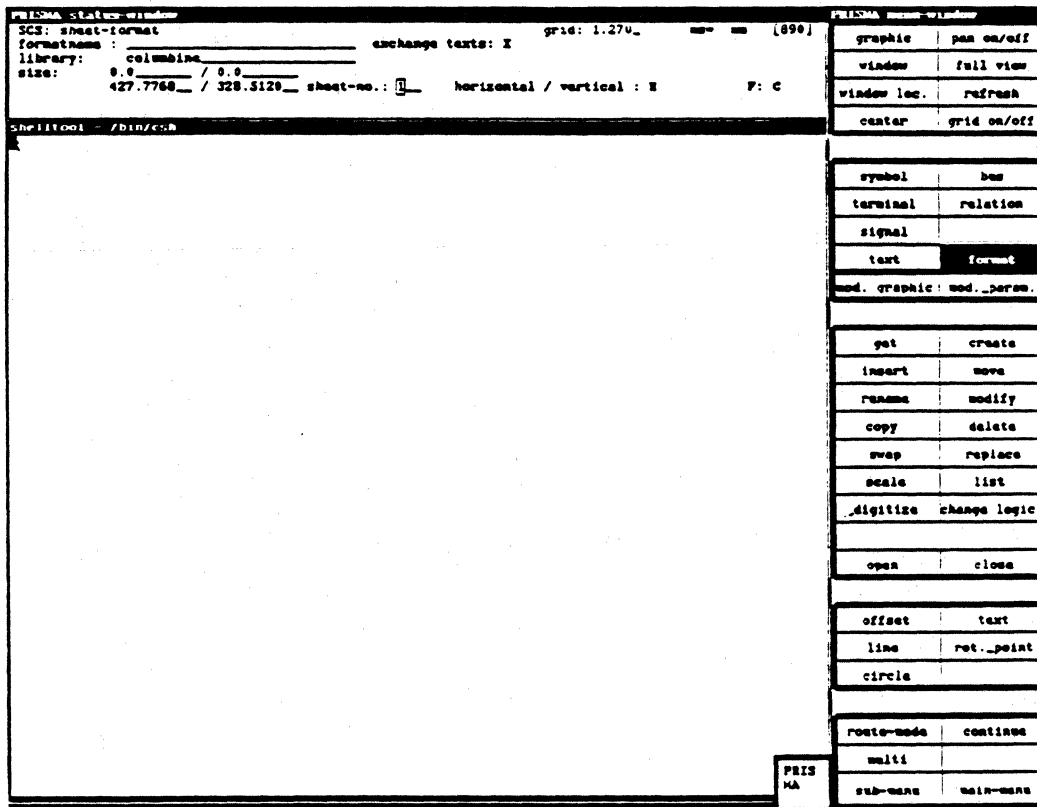
The automatically entered format of the respective texts can be defined in the menu "Technology - text presentation" and will be displayed by pressing <RETURN>.

The text blocks created in this way can be handled like all other objects: they are shifted or deleted by clicking them with <LMB>.

## FORMAT

This function supports the handling of drawing formats (see popup).

The formats should be stored with an informative file name when leaving the program.



## FORMAT POPUP

When clicking "Format", this popup menu becomes active.

It contains the following functions:

**get format:** format input from the library (in the status window)

**digitize:** The formation is made with the function "Digitize format" in connection with the digitizing elements offset, line, circle and text.

**delete format**

**list:** number of sheets



**change sizes:** The outer sizes are changed.

**new sheet**

**change sheet:** change to another existing sheet

**main menu:** Return to the main menu

### **STATUS WINDOW**

The system offers an administration of standard sheet formats. This format supply is combined by the user, as a rule. For the creation of a new format, you can define the format name, the outer size and the orientation (horizontal, vertical) in the status window.

### MODIFY GRAPHIC

This function is useful if you want to adjust earlier defined graphic displays to requirements (new component, new norm, etc.) that have changed in the meantime. The graphic modification is available for symbols, terminals and format.

This function executes the desired graphical changes at the respective objects. After the changes, the symbol frame will be newly defined.

The graphic elements

"offset"  
"text"  
"line"  
"rotation point"  
"circle"

are only available in connection with "Modify graphic" and "digitize".

They make the construction of new schematic symbols possible.

Therefore, you have to get a symbol or terminal first.

The new object can now be completed with "Modify graphic".

Procedure:

- (1) The graphic elements for symbols or terminals can be shifted or deleted.
- (2) With "digitize", new graphic elements can be created.

Explanation of the digitizing help:

- (1) With the option "offset", a reference point is determined for the subsequent drawing. If this function is skipped, the lower left corner of the symbol, i.e. the symbol origin, becomes the reference point. By pressing <LMB>, the reference point is attached to the position of the crosshair.
- (2) With "line", a line is drawn from the fixed reference point. Possible angles are defined with a respective entry in the submenu. With <LMB>, you determine the angle and length of the line. <MMB> deletes the line that was drawn last.
- (3) With "circle", you can add circles or circle parts to the activated symbol. The arcs are fixed with <MMB>.

For all circles or circle parts, the center lies at the crosshair, a point on the circle is determined with "offset". In this way, the circles are determined definitely.

With <RMB>, the arcs are displayed on the screen:

- 1 x <RMB>: creates a quadrant
- 2 x <RMB>: creates a quadrant, but mirrored at the "x axis"
- 3 x <RMB>: creates a semi-circle
- 4 x <RMB>: creates a semi-circle, but mirrored at the "x axis"
- 5 x <RMB>: creates a three quarter circle
- 6 x <RMB>: creates a three quarter circle, but mirrored at the "x axis"
- 7 x <RMB>: creates a complete circle

According to the "last in - first out" principle, the created structures may be deleted gradually by pressing <MMB> several times.

(4)The option "Text" is used for the legend of the drawing.

Further specifications to the text must be entered in the status window (ref. to the menu "Technology").

- Text type (any text, symbol, terminal, or signal name).

With the function "Text", drawings can be lettered fast and easily. As the system automatically adds symbol, terminal, or signal names (determined in the status window by specifying a text type), you have to get a symbol after selecting the "Text" function. In the status window, the text type must then be specified. These types are defined in the menu "Technology - text assignments".

You may choose between the following types:

0: any text (up to 27 characters)

- 1: symbol name
- 2: terminal name
- 3: signal name

The format of the respective texts can be defined in the menu "Technology - text presentation".

The text blocks created in this way can be handled like all other objects: they are shifted or deleted by clicking them with <LMB>.

(5)For each symbol, the rotation point lies in the origin, i.e. the lower left corner.

This standard presetting can be modified, of course. Therefore, you have to select a symbol, switch to "Modify graphic" and choose the option "Rotation point".

The rotation point of the activated symbol is marked with a big cross. With <LMB>, the rotation point is shifted to the crosshair position.

### **STATUS WINDOW**

Here, you have the possibility to change the rotation point of a symbol by entering the new coordinate.

The status windows also allow the creation of graphic or text additions with direct data input. This applies to coordinates (of symbols, terminals), center and reference points (for circles), rotation angle and the coordinates of the rotation point, as well as size, take-over of symbol or pin names, free legends and their length, right or left adjustment, and frame for a specified text.

**MODIFY PARAMETERS**

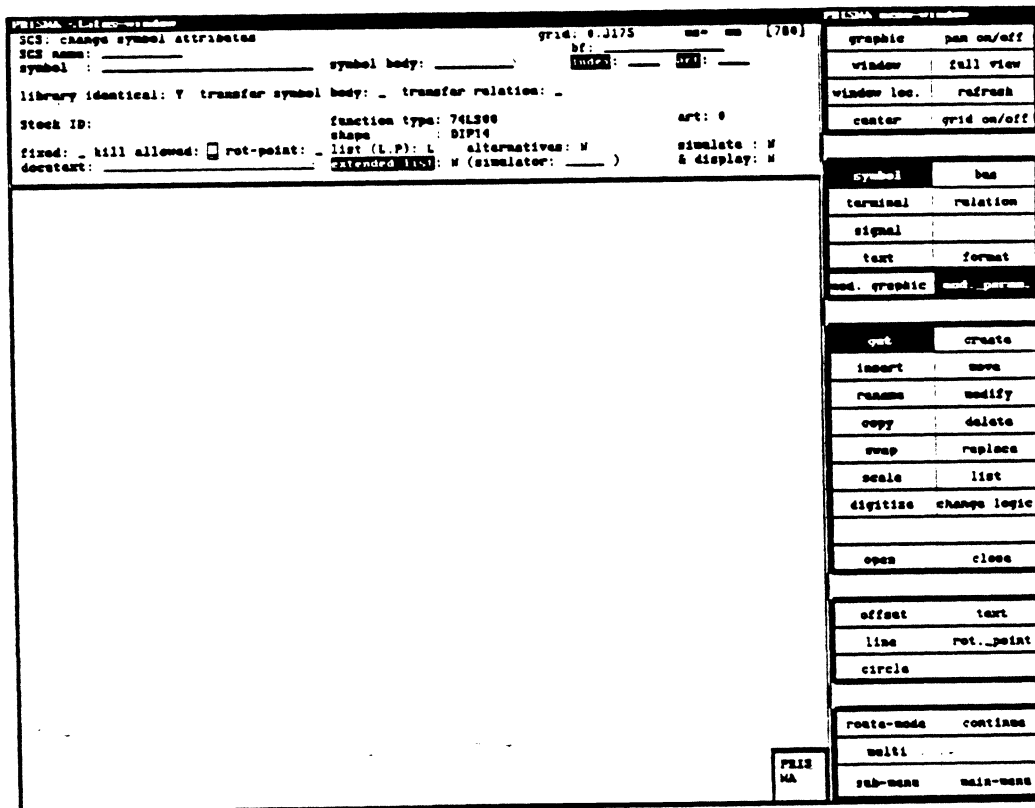
With this function, it is possible to determine attributes for the objects symbol, terminal, signal or change already defined attributes (i.e. "parameters" or global characteristics).

Parameters are handled individually for each object. Therefore, you have to get an object first before you can modify the parameters.

This function can be left by selecting another function.

For the simulation (electrical check), you have to make sure that the necessary technical specifications are entered in the respective status window. For terminals, you have to describe for example if the connection point is used for the input or output, if it is a tri-state or open collector, or a supply connection. For symbols, the parameters for a later access to the library are determined.

**STATUS WINDOW SYMBOL**



The modification of symbol attributes is supported here. These modifications must be effected in the respective status window.

Certain attributes apply to access restrictions to a symbol that was introduced in the schematics. The terms for a search in the library are listed. In the status window, you can also determine if the symbol shall be movable or fixed, if it can be deleted or shall be reserved, or if it is replaceable by an alternative symbol.

Furthermore, the release of a symbol for the transfer of logical component functions to the library "Columbine" is executed here.

In "Docu text", own explanations can be entered (explanations for the user).

Explanation of the terms:

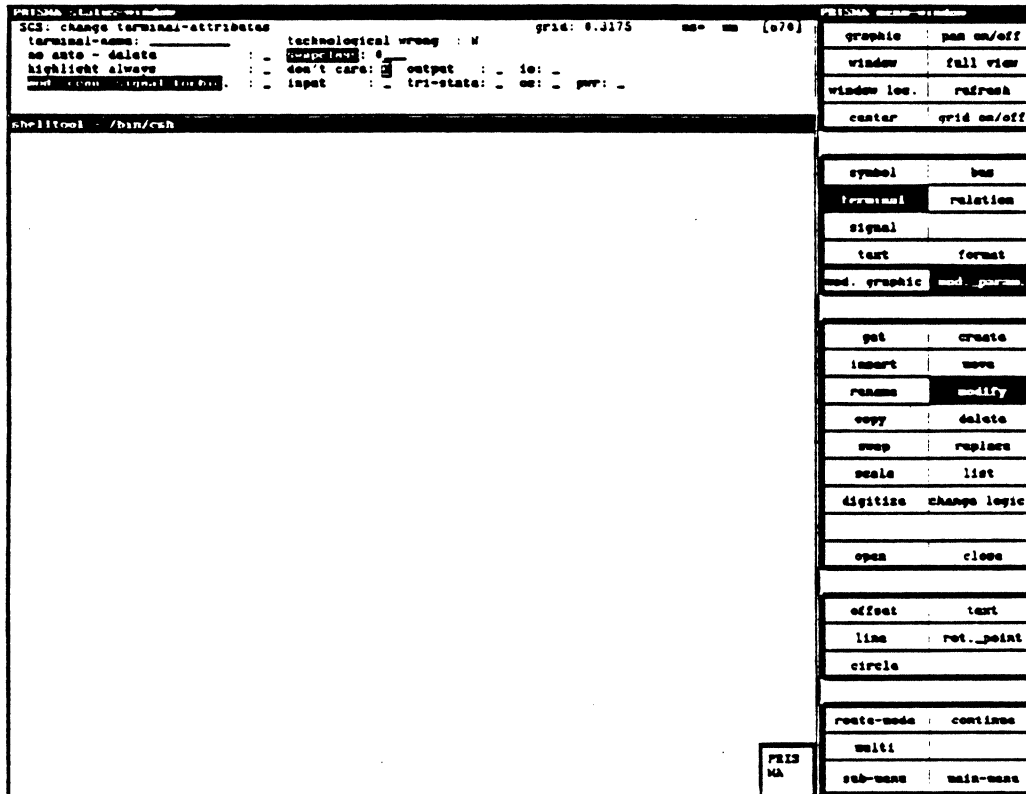
**Symbol:** The logical side of a component. It represents an electrical function element.

**Symbol body:** Consists of the symbol terminals and the symbol extension.

**Function type:** Specifies a function specific standard component (e.g. the 4 NAND component 74LS00) that may exist in one or several variants.

**Shape:** The physical expression of a component that consists of shape body and shape graphics.

## STATUS WINDOW TERMINALS



Terminal attributes that can be modified apply to the following areas:

(1) The logical function of the terminal:

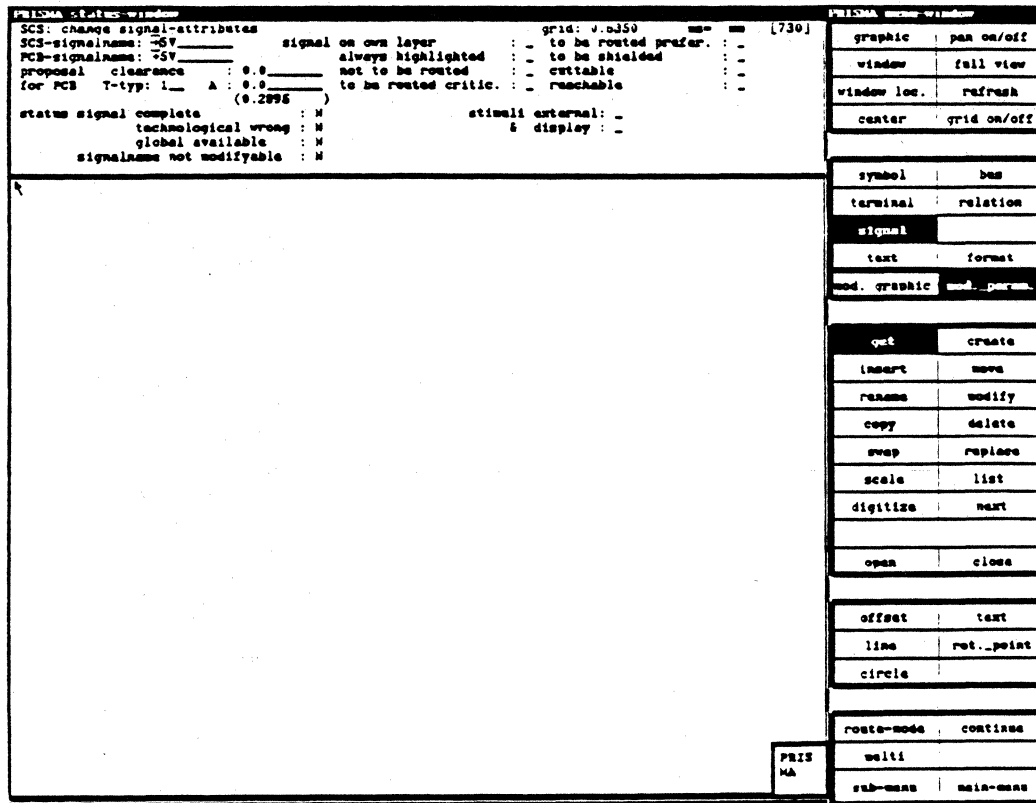
You have to specify for each terminal if it is used as input or output connection (unidirectional), as "tri-state" or supply connection, or as "Don't care" connection. This specification is important for a control simulation and must therefore be definite.

(2) Access protection:

Terminals shall be defined and must still exist when the supplying signal is deleted. If this is the case, you determine in the status window if the terminal can be deleted automatically.

(3) You can also specify if the terminal shall be highlighted permanently.

## STATUS WINDOW SIGNAL



In the menu "Modify parameters", different additional attributes for signals can be defined that have an effect on the PCB part.

Here is agreed:

- Changing the signal name on the PCB and on the SCS side
- Signal on own layer (e.g. for supply signal)
- Preferred routing of the signal
- Highlight the signal permanently during the processing
- Shield the signal from other signals
- Signal shall not be routed
- Signal must be cuttable
- Track type of the signal
- Cross-section
- Marking of signals that are critical to route
- Signal must be reachable

These additional attributes are displayed in the status window for each selected signal.



**ROUTE MODE**

The route mode is helpful for the systematic digitizing of signal lines in the SCS model.

**ROUTE MODE OFF:**

Signal lines from terminal to terminal are marked as airlines. The shortest line between two terminals connected with "Digitize signal" is created. For better identification, each signal line gets a definite number that corresponds to the number of the respective signal. When shifting components, the airlines are shifted as well.

You may use the switched off route mode especially in the beginning to realize the logical connections in the schematics on the screen.

**ROUTE MODE ON:**

When the route mode is switched on, the system tries to route the logical connections (the airlines) on the SCS side.

A complete routing is sometimes not possible so that single airlines cannot be realized in the structured display.

The resolution of the route mode can be varied in the menu "Modes".

**CONTINUE**

With this function, several operations may be combined.

Possible combinations are:

- "get" and "delete"
- "get" and "shift".

**MULTI**

This function facilitates the modification of object groups. After selecting "Multi", a window can be opened with <LMB> (from the lower left to the upper right corner) in which the desired objects are enclosed. The total block can now be modified.

Possible operations are:

- copy
- delete
- shift.

## 2. GRAPHIC SETUP

The screenshot displays the PRISMA software interface. At the top, a window titled 'PRISMA status window' shows a menu of functions: (B) modification (1), (A) misc., (C) change file, (D) update file, (E) graphic, (M) modes, (U) update file, (X) exit, (F) final check, (T) technology, and (S) status. The 'modification' option is selected. Below this, a 'Shelltool - /bin/csh' window is open. On the right side, a vertical menu titled 'PRISMA menu window' lists the following options: modification, graphic setup, final check, layout, misc., modes, technology, change file, save file, exit, and status. A mouse cursor is pointing at the 'graphic setup' option. In the bottom right corner of the main window, there is a small box containing the text 'PRIS MA'.

**WORK MENU**

You return to the superior menu.

**MAIN MENU**

You return to the main menu.

**WINDOW**

Here, you can define a new window.

**Procedure:**

Move the cursor to the place where you want to define the lower left corner of the window. With the left mouse button (called <LMB> in this description), you fix this corner. You can then open the window with the mouse until it has the desired size. The final size of the window is determined by pressing <LMB>. With <RMB>, this procedure can be interrupted. If you press <MMB> in the middle of the screen, the current window will be zoomed. At the edges of the graphic screen, <MMB> may be used as shifting function, i.e. the current window moves in the direction of the crosshair.

**FULL VIEW**

Deletes the current window and shows the total work sheet again.

**WINDOW LOC.**

For very large windows, there is a useful orientation help:

By selecting "Window loc.", the current window is represented by a symbol in relation to the total size of the work sheet. Here, you may choose a new window directly: move the crosshair to the place where the lower left edge of the window is to be defined. Fix this edge with <LMB>. You can then open the window with the mouse until it has the desired size. The final size of the window is determined by pressing <LMB>.

**REFRESH**

With this function, the screen is cleared and you get a new artwork. It contains the current data.

**CENTER**

Sometimes, it is necessary to keep the current scale but to represent a slightly different window.

When selecting this function, the crosshair position becomes the new center of the picture. Note that it is not possible to move the window over the left edge of the sheet (the lower left corner is the coordinate origin). If you enter a name in the status window in "SCS name", the system tries to move the respective element to the center.

### **GRID ON/OFF**

With this function, the grid may be switched on/off. The kind of presentation of the grid can be defined in "Color" in the status window.

If the grid is not needed for the placement, it should be switched off since it has to be drawn anew for each picture; this delays the artwork.

### **RESTORE**

Works like "Refresh" - but the contents of the picture is not calculated anew, it will be newly written by means of the graphic memory (this is faster).

### **ZOOM**

As known from the photo optics, you can determine a part of the picture here that shall be zoomed. With the crosshair, you define the section to be enlarged (with <LMB>) or reduced (with <MMB>) by a constant factor. With <RMB>, you can interrupt the procedure.

### **GRIDS**

According to the user's requirements, different grids are available. The resolution can be changed for the display, the digitizing or the routing. The user can choose between 9 standard settings and one optional setting. The standard settings must have been defined manually in the status window before. The crosshair is moved with the arrow keys or the tab key. Please do not forget to store your entries with <RETURN>!

In the menu, you can select the desired grid with the mouse or with a respective entry in the status window.

Ex.: Setting the "Disp-grid":

- (1) Click "Disp-grid" with <LMB>.
- (2) Click the desired standard setting, e.g. "4" in the menu window with <LMB>.

In the status window, the current values appear for the different grids (digitizing, auxiliary, and routing grid). The measurements (either inches or mm) to be assigned to the numbers can be defined in the menu "Modes".

The current measurement system is shown in the field "ms". (When changing the units, the system automatically changes the entry).

The presentation of the grid - lines or points - can only be defined in the function "Color" at present.

### **DIGI-GRID**

This is the resolution used for the digitizing. The crosshair is moved according to this resolution.

**DISP-GRID**

Specifies the resolution of the auxiliary grid that can be activated with "Grid on/off".

This value cannot be smaller than the value of the "Digi-grid" (must be a multiple of it).

**PLACE-GRID**

This is only available in the model "Layout".

**ROUTE-GRID**

This grid may be chosen freely. In connection with the option "Route mode" (ref. to the menu "Modification"), the resolution with which signals are routed in the SCS-model is defined with the route-grid.

**NUMBER START**

This function is not yet available.

**NUMBER STOP**

This function is not yet available.

**COLOR ON**

This function is activated from the status window. According to the kind of display, colors that have been faded out can be faded in again one by one. Therefore, you click the respective code number in the menu window (0 = all colors).

**COLOR OFF**

This function is also activated in the status window. For better display, it may be useful to fade out some colors.

**COLOR CLEAR**

By clicking the respective code number, all objects of this color will be deleted from the screen. When selecting "Refresh", these objects appear again.

**SET COLOR**

Seven colors are available for the creation of the schematics. With this function, the colors are assigned to the objects (symbols, terminals, crosshair, etc). The assignment of color codes (number between 1 and 7) to the objects is made with a respective entry in the status window.

There are two different kinds of display for the auxiliary grid (display grid): as a net with solid lines (grid type L) or as marking of the nodes (grid type P).

**STD. COLOR**

With this function, the objects will be displayed in the standard color again (symbols red, text and grid white, terminals green, auxiliary lines violet, crosshair blue, frame light-blue).

### SCALE

For a placement true to the scale, a yard-stick is sometimes necessary. Therefore, you can use this function. You determine the starting point of the scale with the crosshair.

The origin of the scale lies exactly in the crosshair. The scale becomes visible with <LMB>. The display unit is shown in the status window.

### STATUS WINDOW:

The length and the axial division of the scale can be defined here.

### OPTIC

This function allows the use of graphic figures as optical support:

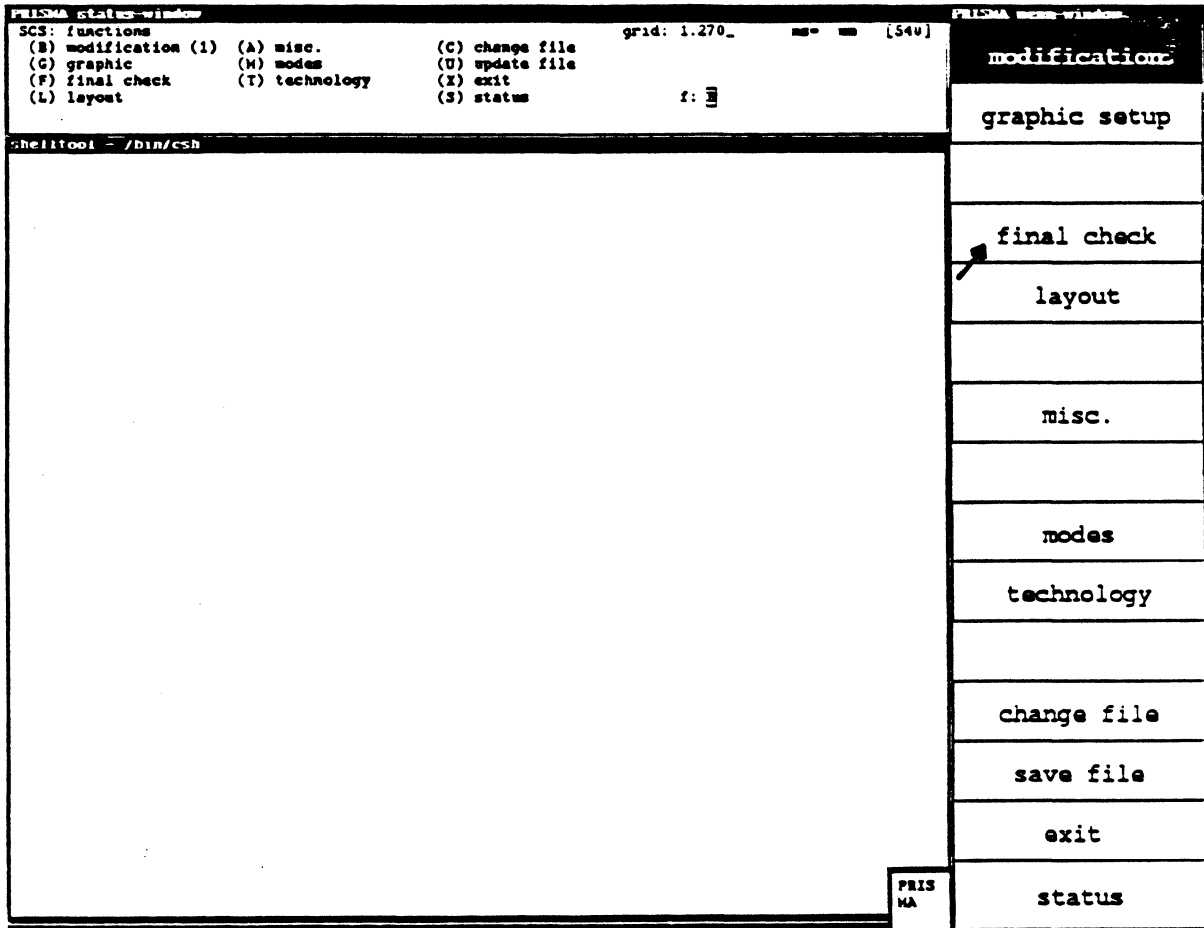
point (P), cross (X), square (Q), circle (K), rhomb (R).

The assignment of an object to a graphic figure is made with a respective entry in the status window. By pressing <RETURN>, the selected figures are displayed.

In "Text-selection", you can choose the text types to be shown on the screen. After selecting "Refresh", the changes are executed.

For comfort reasons, the "Work menu" and the "Main menu" are offered again.

## 3. FINAL CHECK



The following options are offered:

- P packaging
- A man. packaging
- O opt. packaging
- M mapping
- B man. mapping

The selection is made by entering the respective letter in the status window.

The function "Final check" has not yet been implemented for the schematics model.

4. Miscellaneous

PRISMA status window		PRISMA user window	
SCS: functions		grid: 1.270_ [540]	
(B) modification (1)	(A) misc.	(C) change file	
(G) graphic	(H) modes	(U) update file	
(F) final check	(T) technology	(X) exit	
(L) layout		(S) status	f: [ ]
shelltool - /bin/csh		modification	
		graphic setup	
		final check	
		layout	
		misc.	
		modes	
		technology	
		change file	
		save file	
		exit	
		status	
		PRIS MA	



**REPORT**

A submenu shows the reports of the adb-file that can be created at the moment:

- components
- technology
- signals: netlist
- connectors
- PCB - SCS: list of the related components
- statistics: survey of the used elements
- non-related pins
- non-connected pins

**LOG-FILE**

not implemented

**C-shell CALL**

After selecting this function, the menu "shelltool-/bin/csh" appears. With the crosshair, you can delete or modify data in this window. By clicking the upper frame with <LMB>, the window dis-appears.

**SUBWINDOW**

The realtime integration, that PRISMA supports, implies a close relation between the models SCS and PCB.

Especially the data transfer between the two models is accomplished.

The "subwindow" allows an overview of the current layout from the schematics side.

The subwindow will be opened at the place whose coordinates have been entered in the status window (as for all other windows on the screen, you can also modify the starting position of this window with <MMB>: place the mouse on the "submonitor" frame, fix it with <MMB> and "draw" the window to the desired position).

The "subwindow" is only calculated and written once: this means that at present, you cannot see the reactions that changes in the schematics will have in the layout part.

But you may actualize the window by calling "subwindow" again. In order to close the window, click "subwindow" and press <Ctrl C> on the keyboard.

**SHEET FORMAT**

The creation of another sheet within the current "job" is made with the popup menu as a rule.

Popups are called with <RMB>.

Possibly you have to switch on the popup function with <R11> first. The popup menu remains visible as long as you press <RMB>.

By moving the mouse, you can select a function that will be activated when releasing <RMB>.

**Get:** Possibility to call standardized menus from the library. Enter the name of the menu in the status window and press <RETURN>.

**Digitize:** switches directly to the digitizing mode; a modification of the format is possible.

**List:** shows a list of all sheets known to the system.

**Change sizes:** moves the cursor to the respective fields in the status window. Enter the new sizes and press <RETURN>.

**New sheet:** creation of a new sheet

**Change sheet:** Enter the sheet number of the searched sheet in the status window.

**PLOT**

**Status window:**

ICS: hardcopy		grid: 0.3175		(700)	
name of plotfile <u>haha.pl</u>	scale	1.000000			
name of plotter	window	0.0	0.0		
		427.7768	328.5120		
-----					
	grp	txt	trm	air	net
pen	1	1	1	1	1
[Large empty plot area]					
					report
					log-file
					c-shell
					subwindow
					new sheet
					plot
					relation
					main-menu

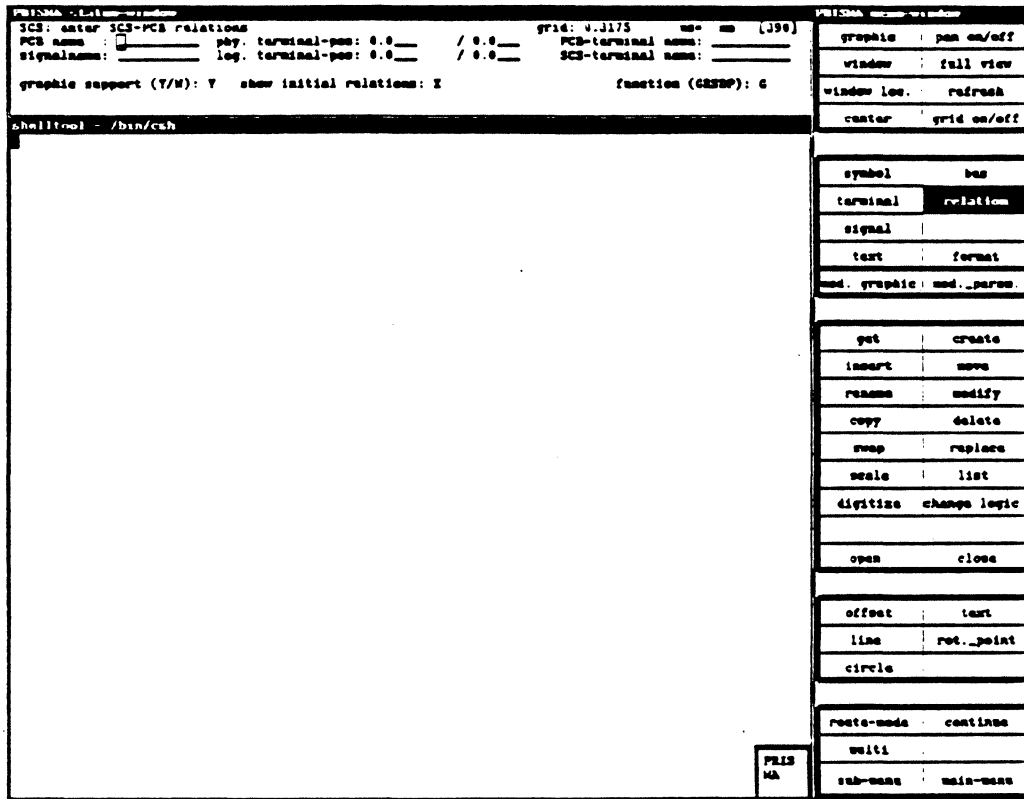
**Factor:** In this field, you can define a factor by which the hardcopy shall be displayed enlarged or minimized.

**Window:** In these fields, the section to be plotted can be defined. The lower left and the upper right corner of the window can also be defined with <MMB>. By pressing <RETURN>, the plot file is created. The window may be deleted with <RMB>.

**Grp, Txt, Trm, Air, Net:** In these fields, plotter pens can be assigned to the different objects.

## RELATION

Status window:



All SCS schematic data and PCB design data are stored in a common data base because of the realtime integration principle in PRISMA. They are combined hierarchically. If a component has been created on the layout side or a symbol on the SCS side, a relation has to be executed in the model "SCS".

### Procedure:

- 1) When selecting "relation" here, the function "relation" of the menu "Modification" is activated automatically.
- 2) Enter the name of the component to be related in "PCB name" and press <RETURN>.
- 3) By moving the mouse, the component can be shifted. Move it close to the symbol. With <LMB>, it can be fixed at the desired place.
- 4) Now select a component terminal with <LMB>. Get the respective symbol terminal with <LMB> and continue the procedure until all terminals have been assigned. For related component-symbol-groups, the respective names are displayed in the

status window from now on.

Non-related pins are displayed after each new placement.

The PCB-terminal-name can also be entered with the keyboard (get SCS terminal first).

Attention:

At present, the relation can only be executed if the symbol and the component are placed with rotation 0.

5. MODES

<pre> PRISMA: status-window SCS: global parameter system - units (I) inch,(M) mm: <input checked="" type="checkbox"/> real-time-integration: <input checked="" type="checkbox"/> grid: 0.3175 mm [420] symbol-frames: _ pop-ups: _ relation-help-graph: <input checked="" type="checkbox"/> ISO-text: <input checked="" type="checkbox"/> crosshair: <input checked="" type="checkbox"/> airlines: <input checked="" type="checkbox"/> rotation: <input checked="" type="checkbox"/> automatic-routing: _ textfield by number: <input checked="" type="checkbox"/> automatic-reroute: _ textfield-size automatic: <input checked="" type="checkbox"/> search range: 0.6350 terminal-help: <input checked="" type="checkbox"/> size: 1.9050  autoclearance to symbols: 2.540 to signal: 0.9525 auto-routing grid: 2.540 display grid: 5.080 interactive grid: 0.3175 F (V): 1 A: 7.2330  auto - enter terminal into signal: <input checked="" type="checkbox"/> auto - reroute if signal - collision: _ auto - enter signal into bus: <input checked="" type="checkbox"/> auto - delete in layout: <input checked="" type="checkbox"/>  auto - create textfield for SCS name: <input checked="" type="checkbox"/> textsize V: min. 2.540 max. 10.160 textfield position ( 0-center 1-up 2-down 3-right 4-left ): 0 minimal length: 0  rate of valid climb orthog.: <input checked="" type="checkbox"/> 45 degrees: <input checked="" type="checkbox"/> all angles: _ br: conn 45 deg: _ 2.540  standard-panic-mode (P,W): W auto-panning-size: _ or fullsize: <input checked="" type="checkbox"/> panning-area (pixels) in X: 1652 (min=1152 max=3456) in Y: 1300 (min=900 max=2700) preference-value: 900 pixels be adequate to 304.80 system - units         </pre>	<pre> PRISMA: menu-window modification graphic setup final check layout TOOLS misc. <b>modes</b> technology change file save file exit status         </pre>
<div style="border: 1px solid black; width: 100%; height: 100%;"></div>	<div style="border: 1px solid black; width: 100%; height: 100%;"></div>

"Modes" are determined in the status window. They are presettings for routine working processes in PRISMA.

The presettings are:

**System-units:** Specification of the measurement unit to be used (inch or mm).

**Real-time-integration:** The reaction that changes in one model will have on another one can be switched on or off.

**ISO-Text:** Specification if the legend is to be effected according to the ISO norm.

**Symbol-frames:** Specification if a frame shall be visible around the symbol. Makes the placement easier because collision possibilities become visible.

**Pop-ups:** Specification if the popup menu is available with <RMB> or not. It can also be switched on with <R11>.

**Relation-help-graph:** Presetting if the relation between SCS and PCB side shall be displayed graphically when selecting "Modification/relation".

**Lowercase char.:** Specification if legends may also contain lowercase (or only uppercase).

**Crosshair:** Crosshair visible or not

**Automatic-routing:** automatic routing

**Textfield by number:**

**Airlines:** Specification if "airlines" are shown or not.

**Rotation:** Specification if the rotation point of an object is to be shown.

**Automatic-reroute:** Specification if all connections are rerouted after the shifting of a symbol.

**Textfield-size automatic.:** Presetting if the text field size is adjusted automatically according to the quantity of the text.

**Search-range:** grid size in which the objects shall be realized.

**Terminal-help:** An additional display help can be switched on (optical help for terminals).

**Size:** Size of the terminal help graphic.

**Autoclearance to symbols:** If the clearance is violated, an error message appears.

**To signals:** If the clearance is violated, an error message appears.

**Auto-routing grid:** grid in which the route mode works

**Display grid:** Here, you can determine the size of the display grid.

**Interactive grid:** Here, you can determine the size of the interactive grid.

**T-Type:** standard presetting for track types

**A:** size of the track cross-section

**Auto-enter terminal into signal:** logical assignment of a terminal to the neighboring signal

**Auto-rename if signal-collision:** When connecting two different signals to a terminal, the message "Signal collision" appears. After pressing <LMB>, the signals are connected nevertheless and get the same name.

**Auto-enter signal into bus:** Specification if signals that shall be connected to a bus wire are connected automatically after the names have been entered or if the connection shall be digitized manually.

**Auto-delete in layout:** Specification if a deletion in the schematics (SCS) shall automatically effect a deletion in the layout (PCB).

**Auto-create textfield for SCS name:** Specification if the symbol legend shall be effected automatically.

**Text size min/max:** Specification of the legend

**Textfield position:** Position of the legend

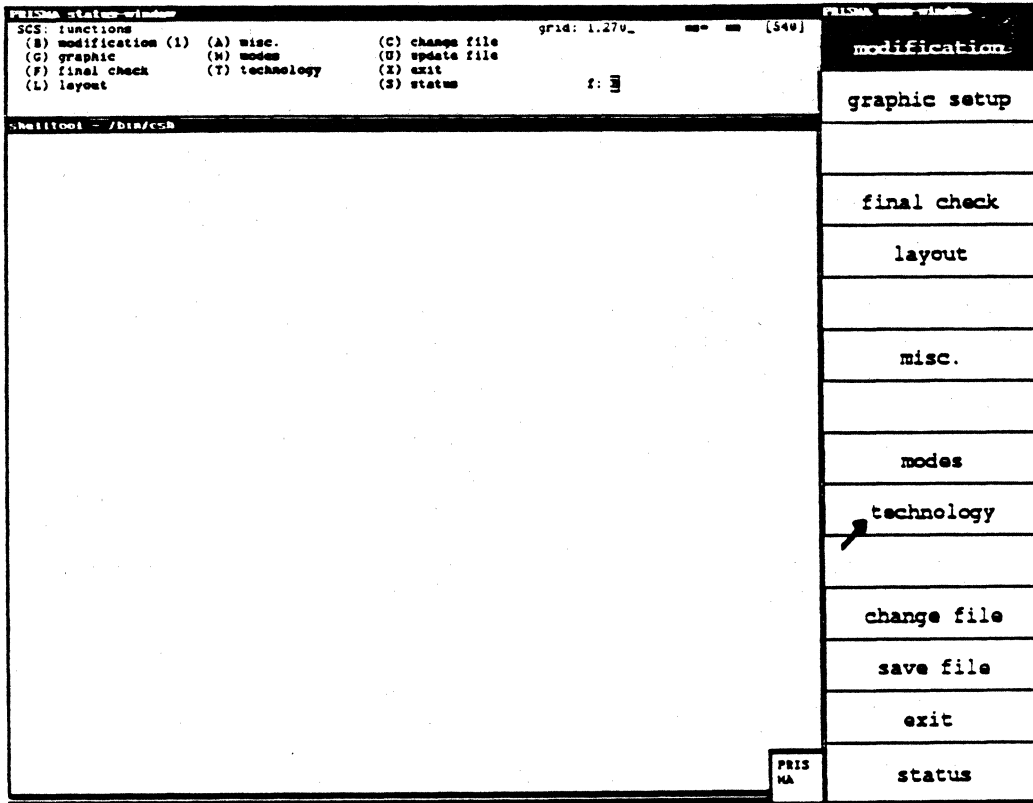
**Minimal length:** minimum length of the text field

**Rate of valid climb:** Specifies the angles that may be drawn during the digitizing (only right angles, only multiples of 45°, all angles).

**Panning-area:** Specification of the PAN area in pixels. Therefore, you have to set standard graphic mode P.

**Standard-graphic-mode:** W = window or P = PAN

## 6. TECHNOLOGY



The menu "Technology" serves for defining the (graphic) representation and the assignment of a representation to an object.

The submenus have the following applications:

- (1) **Line technology:** Definition of line types to which a certain line width and diameter is assigned.
- (2) **Line assignments:** Assignment of a certain line type to the objects signal, bus, element and terminal.
- (3) **Terminal technology:** Definition of certain terminal types
- (4) **Terminal assignments:** Defines which terminal variant is automatically chosen for which orientation (top, bottom, right, left).
- (5) **Text technology:** Definition of text types to which a certain width and height is assigned.
- (6) **Text assignments:** Assignment of these text types to the objects free text, symbol name, terminal name, signal name, pin name, component name and log./physical terminals.
- (7) **Selection mask:** Controls the library access over the dynamic components.



**LINE TECHNOLOGY****Status window**

In this window, you can define line types. A definition consists of an index (three-digit number) to which a line width and a diameter is assigned. If line width = diameter, the end of the line is a rectangle (ref. also to the router manual: track width).

You have to confirm the entered values with <RETURN>.

A starting value may be entered in the field "First number". After the definition has been finished, the system creates the next line index.

**LINE ASSIGNMENTS****Status window**

The line types that have been defined in the submenu "Line technology" can be assigned to the respective objects here.

The assignment is made by entering the respective index of the line type for one of the objects signal, bus, element or terminal.

The entry must be finished with <RETURN>.

**TERMINAL TECHNOLOGY**

Defines the display of terminals

**Terminal assignments  
Status window**

The terminal index determines which terminal graphic is chosen automatically to mark the terminal type.

**Text technology  
Status window**

In this submenu, text types that are characterized by their width and height are defined.

The identification is made according to the text index (a three-digit number).

**Text assignments  
Status window**

In this menu, objects are assigned to the text types that have been defined in the submenu "Text technology". The following objects are available:

free text, symbol name, terminal name, signal name, ISO name, component name and log./physical terminal.

The assignment effects that texts are automatically created in defined formats during the lettering of the plots.

## SELECTION MASK

selection type	range	val.	p.	
		-1		<= value: <, >, * or character defined in "range"
		-1		
		-1		
		-1		priority-range from 0 to 5
		-1		no priority ==> -1
LS		Y	0_	<= value: Y, N or *
S			-1	
HC			-1	
HCT			-1	
F			-1	valid for (S.P):
ADVANCED			-1	enter values and hit <RETURN>-key

line technology
line assignments
terminal technology
terminal assignments
text technology
text assignments
selection_mask
main-menu

By means of the selection mask, the library access is controlled. The selection mask is user dependent (dynamic component selection).

Selection types may be:

- Component elements (TTL, HCMOS, ALS, LS, etc.)

- Shape types (standard, SMD, etc.)

For the library access (ref. to "Insert symbol"), the selection is limited to the specified types in this way. Different priorities must be defined.

7. CHANGE FILE

The screenshot shows a terminal window with a PRISMA status window. The top part of the window displays a menu of functions:

```

PRISMA status window
SCS: functions      grid: 1.270_    [540]
(B) modification (I) (A) misc.          (C) change file
(G) graphic         (M) modes          (U) update file
(F) final check    (T) technology    (X) exit
(L) layout         (S) status
    
```

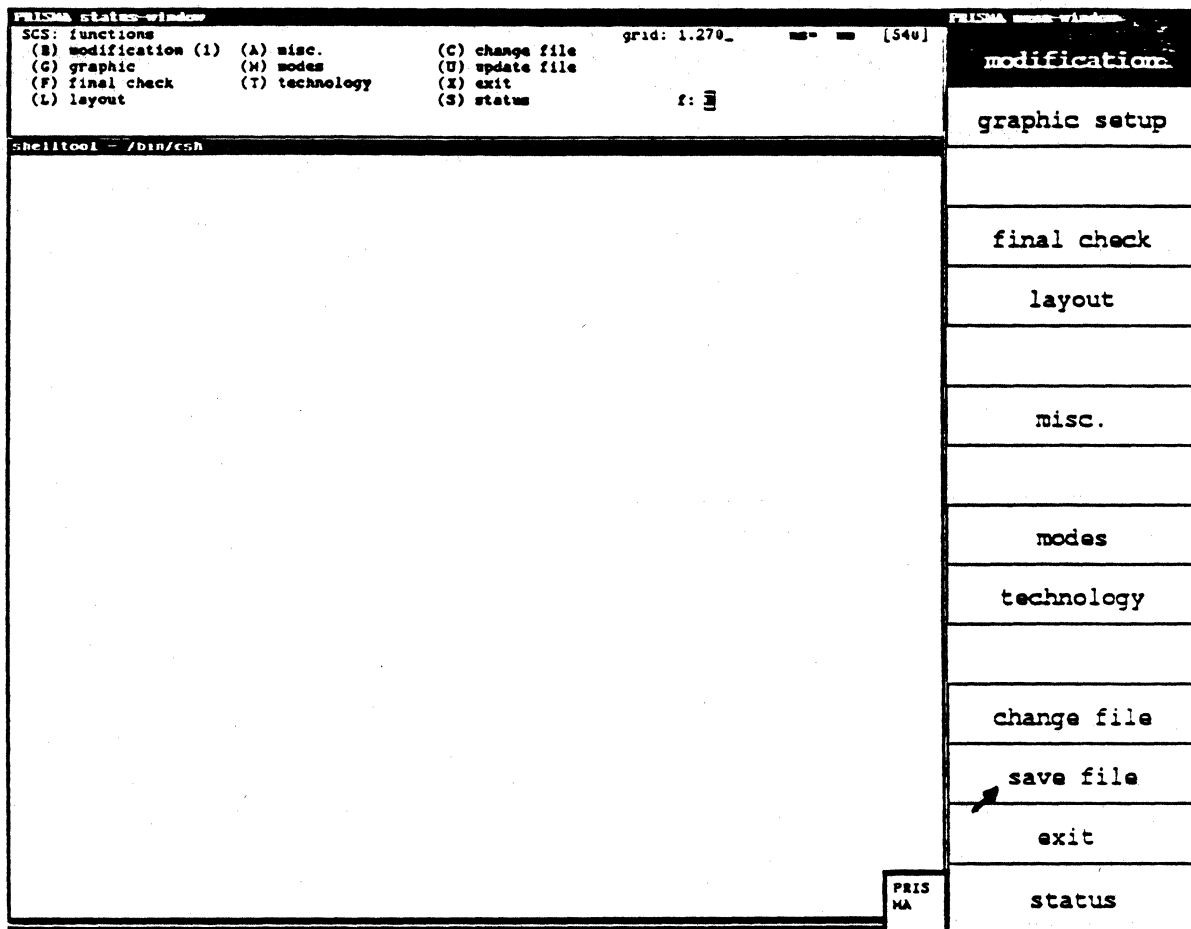
Below this menu is a large empty area labeled "shelltool - /bin/csh". To the right of this area is a vertical list of modification options:

- modifications
- graphic setup
- final check
- layout
- misc.
- modes
- technology
- change file (indicated by a mouse cursor arrow)
- save file
- exit
- status

In the bottom right corner of the terminal window, there is a small box containing the text "PRIS MA".

This function has not yet been implemented.

**8. SAVE FILE**

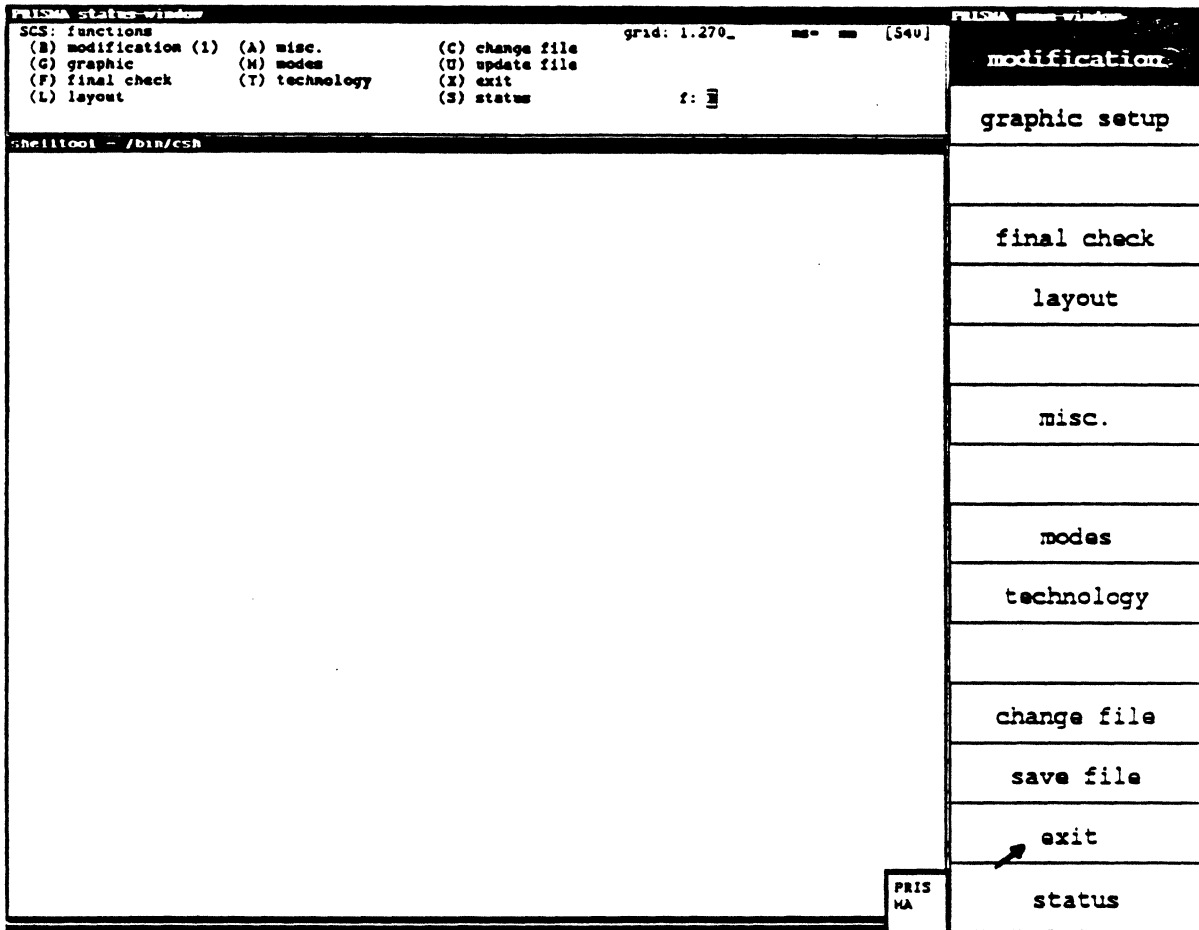


This menu is used for the intermediate storing of the file. For longer files, it is better to create a backup from time to time so that you do not loose all your data in case of a system breakdown.

**Compress data:**

adb-data can be compressed with the UNIX function "compress". Result: the file contains less blocks afterwards (space-saving)

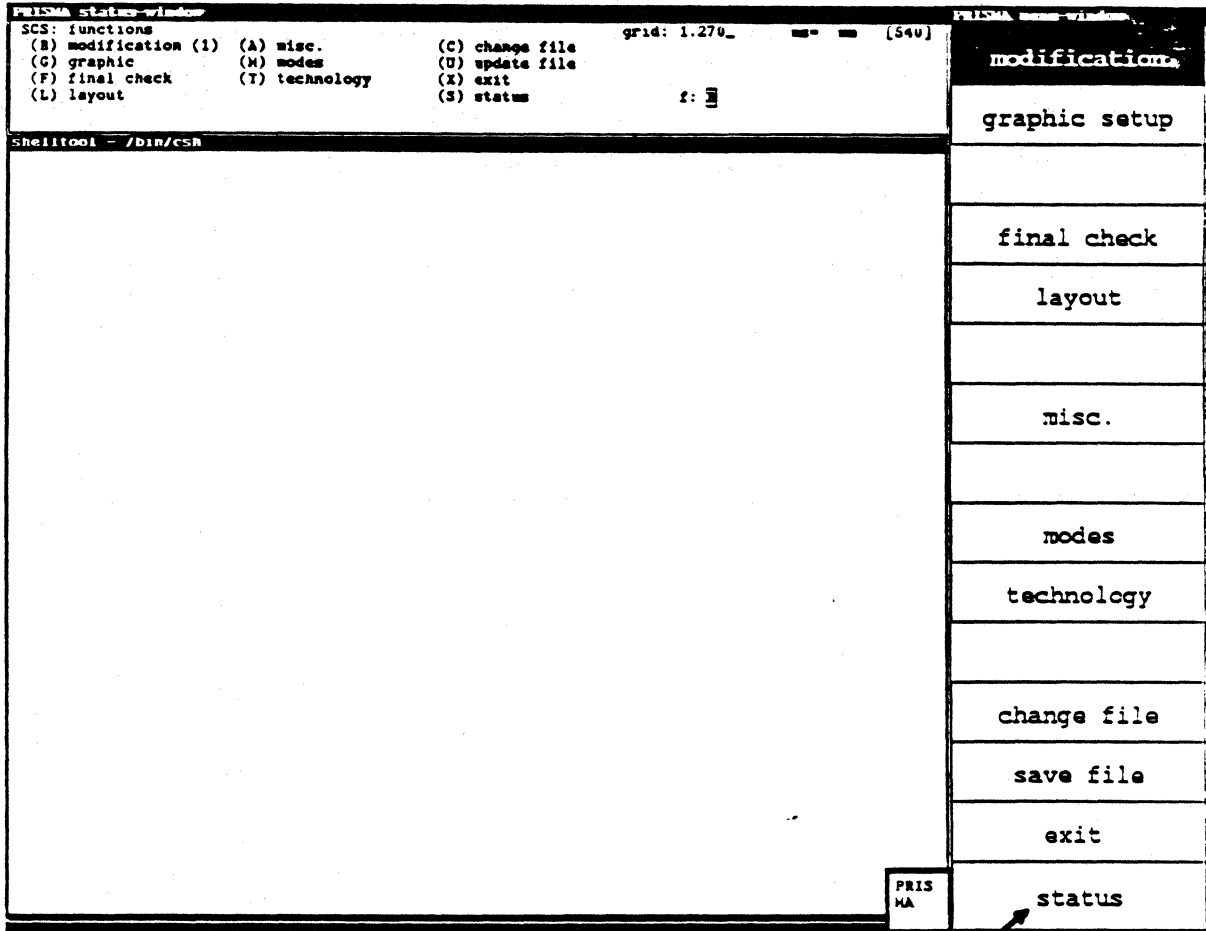
9. EXIT



With this function, the program is finished and you get back to the UNIX operating system.

You can store the data by pressing <RETURN>. If the data shall not be stored, press <ESC><ESC> and type "Y" twice.

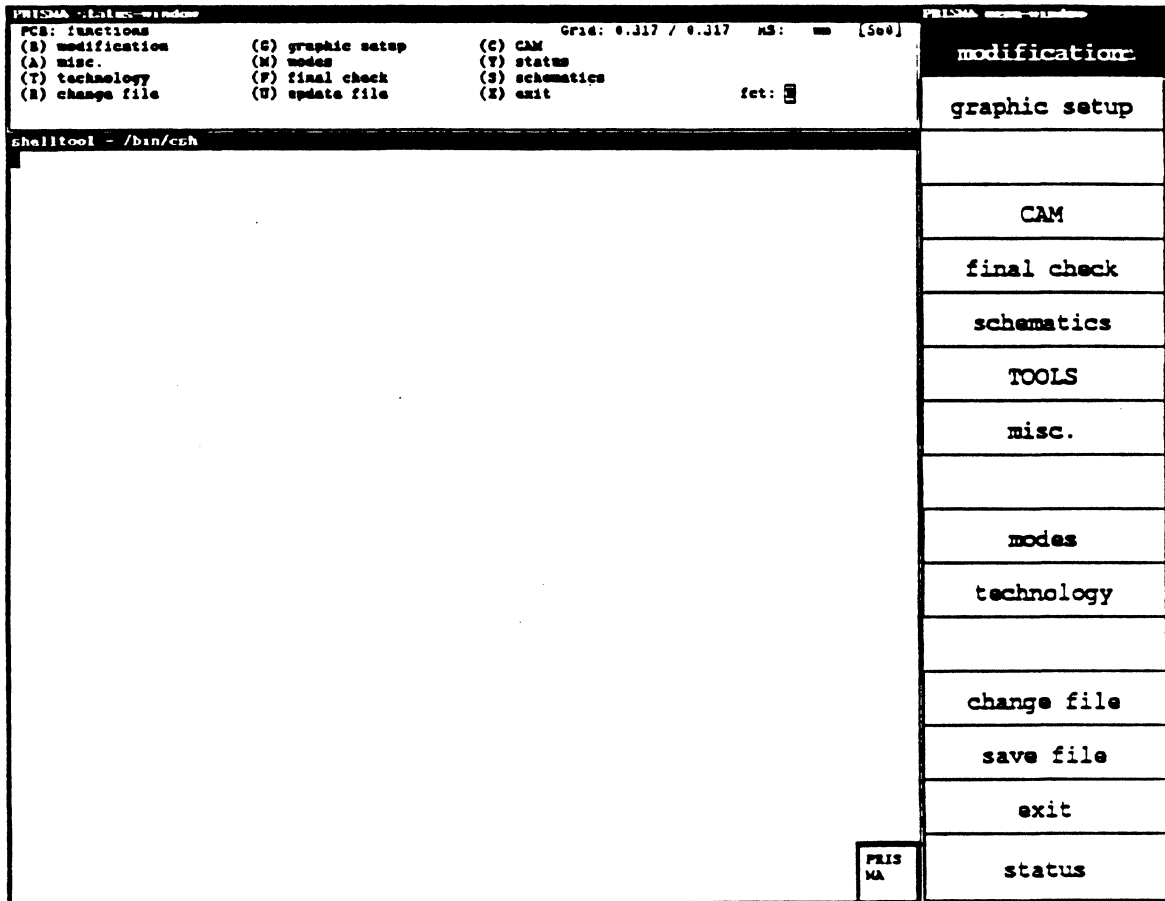
10. STATUS



This function shows the current version number of the software in the status window.

The version number of the program and the menus are displayed. You can leave the menu "Status" by calling another menu.

## 11. Modification



The menu "Modification" which supports the graphics is the largest menu in the system. Standard values like track width, grid size and colors, can be determined in the menus "Graphic setup", "Technology" and "Modes".

The functions do not have a hierarchy so that the user can select all executable functions at any time. These functions can be combined in five units:

- (1) Help for the graphic display (graphic parameters, grids, windows, etc).
- (2) Objects (components, connection points, signals)
- (3) Actions that can be executed with these objects (edit)
- (4) Digitizing help (for making connections)
- (5) Functions that allow the menu selection or the connection of the subordinate items (e.g. a repeat function)

## Graphic selection

After calling the function "GRAPHIC", the menu "Graphic selection" appears in the status window in which the board elements to be displayed (components, terminals, segments, vias, etc.) and the layers are determined.

PRISMA PCB graphic setup				Grid: 0.117 / 0.117		AS: ma [570]		
set: v	components: <input checked="" type="checkbox"/>	(PV)	nets: -	pads: -	X=all	Y=free	work-mask	main-mask
	terminals: <input type="checkbox"/>	(AGQM)	grid: -	frame: - (ZOR)	C=comp.	3=design.	window	full view
	segments: <input type="checkbox"/>	(VFPNT)	airlines: -	restr.: - (YCB)	P=pads	T=term.	window low.	refresh
	vias: <input type="checkbox"/>	(VFPV)	copper: -	texts: - (CPTD)	F=fix	V=var.	center	grid on/off
layer:	COMP <input type="checkbox"/>	X	remaining <input type="checkbox"/>	X	U=unpopulated	X=contour	restart	zoom
	SIG3 <input type="checkbox"/>	X	SHAPE 1 <input type="checkbox"/>	X	A=adjustments	C=via spots		
	SIG4 <input type="checkbox"/>	X	SHAPE 0 <input type="checkbox"/>	X	Q=test spots	M=millling sp.		
	SOLD <input type="checkbox"/>	X			B=via block	Z=short A=rest.		
window:	x: -15.240	y: -25.40	// x: 341.0456	y: 248.2094	O=outline			
comp.:		pin:						

0	1
2	3
4	5
6	7
8	9
ab.-start	ab.-stop
colour on	colour off
colour_clear	
layer on	layer off
display mode:	via mode
0/0	cos on/off
colours	def. colours
work-mask	main-mask

Behind the input fields, the possible entries are listed in parentheses.

The different input possibilities allow an exact specification of the parameters to be displayed.

The input possibilities for the detailed specification of the element parameters are listed on the right side of the menu for your information.

Explanation of some specifications:

- X=all** All parameters of the respective element shall be displayed (corresponds to the specification of all letters in parentheses) - e.g. all components = fix and variable components.
- Y=free** When displaying the element "Restricted area", the 'free' restricted areas (i.e. not assigned to a signal or component) are included.
- C=comp.** When displaying the elements "Frame" and "Text" (i.e. display of component frame and legend), the components are included.
- P=pads** When displaying the elements "Segments", "Vias", "Text", the pads are included.



<b>U=proposals</b>	When displaying the elements "Segments" and "Vias", the 'proposals' (= connection possibilities for the router) are included.
<b>H=heat trap</b>	When displaying the element "Segment", the heat trap segments are included.
<b>B=via block</b>	When displaying the element "Restricted area", the areas in which no vias must be placed are included.
<b>Z=sheet</b>	When displaying the elements "Frame" and "Text" (i.e. display of component frame and legend), the work sheet elements are included.
<b>R=rout.</b>	When displaying the element "Frame", the routing area is included.
<b>O=outline</b>	When displaying the element "Frame", the board contours are included.

In "**Layer:**" in the lower part of the menu, you determine the layers to be displayed in the foreground (ref. to "Top layers", function "Colors") by entering the name or the layer number from the layer plan.

When selecting "**remaining**", all layers that have not been specified are displayed in the background (in color no. 0 of the pcb\_palette file).

Notes:

- For PRISMA stations with UNIX standard operating system, up to 4 foreground layers can be displayed at the same time.
- For PRISMA stations with UNIX OS4 operating system, up to 12 foreground layers can be displayed at the same time.

The last line of the menu contains the **component/pin search function**.

When specifying the component name, possibly the pin specification, <CR>, a new artwork is effected; the component, pin is highlighted and moves into the center of the screen.

The option "Graphic setup" in the modification menu corresponds to the option "Set graphics" in the main menu.

### Work menu

You return to the superior menu "Modification".

### Main menu

You return to the main menu.

### Window

Here, you can define a new window.

#### Procedure:

Move the cursor to the place where you want to define the lower left corner of the window. With the left mouse button (called <LMB> in this description), you fix this corner. You can then open the window with the mouse until it has the desired size. The final size of the window is determined by pressing <LMB>.

### Full view

Deletes the current window and shows the total work sheet again.

### Window loc.

By selecting "Window loc.", the current window is represented by a symbol in relation to the total size of the work sheet. Here, you may choose a new window directly: move the crosshair to the place where the lower left edge of the window is to be defined. Fix this edge with <LMB>. You can then open the window with the mouse until it has the desired size. The final size of the window is determined by pressing <LMB>.

### Refresh

With this function, the screen is cleared and you get a new artwork.

### Center

Sometimes, it is necessary to keep the current scale but to represent a slightly different window.

When selecting this function, the crosshair position becomes the new center of the picture. Note that it is not possible to move the window over the left edge of the sheet (the lower left corner is the coordinate origin).

### Grid on/off

With this function, the grid may be switched on/off. It has no influence on the effectiveness of the grid. The grid may be represented with points or lines. This can be selected in "Graphic/Color setup" in the status window.

If the grid is too fine, it will be suppressed because otherwise the screen would be white.

If the grid is not needed for the placement, it should be switched off since it has to be drawn anew for each picture; this delays the artwork.

### Status window

The optical display support of the menu "Modify graphic" can in some cases only be selected in the status window.

The display support applies to the following areas:

- (1)Color display (e.g. selection of a standard setting)
- (2)Selection of the elements to be displayed: terminals, vias, frame, auxiliary grid, the different display modes of components, etc.
- (3)Selection of the layer to be displayed (of a multilayer board)
- (4)Selection of the window (definition of windows by entering the coordinates of the lower left and the upper right corner)

### Restore

Works like "Refresh" - but the contents of the picture is not calculated anew; it will be newly written by means of the graphic memory (this is faster).

### Zoom

As known from the photo optics, you can determine a part of the picture here that shall be zoomed. With the crosshair, you define the section to be enlarged (with <LMB>) or reduced (with <MMB>) by a constant factor. With <RMB>, you can interrupt the procedure.

### Grids

The grid is an orientation help and facilitates the work at the screen. If you want to place a component for example, the crosshair moves to the point that lies next. The distance between the points may be chosen individually. In the status window, 10 different standard settings with the respective numbers are available that can be defined with the keyboard.

In order to choose of these settings, you have to click the respective number in the menu window. If you do not find a suitable setting, you can either define a new standard setting or enter the distance between the points manually with the keyboard.

In the status window, the current values appear for the different grids (digitizing, auxiliary, and routing grid). The measurements (either inches or mm) to be assigned to the numbers can be defined in the menu "Modes".

The current measurement is shown in the status window in the field "MS". (When changing the units, the system automatically changes the entry).

### Status window

The system offers four different grid types that facilitate your work, especially the digitizing and placement. You can define 10 standard values for the grids which may be stored in the status window with a respective number (0-9). New standard values can be defined by overwriting the old value and pressing <RETURN>.

### Digi-grid

This is the resolution used for the digitizing. The crosshair is moved according to this resolution.

### Disp-grid

Specifies the resolution of the grid that can be activated with "Grid on/off". In difference to the SCS side, this grid is not dependant on the digi grid.

### Place-grid

Specifies the grid for the component placement.

### Temp-grid

Here, you can adjust a temporary grid that remains active until the kind of object (component, signal, etc.) is changed.

### Digits 0-9

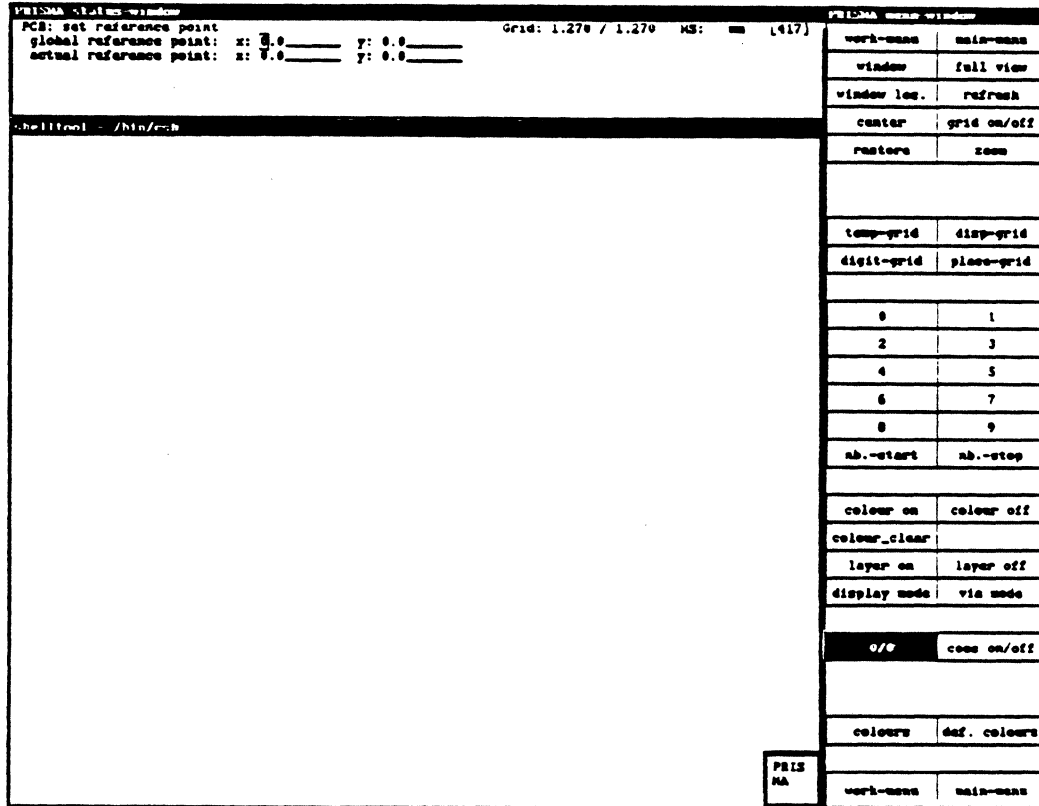
For some functions. e.g. "Grid on/off" and "Color on/off", parameters are represented by numbers in the status window. For changing the parameters, click the respective number in the menu window. The value will be changed accordingly.

Number start  
Number stop  
Color on  
Color off  
Color clear  
Layer on  
Layer off  
Display mode  
Via mode

These functions are not yet available.

0/0

With this function, the coordinate reference point can be determined. The production zero point is the coordinate reference point if nothing else has been defined (ref. to "Technology/Sizes").



**Coos on/off**

When selecting this function, the x and y coordinates of the crosshair are shown in the status window.

**Def. colors**

This function is not yet available.

**Work menu**

You get back to the superior menu "Modification".

**Main menu**

You get back to the main menu.

## COLORS

### I. Definition of the colors

The composition of the 83 available color tones for the PCB graphic is defined on the directory lib/sys in the file "pcb\_palette".

You can edit this file with the vi editor and change the color composition, if necessary.

The pcb\_palette file has the following syntax:

**Color no. red-part, green-part, blue-part color specification (or element to be displayed)**

The part of the three basic colors (red; green; blue) is defined with the respective numbers from 0.0 to 1.0:

0.0 --> color does not exist (zero part)

1.0 --> color has full intensity

Intermediate values are defined with the number behind the decimal point (= 1/10 units).

Note:

The color 0 is reserved for the color definition of the background.

Examples:

green --> 0.0, 1.0, 0.0  
yellow --> 1.0, 1.0, 0.0 (= red + green)  
black --> 0.0, 0.0, 0.0  
white --> 1.0, 1.0, 1.0  
airline --> 0.0, 0.6, 1.0 (= bluish-green)

**II. ASSIGNMENT OF THE COLORS AND THE KIND OF DISPLAY TO THE BOARD ELEMENTS**

The assignment is made in CIS-PCB in the submenu "GRAPHIC".

After calling the function "GRAPHIC", the menu "PCB: Graphic selection" appears in which you can determine the board elements to be displayed (components, terminals, segments, vias, etc.).

For the assignment of the colors that are needed for the display of the PC board elements, you activate the menu function "Colors".

The menu "PCB: Element representation & colors" appears.

PRISMA - Status-window						PRISMA - menu - definition		
PCB: element representation & colors						Grid: 1.270 / 1.270 MS: mm [575]		
set: V	variable:	trk. vias	comp.	: E_ (EM)	grid: 2 (1,2)	C=cont. F=fill	work-menu	main-menu
fix :	C3B	CB (PC,1-4,ED)	pedn :	E_ (EM)	top layers: 2	R=real D=dist.	window	full view
pedn :	F1B	FB (PC,1-4,ED)	vias :	E_ (EM)			window loc.	refresh
proposal:	C2B	CB (PC,1-4,ED)	copper :	E_ (RFCM)	highl. techn.	M=plus mark	center	grid on/off
terminal:	C1B	(PC,1-4,ED)	restr. :	E_ (EM)	wrong: N		restere	zoom
heattrap:	F	(FC)						
colour: airlines	17	highlight	16	frame	8_	remaining	19_	
COMP	2_							
SOLD	1_							
						1=solid		
						2=dotted		
						3=short dashed		
						4=long dashed		
							temp-grid	disp-grid
							digit-grid	place-grid
							0	1
							2	3
							4	5
							6	7
							8	9
							nb.-start	nb.-stop
							colour on	colour off
							colour_clear	
							layer on	layer off
							display mode	via mode
							o/o	coos on/off
							colours	def. colours
							work-menu	main-menu

## 1. SPECIFYING THE ELEMENT REPRESENTATION

In the first part of the menu, the element representation is determined. The specification possibilities are listed in the right part of the menu.

Behind the input fields, the allowed entries are specified in parentheses.

Explanation of some specifications:

- |                            |   |
|----------------------------|---|
| <b>C=cont.</b>             | Only the contours are displayed.  |
| <b>F=fill</b>              | The element is displayed filled.  |
| <b>R=real</b>              | Only the general clearance specified in the menu "Placement & routing" will be considered in the graphic output.  |
| <b>D=dist.</b>             | If an additional clearance has been defined in the track and/or via technology, it is added to the general clearance in the graphic output of the respective elements.  |
| <b>M=plus mark</b>         | The cross marking for rotation points, pin1 mark, etc. is displayed in the graphic output.  |
| <b>Top layers:</b>         | Number of signal layers that shall be displayed in the foreground at the same time (display mode).<br><br>For PRISMA stations with UNIX standard operating system, up to 4 top layers can be specified:<br><br>1 -> only one layer in the foreground (starting with the component side)<br><br>2 -> 2 layers (= 1 layer pair) in the fore-ground (starting with the two outer layers)<br><br>4 -> 4 layers (= 2 layer pairs) in the fore-ground (starting with the 2 outer layer pairs)<br><br>For systems with OS4 UNIX, up to 12 top layers can be specified.<br>With the F5 key, you change from foreground to background and with the F6 key from background to foreground. |
| <b>highl. techn. wrong</b> | Technology violations are highlighted (specification Y/N).  |



## 2. COLOR SPECIFICATIONS

The color assignment is made in the lower part of the menu.

For the color specification, you have to enter the respective color numbers from the pcb\_palette file.

Explanations:

**Airlines and Highlight**

The colors defined for airlines and high-lights in the pcb\_palette file are output here for your information and cannot be changed in this menu.

**Frame**

Here, you can specify the color number for the display of the frames (component frames, text field frames), outlines (board outline, sheet outline) and texts.

**Remaining**

Here, you enter the color number for the display of all board elements that are not in the foreground (ref. to "Top layers").

As the default, color no. 19 (= black) appears.

**COMP SOLD etc.**

The signal layers defined in the layer plan appear here.

With the color assigned to a layer, all segments and areas of this layer are displayed when it is shown in the foreground.

Note:

Vias and through-connected pins are always displayed with color no. 3 (= yellow, according to the presetting in the pcb\_palette file).

### PAN ON/OFF

With the pan (or scrolling) function and the window defined in "Modes", you can go through the complete picture memory (= entire board).

The functions "Window", "Full view", "Window loc.", "Refresh", "Center", and "Grid on/off" are all needed for a better display. They are also available in the submenu "Modification graphic".

### WINDOW

Here, you can define a new window.

Procedure:

Move the cursor to the place where you want to define the lower left corner of the window. With <LMB>, you fix this corner. You can then open the window with the mouse until it has the desired size. The final size of the window is determined by pressing <LMB>.

### FULL VIEW

Deletes the current window and shows the total work sheet again.

### WINDOW LOC.

By selecting "Window loc.", the current window is represented by a symbol in relation to the total size of the work sheet. Here, you may choose a new window directly: move the crosshair to the place where the lower left edge of the window is to be defined. Fix this edge with <LMB>. You can then open the window with the mouse until it has the desired size. The final size of the window is determined by pressing <LMB>.

Not yet implemented.

### REFRESH

With this function, the screen is cleared and you get a new artwork.

### CENTER

Sometimes, it is necessary to keep the current scale but to represent a slightly different window.

When selecting this function, the crosshair position becomes the new center of the picture. Note that it is not possible to move the window over the left edge of the sheet (the lower left corner is the coordinate origin).

### GRID ON/OFF

With this function, the grid may be switched on/off. It has no influence on the effectiveness of the grid. The grid may be represented with points or lines. This can be selected in "Graphic/Color setup" in the status window.

If the grid is too fine, it will be suppressed because otherwise the screen would be white.

If the grid is not needed for the placement, it should be switched off since it has to be drawn anew for each picture; this delays the artwork.

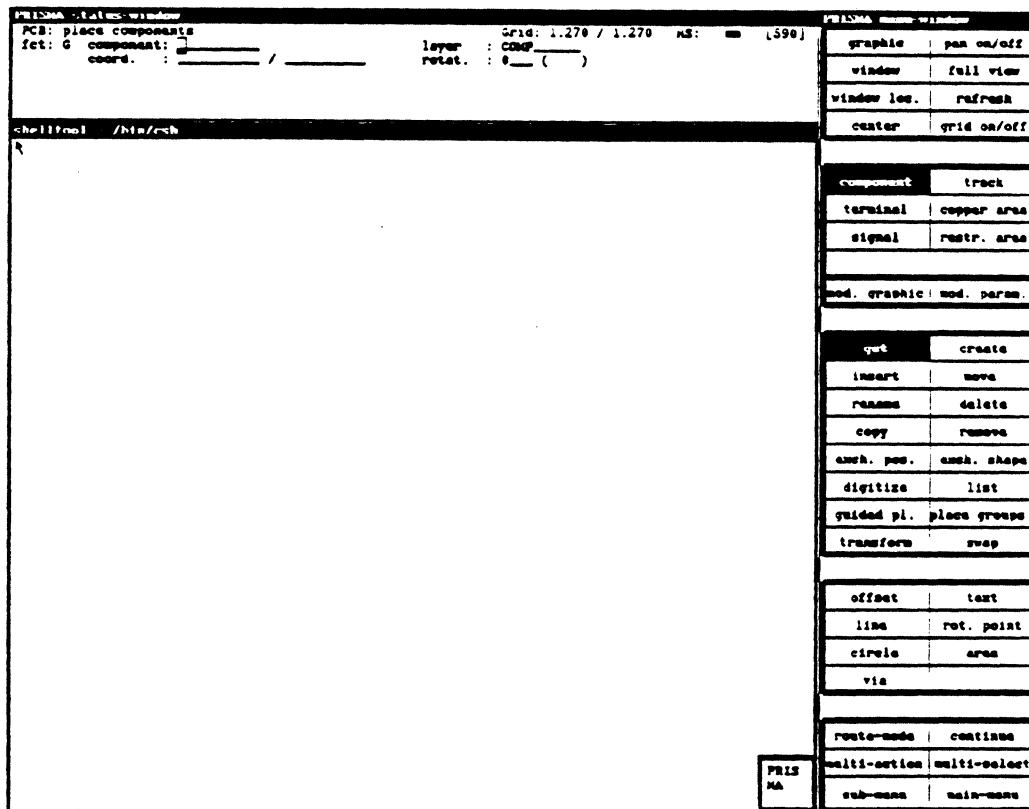
A component can be connected with several single operations that will be described here:

### GET COMPONENT

This function always precedes each operation. "Get" means activating an already generated component for further processing. The activation is achieved by clicking the component with <LMB>.

The activated object will be highlighted (by standard, it will be displayed in blue).

After "Get", the cursor automatically jumps to "Move component".



### Status window:

The status window (basic window) will be shown for all operations with components. All functions can either be activated in the menu window or with a respective keyboard entry.

**CREATE COMPONENT**

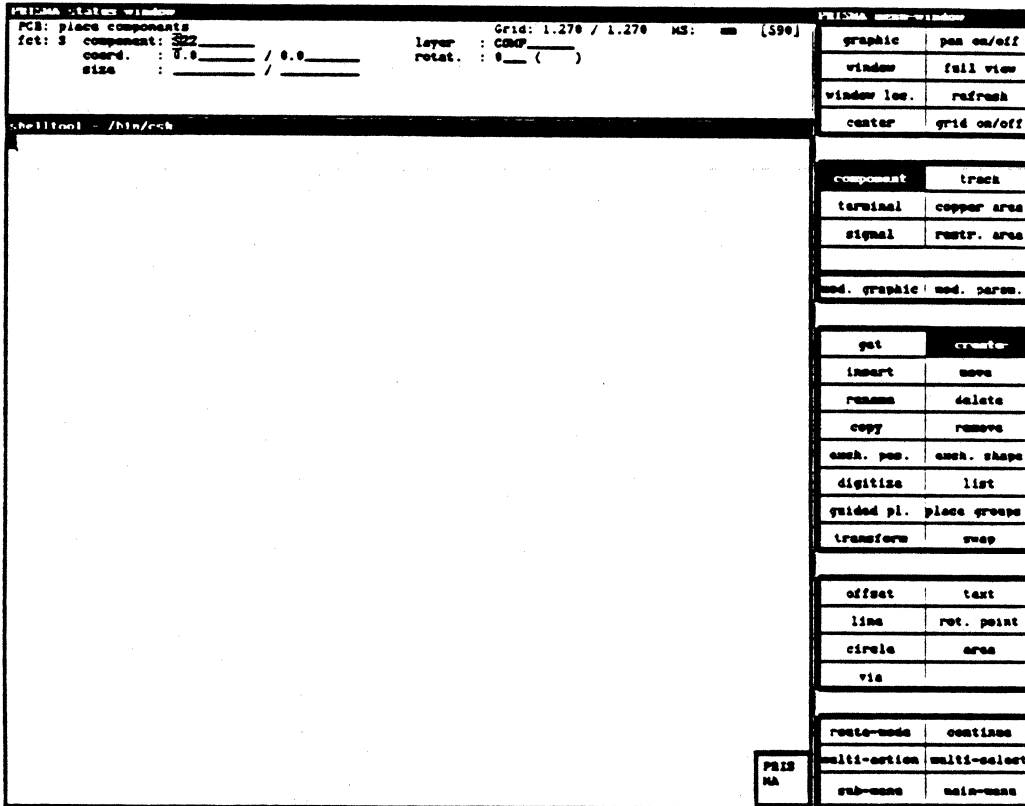
With this function, new components that do not yet exist in the library can be taken into the system.

These new components are only placeholders at first, represented by a rectangle.

With <LMB>, this contour can be opened from the lower left corner to the upper right side.

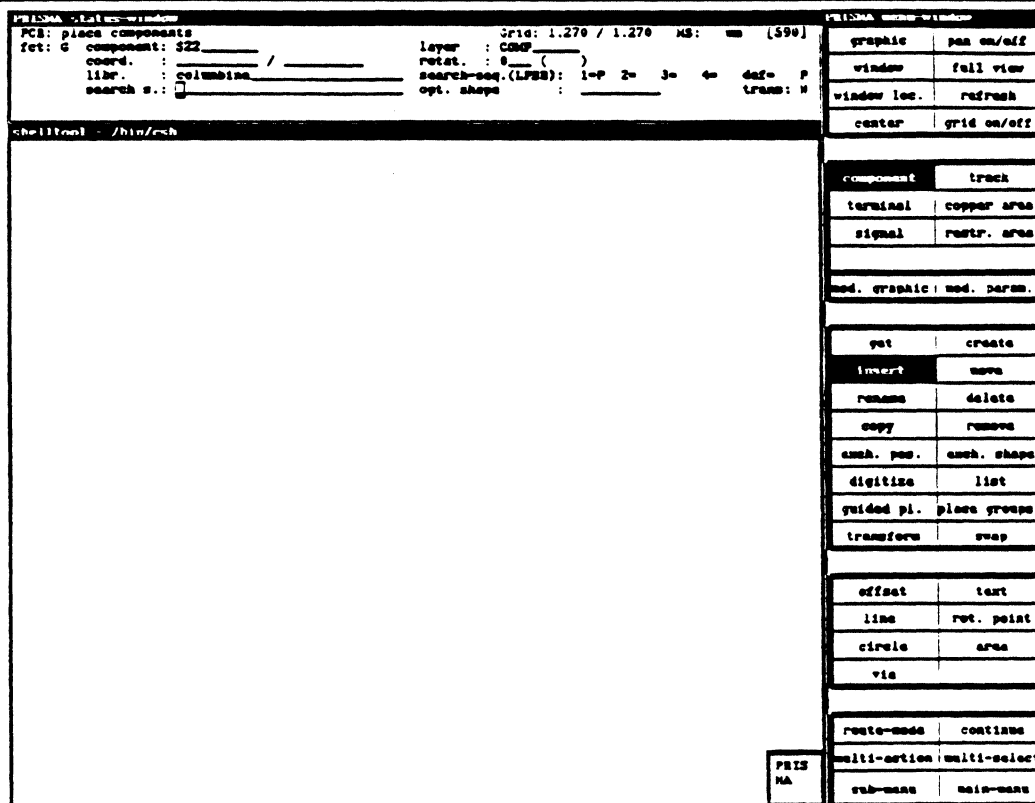
A name is assigned to each generated component.

Afterwards, the cursor moves to "Create terminal".



**INSERT COMPONENT**

This function allows the access to components existing in the library. If you know the name of the component, you can enter it in the status window. The respective component is attached to the crosshair and can be placed. If you do not know the name, the library can be listed in a window and you may choose the desired component.



### Status window

When inserting components in the layout, a read and/or write access to the library COLUMBINE is sometimes necessary.

In order to insert components from the library, there are 4 different search criteria:

- L: logical name (e.g. "NOR", "NAND", etc.)
- P: physical element that realizes this logical function (e.g. "SN74LS00")
- S: item number (internal meaning)
- B: shape (e.g. "DIP16")

### MOVE COMPONENT

With this function, a component can be shifted or rotated in the layout. Therefore, you have to get the respective element first. The angle is specified in  $1/10^\circ$  (e.g. "90°" corresponds to "900"). The direction of the rotation is mathematically positive, i.e. counter-clockwise.

When moving the mouse, the contour of the selected component is also shifted.

Move to the desired position and fix it with <LMB>. Rotations in  $90^\circ$  steps are effected with <MMB>. "Collisions" with other components cause an error message in the status window. A symbol cannot collide with itself so that slight shiftings are possible.

In "Move component", the component is shifted with all attached terminals. The signals are also shifted. If tracks have been digitized, new airlines starting at the connection points of the tracks are connected to the respective pin.

### RENAME COMPONENT

You can assign a name to the activated component if the numbering during the generation becomes tangled. In the status window, you can enter the desired name with the keyboard.

The name may consist of up to 10 characters, but must not begin with \$ because this character is reserved by the system.

### DELETE COMPONENT

An activated component is deleted with <LMB> if "delete immediately" has been activated in the submenu. Otherwise press <LMB> again or <RETURN> for confirmation.

#### ATTENTION!

If you delete a component, all signals and terminals of this component will also be deleted. In addition, the respective symbols, signals and terminals disappear from the schematics.

### REMOVE COMPONENT

Invalid command

### COPY COMPONENT

With this function, a component can be copied to any free place. First a rectangle appears around the component at the crosshair position. With the mouse, it can be moved to the desired position. There, the copy of the component is inserted with <LMB>.

Previously, it can be rotated in 90° steps with <MMB>.

Before placing the component, a new name and a rotation angle may be assigned to it in the status window. If no name is entered, the component is named by the system.

### EXCH. POS.

With this function, a pair of already drawn components can be exchanged. The connections are moved together with the components.

#### Procedure:

1. Get the first component.
2. Select "Exch. pos."
3. Click the second component.

#### ATTENTION!

Already drawn tracks become airlines again. That may destroy your work.

**EXCH. SHAPE**

With this function, you can replace one component by another.

Procedure:

1. Get the component to be replaced.
2. Select "Exch. shape".
3. Click the other component.

**DIGITIZE COMPONENT**

Can only be used with the function "Modify graphic".

**LIST COMPONENT**

A list of component names will be displayed in a text window that appears on the screen. The window cannot be shifted. By clicking the fields "continue" or "previous", paging is possible. The component is selected by clicking the respective name.

You can only continue your work in the layout if a component name has been selected or the window has been closed by clicking "quit".

In connection with the operations "Get component" or "Create component", a listing of the components that already exist on the schematics is output.

With the option "Insert component", a list of all components available in the library is shown.

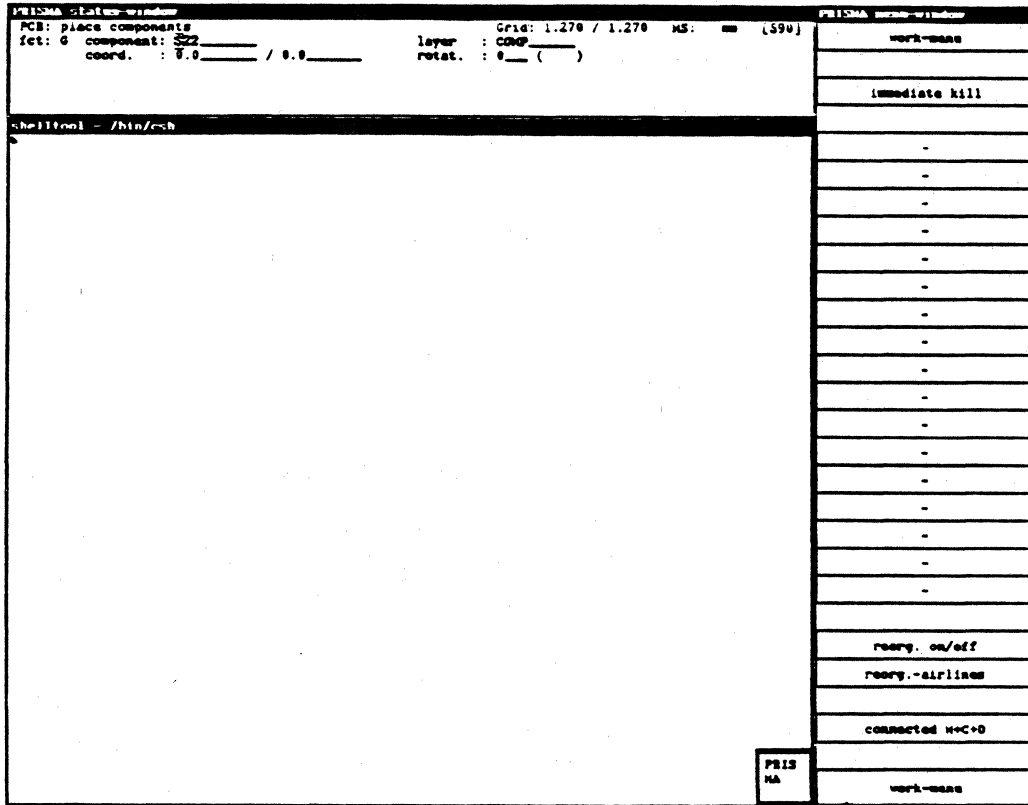
All other combinations require the activation of a symbol with "Get symbol". For this reason, the list of the already used symbols is output each time.

**GUIDED PL.**

With "Guided pl.", the components can be placed anywhere with the mouse. After releasing a component with <LMB>, the next component is automatically offered, i.e. you need not get it first. The sequence in which the components are offered is determined by the system and considers the number of connections to already placed components.

# LAYOUT/Modification

## SUBMENU PLACE COMPONENTS



### IMMEDIATE KILL

If you activate "delete" and this function after getting a component, it will be deleted immediately. It might be necessary to press <LMB> or <RETURN> for confirmation.

### REORG ON/OFF

After activating this function, the airlines of the connections to the component are optimized by the system when shifting the component.

### REORG.-AIRLINES

When calling this function, the system optimizes the airlines.

### CONNECTED M+C+D

When moving (M), copying (C) or deleting (D) a component or component groups, the digitized tracks are also modified.



**PLACE GROUPS**

In the popup menu, the following options are available:

- all components
- no component
- reset
- adjust
- force model
- force vectors
- matrix model
- pairwise interchange
- digital
- previous
- swap
- show gates
- main menu

**ALL COMPONENTS**

With this function, all components are taken up.

**NO COMPONENT**

With this function, you can undo the function "all components".

**RESET**

All activated components are placed outside the board.

**ADJUST**

The activated components are adjusted.

Necessary parameters:

**Adjustment axis:**

- x     The components are placed on a line that is parallel to the x-axis. The x-coordinate of the components will not be changed, the new y-coordinate must be specified in "Coord."
- y     See above, but the line is parallel to the y-axis.

**Coord.:**

x or y-coordinate of the adjustment axis.

**Mode:**

- 0     The components are placed on the adjustment axis.
- 1     The components are placed that the lower (or left) edge of the components lies on the adjustment axis.
- 2     like "1", but the upper (or right) edge of the components lies on the adjustment axis.

## Check:

- N No collision check is effected. The components may overlap.
- Y A collision check is executed. When a collision is found, the function is interrupted.

## FORCE MODEL

All components that have not yet been placed (= all components taken from the board with "Reset") will be placed on the board according to a force model that tries to minimize the connection lengths to the already placed components. This function is only possible if firmly placed components (e.g. connectors) already exist on the board. With this procedure, the components may overlap.

## FORCE VECTORS

The system calculates the main direction of the signals at the activated components and shows it with its force vectors. With a refresh, the red lines disappear.

## MATRIX MODEL

The system divides the board into different areas so that each component gets an area. The system places the components. With a refresh, the red lines disappear.

## PAIRWISE INTERCHANGE

The system changes the placement of the components to get the shortest connections between the different components.

## DIGITAL

Preparation of the board for the digitizing. This function should always be called if tracks are digitized before calling a placement automation.

## PREVIOUS

With "previous" or by clicking the menu field "Component", the function "Place groups" can be left.

## SWAP

With this function, you can swap gates between components or pins within a gate.

### Procedure:

- (1) Click swap
- (2) Click a pin  
either
- (3) Click a pin of the same gate; the two pins swap the signals  
or
- (3) Click a pin of another gate; the two gates swap their position within the components.

The pins or gates to be swapped must be of the same type.

**SHOW GATES**

This function shows which pins belong to a gate.

**MAIN MENU**

With this function, you get directly back to the main menu.

**TRANSFORM COMPONENT**

Newly created components for which no symbol and no relation exists on the SCS side, can be 'copied' to symbols on the schematics with this function.

Symbol and shape have the same graphic; they are related automatically.

**SWAP COMPONENT**

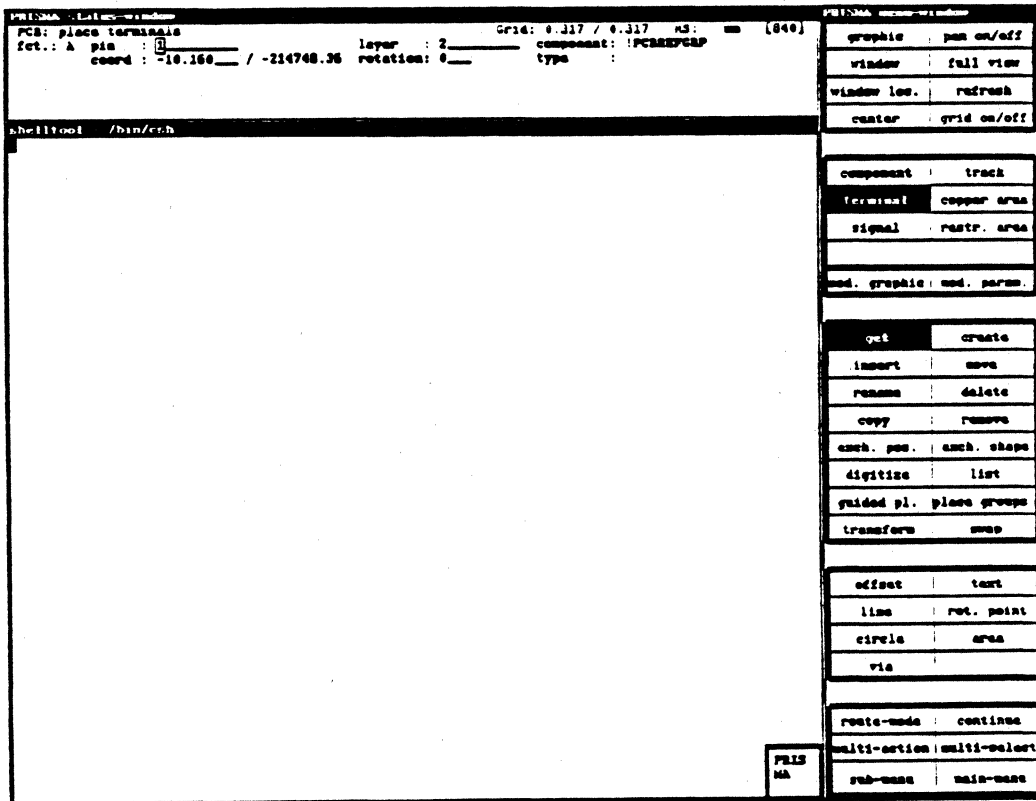
Invalid function

**GET TERMINAL**

The connection points (terminals) are activated with <LMB> for the further processing.

Afterwards the cursor jumps to "Move terminal".

Activated terminals are not moved in the crosshair but in a small square. With <LMB>, the new position is fixed. The terminals are incremented for each component. The identification is made according to the component name and the terminal number in the status window.



**CREATE TERMINAL**

When generating a terminal, the position of the crosshair determines the position of the newly created connection point. Terminals can be placed at all free places, i.e. places without any objects (components, other terminals, signals, tracks, copper areas, restricted areas).

You have to get the assigned component previously.  
If two terminals are too close (or overlap), the error message "Pin collision" is shown in the status window. Terminal groups (e.g. the pin rows of an IC) are created as follows:

- (1) Enter the x and y values for the distance in the status window.
- (2) Enter the desired number of terminals.
- (3) Place the first terminal at the desired place or enter its coordinates in the status window.
- (4) Confirm the entry with <RETURN>.

The origin of the reference system is the placement coordinate of the component.  
The terminal coordinates refer to this origin.

Attention: Plausibility checks concerning the position of terminals within the component will not be executed.

#### **Status window**

If a group of terminals is to be created, you have to enter the distance between the terminals in the field "DIST." (with the respective coordinates).

In "Numb.", you specify the number of terminals to be created.

#### **INSERT TERMINAL**

Invalid operation

#### **MOVE TERMINAL**

Terminals activated with <LMB> can be shifted to a new position with this function.  
With <MMB>, the terminal is rotated in 90° steps. It is not possible to superimpose terminals; you will get a respective error message ("Pin collision").

#### **RENAME TERMINAL**

During the generation, the terminals get a definite name. It consists of the name of the assigned component (e.g. IC8) and the terminal number.

The name can be changed in the status window.

Procedure:

- (1) Get the terminal with <LMB>.
- (2) Activate the function "Rename terminal".
- (3) Change the name of the activated terminal in the status window.
- (3) Confirm the entry with <RETURN>.

### DELETE TERMINAL

Terminals can only be deleted one by one.

Attention: The removal of terminals has no effect on the SCS part.  
In the terminal parameters, there is no deletion protection for terminals.

Terminal groups cannot be deleted (for that reason, the options "Continue", "Multi-action" and "Multi-select" are not allowed for terminals).

### COPY TERMINAL

Activated terminals are copied to the desired place with this function. The copy is assigned to the component shown in the status window and gets the specified serial number. For copying a terminal several times, ref. to "Create terminals".

### REMOVE TERMINAL

Invalid command (see "Delete terminal").

### EXCH. POS.

Invalid command

### EXCH. SHAPE

With this function, you can replace one terminal by another.

Procedure:

1. Get the "master terminal".
2. Select "Exch. shape".
3. Click the terminal to be replaced.

### DIGITIZE TERMINAL

Invalid command

### LIST TERMINAL

Terminals can not only be selected directly (with the crosshair) but also in a terminal list.

Procedure:

- (1) Select "Get terminal".
- (2) Get the component whose terminals are to be listed.
- (3) Select "List terminal".
- (4) Get the desired terminal name with the mouse; you may use the text fields "continue" or "previous" for paging.
- (5) You can only continue your work after leaving the list window. In order to leave the window, you have to select a terminal name. If no terminal name is selected, the window can be left with "quit".

The options

"guided pl.", "place groups", "transform", "swap" are not allowed in connection with terminals.

In the status window, the signal, symbol and component names are specified to which the respective connection point of a terminal refers. Signal and symbol names come from the model schematics (SCS), whereas the component name is the connection to the layout (PCB) model.

The following specifications are used:

SCS terminal  
# SCS signal  
& SCS symbol  
\$ PCB component

### **GET SIGNAL**

A signal in the layout model corresponds to an airline in the schematics model. Signals are the logical significance of a connection. This logical connection can be realized physically as track. Because of these two interpretation possibilities, a connection can be processed as signal (logical) or as track (physical). The activation for the further processing (e.g. digitizing) is made with "get". Therefore, the respective signal is clicked with <LMB>.

The activated signal and the assigned terminals (pins) are highlighted and the cursor moves to "Insert signal".

### **CREATE SIGNAL**

With this function, new signals can be created.

Procedure:

- (1) Enter the name in the status window.
- (2) Press <RETURN>.

### **INSERT SIGNAL**

With this function, the connection of two terminals or copper areas over a signal wire is possible.

Procedure:

- (1) Create the copper area.
- (2) Get the respective signal
- (3) Select "Insert signal".
- (4) Click the terminal/copper area with <LMB>.

Track connections between terminals (pins) are only accepted by the system if both terminals were connected logically by the same signal. If this is not the case, the error message "Pin could not be connected" appears.

### **MOVE SIGNAL**

Invalid operation

### RENAME SIGNAL

When generating signals, they get a definite name by the system (on the PCB side the character "#" followed by a number). If you want another name for the signal, it must be activated with "Get signal". Then you may enter a new name in the status window (up to 10 characters). The name must not begin with "#", as this character is reserved for the system. The input is finished by pressing <RETURN>.

### DELETE SIGNAL

Activated signals can be deleted with this function. It must be confirmed with <RETURN>.

The deletion is irrevocable. Therefore, you should check before using this function if the activated signal shall really be deleted.

### COPY SIGNAL

Invalid function

### REMOVE SIGNAL

Complementary function to "Insert signal". This function removes the assignment of a pin to a (logical) signal. The cursor does not automatically move back to "Get signal", so that complete terminal groups can be removed.

### EXCH. POS.

Invalid operation

### EXCH. SHAPE

Invalid operation

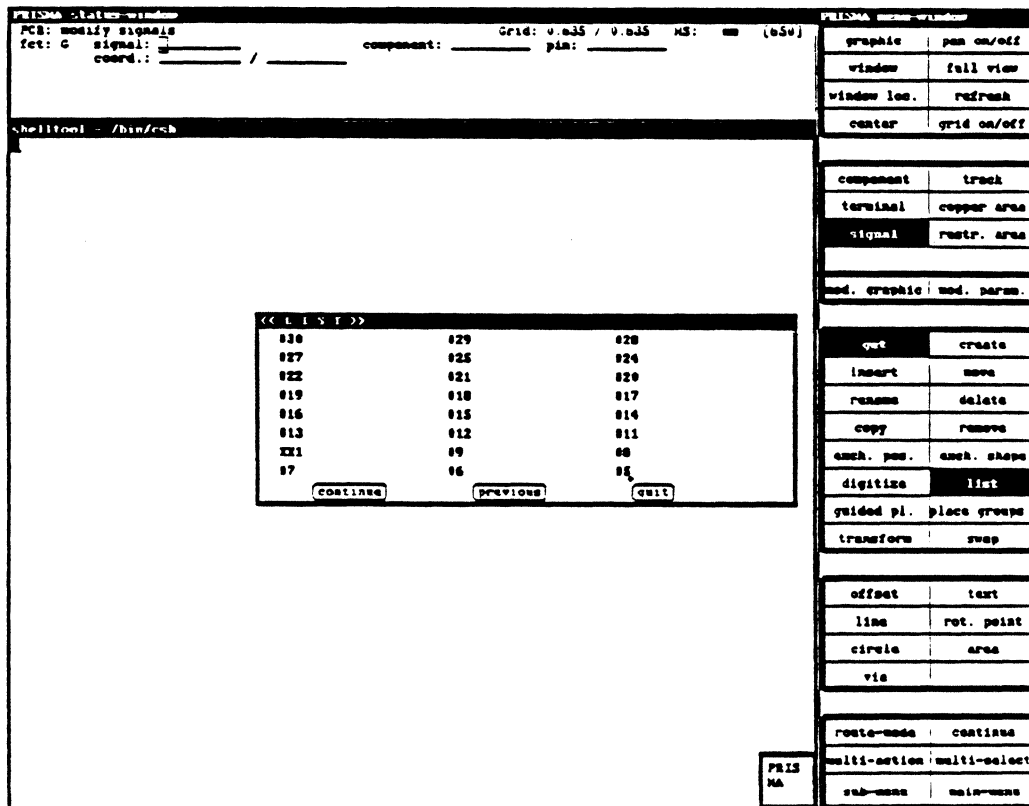
### DIGITIZE SIGNAL

Invalid operation



## LIST SIGNAL

In an additional text window, the names of all signals existing in the layout are listed. With "continue" or "previous", paging is possible. You leave the window with "quit" or by selecting a signal name with <LMB>. The respective signal is highlighted and activated for the further processing. If you get a signal before calling this function, all terminals of this signal will be listed.



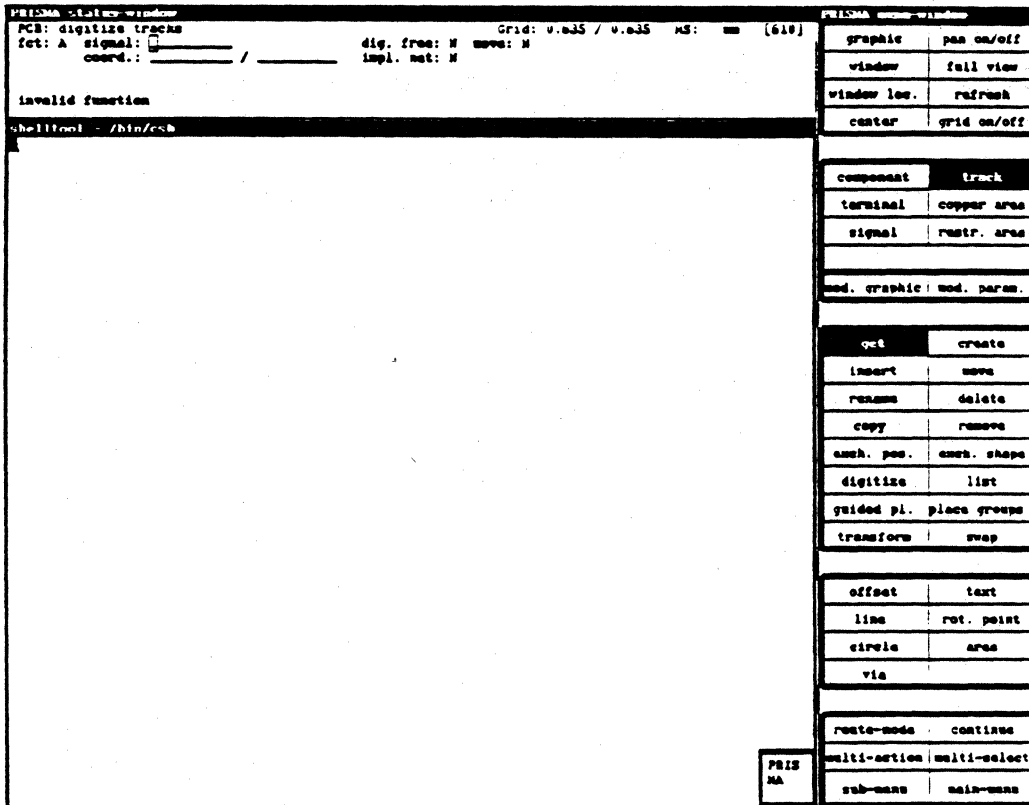
Guided pl.  
Place signal groups  
Transform signal  
Swap signal  
These are invalid functions

The complete digitizing help (modify graphics, digitize and the assigned functions offset, line, circle, via, text, rotation point, area) is not available for signals.

Refer to the track functions.

TRACKS

The signals that you created in the SCS model appear as airlines on the PCB side. Tracks are the physical realization of these airlines. If you click tracks for modification, the function "Get" will be activated first. The object "track" consists of "line" and "via" (through connection). By activating "Line" or "Via", you can either get lines or vias. If no digitizing help is activated, lines will be preferred.



GET TRACK, LINE

This function is used to get an airline or an existing track. In the status window, the name of the signal that is attached to this track appears.

The system moves to "digitize" or "shift" according to the operation that was chosen last.

ATTENTION: It is important for the digitizing at which place you get the track.

GET TRACK, VIA

With this function, you can get a via. The system moves to "digitize" or "shift" according to the operation that was chosen last.

**MOVE TRACK, LINE**

After getting a track, its course can be changed. A line appears on the screen. The part of this line that is vertical to the track determines the shifting. With <LMB>, the shifting is executed. Only a parallel shifting is possible. The neighboring track parts will be lengthened or shortened.

ATTENTION: Vias are fixed points, i.e. they must be shifted separately.

**MOVE TRACK, VIA**

With this function, an activated via can be shifted. The tracks will be lengthened or shortened to the via. If the track is displayed in yellow, the red and the green tracks overlap.

**DELETE TRACK, LINE**

By clicking this function, the activated part of the track can be deleted again. The vias are fixed points and are not effected by this function. A deleted track appears as blue airline on the screen.

The digitizing of a track can be interrupted at any time by clicking "Get". An airline is drawn from this point to the target pin for your orientation.

If a track was digitized incorrectly, it can be deleted from point to point. By pressing <MMB>, the track disappears up to the last point whereas a via consists of two points. The airline to the target pin will be moved accordingly.

For better understanding, the handling of the mouse is explained here:

<LMB> Place the points and digitize the tracks

<MMB> Delete the track from point to point.

<RMB> Place the vias.

If you press <MMB> after getting the track, it has the same effect as clicking the function "Delete".

**DELETE TRACK, VIA**

An activated via can also be deleted by clicking the function "Delete". On the screen, it appears as terminal. The track to the terminal will remain. If the terminal is also deleted, the tracks are replaced by an airline.

### DIGITIZE TRACK, LINE

#### AIRLINE - TRACK

With this function, airlines, i.e. the signals of the schematic model, can be transformed into physical tracks. Therefore, you have to get the respective airline. From the starting pin of the airline, the track is moved together with the crosshair. By pressing <LMB>, you set a marking and the track is digitized between starting pin and crosshair position, the remaining track will be displayed as airline to the target pin. In this way, you can digitize the complete track step by step between starting and target pin. In the submenu, the possible track courses are defined. If for example the auto snap mode and "only XY valid" are active, only horizontal and vertical track parts can be digitized. If two tracks cross, the message "Short-circuit" appears. In this case, place a via (through connection) with <RMB> before the place where the tracks cross and digitize the track on the back. The via appears as yellow spot, the track on the back appears in red. If you want to return to the front, place a new via with <RMB>. The track is displayed in green again. When you have reached the target pin, the cursor moves back to "Get track" and another track can be digitized. If you click "Get" before reaching the target pin, the digitizing will be interrupted. Then an airline appears from the last marking to the target pin.

#### TRACK -> TRACK

Instead of an airline, an existing track can be digitized. With this function, the course of the existing track can be changed. Therefore, you have to get the track part to be changed near the point where you want to start. The procedure is the same as for an airline. But the target of the new track part is not the target pin of the complete track, but the next marking of the original track.

**ATTENTION:** With <MMB>, you can not only delete the new track part from point to point but also the original track after the current marking provided that you have set a new marking.

Note: In "Digitize track, line", vias can also be deleted with <MMB>.

#### VIA

A via is a through connection from one board side to the other.

Vias are placed to avoid track crossings. The track will then be digitized behind the through connection. The change of sides becomes visible by the color change.

Vias are placed in the following way:

- (1) Digitize the tracks
- (2) Press <RMB> before an unallowed crossing: The via is placed and is represented by a filled spot.
- (3) Digitize the track behind the through connection.

Short circuits (track crossings) are shown as error message in the status window as a rule.

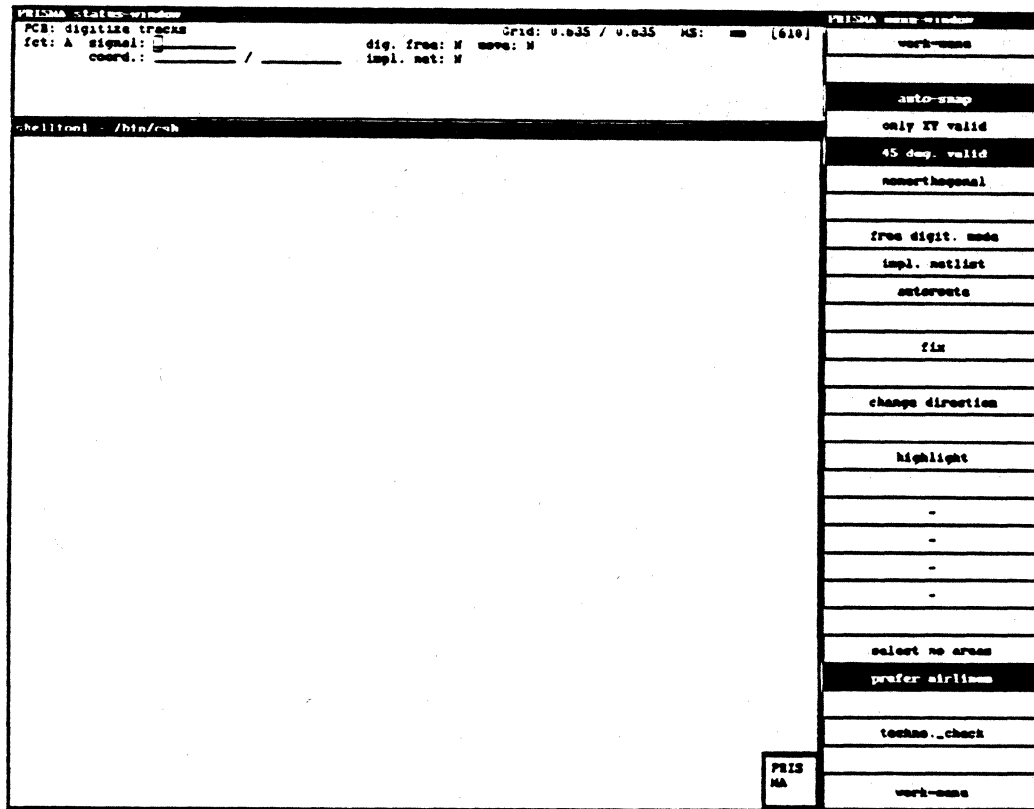
With the option "Via", the through connections can be moved to new positions.

Procedure:

- (1) Select "Get track".
- (2) Switch on "Via".
- (3) Get the respective via with <MMB> (the crosshair must be exactly on the middle of the via).
- (4) The cursor jumps to "Move". Shift the via by moving the crosshair to the desired place and press <LMB>.

It is also possible to delete a via. The connected tracks and the logical signal connections will remain and can afterwards be digitized.

## SUBMENU DIGITIZE TRACKS



### AUTO-SNAP

Special gradients can be defined for the digitizing of tracks (45°, X,Y). Other gradients are not allowed then.

When the auto-snap mode is switched on, the tracks are only drawn with the defined gradients.

The work will be facilitated if the user need not adhere exactly to the gradient.

When pressing <LMB>, the system draws the vertical component. If the auto-snap mode is switched off, the error message "Gradient not allowed" appears if the work was inaccurate.

### ONLY X,Y VALID

As gradient for the tracks, only multiples of 90° are allowed, i.e. 90°, 180°, etc.

**45 DEG. VALID**

As gradient for the tracks, only multiples of 45° are allowed, i.e. 45°, 90°, 135°, 180°, etc.

**NONORTHOGONAL**

All gradients are allowed.

**FREE DIGIT. MODE**

Starting with a terminal, tracks can be digitized freely, i.e. without airline display.

**IMPL. NETLIST**

If terminals that have not yet been assigned to a signal are found during the digitizing, they will be taken into the current signal if this function is active.

**AUTOROUTE**

When shifting vias, the tracing of tracks can be switched on or off with this function.

**CHANGE DIRECTION**

With this option, the starting and target terminal can be exchanged. This is only possible after getting a track.

**HIGHLIGHT**

The complete signal to which the activated track belongs will be highlighted.

**TECHNO. CHECK**

With this function, a track can be checked for interruptions and short circuits.

**SELECT NO AREAS**

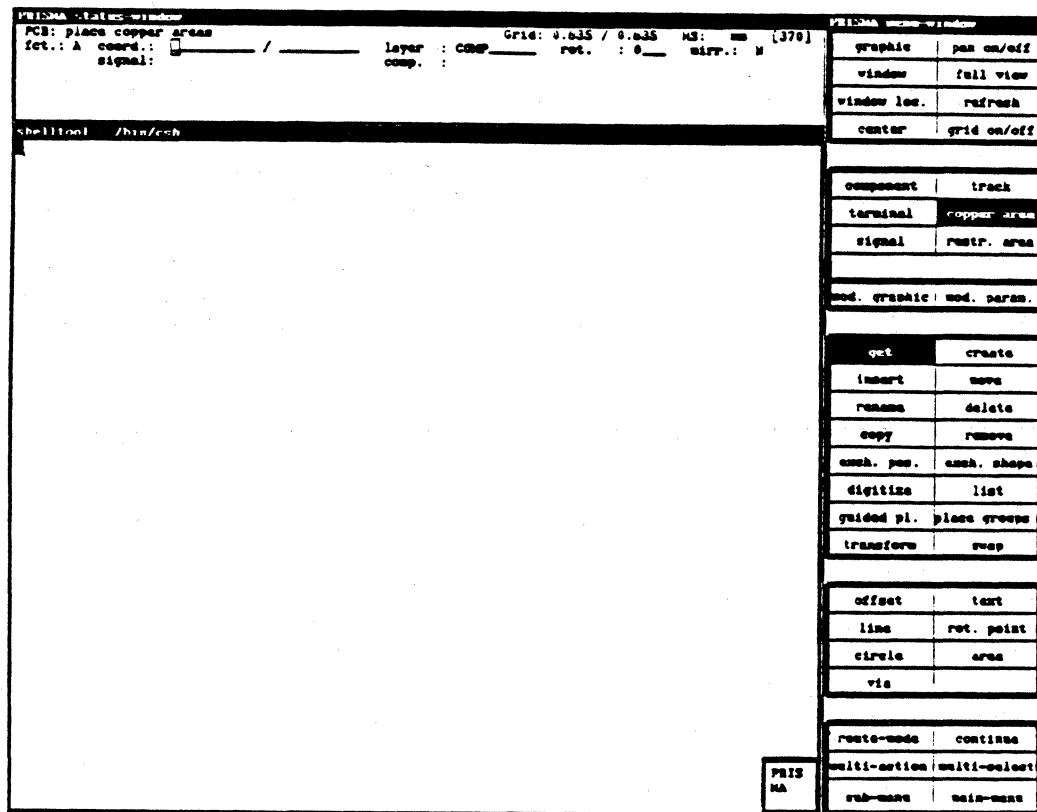
As a rule, copper areas are treated like vias. If this option is active, it is not allowed to get copper areas.

**PREFER AIRLINES**

After activating this option, airlines will be preferred when getting several lines, otherwise tracks.

## GET COPPER AREA

Copper areas that already exist in the layout are activated with "Get" and <LMB> for the further processing. An activated copper area is not highlighted, but a modification symbol with the same shape as the activated area is attached to the crosshair. Afterwards the cursor jumps to "Move copper area".



## CREATE COPPER AREA

At the crosshair position, the lower left corner of a rectangle copper area is created that can be opened by moving the mouse.

With <LMB>, the copper area is fixed and will be displayed as colored area. Please consider that this area has not yet been assigned to a signal. The cursor automatically jumps to "Move copper area".

## INSERT COPPER AREA

Invalid function (ref. to "Create copper area")

## MOVE COPPER AREA

With this function, activated copper areas can be shifted to another place. Short circuits should be avoided - a warning appears for collisions with tracks on the same layer that will be ignored when pressing <LMB> again. Then the operation will be executed.



**RENAME COPPER AREA**

Invalid function

**DELETE COPPER AREA**

This function deletes activated copper areas from the layout by pressing <LMB>. A deletion protection does not exist for copper areas.

**COPY COPPER AREA**

With this function, an activated copper area is copied to the crosshair position by pressing <LMB>.

**REMOVE COPPER AREA**

Invalid function

**EXCH. POS.**

Invalid function

**EXCH. SHAPE**

Invalid function

**DIGITIZE COPPER AREA**

This function only exists in connection with the option "Modify graphic". It is used to change the shape of the copper area. The digitizing help offered in "Modify graphic" (offset, line, circle, area) may be used. The changes are finished with the menu function "STOP". Refer also to "Modify graphic".

**LIST COPPER AREA**

**GUIDED PL.**

**PLACE GROUPS**

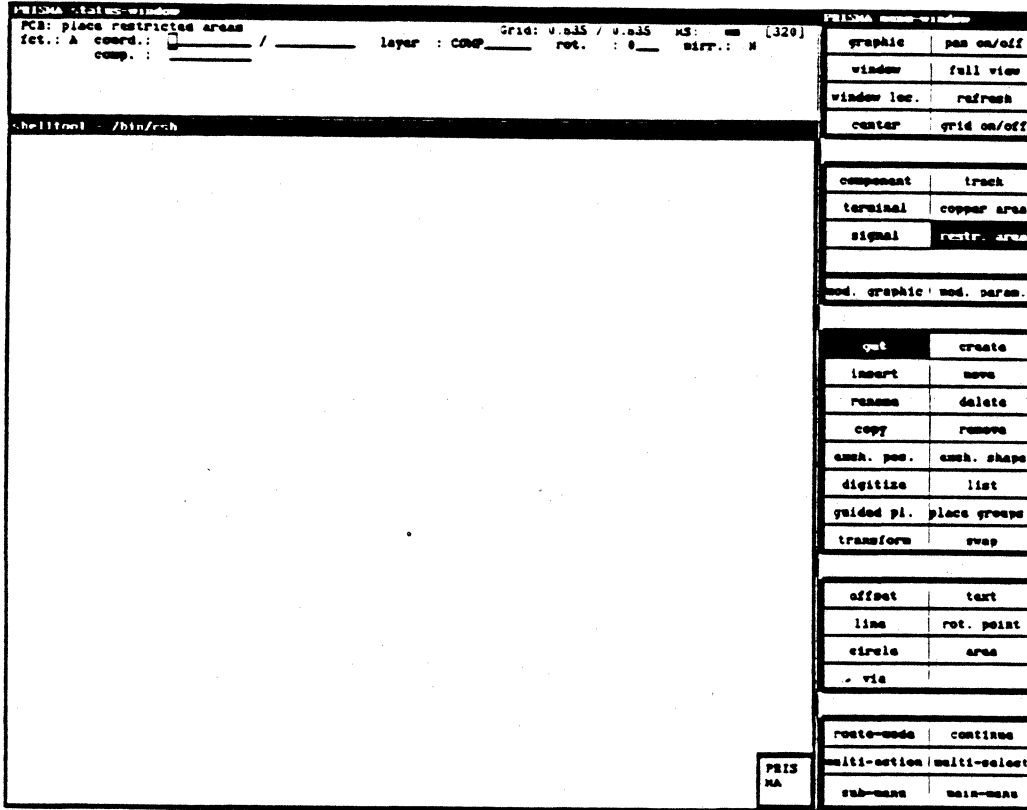
**TRANSFORM COPPER AREA**

**SWAP COPPER AREA**

Invalid functions

## GET RESTR. AREA

Restricted areas that already exist in the layout are activated with "Get" and <LMB> for the further processing. An activated restricted area is not highlighted, but a modification symbol with the same shape as the activated area is attached to the crosshair. Afterwards the cursor jumps to "Move restr. area".



## CREATE RESTR. AREA

At the crosshair position, the lower left corner of a rectangle restricted area is created that can be opened by moving the mouse.

With <LMB>, the restricted area is fixed and will be displayed as colored area. Please consider that no name has been assigned to this area. The cursor automatically jumps to "Move restr. area".

## INSERT RESTR. AREA

Invalid function

## MOVE RESTR. AREA

With this function, activated restricted areas can be shifted to another place. A warning appears for collisions with tracks on the same layer that will be ignored when pressing <LMB> again. Then the operation will be executed.

**RENAME RESTR. AREA**

Invalid function

**DELETE RESTR. AREA**

This function deletes activated restricted areas from the layout by pressing <LMB>. A deletion protection does not exist for restricted areas.

**COPY RESTR. AREA**

With this function, an activated restricted area is copied to the crosshair position by pressing <LMB>.

**REMOVE RESTR. AREA**

Invalid function .

**EXCH. POS.**

Invalid function

**EXCH. SHAPE**

Invalid function

**Status window**

In the status window, you have the possibility to assign new attributes to the defined restricted areas. They apply to the position of the restricted areas and the objects to be blocked.

Parameters for the position:

- Coordinate of a point on the restricted area
- Layer
- Component to which the restricted area belongs

**DIGITIZE RESTR. AREA**

This function only exists in connection with the option "Modify graphic". It is used to change the shape of the restricted area. The digitizing help offered in "Modify graphic" (offset, line, circle, area) may be used.

Refer also to "Modify graphic".

**LIST RESTR. AREA****GUIDED PL.****PLACE GROUPS****TRANSFORM RESTR. AREA****SWAP RESTR. AREA**

Invalid functions

## MODIFY GRAPHIC

This function switches to the working area of the graphic digitizing and offers different drawing facilities. The function cannot be called in connection with the objects "Signal" and "Track".

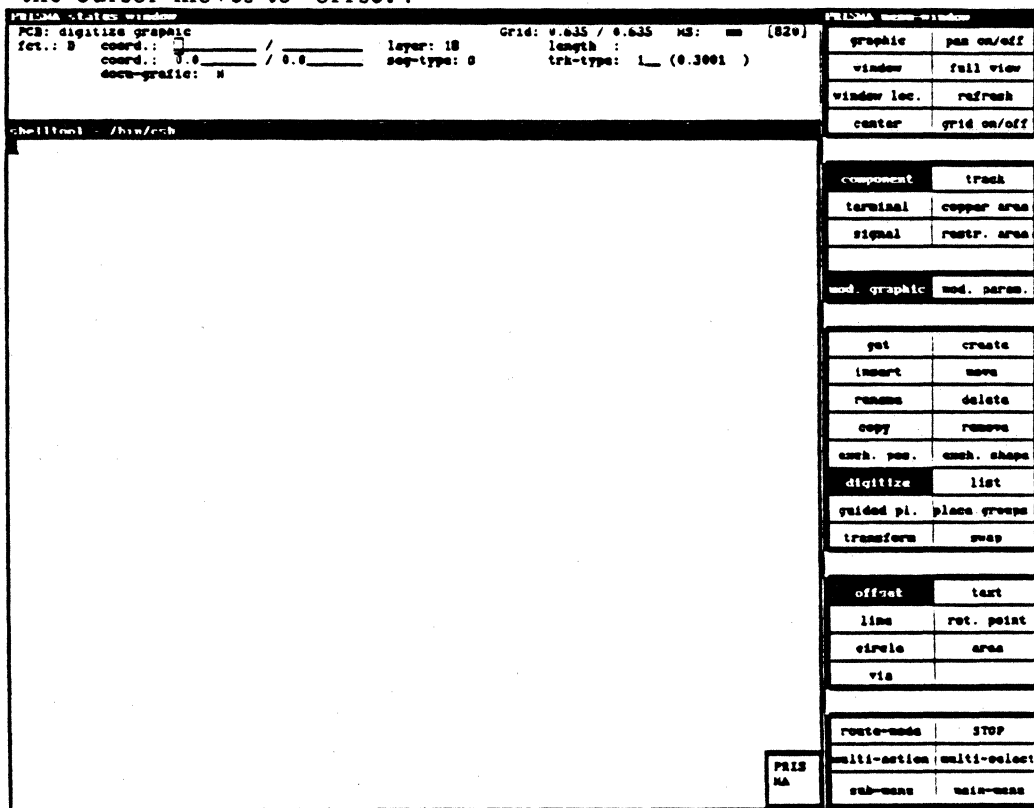
You can activate this function in two different ways:

(1) Click the object type (e.g. "Component");

switch on "Modify graphic";  
get the component with <LMB>;  
the cursor moves to "offset".

(2) Get the object (e.g. "Get component");

switch on "Modify graphic";  
the cursor moves to "offset".



In all cases, "Modify graphic" refers to a special object that has been activated with <LMB> for further processing.

After the activation, the cursor moves to "Digitize offset" and the origin of the activated object is marked with a cross.

For the further modification, you may use the digitizing help offset, line, circle and area.

Independent from the object, the status window "Digitize graphic" appears.

---

**TEXT**

With this function, component legends are created. Only components and terminals can be lettered.

**Procedure:**

- (1) Select the kind of object (e.g. "Component").
- (2) Switch on "Modify graphic".
- (3) Get an object with <LMB>.
- (4) Select the option "Text".
- (5) Specify a text type in the text window. The different text types can be defined in the menu "Technology".

Text type "0" is free text, i.e. the characters can be entered in the status window in "Text".

- (6) Open a window with <LMB> by moving the mouse from the lower left to the upper right corner.
- (7) Other windows can be opened and the created text block can be inserted.

**ROTATION POINT**

The rotation point of the activated objects (components, copper areas, restricted areas and text) is shown when selecting this option. It is symbolized by a cross.

With <LMB>, the rotation point is attached to the crosshair.

If you leave this menu point (e.g. by selecting "Get component") and activate a component in that way, it can be rotated with <MMB>. 90°, 180° and 270° rotations are possible, depending on how often you press <MMB>.

With <LMB>, the new arrangement is fixed.

Legends (texts) are rotated in the following way:

- (1) Get the object to which the legend belongs.
- (2) Select "Modify graphic".
- (3) Choose "Text".
- (4) Select "Rotation point".
- (5) Leave the rotation point in the lower left corner of the object or determine a new rotation point with <LMB>.
- (6) Leave the rotation point by selecting "Get"; get the text block.
- (7) The possible rotations are shown when pressing <MMB> and can be deleted with <LMB>.

**MODIFY PARAMETERS**

With this function, it is possible to determine attributes (parameters) for the PCB objects component, terminal, signal, track, copper and restricted area or change already defined attributes.

When calling this function, you have to get a special object.

You can proceed in two different ways:

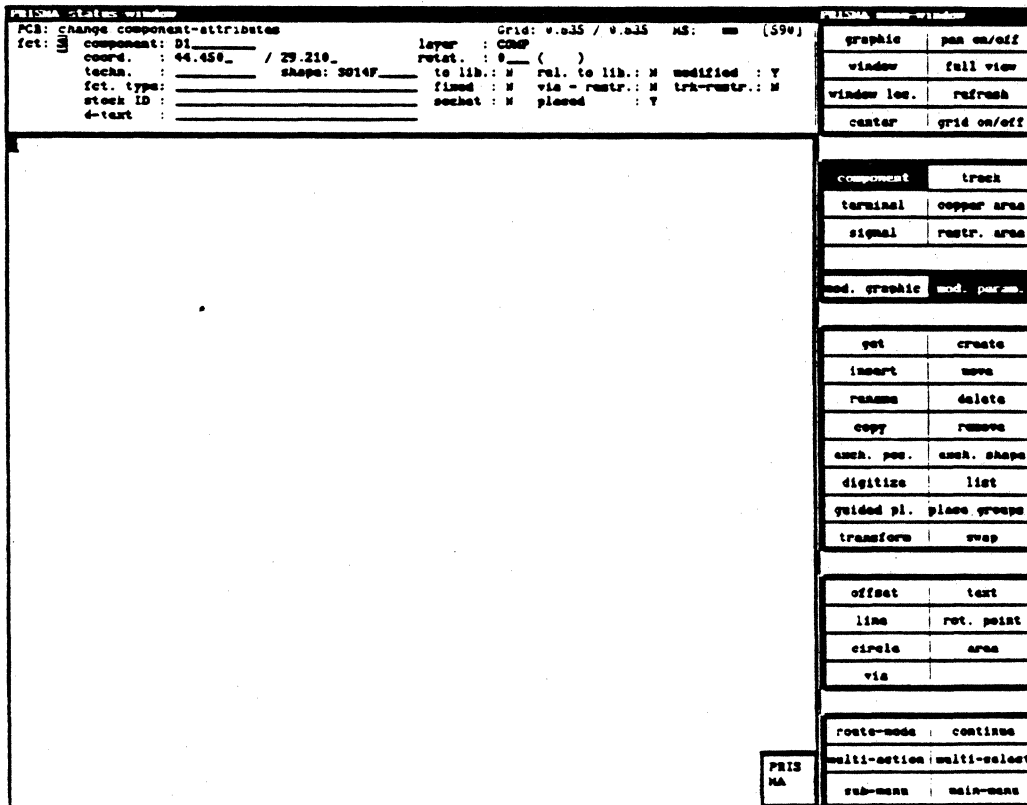
- (1) Select an object and call "Modify parameters".
- (2) Choose an object group (e.g. "Component") without getting a special element in the layout and select "Modify parameters". In the status window, a general modification menu appears in which you have to specify the special object. Then get the element on the layout.

A description of the attributes follows that can be determined for the different object groups with this function.

**MODIFY COMPONENT PARAMETERS**

In the status window, the logical search terms belonging to the component are assigned:

The component can be identified by its name in the layout, by its technology (NMOS; HCMOS, etc.), its shape type, its function type (function specific standard component, e.g. 4 NAND component 74LS00) and a stock ID.



The function "Modify component parameters" is important for the relation of physical and logical component data. Here, the respective component is released.

Procedure for the release:

**1. Release a shape (layout)**

- a. Touch the respective component in the layout submenu "Modification" and activate "mod. param.";

Note: A shape name must be assigned to the component in "Shape" --> during the data transfer, the shape information will be stored in COLUMBINE with this name!

- b. Enter "Y" in "to lib." in the menu "Change component attributes".
- c. Press <RETURN>.

**2. Release a symbol (schematics)**

- a. Get the respective symbol in the schematics submenu "Modification" and activate "mod. param.";
- b. Enter "Y" in "transfer symbol body" and the name with which the symbol body shall be stored in the library in "symbol body" (e.g. BNOR) in the menu "Change symbol attributes";
- c. Press <RETURN>.

**3. Release a related function type (schematics and layout)**

- a. Shape and symbol must be released in the way described in item 1 and 2;
- b. Between the terminals of the symbol and the terminals of the component, a 'relation' must be executed (see below);
- c. On the layout side, you have to assign a function type specification to the component in "fct. type" in the menu "Change component attributes".
- d. On the layout side, you have to enter "Y" in "rel. to lib." in the menu "Change component attributes".
- e. On the schematics side, you have to enter "Y" in "transfer relation" in the menu "Change symbol attributes".

**OTHER PARAMETERS (yes/no fields):**

- To lib.?
- Rel. to lib.?
- Modified?
- Fixed?
- Via-restr.?
- Trk-restr.?
- Socket?
- Placed?

## MODIFY TERMINAL PARAMETERS

The terminal parameters describe the most important technological requirements for a connection point. These are:

- "Fan-In" (factor by which the input current is larger than the current of the ground patter).
- "Fan-Out" (specifies how many inputs of the same ground patter can be driven from the respective output).
- assigned (logical) signal
- terminal group
- cross-section and type of the connected track
- specification if it is a corner connection, e.g. a dual-in-line-shape ("corner pinning").
- symbol representing the terminal (circle, cross, rhomb ...)

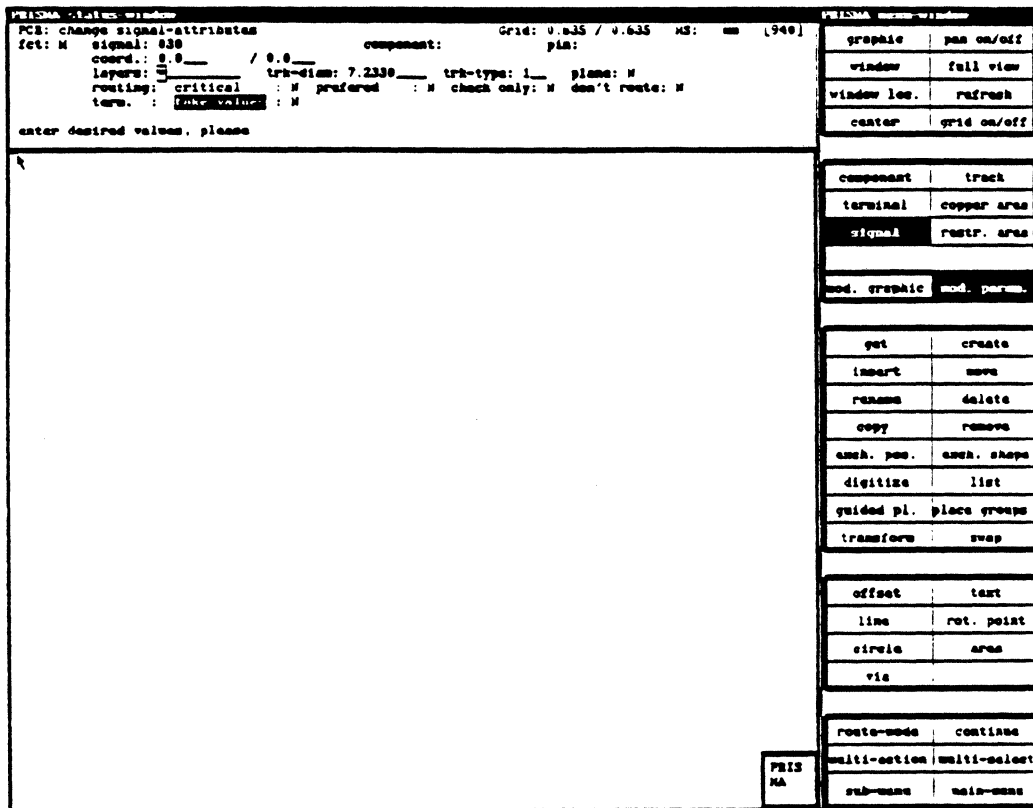
PRISMA - status window		PRISMA - component window	
PCB: modify terminals		Grid: 0.317 / 0.317 x5: mm (840)	
ict.: P pin : GND	layer : 1	component: 2x01	graphic pan on/off
coord : 7.0 / 0.0	rotation: 0	type :	window full view
signal:	sub-sig.:	con-type : (SE)	window loc. refresh
fan-in/out: 0/0	via-lay.:	connect-layers:	center grid on/off
track cross-section: 0.0	-type : 0	pl.-connection: M	
Shelltool / DIALOG			
		component track	
		terminal copper area	
		signal restr. area	
		mod. graphic mod. param.	
		get create	
		insert move	
		rename delete	
		copy remove	
		amsh. pos. amsh. shape	
		digitize list	
		guided pl. place groups	
		transform swap	
		offset text	
		line rot. point	
		circle area	
via			
route-mode continue			
PRIS multi-action multi-select			
MA sub-menu main-menu			



## MODIFY SIGNAL PARAM.

Signal attributes also refer to a later simulation and the routing process. In the status window, the following is agreed:

- layers on which the signal exists
- necessary power cross-section
- power type
- route status (critical, preferred, don't route, check only)



### Status window

In the status window, you have the possibility to assign new attributes to the defined restricted areas.

They refer especially to the position of the restricted areas and the objects to be blocked.

Parameters for the position:

- Coordinate of a point on the restricted area
- Layer
- Component under which routing is not allowed
- Parameter for the selective blocking of special objects
- Segments shall not be placed within the restricted area
- Vias must not be placed within the restricted area
- Components must not be placed within the restricted area

Note: It is also possible to define the area outside the opened window as restricted area. Therefore, you have to enter <Y> in the field "outside" and press <RETURN>.

### **COPPER AREA MOD. PARAM.**

The parameters to be specified here refer to the grid of the copper areas. Specifications concerning the grid size must be entered.

### **RESTR. AREA MOD. PARAM.**

Parameters for the selective blocking of special objects:

- Segments shall not be placed within the restricted area
- Vias must not be placed within the restricted area
- Components must not be placed within the restricted area

Note: It is also possible to define the area outside the opened window as restricted area. Therefore, you have to enter <Y> in the field "outside" and press <RETURN>.

**ROUTE-MODE**

is necessary for the digitizing of tracks and is always active there.

**CONTINUE**

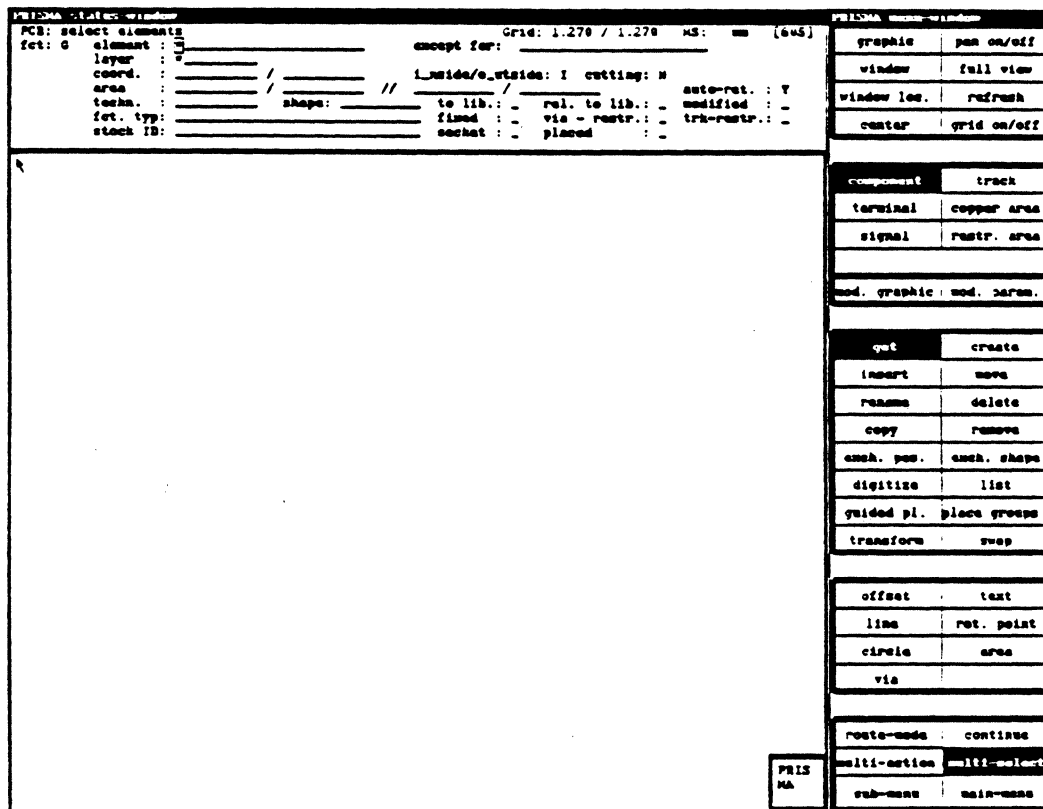
allows the multiple usage of a function (return to "Get").

**MULTI-ACTION**

If several elements have been taken up with multi select, they can be modified after clicking multi action.

**MULTI-SELECT**

With this function, you can get several elements.



In the status window, you have four possibilities to select elements:

- Element:** enter the name of the element
- Layer:** enter the respective layer
- Coord.:** enter the coordinates of the component (the coordinate display can be adjusted in the menu "Graphic setup").

**Area:** enter the coordinates of the diagonal corners of an area with the keyboard. According to the entry, the elements within or outside the areas are activated. Instead of entering the coordinates with the keyboard, the area can be activated by clicking the diagonal corners.

**Except for:** In this field, elements can be defined that shall be excluded from the defined set.

**i\_ nside/  
o\_ utside:** Here, you can determine if the elements are activated inside or outside the defined area.

**Cutting:** In this field, you can determine if elements that lie only partly in the defined area shall also be taken up.

**\*:** The asterisk stands for "all".

### SUBMENU

You get to the submenu.

### MAIN MENU

You return to the main menu.

12. GRAPHIC SETUP

The screenshot displays the PRISMA software interface. At the top, a menu titled 'modification' is visible, listing various options: (B) modification, (A) misc., (T) technology, (R) change file, (G) graphic setup, (M) modes, (F) final check, (U) update file, (C) CAM, (Y) status, (S) schematics, and (X) exit. The 'graphic setup' option is highlighted. Below the menu, a terminal window titled 'shelltool - /bin/csh' is open, showing a blank command prompt. In the bottom right corner, a small box contains the text 'PRISMA'.

### WORK MENU

Return to the menu "Modification"

### MAIN MENU

Return to the main menu

### WINDOW

Serves for defining a new window.

How to proceed: Move the crosshair to the place where you want to define the lower left edge of the window. Fix the edge with <LMB>. Then you may expand the window to the required size. Fix the final window size with <LMB>.

### FULL VIEW

Displays the whole page again.

### WINDOW LOC.

By selecting "Window loc.", the current window is represented by a symbol in relation to the total size of the work sheet. Here, you may choose a new window directly: move the crosshair to the place where the lower left edge of the window is to be defined. Fix this edge with <LMB>. You can then open the window with the mouse until it has the desired size. The final size of the window is determined by pressing <LMB>.

### REFRESH

With this function, the screen is cleared and you get a new artwork.

### CENTER

Sometimes, it is necessary to keep the current scale but to represent a slightly different window.

When selecting this function, the crosshair position becomes the new center of the picture. Note that it is not possible to move the window over the left edge of the sheet (the lower left corner is the coordinate origin).

### GRID ON/OFF

With this function, the grid may be switched on/off. It has no influence on the effectiveness of the grid. The grid may be represented with points or lines. This can be selected in "Graphic/Color setup" in the status window.

If the grid is too fine, it will be suppressed because otherwise the screen would be white.

If the grid is not needed for the placement, it should be switched off since it has to be drawn anew for each picture; this delays the artwork.

Some of the optical popup functions of the menu "Change Graphic" may only be activated in the status window.

These popup functions serve for the following:

- (1) color representation (e.g. selection of a default)
- (2) Selection of the elements to be displayed: terminals, vias, frame, auxiliary grid, the different display modes of components, etc.
- (3) Selection of the layer to be represented (of a multilayer PC board)
- (4) Selection of the window (definition of windows by entering the coordinates of the lower left and the upper right corner)

## **RESTORE**

Works similar to "Refresh", the difference is that the window contents is not newly calculated but newly written by means of the graphic memory (is faster).

## **ZOOM**

As known from the photo optics, you can determine a part of the picture here that shall be zoomed. With the crosshair, you define the section to be enlarged (with <LMB>) or reduced (with <MMB>) by a constant factor. With <RMB>, you can interrupt the procedure.

## **THE GRIDS**

The grid is an orientation help and facilitates the work at the screen. If a component is to be placed, the crosshair moves to the nearest cross point. The distance between the cross points may be defined individually. In the status window, 10 different standard settings with the respective numbers are available that can be defined with the keyboard. If you want to use one of these defaults, then click the respective number in the menu window. If none of the defaults is convenient for you, then you may either redefine a default or specify the clearance "manually" with the keyboard.

The status window shows the current values for the different grids. The measurements assigned to the values (inch or mm) may be defined in the menu "Modes".

The current measurement is displayed in the status window under "MS". (When changing the units, the system performs the change automatically).

The system offers four different grid types that facilitate your work, especially the digitizing and placement. You can define 10 standard values for the grids which may be stored in the status window with a respective number (0-9). New standard values can be defined by overwriting the old value and pressing <RETURN>.

The grid types have the following meaning:

### **Temp-grid**

This serves for specifying a temporary grid, which will be activated until the kind of object (component, signal, etc.) is changed.

### **Disp-grid**

Specifies the resolution of the grid that can be activated with "Grid on/off". In difference to the SCS side, this grid does not depend on the digit-grid.

### **Digit-grid**

This is the resolution with which the system may route tracks. The crosshair moves according to this resolution.

### **Place-grid**

Specifies the grid which is used for placing objects.

### **Digits 0-9**

For some functions, e.g. grid specifications and color on/off, parameters are coded with digits in the status window. In order to change these parameters, click the respective number in the menu window and the values change according to the code.

### **nb.-start**

This function has not yet been implemented.

### **nb.-stop**

This function has not yet been implemented.

### **Color on**

This function has not yet been implemented.

### **Color off**

This function has not yet been implemented.

### **Color Clear**

This function has not yet been implemented.

### **Layer on**

This function has not yet been implemented.

### **Layer off**

This function has not yet been implemented.

### **Display mode**

This function has not yet been implemented.

### **Via mode**

This function has not yet been implemented.



**0/0**

This function serves for defining the coordinate reference point. The lower left edge is the reference point, if not defined otherwise.

**Coos on/off**

After activating this function, the x and y coordinates of the crosshair are shown in the status window. The display is finished when clicking this function again.

**Color**

ref. to page 8.

**Def. Colors**

This function has not yet been implemented.

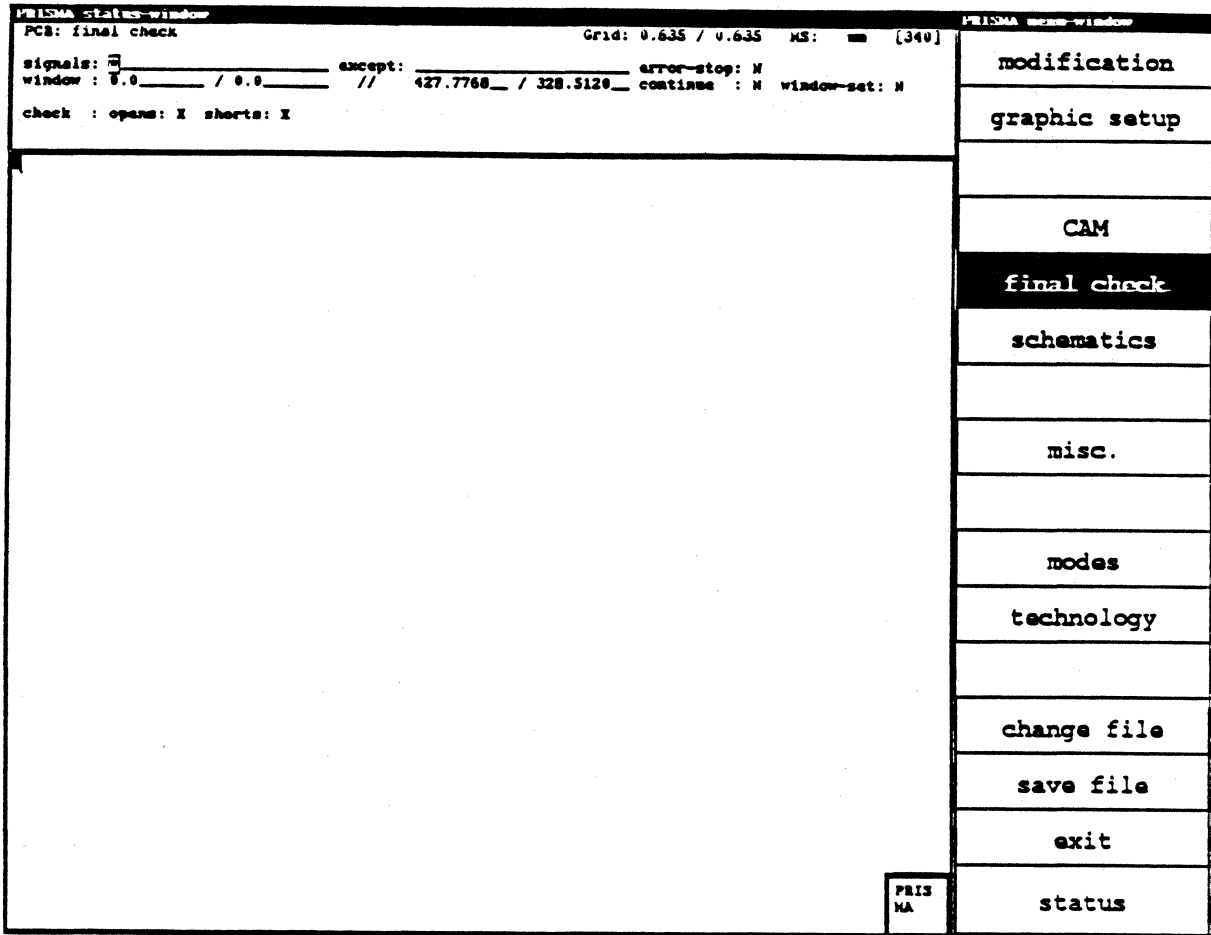
**Work menu**

returns to the superior menu.

**Main menu**

returns to the main menu.

## 13. FINAL CHECK



This menu checks tracks in the layout for interrupts and shortcuts.

The check is performed selectively for certain signals, which are specified with their name, in the area required (rectangle defined by its lower left and upper right edge). You may as well exclude signals from the check. Various system reactions can be defined:

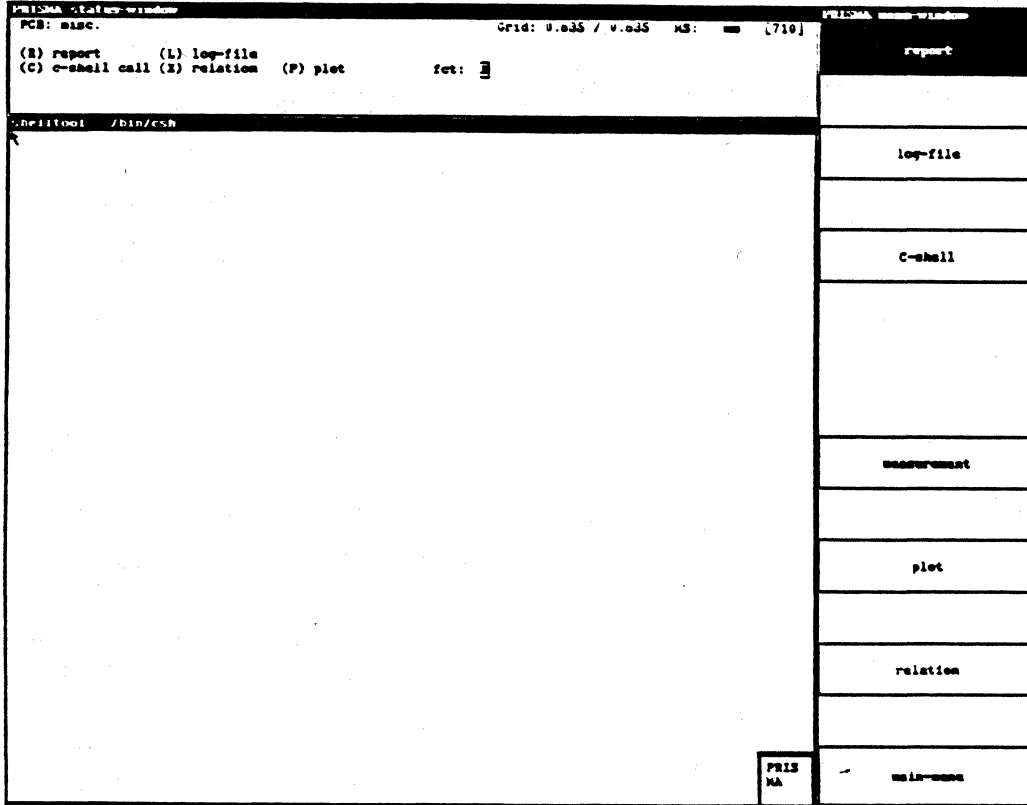
- Stop after each error
- Termination of the check and display of the errors found
- Continue check with <RETURN> after error stop
- Position window on the shortcut (only possible in combination with "error stop: Y".

14. MISCELLANEOUS

PRISMA -- Layout window		Grid: 0.317 / 0.317 MS: mm [560]		PRISMA menu window	
PCB: functions		(G) graphic setup	(C) CAM	modification	
(B) modification	(A) misc.	(M) modes	(Y) status	graphic setup	
(T) technology	(E) change file	(F) final check	(S) schematics	CAM	
		(U) update file	(X) exit	final check	
set: <input type="checkbox"/>				.schematics	
shelltool - /bin/csh				TOOLS	
				misc.	
				modes	
				technology	
				change file	
				save file	
				exit	
				status	
				PRISMA	

REPORT

After calling this function, all used component shapes, logic gates etc. are listed.



LOG-FILE

This function has not yet been implemented.

C-SHELL

After selecting this function, the window "shelltool-/bin/csh" appears. If you move the crosshair to this window, single files may be deleted or edited. By clicking the upper frame of the window with <LMB>, it disappears again.

MEASUREMENT

With this function, you can create measurements - including legend - for components or for the PC board (no specification in "Component").

The use of the popup menu is recommendable.

## PLOT

PRISMA - layout window				Grid: 0.035 / 0.035 MS: mm [510]		PRISMA menu window	
PCB: hardcopy						report	
name of plotfile wfile.p0		scale		1.00000			
name of plotter		window		0.0 0.0			
				556.1996 427.0647			
Comp Airl COMP SOLD						log-file	
pen 1_ 1_ 1_ 1_						C-shell	
						measurement	
						plot	
						relation	
						main-menu	

**NAME OF PLOT FILE:**

Here, the name of the current file is displayed.

**SCALE:**

In this field, you may define the scale with which the hardcopy is to be represented.

**WINDOW:**

In these fields you may define the window which is to be plotted. Instead of specifying the window with the keyboard, the lower left edge and the upper right edge may be defined with <MMB>. <RETURN> generates the plot file. If you made any errors during the generation of the window, you may delete them with <RMB>.

**COMPONENTS, AIRL., COMP, SOLD:**

In these fields, plotter pens may be assigned to the single elements or layers (L1 = layer no. 1 etc).

**RELATION**

This function has not yet been implemented.

**MAIN MENU:**

returns to the main menu

## 15. MODES

PRISMA status-window	PRISMA menu-window
<pre> PCB: global parameters Grid: 0.635 / 0.635 MS: == [430]  units : (I)inch (M)mm (C)=1/20" : <input checked="" type="checkbox"/> searchrange+ : x: 2.2679_ y: 2.2679_ cursor: elements plus crossh.: _ plus terminals: _ digit.-cursor plus crosshair: _ text : default height: 8.0_ automat. width: _ areas : auto - close : _  digit.: auto - routing: _ gradients : 1 (0=I/V only; 1=I/V+1:1; 2=no restr.) misc. : STI : X popups : _ lower case: _  standard-panic-mode (P,W): W auto-panning-size: _ or fullsize: X panning-area (pixels) in X: 1152 (min=1152 max=3456) in Y: 900_ (min=900 max=2700) preference-value: 900_ pixels be adequate to 304.80_ ==                     </pre>	<ul style="list-style-type: none"> <li>modification</li> <li>graphic setup</li> <li>CAM</li> <li>final check</li> <li>schematics</li> <li>misc.</li> <li><b>modes</b></li> <li>technology</li> <li>change file</li> <li>save file</li> <li>exit</li> <li>status</li> </ul>
<div style="border: 1px solid black; height: 200px; width: 100%;"></div>	<div style="border: 1px solid black; padding: 2px; display: inline-block;">PRISMA</div>

Modes are determined in the status window.

**UNITS:**

Here, the unit may be defined with which the system works.

**SEARCH RANGE:**

Area around the crosshair in which you may search for objects.

**ELEMENTS PLUS CROSSHAIR, PLUS TERMINALS, DIGIT.-CURSOR PLUS CROSSHAIR:**

Adjustment of the representation form of activated objects.

**TEXT: DEFAULT HEIGHT, AUTOMAT. WIDTH:**

Default for generation of texts.

**AREAS: AUTO-CLOSE:**

If you activate this function and digitize areas, the digitized line course is closed automatically (between starting and final point) when activating the menu function "STOP".

**AUTO-ROUTING:**

Default for the router mode in the work menu

**LOWER CASE:**

When specifying "Y", lower case may be used for generating or renaming objects.

**GRADIENTS:**

Presetting for the gradient that is allowed during the digitizing (can be changed in the respective submenu).

**RTI:**

This is to switch off Real-Time-Integration. NOTE: With RTI switched off, irreversible inconsistencies may occur between schematics and layout.

**POPUVS:**

In this field you define whether <RMB> switches on popup menus or whether <RMB> is to execute another function. This switch is also possible with <R11>.

**STANDARD-GRAPHIC-MODE:**

W=window or P=pan

**PANNING-AREA:**

specification of the panning-area in pixels. For this, standard graphic mode P has to be specified.

## **LAYOUT**/Modes

---

### **AUTO-PANNING-SIZE:**

The graphic window is not displayed on the complete screen (the menu windows remain).

### **FULLSIZE:**

The graphic window is displayed on the complete screen (covers the menu windows).

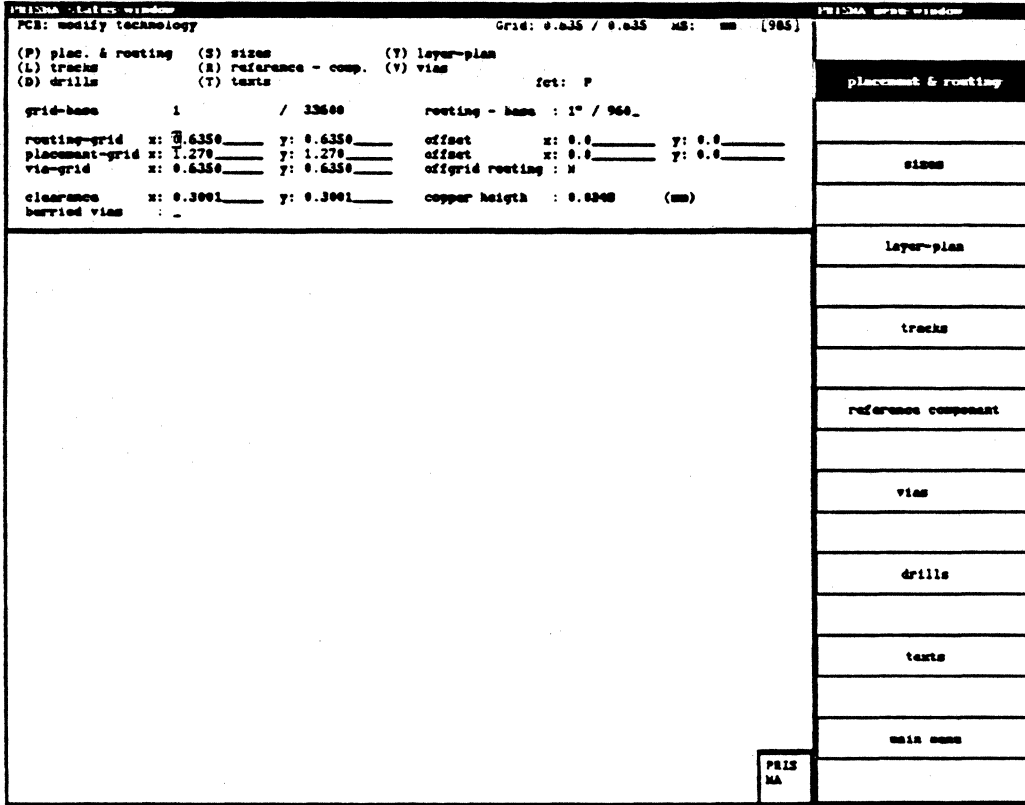


16. TECHNOLOGY

PRISMA -- Layout-window PCB: functions                      (C) graphic setup                      (C) CAM                      Grid: 0.317 / 0.317                      MS:                      [569]		PRISMA menu-window modification. graphic setup CAM final check schematics TOOLS misc. modes technology change file save file exit status
(B) modification                      (M) modes                      (Y) status (A) misc.                      (F) final check                      (S) schematics (T) technology                      (U) update file                      (X) exit                      set:		
shelltool - /bin/csh		
[Empty shell window]		
[Empty shell window]		
[Empty shell window]		
[Empty shell window]		
[Empty shell window]		
[Empty shell window]		
[Empty shell window]		
[Empty shell window]		
[Empty shell window]		
PRISMA		

PLACEMENT & ROUTING

When activating this function in the menu "Technology", a submenu appears in the status window in which you can specify the grid (for placement and routing), the clearance, etc.



**GRID-BASE:**

Specifies the minimum unit for the system internal calculation. This specification is only displayed for your information.

**ROUTING-BASE:**

The routing base is the smallest coordinate distance considered by the router. This unit may be changed.

The lowest routing base is 1/6000 inch, the highest 1/100 inch. 1/960 inch is the standard routing base.

The routing grid must always be an integer multiple of the routing base and the multiplier (M) must fulfill the following condition:

$$2 \quad M \quad 50$$

If this is not the case, the specified routing base value will automatically be rounded off so that this condition is fulfilled.

**Example:**

Routing base: 1/960"; routing grid: 1/70";  
 i.e.  $M = 1/70 : 1/960 = 13.7$   
 Change: routing base: 1/980" ( $M = 1/70 : 1/980 = 14$ ).

**Note:**

- The maximum size of the work sheet (measurement unit in X and Y direction) which can be constructed with CIS depends on the routing base and can be determined exactly by multiplying the routing base by 32767.

**Examples:**

1/960 inch resolution --> 32767 X 1/960" 34" 866 mm  
 1/100 inch resolution --> 32767 X 1/100" 327" 7320 mm  
 1/6000 inch resolution --> 32767 X 1/6000" 5.5" 138 mm

**ROUTING-GRID:**

If no shifting exists, the tracks are routed along the grid lines and the vias are placed on the grid points.

The routing grid must always be an integer multiple of the routing base. If this is not the case, the routing base will internally be changed into a suitable value that fulfills this condition.

According to technology and coordinate unit, the grid unit to be used by the router is automatically entered in "Grid" (e.g. "0.635" for 1/40" technology and unit mm).

The table shows the grid values for the standard Calay routing grids.

unit grid	mm	in.
1/20"	1.27	0.050
1/40"	0.635	0.025
1/50"	0.508	0.020
1/60"	0.423	0.0166
1/70"	0.362	0.0142
1/80"	0.317	0.0125

**VIA-GRID:**

By entering the grid element spacing (in the respective unit --> ref. to the function "Modes"), you can specify the distance between the vias.

If you have a routing grid of x: 0.635 y: 0.635 and enter x: 1.27 y: 1.27 for the via grid, the router can at most place a via on each 2. grid (so you still have the possibility to digitize a track between the vias).

If the specifications in "Routing grid" and "Via grid" are equal, a via can be placed on each neighboring routing grid.

Attention:

If the via grid does not equal the routing grid, it must be an integer multiple of it!

**PLACEMENT-GRID:**

Since routing is bound to the grid, components should be placed on the routing grid or an integer multiple of the routing grid.

A disadvantageous choice of the routing grid may lead to a bad routing result since pins may be placed off-grid. Off-grid pins (i.e. pins which are not placed on a grid point) may effect that no tracks can be digitized between them.

The most sensible grid for the placement of components is 1/10" as it is an integer multiple of all standard routing grids.

sensible grid for the placement							
routing grid	1/10"	1/20"	1/40"	1/50"	1/60"	1/70"	1/80"
1/20"	X	X					
1/40"	X	X	X				
1/50"	X			X			
1/60"	X	X			X		
1/70"	X					X	
1/80"	X	X	X				X

In order to get an optimal bus routing, take care that the components of memory structures do not lie more than 0.3 inches (= 7.62mm) apart from each other.

**OFFSET:**

By entering the respective values, you can define a shifting of the placement grid

This is sensible if elements that are 'on-grid' in the routing grid lie 'off-grid' in the placement grid.

The shifting of the routing grid is not yet possible.

#### OFFGRID ROUTING: Y/N

When entering "Y", the complete routing area with all elements will be shifted half a routing grid in the negative X and Y direction.

This is necessary for the 1/60" routing in order to route 2 tracks of type 1 between 2 IC-pins with a maximum diameter of 1.53 mm.

#### CLEARANCE:

Here you specify the clearance between the PC board elements (tracks, pins, vias, text fields, etc.) and to the routing area and PC board edge.

The clearance, optimized in the standard technologies, can be changed.

Notes:

- In order that segments of the track type with the smallest track width can be routed on each grid line, you have to take care that  
clearance + smallest track width    routing grid
- A subsequent reduction of the initial clearance can effect short circuit messages when executing a technology check afterwards!
- For the single track and via types, an **additional clearance** can be specified in the technology charts that will be added to the general clearance when routing this track or via type.

#### COPPER HEIGHT:

On the base of the copper height and the track width, the track diameter is calculated (ref. to "SCS: Signals & connections"). This parameter is relevant to the simulation.

At present, the standard value 0.0348mm appears here; changes of this specification are not evaluated.

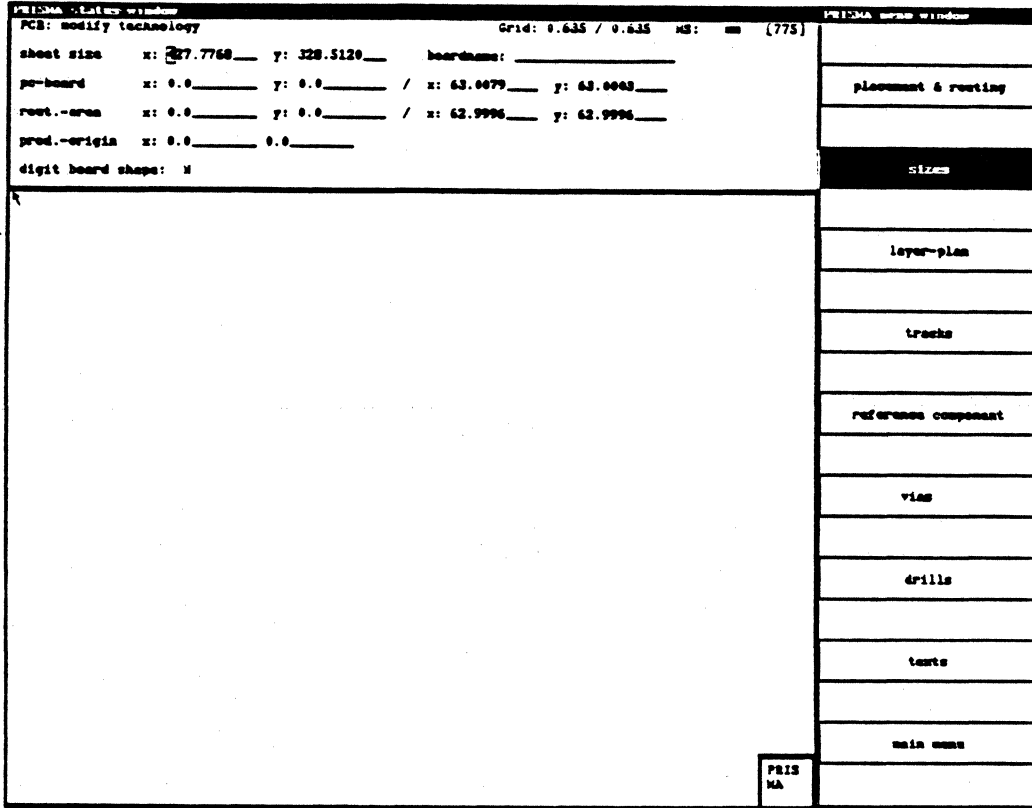
#### BURIED VIAS:

This specification only applies to the interactive digitizing of tracks, not to the routing. By putting a cross in this field, you specify that buried vias shall be used for the digitizing.

## SIZES

The size specifications are entered in the status window.

Reference point for all coordinate specifications in this menu is the zero point of the sheet.



### SHEET SIZE:

When specifying the work sheet sizes, the maximum size considered by the router must not be exceeded.

The maximum size of the work sheet in x and y direction depends on the routing base and can exactly be determined by multiplying the routing base by the number 32767.

### BOARD NAME:

Here, you can enter a user text for characterizing the current PC board that will always appear in this field when calling this file again.

### PC BOARD:

By specifying the coordinates of the origin and the upper right corner of the board, you determine the PC board size here.

**ROUT.-AREA:**

Here, you specify the routing area by entering the coordinates of the origin and the upper right corner of the routing area.

The maximum size of the routing area in x and y direction depends on the routing grid and the form of the routing area:

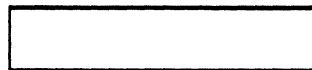
- square area



y = 2048 grid cells

x = 2048 grid cells

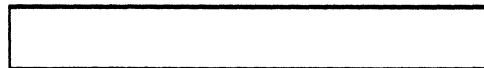
- rectangular area



y = 1024 grid cells

x = 4096 grid cells

- long rectangular area



y = 1024 grid cells

x = 8192 grid cells

**Attention:**

Components must not be placed outside the routing area, otherwise the file cannot be routed (the program is aborted with the message "element outside area").

**PROD.-ORIGIN:**

Coordinate specification for the production zero point.

This point is the **coordinate origin for the postprocessing** and for the coordinate display (when using the graphic menu function "Coos on/off").

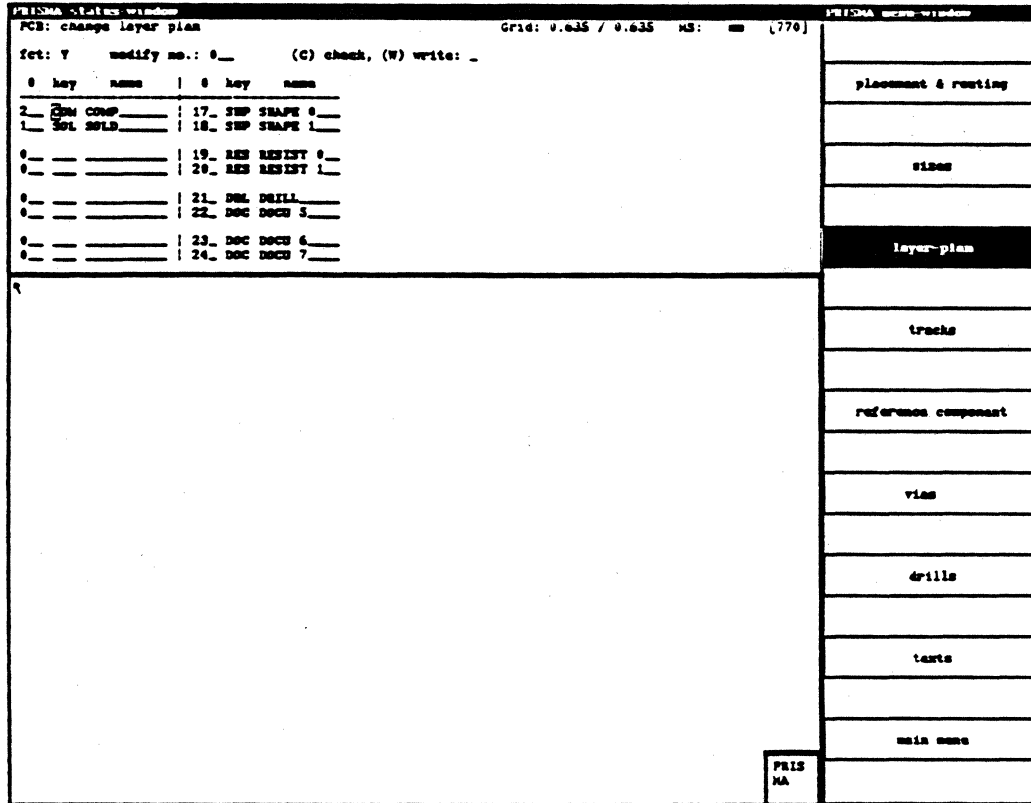
**DIGIT BOARD SHAPE:**

When entering "N", <CR>, the system automatically creates a rectangular board shape according to your coordinate specifications.

When entering "Y", <CR>, the function "mod. graphic - digitize" of the work menu is called automatically and you can freely digitize the board shape.

## LAYER PLAN

When activating this function, the layer plan for the current board appears in the status window.



When creating a new .adb-file, the layer plan for a two layer board appears as the default.

With the respective entries, you can change the layer plan as desired.

From the 255 possible layers, only the first 32 can be considered at present.

#

The layer number is entered here.



**KEY**

Here, the respective specification of the signal or documentation layer is entered.

The specifications that may be used as "key" are determined:

COM --> component side  
SIG --> signal inner layer  
SOL --> solder side  
PLN --> plane  
FRE --> free layer  
SHP --> silk screen documentation layer  
RES --> solder plan documentation layer  
DRL --> drill plan documentation layer  
DOK --> any documentation layer

**NAME**

A user comment can be entered here (e.g. an exact specification of the layer).

**HANDLING**

1. Enter the number of the line (1 to 31) in which you want to make changes in "modify no.", <CR>. The cursor moves to the first input field of the selected row.
2. Now the changes can be made, <CR>.
3. By entering "W" for "write" and pressing <CR>, the entries are stored.
4. With <ESC><ESC>, you leave the menu.

### Effect of the layer plan constellation on the multilayer routing

The router only processes signal layers. The layers specified with "empty" or "plane" in the layer plan will not be considered by the router (and not counted).

If the layer plan of a multilayer contains empty layers and/or planes, the router usually combines the signal layers differently than defined in the layer pair combination of the layer plan.

**Rules for the router internal combination of the layers:**

1. The router starts the routing process on the component side (= 'upper' layer) and continues towards the solder side (= 'lower' layer). The layers are combined to pairs. If the router finds a constellation for which pairing is not possible, it routes the single layer.
2. The component side (= layer 2 in the layer plan) will always be routed first regardless of the other layer the router combines it with.
3. Layer 2 and 1 (= component and solder side) will only be routed together if the layers below are specified as "empty" or "plane".

**In SMD routing**, this constellation is the condition for generating vias for changing layers (track segment + via at SMD pin) in the first layer pair iteration.

If this is not the case, the router combines layer 1 with another layer and does not route them in the first layer pair iteration; correspondingly, no vias for changing layers are created for the SMD pins placed on the solder side.

4. If both layers of the layer pair are specified as signal layers in the layer plan, they are always routed together.
5. If the router finds a layer pair in which one layer has been specified as "empty" or "plane" and the other layer as signal layer, the signal layer is routed alone if the layer pair below does not consist of an "empty" or "plane" layer plus a signal layer.

(This does not apply to the layer pair component/solder side if point 3 is fulfilled).

6. If the router finds two successive layer pairs with a signal layer and an "empty" or "plane" layer when moving from component to solder side, it combines the two signal layers of the two layer pairs.

## CHANGE TRACK TECHNOLOGY

After activating the function "Tracks" in the menu "Technology", a chart with the track technology of the current .adb-file appears in the status window.

When creating a new .adb-file, the chart contains the values stored in the standard technology.

PRISMA - Status window							PRISMA - menu window		
PCB: modify track-technology							Grid: 0.25 / 0.25 MS: mm (96)		
set: L modify no.: 1 (C) check, (W) write: _									
no.	trackwidth	resist	add-clear.	attribute	via				
	overall	core	overall	core			placement & routing		
1	3.001	0.0	0.0	0.0	0.0	_SAD	1		
2	0.602	0.0	0.0	0.0	0.0	_SA	1	size	
3	1.001	0.0	0.0	0.0	0.0	_SA	1		
4	1.499	0.0	0.0	0.0	0.0	_SA	1	layer-plan	
attributes: S=shifted R=routable A=active D=default									tracks
								reference component	
								via	
								drills	
								tests	
								via mask	

From the 255 track types in the menu, only the first 32 may be considered at present.

The widths of the 32 track types, the via types assigned to them and the solder resist mask widths of the standard technology may be changed if necessary.

Max. 16 of these track types can be defined as routable.

Furthermore, you can specify for each of the 32 track types if it shall be shifted half a grid in the negative X and Y direction.

As tracks are also used for the presentation of SMD pins, the track ends and the respective solder resist masks can be defined in different shapes.

NO.

Track type to which the values and parameters in this line apply.

## TOTAL TRACK WIDTH

Defines the real width of the tracks.

The router will use this size as base for the internal display of the track width.

### a. Routable track types

The widths of the max. 16 routable track types are optimized for the router in the standard technologies (1/20", 1/40", 1/50", 1/60", 1/70", 1/80", ref. to the chapter "Standard technologies").

For a free definition of the routable track types, the following restrictions must be considered:

#### 1) Minimum track width

- for non-shifted tracks: 0
- for shifted tracks : 1 x routing grid

#### 2) If

the widest non-shifted track is larger than  
(2 x routing grid) minus (clearance)  
then  
the smallest non-shifted track must be wider than  
(0.83 x routing grid) minus (clearance).

### b. Non-routable track types

The non-routable track types can only be digitized interactively. You may define non-routable track types only within the types 1-31.

## TRACK WIDTH RECTANGLE

The value you enter here determines the width of the core, i.e. the 'rectangle part' of the track segments - and with it the shape of the track ends (in order to display different kinds of SMD-pins).

The following shapes are possible:

standard-SMD-pin

rectangle-pin

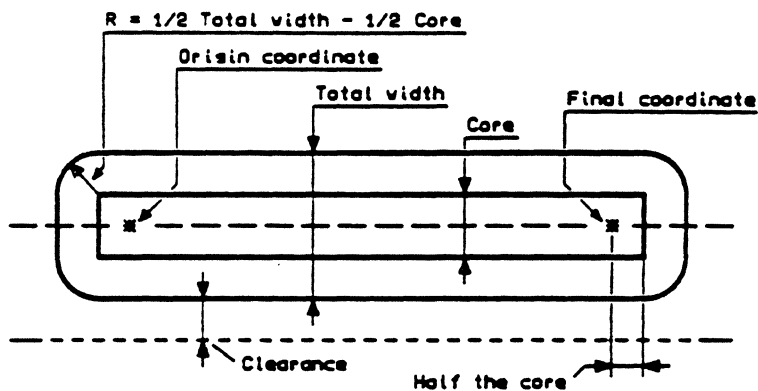
rounded-rectangle-pin



The specifications have the following meaning:

- No entry --> standard-SMD-pin (the radius of the semi-circles representing the track ends = 1/2 total track width).
- Same value as in "Total" --> rectangular SMD-pin (consists only of the core)
- A value > 0 and < the value in "Total" --> rounded-rectangle-pin

The following diagram illustrates the described sizes:



### TOTAL SOLDER RESIST WIDTH

Here, you define the width of the solder resist mask for tracks used for the presentation of SMD-pins.

### RECTANGLE SOLDER RESIST WIDTH

Here, you define the shape of the solder resist mask for tracks used for the presentation of SMD-pins (ref. to "Track width rectangle").

### CLEARANCE:

If you specify a clearance here, it will be added to the general clearance (ref. to "Placement & routing") when routing this track type.

### ATTRIBUTES:

#### Shifted

By entering "S", this track type will be shifted half a routing grid in the negative X and Y direction.

#### Routable

By entering "R", you specify that the respective track type may be routed (max. 16 types!) and will be used as symbol for the router-internal presentation of the pins, vias and non-routable tracks.

#### Active

By entering "A", you specify that the track type may be digitized and routed. The release for the router requires also the specification "R" (= routable).

#### Standard

By entering the attribute "D", the track type with the smallest routable width is defined.  
The attribute "D" must be assigned to a track type but only to one.

### VIA

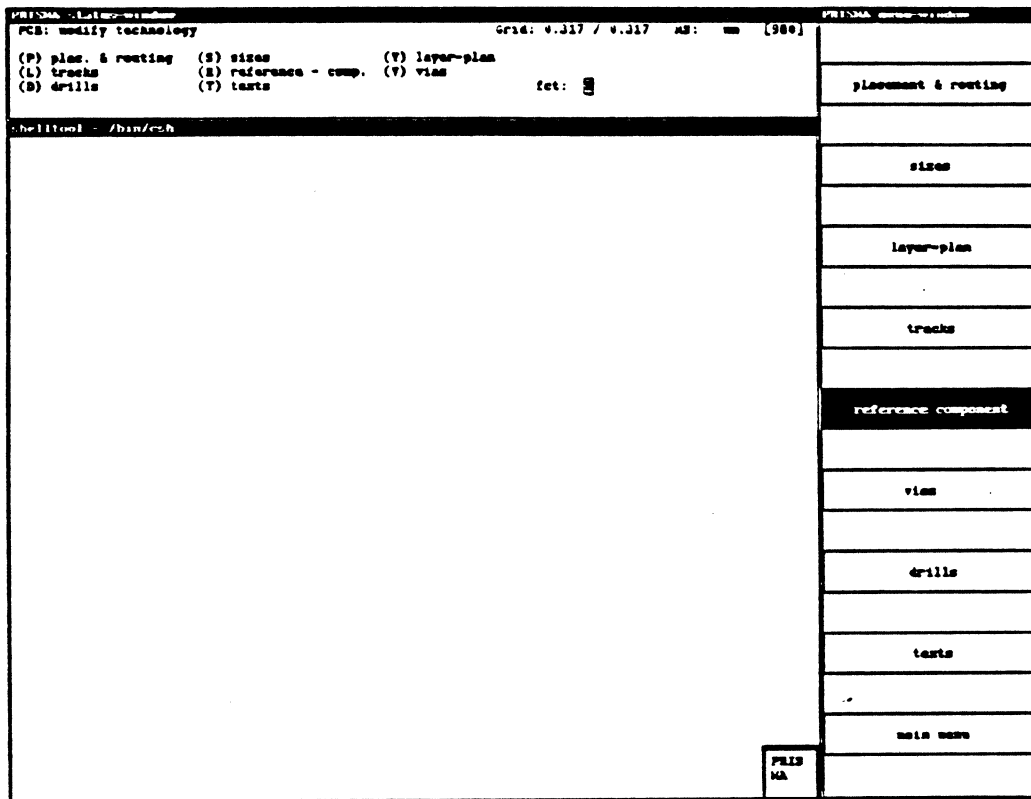
The entered via type (defined in the menu "Change via technology") will be used for changing layers when routing the track type defined in this line.

### HANDLING

1. Enter the number of the line (1 to 31) in which you want to make changes in "modify no.", <CR>. The cursor moves to the first input field of the selected row.
2. Now the changes can be made, <CR>.
3. By entering "W" for "write" and pressing <CR>, the entries are stored.
4. With <ESC><ESC>, you leave the menu.

## REFERENCE COMPONENT

Here, it is possible to pre-define often used terminals. When exchanging terminals, it is possible to use terminals of the reference component as master terminals.



How to proceed:

- (1) Create terminal in reference component (proceed like in "Create terminal").
- (2) Specify "!PCBREFGRP" as component in the status window and the name of the terminal as pin. Select then function "A" or press the <RETURN> key, if the menu field "get" is activated. Then, the system activates "exchange" by itself.
- (3) Click the terminal to be replaced by the reference terminal with <LMB>.

## VIAS

After activating the function "Vias" in the menu "Technology", a chart with the via technology of the current .adb-file appears in the status window.

When creating a new .adb-file, the chart contains the values stored in the standard technology.

PRISMA status window										PRISMA menu-window	
PCB: modify via-technology										Grid: 0.635 / 0.635	MS: mm (907)
set: v modify no.: 0 (C) check, (W) write: _											
no.	via-size overall	via-size core	resist overall	resist core	add-clear	attribute	drl			placement & routing	
1	2.0001	0.0	1.4499	0.0	0.0	3D	1				
2	2.0002	2.0002	2.5997	2.5997	0.0	0	2			sizes	
3	0.2132	0.0	0.5506	0.0	0.0	0	0			layer-plan	
4	0.2132	0.0	0.5506	0.0	0.0	0	0			tracks	
attributes: S=shifted B=routable D=on-every-layer all sizes in mm											
<div style="border: 1px solid black; width: 100%; height: 100%;"></div>										reference component	
										<b>vias</b>	
										drills	
										texts	
										via sizes	

From the 255 via types in the menu, only the first 32 may be considered at present.

The sizes of the 32 via types and their solder resist masks can be changed, if necessary.

Max. 16 of these via types can be defined as routable.

Furthermore, you can specify for each of the 32 via types if it shall be shifted half a grid in the negative X and Y direction and (for multilayers) if it shall go through all layers (drilled through) or only through one layer pair.

With the respective entries in "Total via size" and "Size rectangle", the shape of the vias and the respective solder resist masks can be defined.



NO.

Via type (pin type) to which the values and parameters in this line apply.

In the standard technologies, the via types 1 and 2 are used for changing layers, the types 17 and 18 are used for pins.

Via type 0 is reserved for the router-internal display of technologically false vias (pseudo vias).

### TOTAL VIA SIZE

Specifies the total diameter of the via; this size will internally be used by the router.

### VIA SIZE RECTANGLE

The value you enter here corresponds to the width of the core. It determines the side length, i.e. the shape of the vias.

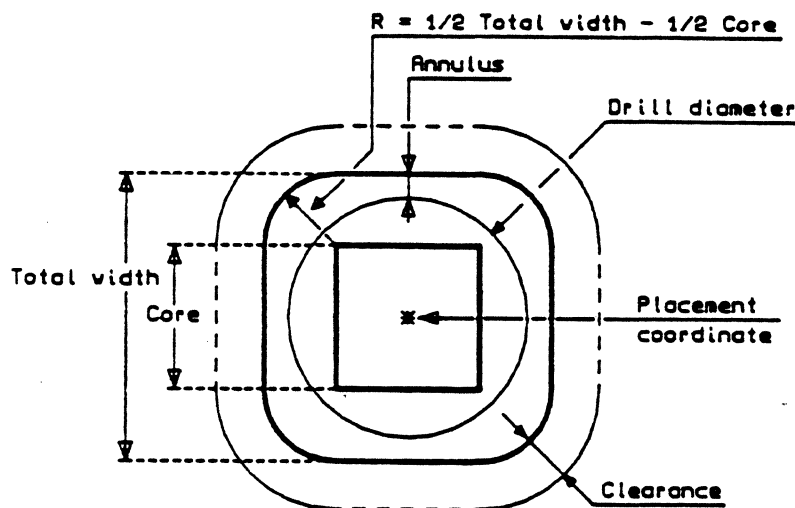
The specifications have the following meaning:

No entry --> round via (pin)

Same value as  
in "Total" --> square via (pin)

A value > 0 and  
< the value in "Total" --> square via with rounded angles

The following diagram illustrates the described terms:



### TOTAL SOLDER RESIST SIZE

Here, you define the size of the solder resist mask for this via type.

### SOLDER RESIST SIZE RECTANGLE

Here, you define the side length, i.e. the shape of the solder resist mask (ref. to "Via size rectangle").

### CLEARANCE:

If you specify a clearance here, it will be added to the general clearance (ref. to "Placement & routing") when routing this via type.

### ATTRIBUTES:

#### Shifted

By entering "S", this via type will be shifted half a routing grid in the negative X and Y direction.

#### Routable

By entering "R", you specify that the respective via type may be routed (up to 16 routable via types are allowed!). If this attribute is not specified, the via type can only be digitized interactively and/or may be used for the pin presentation, but cannot be routed.

#### Through-connected

With the attribute "D", you can determine if the via type shall go through all layers (for pins and stubs from the solder to the component side) or only through one layer pair (=buried; for multilayers).

#### Drill

The entered drill type (defined in the menu "Change drill technology") will be used for the vias (pins) defined in this line (not relevant to the router).

### HANDLING

1. Enter the number of the line (1 to 31) in which you want to make changes in "modify no.", <CR>. The cursor moves to the first input field of the selected row.
2. Now the changes can be made, <CR>.
3. By entering "W" for "write" and pressing <CR>, the entries are stored.
4. With <ESC><ESC>, you leave the menu.

## DRILLS

After activating the function "Drill" in the menu "Technology", a chart with the drill technology of the current .adb-file appears in the status window.

When creating a new .adb-file, the chart contains the values stored in the standard technology.

PRISMA status window				PRISMA menu window			
PCB: modify drill-technology				Grid: 0.035 / 0.035 AS: mm [900]			
set: 0 modify no.: 0_ (C) check, (W) write: _							
no.	diameter	annulus	plated				
0	3.0	0.0	N	placement & routing			
1	7.4997	0.25e2	Y	size			
2	1.3002	0.350	Y	layer-plan			
3	0.0	0.0	N	tracks			
4	0.0	0.0	N	reference component			
5	0.0	0.0	N	vias			
6	0.0	0.0	N	drills			
7	0.0	0.0	N	tests			
8	0.0	0.0	N	main menu			
9	1.0999	0.25e2	Y				

From the 255 drill types in the menu, only the first 32 may be considered at present.

The sizes of the 32 drill types, their annulus widths and the specifications to the parameter "through-connected" can be changed, if necessary.

## HANDLING

1. Enter the number of the line (1 to 31) in which you want to make changes in "modify no.", <CR>. The cursor moves to the first input field of the selected row.
2. Now the changes can be made, <CR>.
3. By entering "W" for "write" and pressing <CR>, the entries are stored.
4. With <ESC><ESC>, you leave the menu.

**TEXTS**

This function has not yet been implemented.

**MAIN MENU**

You return to the main menu.

17. CHANGE FILE

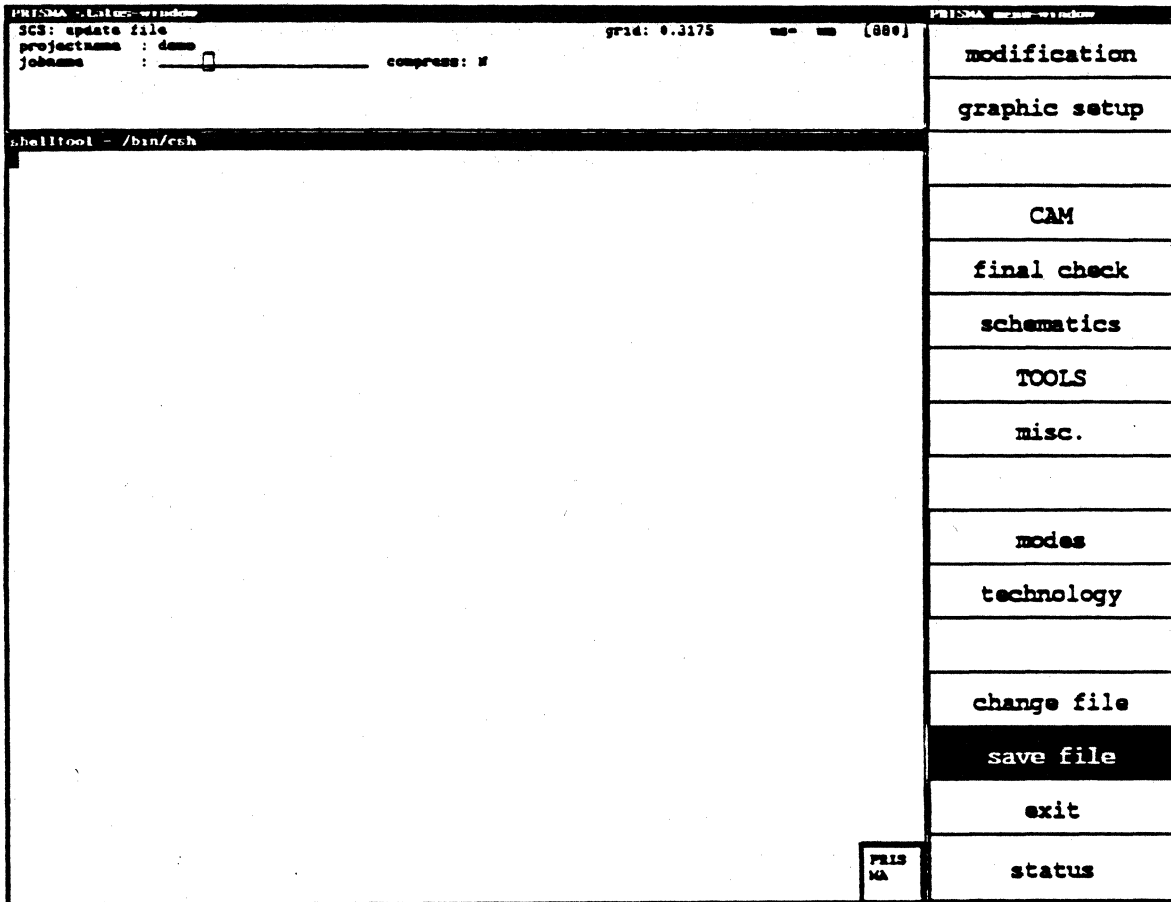
PRISMA - LAY - window PCB: functions (B) modification (G) graphic setup (C) CAM Grid: 0.317 / 0.317 MS: [560] (A) misc. (M) modes (Y) status (T) technology (F) final check (S) schematics (R) change file (U) update file (X) exit set:			PRISMA menu-window <b>modification</b>
			graphic setup
			CAM
			final check
			schematics
			TOOLS
			misc.
			modes
			technology
			change file
			save file
			exit
			status

shelltool - /bin/csh

PRIS MA

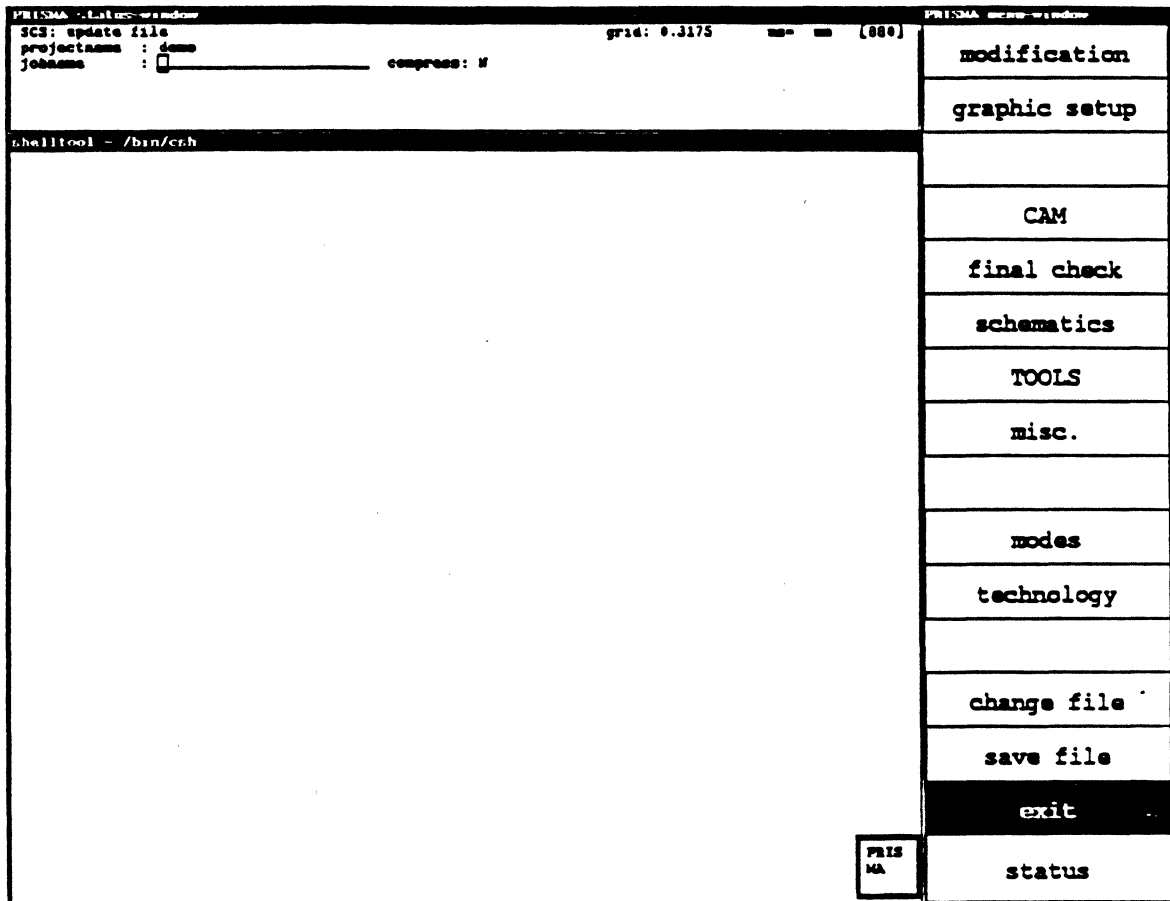
This function has not yet been implemented.

18. SAVE FILE



Stores the current file with the name to be specified in the status window. This kind of interim storage should be done regularly during the modification of a file. For this purpose, it is not necessary to exit the program and the selected menus.

## 19. EXIT

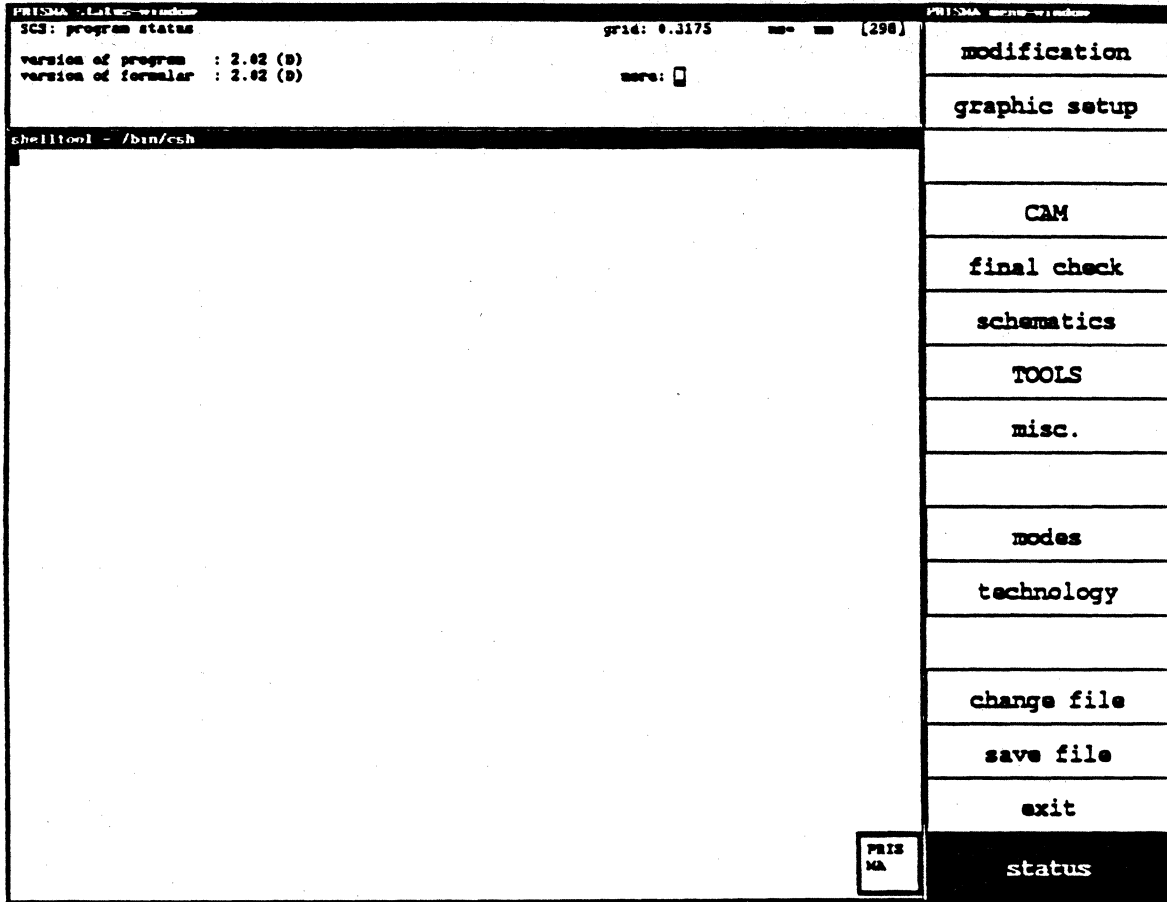


Ends the program after making a backup copy. This step can be aborted by pressing the <ESC> key twice. If the program is to be exited that way, you have to specify "Y" for the questions

"abort storage?" and  
"exit program?"

and to confirm them with <RETURN>.

20. STATUS



Displays the current software version number in the status window.

The version number of the program and the menus of the status window is displayed. Leave the menu "Status" by calling another menu.



# TABLE OF CONTENTS

## C. ROUTER

ROUTER LDROUT . . . . .	1
I. Introduction . . . . .	1
II. Handling . . . . .	2
a. Starting the program . . . . .	2
b. Exit and restart . . . . .	4
c. Options . . . . .	5
1. Router options . . . . .	6
2. Selection of the control file . . . . .	8
d. Standard control file ldroun.Ctl . . . . .	10
Busrouting . . . . .	10
SMD connections . . . . .	10
Glossing . . . . .	11
e. Optimizing router . . . . .	11
III. Output format . . . . .	13
a. Basic information . . . . .	13
b. Additional information . . . . .	14
IV. Router technology . . . . .	16
a. General . . . . .	16
Router symbols . . . . .	16
Display of routable tracks . . . . .	17
Display of vias and non-routable tracks . . . . .	17
b. User defined technologies . . . . .	18
General . . . . .	18
General definition instructions . . . . .	18
Placement & routing . . . . .	20
Sizes . . . . .	24
Layer plan . . . . .	27
Change track technology . . . . .	30
Track width overall . . . . .	31
Via technology . . . . .	34
Drills . . . . .	37
Reference component . . . . .	38
c. Standard technologies . . . . .	39
d. Ldr-file . . . . .	46
V. Multilayer routing . . . . .	48

**ROUTER LDROUT**<sup>1</sup>sectionI**I. INTRODUCTION**

LDROUT is the successor of the CALAY V04 router CARVAR. It is data compatible to GPH-files in the V04 Rev.5 format.

In the definition of the routing technology that is effected in CIS, LDROUT allows a free adjustment of the technology sizes by the user.

LDROUT is a three-stage routing system. With the upgrade V2.06, the pre-router (previously "LDPRE") has been integrated in the main router. The optimizing router ("LDOPT") can either be called automatically after the routing has been finished or as a separate process.

The pre-router is necessary for the sorting of the input data (pin sorting within the nets, net sorting, etc.) and the output of the previous information (routing grid, number of layers and missing connections, etc.).

After the actual routing process, the main router automatically executes the glossing (cost optimization of segments and minimization of the vias).

The optimizing router executes the optimization of the found track courses by automatically shifting or changing the segments or segment parts and vias. The optimization is effected according to cost minimizing, technical, electrical and aesthetical criteria.

The data base for the router is the .gph-file that contains the necessary information (technology, netlist, etc). .adb-files that do not yet exist in the gph-format are transformed system internally into the gph-data-format after the router has been started.

During the routing, the .gph-file is supplied with router information (existing track courses).

The output of the router is the .gph-file extended with the found track courses that may then be retransformed into an .adb-file.

The information that appears on the screen is stored in a routing report file. This file is written on the gph-directory and gets the name of the .adb-file and the extension .ldr.

section end

---

<sup>1</sup>Please note that some of the documentation contains sections marks (e.g. "section 1.0" at the beginning and "section end" at the end to indicate that respective documentation is used as an online-manual).

# ROUTER/Handling

---

## II. HANDLING section IIA

### A. STARTING THE PROGRAM

Under the ROUTER icon, all programs are combined that are necessary for:

- the software routing
- the hardware routing
- the conversion from .gph-files into .adb-files
- viewing the routing reports (\*.ldr) and the error reports for the conversion (\*.ls1 for adb -> gph; \*.ls2 for gph -> adb)
- calling the help function

At first the following menu appears for the function selection:

```
*****  
                PRISMA  ROUTER  SELECTION          v .....  
*****  
                ***  default selection: >> 5 <<  ***
```

1. Run Softwarerouter (ldrout)
2. Run Hardwarerouter (ld400)
3. Run 'gph > adb' conversion
4. View 'list files' (router and conversion)

### 5. Help information

### E. Exit

-----  
your choice ?

Handling instructions:

- Each input must be confirmed with <CR>.
- After entering the selection number and pressing <CR>, a respective submenu for another function selection or a menu for the file specification appears.
- Selection menus contain a default selection in the upper right corner as a rule; the default selection can be activated by pressing <CR>.
- The selection menus can only be left by entering "0" (you get back to the superior menu) or "E" (Exit).
- <CTRL C><CR> can be used for interrupting a process and leaving a file inquiry menu (not for leaving selection menus).
- Hardware routing is of course only possible with the RPR-400. With the support of this hardware accelerator, you will not only get a much faster routing, but also a better routing quality.

After calling the software or hardware router, the following inquiry appears:

**autocreation of .adb file after routing (Y/N)? \_**

When you answer the inquiry with "Y" <CR> or just press <CR> (because "Y" is the default), the .gph-file is automatically retransformed into an .adb-file after the routing (i.e. when the board has been routed completely and smoothed or when the specified number of iterations has been effected).

Afterwards, you are asked for the name of the file to be routed (do not enter an extension!).

Then you are asked for the extension.

**enter the filename [without ext.] :**

**enter the extension [adb, gph] :**

#### Extension specification

- a) If the file was created with the program CIS and has not yet been routed, it only exists in the adb-format.

After entering the .adb extension and the desired options (see below), the file is automatically transformed first.

The transformation is made from the .rai-format into the .gph-format (.adb --> .rai --> .gph).

- b) If an .adb-file has already been routed (completely or partly) and has not been retransformed into an .adb-file and processed in CIS, it already exists in the .gph-format. When routing this file again (= 'restart'), you must enter the extension "gph".

#### Attention:

If the .adb extension is entered although a gph-version of this file is already existing, the program is automatically aborted during the transformation so that the existing gph-version will not be overwritten by the newly transformed file.

After entering the file name with extension and specifying the options, the routing data is transformed and read in from the .gph-file. Subsequently, the router is automatically started.

Note: To avoid that the file is overwritten unintentionally, an .adb-file that has been modified by the router and read in CIS cannot be stored with its original name!

section end

### B. EXIT and RESTART sectionIIB

#### Exit:

When routing without graphic output (i.e. without option "-f"), the routing can be aborted with <CTRL C>. The routing result up to the last finished layer iteration (two layer board) or layer pair iteration (multilayer) is automatically stored.

Note: When using the option `-c:ld1`, the routing should not be interrupted before the 14th iteration, as in this special case the routing result will only be stored automatically after 14 iterations (see page 8).

When routing with graphic output, the routing can only be aborted by placing the mouse cursor on the edge of the LDROUT graphic window and pressing the L10 key.

#### Restart:

With "restart", the repeated routing of an already routed file (extension "gph" as a rule) without the option "-n" is meant. The router starts with the result of the last finished layer iteration (two layer boards) or layer pair iteration (multilayer).

If a file was routed (partly), was then re-transformed into an .adb-file and changed interactively in CIS, you have to specify the new name of the (changed) file and the extension "adb" when restarting the router. After the transformation of this file, the router continues with the routing result of the 'old' file, considering the changes in CIS.

section end

**C. OPTIONS**  
sectionIIC

You may enter several options (switches) before starting the routing process which directly influence the working mode of the router.

The specification may contain several options in a row (without blanks; the "-" character belongs to the option!). You have to take care that the options do not exclude one another.

```
*****
* PRISMA ROUTINGSWITCH SELECTION v... *
*****
```

Router Options :

```
-f      - graphical support
-g      - perform glossing iterations
-i:n    - perform a maximum of n iterations
-k      - routing without restrictions (use with '-n'only)
-l:n    - route only on layerpair n
-n      - new routing
```

----- usable control files: -----

```
-c:ld1  - busrouting - save after 14 cycles through pcb
-c:ld2  - no busrouting
-c:ld3  - perform only 2 final glossing iter. per layerpair
-c:ld4-g - perform only optimizing router
-c:ld5  - standard control file + optimizing router
        (without -c option a standard file will be processed)
```

enter [option][option]

option :

## 1. ROUTER OPTIONS

### **-f (graphical support)**

This option effects the graphic output of the routing process on the screen. As the graphic output appears in the foreground, the report output is not visible.

Notes:

- In the graphic output mode, the routing can only be aborted by placing the mouse cursor on the edge of the LDROUT graphic window and pressing the L10 key!
- White terminals cannot be connected by the router (-> check if they lie within a restricted area or if false connection restrictions have been defined).
- Blue areas indicate clearance violations.

### **-g (perform glossing iterations)**

If the router has completely finished a board, it automatically executes glossing iterations (multilayer: 2 x 2 per layer pair; two layer boards: 2 x 2) (ref. to option '-c', page 8).

When specifying switch "-g", only the respective number of glossing iterations will be executed, regardless whether the board is already routed to 100%.

Note: As a reduction of the segments and vias is effected during the glossing and further routing attempts are executed if the board is not completed, the routing result can be improved as a rule if the router is interrupted and the current provisional result is optimized with pure glossing routing (restart with the option "-g").

But the glossing may also result in channel blockings sometimes as the via reduction is more important for the router than the 'wrong-way'-routing.

### **-i:n (perform a maximum of n iterations)**

Independent from the number of missing connections, the routing will be interrupted after n iterations (possible glossing iterations included!);

(n = 1, 2, 3, etc.).

Without the option '-i:n', the router does not stop before reaching the 100% solution.

**-k (routing without restrictions)**

This option should only be used for boards that are easy to route (with much free space) for an acceleration of the routing. With this option, 'space-saving' restrictions are switched off that are effective in the first five iterations during the 'normal' routing (limitation of the allowed number of vias per net; limitation of the track course to the shortest possible way).

**-l:n (route only on layer pair n)**

For multilayers, you can specify by entering a respective number for "n" that the router may only route on this layer pair.

**-n (new routing)**

This switch causes the router to restart. If the file has already been routed or if variable tracks have been digitized interactively in CIS after the routing, all variable tracks and vias will be deleted and newly routed when starting the router with this function.

Note: For a restart, a busrouting is intended for the first iteration as a rule (ref. to 'Standard control file ldrout.CTL', page 10).



### 2. SELECTION OF THE CONTROL FILE

The proceeding of the router is determined basically by the used .CTL-control file. The .CTL-files are stored on the directory lib/sys.

A .CTL-control file contains:

- parameter sets that control the routing in regard to lower production costs ('cost parameter sets')
- control commands that activate or suppress special functions during the routing (e.g. execution of the busrouting, placement of 'stubs' for SMD routing, activation of the suitable parameter sets, calling the optimizing router, etc.).

The different control files differ only in regard to the function control commands; the cost parameter sets are the same in all control files.

The .CTL-file is called with the option '-c:'. If no specification is made, the standard control file ldroun.CTL will be used automatically (see page 10).

#### **-c:ld1 (save after 14 cycles through pcb)**

When specifying this option, the router uses the control file ld1.CTL for the current routing. The only difference between this control file and the standard control file ldroun.CTL is that

- > the routing result is not saved after every processing of all layer pairs, but only after each 14th iteration.

This effects a considerable acceleration of the routing. But in this case, the routing can only be interrupted after each 14th iteration, otherwise iteration results will be lost.

Note: You should only use this option for two layer boards.

#### **-c:ld2 (no busrouting)**

When specifying this option (= use of the control file ld2.CTL), the busrouting is only executed on the inner layers.

- > no busrouting is effected on the outer layers, i.e. the router can already place through-connections in the first iteration on the outer layers;
- > in the first and second run, only one iteration per layer pair is executed;
- > from the third run, 2 iterations per layer pair are effected.

#### **-c:ld3 (perform 2 glossing iter. per layer pair)**

With this option (= use of the control file ld3.CTL), only two final glossing iterations per layer pair are executed.

In the final glossing iterations, the router does not lay much stress upon the 'wrong-way'-routing. Therefore, this option should only be used for an additional glossing of 100% solved boards.

**-c:ld4-g (perform optimizing router)**

When using this option, only the optimizing routing is executed (see page 11); before the optimizing iterations, the router executes a glossing iteration per layer pair.

**-c:ld5 (standard control file + optimizing router)**

This control file effects the execution of the standard routing with subsequent optimizing routing.

section end

### D. STANDARD CONTROL FILE `ldrout.CTL` sectionIID

If the user does not specify a special .CTL-file with the option "-c:", the standard control file `ldrout.CTL` will always be used automatically. This file controls the routing in regard to cost parameter sets and effects the execution of the following functions:

#### 1. Busrouting

For the first routing or for a new start, only direct connections without vias (bus connections) will be routed in the first iteration in all routing grids.

The routing of the tracks follows the specified directions, provided that the layer plan contains a correct layer assignment:

- on the solder side and the respective inner layers in the X direction
- on the component side and the respective inner layers in the Y direction.

#### 2. SMD connections

If there are SMD components with pins to be connected on the board that shall be routed, the router creates connections to the other layers (called 'stubs') in the first iteration on the outer layer pair by routing short track segments of the respective SMD pins and placing through-connected vias there (ref. also to chapter "Routing with SMD technology). Subsequently, the bus routing is performed in a second iteration on the outer layers.

If the board is a multilayer, the busrouting is executed on the inner layers afterwards. The 'stubs' may be used as bus connections.

In the output of the not found connections, the stub connections are marked with ".\*" [e.g. "SMD1.\*(3)"].

The unused stubs will be deleted during the glossing.

### 3. GLOSSING

After the iteration in which the PC board is finished (i.e. all connections are found), 2 x 2 glossing iterations are executed per layer pair.

During the glossing iterations, special corrections are executed: the number of vias is minimized and the course of the tracks is optimized (see page 6, option "-g").

When restarting the router with the option "-g", the number of missing connections may be reduced since connections can also be routed during the glossing iterations.

section end

### E. OPTIMIZING ROUTER

sectionIIE

The optimizing router LDOPT that is called with the option '-c:ld4' or '-c:ld5' executes the following optimizations in the described order:

- VIA:** Vias will be shifted on the grid to less dense areas in the positive and negative X or Y direction to get more space for the segments.
- AVERAGE:** Segments will be averaged, i.e. they will be shifted to the vertical direction as far as possible (x segments in the negative and positive Y direction, Y segments in the negative and positive X direction).  
As a result, distances may become too large and U-shaped segment topologies may be generated. In this case, the average will only be reached with a subsequent U-optimization.
- U:** U-shaped segment topologies will be minimized or, if possible, deleted.
- VIA CORNER:** Vias that belong to L-shaped segment constellations and lie within a segment will be shifted to the corner where the segments meet.
- T:** T-shaped segment constellations are treated like double L-structures and become V-shaped by slanting them.
- L:** Slanting of corners. The L-structure can be changed into several slants that are separated by steps (= staggered).
- L-Move:** Slants will be shifted until all interrupting steps are deleted.

Notes:

- AVERAGE and the subsequent U-optimization will first be executed for X segments and then for Y segments.

- The 45° slants created during the T and L optimization will not be generated as vectors but as steps (like on the graphic screen). The number of segments is increased respectively after this optimization.  
  
The conversion of these 45° slants into real 45° vectors can be effected in CIS with the CAM function (popup function '45 degrees').
- Vias will only be shifted and segments will only be changed if free space exists, i.e. obstacles cannot be leaped.
- The optimizing router is bound to the grid. Parameters are for example the clearance to be adhered to (1 grid unit for L and T optimization, 2 grid units for all other optimization methods) or the length of the slants (e.g. at most 15 grid units).

section end

---

**III. OUTPUT FORMAT**  
**section IIIA****A. BASIC INFORMATION**

After the read in of the file, the pre-router determines the router technology and checks the technology definition for clearance violations or other technological errors (if there are any, the router will display warnings or error messages on the terminal).

After the check, the following information is output:

- the version number of the router,
- the number of layers (".. layers"),
- the routing grid ("grid = 1/..."),
- the number of missing connections ("... missing connections"),
- the number of missing connections that belong to 'fixed nets' (i.e. to signals that were marked as non routable in CIS) ("... of them in fixed nets")

Attention: Although the router outputs these connections as missing connections, the routing is finished when all other connections have been found; a subsequent technology check in CIS shows the respective number of interruptions!

- the number of terminals that lie offgrid ("... terminals offgrid")

Then, the main router starts the first iteration. During each iteration, a line is output that contains the following basic information:

- iteration specification:    b = bus or stub iteration  
                                  i = standard iteration  
                                  g = glossing iteration
- current number of the iteration,
- the number of calculated connections,
- the number of missing connections,
- the connection points that were 'touched' by the router but could not be connected.

# ROUTER/Output Format

```

i  1( lp  ls  l)  321 / f  152  nnnn  nnnn  nnnn

```

non-connected point  
 number of missing  
 connections  
 number of calculated  
 connections  
 current iteration  
 number

The number of calculated connections is newly displayed after each tenth connection 'touched' by the router. This automatic feature is interrupted if a connection is found or a missing connection is written in the output line.

During the first 9 iterations on a layer pair, the already found connections are not all calculated anew as a rule. This leads to a considerable saving of time.

**Exception:** During glossing iterations, always all connections on the respective layer pair are routed.

(When using the option '-g', the missing connections are also routed).

From the 10th layer pair iteration on, all connections on the current layer pair are calculated anew.

The number of missing connections is always decremented.

If the specification of a not found connection contains "." between the component name and the terminal number, then you have a SMD-pin to which a via for changing layers (= 'stub') has been connected by the router in the first iteration.

section end

## B. ADDITIONAL INFORMATION

sectionIIIB

Behind the current iteration number, the following additional information is output in parentheses:

- total number of bus/stub, standard or glossing iterations executed on this layer pair (the three iteration types are counted separately),
- the cost parameters from the .CTL-file and
- the track type routed in this iteration.

```

i  135 ( 52p  16s  3)

```

track type  
 cost parameters  
 no. of standard iterations  
 executed on this layer

pair

After each completed iteration, the total time needed for this iteration is displayed in a separate line, as well as the time needed for the system internal preparation of the iteration:

**CPU**            **0:01:54.04**            **(37% system)**

preparation part  
1/100 seconds  
seconds  
minutes  
hours

section end



## IV. ROUTER TECHNOLOGY

### sectionIVA0

#### A. GENERAL

The routing technology of an .adb-file is defined in CIS/PCB. Basic routing technology parameters are:

**grid, clearance, routable track and via types.**

The operation of the router is based on the respective routing grid. The universal router LDROUT is not restricted to the CALAY standard grids (1/20", 1/40", 1/50", 1/60", 1/70", 1/80") but can work in freely defined grids.

Beside the grid, all other parameters of a technology are freely definable. The parameter values in the CIS/PCB standard technologies can, therefore, be seen as defaults and may be changed, if necessary.

section end

### sectionIVA1

#### 1. ROUTER SYMBOLS

For the internal display of the layout elements defined in the CIS/PCB technology (pins, tracks, vias) in the grid, the router uses 16 symbols whose diameters are calculated from the widths of the routable track types and the routing grid.

**max. symbol size = 0.000**

**max. symbol size = 2 x routing grid - clearance**

This limitation does not restrict the usable technology sizes for tracks and vias/pins. If an element is larger than the maximum symbol size, it will be composed of several symbols. Then the router always uses an unpaired number of the same symbol (ref. to figure 1). Zero symbols may also be used (3, 5, 7, etc.) - they do not have an extension but use grid cells.

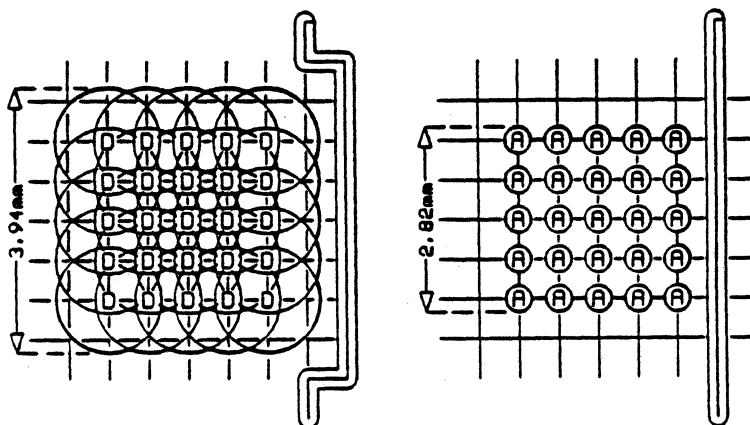


Figure 1

section end

**sectionIVA2****2. DISPLAY OF ROUTABLE TRACKS**

As the number of router symbols is limited to 16, only 16 free routable tracks are available.

Attention:

The routable track types must only be defined within the first 32 types (i.e. type 1-31) at present!

A via type (1-31) that must be used during the routing can be assigned to each free routable track type.

Free routable tracks are always exactly displayed in the routing grid. This also applies to routable tracks whose width is larger than the maximum symbol size.

If the number of defined routable tracks is smaller than the number of symbols (i.e. values unequal 0.00 are entered for less than 16 routable track types in the technology chart), the free symbol sizes are distributed router-internally.

Note: If a routable track type defined in the technology is not used in the layout, the router can change the already defined width of this type for its internal calculation to be able to represent a special via type exactly.

In the technology chart of the routing report file, the width used by the router appears in this case instead of the real CIS width.

section end

**sectionIVA3****3. DISPLAY OF VIAS AND NON-ROUTABLE TRACKS**

Vias (routable and non-routable), as well as non-routable tracks whose widths are larger than the maximum symbol size are displayed router-internally with several symbols of the same kind. Then the router uses the symbol that - composed of an unpaired number - guarantees an almost exact display of the real CIS-size.

Attention:

These via and track types must only be defined within the first 32 types (i.e. type 1-31)!

The parameter sizes entered as the default in the CALAY standard technologies are optimized and guarantee an exact display as a rule.

If the standard parameter values in the technology charts in CIS are changed by the user, the widths and diameters of vias and non-routable tracks might lie between the values that can be calculated by the router. In this case they will be realized with the next higher value that can be displayed (ref. to the chapter ".ldr-file"). The difference between the value with which the router calculates and the value specified in CIS (= inaccuracy) becomes apparent in the layout in a respective increase of the clearance.

When changing the standard values of the CIS technologies, we advise you to check the exactness of the router-internal display of the CIS technology sizes with the .ldr routing report file (ref. to ".ldr-file").

section end

## B. USER DEFINED TECHNOLOGIES

### sectionIVB1

#### 1. GENERAL

The router technology is a part of the .adb-file generated in CIS.

The first step for the creation of an .adb-file in CIS is the read in of an existing technology standard whose name contains the desired routing grid. (The names of the CALAY standard technology .adb-files contain the routing grid value in 1/nn inch : tec20, tec40, tec50, tec60, tec70, tec80.)

An existing technology standard can be read and changed in CIS in the submenu "Technology" of the PCB part.

If you want to change the sizes defined in the technology (routing grid, clearance, track widths, via sizes, etc.), sometimes parameter specific restrictions have to be considered that will be described in this chapter.

If a changed technology is often needed, it is useful to store it as own standard after the definition. For better identification, the respective .adb-file should get a suitable technology specific name.

section end

### sectionIVB2

#### 2. GENERAL DEFINITION INSTRUCTIONS

To avoid routing problems and inaccuracies in the router-internal representation, you should consider the following advice when defining user specific technologies:

- a. Track, via and drill types must only be defined within the first 32 types (i.e. type 1-31).
- b. Routable vias and non-routable tracks can be exactly displayed in the routing grid if they have the same parameters (total width, rectangle width, additional clearance) as a routable track.
- c. A routable track type that is not used in the netlist but will be needed for the digitizing, should be marked as non-routable in the technology chart. In this way, the router can calculate a better size graduation for the symbols.
- d. If a routable track type is neither used in the netlist nor for the digitizing, all width specifications should be 0.00 (for the same reason as in c.)
- e. Tracks and vias shifted half a grid in the negative X and Y direction are always displayed with an even number of grid cells. This results in a router-internal minimum limit of one grid cell for the diameters of these shifted elements.

In the following, the "Technology" menus from the CIS-PCB part will be described.

Although not all technology parameters are router relevant, they will be described here because the CIS documentation does not yet contain a detailed technology description.

The router relevant parameters will be pointed out. Please consider the restrictions that are necessary for some parameter changes.

section end

## 3. PLACEMENT & ROUTING sectionIVB3

When activating this function in the menu "Misc.", the following menu appears in the status window:

```
PCB: modify technology          Grid: 0.026/0.026  MS: in.  [98
(P) plac. & routing  (S) sizes          (Y) layer-plan
(L) tracks          (R) reference - comp. (V) vias
(D) drills          (T) texts          fct: P
grid-base          1 / 1323          routing - base : 1" / 960
routing-grid       x: 0.025 y: 0.025 offset      x: 0.0 y: 0.0
placement-grid    x: 0.050 y: 0.050 offset      x: 0.0 y: 0.0
via-grid          x: 0.025 y: 0.025 offgrid routing: N
clearance         x: 0.011 y: 0.011 copper height : 0.001 (in.
buried vias      : _
```

### GRID BASE

Specifies the minimum unit for the system internal calculation. This specification is only displayed for your information.

### ROUTING BASE

The routing base is the smallest coordinate distance considered by the router. This unit may be changed.

The lowest routing base is 1/6000 inch, the highest 1/100 inch. 1/960 inch is the standard routing base.

The routing grid must always be an integer multiple of the routing base and the multiplier (M) must fulfill the following condition:

$$2 \quad M \quad 50$$

If this is not the case, the specified routing base value will automatically be rounded off so that this condition is fulfilled.

Example:

Routing base:                    1/960"; routing grid: 1/70";  
                                  i.e.  $M = 1/70 : 1/960 = 13.7$

Change: routing base:            1/980" ( $M = 1/70 : 1/980 = 14$ ).

**Note:**

- The maximum size of the work sheet (measurement unit in X and Y direction) which can be constructed with CIS depends on the routing base and can be determined exactly by multiplying the routing base by 32767.

**Examples:**

1/960 inch resolution --> 32767 X 1/960" 34" 866 in.  
 1/100 inch resolution --> 32767 X 1/100" 327" 7320 in.  
 1/6000 inch resolution --> 32767 X 1/6000" 5.5" 138 in.

**ROUTING GRID:**

If no shifting exists, the tracks are routed along the grid lines and the vias are placed on the grid points.

The routing grid must always be an integer multiple of the routing base. If this is not the case, the routing base will internally be changed into a suitable value that fulfills this condition.

According to technology and coordinate unit, the grid unit to be used by the router is automatically entered in "Grid" (e.g. "0.635" for 1/40" technology and unit in.). The table shows the grid values for the standard PRISMA routing grids.

unit grid	mm.	in.
1/20"	1.27	0.050
1/40"	0.635	0.025
1/50"	0.508	0.020
1/60"	0.423	0.0166
1/70"	0.362	0.0142
1/80"	0.317	0.0125

## VIA GRID:

By entering the grid element spacing (in the respective unit --> ref. to the function "Modes"), you can specify the distance between the vias.

If you have a routing grid of x: 0.635 y: 0.635 and enter x: 1.27 y: 1.27 for the via grid, the router can at most place a via on each 2. grid (so you still have the possibility to digitize a track between the vias).

If the specifications in "Routing grid" and "Via grid" are equal, a via can be placed on each neighboring routing grid.

Attention:

If the via grid does not equal the routing grid, it must be an integer multiple of it! At present, the system does not check if this condition is fulfilled!

## PLACEMENT GRID:

Since routing is bound to the grid, components should be placed on the routing grid or an integer multiple of the routing grid.

A disadvantageous choice of the routing grid may lead to a bad routing result since pins may be placed off-grid. Off-grid pins (i.e. pins which are not placed on a grid point) may effect that no tracks can be digitized between them.

The most sensible grid for the placement of components is 1/10" as it is an integer multiple of all standard routing grids.

sensible grid for the placement							
routing grid	1/10"	1/20"	1/40"	1/50"	1/60"	1/70"	1/80"
1/20"	X	X					
1/40"	X	X	X				
1/50"	X			X			
1/60"	X	X			X		
1/70"	X					X	
1/80"	X	X	X				X

In order to get an optimal bus routing, take care that the components of memory structures do not lie more than 0.3 inches (= 7.62 in.) apart from each other.

**OFFSET:**

By entering the respective values, you can define a shifting of the placement grid with regard to the routing grid.

This is sensible if elements that are 'on-grid' in the routing grid lie 'off-grid' in the placement grid.

The shifting of the routing grid is not yet possible.

**OFFGRID ROUTING: Y/N**

When entering "Y", the complete routing area with all elements will be shifted half a routing grid in the negative X and Y direction.

This is necessary for the 1/60" routing in order to route 2 tracks of type 1 between 2 IC-pins with a maximum diameter of 1.53 in..

**CLEARANCE:**

Here you specify the clearance between the PC board elements (tracks, pins, vias, text fields, etc.) and to the routing area and PC board edge.

The clearance, optimized in the standard technologies, can be changed.

Notes:

- In order that segments of the track type with the smallest track width can be routed on each grid line, you have to take care that  
clearance + smallest track width    routing grid
- A subsequent reduction of the initial clearance can effect short circuit messages when executing a technology check afterwards!
- For the single track and via types, an **additional clearance** can be specified in the technology charts that will be added to the general clearance when routing this track or via type.



## ROUTER/Technology

---

### COPPER HEIGHT:

On the base of the copper height and the track width, the track diameter is calculated (ref. to "SCS: Signals & connections"). This parameter is relevant to the simulation.

At present, the standard value 0.001 in. appears here; changes of this specification are not evaluated.

### BURIED VIAS:

This specification only applies to the interactive digitizing of tracks, not to the routing. By putting a cross in this field, you specify that buried vias shall be used for the digitizing.

section end

### 4. SIZES

sectionIVB4

PCB: modify technology	Grid: 0.025/0.025	MS: in.	[775]
sheet size	x: 8.30	y: 5.94	boardname: _____
pc-board:	x: 25.40__	y: 25.40__	/ x: 4.15 y: 4.90
rout.-area	x: 25.40__	y: 25.40__	/ x: 4.15 y: 4.90
prod.-origin	x: 25.40__	y: 25.40__	
digit board shape:	N		

In this menu, the specification of the work sheet and routing area size and the digitized board outline is relevant to the router.

The size specifications are entered in the status window.

Reference point for all coordinate specifications in this menu is the zero point of the sheet.

Attention:

To store the changes of the size specification in this menu, you have to press <CR> first, then <ESC><ESC> and answer the inquiry "Update sizes" with "Y".

### SHEET SIZE:

When specifying the work sheet sizes, the maximum size considered by the router must not be exceeded.

The maximum size of the work sheet in x and y direction depends on the routing base and can exactly be determined by multiplying the routing base by the number 32767.

The sheet frame is displayed automatically on the graphic screen.

**BOARD NAME:**

Here, you can enter a user text for characterizing the current PC board that will always appear in this field when calling this file again.

**PC BOARD:**

By specifying the coordinates of the origin and the upper right corner of the board, you determine the PC board size here.

Attention:

The contour of this area does not appear automatically on the graphic screen! The board outline that is visible on the screen is created with digitizing (ref. to "Digit board shape").

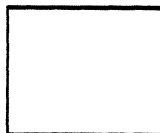
So the board outline can diverge from the rectangular area defined in "PC board"!

**ROUT.-AREA:**

Here, you specify the routing area by entering the coordinates of the origin and the upper right corner of the routing area.

The frame of the routing area is displayed automatically on the graphic screen. The maximum size of the routing area in x and y direction depends on the routing grid and the form of the routing area:

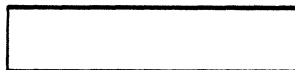
- square area



y = 2048 grid cells

x = 2048 grid cells

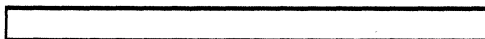
- rectangular area



y = 1024 grid cells

x = 4096 grid cells

- long rectangular area



y = 512 grid cells

x = 8192 grid cells

### Attention:

Components must not be placed outside the routing area, otherwise the file cannot be routed (the program is aborted with the message "Element outside area").

### PROD.-ORIGIN:

Coordinate specification for the production zero point.

This point is the **coordinate origin** for the interactive work in CIS-PCB and for the postprocessing and coordinate display (when using the graphic menu function "Coos on/off").

### DIGIT BOARD SHAPE:

When entering "Y", <CR>, the function "Mod. graphic - digitize" of the work menu is called automatically.

If a standard technology is read in for generating an .adb-file, the board shape is already existing - it appears as inner frame on the screen.

This predefined board shape can be changed by getting and deleting the respective lines first and digitizing the desired shape then.

### Notes:

- When digitizing, take care that the board outline is closed perfectly (check with "STOP")!
- The predefined or digitized board outline delimits also the routing area. The area outside the board outline is a restricted area for the router.
- The area defined in "PC board" need not be identical with the area inside the board outline.

Changing the size specifications in "PC board" does not influence the board outline.

section end

5. LAYER PLAN  
sectionIVB5

```

PCB: change layer plan          Grid: 0.026/0.026 MS: in.      [770]
fct: Y      modify no.: 0__    (C) check, (W) write: _

#  key  name | #  key  name
-----|-----
2__ COM  COMP | 17_ SHP  SHAPE 0__
1__ SOL  SOLD | 18_ SHP  SHAPE 1__

0__  _   _   | 19_ RES  RESIST 0__
0__  _   _   | 20_ RES  RESIST 1__

0__  _   _   | 21_ DRL  DRILL  _
0__  _   _   | 22_ DOC  DOCU  5__

0__  _   _   | 23_ DOC  DOCU  6__
0__  _   _   | 24_ DOC  DOCU  7__
    
```

When creating a new .adb-file, the layer plan for a two layer board appears as the default.

With the respective entries, you can change the layer plan as desired.

From the 255 possible layers, only the first 32 can be considered at present.

With <CR>, paging is possible.

#

The layer number is entered here.

**KEY**

Here, the respective specification of the signal or documentation layer is entered.

The specifications that may be used as "key" are determined:

- COM --> component side
- SIG --> signal inner layer
- SOL --> solder side
- PLN --> plane
- FRE --> free layer
- SHP --> silk screen documentation layer
- RES --> solder plan documentation layer
- DRL --> drill plan documentation layer
- DOK --> any documentation layer

## NAME

In order to define a layer in the layer plan, you must enter a user comment here (e.g. an exact specification of the layer).

### Attention:

At present, the default definition of the documentation layers 17 to 21 must not be changed because OUTPUT can only evaluate this standard.

## HANDLING

1. Enter the number of the line (1 to 31) in which you want to make changes in "Modify no.", <CR>. The cursor moves to the first input field of the selected row.
2. Now the changes can be made, <CR>.
3. By entering "W" for "Write" and pressing <CR>, the entries are stored.
4. With <ESC><ESC>, you leave the menu.

## EFFECT OF THE LAYER PLAN CONSTELLATION ON THE MULTILAYER ROUTING

The router only processes signal layers. The layers specified with "empty" or "plane" in the layer plan will not be considered by the router.

If the layer plan of a multilayer contains empty layers and/or planes, the router usually combines the signal layers differently than defined in the layer pair combination of the layer plan.

### RULES for the router internal combination of the layers:

1. The router starts the routing process on the component side (= 'upper' layer) and continues towards the solder side (= 'lower' layer). The layers are combined to pairs. If the router finds a constellation for which pairing is not possible, it routes the single layer.
2. The component side (= layer 2 in the layer plan) will always be routed first regardless of the other layer the router combines it with.
3. Layer 2 and 1 (= component and solder side) will only be routed together if the layers below are specified as "empty" or "plane".

**In SMD routing**, this constellation is the condition for generating vias for changing layers (track segment + via at SMD pin) in the first layer pair iteration.

If this is not the case, the router combines layer 1 with another layer and does not route them in the first layer pair iteration; correspondingly, no vias for changing layers are created for the SMD pins placed on the solder side.

4. If both layers of the layer pair are specified as signal layers in the layer plan, they are always routed together.

5. If the router finds a layer pair in which one layer has been specified as "empty" or "plane" and the other layer as signal layer, the signal layer is routed alone if the layer pair below does not consist of an "empty" or "plane" layer plus a signal layer.

(This does not apply to the layer pair component/solder side if point 3 is fulfilled).

6. If the router finds two successive layer pairs with a signal layer and an "empty" or "plane" layer when moving from component to solder side, it combines the two signal layers of the two layer pairs.

This example explains the described rules:

PCB: Change layer plan			Grid: 0.025/0.025 MS: in.			[770]
fct: Y		modify no.: 0__	(C) check, (W) write: _			
#	key	name	#	key	name	
2__	COM	COMPONENT	17__	SHP	SHAPE 0__	
80__	FRE	FREE	18__	SHP	SHAPE 1__	
3__	PLN	VCC	19__	RES	RESIST 0__	
4__	SIG	SIG2	20__	RES	RESIST 1__	
5__	SIG	SIG3	21__	DRL	DRILL	
6__	SIG	SIG4	22__	DOC	DOCU 5__	
7__	PLN	GND	23__	DOC	DOCU 6__	
1__	SOL	SOLDER	24__	DOC	DOCU 7__	

In regard to this layer plan constellation, the router proceeds as follows:

- a. Component and solder side (layer 2 and 1) are routed together because the layer under the component side is empty and the layer under the solder side is a plane.
- b. Signal layer 4 is routed alone because it cannot be combined with any other signal layer.
- c. The two signal layers 5 and 6 are routed together.

section end

**6. CHANGE TRACK TECHNOLOGY**  
 sectionIVB6

After activating the function "Tracks" in the menu "Technology", a chart with the track technology of the current .adb-file appears in the status window. When creating a new .adb-file, the chart contains the values stored in the standard technology.

From the 255 track types in the menu, only the first 32 may be considered at present.

The widths of the 32 track types, the via types assigned to them and the solder resist mask widths of the standard technology may be changed if necessary. Max. 16 of these track types can be defined as routable.

Furthermore, you can specify for each of the 32 track types if it shall be shifted half a grid in the negative X and Y direction.

As tracks are also used for the presentation of SMD pins, the track ends and the respective solder resist masks can be defined in different shapes (ref. to "Track width core").

PCB: modify track-technology		Grid: 0.025/0.025		MS: in.		[986]	
fct: L		modify no.: 0__		(C) check, (W) write: _			
no.	track width		resist		add-clear.	attribute	via
	overall	core	overall	core			
1	0.011	0.0__	0.011	0.0__	0.0	_RAD__	1__
2	0.017	0.0__	0.017	0.0__	0.0	_RA__	1__
3	0.039	0.0__	0.039	0.0__	0.0	_RA__	1__
4	0.055	0.0__	0.055	0.0__	0.0	_RA__	2__
attributes: S=shifted R=routable A=active D=default							

**NO.**

Track type to which the values and parameters in this line apply.  
 Track type 0 is reserved for the creation of restricted area outlines.

**TRACK WIDTH OVERALL**

Defines the real width of the tracks.  
The router will use this size as base for the internal display of the track width.

**a. ROUTABLE TRACK TYPES**

The widths of the max. 16 routable track types are optimized for the router in the standard technologies (1/20", 1/40", 1/50", 1/60", 1/70", 1/80", ref. to the chapter "Standard technologies").

For a free definition of the routable track types, the following restrictions must be considered:

- 1) Minimum track width
  - for non-shifted tracks: 0
  - for shifted tracks : 1 x routing grid
- 2) If the widest non-shifted track is larger than (2 x routing grid) minus (clearance) then the smallest non-shifted track must be wider than (0.83 x routing grid) minus (clearance).

**b. NON-ROUTABLE TRACK TYPES**

The non-routable track types can only be digitized interactively. They must only be defined within the types 1-31.

**TRACK WIDTH CORE**

The value you enter here determines the width of the core, i.e. the 'rectangle part' of the track segments - and with it the shape of the track ends (in order to display different kinds of SMD-pins).

The following shapes are possible:

standard-SMD-pin

rectangle-pin rounded-rectangle-pin





The specifications have the following meaning:

- No entry --> standard-SMD-pin (the radius of the semi-circles representing the track ends = 1/2 total track width).
- Same value as in "Overall" --> rectangular SMD-pin (consists only of the core)
- A value > 0 and < the value in "Overall" --> rounded-rectangle-pin

The following diagram illustrates the described sizes:

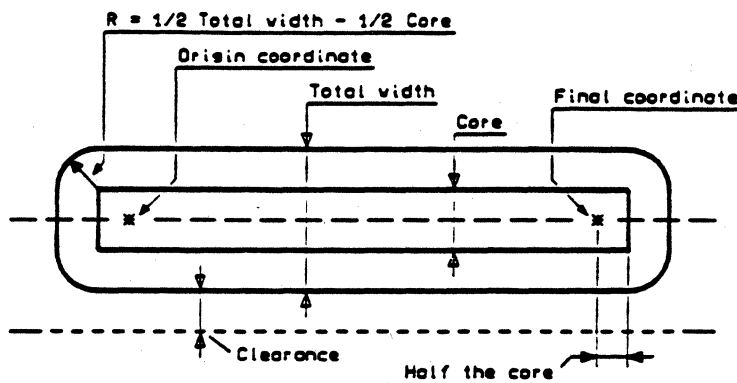


Figure 2

## RESIST OVERALL

Here, you define the width of the solder resist mask for tracks used for the presentation of SMD-pins.

## RESIST CORE

Here, you define the shape of the solder resist mask for tracks used for the presentation of SMD-pins (ref. to "Track width core").

## ADD-CLEARANCE:

If you specify a clearance here, it will be added to the general clearance (ref. to "Placement & routing") when routing this track type.

**ATTRIBUTES:**

**SHIFTED**

By entering "S", this track type will be shifted half a routing grid in the negative X and Y direction.

**ROUTABLE**

By entering "R", you specify that the respective track type may be routed (max. 16 types!) and will be used as symbol for the router-internal presentation of the pins, vias and non-routable tracks.

**ACTIVE**

By entering "A", you specify that the track type may be digitized and routed. The release for the router requires the specification "R" (= routable).

**DEFAULT**

By entering the attribute "D", the track type with the smallest routable width is defined.

The attribute "D" must be assigned to a track type but only to one.

**VIA**

The entered via type (defined in the menu "Modify via technology") will be used for changing layers when routing the track type defined in this line.

**HANDLING THE MENU**

1. Enter the number of the row (1-31) in which changes shall be made in "Modify no.", <CR>. Move the cursor to the desired column.
2. Now the changes can be made, <CR>.
3. By entering "W" for "Write", <CR>, the entries will be stored.
4. With <ESC><ESC>, you leave the menu.

section end

**7. VIA TECHNOLOGY**  
 sectionIVB7

After activating the function "Vias" in the menu "Technology", a chart with the via technology of the current .adb-file appears in the status window.

When creating a new .adb-file, the chart contains the values stored in the standard technology.

From the 255 via types in the menu, only the first 32 may be considered at present. The sizes of the 32 via types and their solder resist masks can be changed, if necessary.

Max. 16 of these via types can be defined as routable.

Furthermore, you can specify for each of the 32 via types if it shall be shifted half a grid in the negative X and Y direction and (for multilayers) if it shall go through all layers (drilled through) or only through one layer pair.

With the respective entries in "Via size overall" and "Via size core", the shape of the vias and the respective solder resist masks can be defined.

PCB: modify via-technology		Grid: 0.025/0.025		MS: in.		[987]	
fct: V		modify no.: 0__		(C) check, (W) write:		_	
no.	via-size		resist		add-clear.	attribute	d
	overall	core	overall	core			
1	0.039	0.0__	0.050	0.0__	0.0	_RD_____	1
2	0.055__	0.0__	0.067	0.0__	0.0	__D_____	1
3	0.0__	0.0__	0.0__	0.0__	0.0	_____	0
4	0.0__	0.0__	0.0__	0.0__	0.0	_____	0
attributes: S=shifted R=routable D=on-every-laye							all sizes in

NO.

Via type (pin type) to which the values and parameters in this line apply.

In the standard technologies, the via type 1 is used for changing layers, the via types 17 to 25 are used for pins.

Via type 0 is reserved for the router-internal display of technologically false vias (pseudo vias).

**VIA SIZE OVERALL**

Specifies the total diameter of the via; this size will internally be used by the router.

**VIA SIZE CORE**

The value you enter here corresponds to the width of the core. It determines the side length, i.e. the shape of the vias.

The specifications have the following meaning:

- No entry --> round via (pin)
- Same value as in "Overall" --> square via (pin)
- A value > 0 and < the value in "Overall" --> square via with rounded angles

The following diagram illustrates the described terms:

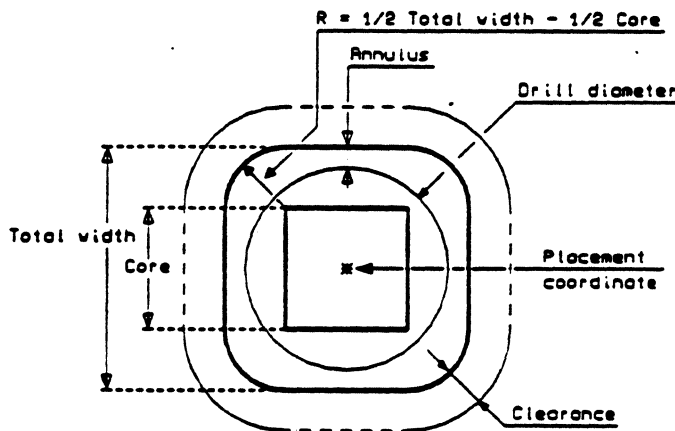


Figure 3

**RESIST OVERALL**

Here, you define the size of the solder resist mask for this via type.

**RESIST CORE**

Here, you define the side length, i.e. the shape of the solder resist mask (ref. to "Via size core").

## ADD-CLEARANCE:

If you specify a clearance here, it will be added to the general clearance (ref. to "Placement & routing") when routing this via type.

## ATTRIBUTES:

### Shifted

By entering "S", this via type will be shifted half a routing grid in the negative X and Y direction.

### Routable

By entering "R", you specify that the respective via type may be routed (up to 16 routable via types are allowed!).

If this attribute is not specified, the via type can only be digitized interactively and/or may be used for the pin presentation, but cannot be routed.

### On-every-layer

With the attribute "D", you can determine if the via type shall go through all layers (for pins and stubs from the solder to the component side) or only through one layer pair (buried via where possible).

## DRILL

The entered drill type (defined in the menu "Modify drill technology") will be used for the vias (pins) defined in this line (not relevant to the router).

## HANDLING THE MENU

1. Enter the number of the row (1-31) in which changes shall be made in "Modify no.", <CR>.
2. Now the changes can be made, <CR>.
3. By entering "W" for "Write", <CR>, the entries will be stored.
4. With <ESC><ESC>, you leave the menu.

section end

**8. DRILLS**  
sectionIVB8

PCB: modify drill-technology      Grid: 0.025/0.025      MS: in.      [988]

fct: D      modify no.: 0\_\_      (C) check, (W) write: \_

no.	diameter	annulus	plated
0	0.0	0.0	N
1	0.024	0.007	Y
2	0.031	0.012	Y
3	0.043	0.012	Y
4	0.0	0.0	N
5	0.0	0.0	N
6	0.0	0.0	N
7	0.0	0.0	N
8	0.0	0.0	N
9	0.0	0.0	N

After activating the function "Drill" in the menu "Technology", a chart with the drill technology of the current .adb-file appears in the status window.

When creating a new .adb-file, the chart contains the values stored in the standard technology.

From the 255 drill types in the menu, only the first 32 may be considered at present.

The sizes of the 32 drill types, their annulus widths and the specifications to the parameter "plated (Y/N)" can be changed, if necessary.

**HANDLING**

1. Enter the number of the line (1 to 31) in which you want to make changes in "Modify no.", <CR>. The cursor moves to the first input field of the selected row.
2. Now the changes can be made, <CR>.
3. By entering "W" for "Write" and pressing <CR>, the entries are stored.
4. With <ESC><ESC>, you leave the menu.

section end

### 9. REFERENCE COMPONENT

#### sectionIVB9

With this function, you can define user specific standard terminals for the current .adb-file that are stored in the 'background' and can be replaced by other terminals of existing components. When activating the function "Reference component", the menu "Place terminals" appears in the status window in which you can define your reference terminals with the usual procedure. These terminals are automatically assigned to the reference component which is named !PCBREFGRP and does not belong to the layout.

After the reference terminals have been defined, you can exchange terminals of any component with terminals of the reference component with the function "Exchange terminals" in the menu "Modification" (the terminals of the reference component remain unchanged):

- a. Activate the function "Terminal";
- b. Enter the name of the reference terminal in "Pin" and "!PCBREFGRP" in "Component" in the status window and press <CR>; the cursor moves to "Exchange".
- c. Click the terminal to be exchanged with <LMT>; the terminal gets the graphics and parameters of the respective reference terminal.

#### Note:

If you do not only want to use the terminals of the reference components for the current .adb-file, it can be generated in one of your standard technologies that must be read in COLUMBINE then. When creating a new .adb-file with this technology standard later, you can also use the respective reference component in the new file then.

section end

**C. STANDARD TECHNOLOGIES**  
sectionIVC1

**1. 1/20" standard technology**

Routing base : 1/960 inch  
 Routing grid : 1.270 mm = 0.050 inch  
 Clearance : 0.011 in.

In the 1/20" standard technology, a track of track type 1 may be routed between 2 IC-pins with a maximum diameter of 1.48 in..

The following chart shows the assignment of the track diameters and the via type to the routable track types

1 to 4. Figure 1 illustrates the assignment.

track type	track width (standard [in.])	via type
1	0.011	1
2	0.017	1
3	0.035	1
4	0.055	1

via type	size [in.]
1	.055

chart 1

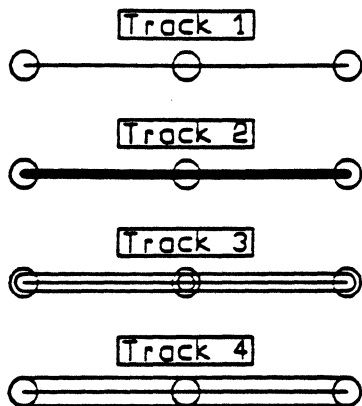


figure 1

section end



**2. 1/40" STANDARD TECHNOLOGY**  
 sectionIVC2

Routing base : 1/960 inch  
 Routing grid : 0.635 mm = 0.025 inch  
 Clearance : 0.011 in.

In the 1/40" standard technology, a track of type 1 may be routed between 2 IC-pins with a maximum diameter of 1.56 in..

The following chart shows the assignment of the track widths and the via type to the routable track types 1 to 4.

Figure 2 illustrates the assignment.

track type	track width (standard) [in.]	via type
1	0.011	1
2	0.017	1
3	0.039	1
4	0.055	2

via type	size [in.]
1	0.039

chart 2

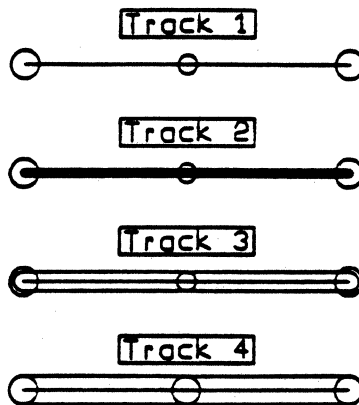


figure 2

section end

3. 1/50" STANDARD TECHNOLOGY  
sectionIVC3

Routing base : 1/1000 inch  
 Routing grid : 0.508 in. = 0.020 inch  
 Clearance : 0.008 in.

In the 1/50" standard technology, two tracks of type 1 may be routed between 2 pins with a maximum diameter of .055 in..

Chart 3 shows the assignment of track widths and the via type to the routable track types 1 to 4. Figure 3 illustrates the assignment.

track type	track width (standard [in.])	via type
1	0.010	1
2	0.015	1
3	0.027	1
4	0.050	1

via type	size [in.]
1	0.050

chart 3

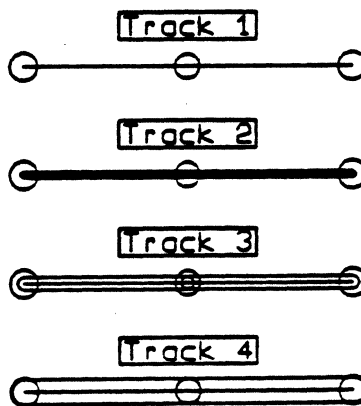


figure 3

section end

**4. 1/60" STANDARD TECHNOLOGY**  
 section IVC4

Routing base : 1/960 inch  
 Routing grid : 0.423 mm = 0.0166 inch  
 Clearance : 0.007 in.

In the 1/60" standard technology, 3 tracks of type 1 may be routed between 2 IC-pins; the maximum pin diameters is 1.06 in..

For a 'shifted' routing, 2 tracks of type 1 can be routed between 2 IC-pins with a maximum diameter of 1.53 in..

The following chart shows the assignment of the track widths and the via type to the track types 1 to 4.

Figure 4 illustrates the assignment.

track type	track width (standard [in.])	via type
1 *	0.008	1 *
2	0.022	1 *
3	0.027	1 *
4 *	0.040	1 *

via type	size [in.]
1	0.040

\*) These track and via types are shifted half a grid in the negative x and y-direction.

chart 4

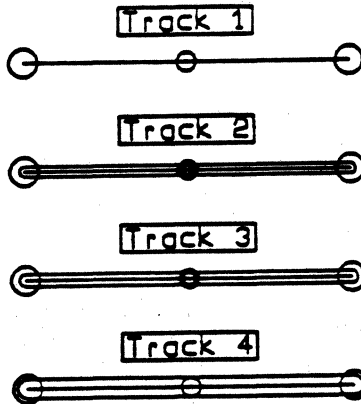


figure 4

## OFFGRID ROUTING

In order to digitize 2 tracks of type 1 between 2 IC-pins with a maximum diameter of .060 in., an offgrid routing must be executed.

Therefore, you have to enter "Y" in "Offgrid routing" in the menu "Placement & routing" (PCB part in CIS).

In this case, the routing area is shifted half a grid in the negative X and Y direction. In regard to the actual grid (on which the pins lie), the tracks will be routed with a shifting of half a grid.

Figure 5 shows that without the offgrid routing, you could not route 2 tracks between 2 pins of .060 in. because the clearance would be violated.

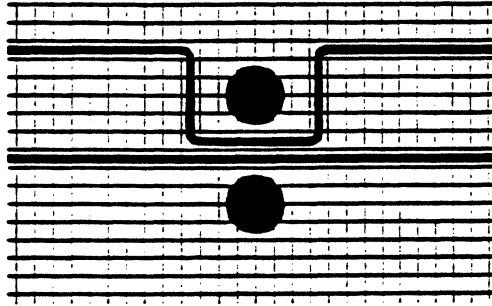


figure 5

section end

**5. 1/70" STANDARD TECHNOLOGY**

section IVC5

Routing base : 1/980 inch  
 Routing grid : 0.362 mm = 0.0142 inch  
 Clearance : 0.007 in.

In the 1/70" standard technology, two tracks of type 1 may be digitized between 2 pins with a maximum diameter of 1.63 in..

Chart 5 shows the assignment of the track widths and the via type to the routable track types 1 to 4.

track type	track width (standard [in.])	via type
1	0.007	1
2	0.013	1
3	0.026	1
4	0.036	1

via type	size [in.]
1	0.036

chart 5

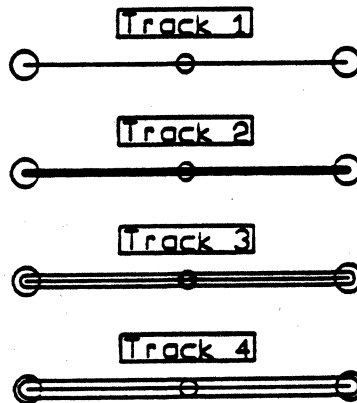


figure 6

section end

6. 1/80" STANDARD TECHNOLOGY  
sectionIVC6

Routing base : 1/980 inch  
 Routing grid : 0.317 mm = 0.0125 inch  
 Clearance : 0.006 in.

In the 1/80" standard technology, 3 tracks of type 1 may be routed between 2 pins with a maximum diameter of .056 in..

Chart 6 shows the assignment of the track widths and the via type to the track types 1 to 4.

Figure 7 illustrates the assignment.

track type	track width (standard [in.])	via type
1	0.006	1
2	0.017	1
3	0.026	1
4	0.031	1

via type	size [in.]
1	0.031

chart 6

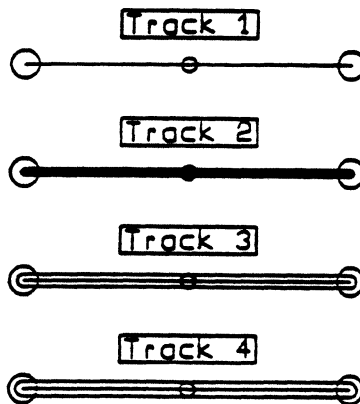


figure 7

section end

# ROUTER/.ldr-file

## sectionIVD

### D. LDR-FILE

The routing report file will be created automatically by the router. It is stored on the `gph`-directory and gets the name of the routed file and the extension `"ldr"`.

Beside the information that is output during the routing (iteration report, error messages, etc.), the `.ldr`-file contains a chart with the CIS technology sizes and their router-internal realization.

The following example shows a `.ldr`-chart which results from routing an `.adb`-file with 'shifted' 1/40" technology (widths and diameters in in.):

track 0: 0.000	1 0= 0.000		via 0: 0.000	1 0= 0.000	symbol 0: 0.000
track 1: 0.011	1 4= 0.011	r 1	via 1: 1.040 <	2 6= 1.040	symbol 1: 0.003
track 2: 0.000	1 0= 0.000		via 2: 0.000	1 0= 0.000	symbol 2: 0.005
track 3: 1.040 <	2 6= 1.040		via 3: 0.000	1 0= 0.000	symbol 3: 0.008
track 4: 0.055	3 2= 0.055		via 4: 0.000	1 0= 0.000	symbol 4: 0.0110
track 5: 0.000	1 0= 0.000		via 5: 0.000	1 0= 0.000	symbol 5: 0.013
track 6: 0.000	1 0= 0.000		via 6: 0.000	1 0= 0.000	symbol 6: 0.014
track 7: 0.000	1 0= 0.000		via 7: 0.000	1 0= 0.000	symbol 7: 0.018
track 8: 0.000	1 0= 0.000		via 8: 0.000	1 0= 0.000	symbol 8: 0.021
track 9: 0.000	1 0= 0.000		via 9: 0.000	1 0= 0.000	symbol 9: 0.023
track 10: 0.000	1 0= 0.000		via 10: 0.000	1 0= 0.000	symbol a: 0.026
track 11: 0.000	1 0= 0.000		via 11: 0.000	1 0= 0.000	symbol b: 0.028
track 12: 0.000	1 0= 0.000		via 12: 0.000	1 0= 0.000	symbol c: 0.031
track 13: 0.000	1 0= 0.000		via 13: 0.000	1 3= 0.000	symbol d: 0.034
track 14: 0.000	1 0= 0.000		via 14: 0.000	1 0= 0.000	symbol e: 0.036
track 15: 0.000	1 0= 0.000		via 15: 0.000	1 0= 0.000	symbol f: 0.0390
track 16: 0.000	1 0= 0.000		via 16: 0.000	1 0= 0.000	
track 17: 0.000	1 0= 0.000		via 17: 0.055	3 2= 0.055	
track 18: 0.000	1 0= 0.000		via 18: 0.055	3 2= 0.055	
track 19: 0.000	1 0= 0.000		via 19: 0.067	3 7= 0.067	
track 20: 0.000	1 0= 0.000		via 20: 0.099	5 0= 0.100 *	
track 21: 0.000	1 0= 0.000		via 21: 0.000	1 0= 0.000	
track 22: 0.000	1 0= 0.000		via 22: 0.000	1 0= 0.000	
track 23: 0.000	1 0= 0.000		via 23: 0.000	1 0= 0.000	
track 24: 0.000	1 0= 0.000		via 24: 0.000	1 0= 0.000	
track 25: 0.000	1 0= 0.000		via 25: 0.000	1 0= 0.000	
track 26: 0.000	1 0= 0.000		via 26: 0.000	1 0= 0.000	
track 27: 0.000	1 0= 0.000		via 27: 0.000	1 0= 0.000	
track 28: 0.000	1 0= 0.000		via 28: 0.000	1 0= 0.000	
track 29: 0.000	1 0= 0.000		via 29: 0.000	1 0= 0.000	
track 30: 0.000	1 0= 0.000		via 30: 0.000	1 0= 0.000	
track 31: 0.000	1 0= 0.000		via 31: 0.000	1 0= 0.000	

a b c d e f g h a b c d e f i j k

The columns have the following meaning:

- a: consecutive track and pin type number
- b: track or via diameter defined in CIS (nominal value)
- c: "<" indicates the shifted types
- d: number of symbols used for the display
- e: marking of the used symbol
- f: track or via width calculated by the router (actual value)
- g: "r" indicates the routeable track types that are really used in the layout
- h: the number specifies the via type that is used as 'stub' when digitizing the track type marked with "r"
- i: the calculated widths marked with "\*" differ from the nominal value
- j: marking of the symbol
- k: diameter of the symbol

**CIS - Initial situation:**

- the track types 1-4 and the via types 1 and 2 are defined as routable;
- track type 3 and via type 1 are shifted;
- in the netlist, only track and via type 1 is used;
- the board contains through-connected pins (realized with via type 17, 18, 19, 20) and SMD pins (realized with track type 4).
- Defined (= with sizes unequal zero) are the track types 1-4, 17, 18 and the via types 1, 2, 17-20.

With this chart, the defined sizes for track and via diameters may be compared to the sizes internally calculated by the router. If the difference between defined and calculated size is too large, you can check by means of the chart if it is possible to assign a better size to the respective track or via type.

**Notes:**

- Besides the technology chart and the information that is displayed on the screen during the routing, the .ldr-file contains some additional information (e.g. to the layer assignment, to the use of 'buried' vias, etc.) that does not appear on the screen.
- Tracks and vias defined in the CIS/PCB technology that are not used on the board (for digitizing variable and fixed tracks or for creating SMD or through-connected pins) are not considered by the router. Zero values appear in the report chart.

section end



section V

## V. MULTILAYER ROUTING

For the router the number of layers of a multilayer is not limited. But in the current version, a limitation to 16 layers is necessary because of the SCSCOM interface.

### A. PROCEEDING OF THE ROUTER (for a new start = option -n)

In multilayer routing, some of the connections are routed on the additional inner layers. In principle, the parameter control is the same for each layer pair, i.e. the first iteration on the outer layers coincides with the first iteration on the inner layer pairs as a rule; the same applies to the following iterations.

#### 1. Outer layers

The router always starts the multilayer routing on the outer layers, independent of the number of layers (if the layer plan constellation or the option "l:n" does not prevent this).

First all connections will be routed that may be realized on the outer layers according to the specifications in the netlist.

When using the standard control file `ldrout.CTL`, the router executes a **bus-routing** in the first iteration on the outer layers as a rule and changes then to the inner layers.

But if there are **SMD** components with netlist connections on the outer layers, only **stubs** (for changing layers) are placed on the outer layers in the first iteration. The busrouting is executed in a second iteration on the outer layers. Then the router changes to the inner layers.

#### 2. Inner layers

After the completion of the iterations on the outer layers, the router tries to distribute the tracks of the still missing connections on the inner layers. In the first iteration on the inner layers, a busrouting is executed, too.

On the inner layers, the procedure is the same as on the outer layers; the router tries to route the non-found connections on the current layer pair provided that these connection layers have been defined in the netlist.

The number of non-found connections on the outer layers must not always correspond to the number of calculated connections on the inner layers. The reason is that connections that have been specified for certain layers will not be calculated on the other layer pairs.

In general, it is possible that the number of missing connections and the number of the connection points differs because the number of missing connections applies to the complete PC board and the specification of the connection points only refers to the calculated connections.

The explanations in regard to the display of calculated connections do not apply to multilayer boards with SMD components (ref.to "Routing with SMD technology", page 51).

### B. IMPORTANT NOTES FOR MULTILAYER ROUTING

1. With the respective layer specification at the signal parameters in CIS, the router can be 'forced' to distribute nets on special layers (default for the routing layers is "\*", i.e. all layers).

By specifying the connection layer at the terminal parameters in CIS, you can define on which layer a terminal shall be connected (default is \*, i.e. all layers). If the specified terminal connection layers do not include the layer(s) on which the pin really exists (SMD pins only exist on the solder or component side), the router ignores the layer specification and tries to connect the pin where it really exists.

2. Vias go either through all layers (drilled through vias) or only through one layer pair (buried vias).

The router can only place buried vias on the inner layers if a via type that is defined as not drilled through has been assigned to the respective track type in CIS (attribute "D" must not be specified).

Note:

In the following cases, the router uses drilled through vias even if a not drilled through via type has been specified in the technology for the track type to be routed:

- a. for SMD plane connections,
- b. when creating 'stubs' for SMD connections,
- c. if layers are routed together that are not in the same layer pair set (e.g. if the layer plan contains empty layers).

3. Multilayer routing may be aborted on any layer pair. When restarting the program without the option "-l:n", the router will start again on the outer layers.

section end

section VI

VI. ROUTING WITH SMD TECHNOLOGY

1. General

When routing with SMD technology, the proceeding of the router differs from the previous description if the standard control file ldout.CTL is used.

2. Stub and busrouting

When using the standard control file ldout.CTL, the router proceeds as follows:

- In the first iteration on the outer layers, the necessary stubs are created.
- In the second iteration, the busrouting is executed on the outer layers; existing stubs can be used as connection possibilities.
- Then the router changes to the inner layers and executes the busrouting there; existing stubs can be used as connection possibilities.

Stub creation

One or several track segments of the SMD pins to be connected will be routed and a through connected via will be placed at the end. This effects that the connection of the SMD pin is now possible on all layers with this via.

Figure 1 shows a possible constellation for the placement of SMD-components. Pins 2 of the SMD components placed on the component and the solder side shall be connected. This connection can only be realized with a stub.

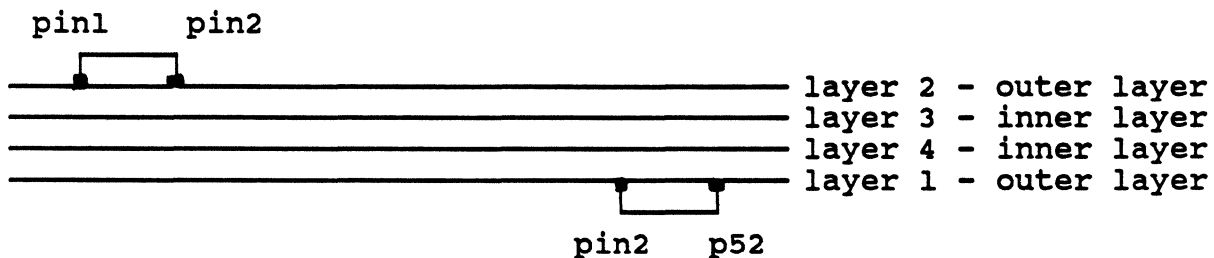


figure 1

In the first iteration, the router generates a track segment with a via at both SMD pins.

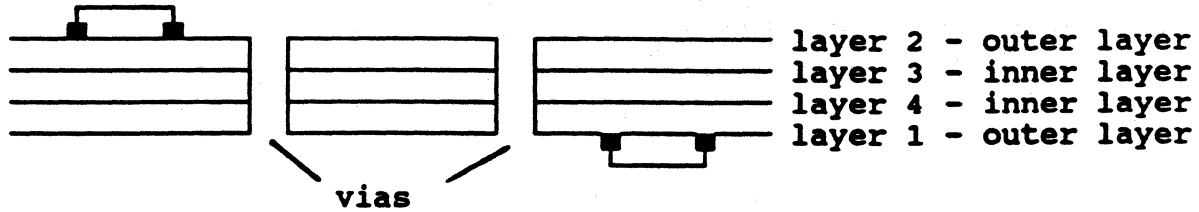


figure 2

The stubs created by the router will be kept until the connection has been realized. If a stub is not necessary for the routing, it will be deleted automatically.

section end

section VII  
**VII. WORKING METHOD OF THE ROUTER**

Before the routing, the pre-router executes a sorting of the nets and connections contained in the netlist.

Within a net, the router will only connect one pin with a target structure. This structure may be the next pin of that net or a track segment which has already been partially realized (see figure 1 and 2).



figure 1

Figure 1 shows a constellation in which IC1 pin 4 is the target structure. The net GND has i.a. the following connection points:  
 /GND: IC1/4, IC2/4, C1/1, C2/1, ...;

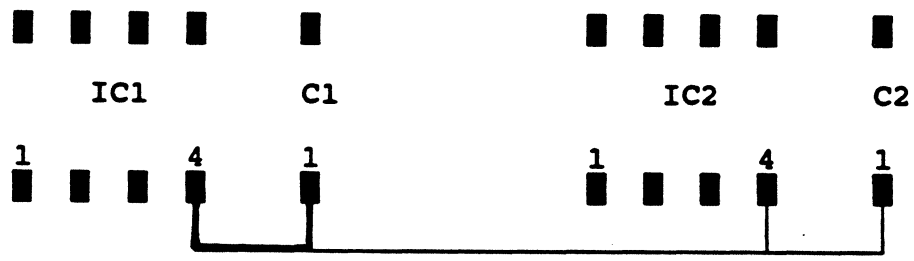


figure 2

Figure 2 shows the target structure (bold type) as a partially realized track (IC 1 pin 4 to C1 pin 1) of the same net.

The two steps illustrated in figure 1 and 2 will ordinarily be performed within one working step.

When defining fixed tracks, take care that the tracks of one net are connected to each other.

Figure 3 illustrates a wrong constellation of fixed tracks. In the net GND, IC1 pin 4 has been connected to C1 pin 1 and IC2 pin 4 to C2 pin 1; respectively by a fixed track. Because of this defined track course, the router assumes that all pins of the net are connected. The two digitized tracks will not be connected to each other; this results in an interruption of the net.

The track course in figure 2 is acceptable for the router. The non-connected pins of IC2 and C2 have been connected to the existing track without any problems.



figure 3

When all connections have been calculated once, a complete iteration is performed.

The several subsequent iterations will be performed with varying parameter sets (cost functions) in order to achieve a 100% routing with the help of different strategies.

The router operates according to the "rip up and re-route" principle. I.e. tracks which have already been routed but prevent the router from further distributions will be routed anew.

During the first iteration (after the router has been started with "-n"), a busrouting is performed. Thereby, only track courses are routed which can be realized without through connections.

To get an optimal busrouting for memory structures, take care that the distance between the pins of the neighboring components is not larger than 0.3 inch or 7.62 mm.

The priority direction for the routing is as follows:

- on the solder side in x-direction
- on the component side in y-direction.

The track types will be routed in descending order of their width (i.e. the widest first, the smallest last).

Minimize the number of "off-grid" pins as far as possible since the router blocks additional grid points when connecting these pins. Besides, "off-grid" pins are not connected to the center.

The router treats text fields on signal layers as blocking areas, i.e. no segments or vias must be placed in the text fields during the routing.  
For text fields on documentation layers, a via blocking can be defined in CIS.

Glue pads of SMD components are also via blocking areas for the router.

### **Plane connections**

For SMD-pins, the router connects track segments to the pin and at the end it generates a via to make a connection to the plane possible. If there is not enough place for the generation of a via, the connection can also be realized by connecting such a pin to an already existing via.

If a net contains through-connected pins defined as plane connection, the router may also realize a plane connection over these pins.

Through-connected pins defined for a plane connection are not considered by the router since these pins go through all layers.

section end



section VIII

### VIII. GENERAL INFORMATION

1. Before starting the router, a better power supply can be achieved by digitizing fixed tracks in a comb structure.
2. In order to achieve optimum routing of a memory structure, the distance between two component pins should not be larger than 0.3 inch or 7.62 mm.
3. The component placement in the layout also determines the routing success of LDROUT. The user should therefore critically observe the first iteration. The reason for an early stagnation may be a placement problem.
4. Offgrid pins should not be used as far as possible.
5. Only after the glossing, the layout is ready for production. Repeated glossings ('restart' with option "-g") increase the quality and minimize the number of through connections.

The provisional results that are created when the routing is aborted should also be smoothed for further processing by restarting the router with the option "-g".

section end

## sectionIX

**IX. ROUTER MESSAGES**

A difference is made between error messages, warnings and comments. Errors abort the program.

Warnings inform the user about problematic constellations; the program will not be aborted. But warnings may also indicate difficulties of the router. The comments are listed in the chapter "Output format, page 13".

At present, not all existing router messages are described here.

**A. ERROR MESSAGES****1. 'missing eof tag'**

The "end of file" marking is missing in the .gph-file.

**2. 'missing header'**

The "header" is missing in the .gph-file.

**3. 'duplicate header'**

The "header" appears twice in the .gph-file.

**4. 'illegal sequence of records'**

The .gph-file contains an unallowed sequence of records.

**5. 'missing gph-file-ID'**

The file has no .gph-file identification.

**6. 'not a V05-file'**

The .gph-file does not have the V04 Rev.5 format.

This message can only occur for .gph-files that were created on RT11 stations. As LDROUT can only work with the Rev.5 format, the .gph-file must be converted with GPHNEW on the RT11 station. Then the converted file can be transferred to the PRISMA system with Ethernet and can be routed.

**7. '.gph-file not converted'**

The .gph-file has V04 format and must be transformed into the Rev.5 format for the routing (ref. to item 6).

**8. 'no part in .gph-file'**

There are no components in the .gph-file.

**9. 'no net in .gph-file'**

The .gph-file contains no net.

**10. 'element outside area'**

One or several components are outside the routing area.

Elimination:

Get the component(s) in CIS/PCB and place them within the routing area.

**11. 'board too large'**

The work sheet for this board is too large.

Elimination:

Change the sizes in the technology (CIS/PCB).

The maximum size in the X and Y direction that can be processed by the router is:

routing base x 32767

(ref. to page 20).

**12. 'missing layer assignment'**

Connection layers are specified in the netlist that are not defined in the layer plan.

Elimination:

Define the respective layers in the technology (CIS/ PCB, function "Layer plan").

**13. 'seg-type not defined'**

For one or several segment types specified in the netlist no widths are defined in the track technology chart.

Elimination:

Enter the necessary widths for the respective track type in the technology (CIS/PCB, function "Tracks").

**14. 'via-type not defined'**

For one or several via types specified in the netlist, no widths are defined in the via technology chart.

Elimination:

Enter the necessary widths for the respective via type in the technology (CIS/PCB, function "Vias").

**15. 'via-type not routable'**

One or several via types that are assigned to routable track types in the technology, were not marked as routable in the via technology chart.

Elimination:

Enter the attribute "R" for the respective via type in the technology (CIS/PCB, function "Vias").

**16. 'too many routable seg-types'**

Too many track types are marked as routable in the track technology chart. At most 16 routable track types may exist.

Elimination:

Reduce the number of segment types to max. 16 with the attribute "R" in the technology (CIS/PCB, function "Tracks").

**17. 'too many routable via-types'**

Too many via types are marked as routable in the via technology chart. At most 16 routable via types may exist.

Elimination:

Reduce the number of via types to max. 16 with the attribute "R" in the technology (CIS/PCB, function "Vias").

**18. 'missing terminal-header'**

In the file, the "header" record is missing for one or several terminals.

**19. 'missing terminal-end'**

In the file, the "end" record is missing for one or several terminals.

**20. 'illegal subtree-assignment'**

The .gph-file contains unallowed subtree assignments.

**21. 'multiple track-widths in subnet'**

Subnets defined in the .gph-file contain several track widths. Each subnet, except for subnet 0, must only contain one track type.

**22. 'terminal is daisy-chain and terminator'**

A connection must not be defined as daisy-chain and terminator at the same time (only "either - or" allowed).

**23. 'track-width in higher subnet is larger'**

The assignment of the track types to the subnets is incorrect. The subnets with larger numbers must contain track types with smaller or the same track width, i.e. the track width for subsignal n+1 must be the same or smaller than the track width for subnet n.

**24. 'subnets not continuously defined'**

The subnet numbers have not been assigned correctly. The numbers must be defined continuously (there must not be 'gaps' in the number sequence!).

**25. 'no subnet 0 in signal'**

During the definition of subnets, the subnet number "0" has not been assigned. If a net contains subnets, subnet 0 must also exist!

**26. 'terminal-connection not routable'**

The track type specified for the connection of a terminal has not been defined as routable in the technology and can therefore not be routed.

**27. '.gph-file not sorted for router'**

If the pre-router has been switched off (option "-x"), the data from the .gph-file is not sorted for the router.

Elimination:

Restart the router with the option "-n".

**28. 'missing control-file .cst'**

The router does not find the file ldroust.cst that is absolutely necessary for the routing.

This file must be on the directory bin/sys. As the router searches it on the gph-directory, a respective "link" to the bin/sys directory must exist. When this error occurs, either the file ldroust.cst or the respective link is missing.

**29. 'no license to run program'**

The license for LDROUT does not exist.

### B. WARNINGS

**1. '... pins off-grid'**

The number of pins that do not lie on the grid is output.

section end  
end of file

# TABLE OF CONTENTS

## D. COLUMBINE

1.	Introduction . . . . .	1
2.	Starting the program . . . . .	6
3.	Library sections . . . . .	7
3.1	Data transfer . . . . .	8
3.1.1	Conditions for the data transfer . . . . .	8
3.1.2	Execution of the transfer . . . . .	10
3.2	Body . . . . .	13
3.3	Logic . . . . .	17
3.4	Function types . . . . .	20
3.5	Devices . . . . .	30
3.5.1	Device parameters for plotter . . . . .	31
3.5.2	Parameters for assembly machines . . . . .	35
3.5.3	Variants of assembly machine . . . . .	37
3.5.4	Parameters for tester . . . . .	39
3.5.5	Parameters for drilling machines . . . . .	41
3.5.6	Parameters for nc-routing machines . . . . .	43
3.6	Graphic macros (for plotter) . . . . .	45
REPORT	. . . . .	48
1.	Introduction . . . . .	48
2.	User specification . . . . .	48
3.	Area selection . . . . .	48
3.1	Function types . . . . .	50
3.2	Body . . . . .	50
3.3	Devices . . . . .	51
3.4	Macros . . . . .	52
3.5	Stock ids . . . . .	52
3.6	Logic . . . . .	53
BTOOLS	. . . . .	54
1.	Introduction . . . . .	54
2.	Handling . . . . .	54

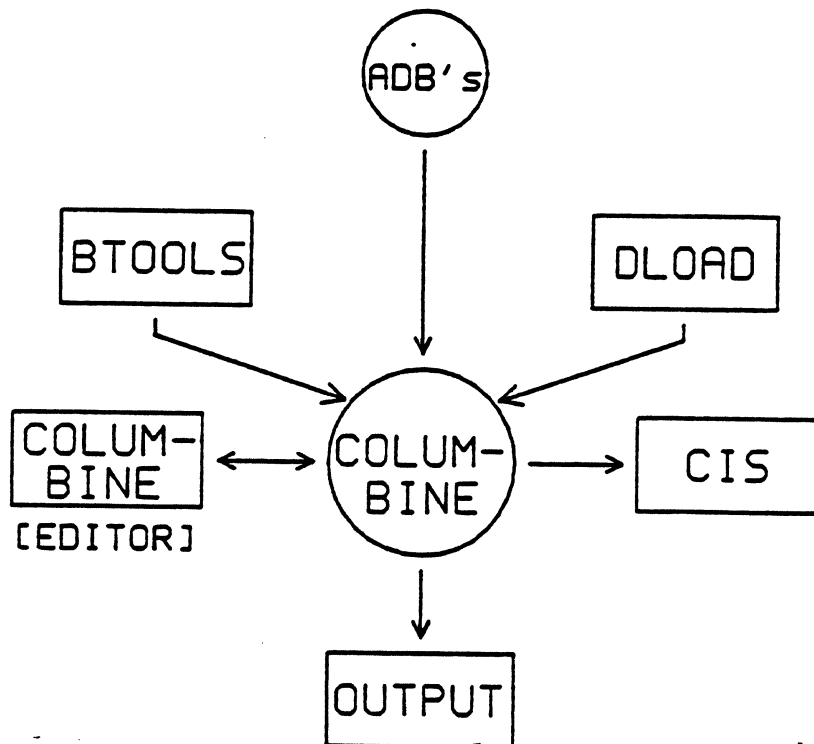
**COLUMBINE (Preliminary version)****1. INTRODUCTION**

COLUMBINE is the central library of the CALAY-PRISMA-system. It contains physical and logical component data, information about peripheral devices (insertion, tester, NC-machine, plotter --> together with information to pens/masks and macros for the board elements).

The program COLUMBINE belonging to the library is used for the data acquisition and change (=library editor).

As the programs CIS and OUTPUT automatically take access to COLUMBINE, the library to be used must always have this standard name.

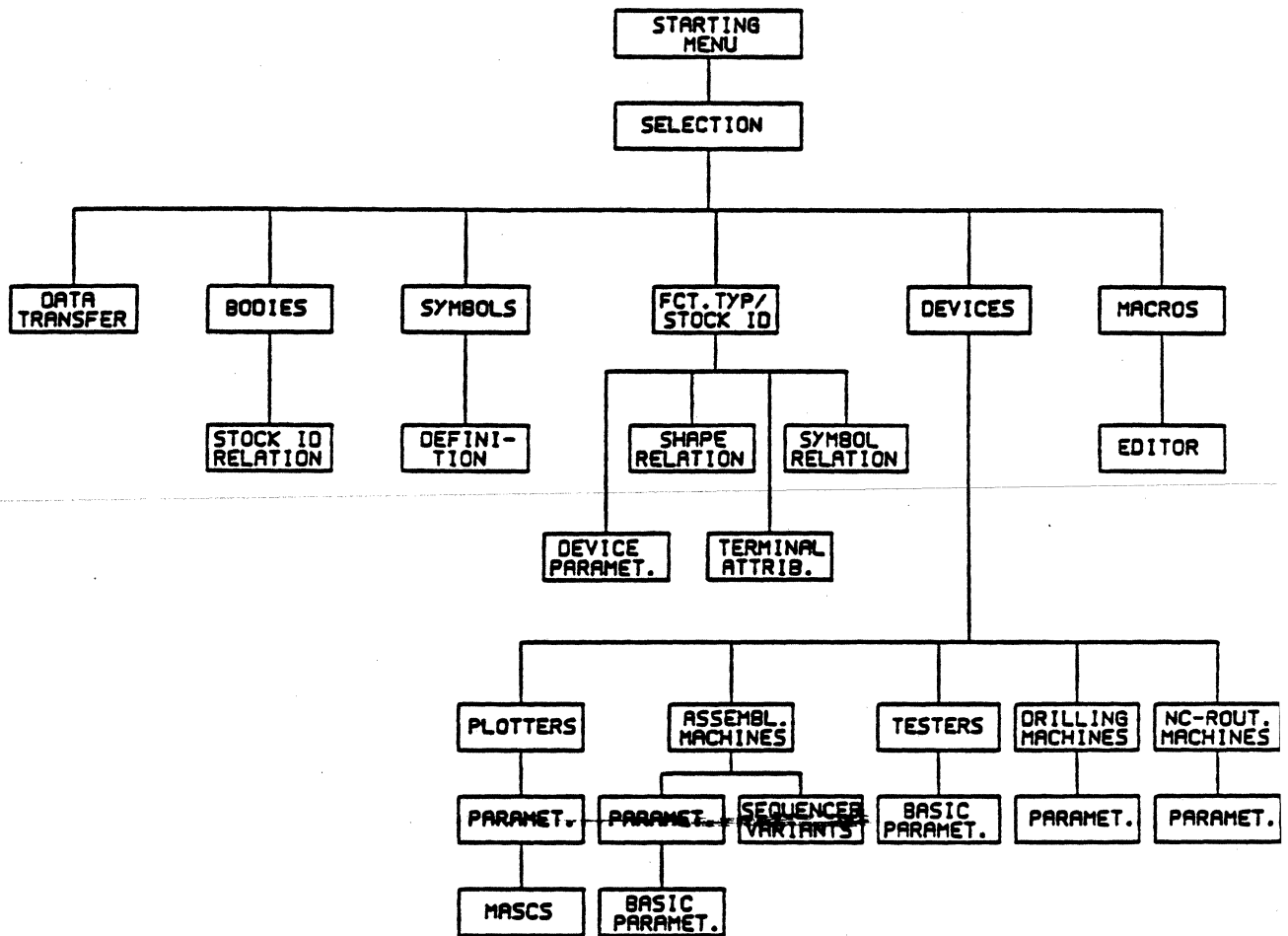
The following illustration shows the connections of COLUMBINE to other PRISMA programs and files.



The program BTOOLS is used for the creation of the library and the definition of the access rights for the user.

With the program DLOAD, external component data in the ASCII format is taken into the library.

The following diagram gives you an overview of the menus and their connections existing in the program COLUMBINE.

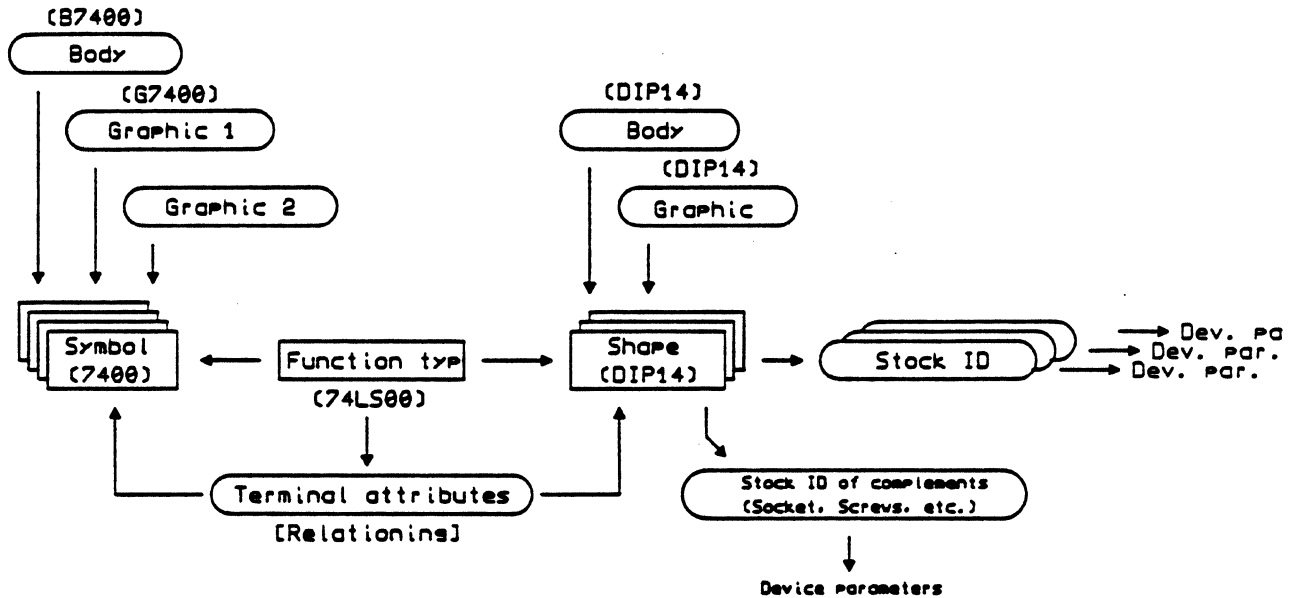


The library COLUMBINE is organized in 5 categories (areas) in which information is stored or can be stored:

- Data transfer** (read in physical and logical component information from .adb-files)
- Body** (information to shape and symbol bodies)
- Logic** (information to logical symbols for electrical function elements)
- Function types** (logical and physical component information)
- Devices** (device definition for plotter, insertion, tester, and NC-machines)



The following diagram shows how COLUMBINE is structured according to function types, shapes, symbols and stock IDs.



**TERM DEFINITIONS:**

The following terms, which are explained here, will be used in the PRISMA programs:

**1. Component**

Stands for the element existing on the PC board (not relevant to COLUMBINE).

**2. PCB name**

Placement name of a component on the PC board (e.g. IC1 -> not relevant to COLUMBINE).

### 3. Shape

Stands for the physical form of a component and consists of the following elements:

#### a. Shape body

Consists of the **shape terminals** and the **shape extension** (the area that the shape needs on the PC board).

The following information belongs to a shape terminal:

- terminal position (refers to the lower left corner of the shape);
- graphic display of the terminal on the respective signal layers in the layout.

notes:

The terminal graphics is determined by the assignment of a via type and/or a track type (incl. segment length) to the terminal effected in CIS; additionally it may contain graphic elements (lines and texts) that are digitized to the terminal by the user with the function "modify graphics" in CIS.

#### b. Shape graphic 1

Consists of the graphic display of the shape frame and the respective text and the additionally digitized graphic elements (lines and text - no terminals).

note:

At the moment, the graphic display of the frame is automatically created during the generation of shapes in CIS --> it corresponds to the extension of the shape.

With the function "modify graphic", the shape graphic consisting only of the frame can be completed or changed.

#### c. Shape graphic 2

This is an alternative graphic display that may be used instead of graphic 1 (e.g. DIP14F graphic as alternative to the DIP14 graphic);

### 4. SYMBOL

Stands for the logical side of a component and represents an electrical function element (e.g. 7400 or NAND). A symbol is defined with the following elements:

#### a. Symbol body

Consists of the **symbol terminals** and the **symbol extension** (the area that the symbol needs in the schematics).

note:

In CIS, the extension of the symbol is not visible when inserting standard symbols (e.g. the symbol 7400) from the library!

**b. Symbol graphic 1**

Consists of the graphic display of the symbol generated in CIS (lines and texts - no terminals).

note:

At the moment, the extension of the symbol is automatically transformed into the graphic display of a respective 'frame' (as marking for the extension) during the generation of symbols in CIS.

With the function "modify graphic", the symbol graphic consisting only of the 'frame' can be completed or changed.

**c. Symbol graphic 2**

Signifies the 'inverse' graphic to graphic 1.

**5. SCS name**

This is the placement name of a symbol in the schematics (not relevant to COLUMBINE).

**6. Function type**

Stands for a function specific standard component (e.g. the 4 NAND component 74LS00) that may exist in one or several 'variants'. According to the number of variants, a function type contains one or several relations between shape(s) (e.g. DIP14, DIP14F, DIP14X, SO14F) and symbol(s) (e.g. 4 NAND symbols 7400) and their terminals, as well as the assignment of a stock ID to each relation. If several relations exist for a function type, the definite selection of one 'variant' (a concrete component) can only be effected by the stock ID.

**7. Stock ID**

Definite identification criterion for a function type 'variant', i.e. for a concrete component (see item 6). The assignment of device parameters for insertion machines and testers is made with the stock ID.

## 2. STARTING THE PROGRAM

The data acquisition program for the library COLUMBINE is menu-driven and is started by 'clicking' the COLUMBINE icon or by entering COLUMBINE.

After you have entered a function, <CR>, you get to a subordinate menu.

The function "0" (=end) finishes your work in a menu and stores the data. You get back to the superior menu or finish the program.

With <ESC><ESC>, you leave the menu without storing your entries and get to the superior menu.

After the program has been started, the following menu appears:

COLUMBINE	Library editor	Version:	2
-----------	----------------	----------	---

---

User	:	_____
Password	:	_____

---

notes:

- In order to avoid an inconsiderate or unauthorized alteration of the library, you have to identify yourself as authorized person with your name and your password to change the library contents.

The program only accepts specifications that are identical with the ones defined in BTOOLS.

It distinguishes between uppercase and lowercase!

After you have entered the user name, you have to press the "Cursor down" key to enter your password in the next line. The password will not be displayed on the screen.

The access rights are determined in the program BTOOLS.  
They can be restricted to different library sections and functions.

- After you have stored your entries with <CR>, you get to menu 14.

**3. LIBRARY SECTIONS**

In menu 14, you select the library section to be modified by entering the respective number.

On the right side of the menu, a library statistics appears from which you can determine the free library capacity.

COLUMBINE	Library editor	Version:	14
-----------	----------------	----------	----

creation date	:	_____	free blocks	:	___	(blocks)
last change	:	_____	deleted blocks	:	___	(blocks)
			number of entries	:	___	
version	:	_____	deleted entries	:	___	

**Functions**

(1) Data transfer	(3) Bodies	(5) Devices
(2) Function types	(4) Symbols	(6) Macro editor (plotter)
(0) End		

Function : >0<

## 3.1 DATA TRANSFER

With this function, physical and/or logical component data can be taken into the library from an .adb-file.

### 3.1.1 CONDITIONS FOR THE DATA TRANSFER

In order to execute the data transfer, the release of the physical and/or logical component or symbol data for the library must have been effected in CIS in the menus "Change component attributes" and/or "Change group attributes".

Physical and logical component data can only be transferred together if a respective 'relation' has been executed in CIS before and a function type has been assigned.

#### Procedure for the release:

##### 1. Release a shape (layout)

- a. Touch the respective component in the layout submenu "Modification" and activate "mod. param.";
- b. Enter -Y- in "to lib." in the menu "Change component attributes".

note:

A shape name must be assigned to the component in "Shape" --> during the data transfer, the shape information will be stored in COLUMBINE with this name!

- c. press <CR>.

##### 2. Release a symbol (schematics)

- a. Touch the respective symbol in the schematics submenu "Modification" and activate "mod. param.";
- b. Enter -Y- in "transfer symbol body" and the name with which the symbol body shall be stored in the library in "symbol body" (e.g. BNOR) in the menu "Change symbol attributes";
- c. press <CR>.

3. Release a related function type (schematics and layout)

- a. Shape and symbol must be released in the way described in item 1 and 2;
- b. Between the terminals of the symbol and the terminals of the component, a 'relation' must be executed (see next page);
- c. On the layout side, you have to assign a function type specification to the component in "Function type" in the menu "Change component attributes".
- d. On the layout side, you have to enter -Y- in "rel. to lib." in the menu "Change component attributes".
- e. On the schematics side, you have to enter -Y- in "transfer relation" in the menu "Change symbol attributes".

**Relation shape <--> symbol to a function type**

If a component has been generated on the layout side and a respective symbol on the schematics side, the relation can be executed in the schematics part. In order to do so, proceed as follows:

- a. Activate the function "misc." in the main menu of the schematics part and the function "relation" in the appearing submenu; subsequently, the menu "Enter SCS-PCB relations" and the function menu for the processing of the schematics appear.
- b. In "PCB name", you enter the name of the component to be related from the layout side and press the <CR> key or the left mouse button (called <LMB> in this description);  
  
the component is now 'attached' to the digitizer and can be released with <LMB> (sensibly near the respective symbol);
- c. Move the crosshair to a terminal of the component to be related and press <LMB>, then you go to the respective terminal of the symbol and press <LMB>; proceed in the same way with all other terminals to be related.

note:

When 'clicking' a terminal, the name and the placement coordinate of this terminal appear in the menu;

- d. If you want to relate another component, activate the function "get", enter the name of the desired component in "PCB name" and relate the terminals of the component with those of the symbol with the same procedure;
- e. After the relations have been finished, you can continue your work with the functions of the schematics menu;

note:

When picking up the symbol after the relations have been executed, the respective component name (PCB name) appears in brackets behind the symbol name (SCS name).

**3.1.2 EXECUTION OF THE TRANSFER**

After you have activated the function (1) "Data transfer" in menu 14, the following menu appears:

<b>COLUMBINE</b>	<b>Library editor</b>	<b>Version:</b>	<b>47</b>
------------------	-----------------------	-----------------	-----------

**DATA TRANSFER**

---

file name : \_\_\_\_\_ .adb            (1) Shapes   (2) Symbols

name : \_\_\_\_\_                    > <   (3) Function types

(1) Data transfer from .adb-file (0) End
---

Function : > <

**File name**

Specify the name of the .adb-file from which the data is to be transferred. After the input of the file name and <CR>, the file is read.

**Name**

Depending on the specification ("1", "2" or "3") in the input field > <, all (\*), only the specified (..) or single (complete name) shapes, symbols or related function types with the respective information will be taken into the library.

**Function**

- "1" activates the data transfer
- "0" finishes this menu and you return to menu 14.

**A. Read in shapes**

- a. In "Name", you specify the shape body name(s) (e.g. DIP\*) entered in CIS in the menu "Change component attributes" in "Shape".
- b. Enter "1" for "Shapes".
- c. Enter "1" in "Function".



**result:**

For all shapes of the specified .adb-file whose name starts with "DIP", the following elements are taken into the library (--> ref. to page 4):

- the shape body (name and extension of the shape body and terminal information);
- the graphic (shape frame)
  - > at the moment, the 'graphic' has the same name as the shape body;
- the function type specification, as far as it has been assigned to the components in CIS in "Function type" in the menu "Change component attributes".

**note:**

If a shape with the same name already exists in COLUMBINE, the inquiry "transfer?" appears for each existing element of the shape (body name, terminals, graphic of the frame, function type specification)

--> if you enter "Y", the respective shape element in COLUMBINE will be replaced by the 'new' one from the .adb-file (overwritten);

--> when entering "N", the element from the .adb-file will not be overtaken.

**B. Read in symbols**

- a. In "Name", you specify the name(s) of the symbol body (bodies) (e.g. B74\*) entered in CIS in the menu "Change symbol attributes" in "symbol body";
- b. Enter "2" for "Symbols";
- c. Enter "1" in "Function".

**result:**

For all symbols of the specified .adb-file whose reference specification (=symbol body specification) starts with "B74", the following elements are taken into the library (--> ref. to page 5):

- the symbol body (name and extension of the body and terminal information);
- the graphic display of the symbols
  - > the 'graphic' name starts with "G" and from the second character on it is identical with the name of the symbol body.
- the specification assigned to the symbols in CIS in the menu "Change symbol attributes" in "Symbol".

**note:**

If a symbol with the same name already exists in COLUMBINE, the inquiry "transfer?" appears for each existing element of the symbol (body name, terminals, graphic and symbol specification)

--> if you enter "Y", the respective element in COLUMBINE will be replaced by the 'new' one from the .adb-file (overwritten);

-->when entering "N", the element from the .adb-file will not be overtaken.

### C. Read in related function types

- a. In "Name", you specify the name(s) of the function type(s) (e.g. 74\*) entered in CIS in the menu "Change component attributes" in "Function type";
- b. Enter "3" for "Function types";
- c. Enter "1" in "Function".

#### result:

For the function types of the .adb-file whose name starts with "74", the following elements are taken into the library (--> ref. to page 5):

#### a. from the layout part

- the shape body (name and extension of the shape and terminal information);
  - the shape graphic (shape frame)
- > at the moment, the 'graphic' has the same name as the symbol body.
- the function type specification assigned to the components in CIS in the menu "Change component attributes" in "Function type".

#### b. from the schematics

- the symbol body (name and extension of the body and terminal information);
  - the graphic display of the symbols
- > the 'graphic' name starts with "G" and from the second character on it is identical with the name of the symbol body.
- the symbol specification assigned to the symbols in CIS in the menu "Change symbol attributes" in "Symbol".

note: see item 1 and 2.

### 3.2 BODY

With the functions of the following menu, you can duplicate, rename, list and delete shape and symbol bodies existing in the library. In addition, device parameters and stock IDs, previously defined in menu 3, can be assigned to a shape for its additional elements (e.g. socket, screws, etc).

COLUMBINE	Library editor	Version:	45
-----------	----------------	----------	----

#### SHAPE BODIES or SYMBOL BODIES

Body name : \_\_\_\_\_ > < (1) shapes (2) symbols

- |            |                                       |
|------------|---------------------------------------|
| (1) Names  | (5) Device parameters (only shapes)   |
| (2) Copy   | (6) Stock ID assignment (only shapes) |
| (3) Kill   |                                       |
| (4) Rename |                                       |
| (0) End    |                                       |

Function : 0

#### (1) Names

When you have entered "1" for "Shapes" or "2" for "Symbols", this function lists all (\*) or the specified (..\*) names of the shape or symbol bodies existing in COLUMBINE.

By 'clicking' the function "next" with the left mouse button (called <LMB> in this description), you get to the next page, with "previous", you get back and with "quit", you finish the listing.

#### (2) Copy

Duplicates the specified shape body (selection "1") or the specified symbol body (selection "2"). You are asked for the name of the new shape or symbol body. The new element contains the information of the old shape or symbol body (no 'graphic!').

### (3) Rename

Renames the specified shape or symbol body. You are asked for the new name of the shape or symbol body. The new element still contains the assigned information (no graphics!). The old name of the shape or symbol body does no longer exist.

### (4) Kill

Deletes the specified name of the shape or symbol body together with the assigned information from the library.

### (5) Device parameters

For the shape type specified in "Body name", 'mechanical' device parameters for assembler and/or tester can be created or existing device parameters can be changed, listed, copied or deleted.

note:

The 'mechanical' device parameters 'belong' to the shape from now on, i.e. when assigning this shape to a function type, the shape specific device information will automatically be overtaken and need not be entered separately with the stock ID. (Only the function type specific ('electrical') device information must be defined with the stock ID --> ref. to page 24).

The following menu appears:

COLUMBINE	Library editor	Version:	162
-----------	----------------	----------	-----

```
DEVICE PARAMETER      Mode :  
(E) Edit, (K) Kill, (L) List, (C) Copy,   select: >E<  
-----  
device      :          # :    type :    (2 = Ass., 3 = Test)  
shape       :  
-----
```

### (L) List

Lists the device parameters assigned to the specified shape.

First the parameters of one device will be listed.

After pressing any button, the parameters (if defined) of further devices will be listed.

With <CR>, you finish the listing and with <ESC><ESC>, you get back to the upper part of the menu.

**(E) Edit**

With this function, you can assign assembling or tester data to the specified shape or change the data. In "Device name", you enter the name of the assembling automat or tester for which the parameters shall be created or changed (the device must be defined in the library area "Device"). In "Type", you enter the device type (2 = assembling automat, 3 = tester), <CR>.

Now you can enter the shape-specific parameters for the respective machine.

You must only use the kind of characters determined during the device definition in "Type"; the number of characters must not be greater than the number entered in "Len" during the device definition.

With <CR>, the current parameters are stored and with <ESC> <ESC>, you get back to the upper part of the menu.

**(K) Kill**

Deletes the device parameters of the specified shape for the device, whose name and type you have entered in "Device name" and "Type", <CR>. With <ESC><ESC>, you get back to the upper part of the menu.

**(C) Copy**

The device parameters of the respective shape will be assigned to another device (tester or assembling automat). First you enter the device for which the parameters already exist, <CR>, then you enter the device which shall get the same parameters.

With <ESC><ESC>, you get back to the upper part of the menu.

**(6) Stock ID assignment**

The stock IDs that may be assigned to a shape in this menu do not stand for the shape itself but for the 'additional elements' like socket, screws, etc.

You can only use stock IDs created in menu 3 to which device parameters for the assembling were assigned in menu 162 (ref. to page 22 and page 24).

The following menu appears:

COLUMBINE	Library editor	Version:	140
-----------	----------------	----------	-----

Shape : _____
Index : >0< Stock ID : _____

- 0
- 1
- 2
- 3
- 4
- 5
- 6
- 7
- 8
- 9

By specifying an index number (0 to 9), you can assign up to 10 stock IDs to a shape. Each stock ID stands for one of the 'additional elements' of the shape.

After entering the index number, <CR>, you get to the field for the input of the stock ID.

After entering the stock ID, <CR>, the assignment appears below the menu and the next assignment can be made.

With <ESC><ESC>, you leave the menu.

### 3.3 LOGIC

With the functions of the following menu, you can duplicate, rename, list and delete symbols existing in the library. In addition, new symbols can be defined (with the assignment of symbol bodies, 'graphics', 'inverse graphics' and function types) and the assignments of existing symbols can be changed.

COLUMBINE	Library editor	Version:	46
-----------	----------------	----------	----

#### LOGIC

Symbol name : \_\_\_\_\_

- |            |                |
|------------|----------------|
| (1) Names  | (5) Definition |
| (2) Copy   |                |
| (3) Kill   |                |
| (4) Rename |                |
| (0) End    |                |

Function : 0

#### (1) Names

This function lists all (\*) or the specified (..\*) symbol names of the library; by 'clicking' the function "next" with <LMB>, you get to the next page, with "previous", you get back and with "quit", you finish the listing.

#### (2) Copy

Duplicates the specified symbol. You are asked for the name of the new element. The new symbol contains the information (symbol body, graphics and function types) of the old one.

#### (3) Rename

Renames the specified symbol. You are asked for the new name of the symbol. It still contains the assigned information. The old symbol name does no longer exist.

#### (4) Kill

Deletes the specified symbol together with the assigned information from the library, provided that only one assignment exists for this symbol. If more assignments exist, they must be deleted one by one in menu 171.

**(5) DEFINITION**

In the menu to this function, you can assign (existing!) symbol bodies, graphics and function types to a symbol name, as well as delete or change existing assignments.

COLUMBINE	Library editor	Version:	171
-----------	----------------	----------	-----

Symbol name: _____	(E) Edit (L) Kill >E< Select	(N) New (+ -) Step
-----------------------	------------------------------------	-----------------------

E    Function type                    Body    Graphic 1    Graphic 2

The assignments are listed below the menu. The symbol name automatically appears in the menu.

>N< New

An additional menu part appears, in which you can assign a body, a graphic ("graphic 1"), an 'inverse' graphic ("graphic 2") and a function type to the specified symbol:

Function	:	_____	Alias name	:	_____
Body	:	_____	2	:	_____
Graphic 1	:	_____			

C0 :		<	>	>	<	B0 :	LS	( )
C1 :	voltage	<	123456	>	<	B1 :	S	( )
C2 :		<		>	<	B2 :	HC	( )
C3 :		<		>	<	B3 :	HCT	( )
C4 :		<		>	<	B4 :	F	( )
C5 :		<		>	<	B5 :		( )
C6 :		<		>	<	B6 :		( )
						B7 :	advanced	( )

- > In "Body" and "Graphic 1", specifications must be made;
- > In "Function type" and "Graphic 2", specifications may be made.

Make your entries in the respective fields and press <CR>. Then you get to the lower part of the menu where you can additionally specify 'dynamic parameters' (ref. to BTOOLS, function "Define user") for the symbol. The 'dynamic parameters' represent electrical selection criteria.



In Cn (n=0-6), electrical characteristics defined in BTOOLS appear that are connected with area specifications (e.g. voltage check, etc). In the brackets, you specify the respective area variants by entering a letter or a number (e.g. "123456" in "C1"); In "> <", you can specify the desired area by entering one of the characters.

In Bn (n=0-7), electrical characteristics defined in BTOOLS appear that may exist or not (e.g. "LS", "HCT", "F", "S", "advanced" etc). If the characteristic shall exist for the current symbol, put an X in the respective parenthesis.

After pressing <CR>, you get back to the menu part where you may enter the symbol body, graphic and function type for further assignments.

Press <ESC><ESC> to finish the procedure. The symbol name and up to ten assignments will be displayed in the lower part of the screen.

With <ESC><ESC>, you get back to the upper part of the menu.

#### **>E< Edit**

The specifications assigned to a symbol can be changed. Therefore, you have to mark the respective assignment line in the lower part of the screen. If more than ten assignments exist for a symbol and if the respective assignment is not displayed on the screen, you have to use the functions "+" or "-" to get to the next or the previous page.

#### **>L< Kill**

With this function, specifications assigned to a symbol can be deleted. After entering "3" in "Select", you have to mark the respective assignment line in the lower part of the screen and press <CR>. If more than ten assignments exist for a symbol and if the desired assignment is not displayed on the screen, you have to use the functions "+" or "-" to get to the next or the previous page.

If all assignments to a symbol have been deleted, the respective symbol name does not appear when listing the names (function "1") in menu 46.

#### **>+<**

If more than ten assignments exist, you can turn to the next page with "+", i.e. up to ten further assignments will be listed in the lower part of the screen.

#### **>-<**

You can turn to the previous page (ref. also to >+<).

**3.4 FUNCTION TYPES**

With the functions of the following menu, you can define, duplicate, rename, list and delete function types and stock IDs. Device parameters for assembler and tester may be assigned to the stock IDs.

In order to define a function type, shapes (body + graphic 1) and symbols (body + graphic 1) that shall be assigned to the function type must exist in COLUMBINE!

The definition is made by specifying a function type name to which first one or several shapes (possibly with the defined stock ID), then one or several symbols, and subsequently the respective terminal attributes may be assigned. (The complete definition of a function type corresponds to the relation symbol <--> component in CIS.)

In this menu, the definition of stock IDs for additional elements to shapes (socket, screws, etc.) is also effected (ref. to page 17). Therefore, you have to enter the stock ID specification in "Function type or stock ID", put an X in ">\_< stock ID?", enter "5" in "Function" and define the device parameters (see below).

After you have assigned a stock ID to a function type, you can also process it with the help of the assigned stock ID instead of the function type specification. Therefore you enter the stock ID in "Function type or stock ID" and put an X in the field ">\_< stock ID?".

COLUMBINE	Library editor	Version:	3
-----------	----------------	----------	---

**FUNCTION TYPES**

Function type or stock ID : \_\_\_\_\_ >\_< stock ID ?

- |                       |  |
|-----------------------|--|
| (1) Names             | (5) Device parameters (only with stock Ds) |
| (2) Copy              | (6) Shape assignment (3) Kill              |
| (7) Symbol assignment | (4) Rename (8) Terminal                    |
|                       | attributes                                 |
| (0) End               |  |

Function : 0

**(1) Names**

lists all (\*) or specified (..\*) function types of the library (by 'clicking' the function "next" with <LMB>, you get to the next page, with "previous", you get back and with "quit", you finish the listing).

By entering a respective specification and putting an X in the field ">\_< stock ID ?", the desired stock IDs and the names of the function types to which they are assigned will be listed.

**(2) Copy**

transfers the shape and symbol assignment, as well as the terminal attributes of the specified function type to a new one, for which you have to enter a name. The information of the old function type concerning tester and assembling automat and stock ID assignment will not be assigned to the new one!

**(3) Kill**

With this function, you can only delete function type names to which a shape, but no stock ID is assigned.

If a shape and a stock ID are assigned to a function type, you have to delete the stock ID (possible in this menu) or the complete assignment (possible in menu 151) first.

In order to delete a function type to which several stock IDs (and shapes) are assigned, you have to delete all assignments in the submenu to the function "Shape assignment" first.

**(4) Rename**

renames the specified function type. You are asked for the new name of the element. The function type still contains the shape and symbol assignment and the terminal attributes of the specified function type.

The information concerning tester and assembling automat, as well as stock ID assignment is still assigned to the function type. The old name does no longer exist.

**(5) Device parameters**

With this function, parameters for assembling automats and tester can be created, changed, copied, deleted or listed.

This function can only be called when specifying the respective stock ID and putting an X in the field ">\_\_< stock ID ?".

notes:

- a) The stock ID can stand for a function type with an assigned shape or for an 'additional element' of a shape:
  - The stock ID assignment for a function type with a specified shape is made in connection with the shape assignment --> ref. to function "6".
  - The stock IDs for the 'additional elements' to the shapes are defined here in menu 3 (ref. to page 22) The stock ID assignment for the 'additional elements' to the shapes (in menu 140) can only be effected after the respective stock IDs have been defined in menu 3 and device parameters have been assigned to them with function (5).

## COLUMBINE/Function Types

---

- b) If device parameters have been assigned to a shape in menu 45, the shape specific device information will be overtaken, i.e. it need not be entered here (ref. to page 15)!

Therefore, the following menu appears:

COLUMBINE	Library editor	Version:	162
-----------	----------------	----------	-----

DEVICE PARAMETER            Mode :

(E) Edit, (K) Kill, (L) List, (C) Copy,    select: >E<

---

device	:	#	:	type	:	(2 = Ass., 3 = Test)
shape	:					

---

**(E) Edit**

With this function, you can assign assembling or tester data to the specified shape or change the data. In "Device name", you enter the name of the assembling automat or tester for which the parameters shall be created or changed (the device must be defined in the library area "Device"). In "Type", you enter the device type (2 = assembling automat, 3 = tester), <CR>.

Now you can enter the machine-specific parameters for the respective device (you may only use the kind of characters determined during the device definition in "Type"); the number of characters must not be greater than the number entered in "Len" during the device definition.

With <CR>, the current parameters are stored and with <ESC> <ESC>, you get back to the upper part of the menu.

**(K) Kill**

For the specified shape, the assigned device parameters of the tester or assembling automat whose name and type you have entered in "Device name" and "Type" will be deleted, <CR>.

With <ESC><ESC>, you get back to the upper menu parts.

**(L) List**

First the parameters of one device will be listed for the shape specified with the stock ID. After pressing any button, the parameters (if defined) of further devices will be listed.

With <CR>, you finish the listing and with <ESC><ESC> you get back to the upper part of the menu.

**(C) Copy**

The defined device parameters of the specified shape are copied to another device (tester or assembling automat). First you enter the device for which the parameters already exist, <CR>, then you enter the device that shall get the same parameters.

With <ESC><ESC>, you get back to the upper menu parts.



In Cn (n=0-6), physical component characteristics defined in BTOOLS appear that are connected with area specifications (e.g. voltage check, tolerance, temperature, etc). In the brackets, you specify the respective area variants by entering a letter or a number (e.g. "C1: TOLERANCE <1234\_\_\_\_\_>" means that the tolerance areas 1,2,3 and 4 are valid). In ">\_\_<", you can specify the desired area by entering one of the characters.

In Bn (n=0-7), component characteristics defined in BTOOLS appear that may exist or not (e.g. "PLCC", "SO", "socket", "standard", etc). If the characteristic shall exist for the current symbol, put an X in the respective parenthesis.

After pressing <CR>, you get back to the menu part where you may enter the symbol body, graphic(s) and stock ID for further assignments.

Press <ESC><ESC> to finish the procedure. The symbol name and up to ten assignments will be displayed in the lower part of the screen.  
With <ESC><ESC>, you get back to the upper part of the menu.

### >E< Edit

The specifications assigned to a symbol can be changed. Therefore, you have to mark the respective shape assignment in the lower part of the screen. If more than ten assignments exist for a function type and if the respective assignment is not displayed on the screen, you have to use the functions "+" or "-" to get to the next or the previous page.

### >L< Kill

With this function, specifications assigned to a function type can be deleted. Therefore, you have to mark the respective assignment in the lower part of the screen. If more than ten assignments exist for a function type and if the desired assignment is not displayed on the screen, you have to use the functions "+" or "-" to get to the next or the previous page.  
If all assignments to a function type have been deleted, the respective function type name does not appear when listing the names (function "1") in menu 3.

### >+<

If more than ten assignments exist, you can turn to the next page with "+", i.e. up to ten further assignments will be listed in the lower part of the screen.

### >-<

You can turn to the previous page (ref. also to >+<).

## (7) Symbol assignment

With this function, you can assign symbol numbers to a function type and symbols to the symbol numbers --> if several logical function elements exist in a component, each function element must have a number (e.g. 4 NAND symbols with no. 1 to 4 for the function type 74LS00).

The symbols must exist in COLUMBINE with the specified names (ref. to the function "Logic", page 18).

COLUMBINE	Library editor	Version:	153
Function type: _____	(E) Edit (L) Kill >E< Select	(+ -) Step	
# : _____	Symbol : _____		

The existing assignments are listed below the menu. The function type name automatically appears in the menu.

After you have selected the respective function, you get to the input field for the number. With <ESC><ESC>, you leave the menu.

### >E< Edit

To create a new assignment, enter the symbol number, <CR>, and the name of the desired (existing!) symbol. After pressing <CR>, the symbol number and name appear below the menu.

In order to change an existing assignment, enter the respective symbol specification and press <CR>. Subsequently, the changes can be executed. If more than ten assignments exist for a function type and if the respective assignment is not displayed on the screen, you have to use the functions "+" or "-" to get to the next or the previous page.

### >L< Kill

With this function, specifications assigned to a function type can be deleted. Therefore, you have to enter the respective number in "Symbol no." and press <CR>.

If more than ten assignments exist for a function type and if the desired assignment is not displayed on the screen, you have to use the functions "+" or "-" to get to the next or the previous page.



**>+<**

If more than ten assignments exist, you can turn to the next page with "+", i.e. up to ten further assignments (symbol numbers and symbol names) will be listed in the lower part of the screen.

**>-<**

You can turn to the previous page (ref. also to >+<).

**(8) Terminal attributes**

With this function, you can assign a name for the layout (PCB) to the single terminals of a function type, a name for the schematics (CAE), and the desired connection width for the tracks to be digitized or routed. In this way, you define a terminal relation between shape and symbol terminals that corresponds to a respective relation in CIS. Already existing relations can be changed. The specifications of this menu will then automatically be taken into the netlist in CIS.

COLUMBINE		Library editor	Version:	160
Terminal	:		Symbol #	:
PCB - name	:		Symbol name	:
CAE - name	:			
Connection width	:	in (mm)		
Signal	:			
Function type	:		Shape:	
Ed	Shape	Stock ID		

If several shapes are assigned to the function type, move the cursor to the shape whose terminal(s) shall be assigned or whose assignment shall be changed and press <CR>.

In "Terminal", you enter the specification of the terminal to be assigned. If the terminal shall belong to a signal, enter the desired number in "Symbol number" (a respective assignment to the function type must have been created with the function "Symbol assignment" first).

After pressing <CR>, the symbol name is automatically entered and you can edit the following fields:

**PCB name:**The specification of this terminal for the layout (e.g. IN1).

**CAE name:**The logical terminal specification for the schematics (e.g. I1). If no symbol number is specified (i.e. symbol no. = 0), the terminal cannot get a CAE name.

**Connection**

**width:** If a signal name has been entered in "Signal", you can specify the width of the track type here with which the terminal is to be connected.

**Signal:** Here, you can specify the name of the net (signal name) to which the entered terminal is to be connected.

This function only makes sense for the specification of plane connections. In this case, you need not enter a symbol number (i.e. "0" in "Symbol no.") and CAE name.

With <CR>, the entered terminal 'attributes' are stored and other terminals of this function type can be related.

With <ESC><ESC>, you leave the menu.

**3.5 DEVICES**

In this part of the library, you can define all devices (plotter, tester, assembling automat, NC-routing and drilling machines), i.e. machine specific parameters, control files, drivers, etc. will be assigned. Parameters may also be changed or deleted and the defined devices can be listed.

The following menu appears:

<b>COLUMBINE</b>	<b>Library editor</b>	<b>Version:</b>	<b>88</b>
------------------	-----------------------	-----------------	-----------

**DEVICES**

---

Device name : \_\_\_\_\_ Type: \_ (1) Plot, (2) Assemble, (3) Test  
(4) Drill, (5) NC-routing, (6) List

---

- (1) Names
  - (2) Copy
  - (3) Kill
  - (4) Rename
  - (0) End
  - (5) Edit parameters
  - (6) Variants of assembly machine
- 

Function : 0

**(1) Names**

Lists the names of all ("Device name:" \*, enter a respective number in "Type") or specified devices. By 'clicking' the function "next" with <LMB>, you get to the next page, with "previous", you get back and with "quit", you finish the listing.

**(2) Copy**

Duplicates the specified device. You are asked for the name of the new device. The information of the old device will also be assigned to the new one.

**(3) Kill**

Deletes the specified device from the library. The specification is made with a respective entry in "Device name" and "Type".

**(4) Rename**

Renames the specified device. You are asked for the new name of the device. The device still contains the assigned information.

**(5) Edit parameters**

With this function, you can set or change parameters for the device entered in "Device name" and specified by the number entered in "Type".

Different menus appear for the different types of devices.

**3.5.1 DEVICE PARAMETERS FOR PLOTTER**

After specifying the plotter name, the selection number (1) in "Type" and "5" in "Function" in menu 88, you get to menu 89 for the parameter definition for this device:

COLUMBINE	Library editor	Version:	89
-----------	----------------	----------	----

```

DEVICE PARAMETERS for plotter: _____ # :__

Driver                : _____ in (mm)
Step length          * 10.000 : _____ in (mm)
Area                 : X _____ Y _____ in (mm)
Stripe               : X _____ Y _____ in (mm)
maximum mask #      : _____
Plotter type         : 1 (1=pen,2=photo,3=grid,4=laser)
Interface            : 0 (0=online,1=offline,2-4=offline)
Interface type       : 0 (0=binary,1=ASCII)
Comment              : _____

Pre-commands         : _____ Program ID : 0

Step (+-) > <
    
```

**- Driver:**

Here, you enter the name of the driver file (without extension!) for the specified plotter.

**- Step length:**

Defines the 'resolution' (smallest step length) for the specified plotter (ref. to the data in the plotter manual). The specification is made in 1/10,000 mm!

**- Area:**

Here, you define the maximum size of the plotting area (i.e. the maximum sheet or foil size).

**- Stripe:**

Divides the plot into stripe sections to prevent ink smearing when working with drum plotters.

**- maximum mask #:**

Here, you enter the maximum number of the masks/pens available in the plotter.

- **Plotter type:**

Choose the desired plotter type by entering the respective number: 1 = pen, 2 = photo, 3 = grid, 4 = laser

- **Interface:**

Specify the kind of data transfer to the plotter.

0 = online, 1 = offline, 2 = offline (512 bytes/block),

3 = offline (256 bytes/block), 4 = offline (384 bytes/block)

- **Interface type:**

Specify the format of the data to be transferred.

0 = binary data, 1 = ASCII-format

- **Comment:**

Here you have the possibility to make notes, etc.

- **Program ID:**

References for other PRISMA programs can be stored here.

When you press the <CR> key, three lines appear for the input of pre-commands.

- **Pre-commands:**

With these commands, you can directly contact the plotter interface (e.g. for the presetting of the light value for a photo plotter). These fields should only be changed or extended in consultation with the distributor.

- **Step (+-) > <**

If "+" is entered, further pre-commands exist (with <CR> you turn to the next page).

If you want to edit further pre-commands, but there is no free line any more, you can generate further pre-command lines by entering "+", <CR>.

With <ESC><ESC>, you get from the above mentioned menu to a menu where you can assign masks and symbols (see next page).

COLUMBINE	Library editor	Version:	98
-----------	----------------	----------	----

## MASK DEFINITIONS

Mask # : \_\_\_\_\_

inner size : X 0.000 Y 0.000 in (mm)  
 outer size : X 0.000 Y 0.000 in (mm)

Mask type :  flash  
 plot

Mask shape :  rectangle  
 circle  
 octagon  
 square  
 special  
 symmetric

In this menu, you enter the parameters of the masks/pens of the plotter (number, size, shape and usage).

These specifications are needed for a correct mask selection for the display of the board elements in OUTPUT.

## - Mask #:

Here, you assign numbers to the masks/pens of the plotter.

You have to enter two values for a mask to get the correct shape (ref. to definition of the track and via shape --> CIS description).

## - Outer size:

The outer diameter is the real diameter of the mask.

## - Inner size:

With this size and the specification in "Outer size", the geometrical shape of a mask is determined (ref. to the illustration on the next page).  
 At present, only symmetrical masks (same value in X and Y) can be defined.

Temporarily, the following mask shapes are possible:

Inner size X=Y=0, outer size X=Y > 0 --> round  
 Inner size X=Y = outer size X=Y --> square  
 Inner size X=Y < outer size X=Y --> square with rounded corners

- **Mask type:**

Here, you specify the usage of the mask.

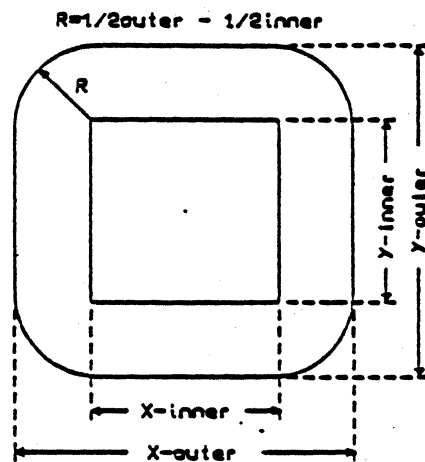
flash --> flash mask  
plot --> plot mask

- **Mask shape:**

By putting an "X" in the respective field, you can specify the shape of the mask.

rectangle  
circle  
octagon  
square  
special  
symmetric

(At present, only "circle", "square" and "symmetric" are possible).





**3.5.2 PARAMETERS FOR ASSEMBLY MACHINES**

After entering the name of the assembly machine, "2" in "Type" and "5" in "Function" in menu 88, the following menu appears for the definition of machine-specific parameters:

COLUMBINE	Library editor	Version:	142
-----------	----------------	----------	-----

```

DEVICE PARAMETERS for assembly machines : _____ # : _____
Driver      : _____
System      : 1 (0=abs, 1=rel)
Code        : _____ (0=EIA, 1=ISO, 2=ASCII, 3=ASCII2, 4=BSH, 5=EBCDIC)
Zero point  : X _____ Y _____ in (mm)
Step length : _____ in (mm) / 10000
Comment     : _____
    
```

**- Driver:**

Here, you enter the name of the driver for the specified assembling automat, <CR>. The data contained in the driver is automatically entered in the respective fields of this menu.

**Attention:**

For changing the entries, please refer to the manual of this machine (if you have doubts or problems, contact the distributor).

**- System:**

Here, you enter if the coordinates for the automat are to be specified absolutely or relatively.

**- Zero point:**

A new zero point for the machine relative to the absolute zero point can be defined (X=0, Y=0 as a rule).

**- Step length:**

Defines the smallest step length of the specified assembling automat (ref. to the manual of the automat).

The specification is made in 1/10,000 mm!

**- Code:**

Defines the data format (ref. to the manual of the machine).

- **Comment:**

Here you have the possibility to make notes, etc.

After you have saved your entries with <CR>, the following menu appears:

COLUMBINE		Library editor	Version:	5
Device name :	_____	# :	__	Type : 2
specification	basic parameters	Un	Len	Type
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__

The specifications in this menu are entered automatically according to the information contained in the driver. For changing the entries, please refer to the manual of the assembling automat (if you have doubts or problems, contact the distributor).

- **Un (Unit)**  
defines the text data unit

- 0 = no unit is defined
- 1 = 0.0001 inch
- 2 = 0.001 inch
- 3 = 0.01 inch
- 4 = 0.1 inch
- 5 = 0.0001 mm
- 6 = 0.001 mm
- 7 = 0.01 mm
- 8 = 0.1 mm

- **Len (Length)**  
Here, you enter the number of characters specified in "Basic parameters".

**- Type**

defines the kind of character of the "Basic parameters". It is entered as the sum of the below mentioned character numbers (e.g. "7" means, all capital and small letters as well as blanks are allowed).

- 1 = capital letters
- 2 = small letters
- 4 = blanks
- 10 = numbers
- 20 = sign characters
- 40 = period
- 100 = special characters

**3.5.3 Variants of assembly machine**

After entering the name of an already defined insertion machine, "2" in "Type" and "6" in "Function" in menu 88, the following menu appears in which different 'variants' for the magazine assignment of the specified assembly machine can be defined, duplicated, renamed and deleted.

COLUMBINE	Library editor	Version:	91
-----------	----------------	----------	----

**DEVICES**

Assembly machine : \_\_\_\_\_ Variant # : \_\_\_\_\_  
New # : \_\_\_\_\_

(1) Names	(5) Define variant
(2) Copy	
(3) Kill	
(4) Rename	
(0) End	

Function : >0<

After entering the desired number in "Variant #" and "5" in "Function", a magazine assignment variant for this assembly machine can be defined in menu 92 (see next page).

The definition is made by assigning magazine numbers (in "M #") to the stock IDs of the respective element (in "Stock ID").

COLUMBINE	Library editor	Version:	92
-----------	----------------	----------	----

VARIANT DEFINITION

Assembly machine: _____	Variant # : _____
Function : >E<	(N) New, (E) Edit, (K) Kill, (+/-) Step
M # : _____	St ID : _____

-->

-->  
-->

- By entering "N", you define a new assignment that will be listed below the menu.
- With "E" (Edit), you get to the listing below the menu where you can change the selected assignment.
- By entering "K", the selected assignment will be deleted.

**3.5.4 PARAMETERS FOR TESTER**

After entering the name of the tester, "3" in "Type" and "5" in "Function" in menu 88, you get to menu 5 for the definition of the specific parameters for this device.

The device parameters to be entered in this menu can be taken from the manual of the test automat (if you have doubts or problems, contact the distributor).

COLUMBINE	Library editor	Version:	5
-----------	----------------	----------	---

Device name : \_\_\_\_\_ # : \_\_ Type : 3

specification	basic parameters	Un	Len	Type
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__
_____	_____	__	__	__

**- Un (Unit)**

Defines the text data unit

- 0 = no unit is defined
- 1 = 0.0001 inch
- 2 = 0.001 inch
- 3 = 0.01 inch
- 4 = 0.1 inch
- 5 = 0.0001 mm
- 6 = 0.001 mm
- 7 = 0.01 mm
- 8 = 0.1 mm

**- Len (Length)**

Here, you enter the number of characters specified in "Basic parameters".

- **Type**

Defines the kind of character of the "Basic parameters". It is entered as the sum of the below mentioned character numbers (e.g. "7" means, all capital and small letters as well as blanks are allowed).

- 1 = capital letters
- 2 = small letters
- 4 = blanks
- 10 = numbers
- 20 = sign characters
- 40 = period
- 100 = special characters

**3.5.5 PARAMETERS FOR DRILLING MACHINES**

After entering the name of the drilling machine, "4" in "Type" and "5" in "Function" in menu 88, you get to menu 143 for the definition of the specific parameters for this device.

Most specifications in this menu are entered automatically according to the information contained in the driver.

For changing the entries, please refer to the manual for this machine (if you have doubts or problems, contact the distributor).

COLUMBINE	Library editor	Version:	143
-----------	----------------	----------	-----

```

DEVICE PARAMETER for drill/NC-routing : _____ # :__
Driver                               : _____
System                               : 1 (0=abs, 1=rel)
Code                                 : _ (0=EIA, 1=ISO, 2=ASCII, 3=ASCII2, 4=BSH, 5=EBCDIC)
Zero point                           : X _____ Y _____ in (mm)
Area in                              : X _____ Y _____ in (mm)
Step length                          : _____ in (mm) / 10000
Maxtools                             : _____
Drill diameter:                      : _____ in (mm) / 10000
Comment                              : _____
    
```

**- Driver:**

Here, you enter the name of the driver for the specified drilling machine.

**- Step length:**

Defines the smallest step length of the specified drilling machine in 1/10,000 units (ref. to the manual of the machine).

**- Area in:**

Here, you define the maximum size of the drilling area.

**- Maxtools:**

Here, you enter the maximum number of drills which fit in the magazine.

**- Code:**

Defines the data format for this machine (ref. to the manual of the machine).

**- System:**

Here, you enter if the coordinates for the machine are to be specified absolutely or relatively.

**- Zero point:**

A new zero point for the machine relative to the absolute zero point can be defined (X=0, Y=0 as a rule).

**- Unit: Drill diameter**

Defines the size unit (= the smallest possible difference of two drills), in which drill diameters are specified (in 1/10,000 units!).



### 3.5.6 PARAMETERS FOR NC-ROUTING MACHINES

After entering the name of the NC-routing machine, "5" in "Type" and "5" in "Function" in menu 88, you get to menu 143 for the definition of the specific parameters for this device.

Most specifications in this menu are entered automatically according to the information contained in the driver.

For changing the entries, please refer to the manual for this machine (if you have doubts or problems, contact the distributor).

COLUMBINE	Library editor	Version:	143
-----------	----------------	----------	-----

```

DEVICE PARAMETER for drill/NC-routing : _____ # :__
Driver      : _____
System      : 1 (0=abs, 1=rel)
Code        : _ (0=EIA, 1=ISO, 2=ASCII, 3=ASCII2, 4=BSH, 5=EBCDIC)
Zero point  : X _____ Y _____ in (mm)
Area in     : X _____ Y _____ in (mm)
Step length : _____ in (mm) / 10000
Maxtools    : _____
Drill diameter: _____ in (mm) / 10000
Comment     : _____

```

- **Driver:**

Here, you enter the name of the control file for the specified NC-routing machine.

- **Step length:**

Defines the smallest step length of the specified NC-routing machine in 1/10,000 units (ref. to the manual of the machine).

- **Area in:**

Here, you define the maximum size of the NC-routing area.

- **Maxtools:**

Here, you enter the maximum number of NC-routing heads which fit in the magazine.

- **Code:**

Defines the data format for this machine (ref. to the manual of the machine).

**- System:**

Here, you enter if the coordinates for the machine are to be specified absolutely or relatively.

**- Zero point:**

A new zero point for the machine relative to the absolute zero point can be defined (X=0, Y=0 as a rule).

**- Unit: Drill diameter**

Defines the size unit (= the smallest possible difference of two NC-routing heads), in which NC-routing heads are specified (1/10,000 units!).

**3.6 GRAPHIC MACROS (for plotter)**

With this function, you can define symbols (macros) for the display of layout elements on the plotter.

The following menu appears:

COLUMBINE	Library editor	Version:	40
-----------	----------------	----------	----

**GRAPHIC MACROS**

Macro - name : \_\_\_\_\_

- (1) Names
- (2) Copy
- (3) Kill
- (4) Rename
- (0) End
- (5) Edit macros

Function : 0

**Macro - name:**

Here, you enter the name of the macro to be defined or changed.

**(1) Names**

Lists the names of all (\*) or the specified (..\*) macros that have already been defined.

**(2) Copy**

Duplicates the macro specified in "Macro - name". You are asked for the name with which the new macro shall be saved in the library.

**(3) Kill**

Deletes the specified macro from the library.

**(4) Rename**

Renames the specified macro. You are asked for the new name of this macro.

**(5) Edit macros**

Calls the graphic macro editor. Subsequently, graphic screen and mouse are activated for the digitizing of the macros.

The following menu appears:

<b>COLUMBINE Library editor Version:</b>		<b>20</b>
<b>MACRO EDITOR</b>  Macro name : _____	<b>Mouse - buttons :</b>  0 : Position 1 : Draw line 2 : Delete line <CR>: End	
<b>Window definition :</b>  lower left corner X: -10__ Y: -10__ upper right corner X: 10__ Y: 10__	<b>Current position :</b>  X :            Y :	

First, only the upper part of the menu appears. After you have entered the symbol name, <CR>, the lower menu part is called.

- **Symbol name:**

Here, you enter the desired name. If you have already entered a symbol name in the previous menu, this name will be taken here. The lower part of the menu appears after the entry of the symbol name and <CR>.

- **Window definition**

Here, you enter the coordinates of the lower left and the upper right corner of the window in which the symbol is to be edited. As by definition the zero point of the coordinates lies in the centre of the rectangular window, the coordinates of the lower left corner must have negative signs and the same number values as the coordinates of the upper right corner.

- **Current position:**

Specifies the coordinates of the current mouse position, referring to the centre of the window (=zero point).

- **Mouse-buttons:**

Shows which functions are assigned to the mouse buttons for the edition process.

**0: Position (= left button)**

By pressing the left mouse button, you determine the current mouse position as starting position of a line.

**1: Draw line (= middle button)**

By pressing the middle mouse button, a line is drawn between the starting position and the current mouse position (=end of the rubber thread). The end of this line automatically becomes the starting point of the next line.

If you want another starting position, move the rubber thread to the desired point and press the left button.

**2: Delete line (= right button)**

By pressing the right mouse button, the last digitized line will be deleted. By pressing this button again, the line before will be deleted, etc.

**<CR>: End**

By pressing <CR>, the edition process for this load symbol is finished and the macro is stored. When entering a new symbol name, another symbol can be edited.

**<ESC><ESC>:**

With this entry, you interrupt the current edition process (without storing it!) and get back to menu 40.



In the upper part of the menu, general information to the library appears (creation date, free memory capacity, number of entries, etc).

REPORT	Library report generator	11
--------	--------------------------	----

Creation date	:	_____	Free data blocks	:	_____	(blocks)
Last change	:	_____	Deleted blocks	:	_____	(blocks)
Version	:	_____	Number of entries	:	_____	
			Deleted entries	:	_____	

What is to be output ? \* \_\_\_\_\_

F_Function type	K_Body	P_Devices
L_Macros	I_Stock ids	S_Logic
E_End		

Function : >E<    Output :        :OUT\_\_\_\_\_

With most of the functions you get to subordinate menus (without pressing <CR>).  
With "E" or <ESC><ESC>, you return to the main menu.

When you activate a function in the submenus, you get back to the main menu.  
To work with another function in the same submenu, you have to activate the respective library area in the main menu again.

Notes:

- The data for each function activated in a submenu is stored in the same output file OUT (Attention: uppercase!) on the lib/sys directory if you leave the predefined file specification unchanged. When creating a new report output file, the old file will be overwritten with the new data contents if the name of the output file (= OUT) is not changed.
- If you want to store the data of single functions in separate output files, enter different names for the output files in menu 11 (you may add different extensions).

Attention:

You only get to the input field for the file specification before entering the function (i.e. press the "arrow down" key twice)!

- You can edit or print the report output files if you want to read them (when using the standard name for the output file, you have to enter the file name OUT in uppercase!).

sectionend

## REPORT

---

### 3.1 FUNCTION TYPES

section3.1

After specifying the desired function types in "What is to be output?" (e.g. "74\*") and entering "F" in "Function", the following submenu appears:

<p>Function type :</p> <p>A Contents B Body assignment C Symbol assignment</p>
--

Function : >A<      Output :      :OUT\_\_\_\_\_

Here, you enter the desired function and the name of the output file.

#### A Contents

A list of the specified function types from COLUMBINE is created and written into the output file.

#### B Body assignment

A list of the specified function types and the shape bodies and stock ids assigned to them is created and written into the output file.

#### C Symbol assignment

A list of the specified function types, together with the assigned stock ids and symbols (with terminal relation and connections) is created and written into the output file.

sectionend

### 3.2 BODY

section3.2

<p>Body :</p> <p>A Contents B Device parameters</p>
---

Function : >A<      Output : OUT:\_\_\_\_\_

#### A Contents

A list of the specified shape types from COLUMBINE is created and written into the output file.



---

**B Device parameters**

A list of the device parameters (for assemblers and testers) assigned to the specified shapes in COLUMBINE is written into the output file.  
sectionend

**3.3 DEVICES**

section3.3

<p>Devices : A Contents B Plotter descriptions C CAM - device descriptions D Lists</p>
--

Function : >A<      Output : OUT: \_\_\_\_\_

**A Contents**

A list of the specified devices from COLUMBINE is created and written into the output file.

**B Plotter descriptions**

The parameters (driver, type, interface, step width, size of the plotting area, masks and pre-commands) assigned to the plotters defined in COLUMBINE are output.

**C CAM - device descriptions**

The parameters (control file, code, step length, etc.) assigned to the assemblers and testers defined in COLUMBINE are output.

**D Lists**

sectionend

## REPORT

---

### section3.4 3.4 MACROS

For this function, there is no submenu. The names of all macros (=load symbols) defined in COLUMBINE are directly written into the specified output file.

sectionend

### 3.5 Stock ids section3.5

**Stock ids :**

- A Contents**
- B Function type assignment**
- C Symbol assignment**
- D Device parameters**

**Function :** >A<      **Output :** OUT: \_\_\_\_\_

#### **A Contents**

A list of the specified stock ids from COLUMBINE is created and written into the output file.

#### **B Function type assignment**

A list of the specified stock ids and the shapes and function types assigned to them is created and written into the output file.

#### **C Symbol assignment**

A list of the specified stock ids and the shapes and function types assigned to them is created and written into the output file.

#### **D Device parameters**

A list of the device parameters (for assemblers and testers) assigned to the shapes in COLUMBINE is written into the output file.

sectionend

---

section3.6  
3.6 LOGIC

Logic :

A Contents

B Element assignment

Function : >A<      Output : OUT: \_\_\_\_\_

**A Contents**

A list of the specified symbols from COLUMBINE is created and written into the output file.

**B Element assignment**

A list of the symbols existing in COLUMBINE and the symbol bodies, symbol graphics and function types assigned to them is written into the output file.

# BTOOLS

---

## BTOOLS

### 1. INTRODUCTION

The program BTOOLS is used for the generation (reservation of the necessary space) of the central PRISMA library 'columbine' and the procedure library 'output.sys', as well as the administration of the library 'columbine' (access definition, etc.). This program can only be called by the system administrator.

### 2. HANDLING

The program is started by clicking on the LIBRARY icon and activating number 3 ('library maintenance') in the selection menu. After pressing <CR>, the following menu appears:

B T O O L S - LIBRARY ADMINISTRATION Version : _____	3
--	---

Library name : : columbine____.____
-------------------------------------

- (1) Generate
- (2) Squeeze
- (3) Define user
- (4) Delete user
- (5) List users
  
- (0) End

Function : >0<

First only the upper part of the menu (heading) appears (as menu 2). You can now enter the name of the library file to be created or processed (the standard name appears as the default). If you want to create an additional library file, you have to specify another name.

#### Attention:

The central library file used for the programs CIS and OUTPUT must either have the standard name or a link to the standard name must exist, otherwise the programs cannot use it.

The procedure library must be named 'output.sys' or must be linked to it so that it is found by the program OUTPUT.

#### Functions:

In order to activate the desired function, you have to enter the respective number. Do not press <CR> afterwards!

**(1) Generate**

With this function, you create a new library file and reserve the necessary space on the device specified in the menu header. After entering "1" in "Function" (no <CR>!), the following menu appears:

<b>B T O O L S - LIBRARY ADMINISTRATION</b> Version : _____	<b>4</b>
---	----------

Library name :        : _____ . ____
--------------------------------------

Size of the data area     : 100\_\_ blocks (min.: 100 max.: \_\_\_\_)

Size of the key chart     : \_\_\_\_\_ blocks (min.: \_\_\_\_ max.: \_\_\_\_)

In 'library name', the name specified in menu 3 appears.

The number of blocks for the new library file, which you have to enter in "Size of the data area", must be within the limits that appear in parentheses (in 'min.:', the minimum number of necessary free blocks and in 'max.:', the available number of free data blocks on the directory 'lib/sys' appears.

After the entry and <CR>, you get to the next line.

The size of the key area lies between 4% and 20% of the blocks specified in "Size of the data area". The limits of the key area are automatically calculated by the system and appear in parentheses. The number of blocks which you have to specify for the key chart must be within these limits.

# BTOOLS

---

## (2) Squeeze

If you want to store new data in an existing library file but there is not enough space in the file, you get the following message:

"Not enough space. Please squeeze!"

With the "Squeeze" function, the different free blocks are combined to one free space.

After entering "2" in "Function" (no <CR>!), the inquiry

"Please confirm "SQUEEZE" (Y/N)"

appears. For the execution of the function, you have to enter -Y-.

Note:

If the message "File inconsistency ....." appears on the screen while working with the COLUMBINE editor, you have to leave the program and squeeze the 'columbine' library. Then you can restart COLUMBINE.

Attention:

The squeezing must not be interrupted, otherwise the library data might be lost.

## (3) Define user

With this function, maximal 255 users can get the access to the library file entered in the menu heading by specifying a user name and a password.

After entering "3" in "Function" (no <CR>!), the following menu appears:

B T O O L S - LIBRARY ADMINISTRATION Version : _____	5
--	---

Library name : : columbine____.____
-------------------------------------

User name : \_\_\_\_\_

The name which you have to specify here can have up to 9 alphanumeric characters.

After entering the user name and <CR>, an additional menu part for the determination of the access rights is called:



## **BTOOLS**

---

In "Bn" (n=1-7) you enter component characteristics (max. 9 alphanumeric characters) that may exist or not (e.g. "on socket", "in pairs", etc). If the characteristic shall exist, put an X in the respective parenthesis.

After pressing <CR>, the cursor moves back to the field for the input of the data area selection and you can execute the same procedure for the second data area.

With <ESC><ESC>, you get back to the part of the menu in which you enter the user name. You may now define a new user and his access rights.

When leaving this menu part with <ESC><ESC>, you get back to menu 3.

### **(4) Delete user**

With this function, you can delete existing user definitions. When entering "3" (no <CR>!), menu 5 appears.

After entering the defined user name and <CR>, the user will be deleted from the list of the accessors.

With <ESC><ESC>, you get back to menu 3.

### **(5) List users**

If you enter "5" in "Function" in menu 3, you get a list of all users and their passwords.

By entering a respective number in "USRID", the succession of the defined users is shown.

The group assignment specified in menu 5 will be listed in "GROUP".



# TABLE OF CONTENTS

## E. SCSCOM

GENERAL . . . . .	1
A.0 INTRODUCTION . . . . .	2
1. Title Specification . . . . .	2
2. Units . . . . .	2
3. Technology Definitions . . . . .	2
4. Sheet Definition . . . . .	2
5. Symbol Definitions . . . . .	2
6. Symbol Placement . . . . .	2
A.1 NAMES/COORDINATES . . . . .	3
1. Characters and names . . . . .	3
2. Coordinates . . . . .	4
A.2 Title . . . . .	4
A.3 Units . . . . .	5
A.4 Technology . . . . .	6
Technology options . . . . .	6
A.5 Schematics sheet . . . . .	7
A.6 Bodies . . . . .	7
A.7 Terminal definition . . . . .	9
A.8 Placement . . . . .	10
1. Component and shape types . . . . .	10
2. Placement specification . . . . .	10
3. Rotation . . . . .	11
4. Graphic elements . . . . .	11
A.9 Graphic elements . . . . .	12
1. Line segments . . . . .	12
2. Text fields . . . . .	12
A.10 Syntax description . . . . .	14
1. General . . . . .	14
2. Basic elements of the syntax . . . . .	14
3. Global structure of a pcb-file . . . . .	15
4. Syntax elements of the global structure . . . . .	15
5. Keywords . . . . .	17
6. Syntax diagrams . . . . .	17

**GENERAL**

**This documentation consists of three parts:**

- Text**
- Syntax Description**
- Syntax Graphs (Not available at this printing)**

**If the text, the syntax description and syntax graphs do not correspond, the syntax graphs are always relevant!!**

**SCSCOM can be regarded as universal software interface to the PRISMA SYSTEM.**

An ASCII-file (.PCB) is supplied which can be used as the input or output file for the CALAY system to communicate with other computer systems, independent from the physical transfer.

This PCB-file may contain all data as text file that is necessary for the symbol description; in addition, it can contain a PCB layout.

As an input file, it enables the description of a design in the predefined syntax not only on the CALAY system. Subsequently, it is transformed into a CALAY-ADB-file with SCSCOM. If the data is inconsistent, a syntax check creates error messages or warnings. The ADB-file can then be processed with the CALAY software.

As an output file, it is created from an existing ADB-file and can be processed and evaluated on other systems if the respective software is available.

The SCSCOM syntax contains the language of PCBCOM V05 as subset. This is described in a separate documentation.

## A.0 INTRODUCTION

A design description contains the following specifications in the following order:

**1. Title Specification**

A title is used for marking all lists created by the compiler.

**2. Units**

Number specifications in SCSCOM are necessary for size specifications or coordinates for example.

Four measuring units are available for the user. From these four units, three can be selected for a design description.

Number specifications do not have measuring units then but symbols that correspond to the selected units.

**3. Technology Definitions**

All parameters of the design technology are defined here.

**4. Sheet Definition**

Specifications to sheet size and border lines are made.

**5. Symbol Definitions**

Symbols and bodies can be defined locally by SCSCOM.

**6. Symbol Placement**

Here, the symbols are placed in a definable rotation; a local definition of bodies is possible.

## A.1 NAMES/COORDINATES

### 1. Characters and names

A PCB-file can contain all printable ASCII characters. The following characters have a special meaning:

Tab (HT)	Separator between
Blank (SPACE)	units
,	(COMMA) of the input language
()	Delimiter for terminal formats, placement formats and terminal names
[]	Beginning or end
% ... %	Comment delimiter, all characters between % ... % are interpreted as comment
;	Delimiter
#, @	Prefixes for numbers (units)
'	(Apostrophe) Name prefix

In a PCB-file, there are 4 groups of elements that must get names:

1. Technology rules
2. Bodies
3. Symbols
4. Terminals

Names must not be double within one group, otherwise the compiler creates an error message.

Terminal names must be definite within one body.

Names consist of up to 30 characters. Uppercase and lowercase will be distinguished, except for the keywords.

Terminal names consist only of up to 10 permissible characters.

All characters are allowed in names.

If a name does not begin with a letter, it must be preceded by a name prefix (apostrophe).

## 2. Coordinates

Coordinate specifications are necessary if symbols are placed or positions for rotation points, text fields, etc. are to be specified.

The coordinates are entered with a number pair, separated by a colon.

Real numbers and prefixes are possible, the assignment of a unit is effected according to the rules for measuring units (ref. to A.0.2: Units).

Examples:

```
20.25:10.75  % Specifications in the standard unit u1 %  
-10 :@100   % x-value      in the standard unit u1 %  
           % y-value      in the unit u2.      %
```

## A.2 TITLE

For all output files created by the compiler, a text can be entered.

If no title is specified, the file name of the PCB-file will be used as headline.

After entering the keyword `.TITLE`, the characters in inverted commas are used as title information.

Example:

```
.TITLE "SCSTST.PCB Test example"
```

This title appears as headline in the `.LST`-file that can be created during the compiling.

**A.3 UNITS**

In a PCB-file, number specifications are possible in three different, selectable measuring units.

Syntax: **.SCUNITS u1,u2,u3 ;**

The following units can be selected for u1, u2 and u3:

<b>MM</b>	Millimeter
<b>MIL</b>	1/1000 inch
<b>INCH</b>	1 inch
<b>CAL</b>	Basic grid

With this command, you determine the standard unit u1 and the special units u2 and u3.

Example: **.SCUNITS CAL,INCH,MM;**

The compiler accepts now number specifications in these three defined units.

The assignment of the respective units to the number specifications is made with the following prefixes:

1. only one value : unit u1 will be used
2. @ followed by a value: unit u2 will be used
3. # followed by a value: unit u3 will be used

Referring to the above example, hence follows:

@17	= 17"
#49	= 49 mm

## A.4 TECHNOLOGY

All technology data must be defined locally in the PCB-file.

### Technology options

**.UNIT**        ivalue defines the basic grid in units per inch. As all coordinate specifications are calculated internally in this basic grid, the maximum resolution is determined with this value. Permissible values are 100 ... 30 000.

**.GRID**        ivalue defines the working grid in units/basic grid

**.CLEARANCE**        ivalue defines the permissible clearance

After the keyword **.LINE**, line types can be defined as follows:

**.LINE** Type    Width 1                      Width 2                      Enlargement

**Type**            is the number of the respective line type to be defined. It must be within 0 ... 15.

**Width 1**            consists of two number specifications. The first one describes the width of the core or the inner square, the second one specifies the total width or the total diameter. The first value must not be larger than the second one. By varying the first parameter between 0 and the value of the second parameter, you can define any forms of rounded line segments, from rectangular to rounded rectangles to circular.

**Width 2**            consists also of two values with which any forms of line widths can be defined to the respective line type (is not evaluated by the system at present).

**Enlargement**        is a single value that defines an additional clearance to elements of this type. This value is added to the general clearance defined with **.CLEARANCE**.

Example:

```
.LINE 2 #0.2 #0.3 #0.0 #0.0 #0.0;
```

Defines line type 2. The width of the core is 0.2, the total width is 0.3; hence follows that segments of this type have slightly rounded corners.

## A.5 SCHEMATICS SHEET

After the keyword **.SCBOARD**, the sheet can be defined.

Example:

```
.SCBOARD
  [
    .SHEET #0.0 : #0.0 #290.0 : #130.0;
  ]
```

```
.SHEET x-value : y-value defines the zero point
        x-value : y-value defines the size of the working area
```

With the help of graphic elements (ref. to chapter A.9), any contours can be assigned to the sheet.

## A.6 BODIES

Bodies can be defined in the local library.

Syntax:

```
.SCLOCLIB [
  ident .FORMAT size [ xy pair ]
  .TERMINAL terminaldef
  .TERMINAL terminaldef ...
  Graphic elements
  ;
  ident ... ;
  ...
]
```



## SEMANTICS:

**.SCLOCLIB** is the keyword that precedes the local symbol definition. The name of the body to be defined follows (ident); it must not have been assigned already. The following specifications apply to this body until the definition is finished with a semicolon. Further definitions may follow.

**.FORMAT** precedes the definition of the body which contains the following specifications:

**Size** The size of the body consists of the specification of a rectangle with x and y length in the form x:y.

**Center of rotation** The center of rotation of the symbol is specified in relation to the zero point of the size, that is the lower left corner. You enter a coordinate pair in the form: x-value;y-value. The default is 0:0 if no other center of rotation is specified.

Then the single terminals of the respective body are defined; bodies without terminals are possible. Any graphic elements may be assigned to the body.

The definitions of the body types are then valid for all symbols of this type.

A terminal consists of one or several line segments.

Example:

```
.SCLOCLIB
[
  TRANS      .FORMAT    @5:@5
             .TERMINAL  '1' 1.0:0
             [.LINE     1  0.2:0 1.0:0]
             .TERMINAL  '2' 2:2
             [.LINE     1  [0] 2:2 2:4];
]
```

After the terminal definition, a section with layout elements may appear that are assigned to the complete body.

**A.7 TERMINAL DEFINITION**

The terminal definition begins with the keyword **.TERMINAL**. The terminal name and position follow. In brackets appear the layout elements for defining the terminal graphics.

Syntax:

```
.TERMINAL      ident      xypair  
                [ ( Graphic elements ) ]
```

## A.8 PLACEMENT

The placement part begins with the keyword `.SCPLACEMENT` and is enclosed in brackets. Here, names are assigned to symbols (symbol-name) and their position on the sheet is defined.

### 1. Component and shape types

First the symbol (e.g. NAND) is specified. Behind the keyword `.TYPE`, a body definition must follow. It may consist only of a body name (e.g. B08) but it can also contain a complete definition as described at `.SCLOCLIB`. In this case, the existing body description with the same name will be overwritten temporarily up to the next semicolon. Afterwards the previous definition from `SCLOCLIB` becomes effective again.

After the symbol specification, the placement list follows that begins with the keyword `.AS`. This list contains at least the names of the symbols belonging to the respective type. Behind each symbol name, the following specifications may appear that are assigned to the respective symbol then:

#### Placement specification

#### Graphic elements

The body is then valid for all component names until the list is finished with a semicolon. The placement of components of another body or symbol may follow.

### 2. Placement specification

The placement specification is enclosed in parentheses and must contain the following parts:

- a) rotation
- b) placement coordinate

If the placement specification is missing, SCSCOM places the respective symbol in the upper right corner of the sheet.

### 3. Rotation

The symbols are rotated counter clockwise around the body's center of rotation, but only in increments of 1/10 degrees. At present, only the following rotations are supported:

0	no rotation
900	90 degrees
1800	180 degrees
2700	270 degrees

Rotation 0 is the position in which the respective body was defined.

### 4. Graphic elements

Graphic elements are text fields or line segments that can be assigned to single symbols (ref. to A.9). These graphic elements follow the placement specification. Graphic elements that are in the placement list (behind the keyword .AS) do not overwrite existing graphic elements of the body but will be added to them.

## A.9 GRAPHIC ELEMENTS

Graphic elements are line segments, circles or text fields that can be assigned to terminals, bodies or symbols.

Placement coordinates are relative to the center of rotation of the symbol or to the placement point of the terminal.

### 1. Line segments

Syntax:

```
.Line      ( Line type  
            Coordinate Coordinate ... Coordinate  
            )
```

Semantics:

Line type type of which the respective line segment shall consist. It refers to the technology definition.

Coordinate at least two coordinates are necessary, namely starting and ending point of the line. But you may also specify several coordinates to describe a complete line with one definition. SCSCOM divides it into several segments then.

Example:

```
.Line      ( 1 1:1 2:2 )  
  
% Line segments of type 1 %  
% from 1:1 to 2:2 %
```

### 2. Text fields

Syntax:

```
.TEXT (Text type Width Position Size Rotation String)
```

Semantics:

<u>Text type</u>	0 free text
	1 symbol name
	2 terminal name
	3 date
	4 distance

<u>Width</u>	The keyword <b>.WIDTH</b> followed by a number specifies the character width in 1/5000 inch. This specification may be omitted.
<u>Position</u>	specifies the coordinate of the lower left corner of the text field.
<u>Size</u>	specifies the size of the text field
<u>Rotation</u>	The keyword <b>.ROTATION</b> followed by a number specifies the rotation in 1/10 degrees. At present, only 0, 900, 1800 and 2700 are allowed. If this specification is missing, 0 will be taken.
<u>String</u>	If text type 0 has been selected, string contains the text between the inverted commas that may consist of up to 80 characters. If the text shall contain the inverted comma (ASCII hex 22), it must be preceded by a "\" (ASCII backslash hex 5C).

### 3. Circles

**Syntax:**

```
.CIRCLE ( Line type Start coordinates
         End coordinate
         Center coord
         Opening angle
         )
```

Line type ----> Line segment

Start coordinate: Starting point of the circle

End coordinate: Ending point of the circle

Center: Center of the circle

Opening angle: 0 .. 3600; 1/10 - degree - steps

## A.10 SYNTAX DESCRIPTION

### 1. General

The syntactical structure of a PCB-file will be described here.

**KEYWORDS** are in bold type in this chapter and do always begin with a point.

Terminal symbols like **[ ] ; ( )** etc. are also in bold type.

Metasymbols like **< > [ ]** are not in bold type.

Non-terminal symbols are also in normal type and lowercase.

Meaning of the metasymbols:

Symbol	Meaning	EBNF	BNF
	or	a = A   B	a = A a = B
[ ]	optional	a = [B] A	a = BA  a = A
< >	repetition	a = <B> A	a = A a = Ba
( )	parentheses	a = (A   B)C	a = AC a = BC

### 2. Basic elements of the syntax

<u>string</u>	:	all printable ASCII characters except for inverted commas (ASCII 042 octal); enclosed in inverted commas.
<u>ivalue</u>	:	any positive, integer or real numbers
<u>xypair</u>	:	coordinate pair, two ivalue optional separated by a colon.
<u>size</u>	:	<u>xypair</u> - size
<u>position</u>	:	<u>xypair</u> - coordinate specification
<u>ident</u>	:	maximal 30 permissible characters
<u>unit-ident</u>	:	MM   MIL   INCH   TECH
<u>layoutel</u>	:	physics

graphics:

```
(.LINE ( ival xypair xypair < xypair > ) )

      |
(.TEXT ( ival [.WIDTH ival] xypair xypair
      |           [.ROTATION ival] [string] ) )
      |

(.CIRCLE ( ival xypair xypair xypair ival ) )
```

3. Global structure of a PCB-file

```
pcb:

[ title ]
[ scunits ]
  scrule
  scboard
<
[ scunits ]
[ scloclib ]
[ scplacement ]
>
[ PCBCOM ]
```

4. Syntax elements of the global structure

```
title : .TITLE [ string ]

scunits : .SCUNITS [ unit-ident , unit-ident ,
                    unit-ident ]

scrules : .SCRULE [ technology ]

technology :

                [.UNIT           ival;]
                [.GRID           ival;]
                [.CLEARANCE      ival;]
                < [.LINE          def           ] >

def :           ival ival
```



ivalue ivalue ivalue ivalue ;

```
scboard : .SCBOARD
          [
                [ .SHEET xypair xypair graphic ]
          ]
```

```
scloclib : .SCLOCLIB [
                < ident format >
          ]
```

```
format : .FORMAT xypair [ xypair ] < terminal |
graphic >
```

```
terminal : .TERMINAL ident xypair [< graphic >]
```

```
scplacement :
          .SCPLACEMENT
          [
                < [ string ] .TYPE ident [ format ] .AS
                list ;>
          ]
```

```
list : <
          ident
          [
                ( ivalue xypair )
          ]
          < graphic >
        >
```

---

## 5. Keywords

Keywords must be preceded by a point. There is no distinction between uppercase and lowercase. Abbreviations are allowed.

.AS	beginning of a placement list
.CIRCLE	circle segment
.CLEARANCE	clearance
.FORMAT	border of a component element
.GRID	routing grid
.LINE	line technology, definition of lines
.ROTATION	rotation of text fields
.SCBOARD	starting the PC board definition
.SCLOCLIB	local declaration of components
.SCPLACEMENT	placement header
.SCRULE	rules for technology of the PC board
.SCUNITS	measuring units
.SHEET	size of the working area
.TERMINAL	terminal definition
.TEXT	definition of text fields
.TYPE	shape type specification in the placement part
.UNIT	grid base
.WIDTH	connection type

## 6. Syntax diagrams

The following pages describe the current SCSCOM syntax by means of syntax graphs.

# TABLE OF CONTENTS

## F. POSTPROCESSING

Introduction . . . . .	1
General . . . . .	1
Function selection . . . . .	1
Notes to the documentation . . . . .	2
Output structure . . . . .	2
Starting the procedure generator . . . . .	6
Procedures and Output . . . . .	7
Define/change procedures . . . . .	7
1.1 Menu 0 . . . . .	7
1.2 Menu description . . . . .	7
For the further proceeding . . . . .	10
Execute procedures . . . . .	11
Output of the plot data on the plotter (online) or in a file (offline) . . . . .	13
Processing the procedure library output.Sys . . . . .	15
Selection output data . . . . .	18
Introduction . . . . .	18
Menu 2 . . . . .	18
Menu description . . . . .	18
Procedure definition plot . . . . .	20
Introduction . . . . .	20
'Sheet' definition . . . . .	21
General . . . . .	21
Menu 3 . . . . .	21
Menu description . . . . .	21
Functions . . . . .	23
Handling instructions . . . . .	24
'Picture' definition . . . . .	25
General . . . . .	25
Menu 4 . . . . .	25
Menu description . . . . .	25
Functions . . . . .	28
Handling instructions . . . . .	29
Display of the picture . . . . .	30
Menu 5 . . . . .	30
Menu description . . . . .	30
General remarks for the further proceeding . . . . .	34
Layer plot . . . . .	35
Layer selection . . . . .	35
Handling instructions . . . . .	35
Selection of the layer elements of signal layers . . . . .	36
Menu 24 . . . . .	36
Menu description . . . . .	36
For the further proceeding . . . . .	38
Solder resist plan . . . . .	39
Layer selection . . . . .	39

Selection of the signal layer elements that are	
Not to be covered . . . . .	40
Silk screen . . . . .	42
Layer selection . . . . .	42
Component selection of component layers . . . . .	43
Menu 21 . . . . .	43
Menu description . . . . .	43
Component display . . . . .	45
Drill plan . . . . .	46
Layer selection . . . . .	46
Selection of the kind of via for the drill plan . . . . .	47
Menu 30 . . . . .	47
Menu description . . . . .	47
Display of the drills . . . . .	48
Production plot . . . . .	48
Control plot . . . . .	49
For the further proceeding . . . . .	50
Negative plan . . . . .	51
Layer selection . . . . .	51
Enlargement of the vias that do not belong to the	
Plane signal . . . . .	52
Display of the vias and heat traps for control	
Plots . . . . .	53
Display of vias/heat traps for production plots . . . . .	54
For the further proceeding . . . . .	55
Element selection for documentation layers . . . . .	56
Mask selection . . . . .	57
Mask selection for control plots (manual) . . . . .	57
Mask selection for the via display in production Plots	59
Segment display in production plots . . . . .	66
Procedure definition drill . . . . .	71
Introduction . . . . .	71
Drill tape definition . . . . .	72
Menu 66 . . . . .	72
Functions . . . . .	72
Selection of the elements to be drilled . . . . .	74
Menu 67 . . . . .	74
Menu description . . . . .	74
Specifications to offset, fix-file, etc. . . . .	76
Menu 69 . . . . .	76
Handling instructions . . . . .	76
Changing the standard values for drill tools . . . . .	78
For vias . . . . .	78
Menu 70 . . . . .	78
Menu description . . . . .	78
For test points . . . . .	80
Menu 71 . . . . .	80
Procedure definition nc-routing . . . . .	81
Introduction . . . . .	81
Tape definition . . . . .	82
Menu 72 . . . . .	82
Functions . . . . .	82
Specifications to offset and fix-file . . . . .	84
Menu 73 . . . . .	84

Handling instructions . . . . .	84
Specifications for nc-routing tools . . . . .	85
Menu 74 . . . . .	85
Menu description . . . . .	85
Introduction . . . . .	91
List selection . . . . .	91
List output in a file . . . . .	93
List output on the terminal . . . . .	94
Output form of the lists . . . . .	95
Netlist . . . . .	95
Component list . . . . .	97
Non connected terminals . . . . .	99
Cross-reference-list (= component connections) . .	100
Component collisions . . . . .	100
Technology . . . . .	100
Statistics . . . . .	101

INTRODUCTION

1. GENERAL

The superior postprocessing function (POSTPROCESS.-icon) contains all data output possibilities of the system that refer to the PC-board production.

2. FUNCTION SELECTION

After clicking the POSTPROCESS.-icon, the main menu for selecting the postprocessing functions appears:

```

PRISMA POSTPROCESSING >>errorstop=on<<      v...

                                *** default selection: >10< ***
1. Execute 'output-procedures'
2. Create or change 'output-procedures'
3. Create plotter data (online/offline)
4. List or delete 'output-procedures'
5. Run 'list-generator'
6. View 'list-files'
7. Transfer ascii/binary data to a serial port
8. Create data for 'genrad tester'
9. Toggle 'errorstop'
10. Help information

E. Exit
-----
your choice ? _
    
```

The main part of the postprocessing is the program package **OUTPUT** (function 1 and 2). From the data base of the adb-file created by the interactive input system CIS and processed by the router, **OUTPUT** can create output data for the creation of layout plots, drill and NC-routing tapes and insertion files.

In addition, the icon "POSTPROCESS." contains:

- the program **PLOT** for creating the plotter control data (from the output data of **OUTPUT** and the respective drivers);
- the program **PROCED** for processing the procedure library;
- the program **CALPRI** for creating different lists from adb/gph-files (netlists, component statistics, etc.);
- a function for viewing all kinds of list files on the screen;
- a 'transfer' function for sending data (ascii/binary) to one of the serial ports (ttya/ttyb) of your system;
- a program for creating control data for 'genrad'-testers;
- a 'toggle' function for switching the 'errorstop' on/off ('errorstop=on' = if an error occurs, the program **OUTPUT** is stopped);
- a **HELP** function with information to the postprocessing function selection.

### 3. NOTES TO THE DOCUMENTATION

The functions 1 to 4 that belong to the program OUTPUT are explained in the following chapters of this manual.

Subsequently, function 5 (= the program CALPRI) will be described.

The other functions are self-explanatory.

Function 8 (setups and hints) refers only to the 'preparatory' postprocessing function selection and file specification, not to the actual programs (OUTPUT, PROCED, PLOT, CALPRI) described in the manual.

### 4. OUTPUT STRUCTURE

The program OUTPUT consists of several modules (**userpp**, **readpp**, **sortpp**, **plotpp**, **drilpp**, **millpp**, **assmpp**, **ercopp**) that are necessary for the creation of 'procedures', the read in and sorting of the input data and the creation of the output data.

With 'procedure', all functions and parameters to be executed when processing an adb/gph-file are meant.

The 'procedures' are defined in a form series, in which you can determine the use of the output data, the output devices and their parameters, as well as the kind and form of the output (e.g. for plots: which PC board elements are to be displayed in which form and by which means).

So the procedure is the 'tool' with which an input file will be processed to get the desired output data.

The 'procedures' are stored in the library **output.sys** and can be called any time for creating output data for a new input file.

(The procedure library can be generated with the program **BTOOLS** - under the **LIBRARY** icon - and must be stored on the directory **lib/sys**).

With the OUTPUT functions, output data can be generated for:

- a. all kinds of layout plots, i.e. repro-fit production copies and control plots (output files with the default extension .gnn; n is a real number from 0 to 9),
- b. the generation of NC-routing and drill tapes (output files with the default extension .bnn or .fnn; n is a real number from 0 to 9),
- c. the selection of assembling desks and insertion equipments (output files with the default extension .bst).

Figure 1 shows the output possibilities of the program OUTPUT and the connection to the other programs of the PRISMA system.

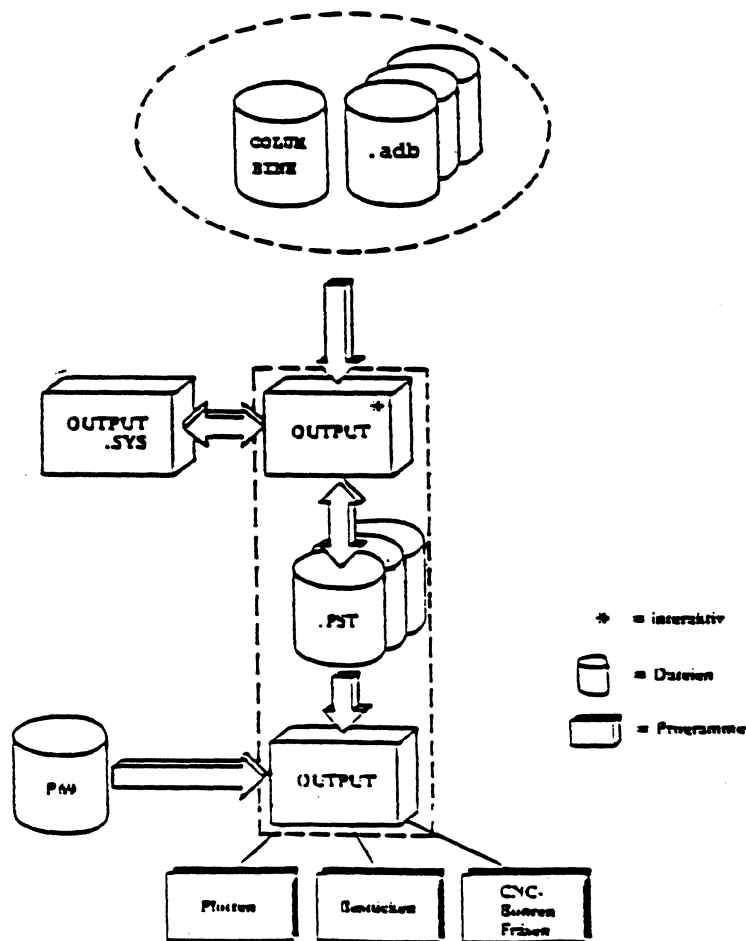


Figure 1



### 5. MENUS FOR THE PROCEDURE DEFINITION

The menu-driven OUTPUT module `userpp` serves for the generation and editing of procedures.

The menus contain functions and data fields for entries.

In the menus, the desired functions are marked with a cross as a rule. The marked functions are activated by pressing <CR> in the sequence in which they are listed. The hierarchical organization of the functions and menus (superior and subordinate ones) with netlike branches (ref. to diagram on page 1-6) effects, that with the activation of a main function, a sub function is called, whose activation calls other subordinate menus, etc.

The hierarchical organization of the menus has an effect on the priority of the contained information, i.e. specifications in a main menu are valid for all processes activated in the following menus, to which they apply (e.g. the parameters 'exactness', 'artwork', etc. of menu 5).

With <ESC><ESC>, you get back to the previous menu (or to the preceding part of the menu). Any entries made in the submenu will not be considered.

With <CR> you store your entries and get to the next subordinate menu.

Figure 2 on the next page shows all menus of the program and their connections.

The diagram illustrates the hierarchical organization of the functions with superior and subordinate menus and shows the possibilities for a movement within the menu.

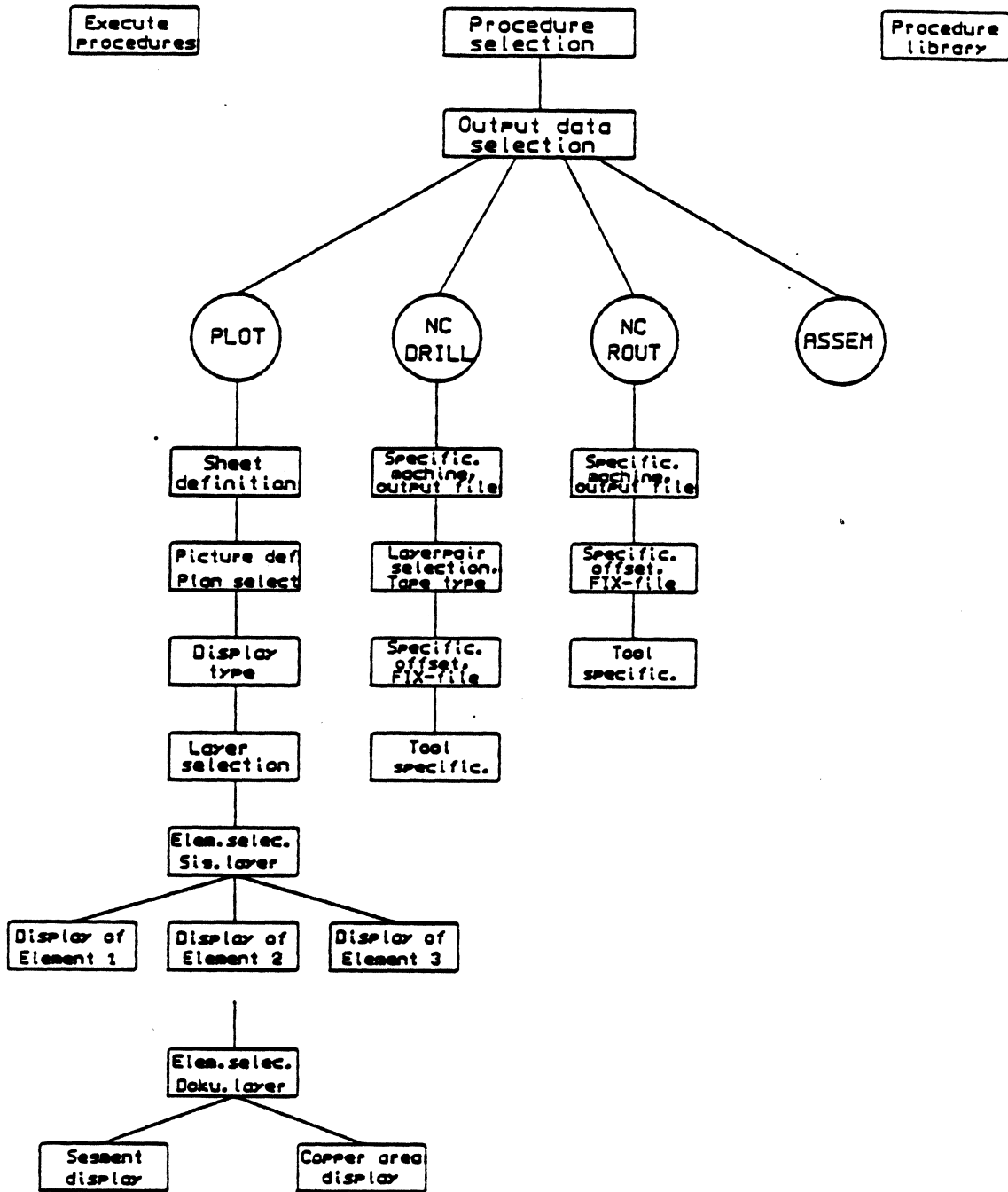


Figure 2

### 6. STARTING THE PROCEDURE GENERATOR

The procedure generator `userpp` is started by clicking on the `POSTPROCESS`-icon and selecting function 2 in the appearing main menu.

After the program has been started, a menu in which you enter the file name (without extension!) appears.

After entering the file name and pressing `<CR>`, you are asked for the extension of the file.

If you enter "adb" as extension, `<CR>`, the .adb-file is transformed into a .gph-file first because `OUTPUT` can only work with the gph-data base at present.

The files in the gph-format are stored on the directory `aph` (not on the gph-directory)!

Subsequently, menu 0 for the procedure selection appears.

**PROCEDURES AND OUTPUT**

**1. DEFINE/CHANGE PROCEDURES**

After activating the POSTPROCESS.-icon, entering the selection number "2" and specifying the file to be processed, you get to menu 0 of the 'procedure generator'. In this menu, you can determine by entering the respective selection number if a new procedure is to be defined or if an existing procedure is to be changed.

**1.1 Menu 0**

O U T P U T      Version _____	0
--------------------------------	---

Name of the .GPH-file : \$GPHDIR:\_\_\_\_\_ .gph

(1) Create new procedure (2) Change existing procedure	Selection (1,2) ==> (2)
Existing procedures : <div style="margin-left: 40px;">                     TEST1    LAYER 1                      TEST2    LAYER 0                      .                      .                      .                 </div>	

With <ESC><ESC>, you can leave the menu if the cursor is on the field for the input of the file name or the selection number; then you get back to the main menu for selecting the postprocessing functions.

**1.2 MENU DESCRIPTION**

**- Name of the .GPH-file:**

Here, you enter the name of the .gph-file, for which the output data shall be generated with the chosen or defined procedure.

Notes:

- The name of the file specified after the program start is entered automatically. If necessary, you can specify another file, provided that it already exists in the gph-format on the directory aph.
  
- When you enter "\*" in "Name of the GPH-file" and press <CR>, the names of all GPH-files existing on the aph-directory will be listed. When pressing <CR> again, you return to menu 0.

### - CREATE NEW PROCEDURE = SELECTION (1)

The group of functions, data, etc. (= the procedure), that will be created in the following menus, is saved in the procedure library `output.sys` with the name you enter in

- Name: \_\_\_\_\_

(max. 6 characters) and will appear again in the list of the existing procedures when calling menu 0 again.

You can assign a comment (max. 39 characters incl. blanks) to the new procedure in

- Comment for the new procedure: \_\_\_\_\_

The part of the menu for the input of these data will appear after activating the selection (1) with `<CR>`. With `<ESC> <ESC>`, the input is interrupted and the cursor moves back to the field where you enter the selection number.

### - Existing procedures:

In the lower part of the menu, the already defined output procedures and their comments will be listed.

You get to this part of the menu by entering the selection number "2" (do not press `<CR>`!).

If "+" appears in the lower right corner, further procedures exist. With `<CR>`, you can turn to the next pages, as long as no procedure was selected.

With `<ESC><ESC>`, you get back to the field where you enter the selection number.

### - Change existing procedure = selection (2)

The selected procedure is loaded from the library into the program and will be used for processing the specified .gph-file. If necessary, the procedure can be changed.

Attention:

If with an already existing procedure another .gph-file than the one used during the definition of the procedure shall be processed, the following must be guaranteed:

- a. its layer plan must be identical with the layer plan of the .gph-file used during the creation of the procedure;
- b. you have to go through all procedure menus that are necessary for the creation of the desired output data (incl. the menus for the mask or tool selection!) even if the function constellation of the procedure shall not be changed;

--> otherwise not the technology of the current .gph-file but the technology of the original .gph-file that was used for creating the procedure will be taken for the creation of the output data!

If you want to prevent the effects that changes within a procedure area ("Plot", "Drill" or "NC-routing") may have, you have to leave the menus 4 (for the "Plot" function), 66 (for the "Drill" function) and 72 (for the "NC-routing" function), in which you get back when the procedure is finished, with <ESC><ESC> instead of "S" (for "Store").

If the changes in the complete procedure shall not be considered, you have to finish the procedure editing in menu 80 without storing the procedure (see next page).

Attention:

If you execute changes in a procedure and

- leave some menus with <ESC><ESC>
- some with "S" and
- store the procedure in menu 80,

the procedure contains then partly 'new' and partly 'old' function constellations!

## POSTPROCESSING/Define/Change Procedures

---

### 1.3 FOR THE FURTHER PROCEEDING

If a new procedure is created, you get directly back to menu 0 after finishing the procedure menus; name and comment of the procedure appear in the procedure list then.

If you edit an existing procedure for changing it and leave at least one of the menus with the command "S" (= Store), the inquiry menu 80 appears before you get back to menu 0.

O U T P U T	version _____	80
-------------	---------------	----

<p>changed procedure :</p> <p>(0) .. don't store procedure in library</p> <p>(1) .. store procedure with old name in library</p> <p>(2) .. store procedure with new name in library</p> <p>&gt;0&lt;</p>
--

By entering the selection number and pressing <CR>, you specify if the changed procedure shall be stored and with which name.

Attention:

If you store the changed procedure with its old name, the old procedure will be overwritten!

Afterwards, you get back to menu 0 where you can edit or create a procedure or leave the menu with <ESC><ESC> and return to the postprocessing selection menu.

## 2. EXECUTE PROCEDURES

When entering "1" in the postprocessing selection menu, the file and procedure inquiry for the processes to be executed appears.

Several processes can be executed (= 'batch-function').

When executing the procedure, the PC board data contained in the input file is transformed into output data for each process according to the selected functions.

Attention:

As a procedure is always created on the basis of an adb/gph input file, a defined procedure already contains a special adb/gph technology.

In order to process another adb/gph-file with this procedure correctly, the technology must be the same!

The names of the output files in the respective OUTPUT menu cannot be changed by the user, all files have the name of the .gph-file from which they were created.

The predefined extensions of the output files and the directories on which they will be stored are different:

- .gnn for plot output files --> on device gdi
- .bnn for drill output files --> on device pst
- .fnn for NC-routing output files --> on device pst

(n is a real number from 0 to 9)

The extensions of the output files can be changed by the user.

Note:

By assigning different extensions to output files that have been created with different procedures from the same .gph-file, you can prevent that the output files will be overwritten.



In addition, OUTPUT creates the following report files:

- an error report in the file `post.log`; here you will find the errors and warnings that occurred when executing the procedure(s).

Attention:

Every time when executing the postprocessing selection function "1", the old error file will be overwritten by the new one. If you want to prevent this, the old `post.log`-file must be renamed previously.

- a drill report in a file with the name of the input file and the extension `.drp` (if the creation of drill tapes was also defined in the procedure);
- a NC-routing report in a file with the name of the input file and the extension `.mrp` (if the creation of NC-routing tapes was also defined in the procedure);

The report files are stored on the `pst`-directory. With the 'view' function (enter "7" in the postprocessing selection menu) or a respective edition (e.g. with the vi editor), these files can be read in.

**3. OUTPUT OF THE PLOT DATA ON THE PLOTTER (ONLINE) OR IN A FILE (OFFLINE)**

The data of the .gmn-files is prepared for the plotter by activating function "3" in the postprocessing selection menu.

Afterwards, the following menu appears:

P R I S M A   P L O T   S E L E C T I O N		v....
*** default selection >> 0 << ***		
1.	: plot hp file (online)	
2.	: create hp file (offline)	
3.	: create gerber file (offline)	
4.	: select plotter driver	
5.	: create selected [ noname] file	
6.	: show plot directory	
0.	: Upper level	
E.	: Exit	
your choice? _		

You can directly choose the online output on a HP plotter and the offline output in a HP or GERBER file.

For the creation of offline files with another driver, you have to select the driver first (selection number "4").

P R I S M A   P L O T T E R   S E L E C T I O N		v...
1.	: aristo	
2.	: awiota	
3.	: benson	
4.	: calcom	
5.	: calcomc960	
6.	: calsoft	
7.	: dmp	
8.	: emma	
9.	: ferranti	
10.	: glaser	
11.	: secrol	
12.	: wx463x	
0.	Upper level	
your choice ? _		

## POSTPROCESSING/OUTPUT On-line and Off-line

---

Of course you can only specify the drivers for which you have a licence.

After the driver specification, you return to the menu for the plot selection; the name of the selected driver appears in point 5 instead of "noname".

When pressing <CR> now, you are asked for the name (with extension) of the plot file (= .gmn-file) from which the offline file shall be created:

enter name of 'plot-file' > \_\_\_\_\_

Here you enter the name, the extension and <CR>.

The subsequently created offline file (= "output file") gets the name of the plot file and an extension to which the driver name is added (e.g. test.g01\_gerber).

When entering the selection number "6", the plot directory gdi will be listed.

4. PROCESSING THE PROCEDURE LIBRARY OUTPUT.SYS

The procedure library can be processed with the program PROCED.

With this program,

- procedures can be deleted from the library;
- the contents of the different procedures can be listed in an ASCII-file;
- the procedure library may be squeezed by moving the free blocks together to get more space.

When selecting "4" in the postprocessing main menu, the following menu appears:

```

PRISMA POSTPROCESSING  list procedures          v....
*** default: >> 0 << ***
1. Delete procedures - create 'procedure-listfiles'
2. View an existing 'procedure-list-file'

0. Upper level

E. Exit

your choice ? _
    
```

By entering "2", you can view a procedure list file created with function "1" on the screen. After selecting this function, you are asked for the name of the list file (without extension).

If you enter "1", the program PROCED is started. The following menu appears:

```

PROCED

Number of procedures      :
Number of deleted procedures :
Number of deleted blocks  :
Number of free blocks     :

- TEST 1      Layer plan 1
- TEST 2      Solder resist 0
- TEST 3      Negative plan GND
    
```

## POSTPROCESSING/PROCED

---

In the upper part of the menu, the number of existing procedures, the number of deleted procedures and blocks and the number of free blocks is output.

In the lower menu part, the names of the procedures and their comments are listed (the specifications shown here are only examples).  
By putting an X in the respective field, you determine the procedure to be processed. Only one procedure can be selected.

If "+" appears in the lower right corner of this list, there are some more procedures. With <CR>, you can page down, as long as no procedure has been selected.

With <ESC><ESC>, you leave this part of the menu and activate the inquiry for the squeezing.

After selecting a procedure, <CR>, the function selection for the processing of the respective procedure is called:

<b>Selected procedure:</b> TEST	Layer plan 1
(1) delete	
(2) list	
(2)	

The name of the selected procedure and the assigned comment are shown in "Selected procedure".

In the input field for the function selection, "2" for "list" appears as the default.

### List

Activating function "2" effects the creation of the list file. It gets the name of the procedure and the extension ".lst" and is written on the pst-directory.

### Delete

In order to delete the selected procedure, enter "1" and press <CR>.

The procedure will be deleted from the library OUTPUT.SYS and you get back to the starting menu.

If you press <ESC><ESC>, the procedure will not be deleted. You return to the starting menu.

In the starting menu, you can select another procedure to be deleted or call the "Squeeze" function by pressing <ESC><ESC>.

### Squeeze

After pressing <ESC><ESC> in the starting menu, the following inquiry appears:

**Squeeze OUTPUT.SYS ? (N)**

If you type in "Y", the free blocks are moved together to one free space.

If you answer the inquiry with "N" or leave the default entry "N" and press <CR>, the following question appears:

**Finish the program ? (Y)**

With <CR>, the program is finished.

If you enter "N", you get back to the starting menu.

## SELECTION OUTPUT DATA

### 1. INTRODUCTION

When you have defined a new procedure in selection menu 0, or when you have selected an existing procedure, <CR>, menu 2 appears, in which you specify the use of the generated output data by selecting the respective function(s).

### 2. MENU 2

O U T P U T	2
-------------	---

Plot	:	-
Drill	:	-
NC-routing	:	-
Assembling	:	-

### 3. MENU DESCRIPTION

- **Plot:**  
Data for the control of pen, photo or laser plotters shall be generated.
- **Drill:**  
Data for the control of NC-drilling machines shall be generated.
- **NC-routing:**  
Data for the control of NC-routing machines shall be generated.
- **Assembling:**  
Data for the control of assembling desks and insertion equipments shall be generated (not yet implemented).

If several functions are selected in this menu, you return automatically to the current menu after processing the subordinate menus and the cursor is moved to the next selected function. If you activate this or another function, you get into the respective subordinate menus and you can extend the procedure with the function and parameter constellation defined there.

If no further functions have been selected, you get directly back to selection menu 0.



## PROCEDURE DEFINITION PLOT

### 1. INTRODUCTION

When you have activated the function "Plot" in the menu for the selection of the output data, you get to several menus for the detailed definition of the plot(s).

With one procedure, several plot output files can be created. They have the name of the input file and the default extension .gnn (n is a real number from 0 to 9).

Each plot output file contains the data base for a 'sheet' to be drawn on the plotter (corresponds to a sheet of paper or a film).

A sheet can have one or several 'pictures'.

For artworks or control plots, a 'picture' contains either the representation of a signal layer and the documentation layers assigned to it or the superposed representation of several signal and documentation layers, i.e. one 'plan' or several superimposed plans.

The upper part of each menu which belongs to the function "Plot" contains information to the following parameters:

- Procedure:** The name of the current procedure is displayed here.
- Plotter:** The name of the selected plotter appears here.
- Picture:** The number of the picture to be defined is shown here.
- Layer:** The number of the current layer (signal or documentation layer) is displayed here.
- Output file:** The name of the plot output file appears here for the current picture.
- Usage:** Here is shown if the current definition refers to a production or control plot.
- Plan:** Here you are informed about the kind of plan (= the plot), which you define.

The specifications to the single parameters appear in the upper part of the menu, in which they are defined.

2. 'SHEET' DEFINITION

2.1 GENERAL

After the activation of the function "Plot", menu 3 appears, in which you determine the name of the plot output file, the output device and the plotter data for the first 'picture'.

2.2 MENU 3

PROCEDURE :	OUTPUT FILE :	<b>3</b>
PLOTTER :	USAGE :	
PICTURE NO. :	PLAN : LAYER :	

Output file : \$OUTDIR:_____g01 Output device :	FIXED PLOTTER DATA
CHANGE PLOTTER DATA (Y/N) : N	Driver name :
Routing area in X: 0.000 (In.) in Y: 0.000 (In.)	Device type :
Speed : 0.000 (In.)/s	Correction x : 0.000 factor y : 0.000
	Step 1/100000 in.:0.000
	Number of masks :
Command (N,E,L,A,S, ? for help) : N Sheet number : 0	

2.3 MENU DESCRIPTION

**Output file:**

Name and extension of the output file are automatically entered.

As a rule, the plot output files are stored on the directory **gdi** and have the name of the .gph-file and the default extension .gnn (n is a real number from 0 to 9).

The extension can be changed, if necessary, e.g. if an output file with the same name is already existing (when the same .gph-file has already been processed with another procedure).

With one procedure, several plot output files can be created that have the same name by default but different numbers in their extensions. Each output file contains the information for a 'sheet' to be drawn on the plotter.

For the first plot (the first 'sheet') the name of the .gph-file and the extension .g01 are automatically entered.

After entering the function "N" or "E" (see below), the cursor moves to the upper part of the menu where you can enter the output device or change the extension of the output file (the file name cannot be changed).

**Output device:**

The name of the output device (=plotter), for which the output data is determined, is entered here. The output device must be defined in the library COLUMBINE.

**Notes:**

- a. If you enter "\*" instead of the name of the output device and press <CR>, all names of the plotter defined in COLUMBINE will be listed for your information. After finishing the listing with <CR>, the first device name is automatically entered in "Output device".
- b. In order to get the desired kind of output data, you have to make sure that the entered output device is defined in COLUMBINE with the respective parameters (ref.to the COLUMBINE description, chapter 3.5.1).

After the output device has been entered, <CR>, some of the parameters of the specified plotter defined in COLUMBINE will automatically be entered in the menu.

If an already defined procedure has been loaded, the respective plotter name automatically appears in "Output device".

**Attention:**

If you use an already defined procedure and change the name of the originally entered output device (i.e. you want to use another plotter and other masks), you have to go through all mask menus and check the mask assignment for the defined plots.

**Fixed plotter data:**

The data entered in "Fixed plotter data" is only shown for information but cannot be changed.

**CHANGE PLOTTER DATA (Y/N):**

- N: The data shall not be changed. After pressing <CR>, you get to the next menu.

**Y:** The entries in "Plotting area" and "Speed" can be changed for the current 'sheet'. For the input of the changes, the cursor is moved to the lower part of the menu. When you have finished your changes, press <CR>.

The measurement unit for the specifications appears automatically - it corresponds to the measurement unit defined in the file calay.usr on the directory etc.

### **Plotting area:**

Here, the maximum plotting area of the specified plotter is automatically entered. If you want to limit it, you have to enter the desired coordinates for the maximum plotting area in the measuring unit specified in the menu.

### **Speed:**

Here, you enter the pen speed of the specified (pen) plotter defined in COLUMBINE. It can be changed, if necessary.

## **2.4 FUNCTIONS**

### **Commands:**

- N:** If you enter -N- here, you can create a new 'sheet' in a new output file. The sheet number and the numerical part of the file extension will automatically be incremented by one.
- E:** After entering -E- and the number of the sheet to be edited, you can change the entries in "Output file", "Output device", "Plotting area" and "Speed" for the specified sheet.
- L:** After entering -L- and the respective number in "Sheet number", the sheet with its entries will be deleted.
- A:** If you enter -A-, the numbers of the already defined sheets will be listed.
- S:** After you have finished the subordinate menus for the 'sheet' definition, you get automatically back to menu 3. By entering -S-, this sheet with all its specifications is stored. You leave menu 3 and get either to menu 2 (if another function was selected there) or directly back to menu 0.
- ?** Explains the commands.

### **Sheet number:**

If you want to edit or delete an existing 'sheet', you have to enter the number of the sheet here for which the respective command shall be executed.

## 2.5 HANDLING INSTRUCTIONS

With <ESC><ESC> in the other parts of the menu, you get back to the field for the entry of the commands.

With <ESC><ESC> in this part of the menu, you get back to menu 2 for the selection of the output data. The data of the current sheet will not be saved in the .pst-file. So you have the possibility to look at an already defined sheet of a procedure and to work with it without storing the data afterwards.

When pressing <CR> after you have finished your entries, you get to the menus for the further determination of the plot(s) for the current output file.

Afterwards you automatically return to this menu. By entering the respective command, you can work with the superior function "Plot" (e.g. generate a new output file = a new 'sheet') or finish the "Plot" function with the command -S- and go back to menu 2 for the selection of the output data.

**3. 'PICTURE' DEFINITION**

**3.1 GENERAL**

After you have defined the current 'sheet' by determining the output file and the plotter data, you get to the menu for the 'picture' definition, in which you can determine the placement and enlargement of a 'picture' on the current sheet as well as the plan of the picture and its use.

**3.2 MENU 4**

PROCEDURE :	OUTPUT FILE :	4
PLOTTER :	USAGE :	
PICTURE NO. :	PLAN : LAYER :	

<p>ENLARGEMENT of the plot :</p> <p>SECTION : (1)                  \$\$PCB (1) \$\$BOARD (2) WINDOW (3)                  Coordinates X1: 0.000 X2: 0.000                  (In.) Y1: 0.000 Y2: 0.000</p> <p>SHIFTING of the plot (In.)                  by section in X: 0 in Y: 0                  with distance in X: 0.000 Y: 0.000</p>	<p>USAGE : (1)                  Production (1)                  Control (2)</p> <hr/> <p>PLAN : (1)                  Layer plot (1)                  Solder resist (2)                  Silk screen (3)                  Drill plan (4)                  Test point plan (5)                  Glue plan (6)                  Negative plan (7)</p>
<p>Command (N,E,L,A,S, ? for help ) : N Picture number : 0</p>	

**3.3 MENU DESCRIPTION**

**Enlargement of the plot:**

Here, you can enter an enlargement factor for the plotting of the current 'picture'. Real numbers between 0, 1 and 100 can be used as enlargement factor. Factor 1 is entered as the default.

**Section:**

By entering the respective selection number and the coordinates if necessary, you determine the section of the working sheet defined in CIS/PCB that shall be displayed in the current 'picture'.

The starting point (zero point) of the plotter is the lower left corner of the defined section.

**\$\$PCB:**

When you specify this section, the entire working sheet will be displayed.

**\$\$BOARD:**

When you select this function, only the PC board area will be displayed.

**WINDOW:**

Only the section (window) specified in "Coordinates" is considered for the plot.

By entering

X1, Y1, you define the lower left corner  
X2, Y2, you define the upper right corner

of the board section to be plotted referring to the zero point of the working sheet.

**SHIFTING the plot:**

Here, you can determine if and in which way the current 'picture' shall be shifted on this sheet.

For the X and the Y coordinate the shifting is entered separately.

**by section:**

Specifies by how many section units the plot is to be shifted in regard to the zero point of the plotter (= lower left corner of the first non-shifted picture) in the +X and/or +Y direction. Only positive integer numbers are allowed.

**with distance:**

Here you can specify, by how many units (defined in calay.usr) the picture is to be shifted in the positive X and/or Y direction.

If specifications larger than 0 are made in "by section" as well as in "with distance", they work together. Here you can specify the distance, which shall exist between the two pictures (so that e.g. during a shifting by one section unit, the edges of the two plots do not collide).

**ATTENTION:**

If you do not enter a shifting, the current plot is drawn on the same field as the preceding one !

**USAGE:**

By entering the respective selection number, you specify here, if the current picture is to be used as artwork or as control plot. The specification has a great effect on the display of the plot. For production plots, only one plan per picture must be plotted, for control plots, several plans can be drawn one upon the other in the same picture.

The specification of the selected usage will be entered in the upper part of the next menu.

**PLAN:**

By entering the respective selection number, you determine which kind of plan shall be displayed in this picture. The specification of the selected plan will be entered in the upper part of the next menu.

**Layer plot (1):**

The elements (tracks + copper areas, vias, texts) of the signal and documentation layer(s) chosen in the menu for the layer selection shall be plotted.

**Solder resist plan (2):**

The solder resist mask for the specified layer(s) shall be plotted.

**Silk screen (3):**

The silk screen for the specified layer(s) shall be plotted.

**Drill plan (4):**

The drill plans for the specified layer pair(s) shall be plotted.

**Test point plan (5):**

At present, it is not yet possible to create test points in CIS/PCB. Therefore, test point plans cannot be created.



**Glue plan (6):**

The glue plan for the glue points of the selected layer(s) shall be plotted.

**Negative plan (7):**

The negative plan (i.e. vias/pins and heat traps) shall be plotted for the specified plane layer(s).

**Attention:**

For 'split plane' layers, the negative print is not created with the function "Negative plan", but with the function "Layer plot"! For more information, see documentation "CIS-SPECIALS".

### 3.4 FUNCTIONS

**Commands:**

- N:** If you enter -N- here, you can create a new 'picture'. The picture number in the upper part of the menu will automatically be incremented by one and you get to the menu parts for entering the plot determination and selecting the usage and plan of the new picture.
- E:** After entering -E- and the number of the picture to be edited, you can change the entries in the left part of the menu for the current sheet.  
  
When defining a new procedure, "E" is automatically entered in "Command" and "1" in "Picture no.". In this case, the entries in the right part of the menu (usage and kind of plan) can also be edited since you define a new 'picture'.
- L:** After entering -L- and the respective picture number, all previous entries for this picture will be deleted.
- A:** If you enter -A-, the numbers of the already defined pictures (picture numbers) will be listed.
- S:** By entering -S- ("Store"), you leave the menu. The entries for this picture are saved and you get back to menu 3.
- ?** Explains the commands.

**Picture number:**

If you want to edit or delete an existing 'picture', you have to enter the number of the picture here for which the respective command shall be executed.

**3.5 HANDLING INSTRUCTIONS**

With <ESC><ESC> in the other parts of the menu, you get back to the field for the entry of the commands.

If the cursor is placed on the command input field and you press <ESC><ESC>, you get back to the menu 3 for the definition of the 'sheet'. The entries made in the subordinate menus will not be saved. So you have the possibility to look at an already defined sheet of a procedure and to work with it without storing the changes afterwards.

When pressing <CR> after you have finished your entries, you get to the menus for the further determination of the current 'picture'. Contents and format of these menus are dependent from the selected plan and its usage. After you have finished your work in the subordinate menus, you automatically return to this menu.

By entering the respective command, you can remain in the menu (e.g. generate a new 'picture', i.e. another plan) or if the work with the pictures of the current sheet shall be finished, you can go back to the menu for the definition of the sheet with the command -S-.

#### 4. DISPLAY OF THE PICTURE

After you have saved your entries with <CR> in menu 4, you get to menu 5 to determine the kind of display.

Here you can specify the parameters "Artwork" and "Bend" for the display of the PC board elements, the numbers of the special pens to be used, etc.

In the upper part of the menu, the specification of the previously selected plan appears automatically.

##### 4.1 MENU 5

PROCEDURE :	OUTPUT FILE :	5
PLOTTER :	USAGE :	
PICTURE NO. :	PLAN : LAYER :	

ARTWORK : ( _ )	BEND : ( _ )
Scheme (1)	Angular (1)
Contour (2)	Slanted (2)
Repro (3)	Rounded (3)
EXACTNESS : -.000 /+.000 (In.)	
MIRROR : mirror everything: _ , DIN-texts: _ , mirror \$\$BOARD: _	
Special pen for texts : 1__	
Special pen for control plots: ,flash (Y/N): load symbol:	

##### 4.2 MENU DESCRIPTION

###### ARTWORK:

By entering the respective selection number, you can define the general display of the board elements here.

###### Scheme (1):

This kind of display applies to control plots.

- tracks of type 0 are displayed with a line;
- track type 1 is displayed with a line;
- all other track types are displayed in their contours (=not filled) and only with the core (= track width "rectangle").
- vias are displayed unfilled.

**Contour (2):**

This kind of display will be used for survey plots as a rule.

- tracks of type 0 are not plotted;
- track type 1 is displayed with a line;
- all other track types are displayed in their contours (=not filled) and only with the core (= track width "rectangle").
- vias are displayed unfilled.

**Repro (3):**

This kind of display is used for repro-fit creations of layouts.

- tracks of type 0 are not plotted;
- all other track types are displayed filled.
- vias are displayed filled.

**BEND:**

With the help of this parameter, the shape of the track angles is determined by entering the respective selection number. The slant or rounding is achieved with a grid cell of the grid defined in the .adb-file.

**Rectangular (1):**

rectangular angles;

**Slanted (2):**

slanted angles (45 degrees);

**Rounded (3):**

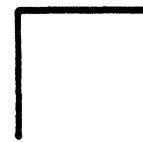
rounded angles



3 = rounded



2 = slanted



1 = rectangular

**EXACTNESS:**

If round or square vias shall be flashed and if the mask is automatically chosen, you can define how far the width of the flash mask may maximally diverge from the width of the vias to be plotted defined in the technology, so that it can still be used for the creation of these vias.

**Attention:**

- This parameter has no effects on the automatic selection of
- the flash masks for the display of square vias with rounded corners;
  - all plot masks.

**MIRROR:**

By putting an "X" in the respective field, you can determine the mirroring and adjustment of the texts.

It is possible to select several functions.

**MIRROR EVERYTHING:**

The total work sheet will be mirrored; i.e. all elements (segments, vias, frame, text fields and texts) will be mirrored.

Mirror axis is the Y-axis in the middle of the work sheet.

**mirror \$\$BOARD:**

The board will be mirrored, i.e. all elements (segments, vias, frame, text fields) belonging to the special component \$BOARD will be mirrored.

Attention: the texts in the text fields will not be mirrored.

Mirror axis is the Y-axis in the middle of \$\$BOARD.

**DIN-texts:**

All texts will be adjusted (rotated and/or shifted) that they can be read in the DIN-direction (i.e. horizontal from left to right and vertical from bottom to top). If "mirror everything" and/or "mirror \$\$BOARD" have also been selected, the texts will first be mirrored and then adjusted.

**Notes:**

- If "mirror everything" as well as "mirror \$\$BOARD" have been selected, all elements belonging to \$BOARD - except for the texts! - will be mirrored twice! In this way, the texts will be mirrored once. But you have to take care that the board (\$BOARD) is placed symmetrically on the 'work sheet' so that the two mirror axes are congruent!
- In the appendix to the POSTPROCESSING description (manual), you will find an example that illustrates the 'MIRROR' function with all possible variations.

**SPECIAL PENS FOR TEXTS:**

For **production plots**, all text fields of the **signal layer** activated for the current 'picture' will be plotted with the specified plot mask. This does not apply to documentation layers.

Additionally, this plot mask will be used for production plots of silk screens in order to plot the component frame.

You cannot specify a pen here for control plots!

**SPECIAL PEN FOR CONTROL PLOTS:**

For control plots with superimposed layer display, the specified mask (= plot and/or flash mask) is used for the single plotting of the vias/pins and/or text fields that exist on several layers at the same time.

A repeated superimposed drawing of these elements is prevented in that way.

Mask number "1" is entered as the default.

You cannot specify a pen here for production plots!

**FLASH (Y/N):**

If the vias/pins are to be flashed with the mask specified in "Special pen for control plots", you have to enter "Y" here. In this case, the mask must be defined as flash and plot mask in COLUMBINE so that the vias can be flashed and the text fields can be plotted!

**LOAD SYMBOL:**

If you do not want to flash, but plot with the mask specified in "Special pen for control plots", you can specify a load symbol (defined in COLUMBINE) for the via/pin display.

If you do not want to flash and no specification is made, the pins and vias are plotted as defined in the via technology.

By entering "\*" in the input field for the load symbol, <CR>, the symbols existing in COLUMBINE can be listed. If you press <CR> again, the name of the load symbol that was listed last will be entered.

### 5. GENERAL REMARKS FOR THE FURTHER PROCEEDING

In principle, the further proceeding is the same for all plans:

- a. After the kind of display of the 'picture' has been determined, the signal and documentation layers to be considered in the respective plan must be selected.
- b. Subsequently, the selection of the layer elements to be modified of the first selected signal layer will be effected.
- c. Then the exact display of the selected elements is determined by specifying the masks and the load symbols.
- d. If further signal layers have been selected, the same procedure (element selection, mask selection) must be executed for the next signal layer, etc.
- e. Afterwards, the element selection for the first selected documentation layer is effected.
- f. Then the display of the selected elements (mask/load symbol selection) is determined.
- g. If further documentation layers have been selected, the same procedure must be executed for the next documentation layer, etc.
- h. Afterwards, you automatically return to menu 4.

#### Further proceeding in this description

The proceeding for the layer selection (point a) and the element selection of signal layers (point b) is not the same in all plans and will therefore be described separately for each plan.

The element selection for documentation layers is identical for all plans and will be described in the chapter "Element selection for documentation layers".

The proceeding for determining the display of the selected elements of signal layers (point c) and documentation layers (point f) is the same for all plans. It will be explained in the chapter "Mask selection".

If the determination of the element display (= mask selection) differs from the general proceeding, it will be explained in the description of the respective plan.

These differences appear in the

- production plot: component display in the silk screen,
- display of drills in the drill plan,
- display of test points in the test point plan,
- display of glue points in the glue plan,
- display of vias and heat traps in the negative plan.

6. LAYER PLOT

After you have saved your specifications for the display of the elements to be plotted and if you have selected "Layer plot" for this picture in menu 4, you get to menu 10 for the selection of the layer(s) to be used for this layer plot.

6.1 LAYER SELECTION

PROCEDURE :	OUTPUT FILE :	
PLOTTER :	USAGE :	10
PICTURE NO. :	PLAN :	LAYER :

Selection of signal layer(s), mark only 1 layer for prod.plots			
[0]	(1) —	[ ]	( ) —
	(0) —		( ) —
[ ]	( ) —	[ ]	( ) —
	( ) —		( ) —
Number of layer pairs :		Paging (+/-) >+<	

Superposing documentation layers:							
(D0)	(D1)	(D2)	(D3)	(D4)	(D5)	(D6)	(D7)
(D8)	(D9)	(D10)	(D11)	(D12)	(D13)	(D14)	(D15)

HANDLING INSTRUCTIONS

The numbers of the layer pairs and signal layers defined in the .adb-file automatically appear in the menu.

You are also informed about how many layer pairs exist (e.g. "Number of layer pairs : 4 ").

If the board has more than four layer pairs, you can select the further layer pairs by pressing <CR> on the input field >+< at the end of this part of the menu. You can go back by entering "-" or pressing <CR> if "-" is already entered.

Several signal layers may only be selected for control plots for the superimposed layer display.

After selecting the signal layer(s), <CR>, you get to the lower part of the menu and can select the documentation layer(s), which superimpose this signal layer(s).



## 6.2 SELECTION OF THE LAYER ELEMENTS OF SIGNAL LAYERS

When you have saved the layer selection or the mirroring specifications, <CR>, menu 24 appears for the selection of the elements to be plotted of the first specified signal layer.

### 6.2.1 MENU 24

PROCEDURE :	OUTPUT FILE :	
PLOTTER :	USAGE :	
PICTURE NO. :	PLAN :	LAYER :

24

Selection of the elements to be plotted

VIAS : \_ SEGMENTS : \_ TEXT FIELDS : \_ COPPER AREAS : \_

Plot only vias which are connected on this layer : \_

Select components : \_ Select signals : \_

### 6.2.2 MENU DESCRIPTION

#### VIAS:

When selecting this function, the specified vias will be plotted (ref. to "Select components / select signals").

#### SEGMENTS:

The specified segments and copper areas of this layer will be plotted (ref. to "Select components / select signals").

#### TEXT FIELDS:

When selecting this function, the specified text fields (ref. to "Select components / select signals") of this signal layer will be plotted.

#### COPPER AREAS:

For PRISMA, this function is contained in the function "SEGMENTS" and should therefore not be selected.

**Plot only vias, which are connected on this layer:**

If this function is selected, only the vias connected on this layer will be displayed in the layer plot.

If you do not select this function, all vias existing on this layer will be plotted.

**SELECT COMPONENTS / SELECT SIGNALS:**

The two selection possibilities (for components and for signals) are connected as an 'AND' link, i.e. **even if only one of the two selections has been activated, the other selection is also active automatically!**

So only 'selection active' (regardless if one or both selection possibilities have been marked) or 'selection switched off' (no selection) is possible.

not selected:

- if both selections haven't been activated, no selection is effected, i.e. all elements marked one menu before (vias, segments + copper areas, texts) will be plotted.
- if only one of the two selections has been activated, no element of the other not activated selection will be plotted!

Note:

If you want to plot e.g. all signals but only some components, you have to mark both selections; after the component selection has been effected, you specify "all but ..." in the signal selection and press <CR>.

selected:

An additional menu appears, in which a selection sorted after components or signals is possible.

Consider all but the specified ones (1)	Show names : _
Consider only the specified ones (2)	
Selection number (1)	
COMPONENT (or SIGNAL)	

Respective to the selection number, which you enter here, only the elements of the signal layer specified in the upper part of the menu will be plotted, which

- (1) do not belong to the specified components or signals. The component or signal names to be omitted are entered in the menu field which appears below.
- (2) belong to the specified components or signals. You enter the desired component or signal names in the menu field which appears below.

### Show names:

When selecting this function (without pressing <CR>!), you get a listing of the signal/component names contained in the file in the upper part of the shell window.

When you are in the lower part of the menu, you have to press the <Cursor up> key to get to this input field.

### Note:

In the appendix to the POSTPROCESSING description, you will find an example that illustrates the effects of the different selection possibilities.

### 6.3 FOR THE FURTHER PROCEEDING

After the selection of the elements to be plotted, the respective menus for the exact determination of the element display are called (ref. to the chapter "Mask selection").

If several signal layers (for control plots) were selected, after the determination of the element display for the first signal layer, the menu for the element selection for the second selected signal layer is called, etc.

If documentation layers were also selected in the menu for the layer selection and the specifications for the plotting of the signal layer(s) are finished, the menu for the selection of the elements to be plotted of the first selected documentation layer appears (ref. to the chapter "Element selection for documentation layers"). After the element selection has been finished, the element display can be determined (ref. to the chapter "Mask selection").

### Attention:

The component and/or signal selection on the signal layers has no effect on the element selection on the assigned documentation layers.

After all documentation layers have been processed, you return to menu 4 for the definition of the 'picture'.

**7. SOLDER RESIST PLAN**

If you have selected the display of the solder resist plan (selection number 2) for a picture to be generated in menu 4 for the picture selection, you get to menu 5 for the determination of the kind of display. In the upper part of the menu "SOLDER RESIST PLAN" is entered. Here, you determine how the board elements for the solder resist plan are to be displayed and which special pens are to be used.

**7.1 LAYER SELECTION**

After you have saved the specifications for the display of the elements to be plotted in the solder resist plan, you get to menu 8 for the selection of the layer(s) to be used for this plan.

PROCEDURE	:	OUTPUT FILE	:	
PLOTTER	:	USAGE	:	
PICTURE NO.	:	PLAN	:	LAYER : <input type="text" value="8"/>

SELECTION of the signal layer(s):	
Solder side	_ (0)
Component side	_ (1)
Mark only one layer for production plots!	

superposing documentation layers:							
(D0)	(D1)	(D2)	(D3)	(D4)	(D5)	(D6)	(D7)
(D8)	(D9)	(D10)	(D11)	(D12)	(D13)	(D14)	(D15)

**HANDLING INSTRUCTIONS**

Both layers (solder and component side) must only be selected for control plots for the display of superimposed layers.

After selecting the layer(s), <CR>, the respective documentation layers are automatically selected by OUTPUT (for the solder side: D2 = layer 19 in the layer plan; for the component side: D3 = layer 20 in the layer plan).

## 7.2 SELECTION OF THE SIGNAL LAYER ELEMENTS THAT ARE NOT TO BE COVERED

After you have stored the layer selection with <CR>, menu 16 appears for the selection of the vias/pins that are not to be covered (= to be plotted) of the first selected signal layer. The number of the activated layer is entered in "LAYER" in the upper part of the menu.

PROCEDURE	:	OUTPUT FILE	:	
PLOTTER	:	USAGE	:	16
PICTURE NO.	:	PLAN	:	LAYER :

SELECTION of the elements that are not to be covered	
pins	: x
SMD-pins	: _
changes	: _

Enlargement/reduction for SMD-pins	: 0.000	(In.)
------------------------------------	---------	-------

### Handling instructions

The elements to be plotted in the solder resist plan (=not to be covered) are chosen by putting an "X" in the respective field.  
OUTPUT uses all solder resist sizes defined for this file in "Track technology" and "Via technology".

### 7.2.1 Special case --> Glue plan for SMD-'reflow'-procedure

In the last line of the menu, you can specify the enlarged/ reduced representation of SMD-pins. This is a special function for the creation of the glue plan for a 'reflow'-procedure (the pins of the SMD component are the glue pads for SMDs). This plan should be created separately from the solder resist plan (select only "SMD-pins" and enter a respective reduction).

#### Attention:

The enlargement/reduction you enter here refers to the solder resist sizes of the respective track types that were defined in the technology!

### 7.3 For the further proceeding

After the selection of the elements to be plotted, the respective menus for the exact determination of the element display are called (ref. to the chapter "Mask selection").

If solder and component side (for a control plot) were selected, after the determination of the element display for the first signal layer, the menu for the element selection for the second selected signal layer is called.

If documentation layers were also selected in the menu for the layer selection and the specifications for the plotting of the solder resist layer(s) are finished, the menu for the selection of the elements to be plotted of the first selected documentation layer appears (ref. to the chapter "Element selection for documentation layers"). After the element selection has been finished, the element display can be determined (ref. to the chapter "Mask selection").

**Attention:**

- The via/pin selection on the signal layers has no effect on the element selection on the assigned documentation layers.
- Sheet frame, board frame, adjustment marks, sizes and legend are only displayed in the solder resist plan if a documentation layer will also be plotted in the respective 'picture'.

After all documentation layers have been processed, you return to menu 4 for the definition of the 'picture'.

## 8. SILK SCREEN

If you have selected the display of the silk screen (selection number 3) for a picture to be generated in menu 4, you get to menu 5 for the determination of the kind of display. In the upper part of the menu "SILK SCREEN" is entered. Here, you determine how the board elements for the silk screen are to be displayed and which special pens are to be used.

### 8.1 LAYER SELECTION

After you have saved the specifications for the display of the elements to be plotted in the silk screen, you get to menu 9 for the selection of the layer(s) to be used for this plan.

PROCEDURE	:	OUTPUT FILE	:	
PLOTTER	:	USAGE	:	
PICTURE NO.	:	PLAN	:	LAYER : <input type="text" value="9"/>

SELECTION of the component layer(s):	
Solder side	_ (0)
Component side	_ (1)
Mark only one layer for production plots!	

superposing documentation layers:							
(D0)	(D1)	(D2)	(D3)	(D4)	(D5)	(D6)	(D7)
(D8)	(D9)	(D10)	(D11)	(D12)	(D13)	(D14)	(D15)

### HANDLING INSTRUCTIONS

Both component layers (solder and component side) must only be selected for control plots for the superimposed display of the two silk screens.

After selecting the component layer(s), <CR>, the respective documentation layer(s) in the lower part of the menu are automatically selected by OUTPUT (solder side -> D0 = layer 17 in the layer plan; component side -> D3 = layer 18 in the layer plan).

#### Attention:

CIS/PCB automatically stores the component frame segments and the text fields with the component names on the assigned component-documentation layer.

On the same documentation layer, user defined elements (segments, text fields) may also exist.

All these elements will not be plotted if the cross that OUTPUT has automatically placed in front of the documentation layer during the layer selection will be deleted by the user!

## 8.2 COMPONENT SELECTION OF COMPONENT LAYERS

After you have stored the layer selection with <CR>, menu 21 appears for the selection of the components of the first selected layer that have to be considered during the generation of the component frames and the component names.

The number of the activated layer is entered in "LAYER" in the upper part of the menu.

### 8.2.1 MENU 21

PROCEDURE	:	OUTPUT FILE	:	
PLOTTER	:	USAGE	:	
PICTURE NO.	:	PLAN	:	LAYER :
				21

Consider all components for the silk screen (Y/N) : N
---

### 8.2.2 MENU DESCRIPTION

Consider all components for the silk screen:

Y:

All components of the signal layer specified in the upper part of the menu are to be plotted.

N:

An additional menu is called, in which the components can be selected (see next page).



Consider all but the specified ones (1) Consider only the specified ones (2) Selection number (1)	Show names : _

Respective to the selection number, which you enter here, only the components of the component layer specified in the upper part of the menu will be plotted, which

- (1) are not specified. You enter the names of the components which are not to be considered in the next menu field.
- (2) are specified. You enter the names of the components to be considered in the next menu field.

**Show names:**

When selecting this function (without pressing <CR>!), you get a listing of the components existing on this layer in the upper part of the shell window.

When you are in the lower part of the menu, you have to press the <Cursor up> key to get to this input field.

### 8.3 COMPONENT DISPLAY

#### PRODUCTION PLOT

As CIS/PCB automatically stores the component frame segments and the text fields with the component names on the assigned component-documentation layer, the respective documentation layer must be marked in the menu.

After the component selection, the menu for selecting the elements to be plotted appears (ref. to chapter "Element selection for documentation layers").

In order to plot the component frames, you have to select 'segments'; to plot the component names, 'texts' must be selected.

The mask selection for plotting the component frames (and all other segments existing on this documentation layer) is effected in the menu that appears next (ref. to chapter "Mask selection").

For production plots, the text fields with the component names are automatically created with the plot mask defined in menu 5 in "Special pen for texts".

#### CONTROL PLOT

If the current 'picture' is a control plot, the mask selection for the display of the component frames and names is effected as usual, i.e a mask is specified for each layer (ref. to the chapter "Mask selection").

**9. DRILL PLAN**

If you have selected the display of the drill plan (selection number 4) for a picture to be generated in menu 4 for the picture selection, you get to menu 5 for the determination of the kind of display. In the upper part of the menu "DRILL PLAN" is entered. Here, you determine how the board elements for the drill plan are to be displayed and which special pens are to be used.

**9.1 LAYER SELECTION**

After you have saved the specifications for the display of the elements to be plotted in the drill plan, you get to menu 11 for the selection of the layer pairs and documentation layers to be used for this plan.

PROCEDURE :	OUTPUT FILE :	
PLOTTER :	USAGE :	11
PICTURE NO. :	PLAN :	LAYER :

Selection of signal layer(s), mark only 1 layer for prod.plots			
[0]	(1) —	[ ]	( ) —
	( ) —		( ) —
[ ]	( ) —	[ ]	( ) —
	( ) —		( ) —
Number of layer pairs :		Paging (+/-) >+<	

Superposing documentation layers:							
(D0)	(D1)	(D2)	(D3)	(D4)	(D5)	(D6)	(D7)
(D8)	(D9)	(D10)	(D11)	(D12)	(D13)	(D14)	(D15)

**HANDLING INSTRUCTIONS**

The numbers of the layer pairs and signal layers defined in the .adb-file automatically appear in the menu.

You are also informed about how many layer pairs exist (e.g. "Number of layer pairs : 4 ").

If the board has more than four layer pairs, you can select the further layer pairs by pressing <CR> on the input field >+< at the end of this part of the menu. You can go back by entering "-" or pressing <CR> if "-" is already entered.

Several layer pairs may only be selected for control plots.

After selecting the layer pairs, <CR>, you get to the lower part of the menu and can select the assigned documentation layer(s).

## 9.2 SELECTION OF THE KIND OF VIA FOR THE DRILL PLAN

After you have stored the layer pair and documentation layer selection with <CR>, menu 30 appears for the selection of the vias/pins to be plotted. If you selected several layer pairs for a control plot, the menu is called for each layer pair. The number of the activated layer pair is entered in "LAYER" in the upper part of the menu.

### 9.2.1 MENU 30

PROCEDURE :	OUTPUT FILE :	
PLOTTER :	USAGE :	
PICTURE NO. :	PLAN :	LAYER : <input type="text" value="30"/>

SELECTION		
drilled through: _	buried: _	blind: _
through-connected: _	not through-connected : _	NC-routing contours : _

### 9.2.2 MENU DESCRIPTION

By selecting the respective functions, OUTPUT gets the information which via and drill types shall be selected from the technology for the drill plan.

#### drilled through:

The drills of all via types that are specified as drilled through (= attribute "D") in the via technology shall be plotted.

This specification applies to the vias and pins that go through all layers, as well as to the mounting holes. These elements can be selected with the parameters "through-connected" and "not through-connected".

#### buried:

The drills for the vias that go only through one layer pair shall be plotted.

#### through-connected:

If this parameter has been selected, the through-connected vias/pins and/or the buried vias will be plotted depending on your entries in "drilled through" and/or "buried". (A drill type defined as "through-connected" in the drill technology must be assigned to the respective via types in the via technology.)

#### not through-connected:

If "drilled through" and this parameter are selected, the mounting holes will be plotted.

**NC-routing contours:**

The tracks digitized for the marking of the NC-routing contours shall be plotted (not yet implemented).

**9.3 DISPLAY OF THE DRILLS**

**9.3.1 PRODUCTION PLOT**

After you have saved the via selection, <CR>, menu 31 for the exact determination of the drill display is called.

PROCEDURE :	OUTPUT FILE :		
PLOTTER :	USAGE :		31
PICTURE NO. :	PLAN :	LAYER :	

DRILL TYPE	DIAMETER	MASK	LOAD SYMBOL
0			
1	0.026	---	_____
2	0.026	---	_____
3	0.026	---	_____
4	0.031	---	_____
5	0.031	---	_____
6	0.031	---	_____
7	0.051	---	_____
		Paging (+/-) >_<	

**HANDLING INSTRUCTIONS**

For the drill types defined in CIS/PCB, the specified diameter values appear in the column "Diameter". Masks (and load symbols) must be entered for these types.

As up to 32 drill types can be defined, you have to page in this menu.

In order to page down, enter "+" in the respective input field. You get to the next page and the cursor remains on the input field so that you can enter "+" again or "-" if you want to page up.

If the cursor is already located on the input field and "+" or "-" is displayed there, paging is also possible by just pressing <CR>.

If all masks have been specified correctly and the cursor is not on the input field for the paging, the information is stored by pressing <CR>.

**PLOTTING THE DRILLS**

For each drill type to be plotted, the number of a plot mask defined in COLUMBINE and a load symbol (= macro) defined in COLUMBINE must be specified.

Note:

By entering "\*" instead of a load symbol, all macros defined in COLUMBINE can be listed. By pressing <CR>, the name of the load symbol that was listed last will be entered.

Attention:

- If no load symbol is specified, OUTPUT expects the number of a flash mask in "Mask"!
- Independent from the drill diameters defined in the drill technology, the macros are always displayed with the size of 2,2 in. if enlargement factor 1 was specified for this 'picture' in menu 4.

If an enlargement factor unequal 1 was specified, the display value 2,2 is enlarged or minimized by this factor.

**FLASHING THE DRILLS**

For each drill type to be flashed, the number of a flash mask defined in COLUMBINE must be specified.

**9.3.2 CONTROL PLOT**

For control plots, all drills of the via kind selected in menu 30 will be flashed with the same mask or plotted with the same pen and the same load symbol.

Therefore, menu 33 appears that corresponds in principle to the standard menu for the element display for control plots (ref. to chapter "Mask selection").

PROCEDURE :	OUTPUT FILE :		
PLOTTER :	USAGE :		33
PICTURE NO. :	PLAN :	LAYER :	

Mask : ____	Load symbol : _____	Flash (Y/N) : N
-------------	---------------------	-----------------

**PLOTTING THE DRILLS**

For each drill type to be plotted, the number of a plot mask defined in COLUMBINE and a load symbol (= macro) defined in COLUMBINE must be specified.

Note:

By entering "\*" instead of a load symbol, all macros defined in COLUMBINE can be listed. By pressing <CR>, the name of the load symbol that was listed last will be entered.

**Attention:**

- If no load symbol is specified, OUTPUT tries to flash with the specified mask even if "N" is entered in "Flash (Y/N)" and the specified mask is a plot mask!
  - Independent from the drill diameters defined in the drill technology, the macros are always displayed with the size of 2,2 in. if enlargement factor 1 was specified for this 'picture' in menu 4.
- If an enlargement factor unequal 1 was specified, the display value 2,2 is enlarged or minimized by this factor.

**FLASHING THE DRILLS**

If the drills are to be flashed, the number of a flash mask defined in COLUMBINE must be specified. You must not enter a load symbol then.

**9.4 FOR THE FURTHER PROCEEDING**

If several layer pairs were selected, after the determination of the element display for the first layer pair, menu 30 for the element selection for the second selected layer pair is called, etc.

If documentation layers were also selected in the menu for the layer selection and the specifications for the plotting of the drills are finished, the menu for the selection of the elements to be plotted of the first selected documentation layer appears (ref. to the chapter "Element selection for documentation layers").

After the element selection has been finished, the element display can be determined for this documentation layer (ref. to the chapter "Mask selection").

The same procedure will be repeated for all other selected documentation layers.

**Attention:**

Sheet frame and board frame are only displayed in the drill plan if a documentation layer will also be plotted in the respective 'picture'.

**10. NEGATIVE PLAN**

If you have selected the display of the negative plan of plane layers (selection number 7) for a picture to be generated in menu 4, you get to menu 5 for the determination of the kind of display. In the upper part of the menu "NEGATIVE PLAN" is entered. Here, you determine how the elements for the negative plan are to be displayed and which special pens are to be used.

**10.1 LAYER SELECTION**

After you have saved the specifications for the display of the elements to be plotted in the negative plan, you get to menu 12 for the selection of the layer(s) to be used for this plan.

PROCEDURE :	OUTPUT FILE :	
PLOTTER :	USAGE :	12
PICTURE NO. :	PLAN :	LAYER :

Selection of signallayer(s), mark only 1 layer for prod.plots			
[ 0 ]	(1) —	[ ]	( ) —
GND	( ) —	[ ]	( ) —
[ ]	(2) —	[ ]	( ) —
	(0) —	[ ]	( ) —
Number of layer pairs : 2		Paging (+/-) >+<	

Superposing documentation layers:							
(D0)	(D1)	(D2)	(D3)	(D4)	(D5)	(D6)	(D7)
(D8)	(D9)	(D10)	(D11)	(D12)	(D13)	(D14)	(D15)

**HANDLING INSTRUCTIONS**

The numbers of the layer pairs and signal layers defined in the .adb-file, as well as the signal names of the planes automatically appear in the menu. You are also informed about how many layer pairs exist (e.g. "Number of layer pairs : 2 ").

If the board has more than four layer pairs, you can select the further layer pairs by pressing <CR> on the input field >+< at the end of this part of the menu. You can go back by entering "-" or pressing <CR> if "-" is already entered.

For production and control plots, you can only select one plane layer for a 'picture'.



**Attention:**

If there are no plane signal names in the layer menu, no planes have been defined in the respective file. In this case, the menu can only be left with <ESC><ESC>.

After selecting a plane layer, <CR>, you get to the lower part of the menu and can select the superposing documentation layer(s).

## 10.2 ENLARGEMENT OF THE VIAS THAT DO NOT BELONG TO THE PLANE SIGNAL

After you have finished the layer selection, menu 34 appears:

PROCEDURE	:	OUTPUT FILE	:	
PLOTTER	:	USAGE	:	
PICTURE NO.	:	PLAN	:	LAYER : <input type="text" value="34"/>

Enlarge VIAS not connected to plane by : 0.00__ (In.)
---

Here, you specify with which enlargement the vias that do not belong to the plane are to be displayed in the negative plan --> this enlargement is not evaluated for the flashing!

**Note:**

After entering the via enlargement in menu 34 and pressing <CR>, a submenu appears for specifying additional plane signals on the respective plane layer. This menu is only intended for Calay V04-systems and is not necessary for PRISMA users!

Please leave this menu with <CR>.

**10.3 DISPLAY OF THE VIAS AND HEAT TRAPS FOR CONTROL PLOTS**

If the current picture was defined as control plot and if you have stored your entries, menu 35 appears:

PROCEDURE	:	OUTPUT FILE	:	
PLOTTER	:	USAGE	:	
PICTURE NO.	:	PLAN	:	LAYER :

35

**Pen assignment for negative plan**

Signal: \_\_\_ Mask: 1\_\_ Load symbol: \_\_\_\_\_ Flash (Y/N): N

Others Mask: 1\_\_ Load symbol: \_\_\_\_\_ Flash (Y/N): N

**HANDLING INSTRUCTIONS**

- a. In "Signal", the signal name for the activated plane layer is automatically entered.

Behind it, you enter either a respective flash mask or a plot mask and a load symbol for the display of the heat traps for the vias/pins that will be connected to this plane layer.

**Attention:**

If you only enter a pen (= plot mask) for plotting the display of the heat traps, but no load symbol, not the heat traps will be plotted but the vias (in the shape and size defined in the technology)!

- b. In the next line (behind "Others"), you enter either a flash mask or a plot mask and a load symbol for the display of the vias/pins that exist on this layer but do not belong to a signal which shall be connected to this plane layer. If you only enter a plot mask (= pen) but no load symbol, the vias/pins that do not belong to the plane signal will be plotted with the shape defined in the technology.

During the plot (= using a plot mask), OUTPUT enlarges the diameter (defined in CIS/PCB) of the vias/pins by the value specified in menu 34.

**Attention:**

The via enlargement specified in menu 34 will not be considered for the flashing. Therefore, the flash mask must be specified with a size that already contains the intended enlargement!

**Notes:**

- The specified masks and load symbols must be defined in COLUMBINE.

## POSTPROCESSING/Negative Plan

- If you enter "\*" instead of a load symbol, all macros defined in COLUMBINE will be listed.  
By pressing <CR>, the name of the load symbol that was listed last will be entered.
- If the heat traps and/or the vias/pins that do not belong to the plane signal are to be flashed, "Y" must always be entered in "Flash".  
The specified mask must be defined as flash mask in COLUMBINE in this case.

### 10.4 DISPLAY OF VIAS/HEAT TRAPS FOR PRODUCTION PLOTS

If the 'picture' for this negative plan was defined as artwork and menu 34 is finished, menu 36 appears.

PROCEDURE :	OUTPUT FILE :	36
PLOTTER :	USAGE :	
PICTURE NO. :	PLAN :	

VIA TYP	WIDTH (In.)	AUTOMAT. MASK SEL.	MASK A	MASK B	DIAMETER (In.)	LOAD SYMBOL	FLASH (Y/N)
0	0.000 0.011	Y	1		A:0.011 B:	_____	N
1	0.000 .040	Y	5		A:0.027 B:	_____	Y
2	0.000 .055	Y	5		A:0.027 B:	_____	Y
3	0.000 0.011	Y	1		A:0.011 B:	_____	N

Vias >>> not <<< connected to plane Paging (+/-) >\_<

The menu corresponds in principle to the standard menu for the via display for production plots (ref. to chapter "Mask selection").

#### Vias that do not belong to the plane signal

In difference to the standard menu, a line appears in the lower part of the menu which specifies that the via types do not belong to the plane signal.

The entered via widths correspond to the widths from the CIS/PCB technology plus the via enlargement specified in menu 34.  
The automatic mask selection refers to these enlarged values!

### Heat traps

After you have finished your entries in menu 36, the same menu (now menu no. 37) is called. Here, you can determine the masks and load symbols for the display of the heat traps of the via types that belong to the plane signal.

In the lower part of the menu, the following line appears:

Vias connected to plane signal	Paging (+/-) >_<
--------------------------------	------------------

The via widths entered in the menu correspond to the widths from the CIS/PCB technology.

**Note:**

For the procedure concerning the mask and load symbol selection, ref. to the chapter "Mask selection".

### 10.5 FOR THE FURTHER PROCEEDING

After the display of the vias and heat traps of the selected plane has been defined, the elements to be plotted of the first selected documentation layer are selected.

Then the element display can be determined for this documentation layer (ref. to the chapter "Mask selection").

If several documentation layers exist, this procedure is repeated for each layer. After all documentation layers have been processed, you return to menu 4 for the definition of the 'picture'.

If the negative plan shall be created for another plane layer, the same procedure will be executed for this plane, as described above.

**11. ELEMENT SELECTION FOR DOCUMENTATION LAYERS**

After you have finished the signal layer(s) or the planes, the standard menu for the element selection of documentation layers appears, independent from the kind of plan.

PROCEDURE	:	OUTPUT FILE	:	
PLOTTER	:	USAGE	:	
PICTURE NO.	:	PLAN	:	LAYER : <input type="checkbox"/>

SELECTION of the elements to be plotted								
SEGMENTS	:	__	TEXT FIELDS	:	__	COPPER AREAS	:	__

**HANDLING INSTRUCTIONS**

This menu appears for the first documentation layer that has been selected; a respective specification is displayed in "Layer".

By selecting the respective functions, you determine which elements of this documentation layer are to be plotted.

Subsequently, you define how the elements shall be displayed (mask/load symbol selection).

In regard to the segment and text field display, the proceeding during the mask selection is dependent from the usage of the 'picture' (control or production plot).

For PRISMA, the function "SEGMENTS" contains also the display of copper areas. The function "COPPER AREAS" has no effect and should not be used.

For further information, refer to the chapter "Mask selection".

If several documentation layers have been selected, the same procedure must be executed for the next documentation layer, etc.

12. MASK SELECTION

12.1 MASK SELECTION FOR CONTROL PLOTS (MANUAL)

If in menu 4 for the picture definition, the selection number (2) was entered in "Usage" for this 'picture' (=control plot), the following menu appears after the selection of the elements to be plotted of a layer (or a layer pair).

The menu number depends on the respective kind of plan.

The mask selection is effected manually (i.e. the specifications must be entered by the user).

The specifications in this menu apply to the display of

- the vias that only exist on this layer;
- the text fields that only exist on this layer;
- all segments that exist on this layer but may also exist on other layers that have been selected for this 'picture'.

PROCEDURE :	OUTPUT FILE :		
PLOTTER :	USAGE :		
PICTURE NO. :	PLAN :	LAYER :	<input type="checkbox"/>

Mask : ___	Load symbol : _____	Flash (Y/N) : N
------------	---------------------	-----------------

IMPORTANT NOTES:

- Mask/pen and load symbol, which you have to specify here, must be defined in COLUMBINE.
- If "Y" is entered in "Flash (Y/N)" in order to flash the vias that only exist on this layer, the specified mask must be defined as flash mask in COLUMBINE.  
  
If you want to plot segments and/or texts of this layer, the mask must also be defined as plot mask in COLUMBINE.
- If you enter "\*" instead of a load symbol, all macros defined in COLUMBINE will be listed. By pressing <CR>, the name of the macro that was listed last will be entered.
- In control plots, the vias/pins that do not only exist on this layer, but also on other layers that have been selected for this 'picture', will only be created once with the pen/mask specified in "Special pen for control plots" (menu 5), even if several signal layers have been selected in menu 10 (ref. to the chapter "Display of the picture", page 4-11).

A repeated superimposed drawing of these vias is prevented in that way.

- In control plots, the text fields that do not only exist on this layer, but also on other layers that have been selected for this 'picture', will only be created once with the pen specified in "Special pen for control plots" in menu 5 (ref. to the chapter "Display of the picture", page 4-11).

Notes:

- After you have finished your entries, <CR>, the selection menu for the elements to be plotted of the next selected signal layer appears.

Afterwards the mask selection for the display of the elements of this layer is called.

When the procedure is finished for the selected signal layers, the same procedure may be effected for documentation layers.

Then you return directly to menu 4 for the definition of the 'picture'.

- If the determination of the element display differs from the general proceeding, it will be explained in the description of the respective plan (drill plan, glue plan, test point plan, negative plan).

**12.2 MASK SELECTION FOR THE VIA DISPLAY IN PRODUCTION PLOTS**

The mask selection for production plots is effected automatically on principle. Flash masks are searched first.

If a 'picture' was defined as artwork and if vias were selected in the menu for the selection of the layer elements, the standard menu appears for the via display in production plots.

The menu number depends on the kind of plan.

If necessary, the masks and load symbols for the display of the different via types in the current picture can be changed manually.

PROCEDURE :	OUTPUT FILE :			
PLOTTER :	USAGE :			
PICTURE NO. :	PLAN :	LAYER :		<input type="checkbox"/>

VIA TYP	WIDTH (In.)	AUTOMAT. MASK SEL.	MASK A	MASK B	DIAMETER (In.)	LOAD SYMBOL	FLASH (Y/N)
0	0.000 0.011	Y	1		A:0.011 B:	_____	N
1	0.000 .040	Y	5		A:0.027 B:	_____	N
2	0.000 .055	Y	5		A:0.027 B:	_____	N
3	0.000 0.011	Y	1		A:0.011 B:	_____	N
						Paging (+/-) >_<	

**MENU DESCRIPTION**

**VIA TYPE:**

Here, the via types contained in the via technology of the adb-file appear.

**WIDTH:**

Here, the via sizes defined in the CIS/PCB technology appear. The first value stands for the via size 'square' (=width of the core). If "0.00" is entered, you have round vias.

The second value entered below stands for the 'total' via size. If both values are the same, you have square vias.



If the value for the 'total' via size is greater than the value for the via size 'square', you have square vias with rounded corners. (Ref. to CIS/PCB technology).

### FLASH (Y/N):

**Y:**

If automatic mask selection has been selected and OUTPUT finds a suitable flash mask or, for square vias with rounded corners, a mask for flashing the 'inner circle' of the via, "Y" is entered here.

If the user enters "Y" for the manual mask selection, OUTPUT uses the specified mask regardless if it is suitable or not, provided that it has been defined as flash mask in COLUMBINE (if it has not been defined as flash mask, a respective message appears).

**N:**

If OUTPUT does not find a suitable flash mask during the automatic mask selection, "N" is automatically entered here.

For the manual mask selection, the user must enter "N" if mask A (existing as plot and flash mask) shall be used.

### AUTOM. MASK SELECTION:

**Y:**

When calling this menu for the first time, "Y" is entered in "AUTOMATIC MASK SELECTION" for each via type whose width is  $> 0$  if the automatic mask selection has been successful.

The proceeding during the automatic mask selection depends on the shape of the via type.

1. For round vias (ref. to CIS/PCB technology)
  - a. If a suitable round flash mask (corresponds to the technology  $\pm$  exactness) is found, it will be used for the via creation.

The number of the selected mask is entered in "MASK A", its diameter appears in "A."; "Y" is entered in "FLASH"; for mask B, no entry is made.

- b. If no suitable flash mask is found, a round plot mask is searched in COLUMBINE whose width corresponds to half of the via width.

If this search was not successful, the next smaller round plot mask defined in COLUMBINE for this plotter will be used for the creation of this via type.

The number of the selected mask is entered in "MASK A", its diameter appears in "A."; for mask B, no entry is made; "N" is entered in "FLASH".

If the smallest plot mask is bigger than half of the via width, the message "Technology violation" appears.

Now "N" is shown in "AUTOMATIC MASK SELECTION"; the cursor automatically moves to the field where this entry appears and expects a respective manual change.

2. For square vias (ref. to CIS/PCB technology).
- a. If a suitable square flash mask (corresponds to the technology  $\pm$  exactness) is found, it will be used for the via creation.

The number of the selected mask is entered in "MASK A", its diameter appears in "A."; for mask B, no entry is made; "Y" is entered in "FLASH".

- b. If no suitable flash mask is found, OUTPUT searches a square plot mask whose width corresponds to half of the via width. With this mask, the via will be plotted, starting from the center.

The number of the selected mask is entered in "MASK A", its diameter appears in "A."; for mask B, no entry is made; "N" is entered in "FLASH".

- c. If this mask is not found, a round plot mask is searched whose width corresponds to half of the via width. If the search was not successful, the next smaller round plot mask will be used.

The number of the selected mask is entered in "MASK A", its diameter appears in "A."; "N" is entered in "FLASH".

In addition, the smallest available round plot mask is searched in order to trace the via contours and fill the remaining gaps (= via corners). The number of the selected mask is entered in "MASK B", its diameter appears in "B:"

If the smallest plot mask is bigger than half of the via width, the message "Technology violation" appears.

Now "N" is shown in "AUTOMATIC MASK SELECTION"; the cursor automatically moves to the field where this entry appears and expects a respective manual change.

3. For square vias with rounded ends (ref. to CIS/PCB technology).
- a. If a suitable flash mask (corresponding to the technology) is found, it will be used for the via creation.

The number of the selected mask is entered in "MASK A", its diameter appears in "A."; for mask B, no entry is made; "Y" is entered in "FLASH".

- b. If no suitable flash mask is found, OUTPUT tries to find a round flash mask whose diameter corresponds to the 'total' via width. With this mask, the 'inner circle' of the via will be flashed.

The number of the selected mask is entered in "MASK A", its diameter appears in "A."; "Y" is entered in "FLASH".

In order to trace the via contours and fill the corners, a round plot mask will be searched whose diameter fulfills the following condition

$$D = \text{'total' via size minus via size 'square'}$$

If this mask cannot be found, the next smaller round plot mask will be used.

The number of the selected mask is entered in "MASK B", its diameter appears in "B."

- c. If no round flash mask was found whose width corresponds to the 'total' via width, a round plot mask will be searched whose width corresponds to half of the via width 'total'. With this mask, the via will be plotted, starting from the center.

If it cannot be found, the next smaller available mask is used.

The number of the selected mask is entered in "MASK A", its diameter appears in "A."; "N" is entered in "FLASH".

For the creation of the remaining part (corners and frame) see "b."

If the smallest plot mask is bigger than half of the via width and/or bigger than the 'total' via width minus via width 'square', the message "Technology violation" appears.

Now "N" is shown in "AUTOMATIC MASK SELECTION"; the cursor automatically moves to the field where this entry appears and expects a respective manual change.

**N:** (= manual mask selection)

- a. If during the automatic mask selection OUTPUT has to use the smallest plot mask and this is larger than half of the via width (or for square via types with rounded corners: larger than the 'total' via width minus via width 'square'), the message "Technology violation" appears and "N" is entered in "AUTOMATIC MASK SELECTION".

The cursor automatically moves to the respective line and you can specify a mask of your choice. Now OUTPUT accepts each mask specification (even the mask that has not been accepted during the automatic selection), provided that the mask has been defined in COLUMBINE.

- b. If you want to change mask specifications or if you want to use a load symbol for the via display in spite of the automatic mask selection, you have to enter "N" in "AUTOMATIC MASK SELECTION".

Subsequently, you can change the mask and load symbol specifications in the respective line. Then OUTPUT uses the specified flash or plot masks (provided that they have been defined in COLUMBINE for the respective device).

- c. If you enter the masks and the load symbol manually and then decide to use the automatic mask selection, just press <ESC><ESC> to effect an automatic mask selection for the respective via type.

**MASK A :**

1. If the automatic mask selection has been successful, the number of the chosen mask appears here automatically.
2. Non-automatic mask selection without load symbol specification

By entering the respective mask number in "MASK A", you specify:

a. for flash masks:

- with which mask this via type is to be flashed (requires the entry "Y" in "FLASH").

**Attention:**

If you have square vias with rounded corners and if only the 'inner circle' shall be flashed with this mask, you have to specify a pen in "MASK B" with which the remaining part of the vias can be plotted.

b. for round plot masks:

- with which round pen the 'inner circle' of square vias  
or
- with which round pen the 'inner circle' of square vias with rounded corners shall be plotted.

**Attention:**

In both cases, a mask specification must be effected in "MASK B", otherwise the via corners will not be plotted and you get round vias in the plot.

c. For rectangular plot masks:

- with which rectangular pen the whole square via is to be plotted.

**Note:**

A specification in "MASK B" is not necessary in this case.

3. Non-automatic mask selection with load symbol specification

By entering the respective mask number in "MASK A", you specify:

- with which pen the contours of the load symbol shall be plotted.

**MASK B :**

1. If the automatic mask selection has been successful, the number of the chosen mask appears here automatically.
2. Non-automatic mask selection without load symbol specification

By entering the respective mask number in "MASK B", you specify:

- with which (round!) plot mask the remaining parts (corners and contours) of the vias shall be plotted.

Refer also to "MASK A", point a. and b.

### 3. Non-automatic mask selection with load symbol specification

By entering the respective mask number in "MASK B", you specify:

- with which (round!) plot mask the load symbol (whose contours have been plotted with mask A) shall be filled.

**Attention:**

If no specification is made in "MASK B", the load symbol will not be filled!

The mask must be defined as round plot mask in COLUMBINE.

**DIAMETER:**

The diameters for mask A (and for mask B), which are defined in COLUMBINE, appear here.

**LOAD SYMBOL:**

For the 'manual' mask specification, you determine the kind of display for each via type by specifying a load symbol (defined in COLUMBINE, section Graphics/"Macros").

If you enter "\*" instead of a load symbol, all macros defined in COLUMBINE will be listed. By pressing <CR>, the name of the macro that was listed last will be entered.

**Paging (+/-):**

In the CIS/PCB technology, up to 32 via types can be defined. Accordingly, the menu for the mask selection of the vias also contains 32 types that cannot be listed on one page.

For that reason, you have to page in this menu.

In order to page down, enter "+" in the respective input field. You get to the next page and the cursor remains on the input field so that you can enter "+" again or "-" if you want to page up.

If the cursor is already located on the input field and "+" or "-" is displayed there, paging is also possible by just pressing <CR>. In this case, the cursor moves to the first input field for the mask selection on the respective page and not on the input field for the paging!

**HANDLING INSTRUCTIONS**

- If the cursor is not on the field >+< or >-< and you press <CR>, the entered data is saved and you leave the menu if there are no technology violations during the automatic mask selection.

Therefore, you should only use the <CR> key for paging if you want to make entries or check the mask selection for further via types.

- With <ESC><ESC> in this menu, you do not leave the menu but activate another automatic mask selection.
- If you want to edit a defined 'picture', for which the automatic mask selection has already been effected, and specify another device name in menu 3, you have to go through all submenus of this 'picture' (incl. the menus for the mask selection) so that the masks of the current device will be accepted.

When you get to the menu described here, you have to press <ESC><ESC> that the automatic mask selection will be executed with the masks of the new device.

**12.3 SEGMENT DISPLAY IN PRODUCTION PLOTS**

The mask selection for production plots is effected automatically on principle.

If a 'picture' was defined as artwork and if segments were selected in the menu for the selection of the layer elements, the standard menu appears for the segment display in production plots.

The menu number depends on the kind of plan.

If necessary, the masks and load symbols for the display of the different via types in the current picture can be changed manually.

PROCEDURE :	OUTPUT FILE :		
PLOTTER :	USAGE :		
PICTURE NO. :	PLAN :	LAYER :	<input type="checkbox"/>

SEG TYPE	WIDTH (In.)	AUTOMAT. MASK SEL.	MASK A	MASK B	DIAMETER (In.)
1	0.000 0.011	Y	1		A: B: A:0.011 B:
2	0.000 0.017	Y	1		A:0.011 B:
3	0.000 .040	Y	5		A:0.027 B:
				Paging (+/-) >_<	

**MENU DESCRIPTION**

**SEG.TYPE:**

Here, the segment types defined in the CIS/PCB technology of the adb-file are automatically entered.

**WIDTH:**

The widths assigned to the segment types in the CIS/PCB technology are automatically entered. The first value stands for the segment width 'rectangle'. The value entered below stands for the "total" segment width. If the first value is "0.00", you have a track type with round (=semi-circular) ends (standard-SMD-pin).

If both values are the same, you have a track type which only consists of the core (e.g. for SMD-rectangle pins).

If the first value is  $> 0 <$  second value, you have a track type with semi-circular (=rounded) ends. (Ref. to CIS/PCB technology).

### AUTOM. MASK SELECTION:

**Y:**

The program OUTPUT automatically searches for suitable pens in the library COLUMBINE and enters "Y" in "AUTOMATIC MASK SELECTION" for all track types whose width is  $> 0$  if the automatic mask selection is successful for these types.

The search of the plot mask depends on the shape of the track types (defined in the CIS/PCB technology).

#### 1. Track types with round corners (rectangle width = 0)

A round plot mask is searched whose width corresponds to the track width. If this search was not successful, the next smaller mask will be used.

The number of the selected mask is entered in "MASK A", its diameter appears in "A."; for mask B, no entry is made.

If the smallest pen defined for this plotter is bigger than half of the track width, the message "Technology violation" appears.

Now "N" is shown in "AUTOMATIC MASK SELECTION"; the cursor automatically moves to the field where this entry appears and expects a respective manual change.

#### 2. Track types with rectangular ends (rectangle width = total width)

Two plot masks (MASK A and MASK B) will be searched:

- a. For the plotting of the track, a round mask (mask A) is searched whose width corresponds to the track width.

If this search was not successful, the next smaller mask will be used. The number of the selected mask is entered in "MASK A", its diameter appears in "A:".

- b. For the plotting of the rectangular ends, the smallest round plot mask defined in COLUMBINE for this plotter will be used.

The number of the selected mask is entered in "MASK B", its diameter appears in "B:".

If the smallest pen defined for this plotter is bigger than half of the track width, the message "Technology violation" appears.

Now "N" is shown in "AUTOMATIC MASK SELECTION"; the cursor automatically moves to the field where this entry appears and expects a respective manual change.



**3. Track types with semi-circular ends (rounded corners)  
(total width > rectangular width)**

Two plot masks (MASK A and MASK B) will be searched:

- a. For the plotting of the track, a round mask (mask A) is searched whose width corresponds to the track width.

If this search was not successful, the next smaller mask will be used. The number of the selected mask is entered in "MASK A", its diameter appears in "A:".

- b. For the plotting of the corners at the segment ends, a plot mask with the diameter

$$D = \text{"total" track width minus track width "rectangle"}$$

will be searched.

If this search was not successful, the next smaller mask will be used. The number of the selected mask is entered in "MASK B", its diameter appears in "B:".

If the smallest plot mask defined for this plotter is bigger than half of the track width, the message "Technology violation" appears.

Now "N" is shown in "AUTOMATIC MASK SELECTION"; the cursor automatically moves to the field where this entry appears and expects a respective manual change.

**N: (= manual mask selection)**

- a. If during the automatic mask selection OUTPUT has to use the smallest plot mask and this is larger than half of the track width (or for segment types with rounded corners: larger than the 'total' track width minus track width 'rectangle'), the message "Technology violation" appears and "N" is entered in "AUTOMATIC MASK SELECTION".

The cursor automatically moves to the respective line and you can specify a mask of your choice. Now OUTPUT accepts each mask specification (even the mask that has not been accepted during the automatic selection), provided that the mask has been defined as plot mask in COLUMBINE.

- b. If you want to change mask specifications in spite of the successful automatic mask selection, you have to enter "N" in "AUTOMATIC MASK SELECTION".

Subsequently, you can change the mask specifications in the respective line.

Then OUTPUT uses the specified plot masks (provided that they have been defined in COLUMBINE for the respective device).

- c. If you enter the masks manually and then decide to use the automatic mask selection, just press <ESC><ESC> to effect an automatic mask selection for the respective track type.

### **MASK A :**

For the non-automatic mask selection, you specify by entering the respective number here with which pen the track segments (independent from the shape of the ends) are to be displayed (ref. to "AUTOMATIC MASK SELECTION", point 1, 2 and 3).

### **MASK B:**

For the non-automatic mask selection, you specify the number of the plot mask here with which the ends of the segments are to be plotted for track types with rectangular or 'semi-circular' ends (ref. to "AUTOMATIC MASK SELECTION", point 2 and 3).

### **DIAMETER:**

The diameters for mask A (and for mask B), which are defined in COLUMBINE, appear here.

### **Paging (+/-):**

In the CIS/PCB technology, up to 32 track types can be defined. Accordingly, the menu for the mask selection for the segment display also contains 32 track types that cannot be listed on one page.

For that reason, you have to page in this menu.

In order to page down, enter "+" in the respective input field. You get to the next page and the cursor remains on the input field so that you can enter "+" again or "-" if you want to page up.

If the cursor is already located on the input field and "+" or "-" is displayed there, paging is also possible by just pressing <CR>. In this case, the cursor moves to the first input field for the mask selection on the respective page and not on the input field for the paging!

### **HANDLING INSTRUCTIONS**

- If the cursor is not on the field >+< or >-< and you press <CR>, the entered data is saved and you leave the menu if there are no technology violations during the automatic mask selection.

Therefore, you should only use the <CR> key for paging if you want to make entries or check the mask selection for further track types.

- With <ESC><ESC> in this menu, you do not leave the menu but activate another automatic mask selection.
- If you want to edit a defined 'picture', for which the automatic mask selection has already been effected, and specify another device name in menu 3, you have to go through all submenus of this 'picture' (incl. the menus for the mask selection) so that the masks of the current device will be accepted.

When you get to the menu described here, you have to press <ESC><ESC> that the automatic mask selection will be executed with the masks of the new device.

## PROCEDURE DEFINITION DRILL

### 1. INTRODUCTION

When you have activated the function "Drill" in menu 2, you get to several menus for the detailed determination of the drill data.

In a procedure, the creation of several drill tapes can be defined. They are stored in different output files.

The output files get the name of the input file and the default extension .bnn; n is a real number from 0 to 9. These files are created during the execution of the procedure (selection number "1" in the postprocessing main menu) and will be stored on the directory pst.

Note:

If the creation of drill tapes was also defined in the procedure, OUTPUT creates a drill report in addition to the respective output files. It is stored in a file with the name of the input file and the extension .drp on the directory pst.

An output file can contain the data for the drilling of buried vias, drilled through vias or the test point holes.

The heading of each menu belonging to the "Drill" function contains information belonging to the following parameters:

- PROCEDURE:** The name of the current procedure is shown here.
- DEVICE:** The name of the selected drilling machine appears here.
- TAPE NO.:** Here, the number of the currently defined drill tape is displayed.
- OUTPUT FILE:** The name of the current output file for the drill tape appears here.
- USAGE:** Here, the usage of the output data (drilling or NC-routing) is shown.
- TAPE TYPE:** Here, you are informed about the elements (buried vias, drilled through vias or test points) selected for this output file.
- LAYER:** The number of the currently treated layer pair is displayed here (e.g. "P1" = layer pair 1)

The specifications to the single parameters will first appear after the menu in which they were defined.

**2. DRILL TAPE DEFINITION**

After the activation of the function "Drill" in menu 2 for the output selection, menu 66 is called.

Here, you enter the name of the drilling machine to be used, the name of the output file and - if desired - a text for the drill tape header. The specified drilling machine must be defined in COLUMBINE, section "Device". In the upper part of the menu, "DRILL" is displayed.

Notes:

- When you enter "\*" in "Name of the drilling machine to be used" and press <CR>, the names of the drilling machines defined in COLUMBINE will be listed. When pressing <CR> again, you return to menu 66; the first device name automatically appears in the menu.
- The extension of the output file can be changed, if necessary; e.g. if an output file with the same name is already existing.

**2.1 MENU 66**

PROCEDURE	:	OUTPUT FILE	:	
DEVICE	:	USAGE	:	66
TAPE NUMBER	:	TAPE TYPE	:	LAYER :

Name of the drilling machine to be used :	
Output file for drill tapes	: \$OUTDIR/_____
User text for drill tape header	:

Command (N,E,L,A,S, ? for help) : N	Tape number : 0
-------------------------------------	-----------------

**2.2FUNCTIONS**

Command:

- N: When you enter -N- here, you can create a new drill tape in a new output file. The tape number and the numerical part of the file extension are automatically incremented by one.
- E: After entering -E- and the number of the drill tape to be edited in "Tape number", you can change the name of the drilling machine, the extension of the output file and the header text for the specified drill tape.
- L: After entering -L- and the desired number in "Tape number", all existing entries will be deleted.

- A:** When you enter -A-, the numbers of the already defined drill tapes (tape numbers) will be listed.
- S:** By entering -S-, you leave the menu. The data of the subordinate menus will be stored. You get back to menu 2 where you can activate another function.
- ?:** Explains the commands.

**Tape number:**

If you want to edit or delete an existing drill tape, you have to enter the number of the tape here, for which the respective command is to be executed.

**3. SELECTION OF THE ELEMENTS TO BE DRILLED**

After you have stored your entries, <CR>, menu 67 for the element selection of the current drill tape appears.

**3.1 MENU 67**

PROCEDURE :	OUTPUT FILE :	
DEVICE :	USAGE :	
TAPE NUMBER :	TAPE TYPE :	LAYER : <input type="text" value="67"/>

Drill tapes for buried VIAS : _	Test points layer 0 : _
drilled through VIAS : _	layer 1 : _
Layer pairs :	

**3.2 MENU DESCRIPTION**

The selection of the elements to be considered for this drill file is made by marking them with "X".

For each output file, only one function may be selected.

**buried VIAS:**

You can select this function if the board has more than one layer pair with buried vias.

After the selection and <CR>, the numbers of the layer pairs are listed in "Layer pairs", according to the constellation in the layer plan. You can now select the layer pair(s) whose buried vias shall be included in the drill tape.

When you have finished your selection, <CR>, an additional menu part appears, in which you specify if only the through-connected vias, only the not through-connected vias or both kinds of vias are to be considered:

through-connected : X	not through-connected : X
-----------------------	---------------------------

### **drilled through VIAS:**

When you select this function, only the drilled through vias are considered in this output file.

When you have finished your selection, <CR>, an additional menu part appears, in which you specify if only the through-connected vias, only the not through-connected vias or both kinds of vias are to be considered:

<b>through-connected : X</b> <b>not through-connected : X</b>
---

### **Test points:**

If drillings for test points are to be created, you have to mark the desired layer (solder or component side). Only one layer may be selected.

#### **Note:**

To create test point drills with OUTPUT, test points must have been defined on the selected layer in the input file, otherwise the message "!!! no test points on this layer!!!" appears.

Up to now, no test points can be created in CIS!



## 4. SPECIFICATIONS TO OFFSET, FIX-FILE, ETC.

When menu 67 is finished and you have saved your entries, menu 69 is called.

### 4.1 MENU 69

PROCEDURE :	OUTPUT FILE :	
DEVICE :	USAGE :	
TAPE NUMBER :	TAPE TYPE :	LAYER :

69

Standard values for drill tools Y/N : Y

Offset for machine zero point in X : 0.000\_\_(in.)

Offset for machine zero point in Y : 0.000\_\_(in.)

If tooling holes, name of fixed file : \_\_\_:\_\_\_ .fix

Drill only fixed file Y/N : Y

Mirror Y/N : Y

### 4.2 HANDLING INSTRUCTIONS

By entering the respective coordinates, you can determine the offset to the machine zero point.

Coordinate origin for the drilling is the production point (defined in CIS/PCB/Sizes).

With tooling holes, drills are meant that may be used for several PC-boards. To avoid a new definition of these fixed 'standard drills' for each board-file, the respective information can be stored in a special file (ASCII-format) with the extension .FIX. It must be stored on the directory lib/sys.

Note:

Documentation about the creation of .FIX-files will be delivered as soon as possible.

If tooling holes are to be generated, you have to enter the name of the .FIX-file in the above menu.

If only tooling holes shall be generated, the default entry -N- in "Drill only fixed file" has to be replaced by -Y-.

The drill coordinates will be mirrored when entering -Y- in "Mirror Y/N". Mirror axis is the vertical to the x-axis that goes through adjustment mark 2.

OUTPUT automatically enters the drill tools to be used (defined by their number), their tool life and the "x" selections as the default when you answer the inquiry "Standard values for drill tools Y/N" with -Y-.

To each drill type, a tool number is assigned that is identical with the number of the drill type. The default value "1290" is assigned as tool life (see menu on next page).

In this case, you get directly back to menu 66 after leaving menu 69. If you do not change the default entry -N-, menu 70 is called for the determination of the drill tool specifications which diverge from the 'standard'.

**5. CHANGING THE STANDARD VALUES FOR DRILL TOOLS**

**5.1 FOR VIAS**

If -N- is entered in "Standard values for drill tools" in menu 69 and a via type (not test points!) was selected in menu 67, the following menu appears after you have stored your entries with <CR>. Here, you can change the standard values of the tools for drilling vias:

**5.1.1 MENU 70**

PROCEDURE :	OUTPUT FILE :	
DEVICE :	USAGE :	
TAPE NUMBER :	TAPE TYPE :	LAYER : <span style="border: 1px solid black; padding: 2px;">70</span>

DRILL TYPE	DIAMETER (in.)	NUMBER	EXECUTE	DRILL TOOL NUMBER	TOOL LIFE (in.)
0	0.024	24	X	1 _____	1290
1	0.031	11	-	2 _____	1290
2					
3					
4					
5					
6					
7					
				Paging (+/-) >--<	

**5.1.2 MENU DESCRIPTION**

All 32 drill types can be called by paging. Therefore, you have to enter '+' (page down) or '-' (page up) in "Paging". If the cursor is already located on the input field and "+" or "-" is displayed there, paging is also possible by just pressing <CR>.

Attention:

If the cursor is not on the input field for the paging, the entries are stored with <CR>.

**DRILL TYPE:**

Here, the numbers of the drill types defined in CIS appear.

**DIAMETER:**

The diameters defined in the CIS technology are entered here for the element selected in menu 67.

**NUMBER:**

The number of drills needed for the element which was selected in menu 67 is output here.

**EXECUTE:**

By entering "X" in this column, you specify that the drills of this drill type are to be executed. If an existing cross is deleted, the respective drill type will not be drilled.

**DRILL TOOL NUMBER:**

Here, you can replace the automatically entered drill tool number by a number of your choice. This drill type will then be executed with the specified drill tool.

**TOOL LIFE:**

Here, you can replace the automatically entered tool life by a tool life of your choice. The figures specify the number of allowed drills; afterwards the tool is used up and must be replaced.

## 5.2 FOR TEST POINTS

If -N- is entered in "Standard values for drill tools" in menu 69 and test points (not vias!) were selected in menu 67, the following menu appears after you have stored your entries with <CR>. Here, you can change the standard values of the tools for drilling the test points:

### 5.2.1 MENU 71

PROCEDURE :	OUTPUT FILE :	
DEVICE :	USAGE :	
TAPE NUMBER :	TAPE TYPE :	LAYER : <input type="text" value="71"/>

Drill test points with tool number: __ , tool life: __
--

Tool number and tool life are automatically entered by OUTPUT. You can change the entries, if necessary.

After you have stored your entries, <CR>, you get back to menu 66 where you can generate a further output file, for example.

If you leave menu 66 with the command -S-, you get back to menu 2 if another function was selected there. Otherwise you get directly back to menu 0 for the selection of the procedure. If you leave menu 70 or 71 with <ESC><ESC>, the entries will not be considered and you get back to menu 69.

## PROCEDURE DEFINITION NC-ROUTING

### 1. INTRODUCTION

When you have activated the function "NC-routing" in menu 2, you get to several menus for the detailed determination of the NC-routing data.

In a procedure, the creation of several NC-routing tapes can be defined. They are stored in different output files.

The output files get the name of the input file and the default extension .fnn; n is a real number from 0 to 9. These files are created during the execution of the procedure (selection number "1" in the postprocessing main menu) and will be stored on the directory pst.

Note:

If the creation of NC-routing tapes was also defined in the procedure, OUTPUT creates a NC-routing report in addition to the respective output files. It is stored in a file with the name of the input file and the extension .mrp on the directory pst.

The heading of each menu belonging to the "NC-routing" function contains information to the following parameters:

- PROCEDURE:** The name of the current procedure is displayed here.
- DEVICE:** The name of the selected NC-routing machine appears here.
- TAPE NO.:** Here, the number of the currently defined tape is shown.
- OUTPUT FILE:** The name of the current output file for the defined tape appears here.
- USAGE:** Here, the usage of the output data (drilling or NC-routing) is displayed.
- TAPE TYPE:** This field is not occupied.
- LAYER:** This field is not occupied.

The specifications to the single parameters will first appear after the menu in which they were defined.

## 2. TAPE DEFINITION

After the activation of the function "NC-routing" in menu 2 for the output selection, menu 72 is called.

Here, you enter the name of the NC-routing machine to be used, the name of the output file and - if desired - a text for the tape header. The specified NC-routing machine must be defined in COLUMBINE, section "Device". In the upper part of the menu, "NC-ROUTING" is displayed.

Notes:

- When you enter "\*" in "Name of the NC-routing machine to be used" and press <CR>, the names of the NC-routing machines defined in COLUMBINE will be listed. When pressing <CR> again, you return to menu 72; the first device name automatically appears in the menu.
- The extension of the output file can be changed, if necessary; e.g. if an output file with the same name is already existing.

### 2.1 MENU 72

PROCEDURE	:	OUTPUT FILE	:	
DEVICE	:	USAGE	:	72
TAPE NUMBER	:	TAPE TYPE	:	LAYER :

Name of the NC-routing machine to be used :	
Output file for NC-routing tapes	: \$OUTDIR/
User text for tape header :	

Command (N,E,L,A,S, ? for help) :	N	Tape number :	0
-----------------------------------	---	---------------	---

### 2.2 FUNCTIONS

Command:

- N: When you enter -N- here, you can create a new tape in a new output file. The tape number and the numerical part of the file extension are automatically incremented by one.
- E: After entering -E- and the number of the tape to be edited in "Tape number", you can change the name of the NC-routing machine, the extension of the output file and the header text for the specified tape.
- L: After entering -L- and the desired number in "Tape number", all existing entries will be deleted.

- A:** When you enter -A-, the numbers of the already defined tapes (tape numbers) will be listed.
- S:** By entering -S-, you leave the menu. The data of the subordinate menus will be stored. You get back to menu 2 where you can activate another function.
- ?:** Explains the commands.

**Tape number:**

If you want to edit or delete an existing tape, you have to enter the number of the tape here, for which the respective command is to be executed.



**3. SPECIFICATIONS TO OFFSET AND FIX-FILE**

When menu 72 is finished and you have saved your entries, menu 73 is called.

**3.1 MENU 73**

PROCEDURE :	OUTPUT FILE :	
DEVICE :	USAGE :	
TAPE NUMBER :	TAPE TYPE :	LAYER :

73

Offset for machine zero point in X : 0.000\_\_ (In.)

Offset for machine zero point in Y : 0.000\_\_ (In.)

If auxiliary NC-routing,  
name of fixed file : \_\_\_\_:\_\_\_\_.FIX

Mirror Y/N : Y

**3.2 HANDLING INSTRUCTIONS**

By entering the respective coordinates, you can determine the offset to the machine zero point.

Coordinate origin for the NC-routing is the production point (defined in CIS/PCB/Sizes).

With auxiliary NC-routing, NC-routings are meant that may be used for several PC-boards. To avoid a new definition of these fixed 'standard NC-routings' for each board-file, the respective information can be stored in a special file (ASCII-format) with the extension .FIX. It must be stored on the directory lib/sys.

Note:

Documentation about the creation of .FIX-files will be delivered as soon as possible.

If auxiliary NC-routings are to be generated, you have to enter a name for the .FIX-file.

The NC-routing coordinates will be mirrored when entering -Y-in "Mirror Y/N". Mirror axis is the vertical to the x-axis that goes through adjustment mark 2.

**4. SPECIFICATIONS FOR NC-ROUTING TOOLS**

After you have stored your entries with <CR>, menu 74 appears for the specifications to the NC-routing tools.

**4.1 MENU 74**

PROCEDURE :	OUTPUT FILE :		<b>74</b>
DEVICE :	USAGE :		
TAPE NUMBER :	TAPE TYPE :	LAYER :	

NC-ROUT. TYPE	DIAMETER (in.)	EXECUTE	NC-ROUTING TOOL NO.	TOOL LIFE (in.)
0				
1	.060	X	1_____	1290
2				
3				
4				
5				
6				
7				
			Paging (+/-) >-<	

**4.2 MENU DESCRIPTION**

In CIS up to 32 NC-routing diameters can be defined (according to the diameters of the via types that have been assigned to the NC-routing terminals for the display of the starting point and the specification of the NC-routing width). OUTPUT assigns NC-routing types to the diameters defined in the input file (starting with NC-routing type 1). The sequence has nothing to do with the via types defined in the CIS technology! OUTPUT refers only to the NC-routing diameters. For example, if different via types for the display of the starting points have been used during the definition of two NC-routing contours and these via types have the same diameter, OUTPUT assumes that both NC-routings belong to the same type.

All 32 NC-routing types can be called by paging. Therefore, you have to enter '+' (page down) or '-' (page up) in "Paging". If the cursor is already located on the input field and "+" or "-" is displayed there, paging is also possible by just pressing <CR>.

**Attention:**

If the cursor is not on the input field for the paging, the entries are stored with <CR>.

OUTPUT automatically enters the NC-routing tools to be used (defined by their number), their tool life and the "x" selections as the default.

To each NC-routing type, a tool number is assigned that is identical with the number of the NC-routing type. The default value "1290" is assigned as tool life.

## **NC-ROUTING TYPE:**

Here, the numbers of the 32 definable NC-routing types appear automatically.

## **DIAMETER:**

The diameter of the NC-routing point defined in CIS for this NC-routing contour is entered here (corresponds to the diameter of the via that was assigned to the respective NC-routing terminal in CIS).

## **EXECUTE:**

By entering "X" in this column, you specify that the NC-routing of this type is to be executed. If an existing cross is deleted, it will not be executed.

## **NC-ROUTING TOOL NUMBER:**

Here, you can replace the automatic entry by a number of your choice (according to the NC-routing point diameter). This NC-routing type will then be executed with the specified tool.

## **TOOL LIFE:**

Here, you can change the automatically entered tool life, if necessary. The number specifies the maximum tool life in in.-units.

After you have stored your entries, <CR>, you get back to the selection menu 2 (if another function was selected there) or directly to menu 0 for the selection of the procedure.

If you leave the menu with <ESC><ESC>, the entries will not be considered and you get back to menu 73.

**CALPRI**

**1. INTRODUCTION**

When selecting function "5" (= 'Run list-generator') in the postprocessing main menu and entering name and extension of the file to be processed, the program CALPRI is started.

With this program, lists and statistics can be created from the data of the specified file.

Note:

If "adb" was entered as extension during the file specification, the .adb-file is transformed into a .gph-file before the program is started. (CALPRI can only process the gph-data base at present.)

After the program start, the following menu appears for selecting the output mode of the lists to be created.

**2. LIST SELECTION**

Lists from the .GPH-file	version:
--------------------------	----------

Name of .GPH-file : _____	
output to terminal (0) or file (1)	1
file on device OUT with extension .lis	
netlist	list type (0,1,2) -
component list	list type (0,1,2,3) -
only solderside (0), only comp.side (1), both (2)	2
missing connections (only for rev. 5-files)	-
non connected terminals	technology -
cross-reference-list	statistics -
collision of component	-
change printer controlling	-
<ESC><ESC> terminates program	

## HANDLING INSTRUCTIONS

- In "Name of the .GPH-file", the name of the specified file appears.

If you enter another file name (without extension!) in this field, you can generate lists from the newly specified file (provided that it exists in the gph-format on the directory gph).

- The list selection is made by putting an "X" in the respective field.

You may select as many lists as desired.

- The list type must also be specified for the netlist and the component list by entering the respective selection number. By means of this specification, the form and the complexity of the information output from these lists is determined (see "Output form"). In one program run, only one type of each list can be created!

For the component list, you have to specify additionally which side(s) shall be considered (default entry = "2", i.e. solder and component side).

- If the list shall be output on the terminal, change the default number to "0".
- Exit the program with <ESC><ESC>.
- The files generated by CALPRI are written onto the directory `pst` and have the name of the `adb` or `gph`-file with the extension `.lis`.

### 3. LIST OUTPUT IN A FILE

If the lists are output in a file, they can be printed or edited (e.g. with the vi-editor).

#### Change printer controlling

As the default, CALPRI creates a page format of 65 lines per page and 80 characters per line for the file output.

But list type 2 and 3 have more lines than the default line length and your printer may require another setting of the line number per page.

With the help of the function "Change printer controlling", you can enter commands for switching the printer to another page and line length.

When selecting this function, the following additional menu appears (see next page):

lines per page for printer	65
sequence for printer at 132 characters	ESCQ__
sequence for printer at 80 characters	ESCN__

- In the input field of the first line, you can specify how many lines the pages of the lists shall contain when printing the file.

The number of lines depends on the printer you use.

The default is the number of lines optimized for printer BINDER 8510 B. Due to this information, the printer will continue with the headline on the next page after 65 lines.

- The maximum line length in the 'normal' operation mode of the printer varies from type to type (e.g. for BINDER 8510 B line printer it is 80 characters).

To get a correct output of the list types 2 and 3 on these devices, it is necessary to compromise the characters.

This is achieved by switching the printer with a command at the beginning of the file.

The input field of the second menu line contains as the default the command sequence for the automatic set back of BINDER 8510 B printer from 132 to 80 characters per line - after printing list types 2 and 3.

If you use another printer, the default command sequences must be replaced by those convenient for your printer. For the respective command, please refer to your printer manual.

The command sequences for switching the printer become active only when activating list type 2 and 3 and output in a .lis-file.

#### 4. LIST OUTPUT ON THE TERMINAL

The output on the printer (.lis-file) should be preferred to the direct output on the terminal!

List type 2 and 3 cannot be represented correctly in the shelltool standard window because their lines are too long. This applies also to the editing of the CALPRI output files (.lis-files) with the vi-editor.

**5. OUTPUT FORM OF THE LISTS**

**5.1 NETLIST**

List type 0

---

netlist alphab. sorted      \*\* test \*\*    01-APR-89      page: 1

---

NETNAME	COMPNAME	TERM	TT	COMPNAME	TERM	TT
GND	\$0	14	2	ST1	11	2
SIG 1	C1	1	1	IC2	5	1
	IC3	5	1	R7	2	2
SIG 2	C4	2	3	IC5	7	3
	IC4	7	2	IC7	12	2
	IC9	3	3			
	etc.					

Notes:

- a. The headline displays the name of the .adb or .gph-file, from which the list has been generated, the date of generation and the page number.
- b. Every line of this list contains the following information from the netlist:
  - names of nets (NETNAME),
  - connection points in net, specified with the component name (COMPNAME) and the terminal name (TERM),
  - track type for connecting this terminal to the net

List type 1

---

netlist alphab. sorted      \*\* test \*\*    01-APR-89      page: 1

---

NETNAME	COMPNAME	TERM	TT	LAYERSIGNAL	SUBSIGNAL	K
SIG 1	C1	1	1	2		P
	IC2	5	1	2		P
SIG 2	C4	2	3	*	0-1	
	IC5	7	3	*	2,3	
	IC9	3	3	*	2-3	
SIG 3	C1	2	2	1		D
	IC1	12	2	1		C
	IC4	6	2	1		D
	etc.					

Note:

Apart from the information in list type 0, list type 1 contains the following information from the netlist:

- number(s) of the connection point permitted (LAYERSIGNAL) --> "\*" = all layers permitted.
- if the connection point belongs to one or more subsignals, it contains the respective subsignal number(s) (SUBSIGNAL),
- if the connection point belongs to a plane or a chain (daisy chain), it contains an according mark in K

(P for plane; D for daisy chain, C for connection point).

List type 2

netlist alphab. sorted	** test **	01-APR-89	page: 1							
NETNAME	COMPNAME	TERM	TT	LAYERSIGNAL	SUBSIGNAL	K	COMMENT	ROUTINGLAYER	CAS	
SIG 1	C1	1	1	2		P	PLANE VCC	*	NNU	
	IC2	5	1	2		P	PLANE VCC	*	NNU	
SIG 2	C4	2	3	*	0-1		RING	3,4	NYA	
	IC5	7	3	*	2,3		RING	3	NYA	
	IC9	3	3	*	1-3		ASTERISK	3	NYA	
SIG 3	C1	2	2	1		D	DAISY	1	YH	
	IC1	12	2	1		C	CONNECTION POINT	1	YH	
	IC4	6	2	1		D	DAISY	1	YH	
etc.										

Notes:

Apart from the information of list type 1, list type 2 contains the following information on signal parameters from the netlist:

- comment assigned to the signal (COMMENT),
- routing layers, i.e., layer(s) on which the signal may be routed (see ROUTINGLAYER),
- additional signal parameters (see CAS), with CAS having the following meaning:
  - > C = critical signal
  - > A = auto-routable
  - > S = signal type (analog, digital, hybrid or not defined)

The following abbreviations may appear under these parameters:

- C -> Y/N for Yes/No
- A -> Y/N for Yes/No
- S -> A (= analog); D (=digital); H (=hybrid); U (=not defined, i.e. without specification)



## 5.2 COMPONENT LIST

List type 0

---

Component list alphab. sorted   \*\* test \*\*   01-APR-89   page: 2

FLAGS: O=N SOCKET, F=FIXED, A=SSEMBLY, V=IABLOCK, D=UMMY COMP.,  
P=LOTNAME, S=OLDERSIDE, L=IBRARY IDENT., I=N-CIRCUIT-TEST

---

COMPNAME	SHAPETYPE	ROT	FLAGS	COMPNAME	SHAPETYPE	ROT	FLAGS
C1	DCRU2A	90.0	FA	IC1	DIP14F	270.0	OAV
R1	WID2L	180.0	A	SM1	DCC16	0.00	S

etc.

Notes:

- a. The headline displays the name of the .adb or .gph-file, from which the list has been generated, the date of generation and the page number.
- b. This list contains 2 columns with the following component information:
  - names of components (COMPNAME),
  - shape type (SHAPETYPE),
  - component rotation for placement (ROT),
  - additional component parameters (see FLAGS); the abbreviations have the following meaning:
    - > O = on socket
    - > F = fixed
    - > A = assembly
    - > V = via blocking
    - > D = dummy component
    - > P = plot name
    - > S = solder side
    - > L = library identical
    - > I = in circuit test

Attention: The flags D, I and L are not correctly evaluated at present!

List type 1

Component list alphab. sorted      \*\* test \*\*      01-APR-89      page: 2

FLAGS: O=N SOCKET, F=IXED, A=SSEMBLY, V=IABLOCK, D=UMMY COMP.,  
P=LOTNAME, S=OLDERSIDE, L=IBRARY IDENT., I=N-CIRCUIT-TEST

COMPNAME	SHAPETYPE	ROT	FLAGS	COMPONENTTYPE
C1	DCRU2A	90.0	F	KO
IC1	DIP14F	270.0	OAV	74LS06
R1	WID2L	180.0	A	WID
SM1	DCC16	0.00	S	SMD/7400

etc.

Note:

Apart from the information in list type 0, list type 1 contains the function type specification (COMPONENTTYPE).

List type 2

Component list alphab. sorted      \*\* test \*\*      01-APR-89  
page: 2

LAGS: O=N SOCKET, F=IXED, A=SSEMBLY, V=IABLOCK, D=UMMY COMP.,  
P=LOTNAME S=OLDERSIDE, L=IBRARY IDENT., I=N-CIRCUIT-TEST

COMPNAME	SHAPETYPE	ROT	FLAGS	COMPONENTTYPE	STOCK IDENTIFIER
C1	DCRU2A	90.0	F	KO	K/00037
IC1	DIP14F	270.0	OAV	74LS06	LNR74LS00
R1	WID2L	180.0	A	WID	W/0003
SM1	DCC16	0.00	S	SMD/7400	XX-8769

etc.

Note:

Apart from the information in list type 1, list type 2 contains the stock ID (STOCK IDENTIFIER).

# POSTPROCESSING/CALPRI/Introduction

## List type 3

---

Component list alphab. sorted      \*\* test \*\*      01-APR-89      page: 2

---

FLAGS: O=N SOCKET, F=FIXED, A=SSEMBLY, V=IABLOCK, D=UMMY COMP., P=LOTNAME  
S=OLDERSIDE, L=IBRARY IDENT., I=N-CIRCUIT-TEST

COMPNAME	SHAPETYPE	ROT	FLAGS	COMPONENTTYPE	STOCK IDENTIFIER	COMMENT
C1	DCRU2A	90.0	F	KO	K/00037	
IC1	DIP14F	270.0	OV	74LS06	LNR74LS00	NEAR OSCILLATOR
R1	WID2L	180.0		WID	W/0003	
SM1	DCC16	0.00	S	SMD/7400	XX-8769	

etc.

### Note:

Apart from the information in list type 2, list type 3 contains comments assigned to the components (COMMENT).

## 5.3 NON CONNECTED TERMINALS

This list contains the terminals of the components from the .adb or .gph-file which are not connected in the netlist.

The component names are output in an alpha-numeric order.

The column COMPONENTTYPE contains the function type specification.

---

list of non connected terminals \*\* test \*\* 01-APR-89 page: 4

---

COMPNAME	TERMINALNAMES	SHAPETYPE	COMPONENTTYPE
	3	DIP14	74LS00
	5 7 9 11		
ST1	7		
	4 8 13		

etc.

**5.4 CROSS-REFERENCE-LIST (= COMPONENT CONNECTIONS)**

This list contains all components (incl. shape, function type specification and stock ID), their terminals and the signals to which the terminals are connected.

The list is output in an alpha-numeric order.

cross-reference-list    \*\* test1 \*\*    01-APR-89    page: 5

output format:

componentname		componenttype				
shapetype		stock id				
terminalname(x)		netname(x)				
IC1		74LS00				
DIP6		ABC123				
1	SIG1	2	SIG12	3	NC.	4
5	SIG17	6	NC.			GND

etc.

(NC. indicates terminals that are not connected.)

**5.5 COMPONENT COLLISIONS**

This list contains the names of the components that collide with each other.

collision of component    \*\* test1 \*\*    01-APR-89    page: 6

R2	<-->	C8	R2	<-->	C9
SM1	<-->	R11			

etc.

**5.6 TECHNOLOGY**

When selecting this function, you get a list with the CIS/board technology contained in the input file. As the output form mainly corresponds to the CIS technology charts, it is not presented here.

Note:

When listing the layer plan, the documentation layers cannot be output.



NUMBER OF PADS

type	number	diameter
1	5	0..038
17	27	1.055
18	3	1.055
-----		
total	35	

NUMBER OF SMD-PADS : 20

-----

NUMBER OF VIAS

type	number	diameter
1	12	1.055
-----		
total	12	

NUMBER OF COMPONENTS

layer	total number	insertable
solder side:	1	1
comp. side :	6	6
-----		
total	7	

NUMBER OF SIGNALS

number	comment
6	signals with 25 terminals
30	terminals not in the netlist
-----	

NUMBER OF SPOTS

	layer	number
test points	:	0
copper points	:	0
glue points	: 0	1
NC-routing points	: 0	1
-----		

M I R R O R

What happens if you mirror?

BOARD:

All elements of \$\$BOARD are mirrored.  
Text fields are mirrored but their contents remains readable.  
'\$\$BOARD' is the area defined in "PC-board" in the menu 'Sizes'.

BOARD, DIN:

Like BOARD, but all text fields are rotated so that they can be read from below or from the right side.

ALL:

All elements of \$\$PCB and \$\$BOARD are mirrored.  
The contents of text fields is also mirrored.  
'\$\$PCB' is the area defined in "Sheet size" in the menu 'Sizes'.

ALL, DIN:

Like ALL, but the contents of the text fields is rotated so that it can be read from below or from the left side.

DIN:

The contents of the text fields is rotated so that it can be read from below or from the right side.

ALL, BOARD:

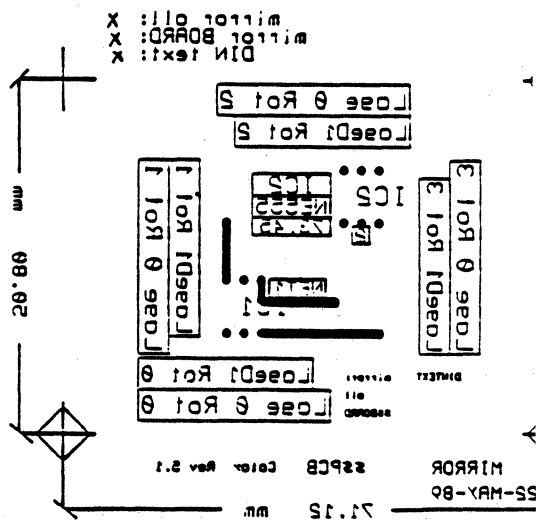
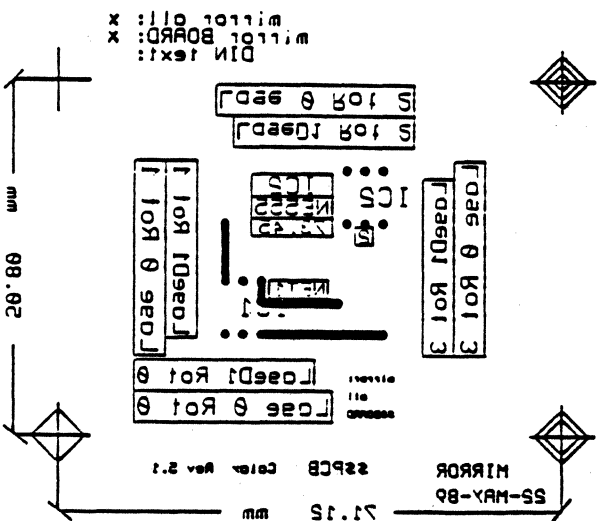
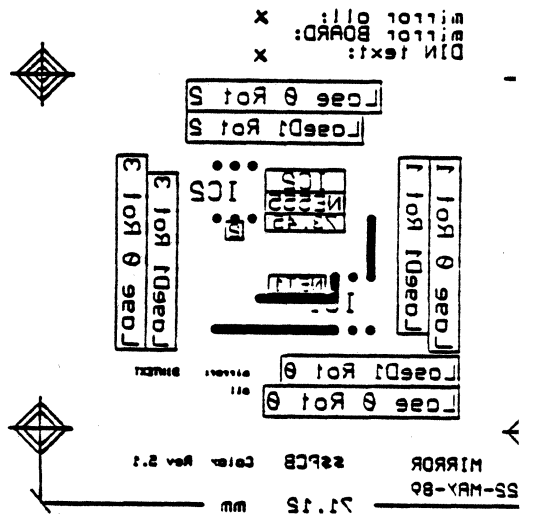
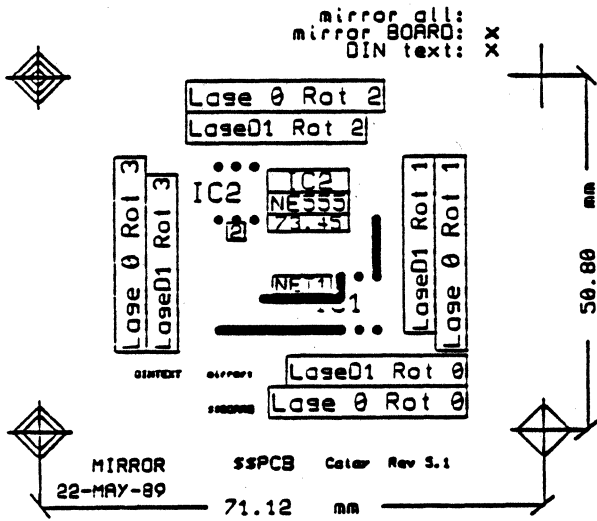
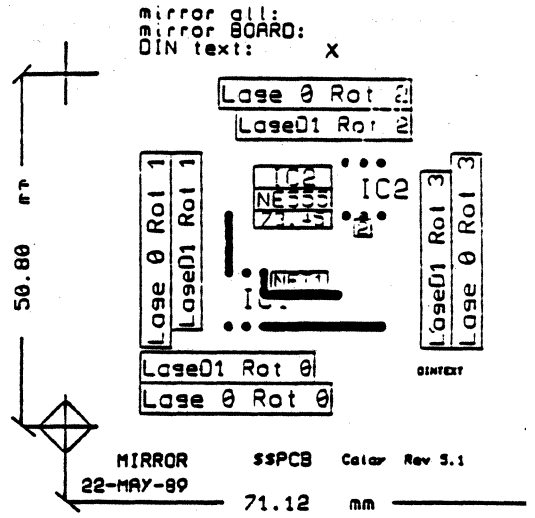
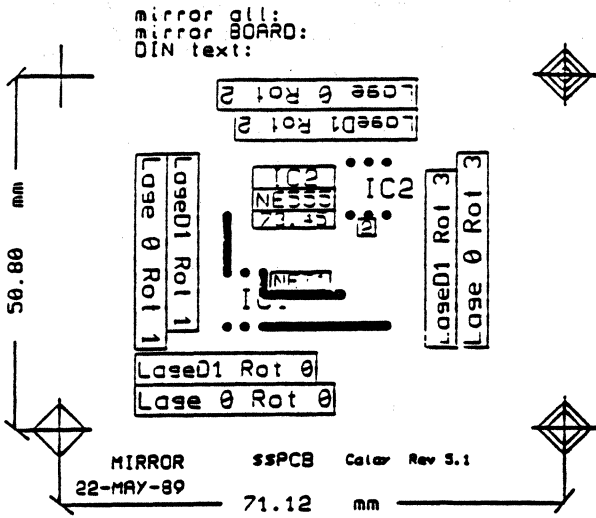
Only text fields or their contents is mirrored, the rest remains unchanged.  
Text fields to \$\$PCB are mirrored with contents.  
Only the contents of text fields to \$\$BOARD is mirrored.

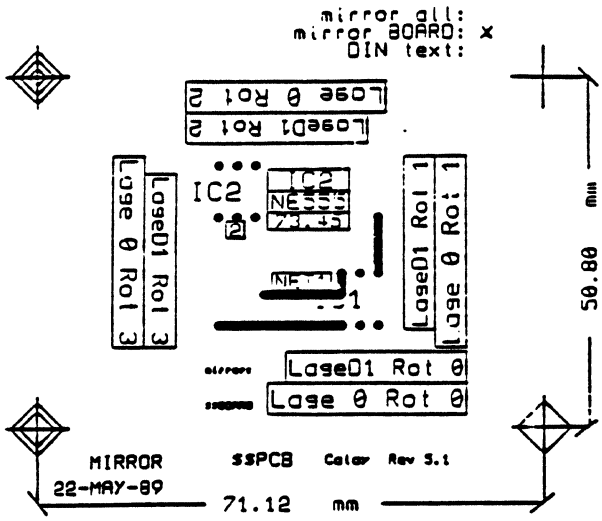
ALL, BOARD, DIN:

Like ALL, BOARD, but the contents of the text fields is rotated so that it can be read from below or from the left side.

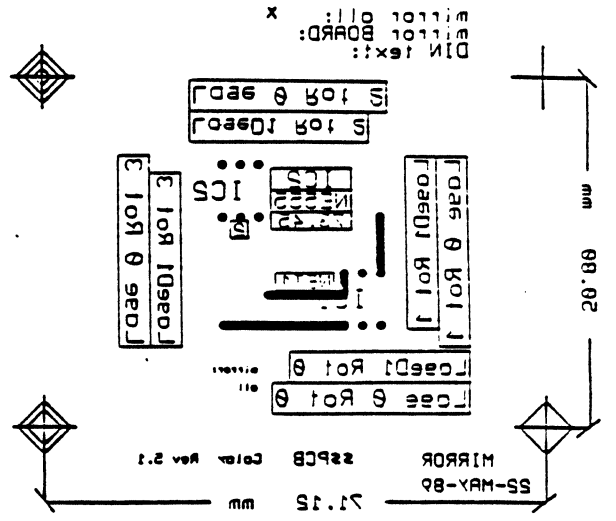


# Example MIRROR (all variants)



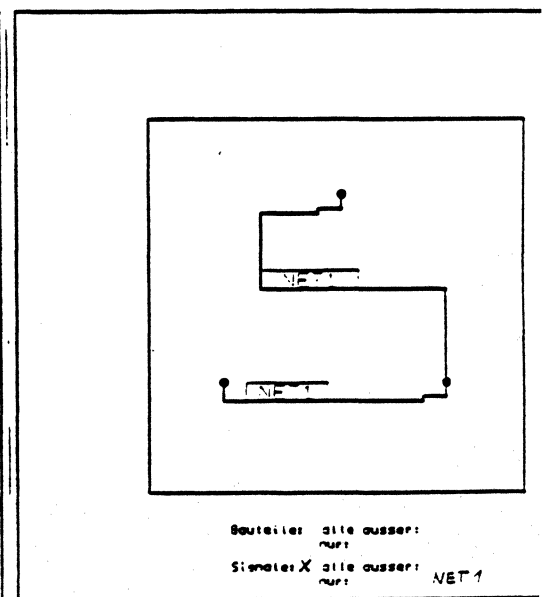
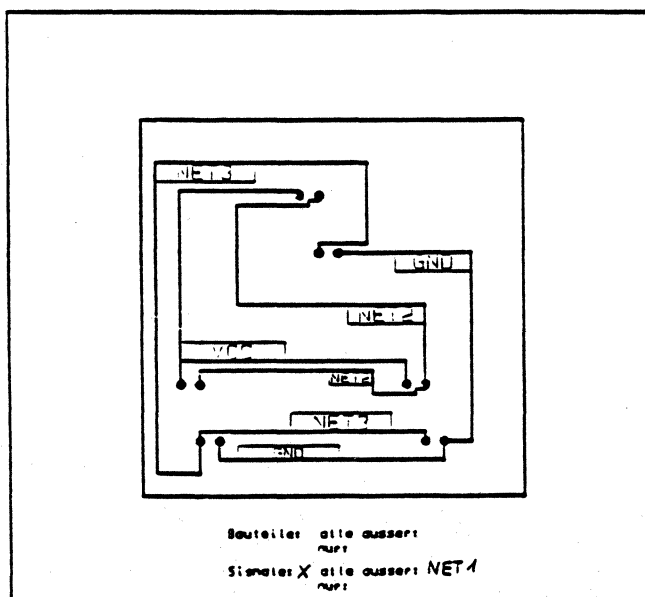
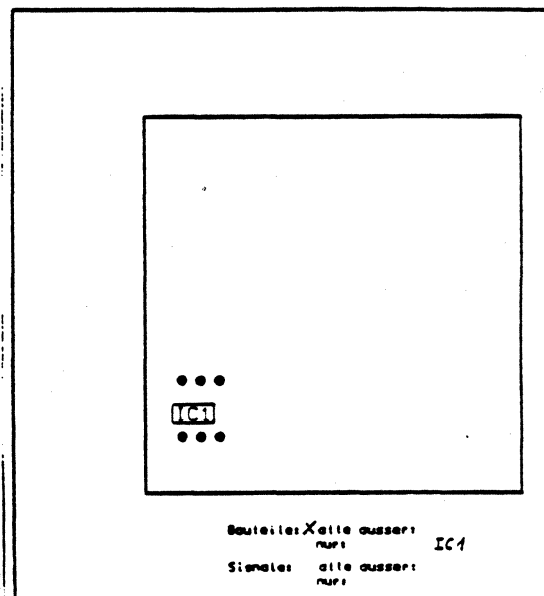
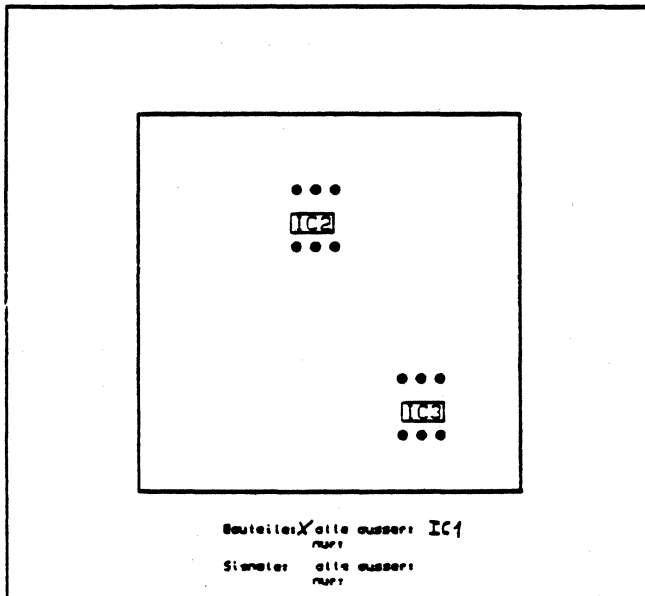
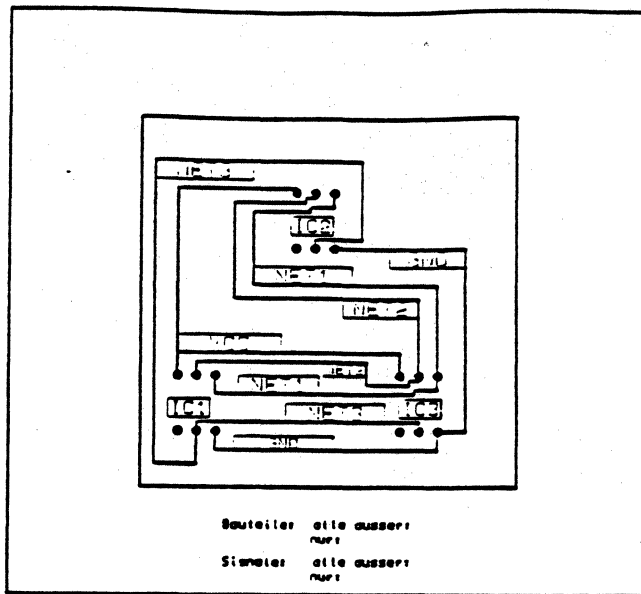


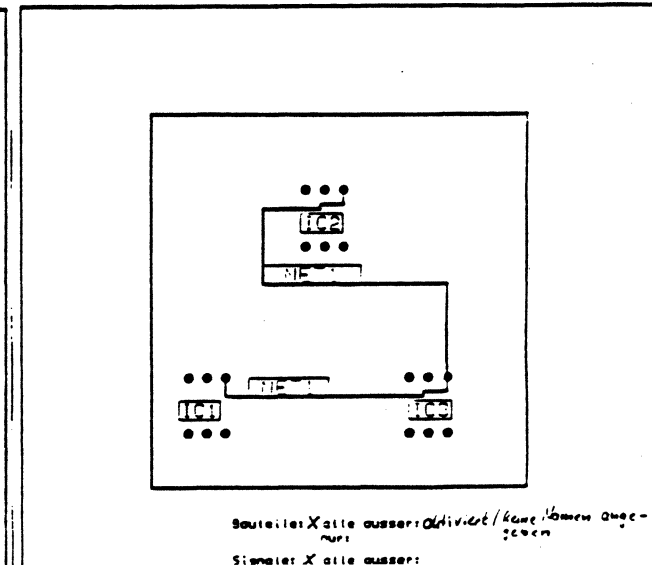
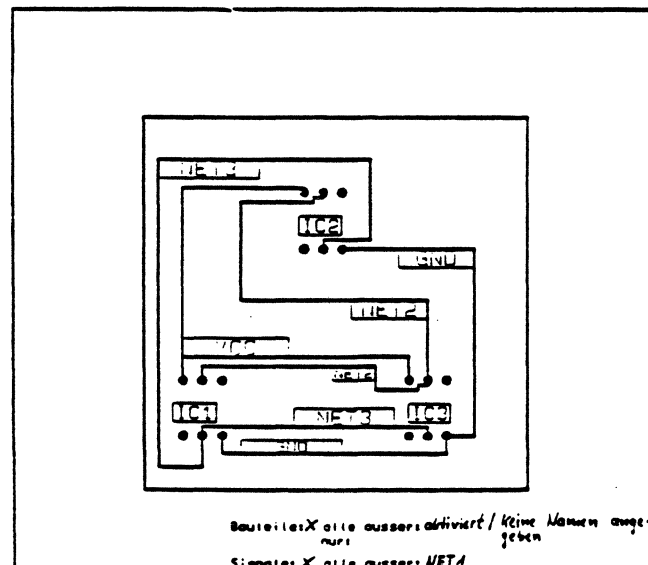
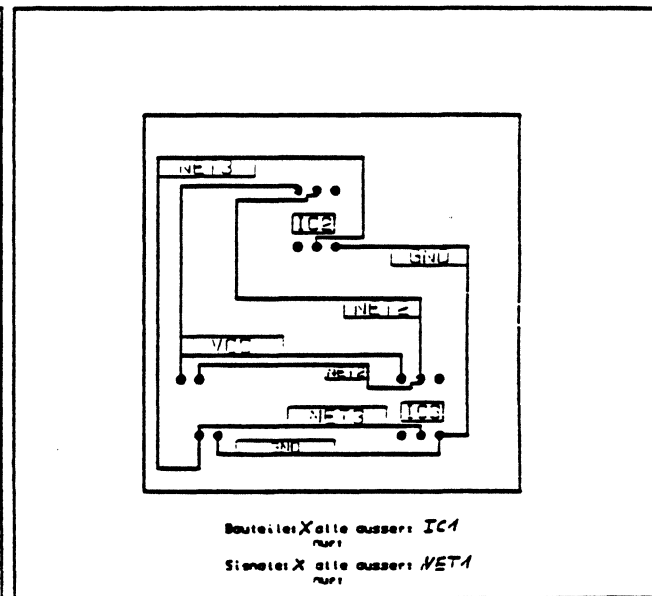
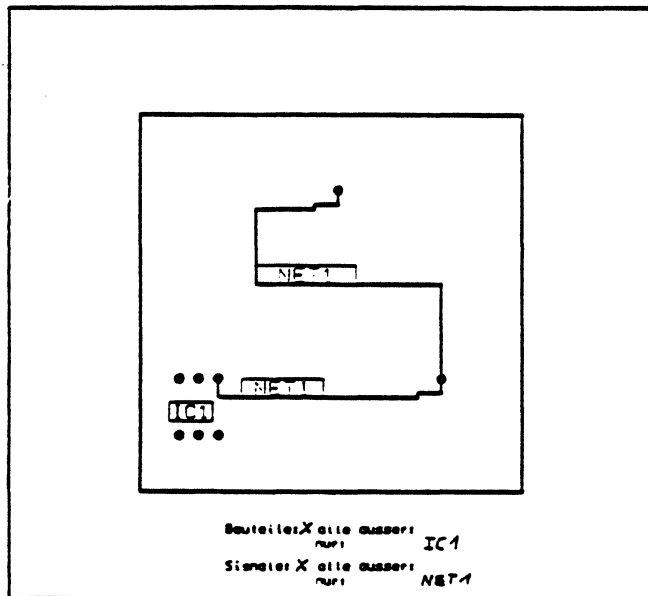
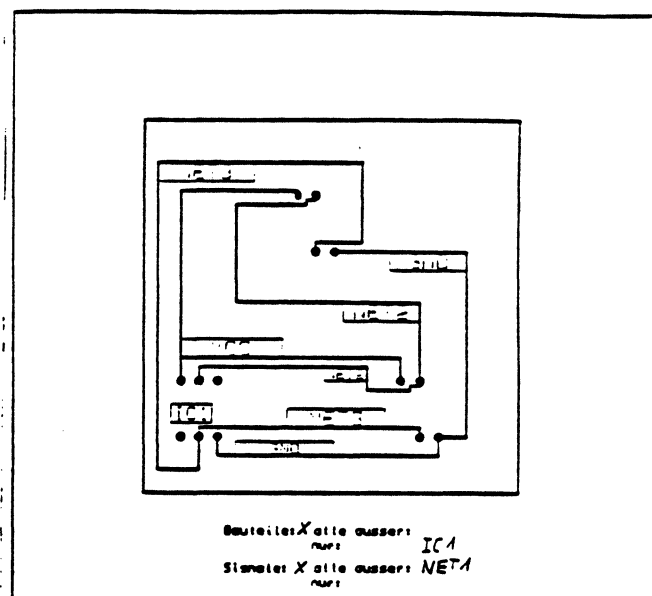
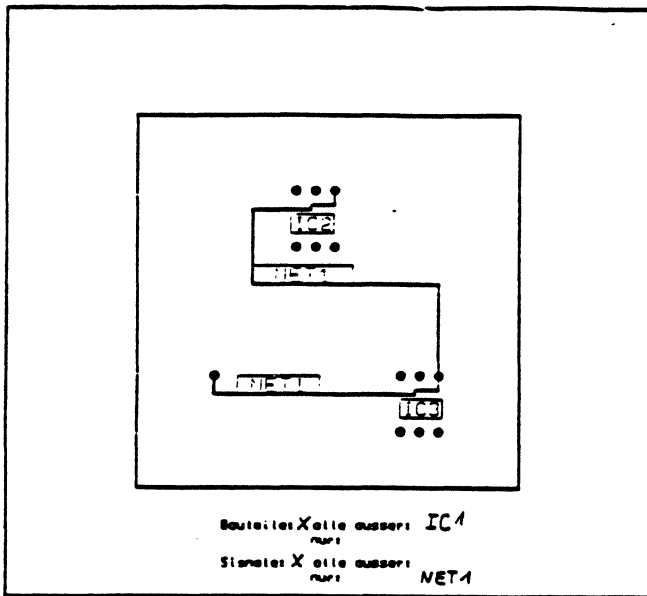
mirror all:  
 mirror BOARD:  
 DIN text:  
 X

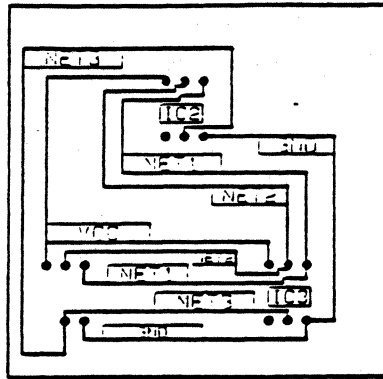


mirror all:  
 mirror BOARD:  
 DIN text:  
 X

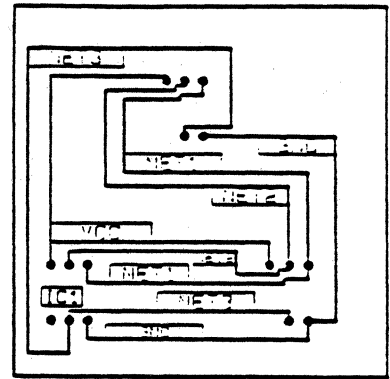
# Example SELEKT Components / signals (all variants)







Bauteile: X alle ausser: IC1  
mark  
 Signale: X alle ausser: aktiviert / keine Namen  
mark ausgegeben



Bauteile: X alle ausser: IC1  
mark  
 Signale: X alle ausser: aktiviert / keine  
mark ausgegeben

**[CIS-0101] falscher Pfadname**  
**[CIS-0101] path wrong**

The environment variables CALHME or CALPRJ have not been defined or have been wrongly defined (they are usually defined during the LOGIN) or the directory or parts of it have been deleted.

**[CIS-0102] mehr als einen Job selektiert**  
**[CIS-0102] more than one job selected**

Several files have been selected for the read in. Only one file is allowed.

**[CIS-0103] falsche Auswahl**  
**[CIS-0103] wrong selection**

An "empty" entry has been selected.

**[CIS-0104] WARNUNG: Syntax des Jobnamens nicht erlaubt**  
**[CIS-0104] WARNING: jobname syntax wrong**

The syntax of the jobname is not allowed. Might have occurred by copying or reading in via the OS or the user. When storing the file, the syntax has to be corrected.

**[CIS-0105] keine Jobs mehr in diesem Directory**  
**[CIS-0105] no more jobs in this directory**

The directory is empty and does not contain any or no more .ADB-files.

**[PCB-2100] kein Bauteil selektiert**  
**[PCB-2100] no component selected**

An attempt was made to execute a command, but no component(s) was (were) selected before. Select a component first by either hitting a mouse button at a location in the layout where the component is placed or by entering its name in the form and execute the command then.

**[PCB-2101] kein Bauteil getroffen**  
**[PCB-2101] no component met**

When the mouse was used to select a component:  
A mouse button was pressed at location where no component exists on the board or an attempt was made to pick up a component though the display of components is switched off in the graphics menu.

When a component name was specified in the form:  
A component with the specified name does not exist. Use the 'List' function to get every component listed on the screen.

## **ERROR MESSAGES**

---

**[PCB-2102] Name schon vorhanden oder falsch**  
**[PCB-2102] name already used or wrong**

While creating or renaming a component the name of an already existing component was entered in the form or the specified name does not yield in a valid components name.

**[PCB-2103] Bauteilkollision**  
**[PCB-2103] component collision**

A collision with another component was detected during the execution of the 'Move' or 'Copy' command. Locate the component at a place, where no other component resides or hit the mouse button at same location again to force a collision (hitting <CR> without changing the coordinates in the form will do). This message appears too, when a terminal would be located on a track or via of a different signal, or when a restricted area for components, tracks or vias lies at the target location. When this error is reported though there are no elements at the target location, make sure that your graphic setup displays every element which might cause a collision. Check your restricted areas and the board outline: they might be 'active' on the 'wrong' side (inside/outside).

**[PCB-2104] illegale Angaben**  
**[PCB-2104] illegal input**

The user tried to select a component without specifying either a components name or a location.

**[PCB-2105] not used yet**  
**[PCB-2105] not used yet**

**[PCB-2106] not used yet**  
**[PCB-2106] not used yet**

**[PCB-2107] not used yet**  
**[PCB-2107] not used yet**

**[PCB-2108] Bauteil besitzt keine Grafik**  
**[PCB-2108] component has no grafic**

The 'Exchange shape' command was called with a master component which has got no graphic at all. Only components with graphic are valid masters for this command.

**[PCB-2109] Bauteil kann nicht relationiert werden**  
**[PCB-2109] can't relate component**

The 'transform' command has been executed on this component before. A second execution is not necessary and not possible either. Please mind, that you have to define all the terminals of a component before you can call the 'transform' command.

**[PCB-2110] Bauteil besitzt keine Pins**  
**[PCB-2110] component has no pins**

The 'transform' command was called for a component with has no terminal at all. The transformation of this component is neither necessary nor possible.

**[PCB-2111] ungueltiger Befehl fuer \$BOARD**  
**[PCB-2111] invalid command for \$BOARD**

The called command is not valid for \$BOARD (e. g. 'Delete', 'Copy', ...).

**[PCB-2112] ungueltige Drehung**  
**[PCB-2112] invalid rotation**

An invalid rotation was entered in the form for the 'Move' or 'Copy' command. Check (and correct) the rotation restrictions for this component in the 'Modify Parameter' form or hit the mouse button at same location again to force this illegal rotation (hitting <CR> without changing the rotation in the form will do). In the latter case the component will be marked as 'technologically wrong'.

HEINZ: bitte kontakte mich wegen dieser fehlermeldung kurz vor Auslieferung der endgueltigen Version: eventuel habe ich das 'force illegal rotation' noch nicht implementieren koennen.

**[PCB-2113] Bauteil ist fixiert**  
**[PCB-2113] component is fixed**

An attempt was made to delete or move a fixed component. Change the 'fixed' mark in the 'Modify Parameter' form first.

**[SCS-1201] Terminalname schon vergeben**  
**[SCS-1201] terminalname already used**

It has been tried to specify a terminal name twice.  
Rules: All the terminals belonging to a symbol have to have a definite name.  
All the sheet-tags existing on one sheet have to have a definite name.



## **ERROR MESSAGES**

---

**[SCS-1202] Terminal nicht gefunden**  
**[SCS-1202] terminal not found**

A terminal has been called via name or coordinate and has not been found by the system.

Cause: Wrong name.  
No terminal on this coordinate.

**[SCS-1203] Funktion fehlerhaft**  
**[SCS-1203] internal error**

see [SCS-1102]

**[SCS-1204] Koordinate fehlt noch**  
**[SCS-1204] coordinate missing**

The input of a coordinate either manually or via mouse is absolutely necessary.

**[SCS-1205] Kein Terminal selektiert**  
**[SCS-1205] no terminal selected**

A command for which an activated (current) terminal is a precondition has been activated too early.

Cause: see [SCS-1105]

**[SCS-1210] Speicherplatz zu gering**  
**[SCS-1210] not enough memory**

see [SCS-1109]

**[SCS-1211] Kollision mit anderem Element**  
**[SCS-1211] collision with other element**

A terminal collided with other elements (terminals, signal connections, symbols) after a MOVE, COPY or CREATE action.

solution: make new location free of any elements

In the future these collisions will be specified better and more closely.

**[SCS-1231] Terminalstatus mehrdeutig**  
**[SCS-1231] terminalstatus ambiguous**

Only one terminal status may be defined from the given possibilities (Input, Output, Tristate..). If several possibilities have been selected simultaneously, the message appears:

```
terminal status ambiguous
*** NEW: status not defined ***
```

the terminal status is set to DON'T CARE.

If the error above did not entail the entry of a new status, (terminalstatus was DON'T CARE), then the message appears:

```
terminalstatus ambiguous
*** Status unchanged ***
```

**[SCS-1232] \*\*\* Status nicht veraendert! \*\*\***  
**[SCS-1232] \*\*\* status not changed ! \*\*\***

see [SCS-1231] and [SCS-1233]

**[SCS-1233] kein Terminalstatus angegeben**  
**[SCS-1233] terminalstatus missing**

A terminal status has to be defined from the given possibilities (Input, Output, Tristate ...). If no possibility has been chosen, the following message appears:

```
terminal status missing
*** NEW: status not defined ***
```

the terminalstatus is set to DON'T CARE

If the above error did not entail the entry of a new status, (terminalstatus was DON'T CARE), then the following message appears:

```
terminalstatus missing
*** Status unchanged ***
```

**[SCS-1234] \*\*\* NEU: Status nicht definiert \*\*\***  
**[SCS-1234] \*\*\* new: status not defined \*\*\***

see [SCS-1231] and [SCS-1233]

## ERROR MESSAGES

---

**[SCS-1235] kein Terminal selektiert**  
**[SCS-1235] no terminal selected**

A terminal has been called via name or coordinate and has not been found by the system

Cause: wrong name  
No terminal on this coordinate

**[SCS-1236] Terminalname fehlerhaft**  
**[SCS-1236] bad terminalname**

The terminal name collides with an existing name or does not correspond to the names' guidelines.

Also see CNORM-NAMES (is yet to be delivered for 3.01).

**[SCS-1237] Speicherplatz zu gering**  
**[SCS-1237] not enough memory**

see [SCS-1109]

**[SCS-1101] Symbol nicht gefunden**  
**[SCS-1101] symbol not found**

A symbol has been called via name or coordinate and could not be found by the system.

Cause: wrong name  
No symbol on this coordinate  
No symbol within the search window

**[SCS-1102] Funktion fehlerhaft**  
**[SCS-1102] wrong function**

This error message has been defined for software engineers and should not occur at the customer's site. In most cases it results from communication errors and last changes prior to a release date. The actual error message will be implemented in the following release after the error has been specified. If such an error occurs, the CALAY quality assurance department should be informed.

**[SCS-1104] Nicht verschoben, gleicher Ort**  
**[SCS-1104] not moved, same place**

The user tried to move a symbol onto the same position using the MOVE-command. Since this often entails a re-routing of the signal connections, the system neglects the first trial. Press the same button again and the command is executed.

**[SCS-1105] Kein Symbol selektiert**  
**[SCS-1105] no symbol selected**

A command for the execution of which a current symbol is needed, has been activated too early.

**Cause:!! Applies to all elements (symbol, terminal, signal) !!**

GET is activated and the system allows for the selection of a working function (handling problem is continually revised, successively improved and within a medium-term period of time, this error message will be superfluous.

or

A previous function sets the system into a state where there is no current selected element which is definitely to be preferred (e.g. after deleting or moving onto an invisible sheet)

**[SCS-1106] Name schon vorhanden oder falsch**  
**[SCS-1106] name already used or wrong**

The symbol name collides with a name already existing in the system or does not correspond to the names' guidelines  
Also see CNORM-NAMES (is yet to be delivered for 3.01)

**[SCS-1107] Kollision**  
**[SCS-1107] collision**

One or more symbols have collided with other elements (terminals, signal connections or other symbols) during a move or copy action.

**Solution:** make new location free of any elements  
We will try to specify the collision better and more closely in the future.

**[SCS-1109] Speicherplatz zu gering**  
**[SCS-1109] not enough memory**

Space required by the system is not available.

**Cause:** SWAPspace too small or internal limit has been reached which has not been eliminated due to safety reasons. For the latter, in most cases storing and recalling the file is helpful.

**[SCS-1110] Logik-Darstellung geaendert**  
**[SCS-1110] logic-representation changed**

see SCS-1111

## ERROR MESSAGES

---

**[SCS-1111] Zweite Logik-Darstellung unbekannt**  
**[SCS-1111] second logic-representation unkown**

It has been tried to call the second representation of a symbol into the foreground (exchange positive against negative logic) without a second representation being assigned to this symbol.

By calling this function again, an identical graphic is generated. Error message no. SCS-1112 appears.

After doing so, calling is possible without error message SCS-1111, the changing of the graphic is now shown by displaying SCS-1110.

**[SCS-1112] Alte Grafik uebernommen**  
**[SCS-1112] old graphic duplicated**

see SCS-1111

**[SCS-1301] Terminal nicht angeschlossen**  
**[SCS-1301] terminal not connected**

The user tried to select a signal at a terminal, which is not connected and which had no signal at the time of the selection.

By activating the permission for implicit terminal to net, this message can be avoided.

**[SCS-1302] Kein Signal aufgenommen**  
**[SCS-1302] no signal selected**

A command for which a selected signal is a precondition has been activated too early.

Cause: see [SCS-1105]

**[SCS-1303] internal error**  
**[SCS-1303] interner Fehler**

see [SCS-1102]

**[SCS-1304] Signal nicht gefunden**  
**[SCS-1304] signal not found**

A signal has been activated via name or coordinate and has not been found by the system.

Cause: wrong name  
no signal on this coordinate

**[SCS-1305] Signal automatisch angelegt**  
**[SCS-1305] signal automatically created**

A signal has been created automatically (see SCS-1306) and a terminal has been assigned to that signal.

Cause: A signal free terminal has been selected and 'implicit SIG -> TRM' was activated in the submenu.

or 'Implicit SIG -> TRM' has been activated indirectly by clicking twice with the same coordinates. In this case, the message SCS-1301 appears first.

**[SCS-1306] Terminal in Signal aufgenommen**  
**[SCS-1306] terminal entered into signal**

A signal free terminal has been assigned to a signal by activating a certain function. At present this message only occurs in connection with SCS-1305. Possible functions are GET and INSERT.

Cause: GET -> see SCS-1305

INSERT -> the function has been selected without any signal being activated.

**[SCS-1307] Kein Terminal an dieser Stelle**  
**[SCS-1307] no terminal at this position**

The function INSERT did not find any terminal on the coordinate specified.

**[SCS-1308] Falscher oder ungueltiger Name**  
**[SCS-1308] wrong or invalid name**

The signal name collides with an existing name or does not correspond to the names' guidelines.

Also see CNORM-NAMES (yet to be delivered for 3.01)

**[SCS-1309] Speicherplatz nicht ausreichend**  
**[SCS-1309] not enough memory**

see [SCS-1109]

**[SCS-1310] Nur ein Signal veraenderbar**  
**[SCS-1310] only one signal modifyable**

## ERROR MESSAGES

---

**[SCS-1311] Terminal mit Signal belegt**  
**[SCS-1311] terminal hold signal**

It has been tried to add a terminal to the current signal with the terminal belonging to another signal

**[SCS-1312] Signal liegt schon an**  
**[SCS-1312] same signal inserted**

A terminal already belonging to the current signal was to be added to the current signal.

**[SCS-1313] Kein Segment selektiert**  
**[SCS-1313] no segment selected**

T-connections may only be on a segment which has been selected before.

**[SCS-1314] T-Verbindung nicht erlaubt**  
**[SCS-1314] t-connection not valid**

On airlines and slopes, T-connections are not allowed.

**[SCS-1315] Signal zusammengefasst**  
**[SCS-1315] signal merged**

Two signals have been merged to one signal.

Rules for the names:

Global signal have the highest priority

Manually created signals are at second place.

Automatic signals have the lowest priority.  
Their age is also considered.

**[SCS-1318] WARNING: no tracktype defined**  
**[SCS-1318] WARNUNG: Leiterbahntyp undefiniert**

This message shows a non-defined track type for the current signal.

Cause: old technology file

or

Changing the technology in PCB.

**[SCS-1319] Signal schon vorhanden**  
**[SCS-1319] signal already used**

This is a message caused by the functions CREATE or RENAME  
The signal name is already known to the system and may not be used a second  
time.

**[SCS-1322] postprocessing restriction**  
**[SCS-1322] postprocessing restriction**

Restrictions withing RELEASE 2.06

**[SCS-1331] Querschnitt ist negativ**  
**[SCS-1331] crosssection is negativ**

The entry for the crosssection is negative.

**[SCS-1332] Mindestabstand ist negativ**  
**[SCS-1332] clearance is negativ**

The entry for the clearance is negative.

**[SCS-1333] kein neues Signal selektiert**  
**[SCS-1333] no signal selected**

A signal has been activated via name or coordiante and has not been found by  
the system.

Cause: wrong name  
no signal on this coordinate.



# INDEX

## C. ROUTER

board 3, 4, 6, 10, 11, 23, 25, 26, 47, 49, 58  
calay 1, 16-18  
check 6, 13, 17, 22, 23, 25, 26, 29, 30, 34, 37,  
47  
cis 1, 3, 4, 7, 12, 13, 16-19, 21, 25, 43, 46, 47,  
50, 55, 58, 59  
columbine 38  
component 10, 14, 26, 28, 29, 36, 38, 50, 51,  
54, 56, 58  
connects 55  
digitized 7, 22, 25, 31, 33, 36, 44, 54  
ldpre 1  
ldrout 1, 2, 4, 6-8, 10, 16, 48, 51, 56, 57, 60, 61  
menu 2, 20, 25, 27, 30, 33, 34, 36-38, 43  
multilayer 4, 6, 10, 28, 48-50  
multilayers 7, 34  
netlist 1, 18, 47, 48, 53, 58  
offgrid 13, 20, 23, 43, 56  
optimized 6, 11, 17, 23, 31  
pads 55  
pcb 5, 8, 16, 18-20, 25, 26, 29, 30, 34, 37, 43,  
47, 58, 59  
popup 12  
prisma 2, 5, 21, 57  
report 1, 6, 17, 46, 47  
resist 26, 29, 30, 32, 34, 35  
routable 13, 16-18, 30, 31, 33, 34, 36, 39, 40,  
41, 44, 46, 47, 59, 60  
save 5, 8  
scs 24  
scscom 48  
size 16-18, 21, 25, 26, 31, 34, 35, 39, 40, 41,  
42, 44, 45, 47, 58  
smd 8, 10, 14, 28, 30-32, 47-52, 55  
unused 10  
window 4, 6, 20, 25, 30, 34, 37, 38

## D. COLUMBINE

board 1, 3, 4, 33  
btools 1, 6, 18, 19, 24, 25, 48, 54  
calay 1  
check 19, 25, 57  
cis 1, 4, 5, 8, 10-12, 20, 28, 33, 54  
columbine 1-3, 5-8, 10, 11, 13, 14, 16, 17, 18,  
20, 22, 24, 26, 28, 30, 31,  
33, 35-39, 41, 43, 45, 46, 48,  
50, 51, 52-54, 56, 57, 59  
command 32  
component 1-5, 8-12, 20, 24-26, 57, 58  
copy 13-15, 17, 20-23, 30, 37, 45  
digitized 4, 28, 47  
dload 1  
menu 6-26, 28-33, 35-39, 41, 43, 45-49, 54-58  
netlist 28  
pcb 3, 9, 28, 57  
prisma 1, 3, 32, 54  
report 48, 49  
schematics 4, 5, 8, 9, 12, 28  
scs 5, 9  
size 31, 33, 41-44, 51, 55  
submenus 49  
test 14, 22, 30, 39  
window 46

## E. SCSCOM

.AS 16  
.FORMAT7  
.TRACK6  
board 17  
calay 1  
check 1  
command 5  
component 10, 17  
pcb 1, 3-6, 14, 15  
prisma 1  
schematics 7  
scscom 1, 2, 10, 12, 17, 18  
size 2, 7, 8, 12-14, 17  
test 4

## F. POSTPROCESSING

artwork 4, 26, 30, 54, 59, 66  
batch 11  
board 1, 2, 11, 25, 26, 30, 32, 35, 39, 41, 42,  
46, 50, 51, 74, 76, 84, 96,  
97, 99  
btools 2  
calay 23, 26, 52  
calcom 13  
calpri 1, 2, 87, 89, 90  
calsoft 13  
check 22, 64, 69  
cis 1, 25, 27, 28, 42, 45, 48, 53, 54, 55, 59-61,  
64, 66, 67, 69, 75, 76, 79,  
84, 85, 86, 96  
columbine 22, 23, 33, 49, 50, 53, 54, 57, 60,  
62, 64, 67-69, 82  
command 10, 21, 23-25, 28, 29, 72, 73, 80, 82, 83, 90  
component 1, 32-34, 37-45, 75, 88, 89, 91, 93,  
94-96  
componentname 96  
componenttype 94-96  
digitized 48  
drilpp 2  
ercopp 2  
gerber 13, 14  
hp 13  
menu 1, 4, 6-25, 27-30, 34, 35, 36-60, 64-66,  
69, 70, 71-87, 89, 90, 99  
millpp 2  
netlist 88, 89, 91, 92, 95, 98  
optimized 90  
pads 40, 98  
pcb 25-27, 42, 45, 48, 53, 54, 55, 59-61, 64, 66,  
67, 69, 76, 84, 99  
plotpp 2  
prisma 1, 3, 15, 36, 52, 56  
proced 1, 2, 15  
readpp 2  
report 12, 71, 81  
resist 15, 25, 27, 39-41  
routable 92  
shelltool 90  
size 49, 50, 53, 59-61, 97, 99  
smd 40, 66, 94, 98  
sortpp 2  
submenus 65, 69  
test 14-16, 25, 27, 34, 58, 71, 74, 75, 78, 80,  
91, 93, 94, 95, 97, 98  
userpp 2, 4, 6  
vi editor 12  
window 25, 26, 38, 44, 90

**A. TUTORIAL**

autoroute 64, 65  
 board 4, 41, 44, 47, 48, 51-53, 55-58, 60, 63, 66, 67, 69-71, 86, 87, 90, 91  
 calay 1, 5  
 check 8, 9, 42, 48, 64, 69  
 cis 1, 3-5, 38, 41, 47, 51, 52, 62, 66, 68, 70, 85, 89  
 columbine 1, 2, 4, 19, 28, 31, 51, 53, 58, 60, 85, 87, 89, 90, 92  
 command 51, 56, 68, 70, 77, 78  
 component 1, 2, 19, 35, 37, 44, 46, 48, 52-61, 63, 66, 67, 71, 86, 87, 88, 90-92  
 copy 2, 16, 20, 29, 31, 35, 37, 53, 55, 61, 85, 90  
 digitized 25, 66  
 gphrai 1, 68  
 ldpre 67, 68  
 ldrou 1, 46, 47, 49, 64, 66-68, 89  
 menu 3, 5, 6, 8, 10-14, 16-18, 21, 23, 24, 26, 28-32, 35-39, 41-46, 50-58, 60-63, 65-67, 69, 70-72, 74-76, 78, 79, 82, 84, 85, 86, 88-90, 92  
 netlist 64, 68  
 nonorthogonal 25, 64  
 offgrid 73  
 opens 19, 42, 64  
 pads 73, 75, 77, 79, 89  
 pcb 1-5, 8, 19, 31, 35, 37, 38, 41-43, 47, 62, 63, 69, 85, 86, 89, 90  
 popup 3, 12, 39, 56, 62, 63  
 prisma 1-3, 30, 67, 68  
 raigph 1, 68  
 report 38  
 resist 48, 49  
 routable 49  
 save 17, 42, 73  
 schematic 1, 2, 4, 5, 8, 35, 37, 39, 51, 53, 85, 89  
 schematics 3, 7, 8, 19, 29, 35, 39, 41, 42, 43, 51, 53, 58, 62, 85-87, 89, 90, 92  
 scs 19, 31, 35, 37, 38  
 scscom 1, 68  
 shorts 42, 64  
 size 14, 36, 44, 47, 49, 57, 60, 61, 86, 90  
 smd 49, 54, 61, 73, 77, 89  
 teardrops 77, 78, 80, 89  
 unused 35  
 window 3, 9, 12, 13, 19, 23, 29, 35, 38, 43, 44, 61, 62, 67-69

**B. CIS REFER MANUAL**

artwork 8, 13, 14, 47, 48, 69-71, 78, 79, 114, 153  
 autoroute 99, 153  
 board 3, 68, 69, 71, 75, 77, 78, 85, 86, 96, 115, 120, 129-132, 150, 152  
 calay 2, 5, 127, 153  
 check 41, 51, 86, 92, 99, 109, 118, 129, 150, 152  
 cis 2-4, 18, 75, 127, 153, 154  
 columbine 3, 4, 18, 42, 81,  
 command 82, 90

**B. CIS REFER MANUAL (cont.)**

component 3-5, 18, 19, 28, 33, 38, 42, 55, 56, 60-62, 68, 69, 71, 72, 76, 77, 79-91, 98, 103-107, 109, 111, 115, 120, 133, 134, 139, 142  
 copy 5, 18, 19, 22, 24, 27, 32, 45, 82, 90, 92, 101, 103, 147, 149  
 digitized 3, 20, 59, 81, 84, 86, 95-97, 99, 123, 128, 136, 138, 142, 153  
 hardcopy 54, 121, 153  
 menu 2, 5-7, 9-13, 22, 31-33, 36, 37, 39, 44, 45, 47-50, 53-55, 58, 60, 61, 64, 66-73, 75-77, 79, 85-87, 101, 105-107, 111, 112, 114, 115-118, 120, 121, 123, 124, 126, 130, 131, 133, 135, 138-140, 142-144, 148, 152  
 multilayer 71, 115, 134, 153  
 multilayers 140, 142, 153  
 netlist 53, 99, 153  
 nonorthogonal 28, 34, 99, 149, 153  
 offgrid 129, 153  
 optimized 84, 129, 136, 153  
 pads 68  
 pcb 3, 5, 15, 16, 18, 19, 23, 30, 35, 44, 53, 55, 56, 58, 59, 69, 74, 75, 77, 91, 92, 94, 106  
 pixels 59, 123, 153  
 placeholders 16, 80, 153  
 popup 2, 5, 22, 23, 33, 36, 53, 58, 85, 114, 115, 120, 123, 149, 150, 153  
 popups 53, 123, 153  
 prisma 2, 3, 5, 53, 55, 58, 69, 76, 153  
 report 53, 120, 150, 152  
 resist 135, 137, 140, 142  
 routable 135, 136, 138, 140, 142, 153  
 save 64, 146, 150-152  
 schematic 38, 55, 96  
 schematics 4, 11, 15, 16, 18-21, 30-32, 42, 45, 49, 51, 53, 59, 82, 83, 87, 91, 107, 123  
 scs 4, 5, 8, 10, 13, 15, 16, 30, 35, 44, 45, 47, 49, 53, 55, 56, 58, 59, 72, 87, 90, 91, 94, 116, 129  
 shelltool 53, 120, 153  
 size 6-8, 13, 20, 37, 40, 47, 58, 59, 67, 70, 78, 110, 114, 124, 127, 130, 131, 136, 140-142  
 smd 62, 134-137, 153  
 submenus 60, 153  
 unix 64, 65, 69, 76, 153  
 vi editor 74  
 window 2, 5, 7-9, 11-14, 16-18, 20, 21-27, 30, 32, 33, 35, 36, 37, 39-45, 47, 48, 49-51, 53-56, 58, 59, 61, 66, 68, 70-73, 78-83, 88-94, 96, 103-106, 109-111, 114-118, 120, 121, 123, 124, 126, 130, 132, 135, 139, 140, 143, 146, 148-150