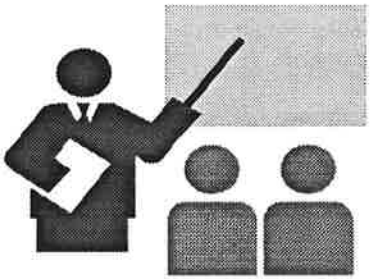
SUPERMAX E-CAD

RELEASE 3.0

IPL BEGINNERS GUIDE



Dansk Data Elektronik
EDA Division
Herlev Hovedgade 199
DK-2730 Herlev
Denmark

Phone (+45) 42 84 50 11
Fax (+45) 42 84 52 20
Telex 3 52 58 DDE DK

English Version

List of Contents:

About this manual

This manual is an introduction to the Supermax E-CAD IPL-system's PCB-layout part.

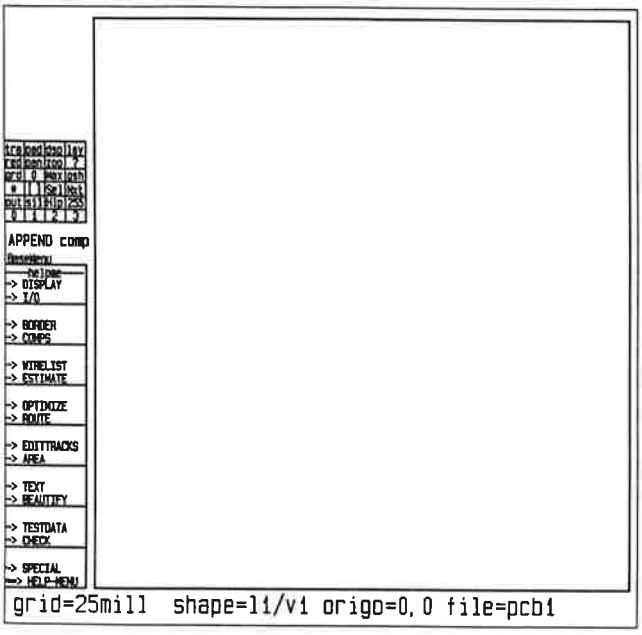
It consists of an tutorial stepping through the design of a fairly standard PCB, with each step explained by the use of a user exercises.

The tutorial is formed like the "TV-kitchen" with pre-prepared PCB examples to each chapter. This makes it possible to use real-world examples directly from the start, thus giving a much better understanding of the powerful features in the Supermax E-CAD IPL-system. It also makes it possible to skip some chapters a first reading, without disturbing the sequence.

When you have worked through the pre-prepared examples, you could reread the tutorial using your own PCB-example.

The remaining part of the manual is devoted to more specialized topics, and can be used as a "cookbook", where solutions to specific problems can be found.

It complements the Supermax E-CAD IPL-reference manual, by giving a solution oriented description of the system. The reference manual gives an exact description of each single command in the system. It is highly recommended that you make yourself acquainted with the reference manual.



You use the cursor to point in the menus.

To do any of the operations place the cursor over the line in the menu and press the white key on the puck.

Please note the 2 lines over the bottommenu:
The lower line shows the name of the menu "BaseMenu", the upper line "APPEND comp" shows the name or number of the current working layer.

When pointing in the menu with the cursor and pressing a button on the mouse the system will either execute some command or change the menu. The lines in the menu starting with '->' will change menu, the others will activate functions.

The menu shown is an overview-menu giving, when read from top to bottom, the normal sequence of operation used to create a pcb.

-> I/O	set working area, save or load jobs, output netlists and componentlists
-> BORDER	define PCB outline
-> COMPS	place components, create/change library
-> WIRELIST	enter/change netlist
-> ESTIMATE	estimate board and connection densities
-> OPTIMIZE	create/refine the component placement
-> ROUTE	automatic route powernets and signal nets
-> EDITTRACKS	manually alter the physical connections
-> TEXT	place free text and change textplacements
-> BEAUTIFY	refine connections
-> TESTDATA	create information for bareboard and incircuit tests
-> CHECK	check that designrules are not violated

Selecting any of these items will bring you into a submenu.

Although the Supermax E-CAD IPLsystem is normally controlled by menus, you can always bypass the menus and access any command just by writing its name or an abbreviation of it. It is also possible to jump forth and back in the menus by typing the menuname.

The upper menu-area, called the topmenu, is for very often used functions.

You should try the functions zoom, pan and redraw:

On the keyboard type cuisine0 and press the <return> or <enter> key. This will execute a commmandfile, that loads a small pcb. The pcb contains small texts and small boxes.

To zoom in on some of texts you should move the cursor to zoo in the topmenu and press a button on the mouse.

> Any button except the green can be used, the green means escape. This is used if you have selected the wrong command or if you want to end a series of repetitive commands. Other ways to escape or stop a command is by pressing the ctrl key and the D or C key.

After pointing to zoom Supermax E-CAD IPL will ask the following questions:

leftlo

This is the lower left point of the rectangle you want to zoom into. To select this point, move the cursor to the chosen point and press the same button again.

righthi

This is the upper right point of the rectangle. You can see the dynamic box displaying the rectangle. When you press the button once more, Supermax E-CAD IPL will redraw the picture displaying only the chosen rectangle.

You can see Supermax E-CAD IPL will allow any zoom.

To pan around in the zoom you can use the pan function. Point to pan in the topmenu.
Supermax E-CAD IPL will ask for:

center

the center of the new picture. If you point close to edges of the current zoom, you will get the neighbor picture with 50% overlap. The new picture will have the same size.

You can also use the so called 'zoom backwards'.
Point to zoo in the topmenu and Supermax E-CAD IPL will prompt you for:

leftlo

in this case you point to some arbitrary point

righthi

now you zoom backwards, that is you point to a point to the left of and/or lower than the leftlo point.

Now Supermax E-CAD IPL will redraw the complete working area showing just the borderline and a small box showing the position of the last zoom. If you made some alteration in the pcb, the last connection touched will also be display. After this has been displayed Supermax E-CAD IPL will ask for a new zoom:

leftlo
the leftlo low point of a new zoom or a arbitrary point.

righthi
here you can select a new rectangle of a new zoom, or you can zoom backwards again. If you zoom backwards again Supermax E-CAD IPL will display the complete workarea.

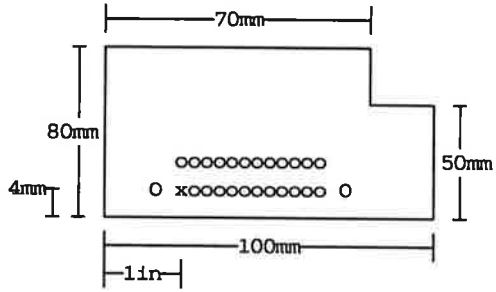
The command red in the topmenu is use to refresh the current zoom. This might be necessary after moving overlapping components.

1. Setting Workspace and creating a border outline

In this chapter you will learn to create the PCB outline and place connectors on their exact locations.

Point to -> Border. This will change the menu to the BorderMenu, where you can define the size of the workspace and create the outline of your PCB.

As an exercise you should create the following PCB from this mechanical drawing:



First the working area should be set. Normally you should create it slightly bigger than your PCB, so in this case a size of *110mm* by *90mm* will be fine.

Point to PCBMAX in the menu. Supermax E-CAD IPL will now pose the following questions in a popup menu with the header PCB maxima:

xmax:

The *xmax* is the distance in the horizontal direction (the x direction). This should be *110mm*, so you change the the value that Supermax E-CAD IPL proposes. To do that you can use the arrowkeys and the functionkeys F6,F7,F8 on the keyboard.

ymax:

The *ymax* is the distance in the vertical direction (the y direction). This should be *90mm*.

layers:

The *layers* parameter is the number of layers that Supermax E-CAD IPL will use simultaneously for autorouting, so this

should be set to the number of layers you will use for signaltraces. In this case set it to 2 by writing 2, by use of the arrowkeys and return or by pointing to the field.

pcbgrid:

The Supermax E-CAD IPL system may be adjusted to any of these grids or to other grids if written. In this case point to *1/40"*.

The PCBmax ratings are altered when execute is pointed and the screen is redrawn with the new working area.

> Supermax E-CAD IPL will recognize the following units:
> mill, inch, mm, um, grid, Modules
> missing unit normally corresponds to mill

Next you define an origo of the PCB that will correspond to the zeropoint of the drawing. It should be placed a small distance from the edge of the working area, because Supermax E-CAD IPL will not accept negative coordinates. In this case a distance of 2 Modules will be sufficient.

Point to set ORIGO and enter *200,200*.

Next you should place cornermarks in the corners of the PCB according to the drawing. You can start by placing the bottom leftmost corner, which should be placed in the zeropoint.

Point to add CORNERmark. Supermax E-CAD IPL will then pose some questions to you:

at:

This is the placement of the cornermark. Since it is placed in the zeropoint you just enter *0,0* (or simply *0*), by typing it directly on the keyboard.

type:

This is the kind of cornermark. Actually it is the name of the figure in the library. Here you should use the name *mis/outercorner*.

dir:

This is the direction of the cornermark. The direction is measured in steps of 90 degrees anticlockwise rotation:

For this cornermark use *0*.

Now Supermax E-CAD IPL fetches the cornermark from the library and places it in the zeropoint.
Next you can place the lower rightmost cornermark. Since Supermax E-CAD IPL is repeating the add CORNERmark command it will be asking you for the next placement.

at:

Again this is the placement of the cornermark. Since the placement of cornermarks is essential you should enter the exact coordinates. Just type the coordinate *100mm,0*.

type:

Again it should be *mis/outercorner*.

dir:

Now it should be rotated 90 degrees anticlockwise, so use *1*.

After answering these questions Supermax E-CAD IPL will copy the already known cornermark and place it exactly in *100mm,0mm*. (just for fun you can check it out by pointing to the ? in the topmenu and the point to the newly inserted cornermark. Supermax E-CAD IPL will the then tell you what is found in the pointed coordinate:

"pad(:corner2-1): at 3937,0,pad paddir 1 shape corner"

You can see that the cornershape is placed in the coordinate 3937,0,pad which is 100mm,0 translated into mills)

Now place the rest of the cornermarks using the add CORNERmark. For the fourth corner you should use the type named *mis/inner-corner* (in dir 0) instead of the mis/outercorner.

The next step is to create the borderline. It will be done by drawing a line between the cornermarks. This borderline will show where the edge of your PCB is placed and it will restrict the autorouting functions. This restriction should be inserted on layer 255 which is equal to 'all' lays.

Point to 255 in the topmenu.

Point to add TRACK. Supermax E-CAD IPL will then ask you the following questions:

from:
This is the start of the borderline, so you can point to the lower leftmost cornermark. Supermax E-CAD IPL will automatically snap into the "center" of the cornermark, so you will be guaranteed that you get the correct coordinates, even if the point is offgrid.

to:
The next point on the borderline. Point to the rightmost lower cornermark.

to:
Supermax E-CAD IPL repeats the command add TRACK until you break it by pointing to a new command in the menu, you press the space-bar or you press the green push-button. Use this facility to connect all the cornermarks together.

> In Supermax E-CAD IPL you can define border outlines in a more advanced manner by creating restriction areas for components, vias and tracks. This will be described in the chapter "Obstruct Areas"

The last step will be placing the connector, since it should also be placed in an exact coordinate.

Point to add CONNECTOR and then answer the following questions:

at:
This is the placement of the referencepoint of the connector. Normally it is pin 1. In this case -according to the mechanical drawing- it is lin,4mm, so you should enter *1in,4mm* (or *1000mi,4mm* or *1000,4mm*).

compname:
Is the name of the component. In this case *J1*.

article:
Is the stocknumber of the component. In this case it is a connector of the kind **readout34**, according to the drawing. Supermax E-CAD IPL will use this information to search in the database for the appropriate physical type. It will also be used in partslists and in Bill of Materials.

device: (only requested if article is not found in database)
This is a description of the component. It is used for the same purposes as the article name.

type: (only requested if article is not found in database)
This is the name of the component kind in the library. The library consist of one small file for each different physical component type (e.g. **con/read-out-m34ab**). **dir:** Like the add CORNERmark it is the direction. Here it should be *1*.

Now you have created the PCB outline and are ready to continue.

2. Placing components I

In this chapter you are introduced to the basic facilities for placing and moving components.

If you did not create the border outline in chapter one you can get a readymade border by typing `cuisine2`. This is a small commandfile that will fetch the readymade border for you.

Now enter the `CompMenu` by pointing to `-> COMPS` in the `BaseMenu`.

To get a good placement you might want to set up a grid of *1*Module. To do that you point to the `grd` in the topmenu. The Supermax E-CAD IPL will show this menu:

```
Usergrid = Double Half 100mill 25mill 20mill 12.5mill
Drawgrid = none 1in 500mill 100mill 50mill 25mill 20mill
Drawcoor = none 5in 1in 500mill 250mill 100mill
```

Here you can point to *100mill* in the Usergrid line, to *100mill* in the Drawgrid line and *1in* in the Drawcoor line. Finally point to `Execute` in order for the system to proceed.

Point to `lay` in the topmenu, and point to `defined` in the topline and then `redraw` in the button line. This will ensure that all component informations are displayed.

Then point to `add COMP` and the following questions are asked:

at:

Point to where you want the component placed. The point is the placement of the component reference, which is normally pin 1 of leaded components and the component center for surface mounted components.

compname:

This is the reference name of the component. You can accept the proposed name by pressing a button on the mouse, or you can use the keyboard to enter a new name, editing the name with Function buttons F6,F7 and F8. Supermax E-CAD IPL will automatically increment the proposed name. A component name can be up to 200 letters long and can contain any letters

except *, ?, [,], \, #, ", ', or ` which are use for special purposes. Try to enter the name *IC1*.

article:
This is the stocknumber of the component. It is used when you create partslists and bill of materials. It is also used a as entry into the Supermax E-CAD IPL-relational database. You can ex. enter the name *7404* from the keyboard.

device: (only requested if article is not found in database)
This is a description of the component. It is used for the same purposes as the article name. Enter any name.

type: (only requested if article not found in database)
This is the name of the component kind in the library. The library consist of one small file for each different physical component type. E.g. use type=*ic/DIP-14-mot*.

dir:
The direction relative to the direction in the library. The component can be rotated 0,90,180 or 270 degrees anticlockwise corresponding to the **dirs** of 0,1,2 and 3. It can also be mirrored by putting a *m* after the number. When a component is mirrored the silkscreen is automatically moved to layer for soldering side silkscreen, and the surfacemount footprints are also moved to the correct layer.

After answering these questions the component is fetched from the library. If you point to close to the edge you will get the message *Outside Border* meaning that a part of the component would have ended outside the working area. If so you will have to try again with a better placement.

Now the system is prepared to place a new component of the same kind, so by pointing to a new placement you will place a similar component (notice that the proposed name was incremented). If you want to place another kind of component just point in the menu on the add COMP again or press the < key which means "restart" of many menucommands.

After placing the component you can move,rotate and mirror it, by using the move COMP, turn COMP and mir COMP. Or you can move multiple components with the move CMPinWIN.

While moving a component it can be rotated 90 degree anti clockwise each time the r key is pressed.

You can change the name of the component with the rename COMP.

You can place components nicely in rows and columns by use of align COMPS, where you select one component as **master** and put the following on the same row or column. The components will be aligned in the direction where there are least difference. In order to specify a new master point to the menufield again or use the < key in order to restart the command.

Two components can be interchanged with the swop COMPS.

When moving components parts of the silkscreen may disappear from the display. Point to red in the topmenu to 'refresh' the picture.

If you want to remove a component you use the rm COMP. This command will ask you for a confirmation before deleting the component. If the component is protected (like the cornermarks and the connector you will not be allowed to remove it, nor will you be allowed to move it).

As an exercise you should try to place some different components for instance the following component list:

Enable the database by pointing to DATABASE y/n in the menu. If it is not already y set it y.

Remove all components by use of rm COMP with * in the topmenu and **confirmed** *y* to accept the deletion of the components. The border components and the connector will remain as they are protected against deletion.

Component name	Article name
IC1	7404
IC2	74LS00
XTAL	XTAL
R1	r33k
R2, R3	r22k
C1, C2	c22nF
J1	connector already placed

This componentlist will be used in the next chapter.

3. Entering netlist interactively

This chapter explains how to create or alter an interconnection list (netlist) interactively on the screen. For further information see the chapter iwl in the post-manual

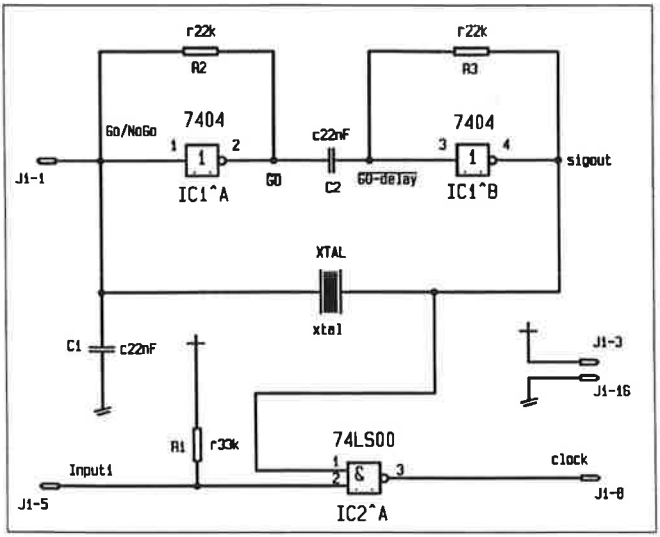
If you did not place the components in chapter two you can get a readymade pcb by typing `cuisine3`.

From the BaseMenu point to `-> WIRELIST`.

Then point to `Show PIN/2` followed by `*` in the topmenu. This will display every second pinnumber on all components. If you use `Show PINhalf` only the highest and the middle pinnumber will be shown (f.x. pin 7 and *14* on a *14pin* package). This information is only drawn as graphics on the screen, so after any redraw, panning or zoom it will disappear. But then you just point to the command again.

A netlist (or wirelist) in Supermax E-CAD IPL is an collection of nets. Each net is an unordered set of pins that should be connected together in the final artwork. That is, the netlist is not organized as a list of from-to pairs, but rather as a pool of pins. The Supermax E-CAD IPL will always know the shortest way to intercon nect these pins. Each net in the netlist is called a **group** and it will have a unique name. When initially a component is placed on the PCB each of the pins will belong to its own group (containing only that single pin). These groups will be given a default name constructed as **compname-pinname** f.x. *IC2-3*. A group can be referred to by its name or by one of the pins in the group f.x. *Gnd* or *IC3-7*. The words group, net, signal and wirelist (wl) are used to connote the same thing.

Now you are ready to create the netlist. You will create a netlist corresponding to this small schematics:



Start by creating the net called *Input1* going from *J1* pin *5* to *IC2* pin *2* and *R1* pin *1*.

Point to Join WL ... . Supermax E-CAD IPL will then ask you the following:

master:

This is the name of the pin or net you would like to add more pins to. You can point to the desired pin or you can type the name of the pin (f.x. *J1-5*).

group:

This is the name of the pin or net you would like to add to the master net. In this case point to *IC2* pin *2* or type *IC2-2*.

group:

Now Supermax E-CAD IPL repeats joining to the master net. In this case point to *R1* pin *1* or type *R1-1*.

Now this net is completed and you should give it the name Input1 according to the schematics. To do that you point to grp RENAME. Then Supermax E-CAD IPL will ask:

oldname:

The old groupname, which was the name of the master group (in this case *J1-5*). Here you can point to one of the pins in the net or you can type the name *J1-5*.

newname:

This is the name you want to call the net. In this case *Input1*. Please still remember that Supermax E-CAD IPL will distinguish small and CAPITAL letter.

Now you can continue to create the other nets from the schematic.

For the nets *0v* and *5v* you can use the command set GRP PROT which will tell Supermax E-CAD IPL that these nets are precious. This will prevent you from accidentally joining it into another net and it will also prevent automatic rerouting of the net.

To inspect the netlist you can use rubberbanding and highlighting:

Point to RB grp and answer the question **group** by pointing to some pin or by typing the appropriate group- or pin-name. Then, if the pin is connected, you will see a rubberband (ratsnest) for this particular net. You could also answer the question by pointing the * in the topmenu. Then you will see rubberbands for all the nets.

The commands rb COMPpair and rb COMP works in a similar way, but gives you the rubberbands between two components or connected to a component respectively.

Point to MARK grp and try answering with either one net or with the *. You will notice that pins not connected are not highlighted.

You can also list the contents of a group by using the command LIST wl. Or you can list the groupnames only with LIST wlNAME.

If you have made some errors (very unlikely) and want to change the netlist you can use the command set PIN single. This command will allow you to disconnect one single pin from a net. An other

way to display information about components and nets is found in the Estimate Menu (point to -> ESTIMATE in the BaseMenu)

Here you can use Estimate which will give you some board statistics:

```
pcbarea: ...
number of pins:...
pindensity:...
area pr. pin:...
component, pad density for each lay:...
the number of components:...
estimated rubberbandlength and density:...
```

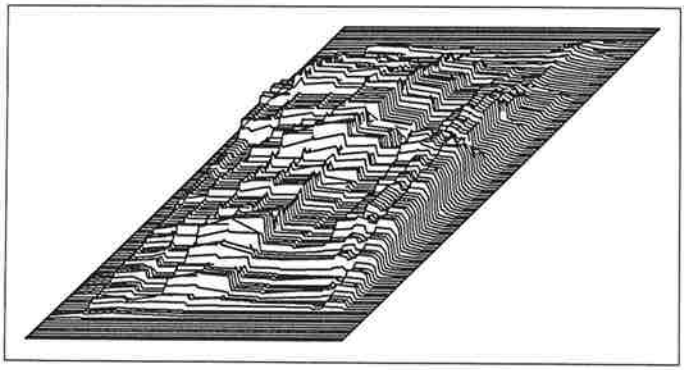
Or you can point to DENSITY and pick one of the options in the Popup:

The **WLcut**, **FROMTOcut**, **MISScut** and **cut+PADS** will display a histogram showing how many connections must cross a given vertical or horizontal cut of the PCB. (**WLcut** looks at the boxes around the groups in the wirelist, **FROMTOcut** looks at the actual expected fromtos, **MISScut** only looks at missing connections while **cut+PADS** counts the numbers of pins placed in the same cuts and add this to the histogram). The histograms can be used to estimate the density and thereby the difficulty of the layout.

Some rules of thumb are:

density top less than 25% - easy automatically routable 2 layer board,
density top between 25% and 35% - more difficult you should consider using innerlayers for powernets,
density higher than 35% - very difficult use multilayers with signals and/or fineline and microvias.

The **AREAdense** will show a twodimensional "histogram" displaying with boxes of different sizes the densities across the board. This is very useful because it gives more precise indications of the problematic areas of the PCB. The **LANDSCAPE** gives the same information, but displays it as a 3-D landscape.



4. Placing components II

> In this chapter you will learn to use the automatic placement routines and placement optimizing commands. It can be skipped at the first reading without interfering with the sequence of chapters.

In this chapter you will use a more complex PCB which will give a better demonstration of the utility of the Supermax E-CAD IPL placement tools. To get this PCB type `cuisine4`. This commandfile will reset the Supermax E-CAD IPL and fetch the bigger PCB.

This PCB is a typical digital design. Actually it is a DMA controller for a small modular computer.

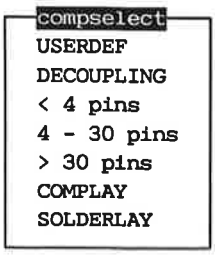
When the board is fetched you can see that the borderline is defined and the connector is placed, but the rest of the components are found in a heap in the middle of the board. This is the typical situation you get after loading a netlist, from the schematic capture system.

First thing to do is to check some board statistics. Go to the `EstimateMenu` by pointing to `-> ESTIMATE` in the `BaseMenu`.

Point to `ESTIMATE` in the `EstimateMenu`. Then you will see the essential statistical numbers on the board density. You should check that the percentage of board area covered by components is less than 100%. Otherwise you should consider using other packagetypes f.x. SMT or placing components on both sides of the board. In this examples there is plenty of space.

Go to the `PlaceMenu` either through the `CompMenu` and `-> PLACE` or by pointing to `-> OPTIMIZE` in the `BaseMenu`.

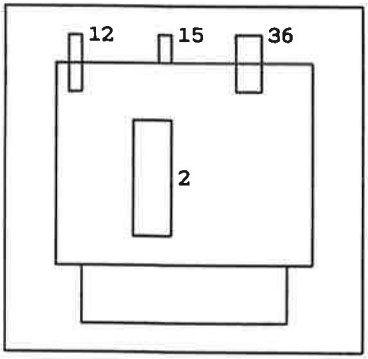
Start by dividing components in different groups depending on their sizes or functions. To do this you point to `sel COMPS` and you get the Popup:



With this commands you can select some components for further processing. In this case point to DECOUPLING, then Supermax E-CAD IPL tells you how many decoupling capacitors are selected.

Next you point to `mov COMP` followed by `Sel` in the topmenu. Place the component near to - but not too close - one of the corners. Then Supermax E-CAD IPL will move ALL the components selected.

Then do the same with `< 4 pins`, `4 - 30 pins` and `> 30 pins` but place them in separate heaps.



The four heaps:

1) 12 decoupling capacitors
2) 15 small components
3) 36 medium sized components
4) 2 big components

You have now separated the components in different groups depending on their sizes.

The small components with less than 4 pins and the decoupling capacitors should not be considered in the beginning, so you

should give them the "dontcare" status.

To do that point to FIX operations and point to FIXWINDOW and DONTCARE in the popups, after which Supermax E-CAD IPL will ask for **leftlo** and **righthi**. Frame in the windows containing all the small components.

The mediumsized components should have the NOT FIX status. Use FIX operations to give them this status.

The big components should be manually placed and should be left untouched by the automatic routines. Therefore they should have the status **FIX**.

To check that you have given all the components the correct status, you can use the DISPFIX in the FIX operations. You will see that dontcared components will display a small "D", the notfixed a "N", the fixed ones a "F" and the protected connector a "P".

Set a usergrid of *1*Module (100mill) with the grd in the topmenu.

Now you can manually move the big components with the mov COMP. You can see that the rubberbands are not going directly to the connector, so therefore you should try placing them in the middle of the board.

Next point to INITPLACE std, and point to **USERDEF** in the popup. Then it will ask some questions:

leftlo and righthi:

The corners in the area in which the components should be placed. That is, in which the component references should be placed, so actually some part of the components can end up outside the area. In this case you can use a point close to the lower left corner of the PCB -just inside the borderline and over the connector, and another point close to the upper right corner of the board.

divx:

The horizontal distance between the gridpoints where initplace will try to place the components. In this example a distance of *5M* will be good because all the medium components are *3M* wide.

divy:
The vertical distance between gridpoints. Here *1.2inch* will be fine.
dir:
The direction of the components relative to the library. Use *0*.

Now initplace will place all the *notfixed* components, one at a time, in the optimal gridpoint. That is, the gridpoint where the rubberbandlength to the already placed components will be at a minimum.

Probably the big components you placed manually are not aligned with the grid that initplace is using, but you can just use the mov COMP to put them in a more convenient place, and then restart the INITPLACE std. To use the same leftlo and righthi as before just hit the return key. This means keep the current value.

Now you can evaluate this initial placement by using RBG comp. You can try different manual placements of the big components and watch the result after restarting the INITPLACE std. The relative high speed of the initplace routine allows you to experiment with many different solutions.

You can also use FIX operations and FIXCOMP to fix some of the mediumsized components so the initplace also will consider these as "seed" components. This might be a good idea with some of the components highly connected directly to the connector. They may even be turned to direction *1* and placed very close to the connector to allow a very easy routing. This is a typical situation for busbuffers.

If your pcb is not too complex, you can also use the **AUTOMATIC** option in the **INITPLACE std** popupmenu. It will automatically divide components in 3 groups: components wider than 5 modules, components between 3 and 5 modules, and slim components not wider than 3 modules. It will then try to place these 3 groups of components using some reasonable grids corresponding to the width of the components.

When you have reached a placement that looks acceptable, you should optimize it further. This is necessary because initplace only places one component at a time and therefore an already

placed component may block for a proceeding one.

In order to see the effect of using optimize you should start by showing the current density figures. Point to <-> ESTIMATE and to DENSITY std.

Go back to the OPTIMIZE menu and point to OPTIMIZE std.

Optimizing will consider all the *notfixed* components and try to interchange them in order to minimize the rubberbandlength.

Optimize will decrease the rubberband length 5% to 10%. It should always be used, even on a completely manual placement, since it will only interchange components and thus not destroying the "appearance" of the placement.

Use the DENSITY std command in the ESTIMATE menu again to see the improvement.

Now you use the FIX operations to fix all the big and mediumsized components and to *notfix* the small components - but not the decoupling capacitors. Then use the command INITPLACE std with a smaller **divx** to initplace them (f.x. use *100mil*), followed by OPTIMIZE std and OPTIMIZE rot to minimize the rubberbandlength.

Last you have the decoupling capacitors left. These can not be placed with initplace with a good result because they are connected to almost all components on the board.

Use sel C.inWINDOW to select the decoupling capacitors in the heap. Then point to move NEXT... which will allow you to manually place the components from the heap - one by one.

Use the R key to turn the components 90 degree while they are being moved.

5. Power routing

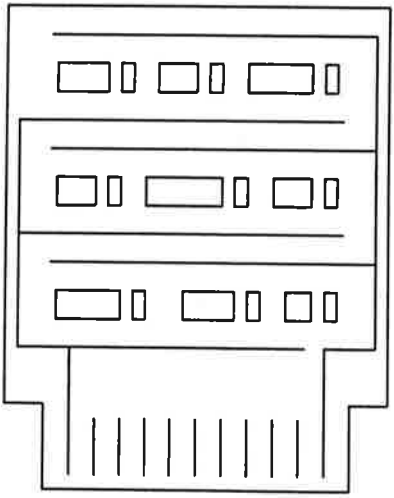
This chapter will teach you how to layout a good Powernet structure. It will also be the first chapter wherein you meet the autorouter.

Now you should reset the working space and a small example should be fetched. To do this enter the IOMenu and point to RESET PCB and answer the question **fast** with *y*. Then fetch the new example by pointing to GET pcb and answer the question **file:** with the name *cuisine3*.

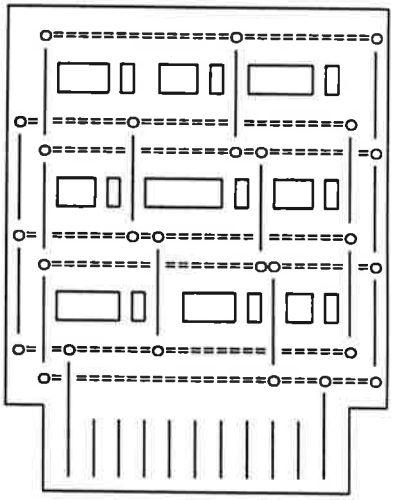
Go to the PowerRoute menu by pointing to ROUTE in the BaseMenu and then point to PowerRoute in the RouteMenu.

Normally a powernet should be made in a special manner using a thick track and placing them in a pattern like a comb or a grid.

COMB:

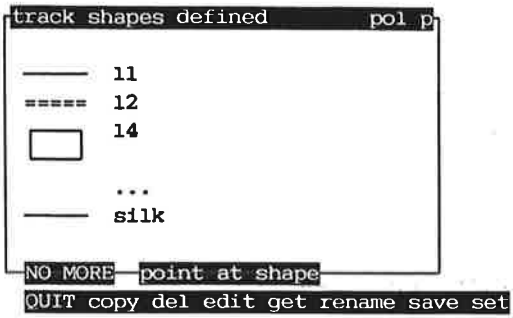


GRID:



To make a layout like this you start by selecting a wide track.

Point to tra in the topmenu. Then you will have a menu looking like this:



Point to set in the bottom line and the point to the desired shapes name, in this case the *14*. Now you see in the statusline in the bottom of the display that your current shapes are the *14* for tracks and *v1* for vias.

> For further information on trackshapes see chapter shapes.

Next you can tell the system that you do not want 45degrees tracks by pointing to 45deg off.

Then you should highlight the *GND* net by pointing to MARK GRP and then point to one pin known to be in the *GNDnet* or by type *GND*.

Now you are ready to make a "sketch" of the ground pattern. The router will -as a default- use the componentlayer (lay 0) as the lay for horizontal tracks, so you should also use layer 0 for horizontal tracks. To select layer *0* point to 0 in the topmenu.

If you have set the grid to *100mill* it might be better to change it to *25mill* or *50mill* using grd in the topmenu.

Point to add track and add a horizontal track starting from one of the marked pins in the *GNDnet*. If you want to change to the vertical direction you should place a via by pointing twice in the same point. Then Supermax E-CAD IPL will place a via and change the lay to solderside (lay 1).

You should add just so many tracks as are needed to sketch the pattern either as a comb or a grid as shown above.

It is very important that all the tracks are connected somewhere to a pin in the *GNDnet*. To check this you point to SHOW LAST and then all the tracks should be highlighted. If they are not connected to a pin in the net the router will not consider them as part of the *GNDnet* and will not used them.

Now you point to route GRP and to one of the tracks. Then Supermax E-CAD IPL will route the rest of the *GND* net using your sketch as a basepattern, yielding a very good powerconnection.

Since you made only a very rough sketch then after the routers have finished some of your tracks may be ending in "nowhere". To cleanup these blind ends you point to rm BLIND.

Segments may be swopped to the other side by use of the command swop CONN.

Now do the same with the *+5V* net.

After finishing the two powernets be sure to allow 45 degrees tracks by pointing to 45deg on and also set the grid to *25mill* using the grd in the topmenu.

6. Signal routing

> In this chapter you will autoroute the rest of the signals on your board.

You will continue with the example from the previus chapter.

Go to the RouteSub from the RouteMenu.

Start by selecting a thin trackshape using the tra in the topmenu (f.x. *11*).

Next point to BUS route. This will route all connections in busses (Supermax E-CAD IPL defines a "bus" as any connection going strictly horizontal or vertical).

Next you should point to SMALL route and * in the topmenu to specify all groups. This command will route all the remaining connections starting with the shortest (smallest) and ending with the longest connections.

In this very simple example the SMALL route should be able to route every connection. To check this, use sel MISS which will select all the remaining not connected nets. It should give you the message *0 selected*.

As you can see the router has been very meticulous using the principle of using layer 0 for horizontal track and layer for the vertical ones. This gives you -especially on an easy board like this- a lot of unnecessary viaholes.

You can have a list of vias used by pointing to list VIANUM.

First you point to the auto SWOP command which will try to swop a track segment from one layer to the other in order to remove viaholes. Try the command list VIANUM to see how many vias was removed. Next you can use the MINI-CLOSE which will remove and reroute all the nets except the powernets (the protected nets). When rerouting the router will use another set of parameters that will allow "wrong" direction tracks if it can avoid placing a via. It will also try to move the connections away from the component pins and try to space the tracks more equally. After

it is finished you can use the list VIANUM again.

The MINI-CLOSE command can be used several times, each one giving some savings in vianumber and some better spacing of tracks. But normally any number more than 2-3 will give only marginal differences.

7. When the router needs help, but helps You anyway

This chapter tells you how the Supermax E-CAD IPL can help you connect the nets, even when the autorouter can not do it fully automatically.

To demonstrate these, probably the most powerful commands, you should try with a "real" PCB. Type `cuisine7` and Supermax E-CAD IPL will fetch a board with some missing connections, that the standard router is unable to complete.

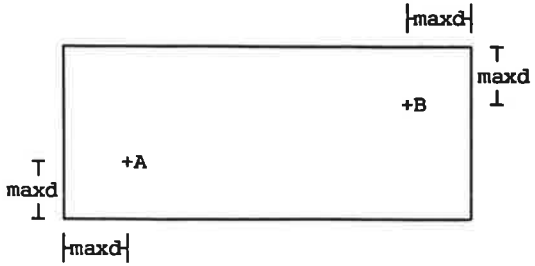
Go to the `RouteMenu` by pointing to `-> ROUTE` in the `BaseMenu` or by pressing `R` on the keyboard.

Start by selecting the missing connections point to `SEL MISS`. This will select the missing nets, so you can systematically step trough them and finish the work. To see the nets point to `MARK/RBM GRP` and answer the question 'group' by pointing to `Sel` in the topmenu.

Use the same trackwidth and viasize as are used in this PCB. To do that point to `gsh` in the topmenu, and then to a via/track in the PCB. Supermax E-CAD IPL will write the names of the track and via shapes you pointed to and make these the current shapes.

You can verify that the router is unable to finish the net(s), if you point to `route GRP` and then point to `Sel` or `Nxt` in the topmenu. In the latter case only the first of the selected nets are tried routed, in the former all the missing ones are tried. When the router is started for the first time after fetching a PCB or after changing the designrules, it will give the message **INST MATX** which means it is creating the correct "maze" for the router. On a big board this can take some time (up to several minutes).

The reason that the router can not complete some of these nets is the standard restriction laid on the router. It is not allowed to use a route that is more than 1/2 inch away from the "box" around the pins in the net. This maxdeviation can be changed. It is called **maxdetour**.



To change it point to MAX DETOUR and enter f.x. *10inch*.

Then try to route the nets again, using the route GRP and Sel.

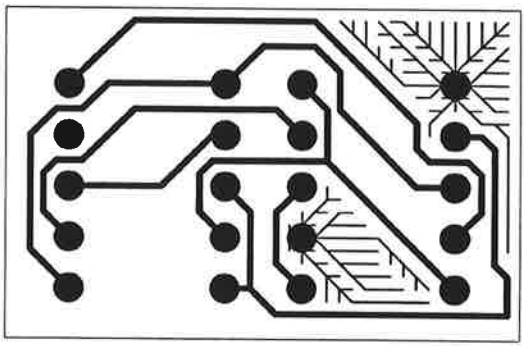
Even when you allow this big detour in the routes made, the router still misses some nets. Select them with SEL MISS.

These net can not be routed because some other tracks are completely blocking for any more connection. To finish the net you will have to move these blocking tracks or vias.

The router will help you find these tracks and allow you to move them.

Now - point to CHK+SKIP if OK which will check the first of the selected nets, and if it is completed unselect it and proceed with the next. In this case it will show the rubberbands for the missing connection. Point to zoo in the topmenu, and zoom "backwards". Then Supermax E-CAD IPL will blank the screen and display the net you are going to finish. To the question **leftlo** just press the green button on mouse or the spacebar.

Point to WAVEroute from and to one of the pads in the net. Supermax E-CAD IPL will now show you where a connection can be found, by displaying the "wave".



wave from one end

wave from other end

When the WAVEroute from is finished you repeat it by pointing to the other parts of the net. After doing this you can easily see where the two (or more) waves are closest together. This area is normally the best area to "clean-up" in order to complete the connection.

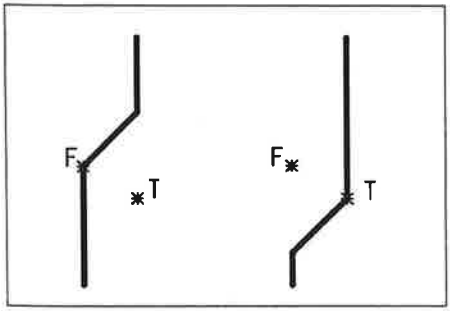
Point to route FROMTO and point to a pad in one part of the net and to one of the lines in the wave close to the "good" area. Be careful that you point to a line with the same color as your current lay. Then the router will make a connection from one part of the net and ending in a "blind" track or via close to the difficult area.

Do the same from the other part of the net.

Now zoom into the problematic area, and highlight the connection with the command SHOW LAST/GRP.

Then you can use the commands MOVE conn, CHLAY/MOVE, JUMP conn and PUSH conn to move the blocking connection away, in order to create a "channel" for the new connection. These commands all have on-line design-rule checking, so you can not make any errors.

MOVE conn: (from F to T)

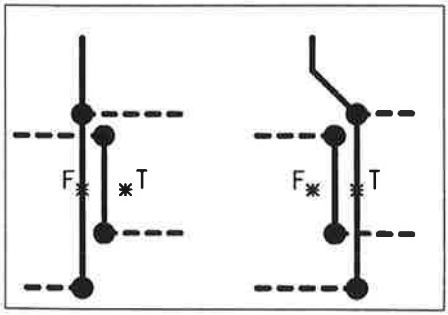


move corner

or

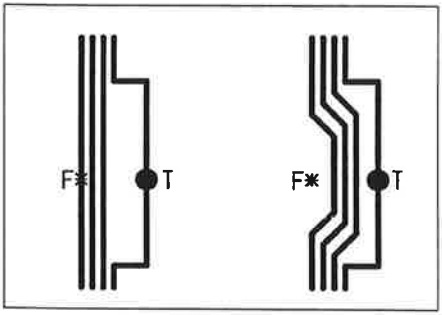
make corner

JUMP conn: (from F to T)



move segment

PUSH conn: (from F to T)



push segment

When you think that there is amble space for the new connection you can use route GRP and point to Nxt in the topmenu. Or you can use MOVE conn to "drag" the connections together.

Then continue with CHK+SKIP if OK and finish the rest of the missing nets.

These steps you have been through now can be automated and the RIPROUTE command in the RouteSub menu will do exactly this. First it finds the place where the waves are closest and then it tries to move the blocking connections. Because it misses the human overview it will do a lot of unsuccessful moving around with tracks, and it is therefore very slow. But it can be used to run overnight. You will learn to do that in the chapter with "Runfiles".

For fun (and coffee-break) you could try to use the riprouter. Type cuisine7 again and change the MAX DETOUR to *10inch* and then start the RIPROUTE directly.

8. Checking

> Here you will learn how to check that all your designrules are met correctly.

To demonstrate the checking commands, type `cuisine8` and Supermax E-CAD IPL will fetch a board with some missing connections, some shortcircuits and some clearance violations.

You start by checking that the physical connections corresponds to the netlist.

Go to the `CheckMenu` from the `BaseMenu`.

Point to `SEL CHECKERR`. Then Supermax E-CAD IPL will select all nets with shortcircuits or with missing connections.

To step systematically trough the incorrect nets, you should use the function `CHK+SKIP if OK`. It will check the first of the selected nets, and mark it, if there is an error.

If the net is missing some connections Supermax E-CAD IPL will show a rubberband between the unconnected parts and all the pins in the selected group. Then you can use the methods from the previous chapter, `Router Help` to finish the net. Sometimes it can easily done with `move conn`.

When you think the net is finished, you point to `CHK+SKIP if OK` again. If you did remove all the errors in this net the next net will be checked.

If there is a shortcircuit from the net to another net, Supermax E-CAD IPL will show all the tracksegments leading to the illegal component pins. Using the 'zoom backwards' (**leftlo = righthi**) or the `erasescreen` to blank the screen, it is very easy to see the error.

Normally you can remove the shortcircuit with the `RM between` command.

Next you should check that the minimums clearance distances are not violated.

Point to SEL CLEARERR. Like the checkerrors, you should step through the nets with errors. To do this use CLCHK+SKIP and again the 'zoom backwards' or the erasescreen to blank the screen, it is very handy to see the errors.

The clearance errors can be corrected with the moveconn, if it is possible to move the connections apart. If not, you might have to change the linewidth on the track. To do that you can set a smaller trackshape with the tra in the topmenu. Use the command RM between to remove the offending track segment and addtrack to insert a new segment with the smaller width.

Or you could use the CHTRA sha GRP in the EditMenu. It will only change the shape of the track if there is room for the replacement.

9. The final finish

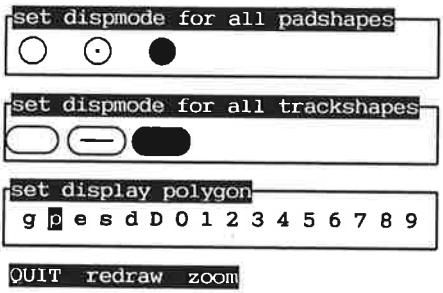
> How to make the layout most **pleasing** from an **artistic** viewpoint.

Type `cuisine9` to get an example, then go to the `BeautyMenu` by pointing to `-> BEAUTIFY` in the `BaseMenu`.

First you can get a list of used tracksegments by pointing to the command `no of TRA/VIA`. Then use the command `BEAUTIFY`. Press `0` and `Max` in the topmenu to indicate the total PCB. It will try to straighten any bended tracks and try to remove any 90 degrees angles. You can list the reduction of tracksegments by using the `no of TRA/VIA` again.

To make it easier to spot any "ugly" connections it is best just to display one single layer at a time. Point to `lay` in the topmenu and remove display of layer 1 by pointing in the field disp for layer 1 (solder side). Then you point to `redraw` in the bottom line and you will only see the pads/vias and the tracks on layer 0 (component side).

To make the display even more alike the final artwork you can point to `dsp` in the topmenu:



Here you can change the way that tracks and via are displayed on the screen. Point to the completely filled pad and the completely filled track. Next you point to `p` in the line saying `disppol`.

This will make the Supermax E-CAD IPL display the information used to create the photoplotterdata. Now point to redraw again.

Now you can do some manual editing of the tracks. Using the command MOVE conn you can move a track around. By pointing to the corners of the connections it is easy to control the movement.

Repeat the process for the solder side.

10. Texts and the Silkscreen

> Here you will learn to place texts and you will get some skills in editing the silkscreen.

Now you should place some text on the layout.

Go to the TextMenu by pointing to -> TEXT in the BaseMenu. Point to the add TEXTstd. Then you can place a text:

at:
point where the lowerleft corner of the text should be.

text:
here you enter the actual text, f.x. "*my 1. PCB*"

size:
the height of the text, f.x. *2mm*.

textsha:
the shape of the text, f.x. *silk*

When you have placed it you can move and rotate it with the commands mov TEXT and turn TEXT. You can even mirror it with the mir TEXT command.

You can also use the r macro to rotate the text, while you are moving it with mov TEXT.

To place a text in any angle or with another width/height relationship or if you want to slant the text, you can use the add TEXT which will ask for all these special parameters with Popup menus.

You can change the size and direction of a text by pointing to edit TEXTparms and type the new values.

To delete a text use the rm TEXT command.

When you have placed some texts, you should also give the silkscreen a final touch.

To display the silkscreen you point to lay in the topmenu and point to none in the topline, then point to the dispfield for layer silk (layer 11) and perhaps layer pads (layer 255), followed by redraw on the bottom line.

Now you have the functions mov TEXT, turn TEXT and align TEXT to move the component texts around.

To check for texts overlapping each other, you can use the Textoverlap.

If you do not want a text to appear on the artwork you can do this by "hiding" it with the command make INVISB. This will not remove the text completely but only make it invisible, so you can always get it back by making it visible again with the command make VISIB. In fact you can not remove a component text completely since it holds the information about the component name, article and device.

To see the text you have made invisible you point to the command disp INVISIB.

For component text you have another usefull tool: the **Textdir**. When you rotate components, you still want the component texts to be seen from the same edge of the PCB. The textdir specify from which edge to se the texts. If you specify *-1* you will be able to rotate the texts as you like.

Last you may want to add some mechanical measurements around the borderline. Change the display back so you can see all the layers and set the layer to the lay for measuments (f.x. *60*). Then use the command add MEASURE:

from:
the first point to measure the distance between. Here point to the lower left cornermark.

to:
the next point to measure. Point to the lower right corner-mark.

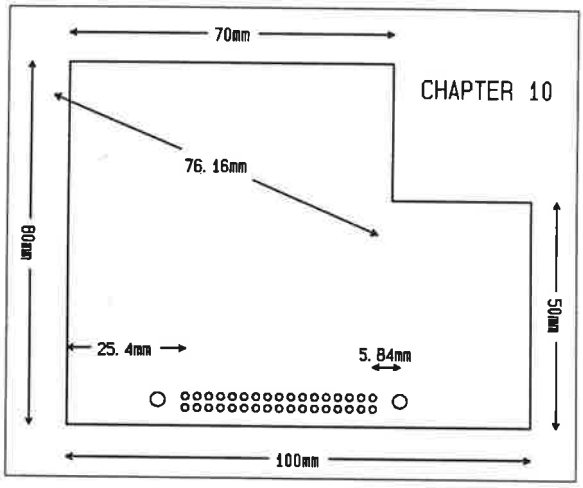
at:
this is the placement of the measurementtext. In this case point a little lower than the borderline.

tol:
an extra text appended to the measurement text. In this case it could be *+-1%*.

way:
the placement of the text. Use *m* to place the text in the middle of the measurementarrows.

openarrow:
the look of the arrow. Answer *y*.
size:
the size of the measument text. Use *3mm*.
textsha:
the shape of the text, f.x. *silk*

More measurements may be made.



11. The End

> The chapter wherein you will meet the MakePost tools and create data for photoplotting and drilling.

Go to the IOMenu by pointing to -> I/O in the BaseMenu.

First you should save the PCB on the harddisk. Point to save PCB and Supermax E-CAD IPL will ask you for the name of the file wherein the PCB will be stored. In this case it could be *MYPCB*. Next it will ask you if it should overwrite the **file** if it already exists. In this case you should answer *n*. If there already exists a PCB called *MYPCB*, Supermax E-CAD IPL will give the message *File error*. In that case you must use another name, or destroy the old one by answering **overwrite** with *y*.

To get an overview of the files on the harddisk you can point to list PCBs.

When the PCB has been saved you point -> POST to move to the PostMenu and then you point to MAKE POST.

Then you will see a small popupmenu:

> *twolay fourlay .....*

Here you should select *twolay*, because it is a two layer pcb.

Next you will be asked if you want edit the standard. In this case answer *n*.

Now the Supermax E-CAD IPL will start the postprocessing program called makepost. It could also be started from a normal alphanumeric terminal.

The makepost program will "remember" what outputdata you want for a specific PCB, so generating outputdata after revisions of the board, will be extremely easy. This information will be saved in a file with the same name as the PCB but with a .d appended.

In this case the standard specifies:

```
1 plot of component side
1 plot of solder side
1 plot of soldermask/isolation mask (Not for SMT)
1 plot of silkscreen

1 file with NC-drilling data

All the photoplots will be placed on as few films as
possible. The films will be chosen as small as possible.

The photoplotterdata will be in EIA gerberformat, using the
standard slide "37".
```

The makepost will place output data in files with the same name as the pcb, but with a number appended. So **MYPCB** will be **MYPCB.*1*** (**MYPCB.*2*** ..). The photoplotter data are placed in a directory called **/usr/ipl/gerber** and NC data are placed in **/usr/ipl/drill**.

When the makepost is finished you can inspect the output made for the photoplotter. Point to `setSIZE+erase` and enter the size of your film. Then point to `plot GERfile`:

file:
the name of the file containing data for the photoplotter. In this case it will be *MYPCB.1*.

getab:
the table describing the apertures on the photoplotter and the format of the data. In this case it is *getab37*.

Now Supermax E-CAD IPL previews the gerber data on the screen, and you can check that it looks correct.

You can also make paperplots of you design with the commands `make CANONplot` or `make HPplot`. These commands will make a plot of the job loaded into the Supermax E-CAD IPL system, or just a window of them.

There is a huge number of different output possibilities, but it is not in the scope of this manual to deal with them.

12. Lay assignments

In this chapter you will learn about the use of the different lays.

The Supermax E-CAD IPL system uses 256 different layers to store all the informations used in a PCB-layout. All these layers are fully userdefined as to what information the lay is used to describe. But it will be very wise to use the default lay assignments the Supermax E-CAD IPL proposes. Every lay can be given a name and some special attributes distinguishing between electric and graphic layers.

The default lay assignment Supermax E-CAD IPL uses is:

0 layer zero is called **comp** and is use for tracks on the component side. Also by default all not mirrored components will be placed on this lay. It is an electric lay, meaning that lines placed on this lay will be tracked by the Supermax E-CAD IPL to form physical connections. It also means that the autorouters will treat lines as obstructions so they will not cross over each other.

1 layer one is called **solder** and is used for tracks on the solder side. Also mirrored components will be placed on this layer. It is like layer 0 electric.

2-7 lays two until seven can be used for internal layers. If so, they will also be electric.

8 is used for extra information regarding the **soldermask**. It is not electric and is therefore treated purely as a graphic lay.

9 contains texts for the **isolationmask**.

10 is called **outline** and is used for the component outline which is the graphics showing the sizes of the components.

11 is called **compname** and is used for the compname with is plotted on the pcb. Because of that it have to be written with a fat track.

12 is called **silkscreen** and is used for the silkscreen on the solderside. Silkscreen is plotted on the pcb, and have to be written with a fat track.

13 is called **compdraw** and contain name and symbol of component used for paper drawing of component placement.

14 is called **article** and is used to hold article names. This can be very useful for creating mounting drawings, where you can plot layer 10 and layer 14 together, given a drawing with the placement and the stocknumbers of the components.

15 is called **device** and contains the devicetexts.

17 is called **block** and is used for displaying boxes around components that are put into blocks.

18 is called **height** and specifying the height of the component.

19 is called **dotsilk** and is a dotted version of lay silk screen used for silkscreen on pcb component side referring to component on pcb solder side. IPL will automatically move the dotsilk between layer 19 and 29 when you mirror a component.

20 is called **soloutl** and is outline for component on the pcb solder side. Supermax E-CAD IPL will automatically move the outline between layer 10 and 20 when you mirror a component.

21 is called **solname** and is compname on the pcb, for component on the pcb solder side. IPL will automatically move the com pname between layer 11 and 21 when you mirror a component.

22 is called **solsilk** and is the silkscreen on the pcb, for component on the pcb solder side. IPL will automatically move the silkscreen between layer 12 and 22 when you mirror a component.

23 is called **soldraw** and is the compdraw for component on the pcb solder side. IPL will automatically move the silkscreen between layer 13 and 23 when you mirror a component.

24-28 not assigned - yet.

29 is called **soldot** and is a dotted version of lay silkscreen used for silkscreen on pcb component side referring to comp onent on pcb solder side.

Information about special components on pcb component side.

30 is called **pastemask** and is additional paste mask information on pcb component side.

31 is called **tpvia** and is vias in testpoints used for paper plot of testpoint placement on pcb component side.

32 is called **carbon** and is information about carbon placement for carbon switches on pcb component side.

33 is called **gold** and is information used for production of gold plated connectores on pcb components side.

Information about special components on pcb solder side.

41 is called **soltpvia** and is vias in testpoints used for paper plot of testpoint placement on pcb solder side.

42 is called **solcarbon** and is information about carbon placement for carbon switches on pcb solder side.

43 is called **solgold** and is information used for production of gold plated connectores on pcb solder side.

Various additional information about special components on pcb component side.

50 is called **gluemask** and is information about placement of glue points for SMD components on pcb components side.

Various additional information about special components on pcb solder side.

60 is called **solglue** and is information about placement of glue points for SMD solder on pcb components side.

Miscellaneous graphic information independent of pcb side.

70 is called **corner** and is special symbols to indicate pcb size & shape.

71 is called **measure** and is mechanical measurements of pcb, used for paper drawing.

72 is called **drilltext** and is identificational text on drillfilm plot.

Additional electrical information independent of pcb side.

80 is called **innerboard** and is boarderlines for plane innerlays, to be plotted together with the relevant routerlays.

90-254
all these layers are free for any userdefined purposes It could be information for bareboard and incircuit testers. Or extra graphics for service documentation. It could be used to store mechanical drawings. Or information for gluespots for SMD insertion.

255 is called **pad** and is used for normal throughhole componentpins and normal vias. It is the ONLY layer that can not be freely assigned since any information on this lay will be treated as though it belongs to all the other lays. It should always be electric.

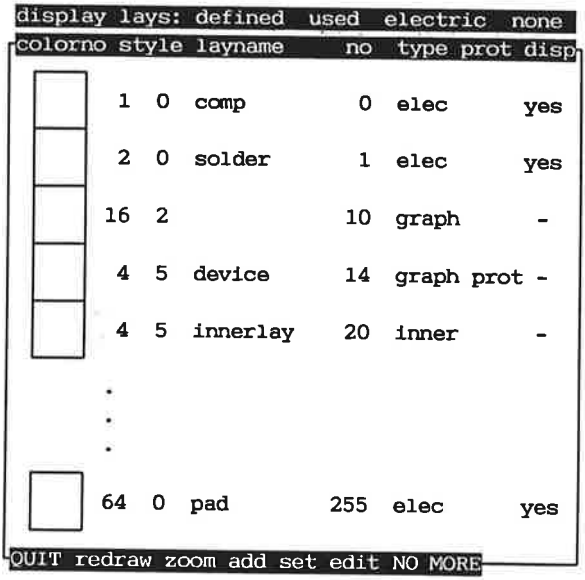
You will notice that there is no lays assigned for apertures for soldermask or drillfilm. Neither do Supermax E-CAD IPL assign lays for solderpaste for SMD components. This is because Supermax E-CAD IPL will store this information in the shape themselves (That is: Supermax E-CAD IPL is not using padstacks, but a much more flexible system. It is described in the chapter Shapes).

When mirroring components or windows, Supermax E-CAD IPL will automatically change the lays. This will be done according to the table called mirmap which is normally equal to:

mirmap=0;1/8;9/10;20/11;21/12;22/13;23/19;29/30;40/31;41/
32;42/33;43/50;60/51;61

meaning that layers 0 and 1 will swop and layers 8 and 9 will swop and so on, when mirroring (or placing components in directions *0m-3m*).

To inspect or change the layassignments, use the laysetup in the topmenu:



Only the lays actually 'used' are included in the laysetup. Here 'used' is a lay given a name, having electrical status or a lay with something displayable on it. To add a new lay point field add and enter an optional name.

13. Shapes

This chapter describes how trackwidths and pad/via sizes are defined. It is a slightly theoretical chapter but very important.

In Supermax E-CAD IPL a shape is the description of a track or a pad/via. It is a collection of different informations that can be extracted by different commands. These informations are for photoplotter, soldermask, drillsize, minimal clearance and graphic appearance on the display plus several userdefined informations like masks for solderpaste and goldplating.

Each of these informations are kept in what is known as a **polygon**.

PADSHAPE:

For a padshape the informations are:

Ppgon	**plotter polygon** used for photo and pen-plotting.
Spgon	**solder polygon** for generating soldermasks.
Gpgon	**graphic polygon** describing the representation on the graphic display terminal.
Epgon	**electric polygon** to define the "electric" size of the pad.
Dpgon	**drill polygon** to define the "drillsize" on a padmaster film.
Opgon	normaly used as **paste mask** for SMT components.
1-9pgon	the polygons 1 to 9 is for user defined purposes.

plus:

size	the drilling **hole size.**
plated	can be *yes* or *no*. Whether the hole should be **plated** or not.
tolerance	the **drill tolerance** string.
dispmode	the way the pad is filled graphically on the screen (just the edge, with a center dot or completely filled)

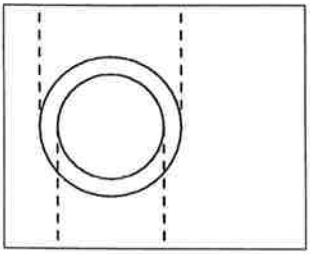
The *plotter* and *solder* polygons are used by the **makepost** and by all the plotting utilities. The information is simply extracted by specifying either *p* or *s* for the plotting polygon. The same goes for all the userdefined polygons.

The *graphical* polygon is used mostly for displaying on the screen. Normally you will define the form different for componentpin and vias, so they are very easy to distinguish on the display. (So you avoid trying to move or delete a component pin, because it is so easy to see what can and what can not be moved). That is why -in the library- all component pins are displayed as squares, even when they are plotted as circles. You could also use it to display the pin 1's in a special manner.

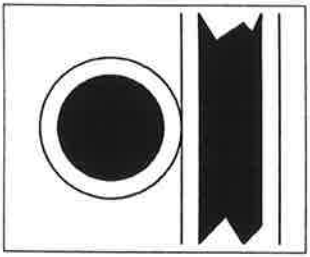
The *electrical* polygon is used for define the minimal clearance distance. In Supermax E-CAD IPL the clearance check commands (the routers, move and jump connection, compact and clearcheck) will allow two electric polygons just to touch but never to overlap. So by specifying the electric polygon bigger than the plotter polygon, the difference will be the clearance distance.

Example of a shape with *plotter*- and *electrical* polygon:

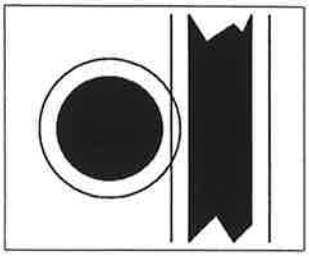
electrical polygon



plotter polygon



Example of a pad and a track placed very close together without errors.

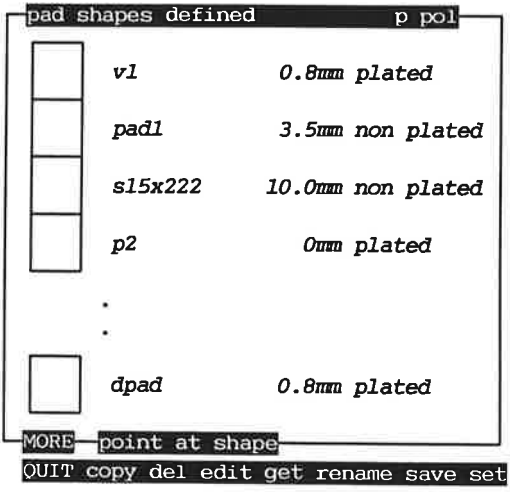


Example of a pad and a track
placed too close
to each other.

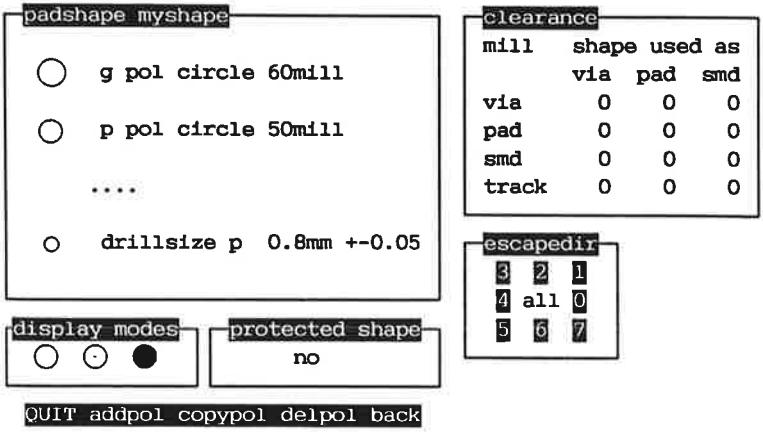
ERROR.

As an example you should create a circular pad with the:
size *70mill*
annular ring to solderresist opening *15mill*
drillsize of *1.0mm, plated*
minimal clearance of *10mill*
square graphic appearance

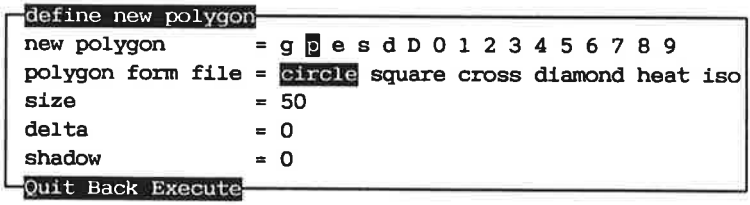
a) To create a padshape point to pad in the topmenu. Then a display like this will be shown:



b) Point to copy in the bottom line and point to the shape to make a copy from (e.x. *v1*). Specify your new shapename (e.x. *myshape*). This shape is now shown alone for further alterations.



c) Point to the p-pol line. A submenu will appear in the lower part of the display.



d) The polygon p and polfile circle should highlight. Point to the size. Alter it to *65mill*. Delta & shadow should be *0*. Point to execute to make the alteration that now will appear in the shape overview.

e) Point to the s-pol line, point to size in the submenu, alter it to *80* (*65* + *15*) and Execute the alteration of the solderresist.

f) Point to the drill size (0.8mm) in the topmenu and alter it to *1.0mm*. If 'np' ' is written to the right of the drillsize then point to it to make it plated, (the displayed *np* will change to *p*). Point to the drill tolerance (+-0.05) and alter it to +-0.06mm.

g) Point to the e-pol line, point to size in the submenu, alter it to *75* (*65* + *10*) and Execute.

h) Point to the g-pol line, square in the the submenu, the size is set to *70* and Execute. Point to QUIT to go back to

Supermax E-CAD IPL.

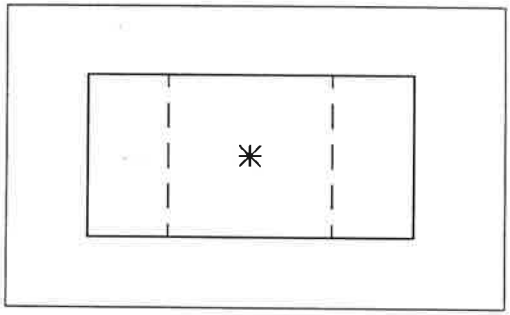
PADSHAPE EXAMPLE 2:

For Surface Mount Technology, SMT, you want to create a padshape with the following measures:

rectangular 3mm long 1mm wide.
10mill clearance
soldershape *equal to* plottershape (wet solder resist)

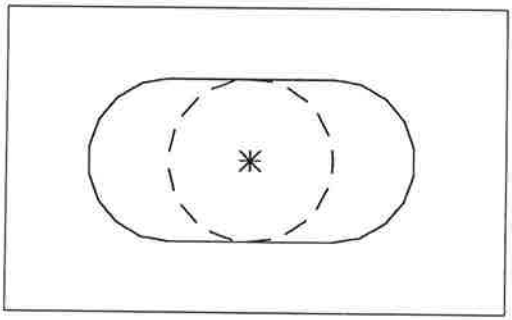
A shape is made longer in one direction by specifying a delta value in the submenu.

d) Set 'polygon form file' to *square*.
e) Set 'size' to *1mm*.
f) Set 'delta' to *2mm* (*3mm - 1mm*).
g) Set shadow to *0*.



a pulled square

delta/2 extra in each end

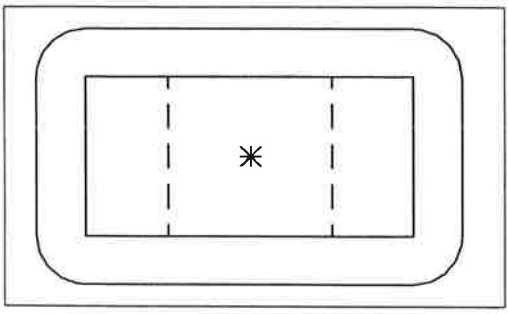


a pulled circle

delta/2 extra in each end

The clearance (electrical polygon) on a elongated shape is done with the shadow parameter.

d) Set 'polygon form file' to *square*. (like above)
e) Set 'size' to *1mm*. (like above)
f) Set 'delta' to *2mm* (*3mm - 1mm*). (like above)
g) Set shadow to *5mill*. (*10mi / 2*). This will make a figure that follow the plotterpolygon but *5mill* away.



Electrical polygon is *10mill* larger that the plotter polygon.

When making connections to a elongated pad, it should normally only be entered in the ends. This is done by pointing to the escapedir field. A direction is legal if it is highlighted. A value is toggled by pointing to the propper direction. The field all will have effect on all directions. Direction *0* (to the right) and *4* (to the left) should be the only highlighted fields. For a SMT it may be convinient to see the center.

TRACKSHAPE:

For a trackshape the informations are:

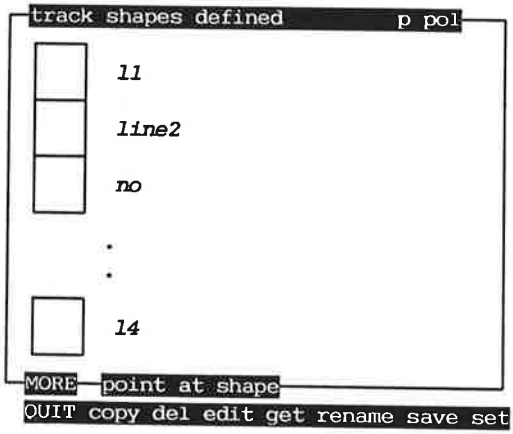
Ppgon **Plotter polygon** for photo- and pen-plotting.
Spgon **Solder polygon** for generating soldermasks (very special).
Gpgon **Graphic polygon** describing the representation on the graphic display terminal.
Epgon **Electric polygon** to define the "electric" size of the track.

Opgon normaly used as **paste mask** for SMT components.
1-9pgon the polygons **0** to **9** is for user defined purposes.
plus:
dispmode the way the track is shown graphically on the screen (square or circle and just the edge, with a center line or completely filled).

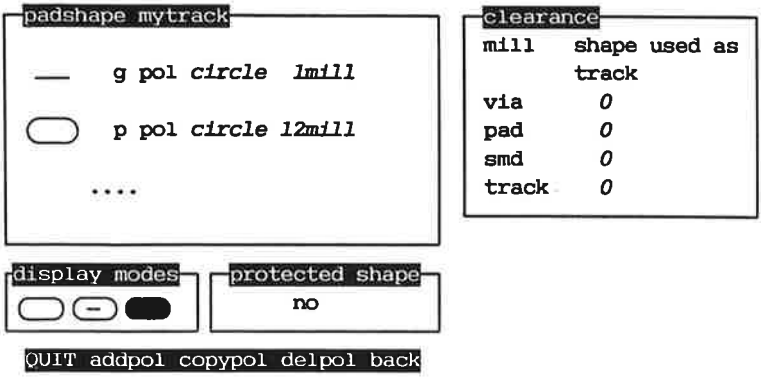
The use and definitions of the trackshape is equal to padshapes.

As an example you should try to define a circular track *8mill* wide, *10mill* clearance. Graphically it should be shown in the *correct size*.

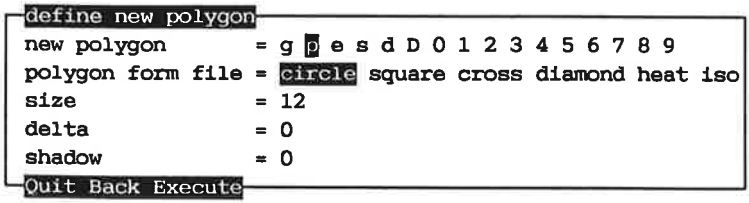
a) To create a trackshape point to tra in the topmenu. Then a display like this will be shown:



b) Point to copy in the bottom line and point to the shape to make a copy from (e.x. *l1*). Specify your new shapename (e.x. *mytrack*). This shape is now shown alone for further alterations.



c) Point to the p-pol line. A submenu will appear in the lower part of the display.



d) The polygon p will highlight. As both plotter- and graphical polygon are to be *8 mill* the 'new polygon' may be set to g and p by writting the two letters followed by Return.

e) The polygon form file should always be *circle* when a trackshape is handled.

f) Point to the size. Alter it to *8mill*. Delta & shadow should be *0*. Point to execute to make the alteration that now will appear in the shape overview.

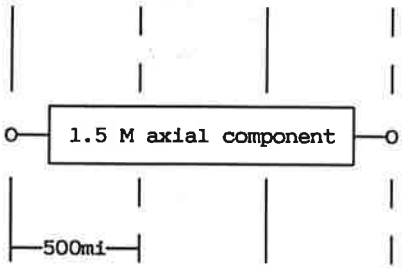
g) Point to the e-pol line, point to size in the submenu, alter it to *18* (*8 + 10*) and Execute. Point to QUIT to go back to Supermax E-CAD IPL.

14. Creating Library

Here you will see how the library is created or modified.

To create a new component in the library you will always use an existing one and modify it.

As an example create the following component:



Go to the LibraryMenu. Before you can create a library component the Supermax E-CAD IPL should be reset.

Point to Reset and answer **fast** with *y*.

Point to get TYPE and type **mis/ref**. This will fetch the component type called **ref** (a component with no pins) in the subdirectory named **mis**.
If you do not type anything or you type an unknown name, the Supermax E-CAD IPL will show you a complete list of the 20 subdirectorys in the library. When you activate the mouse bottom on a subdirectory, you will get the complete list of the subdirectory.

The component will be placed in the middle of the display. To move it to another place use the command move ALL to and point to the pad followed by some place in the middle of the display.

Now you are ready to modify this component. First you should

select a proper shape for the component pins you will use. You learned how to create it in the preceding chapter Shapes. Now you select it by pointing to pad in the topmenu and point to set in the bottom line where after you point to the desired shape.

You can select and display a grid with grd in the topmenu. A grid of 50mill will be very good.

Point to add PAD(s) and a little popup menu will appear:

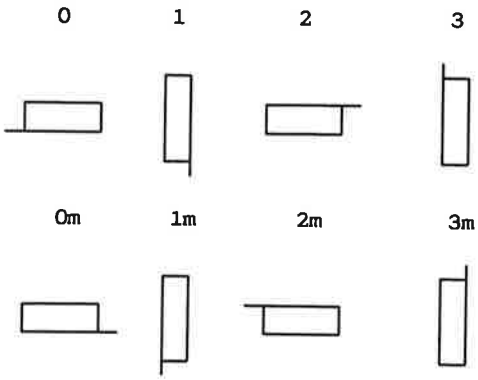
Point to *1* to indicate one pad in a time and the system will ask the following:

at:
the placement of the pad. Point to some gridpoint.

pinno:
the pinnumber of the pad. A pinnumber can contain one letter and several digits. The letter can be to the left of the digits or to the right. A pinnumber must contain at least one digit, thus a pinnumber consisting of one or more letter, and no digits is not allowed. Examples of allowable pinnumbers are 1, 2, 3, 1a, 2a, 3a, a1, a2, a3.

single:
y if the pin is a surface mount pin or a testpad, but *n* for all normal through plated holes.

paddir:
the rotation of the pad. It can be 0,1,2,3 and 0m,1m,2m,3m. This has only meaning for nonsymmetrical shapes f.x. rectangles.



In this case *0* will be the answer.

Supermax E-CAD IPL will repeat the add PAD(s) command so:

at:

the placement of the next pad. Point to a gridpoint 150mill to the right of the previous pad.

Supermax E-CAD IPL has automatically incremented the pinnumber and used the same paddir and single value.

Next you should create the outline placed on layer **outline**. Point to tra in the topmenu and select the shape **outline**. Point to out in the topmenu to select layer **outline** (this could also be done by using lay in the topmenu and pointing to set and the wanted lay).

The outline is a rectangular figure showing the size of component. Point to add BOX and point to a gridpoint 1M to the lower left of the pin 1 and to another point 1M to upper right of pin 2.

To create the silkscreen you select layer **silkscreen** by pointing to sil in the topmenu. Point to tra in the topmenu and select the shape **silk12**. Then use the add TRACK to draw the silkscreen according to the simple and easy understanding drawing. The drawing shoud be inside the outline box, and must be outside the padshapes.

To create the compdraw you select layer **compdraw** by pointing to dra in topmenu. Point to tra in topmenu and select the shape **silk5**. Then use add TRACK to draw the compdraw according to drawing.

Next you should place the reference in pin 1 using move REF.

Then the texts should be placed correctly using mov TEXT.

The component height are to be written in the text on lay **height** by pointing to lay in topmenu and then pointing set and then pointing at the **height lay**. Redraw the screen by pointing red in topmenu, write edittext and point on the text named **height** and enter the component height.

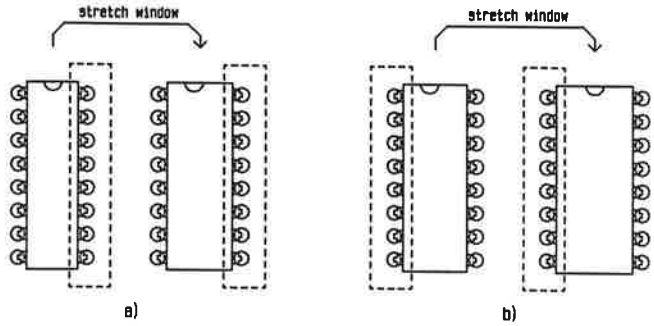
After placing the texts the component is finished and can be saved in the library with the commands save TYPE. It will ask you for the name of the (new) component and ask for overwrite if a component type with this name already exists.

To create a component that looks like an already existing component you can use the functions move PAD and stretch WIN.

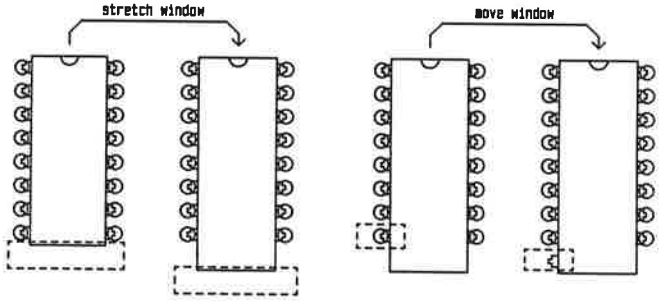
For example you can create a Dual In Line Package with 18 lead and 4 Moduls space between the to rows.

This one resembles a standard 16 pin packages a lot. Lets say that you want to use a DIP16 form Philips as reference.
Start wit Reset then get TYPE type ic/DIP16-phi.

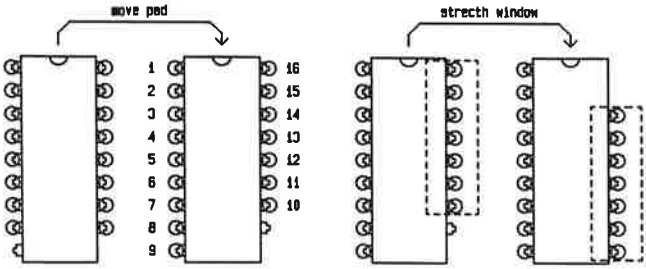
Now you can stretch the component 0.5 Module on etch side with the command stretch WIN



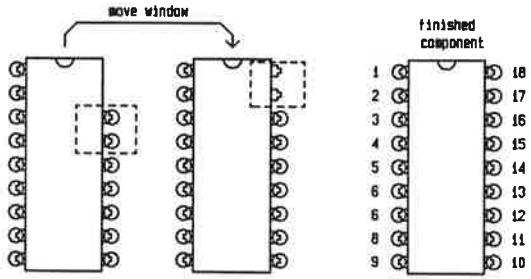
The bottom is now stretch 1 Module down with the command stretch WIN, to make space for the new leads.
Afterwards the drawing of the lead (in compdraw lay) in the left low corner of component are copied from the lead right above. This is done with **movewin ref 0,0 to 0,-100 rot 0 copy y comps n tracks y thislay n**



Now you move the pin 9 to the left row just under pin 8 with the command move PAD.
Next move the rest of the right row down 2 Moduls with the stretch WIN

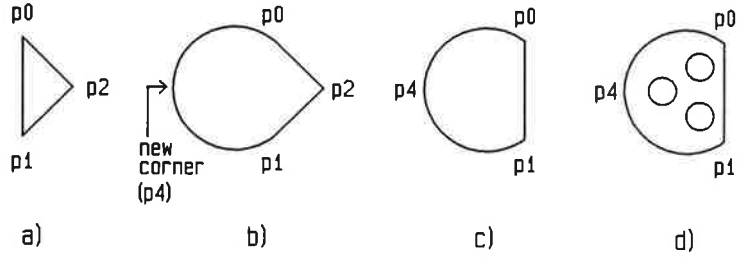


The drawing of the to last lead (in compdraw lay) in the right high corner of component are copied from the lead right belove. This is done with **movewindow.**
Last you should place the pins 17 and 18 with a shape similar to all the others. Point to gsh in the topmenu and and point to one of the pins. This will change your current shape to the one pointed to. Next you use the add PAD(s) to add the two extra pads. - and save the type with the name **DIP18-wide.**



When components with nonrectangular shape are made, an area may be used instead of traks in the outline lay.

Example were you make an transistor with a TO-92 house



First you make a reset and the you get the reference type with **gettype type mis/ref.**
Then you make the outline area with **defarea kind cmpsize dir 0 con 0blind y space 0 group [] areasha outline plotborder 10.**

Make the area as a triangle, where the first and second point (p0 and p1) makes a line similar to the flat side of the TO-92 house. (see fig. a. compared to fig. c.)

Make the round house with the add area CIR in the AreaMenu, use the p0 as refcorner and the back of the TO-92 house as the new corner p4. (see fig. b.)

Then remove p2 with rm area PNT in the AreaMenu. (see fig. c.)

Now the outline area are made, but the pads, the silkscreen and the compdraw are still missing. Starting with selecting used pads in the pad in the topmenu, and the placing them with add PAD(s) in the LibraryMenu.

Make the **silkscreen** as a arc and a track . First select a fat track as **silk12** by pointing tra in the topmenu, then select lay silkscreen (13) by pointing lay in the topmenu.
Then make the arc, by pointing add ARC in the LibraryMenu, and use the coordinate **from p0 through p4 to p1.** Finally a track are added from **p0** to **p1** by pointing add SINGLE tra in the EditMenu.

The **compdraw** are similar to the silkscreen, except the track shape should be **silk5.**

The **refpoint** must be placed at one of the pads with place REF in the LibraryMenu.

The comptexts should be placed near or on the component. (remember that the comptext on lay compname, must be placed outside the outline area)

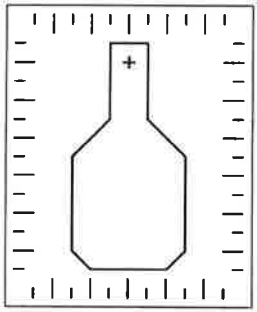
The componenttext on lay **heightlay** must be changed by selecting heightlay on `lay` in the `topmenu` and then point `alter TEXT` in the `TextMenu`.

15. Advanced shapes

This chapter show how to create a shape with completely userdefined form. It also describes some other ways to define clearance distances.

To create a shape with bizarre form or with nonsymmetrical center, you can draw the form as an AREA and then use it in the padsetup command (pad in the topmenu).

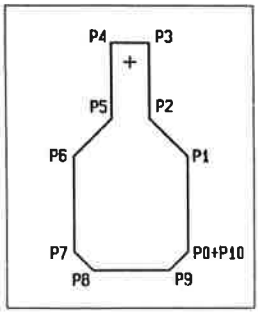
You start by creating a "bottle" for a SMT trimming potmeter. Select an appropriate grid (it do not have to be in scale 1-to-1). In this case you could select *25mill* and draw the grid in *50mill* (point to grd in the topmenu). The bottle looks like:



Go to the AreaMenu through the BaseMenu and point to the OBSTRAREA.

Point to cmp-keep-in. This will in fact create a component keep in area but the kind are of no importance in this case, because you will remove it when the shape has been created.

Now point to the vertices of the figure you want to create. You should point to them in an anticlockwise sequence:



After the area is finished you use the command AREA->POLFILE:

refcorner:
point to one of the vertices in the area.

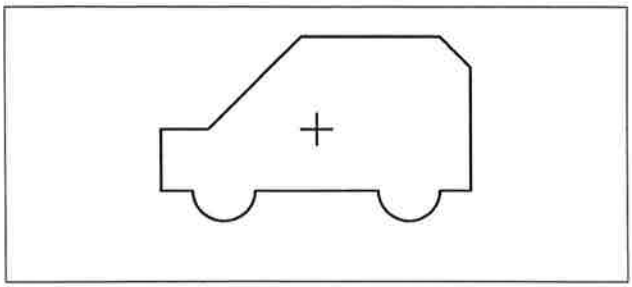
center:
point where you want the center.

polfile:
the name of a file (temporary) containing the shapedescription. In this case it could be */tmp/bottle*.

overwrite:
in this case *y*.

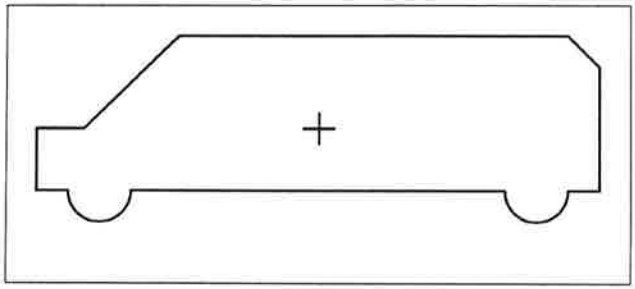
Now you have created a file describing the form of a the "bottle". Use the command rm AREA to remove the area you just created. Then you enter the pad in the topmenu, do the copy of a shape and enter the submenu. Here you can create/change the polygons, but instead of using one of the predefined "polfiles", you enter the name */tmp/bottle*. Now you will see that the polygon is changed to the bottleform.

Another example could be the "car" footprint:



To create this shape you start by defining the rectangle as an area, using the add CMP-K-IN command again. Then you go back to the AreaMenu and enter the AreaEdit by pointing to -> EDITAREA. Here you use the commands to alter the area corners/arcs. Go back to the AreaMenu and use the AREA->POL to save this new form as a polfile f.x. */tmp/car*

When this "car" polygon is used with a delta value a "bus" shape is made:



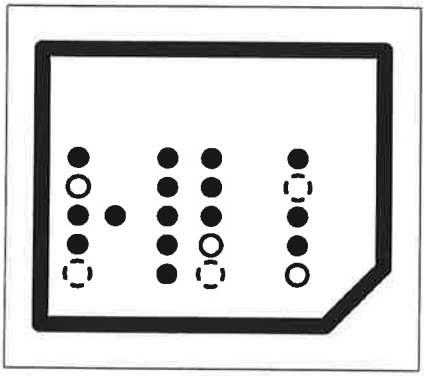
16. Power Innerlayers

> In this chapter you will see how makepost can generate standard innerlayers automatically.

Start by running the command `cuisine16`, which will get a board without any connections on the *GND* and *+5V* groups.

To see that the powernets are not-routed you can go to the `RouteMenu` and use the command `sel MISS`, which will give the message *2 selected*. Then you can point to `MARK/RBM GRP` followed by pointing to `sel` in the topmenu. This will mark the 2 selected nets. (You could also list the netnames with the command `list WLnames` in the `WlMenu`).

For an powerplane you should create a negative plot of the lay. Pads connected to the powerplane should have a heatrelief and pads not connected should be isolated.



In this picture are shown heatrelieved pads, and two different ways of isolating a pad from the plane.

First you must tell Supermax E-CAD IPL that this is a 4-layer pcb, with two powerplanes. You do that with the laysetup.

Point to `lay` in the topmenu, and point to `add`. Add laynumber 2 and give it a name ex. *gndplane*. Add lay 3 and call it *powerplane*.

Then point to the corresponding lines in the menu where it says **graph** and select the field with says **plane**.

Now Supermax E-CAD IPL know that these two lays are special powerplanes, that should be plotted "negative".

Go to the EditMenu. Select a wide track f.x. *14*. Select layer *2* as the current lay by pointing to 2 in the topmenu. Then use the command add TRACK to make the borderline, that will avoid the copper in the powerplane to reach the edge of the board.

Do the same with layer *3*.

Next you should tell Supermax E-CAD IPL that the two powernets are connected by a negative plane. To do this go to the WLsub menu through the WLMenu, and point to WLOPT, which will ask you:

group:
point to the *GND* or the *+5V* group.

A popup menu will now appear:

option	
plane	define planelay
negative	define negshape and planelay
tracksha	specify trackshape
viashape	specify viashape
remove	remove all options from group
userdef	other wloptions

in this case it should be *plane*. The planelay should for the *GND* group be *2*, because the lay it is connected by is layer *2*. For the *+5V* group it should be *3*.

Now try the command Check or the SEL miss again. Then you will get no errors.

Go to the IO menu and save the pcb (not FAST).

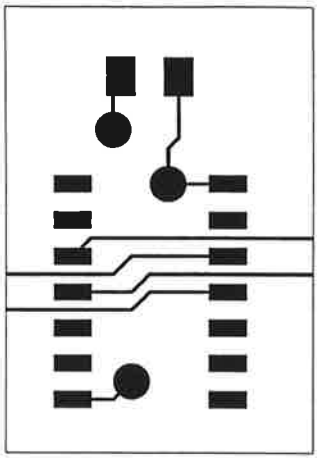
Then point to MAKEPOST in the POSTMENU and select the **4layerstd** standard. Now makepost will create:

```
1 plot of component side
1 plot of solder side
1 plot of ground plane
1 plot of powerplane
1 plot of silkscreen
1 plot of soldermask/isolation mask (Not for SMT)

1 file with NC-drilling data

All the photoplots will be placed on as few films as
possible. The films will be chosen as small as possible.

The photoplotterdata will be in EIA gerberformat, using the
standard slide "37".
```

To inspect the gerberdata you can use the commands setSIZE+erase (use *300mm*, *400mm*) and plot GERfile. The gerberfile name made by MAKEPOST will be the filename specified followed by .1 (2, 3.. for the following films if any).

SMT and powerplanes:

If your pcb contains some SMT components you should be sure that SMT-pins in the powernets are connected to a throughplated pin or a viahole.



You can try this by running cuisinel6smt, which has SMT components. It has already defined the planes and the wirelist options, but the nets are missing the vias.

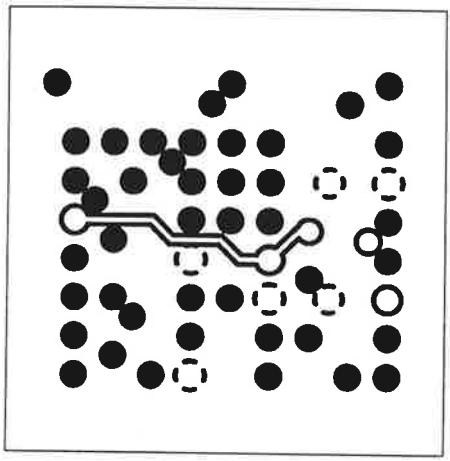
You can see this with the command check.

To make these special connections use the function ROUTE->VIA in the PowerRoute menu.

Signals on powerplanes:

When you have some signals with very special impedance restrictions or if you have a very dense board with a few missing connections, you might have to place some connections buried in the powerplanes.

These connections should be plotted in a special way since the lay is plotted negative.



To place connections in a powerplane you just make them with the commands **add TRACK** and **move CONN**. The check command will recognize the connections on the plane.

To tell Supermax E-CAD IPL and Makepost that these signals should be plotted in a special way, you must give them a **wloption** with option negative on the plane you have used. You must also specify the trackshape to be used by the "isolation".

You can check this by removing some of the nets in the example, and reconnect them on the planes. You don't have to make the entire connection on the same lay.

Signal innerlayer:

A signal innerlay is a standard lay like the componentside or solderside. It is plotted as a positive film.

To tell Supermax E-CAD IPL that a lay is a signal innerlay go to the laysetup menu (lay in the topmenu), and select the laytype to either 'elect' or 'inner' for the lays you want to use. Now IPL will know that these lays have electrical meaning, and the checkcommands will recognize tracks on these lays as connections.

To tell the autorouters that it can use these lays you must use the PCBMAX in the IOMenu, and change the number of layers. When you do this, remember that the routers will use the smallest numbered layers. That is, for a sixlayer board with 4 signal layers and 2 powerplanes, the layassignments should be:

```
pcbmax layers = 4
laytype comp or electric for lay 0
laytype solder or electric for lay 1
laytype inner or electric for lay 2
laytype inner or electric for lay 3
laytype plane for lay 4
laytype plane for lay 5
```

This is also the setup used in the MAKEPOST standard **6layerstd**.

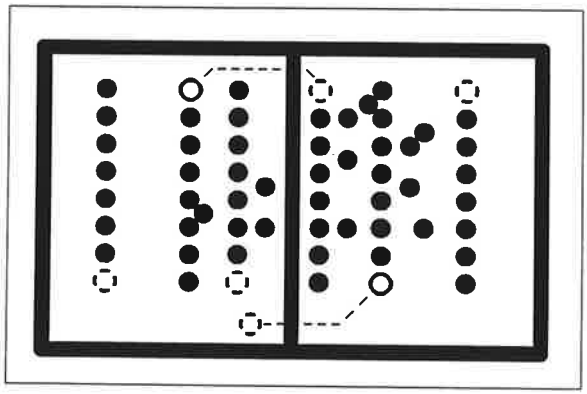
In some cases pads and vias not connected on the innerlayers must be suppressed in photoplot. This will be done if you have specified the lay as an 'inner'. If you specify the lay as an 'elect' the pads and vias are plotted in a normal way.

Multi group Powerplanes:

If several nets must be connected on the same powerplane, f.x. analog ground and digital ground, the different areas should be separated from each other by a track. Also it should be checked that pins from the other nets are really isolated instead of heat isolated.

To do this you should use the special area called "plane". They are placed on the powerplane, and connected to the respective nets. The check routines will verify that pins are connected in the correct way. They will also automatically create the isolating track separating the parts of the powerplane and automatically isolate pins in the "wrong" part of the plane.

Notice that no tracks should be allowed to cross the borderlines on the powerplane. It is legal to connect a pad outside the area to the area on an other lay. The connection can stop in either a pad or via inside the area. The via do not have to be connected.



17. Filled areas

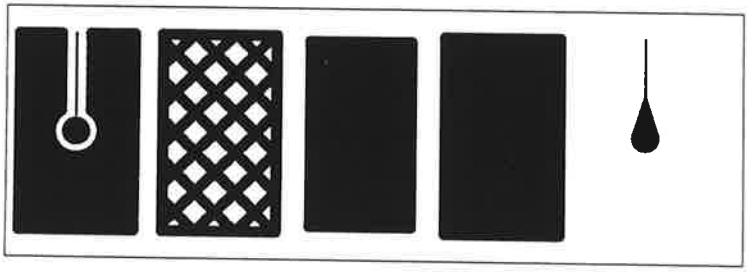
A small chapter for analog people, telling how to create filled areas on the signal layers. Other analog specialties as teardropping and circular tracks are explained.

To make an area filled with copper, you could use add TRACK to place a lot of tracks. But this will use a lot of memory, especially if the area is crosshatched, and it is very difficult to handle, if changes are to be made on it. Therefore a special concept is used to handle filled areas. It is called - yes, you guessed it - areas.

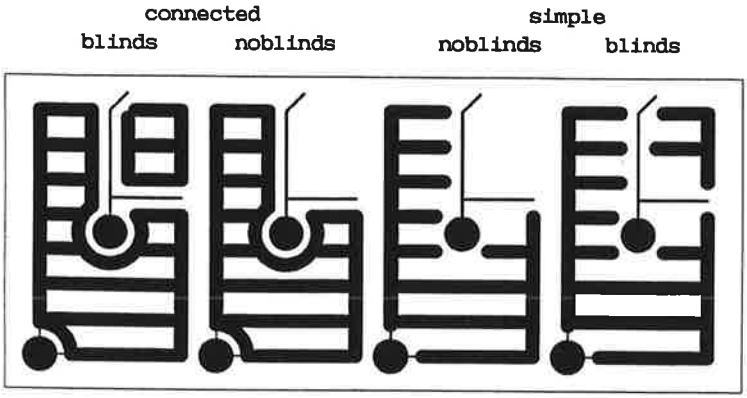
Go to the AreaMenu through the BaseMenu or by pressing A.

In this menu you can define different kinds of filled areas. Point to add HATCH area. It will show some popupmenus, first the kind of filling:

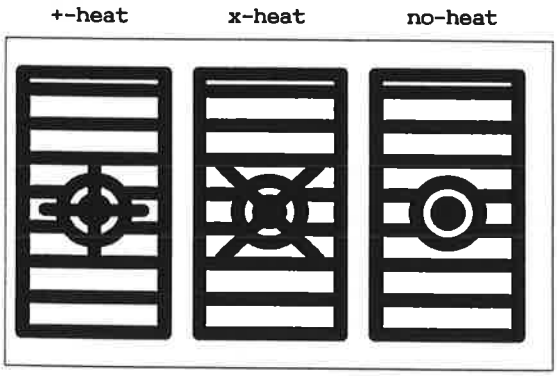
optha cross toedge solid tear



Next the way to treat obstructions inside the area:



And finally how to connect pads:



After these popup's it will ask for some additional parameters:

dir:
the direction of filling 0:—, 1:/, 2:| or 3:\

spacing:
the distance between filling lines. If you specify *0*, Supermax E-CAD IPL will automatically use a spacing that will make the lines overlap.

group
the group that should be connected to this area. If it is a coppersurface for heatsinks, it could be the mounting hole. If you specify the empty group "[]", be careful not to

specify 'noblind'. If you specify the group "*", the area will be completely filled, no matter what obstructions there are inside the area, and independently of the connect/blind and heat.

areasha:
the shape used for filling.

p0
the first corner of the areaborder. This is also the corner that will be the "last" in the border definition.

p1
the second corner. If you want to create a rectangular area, it suffices just to point to the two opposite corners, if the two corners are on the same horizontal line, you will get a circle.

p2
the third corner. In the case of a box or a circle, you should complete the area by pointing to **p0** again or pointing to end in the topmenu or by typing *#end*.

(p3 ..)
the following corners in the area. To complete the area type *#end* or point to end in the topmenu.

The points in an area are always "sorted" in an anticlockwise manner. You should not make a border that crosses it self, since it create holes in the border. If you want holes in the area you should use the joinarea command.

To change the area border you can use the command move corner and the commands add area CORN or add area CIR. The move corner will allow you to move one of the corners. The add area CORN will add one extra corner between the **refcorner** and the next corner on the borderline (remember the anticlockwise direction). The add area CIR will add an arc between the two corners, going through the new point.

If you want a circular corner, you can use the angle=>round, where you specify the radius of the curve.

Normally the area is shown only as a borderline. When the pcb is saved the actual fillpattern is calculated. This makes it easy to work with the connections inside the area.

To see how Supermax E-CAD IPL will make the filling, you can use the function install time and use the all from the popup.

Memory Considerations:

Although defarea is using much less memory than simply filling with tracks, it can use substantial amounts of memory. The factors controlling the memory usage are:

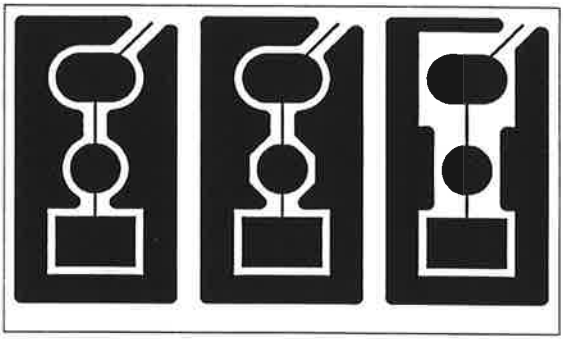
AREA SIZE: Since Supermax E-CAD IPL is installing one area at a time, when saving the pcb, a very big area could be divided into smaller ones. If the area is filled with "noblind", be sure that the group is well distributed in all the smaller areas.

SPACING: Since the number of fill-lines will depend on the spacing, you should use as wide a track as possible. The track should be so wide as to just pass in between the component pins. If the pin spacing varies a lot, it might be better to split the area in two areas with different areashapes. These areas may even overlap each other.

ARCSOLUTION: When using the the "connected" mode every circular pad/via is surrounded by a circle. This circle is made with the arcsolution, so giving an arcsolution of 10 degrees, you will have 36 lines round each pin, but using a solution of 1 degree will give you 360!!.

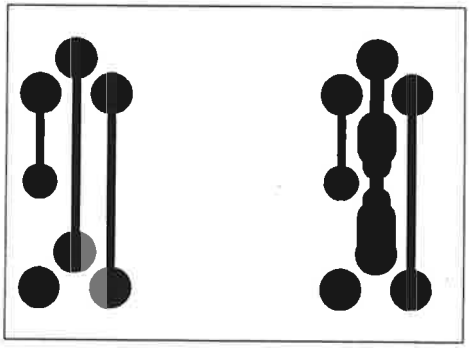
EDGETYPE: Instead of using arcs around pads/vias (edgetype 0), you can specify that circular pins should be surrounded by boxes (edgetype 1) or by octagons (edgetype 2).

To change the edgetype and the arcsolution use the command install mode or the arcsolution and edgetype in the AreaSub menu.



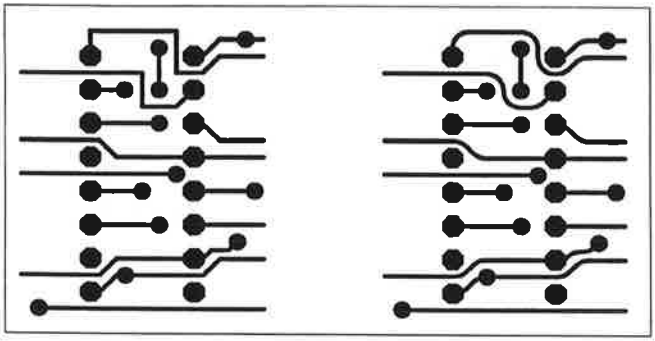
Necking:

For analog designs it is often desirable to make the tracks as wide as possible. This can be done with the CHTRA sha GRP command in the EditSub menu. It can be done several times to get the widest possible linewidth. To check how many lines there are left with the smaller width, the Show TRA shape command can be used.

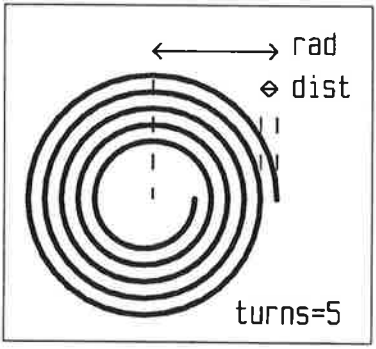


Rounded Tracks:

In the EditSub menu you have also the commands add ROUND, add auto ROUND, rm ROUND and rm auto ROUND. These commands make it possible to substitute a normal angled connection with a circular. This is done by inserting a lot of small tracksegments instead of the original connection. These small segments make it very difficult using the standard editing function move conn, jump conn,... Therefore the function add auto ROUND should be used as a kind of postprocessing. And the functions rm ROUND and rm auto ROUND should be used before any major alterations of the pcb.



Another command is the add SPIRAL. It will calculate and insert a spiral in the pcb calculated from the values radius, turns and distance.



18. Obstruction Areas

Here you learn another method to create borderdefinitions and making restricted areas in the middle of the board.

With the command add OBSTR area in the areamenu, you can create areas on the pcb where it is illegal to place viaholes, wherein all components must be placed or some other restrictions.

When you point to the add OBSTR area, it will show a popupmenu:

areakind	
via-keep-in	keep vias inside
cmp-keep-in	keep components inside
track-keep-in	keep tracks inside
via-keep-out	keep vias out
cmp-keep-out	keep components out
track-keep-out	keep tracks out
cmp-height	maximum component heights
plane	negative plane area
milling	miller route area

The height of a component is specified by adding a comptext on the height lay.
The plane option is used if more than one signal is to be made in the same negative layer (see the chapter inner-lay).
The milling area shows at which edge the PCB is to be cut.

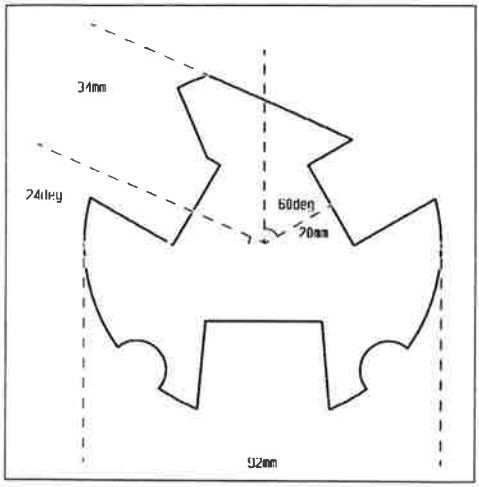
The keep-in areas are for definitions of the pcb borderlines. Here you would start by defining the component keep-in made directly from the mechanical drawing. It should be placed on layer 255. Next you can use the cpy OBSTR area to create track keep-in in a small distance from the borderline, and finally a via keep-in a little further away from the borderline. To make the copy a little smaller than the borderline you should specify a negative shadow.

The keep-out areas are for definitions of illegal areas, e.x. an area where there will be milled a big hole, or an area where

tracks under a component must be avoided because of metallic housing.

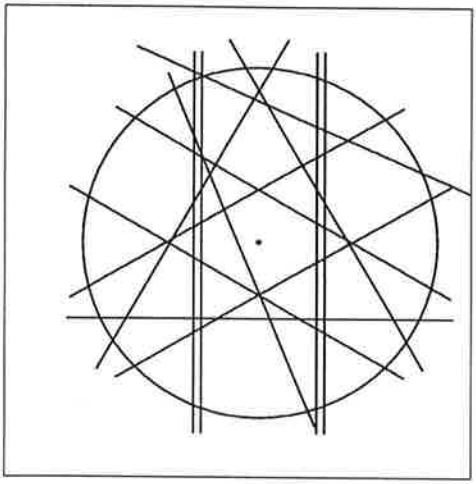
> Speed considerations: The restriction areas are very good to define precisely what restrictions to put on the placement and routing, but since the Supermax E-CAD IPL must check these restriction all the time, they will make the system a little slower. Therefore they should be use with some care.

As an example, you will see how to create the following strange borderline:



Because this form contains a lot of special arcs it is a good idea to make some construction lines/arcs.

First place the origo in the middle of the workspace.
Set arcsolutions to 5 deg.
Create the big circular arc, with add CIRCLE in the ArcMenu (reached from BeautyMenu) Next add construction lines defining all the straight segments of the drawing, this is done with the draw line in the OddMenu.



Next add construction lines going though the centers of the smaller circles and construction lines orthogonal to them, in order to have a crossing point at the center of the circles. Finally place the small circles in these crossing points.

Now you have created a lot of helping lines whose crossing points are defining all the corners of the border line.

Then you use the add OBSTR area and point to all the corners. After that you add the circular parts with the add area CIR.

The helping lines can be removed with rm CONN in the EditMenu and the borderline is finished.

19. menus, macros and runfiles

> How to create menus, macros and commandfiles. This chapter explains the general principles of tailoring menus and minimizing the keyboard entries needed, and some hints regarding advanced commandfiles are given.

Many of the Supermax E-CAD IPL commands has a lot of parameters controlling their behavior, but often you only use a small part of the possibilities. Therefore it is a good idea to 'hide' some of their functionality. Other commands are meant to do some special transformations on the pcb, and are normally used in a sequence of commands. With these commands it is a good idea 'packing' them into commandfiles.

Menus:

Menus are very good for collecting commands that should be used together, or for commands that are used very often. Menus also give you automatic repetition of commands.

A menu can be created from scratch. Or an already defined menu can be modified. The Supermax E-CAD IPL commands used to handle menus are **defmenu**, **swopfields** and **savemenu**.

When creating a menu, some hints could be followed:

*) Notice what commands you are typing all the time, or what menus you are switching between. Create a menu containing these commands.

*) Make the functions in the menu as simple as possible, so you can avoid typing at keyboard.

*) Place related functions in the same box in the menu, so you can switch between them without pointing to the menu.

*) Put the same function twice in the same box, so you can abort and start just by switching mousekey.

*) Do not clobber the menu with too many function, it is better to use different menus. This will make the menus easier to understand.

Macros:

There are two kinds of macros:

1) a macro with only one letter as the name and

2) macros with more letters. The latter are a kind of in-memory commandfiles, the only difference being that the macro name can be abbreviated. The former is more special, because they are executed just by hitting the letter defining them. This makes this kind of macro very suitable for often used commands.

Macros are handled by the commands defmacro and savemacros.

When creating macros some special 'tricks' should be known. If the macro is beginning with \n and it is a one letter macro, it will be executed anywhere you hit the letter defining it. This is good for commands that control the graphics (as zoom, redraw) or for changing layer (setlay). Also many of the special # keywords can be handy (as explained in the next section).

If you want to execute more than one command with the macro, just separate the commands with \n.

If the letter is defined as \ccc where c is a number, it will be interpreted as one letter with the ascii value ccc in octal code. That way it is possible to define one-letter macros using control characters. The characters available are:

```
\002  ctrl-b
\005  ctrl-e
\006  ctrl-f
\007  ctrl-g
\011  ctrl-i or tab
\016  ctrl-n
\017  ctrl-o
\020  ctrl-p
\024  ctrl-t
\025  ctrl-u
\026  ctrl-v
\027  ctrl-w
\030  ctrl-x
\031  ctrl-y
\033  ctrl-[
\035  ctrl-]
\036  ctrl-^ or ctrl-6
\037  ctrl-- or ctrl-- (control minus)
```

A special feature of macros is that the macro has a higher priority than the normal Supermax E-CAD IPL-commands. That is, if you create a macro with the same name as a Supermax E-CAD IPL-command, it will overrule the Supermax E-CAD IPL-command, and you can not access it. This feature is used for protecting your pcb against any hazardous commands (like joinwl, delcomp, reset), or for modifying the behavior of the command. To access to the buildin Supermax E-CAD IPL-command you can use the command buildin that will execute the buildin command without regard to any macros.

Example:

```
defmacro let joinwl macro 'message is "command is disabled"'
defmacro let joinwl macro 'mess is "WATCH OUT!!" \
                                    \n buildin func joinwl'
```

Note: IF you do not use the buildin in the last example you would have an macro that calls itself. This will give you the error message TOO COMPLEX COMMAND STRUCTURE.

Some rules could be followed, when creating a macro:

*) Notice what commands you are typing all the time. Create a macro executing these commands with a single letter.
*) Try to answer as many questions as possible in the macro, so you can avoid typing at keyboard.
*) Avoid using single letter macros beginning with the same letter as some of the buildin commands. (use the chapter ALPHABETIC in the reference manual).
*) Remember that the macro is only defined in-memory, so it will disappear when you leave the Supermax E-CAD IPL-system. If the macro is very smart (AND IT PROBABLY IS ?!), you should place it in your stdusr file.

Be aware that some of the standard Supermax E-CAD IPL-macros (defined in stdmacros) are selfmodifying, so using savemacros could give unexpected results. To avoid this you can run the file stdmacros AFTER your own macro file.

Some ideas of what commands could be made with macros (beside those already defined in stdmacros):

```
defmacro letter ? macro 'where start 0,0'
    give the current coordinate relative to origo
```

```
defmacro letter = macro 'clearcheck group #N \
                                        leftlo #Z righthi #M'
    clearcheck the first selected group

defmacro letter '`' macro 'redraw c -1 w -1 \n showlast'
    refresh the screen and highlight the last connection
    touched.
```

COMMANDFILES:

Commandfiles a the most powerful way to create new and advanced functions on the Supermax E-CAD IPL-system.

Commandfiles can be created with makerunfile, logging, listdevice or with an UNIX texteditor.

A commandfile should be preferred to a multiletter macro or to a complicated menu, since the command file is stored on the harddisk and is easily modified by Your Favorite Editor (YFE-,YFE). The only difference between a macro and a commandfile is that the commandfile will only be recognized by the Supermax E-CAD IPL-system when its name is fully spelled, but - of course - it can be called from a macro.

The rules about when to create a macro also applies to the commandfiles, but commandfiles can also be used for more specialized functions, that not necessarily are used very often. (like the minic commands)

When you create a commandfile you should consider the name very carefully since it will (NORMALLY) be placed in the common directory /usr/ipl/commands, so every one using your system will have access to it. Since a command file always can be called from a macro, it is best to create a long 'selfexplanatory' name for it.

If you define some 'private' runfiles you can use the 'runpath', to specify an alternative directory for the runfiles. This allows many users to have a common directory for normally used commands (i.e. /usr/ipl/commands) and to have a user specific directory. The run path can be setup from the standard runfile 'stdusr'.

A commandfile can be made in several ways. The most obvious way is just to use the editor (YFE). It is also the most powerful, since you can do everything. The only disadvantage is that you should know all Supermax E-CAD IPL-commands, with their parameters, - or you must use the chapter ALPHABETIC in the reference manual.

Another way is to use the command MAKERUNFILE, that will record everything you are doing on the Supermax E-CAD IPL-system. This will relieve you from remembering all the parameters to the commands you use. But since this Supermax E-CAD IPL-command is made specially for creating commandfiles that will run in night, it has the slightly annoying feature of not displaying what is going on. So you will be working in the dark. This will be alright for creating a command sequence for optimizing a component placement or for trying different routing strategies overnight, but for creating a function that will place some tracks in special way it is not suitable. To make some kind of interactive functions you can used the special keyword #ASK. This keyword will make the Supermax E-CAD IPL-system 'forget' the parameter, so when you execute the commandfile it will ask you for the value (notice that in the reference manual, this feature is misunderstood).

A better way is to use the LISTDEVICE command, to create a file containing the commands you want to execute, and then use the editor to modify it to suit your purpose. It has the advantage that the commands are executed simultaneously, so you can see what is going on. And it is easier than using the editor directly from scratch since all the parameters are written to the listfile, so you only have to delete/modify the ones needed.

There are some special keywords that will be recognized in the commandfile, they should be place in the beginning of the line:

#nointeractive
the default mode, where a command only will be executed if all the parameters are written in the file. This has the advantage that if you have missed a parameter or spelled it wrongly it will not prompt you for an answer, but just continue with the next command in sequence. That is, it will not stop in middle of the night and wait for you to type something silly (OR INTELLIGENT). This will of course

never happen since you are always using MAKERUNFILE for overnight commandfiles!.

#interactive
will allow missing keywords in the file and prompt for them, when appropiate. This is almost always necessary for 'real' commandfiles. It will also allow you to use the SETDEFAULT command.

#breakonerr
if any of the commands in the file ends with an error, the commandfile will be breaked immediately. The command file will end in a 'nice' way like a normal buildin Supermax E-CAD IPLcommand. Like #interactive this is also almost always necessary for 'real' commandfiles.

#nobreak
continue the command file even if there are some functions ending with an error.

#setflag
A flag that will be in top until some command end with an error. Then it is lowered to half-mast. This keyword will send it to the top.

#if (command1) command2
#if (FLAG) command2
a conditional command. If the command1 is executed without error, or if the FLAG is not lowered to half-mast, then command2 is also executed, otherwise command2 is discarded. This construction is best used with the the command TEST.

```
#if (command1)
  commands..
#else
  commands..
#fi
```
the same conditional command, but with block of commands and a 'else' part.

```
#while (command1)
  commands..
#done
  the commands.. are executed as long as the command1 is
  without error.

#noisrun
  will fool the Supermax E-CAD IPL-system to think that you,
  yourself, are typing all the commands at the keyboard.
  (IT'S SO EASY TO FOOL). This keyword will suppress the er-
  ror messages (if any) from the runfile, and it will open
  for messages from SAVEPCB,SAVETYPE and echo component/-
  group/text names used in the file. Also any changes to me-
  nus or statusline (APPEND LAY) will be shown on the scre-
  en. (see the EDITMENU example above).

#isrun
  will tell the Supermax E-CAD IPL-system that you stopped
  typing all these silly commands yourself.

#unbreakable
  will force Supermax E-CAD IPL the to execute the runfile -
  even if the user pressed ctrl-C.

#breakable
  normal behavior when pressing ctrl-C
```

Examples:

```
| command file that makes a box
#interactive
askfor parmn leftlo parmt coordinate quest leftlo
askfor parmn righthi parmt coordinate quest righthi
test expr #(leftlo)
x1=#(ipl-x)
y1=#(ipl-y)
l1=#(ipl-lay)
test expr #(righthi)
x2=#(ipl-x)
y2=#(ipl-y)
settrack from #(x1),#(y1),#(l1) to #(x2),#(y1),#(l1)
settrack from #(x2),#(y1),#(l1) to #(x2),#(y2),#(l1)
settrack from #(x2),#(y2),#(l1) to #(x1),#(y2),#(l1)
settrack from #(x1),#(y2),#(l1) to #(x1),#(y1),#(l1)
```

```
| command file that selects and moves 1 component
| from the misspoint
#interactive
windowselect leftlo #(missp)+-1gr,-1gr,0 \
  righthi #(missp)+1gr,1gr,0 group x comps y
select criteria \?&#1
showgrp group #next
compselect compcrit \?&sg
movecomp comp #next rot 0
```

```
| call the external gerber program
#interactive
#breakonerr
askfor parmn ipl-tmp parmt charstr quest tmpfile
askfor parmn savefirst parmt charstr quest savefirst
askfor parmn iplfile parmt charstr quest gerberfile
askfor parmn ipl-app parmt charstr quest append
askfor parmn ipl-lay parmt charstr quest lays
askfor parmn ipl-pol parmt charstr quest pol
askfor parmn ipl-getab parmt charstr quest getab
askfor parmn xoff parmt charstr quest xoff
askfor parmn yoff parmt charstr quest yoff
askfor parmn rot parmt charstr quest rot
#if (test expr #savefirst=y) savepcb file #ipl-tmp \
        overwrite x comment '' jobdep '' postpro ''
file=#ipl-file
ipl-heat=n
sh parm gerber.sh
```

This last example shows the use of both Supermax E-CAD IPL-commandfile and call to shell.

Calling SHELL:

When the Supermax E-CAD IPL-command sh is used the subshell called will have some parameter transferred from the Supermax E-CAD IPL-system. These are transferred from the default values in the Supermax E-CAD IPL, to the environment in the shell.

The default parameters transferred are: xmax, ymax, layers, gridmul, griddiv (from pcbmax), file, comment, jobdep, postpro, xoff, yoff, rot (from savepcb, getpcb), divx, divy, listunit (from usergrid, listunit), plus all parameters beginning with ipl or ending in dir.

These transferred parameters can be used in a shellscript.

To pass parameters back to the Supermax E-CAD IPL-system, you can create a commandfile in the shellscript and execute that file when returning to the Supermax E-CAD IPL.

Note: When starting the Supermax E-CAD IPL-system it creates a unique filename and places it in the default parameter tmpfile.

A better way to pass information back to the Supermax E-CAD IPL-system is using the shellprogram IPLEXEC. IPLEXEC will take an Supermax E-CAD IPL-command on its standard input, pass it to the Supermax E-CAD IPL, and write any messages from the Supermax E-CAD IPL, on its standard output. Normally no graphics will be displayed on the screen, but by using the command #ALPHAMODE ALFA N, the graphic will also be enabled.

Dictionary

In the Supermax E-CAD IPL a lot of different 'words' are defined and used throughout the menus. In order not get too confused a small vocabulary is given here, comparing the Supermax E-CAD IPL terminology with some other systems:

comp, cmp, component
: a physical component with a specific name and mechanical appearance. In other systems it is called shape, figure, nodefig...

group, grp, wl, wirelist, netlist, net
: a list of component pins that should be connected together in the final PCB. The net contains all the pins in an unordered way, that is - the net is NOT divided into sets of frompin-topin. In other systems it is called tree, signal....

pad, pin
: a component pin

connection, conn
: the physical tracks or wires and throughholes that connects the component pins together in the PCB.

track, tra
: one single segment of a connection

via
: a throughhole. The via is part of an connection and can be moved freely around, as opposed to a pad which belongs to a component and is fixed in a relative position to the component.

shape, tracksha, viasha
: a shape is the definition of a size and form of a track or a pad or via. The shape is like a padstack specifying the appearance on the graphic display, the photoplot and the soldermask. It also defines the drillsize for pads/vias. In other systems it is called padstacks or technology specifications.

area
: an area is a description of the borderline of some kind of surface in the PCB. This could be a restriction area disallowing tracks or vias to enter, or a copper filled area for analog purposes. In other systems it is called border, fills, grids or shapes.

Cross Reference:

This chapter gives a cross reference list between the chapters in beginners guide and the corresponding chapters in the reference manual. The chapter called 'general' in ref. manual should be read.

1. **Setting Workspace and creating a border outline**

```
PCBMAX          -> pcbmax
set ORIGO       -> sysparm: origo
add CORNERmark  -> setcomp, fixcomp
?               -> listcontent
add TRACK       -> settrack
add CONNECTOR   -> setcomp, fixcomp
```

you should also read the setlay and the dispon chapters.

2. **Placing components I**

```
grd             -> sysparm: usergrid, drawgrid, drawcoor
lay             -> laysetup
add COMP        -> setcomp
<               -> general
move COMP       -> movecomp
turn COMP       -> movecomp, touchcomp
mir COMP        -> movecomp, touchcomp
move CMPinWIN   -> movewindow
~               -> movecomp, touchcomp
rename COMP     -> renamecomp
align COMP      -> align
swop COMP       -> swopcomp
red             -> redraw
rm COMP         -> delcomp
DATABASE y/n    -> sysparm: dbson (and setcomp)
```

3. Entering netlist interactively

Show PIN/2	-> showpinno
Show PINhalf	-> showpinno
Join WL ...	-> joinwl
grp RENAME	-> grprename
set GRP PROT	-> setprot
RB grp	-> rbgrp
rb COMPpair	-> rb2comp
rb COMP	-> rbcomp
MARK grp	-> markgrp
LIST wl	-> listwl
LIST wlNAME	-> listwl
set PIN single	-> singlewl
ESTIMATE	-> estimate
DENSITY	-> density

4. Placing components II

sel COMPS	-> compselect
FIX operations	-> fixcomp, fixwindow, chfix, dispfix
INITPLACE std	-> initplace
RBG comp	-> rbcomp
mov COMP	-> movecomp
ESTIMATE	-> estimate
DENSITY	-> density
OPTIMIZE	-> optimize
sel C.inWINDOW	-> windowselect
move NEXT	-> movecomp, compselect

5. Power routing

RESET PCB	-> reset
GET pcb	-> showpcb, pcbmax, getpcb
tra	-> tracksetup
45deg off	-> sysparm: 45deg
MARK GRP	-> markgrp
add track	-> settrack
SHOW LAST	-> showlast
route GRP	-> routegrp
rm BLIND	-> blind
swop CONN	-> swopconn

6. Signal routing

tra	-> tracksetup
BUS route	-> busroute
SMALL route	-> smallroute
sel MISS	-> selectgrps
list VIANUM	-> inuse
auto SWOP	-> autoswop
MINI-CLOSE	-> commandfile using reroute,...

7. When the router needs help, but helps You anyway

sel MISS	-> selectgrps
MARK/RBM GRP	-> markgrp, selectgrps
gsh	-> getshape
route GRP	-> routegrp
MAX DETOUR	-> sysparm: maxdetour
CHK+SKIP if OK	-> check, selectgrps
WAVEroute from	-> sysparm: waves (and routegrp)
route FROMTO	-> routefromto
SHOW LAST	-> showlast
MOVE conn	-> moveconn
CHLAY/MOVE	-> chlay proceeded by moveconn
JUMP conn	-> jumpconn
PUSH conn	-> compact
RIPROUTE	-> riproute

8. Checking

SEL CHECKERR	-> check, selectgrps
CHK+SKIP if OK	-> check, selectgrps
move conn	-> moveconn
erasescreen	-> erasescreen, zoom
RM between	-> delbetween
SEL CLEARERR	-> clearcheck, selectgrps
CLCHK+SKIP	-> clearcheck, selectgrps
addtrack	-> settrack
CHTRA sha GRP	-> chshape

9. The final finish

no of TRA/VIA	-> inuse
BEAUTIFY	-> beautify
0 and MAX	-> #zerocoordinate, #maxcoordinate
dsp	-> dispsetup

10. Texts and the Silkscreen

```
add TEXTstd      -> settext
mov TEXT         -> movetext
turn TEXT        -> movetext, touchtext
mir TEXT         -> movetext, touchtext
add TEXT         -> settext
edit TEXTparms   -> edittext
rm TEXT          -> settext
Textoverlap      -> overlap
make INVISIB     -> textinvisible
make VISIB       -> textinvisible
disp INVISIB     -> sysparm: dispinvisible
add MEASURE      -> measure
```

11. The End

```
save PCB         -> savepcb
list PCBs        -> showpcb, UNIX command "ls"
MAKE POST        -> makepost, gerber in the "post manual"
setSIZE+erase    -> erasescreen
plot GERfile     -> gerplot
make CANONplot   -> canonplot in "post manual"
make HPplot      -> canonplot in "post manual"
```

12. Lay assignments

For this chapter you could read the chapters general, laysetup, layattributes, laycolor, layname and the sysparms: laymap, mirmap, layorder.

13. Shapes

This chapter is connected with the commands tracksetup, padsetup, dispsetup, makeshape, chdrill, copypol and sysparm: smoothshadow.

14. Creating Library

Reset	-> reset
get TYPE	-> gettype
move ALL to	-> movewindow
add PAD	-> setpad
add BOX	-> this is a commandfile using settrack
sil	-> setlay
move REF	-> moveref
save TYPE	-> savetype
move PAD	-> movepin
stretch WIN	-> stretchwindow

In connection with this chapter you could use the commands rename to #libtype, maketype and comptotype.

15. Advanced shapes

add OBSTR area	-> defarea
AREA->POLFILE	-> areatopol
rm area	-> defarea

16. Power Innerlayers

sel MISS	-> selectgrps
MARK/RBM GRP	-> markgrp, rbgrp, sysparm: rbmiss
sel in topmenu	-> general: #select
list WLnames	-> listwl
lay in topmenu	-> laysetup
2 in topmenu	-> setlay, dispon
add TRACK	-> settrack
WLOPT	-> wloption
Check	-> check
save PCB	-> savepcb
MAKE POST	-> makepost, gerber in the "post manual"
setSIZE+erase	-> erasescreen
plot GERfile	-> gerplot
ROUTE->VIA	-> routetovia

17. Filled areas

add HATCH area	-> defarea
end in topmenu	-> defarea: #end
joinarea	-> joinarea
move corner	-> movecorner
add area CORN	-> addcorner
add area CIR	-> addcorner
angle=>round	-> roundcorner
install time	-> areainstall
install mode	-> sysparm: edgetype, arcsolution
CHTRA sha GRP	-> chshape
Show TRA shape	-> showshape
add ROUND	-> roundcorner
add auto ROUND	-> autoround
rm auto ROUND	-> autoround
rm ROUND	-> roundcorner
add SPIRAL	-> commandfile using setarc

18. Obstruction Areas

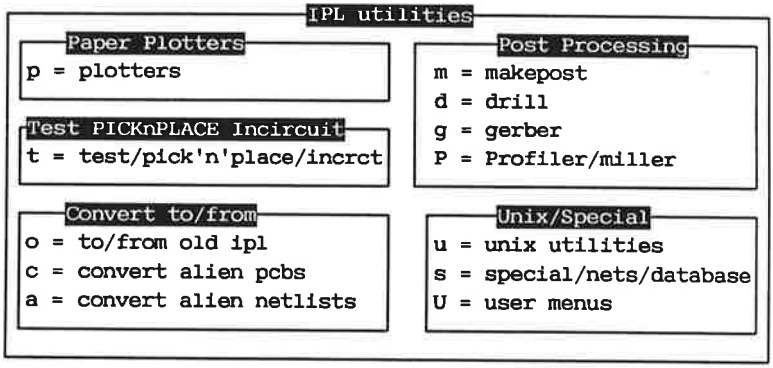
add OBSTR area	-> defarea
comptext	-> comptext
cpy OBSTR area	-> editarea
origo	-> sysparm: origo
arcsolution	-> sysparm: arcsolution
add CIRCLE	-> setarc
add area CIR	-> addcorner

The Helpmenus

This chapter introduces the helpmenus, and gives an overview of what kind of actions that can be made with the helpmenus.

The helpmenus are a set of menus to help you trough the programs that surround the Supermax E-CAD IPL system. The helpmenus can be used from inside the Supermax E-CAD IPL system by pressing the 'H' key. Or you can start the menus from any alphanumeric terminal by writing H followed by the return button.

When you have started the helpmenu, the first menu you see looks like this:



This is the main menu, corresponding to the BaseMenu in the Supermax E-CAD IPL system. It divides the helpmenus into different subjects. To move to one the subjects you press the corresponding letter, e.x. to go to the plotter programs you should press a 'p', to go to the netlist programs you should hit a 's'.

When you have pressed one of the defined keys, a new menu will be shown which further subdivides the programs. You can at any time move a step back by pressing function-key f1 or you can completely leave the helpmenus with function-key F1 (shift f1).

For example, if you hit a 'd' to go to the drilling programs, a new menu will be shown:

```
Drill menu
d = drill output
C = change drill table
c = change drill data
p = copy to tape puncher
P = copy from tape reader
v = verify an drill tape
```

This menu shows the different programs that are relevant for making data for NC drilling machines.

If you press a 'd' again, you will get a menu for making one single file of drilling data. (This would normally be done with the MakePost program, but it also possible to do it without using MakePost). The menu looks like:

```
Drill output
pcb file   = /usr/ipl/iplsave/mypcb

out file   = /tmp/pip

code file  = /usr/ipl/tables/asciiautinch

lay        = -1
rotate     = 0 1 2 3 4 5 6 7
x offset   = 0
y offset   = 0

generate data for drillplot = no yes
  shape file = /usr/ipl/tables/drillshapes
```

First you should notice that the first text on the first line pcb file = /usr/... is written with inverse video (the background color and the lettercolor are swapped). This indicates that you are now able to change the filename. To edit a name you can use the arrowkeys on the keyboard to move forth and back in the line, you can use the function-key f6 to insert a letter and F6 (shift f6) to delete a letter. The function-keys f6,F6,f8 and F8 are also available to move to end and start of line and to

delete rest and whole line.

When you have entered the name you press the return key or the arrow-down key.

If the filename you have specified is acceptable the line is written in a normal way (normal letters on normal background), and the next line is shown with inverse video.

If the filename is not acceptable the helpmenu will ring the bell and wait for you to enter another and better name. If you are in doubt, you can press the F16 (shift f16) function-key. Be careful to use the shift button, since the f16 without shift has a completely different meaning, and using f16 can be dangerous in this moment. Pressing the shift-F16 will make a listing of the files in the directory given by the unacceptable name. E.x. if the the filename is /usr/ipl/iplsave/nothere and the file nothere does not exist, the helpmenu will show a list of files in the directory /usr/ipl/iplsave, that is, all the saved pcbs. Then you can select the correct file with the arrow-keys and the return-key or by pointing with the cursor to the file (X-window systems be careful).

Notice that it is slightly different from the way the Supermax E-CAD IPL system works, because you must give a incorrect filename and it should NOT contain any wildcard characters like *,? and [or].

In some cases, the helpmenu will ask:
  Edit filename before return (y/n):
Normally you should just hit the return-key again. But if you enter y and then return, the helpmenu will start the UNIX editor called 'vi' with the file, so you can make changes to the file. This is very handy, when you enter netlists or gerbertables.

Now the selected file is inserted as a correct filename, and the helpmenu moves to the next line.

The next line is the name of the output file wherein the drilling data is placed. You can use the same method to edit the filename, but the F16 is of no or little use. Normally the output files from the drilling program are placed in the directory /usr/ipl/drill, but often you make the file only for intermediate use, so it could also be placed in the directory

/tmp, which is a directory specificly for such intermediate file (temporary files).

The next question is the name of a file controlling the behavior of the drillprogram. This file will be described in the chapter 'Understanding drill'. For the moment you should just use the file called /usr/ipl/tables/asciiautinch.

The next question is the laynumber that should be used for the generating the drill. It should normally be 255, but could be another lay if the drill should be use for e.x. testpoints.

The answer you give to this question is not checked immediately, so entering something crazy like 'Lange Øreringe', will be accepted by the helpmenu, but -of course- the drillprogram will complain about it.

Now you have moved to the line: rotate = 0 1 2 3 4 5 6 7, and you can see that one of the possible rotations are written with inverse video. This is because, the helpmenu shows all the possible available answer, and you can use the arrow-keys to select one of them (it is like the e.x. the grd command in the Supermax E-CAD IPL system).

The to next questions are for the optional offset of the drill data. They are normally 0. Like the lay question, no checking are made of sanity of answer.

The next question is also a predefined one, where you can answer either yes or no. Normally it should be no, but just to test try to select yes.

When you select yes, the helpmenu will show an additional question: shape file = /... , which could be answered with /usr/ipl/tables/drillshapes.

If you want to change some of the answers you have already given, you can use the arrow-up key to move back to preceding questions. Then you can reenter a new answer and move back again with the arrow-down key.

Use the arrow-key to change the 'generate data for drillplot' to 'no'.

When you have hit return after answering the last question on a helpmenuscreen, all the the questions will be written with normal video.

Now you can start the program by pressing the function-key f16, or you use the arrow-up or -down to change some the answers, or you can completely abandon this menupage by pressing f1.

When you hit the f16 button, the helpmenu will clear the screen and write what program you have started. In this case it will display /usr/ipl/prepostpro/drill. Any messages from the drill program will be shown on the screen.

When the program is finished helpmenu will show the line
`f1 back, F1 exit, f2 help, .....`
at the bottom of the screen, and you have the option to go back to the same menupage, so you can start the program with new parameters, or you can leave the helpmenus with the F1 button.

If the helpmenu was started from within the Supermax E-CAD IPL system, this finishing line will look a little different:
`Quit, Back, Same page`
But the function keys will work in the same way.

The above example, shows the different ways to enter data to the helpmenus and how to move between the different menus.

What programs can be found in the helpmenus:

The content of the helpmenus is constantly changing, because off the ever increasing number of programs that are added to the Supermax E-CAD IPL systems environment. Below is a brief overview of programs that can be found in the helpmenus.

The main menu divides programs in 5 different categories:

1) `Paper Plotters`.

   Programs to make paperplotted documentation. The documentation is automatically extracted from the saved pcb files or schematics, from gerberfiles, from the component library etc. The programs controls various paperplotters of the penplotting, laserplotting or electrostatic type. These programs are described in the chapter Paper Plotting.

2) Test Pick'n'Place Incircuit.
Programs to generate data for testers and placing machines of various kinds.

3) Conversion Programs.
Programs to import data from other cad/cae/cam systems, and -to a lesser extend- export to other systems. The data translated range from ascii netlists to complete binary pcbfiles. Also included are the handling of backannotation and ECO (Engineering Change Orders) to and from many systems.

4) Post Processing.
Programs that handle the classical postprocessing actions: Creating photoplotter data, drilling data and milling data. It is also programs to put the data on appropriate output medium, like floppy disks and punched paper tape. These programs are described in the chapters Understanding Gerber, Understanding Drill and Understanding MakePost.

5) UNIX / Special.
This is actually subdivided further:

5a) DataBase.
Programs that handle the Supermax E-CAD IPL database, like inserting/changing, and listing parts of the database. These programs and menus are described in the IPL Post manual in the chapter DataBase.

5b) Netlist.
These programs are for manipulating the netlists of the Supermax E-CAD IPL system. E.x. comparing netlists in different ways, and checking for correct use of components. Programs to create Part lists and BOM (Bill Of Materials) are also included.

5c) UNIX programs.
These programs handle many of the day-to-day problems, like copying files, renaming files, removing files, making backups on floppydisk etc.

5d) User menus.
This is for user defined menus, which will not be overruled by menus from Supermax E-CAD IPL.

Many of the programs in the helpmenus are not described. They are normally rather selfevident, with very few parameters and a simple function. You should never be afraid to try them out, and -if in doubt- seek help. Also, you should not be afraid ask for new programs to be added to the menus, since often it is very easy to add new facilities.

Paper Plotting

The helpmenuscreens to make paperplots are described.
And the different documentation types are shown.

A paperplot is any plot that is not a photoplot.

The paperplotters supported are:

hp penplotters and hpgl compatible plotters
benson penplotter
calcomp electrostatic plotter
versatec penplotter
canon laserplotters and canon vdi compatible laserplotters.

The kinds of documentation that can be plotted are:

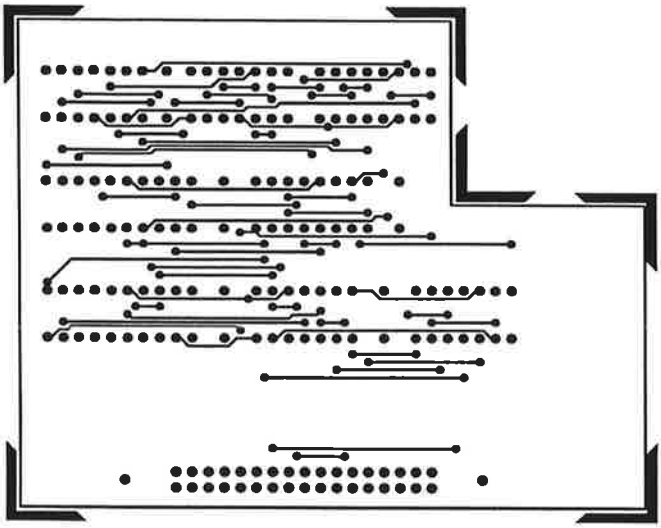
one or many layers from pcb file
one or many layers from schematic file
component placement drawing
drillhole documentations
library documentations
contents of gerberfile

plus extracts and combinations of the above mentioned.

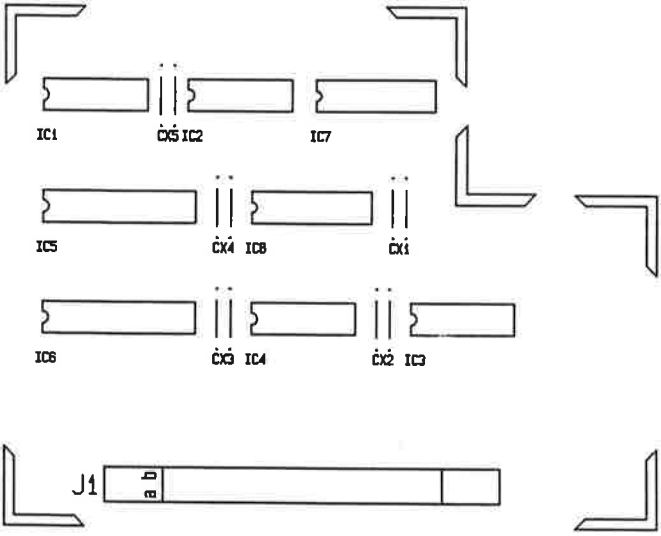
To make any of these plots, you should use the helpmenu. The helpmenu is started with 'H' either from within IPL or from an alphanumeric terminal. Then press a 'p' to go to the Paper Plotters helpmenuscreen.

Standard plots of layers from pcb file:

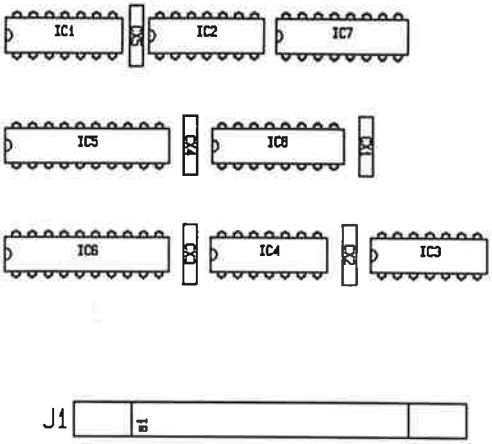
The next pages shows 3 different paperplots. The 1. is of lay 0,255 (component side), the 2. of lay 11,12 (the component silk side), the 3. of lay 13 (pen plotter - documentation), and the 4. is a plot in a window of layers 10,0,1,255.



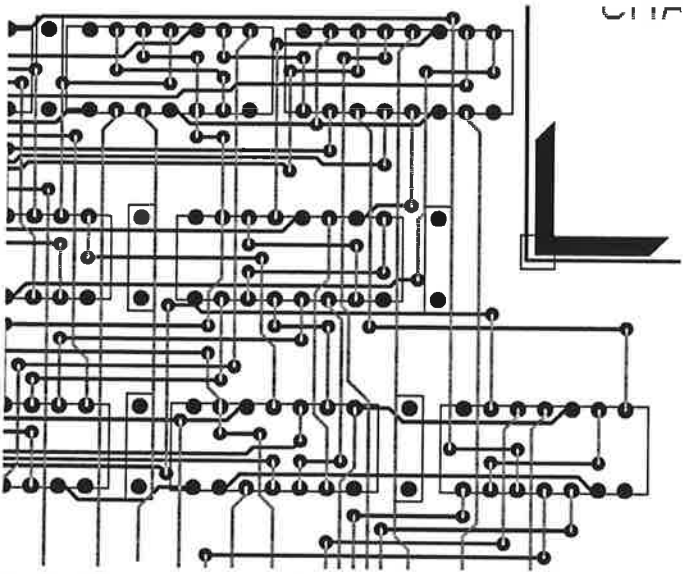
Cuisine 8 lay 0 and 255 with plotter pol, filled.



Cuisine 8 lay 11 and 12 with plotter pol.



Cuisine 8 lay 13 with plotter pol.



Cuisine 8 lay 0,10 and 255 black and lay 1 gray, plotter pol.

To make a plot of a pcb, you select the appropriate plottertype and press the corresponding letter in the helpmenu.

For example, you have a canon laserplotter, so you select the canonplot in the menu by hitting the 'c'. Then you will see a helpmenupage:

```
                    canon output
pcb file            = /usr/ipl/iplsave/mypcb
out file            = /dev/print0

lay                 = 0,255
plot polygon        = p s d g e 0 1 2 3 4 5 6 7 8 9
x offset            = 0
y offset            = 0
rotate              = 0 1 2 3 4 5 6 7
scale factor        = -2
program to fill     = no yes
fill mode           = as_dispmode no fill all
pen width           = 0
color list          = 1,1,1,1,1,1,1
append plot after   = no yes

clip text/tracks    = no yes
  clip lay(s)         = 0,1
  items dist pv10     = pv10
```

The fields you have to answer are:

pcb file

is the name of the file wherein you saved the pcb. It must be specified with full path, (remember you can use the shift-f16).

out file

is the name of the canonplotter or -unusual- the name of a temporary file. The name of the plotter is dependent on the setup of your UNIX system, but it will typically be /dev/-print0, if it is the only plotter/printer in your system, otherwise it could be /dev/canon, or /dev/plotter.

lay

the lay or layers you want to plot on top of each other. If you want to see the silkscreen it would be 12. If you want to see the solderside it would be 1,255 -the 255 is added so you see the pads and vias.

plot polygon

the polygon in the shapes that should be plotted. It will normally be the plotterpolygon 'p'.

x offset and y offset

an eventual offset from the zeropoint of the paper. For the canonplotter it is almost always 0.

rotate

the rotation of the plot on the paper. The rotations 4 to 7 are mirrored plots. For the canonplotter it is almost always 0.

scale factor

the scaling of the drawing relative to true size. Normally you want the plot to be as big as possible, given the papersize. For the canonplotter the papersize is always an A4. If you give the special scale -1 the program will calculate a scaling factor that suits the A4, if you specify -2 it will also rotate the plot in order to make it as big as possible. Otherwise you could enter a number, e.x. 0.60, which will scale the drawing down to 60 percent of the true size.

program to fill

is normally set to no, but if the pcb contains some really artistic and bizarre shapes, the plotter might have trouble filling the shape correctly, and then it should be yes.

fill mode

is how the interior of the pads and tracks should be filled. If you choose nofill, only the contours will be plotted. This is by far the quickest method. The two other possibilities will fill either as specified by track- and pad-setup in the Supermax E-CAD IPL system, or it will fill everything.

pen width

any track thinner than penwidth will be plotted as a single line.

color list

is a funny name for different the linestyles, the canonplotter can use. Each number corresponds to a number in the list of lays given above. The normal is 1,1,1,1... which will plot everything with solid lines. But if you select 2 to 7 the lines will be plotted as dotted lines with different marker/space relationship. If you select 8,15 or 16 filled lines will be plotted with different fillpatterns. The 3 example above, where layers 10,0,1,255 are plotted used the color list 1,15,8,1.

append plot after

specifies whether the paper stays in the plotter for subsequent plots or is flushed out of the plotter. Always no.

clip text/tracks

is a special option for plotting the silkscreen. If yes it will remove all parts of the silkscreen that overlaps pads and/or vias in a given distance.

clip lay(s)

asked only if clip text/tracks is answered with yes. It is the lay(s) where pads and vias are placed. If you are making a component side silkscreen, it should be 0,255. For solder side it should be 1,255.

items dist pv10

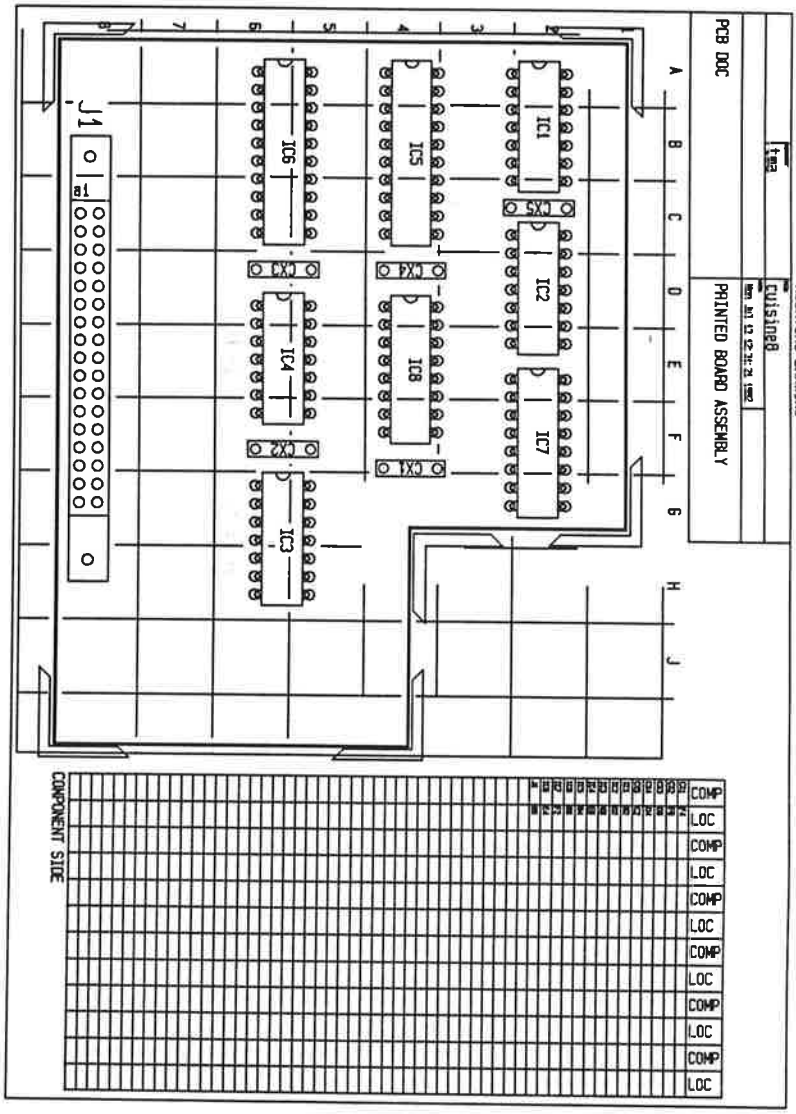
asked only if clip text/tracks is answered with yes. The p specifies that the silkscreen should be clipped against pads, the v specifies for vias and the number specifies the clearance (measured in mill).

After answering all these questions, you press the function key f16, and the plot will be made. Normally you will make separate plots of the component side, the solder side, the silkscreen and perhaps the outlines together with article names or device names for mounting documentation. This is very easy, because when the plot is finished you just return to the helpmenu and then it suffices to change the laylist whereupon you hit the f16 again!

If your plotter is one of the other kinds, there is only a few differences:

The colorlist is a true colorlist, specifying the pennumber.
The scale -1 and -2 does not work since the papersize is unknown.
A penforce parameter might be needed.
If the hp-plotters are directly as the outfile, the hpplot program will start a conversation with the plotter prompting it for its version number and filling capabilities.
The program waits for you to insert paper if the autostart parameter is no.
The plotters might need an offset because the zeropoint often is the center of the paper.

The next page shows a socalled compdrawing plot.


MOUNTING DRAWING
cuisine8
PRINTED BOARD ASSEMBLY
PCB DOC
IC1
IC2
IC3
IC4
IC5
IC6
IC7
IC8
CX1
CX2
CX3
CX4
CX5
J1
COMPONENT SIDE
COMP
LOC

Component placement drawing:

The component placement drawing is a documentation plot of the component placement, it makes automatically a grid wherein the components are referenced.

Go to the compdraw helpmenuscreen.

```
                    Creating compdrawing

pcb file                    = /usr/ipl/iplsave/mypcb

side to plot                = compside solderside
lay to plot (comp)          = 13,255

find pcb max by             = lay comp
  lay(s)                    = 255

compdrawing tablefile       = /usr/ipl/tables/compdraw
cmp locator list in file    = n y
   list file                = /tmp/list

autoscale in drawmax        = n y

plotter device              = canon hp pcb
  plotter port              = /dev/print0
```

Once again you have some questions to answer:

pcb file
the name of the pcb file, with full path.

side to plot
what side to make the documentation plot. If compside then plot the components placed on the component side. If solderside plot the components on the solderside (the mirrored components).

lay to plot
the lays to be plotted. Typically the compdraw and the component pins, that is 13,255 for compside and 23,255 for solderside.

find pcb max by
how the plotter program should calculate the size of the pcb. It is typically done by inspecting a lay (normally lay 255), where the pcb borderline is defined. But is can also

be done by inspecting some components of a special type.

lay(s)
the lay or lays defining the pcb border line. Normally 255, where you have defined a component keep in area or a drawn a track restricting the autorouters.

component type
is only asked if the 'find pcb max by' was answered with 'comp'.

compdrawing tablefile
the name of a file defining the layout(s) of this documentation plot. It defines the placement of the grid and the table. It also defines the size of table. The format of this textfile is described in the Supermax E-CAD IPL POST manual. The default is /usr/ipl/tables/compdraw.

cmp locator list in file
if yes, the contents of the table are also placed in a file.

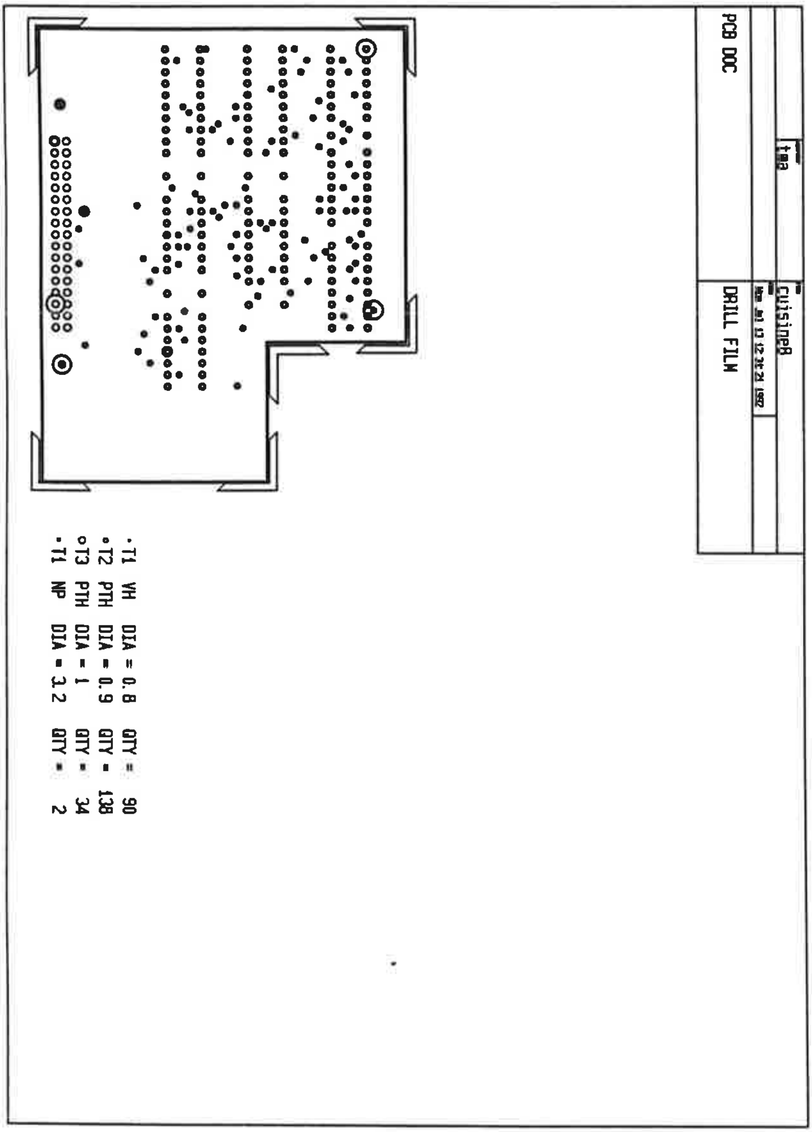
list file
the name of the file with the list.

autoscale in drawmax
if yes the drawing will be scaled so it meets the papersize best possibly.

plotter device
the kind of plotter to be used. It can be canon or hp, in which case it will be just like a standard plot. Or it can be pcb. In that case the program will create a pcbfile which can be plotted with any of the standard plotter programs and it can even be edited with the Supermax E-CAD IPL system.

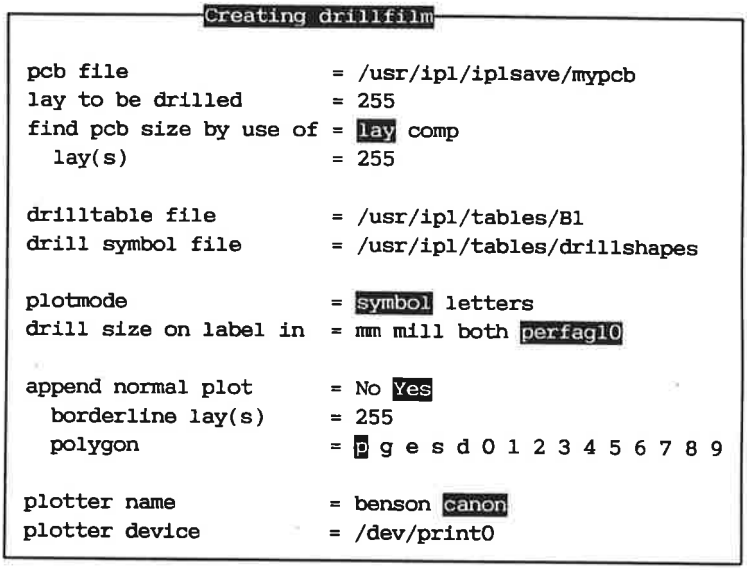
Depending on the the plotter device selected, some extra questions will be asked, specifying the plotters name or the pcbfile name.

The next pages shows two drillhole documentation plots.


PCB DOC
tma
cuisine8
DRILL FILM
•T1 VH DIA = 0.8 QTY = 90
•T2 PTH DIA = 0.9 QTY = 138
°T3 PTH DIA = 1 QTY = 34
•T1 NP DIA = 3.2 QTY = 2

Drillhole documentation:

The 2 plots are made from the drillfilm helpmenuscreen. The name drillfilm is a little misleading, but you can also make this kind of documentation on the gerber photoplotter.



The questions here are as follows:

pcb file
same ol'e stuff.

lay to be drilled
the lay wherefrom the drillinfomation is extracted. It is normally 255, unless you are making a testbench tool.

find pcb size by
how to calculate the pcb size. It is normally calculated from lay 255.

lay(s)
the lay or lays defining the pcb border line. Normally 255, where you have defined a component keep in area or drawn a track restricting the autorouters.

drilltable file
a textfile describing how the drill data should be made. This is explained in the chapter Understanding Drill. In this case it is not very important, so the default /usr-

/ipl/tables/B1 can be used.

drill symbol file
the name of a file describing the different symbols used to represent the different drillholes. It is actually a standard pcbfile, wherein some shapes with the names T1,T2,T3... are defined. These shapes corresponds to the different drill tools used in the drill data. These symbols can be changed by loading the file into the Supermax E-CAD IPL system with getpcb, and then changing the shapes with the command padsetup. The default file is /usr/ipl/tables/-drillshapes.

plot mode
you can choose between letters and symbols.

drill size in
what unit should be used for the drillsizes.

append normal plot
if yes, you can plot some lays from the pcb together with the drilldrawing. This could be the borderline or some lay with measurement drawings.

borderline lay(s)
the lay(s) where you have defined the borderline.

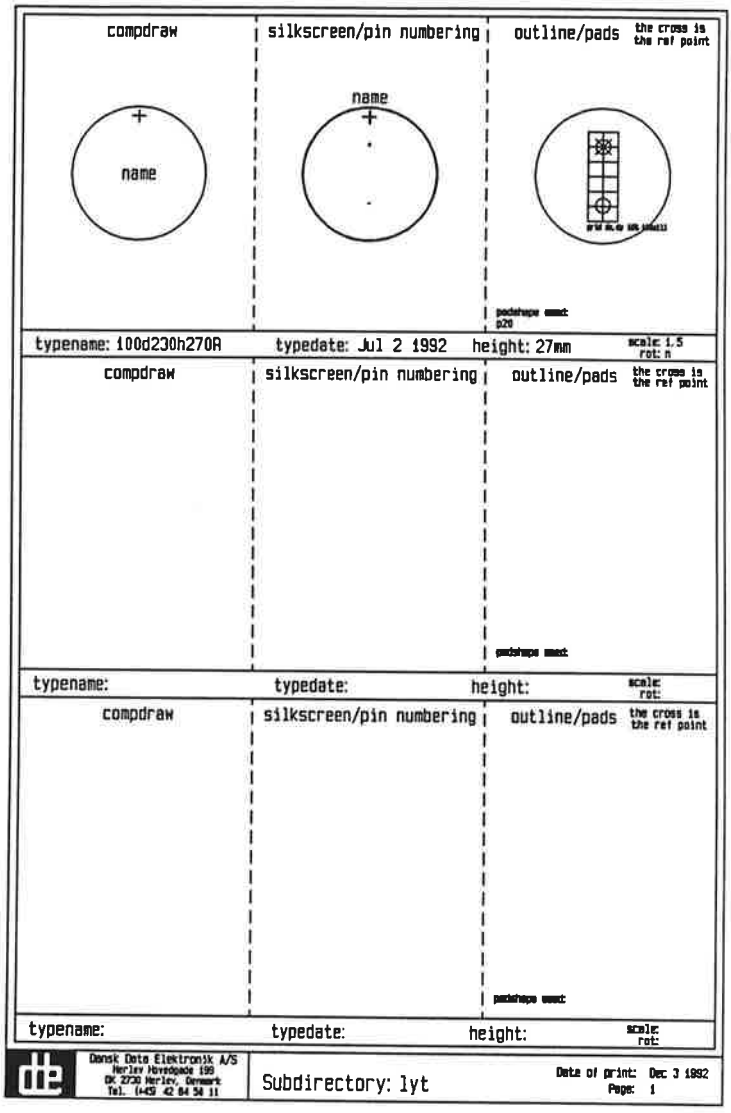
polygon
the polygon to be used for plotting the borderline.

plotter name
here you can chosen between canon, hp, calcomp or benson.

plotter deviceion
the UNIX dependent device name, e.x. /dev/print0 or /dev/-plotter.

If you select one of the other plotters you will also have to define a scale factor and offset.

The next page shows a so called typdoc plot.


compdraw
silkscreen/pin numbering
outline/pads
the cross is the ref point
name
name
typename: 100d230h270R
typedate: Jul 2 1992
height: 27mm
scale: 1.5
rot: n
compdraw
silkscreen/pin numbering
outline/pads
the cross is the ref point
typename:
typedate:
height:
scale:
rot:
compdraw
silkscreen/pin numbering
outline/pads
the cross is the ref point
typename:
typedate:
height:
scale:
rot:
Dansk Data Elektronik A/S
Herlev Hovedgade 199
DK 2730 Herlev, Denmark
Tel. (+45) 42 84 54 11
Subdirectory: lyt
Date of print: Dec 3 1992
Page: 1

Library documentation:

The typdoc allows the user to produce drawings of the component in the library. The typdoc contain a setup file for etch sub directory in the library.
By writing **e** you may create or modify a existing setup file. If a new file is created, a top with an example is added. (the UNIX editor 'vi' is used)
By writing **r** a setupfile can be executed, and the types will be plotted out on the defined plotter.
By writing **c** all the setup files that match with the name of a subdirectory in the library, will be checked and updated for missing types. and ready to be modified in an editor.
The new lines witch is added in the list, starts with '!' and will be ignored while printing.

The parameters in the setup file must be as below, and in the same sequence.

type name: The filenames of the components witch are to be drawn. With full path or relative to the main directory
sheet: The name of the sheet witch the component is to be plotted as (sheet1 to sheet4)
plot: The number of plot for the specified component (1 to 4) (see note)
scale: Scaling of plot for the specified component. If you write 'aut' the typdoc will autoscale the type.
rot: (y/n/x) Rotation of the specified component.
y: rotate the type
n: don't rotate the type
x: (use only with autoscale), The rotation will be fixed where the scale are largest.
gridx: The x-grid in field with outline/pads
gridy: The y-grid in field with outline/pads
p1: Plotted lays in field 1 (compdraw)
p2: Plotted lays in field 2 (silkscreen/pin numbering)
p3: Plotted lays in field 3 (outline/pads)
p4: Plotted paste in field 3 (paste mask) (see note)
text: Text to be plotted in the bottom of field 1, spaces are alouded.

Note: except p4 all the other plot use the p-polygon. P4 use O-polygon witch is the paste mask. When p4 is selected the

paste mask is plotted on top on the other information in field 3. On a canon plotter the 0-polygon is plotted as gray instead of black as for p-polygon.

Plotting a gerberfile:

This program can plot a gerber file on any of the paperplotters. It can be used for checkplots of the gerberfiles, before they are send to the photoplotter. It is a 'hardcopy' version of the Supermax E-CAD IPL command gerplot.

It asks for a file describing the gerberfiles format and the appearance of the photoplotter apertures. This is defined in a so called gerber table. This table is described in the chapter Understanding Gerber.

Understanding Gerber:

This chapter explains how the gerber programs functions and how to control it.

In the Supermax E-CAD IPL system you create shapes. These shapes can have any size and any geometric form. The gerber photoplotter is only able to plot a finite number of geometric forms with a finite number of sizes. These possible forms are called apertures.

When you create data for the photoplotter the program tries to match the shapes with the photoplotter apertures. If a match can be found then the aperture will be used to plot the shape. But if no match can be found, the gerber program constructs the correct shape by drawing lines.

For the gerberprogram to be able to do this, you must specify the geometric forms and sizes of the apertures on the photoplotter. This description is called a gerbertable.

The gerbertable contains a line for each aperture on the photoplotter. This line gives the name/number of the aperture and the form and size, plus some tolerance interval for the size. This tolerance is made so the gerberprogram can make a match within reasonable limits.

The format of these lines is defined in the Supermax E-CAD IPL POST manual, in the chapter gerber and in appendix 2.

To check how the gerber programs will flash or construct the different shapes in a pcb file, you use the program gercheck. Gercheck can be found in the helpmenus (under g=gerber and G=check plotter codes) or it can be called from the PostMenu in the Supermax E-CAD IPL system. The gercheck tells you how many draws and moves, that are necessary to construct the shapes.

In the helpmenu all the programs relevant to making gerber data are collected in the Gerber Menu:

```
Gerber menu
g = gerber output
G = check use of plotter codes
c = convert gerber output
d = gerber disk
C = change gerber table format
v = Convert CV gerber to IPL gerber
V = Convert IPL gerber to CV gerber
P = plot gerber file
```

The menu **Gerber output** looks like:

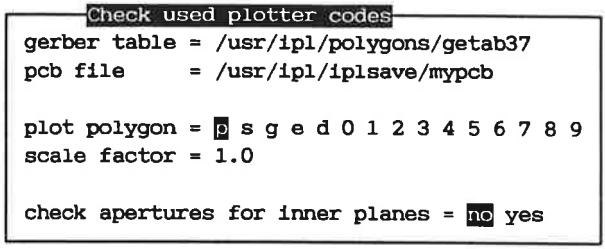
```
Gerber output
pcb file      = /usr/ipl/iplsave/mypcb
out file      = /usr/ipl/gerber/mypcb.1
gerber table = /usr/ipl/polygons/getab37

plot polygon = p s g e d 0 1 2 3 4 5 6 7 8 9
lay          = 0
scale factor = 1.0
rotate       = 0 1 2 3 4 5 6 7
x offset     = 0
y offset     = 0
append       = no yes

generate innerlay (obsolete) = no yes
generate compdrawing = no yes
clip comptext/tracks = no yes
   clip lay(s)     = 255
   clip items pv10 = p25
```

Although you normally will generate gerber data with the makepost program, it is very instructive to see what kind of informations the gerber program uses.

As you can see, the **Gerber output menu** resembles the paperplotter menus a lot. The only real difference is the extra gerber table.

The **append** option makes it possible to create one single gerber data file containing many plots. But you have to specify the offsets correctly, so there will be no overlaps.

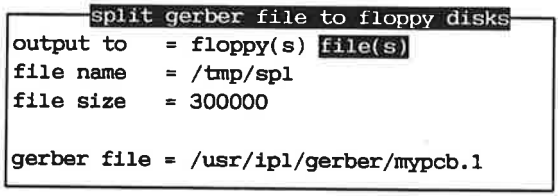
The **Gercheck** menu looks like:



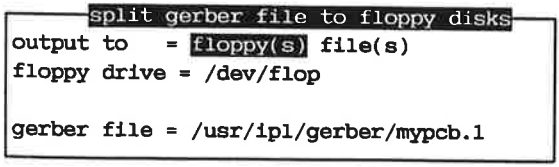
The **Convert gerber output** menu is used for changing the format of an existing gerber file. It is used very rarely, probably only if you have to inspect a gerber file created by an alien system.

The **Gerber disk** menu is rather important, even if you use the makepost program. When you have created the gerber data files for a pcb, they should be transferred to the photoplotter. This is often done with one or more floppy disk (normally pc-compatible floppy disks). Normally a gerber file is quite big, that is bigger than the capacity of the floppy disk. Therefore you will have to split the file into smaller parts. The **gerberdisk** program splits the gerber file in a very smart way, so the single disk can be read in random order.

The menu looks like:

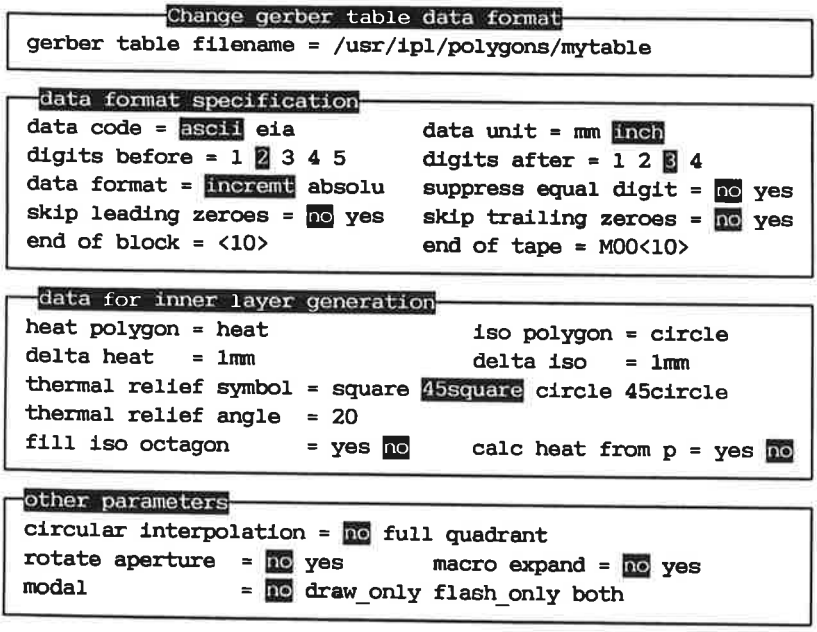


or



As you can see, you can move the gerberfile directly to the floppydisk, or you can split the gerberfile into smaller files with a given maximal size. If you split it into files the new files will have the names e.x. /tmp/spl.01, /tmp/spl.02 etc. These files can then be transferred to pc-floppy disk with the Unix helpmenu.

The **Change gerber table format** menu, is a very complicated menu, specifying a lot of parameters for the gerber program:



The different parameters are described below. But before you despair remember that all photoplotting facilities are able to read almost any data format you give, the only important point being that they are told the format of your data.

Here they come:

gerber table file
the name of the gerber table. If you specify the name of a file that already exists, the menu will be redrawn with the values from the file. This makes it very easy to make a slightly changed copy of an already existing gerbertable: First specify the original name, then move back to the field and give the new name!

data code
the 'alphabet' of the gerber data. The eia is oldfashioned, but some older gerber-photoplotters requires it. The ascii is more standard, and can be read by almost any computer (especially the PC's).

data unit
specifies whether you use metric or imperial units. The inch format is most widely used.

digits before
number of digits before the implicit comma in the coordinates. This limits how long a distance you can move in one step. (specially if you use mm as the data unit).

digits after
number of digits after the implicit comma. It limits the minimum step that can be used, that is the resolution. E.x. if you use inch and 3 digits after, the resolution is 1 mill.

data format
specifies whether each step is given relative to the last position (incremental) or relative to the zeropoint (absolute). The incremental format will theoretically make a smaller file.

suppress equal digits
if yes, coordinates are only included in the file, if they changes. This will make the file smaller.

skip leading zeroes
In order to make the data file smaller, you can skip zeroes in the beginning of the coordinates, e.x. 00120 will be 120.

skip trailing zeroes
Or you can remove the zeroes at the end of the coordinate. E.x. 00120 will be 0012, remember that there is an implicit comma.

end of block

each set of coordinates is terminated by the end-of-block. Normally it is <10> (linefeed) or * or both.

end of tape

The command that will signal the end of all the gerber data. Normally M00<10>.

heat polygon

the name of the aperture 'shape' that can be used for the thermal relief in the powerplanes. If your photoplotter do not have any apertures for heat isolations, you can use the special name empty. Then the gerber program will construct a thermal relief shape (see below).

iso polygon

the name of the aperture 'shape' that can be used for isolating pads electrically from the powerplanes. Normally it is a circle, but if your photoplotter has a doughnut you can specify iso. If you specifies empty the isolation will be constructed.

delta heat

the minimum clearance from the drilled hole to the thermal relief in the powerplane. Normally between 10mill and 40mill.

delta iso

clearance for the isolation pads. 10mill to 40mill.

thermal relief symbol

If you specified empty, for the heat polygon, this symbol specifies how gerber will make the constructed relief.

thermal relief angle

this specifies the opening in the constructed thermal relieves.

fill iso octagon

If yes, the constructed isolation pads will be filled.

calc heat from p

If yes, the size of thermal relieves will calculated from from the bigger of drillsize and plotterpolygon. If no, (the normal case) only the drillsize will be used.

circular interpolation

Some gerber photoplotters (very few), are able to construct circles. This can be used if you are plotting many arcs (very rare). This should normally be set to no.

rotate aperture

Some gerber photoplotters are able to rotate the apertures. It can be used if you have a lot of asymmetrical shapes.

This should normally be set to no.

macro expand

If the plotter is very fast to change aperture (not very likely), it might be better to construct shapes, using different apertures instead of using the default aperture. This should normally be set to no.

modal

Some photoplotters can 'remember' what the last operation was, (flash or draw or move), so it is not necessary to send operation commands all the time. This should normally be set to no.

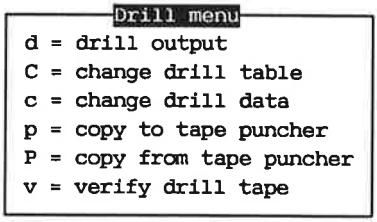
All these many parameters are further explained in the Supermax E-CAD IPL POST manual in the chapters gerber and in appendix2.

The menu **plot a gerber file** has already been described in the Paper Plotter chapter.

Understanding Drill:

> This chapter shows the programs used to control the output of data for drilling machines.

The programs to control the creation of drill data are found in the helpmenu in the drill menu:



The **drill output** menu is an alternative to makepost for creating data for NC drilling machines. It has already been described in the chapter **The helpmenus.**

The **change drill table** is the central tool to control the format of the drill data. Many of the parameters are completely equal to the parameters controlling the gerber output, so a stepwise description of all the parameters is not necessary.

A detailed description is found in the Supermax E-CAD IPL POST manual in appendix A6.

The **change drill data** menu is for converting the format of an already existing drill data file. The most interesting program is the **remove parity.** It removes the parity from the drill data, so it can be inspected with a text editor.

When the data should be move to the drilling machine, it can be done with punched paper tape or with pc-compatible floppy disks.

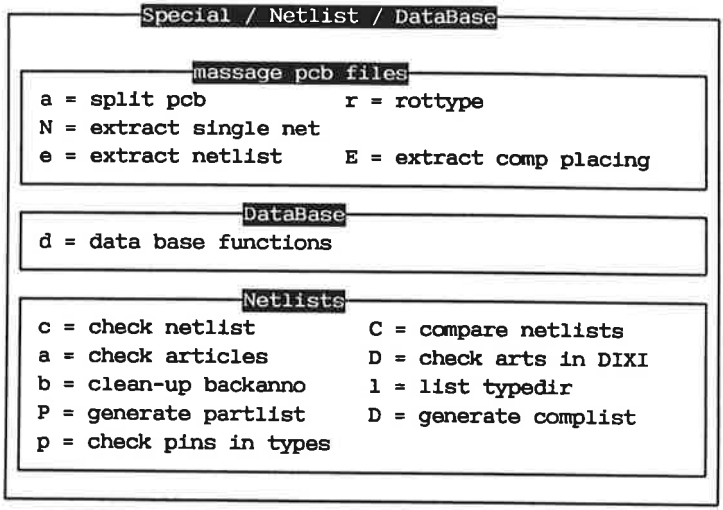
To make a punched paper tape, you should use the **copy to tape puncher** and the **verify drill tape** menus.

To make a pcfloppy you use the pcfloppy menus in the **unix menu.**

Netlist programs:

This chapter describes some of the menus to handle netlists.

The menus to handle netlists can be found in the helpmenu under the **special/netlist/database** menu:



If you have created a netlist with a text editor, or if you have converted a netlist from an alien system with one of conversion programs, it is necessary to check the correctness of the netlist.

This is done with the 3 menus **check netlist**, **check articles** and **check pins in types.**

Check netlist:

The first menu **check netlist** looks like:

```
          wirelist program
in file    = /usr/ipl/owlsave/mynet1
out file   = no yes

file name = /usr/ipl/owlsave/mynet2
format    = pinlist cmpwire cmplist db
```

It will read the original netlist, check if it is syntactically correct, and generate a new, nicely formatted netlist.

The different format options gives:

pinlist
The new netlist will contain a list of used component pins.

cmpwire
The new netlist will have inserted a component list, with a guess of what ipltype the components should have. This is very convenient, if you have only the nets but no component list. This is typically the case if the netlist is converted from an alien system.

cmplist
The new netlist is just reformatted.

db
The components articlename are used to search the DataBase for the device and the ipltype.

If the program detects any syntax-errors in the original netlist, the new netlist may not be complete. So when syntax errors are discovered they should be corrected before you proceed. In this case it is very handy with the shift-F16, function that invokes the 'vi' editor. If you do not feel comfortable with the **vi** editor you should read the UNIX manual called **system V: User guide**. It contain a very good introduction to the **vi** editor!

The next menu is **check articles**. It will read the netlist and check the database for the article name you have used. If they are not found in the database, it generates a textfile that can be used to insert the new components in the database. The program will try to find similar components that are already

defined in the database. E.x. 74HCTL06 will be matched with 7406, so all you have to do, is to correct the device name.

The last menu is **check pins in types**. It will read the netlist and check that the ipltypes used, can be read and that they contain the pins you have used in the netlist. This menu can used before you try to read the netlist with the **inputwl** command in the Supermax E-CAD IPL system.

Extracting netlist:

To extract a netlist from a pcb you have already made, you use the **extract netlist** menu. It will read the pcbfile and generate a netlist. You can choose to have only the component list or only the netlist or both. If you select the **all** option, even wloptions and component attributes are included.

The netlist is extracted in a rather simple format, but you can reformat it with the **check netlist** menu.

The **extract comp placement** can be used, if a complete new netlist must be read into an already made pcb. It makes a Supermax E-CAD IPL runfile, that moves the new components back to the original positions. E.x.

1) The old pcb is saved in a file.
2) The component placement is extracted and the command file /tmp/oldplac is made.
3) All the component (perhaps except the mounting holes) are deleted, with the Supermax E-CAD IPL commands **compselect** and **rm comp**.
4) The new netlist is read with **input wl**.
5) The component placement is restored by running the command file /tmp/oldplac.

Comparing netlists:

You can compare two netlist with the menu **compare netlist**. The parameter **method** changes the way the netlists are compared and the format the differences are listed. The favorites are probably **grpmatch2** and **grpmatch3**, where the last will compare the two nets independent of the netnames.

Creating partslist & component lists:

The two menus **generate partlist** and **generate complist** will make parts and comp lists.

You can change the report format by editing the files **/usr/ipl/tables/mkstk.from** and **/usr/ipl/tables/mkcomp.form.**

Hitchhikers guide to Makepost:

> This chapters goes into greater details on how to use the MakePost program.

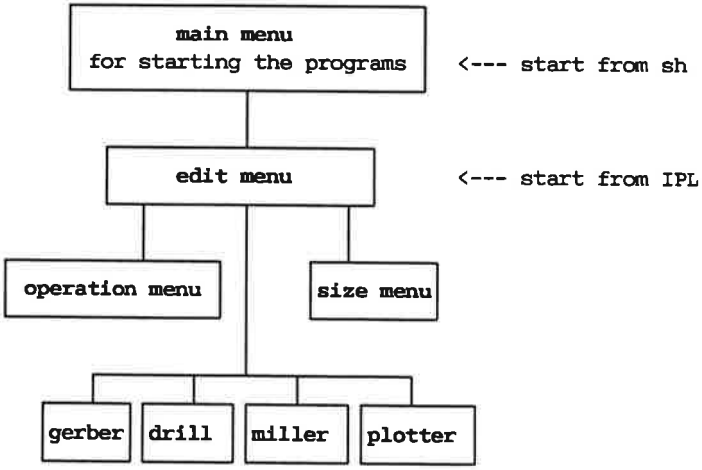
The makepost program is a high level program that controls the basic programs for generating output for photoplotters and drilling machines. It has two very important purposes:

1) It keeps track of placement of plots on films.
2) It keeps for every pcb a small file telling what must be made.

It is possible - and highly desirable - to make standards for the way postprocessing is done.

Makepost uses the gerber, drill, and plotter programs that has already been described.

The makepost programs is controlled from a set of menus. The menus are organized like this drawing:



As you can see a somewhat complicated hierarchy of menus.

When you start the makepost from inside the IPL **PostMenu**, you

will enter the **edit menu** directly. If you start makepost from the helpmenu or from the UNIX shell, you will go to the **main menu.**

The **main menu** will ask you for a pcb file name and a post file name. The latter file is the small file that 'remembers' how to create output data for this pcb. If you have not made any postprocessing on the pcb, makepost will propose a standard file called /usr/ipl/tables/prepostdef. The postfile can be either a file specifically corresponding to the specific pcb, or it can a standard file. The standard files are the ones you see in the popupmenu when running the makepost from the Supermax E-CAD IPL system.

When you have entered the two filenames, you get into **main menu.** From there you can start the actual programs drill, gerber etc.

Or you can enter the **edit menu** by pressing **e.**

From the **edit menu** you can move into the menus:
drill edit, gerber edit, milling edit or **plotter edit.**
These menus specify the information needed to make drill, gerber etc. They partly corresponds to the single helpmenus, where you specify gerbertables, scale factors etc.

From the **edit menu** you can also enter the **size menu.**

The **size menu** tells makepost about the available filmsizes, and about the pcbsize. You can give many alternative filmsizes. This allows makepost to select a filmsize that will give the best usage of film area. The pcbsize can be given either explicitly or makepost can calculate it from one or more lays or from the placement of some corner components. This is like the **compdrawing** or the **drillfilm helpmenu screen** (see the chapter Paper Plotting).

The most important menu is the **operation menu,** it is in the operation menu you specify what makepost should do.

In the **operation menu** you add a line for each photoplot you want to make, and a line for each drill data file. You can even add lines for paperplot, but paperplots are probably easier to make from the helpmenus.

A typical operation menu looks like:

```
comment   = typical example
photoplot = 0,255 p
photoplot = mir 1,255 p
photoplot = 11 p
photoplot = 255 s + 8 p
drill     = 255
```

The only thing to specify is the kind of operation:

comment	just a comment
photoplot	gerber
drill	drill
milling	milling
plot	paperplot

and the laylists and polygons.

Makepost will automatically calculate the placement on the film(s), and start the gerberprograms and drillprogram.

Adding lines to the operation menu is done by pressing function button **f9**.

Then makepost will ask for the kind of operation. Please notice that all the dialog with makepost is done on the bottom line of the screen. When makepost asks for the operation it shows a frame with help, so you can see what can be entered at this moment.

After selecting a kind of operation, makepost asks for the laylist and eventually the polygons. Again a box with helping information is displayed.

Deleting a line from the operation menu is done by pressing **shift-F9**.

Together with the laylists, you can give some extra *keywords* to control some of the special features of the gerberprogram.

Examples:

mirroring the plot is done simply with the word *mir*:

```
photoplot = mir 1,255 p
```

append several plots on top of each other by using the *+*:

```
photoplot = 225 s + 8 p
```

clipping the silkscreen using *clip lay(s) p,v dist*:

```
photoplot = clip 255 p,v 40 11,12 p
```

USERS GUIDE

TO:

SUPERMAX E-CAD VERSION 3.0

RUNNING UNDER X WINDOWS

Dansk Data Elektronik A/S
Supermax EDA Division

This is the user manual on how to use Supermax E-CAD within the X windows environment.

Table of contents:

1: Preface.

This chapter does not have the intention of describing how the X windows system works nor how to specify the huge amount of options provided with the X windowing system. This is very well described in the X window System User's Guide Vol. Three (3rd edition from O'Reilly & Associates, Inc).

The rest of this chapter is split into three paragraphs, each explaining the different window managers Supermax E-CAD can use on different hardware platforms:

Motif on the Supermax platform.
OpenLook on the SUN platforms.
Uwm on the Sony News and 386/486 Interactive Unix platforms.

2: Using Supermax E-CAD with the Motif window manager.

2.1 Login and start up session.

After turning on your X terminal the system prompts you for your user name and password. Enter these to start your X windows session. This will automatically start up several different programs.

2.2 Screen outline.

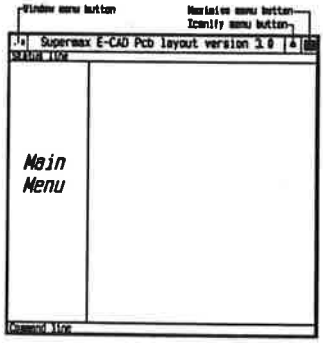
After the login session is completed the X session starts and after some time the following program will appear on the screen:

A clock in the lower right corner.
A helpmenu program (iconified).
A xterm in the lower left corner.
A icon box collector in the lower right corner.
And the Supermax E-CAD window at the upper left corner.

All these program are running at the same time. Another one is running in the background (with no output): it is the Motif window manager. This program is the one that controls the screen or display. Which programs that will be started is described in a setup file. It is called .xsession and is placed in your home directory. For further information read appendix A in vol. three of the X Window System Guide (3rd edition).

2.3 Main Supermax E-CAD window.

The main Supermax E-CAD window will have the outline as shown below:



As you can see there is a border or frame around the window. At

the top there is the 'title bar' with three buttons in it. One at the left and two at the right. The left most button is the *window menu button*. It is used for manipulating the layout of the window. The following functions are available: restore, move, size, minimize, maximize, lower and close. These actions are displayed and selected by moving the pointer (the mouse) and press the left pointer button at the 'window menu button'. A menu will appear and you select the desired item by clicking the first menu button at the function. To skip the menu just click outside the menu.

Some of the functions in the 'window menu' have shortcuts:

Moving the window: Point in the 'window title' with the first pointer button and whilst keeping it pressed move the pointer and release the button at the desired place.

Resizing the window: At the corners you see little corner marks. If you move the pointer at one of them, press the first pointer button and whilst keeping is pressed move the corner to the desired place and release the button.

Iconify or minimize the window: At the right top side of the 'menu title bar' there is two buttons. If you click at the left most the window will iconify. That is the window will disappear on the screen and the icon in the icon box will get a bigger size. Whatever function the Supermax E-CAD program was executing will continue, the graphics will not appear until the window is de-iconified.

De-iconify or restore the window: By double clicking (click twice with a short time between) on the icon the window will re-appear again, at the same place and size as before it was iconified.

For further information read appendix C in vol. three of the X Window System Guide (3rd edition).

2.4 How to look at the text output from Supermax E-CAD.

All listings and messages will appear in the xterm window in the lower left corner. Normally you will only see the last 4 lines, but you could at any time move the pointer in the window and then you can see 24 lines. If you want to see more than 24 lines, move the pointer in the scroll bar (the bar at the left in the xterm) and press the first pointer button and whilst keeping the button pressed move the pointer up and the text will scroll up.

2.5 Root menus.

Root menus are menus that can be called at any time within the X window system. The display not covered by a window is where you can activate the root menu's. If you move the pointer to the root window and press the first pointer a menu will appear. Whilst keeping the button pressed you can move the pointer up and down in the menu fields. By releasing the button you select the action

in the field. If the menu field have a triangle at the right the field have a sub menu. To see this sub menu move the pointer to the field and the sub menu appears. To select a field in a sub menu move the pointer to the field in the sub menu and release the pointer.

The following actions are available with the first button:

Root Menu	Title.
Shuffle up	Rotate the window stack one op.
Shuffle down	Rotate the window stack one down.
Refresh	Send refresh event to all windows.
New Window >	New Window sub menu.
Clients >	Clients sub menu.
b/w xterm	Start a black and white xterm.
Restart...	Restart the Motif window manager.
Exit	Exit the X session.

As you can see the fields 'New Window' and 'Clients' has sub menus. Selecting a field in the 'New Window' sub menu will all create a xterm some in different colors. The 'Client' sub menu also appear when the middle button is pressed in the root window.

The following actions are available in the 'Clients' menu:

E-CAD clients	Title.
Supermax E-CAD	Start Supermax E-CAD.
Helpmenu	Start Helpmenu.
Workview	Start Workview, if installed.
b/w xterm	Start a black and white xterm.

These menus are defined in the Motif setup file .mwmrc and is placed in your home directory.

2.6 Exiting Motif - logout session.

Point in the root window and press the first pointer button and whilst keeping it pressed toggle down to *exit* and release the button. Then move the pointer to the *OK* field and click a pointer button and all your windows will be stopped and a login session is ready to start.

3. Using the OpenLook window manager.

3.1 Login and start up session.

After turning on your Workstation and entering your user name and password, you should enter start and return. Now the X window system OpenWindows is started and will use the OpenLook window manager.

3.2 Screen outline.
After the login session and start up of the X window system the following program will appear on the screen:

A clock in the lower right corner.
A xterm in the lower left corner.
A cmdtool running a shell (acting as the console) in the lower right corner.
And the Supermax E-CAD window at the upper left corner.

All these program are running at the same time. Another one is running in the background (with no output): it is the OpenLook window manager. This program is the one that controls the screen or display. Which programs that will be started is described in a setup file. It is called .openwin-init and is placed in your home directory.
For further information read or skim OpenWindows Version 2 User's Guide, X11/NeWS Version 2 Server Guide and DeskSet Environment Reference Guide.

3.3 Main Supermax E-CAD window layout.
The main Supermax E-CAD window will have the outline as shown below:

As you can see there is a border or frame around the window. At

the top there is the 'title bar' with one button in it, this is called the *window menu button*. It is used for manipulating the layout of the window. The following functions are available: Close, Full Size, Back, Refresh and Quit. To select one of these actions move the pointer to the title bar and press the *menu* pointer button (that is the right most button) and a popup with the mentioned actions is displayed. A action is selected by moving the pointer to the field and clicking the *select* pointer button (that is the left most button). To skip the menu just click outside the menu.

Some of the functions in the 'window menu' have shortcuts:

Moving the window: Point any where at the window border and press the *select* pointer button and whilst keeping it pressed move the pointer and release the button at the desired place.

Resizing the window: At the corners you see little corner marks. If you move the pointer at one of them, press the *select* pointer button and whilst keeping is pressed move the corner to the desired place and release the button.

Iconify or minimize the window: At the left top side of the 'menu title bar' there is a button. If you click at this button with the *select* pointer button the window will be iconified. That is the window will disappear from the screen, but the icon window will appear on the workspace. Whatever function the Supermax E-CAD program was executing will continue only will the graphics not appear until the window is de-iconified.

De-iconify or restore the window: By double clicking, that is click two times with a small time between, on the icon the window will re-appear again, at the same place and size as before it was iconified.

For further information read OpenWindows Version 2 User's Guide.

3.4 How to look at the text output from Supermax E-CAD.

All listings and messages will appear in the xterm window in the lower left corner. Normally you will only see the last 4 lines, but you could at any time move the pointer to the Supermax E-CAD border and press the *menu* pointer button, toggle down to the field *back* and select it. This will get the underlaying xterm in the front and you can now see 24 lines. If you want to see more than 24 lines, move the pointer in the scroll bar (the bar at the left in the xterm) and press the *adjust* pointer button and whilst keeping the button pressed move the pointer up and the text will scroll up. To get the xterm in the back, move the pointer to the xterm border and press the menu pointer button and select the back field.

3.5 Workspace menus.

Workspace menus are menus that can be called at any time within the X window system. The display not covered by a window is where

you can activate the workspace menu's. If you move the pointer to the workspace and press the *menu* pointer button a menu will appear. Whilst keeping the button pressed you can move the pointer up and down in the menu fields. If the field have a triangle at the right a sub menu exist to that field. By releasing the button you select the action in the field.
The following actions are available with the first button:

Workspace	Title.
E-CAD utilities >	Submenu for Supermax E-CAD Utilities.
Programs >	Program submenu.
Utilities >	Utilities submenu.
Properties...	Properties.
Exit...	Exit the OpenWindows system.

As you can see the fields 'E-CAD Utilities', 'Programs' and 'Utilities' all have sub menus.

The following actions are available in the 'E-CAD Utilities' menu:

E-CAD Utilities	Title.
Helpmenu	Start the helpmenu.
Supermax E-CAD	Start Supermax E-CAD.
Command tool...	Start a command tool (shell).

These menus are defined in the OpenLook setup file .initwin-menu placed in your home directory.

3.6 Exiting OpenLook - logout session.

Point in the workspace and press the *menu* button and the 'workspace menu' will appear. Whilst keeping the menu button pressed move the pointer down to the *Exit* field and release the menu button. Then move the pointer to the yes field of the 'confirm popup menu' and click the menu button on it and the OpenLook will exit shutting all programs down. Your are now in the login shell and must enter logout and return to exit this login shell.

4. Using the Uwm window manager.

4.1 Login and start up session.

After turning on your Workstation and entering your user name and password, you should enter xinit and return. Now the X window system is started and will use the Uwm window manager.

4.2 Screen outline.

After a successful login and starting the X window system the following programs will appear on the screen:

A xterm at the lower left corner.
And the Supermax E-CAD window at the upper left corner.

The two programs are running at the same time. Another one is running in the background: it is the Uwm window manager. This program is the one that controls the display or screen. Which programs that will be started is described in a setup file. It is called .xinitrc and is placed in your home directory. For further information read appendix B in vol. three of the X Window System Guide (3rd edition).

4.3 How to look at the text output from Supermax E-CAD.

All listings and messages will appear in the xterm window in the lower left corner. Normally you will only see the last 4 lines, but you could at any time move the pointer in the window and raise the xterm window (see 4.4 for the raise action) and then you can see 24 lines. If you wan to see more than 24 lines, move the pointer in the scroll bar (the bar at the left in the xterm) and press the first pointer button and whilst keeping the button pressed move the pointer up and the text will scroll up. Returning to the Supermax E-CAD window either lower the xterm or raise the Supermax E-CAD window.

4.4 Root menus.

Root menus are menus that can be called at any time within the X window system. If you press the 'alt' key *and* the first button at the pointer the 'WINDOW OPS' will appear. Whilst keeping the buttons pressed you can move the pointer up and down in the menu fields. By releasing the buttons you select the action in the field.

The following actions are available with the first button:

WINDOW OPS	Title.
(De)Iconify	Iconify window.
Move	Move window.
Resize	Resize window.
Lower	Lower window.
Raise	Raise window.

If you press the middle pointer button instead of the first one the 'E-CAD clients' menu appears and the following actions are available :

E-CAD clients	Title.
Helpmenu	Start Helpmenu.
Create Window	Start xterm window.

These menus are defined in the Uwm setup file .uwmrc and is placed in your home directory.

4.5 Exiting Uwm - logout session.

To exit the X window system either to logout or to go to the login shell you have to exit the main Supermax E-CAD program. This is done by selecting the menu field 'EXIT ipl' or 'EXIT schema' in the Supermax E-CAD IOmenu. This will return to the login shell. To end the login session type logout and return and a new login session can begin.

5. General comments.

5.1 Entering composed characters.

To be able to enter composed characters, as ø and ÿ, the applications font must be a 8 bit font. The default fonts specified in the released .Xdefaults file are 8 bit fonts.
To enter a æ use the following key strokes:
ctrl-a and a and e or the compose key and a and e.

5.2 Clearing and redrawing of windows.

Clearing and redrawing is only implemented on the main E-CAD application. Clearing the main E-CAD window - pcb part, use the *home* key. Redrawing the main E-CAD window - hole window, use the *shift home* key.

Appendix A: Setup files for the Motif window manager.

All files should be placed in your home directory.

.Xdefaults	setup file for colors,borders,icons, ...
.Xkeyboard	setup file for the keyboard.
.mwmrc	setup file for root menus and key bindings.
.xsession	setup file for start up session after login.

Appendix B: Setup files for the OpenLook window manager.

All files should be placed in your home directory.

.xinitrc	setup file for X windows start up session.
.openwin-init	setup file for X windows start up session.
.openwin-menu	setup file for workspace menus.
.Xdefaults	setup file for colors,borders,icons, ...

Appendix C: Setup files for the Uwm window manager.

All files should be placed in your home directory.

.Xdefaults	setup file for colors,borders,icons, ...
.uwmrc	setup file for root menus.
.xinitrc	setup file for start up session after login.

Appendix D: Command line options for Supermax E-CAD X window programs.

IPL, main Supermax E-CAD program:

-c install_colormap, do_not_install_colormap
Intsall the specified colors in a private colormap, this will give correct colors for crossing tracks, but will also give color flashing when moving the pointer outside the main ipl window.

-g[eometry] x-size * y-size[+-xoffset+-yoffset]
-geo 80x24
-geo 80x24-0+0 (upper left corner)

-t X[386-uwm], Xm[otif], Xo[penlook]

Default values for the following items are read from the .Xdefaults file:

iplprog.geo	- format as above.
iplprog.helpfont	- font name, as sysparm xwhelpfont.
iplprog.statusfont	- font name, as sysparm xwstatusfont.
iplprog.iplmenufont	- font name, as sysparm xwiplmenufont.

Fonts are specified by the appropriate sysparms.
Colors are specified by the color setup files placed in /usr/ipl/commands and called from the initipl command file.
The bitmap used when the program is in the iconified state is placed in /usr/ipl/bitmaps/pcb_layout.
The bitmap used for the 'busy' cursor is placed in the file /usr/ipl/bitmap/busy_cursor.

The bitmap used for the 'text' cursor is placed in the file /usr/ipl/bitmap/text_cursor.

xhelpmenu, front end to all pre and post processing programs:

-icon[ify] with the bitmap /usr/ipl/bitmaps/helpmenu
-font 9x15
-geo[metry] x-size * y-size[+-xoffset+-yoffset]
  -geo 80x24
  -geo 80x24-0+0 (upper left corner)

Default values for the following items are read from the .Xdefaults file:

helpmenu.geo - format as above.
helpmenu.font - font name.
helpmenu.textcolor - text color name.
helpmenu.menucolor - menu color name.

The bitmap used when the program is in the iconified state is placed in /usr/ipl/bitmaps/helpmenu.

xmakepost, front end for generation of photo, drill, profiling and plotter data:

-icon[ify] with the bitmap /usr/ipl/bitmaps/makepost
-font 9x15
-geo[metry] x-size * y-size[+-xoffset+-yoffset]
  -geo 80x24
  -geo 80x24-0+0 (upper left corner)

Default values for the following items are read from the .Xdefaults file:

makepost.geo - format as above.
makepost.font - font name.
makepost.textcolor - text color name.
makepost.menucolor - menu color name.

The bitmap used when the program is in the iconified state is placed in /usr/ipl/bitmaps/makepost.

End of chapter.