PCB Developer's Individual Assistant

Welcome to PCB Developer's Individual Assistant for Windows
Version 2.1 in June 1996

Installation
General information
Tutorial

Commands
Toolbar
Statusbar

Mouse and keyboard input
Printing and plotting
Hints

License and Purchasing Information

For information on how to use Help, choose Help | Using Help.

Installation

Before you start the installation program you should take care that your system fits to the requirements listed below.

Hardware requirements :
- Windows 3.1 or later
- 386   (387 or 486DX desirable)
- 4 MB RAM
- 2 MB free disk space
  32-bit Edition only
- at least WIN32S (or Windows 95, NT)
- 8 MB RAM

Note : If you are using the 32bit Edition with Windows 3.1, you have to install Win32S before executing setup32.

Installation from floppy disk:

To install the program,

1) Start Windows.

2) Put the distribution disk in your floppy drive.

3) Choose File | Run from Program Manager.

4) Type **a:\setup** (or   **a:\setup32** if you are installing the 32bit Edition).
   (If your floppy drive is not drive A, type the appropriate letter instead.)

5) Choose OK.

6) Follow the instructions on the screen.

Installation from zip-file:
To install the program,

1) Create a temporary directory (e.g. C:\PIATMP):
        md C:\PIATMP   *<enter>*
2) Copy piapcb.zip to the temporary directory.
        copy *<source directory>* \PIAPCB.ZIP C:\PCBTMP   *<enter>*
3) Unzip the file
        pkunzip   C:\PIATMP\PIAPCB   *<enter>*

4) Start Windows (if Windows is not already running).

5) Choose File | Run from Program Manager.

6) Type **C:\PIATMP\SETUP**

7) Choose OK.

8) Follow the instructions on the screen.

Starting PCB Developer's Individual Assistant

Open the P.I.A. group by double clicking the P.I.A. group symbol in Program Manager. Now

start the PCB Editor by double clicking on the icon ( ).

Commands

Available menu commands

<u>File</u>	(load, save, print, plot)
<u>Edit</u>	(move, delete, undo)
<u>Screen</u>	
<u>Objects</u>	(ICs,...)
<u>Items</u>	(pads, lines)
<u>Router</u>	(route, ripup, connections)
<u>Block</u>	
<u>Grid</u>	
<u>Help</u>	

Some commands can be also accessed using the <u>toolbar</u>.

File

File orientated operations
New :   Clear current drawing.
Open :   Open an existing drawing.
Save :   Save current drawing.
Save as :   Save current drawing with a new name.
Append :   Add an existing drawing to your current drawing.
                    You will be asked to select a new origin.
Save configuration : Save current configuration.

Exit :   Exit the program.

Print commands
Print :   Print current drawing.
Parts list :   Create a list with all object IDs and comments.
                    Notepad will be launched and views the temporary file with the parts list
                    (pcb_part.tmp).To print the parts list select Files|Print from within Notepad.
                    Note : Changes made to the temporary file will not be recognized. To
                    change IDs or comments call <u>Edit|Set/edit object text</u>.

Import >   (32bit Edition only)
      Netlist (ASCII) : Create connections from ASCII netlist.

Export >   (32bit Edition only)
      HPGL :   Export current drawing to HPGL.
      NC drill file :   Create NC drill file (ASCII).
      Gerber (auto) : Export current drawing to Gerber. (RS-274X format will be used.)
      Gerber (man) : Export current drawing to Gerber. (User values can be selected in
                  the <u>gerber dialog</u>. RS-274D format will be used)
      Netlist (ASCII) : Save all connections to ASCII netlist.

Related sections:
    <u>Printing and plotting</u>
    <u>Gerber files</u>
    ***<u>NC Drill files</u>
    ***<u>Netlists</u>

Edit

Overview over all available sub topics :

General edit commands
- Undo
- Redo
- Move
- Move single part
- Move point
- Delete
- Delete special

Special edit commands for objects
- Edit / set object text
- Edit pin numbers
- Assign object pin numbers

Special edit commands for lines and pads
- Edit line / corner
- Delete corner
- Width (track)
- Width (single line)
- Edit pad
- Process entire blocks

General edit commands

Undo : Undo last action (10 steps or more possible).

Redo : Restore the previous changes made by Undo.

Move : Execute this command to move any part (pads,lines, ICs,...). ↔
> The first mouse click selects the part. Now move the part to the new location. The second click will fix the part to the new location.
> It is also possible to rotate parts by pressing the mid button. (If do not have a 3 button mouse, select "rotate" from the local menu.)

> Affected lines will be also moved.

> If you have problems to select a line near one or more pads execute 'Edit line' to move this line.

Move   Single Part : Move single parts (pads, ICs,...). Affected lines will NOT be attached.
> (See example below.)
> If you are using the auto router **be careful** with this command.
> Disconnect parts with 'move single parts', if you have moved two parts over each other. Afterwards move all connected lines manually. This is the only purpose for this command.

> If you are not using Route or Ripup (NEVER !), you may also use this command for other purposes.

 Hint   :   If you have selected the wrong part, click on the right mouse button and select "cancel current action" from the local menu.

Example for move / move single :

Origin   Move   Move single
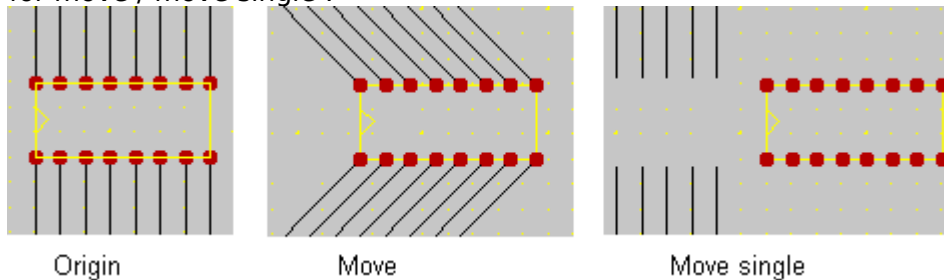
Move point 🐾**:** Use this command to move all items on the selected point to another point. If an item belongs to an object, only the affected item will be moved.
> The intention of this command is to edit objects. This is the only way to edit objects without deleting and reconstructing the object .

Move block : (see block operations)

Edit frames : (see ground frames)

Delete : Delete pads, tracks, objects, connections. (no tracks)

Delete special > X :   Delete only X.
                    (The entire object will NOT be deleted, if an object is affected.)
                    Use this command to delete single parts of an object.

   Note: You can activate a "confirm on delete" requester in the <u>main setup</u> for each
   delete. Otherwise the requester appears only if more than one part would be affected.

Special edit commands for objects

Edit/set object text:
- You can edit your object IDs and comments by this command.
- You can also create an object ID and comment if the object has none.

Click on the object and the <u>object text dialog</u> appears.

Edit pin numbers: Edit single object pin numbers.

Click on a pad which is part of an object. A requester with the old pin number will appear. Change or discard the old pin number. All processed pins will be marked. To remove the marking press the redraw button ( ) .

Assign object pin numbers: Edit single object pin numbers.

The intention of this command is to make it easier to change or assign all pin numbers of an object. Normally all pin numbers are enumerated from 1 to x. With this command you can assign number 1,2,3,... by just clicking on pads. No requester will appear. The pin number starts with 1 and will be increased after each click. If you click on another object the current number will be set automatically to 1. All processed pins will be marked.   To remove the marking press the redraw button ( ) .

The pin numbers will be displayed if you select resolution one or two.

Important : Normal pads (which are no part of an object) can NOT have Pin numbers. If you need a single pad with a pin number use the pad object from the standard library or create your own object. This could necessary for e.g. power supply pads if you are using Netlist Import or Export.

Special edit commands for lines and pads

Edit line/corners : (Move, add or delete corners/ move ends)
    A click on the left mouse button selects a corner for being **move**d.
    The second click fixes the corner. If there is no corner, a corner will be **insert**ed.
    If you have selected the wrong corner , click on the right mouse button and select
    "cancel current action" from the local menu.

    A click on the middle button   (or "m" key if you have none) will **delete** a corner.
    Note : The edit line command affects only the current layer.

Delete corners : A click on the left mouse button deletes a corner.

Width (track): (tracks are connected lines - beginning and end are matching!)
    Use this command to modify the width of an entire track. Click on one single line to
    select a track. The entire track will be processed (until the next   branching or pad).

Width (single line): Change the width of a single line.

Edit pad : Edit diameter , outline, drilling hole of a pad.
    Click on the pad and the <u>edit pad dialog</u> appears.
    Change the displayed values and press OK to modify the highlighted pad. Press
    "entire object" to modify all pads of the affected object if the highlighted pad belongs
    to an object.

Process entire blocks

It is also possible to change pads or line width of an entire block.

    1) Execute Blocks | Mark.
    2) Mark one or more blocks
    3) The marked area will be highlighted
    4) Execute Edit | Edit Pad or Width
    5) All highlighted pads or lines will be processed
    6) Execute Blocks | Unmark or press the redraw button to remove the selection.

Screen

Change settings for the current screen

Redraw: Redraw entire screen.

Change layer: Flip from bottom to top side or vice versa.

Zoom in : Magnify screen (factor 2).

Zoom out: Reduce screen (factor 2).
Resolution X: Set resolution to X   (a higher value means smaller parts).

Colors: Change current colorset

Default drilling holes : Change the default values for drilling holes (ICs,...).

Main setup : Change current settings (layer, metrics, ...).
Auto scroll and fill mode should be deactivated on slow machines.
If the local menu is deactivated the right mouse button will cancel the current action.
The available settings are listed in the Settings sections.

Clean up : Remove unresolved parts from screen.
Be careful with this command. You should **save** your drawings before executing.
Use Clean up only if there is a real problem.
Please read also the special Clean up section before.

Clean up

Clean up : Remove unresolved parts from screen.
Be careful with this command. You should **save** your drawings before executing.
You can use this command to **delete** :
- parts outside the drawing area (could happen easily with the copy/move block command.)
- empty object IDs and comments
- lines with length or width 0
- pads with diameter 0
- object IDs with no corresponding object (should not happen).

(These parts are not displayed. You can recognize them if the counters in the system info are to high.)

Clean up discovers :
- corrupted object structures (move,delete,... do not work properly; should not happen)
- corrupted data (senseless error messages; should not happen).

Command : Screen|Colors

Objects

Related sections :

The following list describes the available commands for placing objects:

Setup object (arrow button) :
 Select standard parameter for objects listed below (e.g. pins , type, RM, pad diameter).
The IC, Tr, R, C buttons will call the corresponding commands.

IC :   Draw IC   (with std parameters).

Tr :   Draw Transistor.

R :   Draw Resistor.

C :   Draw Capacitor.

Other objects (SMDs,...) can be accessed by the object load command (see below),
Note : To draw a line between two normal IC pins, select 62mil diameter.

Bridge: Generate bridge for single sided boards.
(Pads with 62mil / 1.6mm diameter will be created automatically.)
Note : Bridges are treated as objects. The normal move command affects both pads.
To move one pad execute move single point.

Text: Place text.
The available settings are listed in the   Settings section.

Object load : Load object from disk (e.g. sub-d connector, SMDs) and position it.
Object Save : Save a rect in your drawing as an object.
(You will be asked to mark a rect and a hot spot.)
See also : Create your own objects.

Rotate object : To rotate an object press the mid button or   select "rotate" from the local menu   and the object will rotate 90 deg clockwise. ('r' or 'm' on the keyboard)

Note : If you place an object, you will be asked for an ID and comment. You can avoid this dialog with the "hide dialog" option. This option is also available in the main setup dialog.

Create your own objects

The following guidance describes the use of user defined objects (which are not included to the program) for your special purpose.

How to create your own objects :

- Draw all pads , lines, ... for your object.
- Change to the symbol layer (see <u>main setup</u>).
  Draw the symbol for your object using the normal line and edit commands. (The line width will be ignored and set to 1.)
- Execute Objects|Object save.
  Select a rectangle with your object. (Only the elements inside the rectangle will be saved.)
  Select the reference point (e.g. pin 1 or something else).
- Done.

How to use your own objects:

- Execute Objects|Object open.
- Place the object like any other object (ICs, ...).

Note :
- For SMDs square pads are available.
- Object ID and comment will not be saved. You will have to choose them if you place the object.

Libraries

Overview :
- Load library
- Add part to library
- Rename part in library
- Delete part from library
- Create new library

- Open library window
- Open large library window
- How to change the name of an entire library

General:
    A library is a set of many objects. All objects are stored in single library file <name>.lib. (There is also an index file <name>.idx). The advantage of libraries is that you have quick access to all needed objects.
There are also libraries for special purposes available (e.g. smd.lib).
To access a specific library execute "Open library window" or "Open large library window" if the library window is not already open. Press the "library" button and select your desired library.
A preview of the library parts will be shown on the right half of your screen if click on the part's name in the library window.

Load library :
    Execute Objects | Load library to load your desired library. This command is also available from the library window (library button , top line).

Add part :
    This command is used to add a new part to the current library. Execute Objects | Add part to lib. A requester for the part name will appear. You can enter a name with up to 20 characters (this is no file name, spaces are allowed). The following proceeding is the same as in "Object Save".
1) Mark a rectangular area which is containing your new part.
2) Select a reference point (e.g. pin 1). This reference point is used for positioning of lib parts.
3) Done.

Hint : If the area around your new part contains some additional   pads and lines, which are not belonging to your part, use the "copy block" command. Copy the area to another location and delete all pads and lines which are not belonging to your new lib part. Now you can execute "Add part to lib".

A possible error message is "Name already in use" : Choose another name for your new library part. Duplicated names are not allowed.

Rename part:
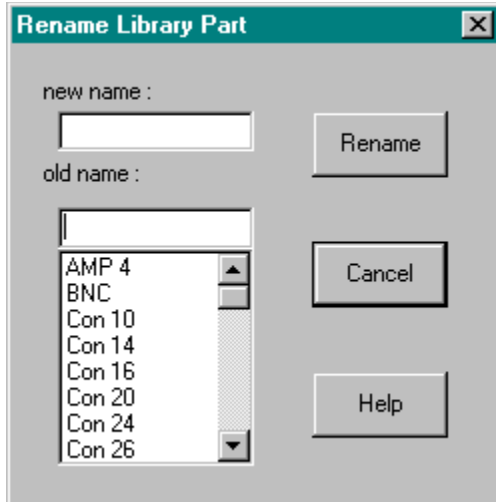
This command is used to change the name of single parts in the current library. Select the old name by clicking on the name in the list box. (Do not try to enter the name in the old name field. This is not allowed.) Enter the new name in the new name field. Press "Rename" to record your changes or "Cancel" to discard   your changes.

Delete part:
Use this command to remove one single part from the current library. A requester with a list of all parts in the current library will appear. Click on the the part which you wish to remove. Press "Remove" to delete the part or "Cancel" to close the dialog without deleting. A confirmation requester will appear before the part is deleted finally. Backup files will be created automatically. If the wrong part has been deleted or if an error occurred while deleting, you can recover the original library by renaming the backup files. (Rename <name>.li2 and <name>.id2 to xxx.lib and xxx.idx.)

Create new library:
To create a new library file execute Objects | Create new library . A requester for the new lib name will appear. Please enter the name without an extension (.lib). The new lib will be created after pressing OK and will become the current library.

Open library window :
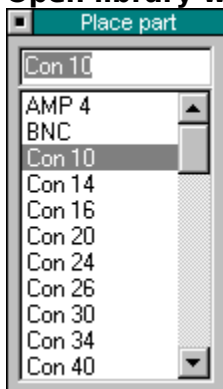
Execute this command to show the library window. To access a specific part of the library click on the part's name in the list field. A preview of the library parts will be shown on the right half of your screen. To place the part move it to the desired position and press on the left mouse button.

Open large library window (right below)**:**
This command will display a larger version of the normal library window. This window has

some additional buttons and fields.
- **library** button : Load another library. A special requester will appear.
- **less** button : Reduce the size of your library window. (The window will be expanded to the largest size if the current window has the smallest size. 3->2->1->3...)
  Three different sizes are available :
  1) library list only
  2) additional lists with standard ICs and standard resistors and capacitors
  3) additional popup lists with lines and pads.
- **list fields** for library parts, ICs, R, C.
  To access a part from those lists just click on its name.
- **list fields** for pads and lines.
  To access a pad or line click on the empty field. A larger list will popup. Now click on the desired pad or lines.
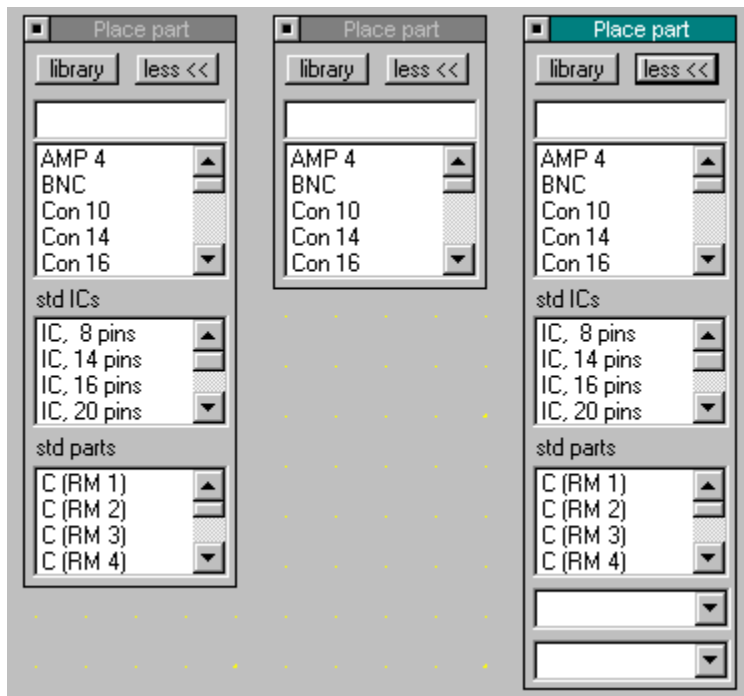
How to change the name of an entire library:
There is no special command for this. But the name can be changed manually. Just rename the lib file (<name>.lib) and the corresponding index file (<name>.idx) with the DOS RENAME command or with File Manager.

Items

Setup element (arrow button) :
> Select standard parameter for pads and tracks (pad diameter and thickness).
> The pad and   track buttons will call the corresponding commands.
> The square pad option is not available for predefined pads.
> Pads normally have drilling holes. If do not need them activate the "no drilling hole" option.
> Note: If the pad is displayed light red instead of red, the drilling hole is larger than the pad !!

Pad : Draw a pad   (with std parameters).
Square Pad :   Draw a square pad   (with parameters set in Pad X).

Line: Draw lines.
> A double click on the left button will terminate the line.
> You can also use "cancel action" from the local menu or the right button if the local menu is deactivated to terminate the line. This will be useful if snap grid is deactivated.
>
> **Note:** The mid button (or 'm' key) will fix the line to the next pad.
> Vias can be generated with the "Via" command from the local menu (or 'v' key). The via will be located at the current cursor position.
>
> Beginning and end should be pads, otherwise ripup is not   possible and block operations could truncate the track.

Circle: To draw a circle   (with std line width):
- click on the center,
- draw a circle with the desired diameter and
- fix the circle with a second click.

Arcs: To draw an arc   (circle segment):
- click on the center,
- click on the beginning ,
- draw an arc with the desired angle and
- click on the end.

> Draw two arcs if the desired angle is larger than 180 deg.
>
> Circles and arcs can be moved with the normal move command. But it is not possible to change the diameter at present. (Delete the old circle and draw a new one in that case.)

Ground frames : There is a special <u>section for ground frames</u> available.

Ground frames

Ground frames :   Draw a rectangle filled with copper. Areas containing lines and pads will not be filled. You may connect some pads and lines to ground (see below) to avoid the gap around those items.

Pad <--> ground : Connect or disconnect a pad. Only the current layer will be affected.

Line <--> ground : Connect or disconnect a line.

Ground frame gap :   Modify the gap around lines / pads in all ground frames.

Copper frames :   Draw a **solid** rectangle (filled with copper).

To speed up screen update, you can activate "draft frames" in <u>main setup</u>.

Edit frames

You can edit a frame by moving the light yellow line in mid of the frame.

- **Move** : Select Move (⬌) ,click in the mid of the line and move the frame to the desired position.
- **Change size** : Move only one end of the yellow line.
- **Delete** : Select Delete ( ) and click on the yellow line.

Example:

Router

Connection :   Draw connections (will become a track after routing).

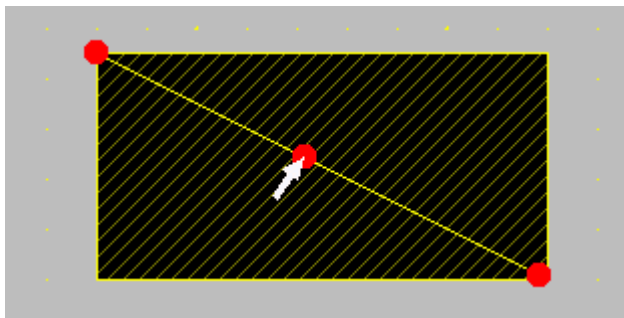**Online route :**   Tool to draw connections. Each connection will be routed immediately. To modify the options execute "setup online router". The router options dialog will not appear .

Setup online router :   Use this dialog to choose all options for online routing.

Route :   Route all connections.(A dialog with the <u>router options</u> will appear.)

Ripup all: All manual drawn tracks and routed connections will become connections.

Ripup single: Ripup single tracks manually.

Route a connection manually : Just draw a normal line and delete the connection afterwards (Edit | Delete special | Delete connection). Beginning and end should be pads. Otherwise Ripup will not be possible.

Note : If you want to limit the routing area draw a line around your PCB (on both sides). This will speed up routing on small PCBs.

Block

Block : A block is a rectangular region   in your drawing.
(You will asked to mark it.)
For move and copy you will be asked for two reference points.
The contents of the block will be moved/ copied from the old one to the new one

 possible operations :
- **Move**
- **Copy**
- **Delete**
- **Rotate**   (rotate block around ref. point 90 deg clockwise)

Blocks for edit pad or line width:
- Mark block
- Unmark block

   (cf**.** Special edit commands for lines and pads)

 Note :
- You can cancel the block operations at any time with cancel action from the local menu.
- The block operation affects entire objects. If one part of an object is inside the block, the entire object will be affected. You could avoid this by selecting ... restrict in the main setup dialog.
- To copy only single sided blocks select "single sided blocks" from   main setup .
- You are able to scroll or change the resolution while a block operation. (use '1'-'5' to change the resolution, zoom in/out cancels the block operation)
- Parts could be moved outside the drawing area. If undo is no longer possible, you can execute Screen|Clean up to remove those parts.

 Example for copy block:
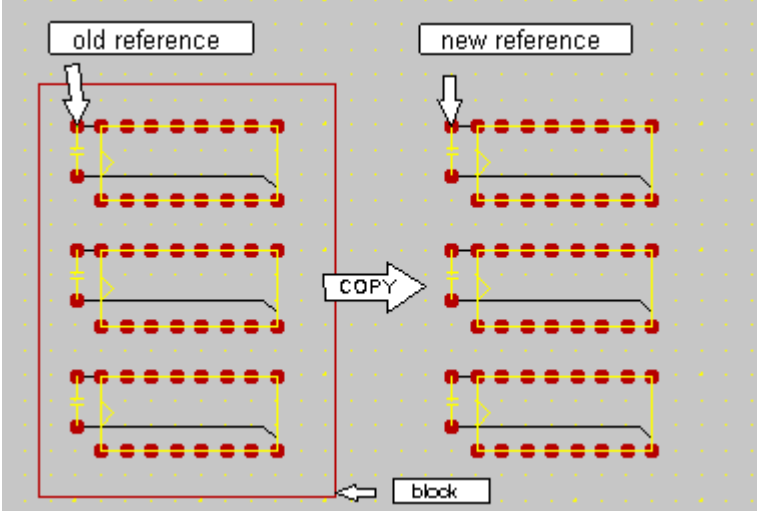
Grid

Grid : Creates a <u>requester for grid</u> (snap grid) options.
    checkboxes :
        - inch :   values in mm or inch
        - rel. coords: relative coordinates (you will be asked to mark the new origin)
        - offset :   the grid will have an offset
        - snap off :   activate / deactivate snap grid
    buttons :
        - up :   grid size *2
        - down :   grid size /2
        - mm/inch :   values in mm or inch (the given values will be changed !)

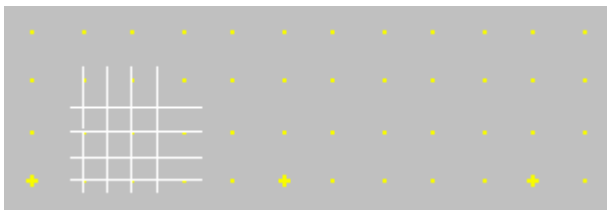**Grid *2 :**   grid size *2
Grid /2 :   grid size /2

Scale all :   Scale X and Y axis with the same value
Scale XY :   Scale X and Y axis with different values

Note : The grid will be displayed always. On/off will affect only the snap grid.

Example :

Only every 2nd point is drawn and every 10th is bigger.

Help

Contents :   Display the help file.

Order Form : Display the Order form. Select print from the file menu to get a printed copy. More Information on how to order is available in the Order section.

Register :   Enter your registration number you received from us. The registration number will be stored.

License agreement : If you do not   agree to the license you must stop using this program.

Using Help : Information on how to use help.

About :   Display information about the program.

Info :   Display general information (number of pads, ...).

Program settings and other dialogs

Main setup dialog

layer group :
- bottom :   solder side (standard for single sided boards)
- top :   component side (ground frames)
- symbol :   symbols for objects (names for ICs, etc.)

screen:
- mm: Use mm instead of inches for parameters (for coordinates and grid see grid setup).
- fill :   Fill pads and lines   (deactivating this function will speed up redrawing screen).
- auto track :   Moving the scroller (bars on the right and bottom side of the window)   will affect the screen immediately. Useful for fast computers with accelerator cards. Otherwise the screen will be updated after releasing the scroller.
- draw holes : Draw drilling holes for pads. Turn this option off for faster screen update.
- draft frames : High speed mode for drawing ground frames.   You will no longer see the gaps around pads and lines.

general:
- local menu : A click on the right mouse button will popup a local menu, otherwise the current action will be canceled.
- confirm on delete :   If this option is activated a requester appears before any part is deleted. Otherwise the requester appears only if more than one part would be affected.
- autodel references :   A pad is reference to a routed connection (the pad is one end of the connection). If this pad is deleted, only the references will be deleted. The lines of the routed connection will still be there. Ripup is no longer possible for those lines.
   Otherwise a requester will appear. You can delete the affected lines and the references or only the references.

   If you are using the auto router you should deactivate this option. Otherwise you can activate this option.

- default operation / move : The program assumes a move operation, if no other operation is selected. (default on)

objects:
- hide text requester : Normally an object text requester appears if you place an object and you will be asked to enter an ID for that object. You can avoid the requester by this option.
- display text : Object IDs will be displayed on the symbol layer . (default on)

blocks:
- ripup after copy : If you copy a block the program will ripup all routed connections.
- single sided blocks   : only lines and pads on the current layer will be affected by block operations.

block borders:
- copy / strict   : Objects and lines will not be copied, if they are not completely inside the block. Nothing outside the block will be affected. Otherwise objects will be copied if one half is inside the block and the other half outside.
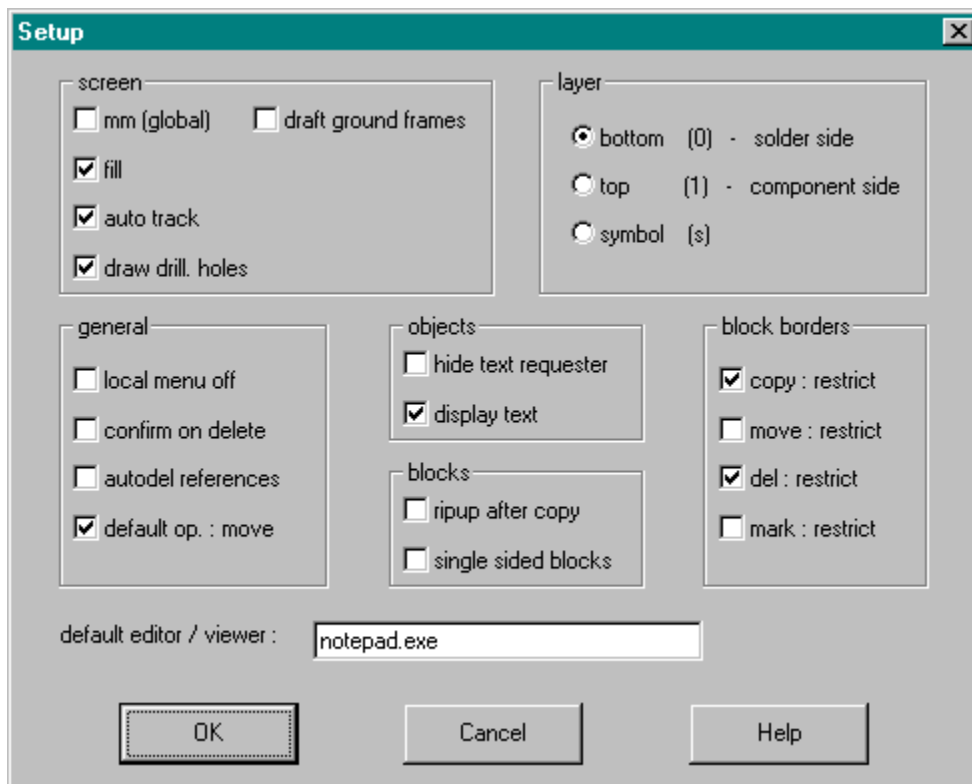
- move / strict   :   cf. copy
- del   / strict   :   cf. copy   (default on)
- mark   / strict   :   cf. copy

default editor / viewer : This program will be executed to view the parts list.
Normally Notepad will be launched. If you prefer another viewer enter the name here
(with the full path).

Note : If no default value is given, the default is off (no mark).

Command : Screen|Main setup

Dialog :

Setup dialog for printed page

layer group :
- bottom :   only solder side (standard for single sided boards)
- top :   only component side
- symbols only: symbols for objects only (names for ICs, etc.)
- symbol plus: symbols for objects (names for ICs, etc.)
              top and bottom layer will be printed shaded

orientation :   normal for exposure (printed side on PCB)

scale group :
- 1:1   no scaling
- 1:1.414   (copy DINA3 to DINA4 for right size)
- 1:2   double size

Other scales : Save your file. Execute Grid|Scale all before printing and reload your file after printing.
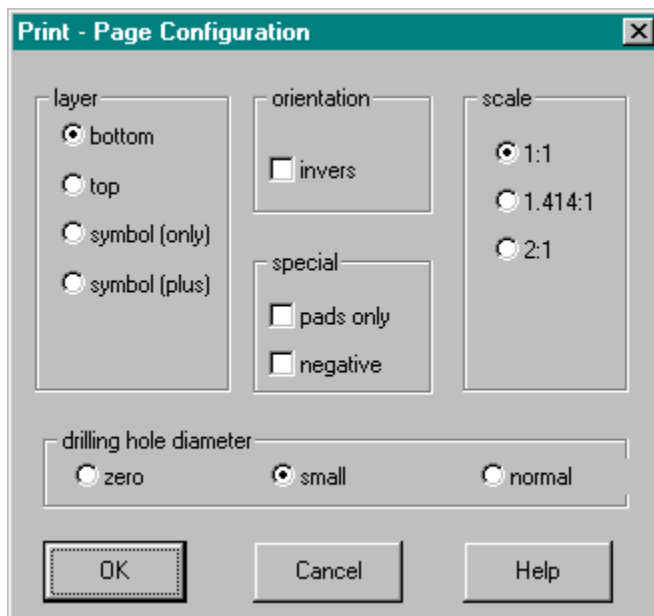
special group:
- pads only   **:** Print only pads (required for solder masks).
- negative **:** White on black printout.

drilling hole diameter group :
- zero   no drilling holes
- small   small holes (for manual drilling)
- normal   the holes in the printout will have the correct   size (large holes).

The page setup dialog will be executed automatically before printing (File|Print).

Dialog :

Setup for HPGL output

layer group :
- bottom : only solder side (standard for single sided boards)
- top : only component side
- symbols only: symbols for objects only (names for ICs, etc.)
- symbol plus: symbols for objects (names for ICs, etc.)
  top and bottom layer will be printed shaded

orientation : normal for exposure (printed side on PCB)

scale group :
- 1:1 no scaling
- 1:1.414 (copy DINA3 to DINA4 for right size)
- 1:2 double size

Other scales : Save your file. Execute Grid|Scale all before printing and reload your file after printing.

special group:
- pads only **:** Print only pads (required for solder masks).
- negative **:** White on black printout.

drilling hole diamter group :
- zero no drilling holes
- small small holes (for manual drilling)
- normal the holes in the printout will have the correct size (large holes).

  pen diameter: Enter the thickness of your plotter pen.
  init string : (Commands to initialize your plotter, only if required.)
    You should slow down your plot speed for best results (e.g. VS3;).
  output file : name of the plot file

Exposure : For best results you should use black india ink instead of normal ink and add reference marks to your drawings.

The page setup dialog will be executed automatically (File | Plot).

Dialog :

Plot - Page Configuration

layer
- ● bottom
- ○ top
- ○ symbol

orientation
- ☐ invers

scale
- ● 1:1
- ○ 1.414:1
- ○ 2:1

Pen diameter (mm !) `0.300000`

INIT string `_____`

[OK]   [Cancel]   [Help]

Grid setup dialog

**Grid :**
checkboxes :
- inch :	Values in mm or inch.
- rel. coords:	Relative coordinates .(You will be asked to mark the new origin.)To change the active origin turn the switch off and on. (Execute the grid dialog twice).
- grid offset :	The grid will have an offset.
- snap grid off :	Activate / deactivate snap grid.

buttons :
- up :	Grid size *2
- down :	Grid size /2
- mm/inch :	Values in mm or inch. (The given values will be changed !)

Command : Grid|Grid

Dialog :
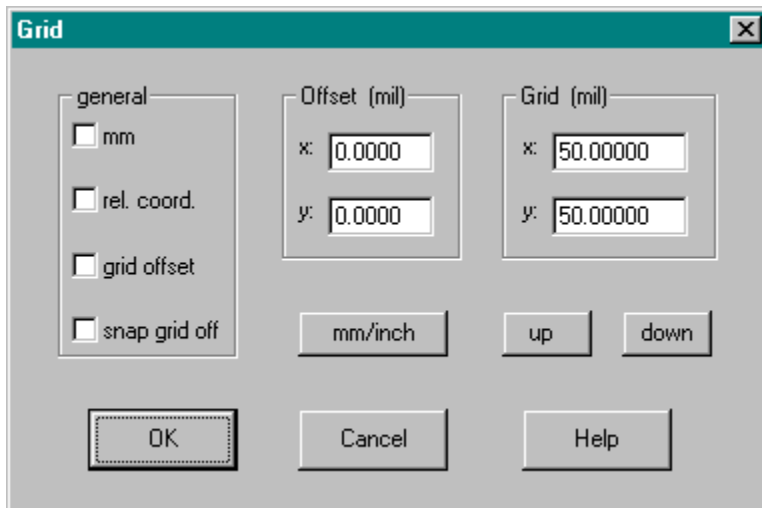
Line / pad setup dialog

Line group :
- select given width or
- select X and enter user width in the field of Line X

Changes will be recorded if you press one of the following buttons : OK , LINE (,PAD).
This setting will be used for the normal line command.

Line button : Click on the line button to record changes and draw the selected line.

Pads group :
- select given diameter or
- select X and enter user diameter in the field of Pad X. If no y value is given the pad will have a symmetrical shape.

Changes will be recorded if you press one of the following buttons : OK , PAD (,LINE).

square option :The pad will have a rectangular shape. The square option is not possible for predefined pads.

drilling hole : Deactivate this option for solid pads (e.g. testing pads, SMDs).

double sided : Deactivate this option for single sided pads (e.g. testing pads, SMDs).

Pad button : Click on the pad button to record changes and draw the selected pad.

General :

Changes will be recorded if you press one of the following buttons : OK , LINE, PAD.
Next time you use the pad or line command the recorded settings will be used.

Command : Items|Setup default line/pad

Dialog :

Text dialog

Text field , valid chars :
        a-z   (will be translated to A-Z)
        A-Z
        0-9
        ( ) : . _
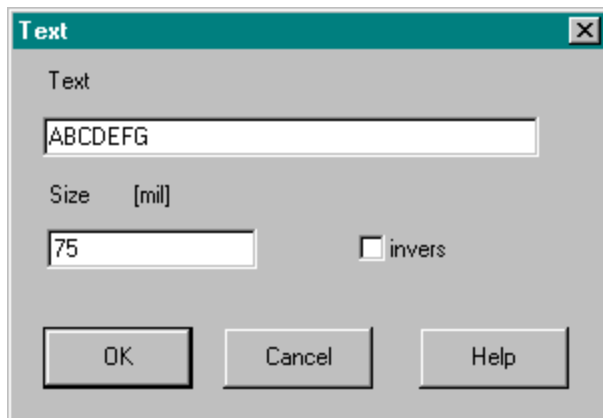   Space

Text size :
- the text size is in mil or mm depending on the <u>main setup</u>

orientation :
- inverse :   for bottom layer
- normal :   for top and symbol layer

Command : Objects|Text

Dialog :

Router setup

Route group :
    Select layer to be routed
- top: route top layer
- bottom :   route bottom layer

2 layers options :
    Select options if you are routing a **double sided PCB.**
- hor / vert :   quick, high percentage routed, but many vias and long lines (complex digital PCBs, fine grid)
- multipass :   slow, best results (use 10 or more counts)
- normal **:**   quick, low percentage routed, only for a few connections.

grid / line width:
    Select grid size for router   / line width :
- fine :   25 mil / 12 mil   (one line between 62mil IC pins)
- normal :   50 mil / 24mil
- rough :   100 mil   / 48 mil
- >>> Max. routing area

Possible warning: Routing grid and snap grid should match together.
    If you receive such a warning after routing, try to move all parts (especially pads) on the routing grid. The routing grid is identical to the snap grid, if you don't use an offset. The routing results should be better now.
    This may happen, if you are using a 50 mil snap grid and the rough routing grid. Move as many parts as possible to the visible snap grid points.

The router setup dialog will be executed automatically before routing (Router|Route).

Dialog :
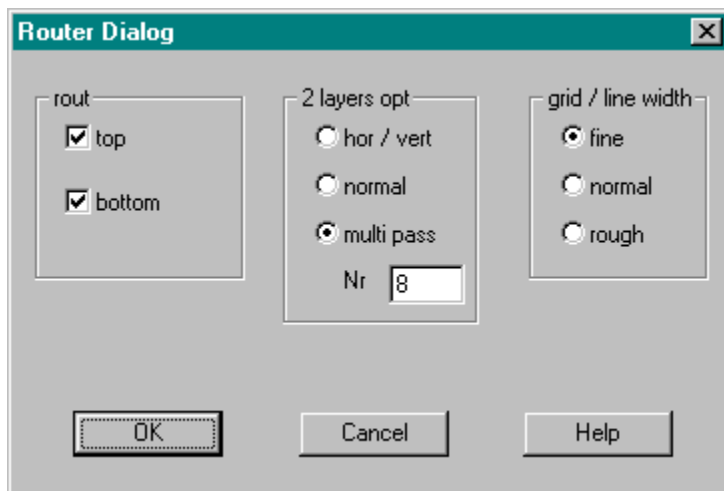
Maximal routing area:

The max. routing area depends on routing grid and program version:

Standard and Extended Edition:
- fine :   6,3" x 6,3"
- normal :   12" x 12"
- rough :   max. drawing area

32Bit Edition :
- fine :   15" x 12"
- normal,rough : max. drawing area

Gerber setup dialog

- **G54 on aperture change :** Insert "G54" command before each aperture change "Dxx".
  (Dependent on photoplotter)

- **Create aperture file :** This option will create an aperture file from the current drawing.
  (For normal raster plotters.)

  Deactivate this option to use a given aperture file. A file requester will appear after
  pressing OK. (For normal vector plotters.)
  Note : You should only use track and pads and lines for which there is a corresponding
  aperture.

- **Match tolerance :** Maximum tolerance for apertures.
  This value will be ignored if "Create aperture file" is activated.

- **Border** : Border around your drawing on the film.

After pressing OK several files will be generated :

    <name>.gb0   Gerber file for bottom layer
    <name>.gb1   Gerber file for top layer
    <name>.gb2   Gerber file for symbol layer
    <name>.map   Aperture file (create aperture file activated)
    <name>.nc   NC drill file.
    <name> is the file name of the current drawing (or noname if the current drawing has
        not been saved until now).

More information is available in the <u>Gerber file</u> section.

Object setup dialog

IC, transistor, capacitor and resistor :
- select type (it will become standard)
  (for R, C you have to select the distance between the two pads)
- the diameter group will affect all items (IC ,R,C and Tr)
- press the corresponding button (IC , R,...) to draw an item and record changes
  or OK for recording only

Other objects (e.g. PLCCs, ...) are available from libraries (Objects| Open Lib window)

General :
Changes will be recorded if you press one of the following buttons : OK , IC, TR, R, C.
Next time you use the IC or transistor, ... command the recorded settings will be
used.

Hint : The needed capacitor or resistor is too large. There is no predefined part.
- Place the largest predefined part.
- Select "Move single points" from the Edit menu.
- Move one or two pads until you have the correct distance.
- To create larger pads place a normal pad with the desired size over the object pad.
(You can save this object as an user defined object.)

Command : Objects|Setup

Dialog :
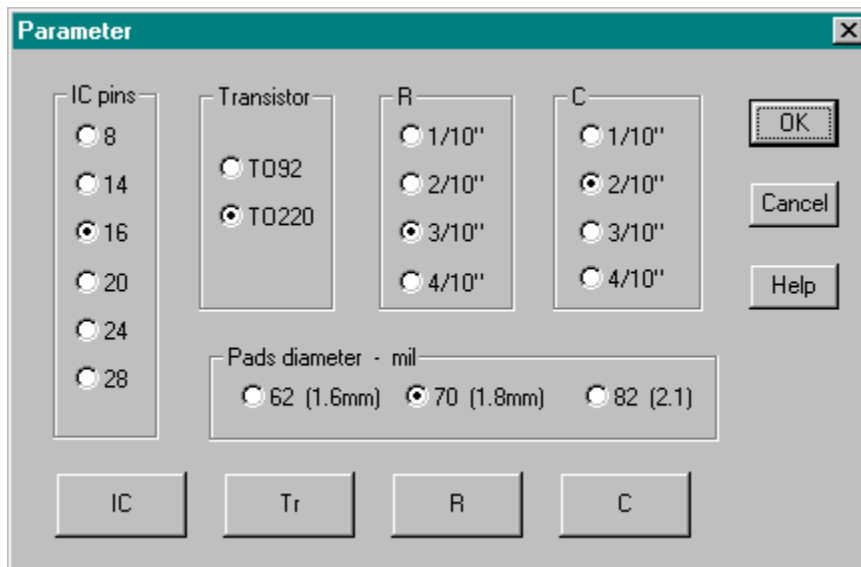
Color setup

set default1 :   Select the default color set 1.(background **light gray**)

set default2 :   Select the default color set 2.(background **black**)
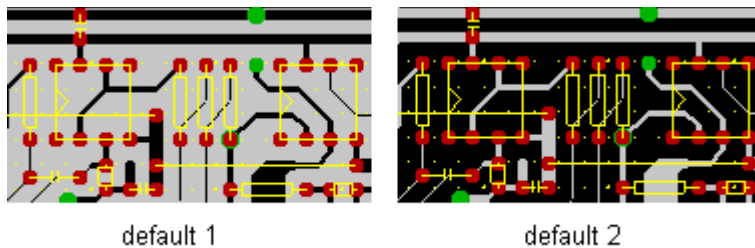
Custom colors :
- Select the color to change.
- Press Change color and select your preferred color.
- To save your custom color set execute File|Save configuration.
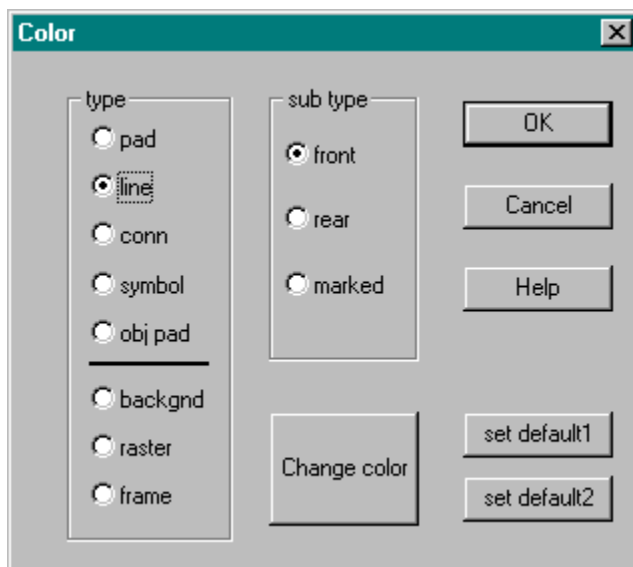
Note : Execute Screen|Save configuration after changing the color set. Otherwise the
changes will be lost after a restart.
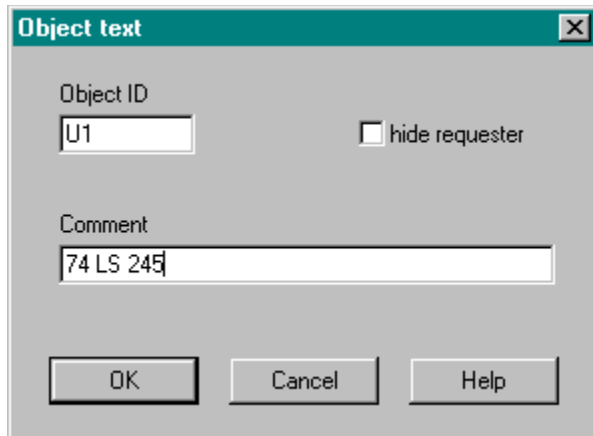

Command : Screen|Colors

Example :

default 1   default 2

Dialog :

Object text dialog

- **ID:**   The ID will be displayed with the object..
              e.g. IC1, R0100
              max. 5 chars

- **Comment:**   This text will not be displayed.
              e.g. 74LS245, 45nF/63V
              max. 50 chars
              The comment will be shown in the parts list.

- **hide requester:** If you don't need IDs for your objects, You should activate this
      checkbox. The requester won't appear if you place an object and the object won't
      have an ID.
      To deactivate this option execute <u>main setup</u>.
  .

Dialog :

Edit pad dialog

Available parameters:

- **Pad diameter**
  x: Diameter horizontal
  y: Diameter vertical

- **square:** Draw a rectangle instead of an octagon.

- **drilling hole** : A normal pad has a drilling hole. For testing pads with no holes or SMDs deactivate this option.

- **change entire object** : If the selected pad belongs to an object, all pads of this object will be changed to the shown values. All those pads will be equal afterwards.

- **hole** : Diameter of the drilling hole (ignored if 'drilling hole' is not set).

Press "OK" to change only one pad (on the current layer) or "entire object" to change **all** pads of the affected object.

Command : Edit |Edit pad.

Dialog :
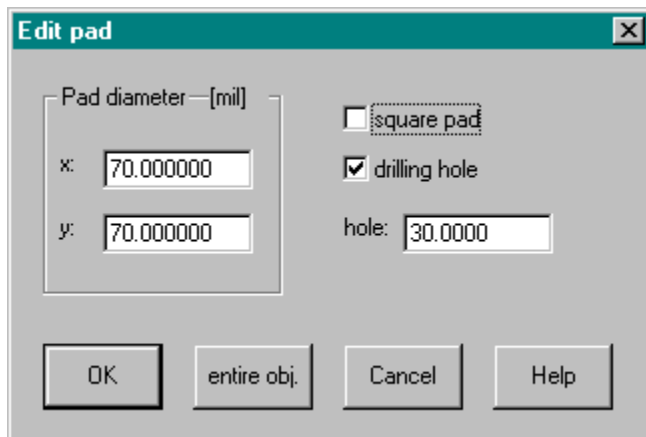
Toolbar

Many menu commands are also available from the Toolbar :

Screen

↔	Redraw screen

↔	Zoom in

↔	Zoom out

↔	Main setup

↔	Change layer

Objects

↔	Setup object

↔	IC

↔	Transistor

↔	Capacitor

↔	Resistor

Parts

↔	Setup lines and pads

↔	Pad

↔	Line

↔	Connection

other objects

↔	Bridge

↔	Text

Edit

↔	Delete

↔	Copy block

↔	Move block

↔	Del block

↔	Edit line

↔	Move

↔	Grid

Statusbar

| Move parts | 0 | 50:50 | 1350:900 | |

The Statusbar displays information on commands, layer, grid and current position :

1) Info text :   shows current <u>command</u> (and hints)

2) Layer :   displays current layer
                 0 : solder side
                 1: component side
                 s: symbol layer
                 To change the current layer select <u>main setup</u> or press 'l'
                 (l toggles between 0 and 1)

3) Grid: displays the current <u>grid</u> in mils
                 (mm are indicated by a leading m)

4) Position :   displays the current position in mils
                 (mm are indicated by a leading m)

(The fields are enumerated from the left to the right side)

Mouse and keyboard input

2 and 3 button mice are supported.

Mouse input : "drag and drop" is not supported. The first click will start the action and the second will terminate it. (like many other CAD programs.) You don't have to keep a button pressed.

left button :
normal action (depends on actual command e.g. draw a pad)
right button :
abort action or entire command
or local menu (if activated)
mid button : (if you have none : press the "m" key)
rotate objects   (draw object)
delete corner   (edit line)
fix on nearest pad   (draw line)

Keyboard input :

Some often needed commands and function can be accessed by keyboard shortcuts.

```
F1        : Help
F2        : Save
Del       : Delete
Backspace: Undo
Esc       : Cancel current action

1,2,3,4,5 : resolution 1,2,3,4,5    (mouse position will become center)
d         : grid * 2   (larger)     (double size)
h         : grid / 2   (finer)              (half size)
l         : flip to top or bottom layer
s         : flip to symbol layer
cursor    : scroll
Shift-cursor : scroll (larger steps)
f         : fill mode on/off
g         : grid on/off
o         : relative coordinates on/off  (origin)
i         : metrics -   mm/inch
r         : rotate objects
v         : generate via (draw line only)
m         : replacement for mid button (2 button mice)
```

Local menu

By pressing the right mouse button you can popup a local menu.
In this menu you can select commands corresponding to the actual command.

The local menu can be turned off in the main setup.
If the local menu is deactivated, a press on the right mouse button will cancel the current action. This could speed up your working (if you are skilled with the program).

Examples :
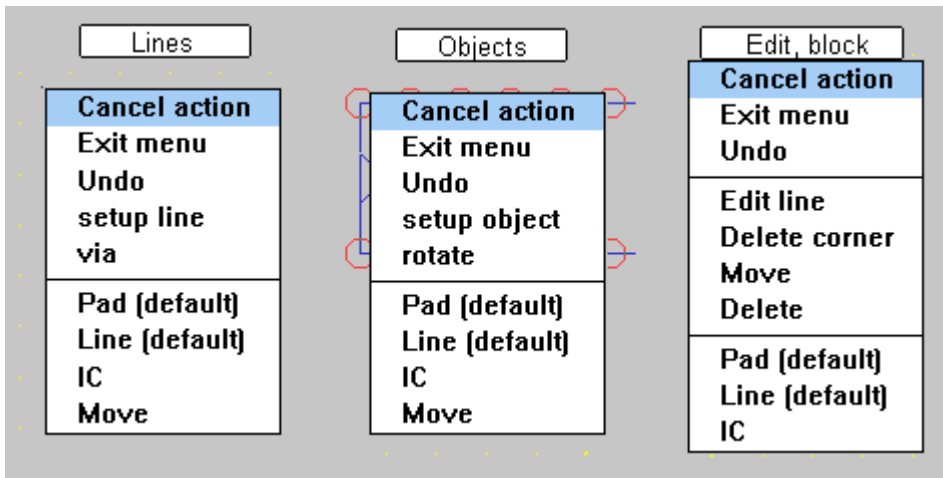
Cancel action :   The current action will be canceled.
                      E.g. terminate a line or exit the copy block function.
Exit menu :   Do nothing. Clear the local menu from screen.

Other menus: Call the corresponding menu commands.

Note :   The delete corner command will affect the position where you pressed the
          right mouse button.

Printing and plotting

About printing

The File | Print command is used to generate drafts of your drawings.
It could be also used to generate final printouts for exposure on a laser printer.
If you are not using transparent paper you will have to photograph the printout to receive a transparent film.

Please read also the "printing problems" section if you have problems with your printer or printout.

About plotting   (32bit Edition only)

The File| Export | HPGL command by-passes the Windows plot drivers. You have to use this function instead of the normal Print command if you are plotting to a HPGL-compatible vector pen plotter. The normal print command will only support raster devices (like laser, dot matrix or ink jet printers).

The HPGL command uses only a few basic commands (SP;PU;PD;PA<x>,<y>;)   which are compatible with many plotter types.
Setting up your plotter to support the HPGL language may be necessary. Consult your plotter documentation for details.

When this command is chosen, a dialog box opens, presenting the plot options.

About photoplotting   (32bit Edition only)

Photoplotters print images directly onto photographic film. The quality is better since no photo-reduction or further photography is required.

Photoplotters are divided into two classes :

Vector plotter: You can compare them with normal pen plotters. They select a drawing instrument and draw   a vector on the film. Normally the apertures are arranged on a wheel. Therefore the number of apertures is limited.
The corresponding command is File | Export | Gerber (man).

Raster plotter: You can compare them with laser printers. They generate a bitmap in their memory first. This bitmap will be printed directly on the film. Those machines do not have problems with a limited number of apertures and are normally much faster than vector plotters.
The corresponding command is File | Export | Gerber (auto).

Both kinds of photoplotters are using the Gerber format.

Printing problems

Wrong scaling:

Many laser, inkjet or dotmatrix printers have tolerances up to 2% or even more. This is could be a real problem for some parts (e.g. large connectors or SMDs) if you use the printout for exposure or toner transfer. Therefore you should make a test printout first. Draw a rectangle out of thin lines (length of borders e.g. 5 inches). Print this test drawing and measure the distances on your printout (horizontal and vertical). If there is a mismatch between drawing and printout you should scale your drawing before printing. Execute Undo after printing. The scaling value in one direction will be :

    distance on drawing / distance on printout

Printing delay

It is possible that there is a delay of some minutes between end of printing (software) and start of printout (printer). This happens especially with some printer driver settings (error diffusion, color correction or   brightness correction). In some cases the printer or spooler icon does not appear. But there should be a correct printout after a few minutes.

Empty pages or misplaced lines

This problem has been reported by some Win95 users. It should be fixed in V2.1. In the case that there are still problems with some strange printer drivers you should follow the listed instructions:

1) Press the "standard values" button for your printer driver (if the property window has such a button).
    Or select standard values for your printer driver if you do not have a "standard values" button. This means: no error diffusion, no color or brightness correction, black and white printing.
2) If 1) did not help change the PrintVer value in the [PIA-PCB-SETUP1] section of your WIN.INI file (located in the Windows directory) to one of the following values:
  Windows 95   10, 4 or 5
  Windows 3.1 or NT 1, 3 or 5
 (Please ensure that the graphics mode of your printer driver is set to raster instead of vector to improve printing speed if you are using mode 5.)

Gerber and aperture files

Gerber format :   (32bit Edition only)

RS274-X: The File | **Gerber (auto)** command crates a gerber file using the RS274X format.
The file will have the name of your current drawing with the extension '.gbr'. If you did not save your drawing before the file name will be 'noname.gbr'.
The apertures are embedded .
The LPC command will be used If your drawing contains ground frames. You should take care that your photoplotter or pcb manufacturer is able to process the LPC command. In that case 3 gerber layers are used for each pcb layer (with ground frames).

The last four layers in your Gerber file will be silk and SMT paste masks (order: silk 0, silk 1, paste 0, paste 1).

Edit the USER.INI file in your program directory to change the parameters of those masks.

```
[GERBER-RS274X]
BorderX=1000            ; border around drawing [mils]
BorderY=1000
CreateSilkMask=1          ; 1enabled / 0 disabled
SilkMaskGap=10          ; gap in mils
CreatePasteMask=1       ; 1enabled / 0 disabled
PasteMaskGap=0          ; gap in mils
```

RS274-D: The File | **Gerber (man**) command crates gerber files using the RS274D format.
5 files will be created :
- [name].gb0   - solder side
- [name].gb1   - component side
- [name].gb2   - symbols
- [name].map   - aperture file
- [name].nc   - NC Drill file

If you did not save your drawing before the file name will be 'noname.*'.

The format is :
- 2:3   (means that all values are in mils)
- leading zero   (leading zeros will suppressed)
- imperial   (inch)

You could avoid the creation of the aperture file and use another aperture file if 'create aperture file' is not activated. This proceedings are necessary for all vector plotters.
(Please consult your photoplotter handbook or ask your PCB manufacturer for the required aperture file.)

Aperture file format (RS274-D only):

Comment

```
Dx              : Aperture number x. X ranges from 11 to 999.
SHAPE           : Shape . Possible values : ROUNDED, SQUARE, OCTAGONAL
XSIZE           : Horizontal size (width or diameter) in inches
YSIZE           : Vertical size (width or diameter) in inches
x               : Unused.
TYPE            : Type of aperture. Possible values FLASH, DRAW, MULTI
x               : Unused. Any data after TYPE will be ignored.
```

Example for aperture file:

#Tool	SHAPE	XSIZE	YSIZE	x	TYPE
D11	OCTAGONAL	0.080000	0.080000	0	FLASH
D12	OCTAGONAL	0.070000	0.070000	0	FLASH
D13	OCTAGONAL	0.060000	0.060000	0	FLASH
D14	ROUNDED	0.012000	0.012000	0	MULTI
D20	ROUNDED	0.040000	0.040000	0	MULTI
D21	ROUNDED	0.030000	0.030000	0	MULTI
D22	ROUNDED	0.020000	0.020000	0	MULTI

Important :   Please note that ROUNDED shapes will be used instead of octagonal shapes for lines in the Gerber output. This is necessary as the Gerber format does not support rotated octagons.

NC Drill files

***

Netlists

Netlist import : You can use this command to import connections from a schematics drawing program.

- Execute "export netlist" in your schematics software
- Select Protel, Protel 2 or Tango format
- Exit your schematics software
- Execute Import | Netlist

All parts listed in the file "net_xref.ini" in your objects directory will be placed automatically. It is possible to add part types to this file (e.g. for user defined parts in your library).

Example :
      [TRANSLATION-PART]
      IDC10P=Con 10
      ;^ Footprint
      ; ^Name in library

      [TRANSLATION-LIB]
      IDC10P=standard
      ;^Footprint
      ; ^name of library (without .lib)

      "Con 10" in library "standard.lib" will be placed for all parts with footprint IDC10P.

The [TRANSLATION-BY-NAME] section of "net_xref.ini" will be used if your schematics software does not add footprints or part types to the netlist. The import command tries to find the footprint using the part description (e.g. 7400 -> DIL14).

If there are still missing connections (due to unknown parts), place all missing parts manually ! (use the same IDs as in your schematics software)

Netlist export : Generates a netlist file (format similar to Protel).

Hints

How to achieve best results for exposure :
> Print drawing on tracing paper or coated overhead films with a laser printer (1:1).
> With most newer laser printers (We have tested it with printers of the HP4-series and OKI OL400ex) you will obtain nearly professional results (if   you don't use   PQFPs or other items with a grid below 50 mil)
> Reference marks are not created automatically. So you should draw them by your own, if you have a double sided PCB.
>
> Many laser, inkjet or dotmatrix printers have tolerances up to 2% or even more. This is could be a real problem for some parts (e.g. large connectors or SMDs) if you use the printout for exposure or toner transfer. Therefore you should make a test printout first. Draw a rectangle out of thin lines (length of borders e.g. 5 inches). Print this test drawing and measure the distances on your printout (horizontal and vertical). If there is a mismatch between drawing and printout you should scale your drawing before printing. Execute Undo after printing. The scaling value in one direction will be :
>> distance on drawing / distance on printout

Output on a plotter :
> Do NOT use the normal print command (it doesn't support plotters) .
> The plot command is supported in the 32-bit version. A separate HPGL file will be generated (only a few basic commands are used and it should work with most plotters).
> Select a low speed (e.g. VS3;) and use   black india ink instead of normal ink for best results.

Terms

line :   single part of a track
track :   list of connected lines
connection : virtual line that becomes a track after performing router. You won't see it in your printings. (It is possible that a connection crosses other connections or lines.)
router :   The router tries to find a way for all connections without crossings

object : composition of lines, pads, etc. like ICs, resistor,...

block : a block is a rectangular region in your drawing

General information

What is **P**CB Developer's **I**ndividual **A**ssistant ?

> It is an easy to use program to develop PCBs. It was designed especially for small and simple boards and for hobby purposes. So it has no varieties of functions or possibilities but it is quick and easy to use.
> Supported features:
> - two normal layers (copper) and one symbol layer
> - up to
>   - 1000 pads, 1400 lines   Standard Edition
>   - 2000 pads, 3000 lines   Extended Edition
>   - 8000 pads, 12000 lines 32-bit Edition
>
> - board sizes up to 15" x 15"
> - grid of 1mil
> - routing (max. area depends on version and routing grid)
> 32-bit Edition only:
> - HPGL file output   (the normal print command will not support plotters).
> - NC drill file output.
> - Netlist import.

Hardware requirements :
> - Windows 3.1 or later
> - 386   (387 or 486DX desirable)
> - 4 MB RAM
> - 2 MB free disk space
> 32-bit Edition only
> - at least WIN32S (or Windows 95, NT)
> - 8 MB RAM

If you are using Windows 3.1 you can use the 32bit Edition with Win32S.
Win32S is an extension from Microsoft to use some 32bit programs with Windows 3.1.
It is available from various FTP servers
(e.g. ftp://ftp.cica.indiana.edu/pub/win3/util/win32s.zip).

Known problems

Problems with graphic output :

- Thin lines on the screen after scrolling :

  Solution : Press the redraw button.

  You should also try to install a newer driver for your graphics card.
  This problem may not occur with another resolution and is dependent on your graphic driver.

- Rests of lines or pads (in background color) after executing Delete or Move.

  Solution : Press the redraw button.

- Rests of object drafts after closing or moving dialogs.

  Solution : Press the redraw button.

How to uninstall

In the unlikely case you wish to remove $P_{CB \, Developer's}$ $I_{ndividual}$ $A_{ssistant}$ from your system, just do the following:

- Delete the PIA directory (or the directory you installed the program into).
- Delete the P.I.A. group from the program manager.

Only one file (ctrl3dv2.dll or ctrl3d32.dll) was copied to your system directory. You should not delete this file. (Another program could be affected.)

Tutorial

Step 1 : How to use the program

The following graphic explains you the main elements of P_{CB} $_{Developer's}$ $I_{ndividual}$ $A_{ssistant}$
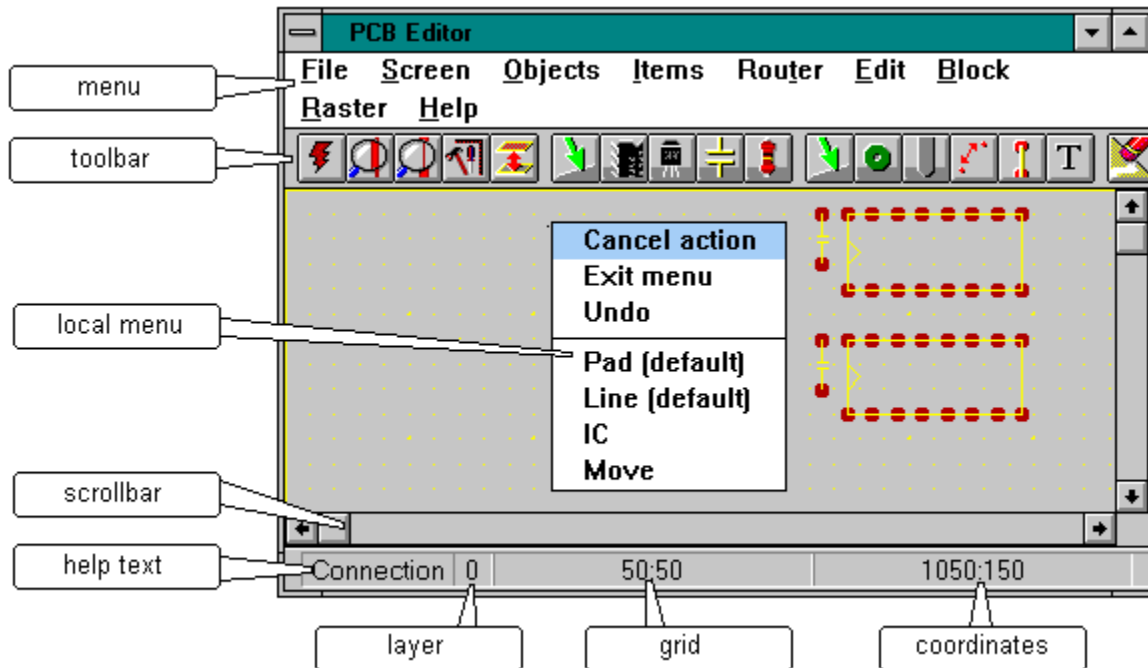
There are 3 possibilities to execute a command :
- menu
- toolbar
- local menu

The statusbar shows you :
- a help text to the current command
- current layer
- snap grid
- coordinates

Screen :
   You can scroll the screen using the scrollbars or the cursor keys.
   To change the resolution click on zoom in / out (↔

↔) on the toolbar or press '1'-'5'.
(Only the zoom in / out buttons on the toolbar will terminate a running command.)

Note : A leading 'm' before the snap grid or coordinates indicates you that the values are in mm.

>>> Step 2

Step 2:   General settings

Select grid :
>   The default snap grid is 50 mils. For easy PCBs with no tracks between two IC pads this is enough. Otherwise the grid can be changed in grid dialog. (select grid|grid from the menu or click on ⟷).

Setting up outlines of your PCB :
>   There are no special outlines.
>   Execute main setup (⟷) and change to the symbol layer.

Just draw normal lines (⟷)around your PCB.You won't see those lines in your normal printout.
If you are using the router, you should also draw lines on the top and bottom layer around your PCB.

Switch to the bottom layer (⟷) and draw the lines. Repeat this for the top layer (if you are using both sides). This will limit the router to this area.
>   You may also add reference marks for exposure. You can draw a few lines on top and bottom side or add the reference object cross.ob (next step).

>   There are also relative coordinates available from the grid dialog (⟷).

Just activate "rel. coords", press OK and click on the new origin. Done.

Examples for outlines and reference marks :
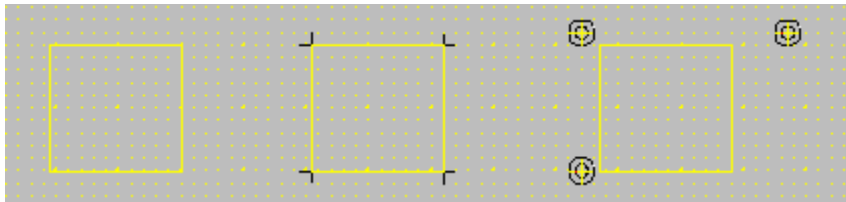
Placement, routing :
>   For single sided boards routing is not useful.
>   So you have to route the tracks manually. Otherwise you would receive to many bridges and worse results.

>   It could be useful to think a few minutes about the best placement and routing for your schematics before starting the program or you have to edit the drawing for several times for best results.

>   There are also several example files included.

>>> Step 3

Step 3 : Place all ICs, connectors, ...

You have for example   two ICs (14 pins) and several resistors and one 25 pin SUB-D connector.

ICs:

- Select Objects|Setup (or 1st green arrow symbol ⬌ on the toolbar)
- Choose the parameters for your object (e.g. 14 pin IC).
- Then press the IC button.
- Move the IC to the desired position and a click on the left button will fix the IC.
- A requester with an ID and comment field will appear. Enter the name of your object (e.g. 74LS00). The IDs will be created automatically. Press OK to continue.

    The next time you need a 14pin IC simply press the IC symbol ⬌ on the toolbar.

To rotate the IC press the mid button ('m' key or "rotate" from the local menu)

Resistors : almost the same.

SUB-D connector :
    Special parts like SUB-D connectors, PGAs, reference marks,... are available as object files.
- Select Objects|Open.
- Load the desired object (subd25.ob).
- Position the connector like an IC or resistor.

Other parts : If you need a part which is not in the library, you can also   create your own objects.

 >>> Step 4

(>>> Step 7)

Step 4 : Draw tracks manually

Set parameters:
- Select Items|Setup (or 2nd green arrow symbol on the toolbar)
  - Choose the thickness for your track (e.g. 30 mil).
  - Then press the Line button.

Draw a track :
- Click with the left button on the first pad.
- Further left clicks will fix corners of your track.
- To terminate the track on the second pad click twice.

Note : Tracks should start and end on a pad.

The SUB-D connector has another grid (stupidly). To fix the end of your track on a pad of the connector press the mid button of your mouse (or the 'm' key if you have none). The line will be fixed to the nearest pad.

The next time you need 30mil line simply press the line symbol on the toolbar.

>>> <u>Step 5</u>

Step 5 : Print

If you have more than one printer and the desired printer is not set as the default printer select File|Change printer.

First printout :
- Execute File|Print.
  A dialog with all print options will occur.
- Select symbols (plus) and press OK.
- All symbols (e.g. ICs, capacitors,..) will be printed and all other parts will be printed shaded.
  This printout is useful to search for errors or for inserting components in the final PCB.

Final printout :
- Execute File|Print.
  A dialog with all print options will occur.
- Select bottom and press OK.
- Everything printed black will appear as copper on the final board.
- Use this printout for exposure.

PS: You should save your file using File|Save of course.

End of the first part.

>>> End

Step 6 : Place all larger parts

(see Step 3)

Step 7 : Draw all connections

Drawing connections is much easier than drawing lines manual.
Just click on the start pad and then on the end pad.

Note : Connections are only possible between pads (and not between lines).

>>> <u>Step 8</u>

Step 8 : Route

Initiate the router:
- Select Router|Route
- A requester will appear
- Mark top and bottom in the route group (Both layers will be used.)
- Select multipass in the "2 layers options" group.
  The multipass counter   should be greater than 8. If you have time and a fast
     computer change it to 30.

- Select the routing grid :
  - fine :   25 mil   (one line between small IC pins)
  - normal :   50 mil
  - rough :   100 mil
  The width of the routed connections depends on the routing grid !
- Press OK. The router will start.

Step 9 : Change placement and reroute

Problems :

There are areas with too many tracks. The router fails because there is not enough place.
- Select Router|Ripup all
- Select Edit|Move.
- Move your components to other locations so that there is more space.
- Select Router|Route again.

There are too many crossing tracks. The router fails.
- Select Router|Ripup all
- Think about another order for   your components.
- Reorder your components using Edit|Move
- Select Router|Route again.

>>> <u>Step 10</u>

Step 10 : Print

(see Step 5)

Edit an existing drawing

Normal   commands for editing (Menu Edit) :

   Undo : If you have done something wrong, this command is able to restore the old state.

   Move: Normal operation ↔
   •   The first click on the left button will selected a part
   •   Moving the mouse will move the part and all affected lines.
   •   The second click will fix the part.
   •   If you have selected the wrong part, click on the right mouse button and select
       "cancel current action" from the local menu.
   Move single : Only the selected part will be moved.
   Delete : normal ↔
   •   A left click will delete the entire part (lines. ICs, ..) under the mouse cursor.
   •   If there is more than one part you will be asked for the right part.
   special : Only the specified part will be deleted (and also single pads from an IC or
       something else).
   Edit a line : ↔
   •   Move corner : Click on an existing corner. Move it. The second click will fix it.
   •   Add corner : Click on a line. A corner will be inserted. Move it. The second click
       will fix it.
   •   Delete corner : A click on the mid button will delete a corner.
   You don't have 3 mouse buttons :
       Move the mouse over the corner.
       Press the 'm' key or press the right mouse button and select "delete corner" from
       the local menu.
   Edit the thickness of a track :
   •   Click on one line of the desired track.
   •   A requester with the old track width appears.
   •   Enter a new value.
   •   The complete track will be modified.
   If you have selected the wrong track press "Cancel" in the requester.

Commands affecting blocks (Menu Block) :

   •   **Copy** ↔
•   **Move** ↔
•   **Delete** ↔ (A confirmation requester appears always)
You will be asked to mark   a rectangular region which will be affected.
       •   The first click will create the first point for the block
       •   The second click will fix the rectangle
       For copy and move you will be also asked to mark an old and a new reference point.
       (The old reference point will become the new one.)

   You can move the screen with the scrollbars or cursor key. The block operation will not be
       intercepted.

>>> End

Example for copy block:

License and Purchasing Information

License Agreement

How to order
Order form

Information on the different editions
How to contact the author

Order form

Product Name: $\text{P}_{\text{CB Developer's}}$ $\text{I}_{\text{ndividual}}$ $\text{A}_{\text{ssistant}}$ V2.1 (valid until 6/97)

Cost : One license   (March 1996, 79DM is about $54US)

      Standard   :   _____   copies @   79DM   = _____

      Extended (disks included) :   _____   copies @   149DM   = _____

      32-Bit   (disks included) :   _____   copies @   249DM = _____

      disks for standard (optional):   _____   copies @   25DM = _____

      European Community residents add 15% sales tax   _____

Total Cost: = _____

Note :   only 3.5" disks are available,   32-Bit : at least WIN32S required

   I would like to pay by: Cheque / Visa/ MasterCard/ American Express

   If credit card : Card No:_____

      Expiry date:_____/19____

      Card holder name:_____

   Full Name:_____

   Company: _____

   Address: _____

   City: _____

   Country: _____

   Phone: _____

   Fax: _____

   Internet/Compuserve address: _____

Terms/Conditions:

   I agree $\text{P}_{\text{CB Developer's}}$ $\text{I}_{\text{ndividual}}$ $\text{A}_{\text{ssistant}}$ is distributed as shareware. No warranty exists, either expressed or implied. No liability is assumed for any damage or loss resulting from the use of this program. No claims are made regarding the accuracy of this program.
 I agree that for users not in Germany only bank checks or creditcards will be accepted.
 Your signature below indicates your acceptance of these terms, conditions and the terms listed in the license file .

 Signed: _____ Date:_____

Mail this form to :   Andreas Waldherr
                  Perlacher Str. 31
                  81539 Muenchen
                  Germany

License Agreement
Copyright   © 1994-1996 Andreas Waldherr (AW)
All rights reserved

License Agreement

INSTALLATION OF PCB DEVELOPER'S INDIVIDUAL ASSISTANT (PIA) ON YOUR COMPUTER SYSTEM IMPLIES AGREEMENT WITH THE TERMS AND CONDITIONS BELOW.

Distribution of PIA, it's accompany programs and documentation is considered as is. AW offers no warranties of any kind, expressed or implied. This includes, but is in no way limited to , warranties of merchantability or fitness for a particular purpose. Under no circumstances will AW be liable for any damages which result from the use of this program or the inability to use it. Exclusion from liability includes, but is not limited to, lost profits, lost savings, or any other incidental or consequential damages.
 Specifically, AW makes no representation or warranty that the software is "error free", or meet any user's particular standards, requirements or needs.
 PIA is distributed as shareware. It is not freeware, or in the public domain. You may use PIA for a trial period two months, at no cost to you, to determine if it fits your needs. If you decide to use PIA regularly, you are expected to register it and pay the applicable registration fee. Individual copies of the unregistered version of PIA may be given to your friends or uploaded to the public section of a public BBS.

You may not modify or disassemble PIA   , nor distribute any modified or disassembled versions of PIA . PIA may not be included with any other product without written permission from AW.

Registered copies of PIA can be used on more than one computer at a time as long as no more than one of these computers is running PIA at the same time. You may make backup copies of PIA as necessary for archival purposes only.

Use, duplication, or disclosure of the software and documentation in this package by the U.S. Government is subject to the restricted rights applicable to commercial computer software (under FAR 52.227-19 and DFAR 252.227-7013).

How to order

Please select your preferred ordering method :

- <u>CompuServe</u>

- Creditcard or cheque from
  <u>Europe</u>
  <u>USA, Canada</u>
  <u>other countries</u>

- or <u>contact the author</u> for other methods of payment

<u>Information on the different editions</u>

Purchase via CompuServe
CompuServe

To register $\mathbf{P}$CB Developer's $\mathbf{I}$ndividual $\mathbf{A}$ssistant complete the following steps:

- Go SWREG #....
  #7624   : Standard Edition
  #7649   : Extended Edition
  #7650 : 32bit Edition
- You will receive the registration notification via CompuServe mail (standard) or the entire program on disks (extended, 32bit).
- To receive the Extended or 32bit Edition also via CompuServe mail
  send a short E-Mail with your preferences to 100606,2612 before you go to the SWREG forum. This will be faster and safer (especially if the snail mail in your country is not very reliable). In this case you will not receive disks.

Note : The prices may vary from the prices listed in the help file. (CompuServe calculates with US$ only.) This could be also an advantage for you. You will see the actual price in SWREG before you order the product. The price will be added to your CompuServe bill.

European order

To register $\mathbf{P}_{\text{CB Developer's}}$ $\mathbf{I}_{\text{ndividual}}$ $\mathbf{A}_{\text{ssistant}}$ complete the following steps:

- You can pay via VISA , American Express, MasterCard or Eurocheque (or Bank Cheque, drawn on a German Bank).
  Please contact Andreas Waldherr for prices for more than 5 licenses
  or other methods of payment..
- Print the order form (click here and select File|Print).
- Complete the order   form (also in the file ORDER.TXT)
- Write a   cheque out to Andreas Waldherr. (If you are not paying by Visa,   Amex or Master)

- Send all to:
      Andreas Waldherr
      Perlacher Str. 31
      81539 Muenchen
      Germany

      or FAX the order form to (if you are paying by creditcard):
        +49-89-6922608

Your registration will be processed as quickly as possible (normally within two working days) and your serial number will be sent to you in the way requested on your registration form (air mail or if possible compuserve mail or fax).

Important : Do NOT send us credit card numbers through the internet ! Normal internet mail is not secure at all. Therefore you will not receive your registration number through the internet.
If you are using PGP you can ask us for our public key.

Note : If you order the Standard Edition you will receive only a registration number. You can order the disks separately, if you need them.
 If you order the Extended or 32bit Edition   you will receive a disk set (no additional fees).
Please contact Andreas Waldherr for delivery via E-Mail.

International order

To register $P_{CB\ Developer's}$ $I_{ndividual}$ $A_{ssistant}$ complete the following steps:

- You can pay via VISA , American Express, MasterCard or via a <u>Bank Cheque</u>.
  Note : 79DM is about $54US (February 1996)
  Please <u>contact Andreas Waldherr</u> for prices for more than 5 licenses
  or for other methods of payment..
- Print the order form (<u>click here</u> and select File|Print).
- Complete the order   form (also in the file ORDER.TXT)
- Ask your bank for a bank check payable to Andreas Waldherr. (If you are not paying by Visa, Amex or Master.)

- Send all to:
  Andreas Waldherr
  Perlacher Str. 31
  81539 Muenchen
  Germany

  or FAX the order form to (if you are paying by creditcard):
  +49-89-6922608

Your registration will be processed as quickly as possible (normally within two working days) and your serial number will be sent to you in the way requested on your registration form (air mail or if possible compuserve mail or fax).

Important : Do NOT send us credit card numbers through the internet ! Normal internet mail is not secure at all. Therefore you will not receive your registration number through the internet.
If you are using PGP you can ask us for our public key.

Note : If you order the Standard Edition you will receive only a registration number. You can order the disks separately, if you need them.
 If you order the Extended or 32bit Edition   you will receive a disk set (no additional fees).
Please <u>contact Andreas Waldherr</u> for delivery via E-Mail.

Please note that a bank cheque is something else than a normal cheque. Your bank will sign this cheque. The check could be also drawn in   US$. Please ask your bank for the equvalent amount in US$.

Requirements and Limitations

Limitations for the different editions:

1. Number of pads and lines:
   - 1000 pads, 1400 lines   Standard Edition
   - 2000 pads, 3000 lines   Extended Edition
   - 8000 pads, 12000 lines   32-bit Edition

     For example : ex_takt1.pcb has 1183 lines (single sided Eurocard)

2. Max. routing area.

3. The 32bit Edition has some additional features:
   - HPGL file output.   (The normal print command will not support plotters.
   - Gerber file output.
   - NC drill file output.
   - Netlist import.
   - Long file names (if you are using Windows95 or Windows NT).

Hardware requirements :
   - Windows 3.1 or later
   - 386   (387 or 486DX desirable)
   - 4 MB RAM
   - 2 MB free disk space
   32-bit Edition only
   - at least WIN32S (or Windows 95, NT)
   - 8 MB RAM

Windows 3.1 + 32Bit Edition:
   If you are using Windows 3.1, you can use the 32bit Edition with Win32S.
   Win32S is an extension from Microsoft to use some 32bit programs with Windows 3.1.
   It is available from various FTP servers
   (e.g. ftp://ftp.cica.indiana.edu/pub/win3/util/win32s.zip).

<<<back

How to contact the author

Software Vertrieb
Andreas Waldherr
Perlacher Str. 31
81539 Muenchen
Germany

FAX :   +49-89-6922608

E-Mail:
- CompuServe : 100606,2612
- Internet : 100606.2612@compuserve.com

Feel free to send us your suggestions, improvements or comments (or bugs).