



LochMaster 4.0 is a developers tools for strip board solutions and can be helpful when...

- optimising the circuit outlines
- fitting the board to the cases
- optimising the components positions
- connecting and wiring the board and its components
- creating components lists
- determination of component types an values
- documentation and archiving

LochMaster supports strip boards with hole distances of 1/10" and 1/5". The maximum board size is 1000 x 1000 mm. Boards with other whole distances can be created. The software provides all typical functions and operations you need, when working with strip boards.

- Selecting parts and components from the predefined, editable [library](#) and placing them on the board. All parts are movable in all three dimensions (x-,y-,z-axis)
- Creating [components](#) groups
- Rotation and duplication of components and [groups](#)
- Using blank or isolated [wires](#), which can be bent and soldered ,as in real life'
- Splitting tracks
- Editable component values and [components list](#)
- Scalable [printout](#) of both board sides
- [Test function](#) for checking wiring of the board and following the signals flow.

Please feel free to send any feedback or take part at our forum.

www.abacom-online.de

What ´s new...?

A quick overview for all users that have already been working with an previous LochMaster version...

- Comfortable 32-bit version with long filenames
- Several boards combined to one project stored in a single file
- "X-ray" view makes components shine through from the opposite board side
- Undo function
- [Unused areas](#) of the board can be displayed
- Several types of strip boards are predefined
- Dockable tools offer customised toolbars
- Exact board dimensions and borders adjustable
- [AutoSave](#) option saves your work regularly

What ´s new in version 3.0

- Bitmaps allow a simple and realistic component design
- File [export](#) for bitmaps or metafiles
- A new [view](#) option allows to look right through the board
- True colour resolution for pens and [brushes](#)
- Integrated text editor for comments, [components list](#), instructions
- Editable [board layout](#) makes LochMaster usable with all types of strips boards
- Lots of predefined board types

- Improved [test function](#) also available for user-defined boards
- [Contour function](#) for automatic bending, rounding and chamfers
- [Component assistant](#) creates standard components with adjustable dimensions
- [Drillings and millings](#) available for cut-outs
- New, extended [library](#) revision with realistic looking components
- Board dimensions up to 300x300 mm.
- Improved [printout](#) of a whole project in one go
- Extensive [bitmap](#) collection for component design

What's new in version 4.0

- Board [size](#) up to 1000mm x 1000mm
- Three user-definable [grids](#) for mm , inch and hole units
- Board [rotation](#) 90° and 180°
- Extendable property dialogues with [extra fields](#), usable as [text variables](#)
- Linking local and [Internet documents and datasheets](#)
- Support of UNICODE characters (international character sets)
- New [soldering point](#) object allows "flying soldering"
- Potential points and [potential](#) colouring
- Copper- and component side editable simultaneously
- Convenient [zoom and scroll functions](#) with mouse wheel
- Individual [view options](#) for each board
- Rotation and [mirror](#) functions
- Object [tree view](#) for structured project view with edit functions
- Export of drilling and milling data ([HPGL](#))
- DDE transfer of [components list](#) to Excel
- Completely updated [print function](#)
- Printout of several board views on one paper sheet
- Individual [print options](#) for each view
- Multi page prints for large projects (banners)
- Print options saved with project
- Free [viewer](#) for LochMaster projects
- And other things...

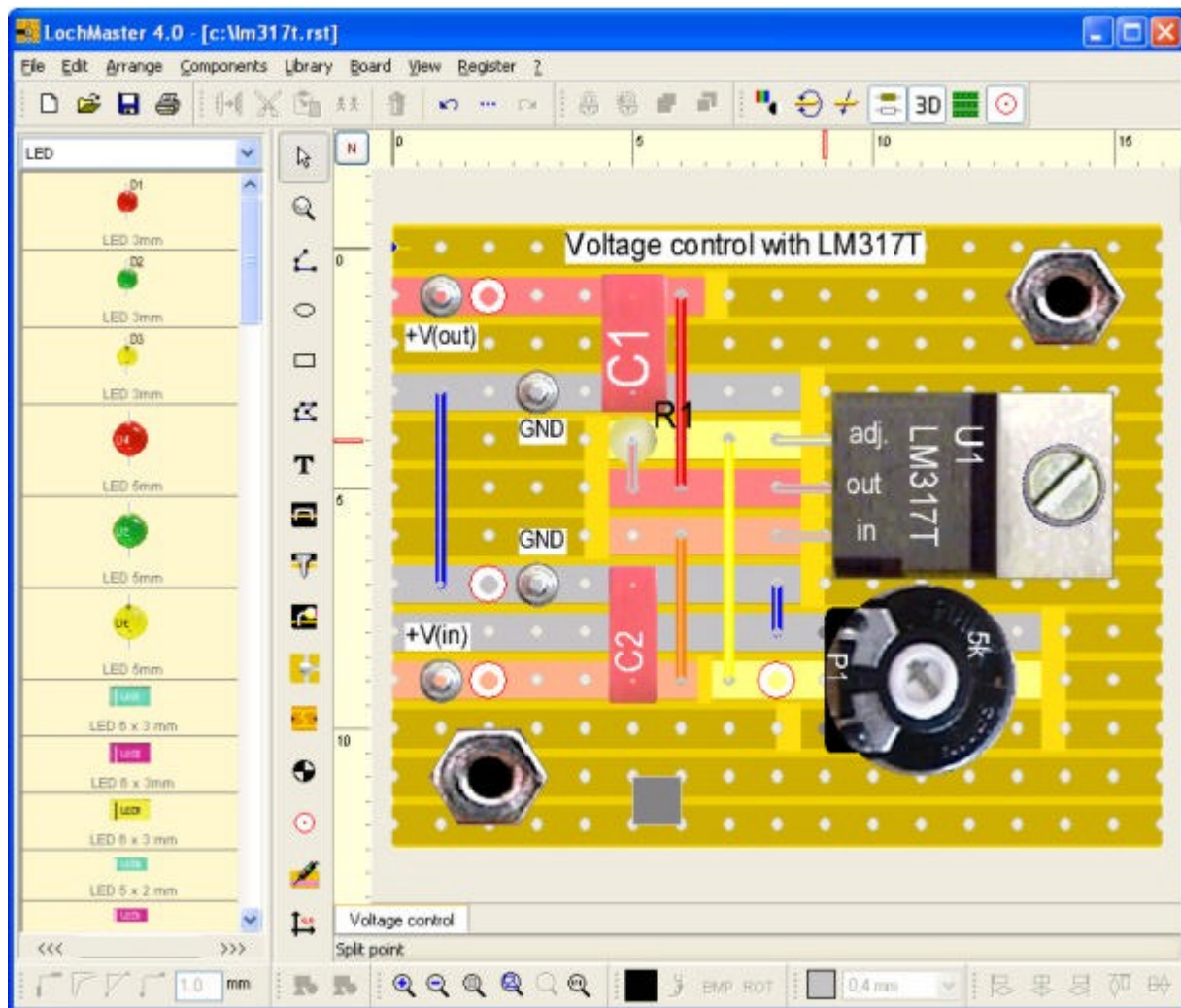
The Viewer

The Viewer is used to open, view and print LochMaster files.

The viewer contains only of one single EXE-file: LOCHMASTER40_VIEWER.EXE

The Viewer is freeware, so everyone can view and print LochMaster projects.
It can be found in the installation directory of your LochMaster software.
It is also available as a free download from our homepage.

Introduction



LochMaster 4.0 – The software and its components, started the first time.

- Menu
- Upper toolbar with tools
- Mode selector for the basic editing and drawing modes
- Components library containing parts and symbols
- Strip board with rulers and parts
- Moveable tools in the editor
- Board selector
- Lower toolbar with tools

LochMaster 4.0 displays helpful hints to all switches and tools, when the mouse is moved to one of these elements.

LochMaster 4.0 allows **individual changes to toolbars** (docking). Changes are saved automatically, when the software is terminated. So the software may look different from Fig.1, when it is restarted, and changes were made to the toolbars.

There are two toolbars above and beyond the strip board editor, containing several tools for changing colours, adjusting elements etc. The position of the tools within its toolbar is changeable and tools can be moved from one toolbar to the other. To do this, you find a small handle on the left of each group of tools. Click on the handle and move the tool to the preferred position, while holding the left mouse button down (drag). Drop the tool at the new position, by releasing the mouse button.

Tools can also be moved to the editor, where they are displayed in a small tool window. You may then close this tool window to hide the tool completely. Hidden tools can be found in the main menu. Select VIEW->SHOW TOOLS to restore hidden tools.

The width of the component library is adjustable. You can decide yourself how much spaced is used for the library. Move the mouse to the right border of the library and drag the border line (between library and vertical toolbar) to a new position. The cursor appears as a double arrow to indicate this function. Hold down the mouse button and release the border line at its new position.

Board properties

When starting the software, the last edited project is being loaded to the editor. A new empty board is being created automatically, if no existing project can be found

You can start a new project at any time with the NEW item of the FILE menu. You may be asked to save your first. After that a file dialogue will offer several board types (*.LMB) that can be opened. An empty board without copper and drillings will be created, if no board type is selected.

Select BOARD->PROPERTIES from the main menu. Enter a short description to the field NAME, like 'power supply'. Enter different width and height, if necessary. Maximum size is 1000mm x 1000 mm.

Ruler / Grid

Three different grids can be used for each board, which are likely to be used:

Units „N“

This is the boards physical grid (hole distance), which is usually 1/10 inch (2.54 mm). Ruler display count units, id this grid is active.

Units „mm“

This grid is useful drawing in metric units. Rulers show mm units, if this grid is active. A value of 0.1 mm seems to be a good choice.

Units „inch“

This grid is useful for drawing in inch units. Rules show inches, if this grid is active. A value of 1/100 inch was found to be useful.

You can choose one of these grids, whenever this seems to be useful in a certain situation.

This is done with the menu command VIEW->RULER/GRID or with key combination "CTRL"+"<" or with the button found in the ruler origin.

Switching the grid will affect the ruler as well as the mouse capture. Drillings are handled with priority to the grid capture. If the mouse cursor is moved onto a drilling, the centre of the drilling is captured, even if it is not on a grid point.

If necessary, the capture can be disabled temporary holding down the SHIFT key.

Origin

The origin marks the zero position fort he ruler. It is also the starting point for all grids.

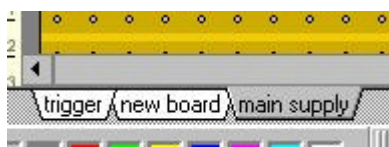
The origin is marked with a blue point on the board.

Offset (Board<->Copper)

You may change the board OFFSET to move the copper layout lightly on the basic material (carrier).

This is because the material sometimes is cut straight through the holes, or sometimes the cut runs between two holes. The offset is set to a default value, when you select a new board type. Change the values if necessary.

You may add additional boards to your project, by selecting BOARD->ADD BOARD from the main menu. You find an entry to each board in the board selector section.



Board selector

Click on one of these entries to show the board in the editor. You find some more operations on Boards

in the main menu. Use them to DELETE or DUPLICATE boards. Use IMPORT to add all boards from an existing LochMaster project file (*.rst) to the current project. Select BOARD->TO LEFT/RIGHT or BOARD->MOVE LEFT/RIGHT from the main menu to sort the entries of the board selector. You can also reach the board operations from a local context menu. Click with the RIGHT mouse button on the board selector, to open the local menu.

Board rotation

The menu command BOARD->ROTATE allows you to turn the board into the preferred position.

Different views



You can change the view of the board, using the tool ZOOM. Use the buttons (+) and (-) to adjust the magnification. The button BOARD adjusts the zoom so that the whole board is visible in the editor. The button COMPONENTS makes all components fit the screen. The button SELECTION displays all selected parts on the screen.

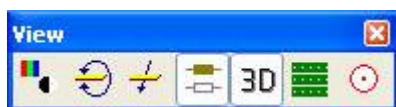
Another way to adjust the zoom is to switch the mode selector to ZOOM-mode and then move the mouse cursor (magnification glass) to the editor. In zoom mode the left mouse button increases the magnification, the right mouse button decreases it. You can also draw a rectangle that includes the area to be zoomed. After you finished zoom adjustment set the mode selector to DEFAULT mode (arrow). If the board does not fit the editor screen, because of its magnification, you get scrollbars on the left and lower edges of the editor. The rulers on the left and upper edges of the editor help you, to count the row and column you are working at.

New in version 4.0:

A mouse wheel is helpful to adjust the zoom. Turning the wheel changes the zoom. Holding down the mouse wheel and moving the mouse scrolls the screen.

The board can be displayed in different views on the screen or printed. The tool VIEW offers the following functions.

- Colour mode and black and white (outline) mode
- Solder and component side of the board (turn over)
- Solder and component side of the board (right trough)
- ‚X-ray‘ makes components on the opposite side shine through
- Bitmap fillings on/off
- Show unused board areas
- Turn potential colouring on and off



The black and white (outline) mode may be useful, to work on parts that are overlapped by other components. This display mode may also be used with b/w-printers to improve the readability of the printout.

[Bitmap fillings](#) should usually be turned on. You can turn them off on older computers that are too slow to process bitmaps without acceptable delay.

You can combine any of these display options freely. They do not affect your project in any way, but simply display your board differently.

New in version 4.0: View options can be adjusted individually for each board in a project.

How to add wires

To realise a wired connection on the board, do the following:



Switch the mode selector to the mode WIRE.

Move the mouse cursor to the board, and click on the wanted start position of the wire. The wire has now been created and you can solder the other end of the wire with another mouse click. You can also use the right mouse button to cancel the wire and restart from another position.

If you wish to bend a wire hold the CTRL-Key on your keyboard down. In that case the end of the wire is not being soldered to the board, but you get a bend point from which you can go on to another position until you release the CTRL-key to finish the wire. If you have to reach a position between the holes, hold the SHIFT key down. As long as the wire is not finished, you can use the right mouse button, to undo the last wire operation.

How to change wires



Switch the mode selector to default mode.

In this mode the mouse cursor may change to inform you about possible operations.

Cursor: Arrow....

You are pointing to an area, where nothing is placed at. A click to this position unselects all components.

Cursor: Pointing hand...

You are pointing to an unselected component. You may click here, to select the component.

Cursor: Arrow...

You are moving on a selected component. You may click and hold the left mouse button down, to move the selected component(s) to another position.

Cursor: Cross + modify

You are pointing to a changeable point of a wire or a line.

Do the following to change a wire:

Click on the wire you want to change. The wire changes its colour to magenta, to show that you have selected it.

If you have selected a wire you can...

...change the start or end point of the wire

Click on one of the extreme points of the wire. The wire is now unsoldered from the board, so you can solder it at another position. See also Step 3 - Wiring

...move the bend point of a wire

Click to a bend point and move it to another position, while you hold the left mouse button down.

...move the wire

Move the mouse on the wire, until the cursor changes to flat hand. Then hold the left mouse button down and drag and drop the wire to its new position.

... change the diameter or the colour of a wire (isolation)

Click on one of the colours from the colour palette tool. The wire will change its colour. Avoid magenta to make sure, that you can see the difference between selected and unselected wires. Select a diameter from the WIDTH tool.



New in version 4.0:

Colour and pen width are adjustable in the property dialogue. The property dialogue opens on a double click to the object or is called from the local popup menu (right mouse button).

...change the contour

Select the corresponding function to assign bending, rounding or chamfers to a wire or other object. Enter a value for radius/length of the rounding/chamfer to the edit field before.



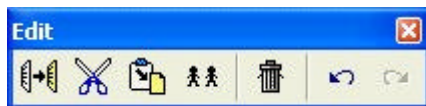
Remember that you have to select an object, before you can assign new values using these functions!

Edit functions



Switch the mode selector to default edit mode. The editing-functions COPY, CUT, DELETE, etc. work, as they do in Windows: First select the object to perform an operation on, and then call the edit function.

Selected objects appear magenta coloured. Select an object with a click. If more than one object has to be selected, hold the SHIFT key down on your keyboard. To deselect an object, click it again, while holding the SHIFT key down. Other object will remain selected. Instead of clicking each individual object to select it, you can also draw a frame to select all object that cross the frame. To deselect all selected objects, click to an area, where NO object is located. Use the main menu entry to SELECT ALL objects on the board. Additional information about selected objects is displayed on the INFO-display tool. After selecting some components of the board, you can perform an operation from the EDIT menu:



DELETE: deletes all selected objects. You can also reach this operation, by pressing the DEL-key on you keyboard. COPY copies the selected objects to the clipboard. As the objects are special components of a strip board (including component data etc.) you can NOT paste these object to other applications. CUT first copies the selected objects to the clipboard and deletes them from the board. PASTE adds components from the clipboard to the strip board. This works similar to adding components from the library, but no component data is requested. DUPLICATE performs COPY and PASTE with a single step. ROTATE: Objects are cut from the board, copied to the clipboard and rotated with an angle of 90 degrees. After that you have to reposition it on the board. While doing this, you can rotate the component again, using the SPACE-key of the keyboard.

Edit functions may be disabled, depending on your selection. For operations like COPY and CUT at least one object has to be selected. ROTATE is only available with a single component or group selected. Edit functions can also be called using the toolbar buttons. If default edit mode is set, you can also reach these functions by pressing the right mouse button, to get a local context menu.

To MOVE components, select them and drag & drop them to the wanted position. Mouse capture can be turned of temporary holding down the SHIFT key.

When cutting or deleting components, the remaining components will NOT be renumbered. Select COMPONENTS->RENUMBER from the main menu, to renumber components. You may undo your last performed operation by selecting UNDO from the main menu or pressing <alt><backspace> on the keyboard.

File operations

The FILE menu contains the well known operations SAVE, SAVE AS, OPEN and NEW, as well as PRINT and QUIT. LochMaster saves your projects to a files with the extension *.LM4. Files with *.RST extension from previous versions can still be loaded. The file includes all boards with their components, including the component data as well as remarks, etc.. Because of the specific file format these files can not be imported to other applications. The current filename is displayed in the programs headline. The file extensions *.RST and *.LM4 are linked with LochMaster automatically when, the software is installed. You can simply click to a project file (*.RST; *.LM4) in the Windows Explorer to start the program and load a project.



Use SAVE to write changes to the file displayed in the programs headline. In that case SAVE AS is called, so you can give a name to your project. Use SAVE AS to rename your project, or save it to another position. If you do not type the extension, the extension *.LM4 will be added to the filename. You better don't use other extensions. If a project file was created by an older version of LochMaster, it will be converted to the new file format. After saving such a file, you can no longer use it with the older version! Always use the latest program version! Use OPEN to read a project from file. The last edited file is loaded automatically, when LochMaster is being started. If you select NEW from the file menu, all boards of your project will be deleted. Before that you will be asked to save your project. To start a new project, you will be asked to enter the properties for the first board.

The EXPORT function can be used to create a graphics file of a board or to export HPGL machine files, which can be used on CNC milling machines.

AutoSave

Select FILE->AUTOSAVE from the main menu. If the option is activated, your project will be saved in the given interval, using the extension *.BAK. If you have to use the BAK-file for some reasons, you can simply open it. File extension will then be changed from *.BAK to *.LM4, so you can continue your work immediately.

New in version 4.0:

Opening an existing project, LochMaster saves an additional backup file with .OLD file extension, which provides some security in case of crashes or unintended changes.

Draw operations

There are basic drawing functions to draw lines, rectangles, circles, polygons and text. Extended drawing functions are PIN, CONNECTION, DRILLING and SPLIT.

Basic drawing modes:



Set the mode selector to LINE, to draw a line. Click to the starting point of the line, then click again to determine the next point of the line, and so on. Use the right mouse button, if you finished the line. As long as line mode is selected, you can continue with more lines, until you select another edit mode. You can easily reach the default edit mode from every other mode, by pressing the right mouse button.

All basic drawing modes use a black pen with a width of 0,1 mm. You can assign different colours or widths from the toolbar. To change colour or width later, select default edit mode, select the objects to change and then adjust colour and width.



Set the mode selector to RECT, to draw a rectangle. Click to the rectangles top left corner and click again to its bottom right corner. If you want the rectangle to be filled, select the option FILL from the toolbar. To select the fill colour click to the colour palette, using the right mouse button.



Set the mode selector to CIRCLE. First click to the circles centre. Move the mouse, to adjust the radius and click again. Adjust width and colours.

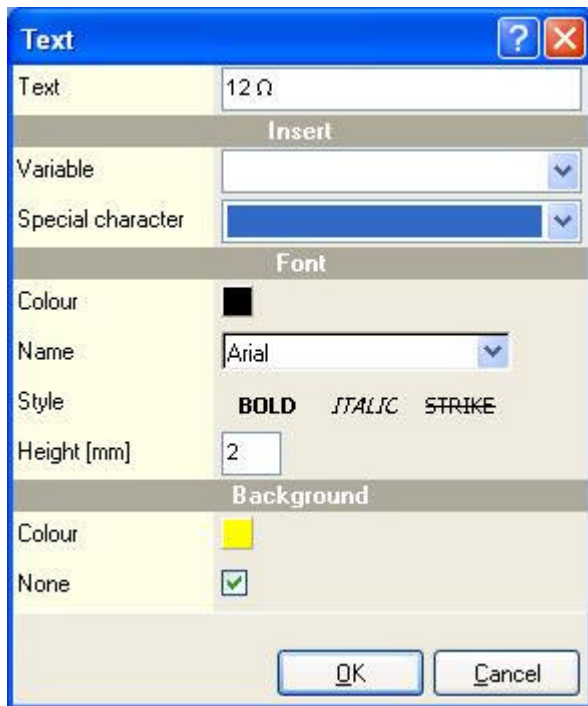


Set the mode selector to POLYGON. Polygons are drawn in the same way as lines, but the figure is closed automatically when finished.

New in version 4.0:

Drawing elements are equipped with property dialogues. These are opened in standard mode with a double click onto an object or called from the local popup menu (right mouse button). The property dialogues can perform additional manipulations, like direct entry of positions or object rotation. You can also enable objects for HPGL export or add extra data fields.

T Set the mode selector to TEXT, to add a text to the board. Click to the text position and then enter the text to the dialog.




New in version 4.0:


Text objects as well as the software itself now support UNICODE character sets. This makes it possible to use special characters (like Ohm character; Omega). Often needed special characters can be chosen from the select list to add them to the TEXT definition.


The VARIABLE selection adds variable fields to the text definition. <variable names> are included into <...> brackets and are replaced with field contents during display process. <Date> will be replaced with the current system date for example. Other variables provide information on system, project or component. Extra fields can be use as text variable as well.

Extended drawing modes:


These modes were designed for use with creation of your own library components.

 WIRES are soldered with both endings to the board. To create your own components you need two more objects, which indicate the kind of connection that is used to contact the component to the board.

 CONNECTIONS consist of a piece of wire, where one end is soldered to the board and the other end is fixed to the component. You find that kind of connections with resistors, capacitors, diodes etc. Drawing connections is like drawing wires. Set the mode selector to CON. The first click marks the soldered end of the connection. Drawing direction always is from the board to the component.


 PINS that are soldered orthogonal to the board and run straight from the board to the component, such as IC-pins To draw a pin, set the mode selector to PIN, and click to the position, where you want a pin to be soldered.

If you are drawing your circuit or component, always make sure, what kind of object you use. WIRE, PIN or CONNECTION. This is very important, to take advantage of the test function and to make your own components work in the way you want it to.

 SOLDERING POINTS are used to connect wires, leads and pins beside their end points. LochMaster handles crossing wires as electrically isolated. To get them still connected place a soldering point at the crossing point. The soldering point will connect all pins, wires and leads that get in touch with it.



In case a copper area is located under a soldering point, the property dialogue lets you choose whether the soldering point connects to the copper or is meant to be „flying“.


 To set a SPLIT to the copper layout, set the mode selector to SPLIT.

A split is set with a single click. To adjust the size of the split, click the left or right half of the split and hold the mouse button down, while moving the mouse.

 Set the mode selector to DRILL mode.

A dialogue requests the drill diameter. You set a drilling with a single click anywhere on the board. This mode is terminated with the right mouse button. Drillings are visible from both sides of a board.

Millings appear as well on both sides of a board. Millings are drawn with the basic drawing function LINE RECTANGLE, CIRCLE and POLYGON. These objects can be enabled for HPGL export in their property dialogue.

 Electrically connected areas can be marked with POTENTIALS, to display them with a certain colour. Set the mode selector to POTENTIAL mode. Enter a NAME and COLOR for the potential. Close the dialog and click to the desired position to add the potential.

Fillings

Closed figures (i.e. [circles](#), [rectangles](#) and [polygons](#)) can be filled with a simple colour or with a bitmap picture and the filling can be turned on and off.



Select some objects that can be filled first. Turn on the fill option for these elements. A simple colour can now be assigned with a click with the right mouse button on the colour palette. A user defined colour can be assigned with a click on colour field of the FILL tool.

The BMP button loads a bitmap from a bitmap file (*.bmp). Lots of useful bitmaps have been installed with the software to the BITMAPS directory. A bitmap is unloaded again when a simple colour fill is assigned to an object. Bitmaps are automatically stretched to a size that fills out the assigned object completely. In case the loaded bitmap has a bad orientation, use the ROT button to rotate the bitmap in steps of 90°.

Important notes for use of bitmaps!

- Minimise the size of bitmap files (resolution, file size) whenever possible. Mostly bitmaps with a size of 100x100 pixels maximum are sufficient for a satisfying result. Especially bitmaps for user defined components should not be greater than a few kByte. Also refer to the examples in the BITMAPS directory!
- Avoid true-colour-resolution of bitmaps and convert them to bitmaps with 256 colours (8 bit palette) with a graphic software, before you use them with LochMaster.

Large bitmaps may otherwise reduce performance speed, especially when running LochMaster software on older computer systems.

Arrange functions

You can arrange objects by setting their z-position and combining them to groups. The functions are settled in the main menu item ARRANGE and can also be called from the toolbar.



To front / To back

Select the objects that have to be arranged. Example: You have added a resistor to your board and then drawn a wire over it. Now you wish to put the wire under the resistor. Select the wire and call **ARRANGE->TO BACK**. You could also select the resistor and call **TO FRONT**, which would have the same effect. If you select more than one object to use with these functions, all selected objects will be set to front or to back, but the relation between the selected objects will stay untouched.

There are two exceptions where z-order of object can not be changed: Text objects will always stay in front, so that they are readable. Splitter will always stay in the back, because they are directly combined with the copper layout.

Building groups

Objects can be combined to groups, to make them selectable and editable with one single click. Groups also protect their elements against unwanted changes in colour etc. Members of a group can not be deleted separately, but the soldered ends of wires and connections are editable. You need at least two elements to build a group and a group can contain one or more sub-groups. To build a group, select the members and call **ARRANGE-BUILD GROUP**.

Splitting groups

If you wish to edit a group member separately you have to split the group. Select the group and then perform **ARRANGE-SPLIT GROUP**. If the group contains subgroups these will stay untouched. You may repeat this action with the subgroup.

Adjust, rotate and mirror functions are also part of the arrange functions. Notice that **PLACE ON OPPOSITE BOARD SIDE** does NOT consider components geometry, but is only performed graphically. Executing this function on a DIL IC for example, will result in a component with a (mirrored) incorrect pinout. In such case the **MIRROR** function needs to be called additionally. An alternative is to perform a **CUT, TURN AROUND BOARD** and **PASTE** on after each other.

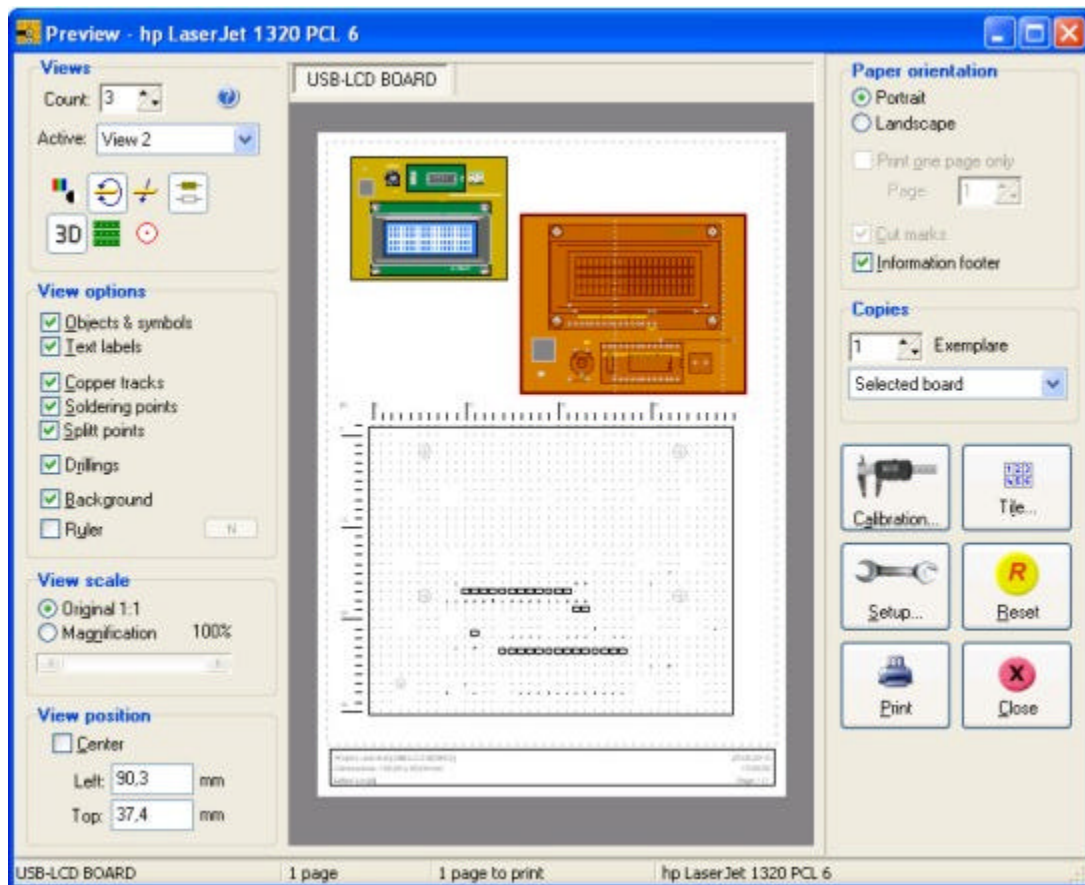
Adjust functions

Use the tool **ADJUST** to adjust simple elements of your drawing like rectangles, circles, text etc. Select all objects that have to be adjusted. The last selected object is the reference object, which will stay at its position. Then click to the **ADJUST** tool to select the direction of adjustment. The reference object can be a complex element like a group or component.



Printing

Select **FILE->PRINT** to get a preview of the printout.



The print function was updated completely. New features are as follows.

- Multiple views of a board can be arranged on a single paper sheet.
- Each view has its own view options, scaling and position, which are independent from other views and editor settings.
- Large boards or views can range over multiple paper sheets. (banner)
- All settings are stored together with the project.

First of all number of views has to be defined (COUNT). In above figure three view were defined (component side, copper side and split points)

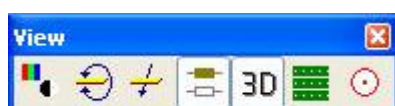
In the centre of the print dialogue, you find a preview of the printout, which allows you to adjust the view positions and to select the ACTIVE VIEW.

All view options are set individually for one certain view, which is the ACTIVE VIEW. The active view is indicated in the preview with a red frame, which will not appear on the paper printout. The active view can be selected with a mouse click onto the desired view in the preview or with the "ACTIVE" select list.

To change the position of a view, you can drag it with the mouse to the desired position, or you can enter the position to the LEFT and TOP fields. The CENTER option adjusts a view to the page centre.

The scale adjustment (size) is individual for each view. A value of 100% will give a printout 1:1 of the board dimension. In case a view exceeds the paper range, additional pages will be added automatically. CUT MARKS can be enabled in this case, to fit pages together exactly after printout. Select PRINT ONE PAGE ONLY to disable this automatic function and select the page to be printed manually.

View options for each view are the same as used for the editor.



- Colour mode and black and white (outline) mode
- Solder and component side of the board (turn over)

- Solder and component side of the board (right trough)
- ‚X-ray‘ makes components on the opposite side shine through
- Bitmap fillings on/off
- Show unused board areas
- Turn potential colouring on and off

In addition you can enable or disable certain object like text, tracks, splits, ruler, etc. for each view.

Last but not least select the number of copies to be printed. Projects with more than one board can be printed in one go or board by board. If desired, you may add information footer to the pages.

The CALIBRATION button enables you to enter correction values, useful compensating little distortions some printers may produce.

The TILE button allows you to printout identical views as tiles beside and/or beneath each other.

The SETUP buttons open the printer's configuration dialogue. You can select a printer (driver) and adjust its individual properties.

The RESET button recalls the default factory setting for ALL print settings.

The PRINT buttons sends the print job to the printer immediately.

The CLOSE button returns to the main program.

Basically the paper printout should produce exactly the same result as the screen preview. Technically spoken screen and printer receive exactly the same graphic commands, executed on different canvases. On the other hand, the printer driver needs to adapt these commands to devices specific given facts. For example colour printouts are converted to grey scale printouts on a black & white laser printer. Not that easy!

In cases the paper printout shows differences against the screen preview, in most cases the print driver is responsible for that, even if everything seems fine in other, more simply applications.

Recently appearing problems may be...

- No printout on EPSON Stylus 660: Chose previous (model 640) driver or driver from Windows CD-ROM.
- Missing lines: Some laser printers accepts black pen colour only.

Try out different drivers that may be available for your printer. (Windows, Manufacturer, Internet). Make sure the driver fits to your Windows release. Check out device dependent printer driver options.

Test function



To test the board set the mode selector to TEST. This mode is helpful to follow the signals flow and find wanted or unwanted connections. Click to one point of the circuit, and the program will find all areas of the board that are connected directly to that point. Wires, connections, and pins will be considered, but the software can not find connection that are build with 0-Ohm-resistors. Connected areas are marked with the colour magenta until you click to another position, or select another edit mode.

Object tree view

The object tree view is a powerful tool offering several advantages. It opens on the menu command BOARD->OBJECT TREE VIEW.

The tree view is a list of objects of the board design. Groups and components appear as a single entry at first, but will drop down when selected, to show the contents of the group.

A click on a list entry will select the object the same way as a click on the object itself in the editor. However the advantage of the tree view is, that objects are reachable that can not be reached with the editor. For example you can select objects out of a group, without having to split the group. This can be very helpful in case you have to manipulate a single object in a group. For example you can easily change the pin of a component.

As well as the editor, the tree view offers a local popup menu (right mouse button). The tree view allow drag & drop operations to change object positions.

Extra fields

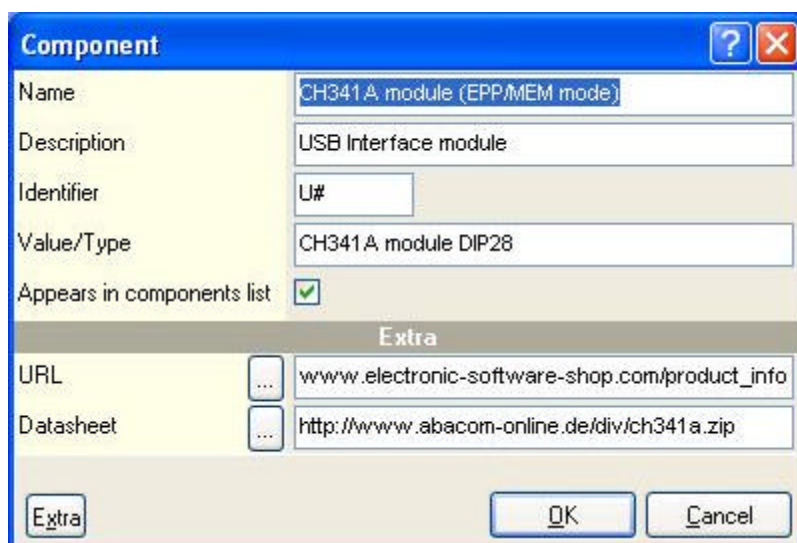
LochMaster offers property dialogues for objects and components that are opened on a double click or with the local popup menu (right mouse button). These dialogues can be extended with user-definable extra fields, to enter additional information, like datasheets, manufacturers, distributors, handling advices, etc.

The EXTRA button can be found in the bottom left corner of the property dialogues.



In case the desired extra field is not yet in the list, use the ADD NEW FIELD button and enter a unique name for the new field.

Checked fields in the list will appear in the property dialog. Unchecked fields will be removed from the property dialogue and their content will be deleted. As soon as you confirm the extra fields dialogue, selected fields will appear in the EXTRA section of the property dialogue.



Now some content can be entered to the extra fields (here "URL" and "Datasheet"). In case the field content is a link to a local or Internet document, LochMaster will add a button "..." in front of the content.

field, which allows you to open the link and view the document directly.

The pool of field definitions is stored together with a board. A field definition– once defined – is available for all objects within a board (or within a library page). Unused field definitions will be deleted automatically. Names of extra fields can be used as <variable> in text objects.

Component lists and annotations

Component lists

The software creates a component list from the components data automatically. Select COMPONENTS LIST from the BOARD menu. There are two kinds of lists that can be created. The first one contains an entry for each component used with the board, the second is an ORDER LIST that collects components with the same data to one entry in the list. You can edit the components list and save as text file (*.txt or *.rtf) or print it. Use the toolbar in the list window. The list is only created temporary and changes are lost if the list is closed. It is recommended using the following ANNOTATIONS option, if you have to save to components list together with your project.

Create Excel list

This function launches Microsoft Excel (if installed) and transfers component data through DDE (Dynamic data exchange) into a new Excel table. This function may fail under restricted user Windows user accounts. The function may take some time to execute. As DDE is known to be not the most reliable mechanism, we recommend saving your project, before executing this command.

Annotations

The menu item ANNOTATIONS can also be found in the BOARD menu. A text editor opens and allows you to enter comments, construction hints, components lists or anything else to each board of your project. The annotations are saved together with your project file automatically. Again you find buttons for creation of component lists and order lists in the toolbar. Annotations are part of your project, which means that they are save when the text editor is closed. You can open and close the annotations whenever you like. Annotations refer to the board that is activated with the board selector. Each board has its own annotations.

Library operations

The library consists of several pages with components on it. You can adapt the library by creating new pages, adding components, renaming, deleting or sorting them.

LochMaster saves libraries automatically to files LIB1.LIB.....LIBx.LIB in Windows directory „AllUsers-AppData“. This means C:\Documents and Settings\All Users\Application data\LochMaster40\Lib\ in a typical Windows XP installation.

Library file are saved automatically, when...

- using [file functions](#)
- changing the library page
- terminating the software

To create a backup of the library files simply copy the lib directory to a save position.

Creating a new library page:

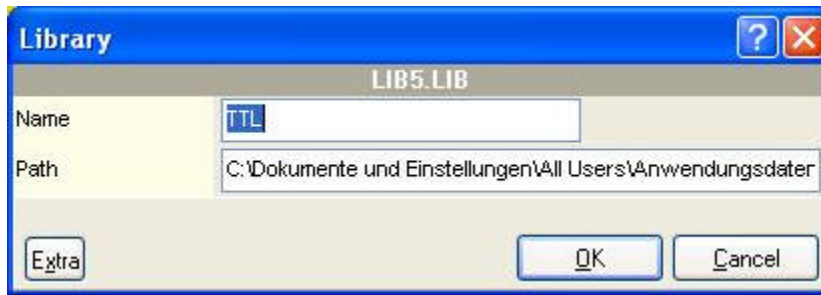
Select LIBRARY->NEW PAGE from the main menu. You are asked to enter a name for the new page. After confirmation the library will contain your new empty page.

Deleting a library page:

Select LIBRARY->DELETE PAGE from the main menu. After confirmation the page will be deleted and is then gone forever.

Renaming a library page:

Select LIBRARY->PROPERTIES from the main menu and enter the new name.

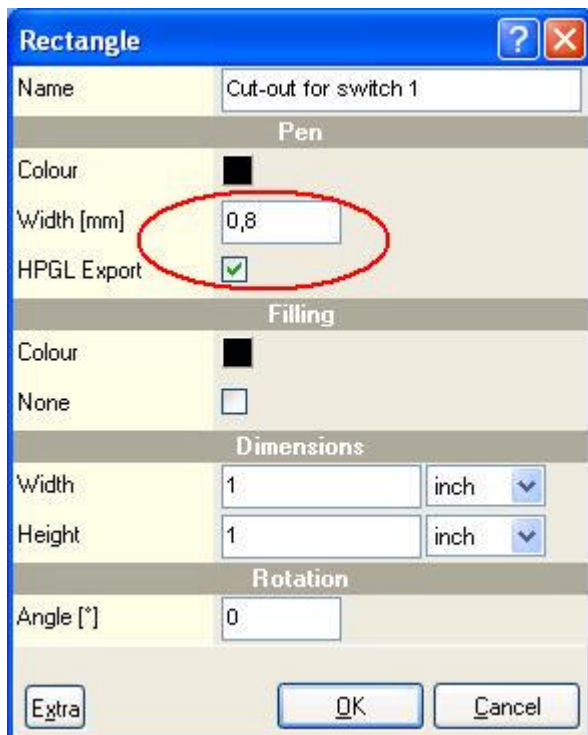


In case you wish to change the library path enter a alternative valid location to the path field. This does NOT change, move, modify or even delete any files. For file operations use the Windows Explorer. The path simply tells the software, in which directory the LIB files are located.

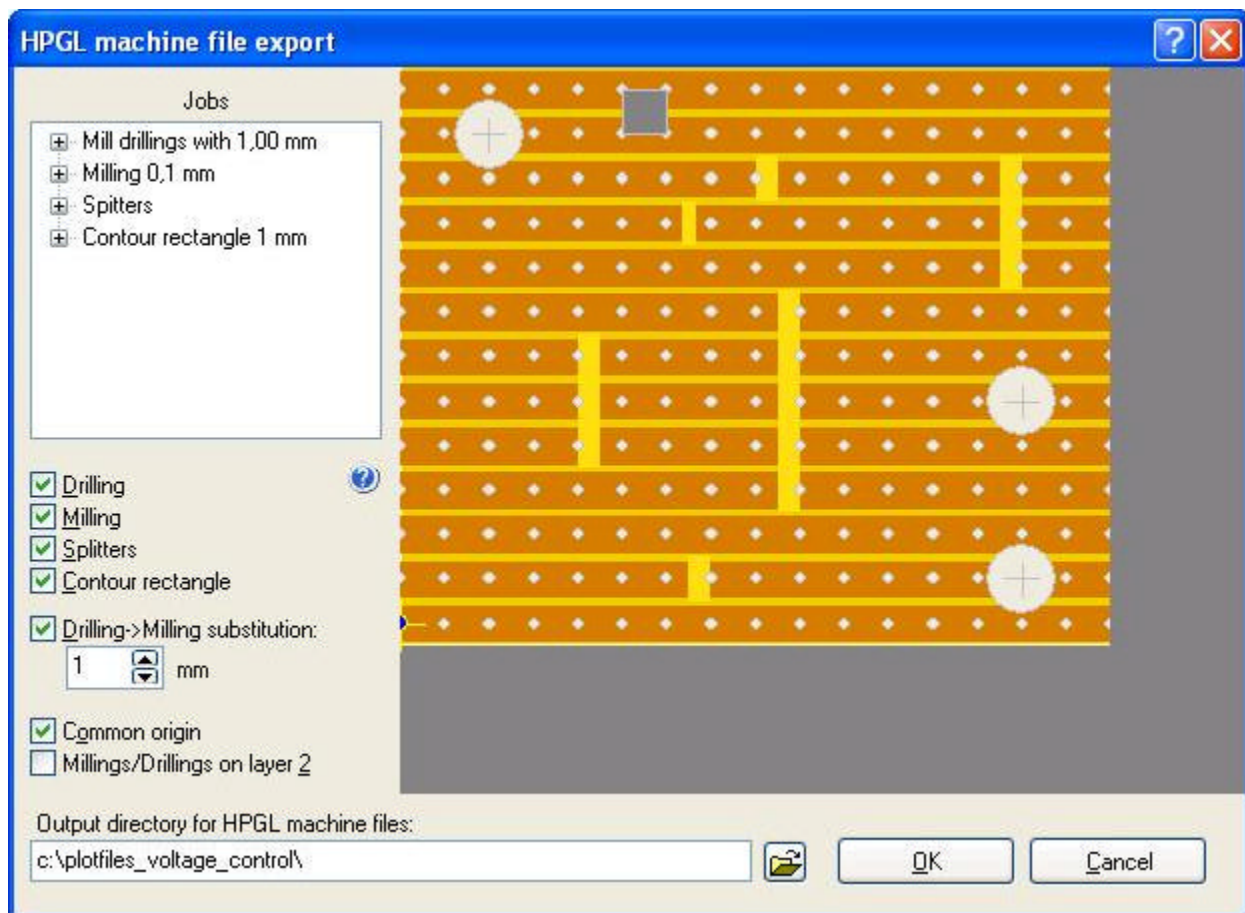
These functions are also reachable by a local context menu, if you click with the right mouse button on a component that is still in the library. Here you will also find the functions TOP and BOTTOM to sort the library. The entry PROPERTIES allows you to modify the component data of a library component. Changes will not effect the components that have already been placed on the board.

HPGL export

LochMaster can export HPGL machine files for use with CNC milling machines. Exported objects are drillings, splits and the board's contour rectangle.



Lines, circles, rectangles and polygons drawn with the basic drawing functions can be enabled for HPGL export as well in their property dialogues. The pen with represents the tool width in this case.



The export dialog is available from the main menu FILE->EXPORT->HPGL MACHINE FILES.

For file export all drillings, millings that use the same tool (diameter) are collected to one so-called job. The dialogue shows a list of all necessary jobs and the object that were collected for the job. You can check jobs that have to be exported: MILLING, DRILLING, SPLITTERS and CONTOUR RECTANGLE. The contour rectangle option creates a plot job that describes the outline rectangle of the board and considers the tool diameter. This is because you can not draw such a rectangle in the editor, because its co-ordinates are outside of board.

As splitters must be milled from the boards copper side anyway, all other jobs are generated to be executed from the boards copper side as well in any case. So there is no need change clamps on the machine between two jobs.

Important: Exported SPLITTERS do NOT consider the tools diameter. Only their outline rectangle is exported. Inside areas are not milled, even if you might expect that. A milling diameter of 0,8 mm was found to be a good choice for the splitters.

If you wish to change the diameter of some jobs that are exported, you can click on the job with the right mouse button. A local popup menu with the entry MODIFY allows you to change tool diameters. The option COMMON ORIGIN adds a PD/PU (pen down/pen up) command at the physical origin (Point 0;0 = left/bottom corner of the board) to each plot job. This option may be helpful to adjust the origins when several jobs are executed one after another.

It may be useful in some cases not to drill drillings with a drilling tool, but to mill these holes instead. In that case the diameter of the milling tool must be of course less than the minimum drill diameter appearing. Therefore the option DRILL-MILL SUBSTITUTION can be activated. You need to specify the diameter for the milling tool that is used to cut the holes. All drill jobs will be replaced with a single mill job, when this option is checked.


The Option MILLING/DRILLING ON LAYER 2 adds a HPGL command "SP2" (select pen 2) to milling jobs and drilling jobs. This option does not have any effect on SPLITTER jobs. Some software products use this option to cut deeper into the material automatically.

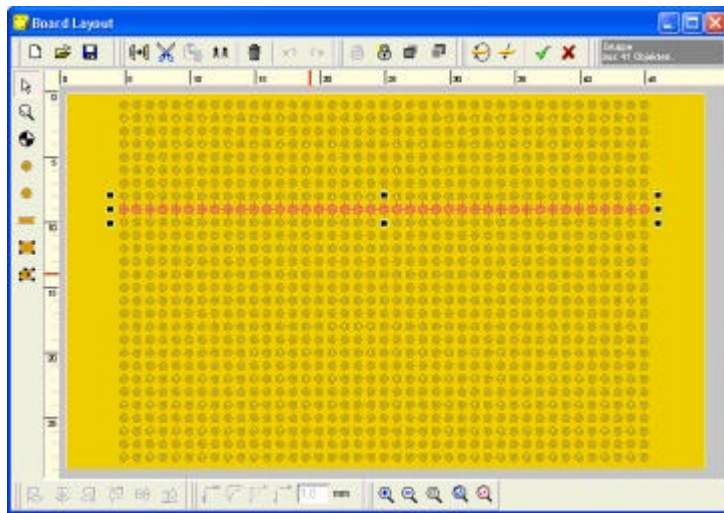
The output directory in which the export files are generated is shown on bottom of the dialogue. The

directory name is generated from the projects filename, and can be modified if necessary. Directories that do not exist will be created. A click on the EXPORT button will create a HPGL file with the name of the job and with the extension .PLT. When export is finished files are displayed in the Windows Explorer HPGL files can be read and processed by almost any plotter device. HPGL viewer programs can be used to verify exported information.

The board layout editor

With LochMaster you can create your own boards.

 The board layout editor is open with a click on the corresponding button in the vertical toolbar.



The functions of the board editor are almost the same as the functions of the [main program](#). The following additional functions are available.

Pads with drilling

A dialog requests values for pad diameter and drill diameter. After that you can place pads with drillings anywhere on the board with a single mouse click. The right mouse button terminates this mode.


Pads without drilling


A dialog requests the pad diameter. After that you can place pads anywhere on the board with a single mouse click. The right mouse button terminates this mode.

Tracks

A dialog requests the width for the track. The first click determines the start position for the track. Move the mouse cursor to the end position of the track and click again. The right mouse button terminates this mode.

The rectangle function  and the polygon function  are the same as in the main program and can be used to draw copper areas as well.

 Click this button to ACCEPT the board layout for your board. The layout editor is closed and the software returns to your current project, with your new board layout.

 Click this button to REJECT the board layout. The layout editor is closed and the software returns to your current project. The board layout will not be changed.

The board layout for each board of your project is saved automatically with the project file (*.rst). The board layout editor allows you save the board layout, independently and separately from your project file as template file (*.LMB). It is strongly recommended to save these files to the BOARD LAYOUTS directory in the installation path of your software, where also the predefined board layouts can be found.

The *main application* saves and opens only files with the extension (*.RST), which are project files that include all data of a project.

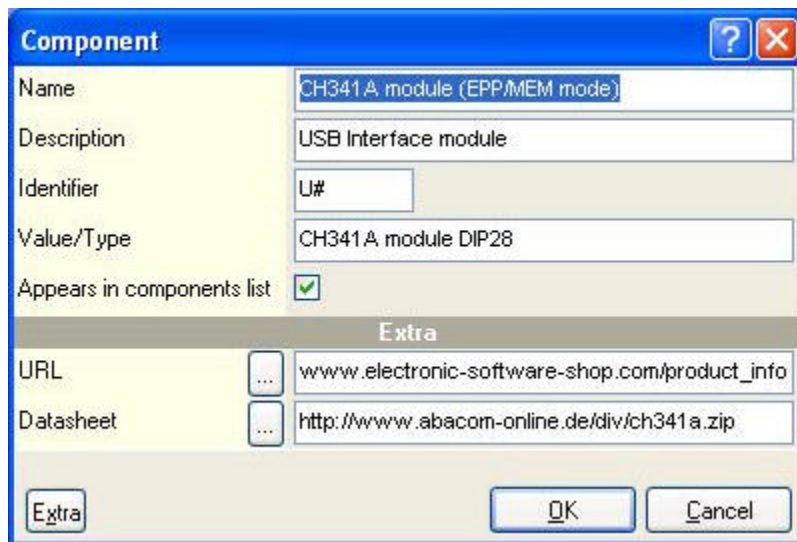
The *board layout editor* saves and opens only files with the extension (*.LMB), which only include the board layout (dimension, copper and drillings). These files are used as templates, when starting a new project or adding a new board to a project.

Components from the library

Components can be selected from the library and can be placed anywhere on the board. You can also use the component assistant to create often needed components with various dimensions.

The library consists of several pages, which contain the predefined components. You will find resistors, capacitors etc. in it. To select a page, use the listbox on top of the library. You can find a page very easily, pressing the first letter of the page name on the keyboard.

If you click on a multipin component in the library, the mouse cursor is moved to the editor and the selected component sticks to the mouse cursor. With a click you can solder it at the wanted position. You may cancel the operation using the right mouse button. As long as the component is not soldered to the board, you can use the SPACE-key on the keyboard to rotate the component. Some components request the entry of some component data (type, value etc.) – Fig. 3



The '#' character in the field IDENTIFIER represents the component number, when using the automatic component numbering option. When you add the component to the board, the software determines the component number for this specific component.

Example: Resistor R1 and R2 already exist on the board. If you add another resistor, its short name is set to R3 automatically. You may override the identifier with an absolute name 'R123' (without '#' character). In that case renumber operations to this component will NOT be performed.

The field DESCRIPTION is used to display the component in the library and the description appears in the components list. The field TYPE/VALUE can be used for additional information to the component. For components like symbols, you may deactivate the option ADD TO COMPONENTS LIST, so that they do not appear in the components list.

New in version 4.0

Components (and other objects) can be given a NAME that is used internally (for dialogue captions and in the object tree view). Leaving the name field empty, LochMaster will display a default name like "wire" or "component" instead. Default names are translated automatically if a project is opened in a different language version, while individual names are certainly not.

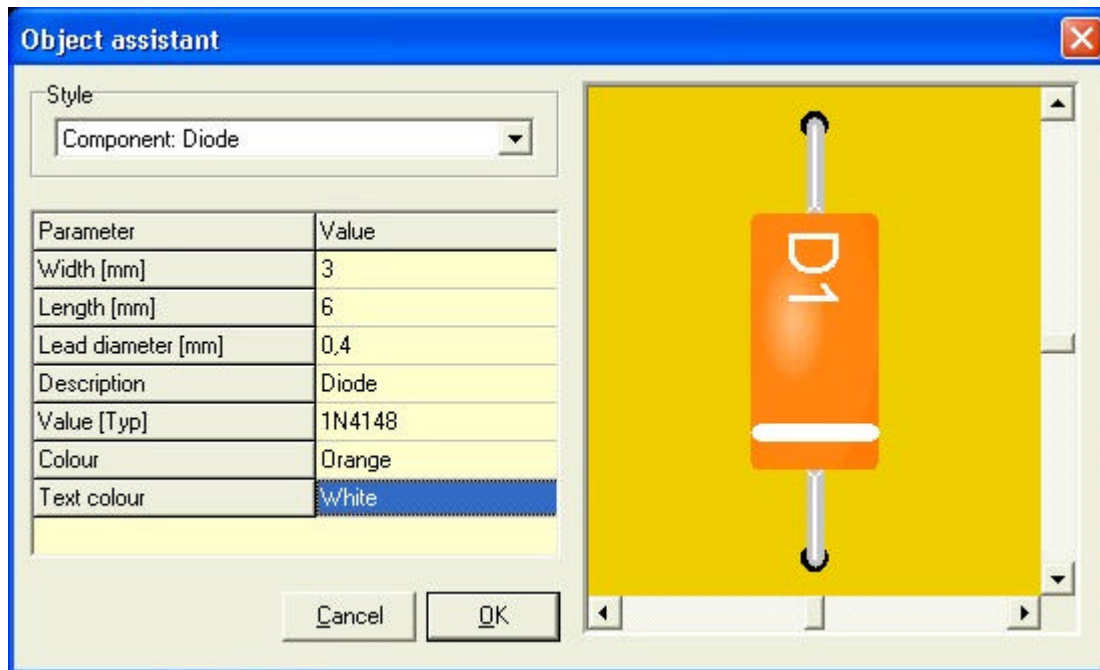
The button EXTRA can be used to add additional data fields to an object.

Coaxial components:

Adding coaxial components like resistors to the board works exactly like adding wires ([see step 3](#)).

The component assistant

You can use the component assistant to create often needed components with various dimensions. Simply select an object from the object list and enter the properties to the parameter list.



The available parameters depend on the object type you have selected. Numeric values can be entered directly from the keyboard. YES/NO selections can be changed using the Y and N key of the keyboard, as well as the SPACE bar, or with a double click. The result can be previewed directly. Select OK to create and add the component to board.

User-defined components

A user-component consists of at least two elements like lines, rectangles, circles, polygons, text, as well as wires, connections, pins and groups of these elements. Basically a component is a group with additional information: The component data. (Identifier, description, type / value)

The first thing to consider is what elements a component consists of. A simple symbol like an arrow may consist of a polygon and a line. To build a component from these two elements you have to mark them and then select ADD TO LIBRARY from the LIBRARY menu.

You now have to enter the component data of your new component. For a simple symbol like an arrow only the field DESCRIPTION is necessary. You can enter a name like 'My Arrow'. The arrow will then appear on the current library page and will show its description 'My arrow'. It can now be used as any other component.

Now we can try to create a simple IC-component, like a timer NE555, which is delivered in a DIL8-case. The case can be drawn as a rectangle, and eight Pins are necessary for the connections.

Additional two text elements are used to display the TYPE ('NE555') and the IDENTIFIER ('IC1'). You already learned how to add a text object to the board. Of course you can add a fixed text 'NE555' to your component, but it is more effective to have the text object filled from the component's data fields, so that text is editable when the component is added to the board. For that reason the variables <ComponentValueType> and <ComponentIdentifier> should better be used in this case. The type and identifier will be filled-in automatically later.

So far we have drawn a rectangle, 8 pins and 2 variable text objects. Now they have to be combined to one component. This is done by selecting the objects, the component consists of, and then calling COMPONENT->BUILD COMPONENT from the main menu. Now the component data for the component is being requested.

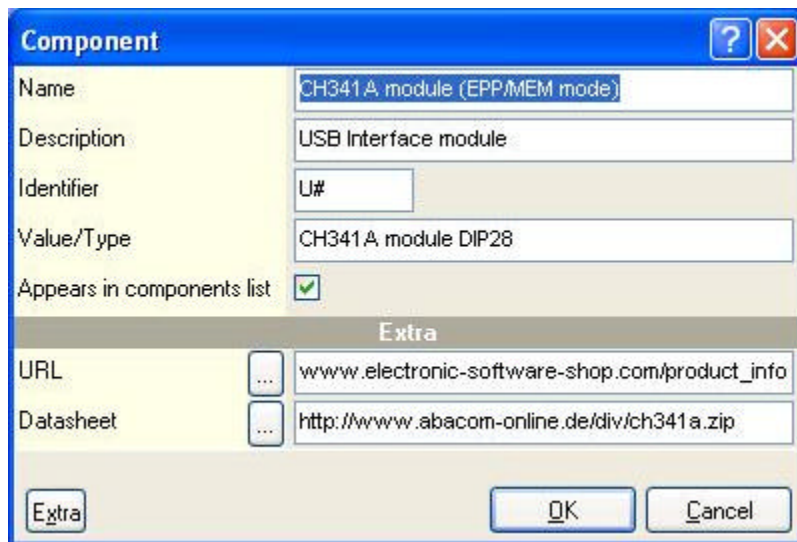
Enter 'IC#' to the field IDENTIFIER. The #-character stands for the component number, and will be replaced automatically when the component is added to the board. Enter 'timer' to the field DESCRIPTION. The field TYPE/VALUE of course you fill with 'NE555'. If you choose the option ADD TO COMPONENTS LIST the component will appear in the components list (default). Disable this option when

building symbols (like arrows) that you do not wish to appear in the components list. If you close the component data dialog with OK, the component is being built from its elements and component data is filled to variable text objects. Instead of COMPONENT->BUILT COMPONENT you could have selected ADD TO LIBRARY as well.

How to modify components

Sometimes you may wish to modify a component. In that case place the component to an empty area of the board and select COMPONENT->SPLIT COMPONENT. The component is then being split to its elements, so you can make your changes on it. When you finished your changes you have to build the component and enter its data again.

If you just want to make change to a component data, you click on it with the right mouse button and select PROPERTIES.



Component	
Name	CH341 A module (EPP/MEM mode)
Description	USB Interface module
Identifier	U#
Value/Type	CH341 A module DIP28
Appears in components list	<input checked="" type="checkbox"/>
Extra	
URL	www.electronic-software-shop.com/product_info
Datasheet	http://www.abacom-online.de/div/ch341a.zip
<input type="button" value="Extra"/> <input type="button" value="OK"/> <input type="button" value="Cancel"/>	

In some cases you will not know the components type or value in advance. In that case you enter a '?' to the data field. This indicates that data will be requested later, when the component is taken from the library.

Special components: Dipoles

Dipoles are defined as components, which have exactly two connection wires, fitting the same axis. (Like resistors, diodes, etc.). Dipoles are drawn and build as any other component, but you must not use pins, wires or splitter in the dipole. Ones again: Dipoles must have two connection wires on one axis.

If you use bitmap fillings as part of the dipole definition, please consider that bitmaps are NOT rotated with the dipole when the dipole's angle changes. It is recommended using only bitmaps with light colour effects, or even better not to use bitmaps with dipoles.