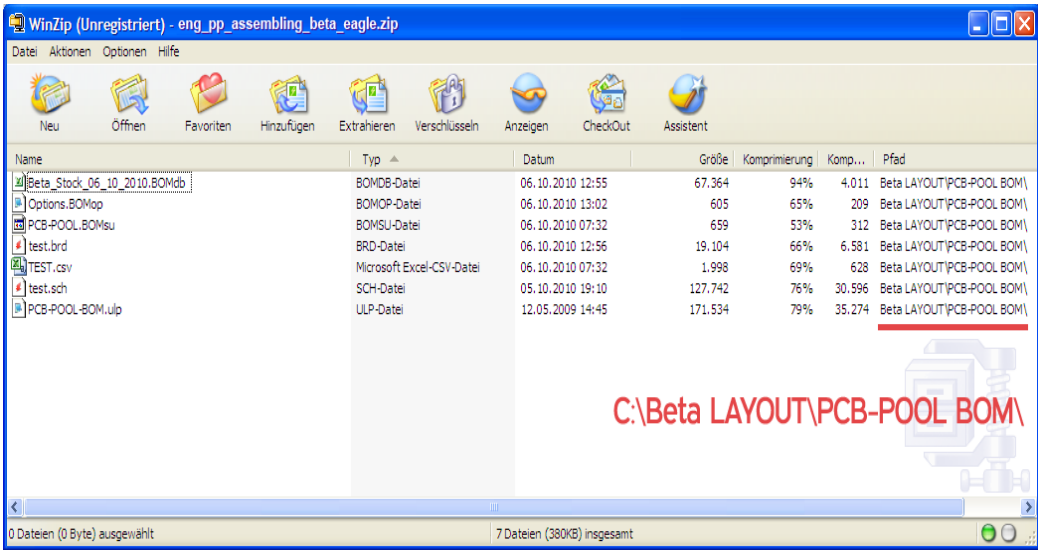
Component assembly from Beta LAYOUT

www.beta-layout.com

In order to avoid errors and time delays caused by faulty or incomplete data we need your "Bill of Materials" file in a special format. The way of creating this file is described below. Some standard components like resistors and capacitors are in our stock and are listed in the file "Beta_Stock_day_month_year.BOMdb". You can merge them with your parts list, if you wish.
Please note, that this will only work, if you used the R-EU_ resistors, C-EU and CPOL-EU parts from the original rcl.lbr library from EAGLE.
If you are using the "Assembly variants" function in Eagle, please note that we require a separate order for each variation.

1 Download the file "eng_pp_assembling_beta_eagle.zip".

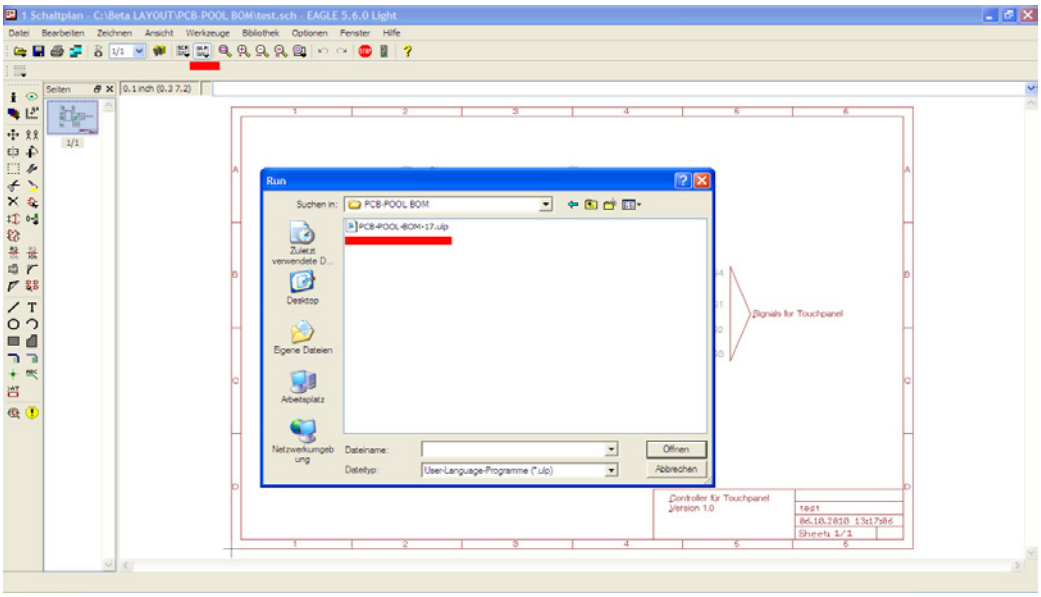
Extract the files to:
C:\Beta LAYOUT\PCB-POOL BOM\



2 Start EAGLE and open the file "test.sch". Make sure, you have your *.brd file in the same folder.

Run the ULP
"C:\Beta LAYOUT\PCB-POOL BOM\PCB-POOL-BOM-17.ulp"

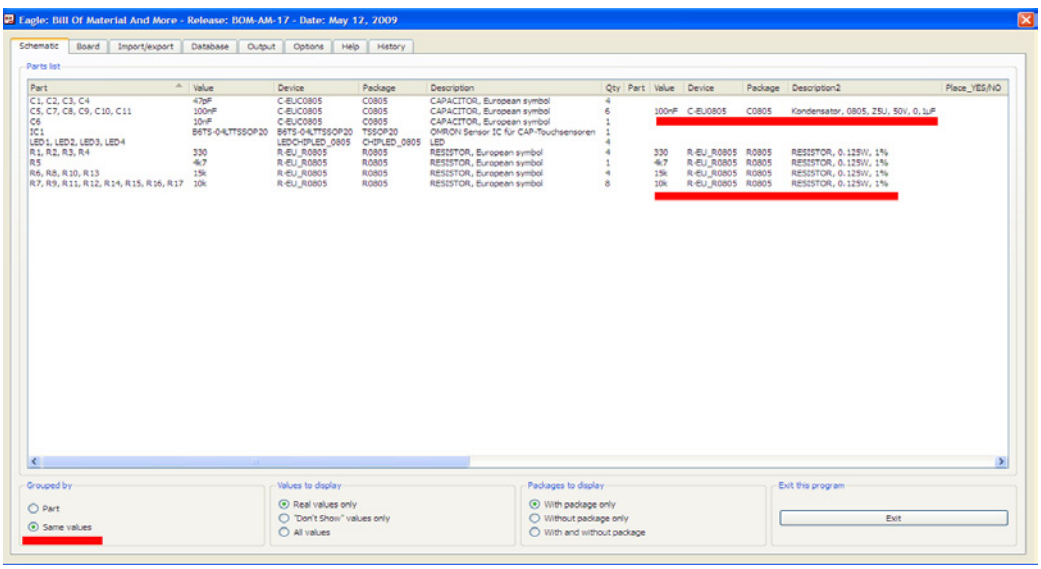
This ULP is initially based on a BOM.ulp supplied with EAGLE in 2004 and was improved over the years by Robert A. Rioja. The original version can be downloaded from the CadSoft website.



3 Your Schematic tab should display the picture on the right.

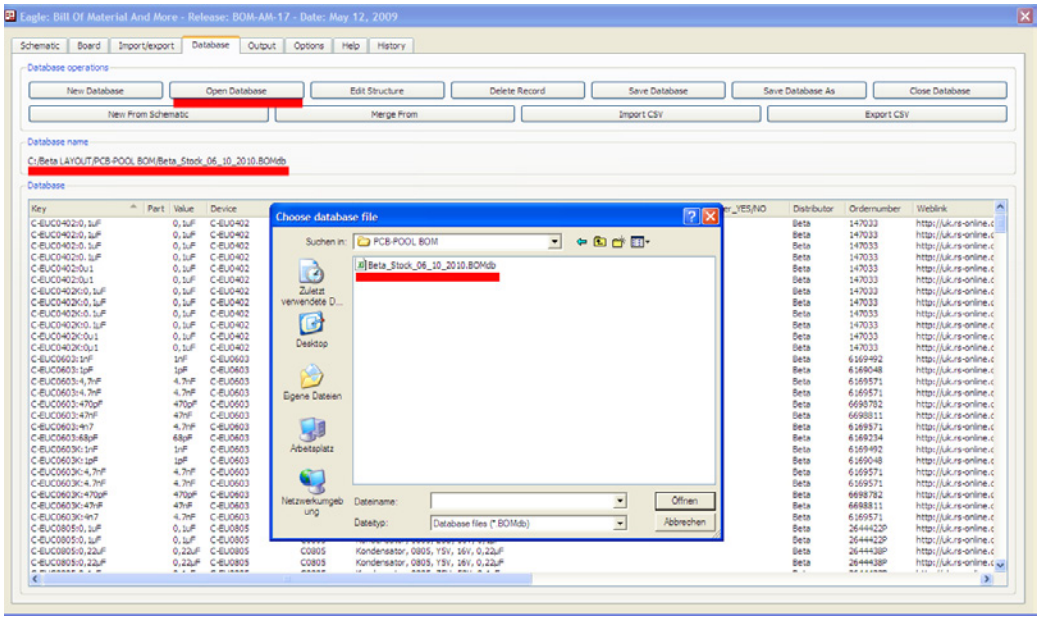
Make sure, that in the lower left corner the "Grouped by" option is set to "Same values".

All parts that were found in the file "Beta_Stock_day_month_year.BOMdb" are automatically merged.



4 Check that the Database tab shows the content of the actual Beta_Stock database.

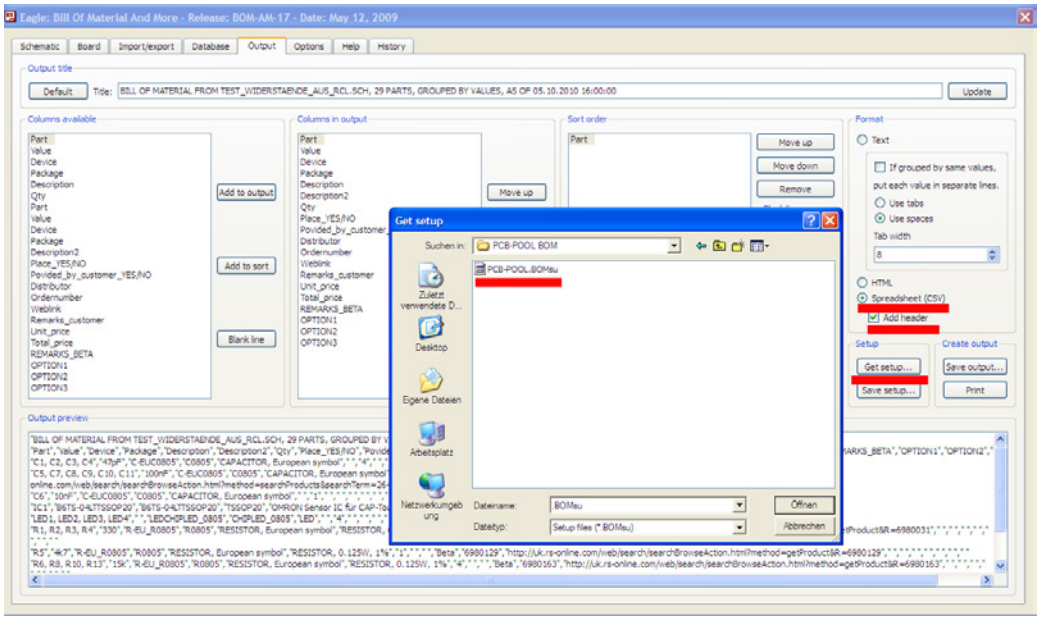
If not, you can load the file "Beta_Stock_day_month_year.BOMdb" with the button "Open Database".



5 Make sure, that in the Output tab the "PCB-POOL.BOMsu" – Setupfile is loaded.

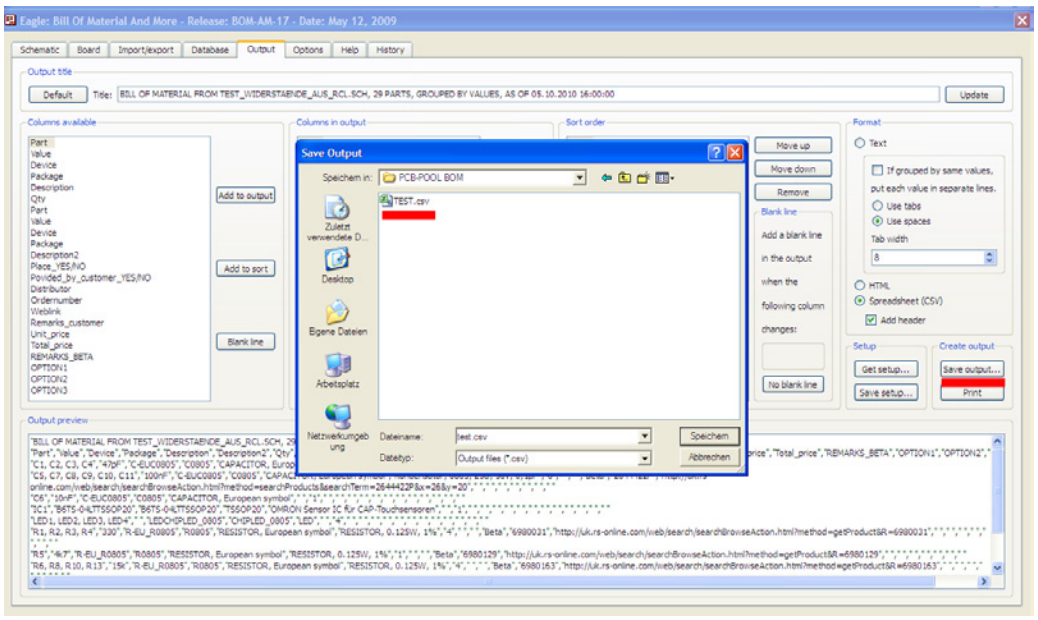
If not, please load it with the button "Get Setup" in the lower right corner.

It is important, that the "PCB-POOL.BOMsu"- Setupfile is loaded after the "Beta_Stock_day_month_year.BOMdb" database file (see 4).



6 Write your "Bill of Materials" file with the button "Save output".

Make sure that you use the Spread-sheet Format.

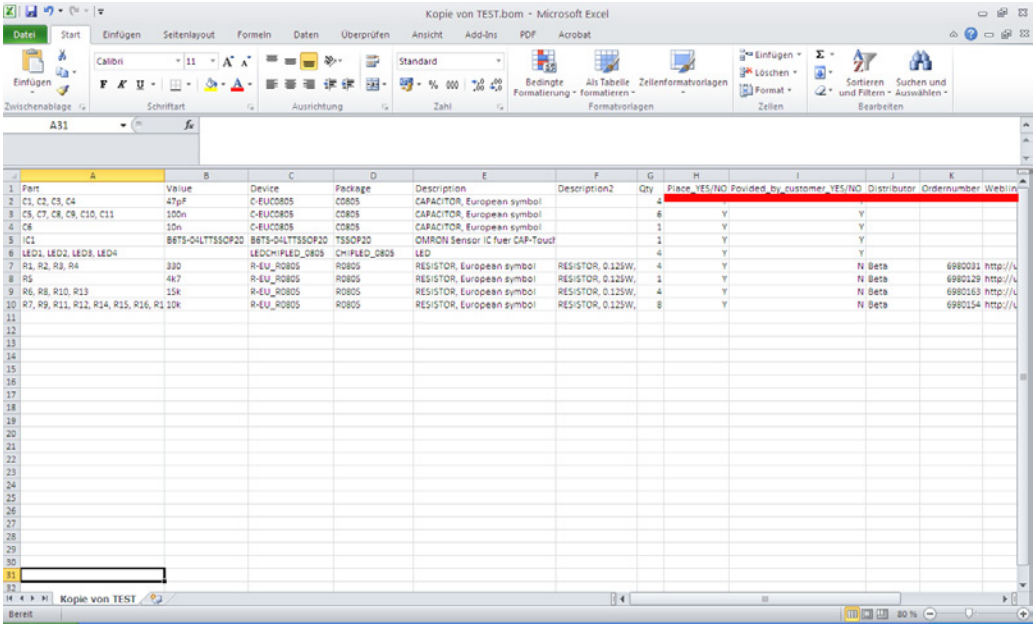


7

Import your "Bill of Material" file in Excel.

If you are experiencing problems reading in a *.csv file (nothing seems separated) try the following:

- rename the *.csv file in *.txt
- open Excel
- click on "Open File"
- choose "all files (*.*)"
- select your *.txt file
- in the following dialog choose "separated by comma (,)"



Now fill in the following fields:

Place_YES/NO	(should we assemble this component?)
Provided_by_customer_YES/NO	(are you shipping this component to us?)
Distributor	(where can we buy this component?)
Ordernumber	(what is the ordernumber of this component when we order it from the given distributor)
Weblink	(help us to find your component, especially if it is a part that is not very common)
Remarks_customer	(anything you want to comment with this component)

The rest of the fields will be filled in by us. Nothing to do here for you.

Please note: We can only accept distributors with an on-line ordering facility (shopping basket) and EURO currency.

Congratulations – you are done!

Use this file to order your assembly service from Beta LAYOUT.

Beta LAYOUT
create : electronics
GmbH
PCB-POOL®
Im Aartal 14
65326 Aarbergen

The layout diagram for your EAGLE file

www.beta-layout.com

We need your layout diagram stored as a PDF as additional information for the assembly of your circuit board.

Please note that the layout diagram is also used, among other things, as a **template for the final visual inspection!**
For this reason, the components, component names and circuit board contours should be easy to recognise in your layout diagram.
The alignment of polarised components is also labelled here (e.g. diodes, electrolytic capacitors, ICs, connectors, etc.).

Which layers do I need to superimpose in EAGLE for this?

Please fade in the following layers for the **TOP side** of the layout diagram:

LY17:	Pads
LY20:	Dimension
LY21:	tPlace
LY25:	tNames
LY48:	Document
LY51:	tDocu

Please fade in the following layers for the **BOTTOM side** of the layout diagram:

LY17:	Pads
LY20:	Dimension
LY22:	bPlace
LY26:	bNames
LY48:	Document
LY52:	bDocu

How do I print the layout diagram as a PDF?

Select the menu item "File" > "print" in your EAGLE layout editor.

Please select a pre-installed PDF writer as the printer and adapt the scale to the DIN A4 format (please ensure that the DIN A4 page is well filled).
Please set the sheet limit in the PDF to "1" when printing.

Please ensure that the **bottom side** is issued as a **mirror image** so that the labels are legible.
A check mark must be placed next to "mirrored" for this in the left area under Options.

Beta LAYOUT
create : electronics

GmbH
PCB-POOL®
Im Aartal 14
65326 Aarbergen