Component Assembly from Beta LAYOUT
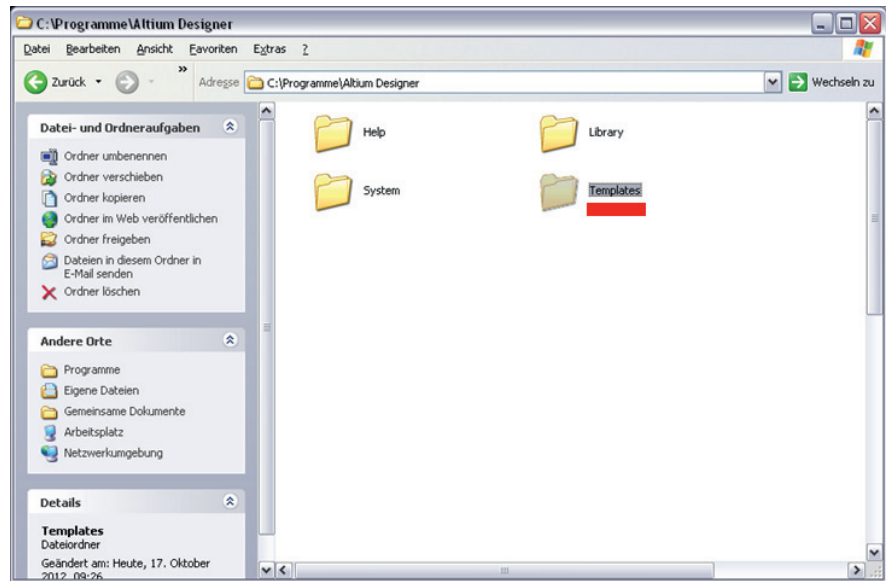
www.beta-layout.com

In order for you to be able to have your printed circuit boards assembled at Beta LAYOUT, we need a bill of materials (BOM) according to our specifications as well as the assembly diagram. The illustrated step-by-step instruction below shows you how to integrate this BOM into your Altium Designer.

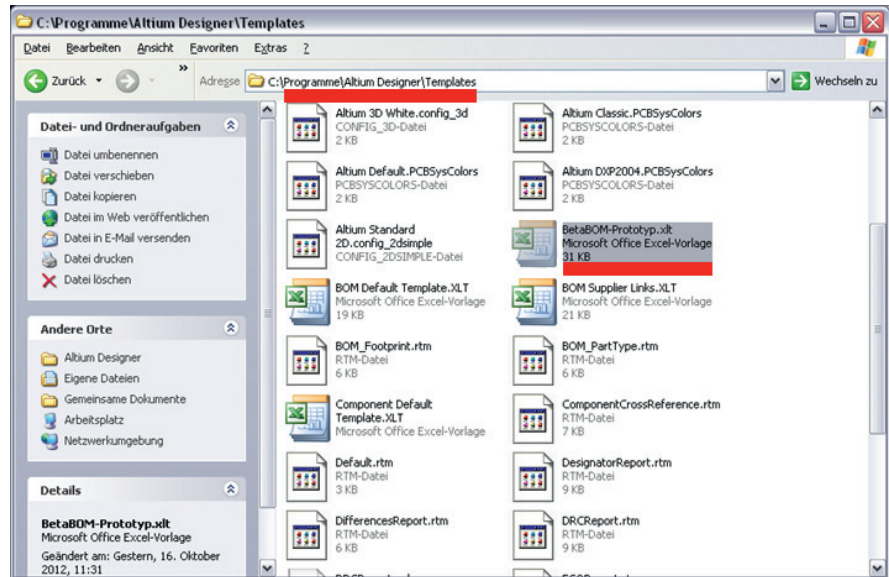**1** Locate the Altium Designer root directory on your hard disk. Here you will find the „Templates" subfolder.

2 In this folder, now save the "assembling_beta_altium.xlt" file, which you have already downloaded.
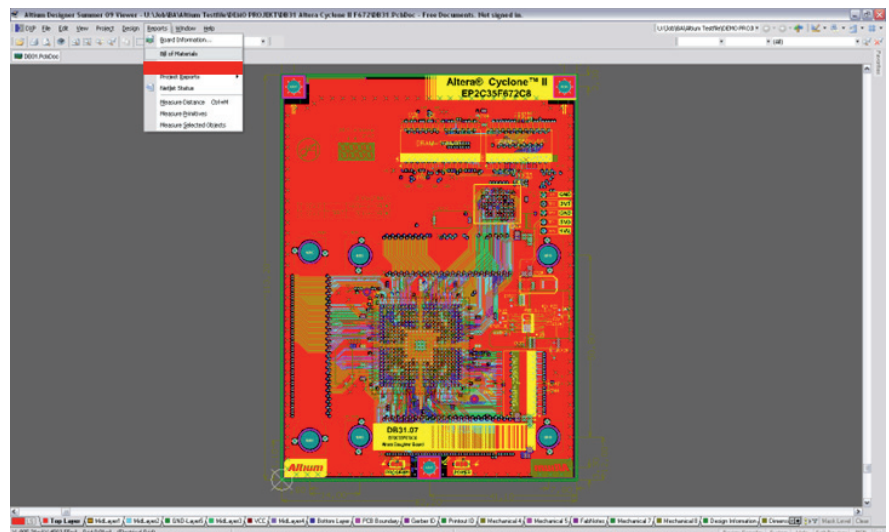
Note that the target path/storage address indicated in the example may differ from that on your computer!

3 Next, open your layout using Altium Designer and, in the menu bar there, select „Bill of Materials" under „Reports".

A new window opens, with the required information about the components that exist in your project.
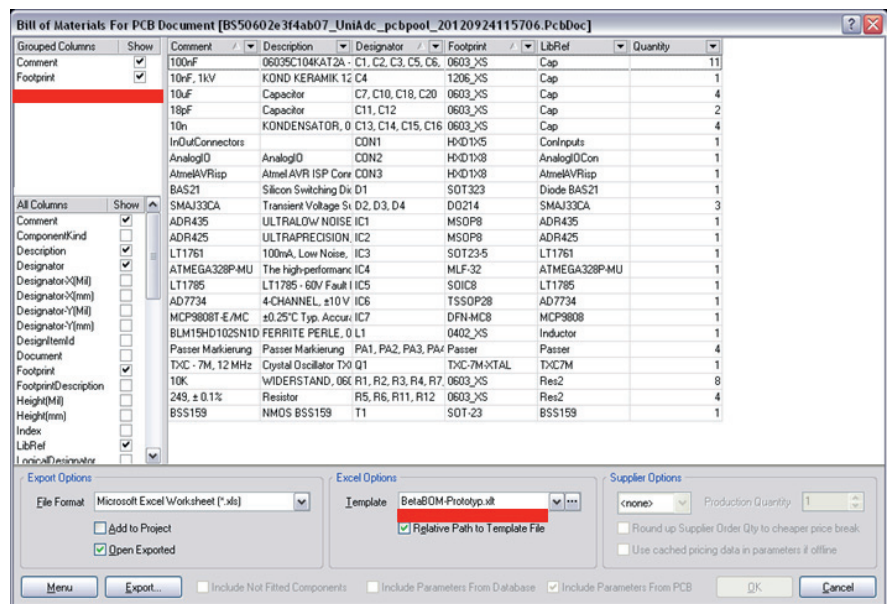
4 In order to export this data to the Beta BOM template in the next step, tick „Comment" and „Footprint" under „Grouped Columns" in the top-left section of the window.

The bottom section contains „Excel Options" with the „Templates" selection box.
Now, select the „BetaBOM-Prototyp.xlt" file, which you saved previously in the subfolder.

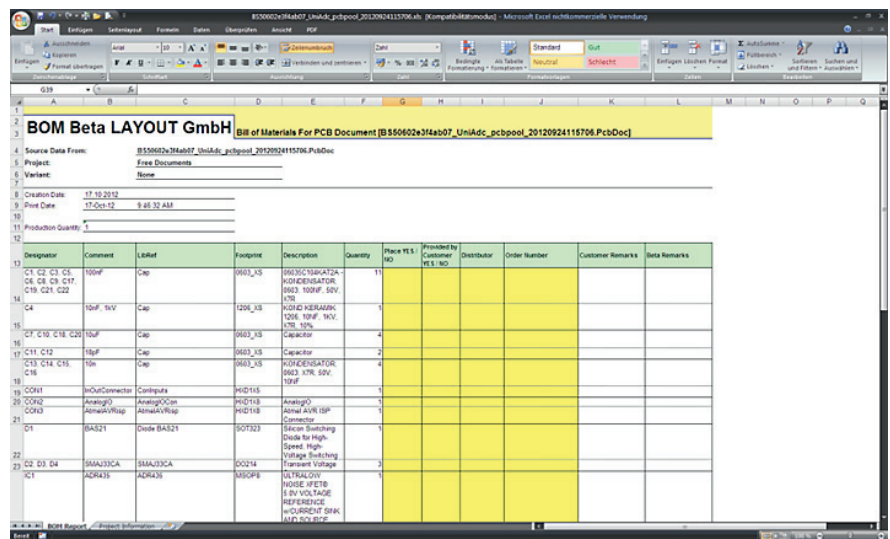Next, press the „Export" button and specify the target directory in which to save the BOM.

5 In this directory, you should find a BOM in Excel format, which matches our specifications.

As a final step, please enter any missing information (cells marked in yellow).

If special information about components is required, you can enter that in the „Customer Remarks" column.

Explanation of the fields to be filled in:

Place YES/NO	(Do you want us to assemble this component for you?)
Provided by Customer YES/NO	(Are you providing this component?)
Distributor	(Where can we buy this component?)
Ordernumber	(What order number does the distributor handle the component under?)
Customer Remarks	(Any comments that you may have about this component)

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

Congratulations – you have done it!
Send this file to us for your assembly service at Beta LAYOUT.

Beta
LAYOUT
create : electronics

GmbH
PCB-POOL®
Im Aartal 14
65326 Aarbergen

The layout diagram for your ALTIUM 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 ALTIUM for this?

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

Top Overlay (should be the first position)
Top Layer
Multi-Layer
Mechanical 1
Mechanical 4
Mechanical 13
Mechanical 15

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

Bottom Overlay (should be the first position)
Bottom Layer
Multi-Layer
Mechanical 1
Mechanical 4
Mechanical 13
Mechanical 15

How do I print the layout diagram as a PDF?

You go to „File" in the menu bar where you select the „Assembly Outputs" sub-item and then „Assembly Drawings".
A window appears with the top and bottom views.

Please select „Configuration" here with the right mouse button and click on „Mirror" for the bottom view.

Then select your PDF writer and print the diagram as a PDF.

Beta
LAYOUT
create : electronics

GmbH
PCB-POOL®
Im Aartal 14
65326 Aarbergen