PCB construction and guide

Odd Rune S. Lykkebø
lykkebo@pvv.ntnu.no

August 23, 2010

This isn't so much a tutorial as a leaflet of various notes, hints and useful stuff to know while designing with Altium and creating a PCB. Think of it as a guide to get up-to-speed on PCB design. Altium is a very powerful PCB-design suite with support for many advanced features such as design simulation and FPGA-integration. The project you are about to embark upon does however not require you to set up simulation models for the components you choose– it is worth noting that in high-speed-designs (i.e. busses clocked at 1MHz or higher), *timing* is one of the primary concerns when designing a PCB, and simulation is a very nice tool to test designs before tape-out.

1 What is a PCB?

Wikipedia can probably tell you more than I can about the subject, but I will never the less give it a go. PCB is short for printed circuit board. It was invented some 70 years ago during the second world war when radio engineers realized that wiring up some 10,000 components by hand weren't much fun.

A PCB consists of 1 or more *layers* of insulating plastic with copper wires on top, known as *traces* which connect the various components. There are two kinds of layers, *internal* and *external*. The external layers are the top and bottom of the PCB, and the internal layers are placed in-between these. The reason for this clear distinction is that external and internal layers have different eletric characteristics– since the external layers are exposed to air, they can handle more heat and as such be thinner than internal layers.

In addition to the layes containing traces, there is also a layer called *silk screen*, that is placed at the very top of the design. This layer should contain all the various markings, component names, name of the group, name of the board, helper lines, etc. The *solder mask* layer is a special layer that goes on top of the bottom and top-layers. This is a layer with a special solder-fobic coating applied between the *pads*. The pads are where you place and solder suface-mount components on the board, one pad per component leg. Having this coating between the pads makes soldering a lot more enjoyable.

Another visible feature of PCBs are the holes. These can be of many sizes and have various properties. Some holes have plates, which makes them conductive. We call these *vias*, and they are used to connect the traces of one layer to another. They can run all the way through (which makes them through-holes), or they can be drilled partially into the layer-stack. This last feature is mostly associated with more advanced designs where one would want to run traces directly below the position of the hole.

All the features of a PCB, i.e. number of layers, trace width, clearance, sizes and number of holes determine the price[1]. You whould strive to reduce the number of different hole-sizes, as each time you use different sized holes the manufacturer needs to change tool while drilling, and this is costly. Trace widths and clearances also rise the price of the PCB, we will look a bit more into these features later when explaining design rules.

2 General PCB design

When designing PCBs there are, at the topmost level, two major tasks. The first is to select the components and connect them at a logical level. "Logical" in this context means at a level above the actual board. Logic layout is done by placing symbolic representations of the components and connect with "wires". Think boxes-and-arrows from software architecture. When the logical design is complete *and verified*, you can move to the next step. *Verification is not something you can skip.* A large part of any engineering effort needs to be put into verification, and so should yours.

Having verified the logical design, one moves to the actual drawing stage. This is done using CAD-tools based on vector graphics. The drawings sent to PCB manufactureres are fairly simple. Each layer of a PCB gets one drawing that shows where copper should be applied, or some other special process such as paint for the silk-screen. [2] Gerber files are a common format for these drawings, and this is what we will use in this project.

Again, I cannot stress the importance of verifying the logic design, and afterwards the finished PCB. Each error caught in verification will save you days worth of chasing bugs and hacking to get around the error.

[1] What, you thought PCBs were free?

[2] Once upon a time, when copper was dirt cheap, a subtractive process was used, where one took a sheet of copper-covered plastic and etched away all the parts where one did not want traces. Today, as copper prices are rising, it is important to have less wasteful processes; thuse one uses an additive process where copper is added to the non-black spots of a gerber file.

3 The Altium wiki

The first thing you should check out is the Altium Wiki, `http://wiki.altium.com/`. It has lots of guides, tutorials and tricks on PCB design. This guide will follow a format where I will point out the most crucial tutorials for you to go through, and provide additional comments on sections I found unclear or simply unexplained in the tutorials.

4 Creating your first layout

`http://wiki.altium.com/display/ADOH/Tutorial+-+Getting+Started+with+PCB+Design` provides the first tutorial you need to get started.

5 Routing

In the previous tutorial you went through a the basic functionality of Altium. `http://wiki.altium.com/display/ADOH/Interactively+Routing+a+Net` give you even more detail on routing, and you should take a look at it to get more in-depth information.

5.1 Rules

One powerful feature of Altium is the *Design Rules*, which you can find under Design, Rules. These rules will give you a headache while routing if not set up properly, and are as such worth spending some time on getting to know. In-depth information is available here: `http://wiki.altium.com/display/ADOH/Design+Rules`, but I will now enumerate some of the core rules you should set up.

1. *Electrical clearance* refers to spacing between features carrying electric current, such as traces and holes. Selecting a proper clearance can be difficult, but if you use a .254 mm clerance between traces you should be fine.

2. *Trace width* refers to the width of the traces. You can set a minimum, maximum and preferred width for various types of nets. I recommend using at least two different rules, one for power nets (with a higher minimum width) and one for signal nets, whose width can be less. I suggest using a minimum of .508 mm for power traces and .254 mm for signal traces.

3. *Mask clearance* refers to the clearance between solder the soldermasks around the pads. This need not be very large, .127 mm should suffice.

5.2 Automatic routing

Altium has a powerfull auto-router, meaning that you can simply move the components where you want them and then have Altium connect them automagically. The way you use it is to enumerate all the rules for your design as you learned in the getting-started tutorial (e.g. select trace width, pad clearance rules, etc.), This can be quite useful, but I suggest trying to route most of the traces manually, as the result tends to be a bit better and less confusing to follow if the need for debugging arises.

5.3 Multitrace routing

Pressing P while in the PCB editor will give you the placement menu. A tool called multitrace routing on this menu is very useful for routing busses, but can be a bit tricky to use. You first have to select each trace that you wish to route by shift+clicking each individual trace. Then press P and select multitrace routing.

6 First component

In the previous tutorial, the components were created for you. But what if you need to create your own components? If you are lucky, Altium already has a component for you. Most of the major manufacturers ship libraries that are compatible with Altium. You can search the Altium libraries at `http://www.altium.com/community/libraries/en/libraries_home.cfm`. If you can't find them there, you might be able to just use another component with identical footprint; a 1206 (a common package for surface mount capacitors) of one make will probably match most other makes, just make sure there aren't hidden traps like mirrored power- and ground-pins.

If you cannot find a proper replacement part, cannot find it in the libraries or on the producers webpages, you will have to make the part yourself. Altium has a nice set of tools for this, and a tutorial for using these can be found at `http://wiki.altium.com/display/ADOH/Creating+Library+Components`.

6.1 Updating footprint

If you at some point discover that you have created an errounous footprint, and you need to update it, Altium has a special tool for that called "Update Parameters From Database", select it under the Tools-menu. For more information on this tool, see `http://wiki.altium.com/display/ADOH/Keeping+Components+Up-To-Date`.