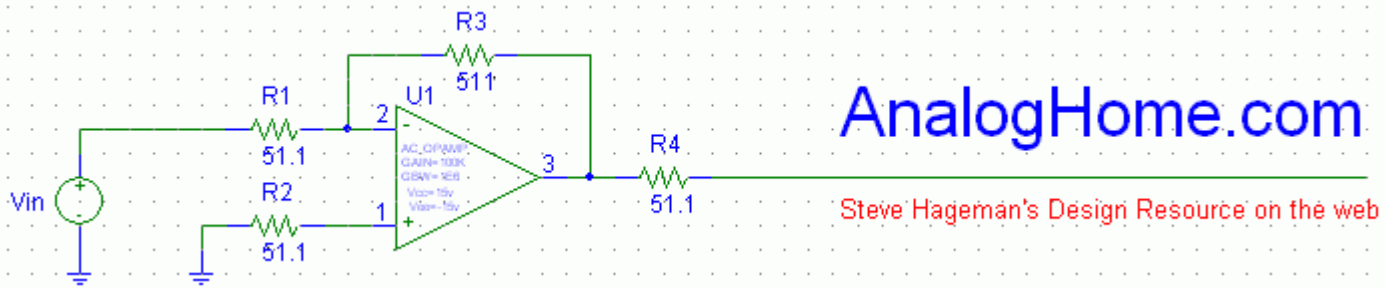# A Program for PCB Estimation with Altium Designer

A Program for PCB Estimation with Altium Designer

By: Steve Hageman
AnalogHome.com

One thing that I have had to do over and over on my new PCB jobs is to make an estimate of how long I think the layout will take, or if it is even possible. I have traditionally done this by roughing out by hand on paper or in a spreadsheet the parts area to the proposed PCB area. This provides me with a kind of utilization ratio which allows me to make a good estimate on the time it will take me to complete the design, or even if the design is possible given the PCB area.

For instance if the utilization ratio is 0.25 the parts only take up 25% of the PCB area and this is pretty easy to route, but if the number gets above 0.5 or even 0.8 these are really getting very dense and it may be difficult to route. But just this number alone is not enough to make a complete estimation. In a simple case you might be building a breakout board for a single IC and the utilization factor might be 0.95, but the design is easy to route.

The other number needed to make a more complete estimate is the number of pins that need to be routed. Everything is easy if you have only 100 pins to route, but as this number goes up so does the total time route the board.

These two numbers lead to another possible metric – the number of routing layers. As the pin count and utilization factor increase the other option to keeping the routing job reasonable or even possible is by adding routing layers.

Having the utilization factor and the number of pins to be routed starts to give the designer real data that can be used in making even better estimates in the future. This is especially true if the method is used to analyze previous designs where the time to completion is well known.

Most designers have some method of estimating like this based on gut feel or past experience or perhaps they make up a spreadsheet and make an estimate, but there is a better way.

A Program for PCB Estimation with Altium Designer

A better way

The better way is to automate the spreadsheet method, so that the results can be seen in seconds instead of manually putting together a spreadsheet in a half an hour or more. Altium Designers "Simple BOM" Report has the most important information in it. For our estimation purposes all we need are the footprint name and the quantity of each part used.

I automated this process by writing a simple standalone C# program that I call the: "Altium Estimator" [1]. This program reads the Altium Simple BOM text file and extracts the quantity for each PCB footprint as it is used on the Schematic. The program then looks for another simple CSV Spreadsheet file that contains the Area and number of pins for each pattern (or footprint) that the user provides. The end result of of this process is to provide an Estimate File, in a Spreadsheet format that contains just the quantity, pattern name, area and number of pins, plus formulas to total all these numbers up (Figure 1).

Simple input data needed

The Pattern database would seem like a hard thing to build, especially if you have libraries consisting of thousands of parts, but features in the program provide a way to built this incrementally as needed. Any given design will probably use very few different patterns. For instance I recently completed a 850 part board that used only 30 different patterns. The Altium Estimator program can determine what footprints were not present in the database file and as a help to the user these missing pattern names can be appended back to the file. Then it is a simple matter to add the area and number of pins for each of the missing patterns. In this way the pattern area spreadsheet fills up quickly and only needs to be incrementally added to.

This is much like making a new schematic from scratch: Usually 80% of the symbols are already made and only 20% need to be created each time, and things proceed quite rapidly.

Running The Altium Estimator

Using the program is quite easy, even for infrequent users. The Graphical User Interface (GUI) lists each step required to produce an output in a top down fashion (Figure 2).

First we need a BOM from the schematic to process. Altium has a command in the Reports menu called "Simple BOM" (Reports -> Simple BOM). This command produces a no frills text file that contains the essential features of a BOM. The Simple BOM should be run on the No Variant version of the schematic as you probably want it to take into account all the patterns that will be on the PCB, because whether they are "no loads" or

A Program for PCB Estimation with Altium Designer

not, you will need to route the part on the PCB.

The Next step is to point the program to the location of the pattern area database file. This file is expected to be in a Comma Separated Values or CSV format. The CSV format is in plain ASCII and can be read into any Spreadsheet or text editor ever written. I chose this format because sometimes when editing files like this it is just as easy to use a simple text editor for editing and this format provides the ultimate flexibility in this regard. The format is also obsolescence proof. As long as their are spreadsheets there will be the CSV format.

The Altium Estimator program has the option to make the estimator file for you automatically or you can browse to a location and file name (Steps 2 and 5). Synchronizing the output to the input BOM directories places the output file in the location where the input BOM file was and changes it's name by appending a: "_Estimator.CSV" to the input BOM file name.

The final step is to hit the Run button, the Estimator then reads both input files, searches for matches, combines like items and then writes the output file including the summation formulas to the specified CSV format file (Figure 1).

If the program finds any missing patterns in the database file it will prompt if you would like to append these missing pattern names back to the database file. This is a really useful and time saving feature, as you don't have to remember the missing pattern names. All you have to do its to is open the patterns CSV file in any spreadsheet or text editor and for each missing footprint add the PCB area and number of pins used. Re-save the patterns file then run the estimator again and the message "Finished with all patterns found" will be given.

Next steps

If you use the Altium Estimator program on some previous designs that you know how long they took to complete and then compare these real world numbers with new designs you will probably find that some pretty strong correlation of data can be found between the PCB Utilization Factor and the pin count versus the time required to complete a design. Having these numbers will help in making better and more accurate PCB estimates for your clients in the future and since the process is automated it won't take long to get real data either.

[1] The free Altium Estimator program can be downloaded from: www.AnalogHome.com/articles/pcb_estimator.html

A Program for PCB Estimation with Altium Designer

	A	B	C	D	E	F
1	Bill of Material for Altimeter3.PrjPcb					
2	On 9/9/2010 at 10:20:55 AM					
3						
4	Quan	Pattern	Area (each)	Pins (each)	Area (total)	Pins (total)
5	17	603	6500	2	110500	34
6	5	805	12800	2	64000	10
7	4	7343-31	105300	2	421200	8
8	2	0603LED	8400	2	16800	4
9	1	USB_MINI	131250	4	131250	4
10	1	HEADER_SIL_2	25800	2	25800	2
11	1	PROGHEADER_DIL	57400	6	57400	6
12	1	SOT-23-OT5_N	22400	5	22400	5
13	1	SSOP-28	159750	28	159750	28
14	2	MSOP-MS8_N	31200	8	62400	16
15	1	DGK8	35000	8	35000	8
16	1	SSOP-SS20_N	106400	20	106400	20
17	2	SOT23_5	22400	5	44800	10
18	1	MS5534BP	188700	6	188700	6
19	1	XTAL_ROUND	32800	2	32800	2
20	1	RESONATOR_3_TH	30000	3	30000	3
21						
22	Total Parts =>	42				
23	Total Area =>	1509200				
24	Total Pins =>	166				
25						
26	Board Area =>	4000000				
27						
28	PCB Utilization Factor =>	0.38				
29						

Figure 1: The output of the Altium Estimator program is a spreadsheet file that contains the information needed to make a better estimate of a given PCB's difficulty. For each pattern the area and pins are multiplied by the quantity of the footprint that were present in the BOM and then these columns are totaled automatically. I work in square mils, but the program doesn't care about units and will work for any consistent units system.
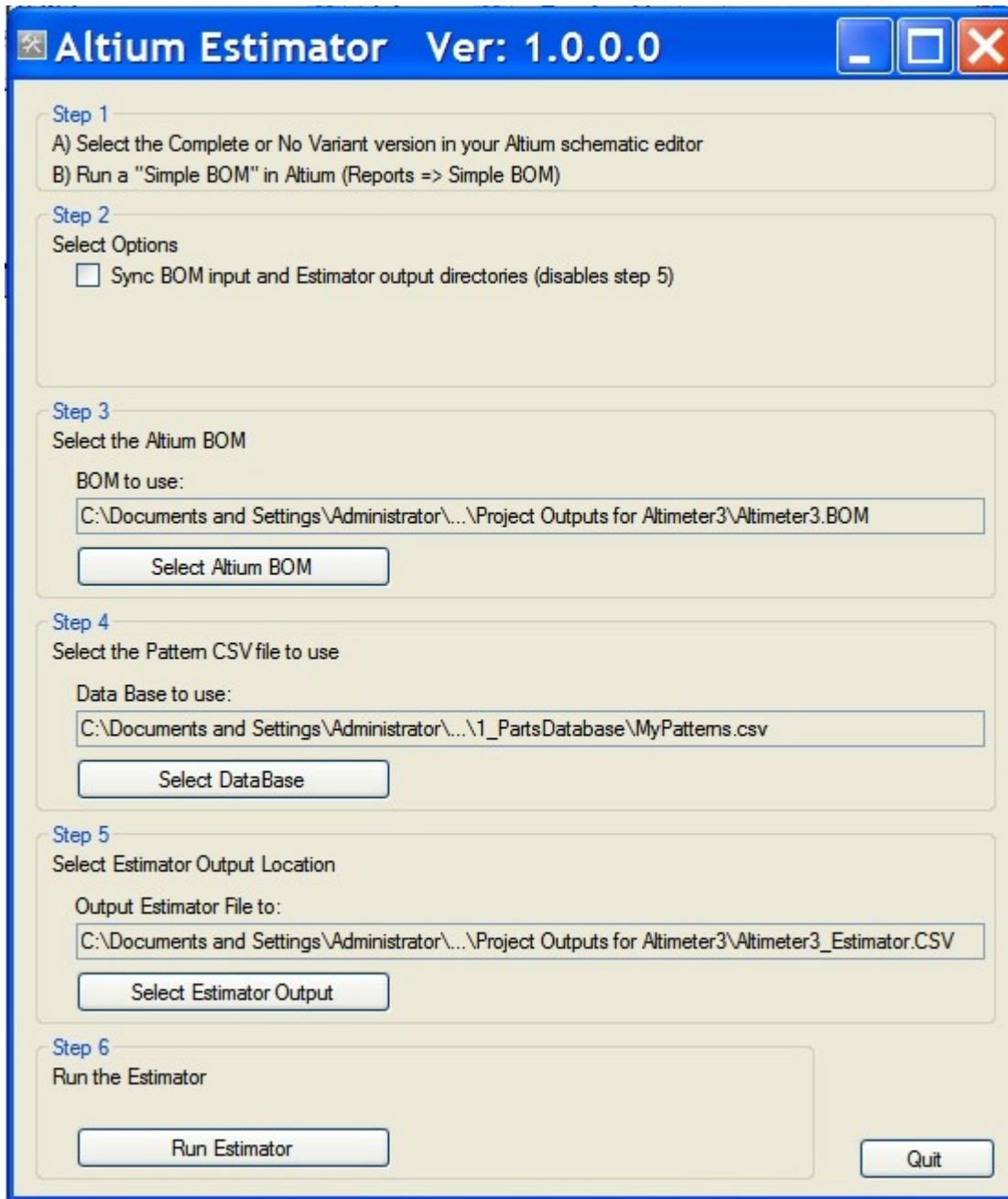
A Program for PCB Estimation with Altium Designer

Figure 2: The User Interface of the Altium Estimator leads the PCB designer to an output in a logical step by step fashion. The program kindly remembers the last location of all the files and settings so this information only needs to be incrementally changed for multiple runs and projects.

Author background: Steve Hageman designed his first CAD based PCB using AutoCAD way back in 1986 and hasn't looked back to doughnut pads and tape since. He has designed everything from switching power supplies and high accuracy analog function modules to embedded systems and dc to 18 GHz radio receivers, oh... and a little software along the way. He may be reached via his website at AnalogHome.com