The Filter Wizard
issue 10: Excel Tunes Up your Schematic Files
Kendall Castor-Perry

In previous posts I've talked about transferring spreadsheet-based circuit design "directly" to the lovely LTSpice simulator, and a reader of The Filter Wizard was keen to understand exactly what "directly" actually meant, and to get some practical information on how to do it.

The approach takes advantage of the ASCII formatting used by LTSpice for its .asc schematic files. My method is nothing more sophisticated than to import this text file into Excel, use standard spreadsheet formulae to manipulate its content, and then write it back out again. But I'm sure you'll find a worked example to be useful, so here goes!

You start with an LTSpice schematic, so here's a simple one in Figure 1. Just a single RC network fed by a voltage source, with boring starting values for the components. You don't even need to put values in for this method to work, but I recommend that you do. That way, you can analyze the circuit and make sure that it basically does what you want without falling over. This approach is definitely useful for more complicated circuits than this!
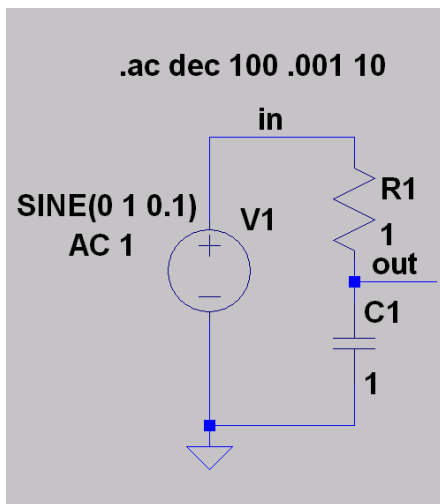

Figure 1: a simple generic circuit to start with

Draw this schematic up in LTSpice and save it as a schematic file. Then open it in a text editor – right-click the file and chose 'Open with...' as the option. On a Windows system you can use NotePad as the .asc files are pretty compact; WordPad works fine too. With my example, you'll see the following text:

```
Version 4
SHEET 1 880 680
WIRE 160 48 112 48
WIRE 224 48 160 48
WIRE 224 64 224 48
WIRE 112 112 112 48
```

```
WIRE 224 160 224 144
WIRE 256 160 224 160
WIRE 288 160 256 160
WIRE 224 176 224 160
WIRE 112 272 112 192
WIRE 224 272 224 240
WIRE 224 272 112 272
WIRE 112 288 112 272
FLAG 256 160 out
FLAG 160 48 in
FLAG 112 288 0
SYMBOL res 208 48 R0
SYMATTR InstName R1
SYMATTR Value 1
SYMBOL cap 208 176 R0
SYMATTR InstName C1
SYMATTR Value 1
SYMBOL voltage 112 96 R0
WINDOW 3 -149 5 Left 0
WINDOW 123 -109 36 Left 0
WINDOW 39 0 0 Left 0
SYMATTR InstName V1
SYMATTR Value SINE(0 1 0.1)
SYMATTR Value2 AC 1
TEXT 42 -8 Left 0 !.ac dec 100 .001 10
```

If you've reproduced my circuit, you may see different numbers in the statements; they are mostly the (x,y) coordinates of drawing points, and aren't important here.

Now, select it all and copy it to the clipboard. Then open Excel (any other spreadsheet program will almost certainly work just as well). Click in cell A1 of your chosen worksheet and paste; you'll get the lines from the .asc file pasted into contiguous cells in column A, as shown in Figure 2.
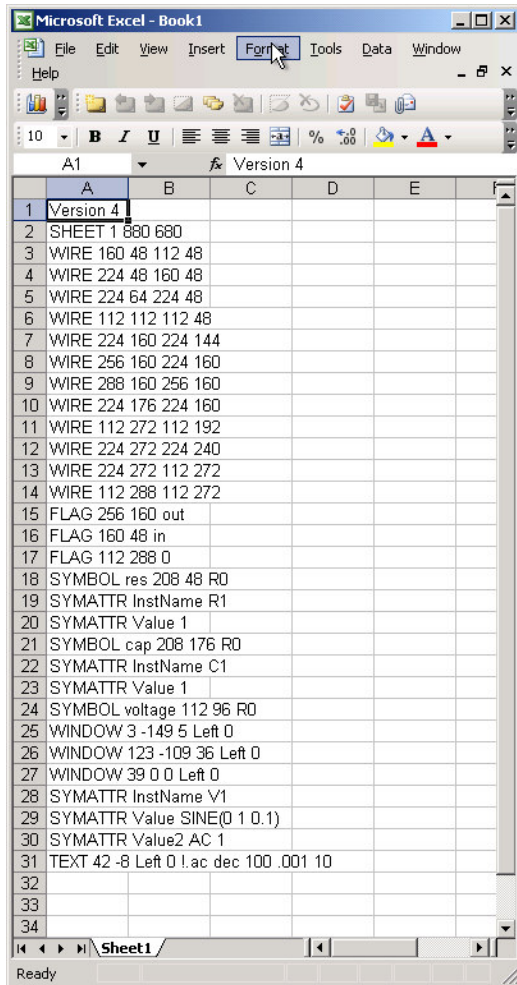
Figure 2: the .asc file imported into Excel

Now, I usually give this a worksheet of its own within the Excel file; it's typically a file that I've already been doing some filter design in. But for this example we'll just do some simple manipulations based on some calculations we can do right in the spare space on the starting sheet.

Let's change our impedance level to 1kohm and push the cutoff frequency to 2121Hz (any phono preamp fans out there? You'll recognize that number). We'll also calculate some suitable plot limits for the analysis, That's done in the central regions of figure 3, which shows how my sample worksheet is going to end up; you can probably reverse-engineer the calculations for yourself.

Now we do the substitution bit, it's easy. Some of the rows in column A are invocations of LTSpice syntax than cause a value to be associated with a component. The two lines

```
SYMATTR InstName C1
SYMATTR Value 1
```

on rows 22 and 23 of Figure 2 set up an instance of a capacitor and set its value to 1. To couple the value invocation into the spreadsheet, we change this into a formula. We use the string concatenation operator & to assemble a cell that contains the original syntax as a string, and the desired value as a numeric picked up from the spreadsheet
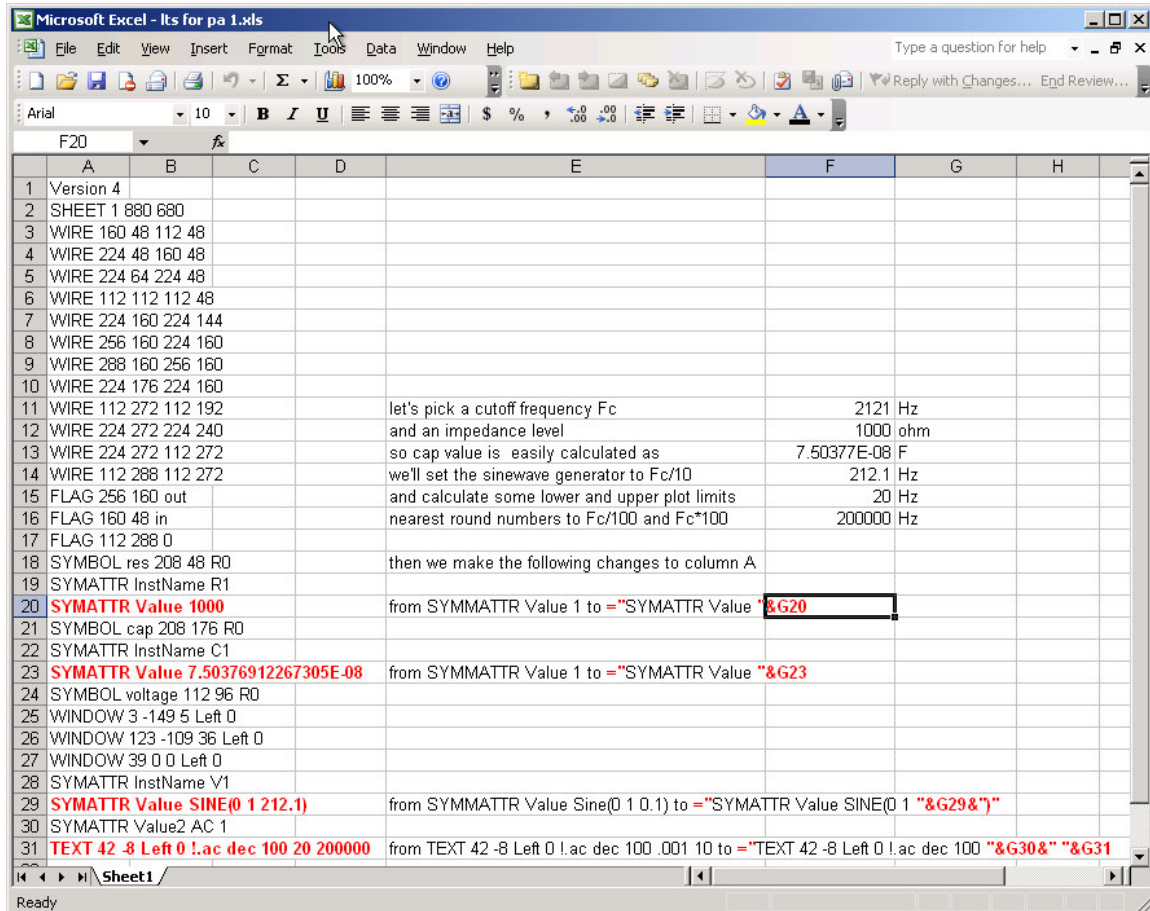
	A	B	C	D	E	F	G	H
1	Version 4							
2	SHEET 1 880 680							
3	WIRE 160 48 112 48							
4	WIRE 224 48 160 48							
5	WIRE 224 64 224 48							
6	WIRE 112 112 112 48							
7	WIRE 224 160 224 144							
8	WIRE 256 160 224 160							
9	WIRE 288 160 256 160							
10	WIRE 224 176 224 160							
11	WIRE 112 272 112 192			let's pick a cutoff frequency Fc	2121	Hz		
12	WIRE 224 272 224 240			and an impedance level	1000	ohm		
13	WIRE 224 272 112 272			so cap value is easily calculated as	7.50377E-08	F		
14	WIRE 112 288 112 272			we'll set the sinewave generator to Fc/10	212.1	Hz		
15	FLAG 256 160 out			and calculate some lower and upper plot limits	20	Hz		
16	FLAG 160 48 in			nearest round numbers to Fc/100 and Fc*100	200000	Hz		
17	FLAG 112 288 0							
18	SYMBOL res 208 48 R0			then we make the following changes to column A				
19	SYMATTR InstName R1							
20	SYMATTR Value 1000			from SYMMATTR Value 1 to ="SYMATTR Value "&G20				
21	SYMBOL cap 208 176 R0							
22	SYMATTR InstName C1							
23	SYMATTR Value 7.50376912267305E-08			from SYMMATTR Value 1 to ="SYMATTR Value "&G23				
24	SYMBOL voltage 112 96 R0							
25	WINDOW 3 -149 5 Left 0							
26	WINDOW 123 -109 36 Left 0							
27	WINDOW 39 0 0 Left 0							
28	SYMATTR InstName V1							
29	SYMATTR Value SINE(0 1 212.1)			from SYMMATTR Value Sine(0 1 0.1) to ="SYMATTR Value SINE(0 1 "&G29&")"				
30	SYMATTR Value2 AC 1							
31	TEXT 42 -8 Left 0 !.ac dec 100 20 200000			from TEXT 42 -8 Left 0 !.ac dec 100 .001 10 to ="TEXT 42 -8 Left 0 !.ac dec 100 "&G30&" "&G31				

Figure 3: change cell contents from text into formulae that reference values

In this case we change

```
SYMATTR Value 1
```

in cell A23 to

```
="SYMATTR Value "&G23
```

because G23 is where the 75nF-ish capacitor value we need has been calculated. When you view the *content* of cell it says

```
SYMATTR Value 7.50376912267305E-08
```

and when we copy out column A, that's what we'll get, plus whatever changes we make in other affected cells. So in Figure 3 I've made the changes I need to, and also shown

comments in column E describing what I did. Note that you can do any formula stuff, not just referring to other cell contents, so you can put entire expressions in here.

The final step is to select column A and copy it to the clipboard in the normal way. Now, open your text editor again, and paste in from the clipboard. Here, by the way, WordPad won't do, because it has some comprehension of Microsoft clipboard formats and will give you a *table*, not a text file. I use NotePad, which works fine. This should give you a text file that reads like the content of column A in the spreadsheet. Save this as a suitably named file with an .asc extension. Now open it with LTSpice and, hey presto, you should see the circuit shown in Figure 4:
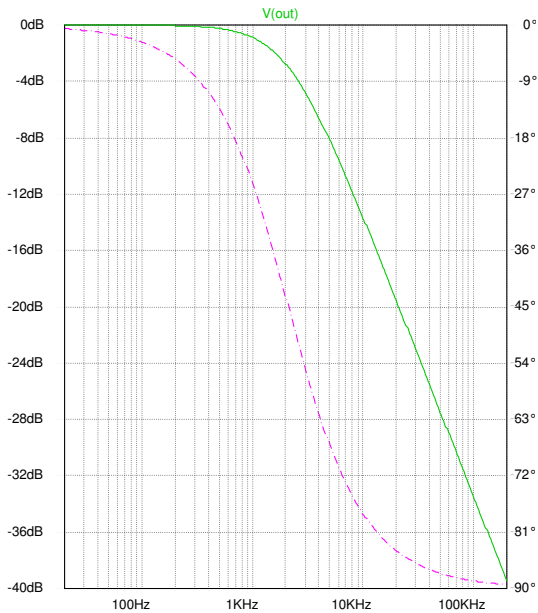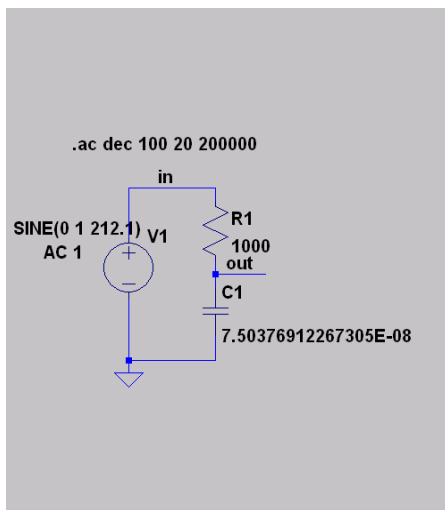
Figures 4&5: open the text file with LTSpice; plot magnitude (green) and phase (pink)

When you click the little running man (LTSpice users will know what I mean) you'll get a response plot very much like Figure 5. Well, it's unlikely to have a chain-dotted pink phase trace, but I'll get to that in the next section. Anyway, this shows a simple example of this technique, but it can be generalized to do some really good work. It's suitable for any CAD environment that uses ASCII file formats containing parameters that you want to change by calculation or lookup.

Importing .wmf plots from LTSpice into Word

I've settled on two pathways for getting LTSpice plots and schematics into Word and PowerPoint documents. For schematics I usually use the "copy bitmap to clipboard" command, paste the clipboard into my ancient copy of PaintShop Pro, and export it as a .png file. These files seem to gel well with MS Office apps, are small in size and, because they are not lossy DCT-based compressions but just run-length compacted, they have none of that blocky-heat-haze fuzziness of a .jpg, which I find so irritating on schematic imagery.

This method works OK for plots as well, but sometimes you want to get in and do extra manipulation, which isn't practical when the data has already been rendered to a grid of pixels. So for plots I use the "export as wmf" option. This creates a Windows Metafile, a combination of pixel and vector graphics.

Now, in the version of Word I use (Word 2003 running under XP), when you use "insert picture from file" on one of these .wmf exports from LTSpice, you get a rather disappointing pixelly rendition of the file. The appearance persists when the document is converted to pdf or html for publication. But here's a great trick to get more out of the process. If you've already got an example in front of you (you *have* been trying this out as you read, yes?), try this. Export the frequency response plot as a .wmf, and import it into Word. You'll probably get something that looks like Figure 6 on your screen:
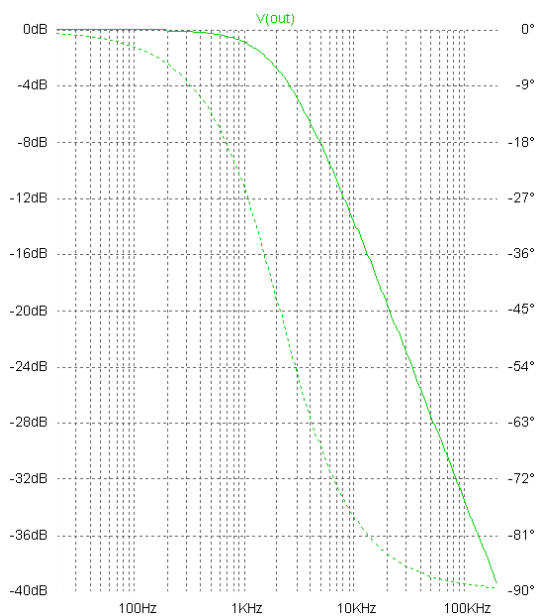
Figure 6: pasty and disappointing initial import

One disappointing thing, though, is that when you use any standard pdf writer, the finely-dotted phase trace will become a solid line, and because it's the same colour, it is hard to tell from the amplitude trace.

Here's the magic bit. Right-click the image, and select 'edit picture'. Straight away, Word exposes the underlying vector graphics for the image – after all, it's the only part of the image it knows how to edit. And now, the image is composed of individual drawing elements that can be edited and even moved around (careful!). Hence the chain-dotted pink phase trace in Figure 5. Incidentally, for that plot, I shrank the image down to the right size before invoking 'edit picture'; if you do this, Word asks you (sometimes) if you want to convert the drawing into separate objects; just answer yes and proceed as before.

So, there you are, two tips in one article for how to get the best out of LTSpice and Excel working in harmony; try it out! / Kendall