

## SRL+XV8.3\_BPF\_Simulation\_V1.1

To further document SRL+XV8.3, Tony has allowed me to post a band-pass filter simulation to the Files > Lite+Xtall v8.3 > BPF Simulation folder.

This version entirely replaces SRL+XV8.3\_BPF\_Simulation\_V1.0. Changes in this version are noted in the Release Notes below.

This is an LT-Spice simulation of the four input band-pass filters for Softrock Lite + Xtall V8.3. The source files are in the .zip, the .pdf contains a summary of the measurement results, schematic, voltage plots in dB and raw error-log .measure results. All four filters are plotted simultaneously to give you an idea of the full coverage possible with the V8.3 receiver.

I now have a SRL+XV8.3 for physical measurement, but I have not had time yet to measure the insertion loss of T100. Therefore, in V1.1 as in V1.0, I've assumed a coupling factor of 0.99 for T100. The series resistances for L100 and the T100 windings were reverse engineered using the freeware mini Ring core Calculator from DL5SWB ([www.dl5swb.de](http://www.dl5swb.de)). The filter component values were taken from:

Lite+Xtall Schematic V8.3\_sheet\_2\_5\_15\_08.pdf

In case you don't have LT-Spice/SwitcherCADIII, it is free and un-crippled from [www.linear.com](http://www.linear.com). After installation, make sure the .asc (schematic) and .plt (plot settings) files from the .zip are in the same subdirectory, then double click the .asc file to open the schematic. Run the simulation from the schematic window via Simulate > Run. The plots will automatically appear automatically provided you have the .plt plot settings file in the same subdirectory as the .asc schematic file. .measure results can be viewed from the schematic window (not the plot window) via View > SPICE Error Log. There is an excellent support group for LT-Spice at:

<http://tech.groups.yahoo.com/group/LTspice/>

Files:

SRL+XV8.3\_BPF\_Simulation\_V1.1.pdf 585kB Documented Results  
SRL+XV8.3\_BPF\_Simulation\_V1.1.zip 3kB LT-Spice Source Files  
SRL+XV8.3\_BPF\_Simulation\_V1.1\_ReadMe.pdf 18kB [This File]

Release Notes:

- 16 May 2008, V1.0, Initial Release
- 18 May 2008, V1.1, Changes:
  - o Mike Collins (KF4BQ), pointed out that V1.0 plots included the 6dB loss in the source and load resistance. V1.1 removes this discrepancy by increasing the AC excitation voltage to 2V from 1V. To accommodate the new plots, the maximum level displayed is +3dBV.
  - o The mini-Ring Core Calculator used to determine the wire lengths, and hence series resistances of L100 and T100 used in the simulation was reporting the number of turns for a given inductance that was sometimes different from the schematic turns-number by one. Specifically: The 80/40m T100 uses 18T from the schematic vs. 19T in mini-Ring, and the 30/20/17m T00 uses 14T from the schematic vs. 15T in mini-Ring. We opted to stick with the schematic values for both the number of turns and inductance in the simulation. So, we fudged the inductance in mini-Ring to get the matching number of turns and wire length. The new values are reflected in the spreadsheet data and the simulation schematic. Yes, the electrical difference is trivial, but proper wire lengths are desired in-case a builder wants to use them for the receiver kit.

- o We turned off the .raw data compression in LT-Spice with the .options plotwinsize=0 option. This allows for the most accurate plots, but the changes are seemingly trivial in this case. We leave compression off anyway.
- o The ReadMe is now in .pdf format and is included as front-matter in the roll-up .pdf file.

73's, David, WB4ONA