

Instructions

Printed Circuit Board Design LTSPICEIV & FREEPCB

V1.1

Created By:		Date:	
Revised By:	Steve Steyn	Date:	14/10/2015



LTSpiceIV

Use LTSpiceIV (schematic capture software) to draw schematic diagrams.

- Run LTSpiceIV.exe to install it on your computer or run the already installed version on **Remote Lab RL-12**
- Accept the license agreement if asked.
- If required, choose a different path to install LTSpiceIV (default path is *C:\Program Files\LTC\LTSpiceIV*)
- You can draw the basic circuit diagrams using LTSpiceIV
- You can export the netlist files in PADS(.net) format and open each one with Notepad.
- From the View menu you can display the Bill of Materials on the schematic.

LTSpice Software Use - Recording by Robert Holm:

<https://au-sas.bbcollab.com/site/external/jwsdetect/playback.jnlp?psid=2016-03-20.1622.M.599F450132D620240643923E99D77C.vcr&sid=2011430>

FreePCB

Use Free PCB (PCB layout editor) to design the PCB layout of the captured schematic diagram.

- Install FreePCB on your computer or run the already installed version on **Remote Lab RL-12 OR Remote Lab RL-2.**
- Note this software is only available for a Windows platform.
- Note the version 1.2 as supplied is adequate for the exercises, so you do not need to spend time upgrading it!
- Import the netlist (created using LTSpiceIV) for the Alarm Circuit (in PADS.net) format from LTSpice IV
- You can Design a two layer pcb for through-hole components using FreePCB.
- Follow the steps in the FreePCB user guide for further assistance.