

WEBENCH[®] Assembly Document

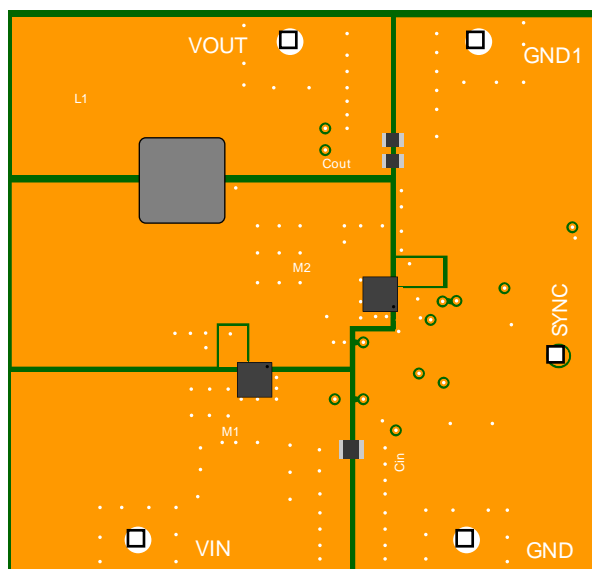
Design : 5153453/61 LM27402SQ/NOPB
LM27402SQ/NOPB 4.0V-12.0V to 2.10V @ 14.0A

General Description

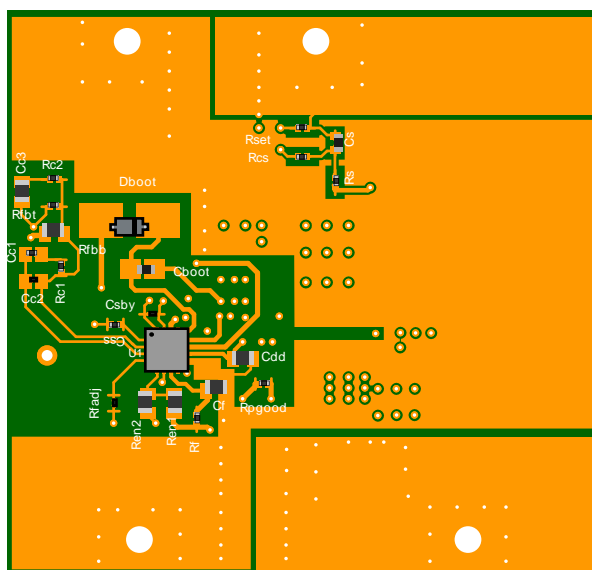
This is a WEBENCH assembly document for LM27402SQ/NOPB to support the Build-It Lite feature offered by WEBENCH Power Designer Tool or the full Build-It feature that includes an orderable blank board from our distribution partners. This document serves as a basic guidance on mounting the Build-It kit components onto a blank board and provides the basic setups for testing the fully assembled evaluation module. For more information on the IC performance and other resource, please refer Design Assistance section below.

My Comments

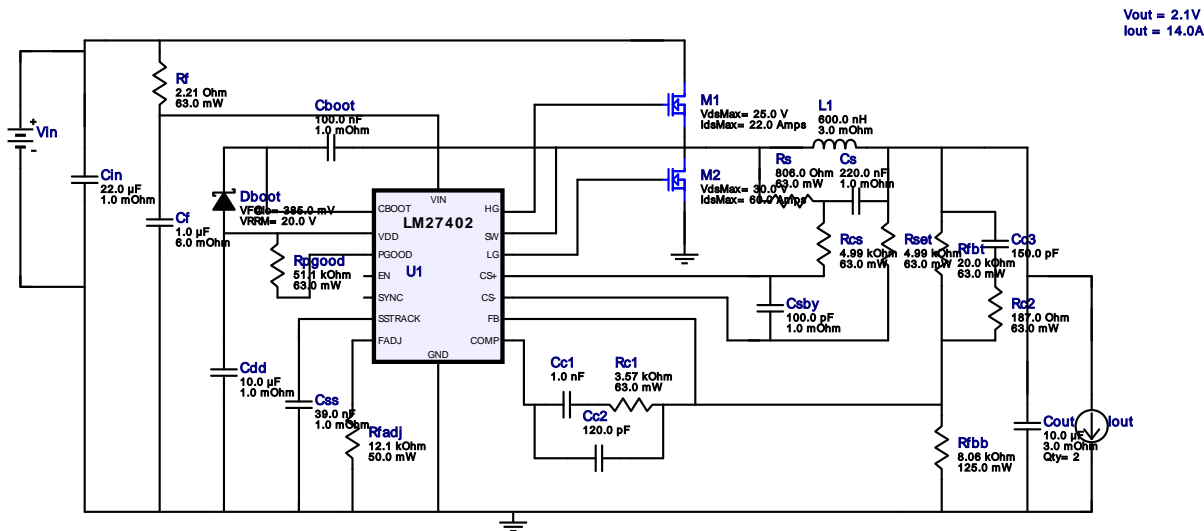
No comments



Top Layer



Bottom Layer



Mechanical BOM

#	WEBENCH Name	Altrium Comp Type	Part Name	FootPrint
No Mechanical BOM Data				

Electrical BOM

#	Component Name(s)	Part Number	Manufacture	Qty	Unit Price	Top View
---	-------------------	-------------	-------------	-----	------------	----------

Component Testing

Some published data on components in datasheets such as Capacitor ESR and Inductor DC resistance is based on conservative values that will guarantee that the components always exceed the specification. For design purposes it is usually better to work with typical values. Since this data is not always available it is a good practice to measure the Capacitance and ESR values of C_{in} and C_{out} , and the inductance and DC resistance of $L1$ before assembly of the board. Any large discrepancies in values should be electrically simulated in WEBENCH to check for instabilities and thermally simulated in WebTHERM to make sure critical temperatures are not exceeded.

Soldering Component to Board

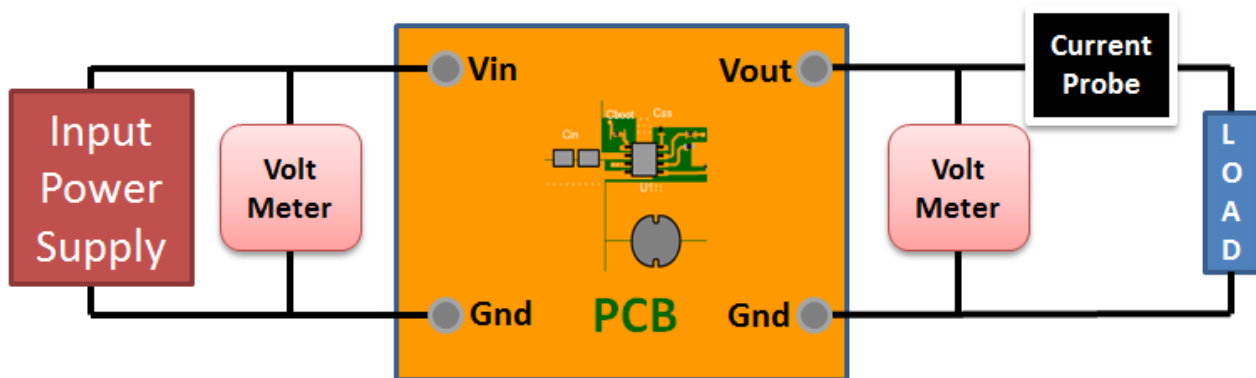
If board assembly is done in house it is best to tack down one terminal of a component on the board then solder the other terminal. For surface mount parts with large tabs, such as the DPAK, the tab on the back of the package should be pre-tinned with solder, then tacked into place by one of the pins. To solder the tab down to the board place the iron down on the board while resting against the tab, heating both surfaces simultaneously. Apply light pressure to the top of the plastic case until the solder flows around the part and the part is flush with the PCB. If the solder is not flowing around the board you may need a higher wattage iron (generally 25W to 30W is enough).

Initial Startup of Circuit

It is best to initially power up the board by setting the input supply voltage to the lowest operating input voltage 4.0V and set the input supply's current limit to zero. With the input supply off connect up the input supply to V_{in} and GND. Connect a digital volt meter and a load if needed to set the minimum load of the design from V_{out} and GND. Turn on the input supply and slowly turn up the current limit on the input supply. If the voltage starts to rise on the input supply continue increasing the input supply current limit while watching the output voltage. If the current increases on the input supply, but the voltage remains near zero, then there may be a short or a component misplaced on the board. Power down the board and visually inspect for solder bridges and recheck the diode and capacitor polarities. Once the power supply circuit is operational then more extensive testing may include full load testing, transient load and line tests to compare with simulation results.

Load Testing

The setup is the same as the initial startup, except that an additional digital voltmeter is connected between V_{in} and GND, a load is connected between V_{out} and GND and a current meter is connected in series between V_{out} and the load. The load must be able to handle at least rated output power + 50% (7.5 watts for this design). Ideally the load is supplied in the form of a variable load test unit. It can also be done in the form of suitably large power resistors. When using an oscilloscope to measure waveforms on the prototype board, the ground leads of the oscilloscope probes should be as short as possible and the area of the loop formed by the ground lead should be kept to a minimum. This will help reduce ground lead inductance and eliminate EMI noise that is not actually present in the circuit.



Design Inputs

#	Name	Value	Description
1.	I _{out}	14.0 A	Maximum Output Current
2.	V _{in} Max	12.0 V	Maximum input voltage
3.	V _{in} Min	4.0 V	Minimum input voltage
4.	V _{out}	2.1 V	Output Voltage
5.	base_pn	LM27402	Base Product Number
6.	source	DC	Input Source Type
7.	T _a	30.0 degC	Ambient temperature

Design Assistance

1. Tip: LM27402 High Current PCB Layout Design Guidance For higher current designs, please take care in designing the PCB layout. Consider good thermal management practices and proper routing of traces. Please see the following for more guidelines. Best Layout Practices for Switching Power Supplies http://sva.ti.com/assets/en/appnotes/national_power_designer114.pdf SIMPLE SWITCHER Layout Guidelines <http://www.ti.com/lit/an/snva054c/snva054c.pdf> Thermal Design by Insight, not Hindsight <http://www.ti.com/lit/an/snva419c/snva419c.pdf>

2. General Description: The LM27402 is a synchronous voltage mode buck controller with inductor DCR current sense capability. Sensing the inductor current eliminates the need to add resistive powertrain elements which increases overall efficiency and allows for accurate continuous current limit sensing. A 0.6V +/-1% voltage reference permits high accuracy and low voltage capability at the output. An operating voltage range of 3V to 20V makes the LM27402 suitable for a large variety of input rails. The LM27402 voltage mode control loop incorporates input voltage feed-forward to maintain stability throughout the entire input voltage range. The switching frequency is adjustable from 200 kHz to 1.2 MHz allowing a flexible design space. A power good indicator provides power rail sequencing capability and output fault detection. Programmable external softstart capability limits inrush current and provides monotonic output control at startup. Other features include external tracking of other power supplies, integrated LDO bias supply, and synchronization capability.

3. General Description: The LM27402 is offered in a 16 pin eTSSOP package and a 4mm x 4mm 16 pin exposed LLP.

4. **LM27402** Product Folder : <http://www.ti.com/product/LM27402> : contains the data sheet and other resources.

Texas Instruments' WEBENCH simulation tools attempt to recreate the performance of a substantially equivalent physical implementation of the design. Simulations are created using Texas Instruments' published specifications as well as the published specifications of other device manufacturers. While Texas Instruments does update this information periodically, this information may not be current at the time the simulation is built. Texas Instruments does not warrant the accuracy or completeness of the specifications or any information contained therein. Texas Instruments does not warrant that any designs or recommended parts will meet the specifications you entered, will be suitable for your application or fit for any particular purpose, or will operate as shown in the simulation in a physical implementation. Texas Instruments does not warrant that the designs are production worthy.

You should completely validate and test your design implementation to confirm the system functionality for your application prior to production.

Use of Texas Instruments' WEBENCH simulation tools is subject to [Texas Instruments' Site Terms and Conditions of Use](#). Prototype boards based on WEBENCH created designs are provided AS IS without warranty of any kind for evaluation and testing purposes and are subject to the terms of the [Evaluation License Agreement](#).