

WEBENCH® Assembly Document

Design : 1382630/540 LM22676TJ-5.0/NOPB
LM22676TJ-5.0/NOPB 18.0V-32.0V to 12.0V @ 3.0A

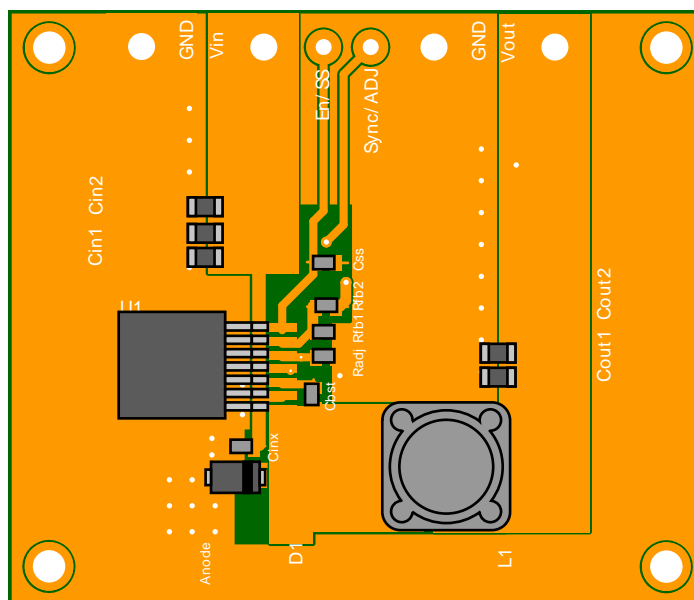


FIGURE 1 - Board Top Side Assembly Diagram

BuildIt BOM

#	Component Name(s)	Part Number	Manufacture	Qty	Unit Price	Top View
1.	L1	MSS1210-333MEB	Coilcraft	1	\$0.0	
2.	Cout	C3216X5R1C106M	TDK	2	\$0.29	
3.	D1	B350A-13-F	Diodes Inc.	1	\$0.53	
4.	Cinx	GRM21BR71H105KA12L	MuRata	1	\$0.45	
5.	Cin	GRM31CR71H475KA12L	MuRata	3	\$0.0	
6.	Rfb1	CRCW0805787RFKEA	Vishay-Dale	1	\$0.09	
7.	U1	LM22676TJ-5.0/NOPB	Texas Instruments	1	\$5.37	
8.	J1,J2,J3,J4	1502-2	Keystone	4	\$0.32	
9.	PC Board	551600233-001	Texas Instruments	1	\$10.2	
10.	Rfb2	ERJ-6ENF1001V	Panasonic	1	\$0.1	

#	Component Name(s)	Part Number	Manufacture	Qty	Unit Price	Top View
11.	Cbst	C0805C103K5RACTU	Kemet	1	\$0.1	

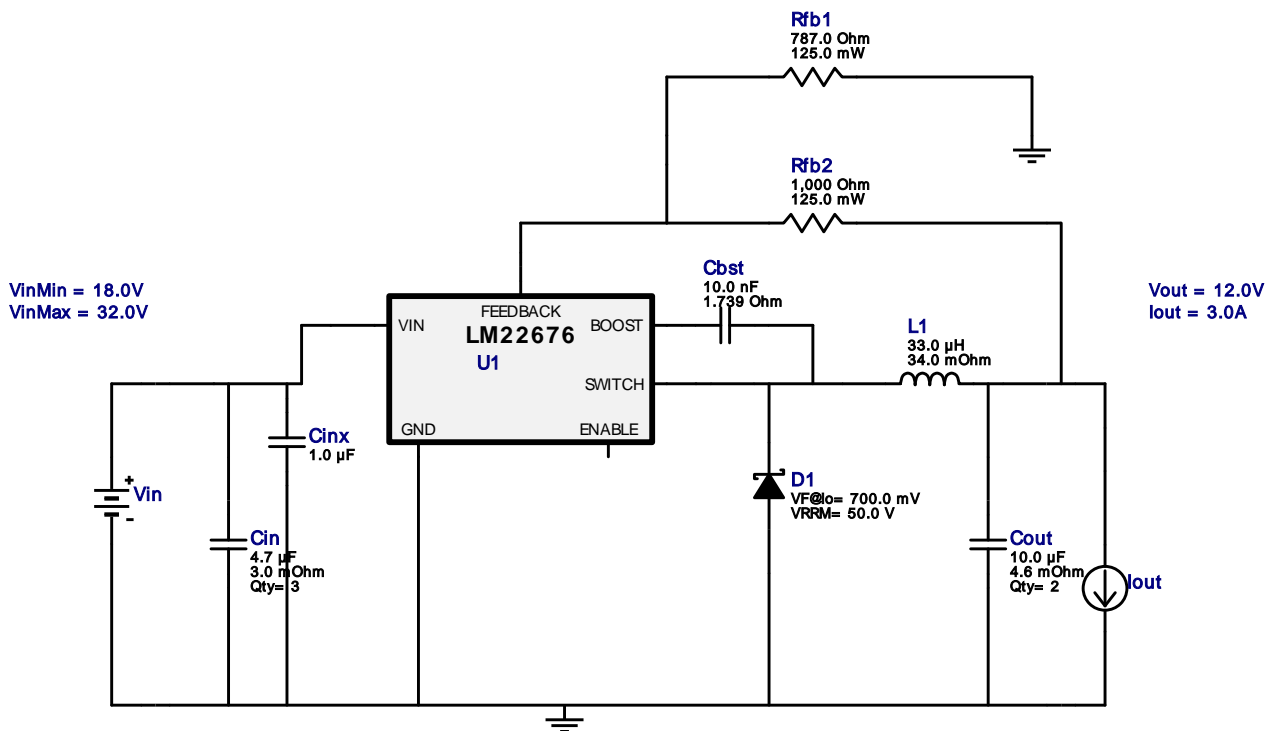


FIGURE 2 - Schematic

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 Cin and Cout, 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 (18.0 V) and set the input supply's current limit to zero. With the input supply off connect up the input supply to Vin and GND. Connect a digital volt meter to Vout 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 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 test setup for load testing is shown in FIGURE 3. The setup is the same as the initial startup, except that an additional digital voltmeter is connected between Vin and GND, a load is connected between Vout and GND and a current meter is connected in series between Vout 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. The four basic test conditions are listed in figure 4. In all cases, Vout should be well controlled within the targeted voltage range of 10.8 Volts to 13.2 Volts: 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. The following figure depicts a method of wrapping voltage probes with uninsulated wire to achieve accurate measurements:

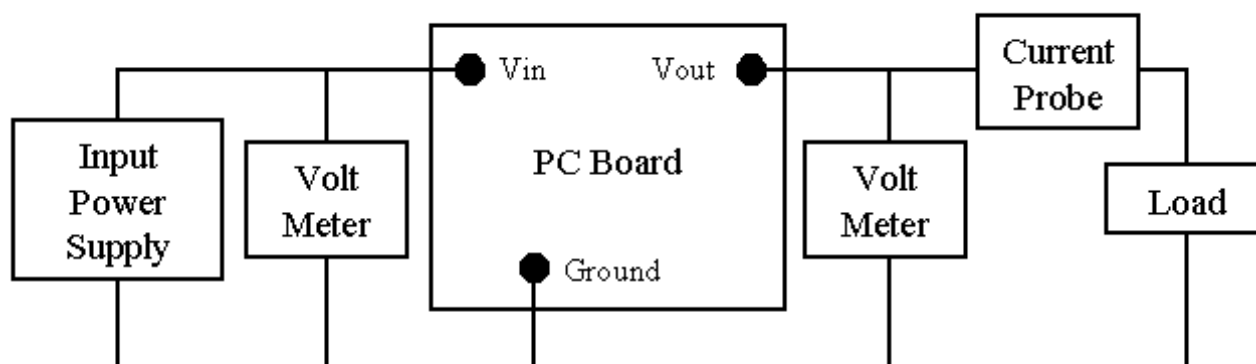


FIGURE 3 - Test setup for load testing

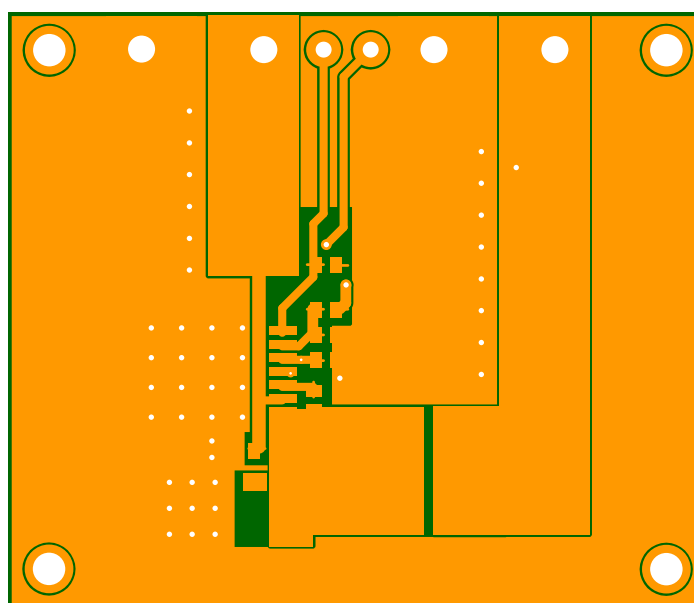


FIGURE 5 - Top Side Copper

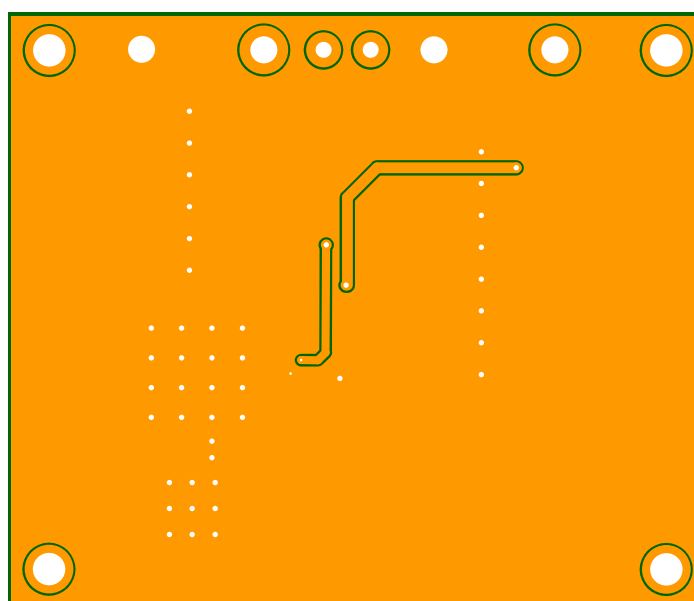


FIGURE 6 - Bottom Side Copper

Design Inputs

#	Name	Value	Description
1.	Iout	3.0 A	Maximum Output Current
2.	Iout1	3.0 Amps	Output Current #1
3.	VinMax	32.0 V	Maximum input voltage
4.	VinMin	18.0 V	Minimum input voltage
5.	Vout	12.0 V	Output Voltage
6.	Vout1	12.0 Volt	Output Voltage #1
7.	base_pn	LM22676	Base Product Number
8.	source	DC	Input Source Type
9.	Ta	30.0 degC	Ambient temperature

Design Assistance

1. Why WEBENCH recommends the 5.0 option for your 12.0V output: The internal compensation for the ADJ version of the LM22676 is optimized for output voltages below 5V. Therefore it is recommended that for outputs greater than 5V, the 5.0 option be used with an additional external resistive feedback divider. Part Description The LM22676 is a monolithic integrated circuit that provides all of the active functions for a step-down (buck) switching regulator capable of driving up to 3.0A loads with excellent line and load regulation characteristics. High efficiency (>90%) is obtained through the use of a low ON-resistance N-channel MOSFET.

2. **LM22676** Product Folder : <http://www.ti.com/product/lm22676> : 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](#).