TPS65023B/TPS650231EVM

The characteristics, operation, and use of the TPS65023B/TPS650231EVM-664 evaluation module (EVM) are described in this document. This EVM is designed to help the user evaluate and test the various operating modes of the TPS65023B/TPS650231. This user's guide includes setup instructions for the hardware and software, a schematic diagram, a bill of materials, and PCB layout drawings for the evaluation module.

Contents

List of Figures

List of Tables

1 Introduction

The Texas Instruments TPS65023B/TPS650231EVM is an integrated power management integrated circuit (IC) for applications that are powered with one Li-ion or Li-polymer cell and require multiple power rails. The TPS65023B/TPS650231 contains three highly efficient switching step-down converters, two LDOs, and additional status and I/O pins. The device is controlled via an I^2C interface (HPA172).

1.1 Requirements

In order for this EVM to operate properly, the following components must be connected and properly configured.

1.1.1 Personal Computer

A personal computer with a USB port is required to operate this EVM. The TPS65023B/TPS650231 interface software, which is run on the personal computer (PC), communicates with the EVM via the PC USB port. The user sends commands to the EVM as well as reads the contents of the TPS65023B/TPS650231 internal registers through the USB port.

1.1.2 Printed-Circuit Board Assembly

The TPS65023B/TPS650231EVM-664 PCB contains the TPS65023B/TPS650231 IC and its required external components. This board contains several jumpers and connectors that allow the user to customize the board for specific operating conditions.

1.1.3 USB-to- I^2C Adapter

The USB-to-I^2C Adapter, also known as the HPA172, is the link that allows the PC and the EVM to communicate. The adapter connects to the PC with the supplied USB cable on one side and to the EVM though the supplied ribbon cable on the other. When the user writes a command to the EVM, the interface program, which is run from the PC, sends the command to the PC USB port. The adapter receives the USB command and converts the signal to an I^2C protocol. It then sends the I^2C signal to the TPS65023B/TPS650231 board. When the user reads a status register from the EVM, the PC sends a command to read a register on the EVM. When the EVM receives the command, it reports the status of the register via the I^2C interface. The adapter receives the information on the I2C interface, converts it to a USB protocol, and sends it to the PC.

1.1.4 Software

Texas Instruments provides software to assist the user in evaluating this EVM. To install the software, go to the Texas Instruments Web page and download the latest software version from the TPS65023B or TPS650231 product folder. Once you have downloaded the software, execute the setup file, and follow the instructions given.

1.2 Features

- Three high-efficiency DC/DC step-down converters (1.7 A/1.2 A/1 A)
- 2 x 200-mA low-dropout (LDO) regulators
- 30-mA RTC (Reel Time Clock) LDO
- I2C Interface (100-ns setup and hold time)
- Dynamic voltage scaling DCDC1

2 Electrical Performance Specification

Table 1. EVM Ordering Options

Orderable EVM Number	Device Part Number	Default Output Voltages				
		DCDC1	DCDC2	DCDC3	LDO1	LDO2
TPS65023BEVM-664	TPS65023B	1.2V / 1.6V	1/8V / 3.3V	1.8V / 3.3V	SeeTable 2	SeeTable 2
TPS650231EVM-664	TPS650231	1.2V / 1.6V	n/a	1.8V / 3.3V	See Table 3	See Table 3

Table 2. Default LDO Output Voltages TPS65023B

DEFLDO2	DEFLDO1	VLDO1	VLDO2
0	0	1.3 V	3.3 V
0	1	2.8 V	3.3 V
1	0	1.3 V	1.8 V
1	1	1.8 V	3.3 V

Table 3. Default LDO Output Voltages TPS650231

DEFLDO2	DEFLDO1	VLDO1	VLDO2
0	0	1.3 V	3.3 V
0	1	2.8 V	3.3 V
1	0	1.3 V	1.8 V
1	1	2.1 V	3.3 V

3 Input/Output Connector Description

This section describes the jumpers and connectors on the EVM, as well as how to properly connect, set up, and use the TPS65023B/TPS650231EVM-664.

J1 – VIN

Input voltage from external power supply, recommended maximum 5.5 V. Input current depends on load but typically is less than 2 A.

J2 – GND

This is the return connection for VIN.

J3 – VINLDO/GND

Input voltage and return for LDO1 and LDO2. Resistor R20 connects this pin to VIN. If an external power supply is used, remove R20. Recommended maximum input voltage is 5.5 V.

J4 – VSYSIN/GND

Input voltage and return for VSYSIN, one of the input voltages for RTC. Resistor R21 can be used to connect this input to VIN. If an external power supply is used, remove R21. Recommended maximum input voltage is 4 V.

J5 – VBACKUP/GND

Input voltage and return for VBACKUP, one of the input voltages for RTC. There are no onboard connections to a voltage input. Recommended maximum input voltage is 4 V.

J6 – VRTC/GND

Output voltage from RTC circuit.

J7 – Fault Outputs

Four fault outputs are available on this connector:

$\overline{\text{PWRFAIL}}$ – Fault occurs when input voltage is less than 3 V. Pulled up to VRTC when safe; low for fail.

$\overline{\text{INT}}$ – Fault occurs when a fail is on an input or output voltage; acts as a sum fail. Pulled up to VIN when safe; low for fail.

$\overline{\text{RESPWRON}}$ – Low reset signal is controlled by SW1, 144 ms. Pulled up to VIN normally.

$\overline{\text{LOWBAT}}$ – Fault occurs when input voltage is less than 3.6 V. Pulled up to VIN when safe; low for fail.

J8 – I2C

This header duplicates the I2C signals from the J20 interface connector. I2C data (SDA) and clock (SCL) can be accessed on this header

J9 – VDCDC1

Output from DCDC1 switching regulator maximum output current 1.7 A; default voltage setting is 3.3 V.

J10 – GND

Return for VDCDC1.

J11 – VDCDC2

Output from DCDC2 switching regulator; maximum output current 1.2 A.

J12 – GND

Return for VDCDC2

J13 – VLDO1

Output from low-dropout regulator VLDO1; maximum output current 200 mA.

J14 – GND

Return for VLDO1

J15 – VLDO2

Output from low-dropout regulator VLDO2; maximum output current 200 mA.

J16 – GND

Return for VLDO2.

J17 – VDCDC3

Output from switching regulator DCDC3; maximum output current 1 A.

J18– GND

Return for VDCDC3.

J19 –

J19 is the interface connector for the I2C interface. Connect a 10-pin ribbon cable between J13 and the USB-to-GPIO interface.

JP1 – DEF1

Sets default voltage for DCDC1, 1.2 V or 1.6 V.

JP2 – DEF2

Sets default voltage for DCDC2, 3.3 V or 1.8 V, in TPS65023B (HPA664-001) configuration. In the TPS650231, no default output voltage options are available. In this case, the output voltage is adjustable with an external voltage divider R3, R2. The default setup is 1.2 V in the factory EVM (HPA664-002) configuration. See section x for advanced voltage scaling options..

JP3 – DEF3

Sets default voltage for DCDC3, 3.3 V or 1.8 V.

JP4 – DCDC1 ON/OFF

EN for DCDC1 converter; default setting is ON

JP5 – DCDC2 ON/OFF

EN for DCDC2 converter; default setting is ON.

JP6 – DCDC3 ON/OFF

EN for DCDC3 converter; default setting is ON.

JP7 – LDO ON/OFF

EN for both LDO1 and LDO2 regulators; default setting is ON.

JP8 – DEFLDO1

Sets default voltage for LDO1 and LDO2 in combination with DEFLDO2. (See Table 2 for TPS65023B and Table 3 for TPS650231 - defaults LDO output voltages.)

JP9 – DEFLDO2

Sets default voltage for LDO1 and LDO2 in combination with DEFLDO1. (See Table 2 for TPS65023B and Table 3 for TPS650231 - defaults LDO output voltages.)

S1 – $\overline{\text{HOT\_RST}}$

S1 is a normally open, momentary pushbutton switch that, when pressed, connects the $\overline{\text{HOT\_RST}}$ input of the TPS650231 to GND, generating the $\overline{\text{HOT\_RESET}}$ pulse. $\overline{\text{HOT\_RESET}}$ pin is pulled up externally.

3.1 Setup

The following steps must be followed before the EVM can be operated.

1. Install the TPS65023B/TPS650231EVM software.
2. Connect input voltages and loads to the EVM.
3. Configure all EVM jumpers to factory setting.

JP4–ON	JP1–1.6V	JP3–1.8V
JP2–1.8V	JP5–ON	JP6–ON
JP7–ON	JP9–High	JP8–High

4. Connect the ribbon cable between the EVM and the USB-TO-GPIO (HPA172) adapter.
5. Connect the USB cable between the computer and the HPA172EVM.
6. Turn on all supplies.
7. Run the TPS65023B/TPS650231EVM software

3.2 Modifications

3.2.1 Setting the Output Voltage

The TPS65023B features two default output voltages. These output voltages can be selected by pulling DEFDCDC2 high – selecting the higher default output voltage – or pulling DEFDCDC2 low – selecting the lower default output voltage.

In addition, the output voltage of DCDC2 can be externally adjusted with the resistor divider network R3 and R6. The default configuration of the TPS65023BEVM-664 is that R3 and R6 are not assembled. The default output voltage of DCDC2 can be selected with JP2.

Note that the default output voltage is selected once at startup of the device. Changing logic level of DEFDCDC2 during operation does not affect the output voltage and is not allowed.

3.2.2 Setting the Output Voltage for DCDC2, TPS650231

The TPS650231 does not feature these default output voltages. The output voltages of DCDC2 is externally adjustable only. The default configuration of the TPS650231EVM-664 is that R3 and R6 are not assembled. JP2 is not assembled.

TPS650231 does not have the default output voltage feature, and this provides the benefit of external voltage scaling options.

3.2.3 Simple Two-Point Voltage Scaling, TPS650231

DCDC2 does not have the previously described preset default output voltages. An external voltage scaling circuit is on the EVM, and the output voltage of DCDC2 can be switched between two preset voltages. This useful feature reduces the power consumption of an application processor in Low Power mode.

The voltage scaling circuit consists of JP2,Q1, R3, R6, and R24. The circuit uses a transistor (Q1) to connect a resistor (R24) in parallel to the lower resistor of the feedback network (R6) of DCDC2.

Modifying the resistor network by paralleling R24 and R6 reduces the overall resistance of the lower resistor and therefore increases the output voltage of the DC/DC converter. See Equation 1 and Equation 2 to design R24. In the factory configuration, the components JP2, Q1, and R24 are not assembled on the board.

3.2.4 Scaling the Output Voltage of DCDC2 From LDO2

Another approach to scale the DC/DC converter output voltage is to use an external adjustable voltage. Any external adjustable voltage source can be used, e.g., output voltage of an digital-to-analog converter. In the TPS65023B/TPS650231, LDO1 and LDO2 can be adjusted via I2C. The TPS65023B/TPS650231 provides the ability to feed the output voltage of LDO2 back to the resistor divider network, using R25, and therefore scale the output voltage of DCDC2 based on the LDO2 output voltage.

In this configuration, R25, R3, and R6 need to be assembled. R24, R26, Q1, and R3 need to be removed.

$$Vout\_DCDC2 = V\_REF \times (1 + \frac{R1}{R2})) + (V\_REF - VLDO2) \times (\frac{R1}{R3}) \tag{1}$$

From Equation 2 it can be seen that maximum DCDC1 output voltage occurs for minimum VLDO2, and minimum DCDC1 output voltage occurs for maximum VLDO2.

To ensure that the desired DCDC1 output voltages can be adjusted, design the resistors R25, R3, and R6 according to and Equation 2 and Equation 2.

$$Vout\_DCDC2{,}min = V\_REF \times (1+\frac{R1}{R2})) + (V\_REF - VLDO2{,}max) \times (\frac{R1}{R3}) \quad (2)$$

$$Vout\_DCDC2{,}max = V\_REF \times (1+\frac{R1}{R2})) + (V\_REF - VLDO2{,}min) \times (\frac{R1}{R3}) \quad (3)$$

The most straight forward way is to choose a value for R6 according to the recommendations in the converter data sheet.

4 Board Layout

This section provides the TPS65023B/TPS650231EVM-664 board layout and illustrations.

4.1 Layout

Board layout is critical for all switch mode power supplies. Figure 1 through Figure 5 show the board layout for the TPS65023B/TPS650231EVM-664 PCB. The nodes with high switching frequencies and currents are short and are isolated from the noise-sensitive feedback circuitry. Careful attention has been given to the routing of high-frequency current loops. See the data sheet for specific layout guidelines.

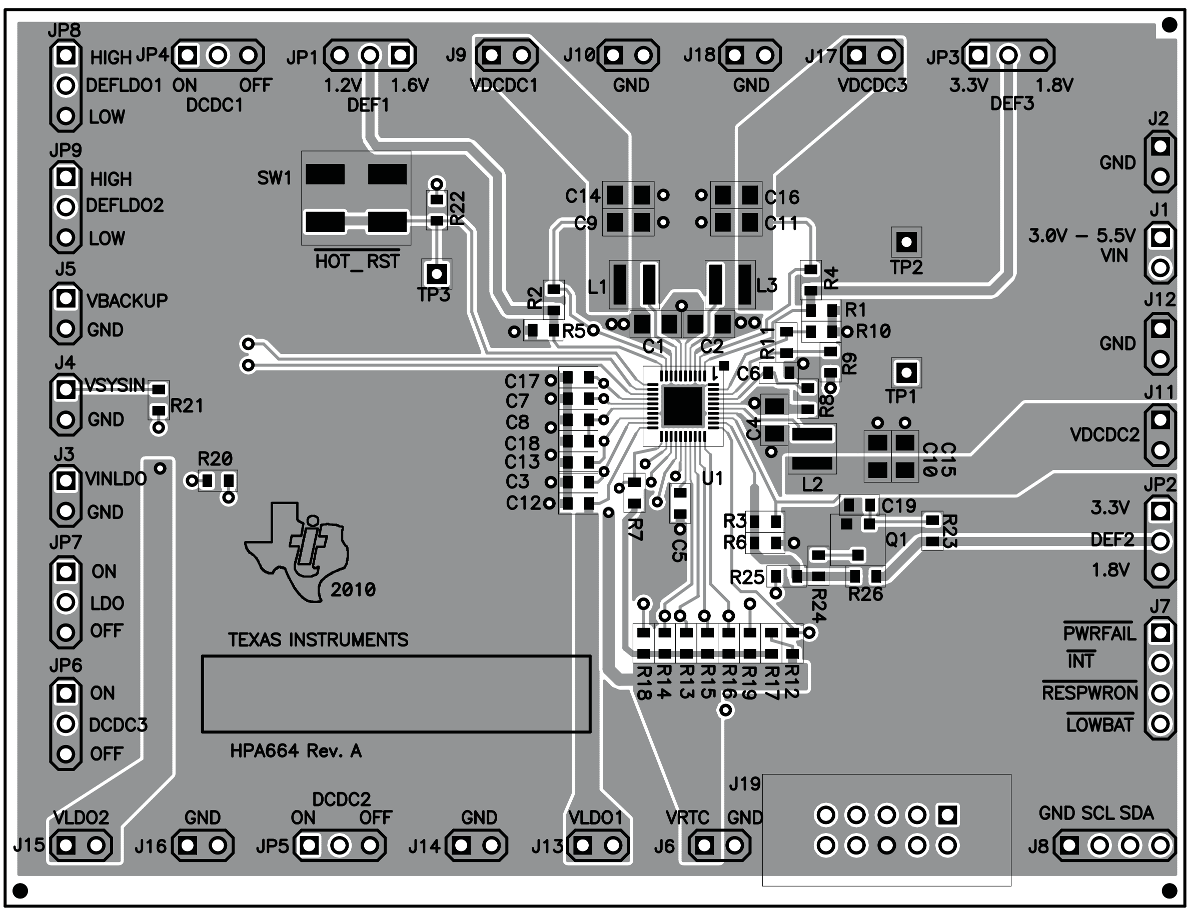

Figure 1. Assembly Layer

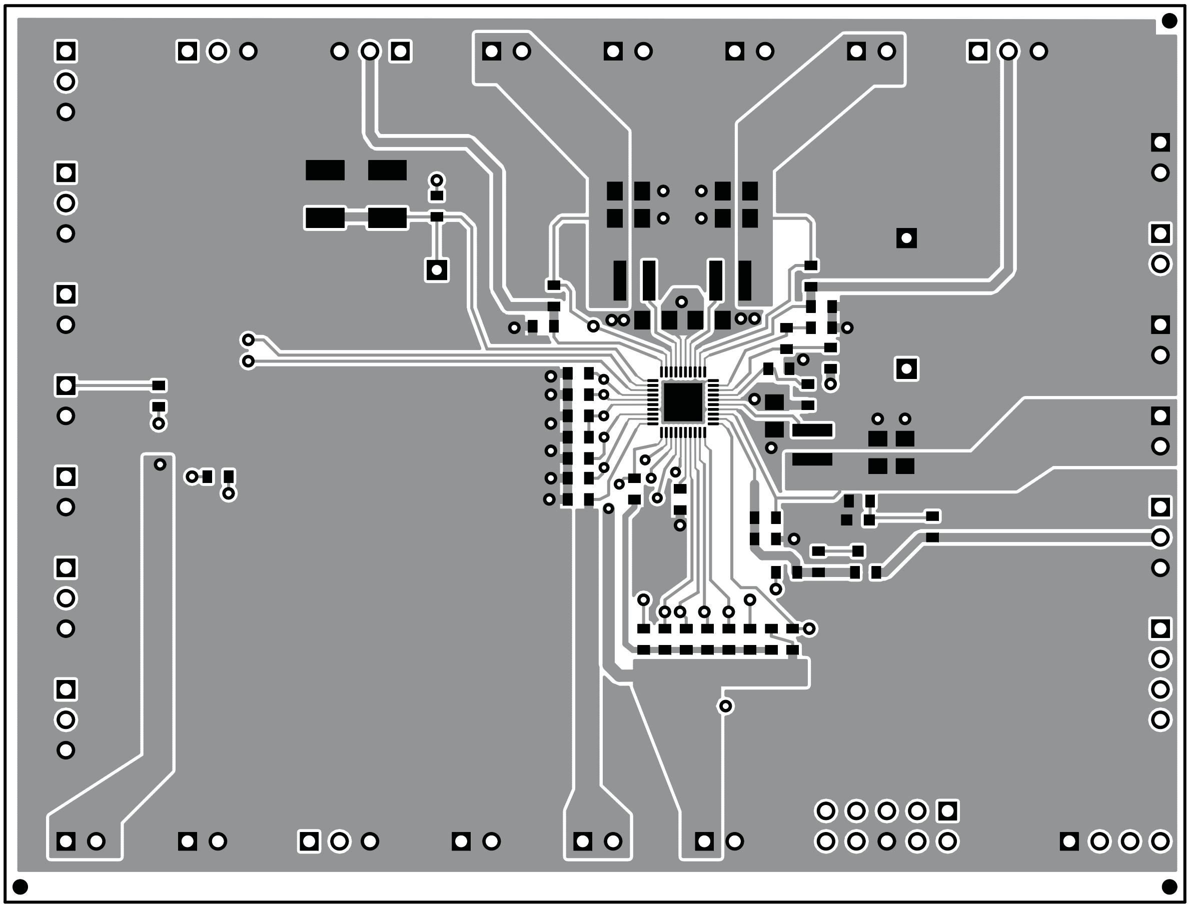

Figure 2. Top Layer Routing

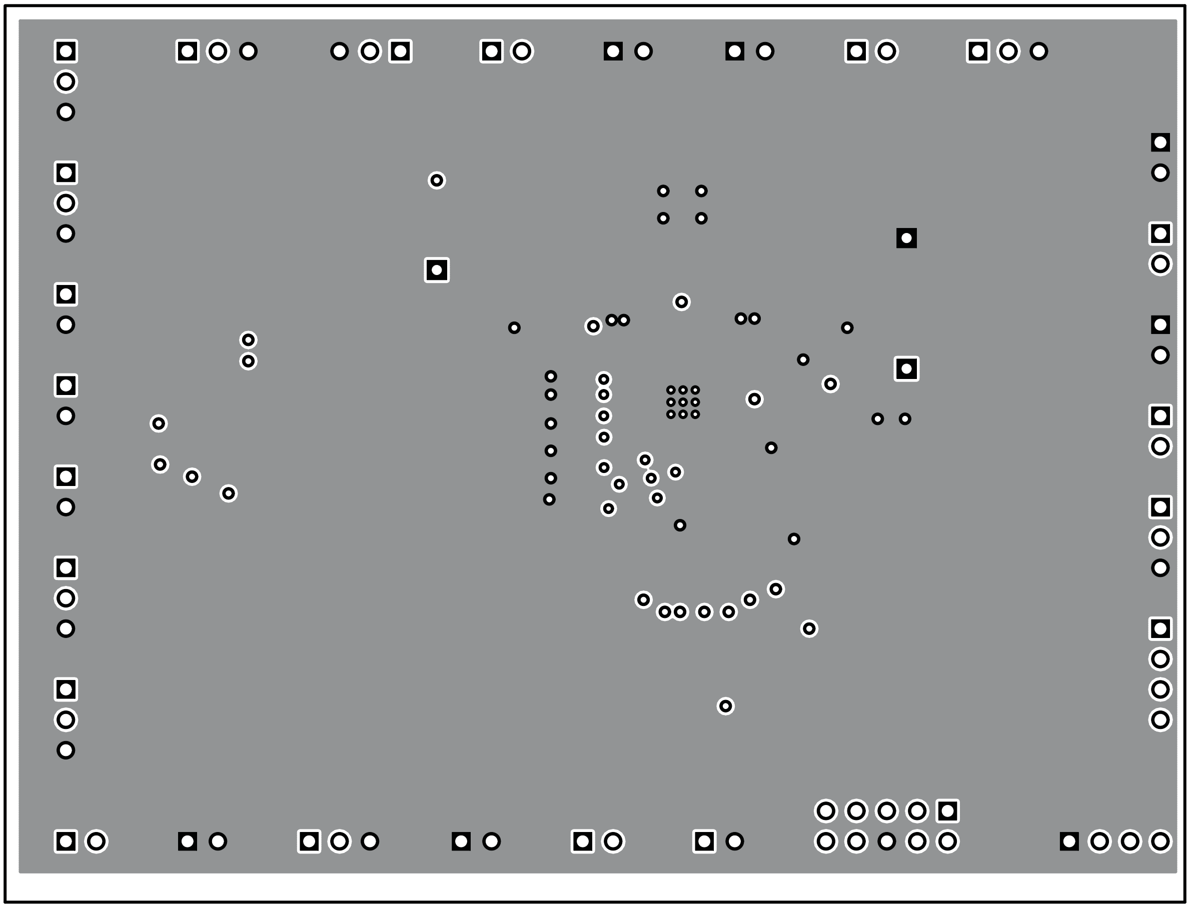

Figure 3. Layer-2 Routing, GND Plane

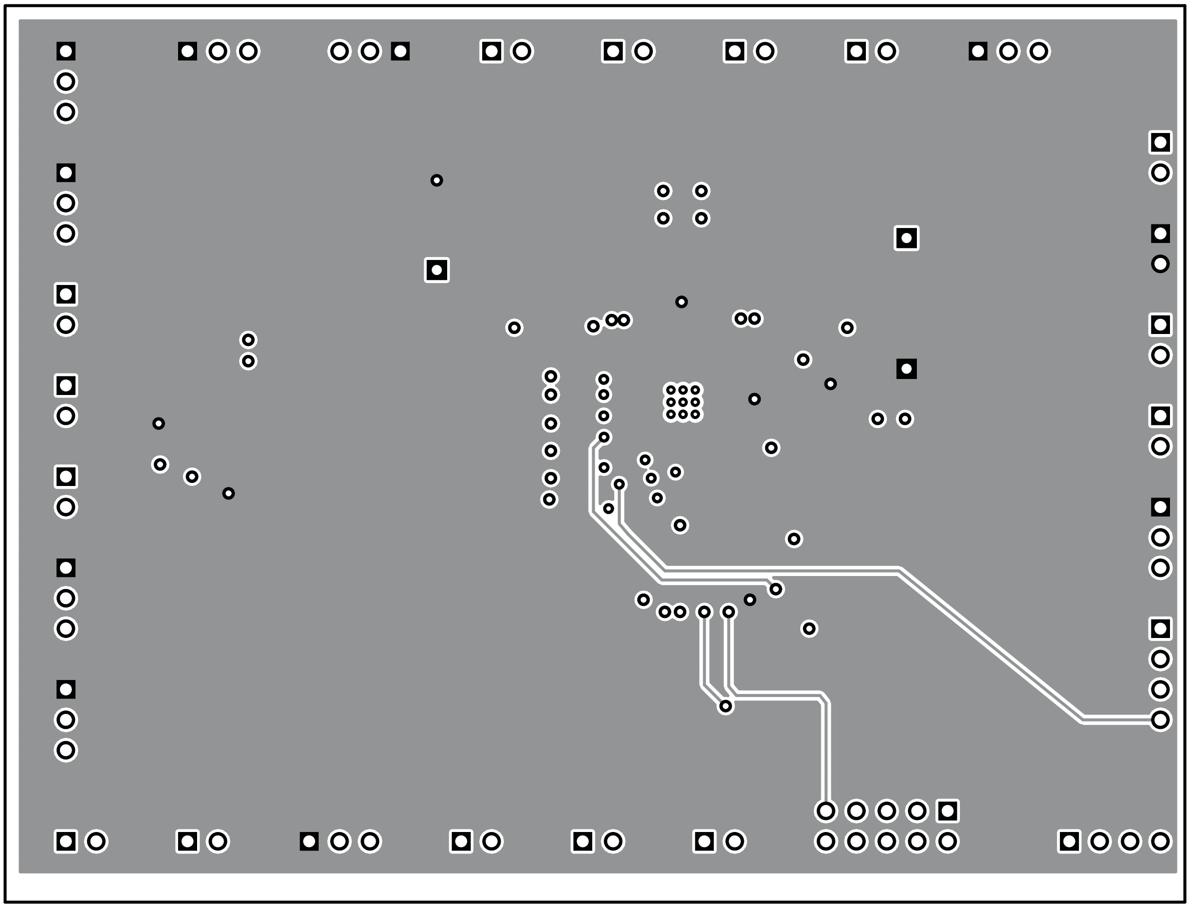

Figure 4. Layer-3 Routing, Vin Plane

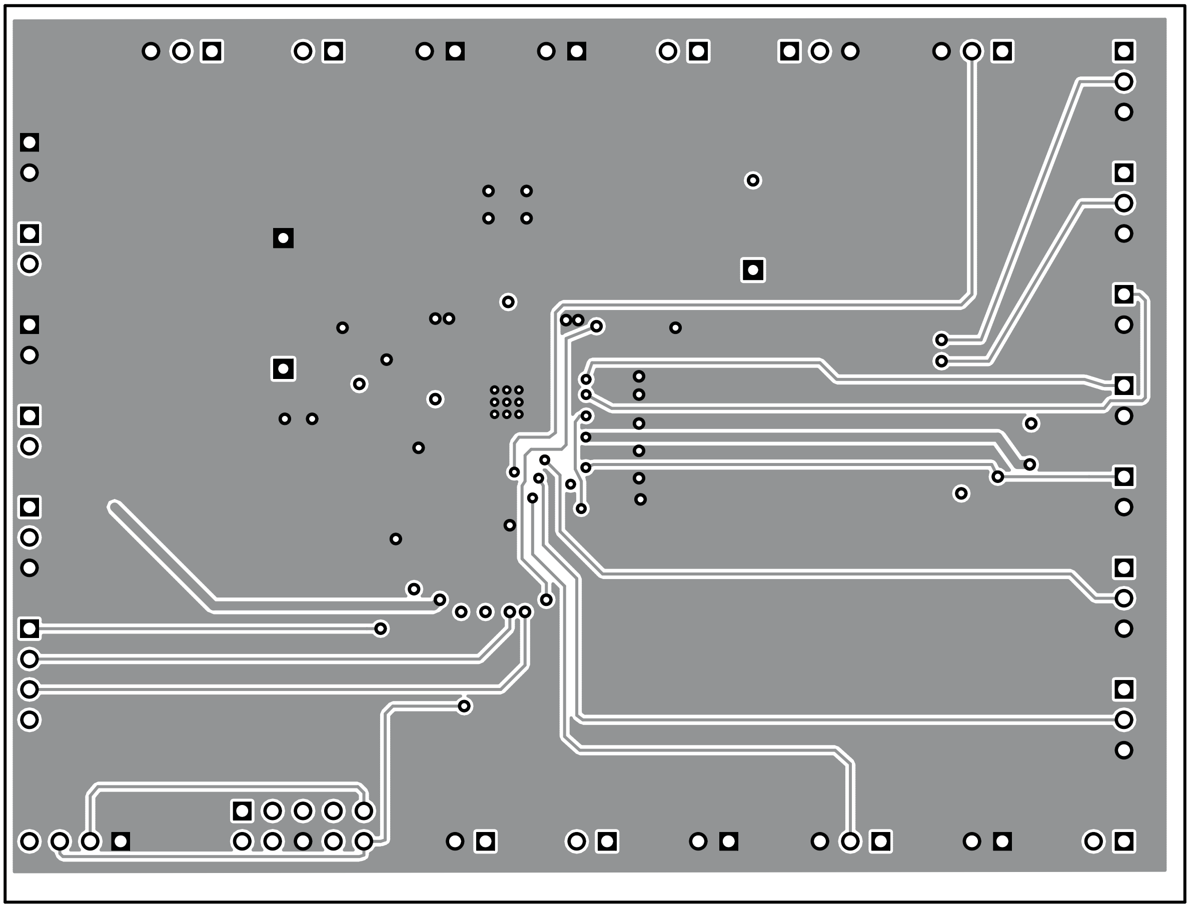

Figure 5. Bottom Layer Routing

5 Schematic and Bill of Materials

This section provides the TPS65023B/TPS650231EVM-664 schematic and bill of materials.

5.1 Schematic

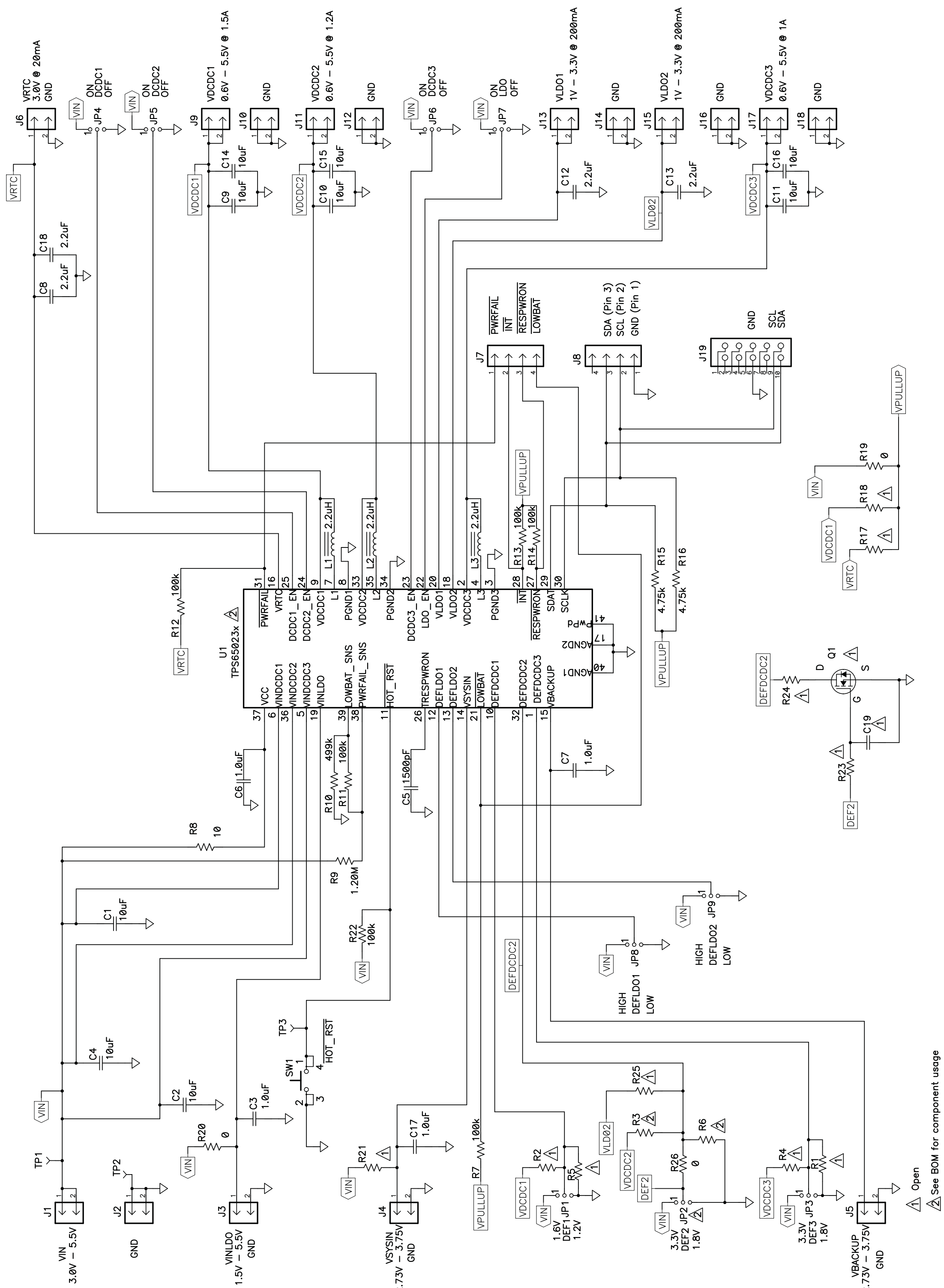

Figure 6. TPS65023B/TPS650231EVM-664 Schematic

5.2 Bill of Materials

Table 4. TPS65023B/TPS650231EVM-664 Bill of Materials

Count		RefDes	Value	Description	Size	Part Number
-001	-002					
9	9	C1, C2, C4, C9, C10, C11, C14–C16	10 µF	Capacitor, Ceramic, 6.3V, X5R, 10%	0805	Std
4	4	C3, C6, C7, C17	1.0 µF	Capacitor, Ceramic, 6.3V, X5R,10%	0603	Std
1	1	C5	1500 pF	Capacitor, Ceramic, 50V, X7R, 10%	0603	C1608X7R1H152K
4	4	C8, C18	2.2 µF	Capacitor, Ceramic, 6.3V, X5R, 10%	0603	C1608X5R0J225K
3	3	L1, L2, L3	2.2 µH	Inductor, SMT, 2.0A, 110mΩ	0.157 x 0.157	LPS3015-222ML
0	0	R1, R2, R4–R6, R17, R18, R21	Open	Resistor, Chip, 1/16W, 1%	0603	Std
0	1	R3	182k	Resistor, Chip, 1/16W, 1%	0603	Std
0	1	R6	365k	Resistor, Chip, 1/16W, 1%	0603	Std
6	6	R7, R11–R14, R22	100k	Resistor, Chip, 1/16W, 1%	0603	Std
1	1	R8	10	Resistor, Chip, 1/16W, 1%	0603	Std
1	1	R9	1.20M	Resistor, Chip, 1/16W, 1%	0603	Std
1	1	R10	499k	Resistor, Chip, 1/16W, 1%	0603	Std
2	2	R15, R16	4.75k	Resistor, Chip, 1/16W, 1%	0603	Std
2	2	R19, R20	0	Resistor, Chip, 1/16W, 5%	0603	Std
1	0	U1	TPS65023BRSB	IC, Power Management IC for Li-Ion Powered Systems	QFN	TPS65023BRSB
0	1	U1	TPS650231RSB	IC, Power Management IC for Li-Ion Powered Systems	QFN	TPS650231RSB

5.3 Related Documentation

TPS650231, Power Management IC For Li-Ion Powered Systems data sheet (SLVSAE3)

Evaluation Board/Kit Important Notice

Texas Instruments (TI) provides the enclosed product(s) under the following conditions:

This evaluation board/kit is intended for use for **ENGINEERING DEVELOPMENT, DEMONSTRATION, OR EVALUATION PURPOSES ONLY** and is not considered by TI to be a finished end-product fit for general consumer use. Persons handling the product(s) must have electronics training and observe good engineering practice standards. As such, the goods being provided are not intended to be complete in terms of required design-, marketing-, and/or manufacturing-related protective considerations, including product safety and environmental measures typically found in end products that incorporate such semiconductor components or circuit boards. This evaluation board/kit does not fall within the scope of the European Union directives regarding electromagnetic compatibility, restricted substances (RoHS), recycling (WEEE), FCC, CE or UL, and therefore may not meet the technical requirements of these directives or other related directives.

Should this evaluation board/kit not meet the specifications indicated in the User's Guide, the board/kit may be returned within 30 days from the date of delivery for a full refund. THE FOREGOING WARRANTY IS THE EXCLUSIVE WARRANTY MADE BY SELLER TO BUYER AND IS IN LIEU OF ALL OTHER WARRANTIES, EXPRESSED, IMPLIED, OR STATUTORY, INCLUDING ANY WARRANTY OF MERCHANTABILITY OR FITNESS FOR ANY PARTICULAR PURPOSE.

The user assumes all responsibility and liability for proper and safe handling of the goods. Further, the user indemnifies TI from all claims arising from the handling or use of the goods. Due to the open construction of the product, it is the user's responsibility to take any and all appropriate precautions with regard to electrostatic discharge.

EXCEPT TO THE EXTENT OF THE INDEMNITY SET FORTH ABOVE, NEITHER PARTY SHALL BE LIABLE TO THE OTHER FOR ANY INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES.

TI currently deals with a variety of customers for products, and therefore our arrangement with the user **is not exclusive.**

TI assumes **no liability for applications assistance, customer product design, software performance, or infringement of patents or services described herein.**

Please read the User's Guide and, specifically, the Warnings and Restrictions notice in the User's Guide prior to handling the product. This notice contains important safety information about temperatures and voltages. For additional information on TI's environmental and/or safety programs, please contact the TI application engineer or visit www.ti.com/esh.

No license is granted under any patent right or other intellectual property right of TI covering or relating to any machine, process, or combination in which such TI products or services might be or are used.

FCC Warning

This evaluation board/kit is intended for use for **ENGINEERING DEVELOPMENT, DEMONSTRATION, OR EVALUATION PURPOSES ONLY** and is not considered by TI to be a finished end-product fit for general consumer use. It generates, uses, and can radiate radio frequency energy and has not been tested for compliance with the limits of computing devices pursuant to part 15 of FCC rules, which are designed to provide reasonable protection against radio frequency interference. Operation of this equipment in other environments may cause interference with radio communications, in which case the user at his own expense will be required to take whatever measures may be required to correct this interference.

EVM Warnings and Restrictions

It is important to operate this EVM within the input voltage range of 3.3 V to 5.5 V and the output voltage range of 0.8 V to 3.3 V .

Exceeding the specified input range may cause unexpected operation and/or irreversible damage to the EVM. If there are questions concerning the input range, please contact a TI field representative prior to connecting the input power.

Applying loads outside of the specified output range may result in unintended operation and/or possible permanent damage to the EVM. Please consult the EVM User's Guide prior to connecting any load to the EVM output. If there is uncertainty as to the load specification, please contact a TI field representative.

During normal operation, some circuit components may have case temperatures greater than 60°C. The EVM is designed to operate properly with certain components above 60°C as long as the input and output ranges are maintained. These components include but are not limited to linear regulators, switching transistors, pass transistors, and current sense resistors. These types of devices can be identified using the EVM schematic located in the EVM User's Guide. When placing measurement probes near these devices during operation, please be aware that these devices may be very warm to the touch.