dsPICDEM™ 1.1 DEVELOPMENT BOARD 3.3V RETROFIT

Modification Instructions

1.1 DSPICDEM 1.1 DEVELOPMENT BOARD 3.3 VDC RETROFIT

This retrofit allows the dsPICDEM 1.1 Development Board (DM 300014, PCB Assembly 02-01719) to be used as a development tool for both dsPIC30F and dsPIC33F devices. The development board is modified to replace the digital voltage regulator, remove the analog voltage regulator and replace transceiver chips with substitute components that support a wider range of operation (2.8–5.5 VDC).

After completing this modification you will be able to interchangeably work with 3.3 VDC or 5 VDC devices by simply adding or removing the jumper on the voltage regulator miniature board.

Note: The LCD is not currently supported when 3.3 VDC operation is selected.

Other development boards that support both dsPIC30F and dsPIC33F devices include:

- dsPICDEM™ 80-Pin Starter Development Board (Part # DM300019)
- Explorer 16 Development Board (Part # DM240001)

If you have, or wish to purchase, one of these boards you probably don't need to retrofit your dsPICDEM 1.1 Development Board.

If you do modify an existing dsPICDEM 1.1 Development Board, you are responsible for maintaining good workmanship practices. To avoid damage to the printed circuit board, it is strongly recommended that components be removed by first cutting the leads and then unsoldering the remaining lead remnants.

Contact your sales representative or DSCD marketing for additional information.

THE DSPICDEM 1.1 DEVELOPMENT BOARD 90-DAY WARRANTY WILL EXPIRE IF YOU MODIFY THE BOARD (REGARDLESS OF WHETHER YOU USE THESE MODIFICATION INSTRUCTIONS OR PERFORM YOUR OWN MODIFICATIONS).

THE MODIFICATION INSTRUCTIONS ARE SUPPLIED BY MICROCHIP "AS IS". MICROCHIP DOES NOT WARRANT THAT THE OPERATION OF THE DEVELOPMENT BOARD, ITS FIRMWARE OR SOFTWARE WILL BE ACCURATE, UNINTERRUPTED OR ERROR-FREE.

It is your responsibility to ensure that the board continues to function properly and any application you develop meets your specifications.

1.1.1 What You Need

The following table lists the parts you need for this modification. Except for the RS422 Transceiver used with the RS-485/RS-422 port, all these parts are contained in a retrofit kit.

Item	Ref. Des.	Qty	Description	Value	Mfg	Mfg Part #
Parts on dsPICDEM 1.1 Development Board						
1	U1,U3	2	IC SMT, Driver, RS232 Xmtr/Rcvr, 16-SOIC(wide)	—	MAXIM	MAX3232CDWR
2	U12	1	IC SMT, 3.3V CAN TX/RX, 8P SOIC (0.150")	—	TI	SN65HVD230DR
3	U4	1	RS422 (See Note 1)	—	MAXIM	MAX3491ECSD
Parts on Regulator Adapter Board						
4	VR1		IC Positive Adj Regulator TO-263	—	National Semi	LM317S
5	J1	1	2 Pin.100, SMT header	—	Samtec	TSM-102-01-S-DV
6	R1	1	Resistor, 1206	412R	Rohm	MCR18EZHF4120
7	R2	1	Resistor, 1206	249R	Rohm	MCR18EZHF2490
8	R3	1	Resistor, 1206	294R	Rohm	MCR18EZHF2940
9		1	PCB Assembly	—	—	—
10		3	Jumper wire	—	—	—

1: This part is not supplied in the retrofit kit. The RS422 Transceiver will only need to be replaced if your application uses this feature with 3.3 VDC operation.

Note: The modification kit does not include a dsPIC33F device. It must purchased separately.

1.2 OVERVIEW OF MODIFICATION

Figure 1 provides an overview of what you need to do to complete this modification. Before proceeding, read the step-by-step procedures carefully to avoid damaging the dsPICDEM 1.1 board or any of the components.

Figure 1: dsPICDEM 1.1 Retrofit Process

CAUTION

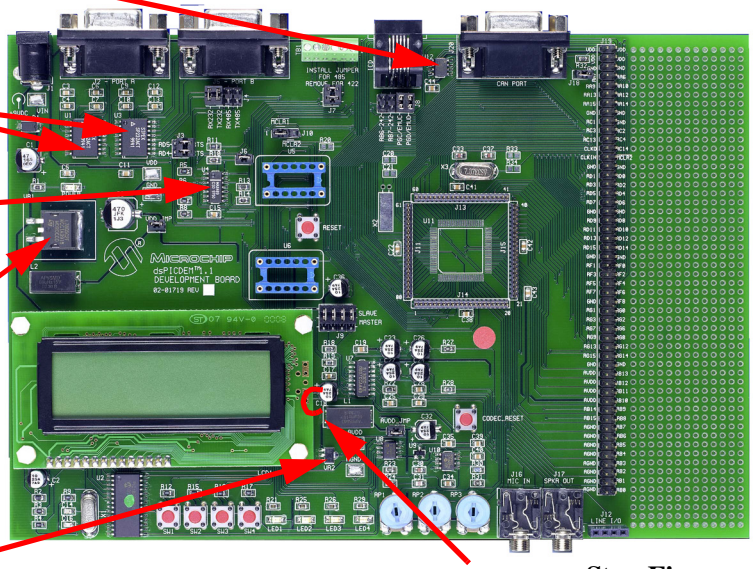
To avoid damage to the PCB, remove components by cutting the leads. Then carefully unsolder each lead.

Step One: Replace CAN Transceiver U12 with the SN65HVD230DR chip in the retrofit kit.

Step Two: Replace UART Transceivers U1 & U3 with the MAX 3232 chips in the retrofit kit. Optionally, replace RS422 Transceiver U4 with MAX 3491 chip.

Step Three: Replace the digital voltage regulator (VR1) with the miniature PCB assembly (refer to step-by-step procedures).

Step Four: Remove the analog voltage regulator (VR2).



Step Five: Install jumper wire from the plus end of C18 to the closest end of L1 to bypass analog reg-

1.3 STEP-BY-STEP PROCEDURES

1.3.1 Step One: Replace CAN Transceiver

1. Remove existing CAN chip U12 by cutting the leads and discarding the chip.
2. Carefully unsolder each lead remnant.
3. Install the TI SN65HVD230DR transceiver chip included.

Note: *Be sure to align pin 1 on the chip to pin 1 on the board.*

1.3.2 Step Two: Replace UART Transceivers

1. Remove existing UART chips U1 and U3 (and, optionally, RS422 chip U4, by cutting the leads and discarding the chip.
2. Carefully unsolder each lead remnant.
3. Solder the MAX3232CDWR transceiver chips into reference designator locations U1 and U3. If appropriate, solder the MAX3491ECSD transceiver chip into reference designator location U4.

Note: *Be sure to align pin 1 on the chip to pin 1 on the board.*

1.3.3 Step Three: Replace Voltage Regulator VR1

1. Cut the leads on the digital voltage regulator chip (VR1), then unsolder the chip from its heat sink pad.
2. Unsolder the lead remnants.
3. Install the miniature regulator board:
   - Solder the miniature board to the dsPICDEM 1.1 board at the four solder points shown in Figure 2. Be sure the PCB assembly does not overlap the VIN and VOUT pads. Use a HOT iron to ensure good solder flow through the holes.
   - Add jumper wires for VIN and VOUT.
4. Place the regulator on the heat sink pad on the miniature regulator board, aligning the pins with the soldering pads and leaving a small area of the heat sink exposed near the regulator's tab (approximately 1.25 mm), as shown in Figure 3.
5. Solder the three regulator leads. Then solder the regulator tab to the exposed area of the heat sink.

1.3.4 Step Four: Remove Voltage Regulator VR2

1. Cut the three leads on regulator VR2.
1. Unsolder the tab to remove the regulator chip.
2. Unsolder the lead remnants.
3. Leave the VR2 socket empty.

1.3.5 Step Five: Install VR2 Bypass Jumper

1. Solder one end of wire jumper to the end of L1 closest to the LCD (see Figure 4).
2. Solder the other end to the + side of C18

Figure 2: Installing the Regulator Miniature Board

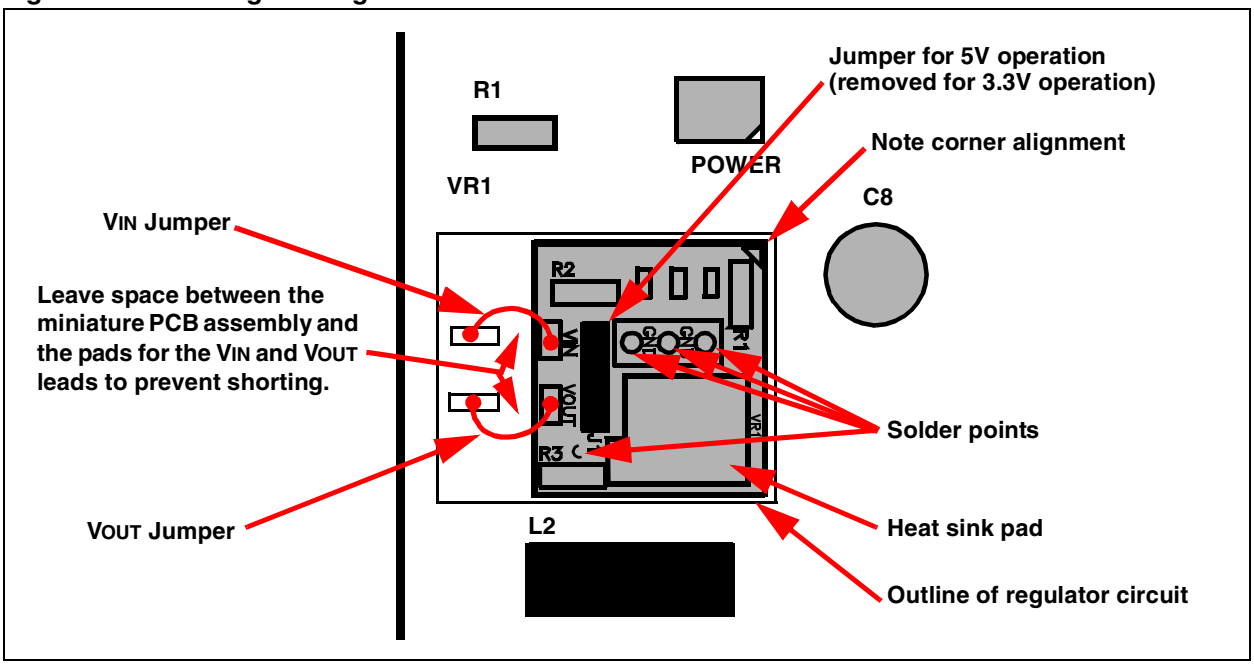


Figure 3: Installing the Regulator Chip

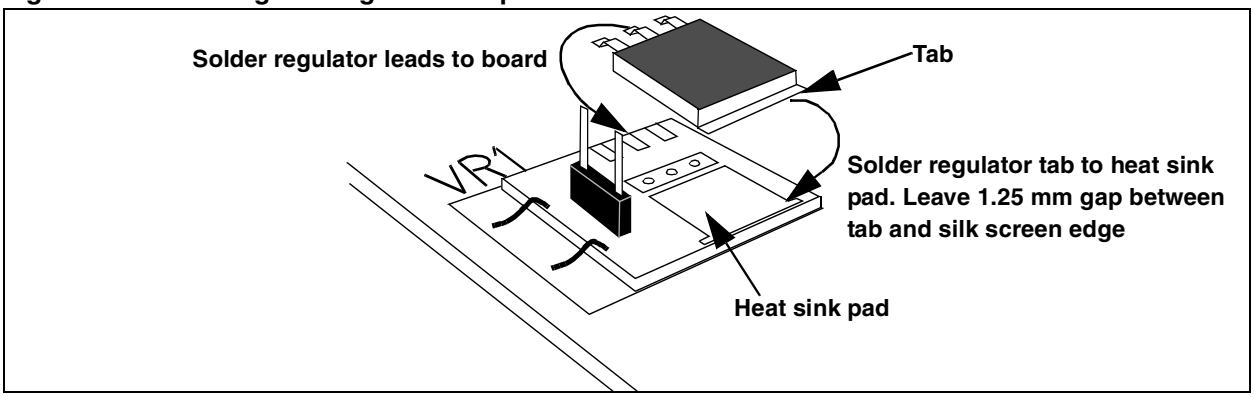
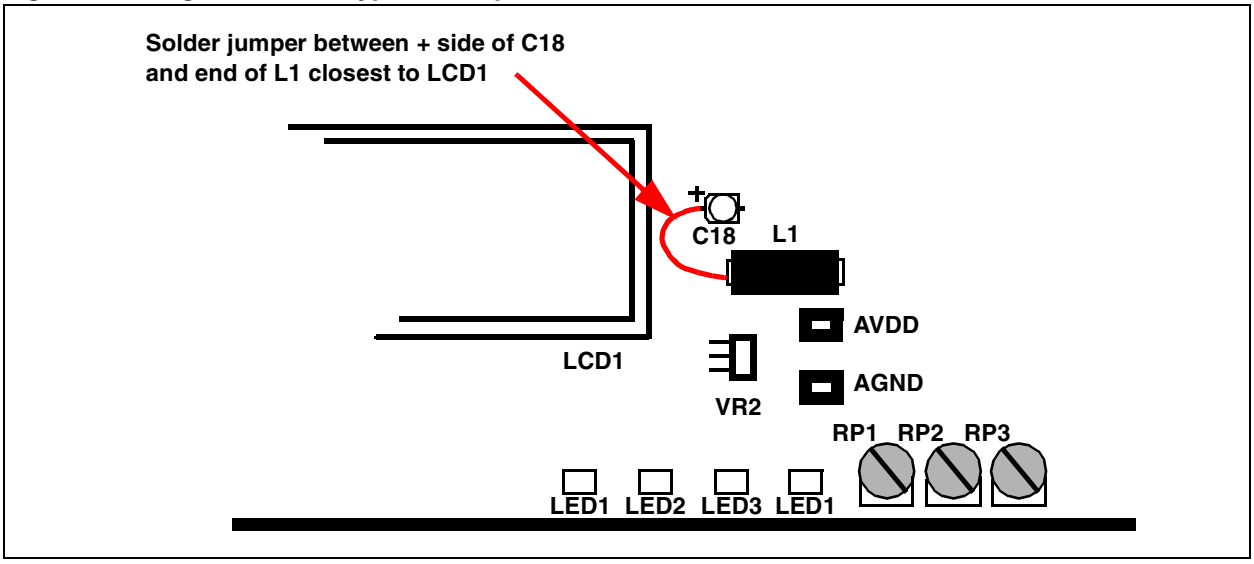


Figure 4: Regulator VR2 Bypass Jumper

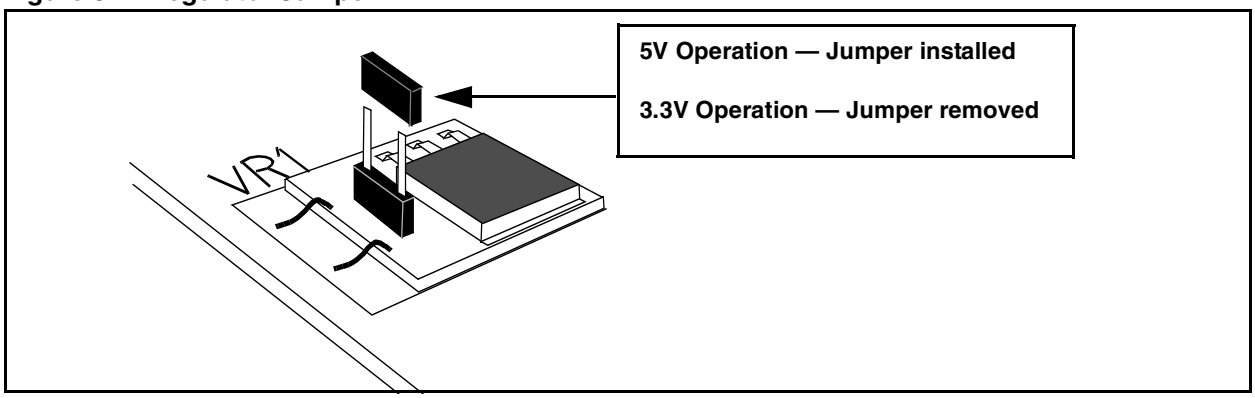


1.4 USING THE BOARD

To use the board with +5 VDC devices, install the regulator jumper (see Figure 5). To use the board with +3.3 VDC devices, remove the regulator jumper.

Note: Remember that the LCD is inoperable when the board is used at 3.3 VDC.

Figure 5: Regulator Jumper



1.5 SCHEMATIC DIAGRAM

Figure 6 is a schematic of the digital voltage regulator replacement circuit.

Figure 6: dsPICDEM 1.1 Development Board 3.3V Modification