

C8051F38X DEVELOPMENT KIT USER'S GUIDE

1. Kit Contents

The C8051F38x Development Kit contains the following items:

- C8051F380 Target Board
- C8051Fxxx Development Kit Quick-start Guide
- Silicon Laboratories IDE and Product Information CD-ROM. CD content includes:
 - Silicon Laboratories Integrated Development Environment (IDE)
 - Keil 8051 Development Tools (macro assembler, linker, evaluation "C" compiler)
 - Source code examples and register definition files
 - Documentation
 - C8051F38x Development Kit User's Guide (this document)
- AC to DC Power Adapter
- USB Debug Adapter (USB to Debug Interface)
- USB Cable

2. Hardware Setup using a USB Debug Adapter

The target board is connected to a PC running the Silicon Laboratories IDE via the USB Debug Adapter as shown in Figure 1.

1. Connect the USB Debug Adapter to the DEBUG connector on the target board with the 10-pin ribbon cable.
 2. Connect one end of the USB cable to the USB connector on the USB Debug Adapter.
 3. Connect the other end of the USB cable to a USB Port on the PC.
 4. Connect the ac/dc power adapter to power jack P1 on the target board.
- Use the Reset button in the IDE to reset the target when connected using a USB Debug Adapter.
 - Remove power from the target board before removing the ribbon cable from the target board. Connecting or disconnecting the cable when the devices have power can damage the device and/or the USB Debug Adapter.

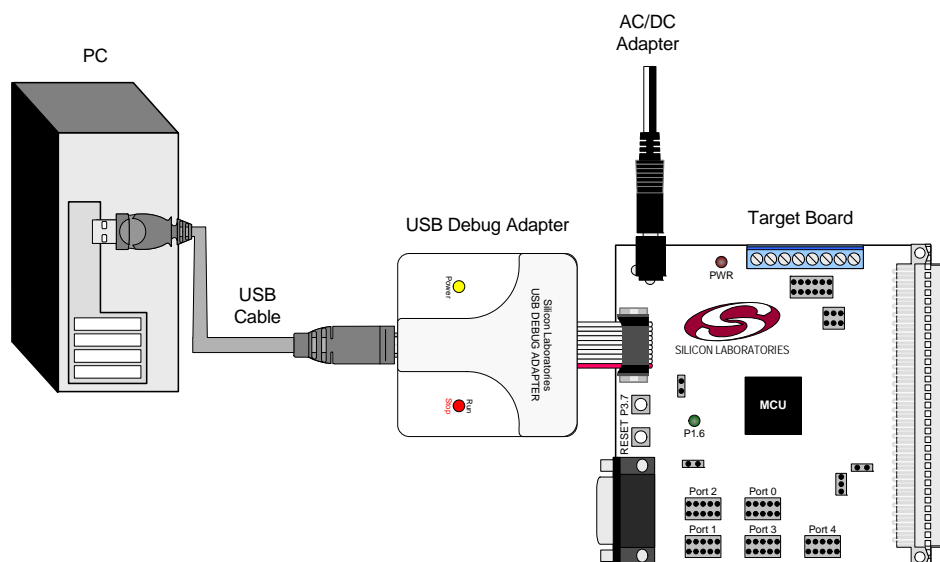


Figure 1. Hardware Setup using a USB Debug Adapter

3. Software Setup

The included CD-ROM contains the Silicon Laboratories Integrated Development Environment (IDE), Keil software 8051 tools and additional documentation. Insert the CD-ROM into your PC's CD-ROM drive. An installer will automatically launch, allowing you to install the IDE software or read documentation by clicking buttons on the Installation Panel. If the installer does not automatically start when you insert the CD-ROM, run *autorun.exe* found in the root directory of the CD-ROM. Refer to the *readme.txt* file on the CD-ROM for the latest information regarding known IDE problems and restrictions.

4. Silicon Laboratories Integrated Development Environment

The Silicon Laboratories IDE integrates a source-code editor, source-level debugger and in-system Flash programmer. The use of third-party compilers and assemblers is also supported. This development kit includes the Keil Software A51 macro assembler, BL51 linker and evaluation version C51 "C" compiler. These tools can be used from within the Silicon Laboratories IDE.

4.1. System Requirements

The Silicon Laboratories IDE requirements:

- Pentium-class host PC running Windows 2000 or Windows XP.
- One available COM or USB port.
- 64 MB RAM and 40 MB free HD space recommended.

4.2. Assembler and Linker

A full-version Keil A51 macro assembler and BL51 banking linker are included with the development kit and are installed during IDE installation. The complete assembler and linker reference manual can be found under the **Help** menu in the IDE or in the "*SiLabsMCU\hlp*" directory (A51.pdf).

4.3. Evaluation C51 "C" Compiler

An evaluation version of the Keil C51 "C" compiler is included with the development kit and is installed during IDE installation. The evaluation version of the C51 compiler is the same as the full professional version except code size is limited to 4 kB and the floating point library is not included. The C51 compiler reference manual can be found under the **Help** menu in the IDE or in the "*SiLabsMCU\hlp*" directory (C51.pdf).

4.4. Using the Keil Software 8051 Tools with the Silicon Laboratories IDE

To perform source-level debugging with the IDE, you must configure the Keil 8051 tools to generate an absolute object file in the OMF-51 format with object extensions and debug records enabled. You may build the OMF-51 absolute object file by calling the Keil 8051 tools at the command line (e.g. batch file or make file) or by using the project manager built into the IDE. The default configuration when using the Silicon Laboratories IDE project manager enables object extension and debug record generation. Refer to Applications Note **AN104 - Integrating Keil 8051 Tools Into the Silicon Labs IDE** in the "*SiLabsMCU\Documentation\Apnotes*" directory on the CD-ROM for additional information on using the Keil 8051 tools with the Silicon Laboratories IDE.

To build an absolute object file using the Silicon Laboratories IDE project manager, you must first create a project. A project consists of a set of files, IDE configuration, debug views, and a target build configuration (list of files and tool configurations used as input to the assembler, compiler, and linker when building an output object file).

The following sections illustrate the steps necessary to manually create a project with one or more source files, build a program and download the program to the target in preparation for debugging. (The IDE will automatically create a single-file project using the currently open and active source file if you select **Build/Make Project** before a project is defined.)

4.4.1. Creating a New Project

1. Select **Project→New Project** to open a new project and reset all configuration settings to default.
2. Select **File→New File** to open an editor window. Create your source file(s) and save the file(s) with a recognized extension, such as .c, .h, or .asm, to enable color syntax highlighting.
3. Right-click on “New Project” in the **Project Window**. Select **Add files to project**. Select files in the file browser and click Open. Continue adding files until all project files have been added.
4. For each of the files in the **Project Window** that you want assembled, compiled and linked into the target build, right-click on the file name and select **Add file to build**. Each file will be assembled or compiled as appropriate (based on file extension) and linked into the build of the absolute object file.

Note: If a project contains a large number of files, the “Group” feature of the IDE can be used to organize. Right-click on “New Project” in the **Project Window**. Select **Add Groups to project**. Add pre-defined groups or add customized groups. Right-click on the group name and choose **Add file to group**. Select files to be added. Continue adding files until all project files have been added.

4.4.2. Building and Downloading the Program for Debugging

1. Once all source files have been added to the target build, build the project by clicking on the **Build/Make Project** button in the toolbar or selecting **Project→Build/Make Project** from the menu.

Note: After the project has been built the first time, the **Build/Make Project** command will only build the files that have been changed since the previous build. To rebuild all files and project dependencies, click on the **Rebuild All** button in the toolbar or select **Project→Rebuild All** from the menu.

2. Before connecting to the target device, several connection options may need to be set. Open the **Connection Options** window by selecting **Options→Connection Options...** in the IDE menu. First, select the appropriate adapter in the “Serial Adapter” section. Next, the correct “Debug Interface” must be selected. C8051F38x family devices use the Silicon Labs 2-wire (C2) debug interface. Once all the selections are made, click the OK button to close the window.
3. Click the **Connect** button in the toolbar or select **Debug→Connect** from the menu to connect to the device.
4. Download the project to the target by clicking the **Download Code** button in the toolbar.
5. **Note:** To enable automatic downloading if the program build is successful select **Enable automatic connect/download after build** in the **Project→Target Build Configuration** dialog. If errors occur during the build process, the IDE will not attempt the download.
6. Save the project when finished with the debug session to preserve the current target build configuration, editor settings and the location of all open debug views. To save the project, select **Project→Save Project As...** from the menu. Create a new name for the project and click on **Save**.

5. Example Source Code

Example source code and register definition files are provided in the “*SiLabsMCUExamples\C8051F38x*” directory during IDE installation. These files may be used as a template for code development. Example applications include a blinking LED example which configures the green LED on the target board to blink at a fixed rate. A Universal Serial Bus (USB) application example is also included with the C8051F38x development kit.

5.1. Register Definition Files

Register definition files *C8051F380.inc* and *C8051F380.h* define all SFR registers and bit-addressable control/status bits. They are installed into the “*SiLabsMCUExamples\C8051F38x*” directory during IDE installation. The register and bit names are identical to those used in the C8051F38x data sheet. Both register definition files are also installed in the default search path used by the Keil Software 8051 tools. Therefore, when using the Keil 8051 tools included with the development kit (A51, C51), it is not necessary to copy a register definition file to each project's file directory.

5.2. Blinking LED Example

The example source files *blink.asm* and *blink.c* show examples of several basic C8051F38x functions. These include; disabling the watchdog timer (WDT), configuring the Port I/O crossbar, configuring a timer for an interrupt routine, initializing the system clock, and configuring a GPIO port. When compiled/assembled and linked this program flashes the green LED on the C8051F380 target board about five times a second using the interrupt handler with a timer.

5.3. Universal Serial Bus (USB) Application Example

The included USB application example requires Windows 98SE/2000/XP operating system. This example consists of three parts: firmware running on the C8051F380 device, a USB device driver running on the host PC, and a host application. The host application communicates with the C8051F380 via USB, allowing the user to view and to change the state of several I/O peripherals on the C8051F380 target board. The example files can be found in the “*SiLabsMCUExamples\C8051F38x\USB_INT*” directory. Located in this directory is the project file for the firmware, the USB driver files, and the host application executable. In addition, three subdirectories are provided that contain the source code for each part of the example application; Host Application Source, Driver Source, and Firmware. Use the following instructions to set up the example USB firmware, install the USB driver, and then run the example host application.

After the target board is connected to the host computer via USB, the example application enumerates and sends data between the target and the host application using Endpoint 0, 1, and 2. The Endpoint 0 packets are control packets as defined in Chapter 9 of the Universal Serial Bus Specification, version 2.0. Endpoints 1 and 2 are used for data IN and OUT, respectively. These data packets consist of 8 bytes used to denote the current status of the buttons, LEDs, Potentiometer, and logic levels of various port pins on the target board. Accessing the USB core (i.e., accessing the USB registers and moving data to and from the Endpoint FIFOs on the target board) is documented in Silicon Laboratories' C8051F38x data sheet.

5.3.1. Setting-up the Firmware

1. Set up the hardware on the C8051F380 target board according to the instructions in Section 2.
2. Create a project using the following steps.
 - a. Open the Silicon Laboratories IDE.
 - b. Click on the **Project→Open Project...** option in the IDE menus.
 - c. Answer **No** to the following message:
 - d. Not all project windows can be restored if not connected.
Do you want to connect now?
 - e. Select the *USB_Int.wsp* project, located by default in the *SiLabs\MCU\Examples\C8051F38x\C\USB_INT* directory, and click the **Open** button.
 - f. Click the **Build/Make Project** button in the IDE toolbar.
 - g. The project should now be compiled and linked successfully, with no errors.
 - h. Connect to the device by clicking the **Connect** button on the IDE toolbar.
 - i. Download the project to the target by clicking the **Download Code** button in the IDE toolbar.
3. Begin program execution on the target by either of the following methods:
 - a. Click the **Go** button in the IDE toolbar.
 - b. Disconnect the IDE from the device by pressing the **Disconnect** button on the toolbar. Next, press the **Reset** button on the C8051F380 target board.

5.3.2. Installing the USB Driver

1. Connect the USB cable between the host computer and the C8051F380 target board, as shown in Figure 2. See Figure 2 on page 6 for the location of the USB connector.
2. At this point, Windows should notify you it has found new hardware and prompt you with a **Found New Hardware** wizard. Press the **Next** button.
3. Select **Search for a suitable driver for my device (recommended)** and press the **Next** button.
4. Select **Specify a location** under “Optional search locations:” and press the **Next** button.
5. Press the **Browse** button to locate the *SiLabs\Int.inf* driver installation file, located by default in the “*SiLabs\MCU\Examples\C8051F38x\C\USB_INT*” directory. Once this file is selected press the **OK** button.
6. Verify that the correct path and filename are shown and press the **Next** button.
7. Press the **Finish** button.

5.3.3. Running the Host Application

1. Once firmware has been downloaded to the target board and the driver has been installed successfully, run the *USBTest.exe* example application located by default in the “*SiLabs\MCU\Examples\C8051F38x\C\USB_INT*” directory.
2. The *USBTest* application will display the status of the potentiometer, the temperature sensor, the two switches, and the lower four bits of Port 0.
3. The application also includes check boxes for turning on the two LEDs and changing the logic levels on the lower four bits of Port 1.
4. If any of the application functions do not operate correctly, verify shorting blocks are installed at the following positions: J3[1-2] (P2.0 Switch), J3[3-4] (P2.1 Switch), J3[5-6] (P2.2 LED), J3[7-8] (P2.3 LED), and J16 and J17 (P2.5 Potentiometer).
5. See Figure 2 on page 6 for the location of headers, LEDs, switches, and potentiometer.

Note: The C8051F380 target board has the ability to be powered through the USB cable. To enable the USB powered mode, short the pins labeled VBUS and VREGIN on the J8 header. Do not short all 3 pins on the J8 header.

6. Target Board

The C8051F38x Development Kit includes a target board with a C8051F380 device pre-installed for evaluation and preliminary software development. Numerous input/output (I/O) connections are provided to facilitate prototyping using the target board. Refer to Figure 2 for the locations of the various I/O connectors.

P1	Power connector (accepts input from 7 to 15 VDC unregulated power adapter)
P2	Analog I/O terminal block
P3	USB connector
P4	RS232 connector
J1	Supply signal header
J2	Port 0 header
J3	Port 1 header
J4	Port 2 header
J5	Port 3 header
J6	Port 4 header
J7	Connects the +3V supply net to the VDD supply net
J8	USB Power or VDD power selection header
J9	Debug connector for debug adapter interface
J10, J11	External crystal enable connectors
J12	Port I/O jumper configuration block
J13	96-pin female connector
J15	Jumper connection for pin 1.5 to capacitors (used when VREF is internally generated)
J16	Jumper connection for potentiometer source to VDD
J17	Jumper connection for potentiometer to pin 2.5
J19	Serial Adapter target board power connector

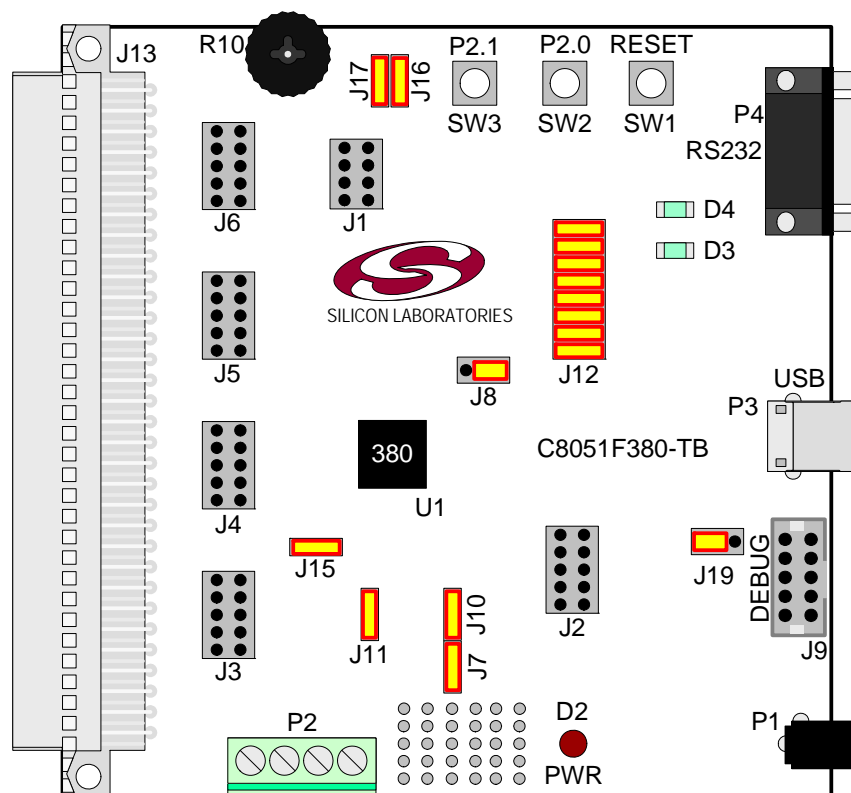


Figure 2. C8051F380 Target Board

6.1. System Clock Sources

The C8051F380 device installed on the target board features a calibrated programmable internal oscillator which is enabled as the system clock source on reset. After reset, the internal oscillator operates at a frequency of 1.5 MHz ($\pm 1.5\%$) by default but may be configured by software to operate at other frequencies. Therefore, in many applications an external oscillator is not required. However, if you wish to operate the C8051F380 device at a frequency not available with the internal oscillator, an external crystal may be used. Refer to the C8051F38x data sheet for more information on configuring the system clock source.

The target board is designed to facilitate the installation of an external crystal. Remove shorting blocks at headers J10 and J11 and install the crystal at the pads marked Y1. Install a 10 M Ω resistor at R1 and install capacitors at C6 and C7 using values appropriate for the crystal you select. Refer to the C8051F38x data sheet for more information on the use of external oscillators.

6.2. Switches and LEDs

Three switches are provided on the target board. Switch RESET is connected to the RESET pin of the C8051F380. Pressing RESET puts the device into its hardware-reset state. Switches P2.0 and P2.1 are connected to the C8051F380's general purpose I/O (GPIO) pins through headers. Pressing P2.0 or P2.1 generates a logic low signal on the port pin. Remove the shorting blocks from the J12 header to disconnect Switch P2.0 and Switch P2.1 from the port pins. The port pin signals are also routed to pins on the J1 I/O connector. See Table 1 for the port pins and headers corresponding to each switch.

Three LEDs are also provided on the target board. The red LED labeled PWR LED is used to indicate a power connection to the target board. The green surface-mount LEDs labeled with port pin names are connected to the C8051F380's GPIO pins through headers. Remove the shorting blocks from the header to disconnect the LEDs from the port pin. The port pin signals are also routed to pins on the J1 I/O connector. See Table 1 for the port pins and headers corresponding to each LED.

Also included on the C8051F380 target board is a 10 k Ω Thumb-wheel Rotary Potentiometer, part number R10. The Potentiometer is connected to the C8051F380's P2.5 pin through the J17 header. Remove the shorting block from the header to disconnect the Potentiometer from the port pin. The port pin signal is also routed to a pin on the J1 I/O connector. See Table 1 for the port pin and header corresponding to the Potentiometer.

Table 1. Target Board I/O Descriptions

Description	I/O	Header
SW1	Reset	none
SW2	P2.0	J12[1–2]
SW3	P2.1	J12[3–4]
Green LED	P2.2	J12[5–6]
Green LED	P2.3	J12[7–8]
Red LED	PWR	none
Potentiometer	P2.5	J17

6.3. Universal Serial Bus (USB) Interface (J14)

A universal serial bus (USB) connector (P3) is provided to facilitate connections to the USB interface on the C8051F380. Table 2 shows the J14 pin definitions.

Table 2. USB Connector Pin Descriptions

Pin #	Description
1	VBUS
2	D-
3	D+
4	GND (Ground)

6.4. PORT I/O Connectors (J2 - J6)

In addition to all port I/O signals being routed to the 96-pin expansion connector, each of the five parallel ports of the C8051F380 has its own 10-pin header connector. Each connector provides a pin for the corresponding port pins 0–7, +3.3 VDC and digital ground. Table 3 defines the pins for the port connectors. The same pin-out order is used for all of the port connectors.

Table 3. J12–J19 Port Connector Pin Descriptions

Pin #	Description
1	Pn.0
2	Pn.1
3	Pn.2
4	Pn.3
5	Pn.4
6	Pn.5
7	Pn.6
8	Pn.7
9	+3 VD (+3.3 VDC)
10	GND (Ground)

6.5. USB Self-powered Configuration (J8)

The C8051F380 target board can be configured as a self-powered USB device to take power from the USB cable instead of the ac/dc adapter connected at P1. To configure the target boards as a self-powered USB device, short the VREGIN and VBUS pins on the J8 header

Note: When the C8051F380 target board is self-powered from the USB, the Serial Adapter is not powered from the target board. The Serial Adapter must be powered directly by connecting the ac/dc adapter to the Serial Adapters' dc power jack. Also, the RS232 Serial Interface (P4) cannot be used when powering the target board from the USB.

6.6. Target Board DEBUG Interface (J9)

The DEBUG connector (J9) provides access to the DEBUG (C2) pins of the C8051F380. It is used to connect the Serial Adapter or the USB Debug Adapter to the target board for in-circuit debugging and Flash programming. Table 4 shows the DEBUG pin definitions.

Table 4. DEBUG Connector Pin Descriptions

Pin #	Description
1	+3 VD (+3.3 VDC)
2, 3, 9	GND (Ground)
4	C2D
5	/RST (Reset)
6	P3.0
7	C2CK
8	Not Connected
10	USB Power

6.7. Serial Interface (P4)

A RS232 transceiver circuit and DB-9 (P4) connector are provided on the target board to facilitate serial connections to UART0 of the C8051F380. The TX, RX, RTS and CTS signals of UART0 may be connected to the DB-9 connector and transceiver by installing shorting blocks on header J12.

J12[9-10]	Install shorting block to connect UART0 TX (P0.4) to transceiver.
J12[11-12]	Install shorting block to connect UART0 RX (P0.5) to transceiver.
J12[13-14]	Install shorting block to connect UART0 RTS (P2.6) to transceiver.
J12[15-16]	Install shorting block to connect UART0 CTS (P2.7) to transceiver.

6.8. Analog I/O (P2)

Several of the C8051F380 target device's port pins are connected to the P2 terminal block. Refer to Table 5 for the P2 terminal block connections.

Table 5. J6 Terminal Block Pin Descriptions

Pin #	Description
1	P1.1 / AIN1.1
2	P1.2 / AIN1.2
3	GND (Ground)
4	P1.5 / VREF (Voltage Reference)

6.9. USB Debug Adapter Target Board Power Connector (J19)

The USB Debug Adapter includes a connection to provide power to the target board. This connection is routed from J9[10] to J19[SER_PWR]. Place a shorting block at header J19[REG_IN-P1_PWR] to power the board directly from an ac/dc power adapter. Place a shorting block at header J19[REG_IN-SER_PWR] to power the board from the USB Debug Adapter. Please note that the second option is not supported with either the EC1 or EC2 Serial Adapters.

7. Schematics

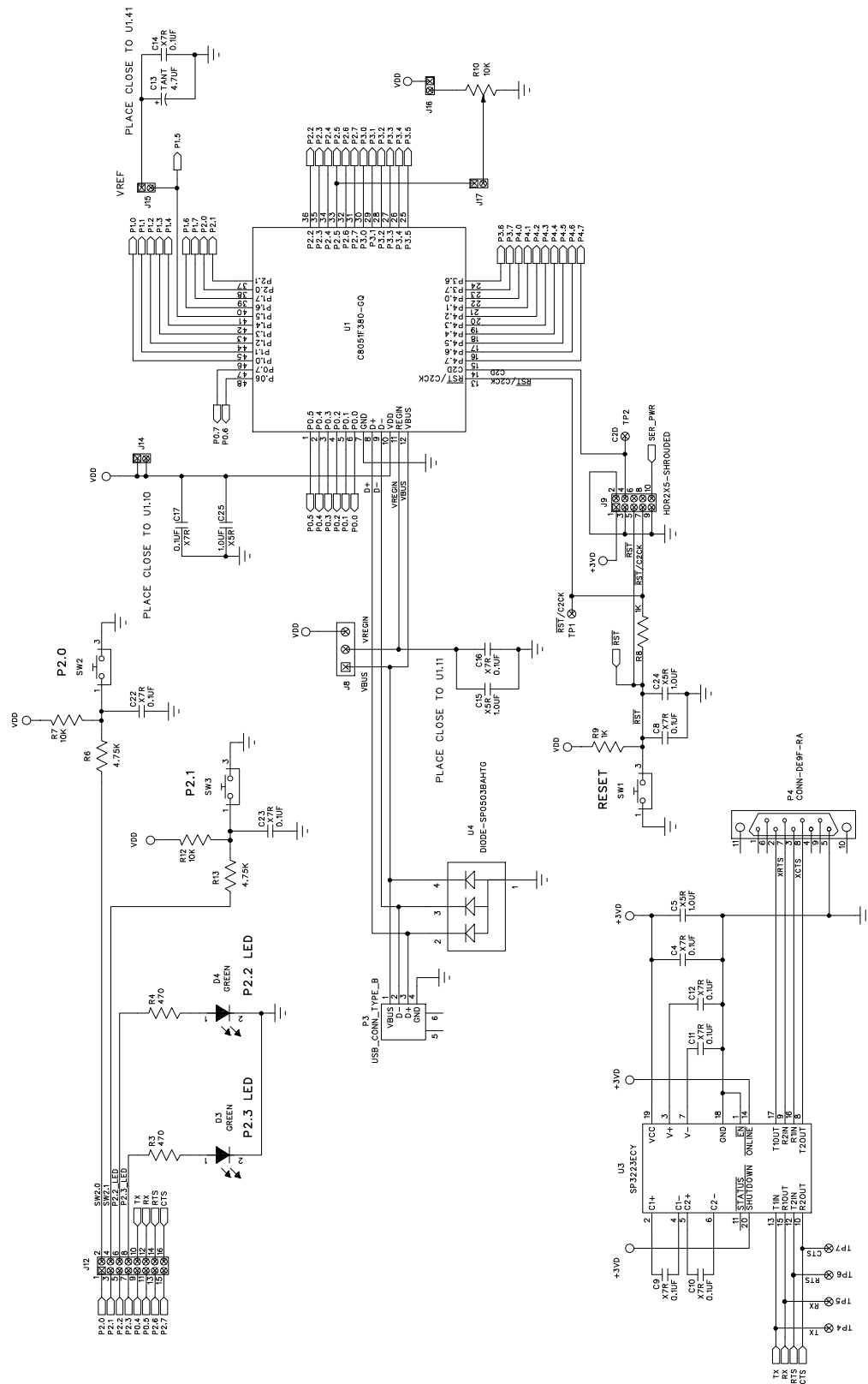


Figure 3. C8051F380 Target Board Schematic (Page 1 of 2)



CONTACT INFORMATION

Silicon Laboratories Inc.
4635 Boston Lane
Austin, TX 78735
Tel: 1+(512) 416-8500
Fax: 1+(512) 416-9669
Toll Free: 1+(877) 444-3032

Please visit the Silicon Labs Technical Support web page:
<https://www.silabs.com/support/pages/contacttechnicalsupport.aspx>
and register to submit a technical support request.

The information in this document is believed to be accurate in all respects at the time of publication but is subject to change without notice. Silicon Laboratories assumes no responsibility for errors and omissions, and disclaims responsibility for any consequences resulting from the use of information included herein. Additionally, Silicon Laboratories assumes no responsibility for the functioning of undescribed features or parameters. Silicon Laboratories reserves the right to make changes without further notice. Silicon Laboratories makes no warranty, representation or guarantee regarding the suitability of its products for any particular purpose, nor does Silicon Laboratories assume any liability arising out of the application or use of any product or circuit, and specifically disclaims any and all liability, including without limitation consequential or incidental damages. Silicon Laboratories products are not designed, intended, or authorized for use in applications intended to support or sustain life, or for any other application in which the failure of the Silicon Laboratories product could create a situation where personal injury or death may occur. Should Buyer purchase or use Silicon Laboratories products for any such unintended or unauthorized application, Buyer shall indemnify and hold Silicon Laboratories harmless against all claims and damages.

Silicon Laboratories and Silicon Labs are trademarks of Silicon Laboratories Inc.

Other products or brandnames mentioned herein are trademarks or registered trademarks of their respective holders.