

---

## C8051T60X DEVELOPMENT KIT USER'S GUIDE

---

### 1. Kit Contents

The C8051T60x Development Kit contains the following items:

- C8051T600 Main Board
- C8051T600 SOIC Socket Daughter Board for programming SOIC devices
- C8051T600 Emulation Daughter Board with C8051F300 installed
- (5) C8051T600-GS Samples
- C8051T60x Development Kit Quick-Start Guide
- Silicon Laboratories IDE and Product Information CD-ROM. CD content includes:
  - Silicon Laboratories Integrated Development Environment (IDE)
  - Evaluation Version of Keil 8051 Development Tools (macro assembler, linker, 'C' Compiler)
  - Source code examples and register definition files
  - Documentation
- AC to DC Universal Power Adapter
- USB Cable
- RS-232 Cable

Also available for purchase separately is a C8051T600 QFN Socket Daughter Board for programming QFN devices.

### 2. About the Daughter Boards

The C8051T60x Development Kit includes two daughter boards: an Emulation Daughter Board (EDB) and an SOIC Socket Daughter Board (SOIC-DB). The EDB has a C8051F300 device installed, which is a Flash-based device that can be used for the majority of C8051T60x code development. The SOIC-DB and the QFN-DB (available separately) are intended to allow both the programming of multiple C8051T60x devices, as well as system-level debugging of these devices. Once a C8051T60x device has been programmed, it cannot be erased, so it is advisable to use the C8051F300 for the majority of code development. Refer to Application Note "AN280: Differences Between the C8051F300 and the C8051T60x Device Family" for more details on how the C8051F300 can be used to develop code for the C8051T60x device family.

### 3. Hardware Setup

Refer to Figure 1 for a diagram of the hardware configuration.

1. Attach the desired daughter board to the main board at connectors P1 and P2.
2. If using the SOIC Socket daughter board or the QFN Socket daughter board, place the device to be programmed into the socket.
3. Connect the AC to DC Power Adapter to connector P3 on the main board.
4. Connect the main board to a PC running the Silicon Laboratories IDE using the USB Cable.

#### Notes:

- Use the **Reset** icon in the IDE to reset the target when connected during a debug session.
- Remove power from the main board and remove the USB cable before removing a daughter board from the main board. Connecting or disconnecting a daughter board when the power adapter or USB cable are connected can damage the main board, the daughter board, or the socketed device.
- Likewise, remove power from the main board and remove the USB cable before removing a C8051T60x device from the socket. Inserting or removing a device from the socket when the power adapter or USB cable are connected can damage the main board, the daughter board, or the socketed device.

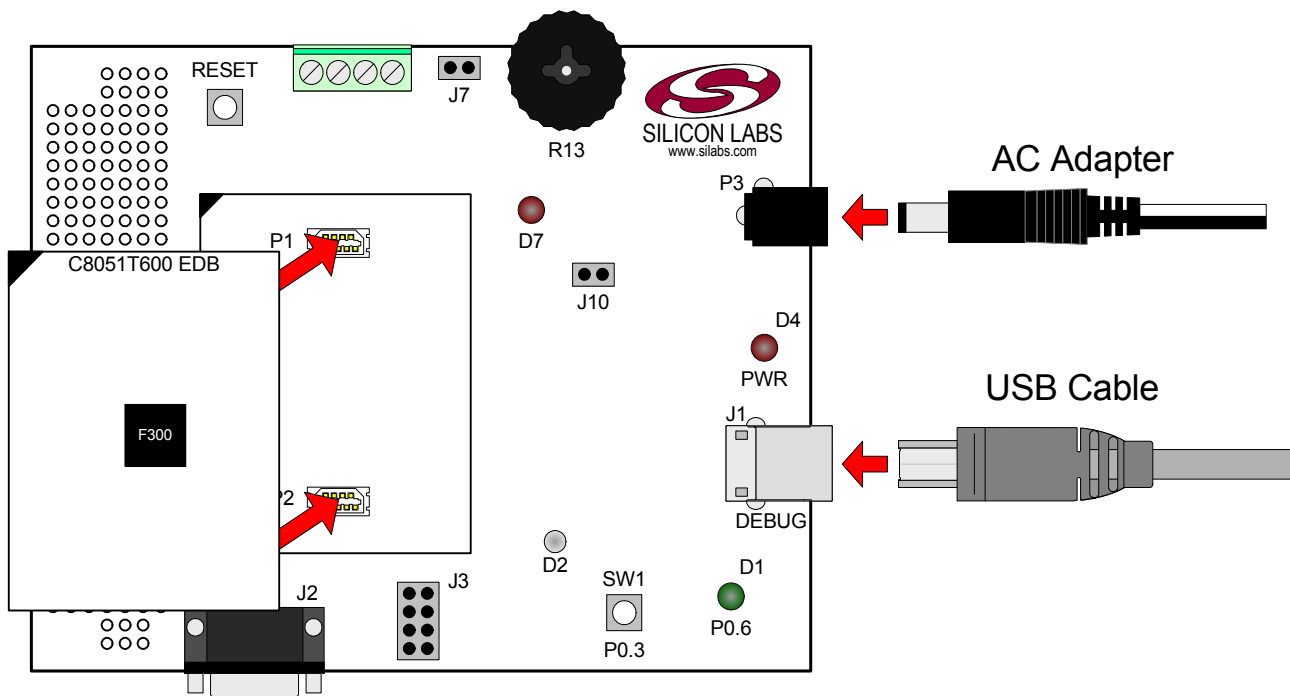


Figure 1. Hardware Setup (Emulation Daughter Board Shown)

## 4. 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 *ReleaseNotes.txt* file on the CD-ROM for the latest information regarding the CD contents.

## 5. 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 an evaluation version of the Keil Software A51 macro assembler, BL51 linker and C51 'C' compiler. These tools can be used from within the Silicon Laboratories IDE.

### 5.1. System Requirements

The Silicon Laboratories IDE requirements:

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

### 5.2. 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 "*SiLabs\MCU\hlp*" directory (A51.pdf).

## 5.3. 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\help*" directory (C51.pdf).

## 5.4. 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 Application Note "**AN104: Integrating Keil 8051 Tools Into the Silicon Labs IDE**" in the "*SiLabsMCU\Documentation\Appnotes*" 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.)

### 5.4.1. 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.

## 5.4.2. 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 “USB Debug Adapter” option. The USB Debug circuitry is integrated onto the C8051T60x main board. Next, the correct “Debug Interface” must be selected. C8051T60x Family devices and the C8051F300 all use Silicon Labs “C2” 2-wire 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.

**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.

5. 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**.

## 6. Example Source Code

Example source code and register definition files are provided in the “*SiLabs\MCU\Examples\C8051T60x*” directory during IDE installation. These files may be used as a template for code development.

### 6.1. Register Definition Files

Register definition files *C8051T600.inc*, *C8051T600\_defs.h* and *compiler\_defs.h* define all SFR registers and bit-addressable control/status bits. They are installed into the “*SiLabs\MCU\Examples\C8051T60x*” directory during IDE installation. The register and bit names are identical to those used in the C8051T60x datasheet. The 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.

### 6.2. Blinking LED Example

The example source files *blink.asm* and *blink.c* show examples of several basic C8051T60x 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 these programs flash the green LED on the C8051T600 main board about ten times a second using the interrupt handler with a timer.

## 7. Development Boards

The C8051T60x Development Kit includes a main board which interfaces to various daughter boards. The "C8051T600 Emulation Daughter Board" contains a C8051F300 device to be used for preliminary software development. The socketed C8051T600 daughter boards allow programming and evaluation of the actual C8051T60x family of devices. Numerous input/output (I/O) connections are provided on the main board to facilitate prototyping. Refer to Figure 2 for the locations of the various connectors on the main board. Figure 3, Figure 4 and Figure 5 show the available daughter boards that can be used with the C8051T600 main board.

- P1, P2 Daughter board connection (Daughter boards are plugged into the P1 and P2 connectors for evaluation and programming)
- P3 Power connector (accepts input from 7.5 to 15 VDC unregulated power adapter)
- J1 USB Debug connector
- J2 RS-232 connector
- J3 LED/Switch/RS-232 connection block
- J6 Analog I/O terminal block
- J7 Connects potentiometer to P0.1
- J8 Direct device pin connection block
- J10 VPP supply connection (used when programming OTP devices)

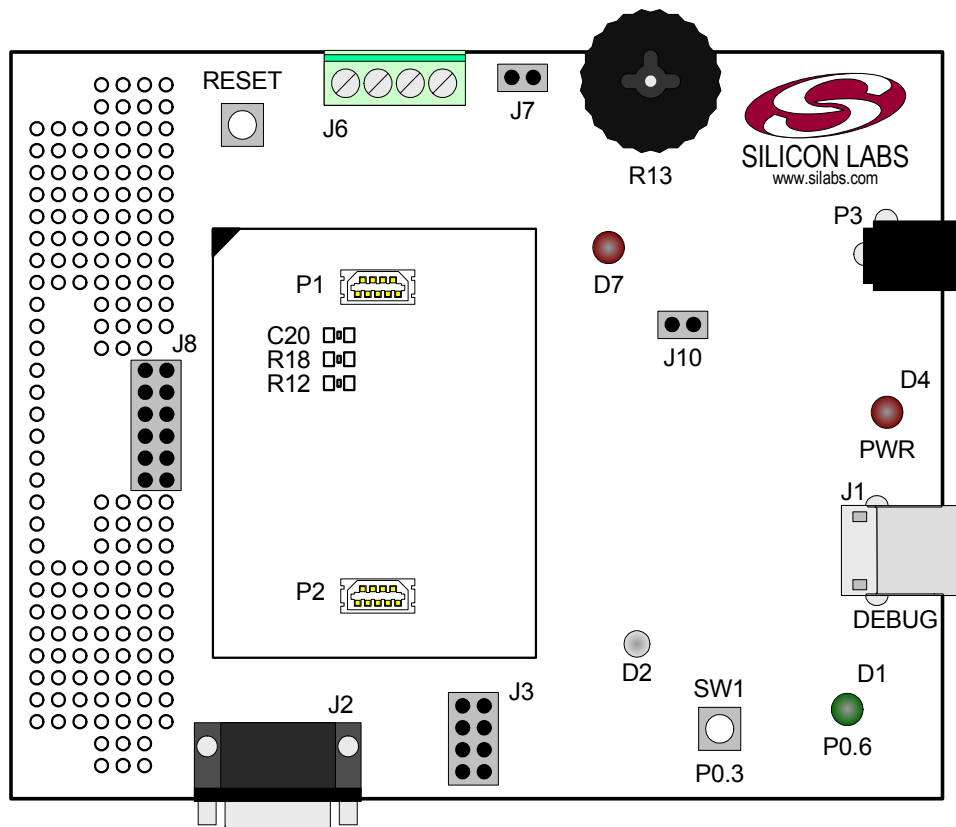
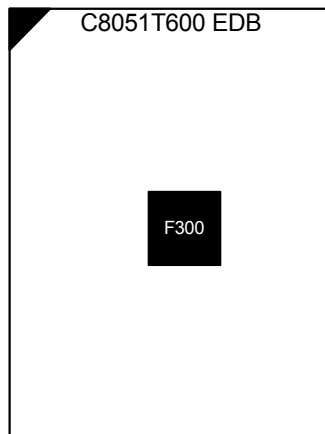
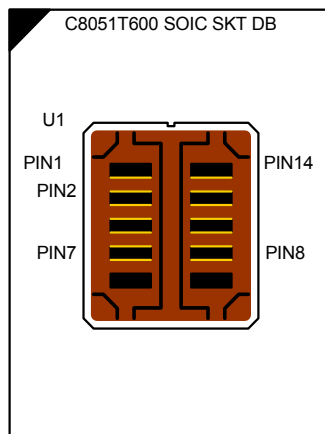


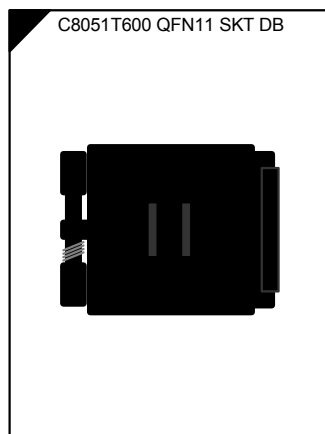
Figure 2. C8051T600 Main Board (Included in Kit)



**Figure 3. C8051T600 Evaluation Daughter Board (Included in Kit)**



**Figure 4. C8051T600 SOIC Socket Daughter Board (Included in Kit)**



**Figure 5. C8051T600 QFN Socket Daughter Board (Available Separately)**

## 7.1. System Clock Sources

The C8051T60x devices feature a calibrated internal oscillator which is enabled as the system clock source on reset. After reset, the internal oscillator operates at a frequency of 24.5 MHz ( $\pm 2\%$ ) 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 C8051T60x device at a frequency not available with the internal oscillator, an external oscillator source may be used. Refer to the C8051T60x datasheet for more information on configuring the system clock source.

The main board is designed to facilitate the use of external clock sources. To use an external CMOS clock source, the clock can simply be applied to P0.3. To implement the RC mode option, placeholders for an 0805-packaged capacitor (C20) and resistor (R18) are supplied on the board. The C (capacitor) clock option can be implemented by using only the capacitor placeholder (C20). To reduce the amount of stray capacitance on the pin, which could affect the frequency in either RC or C mode, resistor R12 can also be removed from the board when using C or RC mode. Refer to the C8051T60x datasheet for more information on the use of external oscillators.

## 7.2. Switches, LEDs and Potentiometer

Two switches are provided on the main board. Switch RESET is connected to the RESET pin of the C8051T60x. Pressing RESET puts the device into its hardware-reset state. Switch P0.3 can be connected to the C8051T60x's general purpose I/O (GPIO) pin through header J3. Pressing Switch P0.3 generates a logic low signal on the port pin. Remove the shorting block from the J3 header to disconnect Switch P0.3 from the port pin.

Four LEDs are also provided on the target board. The red LED labeled PWR (D4) is used to indicate a power connection to the target board. The red/green LED labeled D2 is the run/stop light for the debug circuitry. The red LED labeled D7 indicates when the VPP programming voltage is being applied to the device (see Section 7.7 for additional details). Finally, the green LED labeled P0.6 (D1) can be connected to the C8051T60x's GPIO pin through header J3. Remove the shorting block from the header to disconnect the LED from the port pin.

Also included on the C8051T60x main board is a 10 k $\Omega$  thumb-wheel rotary potentiometer, reference number R13. The potentiometer can be connected to the C8051T60x's P0.1 pin through the J7 header. Remove the shorting block from the header to disconnect the potentiometer from the port pin.

See Table 1 for the port pins and headers corresponding to the switches, LEDs, and potentiometer.

**Table 1. Target Board I/O Descriptions**

Description	I/O	Header
SW1	P0.3	J3[5–6]
SW2	RESET	none
Red LED	PWR	none
Green LED	P0.6	J3[7–8]
Red/Green LED	none	none
Red LED	VPP	J10
Potentiometer	P0.1	J7

## 7.3. USB Debug Interface (DEBUG / J1)

A Universal Serial Bus (USB) connector (J1) is included to provide the on-board debug and programming interface. The debug/programming interface is powered through the USB connector, but requires power to the device via the power connector P3 to be fully functional.

# C8051T60x-DK

---

## 7.4. PORT I/O Connector (J8)

Each of the C8051T60x's I/O pins, as well as VPP, +3VD, GND, and /RST are routed to header J8. This header can be used to easily connect to any signal on the device. Table 2 defines the pins for header J8.

**Table 2. J8 Pin Descriptions**

Pin #	Description
1	+3VD (+3.3 VDC)
2	VPP
3	P0.0
4	P0.1
5	P0.2
6	P0.3
7	P0.4
8	P0.5
9	P0.6
10	P0.7
11	GND
12	/RST

## 7.5. Serial Interface (J2)

A RS232 transceiver circuit and DB-9 (J2) connector are provided on the main board to facilitate serial connections to UART0 on the C8051T60x. The TX and RX signals of UART0 may be connected to the RS-232 transceiver by installing shorting blocks on header J3. The RTS and CTS lines do not have a direct connection available to the C8051T60x, due to limited pin resources on the device. However, these two signals have also been routed through the transceiver, and are available for wiring on-board at test points near the DB-9 connector. The transceiver shifts the UART signals to RS-232 levels, and connects to the appropriate pins on the DB-9 connector.

J3[1–2] - Install shorting block to connect UART0 RX (P0.5) to transceiver.

J3[3–4] - Install shorting block to connect UART0 TX (P0.4) to transceiver.

## 7.6. Analog I/O (J6)

Two of the C8051T60x target device's port pins are connected to the J6 terminal block. Refer to Table 3 for the J6 terminal block connections.

**Table 3. J6 Terminal Block Pin Descriptions**

Pin #	Description
1	GND (Ground)
2	P0.1/AIN1
3	P0.0/VREF
4	VDD



## 7.7. VPP Connection (J10)

The C8051T60x devices require a special 6.5 V programming voltage applied to the VPP pin during device programming. The VPP pin on these devices is shared with P0.2. During programming, the VPP voltage is automatically enabled when needed. Header J10 is provided to allow the user to disconnect the programming circuitry from the VPP/P0.2 pin to avoid interfering with the normal application operation of P0.2. When programming the device, J10 should be shorted with a shorting block. When running normal application code, J10 can be removed. Note that the C8051T600 Emulation Daughter Board does not connect the main board's VPP and P0.2 signals, so removing the shorting block is not necessary when using the Emulation Daughter Board.

## 7.8. Using Alternate Supplies with the C8051T60x Development Kit

For most evaluation purposes, the on-board 3.3 V supply regulator is sufficient to be used as a VDD power supply. However, in applications where a different supply voltage is desired (for example, 1.8 V), an external supply voltage can be applied to the board at the analog connector (J6). Depending on the supply and the desired functionality of the board, it may be necessary to cut the VDD supply voltage trace on the back of the board behind header J9 (not populated). The following should be considered when supplying a different voltage to the device:

1. When programming a C8051T60x device: The on-board 3.3 V regulator should be used for VDD.
2. If a VDD supply voltage between 2.7 and 3.6 V is desired: The ac to dc power adapter should be disconnected from P3, and the desired supply voltage can be applied directly to the GND and VDD terminals of terminal block J6. No board modification is necessary.
3. If a VDD supply voltage between 1.8 and 2.7 V is desired and RS-232 functionality is not required: The ac to dc power adapter should be disconnected from P3, and the desired supply voltage can be applied directly to the GND and VDD terminals of terminal block J6. No board modification is necessary.
4. If a VDD supply voltage between 1.8 and 2.7 V is desired and RS-232 functionality is required: The RS-232 transceiver is not specified to operate below 2.7 V. In this case, the trace under header J9 should be cut. Then, the desired VDD supply voltage can be applied directly to the GND and VDD terminals of terminal block J6, and the ac to dc power adapter can be connected to P3, to provide 3.3 V to the RS-232 transceiver. In this case, it may also be necessary to configure the TX pin on the device for open-drain mode, and add a pull-up resistor to the supply of the transceiver.

## 8. Schematics

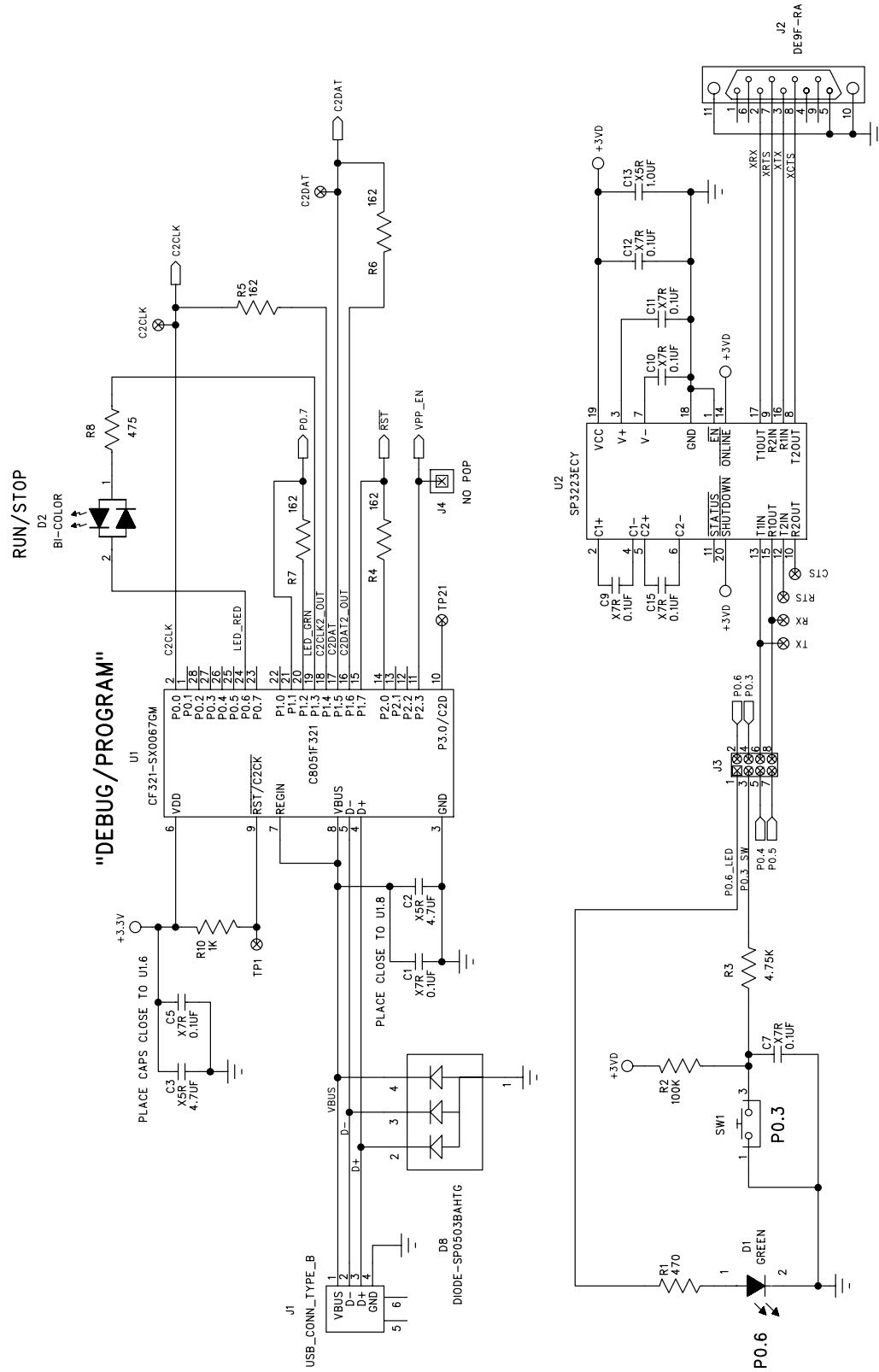


Figure 6. C8051T600 Main Board Schematic (page 1/2)

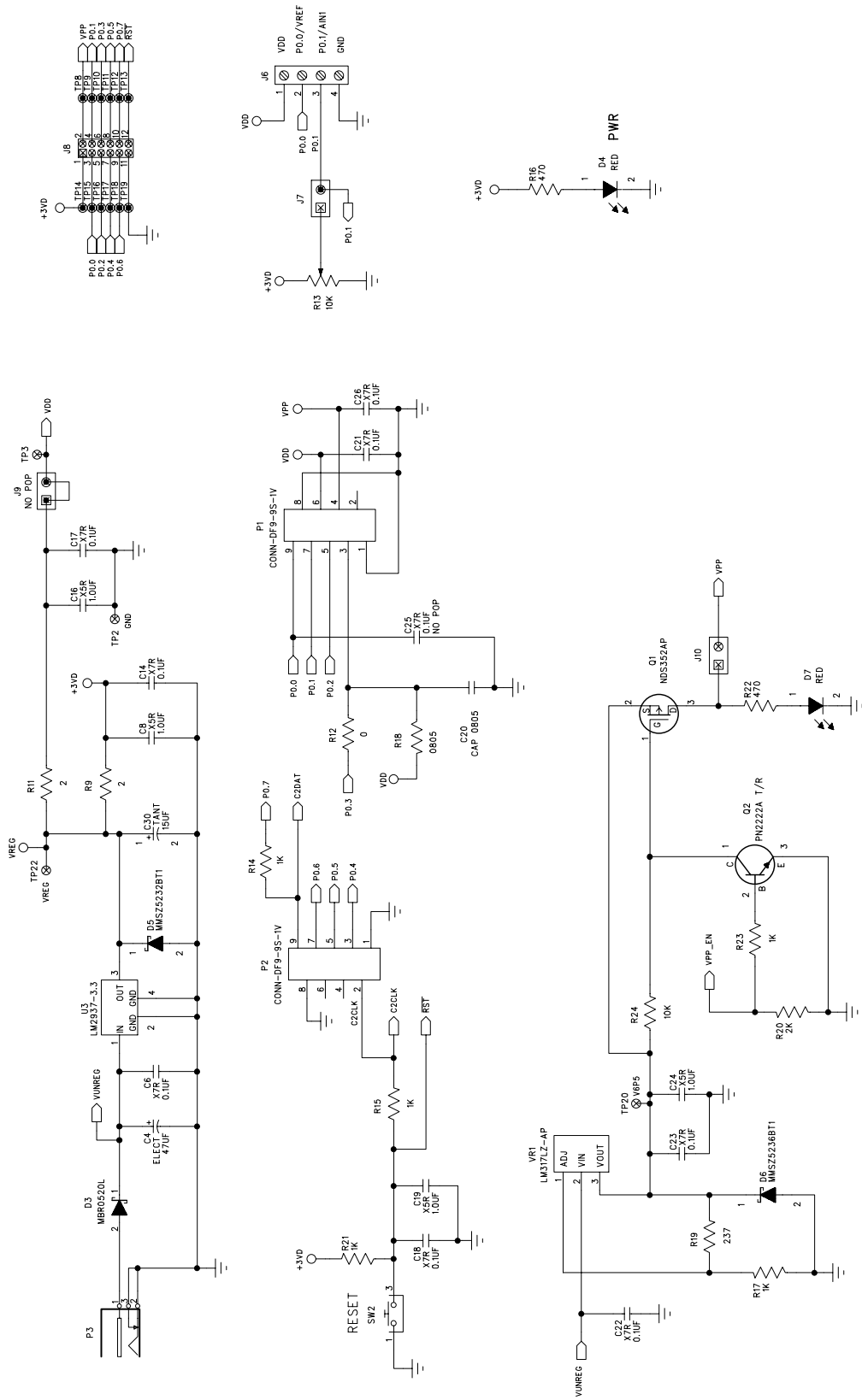


Figure 7. C8051T600 Main Board Schematic (page 2/2)

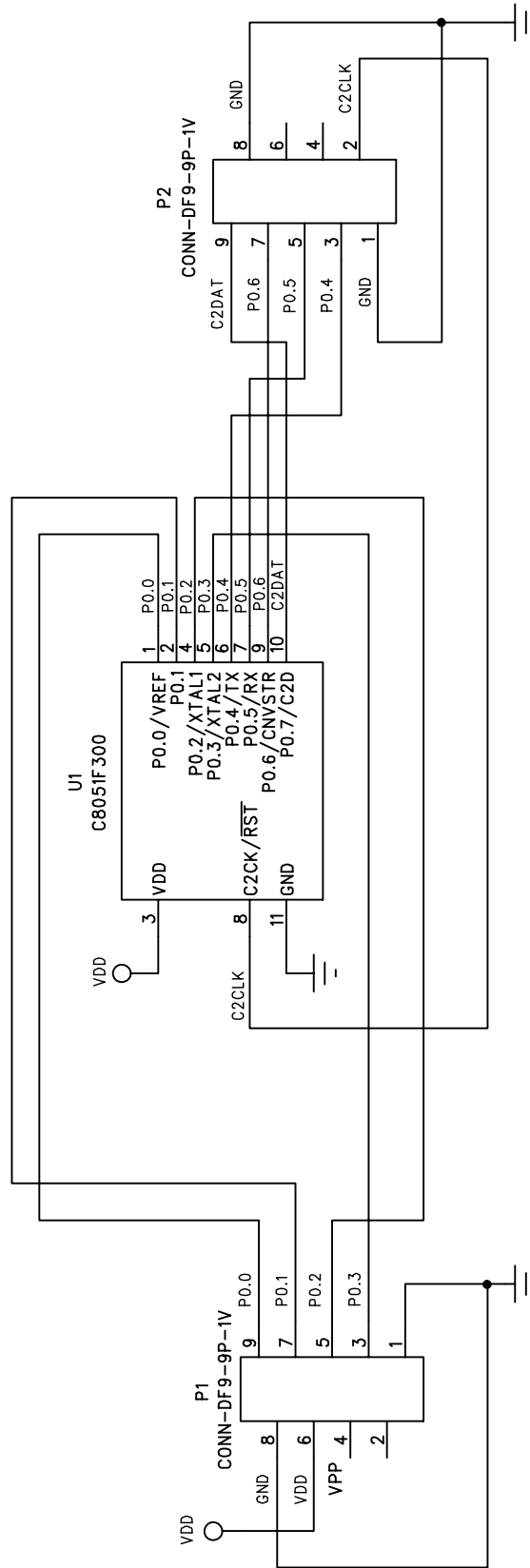


Figure 8. C8051T600 Emulation Daughter Board Schematic

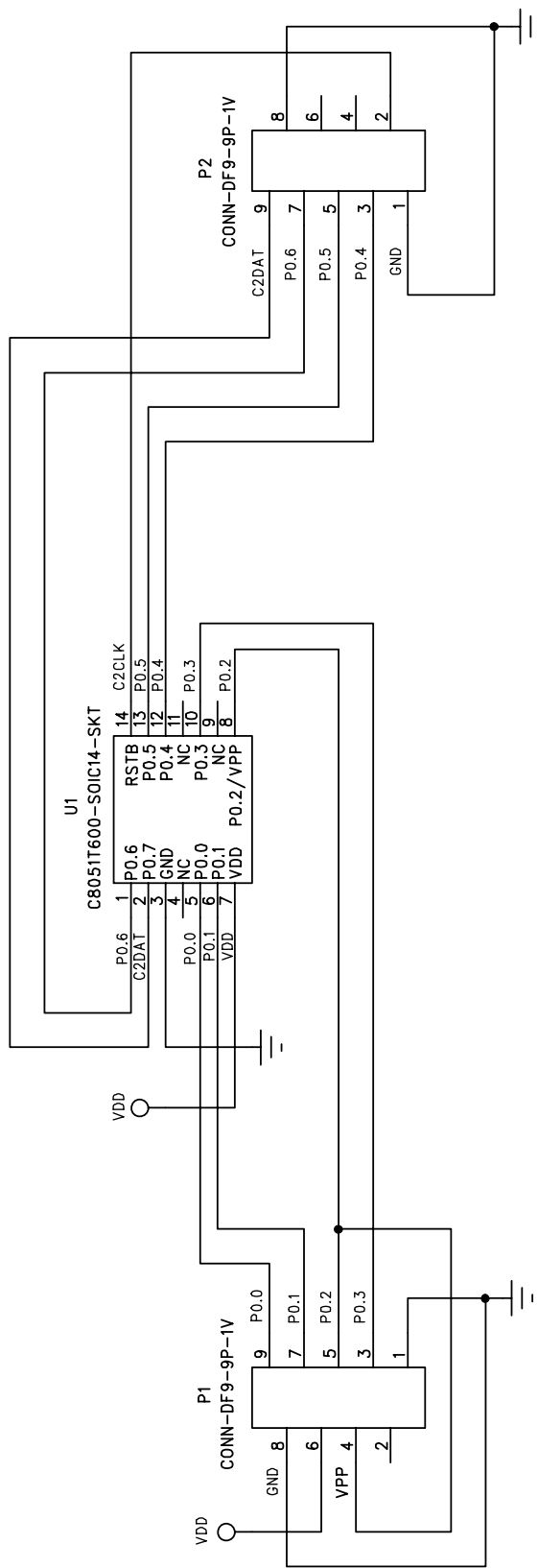


Figure 9. C8051T600 SOIC Socket Daughter Board Schematic

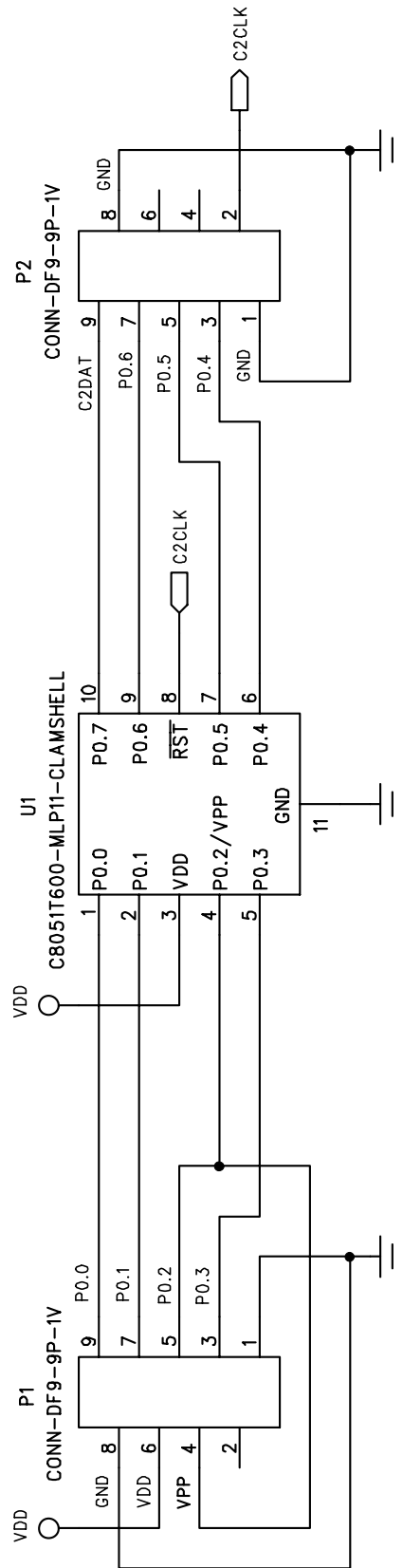


Figure 10. C8051T600 QFN Socket Daughter Board Schematic

**NOTES:**

# C8051T60x-DK

---

## 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  
Email: [MCUinfo@silabs.com](mailto:MCUinfo@silabs.com)  
Internet: [www.silabs.com](http://www.silabs.com)

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.