
C8051F560 DEVELOPMENT KIT USER'S GUIDE

1. Relevant Devices

The C8051F560 Development Kit is intended as a development platform for the microcontrollers in the C8051F55x/56x/57x MCU family. The members of this MCU family are C8051F550, C8051F551, C8051F552, C8051F553, C8051F554, C8051F555, C8051F556, C8051F557, C8051F560, C8051F561, C8051F562, C8051F563, C8051F564, C8051F565, C8051F566, C8051F567, C8051F568, C8051F569, C8051F570, C8051F571, C8051F572, C8051F573, C8051F574, and C8051F575.

- The target board included in this kit is provided with a pre-soldered **C8051F568** MCU (QFN40 package) and a **C8051F550** MCU (QFN24 package).
- Code developed on the C8051F568 can be easily ported to the other members of this MCU family.
- Refer to the C8051F55x/56x/57x data sheet for the differences between the members of this MCU family.

2. Kit Contents

The C8051F560 Development Kit contains the following items:

- C8051F560 Target Board
- C8051Fxxx Development Kit Quick-Start Guide
- Silicon Labs IDE and Product Information CD-ROM. CD content includes:
 - Silicon Labs Integrated Development Environment (IDE)
 - Evaluation assembler, compiler, and linker tools
 - Source code examples and register definition files
 - Documentation
- Optional Third Party Tools CD
- AC to DC Power Adapter
- USB Debug Adapter
- Two USB Cables

3. Hardware Setup

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

1. Connect the USB Debug Adapter to the DEBUG A 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. Verify that shorting blocks are installed on the target board as shown in Figure 4 on page 9.
4. Connect the other end of the USB cable to a USB Port on the PC.
5. Connect the ac/dc power adapter to power jack P4 on the target board.

Notes:

- Use the **Reset** icon in the IDE to reset the target when connected during a debug session.
- Remove power from the target board and the USB Debug Adapter before connecting or disconnecting 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.

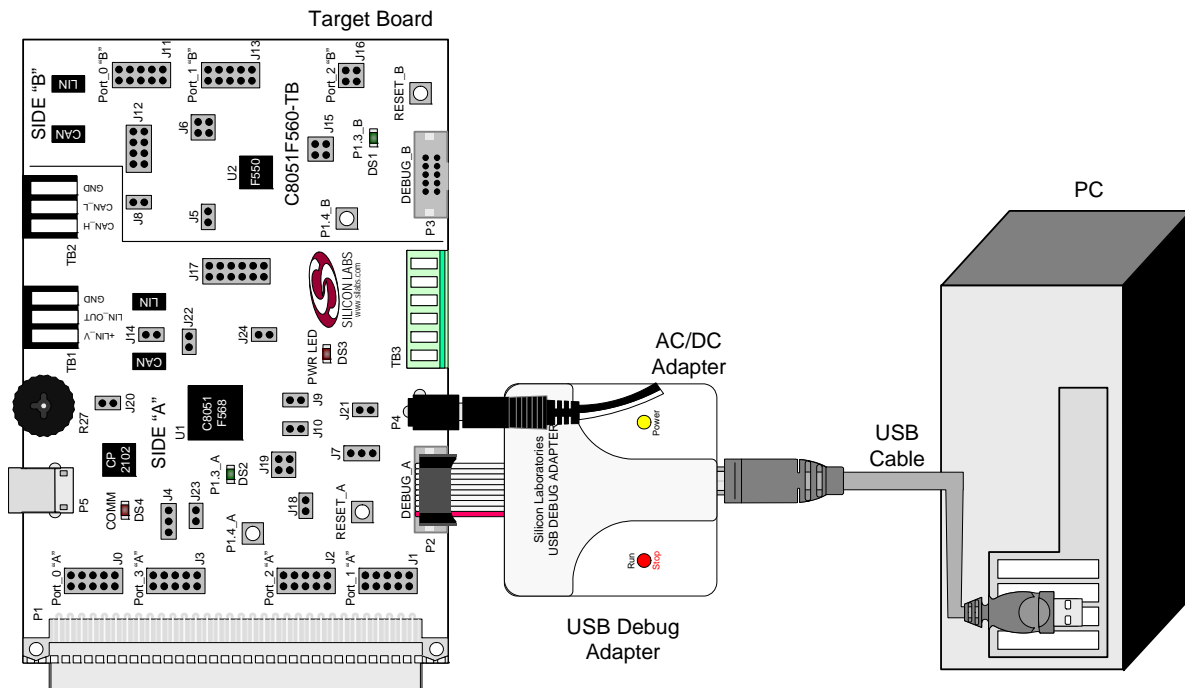


Figure 1. Hardware Setup using a USB Debug Adapter

4. Software Setup

The included CD-ROM contains the Silicon Labs Integrated Development Environment (IDE), evaluation 8051 tools, optional software utilities, 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 known problems and restrictions.

4.1. System Requirements

The following are the system requirements necessary to run the debug and programming tools:

- Pentium-class host PC running Microsoft Windows 2000 or newer.
- One available USB port.

4.2. Development Tools Installation

To install the IDE, utilities, and code examples, perform the following steps:

1. Click on the "Install Development Tools" button on the installation utility's startup screen.
2. In the Kit Selection box that appears, choose the C8051F560DK development kit from the list of options.
3. In the next screen, choose "Components to be Installed". The programs necessary to download and debug on the MCU are the **Silicon Labs IDE** and the **Keil 8051 Evaluation Toolset**. The **CP210x Drivers** are necessary to use the UART capabilities of the target board. See Section 4.3. for more information about installing the CP210x drivers.
4. Installers selected in Step 3 will execute in sequence, prompting the user as they install programs, documentation, and drivers.

4.3. CP210x USB to UART VCP Driver Installation

The C8051F560 Target Board includes a Silicon Labs CP2102 USB-to-UART Bridge Controller. Device drivers for the CP2102 need to be installed before PC software such as HyperTerminal can communicate with the target board over the USB connection. If the "Install CP210x Drivers" option is selected during installation, a driver "unpacker" utility will launch.

1. Follow the steps to copy the driver files to the desired location. The default directory is *C:\SiLabs\MCU\CP210x*.
2. The final window will give an option to install the driver on the target system. Select the "Launch the CP210x VCP Driver Installer" option if you are ready to install the driver.
3. If selected, the driver installer will now launch, providing an option to specify the driver installation location. After pressing the "Install" button, the installer will search your system for copies of previously installed CP210x Virtual COM Port drivers. It will let you know when your system is up to date. The driver files included in this installation have been certified by Microsoft.
4. If the "Launch the CP210x VCP Driver Installer" option was not selected in step 3, the installer can be found in the location specified in step 2, by default *C:\SiLabs\MCU\CP210x\Windows_2K_XP_S2K3_Vista*. At this location, run *CP210xVCPInstaller.exe*.
5. To complete the installation process, connect the included USB cable between the host computer and the USB connector (P5) on the C8051F560 Target Board. Windows will automatically finish the driver installation. Information windows will pop up from the taskbar to show the installation progress.
6. If needed, the driver files can be uninstalled by selecting "Silicon Labs CP210x USB to UART Bridge Driver Removal" option in the "Add or Remove Programs" window.

5. Software Overview

The following software is necessary to build a project, download code to, and communicate with the target microcontroller.

- Silicon Labs Integrated Development Environment (IDE)
- 8051 Toolset

Other useful software that is provided on the development kit CD and the Silicon Labs Downloads (www.silabs.com/mcudownloads) website includes:

- Configuration Wizard 2
- Keil μ Vision2 and μ Vision3 Drivers
- MCU Production Programmer and Flash Programming Utilities

5.1. Silicon Labs IDE

The Silicon Labs IDE integrates a source-code editor, source-level debugger and in-system programmer. The use of third-party compilers, assemblers, and linkers is also supported. This development kit includes evaluation versions of commercial 'C' compilers and assemblers which can be used from within the Silicon Labs IDE.

5.1.1. Third Party Toolsets

The Silicon Labs IDE has native support for many 8051 compilers. Natively-supported tools are as follows:

- Keil
- IAR
- Raisonance
- Tasking
- Hi-Tech
- SDCC

Specific instructions for integrating each of the supported tools can be found in the application notes section of the CD, or on the Silicon Labs web site (<http://www.silabs.com>).

5.1.2. Getting Started With the Silicon Labs IDE

The following sections discuss how to open an example project in the IDE, build the source code, and download it to the target device.

5.1.2.1. Running the F560_Blinky example program

The **F560_Blinky** example program blinks an LED on the target board.

1. Open the Silicon Labs IDE from the Start menu.
2. Select **Project**→**Open Project** to open an existing project.
3. Browse to the *C:\SiLabs\MCU\Examples\C8051F55x_56x_57x\Blinky* directory (default) and select the *F560_Blinky_C.wsp* project file and click **Open**.
4. Once the project is open, 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.
5. 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. Next, the correct “Debug Interface” must be selected. C8051F55x/56x/57x devices use Silicon Labs “C2” 2-wire debug interface. Once all the selections are made, click the **OK** button to close the window.
6. Click the **Connect** button in the toolbar or select **Debug**→**Connect** from the menu to connect to the device.
7. 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.
8. Click on the **Go** button (green circle) in the toolbar or by selecting **Debug**→**Go** from the menu to start running the firmware. The LED on the target board will start blinking.

5.1.2.2. Creating a New Project

Use the following steps to create a new projects. Once steps 1-5 in this section are complete, continue at Step 3 in Section 5.1.2.1.

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. 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**.

5.2. Configuration Wizard 2

The Configuration Wizard 2 is a code generation tool for all of the Silicon Labs devices. Code is generated through the use of dialog boxes for each of the device's peripherals.

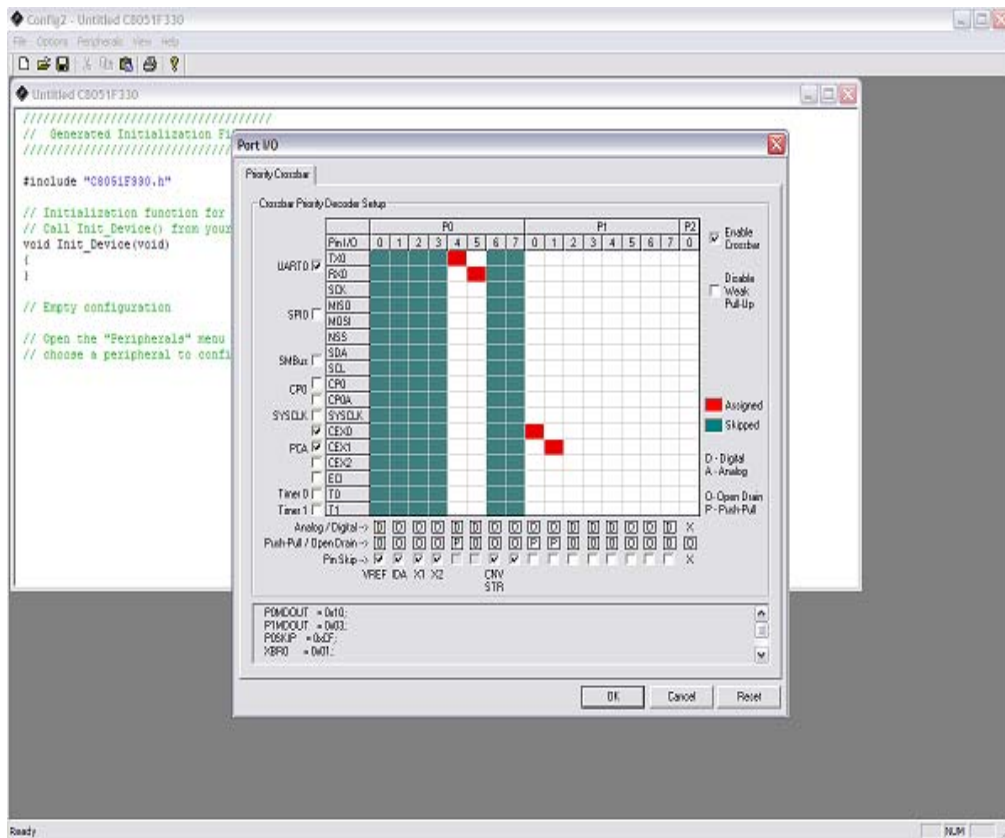


Figure 2. Configuration Wizard 2 Utility

The Configuration Wizard 2 utility helps accelerate development by automatically generating initialization source code to configure and enable the on-chip resources needed by most design projects. In just a few steps, the wizard creates complete startup code for a specific Silicon Labs MCU. The program is configurable to provide the output in C or assembly. For more information, please refer to the Configuration Wizard 2 help available under the **Help** menu in Configuration Wizard 2.

For more information, please refer to the Configuration Wizard 2 documentation. The documentation and software are available from the Downloads webpage (www.silabs.com/mcudownloads).

5.3. Keil μ Vision2 and μ Vision3 Silicon Labs Drivers

As an alternative to the Silicon Labs IDE, the μ Vision debug driver allows the Keil μ Vision IDE to communicate with Silicon Labs on-chip debug logic. In-system Flash memory programming integrated into the driver allows for rapidly updating target code. The μ Vision IDE can be used to start and stop program execution, set breakpoints, check variables, inspect and modify memory contents, and single-step through programs running on the actual target hardware.

For more information, please refer to the μ Vision driver documentation. The documentation and software are available from the Downloads webpage (www.silabs.com/mcudownloads).

5.4. Programming Utilities

The Silicon Labs IDE is the primary tool for downloading firmware to the MCU during development. There are two software programming tools that are intended for use during prototyping or in the field: the MCU Production Programmer and the Flash Programming Utilities. The MCU Production Programmer is installed with the IDE to the directory C:\Silabs\MCU\Utilities\Production Programmer\ (default). The Flash Programming Utilities can be optionally installed from the CD and is installed to C:\Silabs\MCU\Utilities\FLASH Programming\ (default).

6. Example Source Code

Example source code and register definition files are provided in the "SiLabs\MCU\Examples\C8051F55x_56x_57x\" directory during IDE installation. These files may be used as a template for code development. The comments in each example file indicate which development tool chains were used when testing. Example applications include a blinking LED example which configures the green LED on the target board to blink at a fixed rate. Also included are examples for each of peripherals of the MCU, such as the UART.

6.1. Register Definition Files

Register definition files *C8051F560.inc*, *C8051F560_defs.h* and *compiler_defs.h* define all SFR registers and bit-addressable control/status bits. These files are installed into the "SiLabs\MCU\Examples\C8051F55x_56x_57x\Header_Files\" directory during IDE installation. The register and bit names are identical to those used in the C8051F55x/56x/57x data sheet.

6.2. Blinking LED Example

The example source files *F560_Blinky.asm* and *F560_Blinky.c* installed in the default directory "SiLabs\MCU\Examples\C8051F55x_56x_57x\Blinky\" show examples of several basic C8051F560 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 pin. When compiled/assembled and linked, this program flashes the green LED on the C8051F560 Target Board about five times a second using the interrupt handler with a C8051F560 timer.

7. Target Board

The C8051F560 Development Kit includes a target board with a **C8051F568** (Side A) and **C8051F550** (Side B) 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 3 for the locations of the various I/O connectors. Figure 4 on page 9 shows the factory default shorting block positions. A summary of the signal names and headers is provided in Table 12 on page 17.

J7	Header to choose between +5V from Debug Adapter (P2) or +5V from on-board regulator (U6)
J21	Connect V_HIGH node from TB1 LIN header to +5V regulator input for board power
P4	Power connector (accepts input from 7 to 15 VDC unregulated power adapter)
P5	USB connector (connects to PC for serial communication)
TB1	Shared LIN Connector for Side A and B MCUs for external nodes
TB2	Shared CAN Connector for Side A and B MCUs for external nodes
J0-J3	Side A: Port 0 through Port 3 headers
J4	Side A: Connects +5V net or +3.3V net to VIO of the MCU
J9, J10	Side A: External crystal enable connectors
J14	Side A: CAN Transceiver (U3) power connector
J17	Side A: Connects MCU to three separate transceivers (UART(U5), CAN(U3), and LIN(T1))
J18	Side A: Connects VIO to VIO_A_SRC which powers the R27 potentiometer, the RST_A pin pull-up, and P1.4_A Switch pull-up.
J19	Side A: Connects P1.3_A LED and P1.4_A Switch to MCU port pins
J20	Side A: Connects R27 potentiometer to port pin 1.2
J22	Side A: Connects decoupling capacitors C28 and C29 for MCU VREF (P0.0)
J23	Side A: Connects VIO_A power to External Memory Interface Latch (U8)
J24	Side A: Connects +5V net to VIO and VREGIN of the MCU
P1	Side A: 96-pin female connector
P2	Side A: DEBUG connector for Debug Adapter interface
TB3	Side A: Power supply terminal block
J5	Side B: Connects decoupling capacitors C42 and C43 for MCU VREF (P0.0)
J6	Side B: Connects +5V net to VIO and VREGIN of the MCU
J8	Side B: CAN Transceiver (U4) power connector
J11	Side B: Port 0 header
J12	Side B: Connects MCU to two separate transceivers (CAN (U4) and LIN (T2))
J13	Side B: Port 1 header
J15	Side B: Connects P1.3_B LED and P1.4_B Switch to MCU port pins
J16	Side B: Port 2 header
P3	Side B: DEBUG connector for Debug Adapter interface

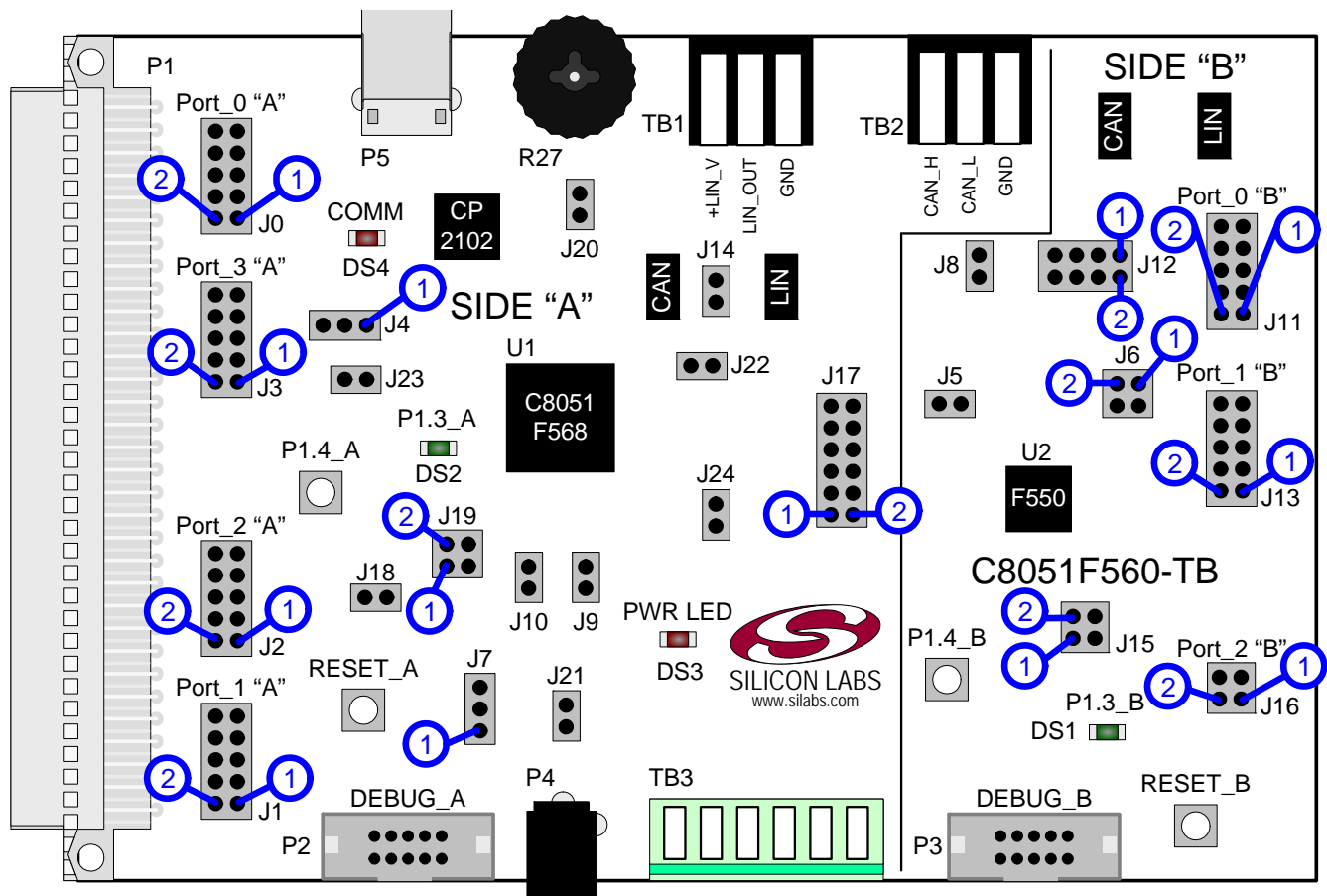


Figure 3. C8051F560 Target Board with Pin Numbers

7.2. Target Board Power Options and Current Measurement (J4, J6, J7, J24, P4, TB1)

The MCUs on the C8051F560 Target Board are powered from a +5 V net. The +5 V net is connected to the headers J4 and J24 (Side A) and J6 (Side B). Shorting blocks can be put on each header to connect the 5V net to the VREGIN and VIO pins on the two MCUs. With the shorting blocks removed, a source meter can be used across the headers to measure the current consumption of the MCU.

The +5 V net on the target board has three possible sources:

1. 12V dc power using the ac to dc power adapter (P4)
2. 5V dc USB VBUS power from PC via the USB Debug Adapter (DEBUG_A)
3. 12V dc power from the LIN external header (TB1)

7.2.1. Using the AC to DC Power Adapter as the Target Board Power Source (P4, J7)

The default configuration of the target board uses the ac to dc power adapter as the source. The 12 V from the adapter is regulated to +5 V using an LDO regulator (U6). The output of the regulator is connected to the +5 V net of the target board through the J7 header. A shorting block should be installed on pins J7[2-3] for this purpose. The +5V net powers the MCUs directly.

7.2.2. Using the USB Debug Adapter as the Target Board Power Source (J7)

The target board can use +5 V provided by the USB Debug Adapter. To enable this source, a shorting block should be installed on pins J7[1-2]. With this shorting block, the output of the LDO regulator (U6) is disconnected from the +5 V net of the target board, and the SER_PWR node is connected to +5 V.

Note: The USB Debug Adapter does not provide the necessary peak power for the CAN transceivers to operate. One of the 12 V dc sources is recommended for CAN transceiver operation.

7.2.3. Using an External +12V LIN Source as the Target Board Power Source (J7, TB1)

The two 12V power sources (LIN and ac to dc power adapter) are ORed together using reverse-biased diodes (Z1 and Z2) and connected to the input of the LDO regulator (U6). The output of the regulator is connected to the +5 V net of the target board through the J7 header. A shorting block should be installed on pins J7[2-3] for this purpose. The +5V net powers the MCUs directly.

7.3. System Clock Sources (J9, J10)

7.3.1. Internal Oscillators

The C8051F568 and C8051F550 devices installed on the target board feature a factory-calibrated, programmable high-frequency internal oscillator (24 MHz base frequency, $\pm 0.5\%$), which is enabled as the system clock source on reset. After reset, the internal oscillator operates at a frequency of 187.5 kHz by default but may be configured by software to operate at other frequencies. The on-chip crystal is accurate for CAN and LIN master communications and in many applications an external oscillator is not required. However, if you wish to operate the C8051F568 device (Side A) at a frequency not available with the internal oscillator, an external crystal may be used. Refer to the C8051F55x/56x/57x data sheet for more information on configuring the system clock source.

7.3.2. External Oscillator Options

The target board is designed to facilitate the installation of an external crystal. Remove shorting blocks at headers J9 and J10 and install the crystal at the pads marked Y1. Install a 10 M Ω resistor at R2 and install capacitors at C6 and C7 using values appropriate for the crystal you select. If you wish to operate the external oscillator in capacitor or RC mode, options to install a capacitor or an RC network are also available on the target board. R2, R3, C6, and C7 are located on the back side of the board, near the Side A MCU. Populate C6 for capacitor mode, and populate R3 and C6 for RC mode. Refer to the C8051F55x/56x/57x data sheet for more information on the use of external oscillators.

7.4. Switches and LEDs (J15, J19)

Two push-button switches are provided on the target board for each MCU. Switch RESET_A is connected to the $\overline{\text{RST}}$ pin of the C8051F568. Switch RESET_B is connected to the $\overline{\text{RST}}$ pin of the C8051F550. Pressing RESET_A puts the C8051F568 device into its hardware-reset state, and similarly for RESET_B and the C8051F550 MCU. Switches P1.4_A and P1.4_B are connected to the MCU's general purpose I/O (GPIO) pins through headers. Pressing either one of these switches generates a logic low signal on the port pin. Remove the shorting block from the header to disconnect these switches from the port pins. See Table 1 for the port pins and headers corresponding to each switch.

Four LEDs are provided on the target board to serve as indicators. The red LED labeled PWR LED indicates presence of power to the target board. The second red LED labeled COMM indicates if the CP2102 USB-to-UART bridge is recognized by the PC. The green LED on Side A is labeled with a port pin name and is connected to a C8051F568 GPIO pin through a header. Remove the shorting block from the header to disconnect the LED from the port pin. Similarly, a second green LED on Side B is connected to the C8051F550 through another header. See Table 1 for the port pins and headers corresponding to each LED.

Table 1. Target Board I/O Descriptions

Description	I/O	Header(s)
RESET_A	Reset (Side A)	none
RESET_B	Reset (Side B)	none
P1.4_A Switch	P1.4 (Side A)	J19[1–2]
P1.4_B Switch	P1.4 (Side B)	J15[1–2]
P1.3_A LED	P1.3 (Side A)	J19[3–4]
P1.3_B LED	P1.3 (Side B)	J15[3–4]
Red LED (PWR)	Power	none
Red LED (COMM)	COMM Active	none

7.5. Target Board Debug Interfaces (P2 and P3)

The debug connectors P2 (DEBUG_A) and P3 (DEBUG_B) provide access to the debug (C2) pins of the C8051F568 and C8051F550. The debug connectors are used to connect the Serial Adapter or the USB Debug Adapter to the target board for in-circuit debugging and Flash programming. Table 2 shows the DEBUG pin definitions.

Table 2. DEBUG Connector Pin Descriptions

Side A - C8051F568		Side B - C8051F550	
Pin #	Description	Pin #	Description
1	Not Connected	1	Not Connected
2, 3, 9	GND (Ground)	2, 3, 9	GND (Ground)
4	P4.0_C2D_A	4	P2.1_C2D_B
5	$\overline{\text{RST_A}}$ (Reset)	5	$\overline{\text{RST_B}}$ (Reset)
6	P4.0_A	6	P2.1_B
7	$\overline{\text{RST/C2CK_A}}$	7	$\overline{\text{RST/C2CK_B}}$
8	Not Connected	8	Not Connected
10	USB Power (+5VDC from P2)	10	Not Connected

7.6. Serial Interface (P5, J17)

A USB-to-UART bridge circuit (U5) and USB connector (P5) are provided on the target board to facilitate serial connections to UART0 of the C8051F568 (Side A). The Silicon Labs CP2102 USB-to-UART bridge provides data connectivity between the C8051F568 and the PC via a USB port. The TX and RX signals of UART0 may be connected to the CP2102 by installing shorting blocks on header J17. The shorting block positions for connecting each of these signals to the CP2102 are listed in Table 3. To use this interface, the USB-to-UART device drivers should be installed as described in Section 4.3. "CP210x USB to UART VCP Driver Installation," on page 3.

Table 3. Serial Interface Header (J17) Description

Header Pins	UART0 Pin Description
J17[3–4]	UART_TX (P0.4_A)
J17[1–2]	UART_RX (P0.5_A)

7.7. CAN Interface and Network (J8, J12, J14, J17, TB2)

Both MCUs on the target board are connected to CAN transceivers (U3, U4) through headers. The port pins assigned to the CAN peripheral on each MCU are P0.6 (CAN_TX) and P0.7 (CAN_RX). The C8051F568 (Side A) is connected to U3 through the J17 header and the C8051F550 (Side B) is connected to U4 through the J12 header. The two CAN transceivers are connected to each other and form a CAN network. Other external devices can be connected to the CAN network through the TB2 interface. The shorting block positions for connecting the MCUs to the CAN transceivers are listed in Table 4. The pin connections for the external CAN devices are listed in Table 5. The CAN transceivers are powered by the +5VREG node and connected through J8 and J14 headers.

Table 4. CAN Interface Headers (J17 and J12) Description

Header Pins	CAN0 Pin Description
J17[5–6]	CAN_TX (P0.6_A)
J17[7–8]	CAN_RX (P0.7_A)
J12[7–8]	CAN_TX (P0.6_B)
J12[5–6]	CAN_RX (P0.7_B)

Table 5. TB2 External CAN Interface Header Description

Pin #	Pin Description
1	GND
2	CAN_L
3	CAN_H

7.8. LIN Interface and Network (J12, J17, J21, TB1)

Both MCUs on the target board are connected to LIN transceivers through headers. These headers assume that the MCU's crossbars are configured to put the LIN TX and RX pins on port pins P1.0 and P1.1 respectively. See the C8051F55x/56x/57x data sheet for crossbar configuration. The C8051F568 (Side A) is connected to the T1 transceiver through the J17 header and the C8051F550 (Side B) is connected to the T2 transceiver through the J12 header. The two LIN transceivers are connected to each other and form a LIN network. Other external devices can be connected to the LIN network through the TB1 interface. The TB1 interface also provides the option for connecting an external power source so that all LIN transceivers can use the same source voltage. This source voltage can also be used to power the target board. If an external voltage source is not provided, the LIN transceivers use the 12V provided through the P4 power adapter connector. See Section 7.2. for more power option details. The shorting block positions for connecting the MCUs to the LIN transceivers are listed in Table 6. The pin connections for the external LIN devices are listed in Table 7.

Table 6. LIN Interface Headers (J12 and J17) Description

Header Pins	LIN0 Pin Description
J17[9–10]	LIN_TX (P1.0_A)
J17[11–12]	LIN_RX (P1.1_A)
J12[3–4]	LIN_TX (P1.0_B)
J12[1–2]	LIN_RX (P1.1_B)

Table 7. TB1 External LIN Interface Header Description

Pin #	Pin Description
1	GND
2	LIN_OUT
3	+LIN_V

Header J21 connects the P4 power-adaptor supply to the V_HIGH node, which is used as the power source for the LIN transceivers (T1, T2). The shorting block on header J21 can be removed to force the LIN transceivers to use the voltage supply externally supplied on the +LIN_V pin on the TB1 header.

7.9. Port I/O Connectors (J0-J3 and J11, J13, J16)

Each of the parallel ports of the C8051F568 (Side A) and C8051F550 (Side B) has its own 10-pin header connector. Each connector provides a pin for the corresponding port pins 0-7, +5 V VIO, and digital ground. The same pin-out is used for all of the port connectors.

Table 8. Port I/O Connector Pin Description (J0-J3, J11, J13)

Pin #	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	+5V (VIO)
10	GND (Ground)

Port 2 on the C8051F550 (Side B) MCU has only two pins and has a reduced header.

Table 9. Port I/O Connector Pin Description (J16)

Pin #	Pin Description
1	Pn.0
2	Pn.1
3	+5V (VIO)
4	GND (Ground)

7.10. Voltage Reference (VREF) Connectors (J5 and J22)

The VREF connectors can be used to connect the VREF pin from the MCU (P0.0) to external 0.1 μ F and 4.7 μ F decoupling capacitors. The C8051F568 (Side A) device is connected to the capacitors through the J22 header and the C8051F550 (Side B) device connects to its own set of capacitors through J5.

7.11. Expansion Connector (P1)

The 96-pin expansion I/O connector P1 is used to connect daughter boards to the main target board. P1 provides access to many C8051F568 signal pins. Pins for VREGIN, VDD, VIO, and 3.3V are also available. See Table 10 for a complete list of pins available at P1.

The P1 socket connector is manufactured by Hirose Electronic Co. Ltd, part number PCN13-96S-2.54DS, Digi-Key part number H7096-ND. The corresponding plug connector is also manufactured by Hirose Electronic Co. Ltd, part number PCN10-96P-2.54DS, Digi-Key part number H5096-ND.

Table 10. P1 Pin Listing

Pin #	Description	Pin #	Description	Pin #	Description
A-1	+3.3V	B-1	GND	C-1	N/C
A-2	N/C	B-2	N/C	C-2	N/C
A-3	N/C	B-3	N/C	C-3	N/C
A-4	N/C	B-4	N/C	C-4	N/C
A-5	N/C	B-5	N/C	C-5	N/C
A-6	N/C	B-6	N/C	C-6	N/C
A-7	N/C	B-7	N/C	C-7	N/C
A-8	N/C	B-8	N/C	C-8	N/C
A-9	N/C	B-9	N/C	C-9	N/C
A-10	N/C	B-10	P0.7_A	C-10	P0.6_A
A-11	P0.5_A	B-11	P0.4_A	C-11	P_0.3_A
A-12	P_0.2_A	B-12	P0.1_A	C-12	P0.0_A
A-13	P3.7_A	B-13	P3.6_A	C-13	P3.5_A
A-14	P3.4_A	B-14	P3.3_A	C-14	P3.2_A
A-15	P3.1_A	B-15	P3.0_A	C-15	P3.7_A_L
A-16	P3.6_A_L	B-16	P3.5_A_L	C-16	P3.4_A_L
A-17	P3.3_A_L	B-17	P3.2_A_L	C-17	P3.1_A_L
A-18	P3.0_A_L	B-18	P2.7_A	C-18	P2.6_A
A-19	P2.5_A	B-19	P2.4_A	C-19	P2.3_A
A-20	P2.2_A	B-20	P2.1_A	C-20	P2.0_A
A-21	P1.7_A	B-21	P1.6_A	C-21	P1.4_A
A-22	P1.2_A	B-22	P1.3_A	C-22	P1.5_A
A-23	P1.1_A	B-23	P1.0_A	C-23	N/C
A-24	P4.0_C2D_A	B-24	N/C	C-24	N/C
A-25	RST_A	B-25	GND	C-25	N/C
A-26	GND	B-26	N/C	C-26	N/C
A-27	N/C	B-27	N/C	C-27	N/C
A-28	N/C	B-28	N/C	C-28	N/C
A-29	VIO_A	B-29	VDD_A	C-29	VREGIN_A
A-30	N/C	B-30	N/C	C-30	N/C
A-31	N/C	B-31	N/C	C-31	N/C
A-32	N/C	B-32	AGND	C-32	N/C

The Silicon Labs add-on boards designed to interface to the 96-pin interface are primarily designed to operate at 3.3 V and to use a non-multiplexed external memory interface. When interfacing to a +3.3 V add-on board, it is recommended to put the shorting block on J4[2-3] to connect VIO of the C8051F568 to +3.3 V instead of the default +5 V. The C8051F560 target board also includes an 8-bit latch (U8) for use with the multiplexed mode of the

external memory interface. The output pins of the latch are connected to the 96-pin header and include an _L suffix in the pin name.

7.12. Potentiometer (J20)

The C8051F568 (Side A) device has the option to connect port pin P1.2 to a 10K linear potentiometer (R27). The potentiometer is connected through the J20 header. The potentiometer can be used for testing the analog-to-digital (ADC) converter of the MCU.

7.13. Power Supply I/O (Side A) (TB3)

All of the C8051F568 target device's supply pins are connected to the TB3 terminal block. Refer to Table 11 for the TB3 terminal block connections.

Table 11. TB3 Terminal Block Pin Descriptions

Pin #	Description
1	VIO_A
2	VREGIN_A
3	VDD_A
4	VDDA_A
5	GNDA_A
6	GND

7.14. Alternate Power Supply Headers (J18, J21)

The C8051F560 Target Board includes two headers that allow for alternate power sources and power measurement. Header J18 connects the VIO voltage supplied to the Side A MCU to other peripherals on the board, such as the P1.4_A push-button switch pull-up, and the R27 potentiometer source. To enable current measurement, the shorting block on J18 can be removed so that the VIO_A node only powers the VIO pin on the MCU. Another voltage source will need to be applied to the VIO_SRC node to power the other peripherals.

7.15. C2 Pin Sharing

On the C8051F568 (Side A) and the C8051F550 (Side B), the debug pins C2CK and C2D are shared with the pins RST and P4.0/P2.1 respectively. The target board includes the resistors necessary to enable pin sharing which allow the pin-shared pins to be used normally while simultaneously debugging the device. See Application Note "AN124: Pin Sharing Techniques for the C2 Interface" at www.silabs.com for more information regarding pin sharing.

7.16. Target Board Pin Assignment Summary

Some GPIO pins of the C8051F568 MCU can have an alternate fixed function. For example, pin 38 on the C8051F568 MCU is designated P0.4, and can be used as a GPIO pin. Also, if the UART0 peripheral on the MCU is enabled using the crossbar registers, the TX signal is routed to this pin. This is shown in the "Alternate Fixed Function" column. The "Target Board Function" column shows that this pin is used as TX on the C8051F560 Target Board. The "Relevant Headers" column shows that this signal is routed to pin 3 of the J17 header and pin 5 of the J1 header. More details can be found in the C8051F55x/56x/57x data sheet. Some of the GPIO pins of the C8051F568 have been used for various functions on the target board. All pins of the Side A MCU also connect to the 96-pin (P1) expansion connector which is not explicitly listed below. Table 12 summarizes the C8051F568 MCU pin assignments on the target board, and also shows the various headers associated with each signal.

Table 12. C8051F560 Target Board Pin Assignments and Headers

MCU Pin Name	Pin#	Primary Function	Alternate Fixed Function	Target Board Function	Relevant Headers
P0.0	8	P0.0	VREF	VREF	J0[1], J22[1]
P0.1	1	P0.1	CNVSTR	CNVSTR	J0[2]
P0.2	40	P0.2	XTAL1	XTAL1	J0[3]*, J9[2]
P0.3	39	P0.3	XTAL2	XTAL2	J0[4]*, J10[2]
P0.4	38	P0.4	UART_TX	TX_MCU	J0[5], J17[3]
P0.5	37	P0.5	UART_RX	RX_MCU	J0[6], J17[1]
P0.6	36	P0.6	CAN_TX	CAN_TX	J0[7], J17[5]
P0.7	35	P0.7	CAN_RX	CAN_RX	J0[8], J17[7]
P1.0	34	P1.0		LIN_TX	J1[1], J17[9]
P1.1	33	P1.1		LIN_RX	J1[2], J17[11]
P1.2	32	P1.2		POTENTIOMETER	J1[3], J20[1]
P1.3	31	P1.3		LED	J1[4], J19[4]
P1.4	30	P1.4		SWITCH	J1[5], J19[2]
P1.5	29	P1.5	ALE (U8)	GPIO	J1[6]
P1.6	28	P1.6		GPIO	J1[7]
P1.7	27	P1.7		GPIO	J18]
P2.0	26	P2.0		GPIO	J2[1]
P2.1	25	P2.1		GPIO	J2[2]
P2.2	24	P2.2		GPIO	J2[3]
P2.3	23	P2.3		GPIO	J2[4]
P2.4	22	P2.4		GPIO	J2[5]
P2.5	21	P2.5		GPIO	J2[6]
P2.6	20	P2.6		GPIO	J2[7]
P2.7	19	P2.7		GPIO	J2[8]
P3.0	18	P3.0		GPIO	J3[1]

Table 12. C8051F560 Target Board Pin Assignments and Headers (Continued)

MCU Pin Name	Pin#	Primary Function	Alternate Fixed Function	Target Board Function	Relevant Headers
P3.1	17	P3.1		GPIO	J3[2]
P3.2	16	P3.2		GPIO	J3[3]
P3.3	15	P3.3		GPIO	J3[4]
P3.4	14	P3.4		GPIO	J3[5]
P3.5	13	P3.5		GPIO	J3[6]
P3.6	12	P3.6		GPIO	J3[7]
P3.7	11	P3.7		GPIO	J3[8]
P4.0_C2D	9	C2D	P4.0	GPIO	P2[4], P2[6]*
$\overline{\text{RST}}/\text{C2CK}$	10	$\overline{\text{RST}}$	C2CK	$\overline{\text{RST}}/\text{C2CK}$	P2[7], P2[5]*
VIO	2	VIO		VIO	J4[2], J18[1], TB3[1] J0-J3[9]*, J23
VREGIN	3	VREGIN		VREGIN	J24[2], TB3[2]
VDD	4	VDD		VDD	TB3[3]
VDDA	5	VDDA		VDDA	TB3[4]
GND	6	GND		GND	J0-J3[10], TB3[6]
GNDA	7	GNDA		GNDA	TB3[5]
*Note: Headers denoted by this symbol are not directly connected to the MCU pin; the connection might be via one or more headers and/or pin-sharing resistor(s). See board schematic for details.					

8. Schematics

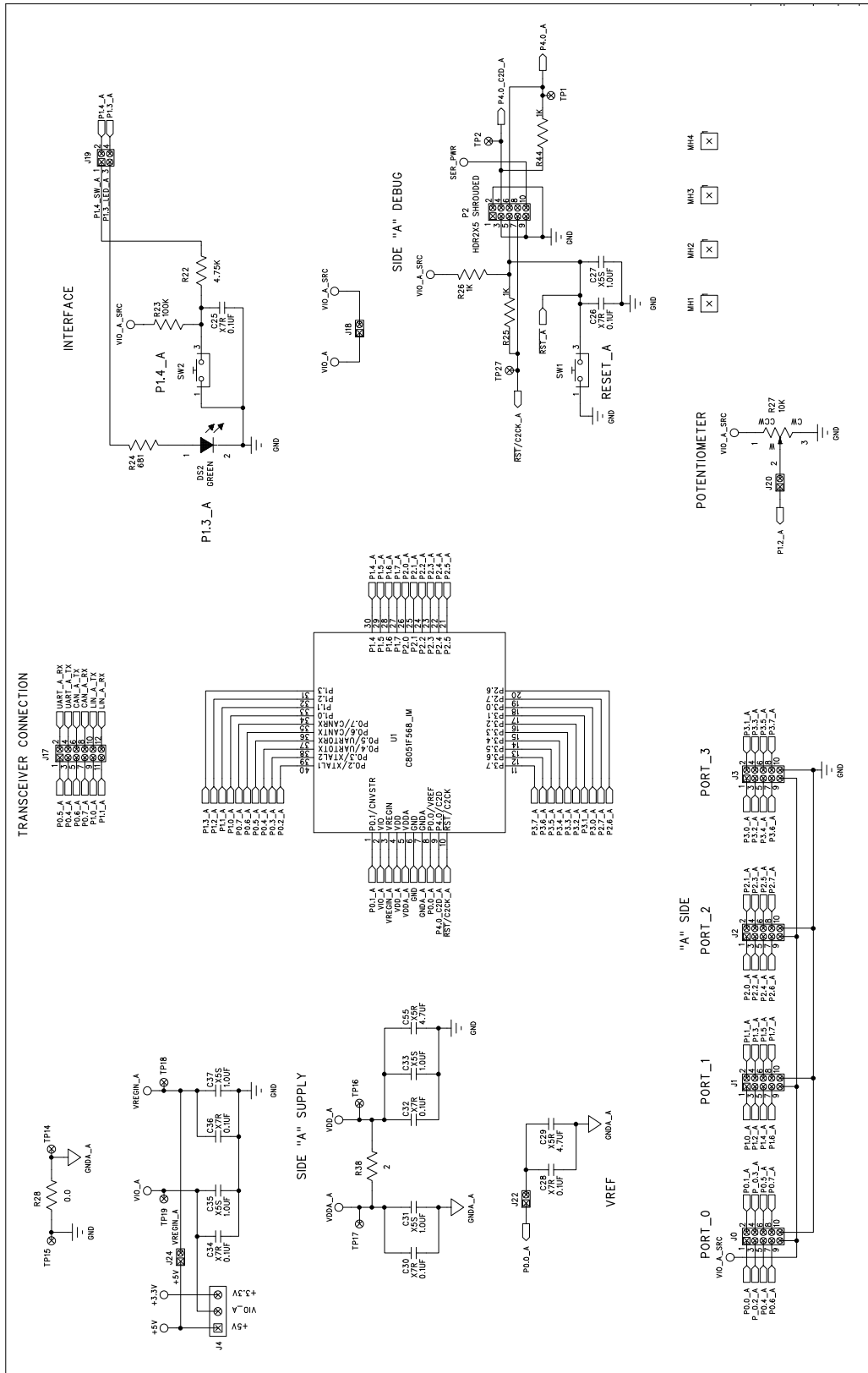


Figure 5. C8051F560 Target Board Schematic (Page 1 of 4)

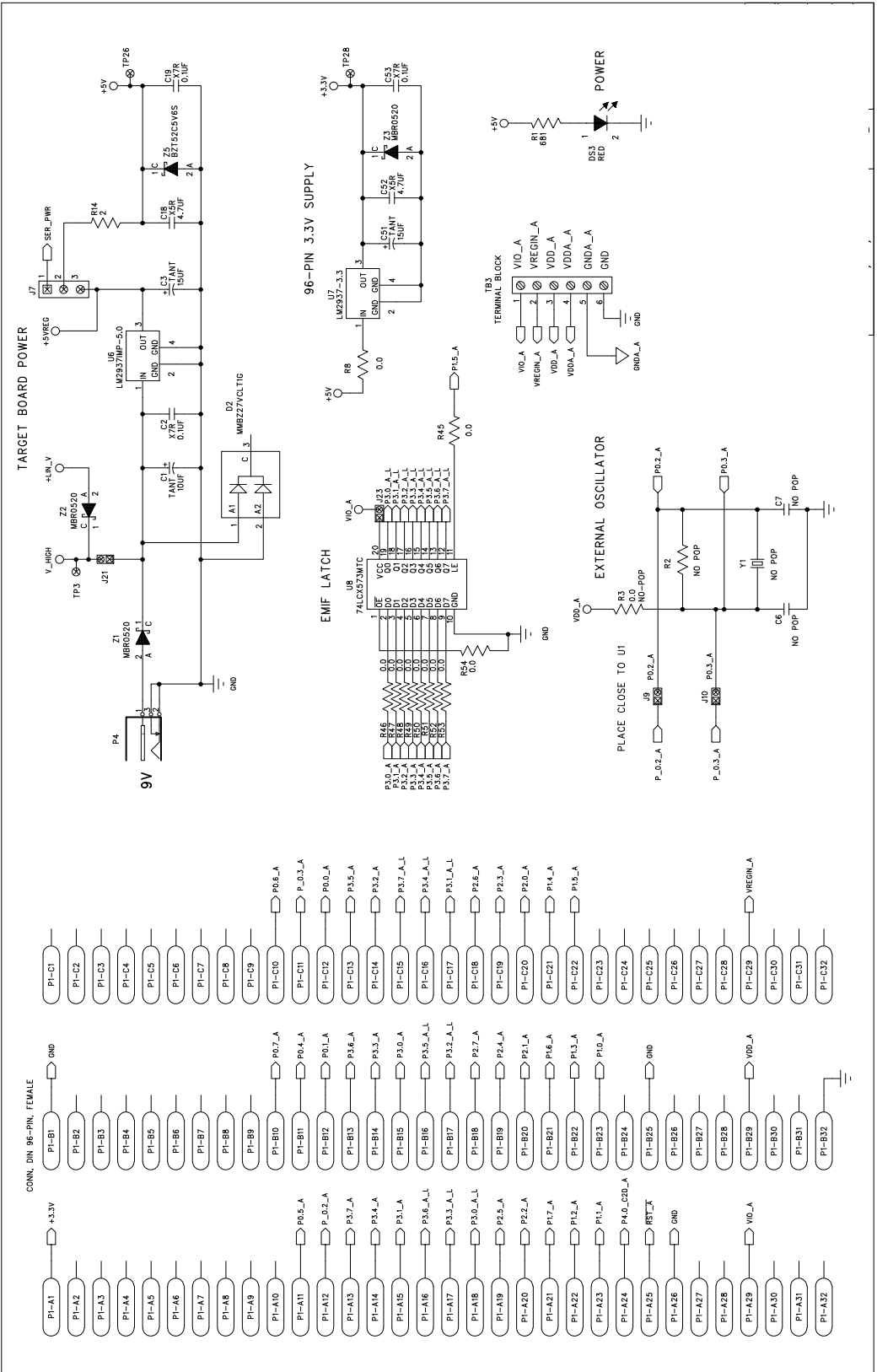


Figure 6. C8051F560 Target Board Schematic (Page 2 of 4)



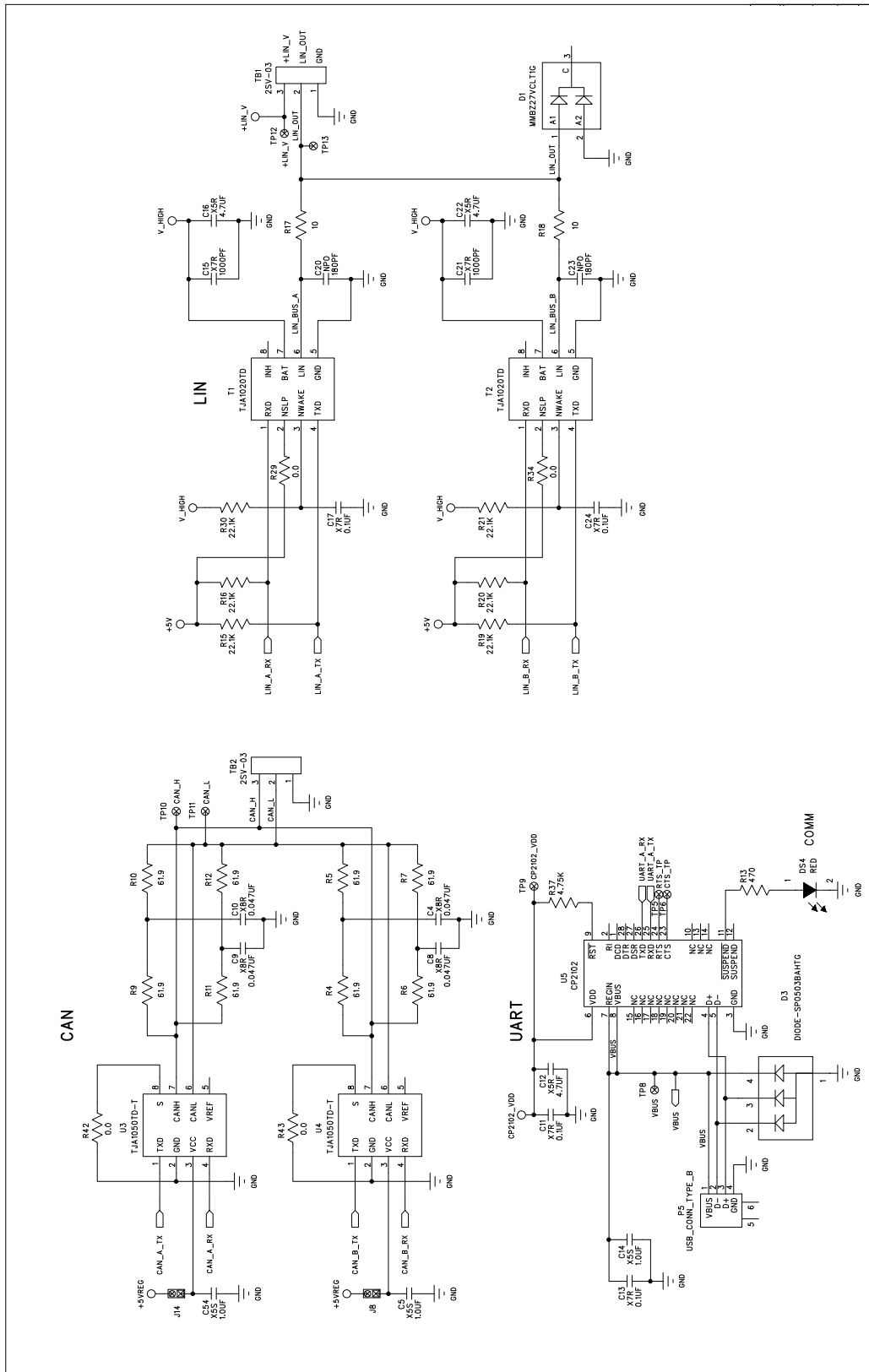


Figure 8. C8051F560 Target Board Schematic (Page 4 of 4)

NOTES:

CONTACT INFORMATION

Silicon Laboratories Inc.
400 West Cesar Chavez
Austin, TX 78701
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.