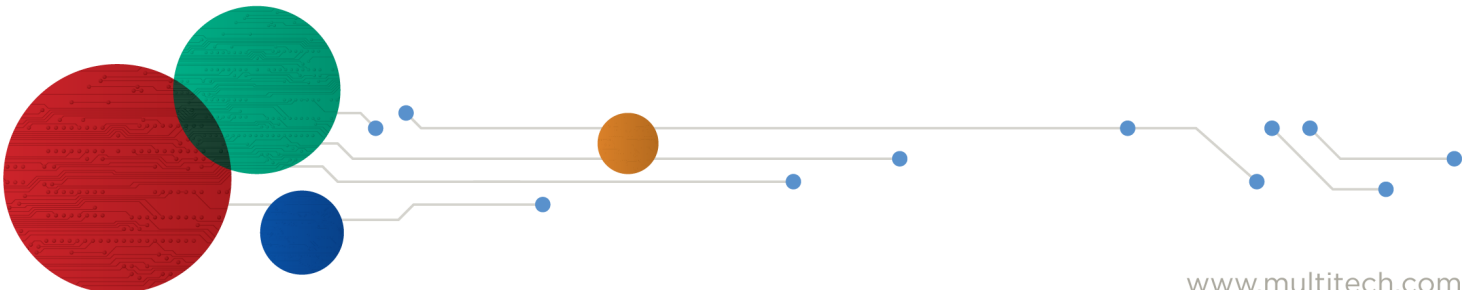


Universal Developer Kit

MTUDK2-ST-CELL.R1 Developer Guide



Universal Developer Kit Developer Guide

Models: MTUDK2-ST-CELL.R1

Document Part Number: S000779 Rev. 1.2

Copyright

This publication may not be reproduced, in whole or in part, without the specific and express prior written permission signed by an executive officer of Multi-Tech Systems, Inc. All rights reserved. Copyright © 2025 by Multi-Tech Systems, Inc.

Multi-Tech Systems, Inc. makes no representations or warranties, whether express, implied or by estoppels, with respect to the content, information, material and recommendations herein and specifically disclaims any implied warranties of merchantability, fitness for any particular purpose and non-infringement.

Multi-Tech Systems, Inc. reserves the right to revise this publication and to make changes from time to time in the content hereof without obligation of Multi-Tech Systems, Inc. to notify any person or organization of such revisions or changes.

Trademarks

Multi-Tech and the Multi-Tech logo, DeviceHQ, SocketModem, and Conduit are registered trademarks of Multi-Tech Systems, Inc.

mPower, mCard, and mDot are trademarks of Multi-Tech Systems, Inc.

All other brand and product names are trademarks or registered trademarks of their respective companies.

Legal Notices

The MultiTech products are not designed, manufactured or intended for use, and should not be used, or sold or re-sold for use, in connection with applications requiring fail-safe performance or in applications where the failure of the products would reasonably be expected to result in personal injury or death, significant property damage, or serious physical or environmental damage. Examples of such use include life support machines or other life preserving medical devices or systems, air traffic control or aircraft navigation or communications systems, control equipment for nuclear facilities, or missile, nuclear, biological or chemical weapons or other military applications ("Restricted Applications"). Use of the products in such Restricted Applications is at the user's sole risk and liability.

MULTITECH DOES NOT WARRANT THAT THE TRANSMISSION OF DATA BY A PRODUCT OVER A CELLULAR COMMUNICATIONS NETWORK WILL BE UNINTERRUPTED, TIMELY, SECURE OR ERROR FREE, NOR DOES MULTITECH WARRANT ANY CONNECTION OR ACCESSIBILITY TO ANY CELLULAR COMMUNICATIONS NETWORK. MULTITECH WILL HAVE NO LIABILITY FOR ANY LOSSES, DAMAGES, OBLIGATIONS, PENALTIES, DEFICIENCIES, LIABILITIES, COSTS OR EXPENSES (INCLUDING WITHOUT LIMITATION REASONABLE ATTORNEYS FEES) RELATED TO TEMPORARY INABILITY TO ACCESS A CELLULAR COMMUNICATIONS NETWORK USING THE PRODUCTS.

Disclaimers

Information in this document is subject to change without notice and does not represent a commitment on the part of Multi-Tech Systems, Inc.. Multi-Tech Systems, Inc. provides this document "as is," without warranty of any kind, expressed or implied, including, but not limited to, the implied warranties of fitness or merchantability for a particular purpose. Multi-Tech Systems, Inc. may make improvements and/or changes in this manual or in the product(s) and/or the software described in this manual at any time.

Contents

Chapter 1 Developer Kit Introduction	5
Overview	5
Features	5
Related Documentation	5
MTUDK2-ST-CELL Developer Kit Contents	6
Chapter 2 Board Components	7
Developer Board	7
Developer Board Dimensions	8
Developer Board Connectors	9
Board Components	9
Additional Connector Information	12
LED Indicators	12
Chapter 3 Installation and Operation	13
Installing a SIM Card on a SocketModem Cell	13
Installing a SIM Card on a SocketModem	13
Installing a SIM Card on SocketModem MTQN	14
Installing a SocketModem Cell on the Developer Board	15
Installing a SocketModem MTQ on the Developer Board	17
Installing a SocketModem MTQN on the Developer Board	19
Installing an Arduino Shield with a SocketModem MTQ	20
SocketModem MTQ Arduino Pins	22
SocketModem MTQN Arduino Pins	23
Attaching Power Supply Blades	23
Power Supply and Blades	23
Attaching the Blades	24
SMA to U.FL Cables	25
Connecting an Antenna through the Developer Board Connectors	25
Chapter 4 Diagrams and Schematics	27
Assembly Diagram	27
Block Diagram	28
Schematics - Power	29
Schematics - RS-232	30
Schematics - Sockets	31
Schematics - ST Link	32
Schematics - LEDs	33
Schematics - USB	34

Chapter 5 Design Considerations 35

Noise Suppression Design 35

PC Board Layout Guideline 35

Electromagnetic Interference 35

Electrostatic Discharge Control 36

USB Design 36

Warranty..... 37

Contact Information 37

Revision History 37

Chapter 1 Developer Kit Introduction

Overview

The MTUDK2-ST-CELL.R1 (UDK2.R1) Universal Developer Kit supports development with cellular SocketModem Cell (MTSMC), SocketModem MTQ, and SocketModem MTQN. Use the developer board to streamline your development efforts and evaluate your products and applications. Easily plug in your communications device and use the developer kit for testing, programming and evaluation.

Note: Unless otherwise specified, features labeled as MTQ on the Developer Board, the feature can be used by the MTQN.

Features

- 6V-12V power input
- Selectable 3.3V or 5V on board power supply
- USB and serial interfaces
- USB port for mbed development environment
- RS-232 DB-9 connector for serial interface
- Arduino shield socket

Related Documentation

This User Guide provides information about the Developer Kit. You will also need:

- Device Guide for your SocketModem Cell or SocketModem MTQ model for specifications, pin information, mechanical drawings, labeling, regulatory information, and other model specific details.
- AT Command Reference Guide for the radio on your device
- USB Driver Installation Guide for USB models, for driver installation steps.

These documents are available on the MultiTech website. Follow the relevant link below, select your model, and select the specific documentation for your device.

- [SocketModem Cell](#)
- [SocketModem MTQ](#)
- [SocketModem MTQN](#)

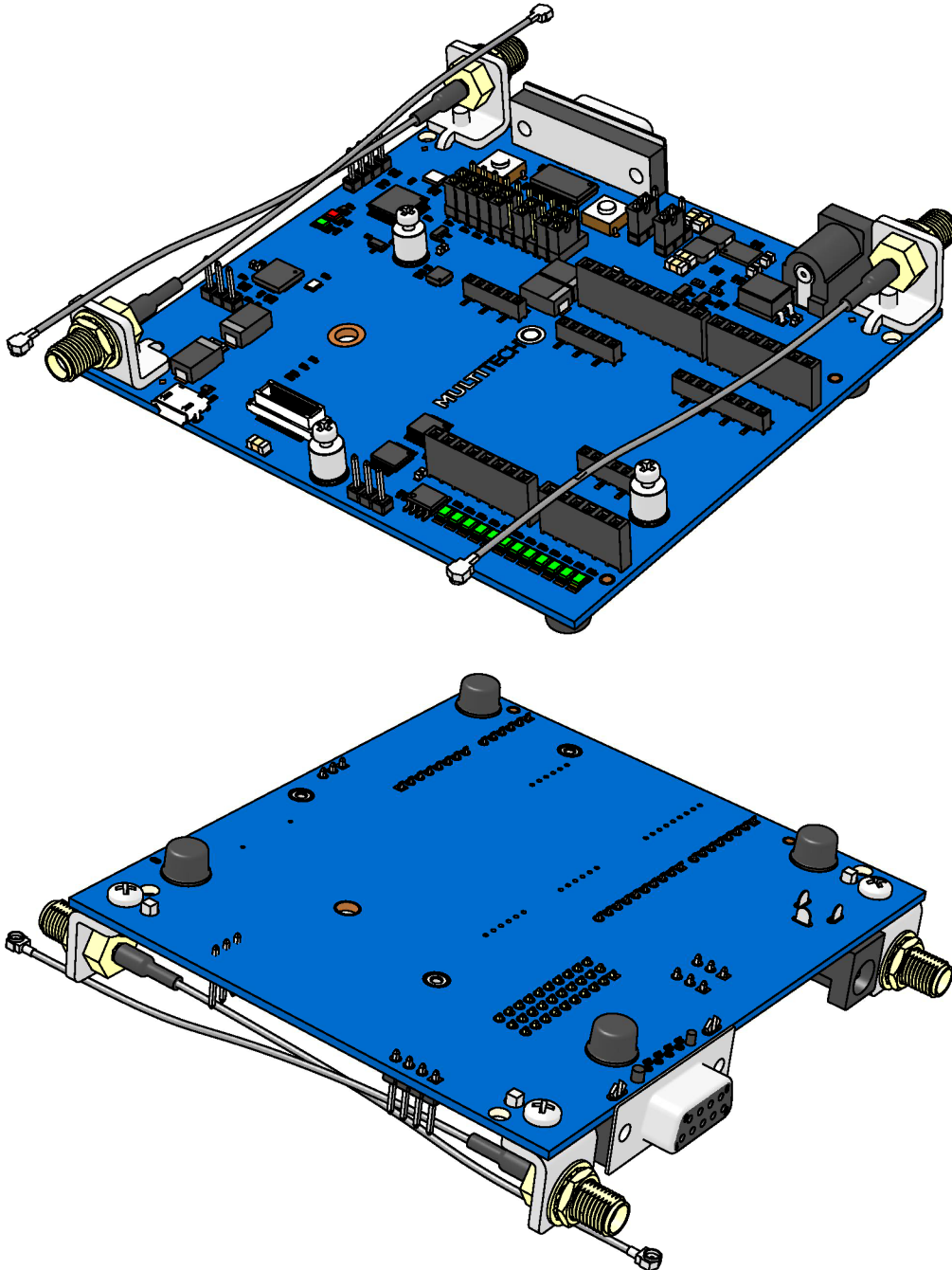
MTUDK2-ST-CELL Developer Kit Contents

The MTUDK2-ST-CELL Developer Kit includes the following:

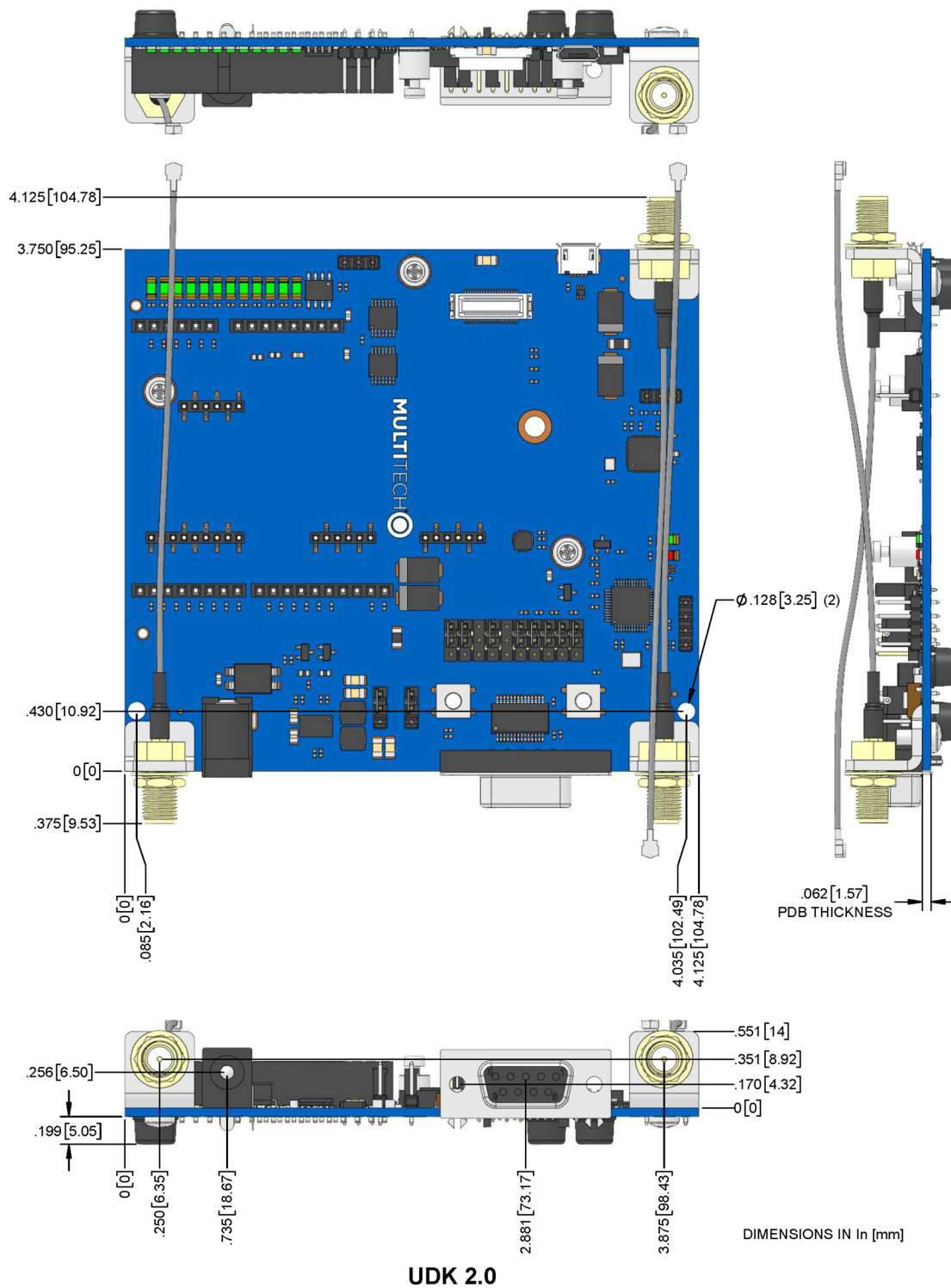
Developer Board	1 - MTUDK 2.0 Cell Developer Board
Power Supply	1 - 100-240V 9V-1.7A power supply with removable blades
	1 - US blade/plug,
	1 - EU blade/plug
	1 - GB blade/plug
	1 - AU/NZ blade/plug
Cables	1 - Micro USB Cable
	3 - SMA-U.FL Antenna Cables (attached to developer board)
Antennas	1 - 3.3V magnetic GPS Antenna
	2 - 700-2600 MHz Antennas
Customer Notices	Quick Start Guide
Additional	One promotional screwdriver

Chapter 2 Board Components

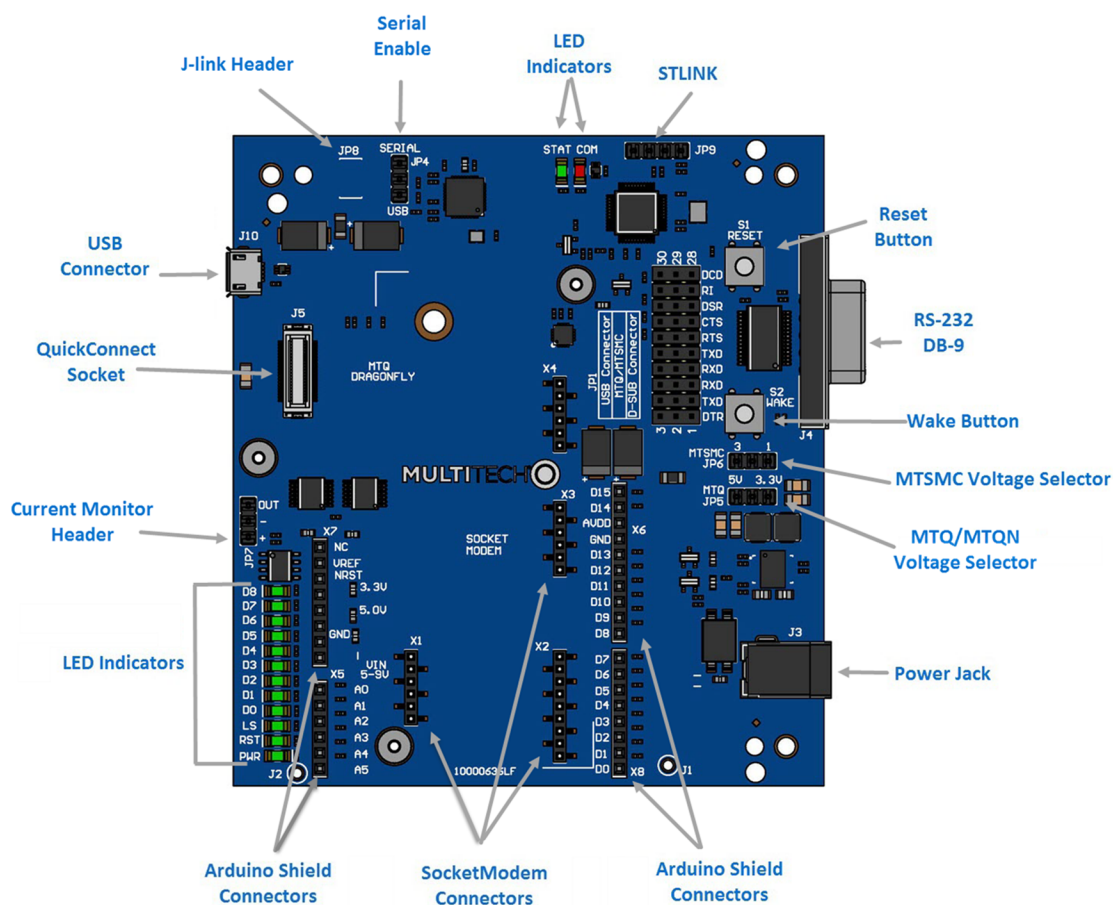
Developer Board



Developer Board Dimensions



Developer Board Connectors



Board Components

Label	Description
J1	Scope probe ground connection
J2	Scope probe ground connection
J3	Power Jack
J4	RS-232 Connector
J5	QuickConnect Socket (for MTQ/MTQN devices).
J10	USB connector for mbed, serial, and SocketModem Cell. For the MTQ/MTQN, use the USB connector on the MTQ/MTQN. For information on connecting to and using mbed, refer to the device guide for your MTQ/MTQN model. (Not available for SocketModem Cells.)
JP1	3-row connector: USB Connector, MTQ/MTSMC, D-SUB Connector. Refer to MTUDK2-ST-CELL.R1 Developer Guide for additional information.
JP4	For the MTSMC, used to disable the USB interface. Default jumper position is 1-2 for USB.

Label	Description
JP5	Selects between the on-board 3.3V or 5V regulator for powering an MTQ or MTQN. Jumper Pins 1-2: Target voltage 3.3V. Jumper pins 2-3: Target voltage 5V.
JP6	Selects between the on-board 3.3V or 5V regulator for powering an MTSMC. Jumper Pins 1-2: Target voltage 3.3V. Jumper pins 2-3: Target voltage 5V.
JP7	Used with the Current Monitor (U4) to for monitoring the current for the device.
JP8	J-link Header
JP9	4-pin Header STLINK
S1	Reset Button. Use to reset the processor of the device attached to the board.
S2	Wake Button. Use to wake the MTSMC/MTQ/MTQN.
X1	SocketModem Cell, USB Connector.
X2	SocketModem Cell Serial Connector.
X3	SocketModem Cell, GPIO (not connected).
X4	SocketModem Cell Power Connector.
X5	Arduino Shield Connector.
X6	Arduino Shield Connector.
X7	Arduino Shield Connector.
X8	Arduino Shield Connector.

CAUTION: Take care when connecting or disconnecting USB cables to avoid detaching the connector from the board.

Main Regulator

The main regulator supplies 3.3V and 5V to the board from the main barrel jack or from USB if the Host can deliver enough power.

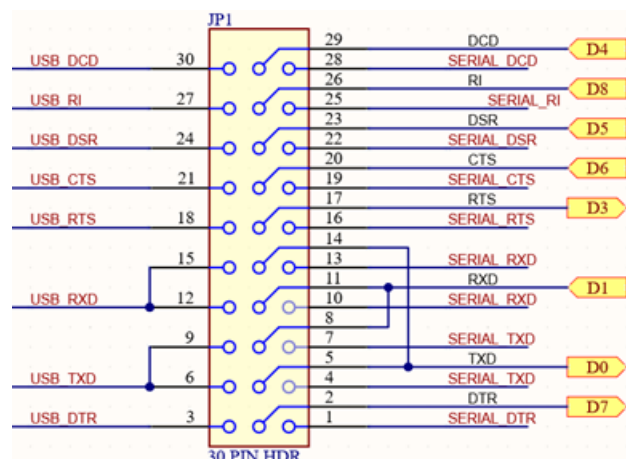
The switching frequency is set to 1.05MHz, but is adjustable from 300 KHz to 2MHz. Any changes to the frequency may require inductor or capacitor changes for proper operation.

The part also supports a spread spectrum mode that sweeps between the set value and 20% higher than set frequency over a 5 KHz rate. This mode could be used to reduce emissions from the power supply.

This supply is not meant to be a reference for general MTQ or Socket Modem designs as generally users pick 3.3 or 5V and don't need to have high current capabilities on both voltages.

JP1 USB Connector

Use the JP1 USB Connector to route the serial connection from the MTQ/MTSMC to either the USB or the DB9



To select USB for serial communication from the modem:

- Use the left side of the connector (3-30) with shunts to connect the modem to the USB transceiver.

To select the 9-pin serial connector:

- Use the right side of the connector (1-28).

Note: There are extra pins for RX and TX so you can swap these signal directions.

Default Jumper Positions

By default, USB is selected with the following jumper positions installed:

- DCD: 29-30
- RI: 26-27
- DSR: 23-24
- CTS: 20-21
- RTS: 17-18
- Open
- RXD: 11-12
- Open
- TXD: 5-6
- DTR: 2-3

The 9-pin serial connector is selected with the following jumper positions installed:

- DCD: 28-29
- RI: 25-26
- DSR: 22-23
- CTS: 19-20
- RTS: 16-17
- Open

- RXD: 10-11
- Open
- TXD: 4-5
- DTR: 1-2

Configuring Headers

Headers can be configured to act like a USB to serial converter for other devices. To do this, use jumper wires.

- Connect pin 6 to pin 10
- Connect pin 4 to pin 12

Additional Connector Information

Voltage Selection Connectors

Three position headers are used to select incoming voltage for the MTQ (JP5) and the Socket Modem (JP6). They allow you to select 5V, 3.3V, or an external voltage (using Pin 2 of JP5 or JP6). These headers can also be used with the current monitor (U4) to measure the current consumed by the device.

USB from MTQ

On the MTUDK2-ST-CELL.R1, there is one USB connection for all communications to the board. If you are working with an MTQ or MTQN, DO NOT use the USB connection to the device while it is connected to the developer board.

Arduino Connectors

The Arduino connectors are intended to allow the user to integrate sensors or other I/O to an MTQ or Socket Modem.

Current Monitor

You can use the current monitor (U4) to monitor current of any device connected to JP7 as long as the voltage and current are within its capabilities of +/-5A and less than 420V. The layout is not designed to have high voltage isolation so there may be issues with voltages over 75V.

To monitor the current used by an MTQ with a 5V input. Connect Pin 3 of JP5 to pin 1 of JP7 and Pin 2 of JP7 to pin 2 of JP5.

To approximate the current in Amps = (Analog Voltage from JP7 pin 3 – 3.3V / 2) / 0.264. For the actual value for your board, measure the voltage on JP7 pin 2 with no current.

LED Indicators

LED indicators are labeled on the board. STAT and COM LEDs are next to JP9. The rest of the LED indicators are next to the Arduino connectors. For more LED information, consult the [schematics](#).

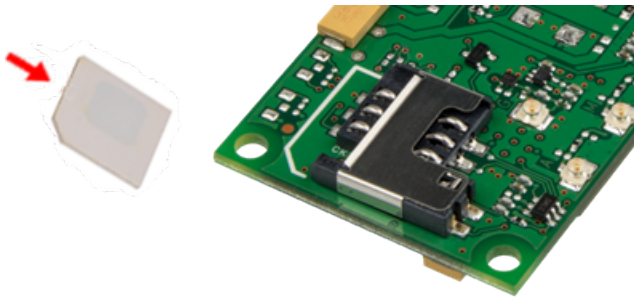
Chapter 3 Installation and Operation

Installing a SIM Card on a SocketModem Cell

Note: When using the SocketModem Cell with a developer board, mount the SocketModem Cell on the developer board before installing the SIM card.

To install the SIM Card:

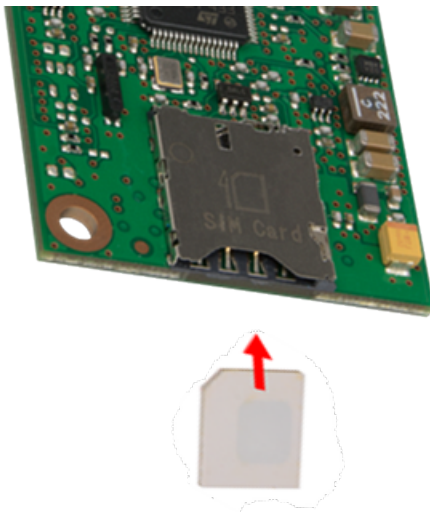
- With the contact side facing down, align the notched edge as outlined on the SocketModem Cell and slide the SIM card completely into the SIM holder.



Installing a SIM Card on a SocketModem

When using the SocketModem with a developer board, install the SIM card before mounting the SocketModem on the developer board.

With the contact side facing down, align the notched edge as shown on the SocketModem's SIM holder and slide the SIM card completely into the SIM holder.



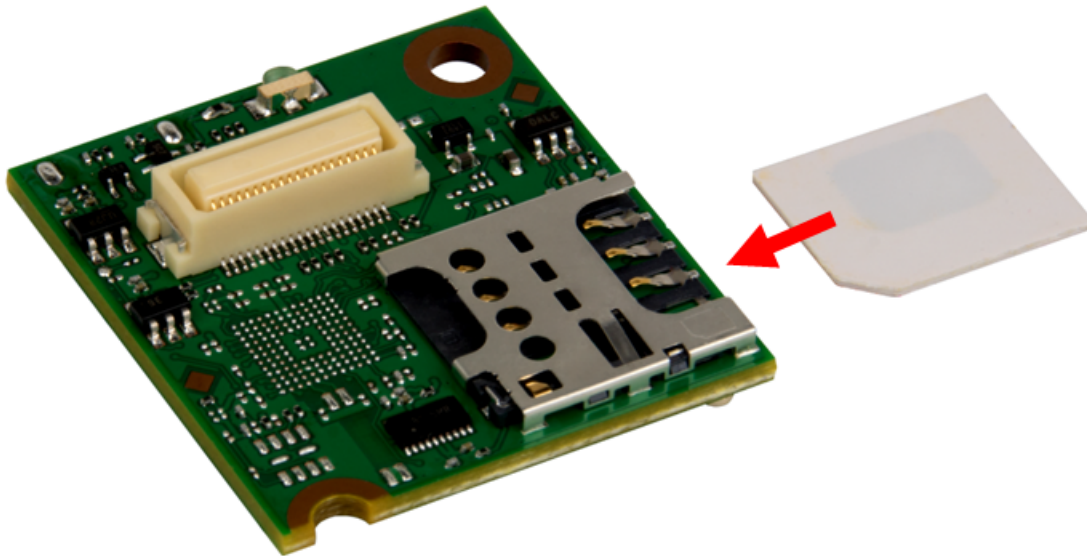
Installing a SIM Card on SocketModem MTQN

Note: When using the SocketModem MTQN with a developer board, install the SIM card before mounting the SocketModem MTQN on the developer board.

Note: All SocketModem MTQN models require the use of a Micro SIM (3FF) card.

To install the SIM card:

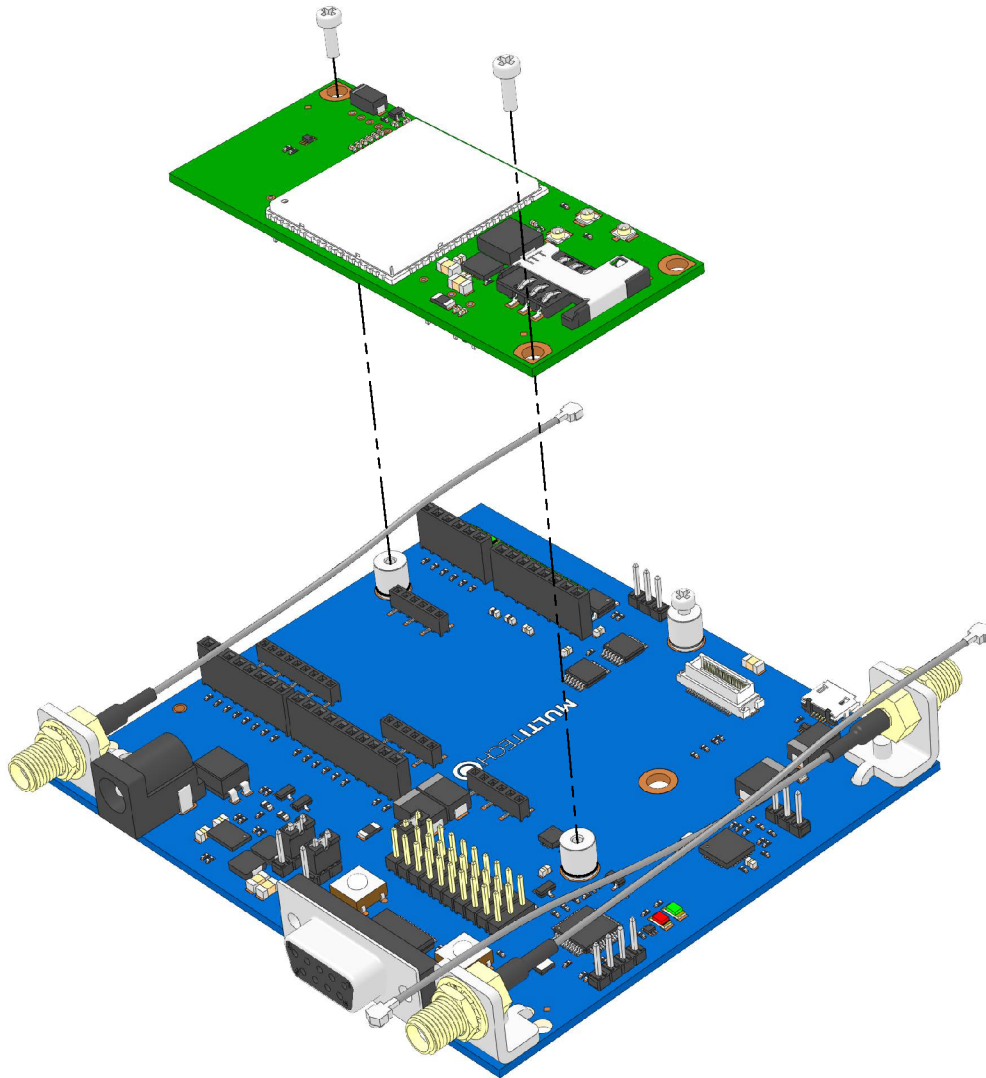
- Refer to the image below on how to install the SIM card.
- Slide the SIM card completely into the SIM holder.



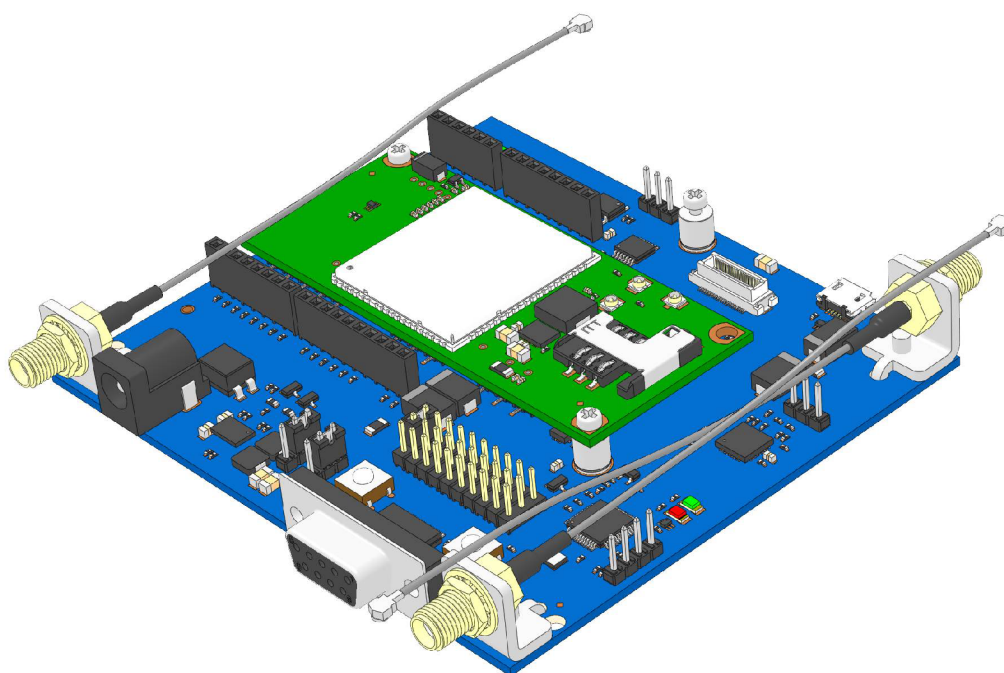
Installing a SocketModem Cell on the Developer Board

To install a SocketModem Cell:

1. Remove the screws from the developer board.
2. Align the SocketModem Cell on the developer board as shown.



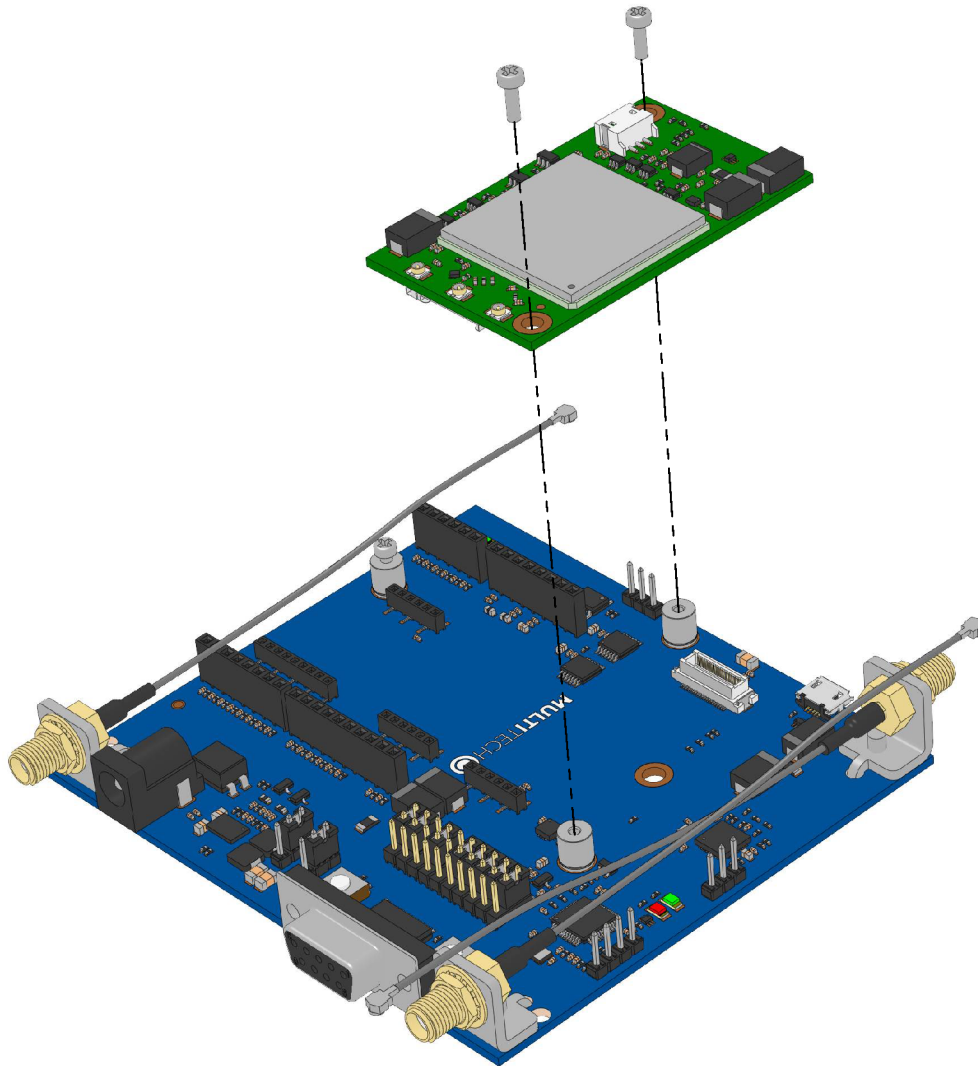
3. Secure the SocketModem Cell with the screws you removed in Step 1.



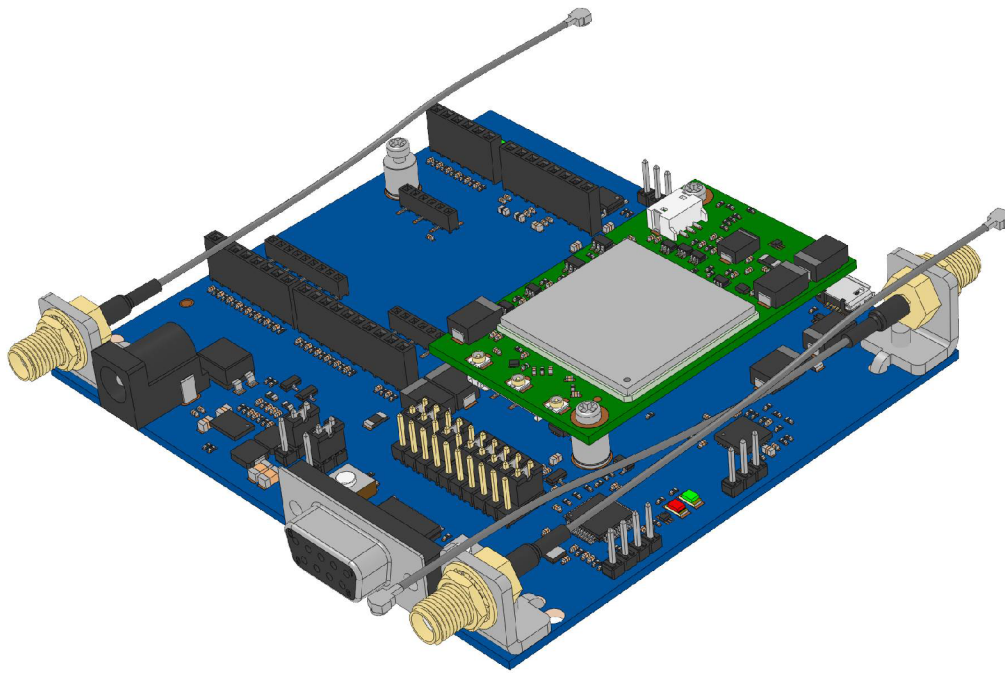
Installing a SocketModem MTQ on the Developer Board

To install a SocketModem MTQ:

1. Remove the screws from the developer board.
2. Align the SocketModem MTQ on the developer board as shown.



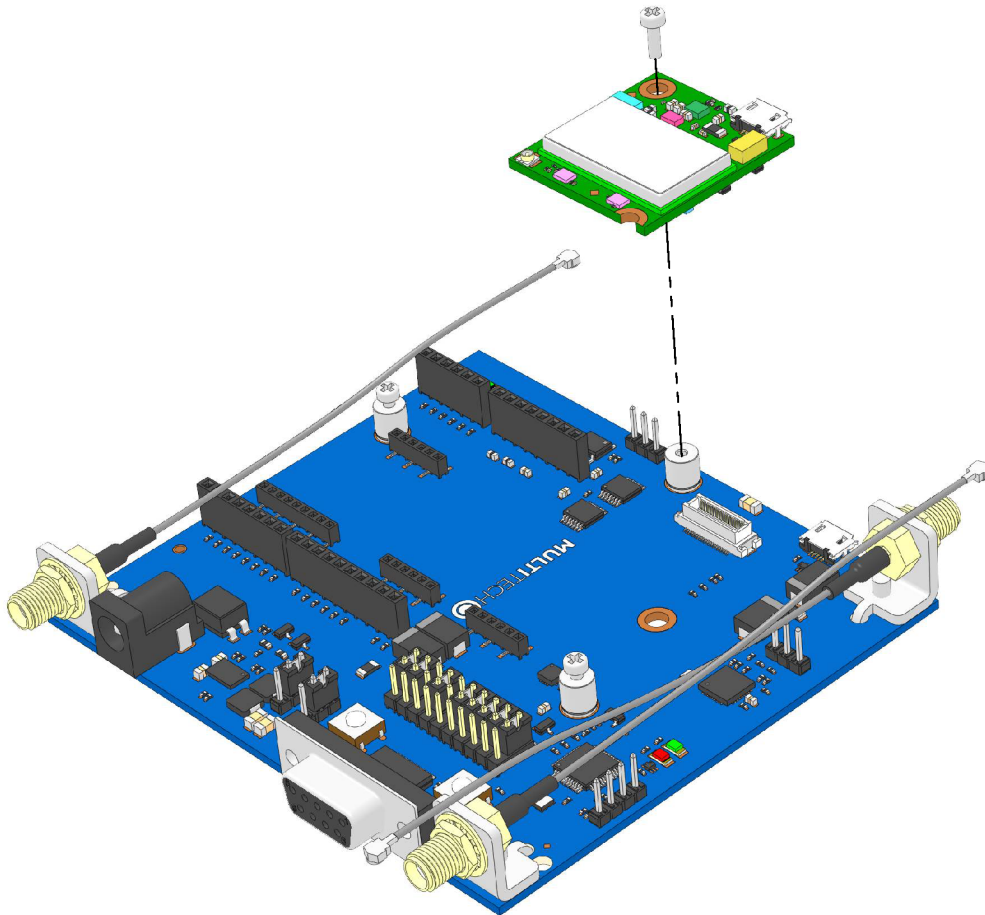
3. Secure the SocketModem MTQ with the screws you removed in Step 1.



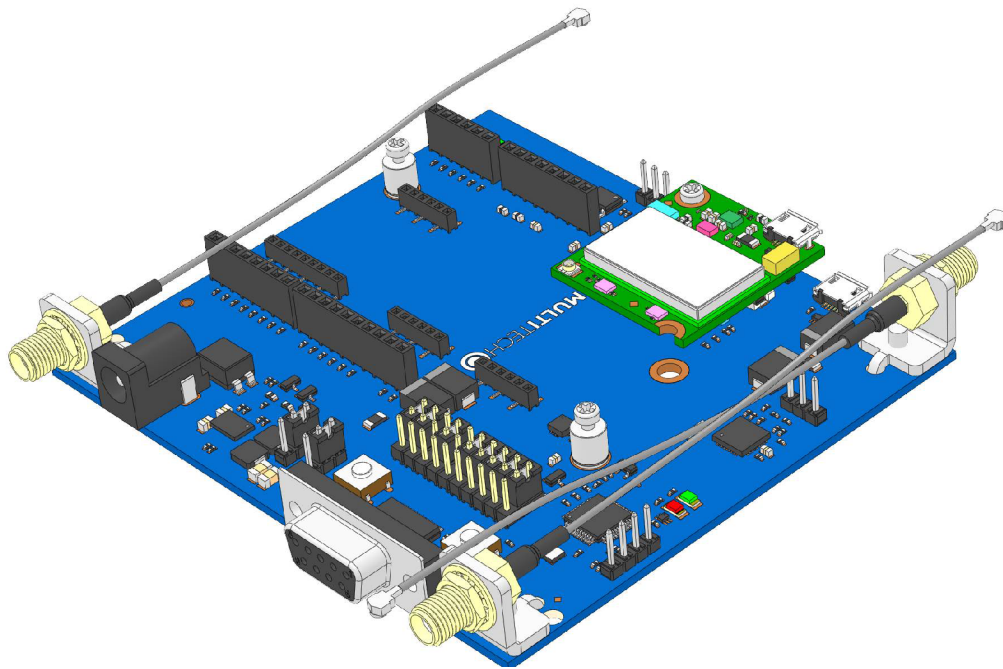
Installing a SocketModem MTQN on the Developer Board

To install a SocketModem MTQN:

1. Remove the screws from the developer board.
2. Align the SocketModem MTQN on the developer board as shown.



3. Secure the SocketModem MTQN with the screws you removed in Step 1.

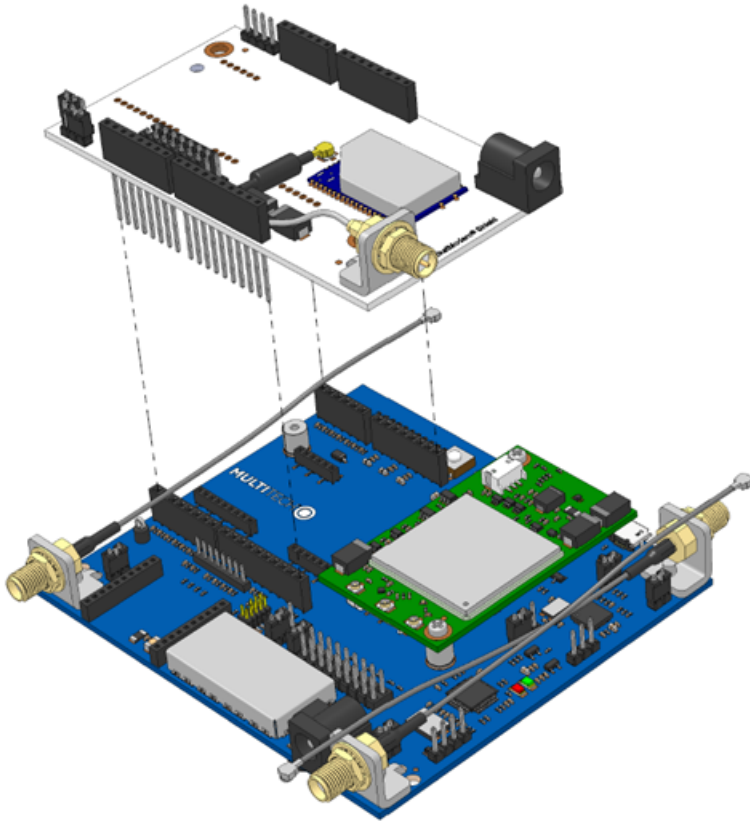


Installing an Arduino Shield with a SocketModem MTQ

Note: When using an Arduino Shield with a SocketModem MTQ, install the SIM card in the SocketModem MTQ and then install the SocketModem MTQ on the developer board before installing the Arduino shield.

To use an Arduino Shield with a SocketModem MTQ

1. Disable the developer card's serial port.
2. Align the Arduino Shield on the developer board as shown. It will overlap the SocketModem MTQ.



SocketModem MTQ Arduino Pins

Signals (MTQ-B01, MTQ-B02 Models)		Arduino Shield			Signals STM32F411RE processor (MTQ-B01 Models)
			D15		D15 (PB8)
			D14		D14 (PB9)
			AVDD		NC
			GND		Ground
		NC	D13	X5	D13 (PA5)
NC		VREF	D12		D12(PA6)
nReset, from pushbutton		nRST	D11		D11 (PB5)
NC		3.3V	D10		D10 (PC8)
5.0V		5.0V	D9		D9 (PB13)
Ground		GND	D8		D8 (PB1)
Ground		GND			
NC		VIN	D7		D7 (PA8)
			D6		D6 (PA1)
A0 (PC2)		A0	D5		D5 (PA9)
A1 (PC0)		A1	D4	X7	D4 (PA7)
A2 (PC4)		A2	D3		D3 (PA0)
A3 (PB0)		A3	D2		D2 (PB15)
A4 (PC1)		A4	D1		D1 (PA2)
A5 (PC9)		A5	D0		D0 (PA3)

SocketModem MTQN Arduino Pins

Signals (MTQN-B01, MTQN-B02 Models)		Arduino Shield			Signals STM32L471QG processor (MTQN-B01 Models)
			D15	X5	D15 (PB8)
			D14		D14 (PB9)
			AVDD		NC
			GND		Ground
	X8	NC	D13	X5	D13 (PG2)
NC		VREF	D12		D12(PG3)
nReset, from pushbutton		nRST	D11		D11 (PB5)
NC		3.3V	D10		D10 (PC8)
5.0V		5.0V	D9		D9 (PB10)
Ground		GND	D8		D8 (PB0)
Ground		GND			
NC		VIN	D7	X7	D7 (PG7)
			D6		D6 (PA1)
A0 (PC2)	X6	A0	D5		D5 (PA9)
A1 (PC13)		A1	D4		D4 (PA7)
A2 (PC4)		A2	D3		D3 (PA0)
A3 (PE6)		A3	D2		D2 (PB15)
A4 (PA6)		A4	D1		D1 (PA2)
A5 (PG8)		A5	D0		D0 (PA3)

Attaching Power Supply Blades

Power Supply and Blades

If your device shipped with a power supply, attach the blades for your region.



Power Supply no blades



Power Supply with EU blade



Power Supply with US blade



Power Supply with GB blade

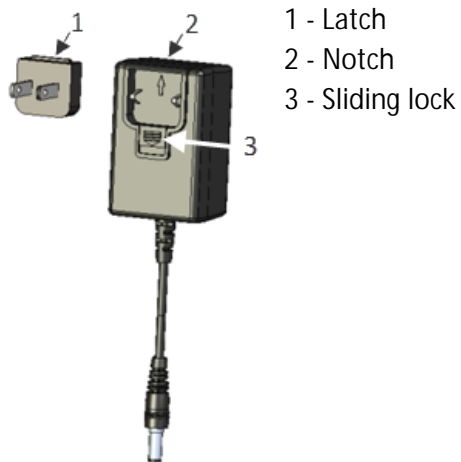


Power Supply with AU-NZ blade

Attaching the Blades

To attach a power supply blade:

1. Remove the power supply cover (not shown). To do this, slide the lock down and hold it while you lift off the cover.
2. Insert the latch on the blade into the notch on the power supply.
3. Slide the lock down and hold it while you press the blade in place. Then, release it.



SMA to U.FL Cables

The developer kit includes three 4.5" SMA to U.FL cables which are preinstalled on the developer board. Consult the mechanical drawings for your device to determine which antenna to connect to which U.FL connector on the device.



Connecting an Antenna through the Developer Board Connectors

To connect an antenna to the device through the developer board:

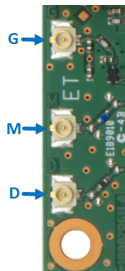
1. Determine which SMA connector you want to use for the antenna.
2. Finger tighten the antenna to the SMA connector.
3. Attach the U.FL connector from the cable to the connector on the device.



G = GNSS (may not apply to your device)

M = Main

D = Auxiliary/Diversity (may not apply to your device)





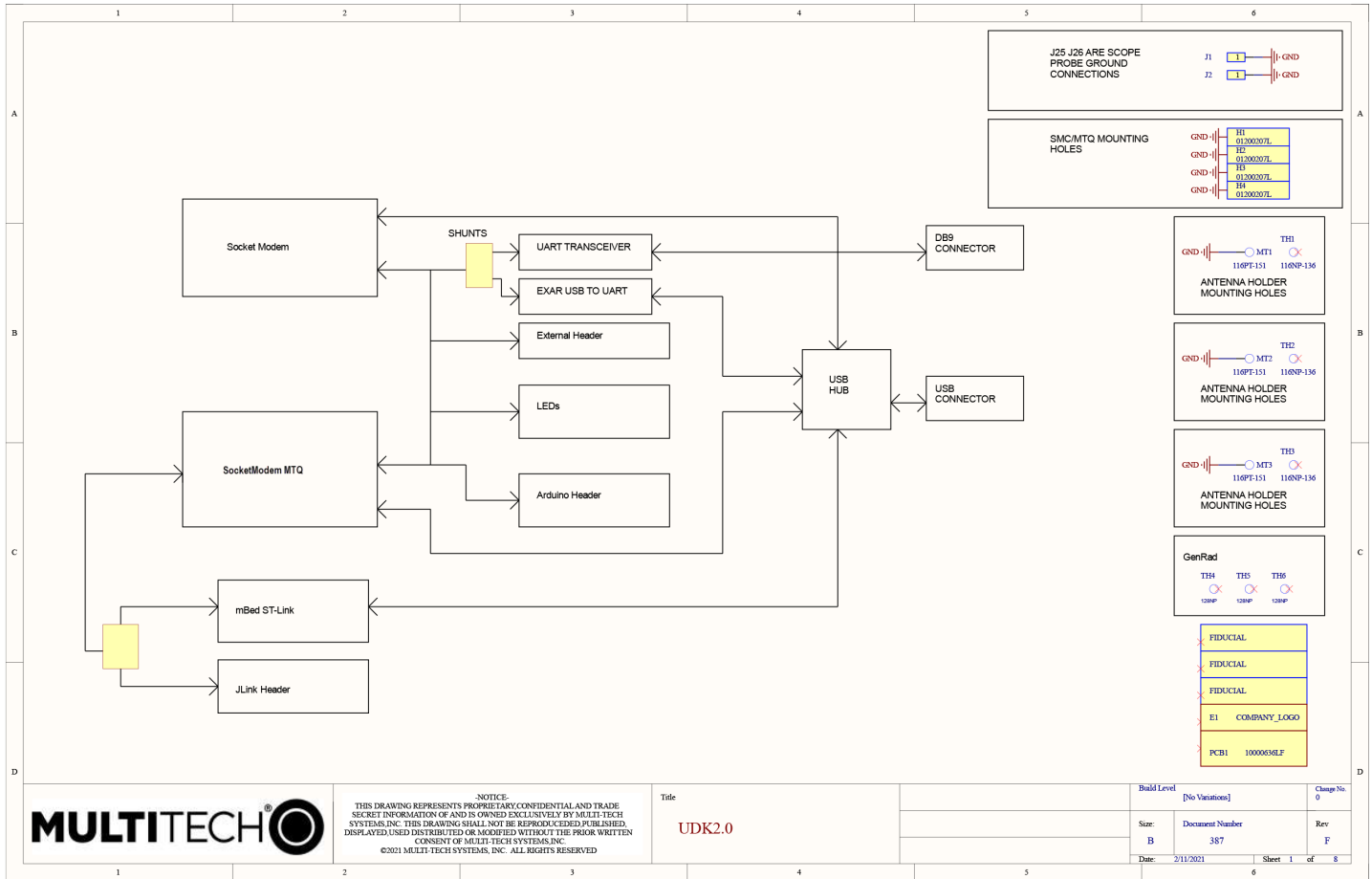
NOTICE:

THIS DRAWING REPRESENTS PROPRIETARY, CONFIDENTIAL AND TRADE SECRET INFORMATION OF AND IS OWNED EXCLUSIVELY BY MULTI-TECH SYSTEMS, INC. THIS DRAWING SHALL NOT BE REPRODUCED, PUBLISHED, DISPLAYED, USED OR DISTRIBUTED OR MODIFIED WITHOUT THE PRIOR WRITTEN CONSENT OF MULTI-TECH SYSTEMS, INC.

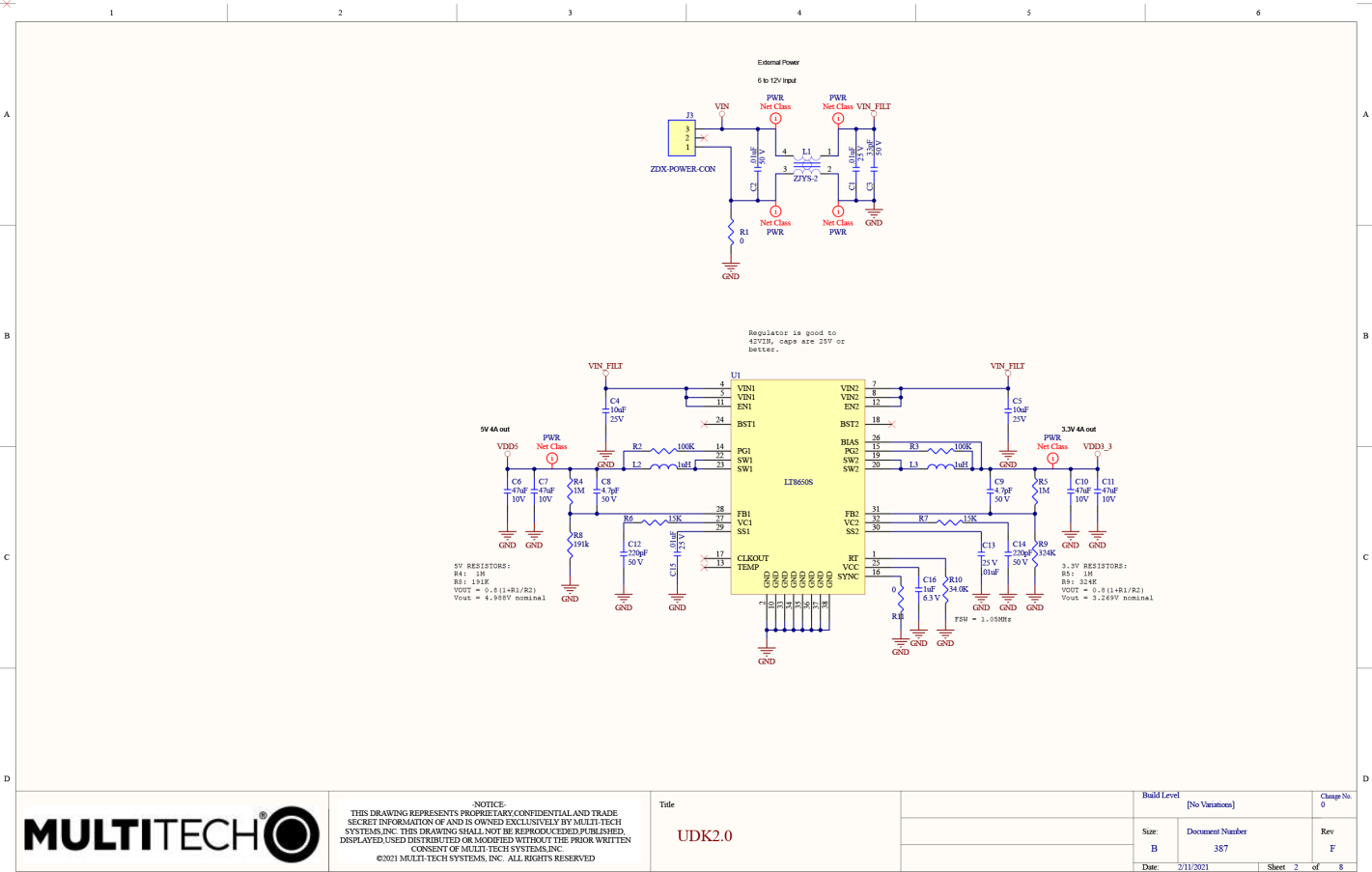
©2020 MULTI-TECH SYSTEMS, INC. ALL RIGHTS RESERVED

TITLE		SCALE: 2:1		SHEET 1 OF 1	
UDK2.0	SIZE	Document Number	Rev		
	B	487P	E		

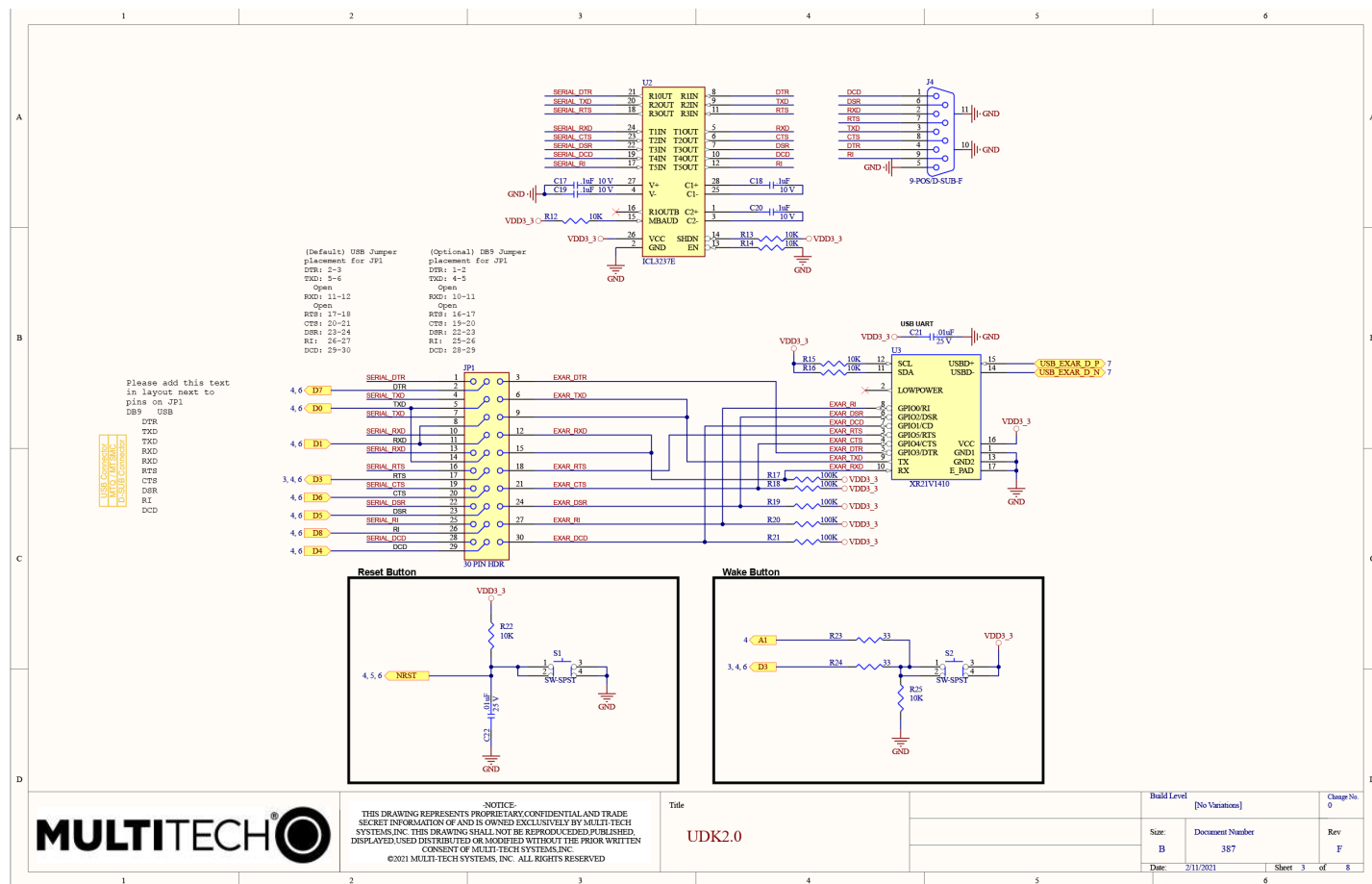
Block Diagram



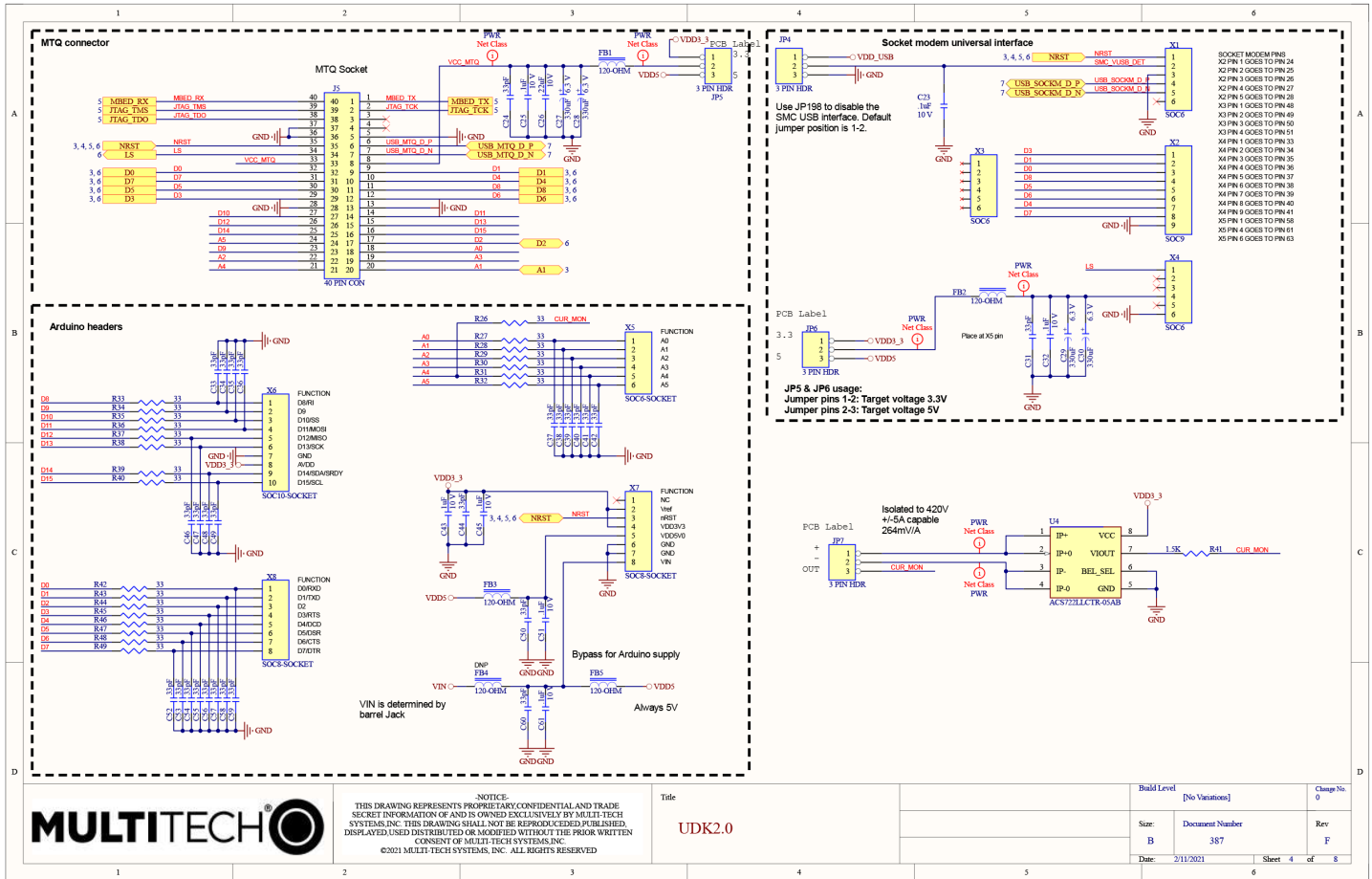
Schematics - Power



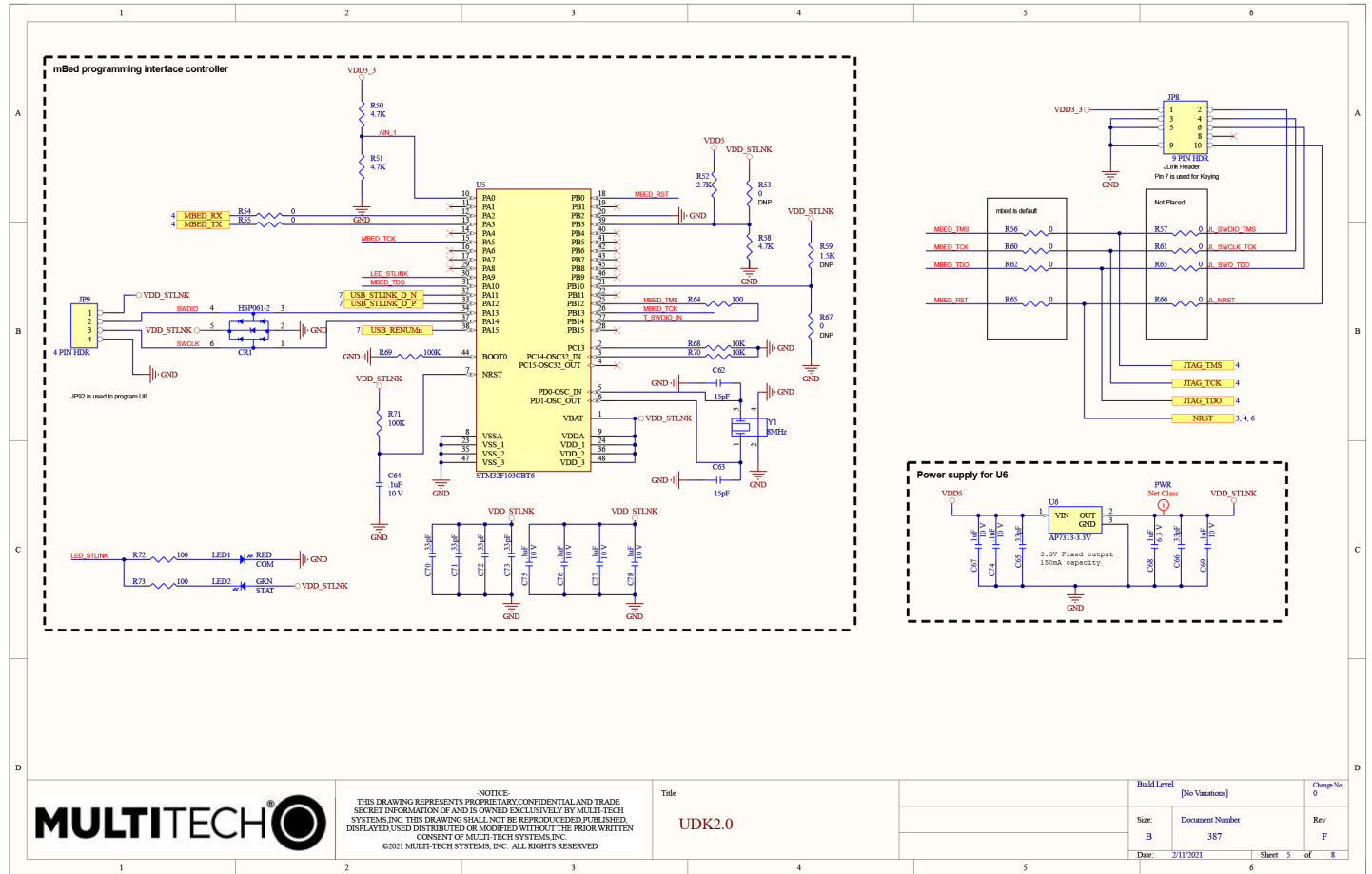
Universal Developer Kit MTUDK2-ST-CELL.R1 Developer Guide
S000779 Rev. 1.2



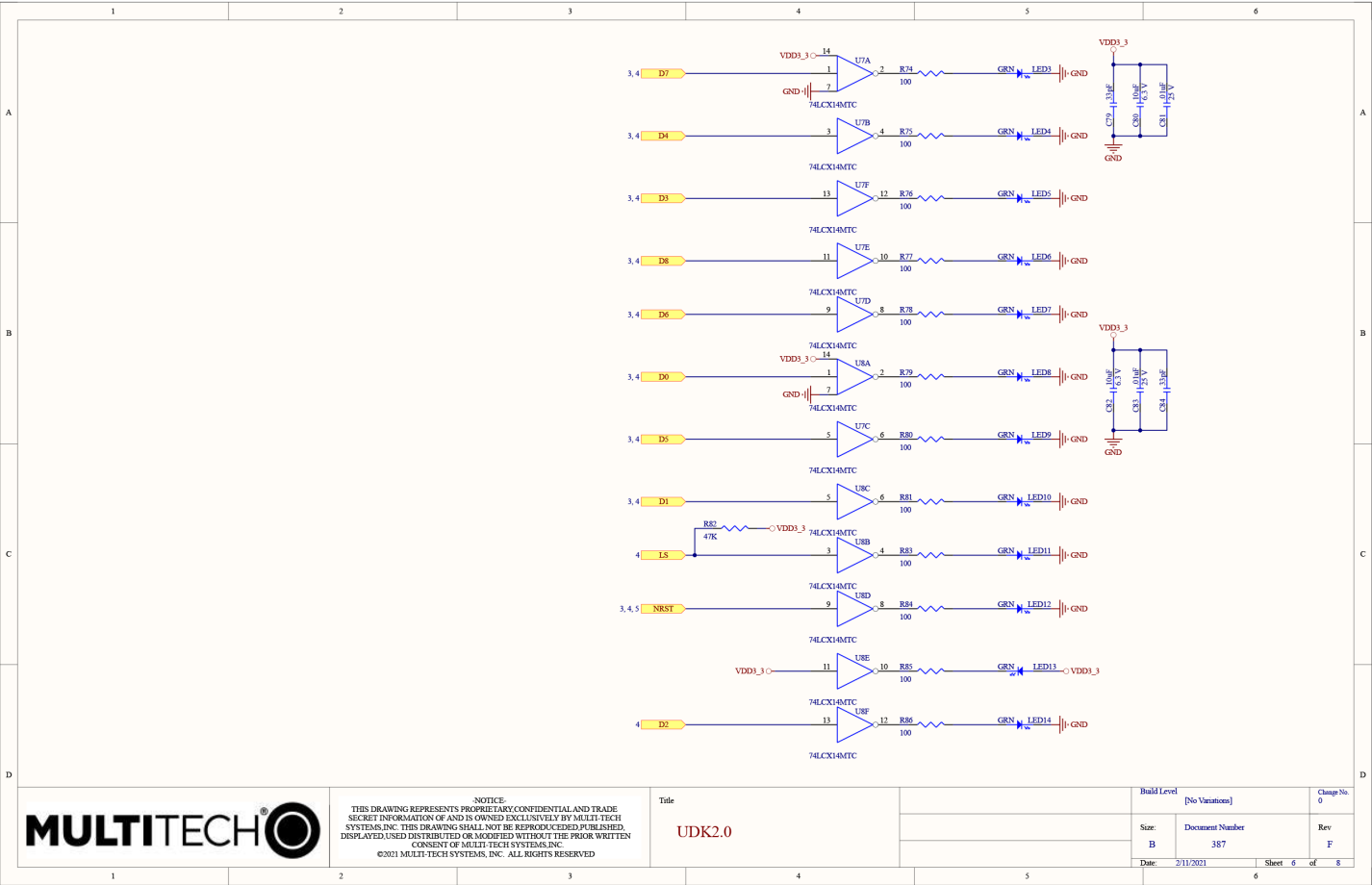
Schematics - Sockets



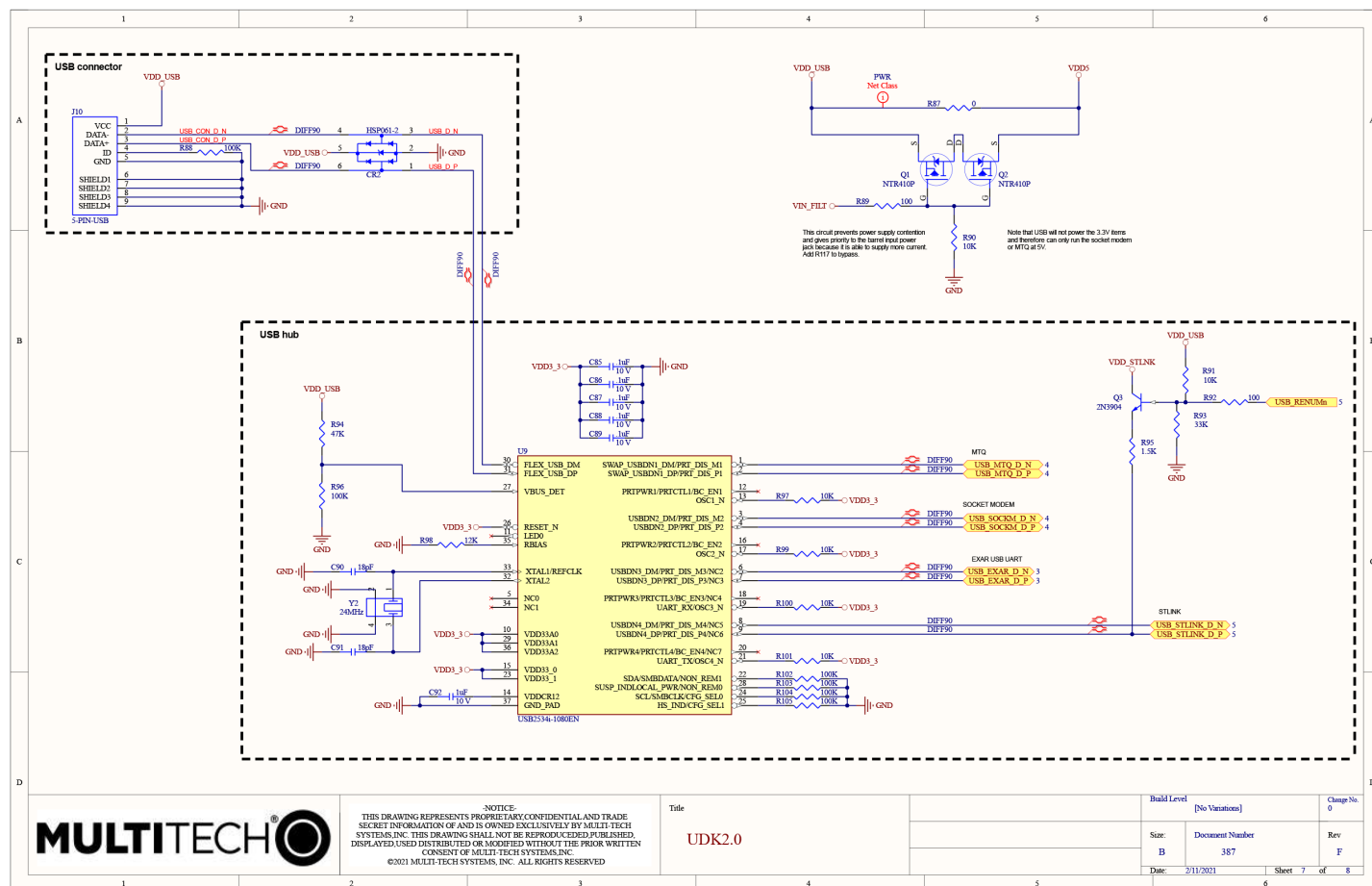
Schematics - ST Link



Schematics - LEDs



Universal Developer Kit MTUDK2-ST-CELL.R1 Developer Guide
S000779 Rev. 1.2



Chapter 5 Design Considerations

Noise Suppression Design

Adhere to engineering noise-suppression practices when designing a printed circuit board (PCB). Noise suppression is essential to the proper operation and performance of the device and surrounding equipment.

Any OEM board design must consider both on-board and off-board generated noise that can affect digital signal processing. Both on-board and off-board generated noise that is coupled on-board can affect interface signal levels and quality. Noise in frequency ranges that affect device performance is of particular concern.

On-board generated electromagnetic interference (EMI) noise that can be radiated or conducted off-board is equally important. This type of noise can affect the operation of surrounding equipment. Most local government agencies have certification requirements that must be met for use in specific environments.

Proper PC board layout (component placement, signal routing, trace thickness and geometry, and so on) component selection (composition, value, and tolerance), interface connections, and shielding are required for the board design to achieve desired device performance and to attain EMI certification.

Other aspects of proper noise-suppression engineering practices are beyond the scope of this guide. Consult noise suppression techniques described in technical publications and journals, electronics and electrical engineering text books, and component supplier application notes.

PC Board Layout Guideline

In a 4-layer design, provide adequate ground plane covering the entire board. In 4-layer designs, power and ground are typically on the inner layers. Ensure that all power and ground traces are 0.05 inches wide.

The recommended hole size for the device pins is 0.036 in. +/-0.003 in. in diameter. Use spacers to hold the device vertically in place during the wave solder process.

Electromagnetic Interference

The following guidelines are offered specifically to help minimize EMI generation. Some of these guidelines are the same as, or similar to, the general guidelines. To minimize the contribution of device-based design to EMI, you must understand the major sources of EMI and how to reduce them to acceptable levels.

- Keep traces carrying high frequency signals as short as possible.
- Provide a good ground plane or grid. In some cases, a multilayer board may be required with full layers for ground and power distribution.
- Decouple power from ground with decoupling capacitors as close to the device's power pins as possible.
- Eliminate ground loops, which are unexpected current return paths to the power source and ground.
- Locate high frequency circuits in a separate area to minimize capacitive coupling to other circuits.
- Locate cables and connectors to avoid coupling from high frequency circuits.
- Lay out the highest frequency signal traces next to the ground grid.
- If using a multilayer board design, make no cuts in the ground or power planes and be sure the ground plane covers all traces.
- Minimize the number of through-hole connections on traces carrying high frequency signals.

- Avoid right angle turns on high frequency traces. Forty-five degree corners are good; however, radius turns are better.
- On 2-layer boards with no ground grid, provide a shadow ground trace on the opposite side of the board to traces carrying high frequency signals. This will be effective as a high frequency ground return if it is three times the width of the signal traces.
- Distribute high frequency signals continuously on a single trace rather than several traces radiating from one point.

Electrostatic Discharge Control

Handle all electronic devices with precautions to avoid damage due to the static charge accumulation.

See the ANSI/ESD Association Standard (ANSI/ESD S20.20-1999) – a document “for the Development of an Electrostatic Discharge Control for Protection of Electrical and Electronic Parts, Assemblies and Equipment.” This document covers ESD Control Program Administrative Requirements, ESD Training, ESD Control Program Plan Technical Requirements (grounding/bonding systems, personnel grooming, protected areas, packaging, marking, equipment, and handling), and Sensitivity Testing.

MultiTech strives to follow these recommendations. Input protection circuitry is incorporated in MultiTech devices to minimize the effect of static buildup. Take precautions to avoid exposure to electrostatic discharge during handling.

MultiTech uses and recommends that others use anti-static boxes that create a faraday cage (packaging designed to exclude electromagnetic fields). MultiTech recommends that you use our packaging when returning a product and when you ship your products to your customers.

USB Design

MultiTech recommends that you review Intel's High Speed USB Platform Design Guidelines for information about USB signal routing, impedance, and layer stacking. Also:

- Shield USB cables with twisted pairs (especially those containing D+/D-).
- Use a single 5V power supply for USB devices. See Power Draw for current (ampere) requirements.
- Route D+/D- together in parallel with the trace spacing needed to achieve 90 ohms differential impedance for the USB pair and to maintain a 20 mil space from the USB pair and all other signals.
- If power is provided externally, use a common ground between the carrier board and the device.

Warranty

To read the warranty statement for your product, visit <https://www.multitech.com/warranty>.

Contact Information

General Information	info@multitech.com https://multitech.com/contact-us/
Sales	+1 (763) 785-3500 sales@multitech.com
Technical Support Portal	+1 (763) 717-5863 https://support.multitech.com
Website	www.multitech.com
World Headquarters	2205 Woodale Drive Mounds View, MN 55112 USA

Revision History

Revision Number	Description	Revision Date
1.2	Updated marketing brand.	January 2025
1.1	Updated Arduino pin definitions.	July 2024
1.0	Original publication.	September 2022