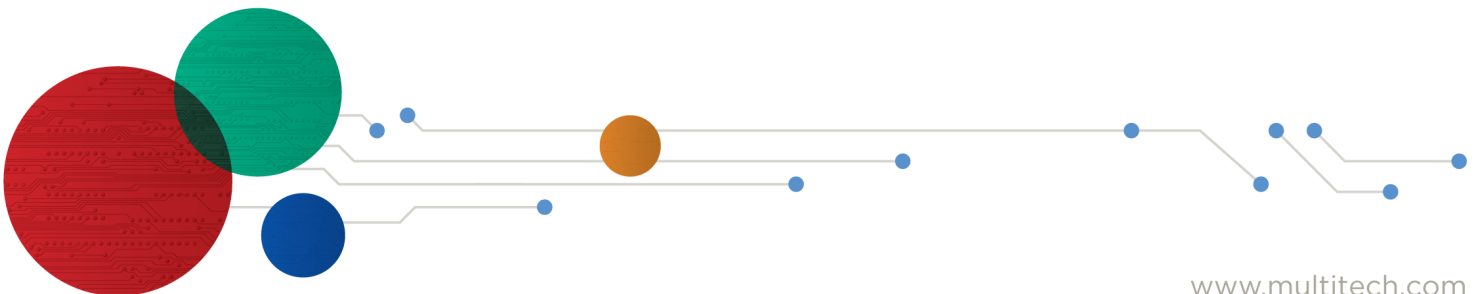


Universal Developer Kit 2.0

MTUDK2-ST-CELL.R1 Developer Guide



Universal Developer Kit Developer Guide

Models: MTUDK2-ST-CELL.R1

Part Number: S000779, Version 1.0

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 © 2022 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 and Registered Trademarks

MultiTech, MultiConnect, SocketModem, and the MultiTech logo are registered 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.

The MultiTech products and the final application of the MultiTech products should be thoroughly tested to ensure the functionality of the MultiTech products as used in the final application. The designer, manufacturer and reseller has the sole responsibility of ensuring that any end user product into which the MultiTech product is integrated operates as intended and meets its requirements or the requirements of its direct or indirect customers. MultiTech has no responsibility whatsoever for the integration, configuration, testing, validation, verification, installation, upgrade, support or maintenance of such end user product, or for any liabilities, damages, costs or expenses associated therewith, except to the extent agreed upon in a signed written document. To the extent MultiTech provides any comments or suggested changes related to the application of its products, such comments or suggested changes is performed only as a courtesy and without any representation or warranty whatsoever.

Contacting MultiTech

Country	Sales		Support	
Europe, Middle East, Africa:	sales@multitech.co.uk	+ (44) 118 959 7774	support@multitech.co.uk	+ (44) 118 959 7774
U.S., Canada, all others:	sales@multitech.com	(800) 328-9717 or (763) 785-3500	support@multitech.com	(800) 972-2439 or (763) 717-5863

Website

www.multitech.com

Knowledge Base

For immediate access to support information and resolutions for MultiTech products, visit <http://www.multitech.com/kb.go>.

Support Portal

To create an account and submit a support case directly to our technical support team, visit: <https://support.multitech.com>.

Warranty

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

World Headquarters

Multi-Tech Systems, Inc.
2205 Woodale Drive, Mounds View, MN 55112

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	13
Installing a SIM Card on a SocketModem	13
Installing a SIM Card on a DragonFly	13
Installing a SIM Card	14
Installing a SocketModem on the Developer Board	15
Installing a Dragonfly on the Developer Board.....	17
Installing a Dragonfly Nano on the Developer Board	19
Arduino Shield.....	20
Installing an Arduino Shield with a Dragonfly.....	20
Dragonfly Arduino Pins	22
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

Chapter 1 – Developer Kit Introduction

Overview

The MTUDK2-ST-CELL.R1 (UDK2.R1) Universal Developer Kit supports development with cellular SocketModem (MTSMC), Dragonfly (MTQ), and Dragonfly Nano (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 or Dragonfly 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 through your model's pages at:

- [SocketModem](#)
- [Dragonfly](#)
- [Dragonfly Nano](#)

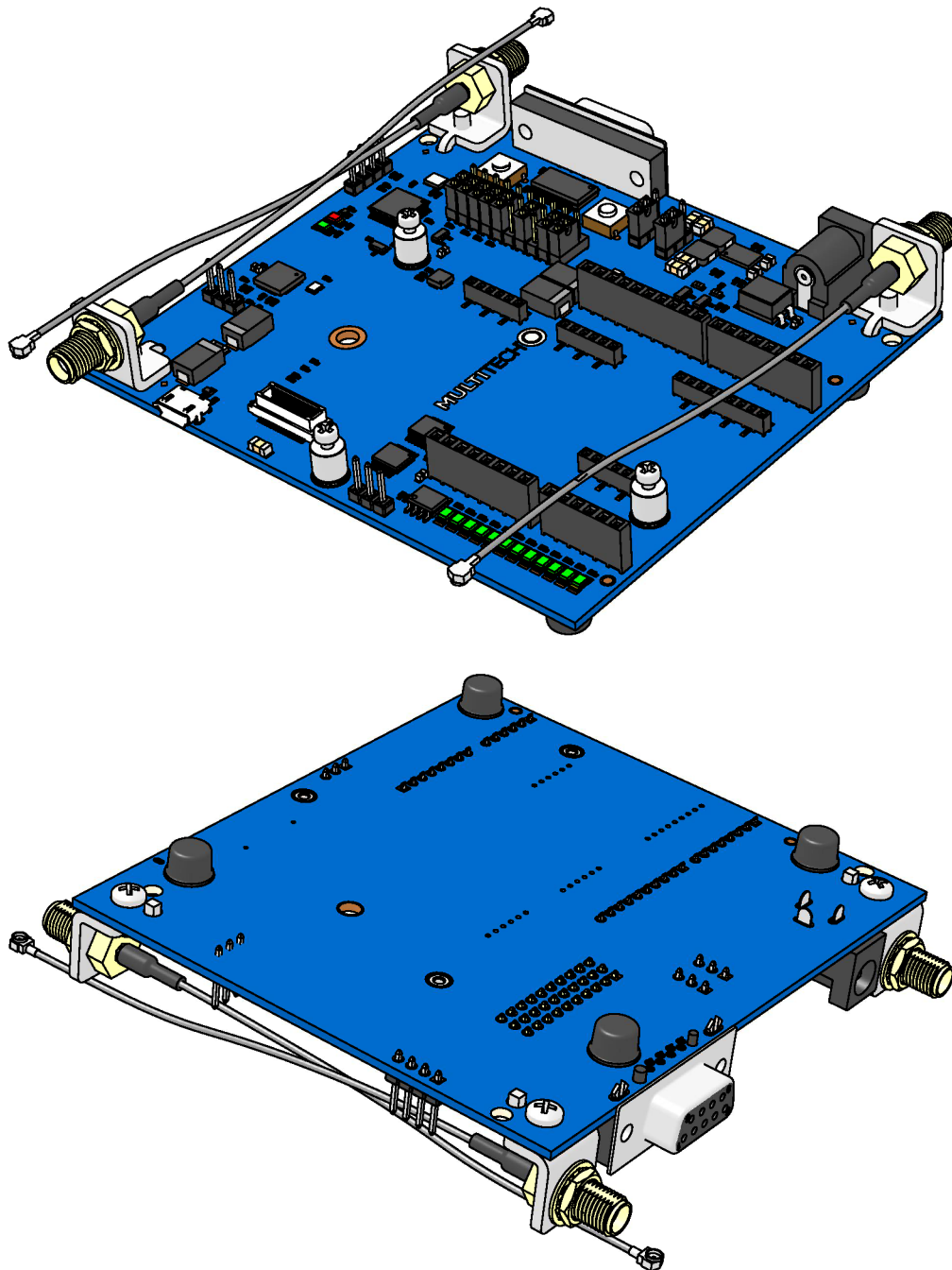
MTUDK2-ST-CELL Developer Kit Contents

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

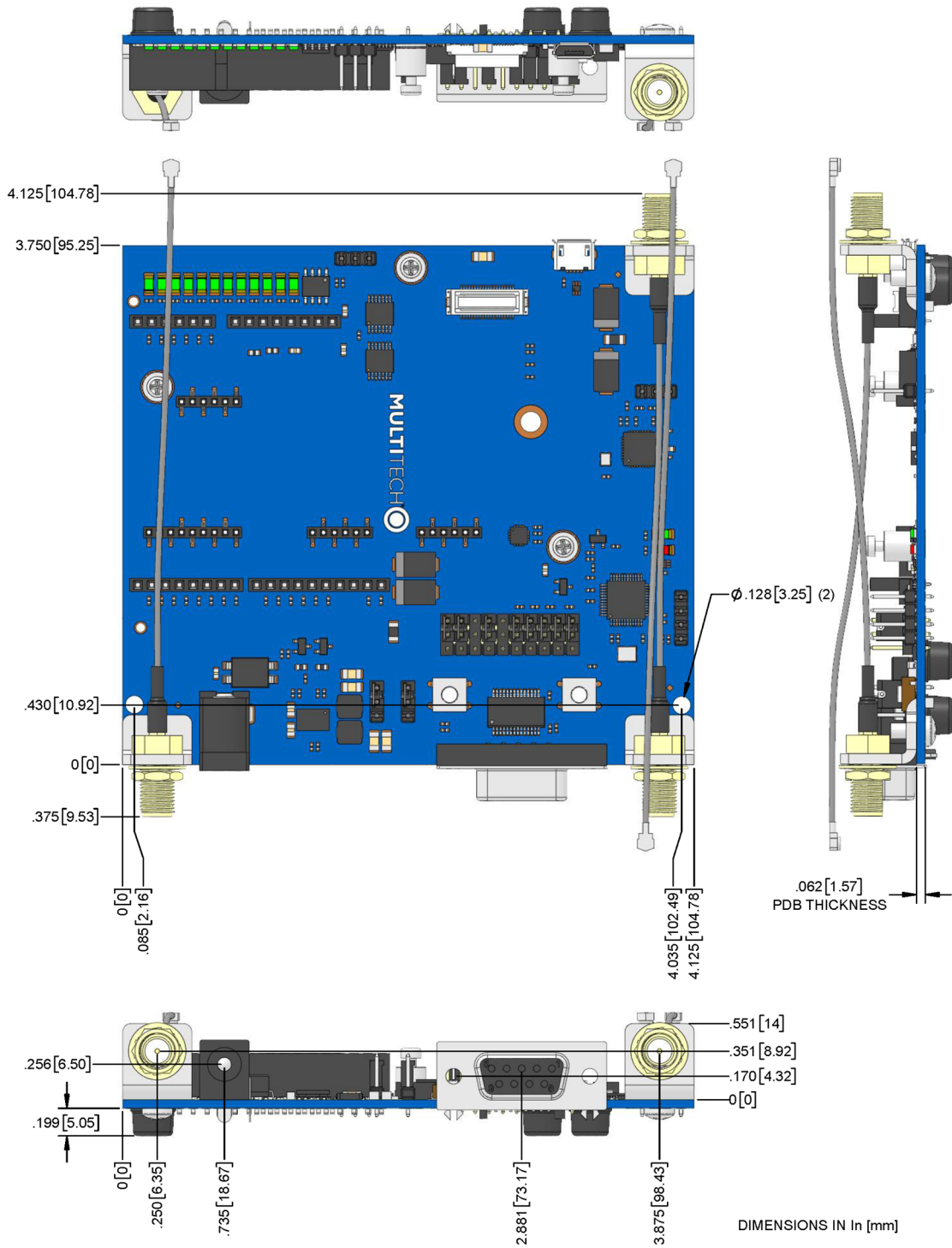
Developer Board	1 - MTUDK 2.0 Cell Developer Board
Power Supply	1 - 100-240V 9V-1.7A power supply with removable blades
	1 - NAM blade/plug,
	1 - EURO blade/plug
	1 - UK blade/plug
	1 - AU/NZ blade/plug
Cables	1 - Micro USB Cable
	3 - SMA-U.FL Antenna Cables (attached to developer board)
Antennas	1 -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

Chapter 2 – Board Components

Developer Board

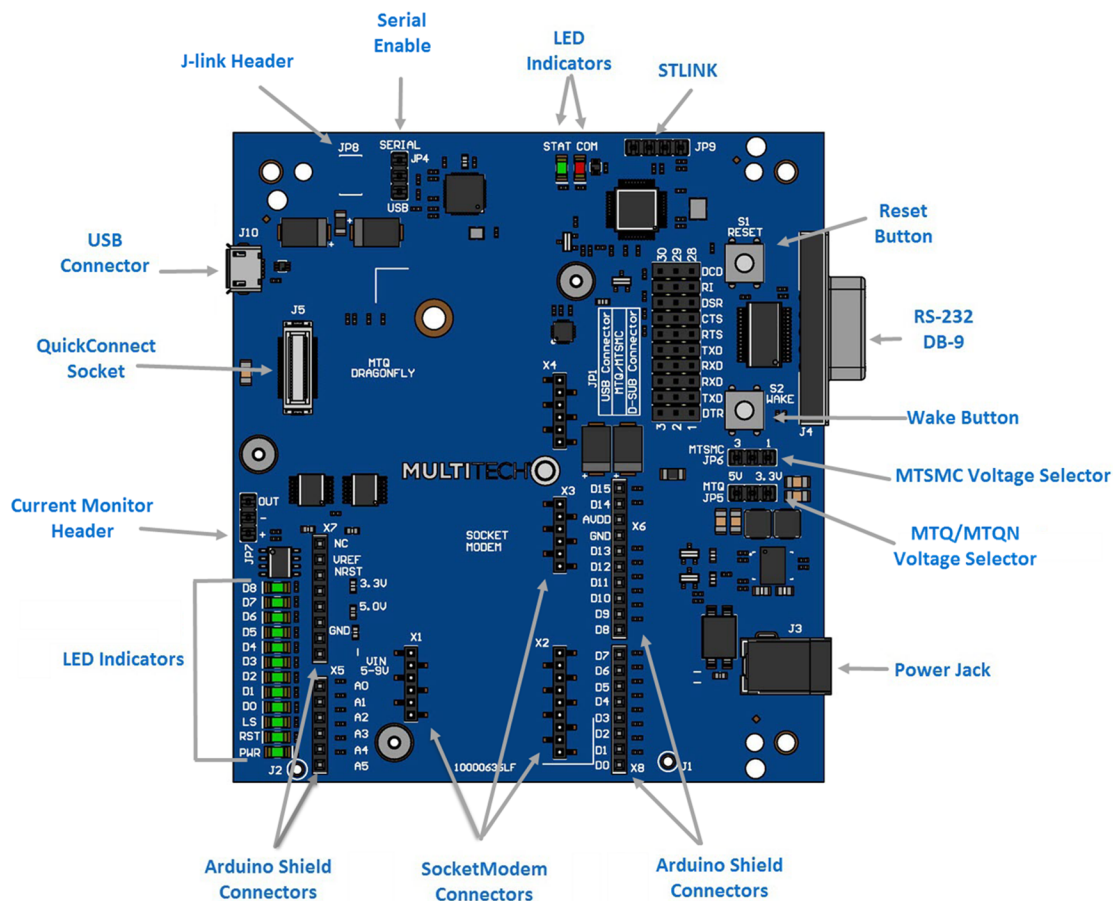


Developer Board Dimensions



UDK 2.0

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. 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 SocketModems.)
JP1	3-row connector: USB Connector, MTQ/MTSMC, D-SUB Connector. Refer to JP1 USB Connector 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, USB Connector.
X2	SocketModem Serial Connector.
X3	SocketModem, GPIO (not connected).
X4	SocketModem 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.

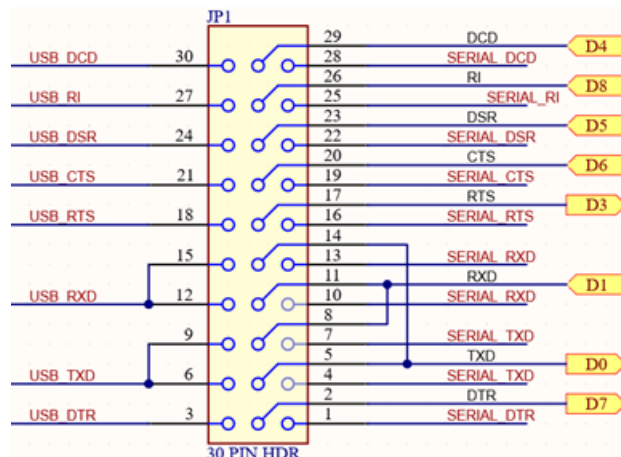
The switching frequency is set to 1.05MHz, but is adjustable from 300 KHz to 2MHz. Any changes to the frequency may require inductor or capacitor changes for proper operation.

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 UDK2.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

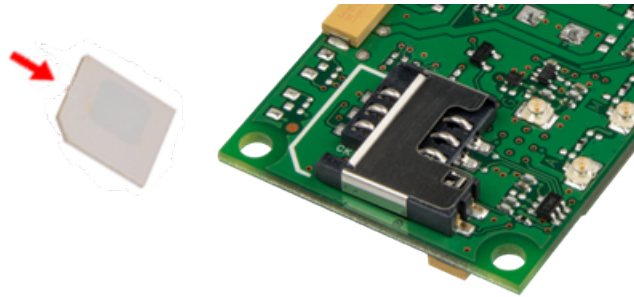
Installing a SIM Card

Installing a SIM Card on a SocketModem

Note: When using the SocketModem with a developer board, mount the SocketModem 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 and slide the SIM card completely into the SIM holder.

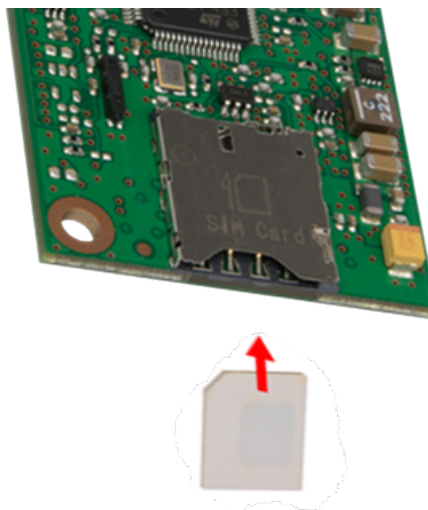


Installing a SIM Card on a DragonFly

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

To install the SIM card:

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



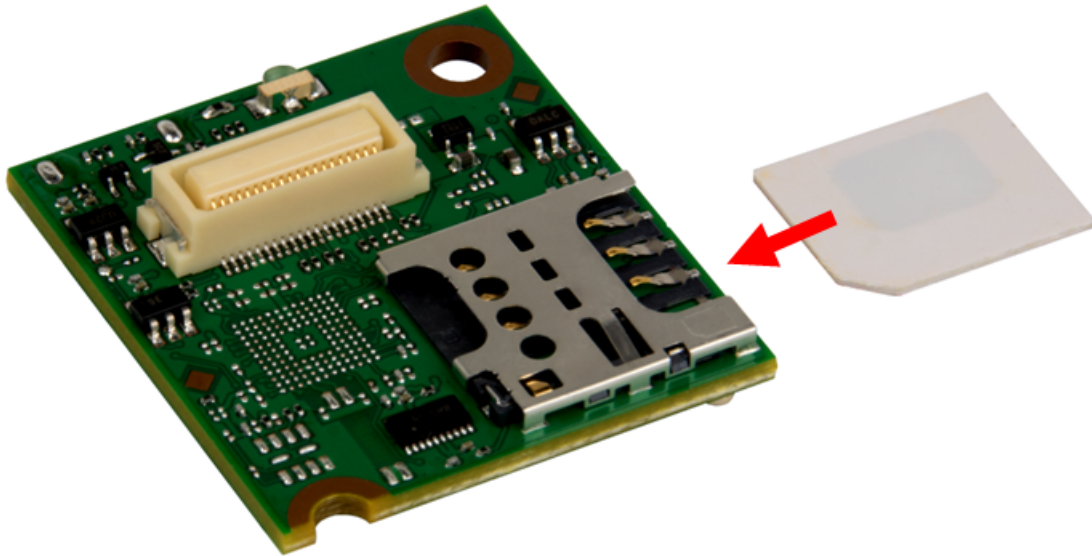
Installing a SIM Card

Note: When using the Dragonfly™ Nano with a developer board, install the SIM card before mounting the Dragonfly™ Nano on the developer board.

Note: All Dragonfly™ Nano models require the use of a Micro SIM (3FF) card.

To install the SIM card:

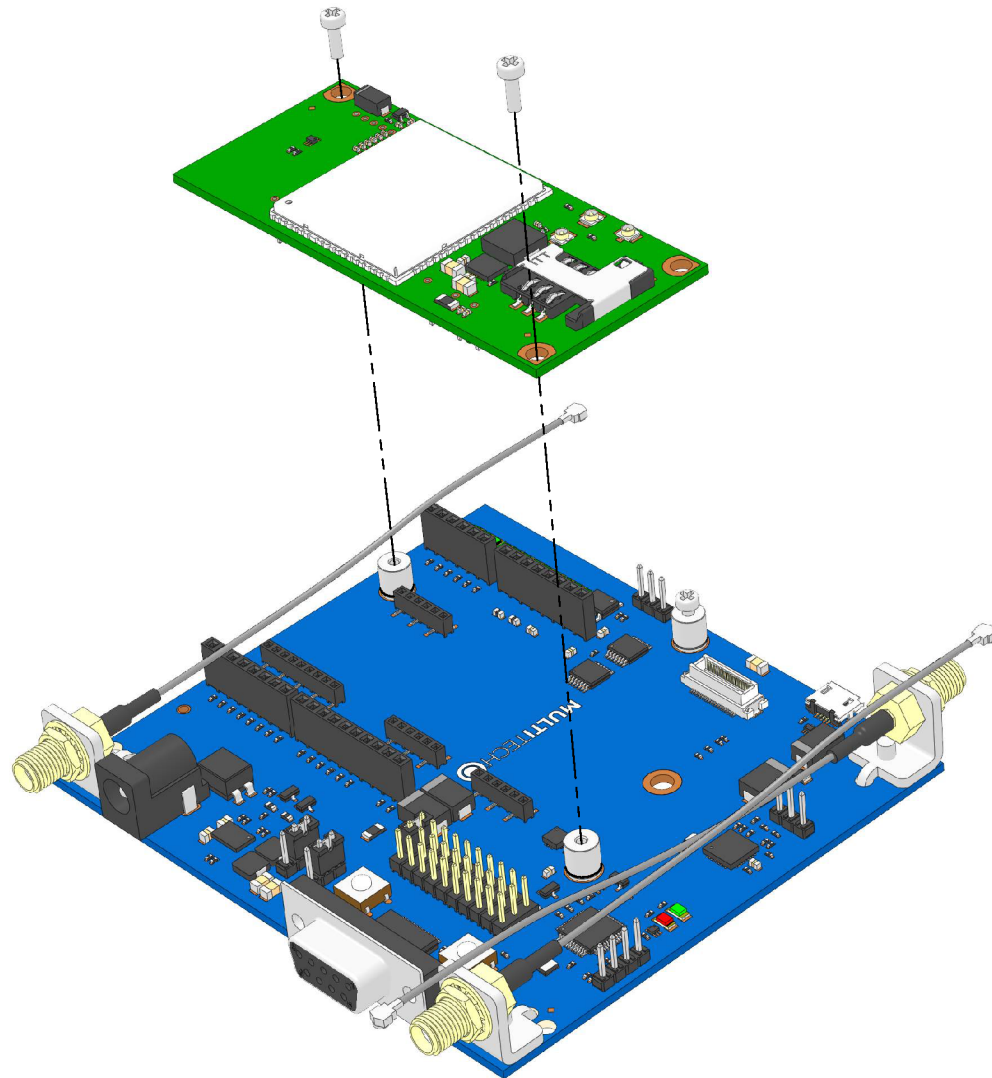
- Refer to the image below on how to install the SIM card.
- Slide the SIM card completely into the SIM holder.



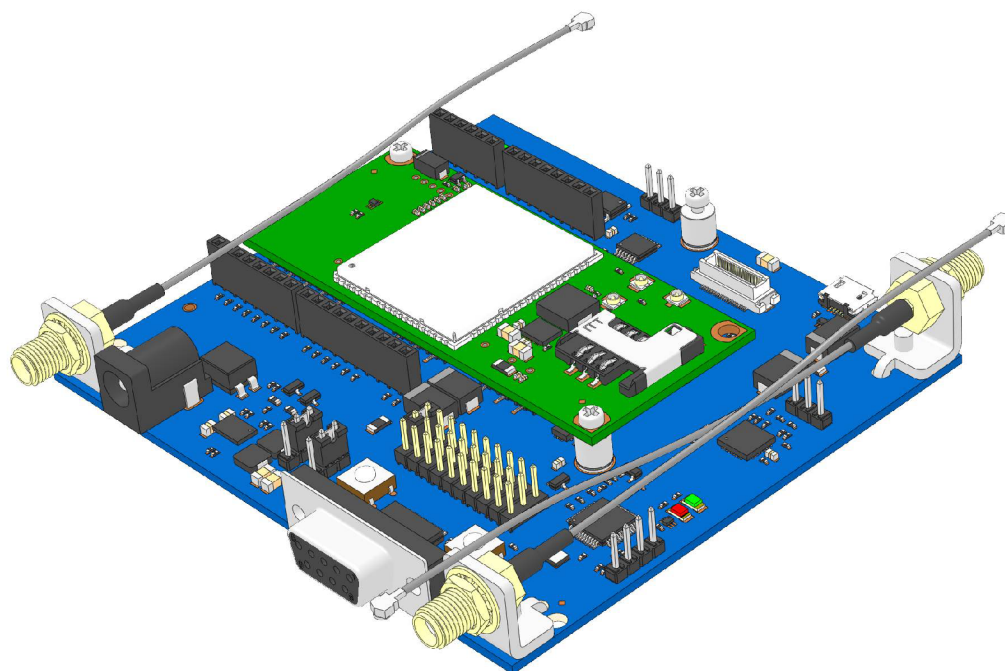
Installing a SocketModem on the Developer Board

To install a SocketModem:

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



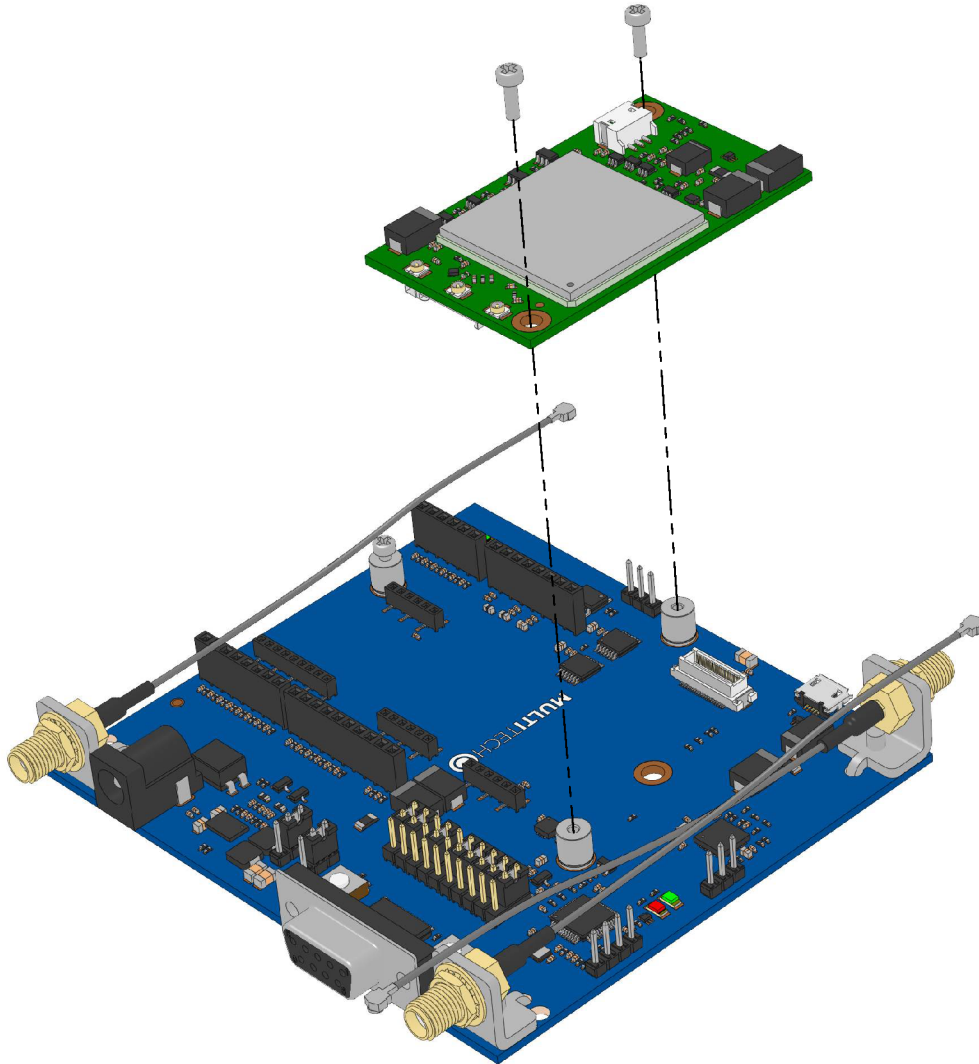
3. Secure the SocketModem with the screws you removed in Step 1.



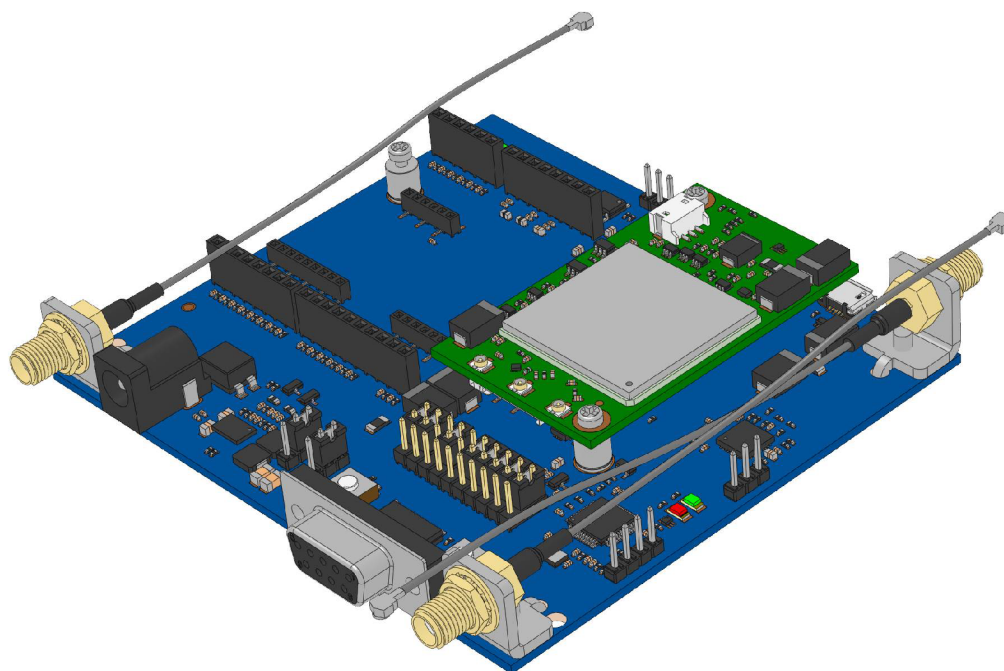
Installing a Dragonfly on the Developer Board

To install a Dragonfly:

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



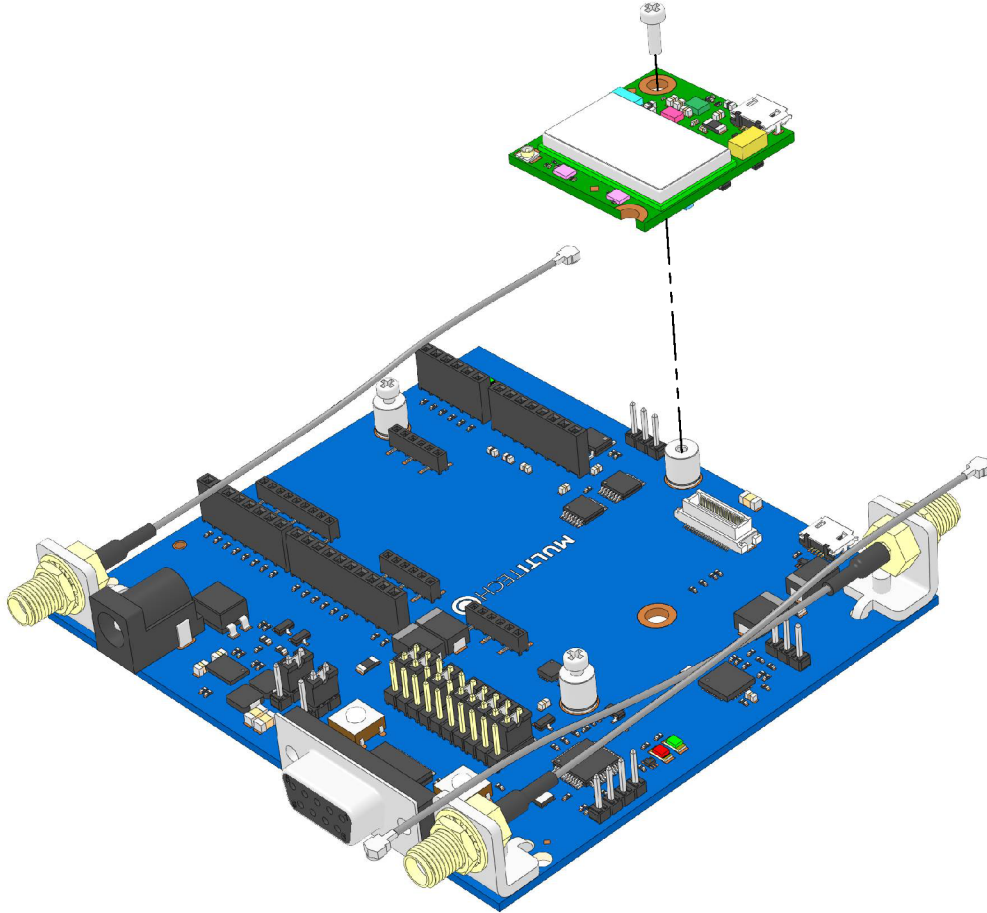
3. Secure the Dragonfly with the screws you removed in Step 1.



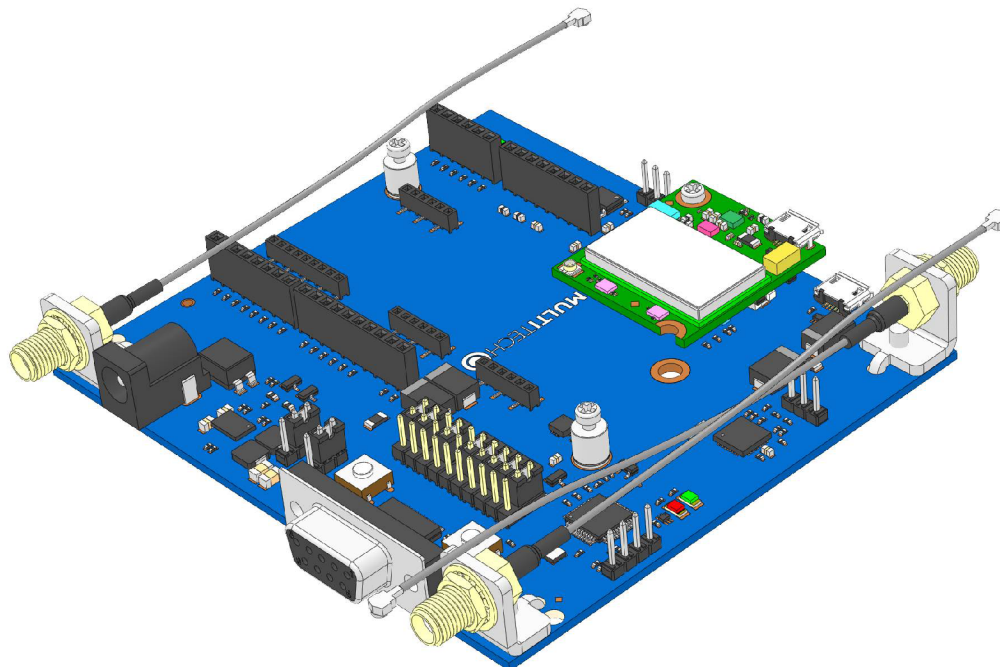
Installing a Dragonfly Nano on the Developer Board

To install a Dragonfly Nano:

1. Remove the screws from the developer board.
2. Align the Dragonfly Nano on the developer board as shown.



3. Secure the Dragonfly Nano with the screws you removed in Step 1.



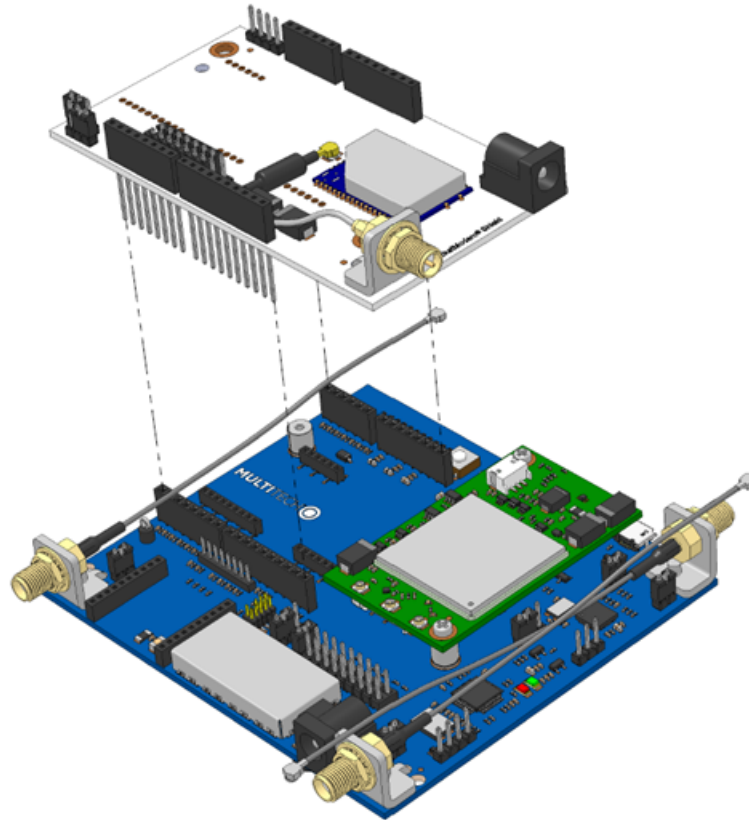
Arduino Shield

Installing an Arduino Shield with a Dragonfly

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

To use an Arduino Shield with a Dragonfly

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



Dragonfly Arduino Pins

Signals (B01 & B02)		Arduino Shield		Signals (STM32F411RE processor- B01 only)	
			D15	X6	D15 (PB8)
			D14		D14 (PB9)
			AVDD		NC
			GND		Ground
	X9	NC	D13	X8	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
A0 (PC2)	X7	A0	D5	X8	D5 (PA9)
A1 (PC0)		A1	D4		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)

Dragonfly Nano Arduino Pins

Signals (B01 & B02)		Arduino Shield		Signals (STM32L471QG processor- B01 only)	
			D15	X6	D15 (PB8)
			D14		D14 (PB9)
			AVDD		NC
			GND		Ground
	X9	NC	D13	X8	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		D7 (PG7)
					D6
A0 (PC2)	X7	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 NAM blade



Power Supply with UK blade

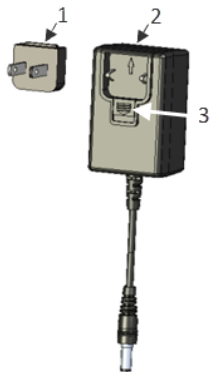


Power Supply with AU-NZ blade

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.



- 1 - Latch
- 2 - Notch
- 3 - Sliding lock

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:

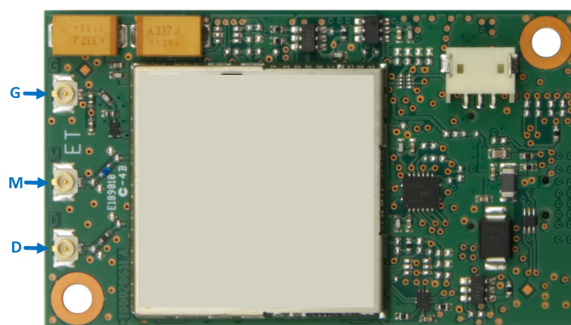
1. Determine which SMA connector you want to use for the antenna.
2. Finger tighten the antenna to the SMA connector.
3. Attach the U.FL connector from the cable to the connector on the device.



G = GPS (may not apply to your device)

M = Main

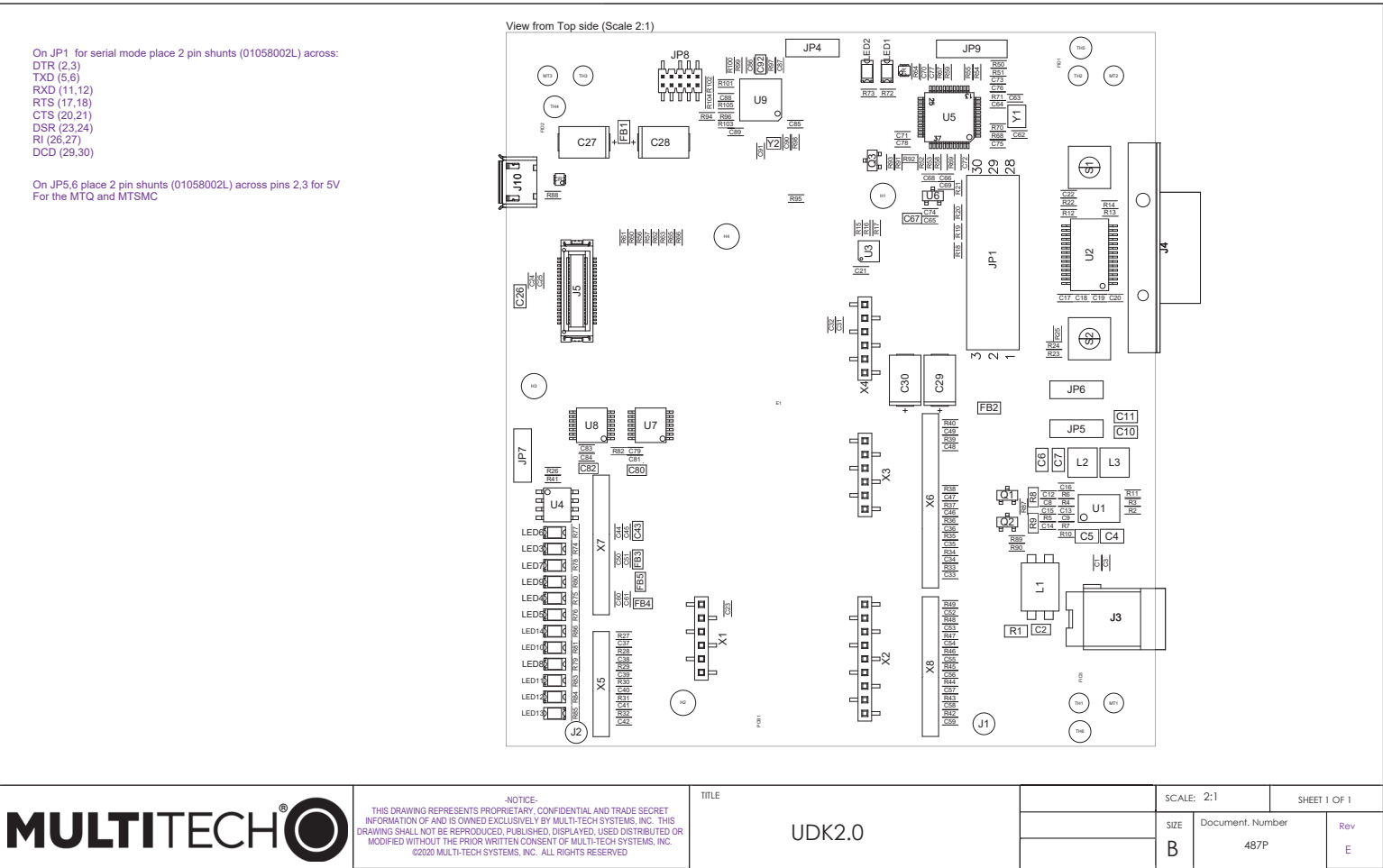
D = Diversity



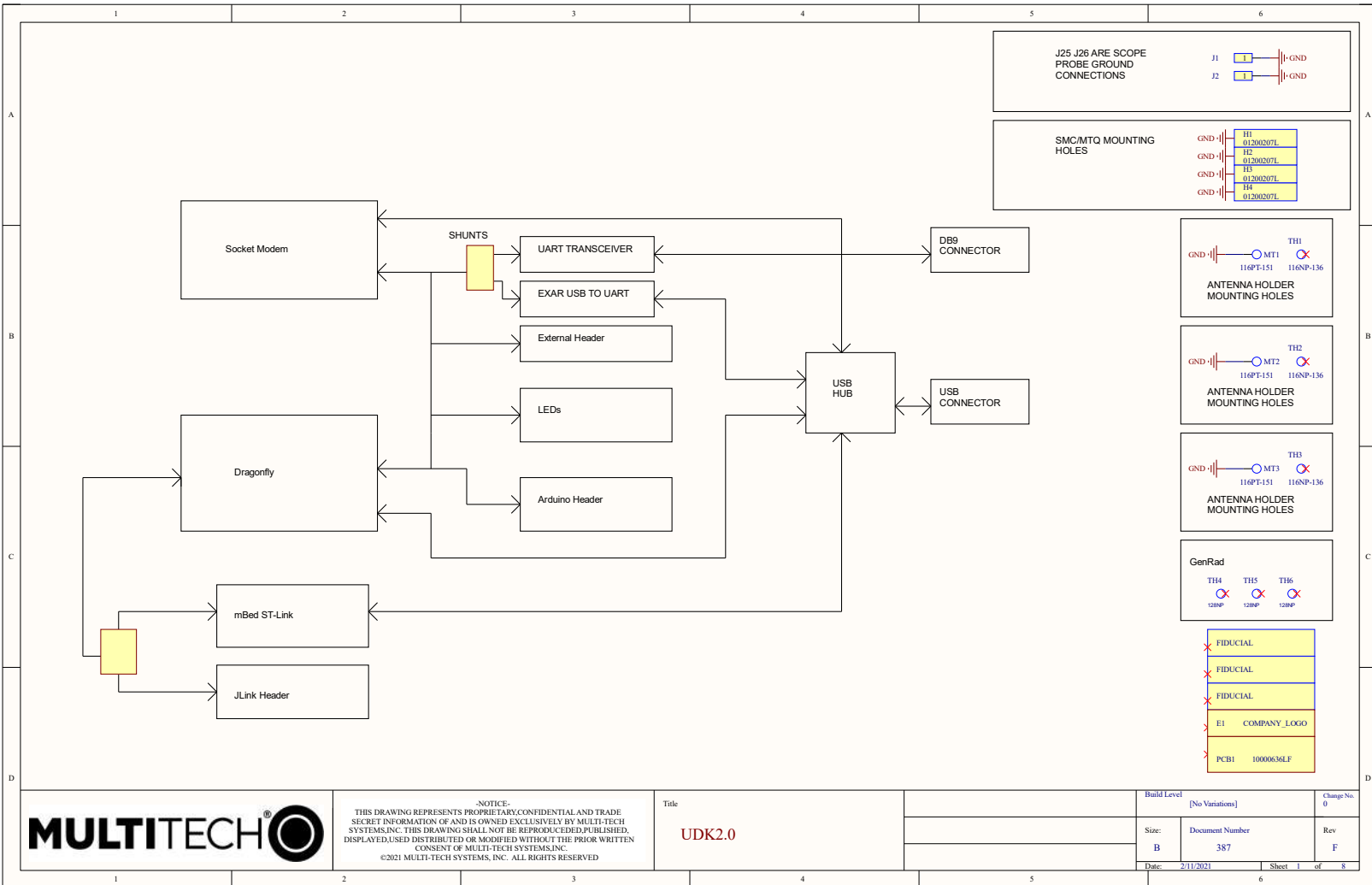


Chapter 4 – Diagrams and Schematics

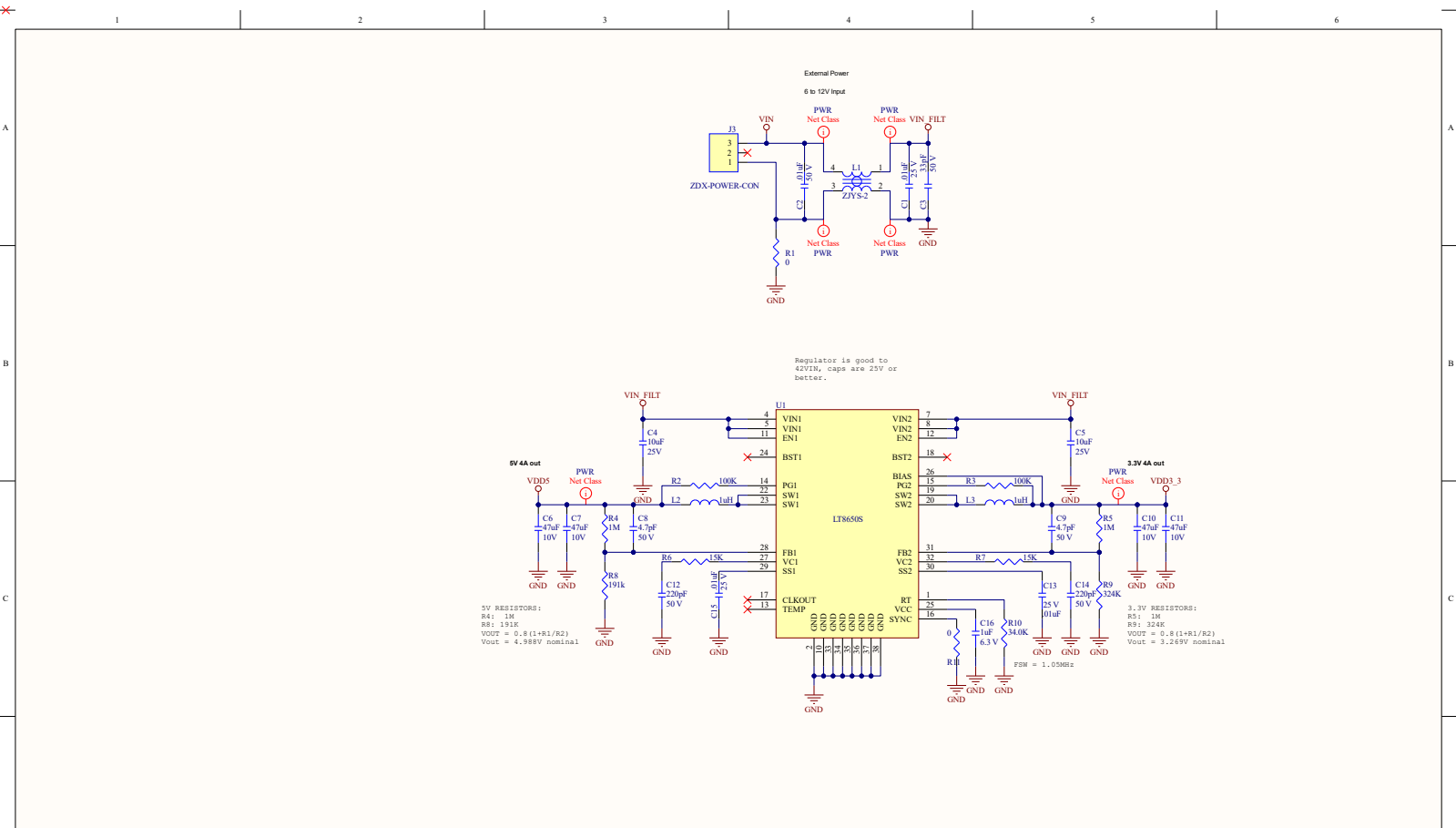
Assembly Diagram



Block Diagram



Schematics - Power



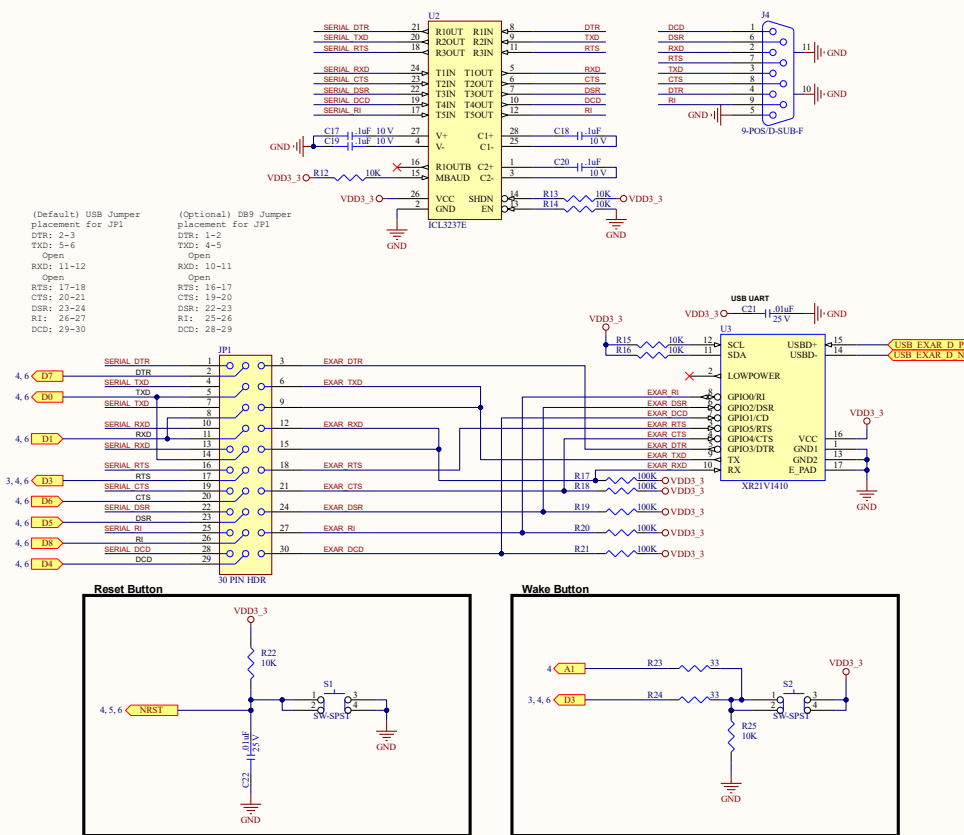
NOTICE:
THIS DRAWING REPRESENTS PROPRIETARY, CONFIDENTIAL AND TRADE SECRET INFORMATION OF AND IS OWNED EXCLUSIVELY BY MULTI-TECH SYSTEMS, INC. THIS DRAWING SHALL NOT BE REPRODUCED, PUBLISHED, DISPLAYED, USED, DISTRIBUTED OR MODIFIED WITHOUT THE PRIOR WRITTEN CONSENT OF MULTI-TECH SYSTEMS, INC.
©2021 MULTI-TECH SYSTEMS, INC. ALL RIGHTS RESERVED

Title
UDK2.0

Build Level	[No Variations]	Change No.	0
Size:	Document Number	Rev	F
B	387		
Date:	2/11/2021	Sheet	2 of 8

Schematics - RS-232

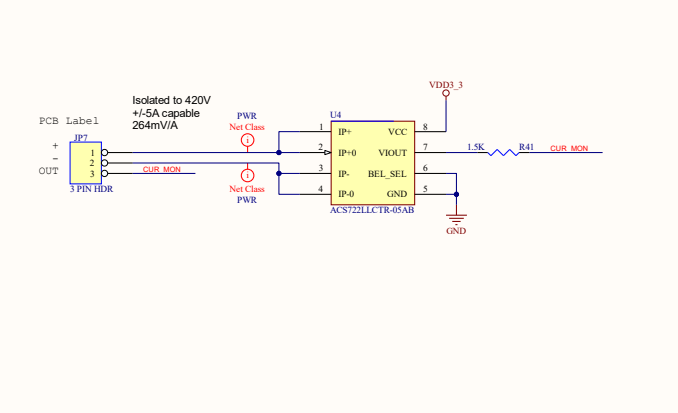
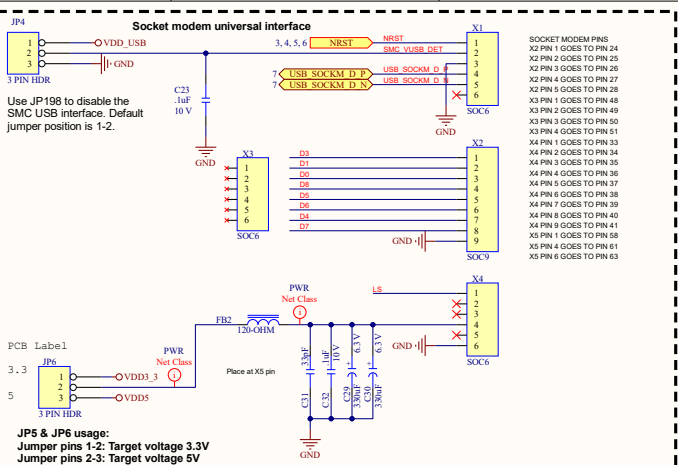
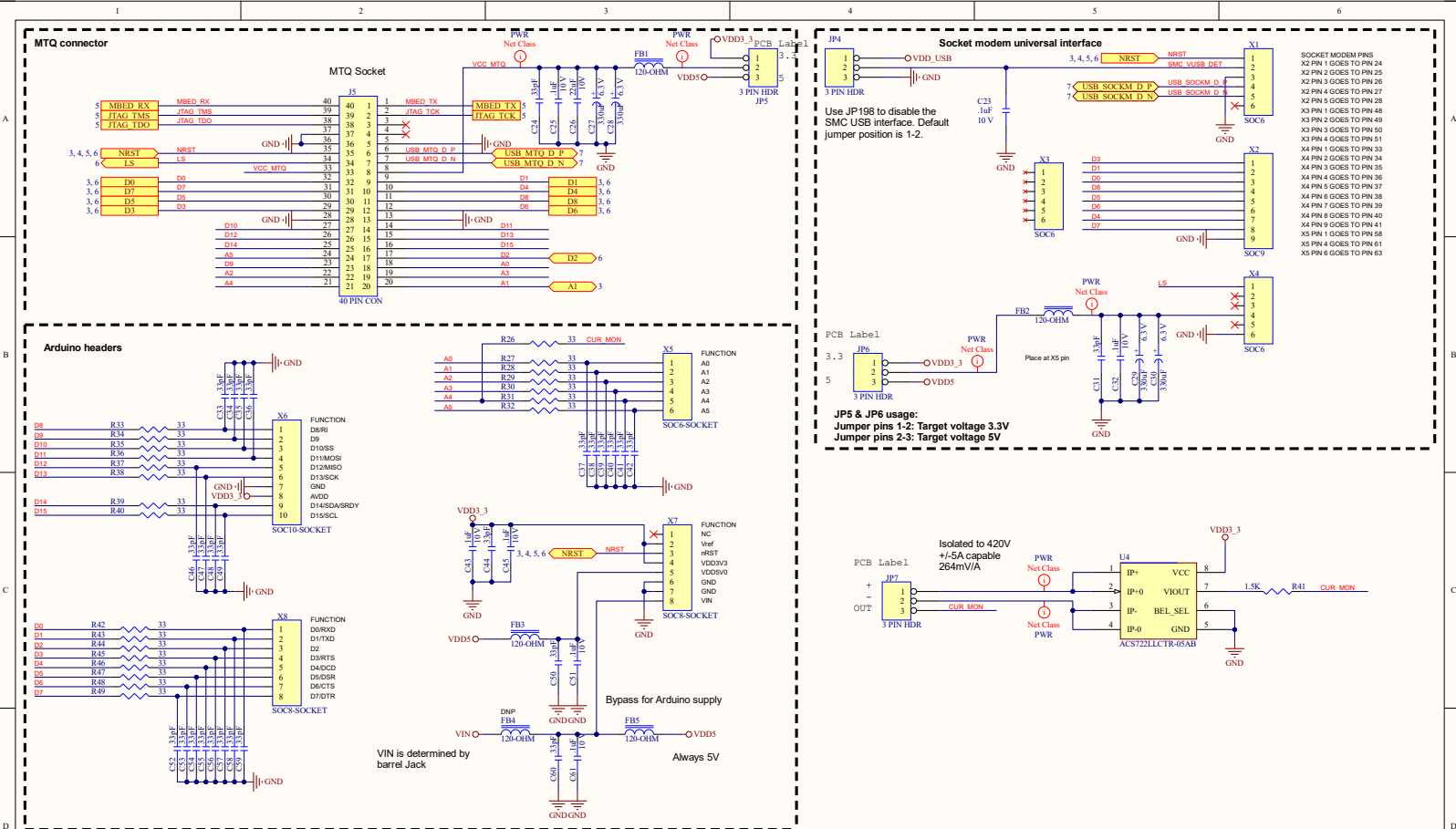
Please add this text in layout next to pins on JP1
 DB9 USB
 DTR TXD
 RXD RSD
 RTS
 CTS
 DSR
 RI
 DCD



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, DISTRIBUTED OR MODIFIED WITHOUT THE PRIOR WRITTEN CONSENT OF MULTI-TECH SYSTEMS, INC.
 ©2021 MULTI-TECH SYSTEMS, INC. ALL RIGHTS RESERVED

Title		Build Level		Change No.
UDK2.0		[No Variations]		0
Size:	Document Number	Rev		
B	387	F		
Date:	2/11/2021	Sheet	3	of 8

Schematics - Sockets

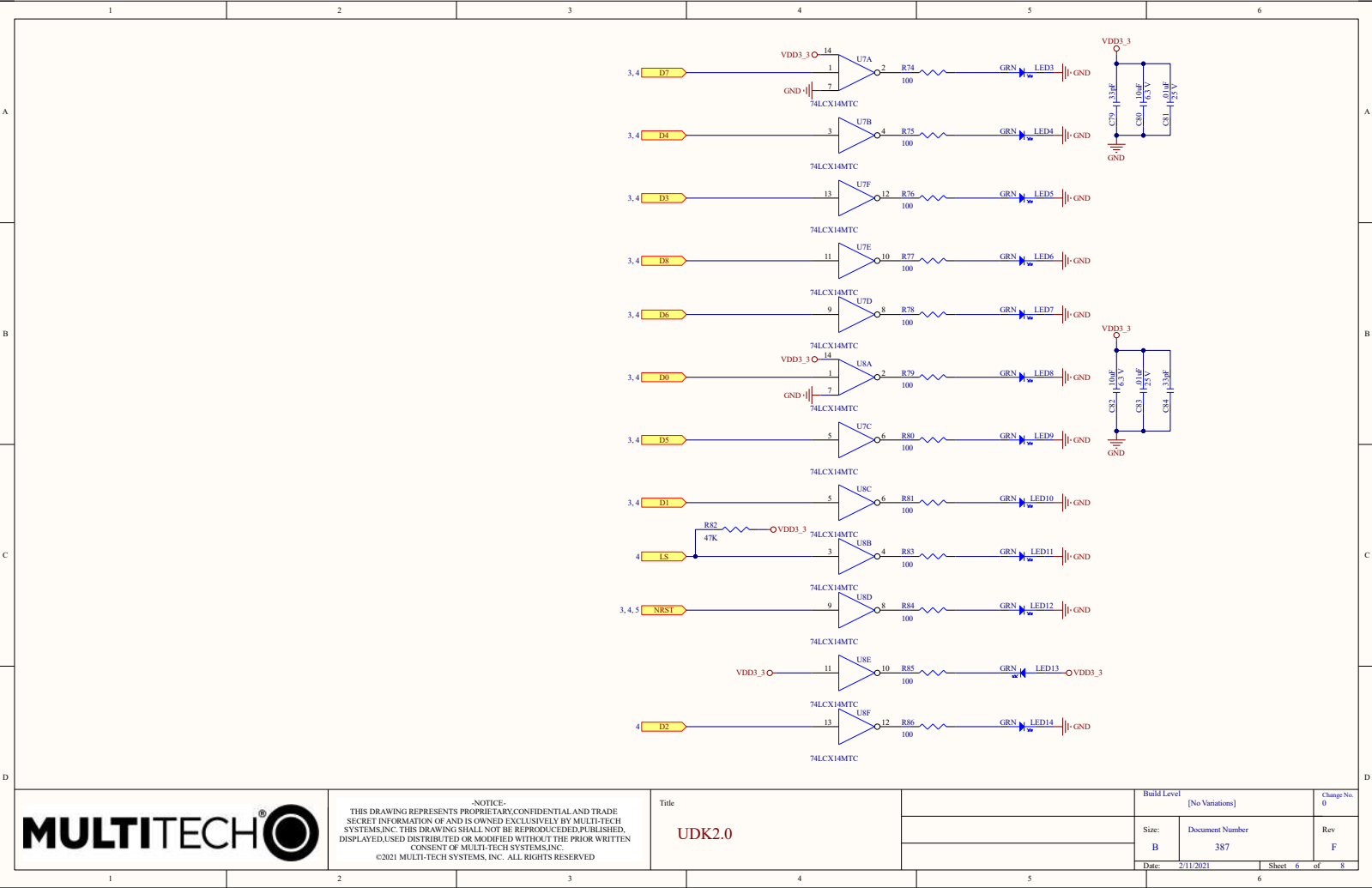


NOTICE:
 THIS DRAWING REPRESENTS PROPRIETARY, CONFIDENTIAL AND TRADE SECRET INFORMATION OF AND IS OWNED EXCLUSIVELY BY MULTI-TECH SYSTEMS, INC. THIS DRAWING SHALL NOT BE REPRODUCED, PUBLISHED, DISPLAYED, USED, DISTRIBUTED OR MODIFIED WITHOUT THE PRIOR WRITTEN CONSENT OF MULTI-TECH SYSTEMS, INC.
 ©2021 MULTI-TECH SYSTEMS, INC. ALL RIGHTS RESERVED

Title
UDK2.0

Build Level	[No Variations]	Change No.	0
Size:	Document Number	Rev	F
B	387		
Date:	2/11/2021	Sheet	4 of 8

Schematics - LEDs

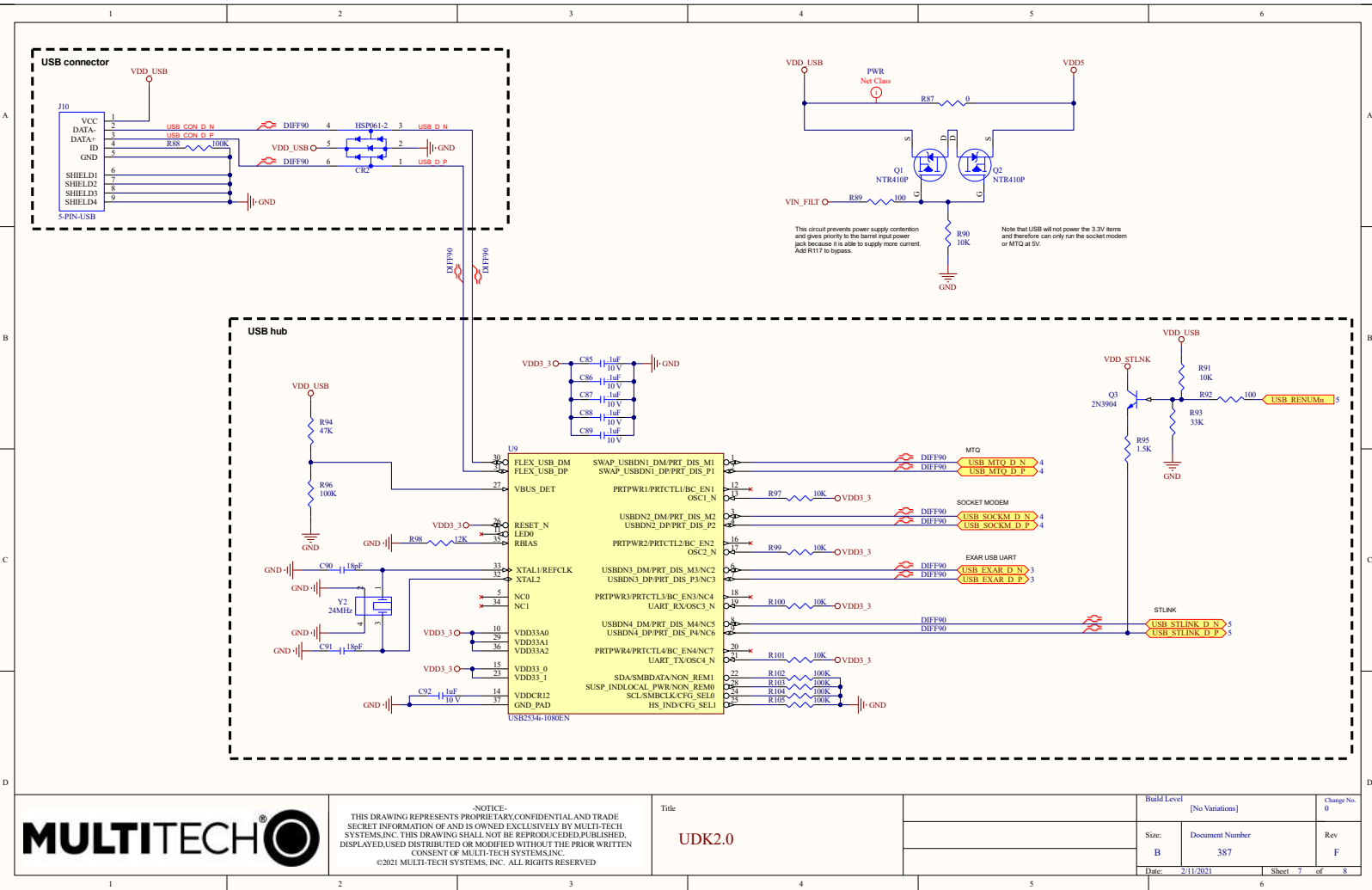


NOTICE:
THIS DRAWING REPRESENTS PROPRIETARY, CONFIDENTIAL AND TRADE
SECRET INFORMATION OF AND IS OWNED EXCLUSIVELY BY MULTI-TECH
SYSTEMS, INC. THIS DRAWING SHALL NOT BE REPRODUCED, PUBLISHED,
DISPLAYED, USED, DISTRIBUTED OR MODIFIED WITHOUT THE PRIOR WRITTEN
CONSENT OF MULTI-TECH SYSTEMS, INC.
©2021 MULTI-TECH SYSTEMS, INC. ALL RIGHTS RESERVED

Title
UDK2.0

Build Level	[No Variations]	Change No.	0		
Size:	B	Document Number	387	Rev	F
Date:	2/11/2021	Sheet	6	of	8

Schematics - USB



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, DISTRIBUTED OR MODIFIED WITHOUT THE PRIOR WRITTEN CONSENT OF MULTI-TECH SYSTEMS, INC.
©2021 MULTI-TECH SYSTEMS, INC. ALL RIGHTS RESERVED

Title
UDK2.0

Build Level	[No Variations]	Change No.	0
Size:	Document Number	Rev	F
B	387		
Date:	2/11/2021	Sheet	7 of 8

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 modem 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 modem 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 modem 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.
- Decouple the telephone line cables at the telephone line jacks. Typically, use a combination of series inductors, common mode chokes, and shunt capacitors. Methods to decouple telephone lines are similar to decoupling power lines; however, telephone line decoupling may be more difficult and deserves additional attention. A commonly used design aid is to place footprints for these components and populate as necessary during performance/EMI testing and certification.
- Decouple the power cord at the power cord interface with decoupling capacitors. Methods to decouple power lines are similar to decoupling telephone lines.

- 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.