

Universal Developer Kit 2.0

MTUDK-ST-Cell Developer Guide



Universal Developer Kit 2.0 Developer Guide

Models: MTUDK-ST-Cell

Part Number: S000610, Version 2.6

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 © 2018 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

Knowledge Base

The Knowledge Base provides immediate access to support information and resolutions for all 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>.

Support

Business Hours: M-F, 8am to 5pm CT

Country	By Email	By Phone
Europe, Middle East, Africa:	support@multitech.co.uk	+(44) 118 959 7774
U.S., Canada, all others:	support@multitech.com	(800) 972-2439 or (763) 717-5863

Warranty

To read the warranty statement for your product, visit www.multitech.com/warranty.go. For other warranty options, visit www.multitech.com/es.go.

World Headquarters

Multi-Tech Systems, Inc.
 2205 Woodale Drive, Mounds View, MN 55112
 Phone: (800) 328-9717 or (763) 785-3500
 Fax (763) 785-9874

Contents

Chapter 1 – Developer Kit Introduction.....	4
Overview	4
Features	4
Device Specific Documentation	4
MTUDK2-ST-CELL Developer Kit Contents	4
Chapter 2 – Board Components	6
Developer Board	6
Developer Board Connectors	7
Board Components	7
LED Indicators	8
Chapter 3 – Installation and Operation	9
Installing a SIM Card	9
Installing a SIM Card on a SocketModem	9
Installing a SIM Card on a DragonFly	9
Installing a SocketModem on the Developer Board	10
Installing a Dragonfly on the Developer Board.....	11
Arduino Shield.....	12
Installing an Arduino Shield with a Dragonfly.....	12
Dragonfly Arduino Pins	13
mDot Arduino Pins.....	15
Installing an Arduino Shield with a SocketModem	16
Attaching Power Supply Blades	17
Power Supply and Blades.....	17
Attaching the Blades	17
SMA to U.FL Cables	18
Connecting an Antenna through the Developer Board Connectors.....	18
Chapter 4 – Block Diagram and Schematics.....	19
Block Diagram	20
Schematics	21
Chapter 5 – Design Considerations.....	28
Noise Suppression Design	28
PC Board Layout Guideline	28
Electromagnetic Interference	28
Electrostatic Discharge Control.....	29
USB Design	29

Chapter 1 – Developer Kit Introduction

Overview

The MTUDK2-ST-Cell Universal Developer Kit supports development with cellular SocketModem, Dragonfly (including Dragonfly Nano), and mDot devices. 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: Use this Developer Guide for developing with all SocketModem or Dragonfly models. If developing with an mDot, use the MultiConnect® mDot™ Developer Guide, which includes device and developer board information specific to mDots. Go to the mDot Developer Kit page at <http://www.multitech.com/models/94558010LF>.

Warning: Do not use an mDot and a Dragonfly or Socketmodem on this board at the same time. Doing so may damage both devices.

Features

- 5V-9V power input
- Selectable 3.3V or 5V on board power supply for mDot and SocketModem and 5V only for Dragonfly models
- USB and serial interfaces
- USB port for mbed development environment
- Arduino shield socket

Device Specific Documentation

Refer to the Device Guide for your SocketModem or Dragonfly model for specifications, pin information, mechanical drawings, labeling, regulatory information, and other model specific details.

MTUDK2-ST-CELL Developer Kit Contents

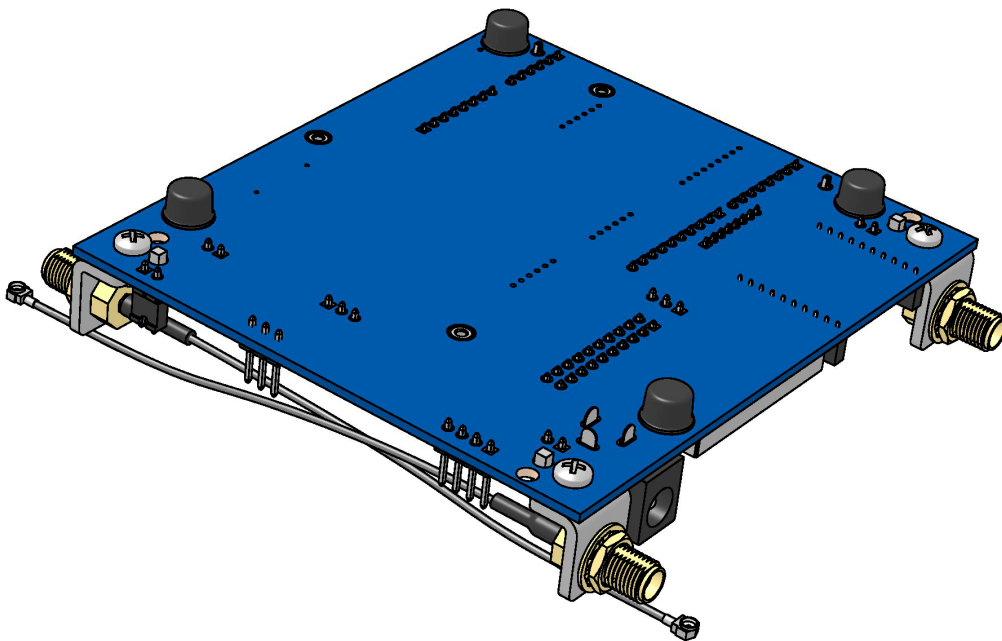
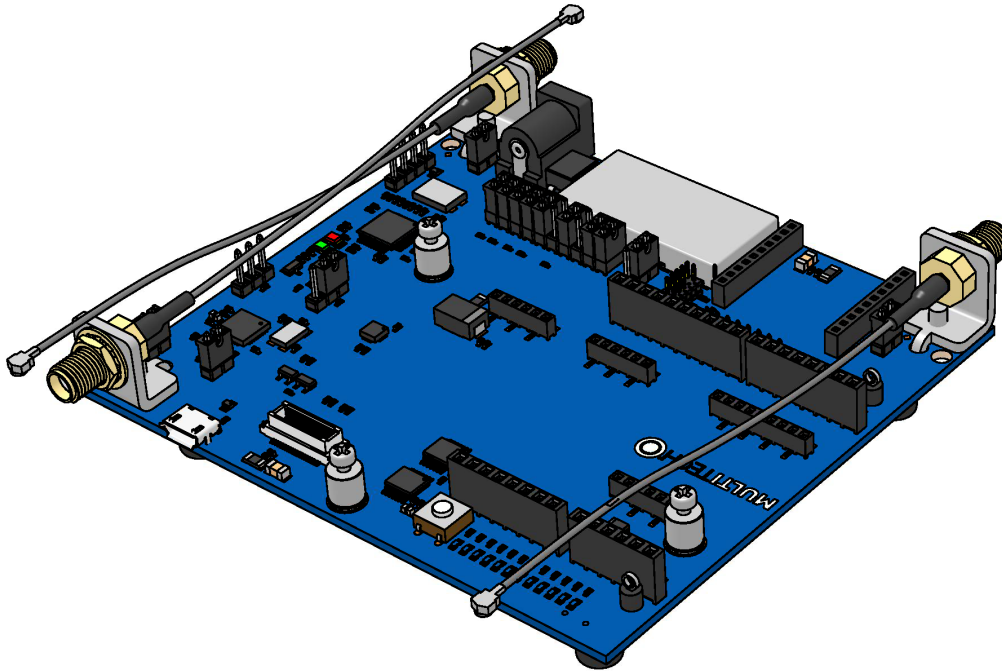
The MTUDK2-ST-CELL Developer Kit includes the following:

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

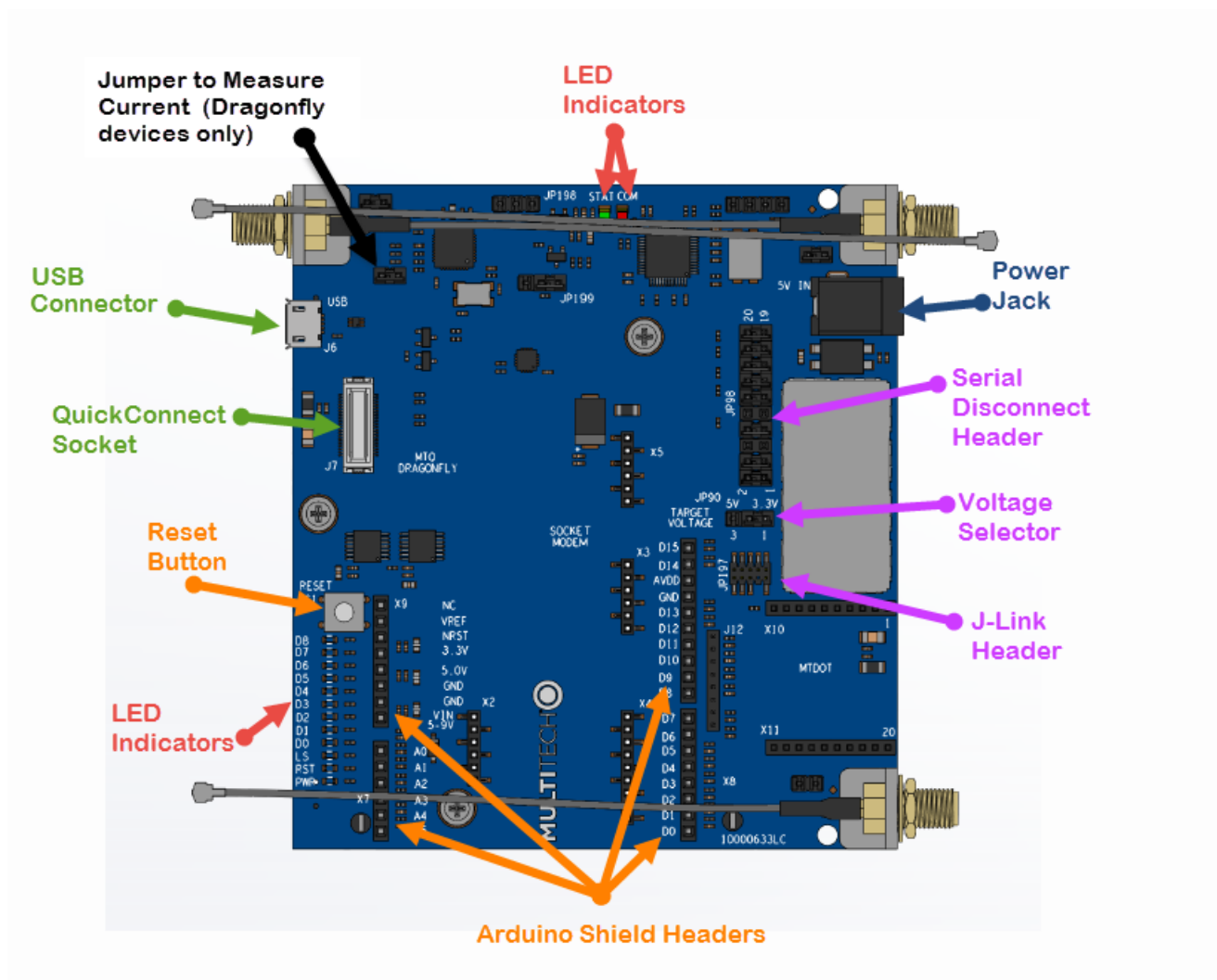
Additional	One promotional screwdriver
------------	-----------------------------

Chapter 2 – Board Components

Developer Board



Developer Board Connectors



Board Components

Label	Description
Voltage Selector	Selects between the on-board 3.3V or 5V regulator for powering a SocketModem, or mDot. Factory default operating voltage is 5V. (Note: Does not apply to Dragonfly devices which use the QuickConnect Socket that is 5V only.)
J6	USB connection for mbed, serial, and SocketModem. For the Dragonfly, use USB connector on the Dragonfly. For information on connecting to and using mbed, refer to the device guide for your Dragonfly model. (Not available for SocketModems.)
J7	QuickConnect Socket (for Dragonfly devices).
JP98	Serial Disconnect Header.
JP197	J-Link Header.
S1	Reset Button. Use to reset the processor of the device attached to the board.

Label	Description
X2	SocketModem, USB Connector.
X3	SocketModem, GPIO (not connected).
X4	SocketModem Serial Connector.
X5	SocketModem Power Connector.
X6	Arduino Shield Connector.
X7	Arduino Shield Connector.
X8	Arduino Shield Connector.
X9	Arduino Shield Connector.
X10	MTDOT Connector.
X11	MTDOT Connector.
J12	MTDOT Programming Header.

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

LED Indicators

Label	LED	Location
STAT	LED1	Near JP198
COM	LED2	Near JP198
D7	LED3	Above the SMC-USB connector
D4	LED4	Above the SMC-USB connector
D5	LED5	Above the SMC-USB connector
D8	LED6	Above the SMC-USB connector
D6	LED7	Above the SMC-USB connector
D3	LED8	Above the SMC-USB connector
D0	LED9	Above the SMC-USB connector
D1	LED10	Above the SMC-USB connector
LS	LED11	Above the SMC-USB connector
RST	LED12	Above the SMC-USB connector
PWR	LED13	Above the SMC-USB connector
D2	LED14	Above the SMC-USB connector

Chapter 3 – Installation and Operation

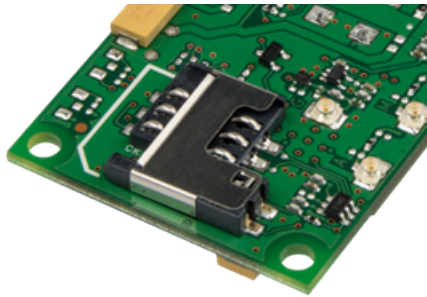
Installing a SIM Card

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.

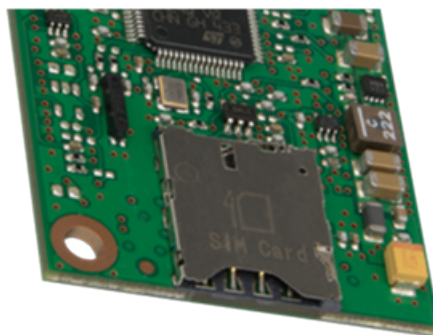


Installing a SIM Card on a DragonFly

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:

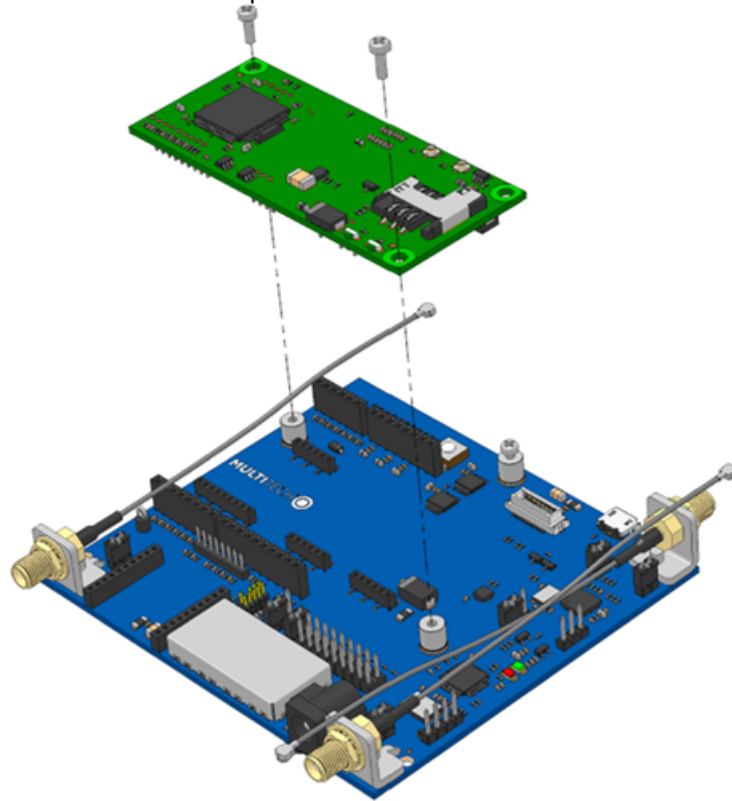
- With the contact side facing down, align the notched edge as shown on the Dragonfly's SIM holder and slide the SIM card completely into the SIM holder.



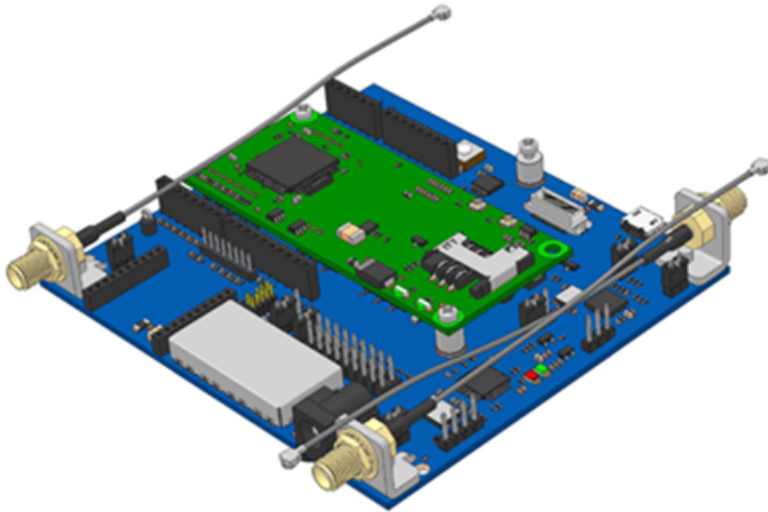
Installing a SocketModem on the Developer Board

To install a SocketModem:

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



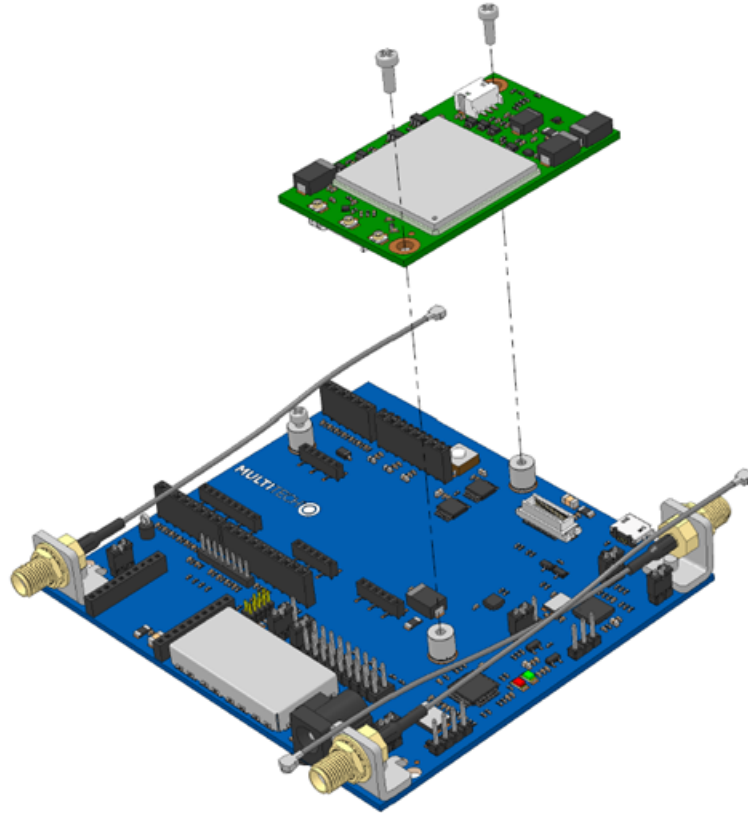
3. Secure the SocketModem with the screws you removed in Step 1.



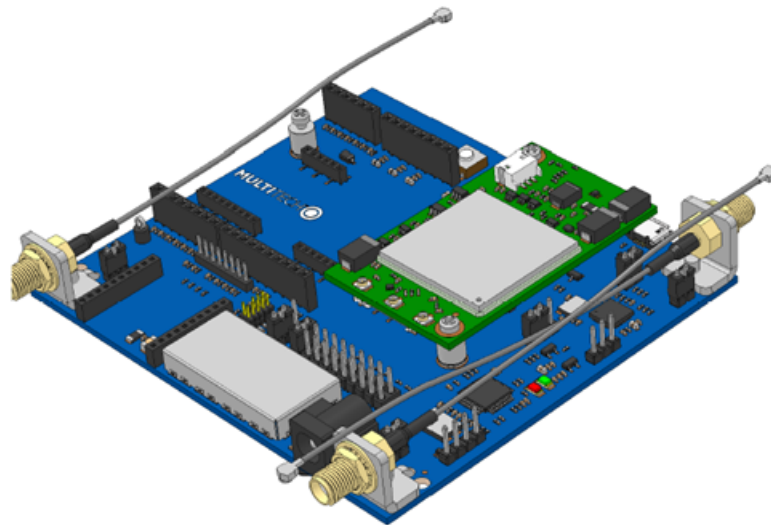
Installing a Dragonfly on the Developer Board

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.



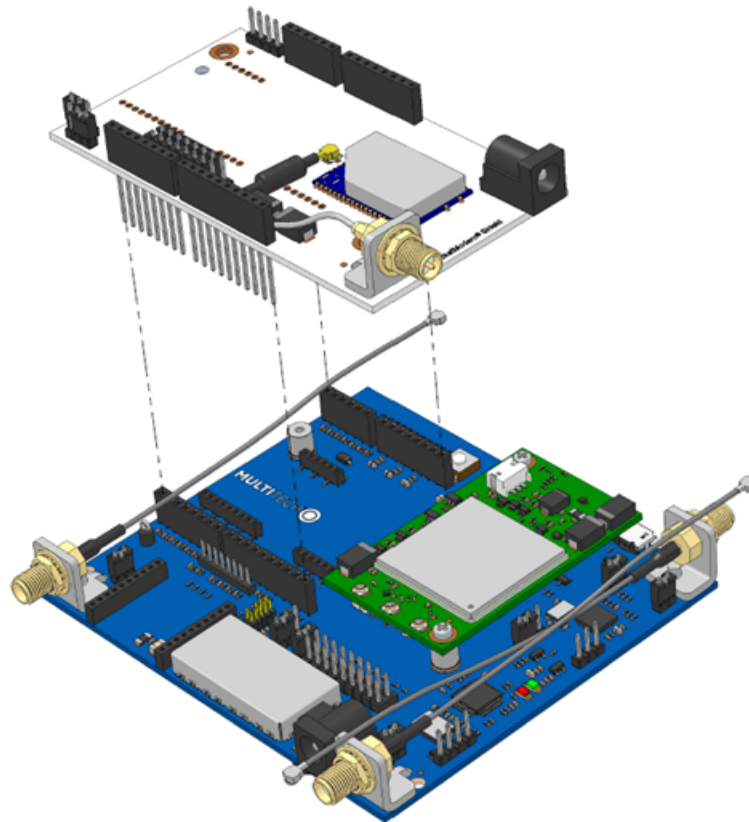
Arduino Shield

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	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	X6	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	X8	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)

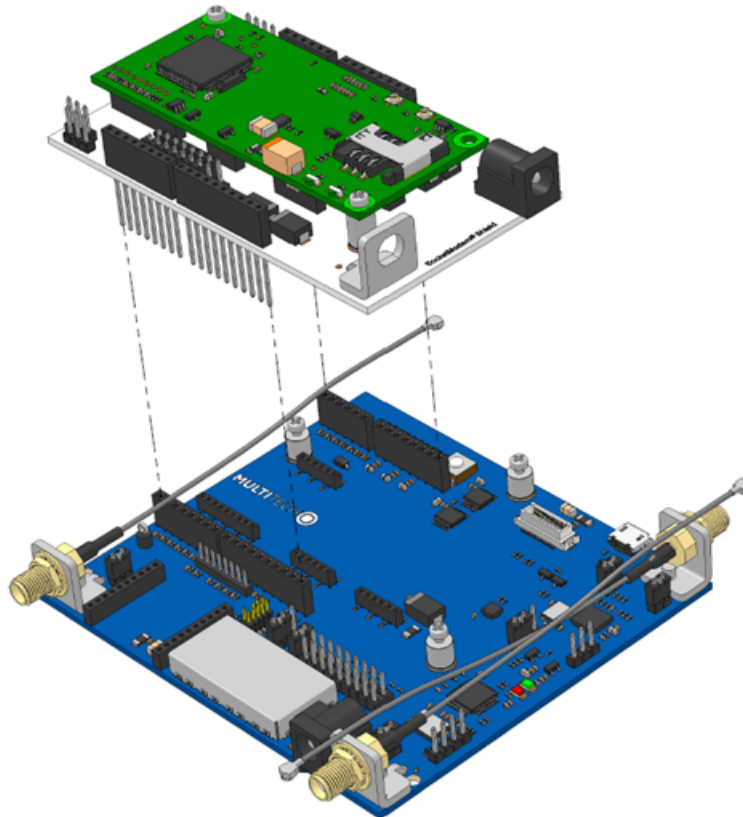
mDot Arduino Pins

Signals (module pin) micro pin		Arduino Shield					Signals (module pin) micro pin
					D15		PWM0/RSSI/I2CSCL (6) PA8
					D14		PWM1/I2CSDA (7) PC9
					AVDD		3.3V
					GND		Ground
3.3V			NC		D13	X6	AD2/DIO2/SCK (18) PA5
nReset, from pushbutton			VREF		D12		DO8/MISO (4) PA6
3.3V			nRST		D11		AD4/DIO4/MOSI (11) PA7
3.3V			3.3V		D10		AD3/DIO3/SNSS (17) PA4
5.0V	X9		5.0V		D9		no connect
Ground			GND		D8		no connect
Ground			GND				
5-9V input from J3			VIN		D7		nDTR/SleepRQ/DI8 (9) PA11
					D6		nRTS/AD6/DIO6 (16) PA1
AD0/DIO0 (20) PB1			A0		D5		no connect
AD1/DIO1 (19) PB0			A1		D4	X8	no connect
Associate/AD5/DIO5 (15) PC1	X7		A2		D3		nCTS/DIO7 (12) PA0
no connect			A3		D2		ON/nSleep (13) PC13
no connect			A4		D1		Dout (2) PA2
no connect			A5		D0		Din (3) PA3

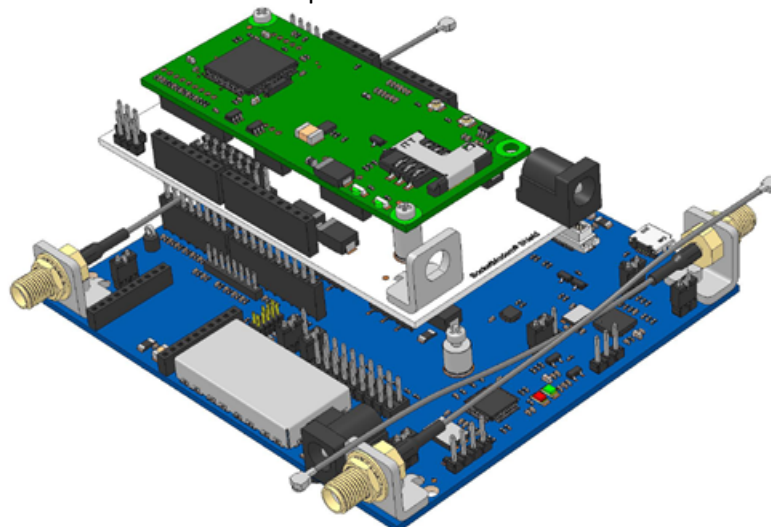
Installing an Arduino Shield with a SocketModem

If using an Arduino Shield with a SocketModem:

1. Move jumpers for JP98 as follows:
 - Move Jumper 3-4 to Jumper 5-6
 - Move Jumper 7-8 to Jumper 9-10
2. Mount the MTSMC device on the Arduino shield as shown in the following image.



3. Connect the Arduino Shield to the developer board.



Attaching Power Supply Blades

Power Supply and Blades

If your device shipped with a power cord, 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

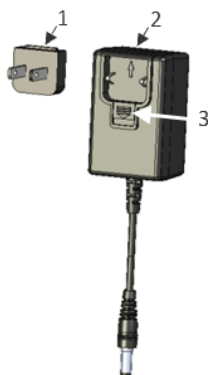


Power
Supply with
AU-NZ blade

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)

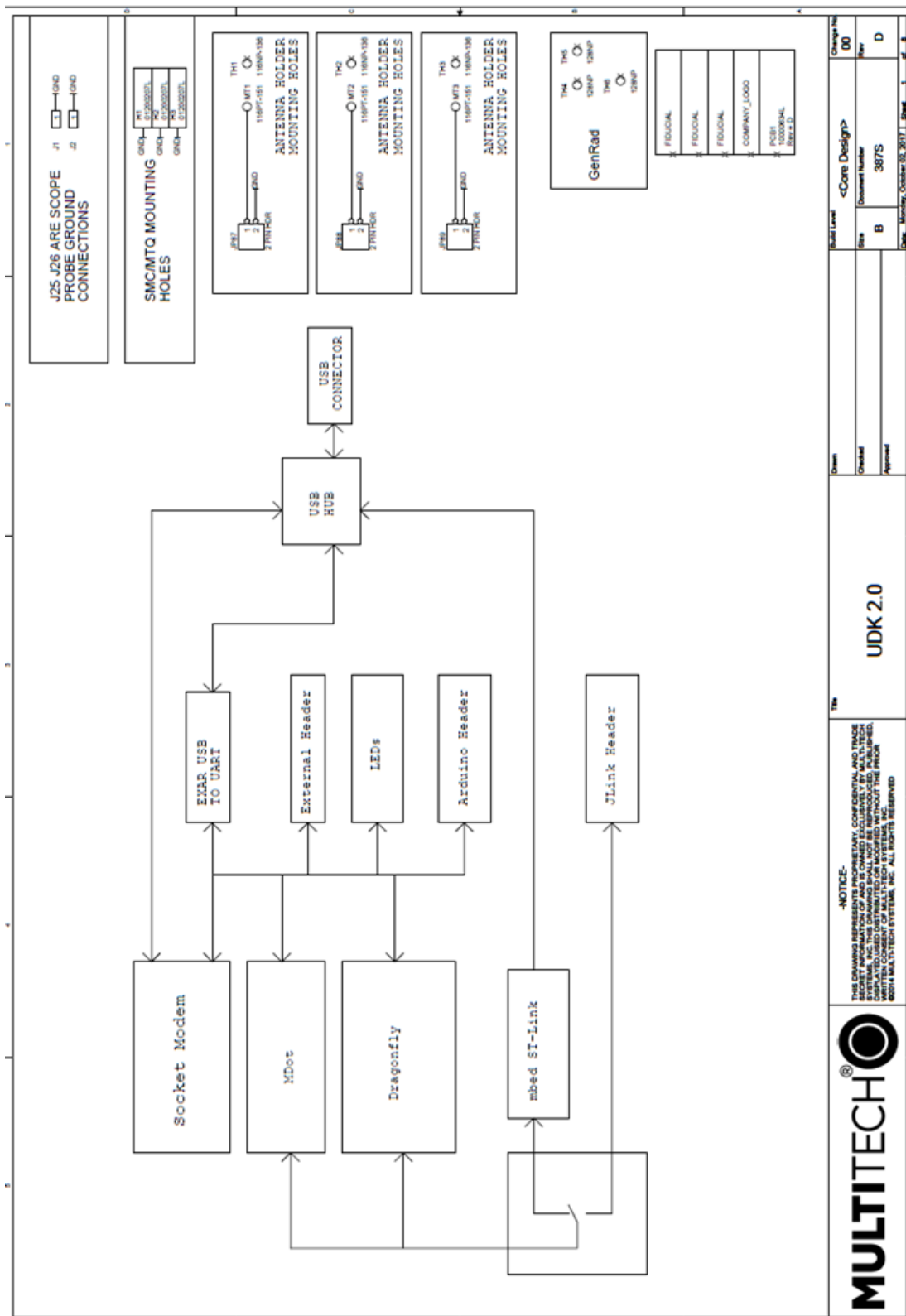
M = Main

D = Diversity



Chapter 4 – Block Diagram and Schematics

Block Diagram

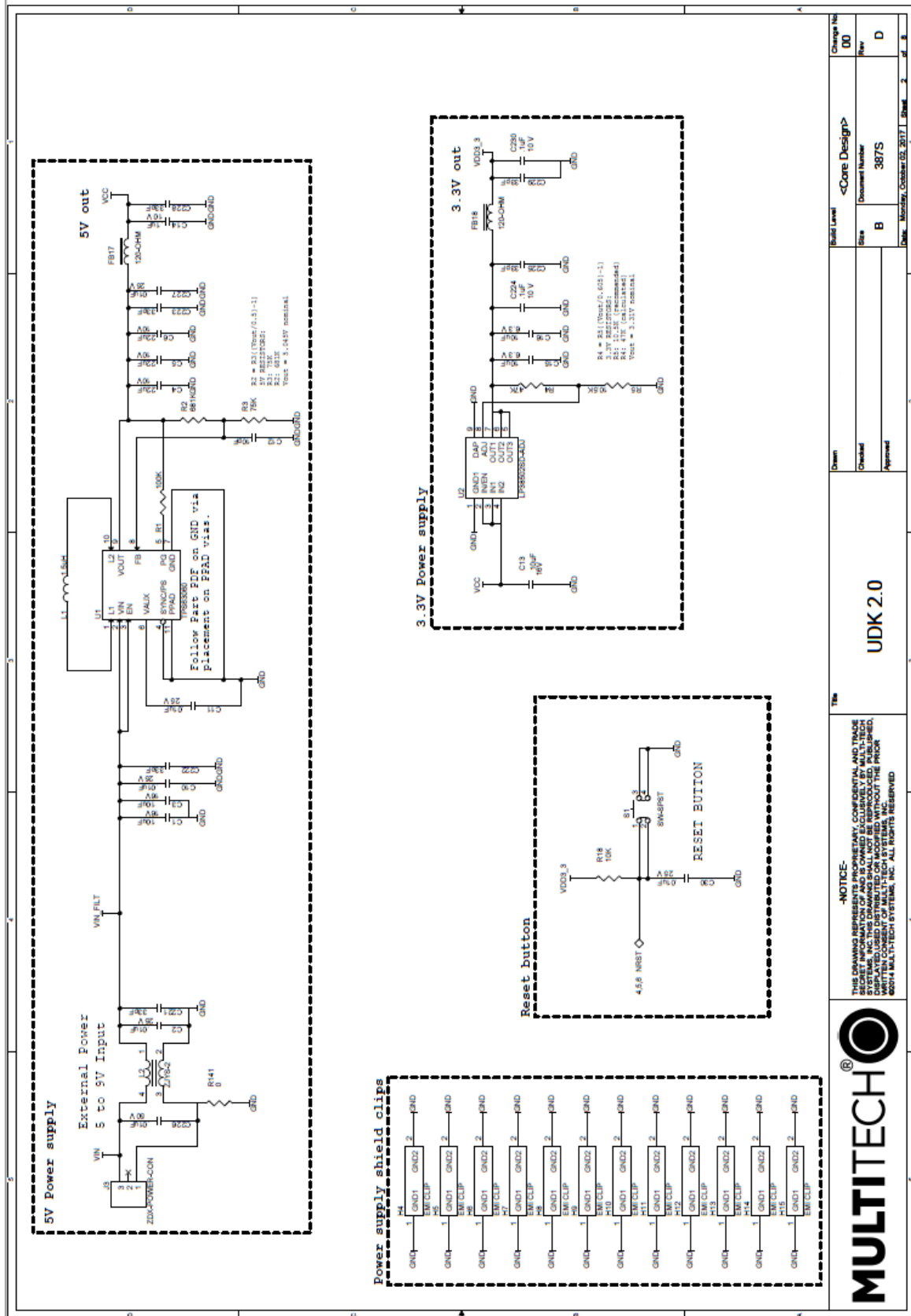


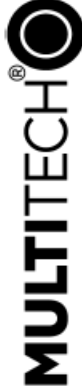
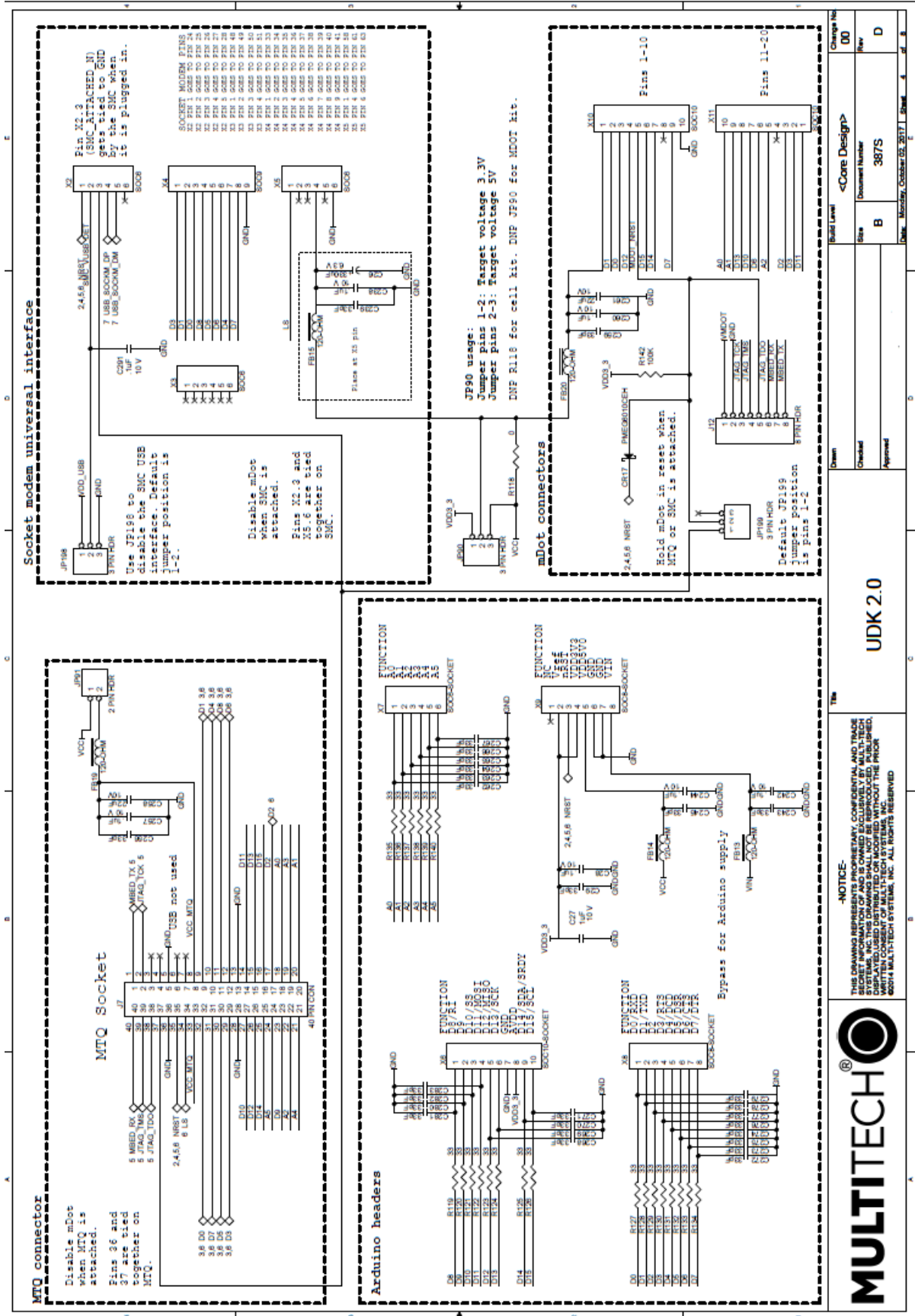
UDK 2.0

MULTITECH
 THIS SCHEMATIC REPRESENTS CONFIDENTIAL AND TRADE SECRET INFORMATION OF AND IS OWNED EXCLUSIVELY BY MULTITECH SYSTEMS, INC. NO PART OF THIS DRAWING SHALL BE REPRODUCED, COPIED, PUBLISHED, OR TRANSMITTED IN ANY FORM OR BY ANY MEANS, WITHOUT THE WRITTEN CONSENT OF MULTITECH SYSTEMS, INC.
 ©2011 MULTITECH SYSTEMS, INC. ALL RIGHTS RESERVED

Drawn	
Checked	
Approved	
Scale	B
Document Number	3875
Revision	D
Change No.	00

Schematics



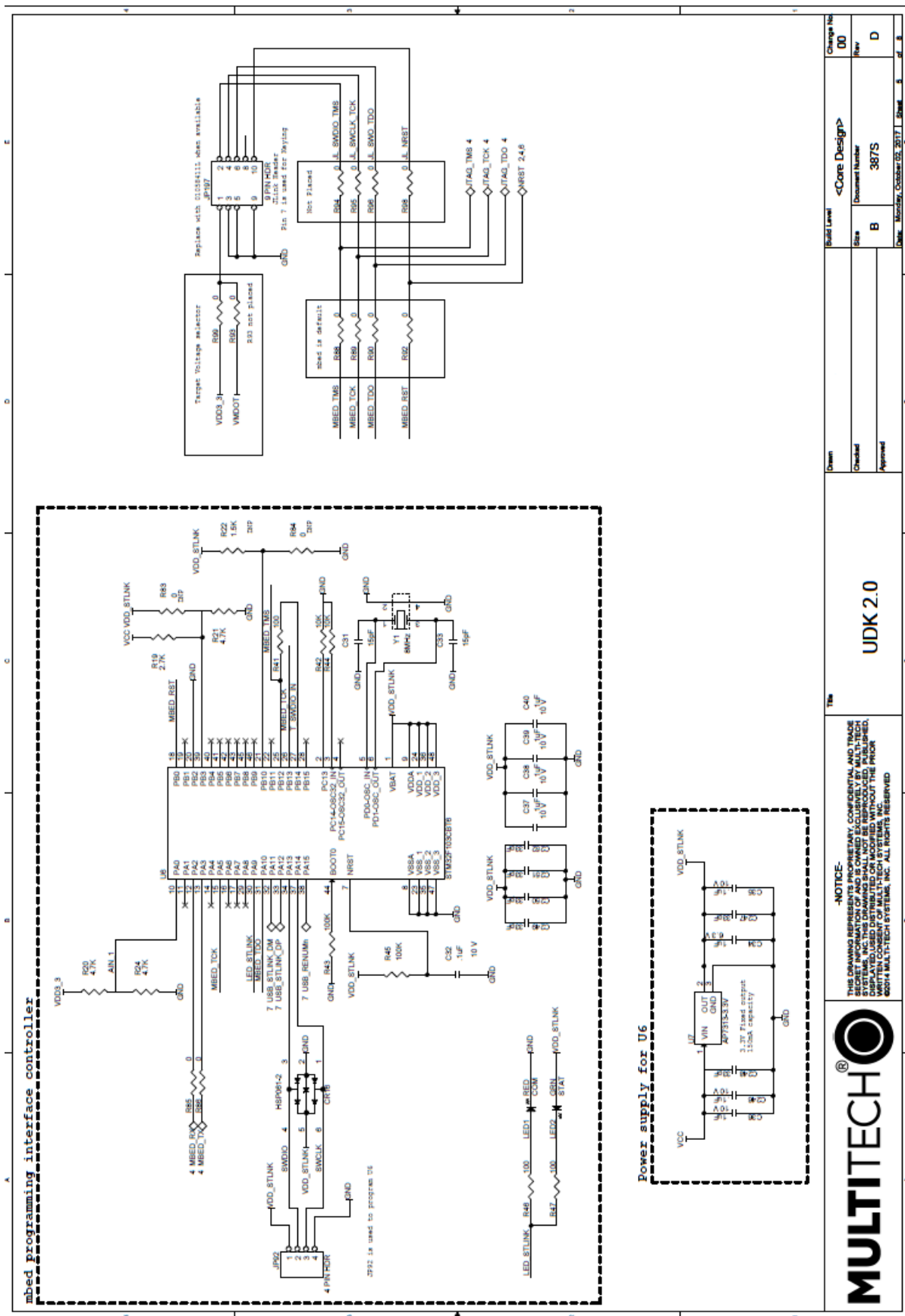


UDK 2.0

Rev
 387S
 B
 00

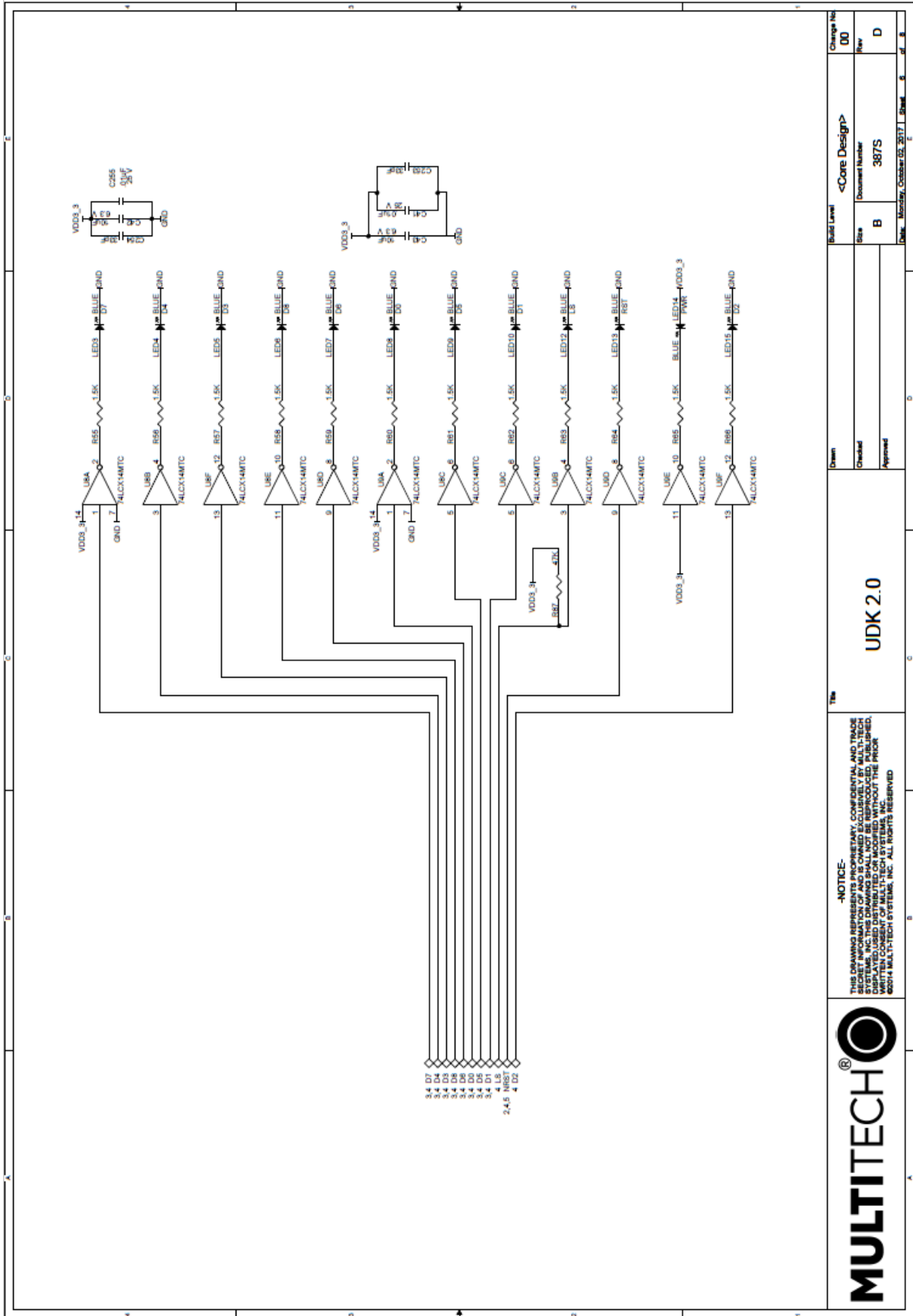
Change No
 Rev
 D

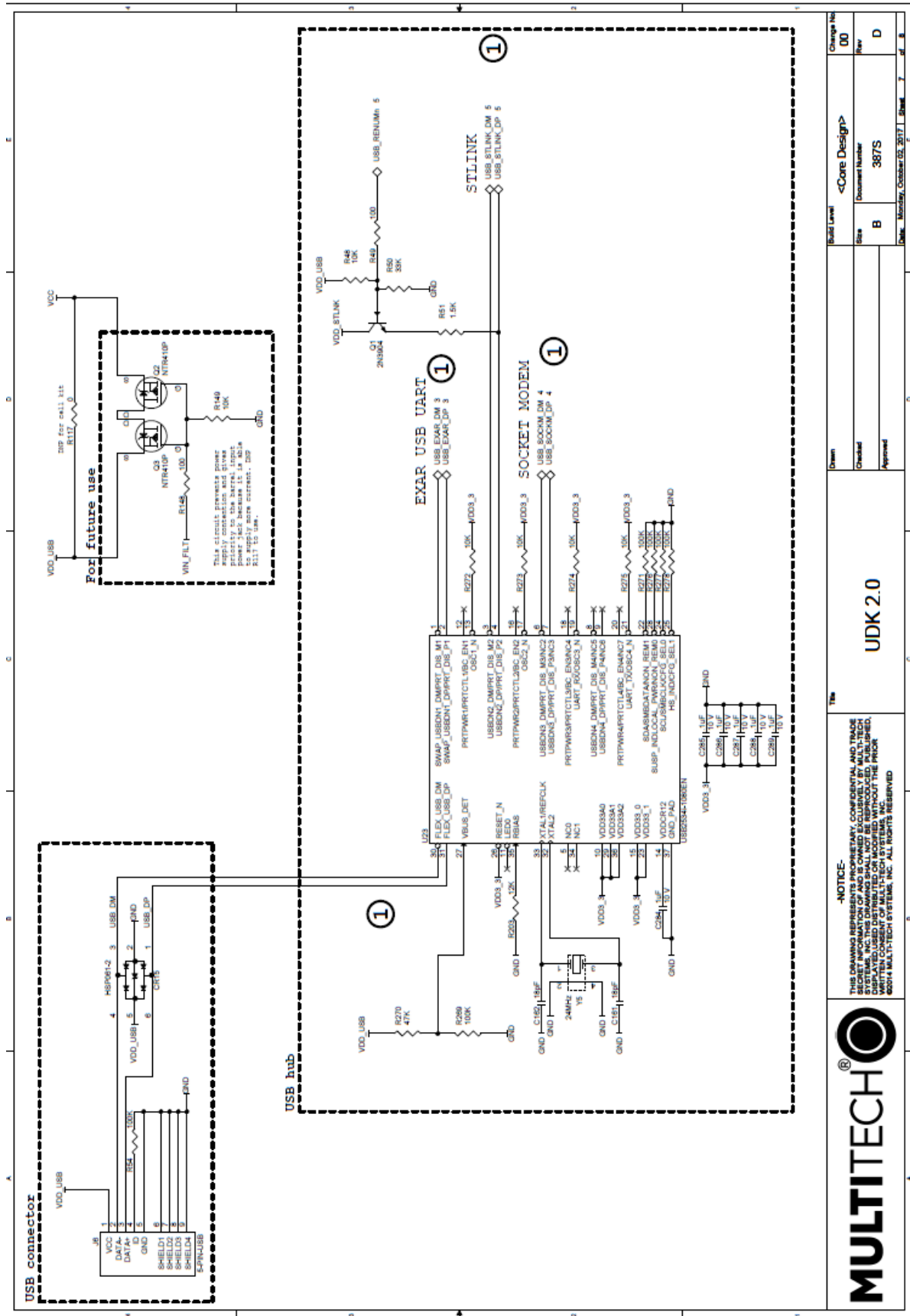
Doc. Monday, October 02, 2017 Sheet 4 of 8



Drawn	Valid Level	<Core Design>	Change No.
Checked	Site	Document Number	Rev
Approved	B	387S	D
Date: Monday, October 02, 2017		Sheet	5 of 8

UDK 2.0





Title		Revision	
UDK 2.0		Drawn	<Core Design>
		Checked	Document Number
		Approved	3675
Date: Monday, October 02, 2017 1:36pm		Size	B
Sheet		Change No.	00
		Rev	D

NOTICE:
 THIS DRAWING REPRESENTS PROPRIETARY, CONFIDENTIAL AND TRADE SECRET INFORMATION ON BEHALF OF MULTITECH SYSTEMS, INC. AND IS NOT BE REPRODUCED, COPIED, TRANSMITTED, OR OTHERWISE DISCLOSED IN ANY MANNER WITHOUT THE WRITTEN PERMISSION OF MULTITECH SYSTEMS, INC. ALL RIGHTS RESERVED.
 ©2014 MULTITECH SYSTEMS, INC. ALL RIGHTS RESERVED



<p>NOTES:</p> <ol style="list-style-type: none"> Route all USB traces as differential pairs. Use 90 differential impedance traces. <p>Changelog, RevB to RevC:</p> <ul style="list-style-type: none"> Removed micro-USB connector J6 and replaced with more mechanically robust J13 Added circuit to keep mDot in reset whenever an MTQ or SMC are plugged in. Fixed net names of TXD and RXD on Arduino shield connectors. D0 should be RXD, and D1 should be TXD. Fixed JTAG_RST and NRESET so that the interface controller can reset the targets. Added a circuit to allow auto-switchover from USB to barrel-jack power. Removed RS232 transceiver U5 and supporting components. Removed DB9 connector and supporting components. Fixed the power sequencing issue that caused the "12KB disk error." Power the STM32 (U6) from VCC only; this will prevent it from turning on before the target is powered. Removed SMC micro-USB connector J5. Connected SMC USB connection to the USB hub instead. 	<table border="1"> <tr> <td data-bbox="1312 212 1339 814"> Drawn Checked Approved </td> <td data-bbox="1312 814 1339 1696" rowspan="2"> Title <h2 style="text-align: center;">UDK 2.0</h2> </td> <td data-bbox="1312 1696 1339 1913"> Change No. 00 Rev D </td> </tr> <tr> <td data-bbox="1339 212 1421 814"> Build Level <Core Design> Document Number 3875 Size B Date Monday, October 02, 2017 8:56 AM </td> <td data-bbox="1339 814 1421 1696"> This drawing represents proprietary, confidential and trade secret information of and is owned exclusively by Multi-Tech Systems, Inc. No part of this drawing may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, mechanical, photocopying, recording, or by any information storage and retrieval system, without the prior written permission of Multi-Tech Systems, Inc. All rights reserved. ©2014 MULTI-TECH SYSTEMS, INC. ALL RIGHTS RESERVED. </td> </tr> </table>	Drawn Checked Approved	Title <h2 style="text-align: center;">UDK 2.0</h2>	Change No. 00 Rev D	Build Level <Core Design> Document Number 3875 Size B Date Monday, October 02, 2017 8:56 AM	This drawing represents proprietary, confidential and trade secret information of and is owned exclusively by Multi-Tech Systems, Inc. No part of this drawing may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, mechanical, photocopying, recording, or by any information storage and retrieval system, without the prior written permission of Multi-Tech Systems, Inc. All rights reserved. ©2014 MULTI-TECH SYSTEMS, INC. ALL RIGHTS RESERVED.
Drawn Checked Approved	Title <h2 style="text-align: center;">UDK 2.0</h2>	Change No. 00 Rev D				
Build Level <Core Design> Document Number 3875 Size B Date Monday, October 02, 2017 8:56 AM		This drawing represents proprietary, confidential and trade secret information of and is owned exclusively by Multi-Tech Systems, Inc. No part of this drawing may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, mechanical, photocopying, recording, or by any information storage and retrieval system, without the prior written permission of Multi-Tech Systems, Inc. All rights reserved. ©2014 MULTI-TECH SYSTEMS, INC. ALL RIGHTS RESERVED.				

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.