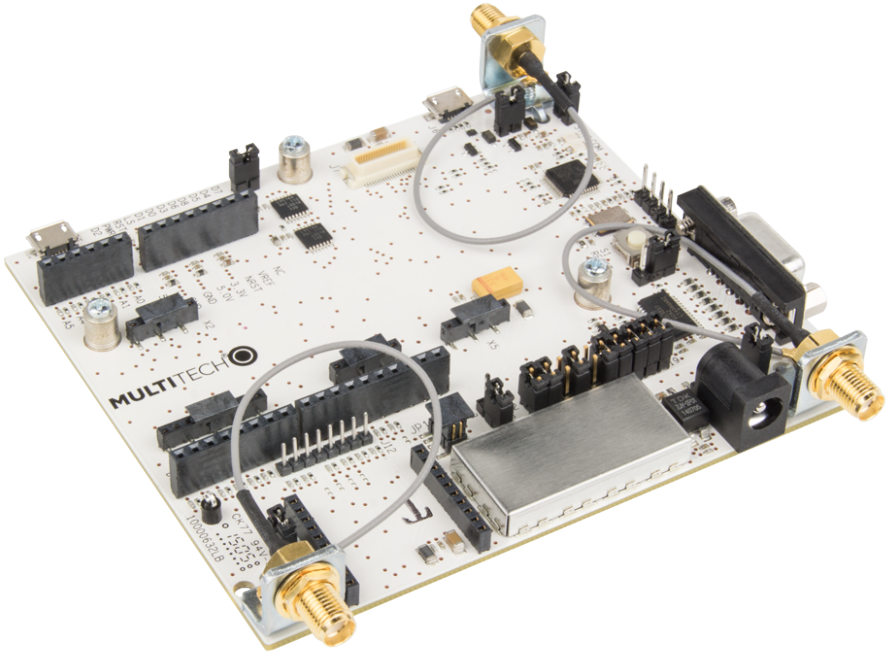
MULTITECH



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 1.0

Copyright

Trademarks and Registered Trademarks

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 MULTI-TECH 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

Developer Board Installation and Operation

Overview

The MTUDK2-ST-Cell Universal Developer Kit supports development with cellular SocketModem and Dragonfly 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.

Features

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

Device Specific Documentation

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

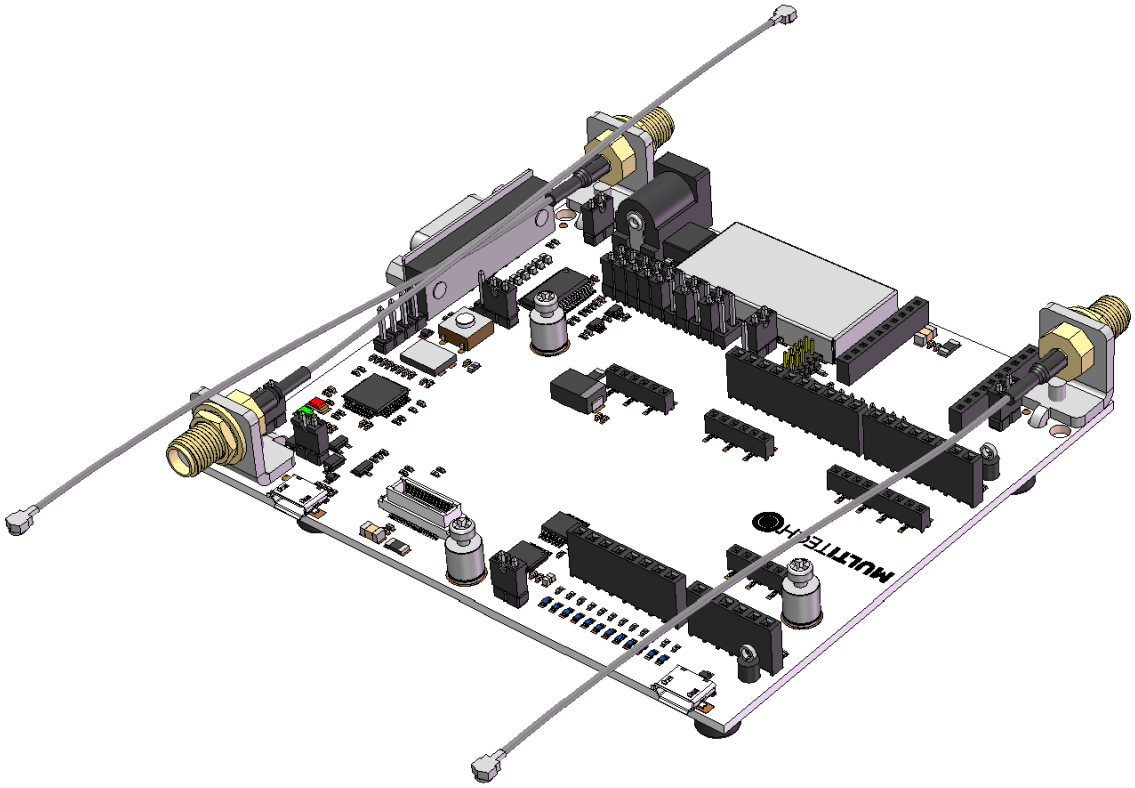
Developer Kit Contents

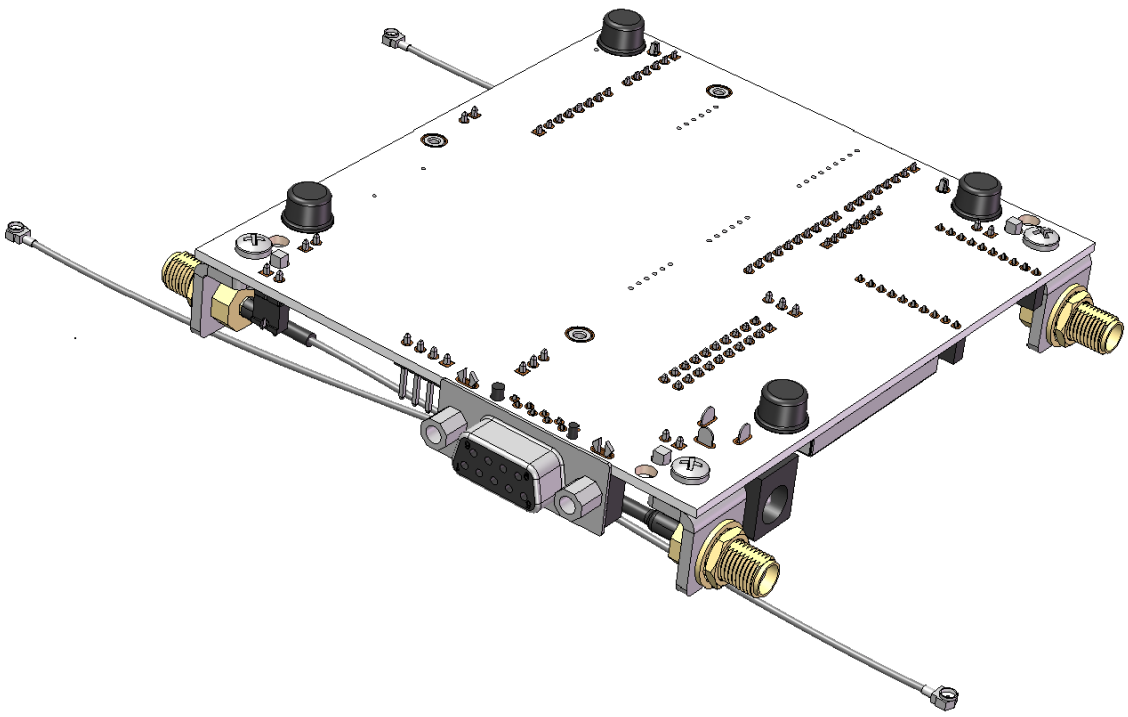
Your Developer Kit (MTUDK2-ST-Cell) includes the following:

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

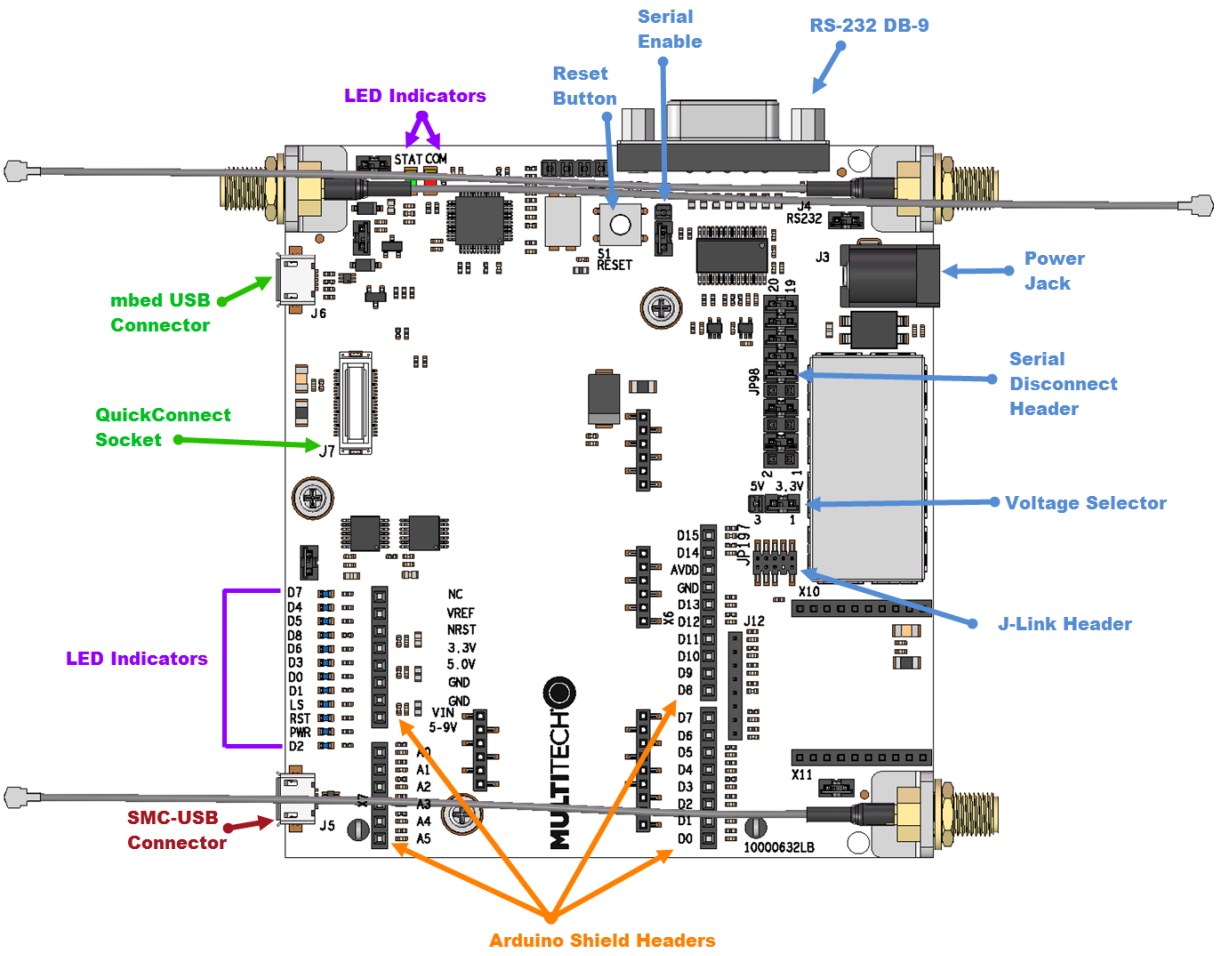
Board Components

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.
J4	RS232 DB-9 Serial Connector
J5	SMC-USB Connector
J6	mbed USB Connector
J7	QuickConnect Socket
JP98	Serial Disconnect Header
JP197	J-Link Header

Label	Description
S1	Reset Button
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

LED Indicators

Label	LED	Location
STAT	LED1	Same side as RS-232 DB-9 connector
COM	LED2	Same side as RS-232 DB-9 connector
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

Installation and Operation

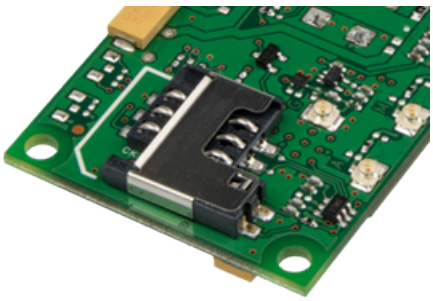
Installing a SIM Card

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.

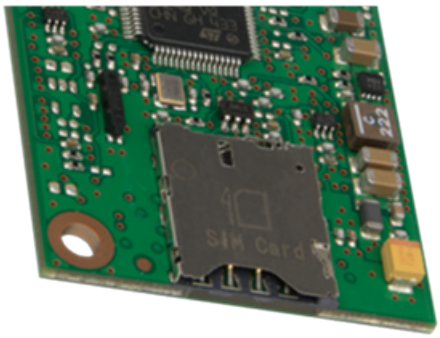


Installing a SIM Card on a DragonFly

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:

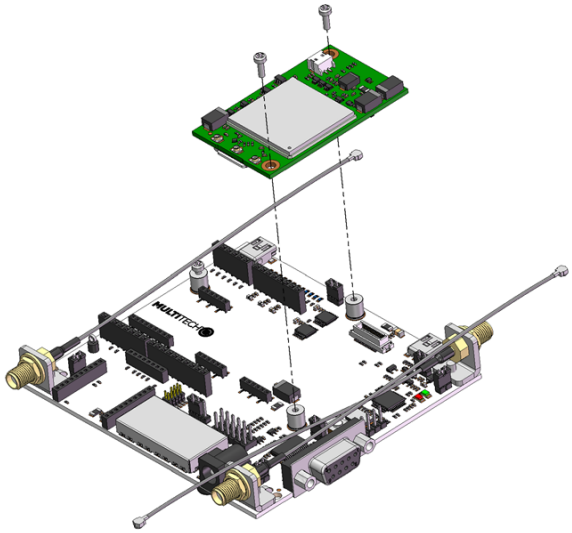
- With the contact side facing down, align the notched edge as shown on the Dragonfly's SIM holder and slide the SIM card completely into the SIM holder.



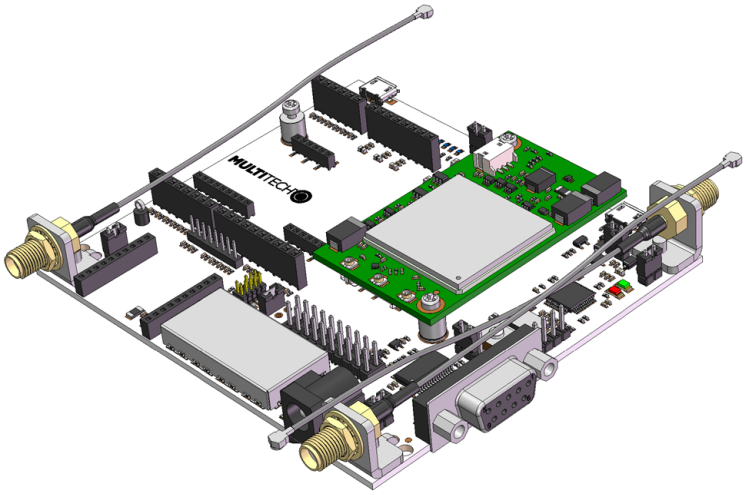
Installing a Dragonfly on the Developer Board

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

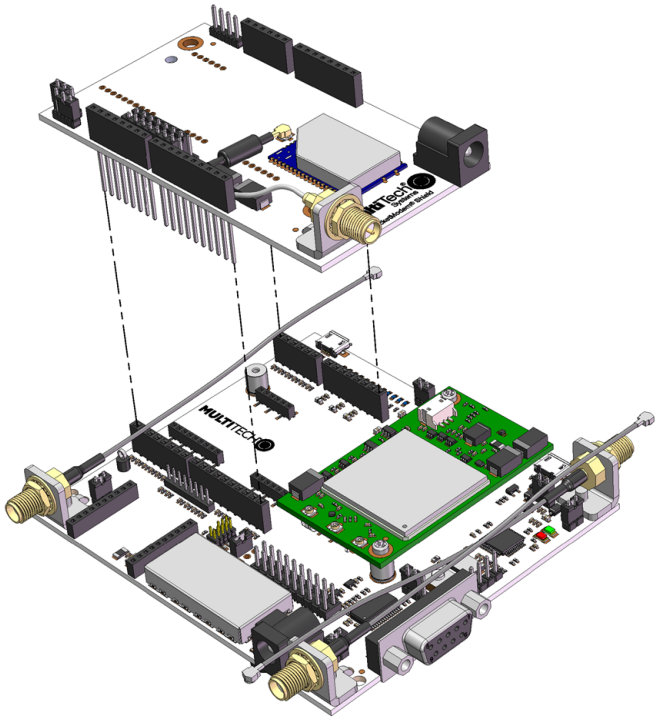
Installing an Arduino Shield with a Dragonfly

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



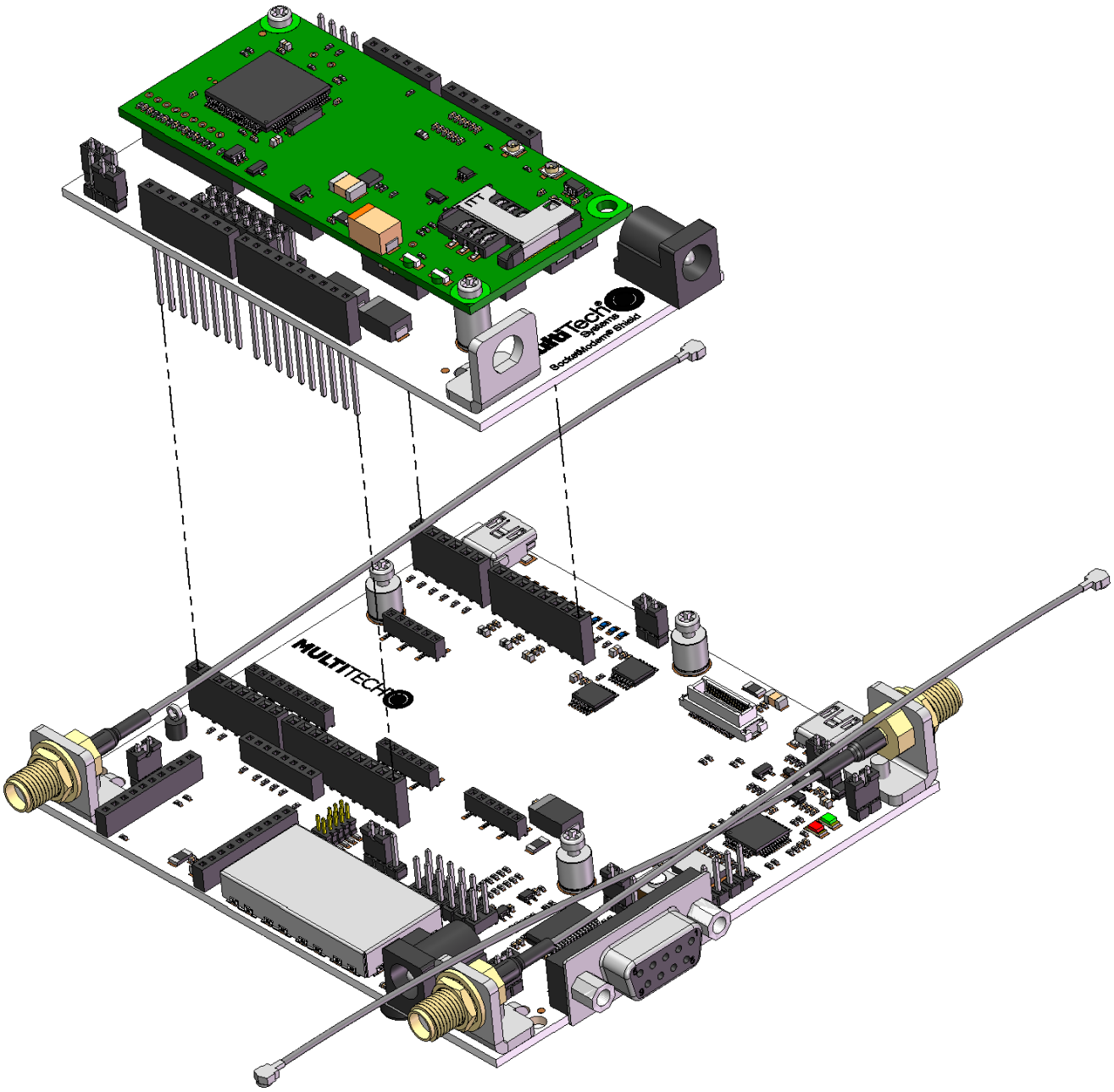
Dragonfly Arduino Pins

Signals (micro pin)		Arduino Shield					Signals (micro pin)
				D15			D15/SCL/SS1 (PB8)
				D14			D14/SDA/SRDY (PB9)
				AVDD			3.3V
				GND			Ground
			NC	D13		X6	D13/SCK (PA5)
3.3V			VREF	D12			D12/MISO (PA6)
nReset, from pushbutton			nRST	D11			D11/MOSI (PB5)
3.3V			3.3V	D10			D10/SS2 (PC8)
5.0V	X9		5.0V	D9			D9 (PB13)
Ground			GND	D8			D8/RI (PB1)
Ground			GND				
5-9V input from J3			VIN	D7			D7/DTR (PA8)
				D6			D6/CTS (PA1)
A0 (PC2)			A0	D5			D5/DSR (PA9)
A1 (PC0)			A1	D4		X8	D4/DCD (PA7)
A2 (PC4)	X7		A2	D3			D3/RTS (PA0)
A3 (PB0)			A3	D2			D2 (PB15)
A4 (PC1)			A4	D1			D1/RXD (PA2)
DIO (PC9)			A5	D0			D0/TXD (PA3)

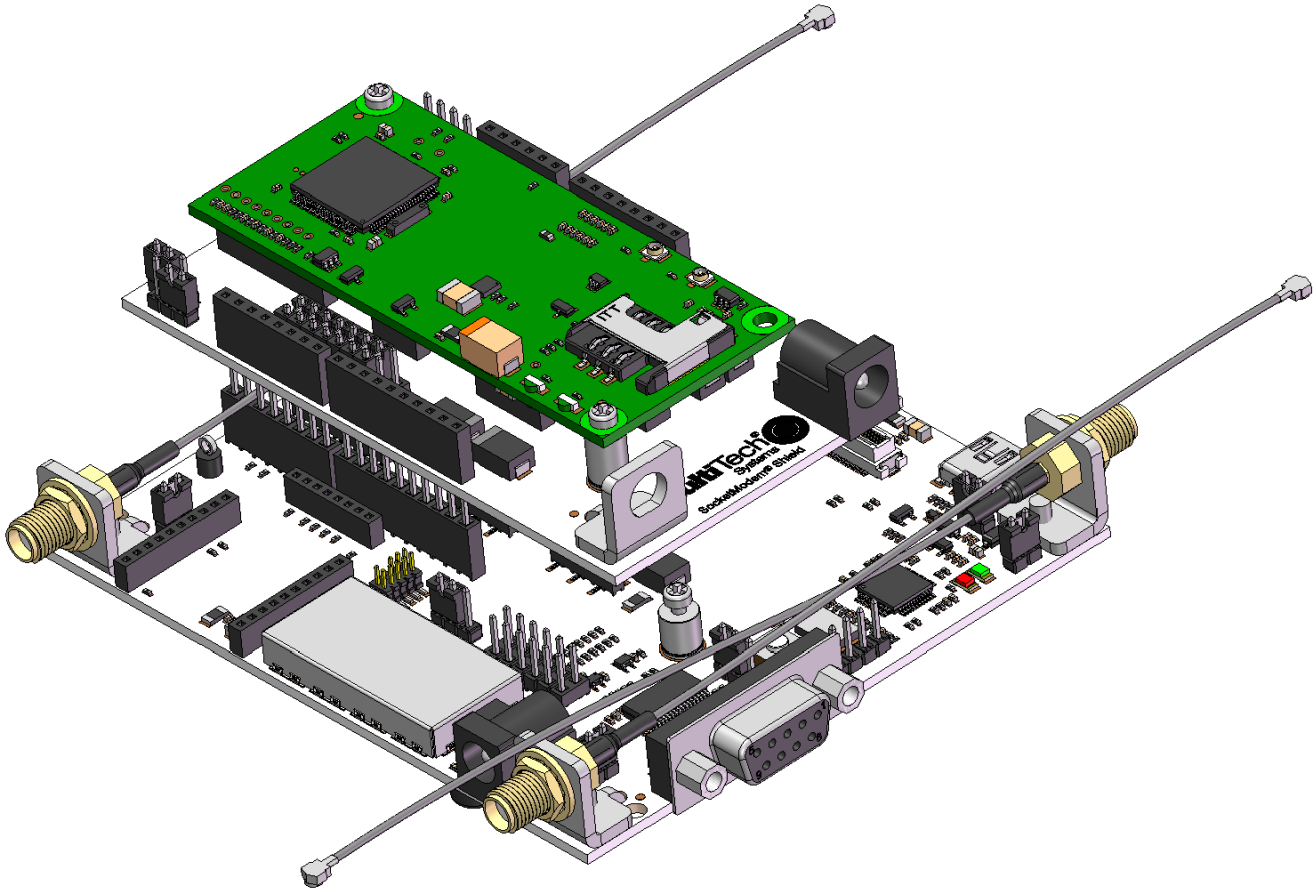
Installing an Arduino Shield with a SocketModem

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 as shown in the following image.



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

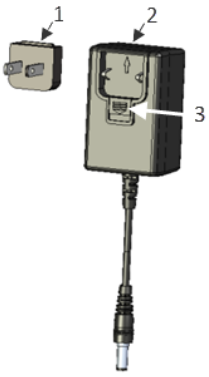
Power Supply with UK blade

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

To connect an antenna to the device:

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 SMA to U.FL cable to the connector on the device.

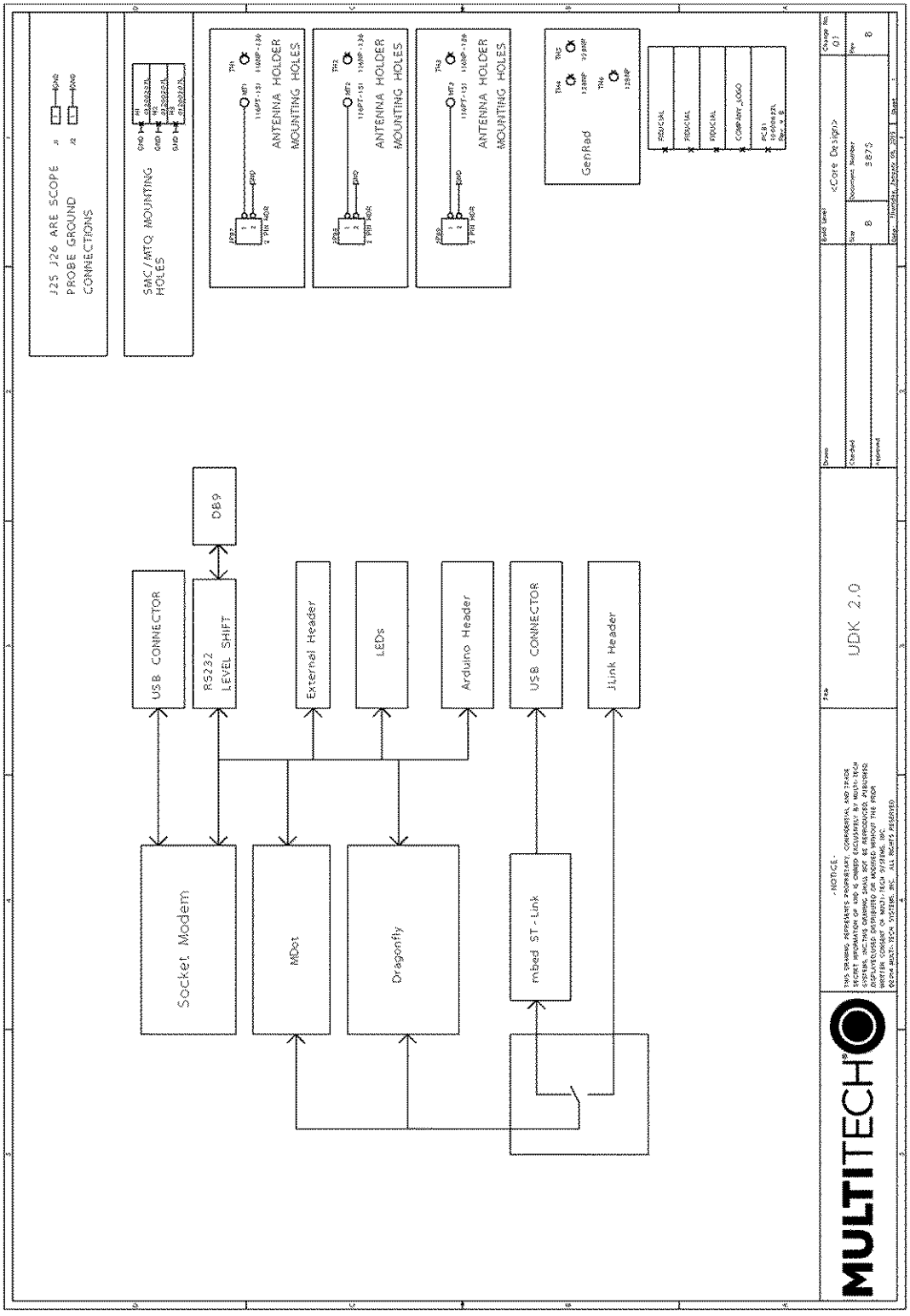
Modem Cable DTE to DCE

DTE Device (Processor)			Connections	DCE Device (Modem)		
PIN#	**DB9 RS-232 Signal Names**		**Signal Direction**	**Pin#**	**DB9 RS-232 Signal Names**	
1	Carrier Detector (DCD)	CD	←	1	Carrier Detector (DCD)	CD
2	Receive Data (Rx)	RD	←	2	Receive Data (Rx)	RD
3	Transmit Data (Tx)	TD	→	3	Transmit Data (Tx)	TD

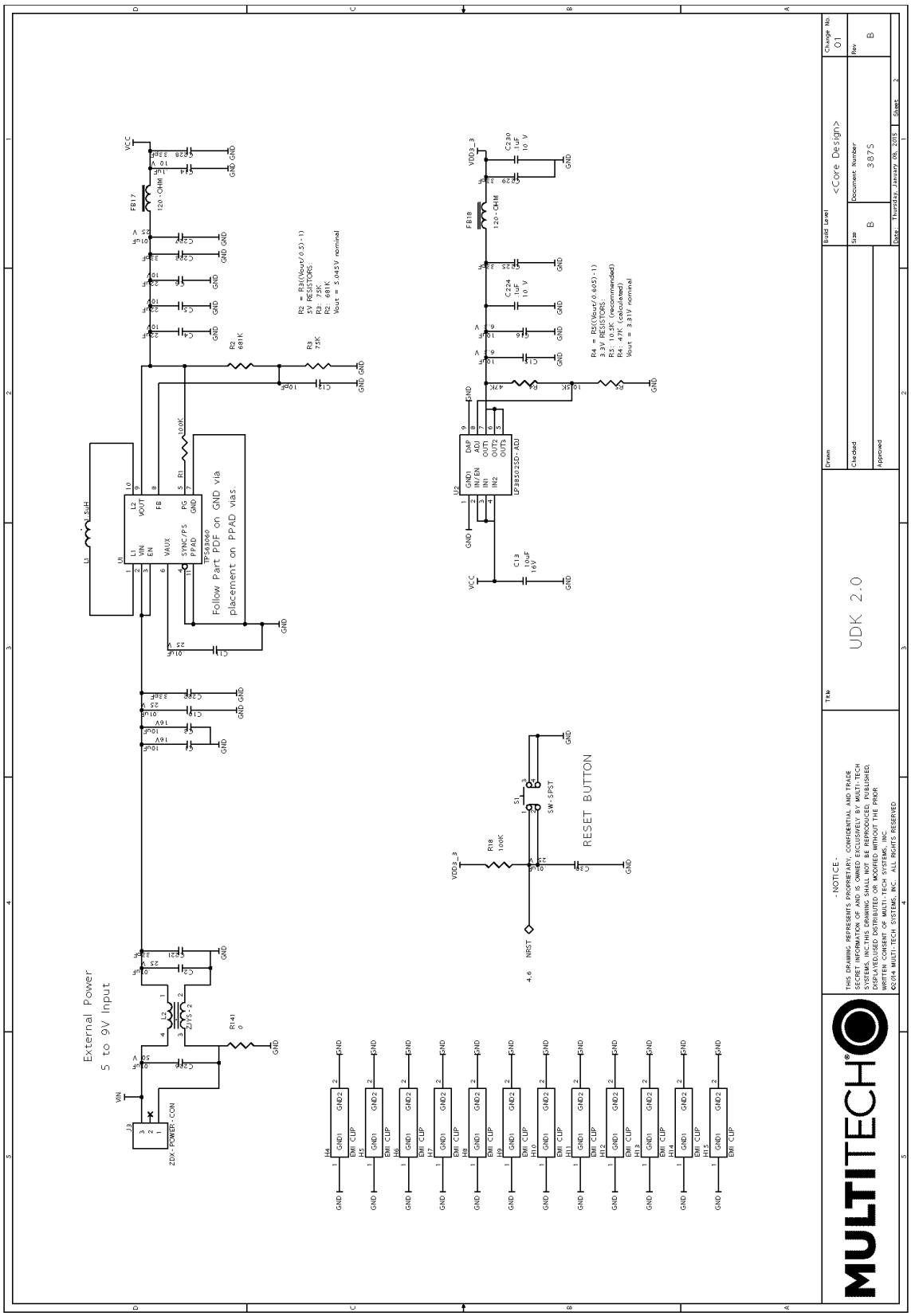
DTE Device (Processor)			Connections	DCE Device (Modem)		
4	Date Terminal Ready	DTR		4	Date Terminal Ready	DTR
5	Signal Ground/Common (SG)	GND		5	Signal Ground/Common (SG)	GND
6	Data Set Ready	DSR		6	Data Set Ready	DSR
7	Request to Send	RTS		7	Request to Send	RTS
8	Clear to Send	CTS		8	Clear to Send	CTS
9	Ring Indicator	RI		9	Ring Indicator	RI
	Soldered to DB9 Metal-Shield	FGND				

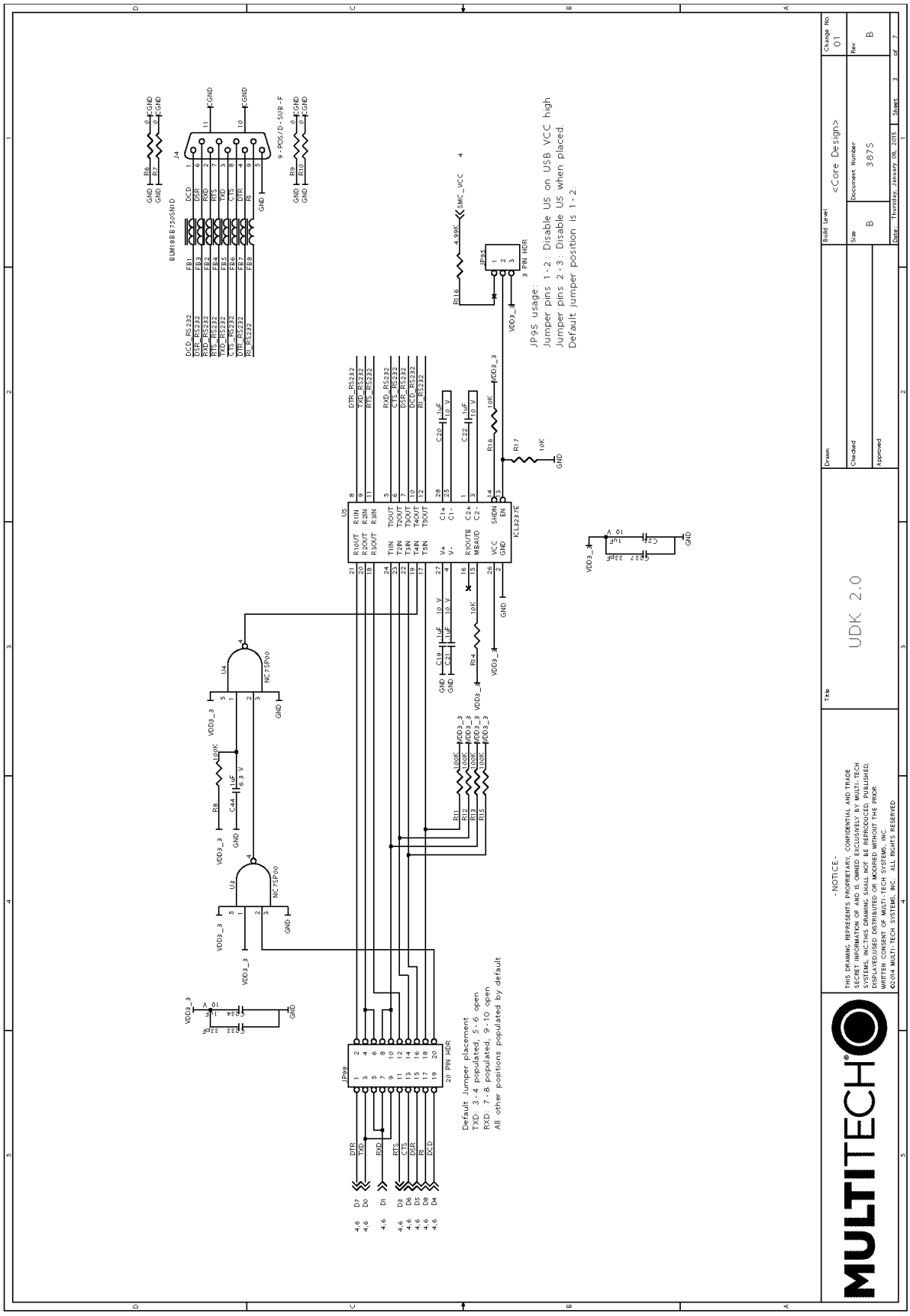
Developer Board Block Diagram and Schematics

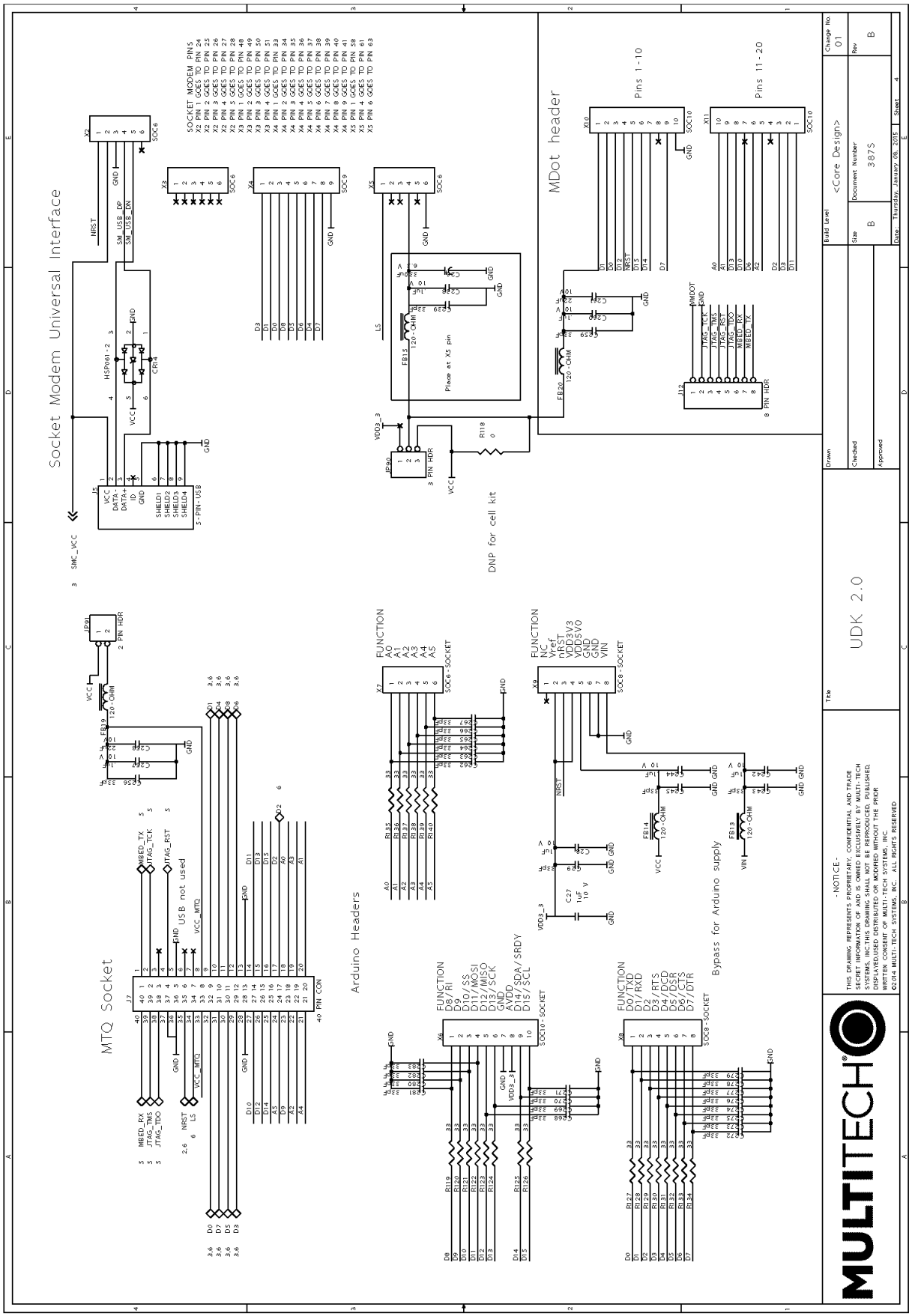
Block Diagram

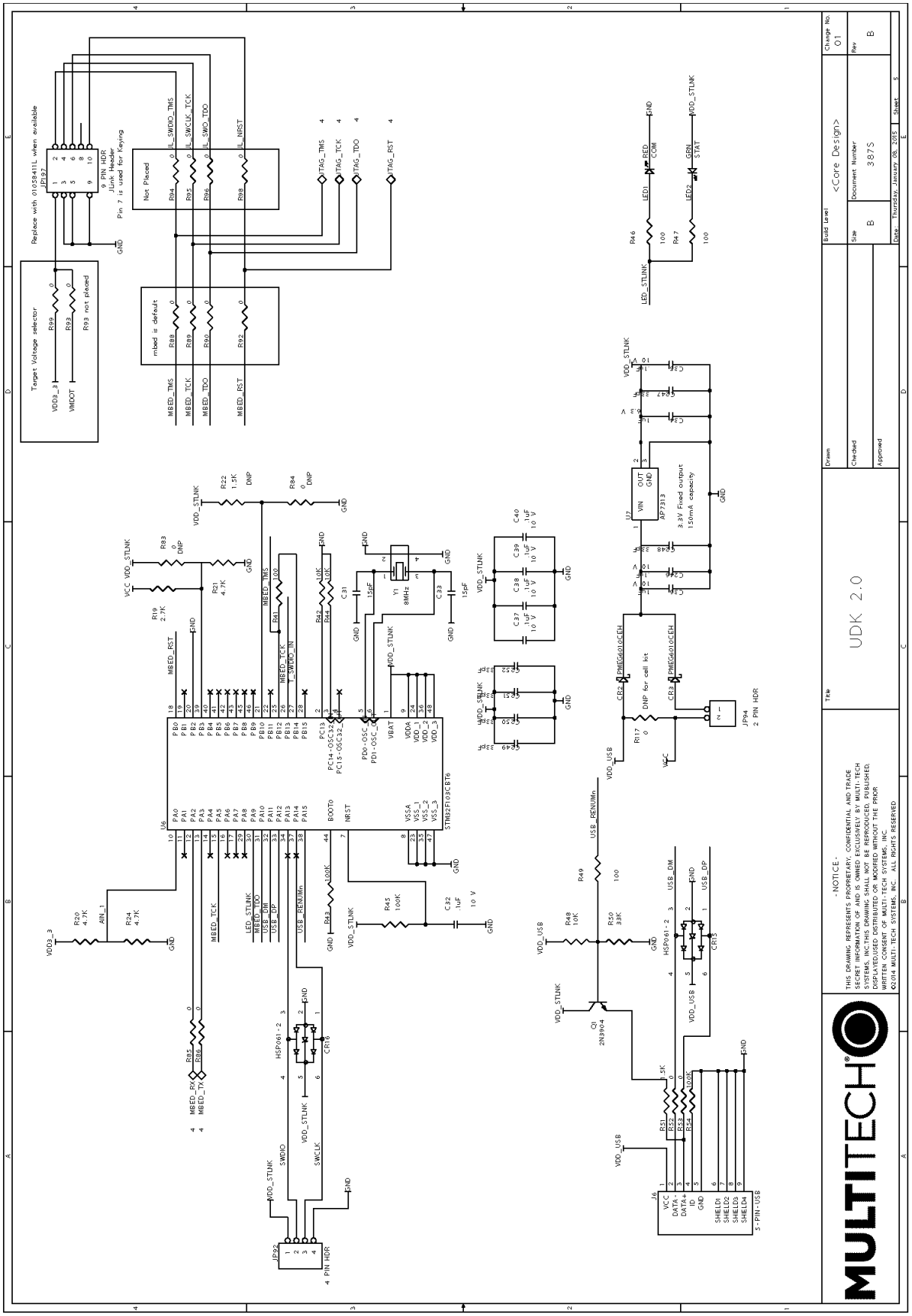


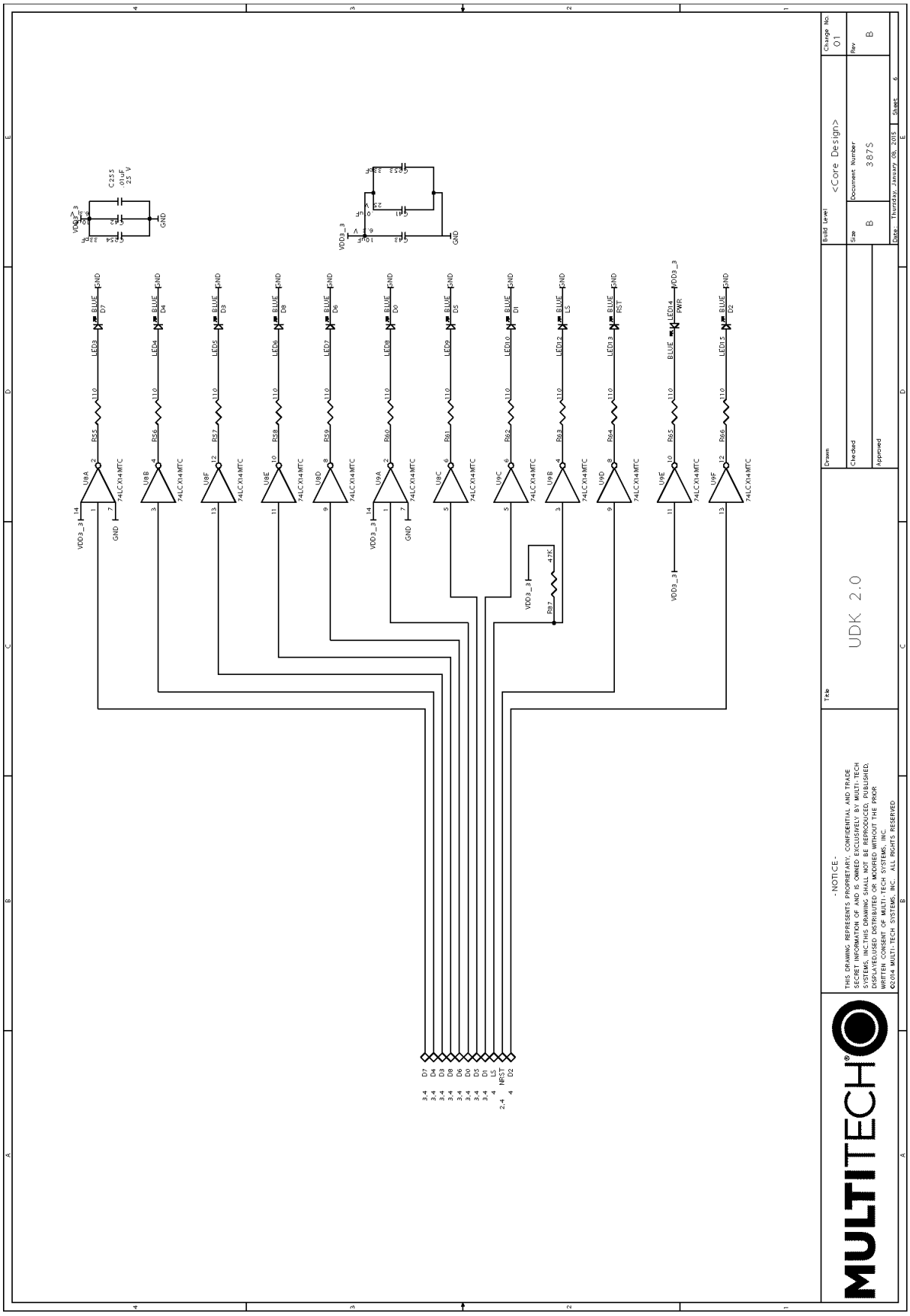
Schematics


UDK 2.0
<Core Design>
RESET BUTTON
External Power
5 to 9V Input
Follow Part PDF on GND via placement on PPAD vias
MULTITECH


MULTITECH
UDK 2.0
<Core Design>
Default jumper placement
TxD: 3-4 populated, 5-6 open
RxD: 7-8 populated, 9-10 open
All other positions populated by default
JP95 usage:
Jumper pins 1-2: Disable U5 on USB VCC high
Jumper pins 2-3: Disable U5 when placed
Default jumper position is 1-2.
- 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, DISCLOSED TO OTHERS, COPIED FROM OR MODIFIED WITHOUT THE PRIOR WRITTEN CONSENT OF MULTI-TECH SYSTEMS, INC.
©2014 MULTI-TECH SYSTEMS, INC. ALL RIGHTS RESERVED


Socket Modem Universal Interface
MDot header
Pins 1-10
Pins 11-20
DNP for cell kit
MTQ Socket
Arduino Headers
Bypass for Arduino supply
UDK 2.0
MULTITECH


UDK 2.0
<Core Design>
MULTITECH
- 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, DISCLOSED TO OTHERS, COPIED OR USED IN ANY MANNER WITHOUT THE PRIOR WRITTEN CONSENT OF MULTI-TECH SYSTEMS, INC.
©2014 MULTI-TECH SYSTEMS, INC. ALL RIGHTS RESERVED
Target Voltage selector
Replace with 0105841IL when available
9 PIN HDR
Jlink Header
Pin 7 is used for Keying
Not Placed
mbed is default
R93 not placed
MBED_TMS
MBED_TCK
MBED_TDO
MBED_RST
JTAG_TMS
JTAG_TCK
JTAG_TDO
JTAG_RST
IL_SWDIO_TMS
IL_SWCLK_TCK
IL_SWO_TDO
IL_NRST
VDD_STLINK
VCC VDD_STLINK
LED_STLINK
LED1
LED2
RED COM
GRN STAT
AP7313
3.3V Fixed output
150mA capacity
DNP for cell kit
CR2 DMEG6010CEH
CR3 DMEG6010CEH
STM32F103C8T6
USB_DP
USB_DM
USB_PENUMn
VDD_USB
2N3904
HSPO61-2
4 PIN HDR
2 PIN HDR
5-PIN-USB
SWDIO
SWCLK
MBED_RX
MBED_TX
Y1 8MHz
Build Level
Size B
Document Number 387S
Rev B
Change No. 01
Date: Thursday, January 08, 2015
Sheet 5
Drawn
Checked
Approved
Title


UDK 2.0
<Core Design>
MULTITECH
- NOTICE -
THIS DRAWING REPRESENTS PROPRIETARY, CONFIDENTIAL AND TRADE SECRET INFORMATION OF AND IS OWNED EXCLUSIVELY BY MULTI-TECH SYSTEMS, INC. THIS DRAWING MAY NOT BE REPRODUCED, PUBLISHED, DISPLAYED/USED, DISTRIBUTED OR MODIFIED WITHOUT THE PRIOR WRITTEN CONSENT OF MULTI-TECH SYSTEMS, INC.
©2014 MULTI-TECH SYSTEMS, INC. ALL RIGHTS RESERVED

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.

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

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

USB Design

Multi-Tech 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.