cpNode-ProMini Assembly Instructions

Revision v1.2 05/07/23

Model Railroad Control Systems
www.modelrailroadcontrolsystems.com
Seth Neumann, *sales@modelrailroadcontrolsystems.com*

Introduction

This document describes the functional blocks of the cpNode Pro-Mini and how to assemble the cpNode PM.

Revision History

v1.0 02/21/22 SCN initial version
v1.1 05/07/23 SCN revisions to assembly order

Table of Contents

1. Table of tables

2. Table of Figures

3. CPNODE SYSTEM OVERVIEW

The cpNode consists of four main sections.

- Arduino processor board header connectors for I/O ports and system power.
- CMRInet RS-485 drivers, automatic transmit enable circuit (AutoRTS), and network cable terminal pads.
- I2C serial interface for I/O extender cards.
- I/O port option area and connection pads.

All components are through-hole technology for ease of assembly and repair.

Pads for I/O connections are standardized on .100" centers. This provides a wide range of interconnect options and components. The I/O connection design is followed throughout the boards created in the cpNode system. Port connection schemes include screw terminal blocks, header pin connectors (male and female), soldered right angle headers, and direct soldered wires.

The solder pad option area has one configurable area for each I/O port. The option area consists of through-hole resistor pads and a solder jumper. The option area is not connected by default. To connect an I/O pin from the Arduino to a connection pad, either a solder jumper or resistor (normally used for LED current limiting) must be present. Wires can be inserted into the resistor pads if soldering the jumper pad is not desired.

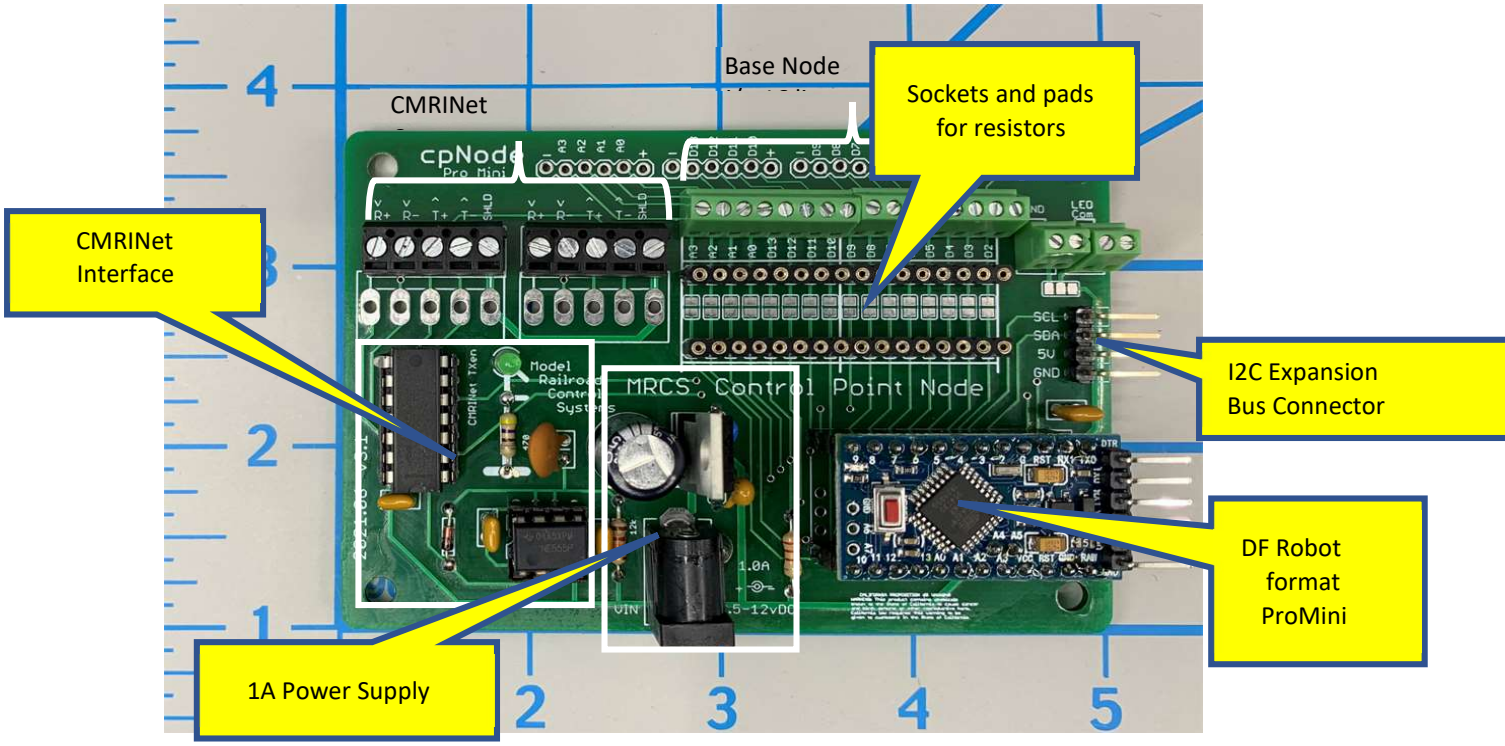

Figure 1- cpNode-ProMini Component Locations

4. ARDUINO™ PROCESSOR BOARD INTERFACE

The cpNode ProMini motherboard provides the physical interconnect to the Arduino ProMini and the input/output ports.

5. CMRINET (RS-485/RS-422) INTERFACE

The physical communication connection to CMRI serial nodes is RS422 / RS485, 4-wire, halfduplex. Network speeds of 9600, 19200, 28800, 38400, 57600, and 115200 BPS are supported. Transmit enable for the network driver chip is controlled by an AutoRTS circuit. The network connections are daisy chained from node to node for a maximum of 128 nodes. The cpNode supports the CMRI protocol through the RS-485 wire interface. The master or host end of CMRInet has an RS-232 or USB to RS485 interface, commonly referred to as a "dongle".

CMRI nodes are connected in daisy chain fashion with four-wire shielded cable. Four-wire cable connects to the RS485IN and RS485OUT terminals on the cpNode. The 5th position in the pad area provides a convenient tie point for the shield drain wire. This pad is not connected to any circuit on the board.

Three pad configurations are available; 0.156" (traditional Molex) spaced pads, 3.5mm spaced pads and 0.100 spaced pads for screw terminal blocks. All similar signals are wired in parallel. Either pad area can be used for the network in or network out connection. A 5th pad is present to tie the shield or drain wires together. This pad does not connect to any ground plane on the cpNode board and is provided for cabling convenience.

CMRInet signal names are TX+,TX- (Chubb OUT+, OUT-) and RX+,RX- (Chubb IN+, IN-).

A Maxim MAX489 or MAX 3465 driver chip (U1) provides the connection between the Arduino serial pins D0/RX and D1/TX, and the CMRI network. A socket is recommended for this chip.

6. AUTORTS INTERFACE

CMRInet operates in Master/Slave transmission mode. Each node has a unique numerical address. The Master polls each node, requesting data. The polled node responds with data. All nodes on the network receive the poll message, but only the addressed node responds. All nodes' transmit pairs are connected in parallel, as are the receive pairs.

The MAX489 (U1) driver chip contains a pair of line driver circuits, one for transmit and one for receive. The receive side is always enabled. The transmit side is only enabled when the node needs to transmit data to the Master. Traditional hardware implementations would dedicate a software controlled I/O pin to enable/disable the transmit driver.

To conserve I/O pins, transmit enable (TXEN) is controlled by a NE555 (U2) timer chip. The timer is set by the first bit of a byte being sent, holding TXEN high for the duration of the transmitted byte. Additional hold time is added to insure all of the bits are sent before the transmit drivers are disabled. The total enable time is line speed dependent and the timeout interval is set by R1 and C1.

A table of R1/C1 values by line speed is on the schematic. For 9600 and 19200 BPS, a value of .1uf for C1, and a value of 12k for R1 can be used. Testing at both these speeds confirm proper operation.

* AutoRTS Resistor Value

BPS	R1	C1
9600	12000	.1 uF
19200	6000	.1 uF
28800	3900	.1 uF
38400	3000	.1 uF
57600	2000	.1 uF
115200	1000	.1 uF

Table 1- AutoRTS Timing Component Values

7. I2C INTERFACE

To provide additional input and output ports to a cpNode, I/O extender boards (IOX) connect to the I2C serial bus. This serial bus is tailored to driving low-level devices like LED's and contact sensors. The bus is present in some automobiles and is the interface for many dashboard and interior control functions. The cpNode connects to the input/output extender boards with four-wire cable though the I2C connector (I2CHEADER). Four wire cable harnesses are available from various suppliers like Pololu and Sparkfun. A Molex 2.54mm KK style header will also work.

IOX boards are interconnected in daisy chain fashion using four wire .100 header cables.
An established color code for I2C wires is:

Pin 1 SCL Yellow
Pin 2 SDA White
Pin 3 +5v Red
Pin 4 Gnd Black

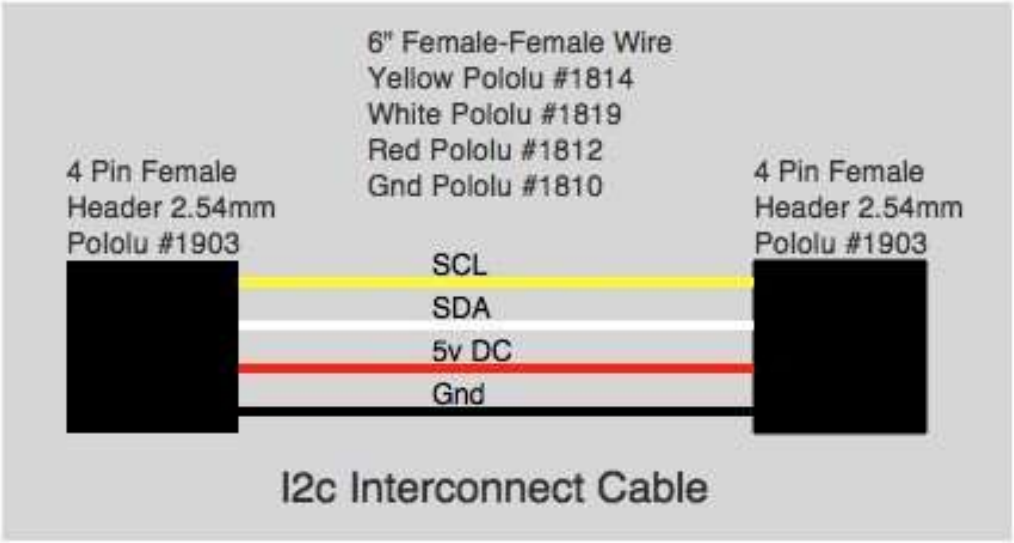

Figure 2- I2C Interconnects

8. GND AND LEDCOM CONNECTORS

There are two common tie points on the logic board. Both sets of pins were intended to provide a convenient GND/common tie point for wire connection at the node location on the layout.

GND is intended to be used as the input line for contact sensors. Just run the GND to one side of a push button, the other side of the button goes to a port defined as INPUT_PULLUP. A built-in weak pull-up resistor is enabled when a port is defined this way.

The LEDCom pins are connected to the CA/CC (Common Anode/Common Cathode) solder jumper pads. This connection is intended to provide a separate connect point for LEDs, either providing sourcing (+5v) or sinking (GND) which is the same common logic ground as the GND pins. The node design is focused on driving LED based signal heads.

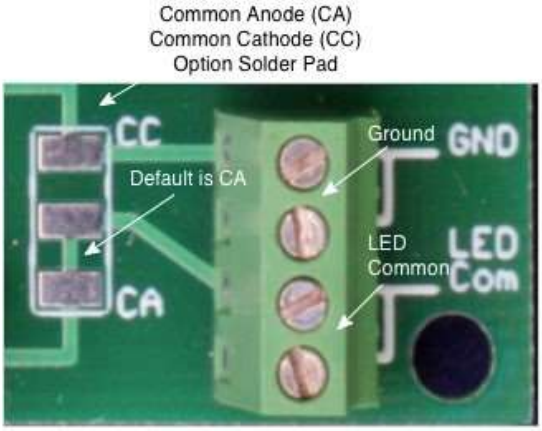

Figure 3- Ground and LED Common Connector

9. Bill of Materials

Qty	Value	Device	Package	Parts	Description
4	0.01uF	CAPPTH2 Monolithic	C-PTH-0.100	C2, C3, C4, C5	
2	0.100	CONNECTOR-M08LOCK_LONGPADS	1X08_LOCK_LONGPADS	J1, J2	Header 8
2	0.100	M02POLAR_LOCK	MOLEX-1X2_LOCK	GND, LEDCOM	Header 2
4	0.100	M081X08	1X08	IO1, IO2, IO3, IO4	Header 8
1	0.100	PINHD-1X4/90	1X04/90	I2CHDR	PIN HEADER
1	0.1uF	CAPPTH2 – Disk Ceramic	C-PTH-0.100	C1	
1	1.0uF	CAPPTH1 Monolithic	C050-024X044	C7	
1	10uF	CAPPTH1 Monolithic	C050-024X044	C6	
1	12k	RPTH04 – Auto RTS resistor	AXIAL-0.4-RES	R1	Auto RTS resistor
1	150uF	CAP-POLPTH-5MM	E5-10.5	C8	
1	1N4148	DIODE1N4148	DIODE-1N4148	D1	Diode
1	2.1mm	CONNECTOR-DC-POWER-RA	DCJ0202	J3	DC POWER JACK
1	2k2	RPTH04	AXIAL-0.4-RES	R3	Pull up for D13
2	3.5mm	CONNECTOR-CMRINET3.5MM	CMRINET-3.5MM	RS485-3, RS485-4	CMRINet Top 3.5mm Screw Terminal 5 pos
1	470	RPTH04	AXIAL-0.4-RES	R2	LED limiting resistor
1	5v	VREG-PTH-RA-	TO-220-RA	VR5.0	Voltage Regulator
1	BOARD	BOARD69X100MRCS	BOARD-DINRAIL-2.71IN-MRCS	BOARD1	
1	DAVEK	ARDUINO-PRO-MINIDAVESROBOT	ARDUINO_PRO_MINI_DAVEK	ARDUINO1	
1	G	LED3MM	LED3MM	TX	
1	MAX3465CPD	MAX3465CPD	DIL14	U1	+5V, Fail-Safe, 40Mbps, Profibus RS-485/RS-422 Transceivers
1	NE555P	NE555P	DIL-08	U2	General purpose bipolar Timer
1	socket 8		DIL-08		For 555

1	socket 14	For MAX3465/489
2	female header 12	
1	female header 2	

Table 2- Bill of Materials

10. ASSEMBLY

Start with the low parts and work towards the higher to support parts as you flip the board to solder from the bottom:

1. [ ] R1-R3, and D1 – I put the boards over a small box like a shake-the-box kit box. Tack solder the parts on the top to ensure everything is sitting down before you flip the board over and solder properly. I use very thin (0.015") rosin core solder to control the amount solder flowing in. **R3 – see notes on D13**
2. [ ] Add the sockets (1 8 pin, 1 14 pin). Do one pin first and check that it's sitting down properly before soldering the rest. Again, fine solder helps control the flow.
3. [ ] Install the 16 pin machined pin sockets and the I2C header.
4. [ ] Install the transmit activity LED, note the anode (long lead) goes towards the trace to the resistor
5. [ ] Install the monolithic capacitors. Note that the bare leads should be visible about 1/32nd above the board, so the components already installed should hold the board up. Do not install the disk ceramic yet.
    a. [ ] install C2-C5 .01 UF (103) note C5 is way over on the right by the Arduino
    b. [ ] install C7 1uf (105)
    c. [ ] install C8 10uF (106)
6. [ ] Install the disk ceramic (C1) capacitor
7. [ ] Install the Pro Mini headers: 2x 12 pin + 1 x 2 pin, you can use a ProMini as a fixture to hold them in alignment. Hint: pull the 2 pin header down a little when seating the headers to help locating the headers.
8. [ ] Install any 0.100 headers for output, and the LED and COM and the CMRI net connectors (if using 3.5 mm or 0.100 screw terminals or headers)
9. [ ] Install the DC power jack: use regular (~ 0.030") solder for the jack, allow the solder to form a meniscus and fill the holes
10. [ ] Install the Electrolytic Capacitor (>150 uF but anything upto 470 uF is fine)
11. [ ] Install the 5 Volt Regulator, align the fin with the bar on the board
12. [ ] Install Molex connectors for CMRINet if you are using them
13. [ ] Now is a good time to test the power supply: use a 12V wall wart or bench supply set to 12V and check that you are drawing about 3 mA, and that you are seeing 4.85 – 5.05 volts across the LED Com and I2C connectors.
14. [ ] Disconnect power and install the 555 and Max 1489 chips. Check the power again, you should be drawing about 5 mA.
15. [ ] Program the ProMini and assemble the pins (use the headers on board as a fixture) using the sketch.
16. Disconnect power from the cpNode ProMini and install the ProMini.
17. [ ] Power up again, you should see 18-23 mA drawn.