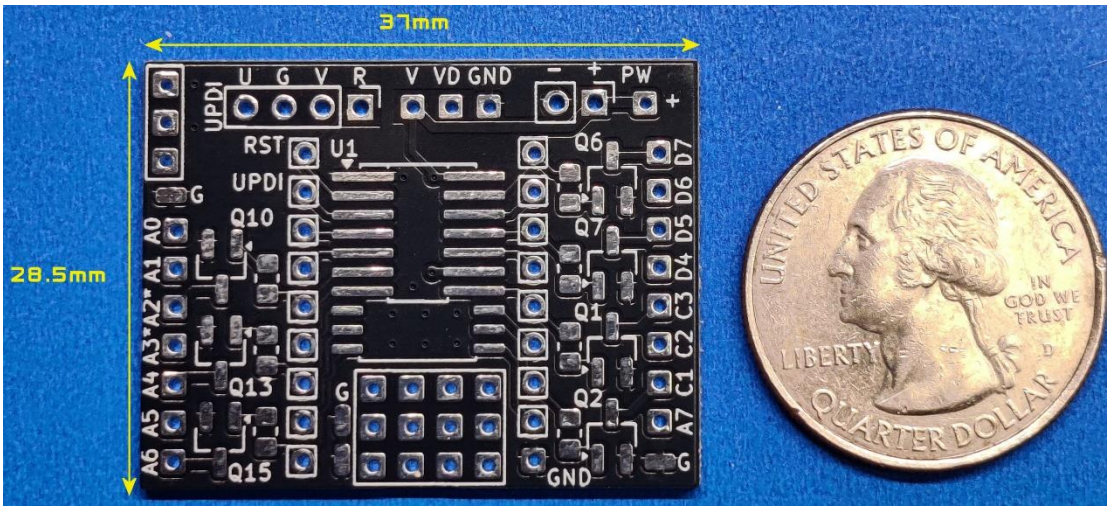
AVR SOIC20 Development Board

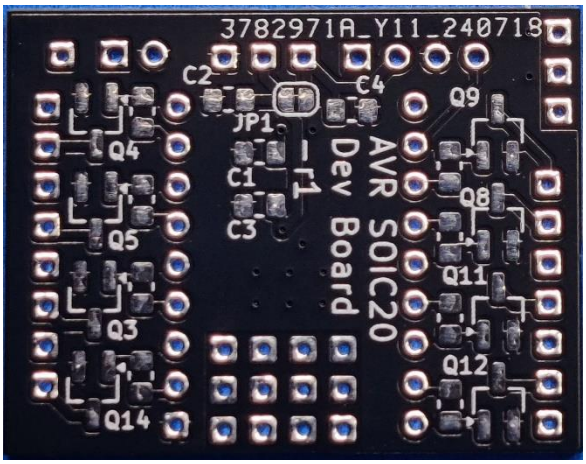
Contents

Description

Front



Back



The AVR SOIC20 development board supports a variety of AVR DD Series 14 and 20 pin SMD parts with a UPDI programming option (See the Board Details for a list of devices). The SOIC14 parts are in a narrow package with VDD on pin 14, GND on Pin 1, and the UPDI on pin 3. The SOIC20 parts are a wide package with VDD on pin 20, GND on Pin 1, and the UPDI on pin 3.

Connections to all IO pins are provided (Referenced as RST, UPDI, An/Cn/Dn)) as well as a sub set of connections through SMD resistors and MOSFET drivers. A total of 15 MOSFET transistors are available for the 20-pin package or optionally a bipolar junction transistor (BJT) can be used. For the 14-pin package up to 9 transistors are available. The only outputs that do not have a transistor option are Pins 2 & 3. See the *IO Pin Connection Planning Guide* at the end of this document for details.

A dedicated power connection is provided (PW) and other connection areas are marked as power (+) and ground (GND/G) for sourcing other components. The power supply range is determined by the selection of U1, typically 1.8-5.5 volts.

The IO outputs are setup to support 2.54mm pin strip headers so that the board can be used in breadboarding a design.

Programming can be accomplished using the UPDI header to program the mounted part on the board. I also have available a soft touch programming cable (No UPDI socket needed). There are many YouTube videos on Arduino programming options and I discuss my favorite in the UPDI Header section.

This board was designed to be as small as possible while providing numerous connection options and a set of MOSFET drivers for LED lighting in scale model builds. Typically used in scale model builds, dioramas or other areas where a small compact SoC (System on a chip) is needed.

Board Details

- Dimensions: 37 x 28.5 mm
- Parts Supported: Designed for the AVR DD series with 14 pin narrow or 20 pin wide package, VDD pin 14/20, VDD2 pin 9/15, GND pin 1.
    - UPDI pin 3 Programming
    - AVR16DD14/20, AVR32DD14/20, AVR64DD14/20
- Support for a separate VDDIO2 available
- Up to 15 MOSFET or BJT drivers supported with 20 pin packages or 9 with the 14 pin packages, see the *IO Pin Connection Planning Guide* at the end of this document for the fixed outputs.
- Programming using the UPDI port.

Parts List

Part **Reference**	**Quantity**	**Value**	**Description**	**Source Links**
U1	1		SOIC14 – Narrow SOIC20 - Wide AVR16DD14/20 AVR32DD14/20 AVR64DD14/20	Mouser Electronics https://www.mouser.com/c/semiconductors/embedded-processors-controllers/microcontrollers-mcu/8-bit-microcontrollers-mcu/?q=AVR&package%20%2F%20case=SOIC-14~~SOIC-20&series=AVR16%2F32DD14%2F20%7C~AVR64DD14%2F20&instock=y&rp=semiconductors%2Fembedded-processors-controllers%2Fmicrocontrollers-mcu%2F8-bit-microcontrollers-mcu%7C~Package%20%2F%20Case
Q1- Q15	9 15	A2SHB	MOSFET A2SHB/SI2302 SOT23 SMD 9 – SOIC14 15 – SOIC20	Mouser Electronics https://www.mouser.com/ProductDetail/Vishay-Semiconductors/SI2302CDS-T1-E3?qs=%252BPu8jn5UVnHNrjAmGCs%2Fuw%3D%3D AliExpress
R1-15 (No Board Reference)	9 15	470Ω	0805 SMD/TH Resistor for transistor drivers	Mouser Electronics https://www.mouser.com/ProductDetail/Vishay-

Part Reference	Quantity	Value	Description	Source Links
			9 – SOIC14 15 – SOIC20	Draloric/RCG0805470RJNEA?qs=vOeJqewp7jBU33bjXc%252BrVQ%3D%3D
C4	1	1uf	0805 SMD Decoupling cap	Mouser Electronics https://www.mouser.com/c/passive-components/capacitors/ceramic-capacitors/mlccs-multilayer-ceramic-capacitors/multilayer-ceramic-capacitors-mlcc-smd-smt/?q=0805%20capacitor&capacitance=22%20pF%7C~0.1%20uF%7C~1%20uF&instock=y
C1	1	0.1uf	0805 SMD Decoupling cap	Mouser Electronics
C2	1	1uf	0805 SMD Decoupling cap - Optional, for VDDIO2	Mouser Electronics
C3	1	0.1uf	0805 SMD Decoupling cap - Optional, for VDDIO2	Mouser Electronics
UPDI	1	1x4	1x4 pin socket 2.54 mm pitch - Optional for programming	Male/Female socket header based on programming needs.
PCB	1		AVR SOIC20 Dev Board	

Pre-Assembled Boards

If you purchased an assembled PCB your board will be assembled based on the option you selected:

- **Option 1/2**: Board assembly with
  - AVR 32DD14 or AVR64DD20
  - All decoupling capacitors
  - This option will include all 9/15 MOSFETS loaded with their corresponding 470Ω resistors
  - Parts listed as OPTIONAL will NOT be loaded (UPDI/PW Header, C2, C3 (VDDIO2))

- To test the board a test program will be loaded to U1 and used to check all output pins. It is a simple high/low test pattern applied to each pin about every ¼ second or so.
- You should be able to power up your board and see that same test pattern if your assembled board included U1. I'd recommend doing this before you reprogram the part in case something happened in transit.
- See the Board Options/Configuration section for configuring MOSFET outputs.

Assembly Guide

Caution: Electrostatic discharge (ESD) is a sudden and momentary flow of electric current between two differently-charged objects when brought close together or when the dielectric between them breaks down, often creating a visible spark associated with the static electricity between the objects. [1]

This type of shock can cause damage to ESD sensitive parts such as those used in this build especially U1. Proper ESD protection and soldering equipment should be used to prevent damage to parts during assembly and implementation into your project.

Assembly Planning

The smallest components are 0805 and while small can still be hand soldered with care and patience. A fine tip soldering iron is useful along with 0.015" (0.38mm) flux core solder and extra flux if needed. See the references section for a YouTube video link on assembling this and other boards.

A note on connector sockets: The UPDI and PW locations support 2.54mm pitch sockets. However, I have found that these can cause a height issue with scale models as space can be very limited. For flexibility I usually wire directly to the board or use in-line connectors to keep the board height to a minimum. An angled 1x4 header for the UPDI can be better than a vertical one or the use of a soft touch programmer cable eliminates the need altogether.

PCB Assembly

- PCB assembly can be completed in any order.
- If using a hot plate or reflow heater I usually start with the side with the most SMD or hardest parts to hand solder and then hand solder the other side.
- If you are completely hand soldering my recommendation is to complete the back of the board first by mounting the capacitors C1 & 4. If you are using VDDIO2 as a separate supply then mount C2 & 3.
- Next install any transistor output drivers for LED's or other needs. Depending on your design of input and output signals you may not want to mount all of the MOSFET or BJT parts and the associated resistors. *I've included a design planning table at the end of this document that can be used to help lay out your design and connection options.*
    - SOIC14 bottom: Only uses Q4,5,8,9,11.
- Continue to mount the 470Ω SMD (or a value of your choice in SMD or TH) resistors for each transistor installed.
- Moving to the top of the board install U1.
- Repeat the mounting of any other resistor/transistor pairs for the top layer.
    - SOIC14 top: Only uses Q1,6,7,10.

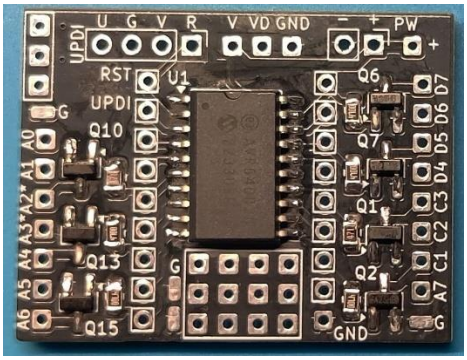
  A note about MOSFET transistors. To keep the board size as small as possible I did not include any gate pull down resistors. These are usually used to prevent signal instability when U1 initializes the output pins on power up. If you feel these are needed for your design an appropriate value resistor can be added across the

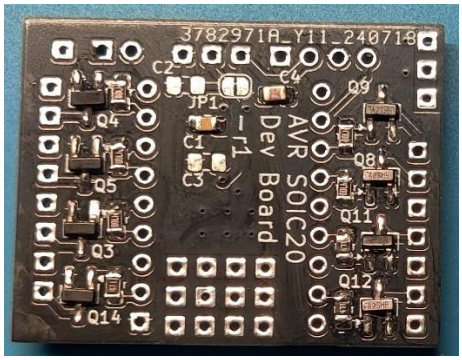
[1] Definition provided by From Wikipedia, the free encyclopedia. For more information on ESD see https://en.wikipedia.org/wiki/Electrostatic_discharge

gate/source pins of the MOSFET. They are generally not needed when using BJT transistors.

- Determine how you will program the part and if needed install a 1x4 (2.54mm pitch) header for UPDI.
- If you will be using a connector for power then install a 1x2 (2.54mm pitch) header at PW.
- See the Board Options/Configuration Section if you will be splitting the power supply to the device.

Assembled Images (SOIC 20 pin part as an example)





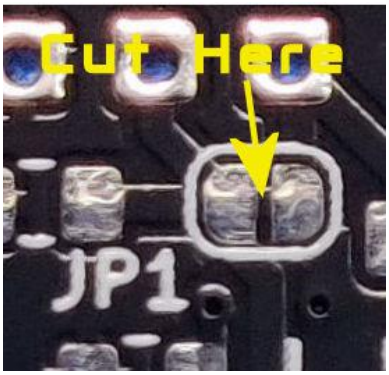
Board Options/Configuration

This section should be reviewed for those that are assembling the board themselves or purchased a pre-assembled version.

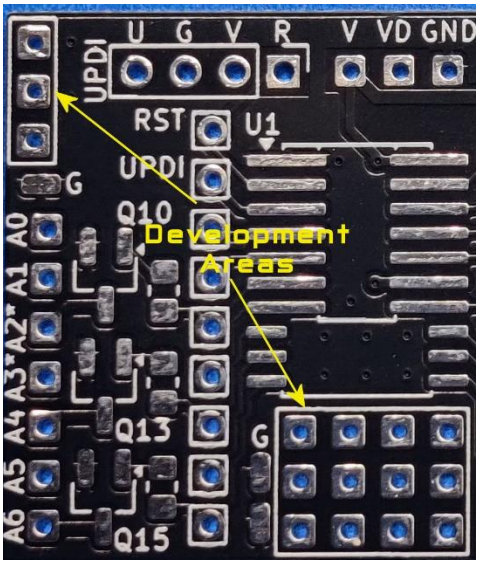
VDDIO2 Option

By default, the VDDIO2 supply is connected to the main board supply. JP1 can be used to break this connection so a separate supply voltage can be used for VDDIO2. The decoupling capacitors C2 and 3 should be mounted and then the VD pad can be connected to the appropriate voltage.

To cut JP1 use a Xacto knife or Dremel cutter to carefully cut the trace between the two pads. Should you decide not to use the split voltage at a later date then you can use a solder bridge to reconnect the JP1 pads.



IO Connections

Power	**PW +/-** Connect an appropriate power source to the PW connector. There are also extra power and ground connections for other needs. (+ / GND, G)
Reset	**RST** The reset can be used as an input.
UPDI	Programming connection for a serial programmer to program U1 on the board. Supports UPDI mode.
IO Ports	**Round Pads** **UPDI pin** This connection does not have transistors to drive a load. **An/Cn/Dn** These remaining connection points with round pads are to the device IO lines and bypass the transistor and resistors. Note: Reference A2*/A3* will be port C1/2 for 14 pin parts. See IO Pad Sections below for examples.
Transistor Outputs	**Square Pads** **An/Cn/Dn** These connections align with the devices port outputs and have a resistor/transistor to drive a load. The transistors (MOSFET/BJT) sink to ground. See IO Pad Sections below for examples.
Development Areas	This board used some available free space as a development area for adding other components (Resistors, transistors, diodes, sensors, connectors, etc…). There are even some surface ground pads for attaching connections. 

IO Pad Sections

Each grouping of IO ports has two connection options.

- IO Port Output - Connect to the port IO output directly (Round Pad - No resistor or transistor driver)
- Transistor Output - Connect to the transistor output driven by this port (Square Pad)

.

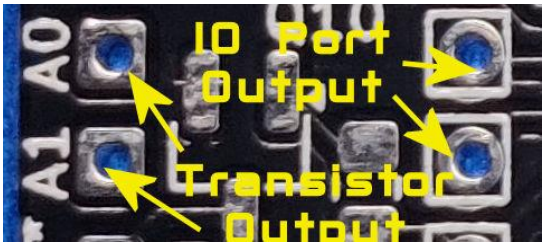
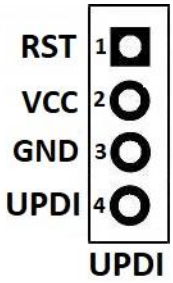
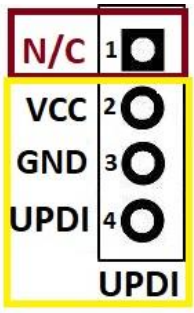


Figure 1 Example for A0 & A1

UPDI Header

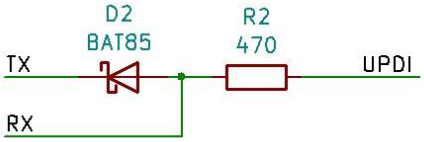


The UPDI connector follows this layout which is the recommended pin out from Microchip. There are a number of YouTube videos showing various methods for programming including using a USB to TTL Serial Converter such as the CH340 and a few components on the TX/RX lines.

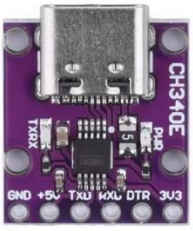


I also offer a soft touch programming cable to eliminate the UPDI header and connect directly to the board.

It is a 3-pin cable and connects to pins 2-4 of the UPDI header and includes the resistor and Schottky diode on the TX/RX lines.



I have used the CH340E and CH9340C USB C converters and I like the CH340E the best. They can be acquired from AliExpress or from my store.



Mounting Options

Hot glue is my go-to option for PCB mounting in models. It has great hold and sets up quickly. It can easily be removed and reapplied. Double sided tape or possibly Velcro could also be used.

References

- **Github: Development board documentation and schematics.**
  - **https://github.com/JohnnyElectronic/Dev_Boards/**
- **YouTube: Board assembly and project videos that are related to this board.**
  - **https://www.youtube.com/@Johnny_Electronic**
- **SerialUPDI**
  - **https://github.com/SpenceKonde/AVR-Guidance/blob/master/UPDI/jtag2updi.md**
- **Arduino IDE board files**
  - **DxCore - https://github.com/SpenceKonde/DxCore**
  - **It can be installed manually or through the Boards Manager**

Revisions

R1	First release

Disclaimer

This information is provided “as-is” with no representation or warranty of any kind whether express or implied. However, I’ve tried to make this document (as well as the supporting videos) as useful and accurate as possible. If you find something that is incorrect or confusing, please let me know as I would like to make the correction so others will not have the same issue.

Feel free to email me for issues you may have with this board or if you need extra help with coding, programming, or just design ideas for your latest project please check out my Patreon page.

johnnyelectronic1@gmail.com

Legal note

IO Pin Connection Planning Guide

Use this guide to help plan out your inputs/outputs/LED driver connections. It has helped me during planning and final assembly.

AVR SOIC20

- RED Sections: Pin 2,3, No transistor option.
- When directly driving an output (No MOSFET/BJT) there is a 20ma max per pin and a 100ma total for all pins or per device.
- MOSFET's can handle a few 100 mA without issue, the ones selected are rated to over 2 Amps but watch for excessive heat.

CONNECTED TO WHAT?	INPUT/OUTPUT PINS/PORTS		CONNECTED TO WHAT?
N/A - GND	1	20 / 14	N/A - VDD
	2 (RST)	**19 / 13** (D7) Q6	
	3 (UPDI)	**18 / 12** (D6) Q4	
	4 (A0) Q9	**17 / 11** (D5) Q7	
	5 (A1) Q8	**16 / 10** (D4) Q5	
	6 (A2* / C1) Q10	15 / 9	N/A – VDDIO2
	7 (A3* / C2) Q11	**14 / 8** (C3) Q1	
	8 (A4) Q13	**13** (C2) Q3	
	9 (A5) Q12	**12** (C1) Q2	
	10 (A6) Q15	**11** (A7) Q14	