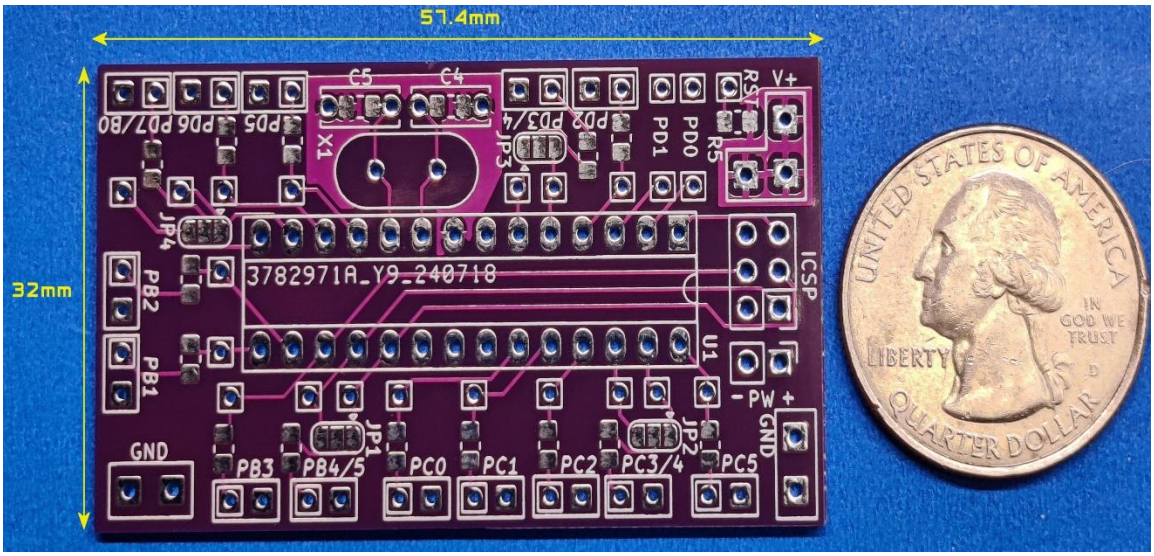
ATMega 328 Development Board

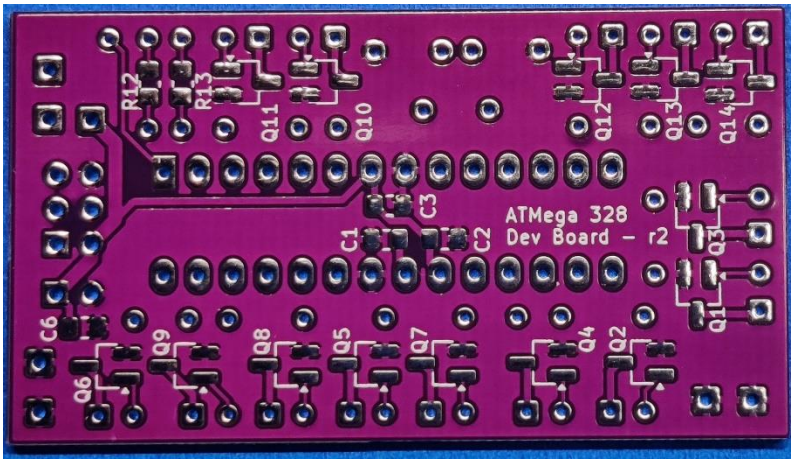
Contents

Description

Front



Back



The ATMega 328 development board supports the ATMega44/88/168/328 28 pin through hole (TH) parts although the 328 is probably the only one easily available. Connections to all IO pins are provided as well as a sub set of connections through SMD resistors and MOSFET drivers. A total of 14 MOSFET transistors are available or optionally a bipolar junction transistor (BJT) can be used. Some outputs share a MOSFET and the output pin can be selected by a solder jumper (JP1-4). Resistor pads are available for the RX and TX lines if needed for serial communication or they can be used for general IO. 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 (V+) and ground (GND) for sourcing other components. The power supply range is determined by the selection of U1, typically 1.8-5.5 volts. However, support for the 16 Mhz crystal requires a minimum of 4.5 volts although I've used it as low as 3.7 volts. If you need to use lower voltages you may need to look at running the device at a lower frequency.

Programming can be accomplished using a chip programmer for the ATMega part or using the In-Circuit Serial Programming (ICSP) port to program the mounted part on the board. I also have available a soft touch ICSP programming cable (No ICSP socket needed) that can be used with an Arduino UNO or NANO type of device as the programmer. You can also optionally use a 28-pin socket and then remove the part for programming. There are many YouTube videos on Arduino programming options.

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: 57.4 x 32 mm
- Parts Supported: ATMega 44 / 88 / 168 / 328
- Up to 14 MOSFET or BJT drivers supported, see the IO Pin Connection Planning Guide at the end of this document for the 10 fixed outputs and the 4 sets of outputs (8 I/O ports) that can be configured.
- Programming using the ICSP port or directly to the removed part.

Parts List

Part Reference	**Quantity**	**Value**	**Description**	**Source Links**
U1	1		ATMega 28 pin DIP ATMEGA328-PU ATMEGA328P-PU	Mouser Electronics https://www.mouser.com/ProductDetail/Microchip-Technology/ATMEGA328P-PU?qs=K8BHR703ZXguOQv3sKbWcg%3D%3D
Socket	1		28 pin DIP TH Socket 2.54 mm pitch - Optional	Mouser Electronics https://www.mouser.com/ProductDetail/Adam-Tech/ICS-328-T?qs=FG09h9tFCuC6Ck%2FzUH9arw%3D%3D
X1	1		16Mhz Crystal (TH)	Mouser Electronics https://www.mouser.com/ProductDetail/ABRACON/ABL-16.000MHz-B4Y?qs=sGAEpiMZZMsBj6bBr9Q9acsm1aZFaUGXsH5khlLoENx8BbEll4UD1w%3D%3D

Part Reference	Quantity	Value	Description	Source Links
Q1- Q14	14	A2SHB	MOSFET A2SHB/SI2302 SOT23 SMD	Mouser Electronics https://www.mouser.com/ProductDetail/Vishay-Semiconductors/SI2302CDS-T1-E3?qs=%252BPu8jn5UVnHNrjAmGCs%2Fuw%3D%3D AliExpress
R (Unmarked)	14	470Ω	0805 SMD/TH Resistor for transistor drivers - Can also use 1/8 or 1/4 watt through hole parts.	Mouser Electronics https://www.mouser.com/ProductDetail/Vishay-Draloric/RCG0805470RJNEA?qs=vOeJgewp7jBU33bjXc%252BrVQ%3D%3D
R5	1	10kΩ	0805 SMD - Reset pullup	
R12,13	2	TBD	0805 SMD or TH - Optional load - Can also use 1/8 or 1/4 watt through hole parts.	
C1,C6	2	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
C2,C3	2	0.1uf	0805 SMD Decoupling cap	Mouser Electronics
C4,C5	2	22pf	0805 SMD or TH	Mouser Electronics
ICSP	1	2x3	2x3 pin socket 2.54 mm pitch - Optional for programming	Male/Female socket header based on programming needs.
PCB	1		ATMega Dev Board	

Pre-Assembled Boards

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

- **Option 1**: Board assembly with supporting parts only, **NO ATMega328**.
  - 28 pin DIP Socket
  - 16 MHz Crystal with 22pf capacitors
  - All decoupling capacitors
  - R5, 10kΩ resistor
  - This option will include all 14 MOSFETS Loaded with their corresponding 470Ω resistors.
  - These OPTIONAL parts will NOT be loaded (ICSP/PW Header)
- **Option 2**: Board full assembly with supporting parts and ATMega328.
  - This is all of Option 1 and includes the ATMega328, ATMEGA328-PU
  - No bootloader installed

- To test the board a test program will be loaded to U1 (Either the one shipped with the assembled board under option 2 or my own test chip if you selected option 1) 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 selection 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.

The board was not specifically designed to support through hole resistors but since connections are available at the IO pin and transistor base/gate connection a through hole resistor could be used instead of the SMD version see the IO Pad Sections for examples.

A note on connector sockets: The ICSP 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 2x3 header for the ICSP can be better than a vertical one or the use of a soft touch programmer cable eliminates the need altogether.

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

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 (C1,2,3,6).
- Next determine if you will use the optionally loaded R12/13 resistors. They can be whatever is needed for your design or used for serial port connections.
- 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.*
- Moving to the top of the board, install R5 for a stronger reset pullup. The reset pin has a weak internal pull up but a stronger one may be desired.
- Continue to install the remaining 470Ω SMD (or a value of your choice in SMD or TH) resistors for each transistor installed. These are un-referenced parts to reduce the board size.
- 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 gate/source pins of the MOSFET. They are generally not needed when using BJT transistors.
- You can now install the crystal (X1), capacitors (C4/5) and the 28 pin socket if used. Note that C4 and C5 can either be SMD or through Hole (TH) parts but do not install both!

  It can be useful to bend the crystal pins and a few of the socket pins to hold the part in place for soldering. Once one pin is soldered check that the part is flush with the board. If not just reheat the connection while pressing the part flush with the board. You can then solder the remaining pins/wires.
- Determine how you will program the part and if needed install a 2x3 (2.54mm pitch) header for ICSP.
- If you will be using a connector for power then install a 1x2 (2.54mm pitch) header at PW.

Board Options/Configuration

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

Using the table below determine which shared drivers will be used, if any, and solder the appropriate side of the jumper to make the connection.

Jumper Options

JP1	1-2 – PB5 2-3 – PB4	Q4
JP2	1-2 - PC4 2-3 - PC3	Q9
JP3	1-2 - PD4 2-3 - PD3	Q10
JP4	1-2 – PD7 2-3 – PB0	Q14

Making a solder bridge

You can make your connection by selecting which half of the bridge to connect but make sure you do not connect both parts. The center pad will connect to either the upper or lower pad, ie pad 1-2 or 2-3. Once you determine the pads to connect add some solder to each pad then continue to heat both pads adding more solder if needed until the two pads are connected. The images below show some examples:

Step 1
Example bridging pad 1 & 2

Step 2
Completed Bridge

Bad Solder Bridge
All pads connected

IO Connections

Power	**PW +/-** Connect an appropriate power source to the PW connector. There are also extra power and ground connections for other needs. (V+ / GND)
Reset	**RST** This connection does not have a transistor to drive a load. The reset can be used as an input and depending on your design you may want to remove or change the R5 pull up resistor. *Use care if setting this port as an output as this is one of the programming pins and cannot easily be reverted back to be a Reset.*
ICSP	Programming connection for a serial programmer to program U1 on the board.
IO Ports	**PD0/1** These connections do not have transistors to drive a load but do have a resistor pad available (R12/13). **PD/PC/PB** These remaining connection points are to the device IO lines and bypass the transistor and resistors. See IO Pad Sections below for examples.
Transistor Outputs	**PD/PC/PB** These connections align with the devices port outputs and have a resistor/transistor to drive a load. The transistors (MOSFET/BJT) sink to ground. If you only wanted to use the resistor in-line with the output pin you can remove the transistor and add a jumper. This could allow using the output line to drive a single LED or other lower current parts. You also have the option of using through hole resistors instead of surface mount. See IO Pad Sections below for examples.

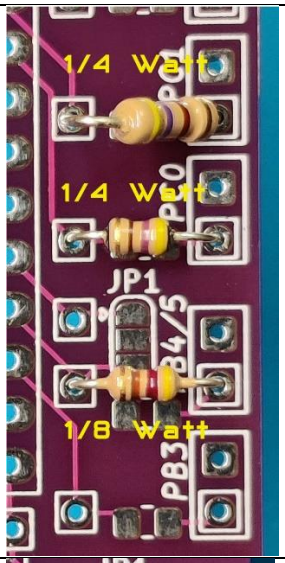
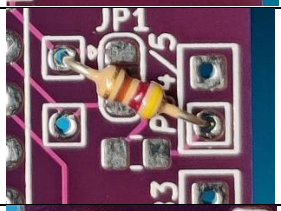
IO Pad Sections

Each grouping of IO ports has three connection options.

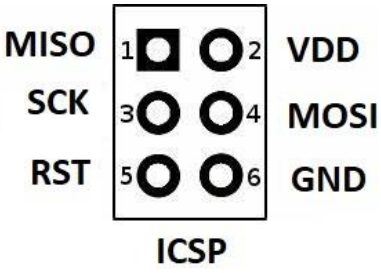
- IO Port Output - Connect to the port IO output directly (Round Pad - No resistor or transistor driver)
- Resistor/Gate Output - Connect to the port IO output through a resistor
- Transistor Output - Connect to the transistor output driven by this port (Square Pad)

See Figure 1 below for an example for PC0.

Figure 1 Example for Port C-0

Each pad section that has a resistor/transistor combination has the option to use through hole (TH) resistors instead of surface mount. This example shows using an 1/8 watt and two types of 1/4 watt resistors in place of the SMD.	
When using TH resistors for sections that have a solder jumper you can just make the connection directly to the output you want.	
If you are only going to use the resistor and not load the associated transistor then one end of the resistor can be bent to connect to the adjacent pad and that pad used for a wire connection.	

ICSP Header



The ICSP connector follows this layout which is the same used for Arduino boards. There are a number of YouTube videos showing various methods for programming including using UNO or Nano boards as an AVR In System Programmer using the ArduinoISP sketch.

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

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**

Revisions

R1	First release
R2	Improved board layout

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.

ATMega168/328 (28 pin DIP)

- ~ Marks PWM pins
- Colored sections share the transistor (Q#) listed on the ATMega Dev Board
- RED Sections: Pin 1-3, RST, TX, RX.. 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?
10k Reset Pull Up Reset, can be used for other purposes.	**1** (PC6) RST	**28** (PC5) D19/A5 Q6	
Optional in-line resistor I/O, no driver	**2** (PD0) RX D0	**27** (PC4) D18/A4 Q9 -JP2 (1/2)	
Optional in-line resistor I/O, no driver	**3** (PD1) TX D1	**26** (PC3) D17/A3 -JP2 (2/3)	
	4 (PD2) D2 Q11	**25** (PC2) D16/A2 Q8	
	5 (~PD3) D3 Q10 -JP3 (2/3)	**24** (PC1) D15/A1 Q5	
	6 (PD4) D4 -JP3 (1/2)	**23** (PC0) D14/A0 Q7	
N/A - VCC	7	22	N/A - GND
N/A - GND	8	21	N/A - ARef
N/A - Crystal	9	20	N/A - VCC
N/A - Crystal	10	**19** (PB5) D13 Q4 -JP1 (1/2)	(Typical Blink LED location)
	11 (~PD5) D5 Q12	**18** (PB4) D12 -JP1 (2/3)	
	12 (~PD6) D6 Q13	**17** (~PB3) D11 Q2	
	13 (PD7) D7 Q14 -JP4 (1/2)	**16** (~PB2) D10 Q3	
	14 (PB0) D8 -JP4 (2/3)	**15** (~PB1) D9 Q1	