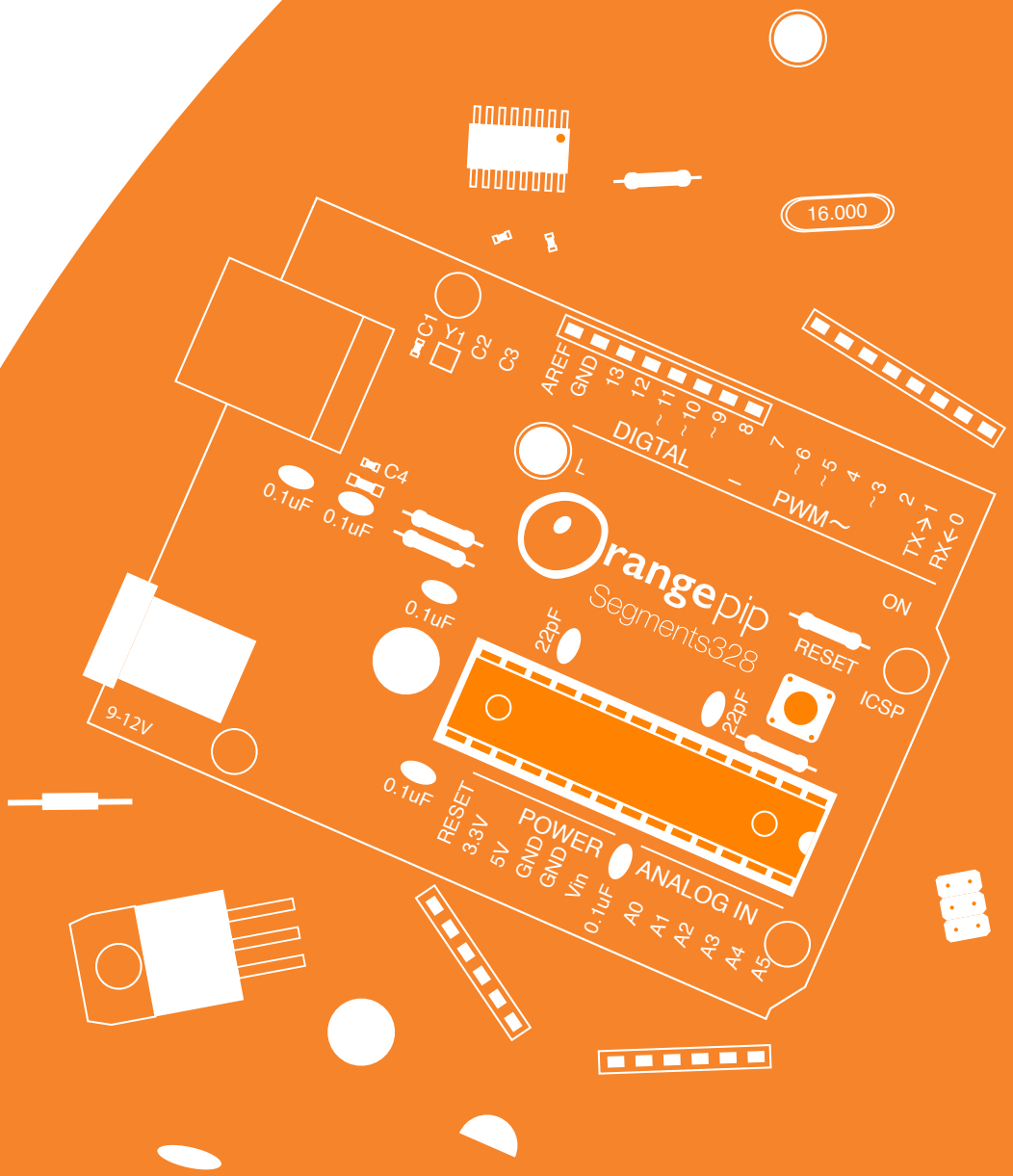
Build Instructions



rangepip®
Segments328

Check contents before you build!

1x

Orangepip Segments PCB

1x

16MHz crystal

1x

ATmega328 Microcontroller

1x

Dual in line socket

1x

Tactile switch

1x

Red LED

1x

Green LED

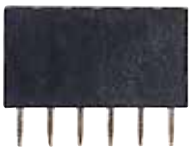
1x

6-pin dual row header

2x

8-pin single row sockets

2x



6-pin single row sockets

1x

USB socket

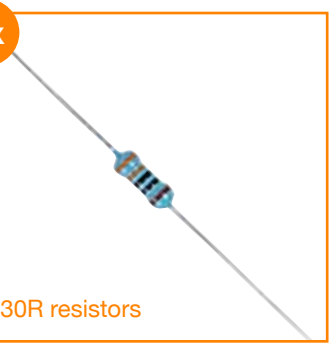
1x

Power supply jack socket

1x

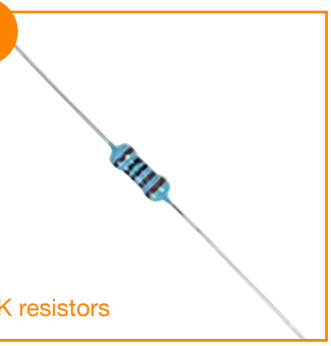
10K resistor

2x



330R resistors

2x



1K resistors

2x

47uF electrolytic capacitors

1x

1N4007 diode

1x

7805 voltage regulator

1x

7133 voltage regulator

1x

PTC resettable fuse

5x

0.1uF ceramic capacitor

2x

22pF ceramic capacitor

How to build your …
Orangepip Segments328

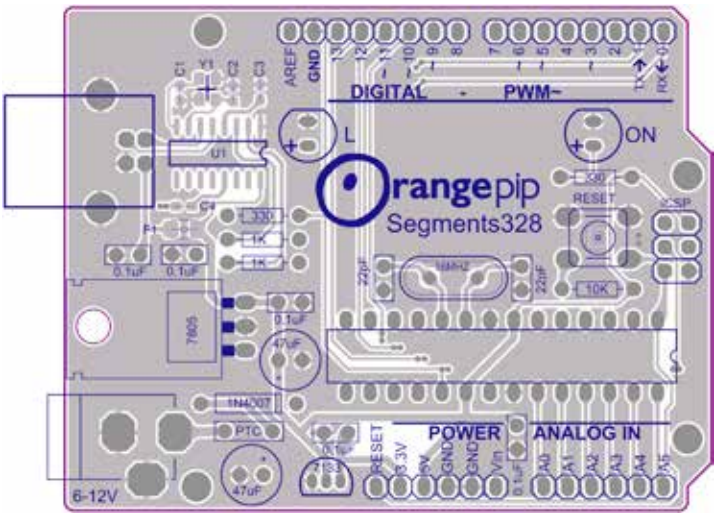
To put your Orangepip Segments328 together you will need basic soldering skills. When soldering you should always work on a flat surface in a well-ventilated environment. Make sure you have plenty of room on your workspace.

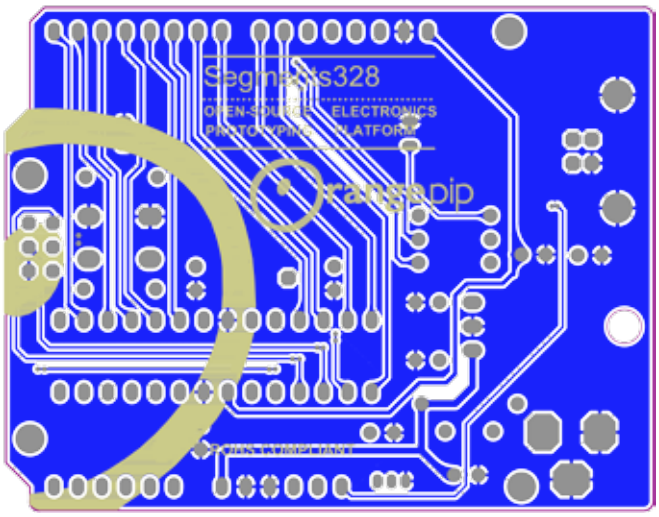
Equipment you will need …

					
Soldering Iron	Solder	Side Cutters	Snipe nose pliers	Multimeter	Assembly aid/ helping hand

Check your kit before starting, to make sure you are not missing any components.

The circuit diagram for your Orangepip Segments





1.0

Locate the bag containing the resistors. This will include 5 pieces in total. Sort them carefully into their respective values using the illustrations below.

330R x 2 pieces.
Orange, Orange, Black, Black, Brown

1K x 2 Pieces. Brown, Black, Black, Brown, Brown

10K x 1 piece. Brown, Black, Black, Red, Brown

There are three ways to identify the value of a resistor:

1 Using a Multimeter

- Polarity doesn't matter to the measurement.
- It removes the possibility of misreading one of the resistor colour bands.
- It can be a quick way to check when you have multiple resistors.

2 Resistor Card

- It doesn't need power or the Internet so you can use it anywhere
- It teaches you to read the values without the aid of any equipment

3 Online guide

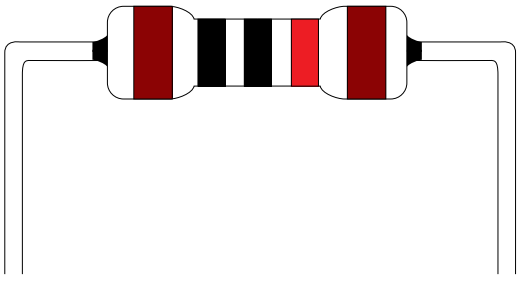
- Inputting the resistor colour bands will give you the exact value of the resistor
- The calculations are done for you which makes it easier to use than a resistor card.
- You just need to accurately identify the colours to get the correct result.

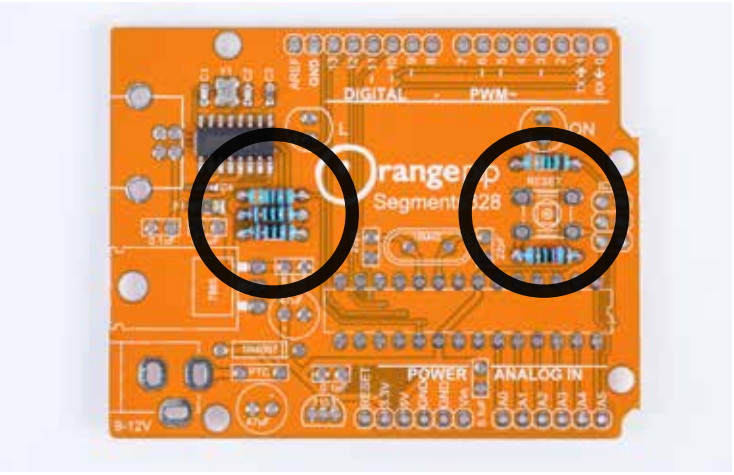
TIP

To be 100% sure you can use a multimeter to check the value of the resistors. It can be easy to misread a 1% tolerance brown band as the 1st digit!
SEE OUR MULTIMETER VIDEO FOR DETAILS.

1.1

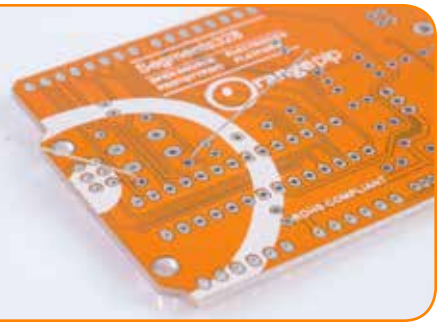
Bend the resistor legs 90° and then insert each resistor into the indicated location on the board and solder in place





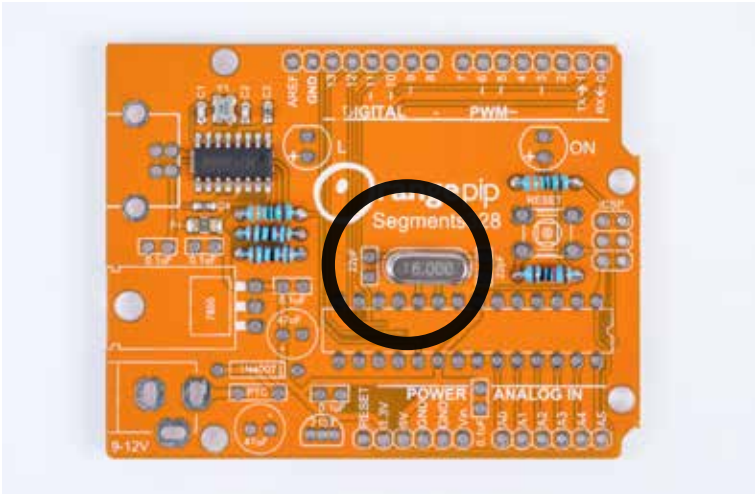
TIP

Bend the legs outwards to prevent the components from dropping out of the PCB while soldering.



2.0

Locate the crystal and place into the board. Use the leg bending tip from earlier to keep the crystal flush to the PCB and solder in place.

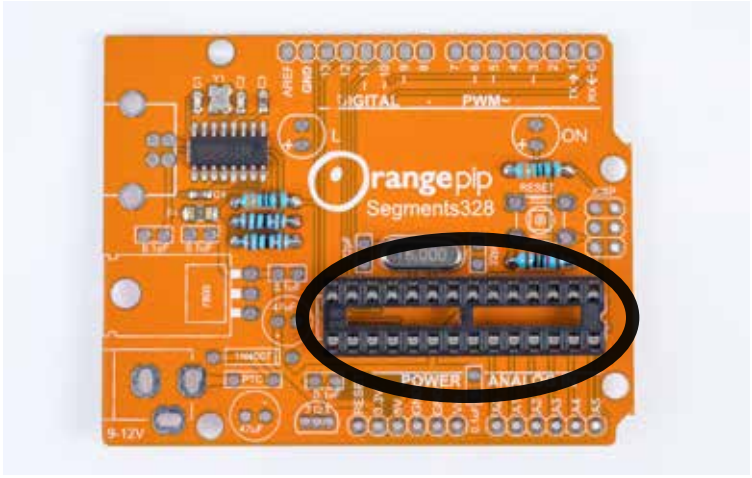


2.1

Locate the bag containing the microcontroller and the DIL socket.

The chip carrier socket is not polarity sensitive, but you will notice that there is a notch in one end. This is used to assist with the correct placement of the Microcontroller, which is polarity sensitive. The PCB, the DIL socket and the Microcontroller all share the notch marking to aid correct placement.

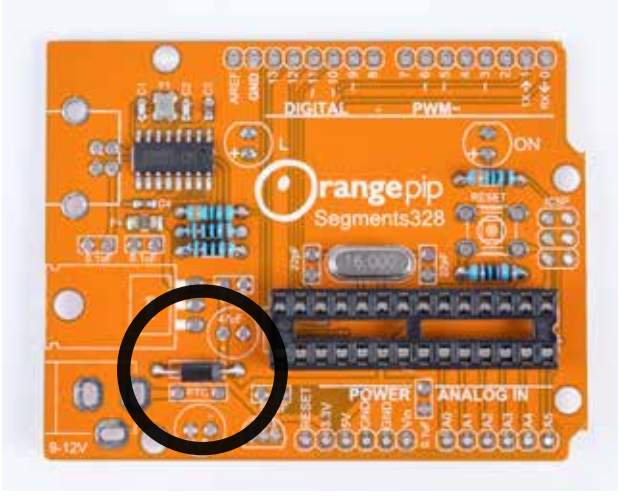
Place the correctly orientated DIL socket into the PCB and solder into place.



2.2

Next it's the 1N4007 diode. This is the first polarity sensitive item that we are going to add to the board. A polarised component must be connected to the circuit the correct way round. Diodes only allow current to flow in one direction from the anode to the cathode. We can identify the cathode by the silver band. The PCB has a corresponding mark to match this up with.

Place the component into the PCB with the silver band matching the marking on the PCB and solder in place.



2.3

Separate the ceramic capacitors from the other components. The 0.1uF capacitors are marked with the number **104** and the 22pF are marked with the number **22**.

2.4

Place the 0.1uF (104) capacitors into the marked locations on the PCB and solder in place.

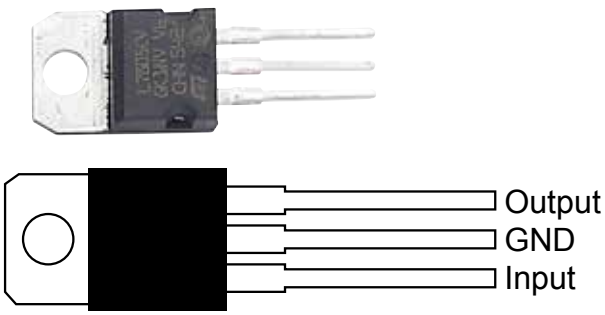
2.5

Place the 22pF (22) capacitors into the marked locations on the PCB and solder in place.

2.6

Locate the 7805 voltage regulator (L7805CV).

Just like the diode the voltage regulator needs to be placed in the PCB the correct way round. The diagram on the right shows the pinout.



2.7

In the image on the right you can see that the component has been laid flat against the PCB to offer a lower profile. This can be done with a pair of needle nose pliers or by pushing the component into the board and applying pressure to flatten the device.

TIP

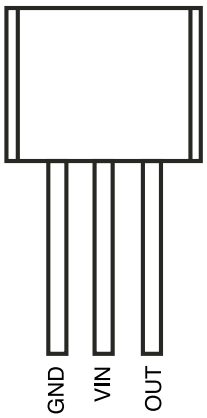
If you have any doubts about doing this, the product can be soldered upright with no negative effects other than an increased height profile. If you decide to do this then we recommend saving it until last.

2.8

Next, the tactile switch. Push the tactile switch into the PCB. The component is not polarity sensitive and can be inserted either way round. The kink in the legs will ensure it stays in place while being soldered.

2.9

Locate the 7133 regulator. This regulator is polarity sensitive and needs to be inserted into the PCB the correct way round. The marking on the PCB is shaped to match the component. The flat face of the regulator should be facing away from the Orangepip logo towards the edge of the board.

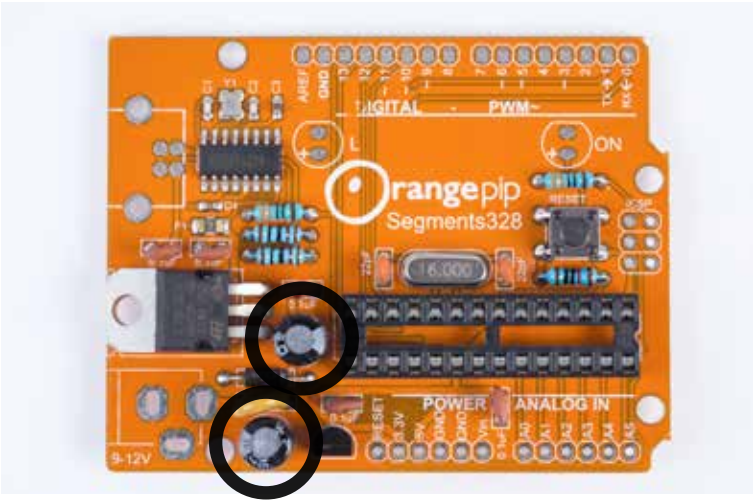


2.10

Locate the PTC resettable fuse.
This component is not polarity sensitive so place into the PCB and solder in place.

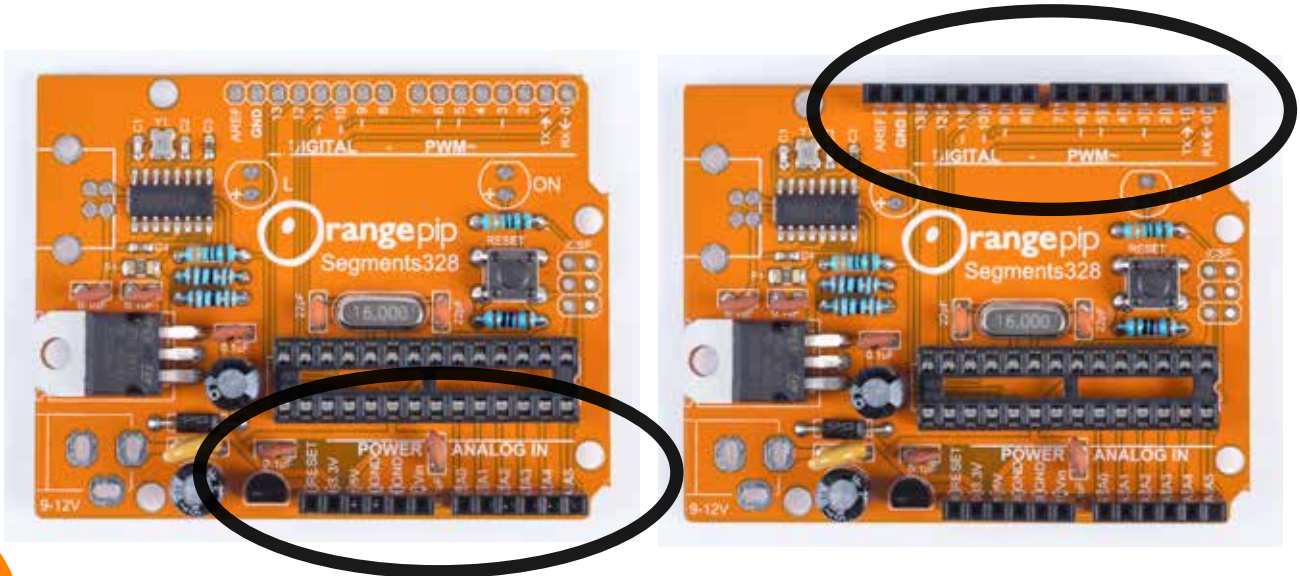
2.11

Locate the 47uF electrolytic capacitors. These are polarity sensitive, so you need to ensure they are mounted the correct way round. To assist with this the capacitors have two indicators to help identify the orientation. The first is that the positive (anode) leg is longer than the negative (cathode). The second is that the body is marked with a silver line with minus signs to indicate the negative (cathode). Match the longer leg with the + marking on the PCB.



3.0

Locate the I/O sockets. There are 2x 6 way and 2x 8 way. The 8 way sockets are to be placed in the 'DIGITAL' and 'PWM' slots shown at the top of the picture. The 6 way sockets are to be placed in the 'POWER' and 'ANALOG IN' slots shown at the bottom of the image.



3.1

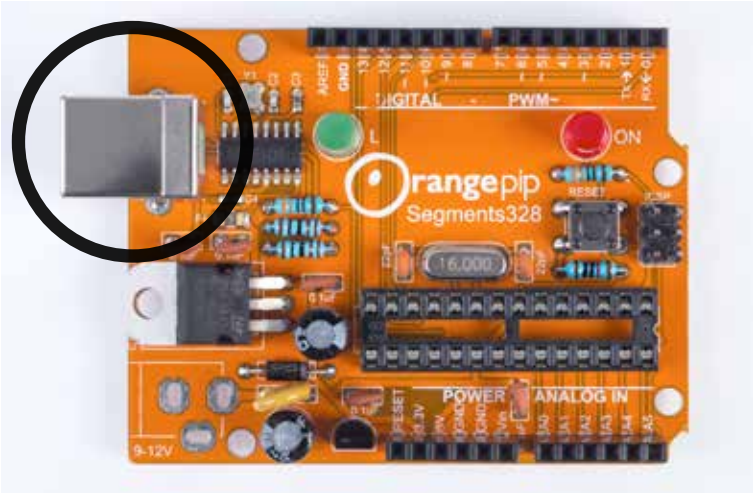
Locate the ICSP 6 pin header. This is not polarity sensitive, but you will notice that the legs either side of the black plastic are different lengths. The shorter ends should be on the underside of the boards with the longer ends accessible from the top.

3.2

Locate the green and red LEDs. The LEDs are polarity sensitive and need to be inserted into the PCB the correct way round. We can use a similar method to the electrolytic capacitors to determine the correct orientation. The positive (anode) is the longer leg and the negative (cathode) is the shorter leg. Match the longer leg with the + mark on the PCB. It doesn't matter which LED you solder in which location.

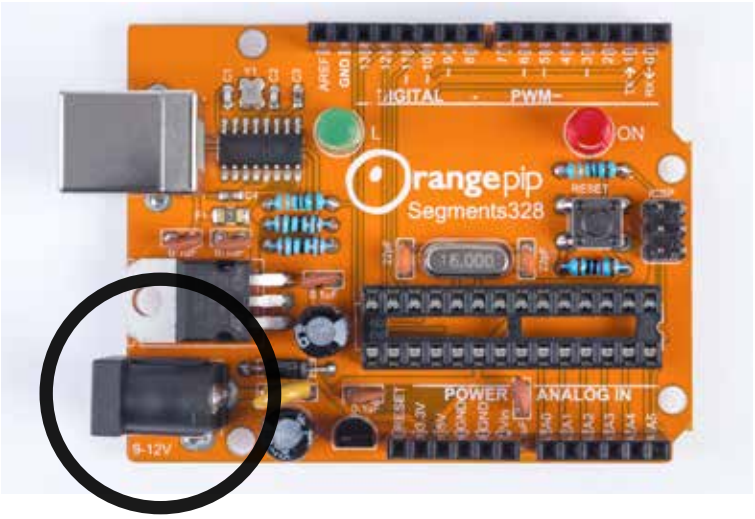
3.3

Locate the USB-B connector. The two larger pins act as anchor points and ensure that the connector can stand up to repeated insertion and removal of the USB cable. They are also useful to hold the connector in place during soldering.



3.4

Locate the 2.1mm DC socket. Insert it into the PCB and solder in place.



3.5

The last job is to insert the ATmega328 microcontroller into the chip carrier socket you soldered earlier. The microcontroller is polarity sensitive. Find the end that has the notch and line this up with the notch on the microcontroller.

Place the microcontroller gently into the socket. Ensure that the pins are lined up with those of the socket and apply firm but even pressure to seat the microcontroller in the socket. Not applying even pressure can cause the pins to bend out of shape and damage the microcontroller.

4.0

Your board is now complete, and you can begin prototyping. Click here for our guide on installing the software and uploading your first program.