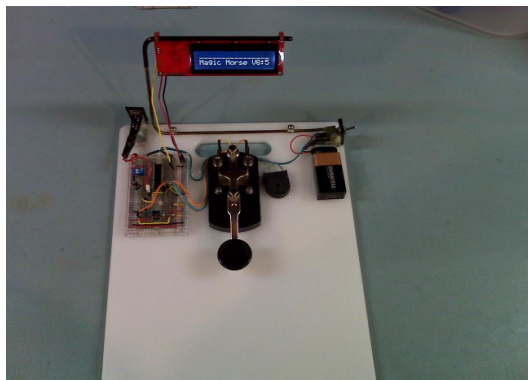


Magic Morse Code Trainer

PART NO. 2169467



Many educators will advise that the best way to learn a language is to listen first to a phrase, then associate the meaning, and then pronounce the phrase or response. Unfortunately, when an individual is along trying to learn Morse Code, the try and true is difficult to implement. The frustration comes when the student attempts to key a word or phrase and has no direct feedback if the keying was correct. Magic Morse eliminates this barrier to learning by providing immediate feedback and in a manner that would be the way a trained listener would recognize the keying. By providing positive and immediate feedback, the student learns to key correctly and can spend more effort on accuracy and rhythm. Magic Morse is implemented in a single microcontroller chip (PICAXE 20X2 - PIC18F14K22 uC) a wide band tone demodulator and a Morse character decoder. It can even be used with a PC provided, it provides standard PC serial port, COM1 or COM2.

It can be also be driven from the PC sound card provided that the audio level is sufficient, which allows for the decoding of ARRL test files which are hosted as MP3 and WAV formats on the Internet. This provides a kind of safety-net while learning such that if the student is listening and writing the code on paper, he/she can refer to the LCD display (optional) or the PC terminal display to make a positive association of the correct character with the Morse Code sounds.

Time Required: 4 - 6 hours depending on experience

Experience Level: Intermediate

Required tools and parts:

PICAXE 20x2 microcontroller
PIC programmer
Telegraph Key
Soldering iron and solder
Diagonal wire cutters
Wire strippers
9V battery
Wooden/plastic cutting board or plywood base (optional)
Wood screws, machine screws, coat hanger or stiff copper wire (optional)
Needle nose pliers

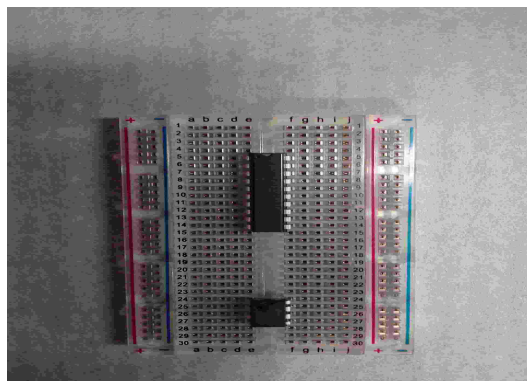
Bill of Materials:

Qty	Jameco SKU	Component Name
1	20601	BREADBOARD,3.25"x2.125",400pnt
1	51182	IC,78L05,+5V,TO-92
1	93761	Capacitor, RADIAL,100uF,25V,20%,85C
2	15229	Capacitor, CERM,DISC,.01uF,50V,20%
1	34673	LED,RED/GRN,WhtDIF,T1-3/4,697
4	2157175	RESISTOR,CARBON FILM,100 OHM,1/4 WATT,5%
1	27423	IC,LM555CN, DIP-8
4	691104	RESISTOR,CARBON FILM,10K OHM,1/4 WATT,5%
1	38360	Transistor, 2N3904, TO-92
1	544913	Capacitor, CONF,X7R,.047UF,100V,10%
1	11280	Battery Clip, for 9V

1	2098523	BUZZER,PIEZO,5Vp-p,80dB
1	2154927	DISPLAY,LCD,16X2,SERIAL,5V,I2C
1	228494	STEREO-3.5mm MALE PLUG (optional)
2	661183	Resistor, CF,47 OHM,1/2 WATT,5% (optional)
1	2152921	WIRE,22AWG,SOLID,YELLOW,25'

Step 1 - Insert ICs into Solderless Breadboard

- 1) Orient the prototype solderless board so that the row numbers are vertical, 1 - 30, across the left and the right sides. This will also orient the power connections so that they too run vertical.
- 2) In the Magic Morse packet with the parts are several documents. Remove the one labeled "Schematic" and place it where you can reference the document throughout the remaining steps.
- 3) Take reasonable precautions regarding static electricity prevention. There is an advisement printed and inserted with these assembly instructions. Take a moment and read this advisory.
- 4) There are 2 ICs, on antistatic foam. One has 20 pins total and the other has 8 pins total. The 20-pin IC is the microcontroller that is programmed with the Magic Morse software. The 8-pin IC is the 555-timer which will be used as the tone oscillator for our project.
- 5) The 555-timer, 8-pin IC, is to be inserted into the solderless breadboard such that it straddles the middle indentation (runs vertically). Before inserting the pins into the breadboard, take a moment and orient the chip so that PIN #1 is at the top-left. PIN #1 is identified by a small dimple molded in the top of the plastic case. When the IC is oriented correctly, the molded "U" shaped notch represents the top side. Refer to the component pictures, if necessary, to ensure the correct orientation and then press the IC into the holes numbered 24-27. You may find it easier to gently insert the left or right side first and then use your thumb to press the metal "legs" so that they are easy to insert into the breadboard. Another "trick" is to use the edge of a table to press the left side and then the right side of the chip until the metal legs are completely vertical and easy to insert. When all 8 pins are inserted such that the IC straddles the trough, press the top of the IC with your thumb and make certain that it is firmly seated against the breadboard.
- 6) Applying the same techniques you used in #5 above, insert the 20-pin microcontroller in the breadboard. First, orientate the chip so that PIN #1 is top-left. Remember that PIN #1 is marked by a dimple in the plastic case. When properly oriented, the "U" notch is also at the top center. As there are more pins per side, proceed carefully starting with PIN #1 and PIN #20 in breadboard connection #5 (straddling the center groove of the breadboard.) IC PIN #10 and PIN #11 should be in breadboard hole #14.



Step 2 - Add the LED and current limiting resistors

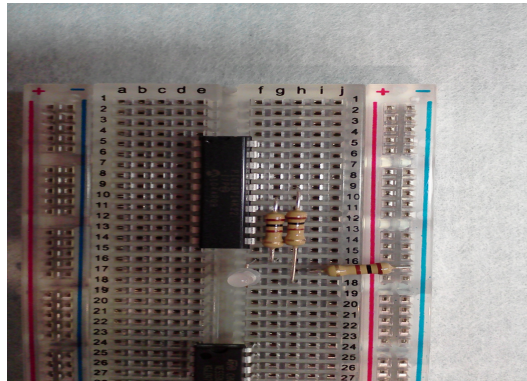
The LED used in this project is a 3-wire device with a clear or translucent plastic housing. With the plastic facing up, the three connection leads are pointed down and the center lead is the cathode. The two outside leads are Red and Green with the Green LED anode connection being the "shortest" of the three wires. The Green LED should go to the PIC chip pin#16 and the Red LED should go to PIC chip pin#15. BOTH first connecting with a 100 Ohm resistor and then the resistor connecting with the representative PIC data pin. For example, the Green (shortest lead) of the LED connects to a 100 Ohm resistor and the other end of the resistor connects to PIC Pin#15.

- 1) Insert the LED in the breadboard, right side of center, column "F" and rows 16, 17, and 18.
- 2) Lay one of the 100 Ohm resistors (Brown / Black / Brown) along the breadboard and even the length of the resistor leads from LED

row 16 to PIC row 10. Using your wire cutters, cut both leads of the resistor with a minimum of 1/4 inch extra on each side of the resistor to allow the ends to be bent and inserted into the breadboard. See graphic before cutting the leads; double-check and cut the resistors leads. Using a long nose pair of pliers, bend the resistor leads on both sides at right angles and insert into vertical row "G" and horizontal holes 10 and 16.

2) Repeat the above for a second resistor, but center this resistor between horizontal row 18 (bottom of the LED) and PIC row 9. Cut the resistor, bend, and insert it in column "H" row 9 and column "H" row 18.

3) The third 100 Ohm current limiting resistor will be placed horizontally between the center LED connector (cathode) and circuit Ground, represented by the rightmost vertical blue bar with the "-" symbol at the top/bottom of the breadboard. Measure, cut, and insert in column "I" row 17 to Ground.



Step 3 - Add the input "clipper" transistor

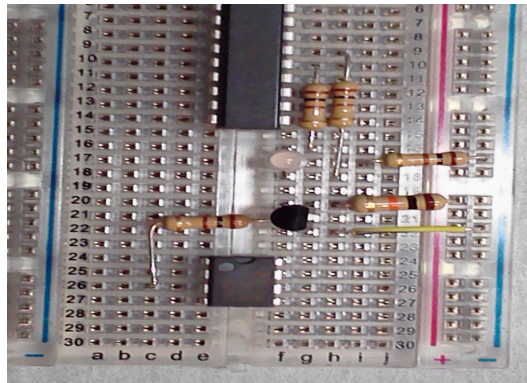
1) Insert the transistor, flat side facing to the right of the breadboard, into column "G" and rows 20 (collector), 21 (base), and 22 (emitter.)

2) Locate one of the 10K resistors (Brown, Black, Orange) and bend both leads close to the body, cut no shorter than 1/4" and insert one lead into column "H" row 20 and the other lead into the red vertical positive power bus area, labeled "+".

3) Cut a length of hook-up wire to be approximately 1" in length and remove 1/4 inch of insulation from each end of the wire. The wire ends should be bent at right angles and inserted into row #22 column "H" Row #22 (the transistor Emitter) and the Blue "-" ground column.

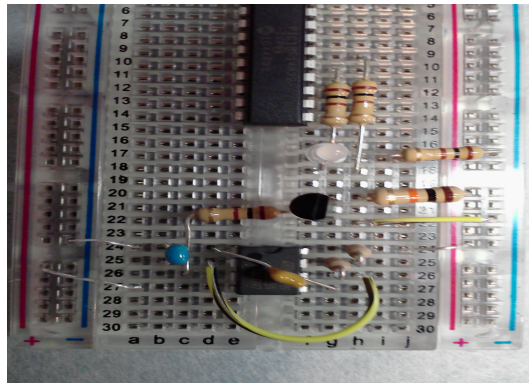
4) Locate a 100 Ohm resistor (Brown, Black, Brown) and trim one end of the lead to 1/2" and bend the endmost 1/4" to fit into Column "F" Row #21 (NOTE: the resistor will connect to the Base of the transistor but will be positioned to the left of the center divide in the protoboard.) DO NOT insert the resistor yet.

5) Bend the left-most lead of the resistor downward and cut the lead 1/4" beyond row #26. Now bend the resistor lead just cut at right angles and insert it into left-row #26 column "C". Insert the transistor end of the resistor into column "F" row #21 (the transistor base - center connection.)



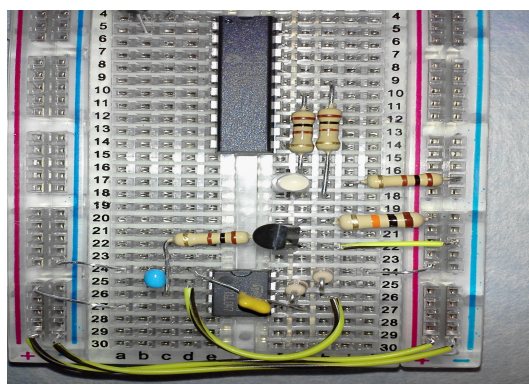
Step 4 - Complete the Code Practice Oscillator

- 1) Select a 10K resistor (Brown, Black, Orange) and clip one end to be 1/4" in length. Bend the top wire over against the resistor and clip the second lead to be even with the first lead; that is, the length beyond the transistor end is 1/4".
- 2) Repeat #1 for the second 10K resistor.
- 3) Insert one resistor vertically into column "H" row #24 and the second lead into column "H" row #25.
- 4) Insert the second resistor vertically into column "G" row #25 and the second lead into column "G" row #26.
- 5) Identify the 47nF ceramic capacitor - this capacitor may be marked as "473K" or .047uF or 47000pF. Clip both leads so that they are 1/4" long. Insert the capacitor into column "B" rows #24 and #25 (across pins #1 and #2 of the 555.)
- 6) Verify that the capacitor in step #5 does not touch the resistor lead of the 100 Ohm resistor that connects the 555 pin #3 to the base of the transistor. Bend the resistor lead if necessary to ensure that no accidental short occurs.
- 7) Identify the 10nF capacitor - may be marked as 10000pF or .01uF. The leads on this capacitor must be longer to reach across the 555 IC from pin #1 to pin #5. Therefore, clip the leads of this capacitor to 1/2" each. The leads will need to be bent outward from the bottom to form an arc of approximately 90 degrees between the two wires. Then, use long nose pliers to adjust the angle of the two leads to be 1/4" straight to allow insertion into the pegboard. Insert this capacitor into column "E" row #24 and column "G" row #27.
- 8) Strip a bare 5/8" section of jumper wire. Connect this bare wire between column "A" and row #24 and the blue left-hand Ground "-" bus.
- 9) Strip a bare 1" section of jumper wire. Connect this bare wire between column "A" and row #27 and the red left-hand Positive "+" bus.
- 10) Strip a bare 5/8" section of jumper wire. Connect this bare wire between column "J" and row #24 and the right-hand Positive "+" bus.
- 11) Strip 1/4" from each end of a 1 1/2 inch length of insulated jumper wire. Connect one end to column "D" row #25 (555 pin #2). Route the wire around the end of the IC and connect the other end to column "H" row #26 (555 pin #6).
- 12) Carefully inspect your work; especially step #7 to ensure that the capacitor leads connection the 555 IC pin #1 and pin #5 do not touch adjacent pins #2 and #6. Use the needle nose pliers to adjust the bend away from the IC as necessary.



Step 5 - Add the Voltage Regulator

- 1) Identify the 78L05 IC; the part is a 3-wire device and is labeled on the "flat" side. It looks identical to the 2N3904 transistor - so be careful that the right parts are used in the right location.
- 2) Insert the 78L05 on the left-side of the breadboard with the flat side facing to the right. Position the part and press the 3 wires into Column "B" rows #3, #4, and #5. Note that Row #5 is also pin #1 of the PIC microcontroller IC.
- 3) Identify the 100uF filter capacitor. Notice that the pins exit the bottom of the device and that one side is marked with "---" to signify the negative side of the part. It is critical that the part be installed correctly. If the part is marked with "+++", then that side of the part is positive... markings differ by manufacturer, so pay particular attention. Insert the 100uF capacitor so that the "+++" (or unmarked side) is on row #3 (left-side of board adjacent to the 78L05) and the "---" (or unmarked side) is on row #4. The capacitor bridges the 78L05 pins #1 and #2. Do not worry about which columns ("C" or "D") used, just ensure that the device is inserted firmly and that the polarity markings are correct - pin#2 of the 78L05 is the ground connection also referred to as the negative ("-.") connection.
- 4) Using a 1/2" bare jumper wire, connect column "A" row #4 to the left-side Ground bus "-".
- 5) Connect the two positive ("+") solderless breadboard buses. Run a 2 1/2" (1/4" stripped ends) wire from the last "+" row #30 on the left of the board to the last "+" row #30 on the right hand side of the board.
- 6) Connect the negative bus sides of the breadboard in a similar fashion to step #5 using a 2 1/2" piece of insulated wire (1/4" stripped from each end.) The wire should connect the lower "- negative bus on the left with the lower "- negative bus on the right.
- 7) Carefully inspect the two wires to ensure that the stripped ends of the wire are firmly and completely inside the press-in connection. If bare wire is exposed, remove the wire, trim a small amount and reinsert until there is no chance of a short between the two power rails.

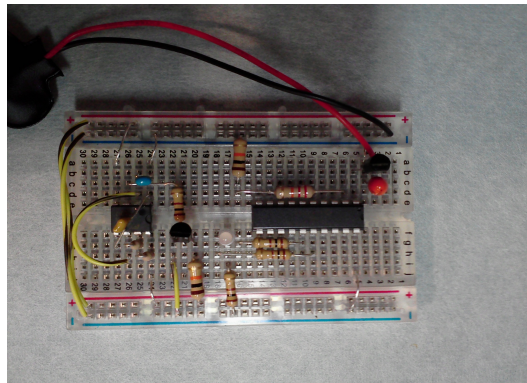


Step 6 - Complete the PIC Microcontroller wiring

- 1) Using a 5/8" section of bare jumper wire, connect breadboard location column "J" row #5 to the right-hand ground "-" bus.
- 2) Identify the 22K resistor (Red, Red, Orange) Clip both leads to be 1/2" long and bend 1/4" of each lead to insert into the breadboard. One lead in column "D" row #6 and the other lead in column "D" row #16
- 3) Identify a 10K resistor (Brown, Black, Orange) and bend both leads at right-angle to the resistor body. Clip the leads to 1/4" each. Insert one end of the resistor into column "D" row #16 and insert the other end into the left-hand Ground bus "-".
- 4) Identify the 9V battery connector. The red wire is "+" positive and the black wire is "-" negative. DANGER - APPLYING 9V TO THE PIC MICROCONTROLLER IC WILL DESTROY THE DEVICE. The battery voltage must be regulated by the 78L05 to 5V. Remove 1/4" of insulation from the ends of both wires. Insert the Red "+" lead into column "A" row # 3. Insert the Black "-" lead into the left-hand negative bus "-".

Note: If the wires for the battery connector are not pre-tinned, you may wish to apply a light coat of solder. Otherwise, simply use your fingers to tightly wrap the wires together and press the wire into the solderless connectors being careful not to unravel the wire. If you wish, after testing is completed, you may use a drop of fingernail polish or airplane glue to fix the leads firmly into the breadboard. DO NOT USE any kind of "super glue" as all forms of this type of glue are corrosive to electronic connections.

At this time only a small amount of wiring remains. BEFORE continuing, a quick check of the progress so far will be done in the next step.



Step 7 - Sanity Testing

1) Perform a visual inspection. Review the previous steps, perhaps have a friend or family member read them out and verify that parts are in the correct location and that component leads are not touching each other. All connections should be made through the solderless breadboard. Pay particular attention around the 555 IC since two of the resistors are vertically mounted and one capacitor is above the IC.

2-A) Verify the 9V battery termination into the breadboard. Now, using a new 9V alkaline battery, plug the battery into the connector and observe the bi-color LED. Both the Red and the Green should be illuminated and should go out in approximately 10 seconds. If so, jump to #3 below.

2-B) IF the LED did not extinguish in 10 seconds, repeat #2 and wait an additional 5 seconds. If the LED is still ON after 15 seconds, there is a wiring issue with the microcontroller. Review the LED resistors (three) and the 78L05 power regulator IC. Verify that the 10K and the 22K resistor between Ground and uC pin #2 are correct.

Note that at this time, the 5V positive connection to the left-side 5V "+" bus is not active, so no power should be going to the 555 tone oscillator.

If you have a Volt-Ohm-Meter or a Digital Multimeter, DMM, then do the following:

Remove the 20-pin PIC from the breadboard by gently prying a small screwdriver underneath the chip in the channel that the IC straddles. With the IC removed, connect the 9V battery and put your fingers on the 9V regulator. If it gets warm or hot-to-the-touch immediately disconnect the battery as you may likely have the IC reversed. Verify the connections and that the IC has the flat side toward the "right" of the board.

Using a DMM or VOM, set the meter on a DC voltage scale greater than 9V. Measure the 9V battery and it should read in excess of 9V, perhaps as high as 10V. Connect the battery to the battery connector and measure the voltage from either of the negative rails

"-" to 78L05 output lead which should be in "B5", this voltage must be between 4.9V and 5.5V. Anything outside of this range is not acceptable. Remove the capacitor from Row #3 and Row #4 and repeat the measurement. If the voltage is now correct, the 100uF capacitor is likely damaged and will need replacing. If the voltage is still not in the 4.9V to 5.5V range, the 78L05 is damaged and will require replacing.

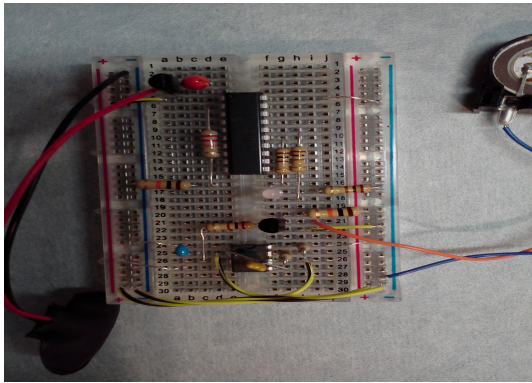
3) You are here because test #2 was successful. Place a small jumper wire between column "A" and row #5 (5V output) to the left-side positive rail "+".

4) Connect the 9V battery to the battery connector. Observe that the LED illuminates and goes off in under 10 seconds. If the LED does not come on, remove the wire in #3 and repeat #4. If the LED now illuminates, there is a wiring problem with the lower part of the board. Review all steps and correct.

5) With the jumper wire in #3 in place and verification that the LED came on and then went out, the microcontroller is working correctly. We will now verify that the oscillator is working.

6) Using the piezo sounder supplied in the kit, connect one of the wires to either negative / ground bus "-" and connect the second lead to column "H" row #21. A tone should be heard in the piezo sounder. If the tone is heard, Congratulations - testing has been successfully completed.

If a tone was not heard in #6, then the oscillator is not working. Review all of the steps completed previously in construction. If you have a multimeter, read the voltage and ensure that the 555IC pins #4 & #8 both show +5V in reference to 555IC pin #1. Review and correct any wiring issues and repeat all tests until step #6 is successful.



Step 8 - Configurations for use

If the default communications settings are adequate for your usage, then no changes are required. The RS232 protocol is 9600, N,8,1 and most common LCD displays are in the "Idle-High" signaling protocol.

1a) If you intend on using a PC as a terminal, then the PIC pin #4 must have a jumper to ground "-".

1b) If you intend on using an LCD with idle-Low (also in the case of a PC), then you must have a jumper from PIC pin #3 to ground

2) The RS232 output for serial LCD or PC is PIC pin #19 (row #6 on right-hand side of board) and Ground. The 78L05 can provide DC for most low-powered LCD displays: that is, those with no low-current backlights.

ALL COMMUNICATIONS SETTINGS ARE ONLY READ BY THE MICROCONTROLLER AT POWER-ON, SO CHANGES WILL REQUIRE A POWER CYCLE.

3) The Magic Morse PIC microcontroller can be configured to work with a Telegraph Key, the default usage or can be used with a PC (or other suitable audio device such as a portable CD player or high fidelity tape recorder) to decode recorded Morse Code signals for training purposes. For example, the ARRL has numerous MP3 files available at various speeds for downloading and these files can be played on your PC. If the PC has the ability to output the audio signal at 1.6 volts or greater from the earphone jack, then Magic Morse can decode from 5WPM to 15WPM.

4) For ALL usage, remove the end of the 100 Ohm resistor at location column "C" row 26 leaving the end at "F21" connected.

Reconnect the loose end to "C20".

5) For ALL usage, connect the output of the NPN-clipper transistor to the PIC input. Cut a 1 1/4" hookup wire and strip 1/4" from the ends. Connect one lead to column "F" row #20 (the transistor collector) and connect the other end to column "I" row #14 (PIC pin #21.)

6) For telegraph key operation, connect the key from "C26" to "A20"

7) For PC earphone output to kit input, you will need an appropriate cable. If stereo, the wire the Left and Right channels together through two 20 Ohm resistors. The junction of the resistors should go to "A20" and the shield should be grounded to the negative "-" connection. Your PC must produce 1.6V or greater for this circuit to work.

Refer to the user manual for other valuable information.

