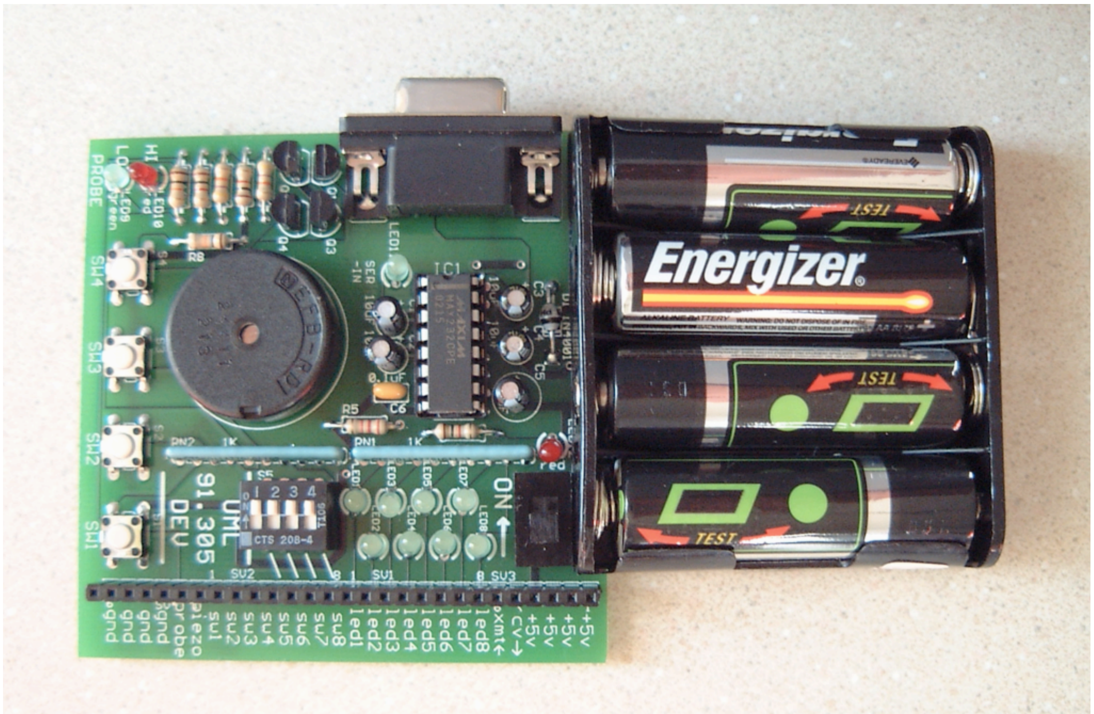
UML305DEV BOARD ASSEMBLY GUIDE



INTRODUCTION

This document provides an overview of soldering technique and then gives step-by-step instructions for assembling the UML305DEV board.

SOLDERING TECHNIQUE

There are three aspects to soldering properly: safety, care of the soldering iron, and the actual soldering method.

Safety

Soldering involves heating the metal solder to its liquid melting point. Clearly, molten solder is hot and dangerous. The most important safety rule is:

1. WEAR EYE PROTECTION

AT ALL TIMES WHEN SOLDERING

OR

WHEN SITTING NEXT TO PEOPLE WHO ARE SOLDERING !!!

You may either wear the lab safety glass which are provided in the lab, or your own eye glasses.

The next safety point is based on the fact that solder is made from a mixture of lead and tin. Lead is a known toxic substance.

2. WASH HANDS AFTER SOLDERING.

Prevent yourself from ingesting lead by bringing your hands to your mouth after you have soldered.

The last safety point is simple. The soldering iron is hot, and you can easily burn yourself with it.

3. BE CAREFUL WITH THE SOLDERING IRON. IT IS HOT.

Care of the Soldering Iron

The soldering iron tip must be kept clean and shiny in order to perform properly.

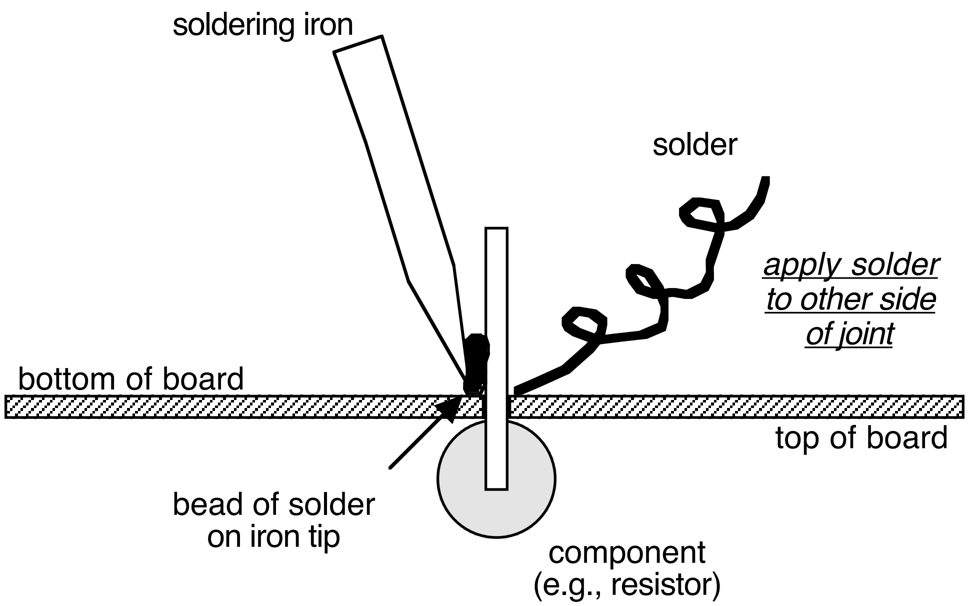
Every few minutes, or more frequently, clean the tip by wiping it across a damp sponge.

Note: Please make sure the sponge is wet! If you arrive at the soldering station and the sponge has dried out, please apply some water.

After wiping the tip, apply a small bead of solder to the end of the tip. This bead of solder will then make a good thermal connection to your solder joint. This is called “tinning the tip.”

If the soldering iron tip does not appear clean after wiping it across the damp sponge, you may insert the fully heated tip into the small dish of solid chemical cleaner. The cleaner will vaporize and clean the tip. Then, wipe the tip across the damp sponge, and apply the bead of solder.

Finally, **please turn off the soldering iron when you are done using it.** There’s nothing that will wear out the iron faster than having it left on for overnights unattended.



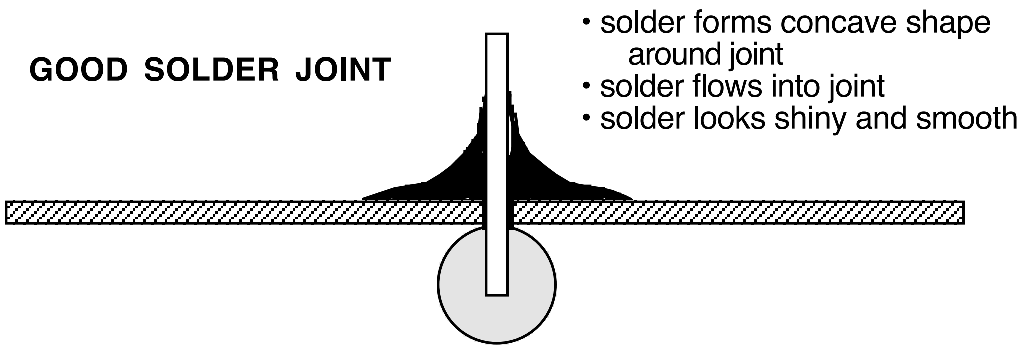
Soldering Method

The figure above shows the basic method for proper soldering. Here is the sequence of actions to perform:

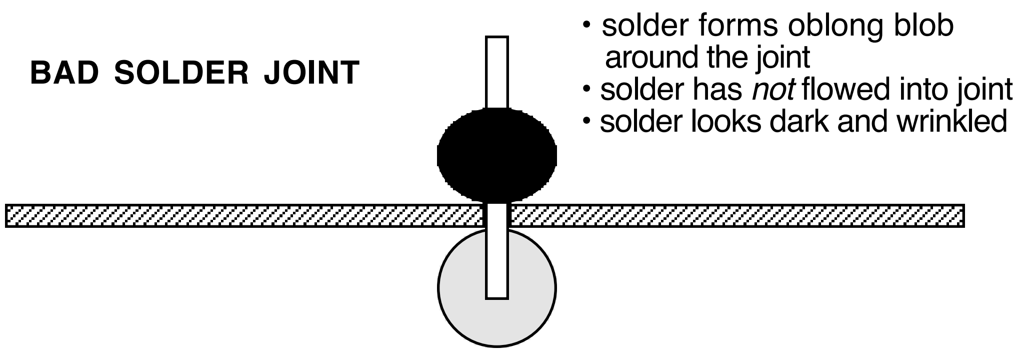
- Insert the component into the circuit board from the top side.
- Turn the board upside down and spread the component leads slightly so it does not fall out.
- Clean the soldering iron tip, and apply a small bead of new solder to the end of the tip ("tinning the tip").
- Press the bead of solder into the joint. Make sure the bead touches both the component lead and the pad on the circuit board.
- Apply solder to the joint from the other side of the joint. (Don't add solder to the iron and try to "paint it on"; that won't work.)
- Add solder till the joint fills up.
- Stop adding solder, but keep the iron pressed into the joint for an additional 2 – 3 seconds, allowing the solder to fully melt and flow.
- Remove the iron.
- After the joint cools, clip the component lead.

Good and Bad Solder Joints

The following two diagrams illustrate a proper solder joint and an incorrect one. It's worth studying these diagrams so that you can identify a good joint and fix a bad one!



With a proper solder joint, the solder forms a concave "volcano" shape around the component lead. It has flowed through the joint and may be visible from the top side of the board. It clings to the component lead and the board. The solder has a smooth and shiny surface.



With a bad solder joint (sometimes called a "cold solder joint"), the solder has balled up around the component lead. It has not flowed through the board, and it does not adhere to the board. The solder appears dark and wrinkled.

To fix a bad solder joint, try inserting the cleaned and tinned iron tip into the joint and see if you can get the solder to melt and flow. It may be necessary to remove the solder and start over.

BOARD ASSEMBLY

Following are step-by-step instructions for assembling the board. The basic strategy is to install the flattest components first and finish with the thicker ones.

You may wish to check off each step as it is completed.

Resistor Description

Let's start with the resistors. They're marked with four colored bands indicating their value. Three bands indicate the value according to the following table. (The fourth band is gold or silver, indicating the precision of the value [gold=5%, silver=10%]):

Color	**Value**
black	0
brown	1
red	2
orange	3
yellow	4
green	5
blue	6
violet	7
gray	8
white	9

Resistor Color Codes

Arrange the resistor so the gold or silver band is on the right. Reading from the left, the **first two bands are the mantissa** and the **third band is the exponent** of the value.

So, for example, if a resistor reads:

brown, black, red

This translates to:

1, 0, 10^2

or 10 x 10^2

or 1000 ohm, commonly abbreviated as 1Kohm, 1KΩ, or simply 1K.

yellow, violet, red is 4 7 x 10^2 or 4700 ohms or 4.7K.

See `http://www.dannyg.com/examples/res2/resistor.htm` for a live resistor color code demo.

Resistor Installation
Before starting installation, please compare the 1K (*brown, black, red*) and the 10K (*brown, black, orange*) so you can discern the difference between the red and orange exponent bands.

☐ R2, 1K, *brown, black, red*
☐ R4, 1K, *brown, black, red*
☐ R6, 1K, *brown, black, red*
☐ R5, 2.2K, *red, red, red*
☐ R7, 10K, *brown, black, orange*
☐ R8, 10K, *brown, black, orange*
☐ R1, 47K, *yellow, violet, orange*
☐ R3, 47K, *yellow, violet, orange*

Resistor Networks

These components package a bunch of resistors into one device. The ones we are using are polarized because they contain 9 resistors, with one common pin. This pin is marked with a dot.

Make sure to install the dotted pin in the PCB hole marked with a tiny "1".

☐ RN1, 1K, *labeled "ctsk…102"*
☐ RN2, 1K, *labeled "ctsk…102"*

LEDs

The green and red LEDs are installed next. These are polarized so don't install them backward.

The short lead of the LED is negative, and this goes in the square hole. Note that some LED positions have their square hole to the left, while others are on the right!

☐ LED1, HLMP1790, *green*
☐ LED2, HLMP1790, *green*
☐ LED3, HLMP1790, *green*
☐ LED4, HLMP1790, *green*
☐ LED5, HLMP1790, *green*
☐ LED6, HLMP1790, *green*
☐ LED7, HLMP1790, *green*
☐ LED8, HLMP1790, *green*
☐ LED9, HLMP1790, *green*
☐ LED11, HLMP1790, *green*

☐ LED10, HLMP1700, *red*
☐ LED12, HLMP1700, *red*

IC Socket

The 16-pin DIP socket goes in position for the MAX232 chip.

While the socket itself is not polarized, the MAX232 chip must be installled in the correct (of two choices) orientation. Therefore it is desireable to install the socket in the way that indicates the correct position for the chip it holds.

Look closely at the socket, and notice the little half-circle dimple along one of the short edges. Then find this same dimple in the markings on the circuit board (next to the text “IC1”).

When you install the socket, line up the dimples. Later, when you install the MAX232 chip, the dimple on the chip will line up with the dimple on the socket.

☐ DIP1, 16-pin socket, in position of IC1

Pushbutton Switches

Switches 1 through 4 are pushbutton switches. These are slightly oblong (rectangular) when inserted into the board. When you go to insert these into the board, they should fit easily. If you have to bend the pins excessively—you’re doing it wrong. Turn the switch 90 degrees and try that position.

☐ SW1, pushbutton switch
☐ SW2, pushbutton switch
☐ SW3, pushbutton switch
☐ SW4, pushbutton switch

Diode

Diode D1 limits the amount of current that can flow from the batteries and also drops 0.6v to bring the 4AA supply into a better range for use with 5v logic.

The diode is polarized. **Line up the band on the diode with the band on the component marking (i.e., toward the right).**

☐ D1, 1N4001 diode

Capacitors

There are two different varieties of capacitors used. The small cylindrical type are electrolytics. These are polarized and must not be installed backward. The flat type is a monolithic. This is non-polarized and may be installed in either rotation.

For the electrolyics, find the band on the device with a minus symbol. This is the negative (–) lead. The other lead is the positive (+) lead. **Insert the positive lead into the pin marked with a “+” symbol on the circuit board.**

☐ C5, 47μF, electrolytic capacitor

☐ C1, 10μF, electrolytic capacitor
☐ C2, 10μF, electrolytic capacitor
☐ C3, 10μF, electrolytic capacitor
☐ C4, 10μF, electrolytic capacitor
☐ C6, 0.1μF, monolithic capacitor

Transistors

There are two different types of transistors in the design—two NPN's and two PNP's. If you put the PNPs in the NPN positions (or vice-versa), the logic probe circuit won't work. **Note that the component numbering is NOT sequential, e.g., the NPNs are Q1 and Q4.**

The component markings on the circuit board indicate how the transistors should be aligned. Bend the middle pin backward to fit into the hole between the two outer pin positions.

☐ Q1, NPN transistor, 2N3904
☐ Q4, NPN transistor, 2N3904

☐ Q2, PNP transistor, 2N3906
☐ Q3, PNP transistor, 2N3906

Slide Switches

The two remaining switches are SW6, the power switch, and SW5, the 4-position DIP switch.

The power switch can only be installed one way.

☐ SW6, SPDT slide switch

The DIP switch will work either way, but let's all do it the same way so all of our boards come out the same. **Please line up the numeral "1" on the switch with the label "S5" on the circuit board.**

☐ SW5, 4–position DIP switch

Piezo Beeper

The large round beeper goes in the component position marked X1. Orientation does not matter.

☐ X1, piezo beeper

DB9 Jack

The DB9 jack mounts in the position labeled J1. Push it through carefully but firmly, making sure all nine pins go into their holes properly. Check that all nine pins are through their correct holes on the circuit board before soldering!

Add lots of solder to the two through-the-board mounting pegs. These provide a strong mechanical connection to hold the jack in place.

☐ J1, DB9 connector

Strip Header

The strip header provided has 36 pin positions. The circuit board requires 28 pins. The extra 8 must be cut off.

There are a couple of ways to do this.

1. Use a razor knife to score the header in the groove where you want to it to split. Repeatedly score both sides of the same groove, then firmly snap the part in two. Note: you must score the groove repeatedly in order to get it to snap properly. If you just try to snap it without scoring, or with not enough scoring, it may break somewhere else, or destroy the edge pin near the snap.

2. Using diagonal cutters, cut the strip in half. However, don't try to cut it at a groove, because one side of the split will get damaged, and you won't be able to control whether it's the side you care about (the 28-long piece) or the side you don't (the 8-long piece). So cut in the middle of a pin position, make sure that one side of the strip will still have 28 intact pins. You should end up with a 28-long piece and a 7-long piece.

After you produce the intact 28-pin piece, solder it into the positions marked SV1 through SV4. **Solder the two ends and make sure the socket is flat with the circuit board before soldering the whole thing!**

☐ SV1-4, 28-long strip header socket

Battery Pack

The battery pack mounts down through the top of the board like the other components. Its leads are inserted into the hole marked with a minus sign "–" and the adjacent square hole. Both are located along the top edge of the board, toward the left.

☐ BATT1, 4-AA cell holder

MAX232 Chip

The MAX232 chip can now be installed in its socket. **Make sure the notch on the chip is toward the left.**

☐ U1, MAX232CPE serial transceiver

Assembly is now complete! We're ready for testing.

Turn this page in as evidence of completed operation.

Name ____________________________________ ID#________________

I attest that all tests checked off were completed successfully.

Signature__________________________________ Date _______________

TESTING THE UML305DEV BOARD

The following sequence of steps will determine that the board is working properly.

Power

☐ Put 4 AA cells into the battery holder, and turn the board on (move the power switch to the left). The red power indicator LED (LED12) should light.

Logic Probe

Cut a length of wire about 6 inches long, and strip about 1/3 inch of insulations from both ends. Insert one end of the wire into the strip socket labeled “PROBE”.

☐ Touch the other end of the wire to any of the strip sockets labeled “+5V”. The red “HI” probe indicator should light.

☐ Touch the other end of the wire to any of the strip sockets labeled “GND”. The green “LO” probe indicator should light.

Status LEDs

Remove the wire from the PROBE position (make sure to pull it straight up, so it doesn’t snap off in the socket). Insert one end into one of the +5V positions. Now touch the other ends to each of the LED1 through LED8 positions. The corresponding green LEDs should light up.

☐ LED1
☐ LED2
☐ LED3
☐ LED4
☐ LED5
☐ LED6
☐ LED7
☐ LED8

Output Switches

Insert your test wire back into the PROBE position. Put the other end in the SW1 position. The green “LO” probe LED should light. Press SW1. Now the red “HI” probe LED should light.

Do this test for all eight switches.

☐ SW1—released:green; pressed:red
☐ SW2—released:green; pressed:red
☐ SW3—released:green; pressed:red
☐ SW4—released:green; pressed:red
☐ SW5—off:green, on:red
☐ SW6—off:green, on:red
☐ SW7—off:green, on:red
☐ SW8—off:green, on:red

Serial Line Converter Circuit

You will need to connect the dev board to a PC to do this test.

Connect the dev board to the PC using the DB9-male-to-DB9-female cable. The following instructions assume you are connected to the COM1 serial port. If you are connected to a different port, change accordingly.

Now, load and run the software “HyperTerminal”. This software is included with Microsoft Windows 95, 98, and NT. If running a different version of Windows, you may download a copy for free personal use from `http://www.hilgraeve.com/htpe/` .

(If you are using a Macintosh computer, you may need a USB-to-serial converter and/or adapter cables. Use the software ZTerm instead, available at `http://homepage.mac.com/dalverson/zterm/` .)

(If running Linux, use minicom, available at `http://packages.debian.org/stable/comm/minicom.html` .)

Whichever software package you are using, the procedure is the same:

- Start a connection to the serial port; e.g., “Direct to COM1”.
- Make the port settings 300 baud, 8 data bits, no parity, and no flow control.
- If there is a setting for “local echo,” make sure it is OFF.
- Open the connection.

At this point (when the connection is open) the green LED11 labeled SER-IN on the dev board should turn on. (It may have already turned on once you had plugged the dev board into the PC—that’s OK).

Now, start typing on the PC keyboard. You should see:

- With each keystroke you type, the green SER-IN LED should flash.

Next, on the dev board, *use a wire jumper to connect the socket pins labeled RCV and XMIT.* This will loop the PC’s serial output back to its input, so the PC should receive an echo of whatever it sends out the serial port. *Turn on your dev board.*

Now type on the PC keyboard, and:

- Each key you type should appear on the screen! The characters are actually being sent out the serial port, converted by the MAX232 chip, and then run back into the MAX232, converted back, and routed back to the PC.

☐ serial loopback test—chars echo when jumper attached and board turned on; do not echo when jumper removed.

Piezo Beeper

This test is easy. Connect a jumper wire from the PIEZO pin to one of the pushbutton switches (e.g., SW1). When you press the switch, you should hear a little “click” from the piezo. When you release the switch, it should click again. Note: it’s just a little click, not a strong tone!

☐ piezo beeper—clicks gently when switch is pressed and released.