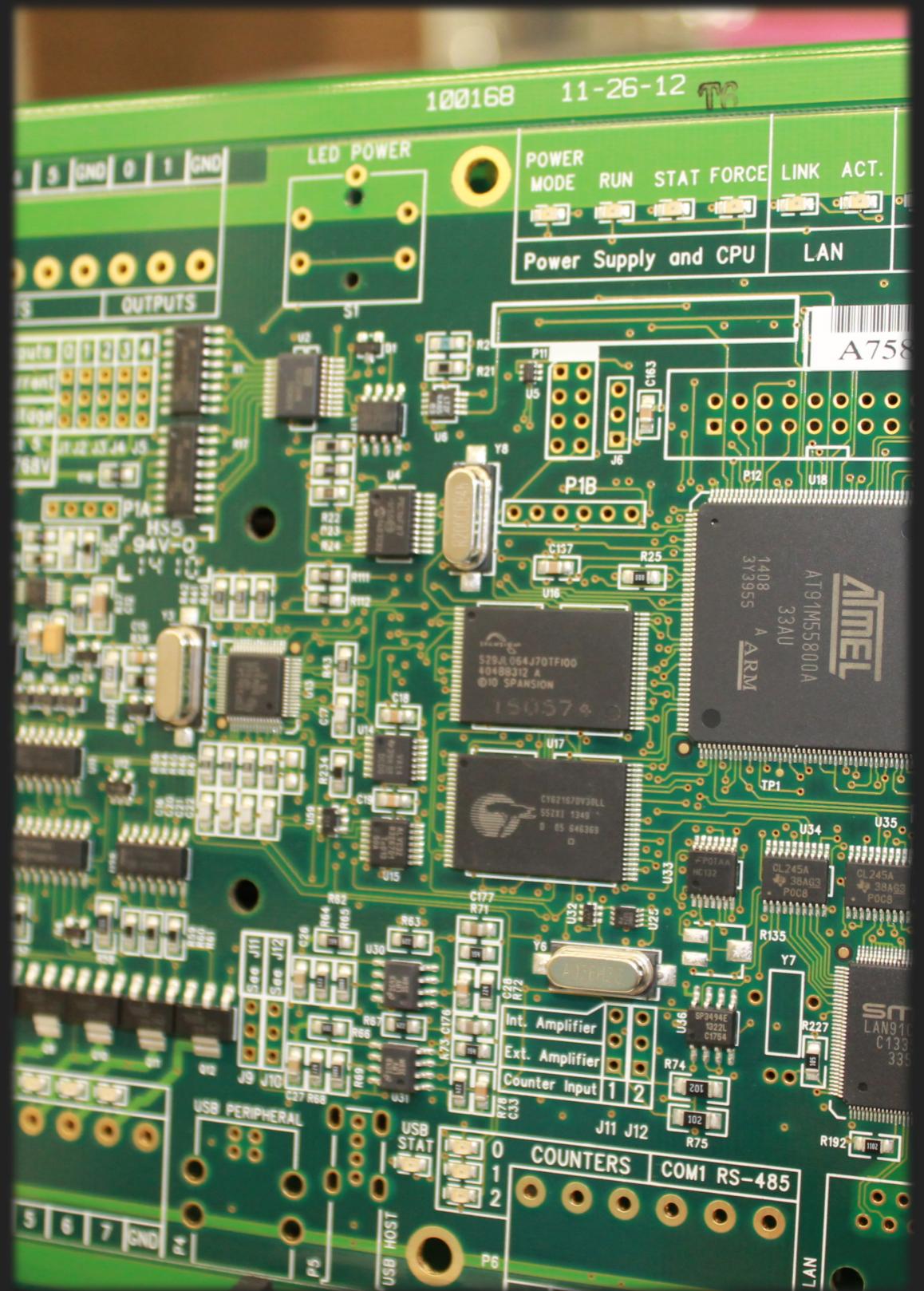


2.131 CRASH COURSE TO PCB DESIGN

DESIGNING PRINTED CIRCUIT BOARDS USING EAGLE CAD

PRINTED CIRCUIT BOARDS (PCB)

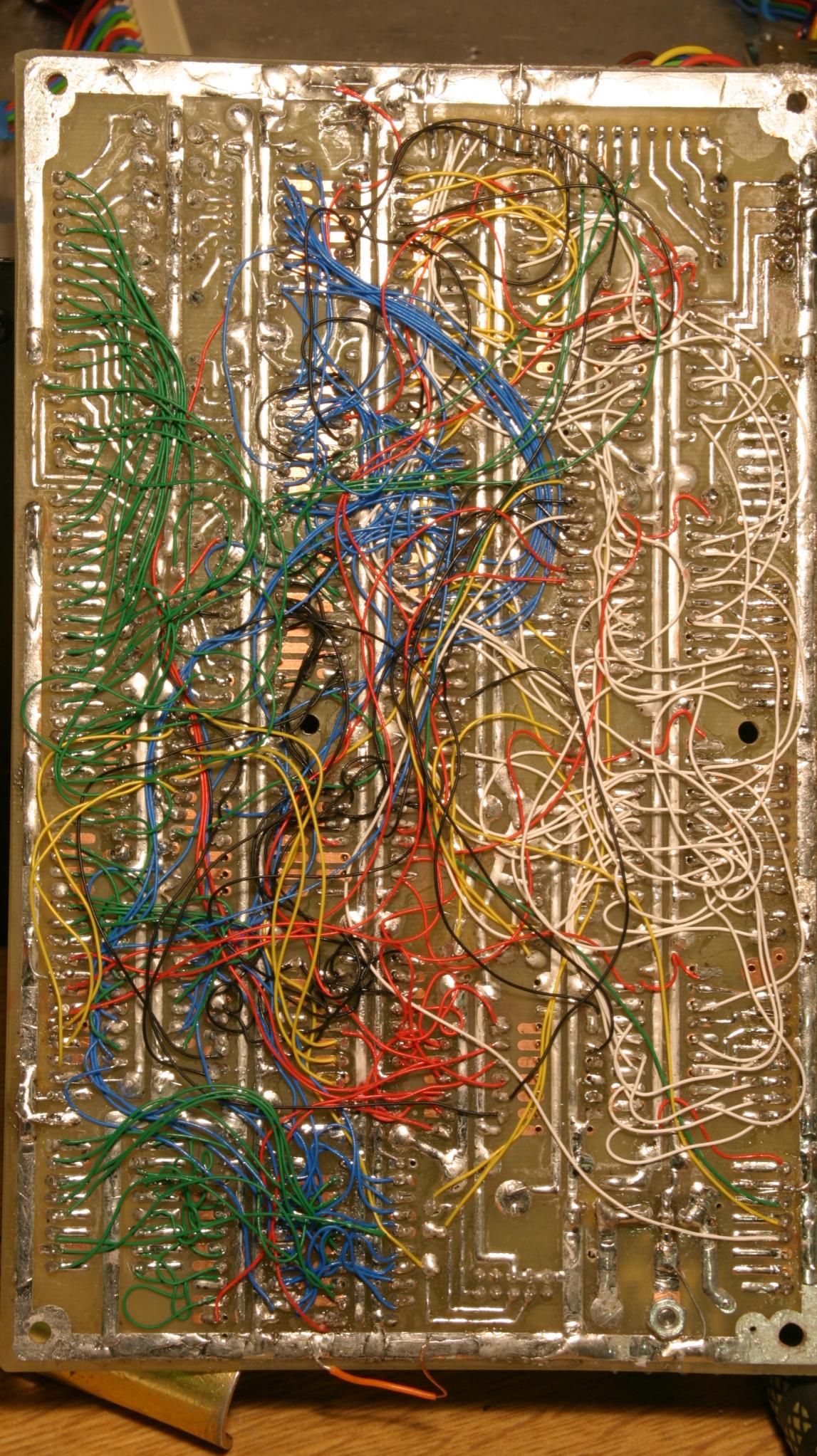
- ▶ Mechanically supports and electronically connects electrical/electronic components
- ▶ Connections between electronic components are usually copper tracks
- ▶ Components are usually soldered to board





BEFORE PCB

**POINT-TO-POINT
CONNECTION**



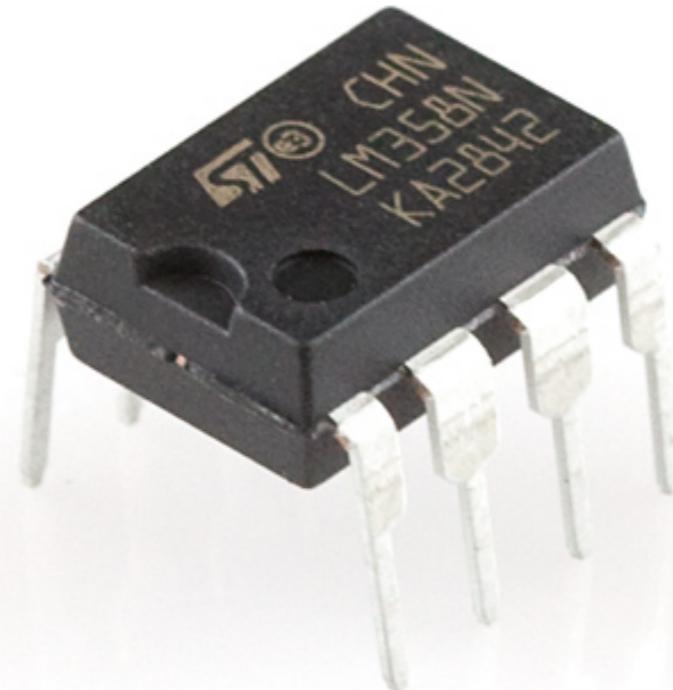
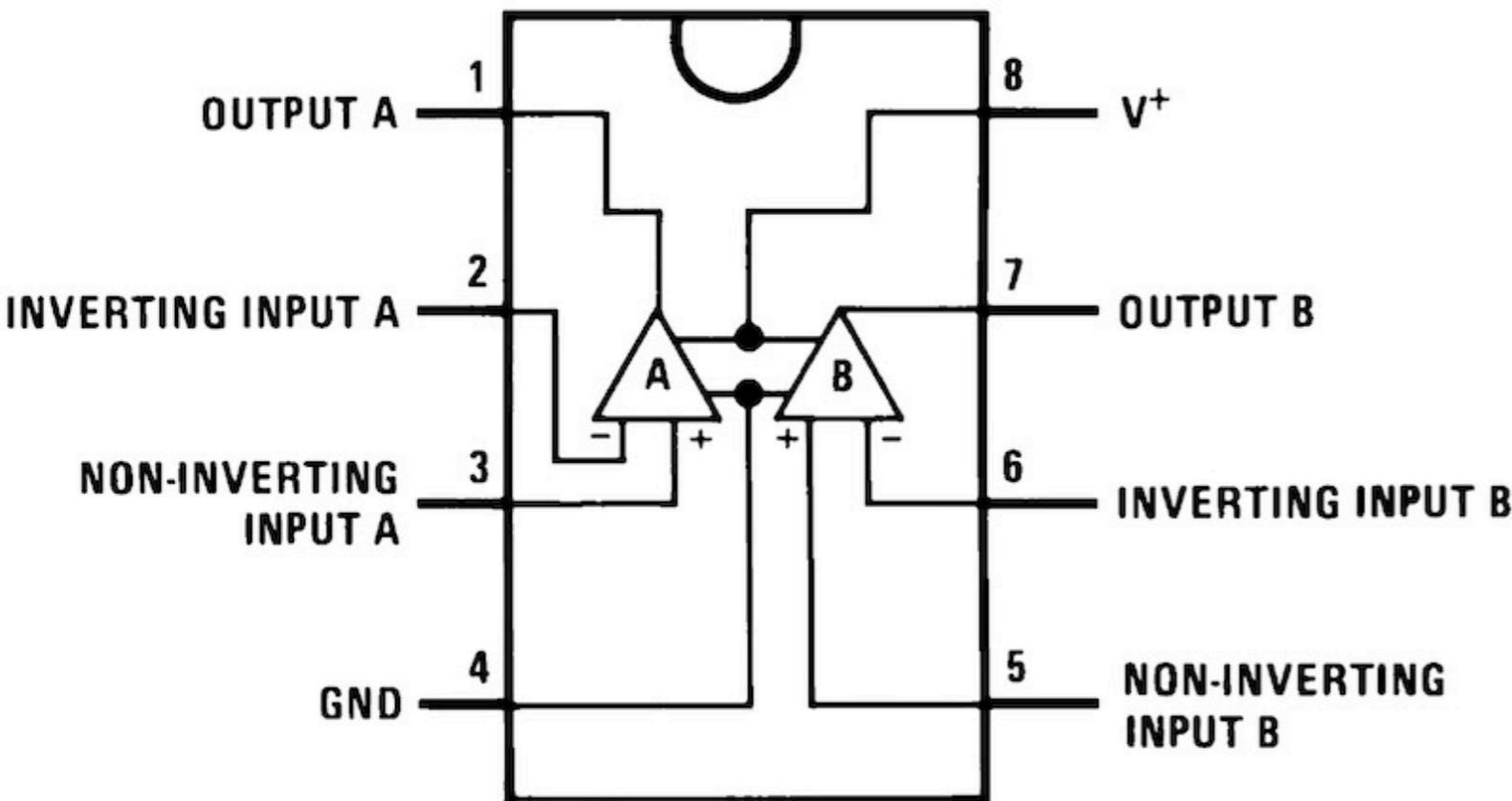
BEFORE PCB

WIRE WRAPPING

TYPES OF PACKAGING FOR ELECTRONIC COMPONENTS

- ▶ There are literally millions of electronic component types
- ▶ We're going to focus on electronic chips (not resistors, capacitors, switches etc)
- ▶ 2 types of packaging for electronic chips:
 - ▶ Through-hole device
 - ▶ Surface-mount device

Through-hole



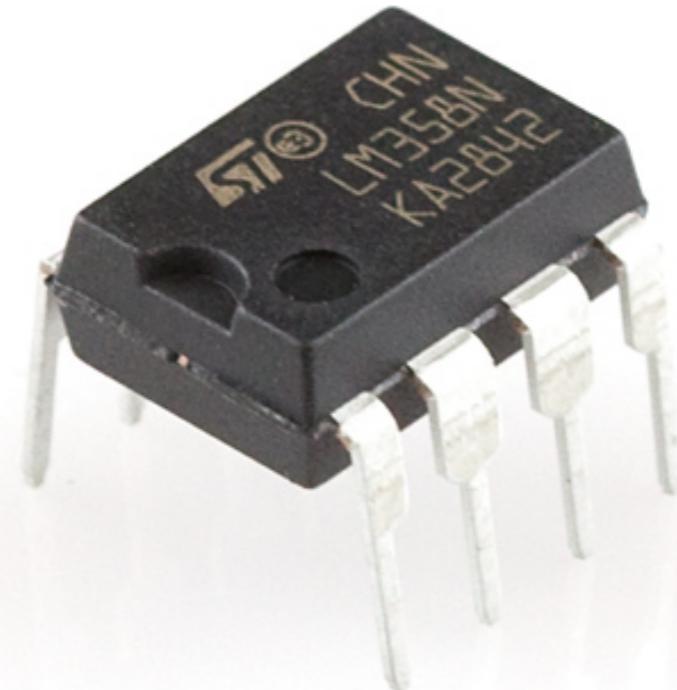
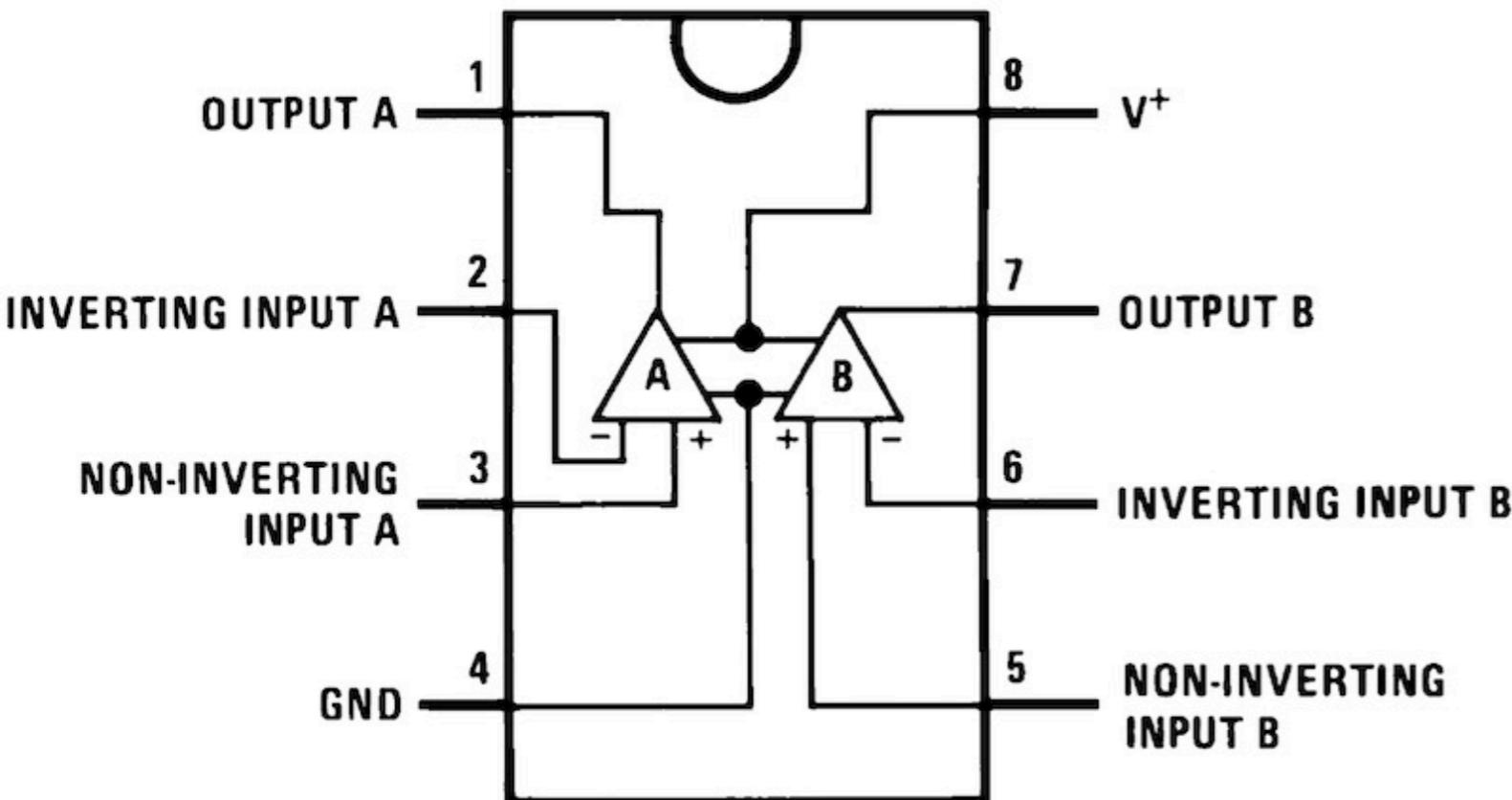
Advantages of SMD:

- Significantly smaller packaging
- Fewer holes need to be drilled
- Faster assembly
- Lower resistance and inductance and connection point
- Better performance under shake and vibration conditions

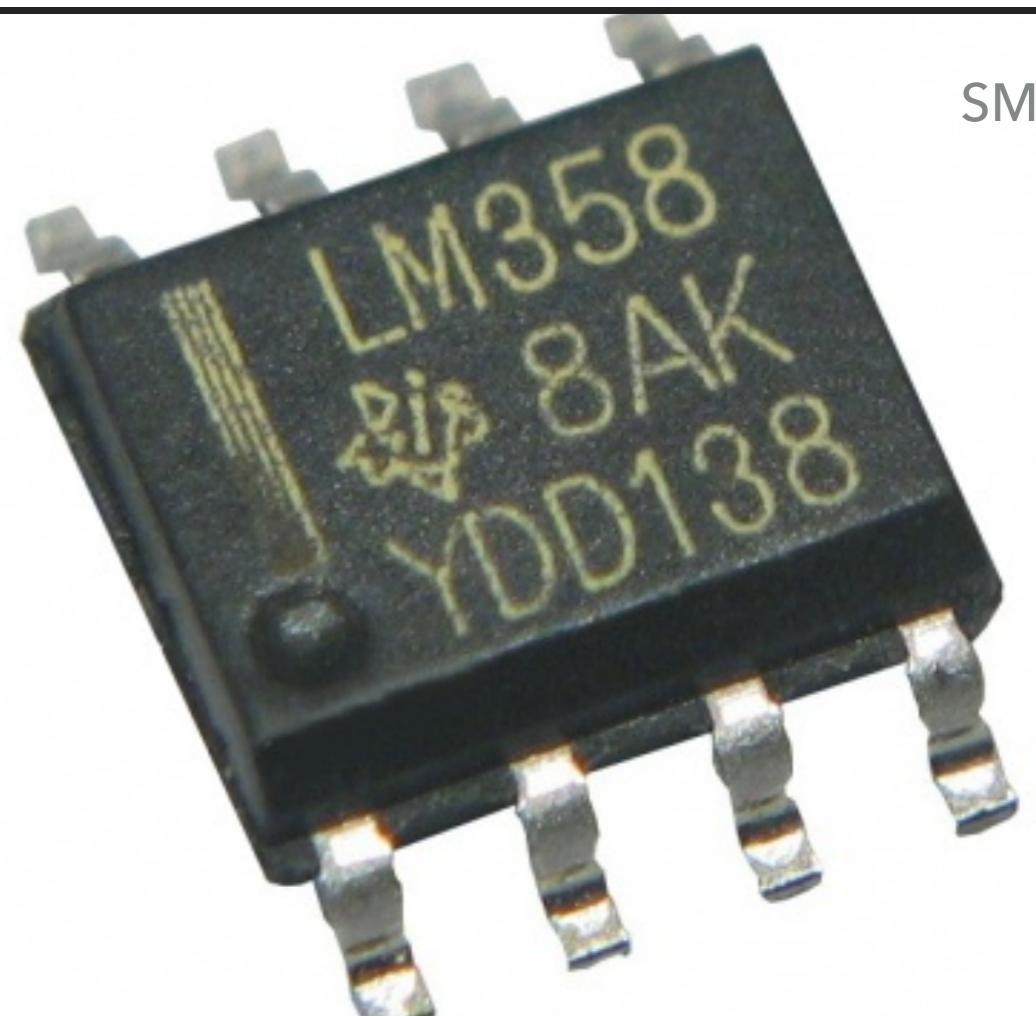
SMD



Through-hole



SMD



Disadvantages of SMD:

- ▶ Repairing/Troubleshoot/Debugging your board ain't easy
- ▶ SMD can't be used on breadboard
- ▶ SMD may not be suitable for high-power projects

EAGLE CAD

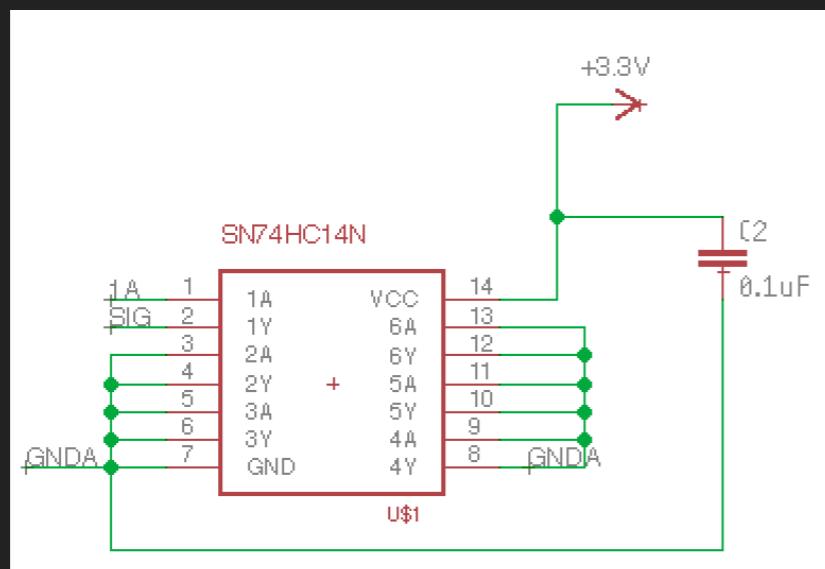
- ▶ Electronic design application (EDA)
- ▶ Features:
 - ▶ Schematic Capture: Visual representation of the physical connections of an electrical circuit
 - ▶ PCB Layout of Components
 - ▶ Autorouter: Automate the process of connecting components
 - ▶ Computer-aided Manufacturing (CAM): Generate files for fabrication



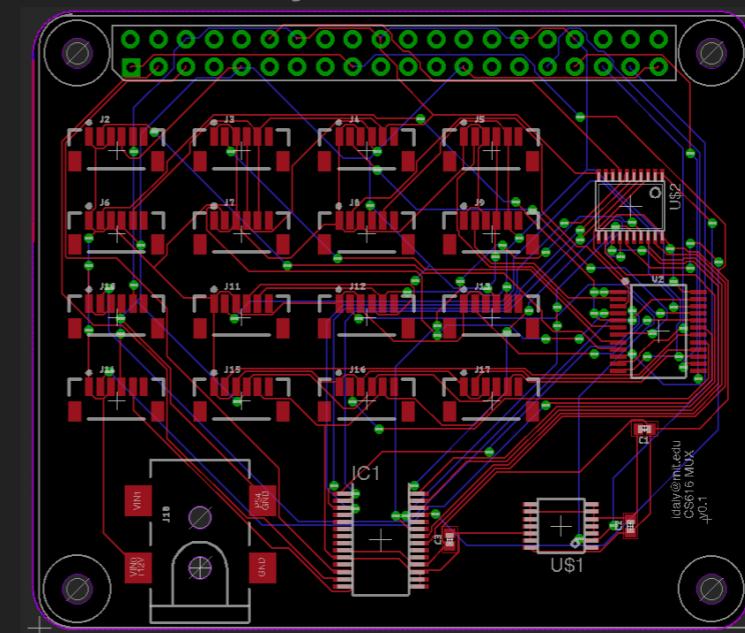
2.131 CRASH COURSE TO PCB DESIGN

EAGLE CAD

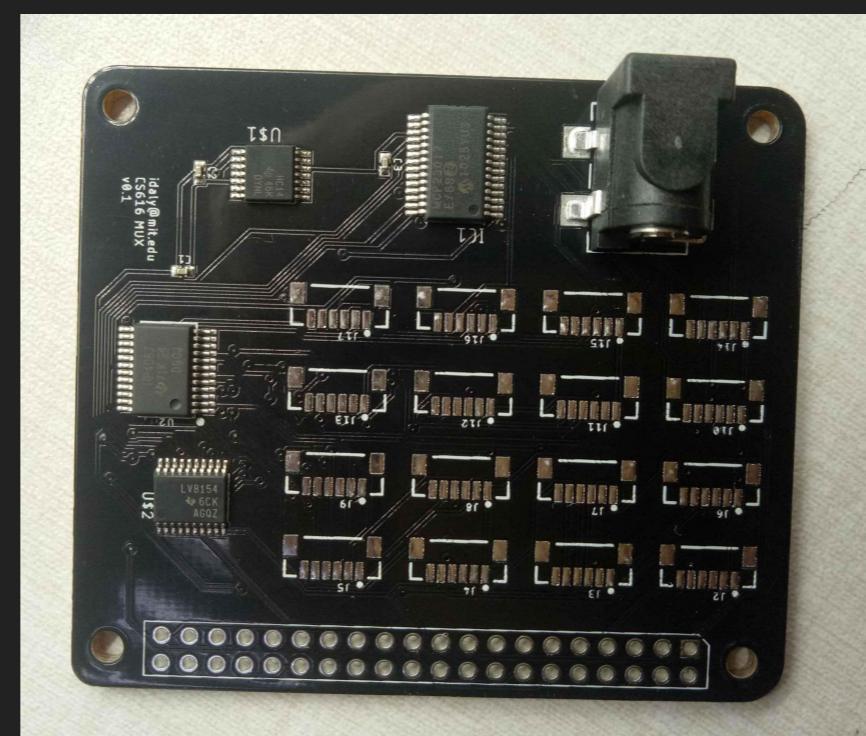
Schematic



PCB Layout



An actual PCB!



LOGISTICS

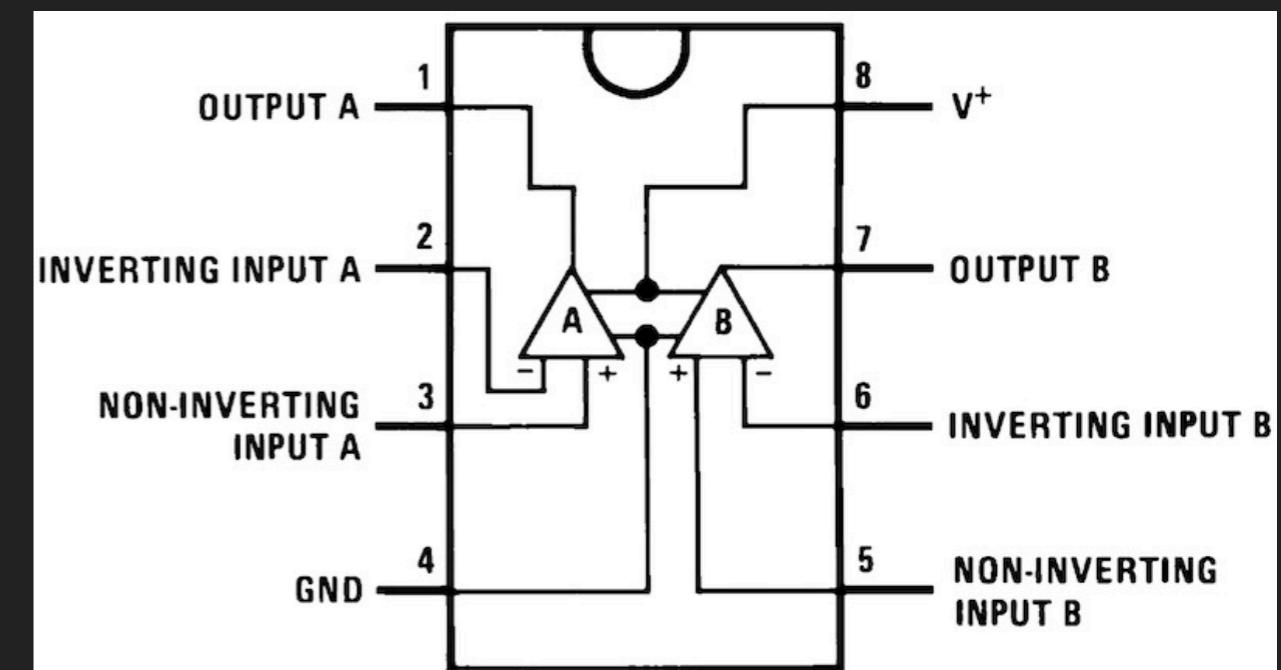
- ▶ EAGLE => <https://www.autodesk.com/products/eagle/overview>
- ▶ Sparkfun EAGLE Library => <https://github.com/sparkfun/SparkFun-Eagle-Libraries>
- ▶ OSH Park Design Rules (oshpark2layer.dru)=> <http://docs.oshpark.com/design-tools/eagle/design-rules-files/>
- ▶ Download OSH Park EAGLE 7.2 CAM (oshpark 2layer.cam) => <http://docs.oshpark.com/design-tools/eagle/generating-gerbers/>

LOGISTICS

- ▶ Design Rules (.dru) defines all the parameters that the board layout has to follow.
- ▶ Computer-Aided Manufacturing (.cam) defines the parameters to generate Gerber files
- ▶ Gerber: De facto standard used by printed circuit board (PCB) industry software to describe the printed circuit board images: copper layers, solder mask, legend, etc

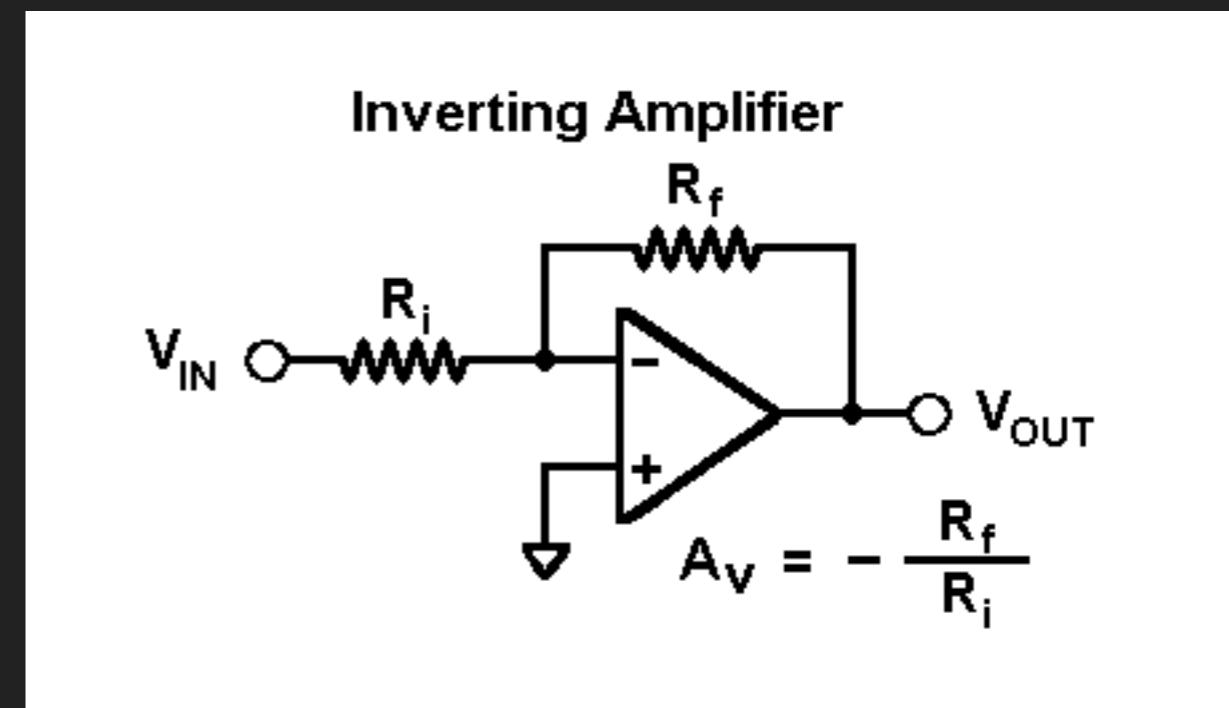
LM358 DUAL OP-AMP

- ▶ Input power supply, V+: 3-32 VDC
- ▶ Dual Op-amp
- ▶ Source up to 20mA



ASSIGNMENT: INVERTING OPERATIONAL AMPLIFIER

- ▶ THIS IS NOT AN ELECTRONICS CLASS!
- ▶ Scale a signal to any voltage range we wish by adjusting the gain accordingly
- ▶ Gain = - R_f/R_i
- ▶ Assumption 1 => We have a regulated 3.3V power supply
- ▶ Assumption => Input Voltage Signal V_{IN} is 0.01V

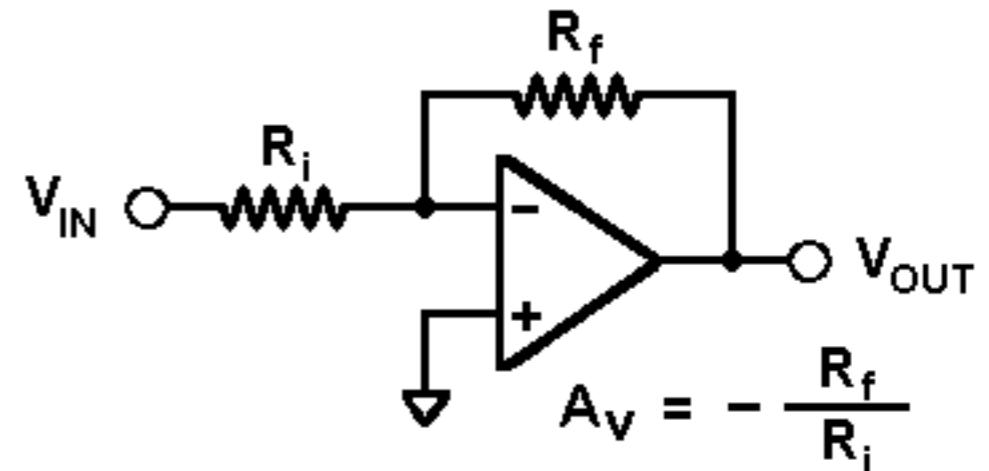


- ▶ Goal: We want gain to be -100
- ▶ $V_{OUT} = -1V$
- ▶ $R_i = 100 \text{ Ohm}, R_f = 10\text{kOhm}$

ASSIGNMENT: NON-INVERTING OPERATIONAL AMPLIFIER

- ▶ Scale a signal to any voltage range we wish by adjusting the gain accordingly
- ▶ Gain = $- R_f / R_i$
- ▶ Submit screenshot of schematic and board layout to idaly@mit.edu

Inverting Amplifier



RECOMMENDED READINGS:

- ▶ Sparkfun EAGLE Tutorial (Basic):
 - ▶ <https://learn.sparkfun.com/tutorials/using-eagle-schematic>
- ▶ SMD PCBs, Creating custom footprints etc...
- ▶ Instructables EAGLE Tutorial:
 - ▶ <http://www.instructables.com/id/how-to-use-eaglecad/>