Printed Circuit Board (PCB) Design with Kicad
Printed circuit boards are in essentially all consumer electronics.
Printed circuit boards are in essentially all consumer electronics.
Printed circuit boards are in essentially all consumer electronics.
Printed circuit boards are in essentially all consumer electronics.
The whole concept is that you solder most or all of your components securely to a single board and metal conductive traces on the PCB connect them all together rather than wires.
PCB Design

PCB design programs are called **EDAs** for Electronic Design Automation.

There are many. They are almost all either expensive, or free.

Most of the free ones, or free versions of good ones, are very not recommended.
PCB Design

As far as free EDAs go KiCad is by far the most popular and the best. We are going to do KiCad.

Eagle used to be the most popular but they limited the free version to the point you can only make really small boards with it.
PCB Design

Kicad

Kicad is free and open source. Free unlimited use for however complex your PCB designs get.

It’s a little less painful to use than Eagle. Combined with the fact its free means more people are switching to it.
PCB Design

Eagle

Eagle is a PCB design software that has had a free version that has been popular forever. It currently limits you to 12 square inch boards.

Also everyone has hated Eagle forever because it’s UI is terrible.
PCB Design

The PCB design process in Kicad is conceptually simple.

1) You design a schematic for what you eventually want to make into a PCB.
The PCB design process in Kicad is conceptually simple.

1) You design a schematic for what you eventually want to make into a PCB.
2) Create a **netlist**, which tells it what physical parts correspond to the symbols in your schematic.

The drawing of the physical part, which will be etched and its holes drilled into your actual PCB, is called that part’s **footprint**.
2) Create a **netlist**, which tells it what physical parts correspond to the symbols in your schematic.

The drawing of the physical part, which will be etched and its holes drilled into your actual PCB, is called that part’s **footprint**.
There is a huge library of footprints included in Kicad, and you can download many more online. But often you will have to make your own, or edit/alter existing ones, which is a whole separate topic for later.
Some programs like Eagle are more rigid about this - any one symbol can only connect to one footprint unless you make a bunch of copies. This can become a problem when you eventually want to use the same symbol with a different footprint, or vice versa.
3) Layout the PCB. It will put all your footprints into a single drawing. You have to drag them around to wherever you want them and draw the **traces** (connections) between them.

The first time you do this with anything more complex than an intro tutorial, it will take a while. Start small with a tutorial like the Digikey tutorial linked on the class page. Once you do all that once or twice it gets much easier.
4) Convert the finished PCB into **Gerber** files and a **drill file**.

**Gerbers** are the standardized files all EDAs use which you send to a manufacturer to have them make your PCBs for you. The drill file tells the PCB machines where to drill holes.
4) Convert the finished PCB into **Gerber** files and a **drill file**.

Kicad comes with a tool for viewing your Gerber and drill files to make sure they’re actually right.
5) Give the info to a PCB manufacturer ([www.oshpark.com](http://www.oshpark.com) above) for an instant quote and if you’re happy order them.