There U Glow's Glow Bug Badge Quickstart Guide!

This Art PCB was created by Darcy Neal as a fully open source workshop aid and gift to teach people about LEDs and soldering, as well as teach PCB (KiCad) design and manufacturing basics.

Step 1. Gather all the bits you need (from a workshop lead):
- **ONE** round Glow Bug PCB
- **ONE** small switch
- **ONE** purple lanyard
- **TWO** AAA single battery holders
- **TWO** AAA batteries
- **THREE** color changing LEDs, two bent long, one bent short

Common Mistakes: Know which side is the front, and which is the back. Try not to bridge any pins or copper traces to other pins, with solder, particularly with the small switch. Place your battery holders in the correct direction. Ask for help if needed!

Step 2. Solder! In no particular order you will solder:

- The **switch** onto the bottom of the badge. It has three small legs that poke through the PCB. Don't bridge them! (meaning, don't get solder stuck between them, making them touch)

- The **battery holders** onto the back, make sure they are placed IN THE CORRECT DIRECTION. Positive and negative are marked on the board, and on the battery holders.

- The two **long-bent LEDs** onto the back, through the holes. They should point down, towards the switch. (The badge in the photo below is slightly outdated, so pay attention to *your* badge and point them down!)

- Finally, flip the badge over and solder the single **short-bent LED** onto the front, pointing down towards the switch, as the glowing butt of the glow bug.

- Test your badge with the batteries, add a lanyard. Troubleshoot if needed. It's ok if it doesn't work at first! We can fix any mistakes, and we are here to help you.