

# How to Work with Provided Eagle PCB Files

## Download and Install Eagle

1. Visit the [Autodesk Eagle Download Page](#)
2. Create an Autodesk account if you don't already have one.
3. Download the installer compatible with your operating system (Windows, macOS, or Linux).
4. Install the application following the on-screen instructions.
5. Note: Eagle is now bundled with \*Fusion 360 Electronics\*, but you can still access Eagle independently.

## Open the Provided Files

1. Launch Eagle.
2. Navigate to File → Open → Project.
3. Locate the downloaded project folder containing the schematic and board files.
4. Open the .sch (schematic) file first to ensure netlists are correct.
5. Open the corresponding .brd (board) file to view the PCB layout.

## Add Any Missing Libraries

1. Identify missing libraries from Control Panel → Libraries → Used Libraries.
2. Download required .lbr files from trusted sources or request them from the project provider.
3. In Eagle:
  - a. Open Control Panel → Libraries.
  - b. Right-click and select Open Library Manager.
  - c. Under the Available tab, click Browse to add the .lbr files.
  - d. Enable the library by clicking the green checkmark.

## Review and Edit the PCB Layout

1. Verify that all components are placed properly.
2. Check routing and connectivity visually.
3. Use Tools → Ratsnest to ensure all nets are correctly updated.

## Run the Autorouter (if Necessary)

1. Select Tools → Autorouter.
2. Configure settings:
  - a. Layers (e.g., TOP and BOTTOM for two-layer boards).

- b. Effort Level (start with "Medium").
4. Click Start to run the autorouter.

## Introduce or Verify Ground Planes

1. Select the Polygon tool from the toolbar.
2. Check if a GND polygon already exists.
3. If not, draw a rectangle surrounding the board outline.
4. Set the Signal Name to GND.
5. Click Ratsnest to fill the copper pour.

## Run Design Rule Check (DRC)

1. Go to Tools → DRC.
2. Load manufacturer-specific DRC settings if provided.
3. Click Check to run the analysis.
4. Fix any errors (and seriously review warnings).

## Export Gerber Files for Manufacturing

1. Open \*\*File → CAM Processor\*\*.
2. Use a predefined Gerber job (if available) or set up your own layer stack.
3. Export:
  - a. Copper layers (Top/Bottom)
  - b. Silkscreens
  - c. Solder masks
  - d. Drill files (NC Drill)
4. Package all exported Gerber files into a ZIP folder ready to send to a PCB manufacturer.