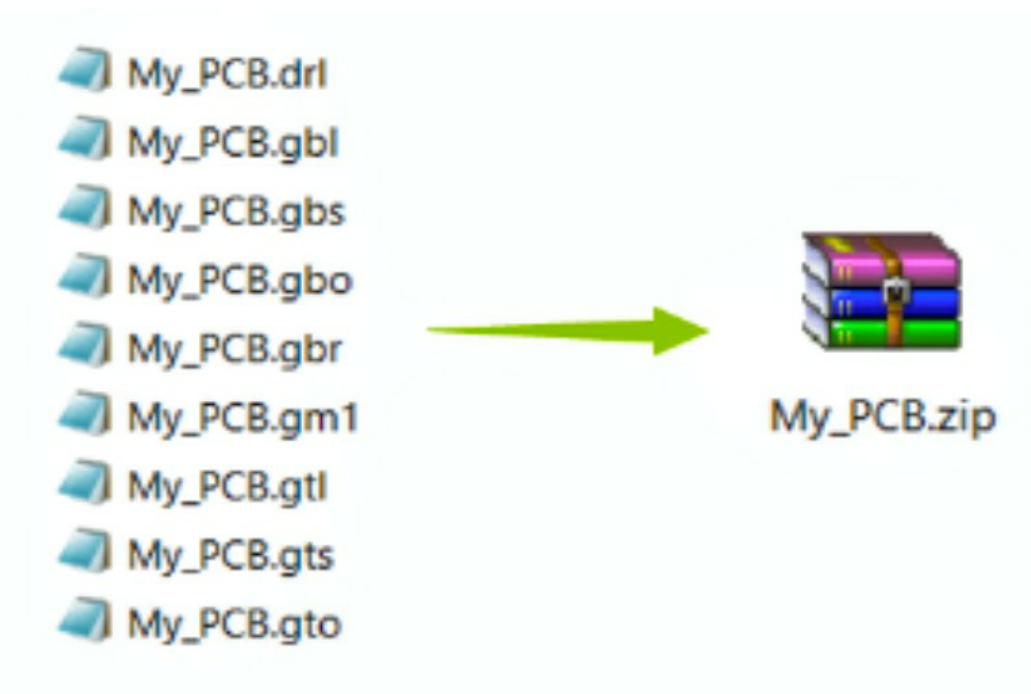


DAILY ASSESSMENT FORMAT

| | | | |
|-----------------------|----------------------------|------------------------|-----------|
| Date: | 13-06-2020 | Name: | Abhishek |
| Course: | PCB Design using Kicad | USN: | 4a17ec001 |
| Topic: | 1]Prepare production files | Semester & Section: | 6 & 'A' |
| Github Repository: | Abhishek-online-courses | | |

FORENOON SESSION DETAILS

Image of session



Report –

Prepare production files :

- Production files need to be generated are **Gerber** and **Drill** files.
- Steps in creating these files are,
 - ✓ Select the 'Plot' button
 - ✓ Make sure the Plot format is set to 'Gerber' and all the aforementioned layers have been selected.
 - ✓ Once our gerber files have been generated, they can be reviewed to catch any potential errors.
 - ✓ Click on File and select Load Gerber file.
 - ✓ Select all of the layers shown and click 'Open'.
 - ✓ In the same manner, click on 'File' > 'Load EXCELLON Drill File'.
 - ✓ Select the drill file(s) and click 'Open'.

