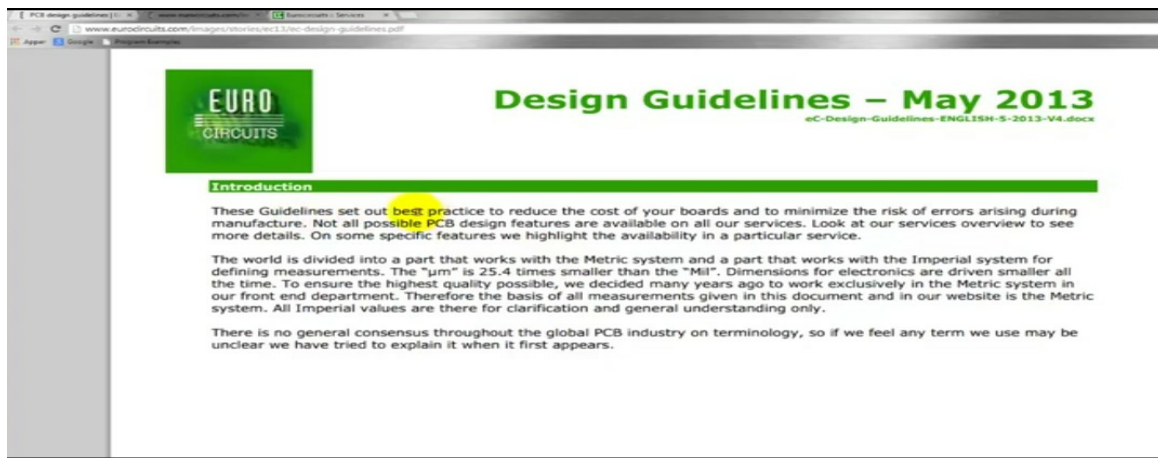


DAILY ASSESSMENT FORMAT

Date:	12-06-2020	Name:	Abhishek
Course:	PCB Design using Kicad	USN:	4a17ec001
Topic:	1]Create PCB footprint component 2]Prepare production files	Semester & Section:	6 & 'A'
Github Repository:	Abhishek-online-courses		

FORENOON SESSION DETAILS

Image of session



Lectures

More



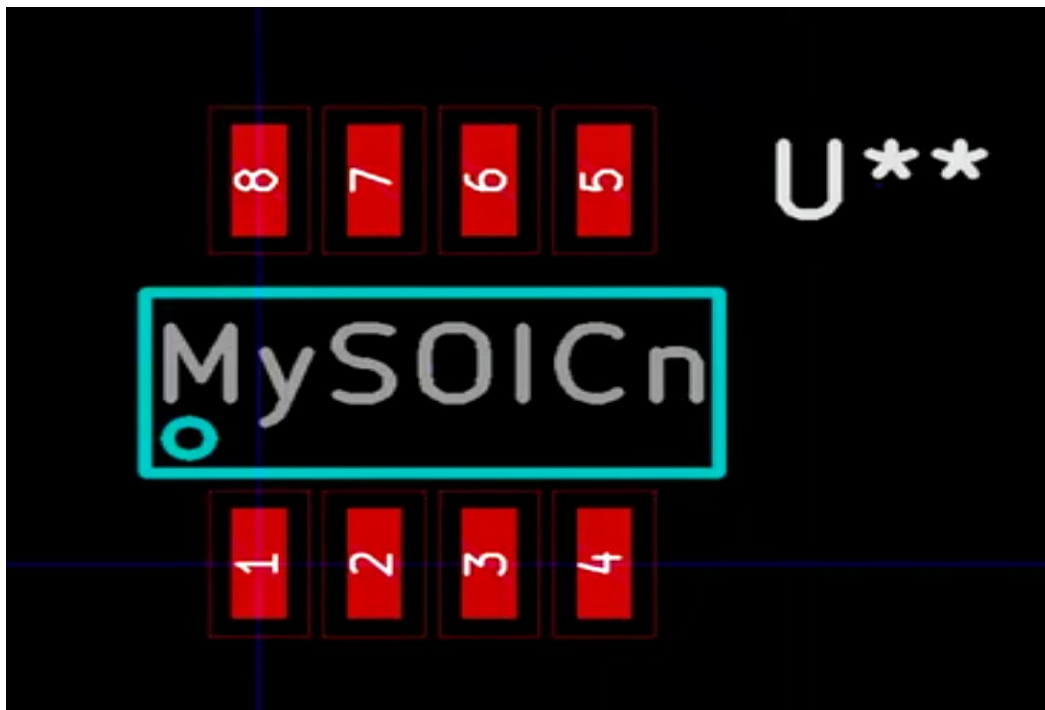
- 1 ☒ Start a new project.
Video - 17:38 mins
- 2 ☒ Netlist and footprint association a...
Video - 16:12 mins
- 3 ☒ Silk-screen and copper pour.
Video - 08:41 mins

Report –

KiCad is a free and open source Electronic Design Automation (EDA) software package used to draw schematics (known as schematic capture) and for PCB design and layout.

Create PCB footprint component :

- Steps in creating a PCB footprint component are,
 - ✓ In the footprint editor, click on new footprint button.
 - ✓ Name the new footprint.
 - ✓ Add boundaries and pins.
 - ✓ The pad button allows you to add pads on the canvas.
 - ✓ Use the polygon tool to draw the silkscreen shape that will indicate the border of the custom footprint.
 - ✓ Use the text label tool to label the pins.
 - ✓ Save the footprint.
- Example on creating a new footprint component,



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'.

