

DAILY ASSESSMENT FORMAT

Date:	12/06/2020	Name:	Nishanth
Course:	Pcb design	USN:	4a117ec063
Topic:	1.prepare production files	Semester & Section:	6 th b-section
GitHub Repository:	nishanthvr		

FORENOON SESSION DETAILS

- Tool lists for drill files are **ALWAYS** read by our CAM systems as finished hole sizes (ENDSIZE).
- All PCB drills are manufactured in increments of 0.05 mm. So we convert the drill sizes given in the drill files or lists into millimeters and round to the nearest 0.05mm.
For example:
 - Drill size of 31mil is converted to 0.7874mm and then rounded to 0.80mm.
 - Drill size of 32mil is converted to 0.8128mm and then rounded to 0.80mm.
 - Drill size of 33mil is converted to 0.8382mm and then rounded to 0.85mm.
- If possible, provide separate drill files for plated (PTH) and non-plated (NPTH) holes. If this is not possible, always specify different tools for PTH and NPTH holes and mark clearly which tools are PTH and which tools are NPTH.
- When no PTH/NPTH info is given we use the following rules to determine PTH/NPTH:
For 0-layer and 1-layer boards: → ALL holes are considered as NPTH by default.
For 2-layer and multilayer boards: → ALL holes are considered PTH except the following cases which are considered NPTH:
 - Non-connected holes without copper pads.
 - Non-connected holes where the copper pad size is equal to or smaller than the drill TOOLSIZE (the copper pad will be removed in single image preparation)
 - Connected holes with a copper pad on 1 side (outer), no connection on any other layer (outer or inner) and no copper pad on the other side (outer).

Lectures

More



	Video - 16:12 mins	
3	Silk-screen and copper pour.	
	Video - 08:41 mins	
4	Mounting holes.	
	Video - 03:31 mins	
5	Create a library and put your ...	
	Video - 08:30 mins	
6	Create PCB footprint compon...	
	Video - 11:53 mins	
7	Add Footprint search path	
	Video - 01:50 mins	
8	Prepare production files.	

- What is gerber file

Gerber files are the artwork of the ways layers that will be used to construct the board once its being fabricated.

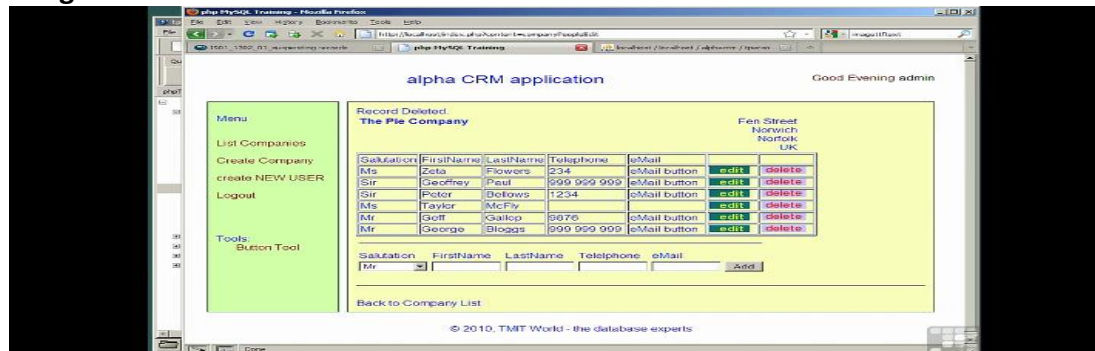
Generating gerber files

1. select the plot button
2. make sure plot format is set to gerber and all the aforementioned layers have been selected.
3. under the messages windows will show where the drill files have reviewed to catch any potential error before being sent to us for pcb fabrication quote.

Date: 12/06/2020 **Name:** Nishanth
Course: Beginner PHP and MySQL **USN:** 4a117ec063
1.00 Programming - **Semester & Section:** 6th and b section

AFTERNOON SESSION DETAILS

Image of session



Lectures

More

Section 13 - Real Life PHP Introduction

- 142 ☒ Real Life Introduction
Video - 01:29 mins
- 143 ☒ Building PHP Tools - Button ...
Video - 05:30 mins
- 144 ☒ Building PHP Tools - Embedd...
Video - 03:59 mins
- 145 ☒ Managing Deletions - Suspen...
Video - 03:29 mins
- 146 ☒ Managing Deletions - Suspe...

TODAY I LEARN THIS CONCEPTS :

1. Email with PHE,Email With PHP And MySQL,Sendmail SMTP Server,Sendmail mail Function,PHP mail CC And BCC,HTML email Content,Email Out - Hiding SMTP Address,Email Out - Embedded Form,Logging Sent Email
2. Real life PHP IntroductionReal Life Introduction,Building PHP Tools - Button Maker,Building PHP Tools - Embedded Tool,Managing Deletions - Suspension Fields,Managing Deletions - Suspend Record,Managing Deletions - Restore Records,OO Programming - A Caveat,OO Programming Model - DB Connection,OO Programming - DB Examples,CMS Open Source,CMS Joomla,CRM Download And Extract,CRM vTiger Install,CRM Modifying vTiger