



Försättsblad Prov Original

Kurskod	Provkod	Tentamensdatum
E T 1 0 0 G	T 1 0 1	2 0 1 8 - 0 3 - 1 9
Kursnamn	Elektroteknik GR (C), Prototypkonstruktion av inbyggda sy...	
Provnamn	Skriftlig tentamen	
Ort	Sundsvall	
Termin	V18	
Ämne	Elektroteknik	

Exam in “Prototyping of Embedded Systems”

Course: “Prototyping of Embedded Systems”: ET100G

Date: 19th of March, 2018

Examiner: Peng Cheng Office Tel: 010-1428495

Allowed to use: Pens, erasers, ruler.

Marks: All questions give 2 p. Write in as clear as possible English!

Number of marks (p)	Grade
>=38	A
>=34	B
>=30	C
>=26	D
>=22	E (Approved) ☺
<22	F

- 1) What is PCB used for? What is it usually made of (two things)?
- 2) What layers cannot be used to mount surface-mount components in a four-layer PCB? Top? 2nd layer? 3rd layer? Bottom? And why?
- 3) What is via used for in PCB? What are the three different kinds of vias and describe their differences in construction.
- 4) Draw a via, show out what is the via ring thickness and what is the via hole diameter.
- 5) What are meanings of clearance and minimal trace width setting in PCB design? And why are they needed?
- 6) What are the PCB design differences for mounting of surface-mount components and through-hole-mount components? Which one is more popular in PCB design now?
- 7) What is solder mask in PCB production and assembly? And what is it used for?
- 8) What is silkscreen in PCB production and assembly? And what is it used for?
- 9) What is paste mask in PCB production and assembly? And what is it used for?
- 10) What is board outline in PCB production and assembly? And what is it used for?
- 11) What is ground flooding in PCB design? And what is it used for?
- 12) What are the plated drill and non-plated drill? And what is the difference between them?
- 13) Is it ok for silkscreen to overlap mounting pads in PCB design? And why?
- 14) What is used to connect and mount terminals of components to their mounting pads on PCB? And briefly describe how it is used to do so.
- 15) Why are power supply and grounding traces in PCB design usually wider than signal traces?
- 16) For high speed signal trace routing, people typically want the trace to be routed thin, short and close to ground, which three factors are the in the consideration? And why?
- 17) Why different grounds are recommended to be mostly separated and only be joined at the final power supply ground point? What are in the consideration?
- 18) Are there all values available to buy for the passive components on the market? And why?
- 19) What are the meanings of ideal and non-ideal passive components?
- 20) Can a 10Ω and a 1 MΩ resistors have the same small component size of 0402? And can a 10pF and a 100μF capacitor have the same small component size of 0402? And why?
- 21) Why resistor has the power range rating? Why capacitor has the voltage range rating? How would they fail if you use them above these ratings? (describe their failure mode)
- 22) What is the meaning of tolerance rating for the passive components? How high the percentage can the tolerance ratings of commercial resistor and capacitor be in market?