

Skills: 16 Electronics

Information for the Task „HW-Design“

1. Introduction

In the task HW-Design you need to design a printed circuit board (pcb). Read this document carefully, so you have good chances to succeed. You can prepare yourself for this task right now.

2. Parts Library

You see in the table below the electronic parts we will use for this task. There is no time to generate all these parts in your layout tool while the competition, so you need to do this job until the start of competition.

At least one new part you will have to create while the competition.

Pos.	Part	Typ	Producer	Footprint	Distributor	Part-No.
1	ESP32-Pico-Kit	WLAN Development Tools (802.11) ESP32-PICO-D4 Development Board	Espressif Systems	DIL40-W	Mouser	356-ESP32-PICO-KIT
2	IC: OPV	MCP6002T-I/SN, 2 channel, 1.8 .. 5.5V, 23mA, 1MHz	Microchip	SOIC-8	Mouser	579-MCP6002T-I/SN
3	IC: ADC	MCP3428-E/SL, ADC, 16bit delta-sigma, 4 channels	Microchip	SOIC-14	Mouser	579-MCP3428E/SL
4	IC: Timer	NE555DR, Precision Timer, TTL, 4.5V ... 16V	TI	SOIC-8	Farnell	3121192
5	IC: Logic Multifunctional	SN74LVC1G97DBVR, Onfgible Mult Function Gate, 1.65V ... 5.5V	TI	SOT-23-6	Mouser	595-SN74LVC1G97DBVR
6	IC: Logic D-FF	SN74LVC1G175DBVR, D-FlipFlop, 1.65V ... 5.5V	TI	SOT-23-6	Mouser	595-SN74LVC1G175DBVR
7	IC: Logic Inverter	SN74LVC1G14DBVR, SchmittTrigger Inverter, 1.65V ... 5.5V	TI	SOT-23-5	Mouser	595-SN74LVC1G14DBVR
8	Transistor	BC817-40; NPN, 300mW, 45V, 500mA, 250nFE, 170MHz	Multicomp	SOT-23	Farnell	1798081
9	Schottky-Diode	SD0805S020S1R0, 1A Vf=0.45V, Vr=20V	AVX	0805	Mouser	581-SD0805S020S1R0
10	TFT-Display	TFT01-1.65P, 1.8", TFT, 160x128, SPI, 3.3V, ST7735, SD-Card 2GB	-	-	Boxtec	45060
11	Ambient Light Sensor	TEMT6200FX01, Ohot Sensor, Top View, 550nm,	Vishay	0805	Digikey	751-1056-1-ND
12	IR-Receiver Module	TSOP38138, 2.5 .. 5.5V, 36kHz, 45m, 45°	Vishay	THT-5x4.8x6.95mm	Farnell	2251356
13	IR-Transmitter	TSAL4400, 3mm, 25°, 100mA, 1.35V, 940nm, 800ns	Vishay	T-1	Farnell	1045420
14	RGB-LED	WS2812B, RGB-LED, 5V	Worldsemi	SMD	Digikey	1568-COM-16346-ND
15	LED	different Types	-	0805	-	-
16	Piezo	AT-1224-TWT-5V-2-R, 5V, 40mA, 2.4KHz, 87dB	PUI Audio Inc.	THT	Digikey	668-1470-ND
17	Switch On/Off	1101M2S3AQE2, SPDT, 0.4VA, 250V, 6A	C&K	THT	Farnell	1437701
18	Button	MCDTS2-4N, SPST, NO, 12V, 50mA	Multicomp	THT	Farnell	9471634
19	DIP-Switch 8-channel	A6H-8102, SPST, 8Pos, 24VDC, 25mA	Omron Electronic	SMD-RM=1.27mm	Digikey	SW1119-ND
20	Trimpotentiometer	3314J-1-103E, 10k, 250mW, 240°, linear	Bourns	SMD-4mm*2	Mouser	652-3314J-1-103G
21	Resistor	different values	-	0805	-	-
22	Capacitor	different values	-	0805	-	-
23	Pin-Header ?x?	MTSW-xxx, x row, x position, 4.7A	Samtec	THT-RM=2.54mm	Mouser	200-HMTSW11022TS390*
24	Pin-Socket ?x?	SSW-xxx, x row, x position, 4.7A	Samtec	THT-RM=2.54mm	Mouser	200-SSW11001GS*

You will get the effective amount auf parts during the IndustrySkills 2020

You will get the effective value/types of R, C, & LEDs during the IndustrySkills 2020

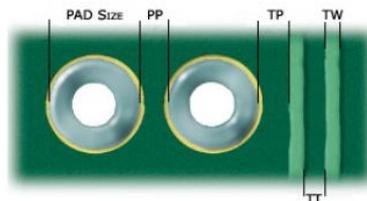
You will have to add at least one more part to the BOM during the IndustrieSkills 2020

* The headers and sockets are not defined yet if one or two row and the amount of contacts and straight or angle => be flexible, normally you can choose them in your library (2.54mm Headers)

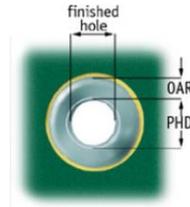
In the e-mail you find the original excel sheet.

3. Design Guidelines

You need to take care for the following listed requirements. The requirements are important, that your pcb can be produced. It is up to you if you want to start and configure already a project.



The minimum track width (TW) is 100µm. The minimum distance from tracks and pads (TT, TP & PP) is 100µm



The smallest production hole diameter (PHD) is 200µm

Think about if it is worth to go to these limits. Possibly there is no solder mask and no vias and you need to do each via by your own!

Tip: <https://circuitpeople.com/Blog/PcbManufacturingChecklist.aspx>

4. Gerber Files

We will produce during the competition a real pcb from your pcb-design and everyone has to assemble his own board in the second part (day 3). For this reason, we need at the end of the first part (day 1) your Gerber-files. Make sure you can generate this file with your SW-tool. We need the following files from you:

- Copper-Top-Layer
- Copper-Bottom-Layer
- Board-Outline-Layer
- Drill-Layer
- Ev. Solder mask-Top

Put your files in to a folder, name it with "SurnameName", and zip this folder.

The experts want to prove your result before we send this zipped folder to the producer. We decide if it is worth to produce or not.

For proving, we use an online Gerber viewer like <https://www.gerber-viewer.com/> (Register / Login needed). If there are problems, choose another **online** Gerber viewer (we do not install viewer!) and put a link as well an information if we need to register to use it (link.txt / Link.doc)

Please check if your tool generates files that can be viewed (maybe with your homework). The experts and our producer have no time to find out what kind of file you send us. We put the zipped folder to the online Gerber viewer and decide of them what we can see there.

Our checklist

- We get all the Gerber files that are needed to produce a pcb
- They are zipped in one folder, so we can throw it to the online Gerber viewer in one step. If we need more than three minutes to see your results on the screen you will get less points.
- The folder is named with "SurnameName"
- The work seemed to be finished so it could work

Think about, that we need to start the production at that evening. If we cannot send your proved files to the producer, there is no chance to produce it. If the producer has questions, there is no time to ask and answer, he decide if he can and will do. It is not only your layout, there are 13 others.

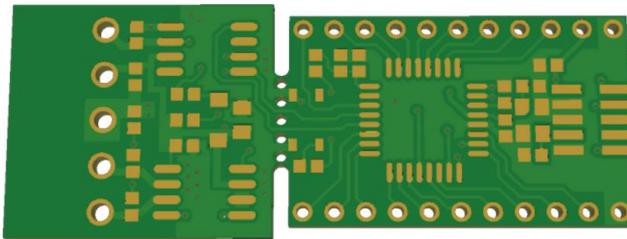
5. Documentation

You need to generate the following PDF-Files from your pcb-design-SW-tool:

- Scheme
- Bill of material
- Assembly of top and bottom side

6. Hints

- If you have never designed a breakable pcb, it is a good idea to learn how you can realize it (Board outline). The whole work is split in more little works, so we can produce at least one pcb. As we cannot produce multiple pcs we have to make it breakable. If you are able to make fast layouts, you can do more layouts and get more points.



<https://electronics.stackexchange.com/questions/473037/how-to-design-a-snap-breakable-pcb-module>

If you prefer to saw there will be one to do this 😊.

- You need to realize solder jumpers with two and three pads. Choose a scheme symbol and a footprint of your own choice.

