Skill 16: Electronics

HW Design
Part 1 "Engineering" (Day 1)

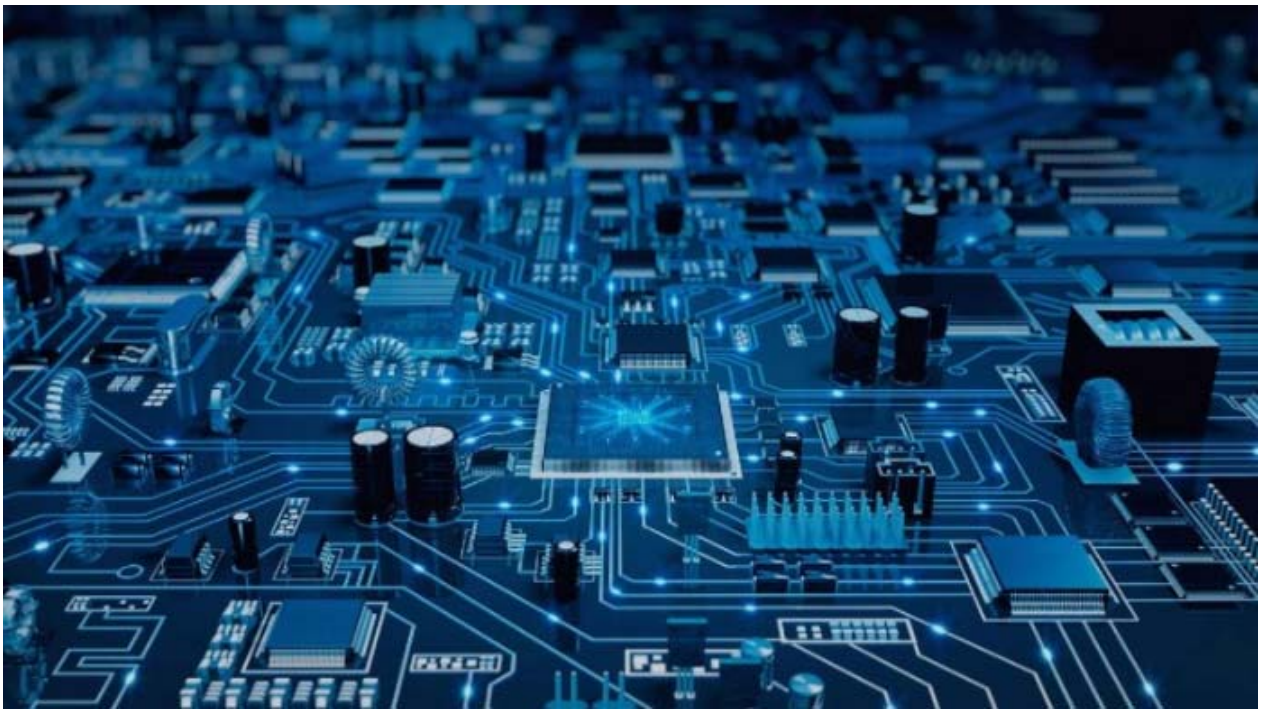

Fig. 1: Cover picture

Table of Contents

1 Introduction

You have to develop a functional PCB within the available time, which meets the desired requirements.

The following steps are required for this:

- Day 1 (full day)
  - Analysis of the task and the existing circuit
  - Complete the component library
  - Expand the circuit according to requirements
  - Draw schematics
  - Create a layout
  - Create gerber files for production (ZIP file) and submit them
  - Submit production files in PDF format

- Day 2
  Schedule according to invitation => other sills are checked.
  Your printed circuit board will be manufactured externally on this day.

- Day 3 (afternoon)
  On the 3rd day, the assembly & mounting, as well as the commissioning & the function test of the printed circuit board take place. The corresponding task will be distributed on that day.

2 Goal

In task HW-Design, among other things we want to see, how well you can assess your skills and where your quality requirements are. The ideal candidate meets all requirements in very good quality in the available time. However, who is Mr. Perfect?

Be aware that certain misjudgments can take revenge later on.

If anyone told you, you could choose between your own layout and an official solution in the past that may have been the case in the past. There is NO official solution available. The alternative to your own layout/PCB is a prototyping board and enameled copper wire.

In the end, the features that work will count, not the ones you planned to realize. You will get more points for working parts than partially finished.

3 Block diagram

The following block diagram provides a first picture of the hardware to be developed

4 Complete the component library

The following components are new and have not been included in the BOM we originally submitted. The BOM and part library must be updated with these components.

4.1 New components

The following two components must be integrated into the project. The corresponding information can be found in the BOM and in the corresponding data sheet in the appendix:

- X1 USB Mini Connector
- S2 Tactile Switch 90°

4.2 Pin & Socket Headers

In the BOM from the invitation, the pin & socket headers have not yet been precisely defined. Also, check the required information in the BOM and the data sheets of Samtec.

5 Schematic and circuit extension

A part of the schematic is given and must be implemented. It is possible to extend the circuit and thus gain additional points. If not all extensions can be implemented due to time constraints, this is not too bad as long as the basic function has been realized.

5.1 Schematic part "Base function"

The schematic in the appendix must be fully implemented. If a part is missing, this task is considered unfulfilled.

5.2 Schematic part "Extensions"

The purpose of the extensions is to be able to use certain HW & SW resources from the ESP32 for other applications. Whether you deliver your solutions on paper or directly in the schematic is up to you.

Only the components in the BOM in the appendix are available to you. For resistors, values from the E12 series (10Ω - 1M) and for capacitors values from the E6 series can be selected (1pF - 1uF). Both components are available in 0805 packages.

5.2.1 Save I/O-Pins

We want to be able to provide the inputs I_34, I_35, I_37 and I_38 to another application if necessary. However, we would still like to be able to use the buttons S2 to S6. Develop a simple circuit with which allows selecting between 5 digital inputs (current) or an analog input (I_39)

Initial situation & requirements:

- 6 different analog voltages must be generated. The minimum distance of the 6 analog voltages to each other is 0.5V.

  For example: S2 = 0V, S3 = 0.6V, S4 = 1.2V, S5 = 1.8V, S6 = 2.4V and no switch = 3V.

  It is not necessary to consider that several buttons could be pressed at the same time.
- The supply voltage is 3.3V (max. voltage)
- It must not flow more than 1mA of current in any constellation
- The resistor values specified in the submitted schematic (R3 to R7 = 10kΩ) can be ignored, as this applies only to those who do not solve this task.
- For the evaluation, it is important that both the digital mode (5 I/O => I_34, I_35, I_37, I_38 and I_39) as well as the analogue mode (I_39) can function independently of each other. The choice of one of these modes must be possible via S7 (only S7.1 – S7.4) and/or via solder bridges.
- The more economical the solution is, the more points you get. The best-submitted solution gives the maximum points. Those who need more or more expensive components (parts and solder bridges) in comparison will receive a deduction from the criterion "economic efficiency" per component.
- It must be clear which analog voltage is to be expected at which key press and how it was calculated.

PS: Points are awarded for each individual requirement met.

5.2.2 Save SW resources for the acoustic output

For generating a frequency or a tone sequence, we want to unload the ESP32. It should only have to generate an enable signal for the duration of the acoustic output.

This task can be divided into different stages. Points can be scored for each stage goal reached.

1. Stage "f1"

   Develop the simplest possible circuit for a rectangular signal generator with the frequency 554Hz, with which you can control the piezo speaker LS1 via R8 and T2. The piezo should sound as long as the signal "Piezo" (IO_26) is high.

2. Stage "f1, f2 and f3"

   We need two additional frequencies of 659Hz and 880Hz. If you do not want to/cannot realize the other milestones, use solder bridges to choose between the 3 frequencies.

3. Stage "Switching frequencies by ESP32"

On each rising edge of the ESP32 signal "Piezo" (IO26) we switch between states f1, f2, f3 and OFF.

4. Stage "Automatic switching"

   As long as the enable signal is in place, your circuit automatically switches between f1, f2, f3 and OFF. Each frequency output takes about 2 seconds (tolerance ±0.5s).

5. Stage "Adjustable switching"

   With the help of a potentiometer (10KΩ) the output duration can be set in the automatic switching in the range of approx. 0.5 - 3s (tolerance -0.2s / + 0.5s).

5.3 General Notes

The following hints must be taken into account in the schematic:

- For the test points, it is only important that they be connected to the correct network. You can choose your own standards for the schematic symbol and SMD pad. You can also add any additional test points at your own discretion.
- During the assembly, you will receive the material in bags. The corresponding BOM refers to the part identifiers specified here (R1, X2, C3, and so on). If you choose a different part numbering, the part sorting using the specified identifiers will be more difficult later. It is up to you to make the best decision for yourself.
- The appearance and completeness of the schematic are also evaluated.
- The solutions for the extensions must be submitted before lunch (notes, calculations, schematic, etc.). For the evaluation of the development part, the documentation provided at this time is relevant.

6 Appendix

For the first part, you get the following additionally information:

- Scheme
- BOM "Basic" and parts for development
- Datasheet "USB-Connector" (PDF)
- Datasheet "Tactile switch 90°" (PDF)