2025

BRICS SKILLS COMPETITION

(BRICS+ FUTURE SKILLS & TECH CHALLENGE)

Intelligent Electronic Products Design and Manufacturing

BRICS-FS-46

Test Project

(International Finals_Online)

August, 2025

金砖国家职业技能大赛（金砖国家未来技能和技术挑战赛）

Contents

BRICS Skills Competition（BRICS Future Skills Challenge）
金砖国家职业技能大赛（金砖国家未来技能挑战赛）

Module A: Electronic Product Design (30 points)

Note: Complete the corresponding design documents according to the task description requirements. The result files of this module need to be saved to the "D:\Submitted Materials\Module A" folder on the server computer. At the same time, copy the entire folder to the root directory of the USB flash drive uniformly distributed in the competition. After the competition, the USB flash drive will be submitted as the competition result.

Task Description

Complete the circuit schematic design and PCB drawing according to the provided materials. The specific tasks are as follows:

Use EDA software to complete the schematic diagram and PCB circuit drawing according to the provided functional circuit schematic PDF;

Draw 3D models or import 3D models according to the specified components.

Task Requirements

1.Establish project files

Establish a project file, which includes schematic files and PCB files. The project file is saved in the "Intelligent Switch Module A Submitted Materials" folder. The file name is the contestant's competition station number, and the names of the schematic diagram and PCB are also the corresponding contestant's competition station number. After completion, take screenshots accordingly.

2.Draw schematic component package diagrams

1). Draw optocoupler

The drawn optocoupler is named "UK1", and its pins and size (grid size is 10) are shown in Figure 1. After completion, take a screenshot of the complete interface of "Component Library \ Symbols".

Figure 1 Drawing of optocoupler style

2). Draw relay

The drawn relay is named "K2", and its pins and size (grid size is 5) are shown in Figure 2. After completion, take a screenshot of the complete interface of "Component Library \ Symbols".

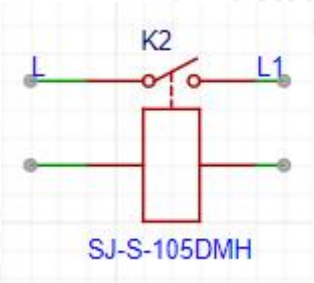


Figure 2 Drawing of relay style

3.Draw schematic diagram

1). Draw the schematic diagram of the intelligent switch. The optocoupler and relay are drawn by the contestant, and the symbols of other components can be the default component library symbols, or can be appropriately adjusted according to the drawings.

2). According to the functional description of the intelligent switch, the input end is required to use optocoupler isolation, and the triode S8050 is used to control the on-off of the relay (SJ-S-105DMH). After completing the schematic design, export the schematic diagram in PDF format, name it "Intelligent Switch Schematic Diagram", and save it in the

"Intelligent Switch Module A Submitted Materials" folder.

Note: It should also be connected with the components outside the box to realize the circuit function.

4.Draw PCB component package diagrams

1). Draw optocoupler package

The reference style of the optocoupler package is shown in Figure 3. The package is named "PS2501L-1". After completion, take a screenshot of the drawn optocoupler package. The package size of the optocoupler can be found in the component size diagram or manual provided in "Module A Venue Materials".

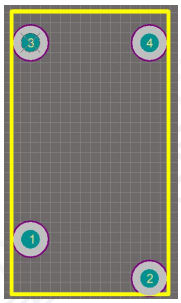


Figure 3 Optocoupler package style

5.PCB layout design

1). Draw the frame

Draw the circuit board frame diagram according to the provided shell size diagram and 3D model diagram, and take a screenshot after completion. The screenshot includes the size of the frame and the shape requirements of the frame.

2). Insert picture

Select the top silkscreen to insert the LOGO picture provided by the venue on the PCB board. The graphic is required to be reversed, and the picture size is 42*55mm. Take a screenshot of the setting window.

3). PCB layout

(1) According to the shell size and assembly requirements, divide into high-voltage board and low-voltage board;

(2) The low-voltage board is mainly the wireless module to control the touch buttons, and the high-voltage board is mainly the input and output of high voltage, and the relay on the high-voltage board is controlled by the low-voltage board.

(3) The high-voltage board and low-voltage board are connected through J1 and J3.

(4) For the low-voltage board, only the touch pad (K1) and touch indicators (D4, D5) are required to be on the top layer, and other components are mostly on the bottom layer; for the high-voltage board, only the terminal J2 is required to be on the bottom layer, and other components are on the top layer;

(5) The touch pad (K1) on the low-voltage board is required to be in the center of the PCB, and there should be no copper cladding within 8mm around the pad;

(6) The terminal J2 needs to be placed with neutral wire and live wire identification characters; the programming interface needs to be placed with VCC and GND identification characters. Place the station number in mirror image on the bottom layer, and the font size is unlimited;

(7) Export PCB diagram

The exported PCB document requires that the top and bottom solder paste layers and solder mask layers are not selected, and all other items are selected. Export the PCB diagram in PDF format, name it "Intelligent Switch PCB" and save it in the "Intelligent Switch Module A Submitted Materials" folder. After completion, take a screenshot of the PCB.

6 .PCB wiring design

1) Add rules

Add the rule "GND" with a line width of 20mil, a spacing of 8mil, a hole outer diameter of 20mil, and a hole inner diameter of 12mil. Take a screenshot of the setting window.

2) Copper cladding

After completing the layout and wiring, perform copper cladding on the top and bottom layers of the PCB, and take a screenshot of the PCB window after copper cladding is completed.

7.Production data output

1) DRC check

After completing the PCB file, check the DRC, which is required to have no errors, and take a screenshot of the design manager interface.

2) "V" cutting requirements

The circuit board is required to be "V" cut during production; the panel is in 2*2 mode; the row spacing and column spacing are 2mm, and it is not required to create a boundary. Take a screenshot of the setting window.

3) Export coordinate file

Name it "Relay Control Coordinate File" and save it in the "Relay Control Module A Submitted Materials" folder. It is required to include the coordinate file after panelization.

Module B: Simulation Design and Development (30 points）

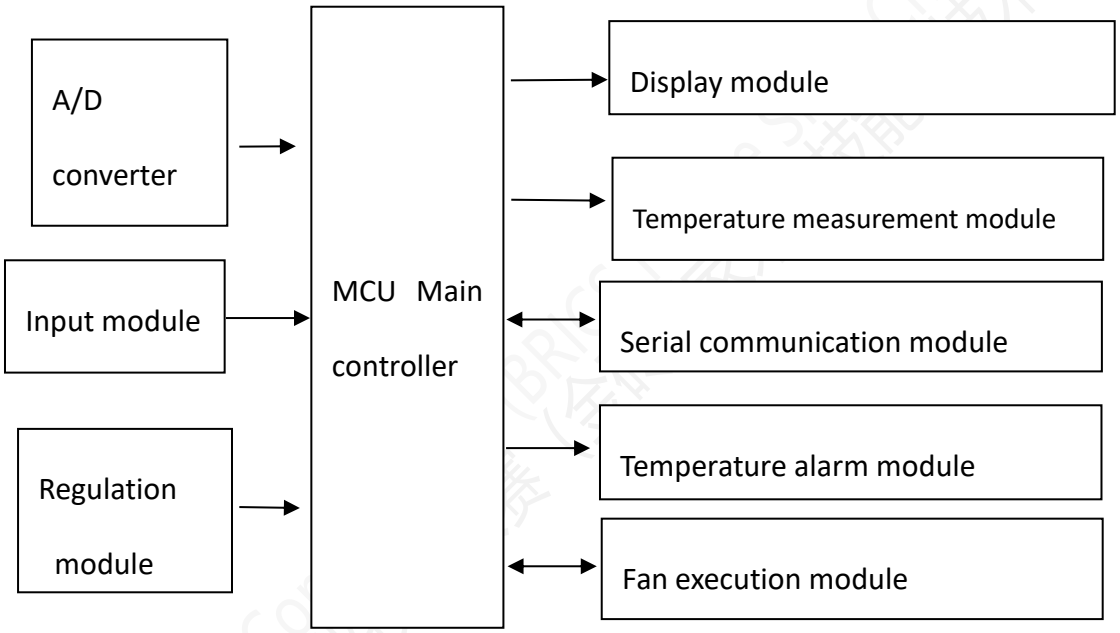
Function Description

Realize a functional simulation system: an intelligent temperature control system based on auxiliary simulation software.

Function Requirements

1.Build a simulation circuit

It has functions such as temperature detection, data display, temperature alarm, and temperature regulation.



2.Write program logic functions

According to the task function requirements, the participating teams need to realize the functions, and with reference to the materials (partial driver code) issued in the competition, use the MCU programming software (it is recommended to use keil) to write the program. Click the "MCU Programming" button on the experiment panel to burn the compiled HEX file into the MCU, click "Run Experiment" and combine the completed functional simulation for debugging to complete the functions required by the task function requirements.

(3) Task function requirements

1.Build a simulation circuit

Build the circuit with reference to the simulation circuit design diagram, correctly configure the MCU input and output IO, and correctly connect the lines.

(1) Add MCU (STM32).

(2) Add buttons.

(3) Add A/D analog input module.

(4) Add a serial debugging assistant module to realize data collection and display.

(5) Add a 1602 LCD screen to display the task content normally.

(6) Add a sound module, correctly configure the sound module peripheral IO port, and can prompt normally by voice.

(7) Add two LED lights (temperature alarm module), which are green and yellow respectively. The LED lights are activated at high level by default.

(8) Add a DC motor to simulate the rotation of the fan motor, which can control the speed to realize forward and reverse rotation.

2.Write program logic functions

(1) Realize temperature collection and real-time display, display of upper and lower temperature limits (lower limit 25℃, upper limit 30℃), display of current mode, etc.

(2) In the automatic mode, it has the function of sound and light alarm when the temperature exceeds the upper and lower limits.

(3) The upper and lower temperature limits can be set, with power-off storage function.

(4) The system has automatic control/manual control functions.

Automatic control function: When the temperature exceeds the upper and lower limits, the corresponding regulation module will work (the regulation module can choose a motor, relay, etc., to simulate cooling, heating and other equipment).

Manual control function: The input module can be used to separately control the regulation module.

(5) With remote monitoring function.

The remote temperature, status, and regulation module can be monitored through the serial debugging assistant.

Remote mode switching, temperature upper and lower limit setting, and separate control of the regulation module can be realized through the serial debugging assistant.

Module C: Embedded Application Software Development (30 points)

Task Description

Complete the relevant application software development according to the package library and materials provided by the venue. The software interface is required to display sensor data, be able to control actual/simulated equipment, set relevant scenarios, and upload data to the server.

Task Requirements

1.Complete the software login and main interface construction.

2.Display real-time sensor data on the main interface, including but not limited to temperature and humidity, illuminance, CO2, etc.

3.Be able to control the on-off of the relay on the main interface and feed back the real-time status.

4.Custom scenarios can be set on the main interface.

金砖国家职业技能大赛（金砖国家未来技能和技术挑战赛）