Test Project

Module A2 - PCB Design

Electronics

Submitted by:
Shuichang Yang – Sanxiang, Independent Test Project Designer
Steve Williams, Independent Test Project Validator

WSC2026_TP16_en

Contents

Test Project files 3
Introduction 3
Instructions: Symbol and Footprint Creation 3
Instructions: PCB Layout 3
Gerbers 4
Drill files 5
BOM File 5
Marking Scheme 5

© WorldSkills International

Test Project files

The test task includes the following files:

- WSC2026_HWD_Datasheets.zip
- WSC2026_TP16_A2_actual_en.docx — This file
- WSC2026_TP16_HW_A2_StartingPoint.fsch — Fusion schematic file

Introduction

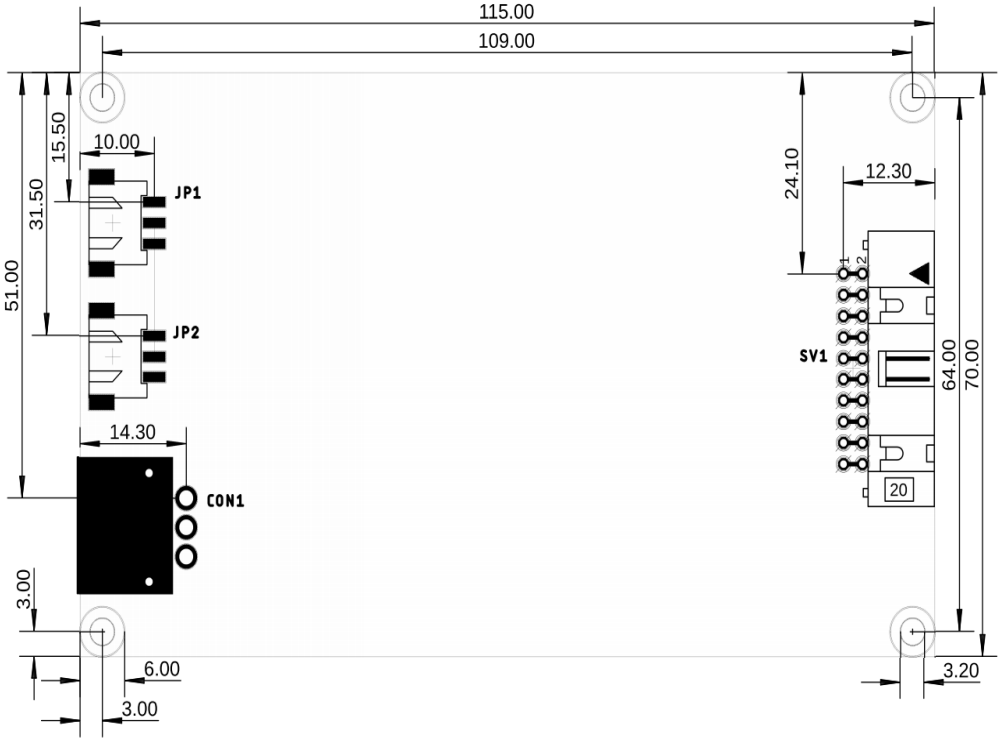
At this stage of the testing project, you will design the PCB layout for the left wheel diver circuit board of bipedal wheeled robot. You will receive an incomplete schematic (WSC2026_TP16_HW_StartingPoint.fsch), which is missing one component. You need to create schematic symbols and footprint for missing components, and then use this starting schematic to complete the PCB layout.

Instructions: Symbol and Footprint Creation

Use the provided data sheet (WSC2026_TP16_datasheet_MR30PW-M30.G.Y_en. pdf) to create the Power Block MR30PW-M30.G.Y Create schematic symbols and associated footprint as shown on the datasheet.

Instructions: PCB Layout

Load the schematic of "WSC2026_TP16_HW_StartingPoint.fsch" and insert the MR30PW-M30.G.Y component you created. Start PCB layout. Use the following diagram to determine the size of the PCB, the position and dimensions of the mounting holes, and the positions of the MCU module, motor, and servo connectors.



© WorldSkills International

Layout the PCB as a double-sided circuit board. Try to keep your route as short as possible and place the ground on the top and bottom floors.

Please integrate the measurement pins and the following signals, place the labels next to them, and write the names on them:

GND (TP0)	OUTA (TP4)	M_V_CS (TP8)
INA (TP1)	OUTB (TP5)	M_W_CS (TP9)
INB (TP2)	SPWR (TP6)	GND (TP10)
VSSA (TP3)	M_U_CS (TP7)	

The PCB must also have the following items:

- 3-pin servo connector (JP1), used to connect the left front servo
- 3-pin servo connector (JP2), used to connect the left rear servo
- 3-pin motor connector (CON1), used to connect the left motor
- 20 pin connector (SV1), used to connect the MCU module

Design Rules

- Minimum clearance: 0.254 mm
- Minimum trace width: 0.254 mm
- Minimum drill: 0.6 mm
- Minimum track width - signals: 0.254 mm
- Minimum track width - Power Supply: 0.6 mm
- Minimum track width - Motor Power: consider 1.0 mm
- Polygon power/ground plane: minimum width 0.3 mm
- Thermal Isolation width: 0.4064 mm
- Minimum via: 0.6 mm
- Maximum via: 1.0 mm (except connectors)
- The loop length of motor power lines should be as short as possible
- Vias do not need thermal relief
- The diameter of the mounting hole is 3.2 mm, a minimum restricted area of 6mm is requires around the mounting hole
- Include your two-letter country code on the copper layer of your PCB.
- Save your work in the competition directory

Gerbers

Create the Gerber files needed for fabrication as requested below:

- copper
- profile
- soldermask
- silkscreen
- gerber_job
- solderpaste

© WorldSkills International

Drill files

Create the NC drill file needed for fabrication as requested below:

- Three Integer and 3 Decimal format

BOM File

Create the Bill of Material file needed for fabrication as requested below:

- List type: Group by Value
- Format: .xlsx

Marking Scheme

SUB CRITERION	SUB CRITERION NAME	WSOS
A2	Design of PCB board layout - Phase 2	3, 4, 7

© WorldSkills International