Test Project

Module A - Development of circuits

Electronics

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

WSC2026_TP16_en

Contents

Introduction to Test Project **3**
Introduction **3**
Description of project and tasks **4**
Task 1: Constant current charging circuit (LTspice） **5**
Task 2: Voltage regulator（LTspice） **6**
Task 3: Power amplifier circuit (LTSpice) **7**
Task 3A 7
Task 3B 7
Components List **10**
Marking Scheme **11**

© WorldSkills International

Introduction to Test Project

This testing project includes the following documents:

- HWD_Datasheets.zip
- WSC2026_TP16_A1_actual (This file)
- WSC2026_HWD_BOM_en.xlsx (Bill of Materials)
- WSC2026_HWD_Task1_starting_point.asc (LTSpice schematic)
- WSC2026_HWD_Task2_starting_point.asc (LTSpice schematic)
- WSC2026_HWD_Task3_starting_point.asc (LTSpice schematic)
- WSC2026_HWD_Task4_starting_point.asc (LTSpice schematic)
- WSC2026_HWD_Task5_starting_point.asc (LTSpice schematic)
- CountryCode-WSC2026_TP16-A1_answer sheet

Introduction

You need to complete five designs, please remember the following points:

- All tasks are independent of each other and do not rely on previously completed tasks.
- Only use components from the parts list during design.
- If you have used all the components in one task, you cannot use them again in another task.
- Your 'Answer Pack' will consist of answer sheets (. docx), simulation circuit files (. asc):
    - The simulation results will be presented in the form of screenshots.
    - The captured image must include the circuit, and the result must clearly display the correct operating status of the circuit (see Figure 1).

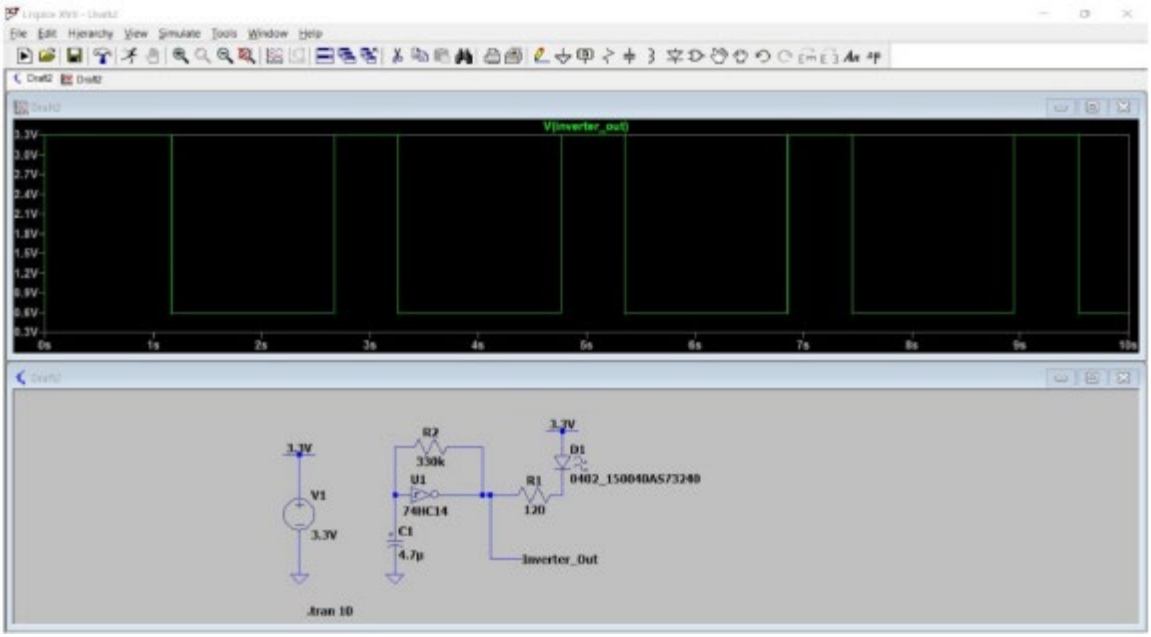


Figure 1: Example docx file, including LTSpice screenshot, showing circuit structure and simulation results.

- Paste your simulation results into the file 'CountryCode-WSC2026_TP16-A1_answer sheet. docx'.
- The simulation answer sheet has corresponding filling areas for each task simulation.

© WorldSkills International

- All simulation results must be saved in the same. docx file.
- Mark your name and two letter country code on all answer sheet pages.
- After the file is completed, or before the deadline, convert the docx document to PDF format and submit two copies for evaluation.
- Please make sure to save LTSpice files containing the correct simulation parameters, as these files can be used to evaluate your design (experts will not manually input simulation parameters).
- The LTSpice file is saved in the format of CountryCode-HWD (TaskX. asc).
- Place the LTspice file (. asc) in your Competitor results directory.
- You must complete the production of the above documents within the task time. After the time is up, experts will collect your work results for evaluation.

All files must be saved in the 'Competitor results' directory. The system will not provide additional time for generating these files.

The calculation results can be saved using the formula editor in Word at:

CountryCode_WSC2026_HWD_A1_AnswerSheet .docx The designated area in the file, or directly recorded by Competitors on the provided calculation paper.

Description of project and tasks

With the continuous iteration and upgrading of intelligent robot technology, bipedal wheeled robot have been widely used in various fields such as industrial inspection, smart logistics, public security, agricultural operations, scientific research and training, and livelihood services, relying on their unique advantages of wheel foot switching, all terrain adaptation, and flexible obstacle crossing. This project requires the design of an intelligent bipedal robot that integrates self balancing control and remote control, achieving an integrated design of stable driving, intelligent interaction.

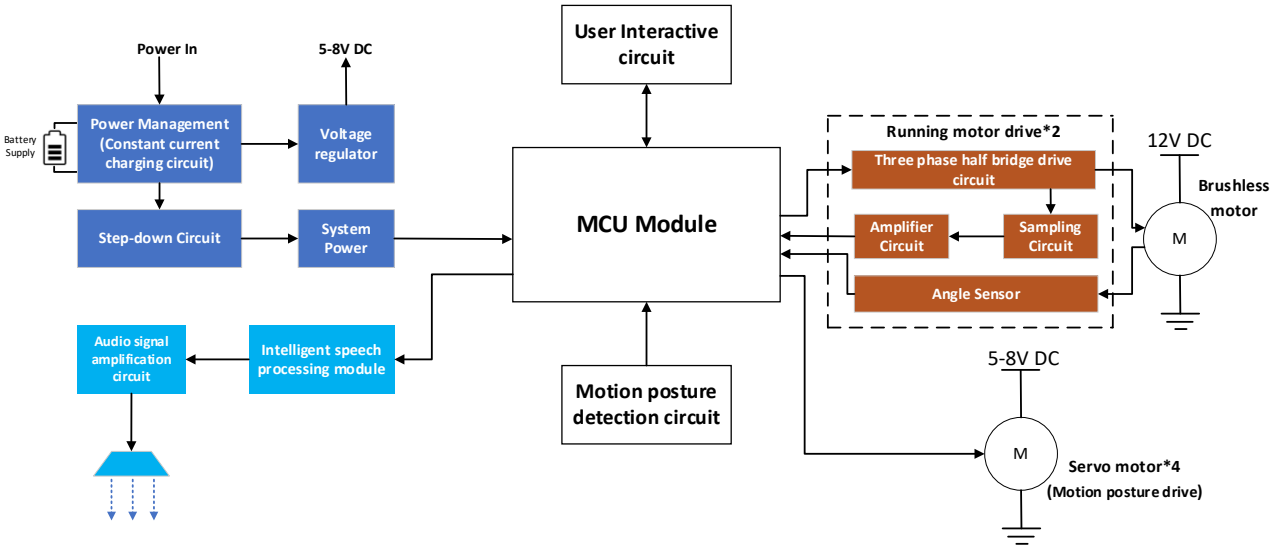


Figure 2 Block Diagram of the Control Circuit for the bipedal wheeled robot

This system is centered around the MCU module: the power module provides stable power supply for the entire system; Motion posture control circuit, motor current sampling, and angle sensors provide feedback on the robot and motor status to the MCU module; The MCU module outputs a driving signal based on this, and the motor is

controlled by the driving circuit to achieve self balancing; The joystick sends commands to the MCU module enable human-machine control operations, and achieve closed-loop control and multimodal interaction.

Task 1: Constant current charging circuit (LTspice)

You have to design a constant current charging circuit that can quickly charge lithium batteries, the circuit with the following characteristics:

- Supply: +18 VDC
- Battery charging maximum current: 2A (+/- 0.2A)
- Battery charging maximum Voltage: 12.6V (+/- 0.1V)
- Resistors as much as you need (see resistor kit)

Using the LTSpice schematic
"WSC2026_HWD_Task1_starting_point.asc" (Figure 3) complete the schematic and simulate as requested.

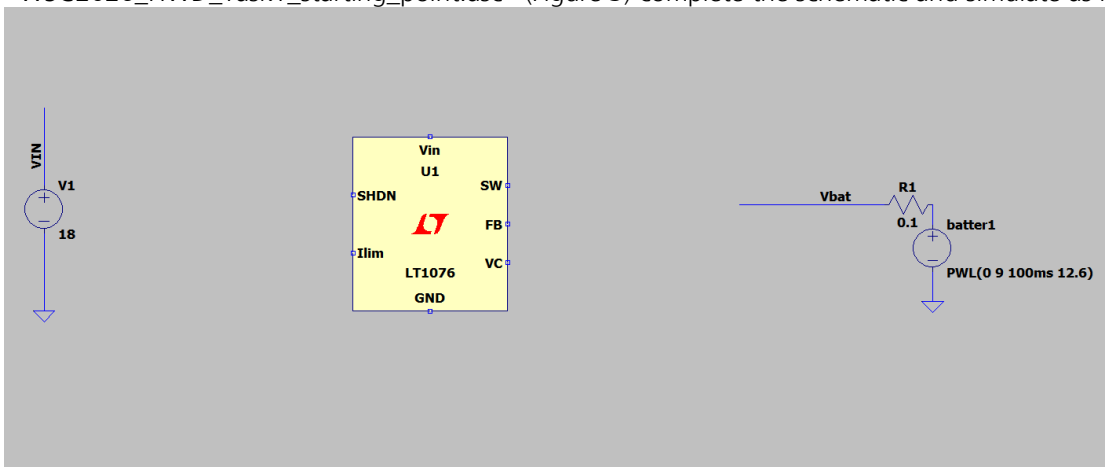


Figure 3 Incomplete task 1 (LTSpice)

Note: Simulated Battery has been provided in "WSC2026-HWD_Task1_starting_point. asc". The battery consists of an internal resistance R and a Piece-Wise Linear Voltage Source PWL (with an internal resistance of 0.1 ohms and a charge discharge range of 9-12.6) V) ,Competitor needs use this Simulated Battery complete the schematic and simulate.

Rename the LTSpice. asc file to "CountryCode_ WSC2026-HWD_Task1. asc" and save the file to the competition directory.

Screen Capture : Insert a screen capture of your simulation results with associated schematic shown at the same time into the "CountryCode_WSC2026_TP16_A1_answer_sheet.docx" under the Task 1 heading that clearly shows the Simulated Battery relationship between charging voltage and current is such that the current remains constant within the range of 2A (+/- 0.2A) during charging. When the battery voltage rises to 12.6V(+/- 0.1V), the charging current gradually decreases by nearly 0A.

© WorldSkills International

Task 2: Voltage regulator（LTspice）

You need to design a voltage regulator that can adjust the output voltage of the battery to the power supply of the servo motor, the circuit with the following characteristics:

- Supply: +12 VDC
- Voltage output: +5V to 8VDC(+/-0.2V)
- Maximum output current: ≤ 2A
- Resistors as much as you need (see resistor kit)

Using the LTSpice schematic

"WSC2026_HWD_Task2_starting_point.asc" (Figure 4) complete the schematic and simulate as requested.

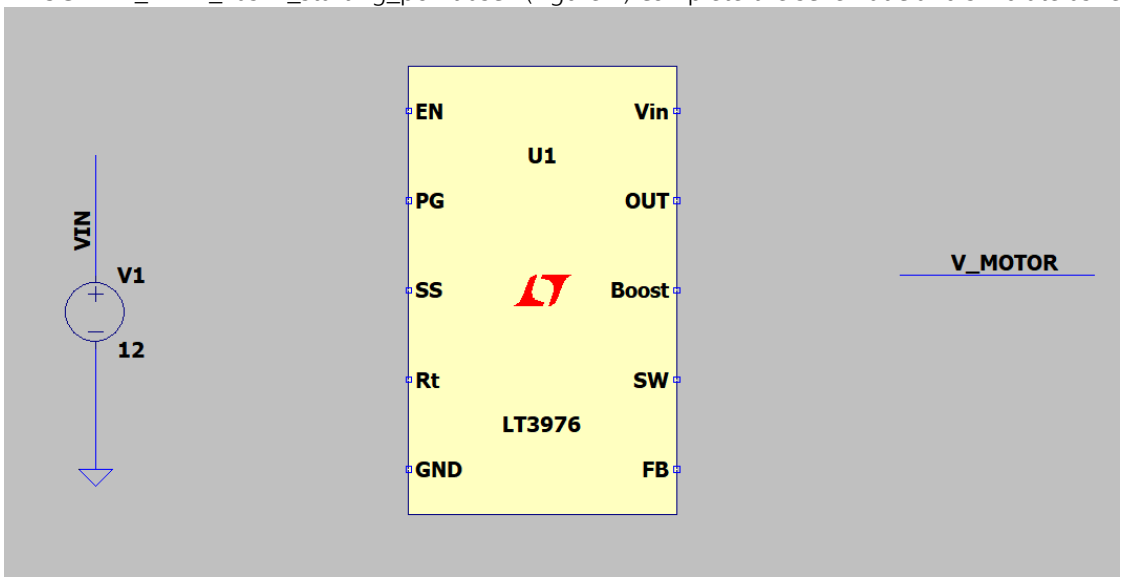


Figure 4 Incomplete task 2 (LTSpice)

Screen Capture1: Insert a screen capture of your simulation results with associated schematic shown at the same time into the "CountryCode_WSC2026_TP16_A1_answer_sheet.docx " under the Task 2 heading that clearly shows the Minimum voltage output.

Screen Capture2: Insert a screen capture of your simulation results with associated schematic shown at the same time into the "CountryCode_WSC2026_TP16_A1_answer_sheet.docx " under the Task 2 heading that clearly shows the Maximum voltage output.

Save the LTspice file with settings needed to produce the requested simulation in the competition directory as "CountryCode_ WSC2026_HWD_Task2 .asc".

© WorldSkills International

Task 3: Power amplifier circuit (LTSpice)

You have to design an speaker power amplifier with the following characteristics:

- Supply: +12V
- High-pass cutoff frequency: 40 Hz(+/-2Hz)
- Gain $_{(DC)}$: -20（+/-5%）
- Load: 4Ω
- Load peak power：≥3W

This task consists of 2 sub-tasks, task 3A and 3B.

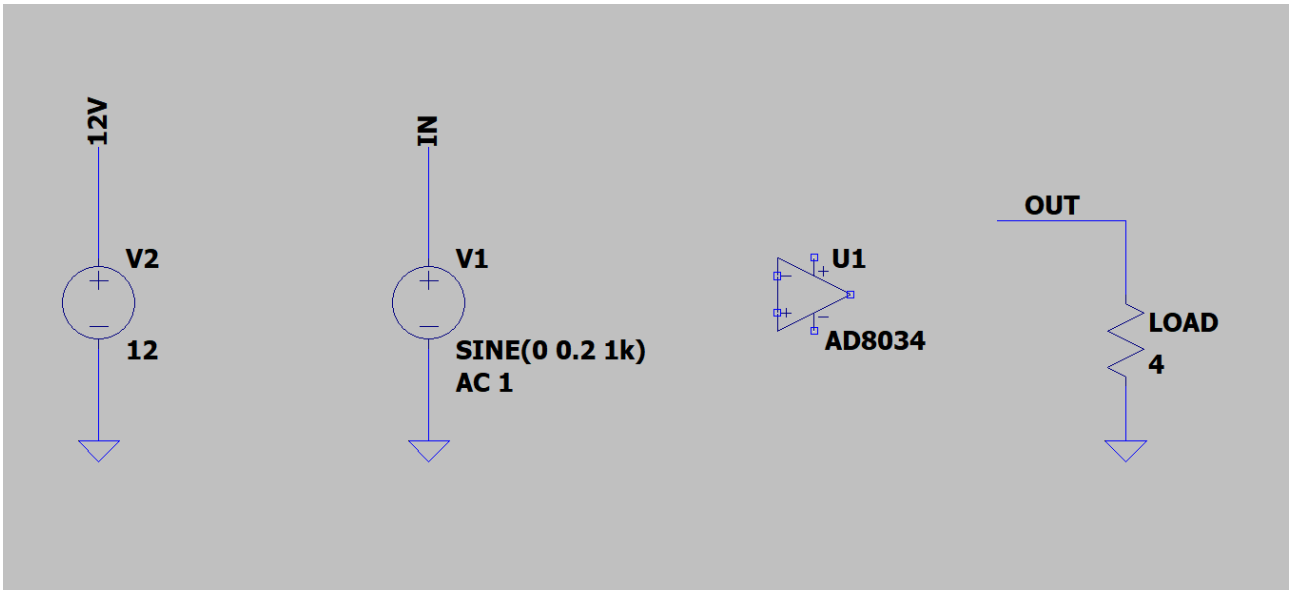


Figure 5 Incomplete task 3 (LTSpice)

Task 3A

Design a pre-amplifier circuit using AD8034 to provide an output as close to 8.0 Vp-p and DC offset at 6V. Output is based on an input sine wave signal of 0.4Vp-p at 1kHz(DC offset=0V). Draw and simulate your circuit in LTspice.

Screen Capture 1: Insert a screen capture of your simulation results with associated schematic shown at the same time into the "CountryCode_WSC2026_TP16_A1_answer_sheet.docx" under the Task 3A1 heading that clearly shows the input and output of amplifier.

Screen Capture 2: Insert a screen capture of your simulation results with associated schematic shown at the same time into the "CountryCode_WSC2026_TP16_A1_answer_sheet.docx" under the Task 3A2 heading that clearly shows the hi-pass cutoff frequency of the amplifier,Perform an AC analysis decade sweep plot from 40 Hz to 100 kHz of your circuit in LTspice.

Task 3B

Design a current amplification circuit for the AD8034 to drive the 4 Ω load(Speaker).This circuit consists of a push-pull amplifier built with two Darlington transistors, it amplify the current signal output by AD8034.This task based on the circuit of Task 3A,modify one wire in the task 3A circuit ,the output current can drive a 4Ω load

© WorldSkills International

without signal distortion,and the peak power of the load shall not be less than 3W.You may use diodes, transistors and other components from the component list, draw and simulate your circuit in LTspice.

Screen Capture : Insert a screen capture of your simulation results with associated schematic shown at the same time into the "CountryCode_WSC2026_TP16_A1_answer_sheet.docx" under the Task 3B heading that clearly shows the power(Vload*Iload) situation of the Load.

Save the LTspice file with settings needed to produce the requested simulation in the competition directory as "CountryCode_ WSC2026_HWD_Task3 .asc" .

Task 4: Current sampling circuit（LTspice）

You need to design a current sampling amplification circuit to convert the current into the corresponding voltage output, the circuit with the following characteristics:

- Supply: 3.3V DC
- Current input: 1.65A AC
- Voltage output: 0-3.3V
- Current voltage conversion ratio: 1(A):1(V)
- To simulate use the AD8031 as linear circuit instead of GS8591,R1 is sampling resistor.
- Resistors as much as you need (see resistor kit).

Using the LTSpice schematic

"WSC2026_HWD_Task4_starting_point.asc" (Figure 5) complete the schematic and simulate as requested.

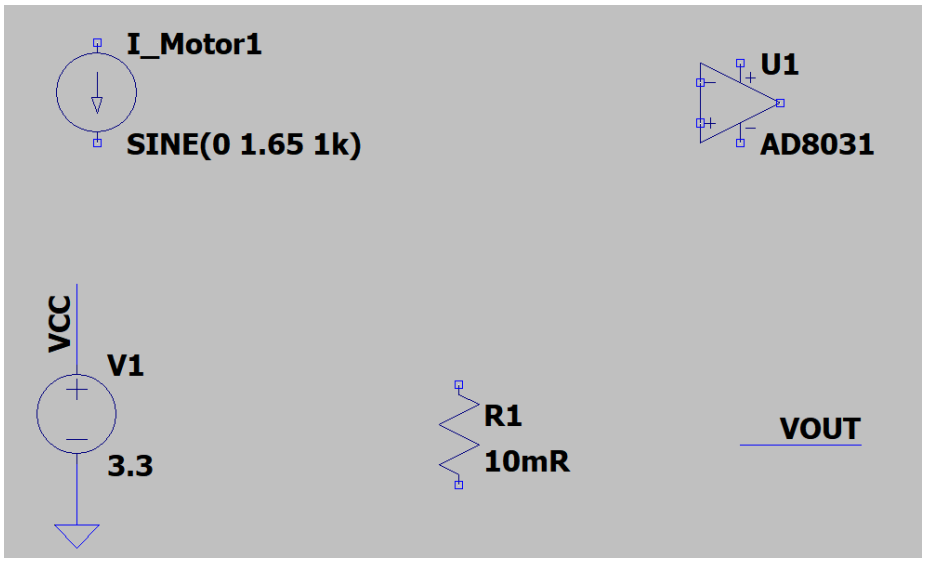


Figure 5 Incomplete task 4 (LTSpice)

Note: The sampling resistor (R1) is 10m Ω, and I_Motor simulates the current (sine wave variation) of one phase of the motor.

Screen Capture: Insert a screen capture of your simulation results with associated schematic shown at the same time into the "CountryCode_WSC2026_TP16_A1_answer_sheet.docx " under the Task4 heading that clearly shows the relationship between sampling current and output voltage.

© WorldSkills International

Task 5: Drive circuit（LTspice）

Two PWM signals（P and N） are passed through a half bridge drive circuit to generate the drive signal for the motor. The input and output relationship is shown in the figure6， the circuit with the following characteristics:

- Supply: 12VDC and 3.3VDC
- Output voltage:VH=12V,VL=0V,
- Dead Time =500ns (+/-5%)
- To simulate use the ADuM4221A as linear circuit instead of UCC21520 .
- Resistors as much as you need (see resistor kit)

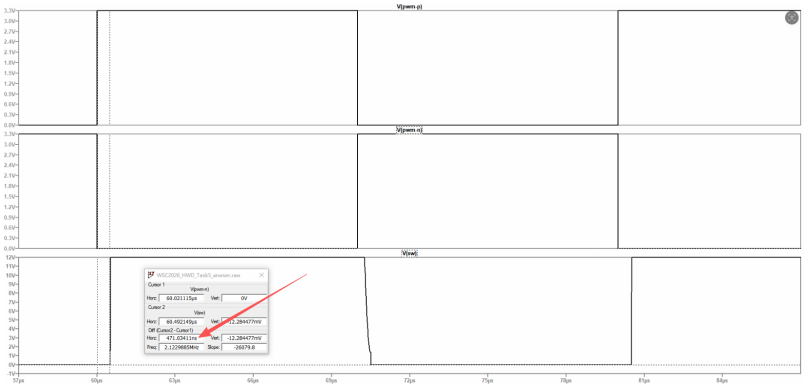


Figure 6 Waveforms of the PWM signal drive

Using the LTSpice schematic
"WSC2026_HWD_Task5_starting_point.asc" (Figure 7) complete the schematic and simulate as requested.

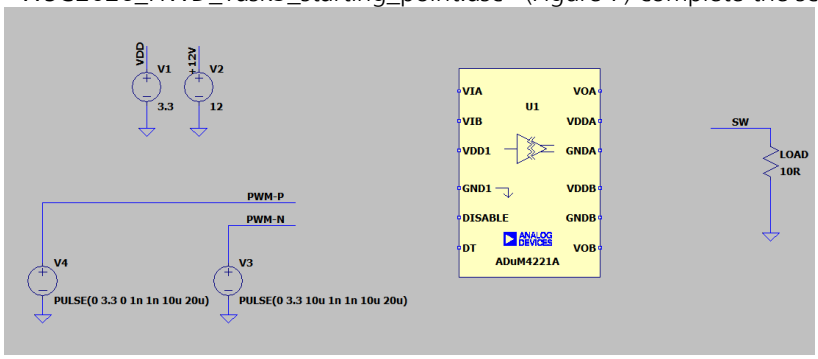


Figure 7 Incomplete task 5 (LTSpice)

Screen Capture: Insert a screen capture of your simulation results with associated schematic shown at the same time into the "CountryCode_WSC2026_TP16_A1_answer_sheet.docx " under the Task5 heading that clearly shows the relationship between Inputs and output signal.

© WorldSkills International

Components List

DESCRIPTION	VALUE	PACKAGE	QUANTITY
2A Step-Down Switching Regulator	LT1076	TO263-5	1
40V, 5A, 2MHz Step-Down Switching Regulator with 3.3uA Quiescent Current	LT3976	MSOP-16	1
80 MHz rail-rail op amp	*GS8591	SOP8	1
Isolated Half Bridge Gate Drivers with Adjustable Dead Time	**UCC21520	SSOP-16	1
N-Channel Power MOSFET	BSB014N04LX3	MG-WDSON-2	2
Schottky	1N5819	SOD-123	2
Schottky	MBRS340	DO-214AB	2
FastFET Op Amps	AD8034	SOIC-8	1
FAST SWITHING DIODES	1N4148	SOD-123	2
Middle Power Transistor	2SAR554P	SOT-89	1
Middle Power Transistor	2SCR554P	SOT-89	1
PNP TRANSISTOR	2N3906	SOT-23	1
NPN TRANSISTOR	2N3904	SOT-23	1
Polarized Capacitor (Radial)	16V/10 uF	4*5.4mm	4
Polarized Capacitor (Radial)	16V/470 uF	6.3*7.7mm	2
Capacitor	1000uF (±5%)		1

* For LTSpice simulation, use AD8031

** For LTSpice simulation, use ADuM4221A

Other components based on WSC2026_HWD_BOM_en.xlsx

© WorldSkills International

Marking Scheme

SUB CRITERION	SUB CRITERION NAME	WSOS
A1	Development of circuits - Phase 1	1 , 3 , 7

© WorldSkills International