Product Overview:

ELab v1 Mini Electronic Laboratory is an integrated experimental platform dedicated to closed-loop testing of electronic components and modules. Its core functions include Bode plot analysis, FFT spectrum analysis, frequency spectrum scanning, voltage scanning, and synchronous online acquisition and debugging with MATLAB, enabling comprehensive and precise performance testing of electronic devices.

This device supports simultaneous reception of multi-channel signals by a single PC and can aggregate data to higher-level systems via the TCP protocol, ensuring efficient integration and centralized management of test data. In terms of compatibility, it seamlessly integrates with mainstream data analysis platforms such as Python and MATLAB, offering users flexible options for data analysis tools.

Whether for principle demonstration in classroom teaching, equipment pre-research in scientific research, or prototype trial production in engineering practice, ELab v1 precisely meets the requirements of diverse scenarios, providing stable and reliable technical support for teaching, R & D, and practical work in the field of electronics.

1. Input Measurement Function

- **Channel Configuration**: Dual - channel single - ended input (CH1, CH2) with DC coupling.
- **Input Impedance**: 300 kΩ ~ 2 MΩ (adjustable), with capacitive reactance ≤ 10 pF.
- **Measurement Range**:
  - Low gear: 0 ~ 10 V (high - precision mode)
  - High gear: 0 ~ 100 V (expanded mode)
- **ADC Accuracy**: 12 - bit resolution to ensure high - precision signal acquisition.
- **Sampling Rate**: 50 kS/s ~ 15 MS/s (programmable adjustment) to meet different bandwidth requirements.

2. Output Function

- **Channel Configuration**: Two independent outputs (OUT1, OUT2), each composed of two sub - channels in parallel.
- **Waveform Generation**:
  - Square wave: amplitude 0 ~ 3 V, frequency 1 kHz ~ 300 kHz, with adjustable duty cycle and phase.
  - Sine wave/triangle wave/custom waveform: amplitude 0 ~ 3 V, frequency 100 Hz ~ 30 kHz.

- **Power Output**:
  - 2.5 V / 5 V DC output: supports closed - loop testing of electronic modules, with a maximum output current of 50 mA for 5 V.
  - OUT1 current measurement: maximum measurable current is 3 mA.

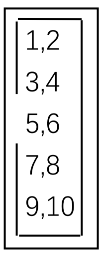
3. Data Transmission and Communication

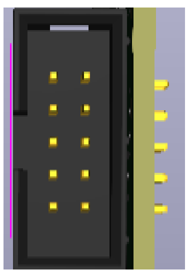
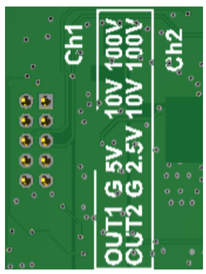
- **USB Interface**:
  - Supports parallel operation of multiple devices.
  - Real - time data is uploaded to the server via the TCP protocol, supporting synchronous access by multiple clients (PCs).
- **Software Support**:
  - Provides Windows & Ubuntu drivers and APIs, compatible with mathematical software such as MATLAB, facilitating secondary development.
  - The server side supports data cascading, aggregation, and advanced analysis.

4. Safety and Limitations

- **Input Protection**:
  - The input voltage is strictly prohibited from exceeding 100 V, otherwise, it may cause permanent damage to the device.
  - It is forbidden to directly connect the device's GND to 110 V/220 V mains to avoid burning the device or PC.
- **Error Description**:
  - The error range is 1% ~ 5%.
- **Trigger Function**:
  - No hardware trigger support, adopting the direct acquisition and display mode.

5. Hardware Wiring Diagram




Side,


Back

- Input terminals: CH1:1,3、CH2:2,4;1: 100V、2:100V、3:10V、4:10V
- Output terminals: OUT1:9; OUT2:10

- Power interface: 5V_OUT:5; 2.5V_OUT:6; GND:7,8
- USB: Micro - USB interface

Application Scenarios

- Electronic circuit debugging, sensor signal acquisition
- Embedded system testing (such as PWM waveform generation)
- Educational experiments, electrochemistry, scientific research data analysis

Program Installation and Startup Guide

1. Program Installation (Portable Version)

Download Verification

1. Download the program package: `qiftech.cn.lab.tar`
2. Verify file integrity by running the following commands in the terminal:

bash
```
# Calculate MD5 checksum
CertUtil -hashfile qiftech.cn.lab.tar MD5
# Calculate SHA1 checksum
CertUtil -hashfile qiftech.cn.lab.tar SHA1
```

3. Compare the output hashes with the official values to ensure file integrity.

Program Startup

- **Windows/Linux**:
  - Extract the package and run `qiftech.cn.lab` directly (no installation required).

   - **Linux Recommendation**: Run with `sudo ./qiftech.cn.lab` to avoid permission issues.

Troubleshooting USB Connection Issues

If the USB connection fails, try:

1. Reconnecting the USB device.
2. Closing and restarting the `qiftech.cn.lab` program.

2. Driver Installation

Linux Systems

- Most Linux distributions include the necessary drivers.
- **Ubuntu Additional Support**:

  Install the `libusb` library by running:

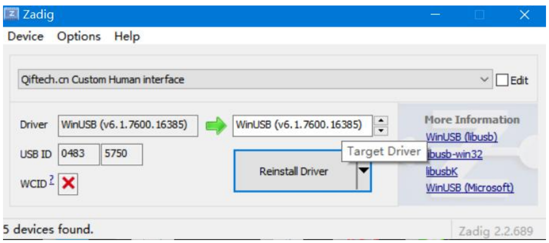
  bash

- `sudo apt-get install libusb*`
-
- **Device Connection Check**:

  After connecting the device, run `sudo lsusb -t` to verify device recognition.

Windows Systems (Win7/10)

1. Use the Zadig tool to install drivers.
2. Select the **WINUSB (recommended)** driver and complete the installation.
3. The system will recognize the device after successful installation.



3. Device Connection Optimization

Direct PC Connection:

- It is recommended to plug the ELab v1 directly into the computer's USB port, avoiding the use of USB hubs.
- Ensure the device exclusively uses the USB high-speed bus (not shared with other high-speed devices).

Power Supply Requirements:

- Typical operating current <500 mA; it is advisable to use a high-quality USB 2.0 cable.
- Supports parallel use of 3 to 5 ELab v1 units, but stable power supply must be ensured.

Setting Saving:

- When exiting the program, the current configuration will be automatically saved and restored upon the next startup.

4. Device Selection and Interface Operation

1. After launching the program, navigate through the menu:
   Tools → Boards → Select "ELab v1 Mini Electronic Laboratory".
2. Once the device connection status is confirmed, you can start using it.

Troubleshooting Common Issues

Unstable USB Connection:

- Replace the USB port or cable.
- Turn off other devices that occupy the high-speed bus (such as cameras, capture cards).

Driver Installation Failure:

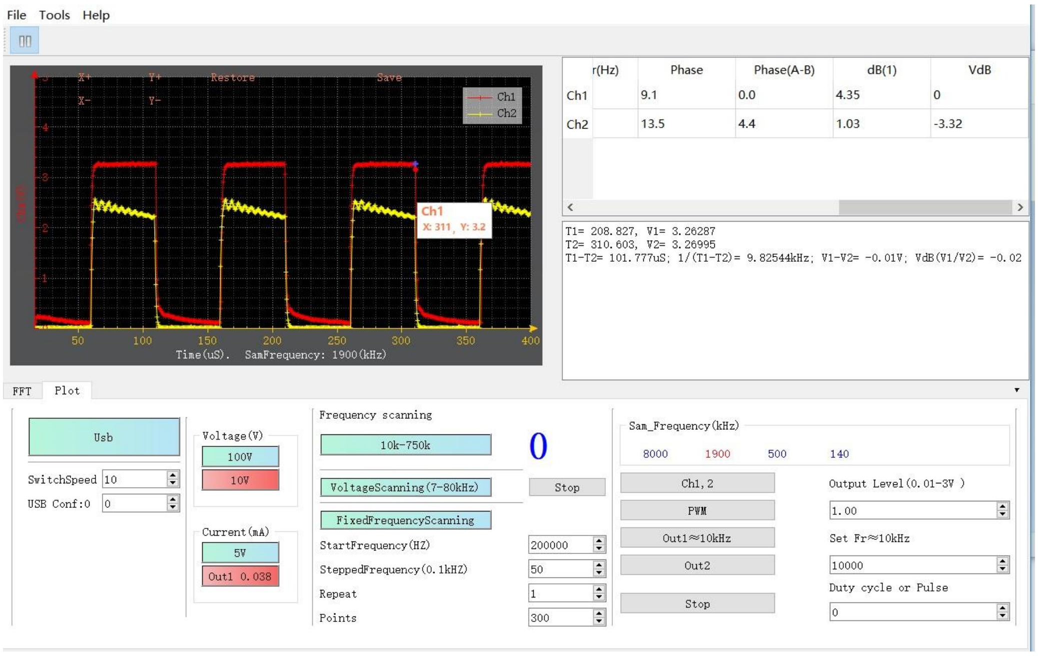
- On Windows, manually update the driver in Device Manager and select WINUSB.
- Linux users can try reloading the libusb module:

  Bash

```
sudo modprobe usbcore
```

Quick Start Guide

1. **Hardware Connection**
   Connect the CH1 and CH2 interfaces of the two probes to the inductor respectively, and then connect them together to the output terminal OUT1.
2. **Waveform Setting**
   Enter the PWM waveform setting interface, input the parameter of 10000Hz, and the device will automatically generate and display the square wave trajectory.
3. **Frequency Measurement**
   Click the pause button to freeze the waveform, then click any two points on the square wave trajectory, and the system will automatically calculate and display the frequency value between the two points.



Voltage Measurement

- **Single-ended voltage measurement**: Supports simultaneous measurement of single-ended voltages between CH1, CH2 channels and GND.
- **Data display**: Can show channel name, voltage range, frequency spectrum range, peak-to-peak value, DC voltage component, AC voltage component, decibel value, phase and phase difference, dB and dB difference. It also supports calculation of CH1-CH2 difference, CH2/CH1 ratio, and Abs (absolute signal value).

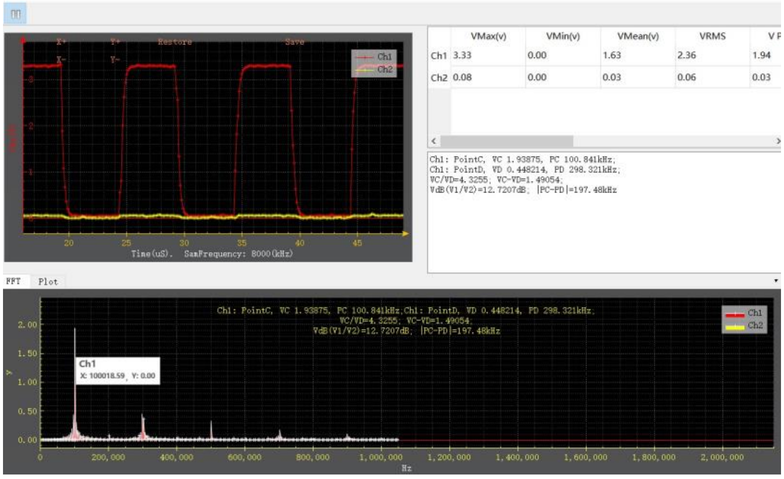
Current Measurement

- **5V output current measurement**: Designed specifically for testing 5V electronic products, with a maximum current supply of 50mA.
- **OUT1 output current measurement**: Suitable for testing low-power devices, with a maximum current supply of 3mA.

FFT Spectrum Analysis

- **Nyquist frequency prompt**: The sampling frequency must satisfy the Nyquist sampling theorem (sampling frequency ⩾ 2 times the highest signal frequency). Otherwise, signals higher than the Nyquist frequency will cause aliasing, resulting in high-frequency components being mirrored to the low-frequency region.

- **Spectrum calculation operation**: After clicking pause, double-click any two points on the spectrum diagram with the mouse, and the system will automatically calculate and display the FFT spectrum-related values between the two points.



Frequency Sweep Function Description

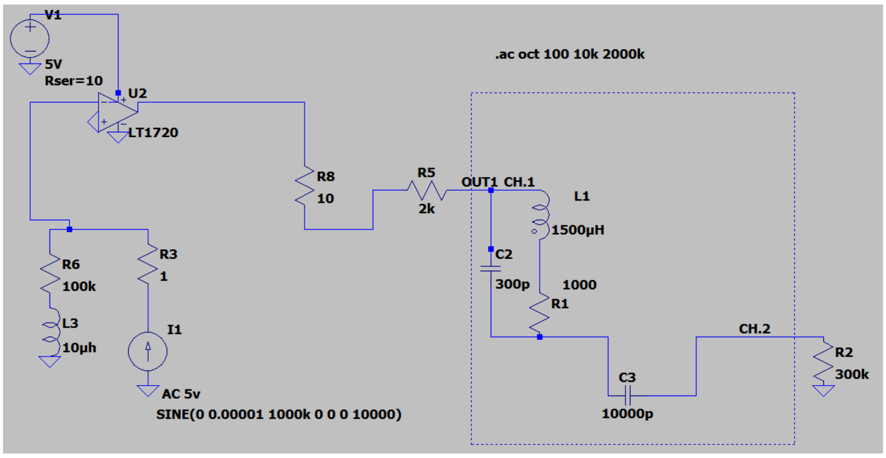
Frequency Range Setting

The frequency sweep range is 1kHz-1.5MHz, and each sampling channel can be set independently. The following is an operation guide for frequency sweep of LC resonant circuits:

LC Frequency Sweep Configuration

1. **Hardware Connection Reference**: Please refer to the LTspice simulation schematic for wiring. The following key components need to be added:
    - L1: Three-pin boost inductor (1.5mH, I-shaped inductor)
    - C3: 10nF capacitor
2. **Device Connection**:
    - CH1: Connect to the signal input terminal of the circuit under test
    - CH2: Connect to the signal output terminal of the circuit under test
    - OUT1: Connect to the excitation source input terminal of the circuit under test

Note: Components not marked in the schematic are parasitic parameters of the circuit and do not need to be added separately during actual testing.



Sampling Frequency Setting

- Set the sampling frequency of CH1 and CH2 to 8MHz (to ensure compliance with the Nyquist sampling theorem).

Sweep Range Setting

- **Start Frequency**: Enter 100kHz
- **Stepped Frequency**: Enter 5kHz (i.e., 50 times 0.1kHz)

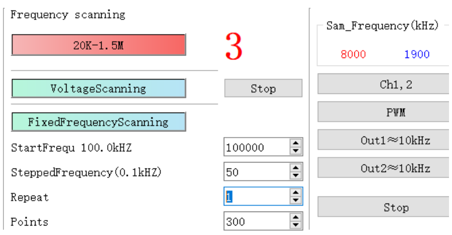
Sweep Accuracy Setting

- **Repeat**: Set to 1 (increasing this value can improve result accuracy but will reduce sweep speed)
- **Points**: It is recommended to set 100-300 points

Startup and Result Viewing

After completing the settings, click the "Start" button. Once the sweep is finished, you can view the frequency sweep results by opening "Bode Plot DEMO2".

.



Bode Plot Operation Guide

Basic Operations

- **Data Viewing**: After completing the frequency sweep, open the Bode plot interface. Click any point on the curve to view the corresponding X-axis (frequency) and Y-axis (amplitude/phase) values.

Demonstration Cases

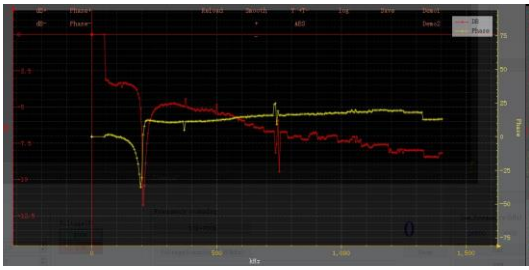
- **DEMO1**: Frequency sweep results of a series circuit consisting of a 68mH boost inductor and a 10nF capacitor.
- **DEMO2**: Frequency sweep results of a series circuit consisting of a 1.5mH boost inductor and a 10nF capacitor (this case is recommended for observing resonance phenomena).

Function Description

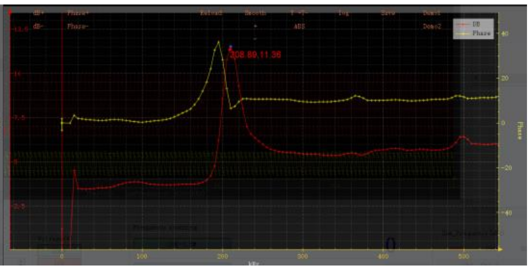
- **Log Mode**: Switch to logarithmic coordinate display.
- **ABS (Absolute Value)**: Display the absolute value of the signal.
- **Smooth**: Smooth the curve using algorithms to enhance visualization (Note: Excessive smoothing may cause distortion of key data).

Operation Recommendations

1. Open the DEMO2 case.
2. Apply the smoothing function moderately to optimize the curve display.
3. Observe the amplitude peak and phase mutation characteristics at the resonant frequency point (around 200kHz).



Real data



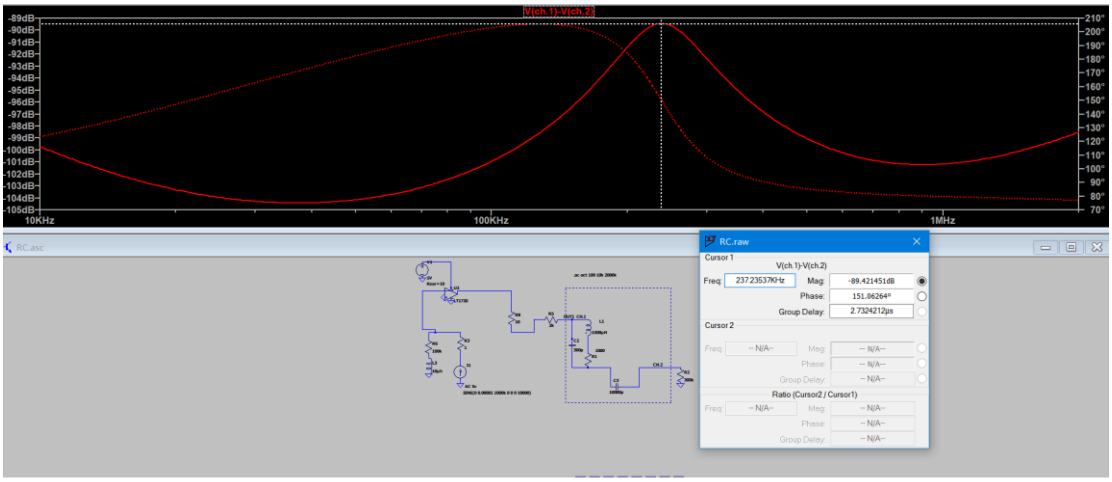
smoothing function, etc

Comparison Analysis of LTspice Simulation and Actual Measurement

A comparison between the LTspice simulation results and the actual measurement data of DEMO2 shows the following:

- **Resonant frequency consistency**: Both show an obvious resonant peak around 200kHz, with a high degree of matching between the theoretical calculation and the actual measurement results.
- **Curve shape difference**: The LTspice simulation curve is smoother, while the actual measurement curve has slight fluctuations due to noise interference. The display effect can be optimized through the device's "Smooth" function.
- **Quality factor (Q value)**: The actual measured Q value is slightly lower than the simulated value, reflecting that the actual circuit has additional losses (such as parasitic resistance).

Conclusion: The device measurement results are highly consistent with the LTspice simulation, verifying the accuracy of the circuit design.



Voltage Sweep Function Description

Basic Configuration

- **Current Selection**: Supports switching measurement modes, allowing selection of 5V output current or OUT1 output current.
- **Core Principle**: Voltage regulation is achieved by adjusting the PWM duty cycle, and a capacitor is required to complete the voltage sweep process.

Diode Sweep Experiment Configuration

1. Component Preparation (Additional Components Required)

- C1: Electrolytic capacitor
- U2: 9013 transistor
- U4: IN4007 diode

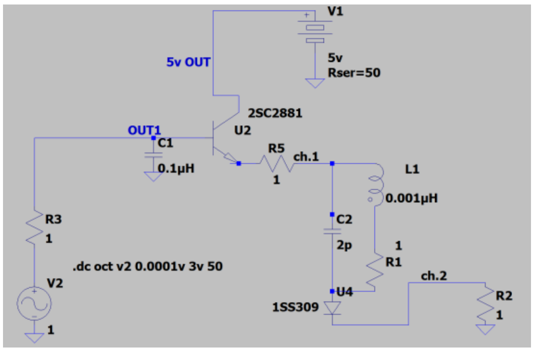
(Note: Other unmarked components in the circuit are parasitic parameters and do not need to be additionally connected.)

2. Hardware Connection

Complete the wiring according to the diagram, with specific interface correspondences as follows:

- 5V output terminal
- CH1, CH2 signal measurement terminals
- OUT1 signal output terminal
- GND (ground) terminal

After connection, the voltage sweep can be started. By monitoring the relationship between PWM duty cycle changes and voltage response, the analysis of the diode circuit characteristics can be realized.



Guide for Voltage Sweep Parameter Configuration

1. Sampling Frequency Setting

- Set the sampling frequency of CH1 and CH2 to "NO2 gear" (a preset gear suitable for the current experiment).

2. Current Mode Selection

- **Current(mA)**: Select the "5V" mode (corresponding to 5V output current measurement).

3. Sweep Range Setting

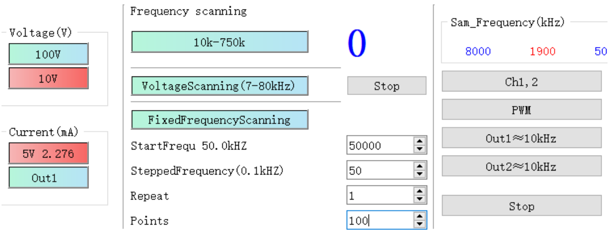
- **Start Frequency**: Enter 50kHz
- **Stepped Frequency**: No need to set in voltage sweep mode; the system will automatically ignore this parameter.

4. Sweep Accuracy Setting

- **Repeat**: Set to 1 (increasing this value can improve result accuracy but will reduce sweep speed)
- **Points**: Set to 100 points

5. Startup and Result Viewing

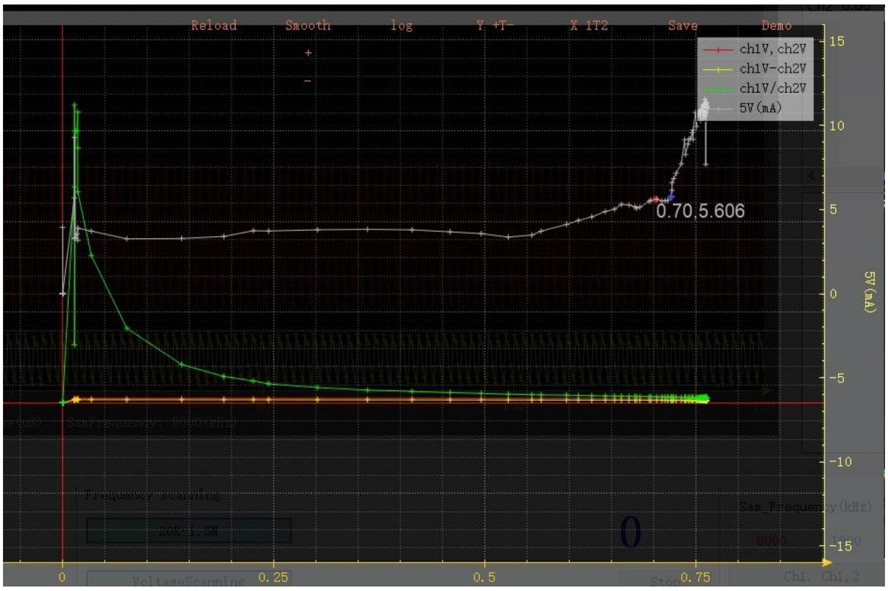
After completing the settings, click the "Start" button. Once the sweep is finished, open the "Voltage Sweep Graph" to view the results. The DEMO example shows typical results of frequency-voltage sweep.



After opening the voltage sweep result DEMO, you can adjust the view according to the following steps:

1. Click the menu "X1T2" to flip the XY coordinates;
2. Select "Y+T-" to flip the positive and negative directions of the Y-axis;
3. Enable the "LOG (logarithmic)" mode to obtain the target view (as shown in the figure below).

Through coordinate transformation and display mode adjustment, the variation law of voltage with sweep parameters can be observed more clearly, which facilitates the analysis of the voltage response characteristics of the circuit.

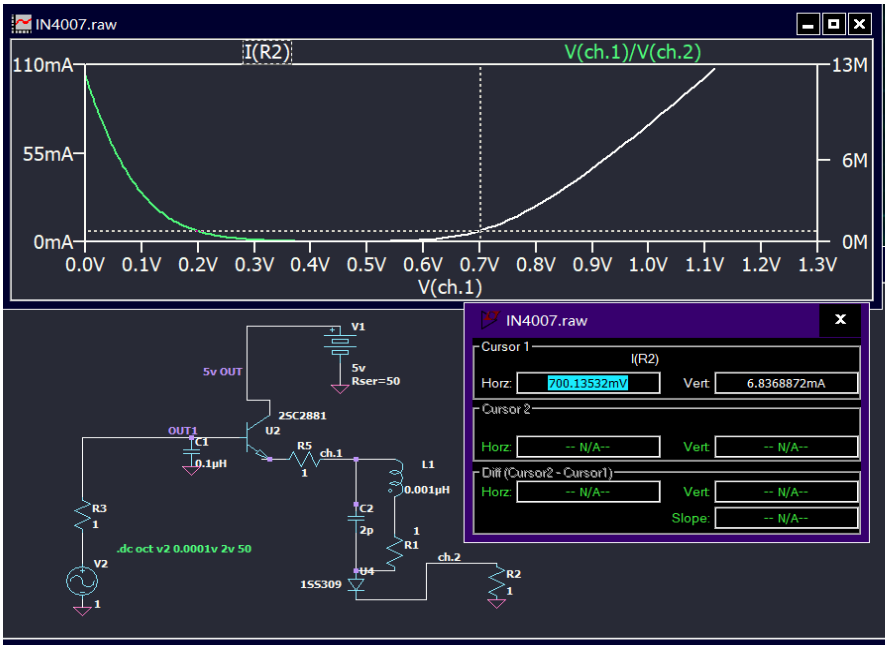


Comparative Analysis of LTspice Simulation and Actual Measurement Results

By comparing the actual measurement data with the LTspice simulation results, the corresponding relationships and characteristics of key parameters are as follows:

- **Voltage ratio correspondence**: The CH1/CH2 ratio in actual measurements corresponds to the calculated value of V(CH1)/V(CH2) in the simulated circuit.
- **Current parameter correspondence**: The actual measured 5V output current (in mA) corresponds to the current I(R2) flowing through resistor R2 in the simulated circuit.

The comparison shows that the two are highly consistent at the key inflection points of the diode characteristic curve: when the voltage across the diode reaches the conduction threshold of 0.7V, the current shows an obvious upward trend, and the current value at the inflection point stabilizes in the range of 5-6mA. This verifies the accuracy of the actual measurement data and the reliability of the circuit model.



Operation Guide for Fixed-Frequency Sweep

1. Function Description

Fixed-frequency sweep is used for current measurement under conditions of fixed frequency and duty cycle, supporting the selection of 5V output current or OUT1 output current.

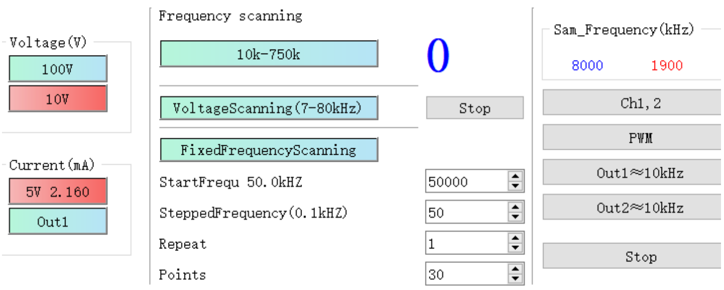
2. Parameter Settings

- **Sampling Frequency**: Select "NO2 gear" for the sampling frequency of CH1 and CH2.
- **Frequency**: Set to 50kHz (keep constant).
- **Number of Sampling Points**: Set to 30 points.

3. Operation Steps

- After completing the parameter settings, click the "Start" button.
- After the sweep is completed, open the "Voltage Sweep Graph" to view the results.
- Click the "SAVE" button to save the current results.

This mode is suitable for analyzing circuit characteristics at specific frequency points, such as measuring resonance point current, verifying specific frequency responses, and other scenarios.



Custom Waveform Function Description

1. Input Specifications

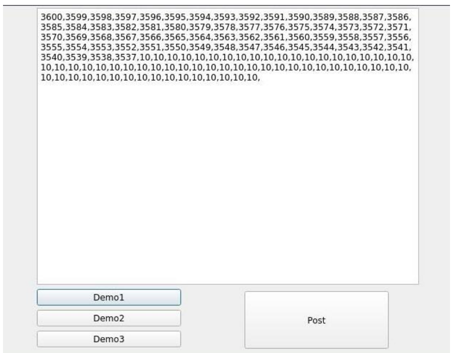
- An array with a length of 127 must be input.
- Each element in the array ranges from 0 to 4095 (12-bit precision).

2. Function Applications

By customizing the array values, the following functions can be achieved:

- Adjust the amplitude of the output signal.
- Generate arbitrary waveforms (square waves, triangular waves, sawtooth waves, etc.).
- Design complex signals with specific frequencies/phases.

After inputting the array into the system, the output effect can be verified through the waveform preview function. It is suitable for scenarios such as signal processing experiments and device driver testing.



Output Function Description

Output Channels and Waveform Types

OUT1 and OUT2 support the output of PWM waves, sine waves, and triangular waves, with specific modes as follows:

- **PWM-PWM3 Mode**: OUT1 and OUT2 are in parallel output.
- **PWM4-PWM5 Mode**: OUT1 and OUT2 are in series output.

Square Wave Output Characteristics

- **Parameter Range**: 3V output amplitude, frequency 1kHz-0.3MHz, supporting duty cycle and phase adjustment.
- **Adjustment Method**: After switching the PWM mode, parameter configuration is realized by adjusting "Duty cycle" or "Pulse" (pulse parameters).

Detailed Mode Description

1. PWM Mode

- **Phase Difference**: The phase difference between OUT1 and OUT2 is 0°.
- **Duty Cycle**: OUT1 is fixed at 1/2, and OUT2 can be adjusted flexibly.

2. PWM1 Mode

- **Phase Difference**: The phase difference between OUT1 and OUT2 is 0°.
- **Duty Cycle**: Both channels support independent adjustment.

3. PWM2 Mode

- **Phase Difference**: The phase difference between OUT1 and OUT2 is 90°.
- **Duty Cycle**: Both channels support independent adjustment.

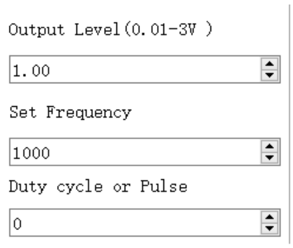
4. PWM3 Mode

- **Phase Difference**: The phase difference between OUT1 and OUT2 is 0° (phase can be adjusted).
- **Duty Cycle**: Fixed at 1/2.
- **Other Waveforms**: Supports 0V-3V output, frequency 100Hz-30kHz, including sine waves, triangular waves, and custom waveforms (output amplitude is adjusted via "Output Level").

5. PWM4 and PWM5 Modes

- **Channel Characteristics**: Single channel independent output (OUT1 corresponds to PWM4, OUT2 corresponds to PWM5).
- **Parameter Adjustment**: Frequencies are set independently, and duty cycles can be adjusted flexibly.

Through multi-mode switching and parameter configuration, it can meet diverse needs such as complex circuit testing, waveform synthesis, and phase characteristic analysis.



Guide to Data Export and MATLAB Collaboration

Data Export Function

- **Text Export**: The text window on the right supports exporting measurement samples as TXT format files, which can be directly used in software such as MATLAB and Excel for subsequent data processing and analysis.

Steps for Collaboration with MATLAB

1. Hardware Connection

- **Device Connection**: Connect the OUT1 output terminal to the CH1 channel, and the OUT2 output terminal to the CH2 channel.

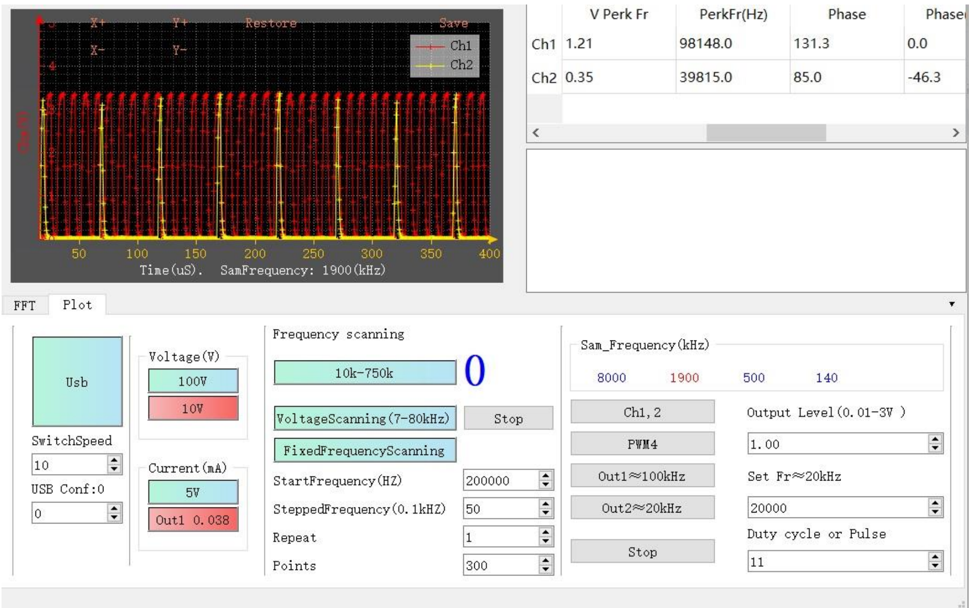
2. Parameter Configuration

- Set the CH1 signal as a 100kHz pulse (simulating a clock signal).
- Set the CH2 signal as a 20kHz pulse, and modify the pulse width parameter so that a trigger signal is generated at regular intervals (to realize the synchronous linkage between the clock and the trigger).

3. Data Acquisition and Processing

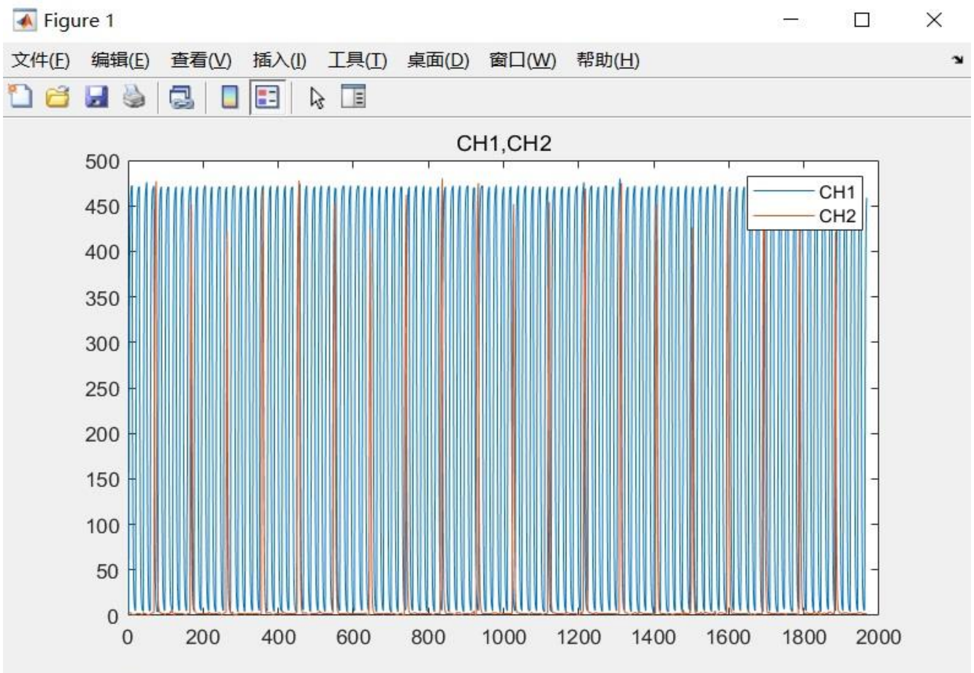
- After starting the device to collect data, export the file in TXT format.
- Import the data into MATLAB. Through writing scripts, in-depth processing such as timing analysis of clock signals and trigger signals, pulse interval calculation, and synchronization verification can be realized.

This solution is suitable for experimental scenarios such as timing logic verification and trigger mechanism simulation, providing a convenient software-hardware collaborative platform for digital circuit design and signal synchronization research.



Open qiftech.cn.yanan.server.exe

Open MATLAB online free version, you can online joint.



test.m

```
url = 'http://127.0.0.1:8010/';
data= webread(url);
datanum=split(data,",")
datanum=datanum(strlength(datanum)>0)
datanum=str2double(datanum)
if (datanum(7)==1)
    plot(datanum(127:2:end-64));%CH1
    hold on
    plot(datanum(128:2:end-63));%CH2
    hold on
    title('CH1,CH2')
    legend('CH1','CH2');
    hold off
end
```