

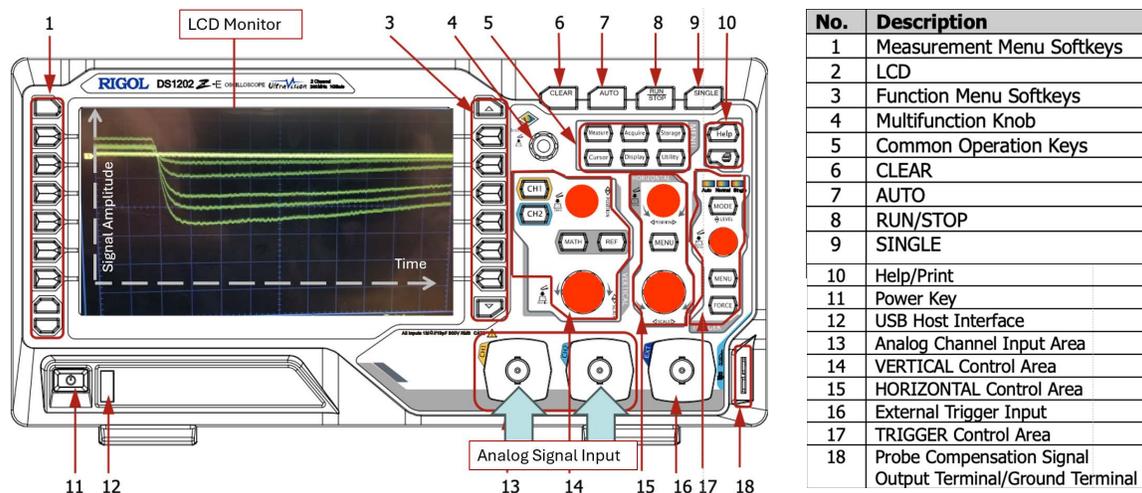
ANSEL Experiment Analog & Digital Electronics

Experimental Tasks (2 Periods)

- I. **Digital oscilloscope, practice basic operations.**
- II. **Analog electronics, practice spectroscopic applications.**
- III. **Run the Data Acquisition, pulser calibration.**
- IV. **Digital electronics, cables and terminators, set up logic operations.**

I. The digital oscilloscope

1. Ascertain that the Teaching Associate (TA) has set up for your use an analog (spectroscopy) pulse generator ("pulser"), for example, an ORTEC model 419 pulser with 50- Ω BNC cable(s) to use with scope operations. **Assign 2 identical cables (same type, same length) as your scope test cables.**



2. Familiarize yourself with the layout of the front panel of the oscilloscope ("scope"). The scope has a display area with variable function buttons and a control area. Properly set up, the display screen will image signal amplitude (vertical ordinate) vs. time (horizontal, abscissa). It will also display numerical values for vertical (V) and horizontal sensitivity (s) per scale division. The control area contains BNC type analog signal inputs, as well as control knobs for vertical and horizontal position and sensitivity, separately for each of the color coded scope "channels." Note the input resistance (50 Ohms or Meg Ohms). Menu and function buttons are used to select the desired modes of operation and display. The knob "Trigger" (Level) is used to control the minimum signal amplitude displayed by the scope. In operation, the value of the trigger level is indicated by an arrow on the right side of the display area and by its numerical value on the bottom of the screen.

If at any time during the lab the scope appears to be malfunctioning, press the "Auto" button, which will reestablish scope default settings.

3. **Without cables connected** to any of the scope inputs, switch on the scope power by pressing a button on the front panel. The monitor screen will show the previous or default setup. It may display one or several traces as colored horizontal lines.
4. Press the various menu buttons (Trigger and Channel menus) and note the available options. **Auto** triggering occurs with reference to the 60Hz electrical power cycle.
5. Select one channel (Ch 1, yellow) and deselect the other channels by pressing the corresponding menu buttons. Press the Trigger Menu button for Ch 1 and select ground (GND) for Ch 1 coupling. Trace 1 should appear vertically centered. It may indicate some electronic noise fluctuations.
6. Move the trace across the screen by operating horizontal and vertical position controls. Change horizontal and vertical scales and note the values displayed on the screen for sensitivities and offsets.
7. Adjust the scope trace to the left and vertical center of the screen. Choose 1 V/division and 1 μ s/division for vertical and horizontal sensitivities, respectively. For the trigger level choose DC coupling, positive signal slope, and 1 M Ω for the Ch1 input coupling.
8. **Using a BNC cable**, connect the pulser (direct) output to Ch 1/AC input with and observe the signal trace on the scope screen.
9. Determine presence or absence of a DC level on the pulser output by changing the trigger mode between AC to DC. Measure the signal amplitude, as well as rise and decay times of the pulser signal.
10. With BNC cables connect the direct output of the pulser to scope Ch 1 and the attenuated output to Ch 2 display both Ch 1 and Ch 2 traces on the screen.
11. From the trigger menu select trigger source Ch 1.
12. Change amplitude and decay times of the pulser signals and observe the corresponding changes of the signal images on the scope screen.
13. Calibrate approximately the pulser control dial and attenuator switches in terms of the voltage indicated on the scope screen.

This completes the scope tests. Following should be tests of a pulse shaping with main amplifiers. Document all control settings of all modules used.

II. Analog electronics, use a spectroscopic signal amplifier and single-channel analyzer.

1. Produce a low-amplitude pulser signal from the attenuated output. Record signal pulse shape (rise and decay times) and amplitude. Insert this pulser signal into an ORTEC 571 or 572 Spectroscopy Amplifier (or similar). Set the amplifier to lowest coarse and fine gains. Select the appropriate input polarity and set the amplifier base line restorer toggle switch (BLR) to automatic (AUTO). Select No Delay on the toggle switch at the unipolar output.
2. Select a 3- μ s shaping time (to be varied later).
3. View the unipolar output of the amplifier on the scope.
4. Set the pulser signal amplitude such that the amplified (unipolar) signal is of the order of 1 V.
5. With the scope (DC, 1M Ω) check for a DC level on the amplifier output. If necessary, correct it to zero level by turning the DC screw driver potentiometer on the front panel.
6. On the scope inspect the trailing slope of the amplifier unipolar output signal for under or overshoot. Observe the response of the signal shape to turns of the pole-zero adjust screw driver potentiometer (PZ). Optimize for fastest signal return to base line with no crossing.
7. Observe (and record) the response of the amplifier output signal to different shaping time constants selected with the Shaping Time selector varied from 0.5-10 μ s.
8. On the scope channels 1 and 2, triggered on Ch 1, inspect the bipolar amplifier output signals and their relation to the unipolar signals. Note that the unipolar signal can be delayed by a preset time, depending on the position of the toggle switch DELAY.
9. Set up an ORTEC 551 Timing Single Channel Analyzer (TSCA) or 590A amplifier-plus-TSCA in integral mode, to trigger on amplifier output signals of minimum amplitude ("threshold").
10. Feed the bipolar amp output into the TSCA input. Display the TSCA output signal on scope Ch 1 (trigger on Ch1, note polarity). Display the delayed unipolar amplifier output signal on scope trace Ch 2. An amp signal will appear on CH 2 only when a corresponding TSCA signal has triggered Ch 1.
11. Varying the lower threshold of the TSCA will restrict the pulse height amplitude of signals that are displayed on the coincidence trace Ch 2.
12. Demonstrate similarly the function of a TSCA in windows (E-DE) mode.

This completes the analog electronics tests. Following should be a measurement of a pulse generator output signal using the DDC-8 based data acquisition module. Document well in your log book all control settings used by all modules used.

III. Run the Data Acquisition, perform a DDC-8 pulser calibration.

1. Confirm with a TA that you are producing proper Signal and Trigger from the amplifier and TSCA output.
2. Follow the procedure in the *DDC-8/Blackbox Quickstart User's Guide* to initialize hardware and software for measuring pulser signals in histogram mode.
3. Set the pulser output to DDC-8 mid acceptance range (+1 V amplitude) using pulser attenuators and fine adjust knob. Check signal amplitude with scope.
4. Connect the pulser output to DDC-8 channel CH_0 and the trigger signal to NIM_IN_0.
5. Start the measurement and let signals be accumulated for a few minutes.
6. Stop the measurement and view the acquired data on the computer monitor.
7. Measure the linearity of pulser and DDC-8 by acquiring pulser signals for 10 different amplitudes, each checked on the scope and acquired for 1-2 minutes ("pulser fence"). Start and stop each measurement by connecting and disconnecting the trigger signal to and from the DDC-8.
8. Determine the range of the DDC-8 analyzer which ideally should be 0-2 V. Plot signal amplitude in volts vs. DDC-8 channel number (bin #).

This completes the DAQ test runs with a pulse generator. Following should be tests with NIM logics and cables.

IV. Generate and process digital signals, set up logical requirements. Before further processing check signals on scope. Note that Delay-Gate Generators have the functions of delaying a signal and/or converting it to a wide "gate" signal.

1. Confirm that you have two identical scope cables.
2. Feed an amplified pulser signal into a discriminator (TSCA) and its output to a logical/linear FAN IN/FAN OUT (FIFO) module.
3. Inspect the various FIFO NIM output signals on the scope with **DC** and **50Ω** trigger mode. Note the signal levels.
4. Observe changes in scope response when changing the trigger mode, first to **AC**, then to **1MΩ**.
5. Observe signal reflections on open cables by connecting an additional, loose cable with one end and a BNC/LEMO Tee to the scope input.
6. Terminate the loose cable with a 50Ω terminator and a barrel.
7. Connect one FIFO NIM output signal to scope Ch 1 (AC, 50Ω). Trigger on Ch 1. Feed another FIFO NIM output signal to a (ORTEC or LeCroy) Gate/Delay-Gate Generator (GG). Observe the GG output signal(s) on Ch 2 of the scope.
8. Connect one FIFO NIM output signal to a Gate Generator (GG) to produce a 100-200 ns wide "gate" signal. Connect the other to a Delay Generator (DG) to produce a delayed signal.

9. Observe the two signals on scope channels 1 and 2, respectively, triggering with the gate pulse.
10. Insert the two signals into inputs of a coincidence unit. Select two-fold coincidence mode. Observe the coincidence unit output on the scope.
11. Varying the delay of the delayed signal, measure the coincidence resolution time. Compare it to the sum of the widths of gate and delay pulses.
12. Insert the wide gate pulse into the Veto input of the coincidence unit. Select single-fold (or) mode of the unit. Observe the coincidence unit output on the scope.
13. Varying the delay of the delayed signal, measure the anticoincidence efficiency and resolution time. Compare the latter to the coincidence resolution time.