

Testing Switch Mode Power Supplies

Using the Infiniium EXR-Series Oscilloscope

Switch mode power supplies (SMPS) have surpassed linear power supplies in efficiency and functionality, as well as their convenient size. Mid- and high-end oscilloscopes have optional power analysis software that can help you test and characterize your power supplies faster and more thoroughly. Many Keysight oscilloscopes offer a broad range of power related measurements with the optional power measurements application.

This application note will show you how to test switch mode power supplies with the Infiniium EXR-Series oscilloscope. More specifically, it will cover the fundamental measurements of input, switching, and output analysis performed by this oscilloscope using the optional power analysis software (P/N: D9010PWRA).

Table of Contents

Required Equipment	3
Probing tips	4
AC Input Analysis	5
Power quality analysis.....	6
Current harmonics.....	7
Inrush current	9
Switching Analysis	10
Switching losses.....	11
Effective resistance	13
Slew rate	14
Output Analysis	15
Output ripple.....	15
Turn-on and turn-off time	16
Transient response analysis	18
Efficiency	20
Conclusion.....	23

Required Equipment

The following list constitutes an appropriate set of hardware, software, probes, and accessories that you will need for SMPS testing:

- Keysight EXR054A or EXR058A Oscilloscope
- D9010PWRA Power Supply Test Software
- U1880A Deskew Fixture with USB cable for power
- N2790A high-voltage differential active probe
- 1147B (up to 50 MHz/15A), N2893A (up to 100 MHz/15A), or N7026A (up to 150 MHz) AC/DC current probe
- 10:1 passive voltage probe (e.g. N2873A)
- Your switch mode power supply (SMPS)



Figure 1. An 8-channel Infiniium EXR-Series Oscilloscope



Figure 2. Some of the probes required to perform switch mode power supply analysis (Left: Keysight's N2790A 100 MHz high-voltage differential active probe. Right Keysight's 1147B 50 MHz, 15-A AC/DC current probe).

Probing tips

Depending on the voltage levels you are working with, you may need to use a high-voltage differential active probe. Be sure to select the correct probe for your signals. Note that the N2790A high-voltage differential probe can be used with a 50:1 attenuation setting ($\pm 140\text{V}$ max) or with a 500:1 attenuation setting ($\pm 1400\text{V}$ max). However, the scope will only detect a 50:1 setting. If you use the 500:1 attenuation setting for higher voltages, you must manually enter this attenuation factor in the scope's probe menu for accurate voltage measurements. (For correct measurements, you must match the oscilloscope's probe attenuation factor settings with the attenuation factors of the probes being used.)

When using your current probe, do not forget to de-magnify (degauss) it by using the DEMAG button near the base of the probe after disconnecting it from the device under test (DUT). Also, if using the 1147B current probe, you can zero the offset by rotating the ZERO ADJ thumbwheel until the baseline current waveform aligns with the ground indicator on the scope's display with the probe disconnected. If using the N2893A or N7026A current probe, offset error can be calibrated out automatically within the scope's probe menu.

Timing skew between voltage and current probes can induce significant errors and distortion in your power measurements, especially switching loss measurements. Deskewing is critically important since a small offset in the timing of the voltage and current traces can cause a large error in the instantaneous power reading. To deskew a pair of probes, drive both voltage and current probes with the same pulse signal using the U1880A deskew fixture. With a single click on the menu, deskewing is automatically performed and its values are saved in the power measurement software.

Once it is saved, next time you launch the power measurement application, you can use the saved deskew values or perform the deskewing again. Thus, in order to get accurate measurements, do not forget to de-skew your current and voltage probes. Here's how, using the U1880A Deskew Fixture:

1. Connect the U1880A's USB cable between the Deskew Fixture and one of the USB ports on the back of the oscilloscope.
2. Select the **Analyze menu**, select **Power Analysis**, and then open the **Setup Wizard** at the top of the window.
3. Select **Deskew Cal** to proceed with the calibration.
4. Set the S1 switch on the deskew fixture to the "Small Loop" selection, which is to the right on the fixture.
5. Connect the N2790A high-voltage differential active probe (using the 50:1 attenuation setting) between the scope's channel-1 input to J6 (red lead to red test point) and J7 (black lead to black test point) on the U1880A Deskew Fixture as illustrated in Figure 3.
6. Connect the current probe from the scope's channel 2 input to the "Small Loop" in the direction indicated on the Deskew Fixture, with the current flow towards the top of the board.
7. Select **Calibrate**.

For each of these sections, we will assume channel 1 for voltage and channel 2 for current. Please use this as a reference, and feel free to use the most convenient channels in your test setup.

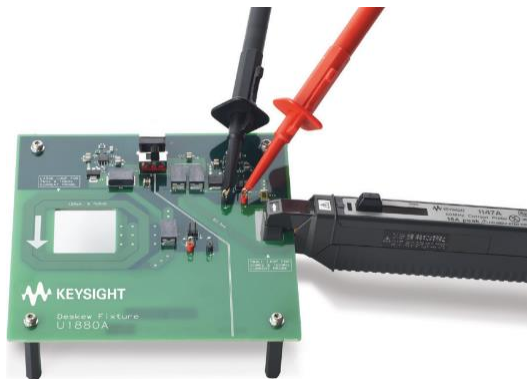


Figure 3. Keysight's U1880A Power Measurement Deskew Fixture

AC Input Analysis

AC input analysis is relevant to the power quality of the input AC signal, current harmonics, and inrush current. However, for each of the three input analysis measurements covered below, you need to set up the following for the best results:

1. Connect the N2790A voltage probe between the scope's channel 1 input to Line (red lead) and Neutral (back lead).
2. Connect the current probe between the scope's channel 2 input to a wire-loop of the input Line signal. You may need to create your own wire current loops on your DUT.
3. Before performing any measurements, ensure that Voltage is set to channel 1 and current is set to channel 2 in the Signals section as shown in Figure 4.

Now you are ready to perform power quality measurements, current harmonics, and inrush current on the input AC signal of your power supply.

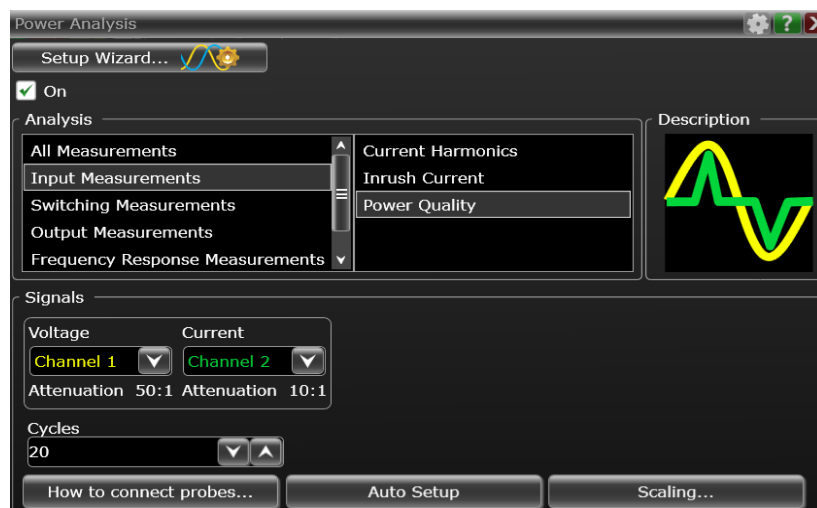


Figure 4. Power quality setup

Power quality analysis

Real power, apparent power, reactive power, power factor, crest factors, and phase angle are measurements that represent signal quality and enable continuous monitoring, early excursion detection, root cause analysis, and timely corrective actions that improve overall AC grid reliability. Assuming you have set up probes in steps 1 – 3 above, you are ready to perform a power quality measurement on the AC input:

1. Press the **Default Setup** front panel key so you start from the same reference point as these instructions. (The On button at the top of the menu defaults to Off. Although you can select On at any time, it is more efficient to select the particular power measurement, set it up (select channels, settings, Auto Setup, etc.), and then select On).
2. Select the **Analyze menu** and then select **Power Analysis**.
3. Select **Input Measurements** and then select the **Power Quality** measurement. You should have a window that looks like the Power Analysis window in Figure 4.
4. The number of cycles in the Power Analysis window is what your measurements will be based on.
5. Select **Auto Setup**, which provides optimum scaling.

The Infiniium EXR-Series scope will automatically set up vertical and horizontal scaling for you, turn on the power waveform ($V \times I$), and then measure all power quality parameters as shown in Figure 5. The power waveform is generated using the scope's "waveform math" ($P=V \times I$). If the power quality measurement is turned on, the EXR oscilloscope will automatically provide measurements of real power (W), apparent power (VA), reactive power (VAR), power factor, crest factors for voltage and current, RMS voltage, phase angle, and frequency.



Figure 5. Input power quality measurements. Here, the yellow sinusoidal waveform is voltage, the green pulsed waveform is current, and the purple is the power waveform, which uses the scope's waveform math ($P=V \times I$).

Current harmonics

Current harmonics refer to the amplitude of frequency components that can be injected back into the AC lines that can potentially disturb other equipment on the same power grid/network. Therefore, harmonics should be measured to ensure compliance with IEC standards. To measure current harmonics, the oscilloscope performs FFT measurements of the current waveform and then creates a table of current measurements in IRMS up to the 40th harmonic. The scope then compares amplitudes of each harmonic against published IEC specification limits along with pass/fail results.

1. Press the **Default Setup** front panel key so you start from the same reference point as these instructions. (The On button at the top of the menu defaults to Off. Although you can select On at any time, it is more efficient to select the particular power measurement, set it up (select channels, settings, Auto Setup, etc.), and then select On).
2. Select the **Analyze** menu, and then select **Power Analysis**.
3. Select **Input Measurements**, and then select the **Current Harmonics** measurement.



Figure 6. Current harmonics measurements in a tabular format

4. Select **Auto Setup**. If you use the default settings, the scope will display 20 cycles of the input line voltage (yellow trace) and current (green trace) waveforms on the scope's display. Autosetup will optimally scale the vertical and horizontal.

The Infiniium EXR-Series will display measured RMS values, RMS limits, margins, and status as shown in Figure 6.

Note that under the **Settings** section, you can set the frequency to either 50 Hz, 60 Hz, 400 Hz, or use **Automatic Detection**, depending upon the line frequency in your part of the world (or application). You can also select the appropriate IEC standard to test against in this menu (**class A, B, C or D**) as well as the **Display Mode**, as shown in Figure 7.

If the current harmonics measurement is turned on, the EXR oscilloscope performs a fast Fourier transform (FFT) math function on the current waveform. It also creates a table of actual current measurements in IRMS. Scroll down in the table to see up to the 40th harmonic. The status in the tabular format shows whether the harmonic has passed (green) or failed (red) or is on the edge of failure (yellow). Alternatively, you can view the current harmonics measurements in a bar chart format as shown in Figure 8.



Figure 7. Current harmonics settings options

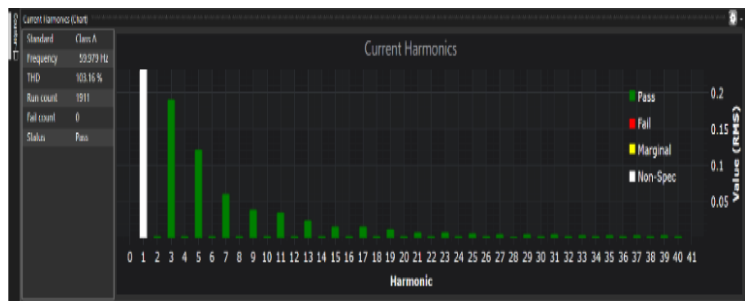


Figure 8. Current harmonics measurements in a bar format

Inrush current

Inrush current is the initial surge of current drawn by the power supply when it is first turned on. This maximal or minimal instantaneous value is captured by a single shot acquisition of the oscilloscope. The first task in making an inrush current measurement is to probe the input with voltage and current probes. You will typically need a high voltage differential probe and a clamp-on Hall-Effect current probe. (The arrow on the current probe should match the direction of the current flow.) Keysight's clamp-on current probes sense current flowing through a conductor and convert it to a voltage that can then be viewed and measured on an oscilloscope. These probes use a hybrid technology that includes a Hall effect sensor (which senses the DC current) and a current transformer (which senses the AC current). Using split core construction, the current probes easily clip on and off a conductor, making it unnecessary to make an electrical connection to the circuit. Measurement bandwidths from DC to 150 MHz are available.

1. Press the **Default Setup** front panel key so you start from the same reference point as these instructions. (The On button at the top of the menu defaults to Off. Although you can select On at any time, it is more efficient to select the particular power measurement, set it up (select channels, settings, Auto Setup, etc.), and then select On).
2. Select the **Analyze** menu, and then select **Power Analysis**. Then you need to select which measurement you want to perform.
3. In this case, select **Input Measurements** from the menu, and then **Inrush Current** in this menu.
4. Set **Max Vin** and then select the **Expected** option in the menu and set what your expected maximum inrush (or peak current) is going to be. Just approximate the expected surge current for your device under test if you don't know. You may need to run the test a few times if your expected peak current is unknown.
5. To begin the measurement, select **Auto Setup**, and then follow the on-screen step-by-step instructions, which are repeated below:
 - a. Turn OFF your power supply and give it a few seconds to drain the capacitors of energy, then select **Next**.
 - b. Turn ON power then select **Next**.
 - c. Repeat several times in order to obtain the worst-case peak inrush current as shown in Figure 9. In this example, the value of the peak current is equal to -33.26 A.

You may need to turn up the intensity or zoom in to see the narrow negative spike in the waveform shown in figure 9.



Figure 9. Inrush current measurements. The yellow signal above is the voltage waveform turning on. The green one is the current wave going from zero amps up and then to a steady state.

Switching Analysis

Switching device analysis is relevant to the study of power and energy losses of your switching device/transistor during one switching cycle, the effective resistance of power field effect transistors (FET), and the slew rates of voltage and current switching signals.

For each of your switching analysis measurements, make sure of the following:

1. Connect the N2790A high-voltage differential probe between the scope's channel 1 input to the drain/ collector (red lead) and source/emitter (black lead) of your switching transistor.
2. Connect the current probe between the scope's channel 2 input to a wire-loop from the source/emitter trace. Note that before connecting the current probe to the current loop, this might be a good time to de-gauss the current probe.
3. Before performing any measurements, ensure that Voltage is set to channel 1 and current is set to channel 2 in the Signals section.

Switching losses

Visualizing the voltage and current during conduction and non-conduction phases will give you a sense of the power losses during the turn on and turn off times of the power supply. Probe the switching device with voltage and current probes. You will need a high-voltage differential probe and a clamp-on current probe to connect to your switching device (e.g., transistor/MOSFET driver chip). You need to connect your voltage probe across the drain and source of your switching device in the power supply. You may also need to create your own current wire loop to measure the current supply that is sourcing the switching device. The arrow on the current probe should match the direction of the current flow.

1. Press the **Default Setup** front panel key so you start from the same reference point as these instructions. (The On button at the top of the menu defaults to Off. Although you can select On at any time, it is more efficient to select the particular power measurement, set it up (select channels, settings, Auto Setup, etc.), and then select On).
2. Select the **Analyze** menu, and then select **Power Analysis**.
3. Select **Switching Measurements**, and then select the **Switching Loss** measurement.
4. Select **Auto Setup**.

The scope will establish optimum vertical scaling on the voltage (yellow trace) and current (green trace) waveforms around the switching device, set the main time base (upper display) to show two switching cycles, and set the zoom time base (lower display) to show one switching cycle. (The scope performs measurements across the single switching cycle).



Figure 10. Power and energy losses over one switching cycle

5. Select **Start Offset Calibration**, and then **Next** to confirm that the scope's vertical settings have been established for the switching loss measurement.
6. Following on-screen directions, disconnect your differential voltage probe from the power supply and short the tips together (0.0 V), then press **Next**.
7. When the voltage offset measurement has completed, disconnect the current probe for the power supply, clamp the jaws fully closed, and then press the **DEMAG** button on the probe.
8. Select **Next** to perform the current offset error measurement at 0.0 amps.
9. Reconnect probes to the DUT and select **Finish**.



Figure 11. Power and energy losses during the conduction phase. The transistor's saturation voltage level (circled in red) is magnitudes lower than the peak-to-peak voltage (circled in pink at right) during the conduction phase of the switching device (e.g., transistor) in the power supply.

The scope should now be measuring the power and energy losses across one switching cycle, as shown in Figure 10. To measure losses during specific phases of switching, simply adjust the zoom window for the desired phase, such as the conduction phase or the turn-off phase. Figure 11 shows an example of measuring the losses during the conduction phase.

The reason that offset calibration is required is that all scopes and probes together have a certain degree of offset error. Even when the scope is within calibration, it is very difficult for the scope to accurately measure the small voltage across the transistor when it is in saturation relative to the scope's vertical scaling which has been automatically scaled for the much larger switching voltage. A small amount of oscilloscope system offset error can produce a relatively large amount of conduction phase

power loss. You must perform a precision offset calibration to minimize conduction phase power loss error. In addition, most of the switching losses will typically occur when the transistor switches. This means that the timing of the voltage and current waveforms are also very critical for getting very accurate switching loss measurements during the turn off phase. So, it is important that you ensure that your voltage and current probes are properly deskewed for this measurement.

Effective resistance

The effective resistance R_{ds} (drain to source) of the switching transistor during the conduction phase is determined by dividing the average voltage by the average current during the center portion of this phase. You can then use this value to automatically calculate the losses during the conduction phase based on the I^2R formula as opposed to power measured as $P=V \times I$.

1. Press the **Default Setup** front panel key so you start from the same reference point as these instructions. (The On button at the top of the menu defaults to Off. Although you can select On at any time, it is more efficient to select the particular power measurement, set it up (select channels, settings, Auto Setup, etc.), and then select On).
2. Select the **Analyze** menu, and then select **Power Analysis**.
3. Select **Switching Measurements**, and then select the **$R_{ds(on)}$ & $V_{ce(sat)}$** measurement.
4. Perform the offset calibration in a similar fashion as in the previous section on switching losses.
5. Select Auto Setup.

The scope computes $R_{ds(on)}$ as V/I as shown in Figure 12.



Figure 12. Measuring $R_{ds(on)}$

Slew rate

Slew rate refers to how fast a device can turn on and/or turn off. Thus, if this rate is high, the losses during switching are minimized.

1. Press the **Default Setup** front panel key so you start from the same reference point as these instructions. (The On button at the top of the menu defaults to Off. Although you can select On at any time, it is more efficient to select the particular power measurement, set it up (select channels, settings, Auto Setup, etc.), and then select On).
2. Select the **Analyze** menu, and then select **Power Analysis**.
3. Select **Switching Measurements**, and then select the **Slew Rate** measurement.
4. Select the desired **Slew Rate Source** (Voltage or Current) under the **Settings** section.
5. Select **Auto Setup**.

Your scope's display should now look similar to Figure 13. The voltage and current waveforms will be optimally scaled, the main time base (upper display) will show two switching cycle across the screen, and the zoom time base (lower display) will be windowed-in on the turn-off phase of the switching voltage and current waveforms.



Figure 13. Slew rate measurement of the voltage waveform during the transistor's turn-off phase.

Output Analysis

Output analysis is relevant to studying the output ripple, turn-on and turn-off time, as well as transient response analysis.

For each of your output analysis measurements, make sure of the following:

- Connect a standard 10:1 passive voltage probe between the scope's channel 3 input to the DC output signal of your DUT and ground.

Output ripple

Output ripple refers to the switching noise on your DC outputs and is a crucial power rail integrity measurement, especially for high-speed digital devices. This may sound simple, but there are some important things to consider in order to make this measurement accurate. You may want to use a shortened ground lead, otherwise the standard ground lead can act like an antenna.

1. Press the **Default Setup** front panel key so you start from the same reference point as these instructions. (The On button at the top of the menu defaults to Off. Although you can select On at any time, it is more efficient to select the particular power measurement, set it up (select channels, settings, Auto Setup, etc.), and then select On).
2. Select the **Analyze** menu, and then select **Power Analysis**.
3. Select **Output Measurements**, and then select the **Output Ripple** measurement.
4. Under the **Signals** section, change the **Voltage** setting to channel 3.
5. Select **Auto Setup**. Your scope will optimally scale and turn on AC coupling and expand the output ripple as much as possible on the screen.

Your scope should now display something like Figure 14. For a best-case scenario, you can turn on bandwidth limit. The type of probe you use and the way you ground it can make a big difference.



Figure 14. Output ripple measurements

Turn-on and turn-off time

Turn-on time is the time for the output voltage to reach 90% of the expected level after the input voltage is applied to the switch mode power supply. Turn-off time is the time for the output voltage to drop to 10% of its normal output level after the input voltage is removed. This step focusses on the turn-on and turn-off time measurement on the DC output relative to when AC power is applied on a SMPS. You need to probe the AC input and DC output of the SMPS. Use a high-voltage differential probe for the AC input and a standard 10:1 passive voltage probe for the DC output.

1. Connect the N2790A high-voltage differential active probe between the scope's channel-1 input and Line (red lead) and Neutral (black lead) on your switch mode power supply (SMPS). Connect the passive probe between the scopes channel-3 input and the DC output of the power supply.
2. Press the **Default Setup** front panel key so you start from the same reference point as these instructions. (The On button at the top of the menu defaults to Off. Although you can select On at any time, it is more efficient to select the particular power measurement, set it up (select channels, settings, Auto Setup, etc.), and then select On).
3. Select the **Analyze** menu, and then select **Power Analysis**.
4. Select **Output Measurements**, and then select **Turn-On/Off Time**.
5. Under the **Signals** section, set the **Input Voltage** to channel 1 and the **Output Voltage** to channel 3.
6. Under the **Settings** section, enter values for expected **Max Vin** (peak-to-peak) and expected steady-state DC output (**Vout**). These values will be used to determine the scope's vertical

scaling when Auto Setup is selected (next step). Also, you may want to begin your test with the default duration setting of 500 ms, which will determine the scope's time base setting for this measurement.

7. Select **Auto Setup**, then follow on-screen instructions for the Turn On measurement repeated below:
 - a. Turn OFF the power supply then press **Next**.
 - b. Turn ON the power supply then press **Next**.

If the turn-on time of your device under test is less than 400 ms, then your scope's display should look similar to Figure 15. To see the detail, you can zoom in on the rise time of the power supply with the touch screen and horizontal zoom.

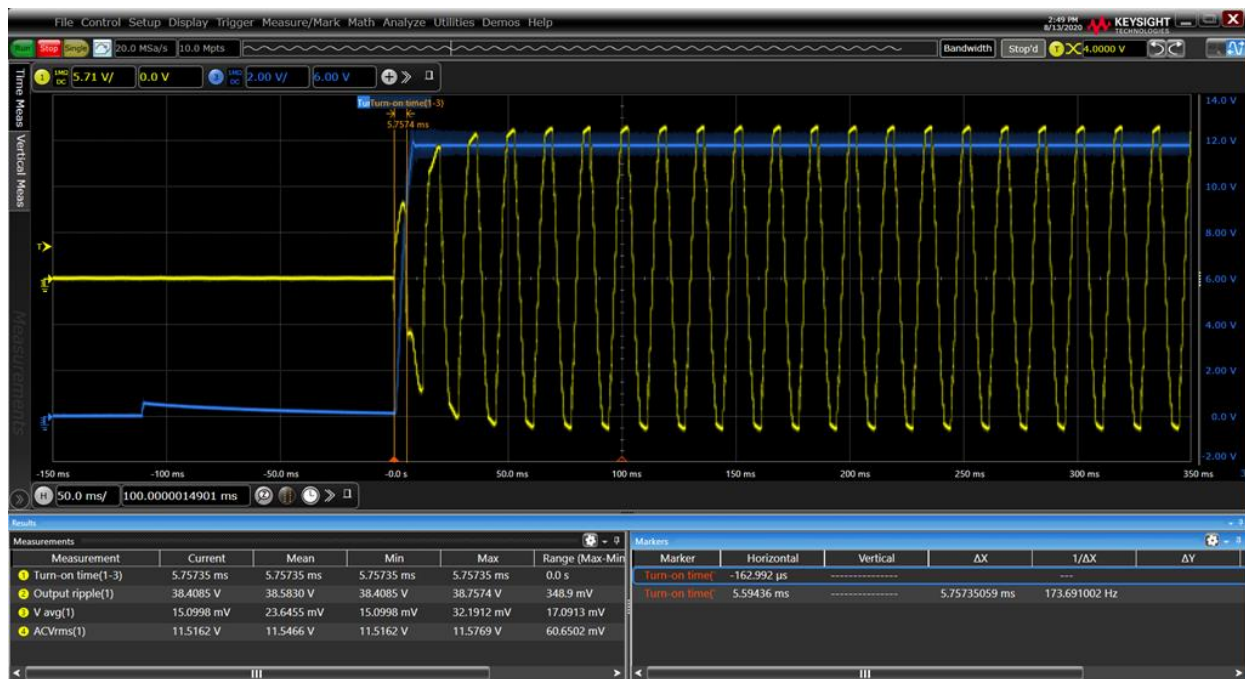


Figure 15. Turn-on test using a default duration time setting of 500 ms. The yellow waveform is the AC input. The blue waveform is the DC output that starts at 0 v and then ramps up quickly.

Testing Turn Off:

1. Continuing from the previous Turn-on time measurement, under the **Settings** section, press the **Test** softkey then change from Turn-on to **Turn-off**. Note that the default duration time setting of 1.00 second may not be a valid setting for your power supply. This measure can be repeated until the duration time is correctly set.
2. Select **Auto Setup** and follow the step-by-step on-screen instructions, which are repeated below:
 - a. Turn ON the power supply (if it's already on, do nothing) then press **Next**.
 - b. Turn OFF the power supply then press **Next**.

Your scope's display should now look similar to Figure 16, as the power measurement software has captured the drop of the DC output (blue line).

Repeat this measurement several times to determine the best- and worst-case turn-off times. Note that you may need to adjust the **Duration** time setting for this test.

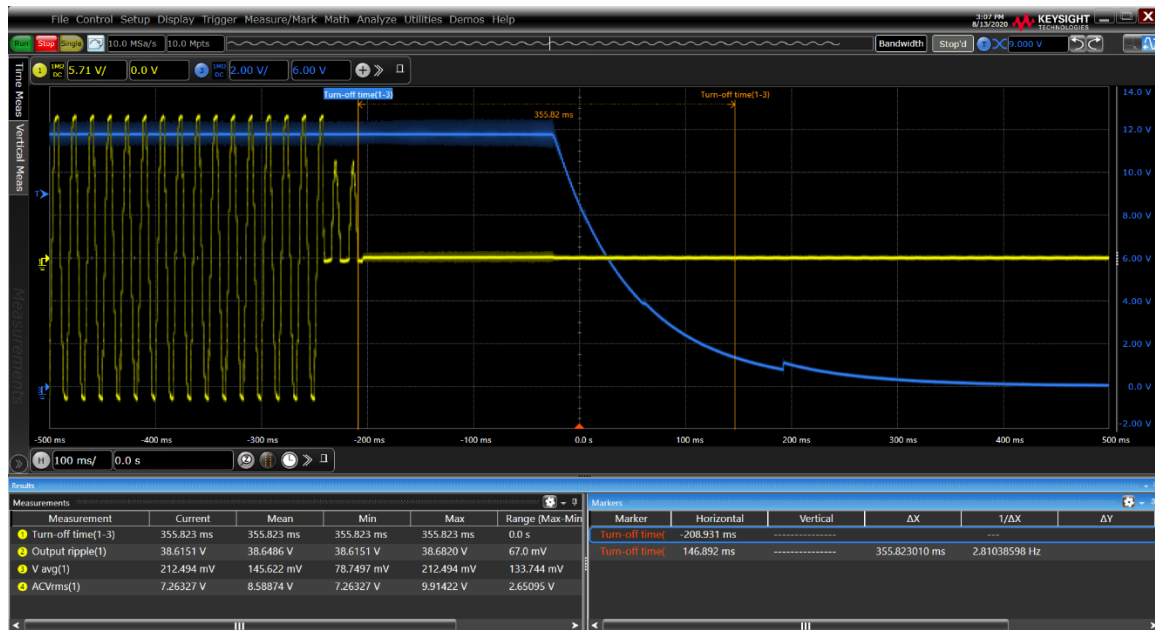


Figure 16. Turn-off time test

Transient response analysis

Transient response analysis measures the transient response settling time of the DC output voltage. When there is a sudden increase or decrease in the output current (e.g., load change), the scope triggers and compares the values to the tolerance entered by the user. Probe the DC output with the standard 10:1 voltage probe and an AC/DC clamp-on current probe.

1. Press the **Default Setup** front panel key so you start from the same reference point as these instructions. (The On button at the top of the menu defaults to Off. Although you can select On at any time, it is more efficient to select the particular power measurement, set it up (select channels, settings, Auto Setup, etc.), and then select On).
2. Select the **Analyze** menu, and then select **Power Analysis**.
3. Select **Output Measurements**, and then select the **Transient Response** measurement.
4. Connect the current probe (e.g., N7026A) between the scope's channel-2 input and the DC output on your power supply.
5. Connect the 10:1 voltage probe between the scope's channel-3 input and the DC output on your power supply.
6. Under the **Signals** section, set the **Voltage** to channel 3 and the Current to channel 2.

7. Under the **Signals** section, set the expected steady-state dc output voltage (**Vout**) for your power supply.
8. Under the **Settings** section, set **Initial I** (current) to the approximate low-load output current level—assuming that the load power supply under test is in a low- load/low current status. Also set the **New I** to the approximate high-load output current level. Your approximate, expected values are not critical to get correct, because the scope uses those values to set the trigger level between the expected high and low values.
9. Select **Auto Setup**, then follow the on-screen step-by-step instructions, as repeated below:
 - a. Increase the output load (i.e., higher current).
 - b. Press **OK**.

Your scope's display should now look similar to Figure 17. When output current measured by the channel-2 current probe (green trace) suddenly increases, output voltage (blue trace) should momentarily drop and then settle back to within 10% of the expected output voltage level in approximately 10 ns.

You can also perform this transient response settling time measurement for when the output load decreases (i.e., lower output current), in which case your scope's display should look similar to Figure 18. When output current (green trace) suddenly drops, output voltage (blue trace) momentarily increases and then settles back to within 10% of the expected output voltage level in approximately 9 ns.

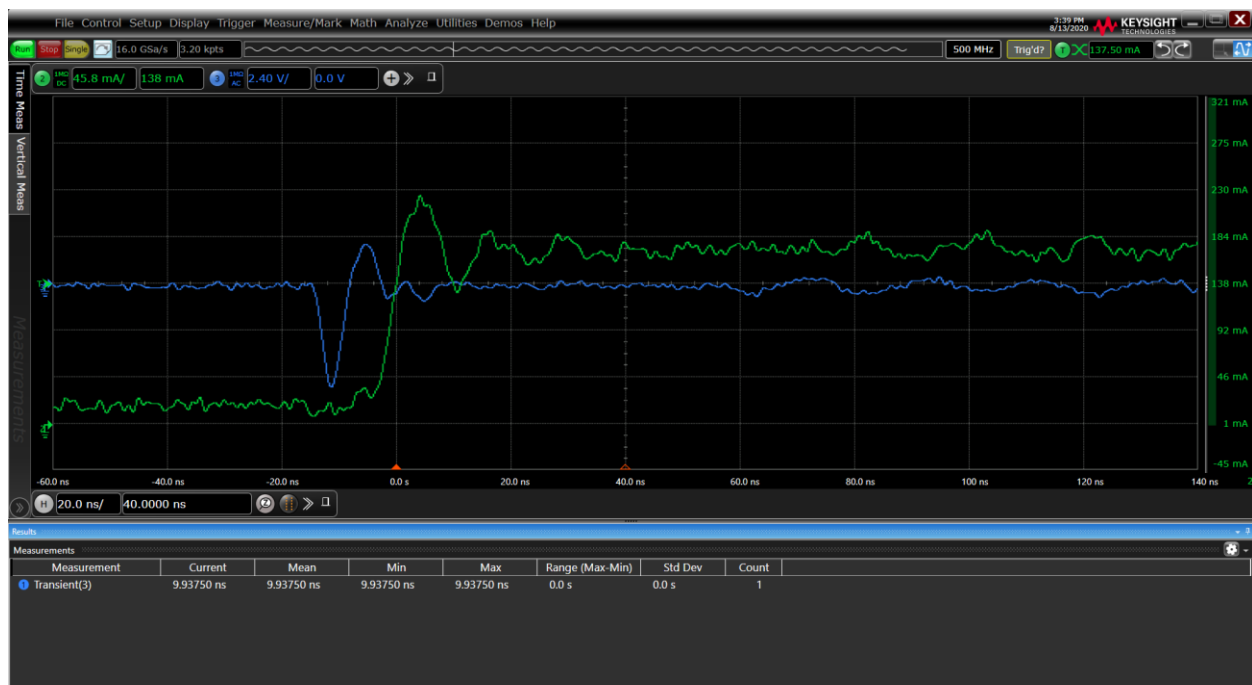


Figure 17. Transient response settling time measurement when output load increases (output current suddenly increases)

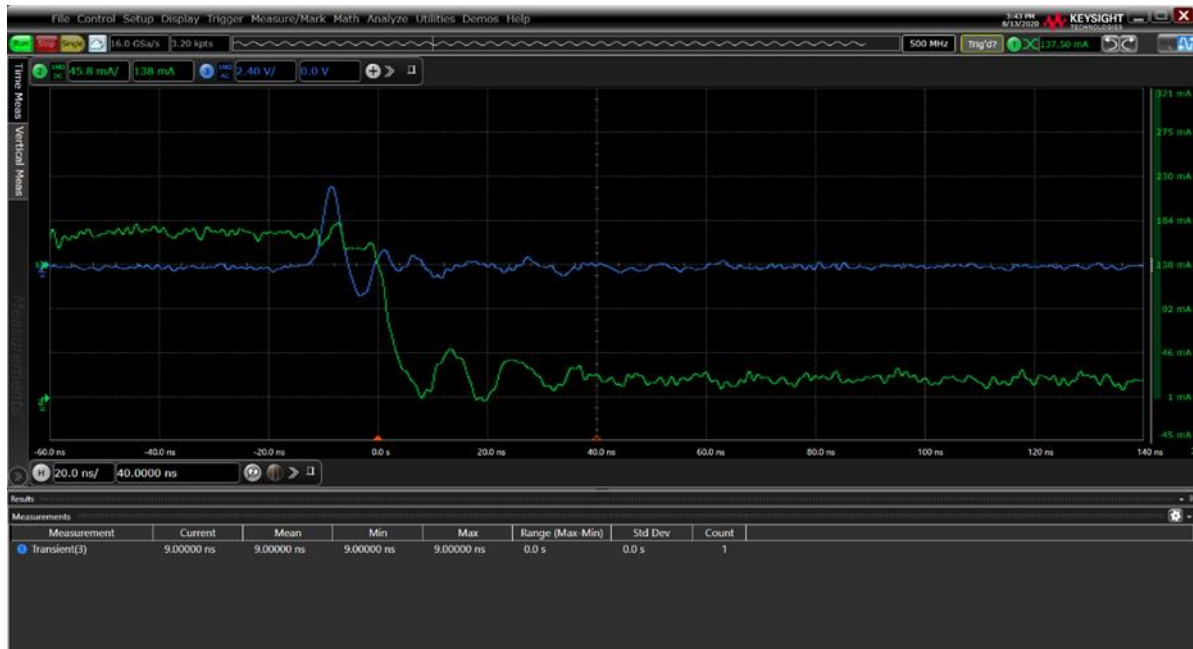


Figure 18. Transient response settling time measurement when output load decreases (output current suddenly decreases)

Note that the transient response settling time measurement for both increasing and decreasing load conditions should be repeated several times in order to characterize best- and worst-case settling times.

Efficiency

Efficiency is the last specification that is usually the most critical for any SMPS design. The lower the wasted energy in your power supply or battery, the lower the operational cost of your device and the longer the life span. The efficiency analysis measures and calculates the real power ($P_{IN} = V_{IN} \times I_{IN}$ and $P_{OUT} = V_{OUT} \times I_{OUT}$) of your input and output, then calculates the efficiency as P_{OUT}/P_{IN} . To measure efficiency, you will need two current probes and two voltage probes. Note, you may need to create a current loop out of wire to test your power supply with a clamp-on current probe to measure current.

1. Press the **Default Setup** front panel key so you start from the same reference point as these instructions. (The On button at the top of the menu defaults to Off. Although you can select On at any time, it is more efficient to select the particular power measurement, set it up (select channels, settings, Auto Setup, etc.), and then select On).
2. Select **Analyze**; then select **Power Analysis**.
3. Select **Output Measurements**; then select the **Efficiency** measurement.
4. Connect the N2790A high-voltage differential active probe between the scope's channel-1 input and probe tips to Line (red lead) and Neutral (black lead).
5. Connect a clamp-on current probe between the scope's channel-2 input to the input AC Line current loop.

6. Connect a voltage probe (e.g., 10:1 passive voltage probe) between the scope's channel-3 input to the DC output voltage.
7. Connect another current probe between the scope's channel-4 input to the output DC current loop.
8. Under the **Signals** section, set the **Voltage Input** to channel 1, the **Current Input** to channel 2, the **Voltage Output** to Channel 3 and the **Current Output** to Channel 4, as shown in Figure 19.
9. Select **Auto Setup**. The scope will automatically set up the AC input signals as well as the DC output signals and put them all up on the display.



Figure 19. Efficiency settings

Your scope's display should now reflect the efficiency of your power supply. If you don't have four probes, you can still measure efficiency. It is possible to perform the efficiency measurement with only one active current probe. However, to do that, first measure the input power using the set up described above, with nothing connected to channel 4. Your scope's display should now be similar to that in Figure 20. In this case, $P(in) = 3.35W$.

Next, move the current probe that is connected to the scope's channel-2 input to channel 4. Then connect it to the output DC current loop. **Manually reposition** and rescale Channel 3 (V out) and Channel 4 until both DC traces can be observed on-screen. Now take note of the output power results, as shown in Figure 21. In Figure 21, $P(out) = 2.20W$.

You can then find the efficiency of your power supply using the equation $P(out)/P(in) \times 100\%$.



Figure 20. Input power measurement



Figure 21. Output power measurement

Conclusion

Each of these measurements is important for testing and characterizing switch mode power supplies. Other relevant measurements not covered in this application note include the Power Supply Rejection Ratio Analysis (PSRR) and Control Loop Response Analysis.

¹ The probe attenuation factor setting changes the vertical scaling of the oscilloscope so that the measurement results reflect the actual voltage levels at the probe tip. If the channel's menu is not currently displayed, press the channel key ([1], [2], [3], or [4]). In the Channel menu, press Probe. Continue pressing the Probe softkey or turn the entry knob to select the appropriate attenuation factor.

Learn more at: www.keysight.com

For more information on Keysight Technologies' products, applications or services, please contact your local Keysight office. The complete list is available at: www.keysight.com/find/contactus

