

Online Help



DPOPWR Power Measurement and Analysis Software

Adapted from the DPOPWR Power Measurement and Analysis Online Help,
077-0018-00, Version 1.0.

Copyright © Tektronix, Inc. All rights reserved. Licensed software products are owned by Tektronix or its suppliers and are protected by United States copyright laws and international treaty provisions.

Tektronix products are covered by U.S. and foreign patents, issued and pending. Information in this publication supercedes that in all previously published material. Specifications and price change privileges reserved.

TEKTRONIX and TEK are registered trademarks of Tektronix, Inc.

Contacting Tektronix

Tektronix, Inc.
14200 SW Karl Braun Drive
P.O. Box 500
Beaverton, OR 97077
USA

For product information, sales, service, and technical support:

- In North America, call 1-800-833-9200.
- Worldwide, visit www.tektronix.com to find contacts in your area.

Table of Contents

GENERAL SAFETY SUMMARY	VII
TO AVOID FIRE AND PERSONAL INJURY	vii
Terms in this Manual	vii
Terms in the Application.....	vii
INTRODUCTION	1
WELCOME	1
ONLINE HELP AND RELATED DOCUMENTATION	2
USING ONLINE HELP	2
PRINTING FROM ONLINE HELP	3
RELATED DOCUMENTATION.....	3
CONVENTIONS.....	4
FEEDBACK.....	4
GETTING STARTED	7
INTRODUCTION AND PRODUCT DESCRIPTION	7
COMPATIBILITY.....	7
REQUIREMENTS AND RESTRICTIONS	8
ACCESSORIES	8
CURRENT PROBES	9
DIFFERENTIAL PROBES.....	9
UPDATES THROUGH THE WEB SITE.....	10
INSTALLATION PROCEDURES.....	10
Installing the Application.....	10
UTILITIES	11
DESKEWING PROBES AND CHANNELS	11
Static Deskew.....	11
Deskewing in DPO7000 series Oscilloscopes.....	13
Deskewing Using a Power Deskew Fixture	15
SOA OVERLAY	16
OPERATING BASICS	19
APPLICATION INTERFACE	19
APPLICATION INTERFACE MENU CONTROLS	19
BASIC APPLICATION FUNCTIONS.....	20
USING ONLINE HELP	20
APPLICATION DIRECTORIES AND FILE NAMES	20
FILE NAME EXTENSIONS	21
RETURNING TO THE APPLICATION.....	21
SETTING UP DPOPWR TO TAKE MEASUREMENTS	21
TABLE OF OPTIONS-SOURCE CONFIGURATION	22
TABLE OF MEASUREMENTS AND CONFIGURATIONS-POWER DEVICE ANALYSIS	23
TABLE OF MEASUREMENTS AND CONFIGURATIONS-MAGNETICS	26

Table of Contents

TABLE OF MEASUREMENTS AND CONFIGURATIONS-INPUT/OUTPUT ANALYSIS.....	27
TAKING A NEW MEASUREMENT.....	31
ANALYZING THE RESULTS	31
SELECTING A MEASUREMENT	31
SOURCE CONFIGURATION PANEL.....	32
MATH SETUP.....	34
SOA MASK EDITOR.....	36
POWER DEVICE ANALYSIS	38
HI-POWER FINDER	38
Selecting and Configuring Measurements	38
Viewing Hi-Power Finder Results	43
Generating Reports-Hi-Power Finder	45
SWITCHING LOSS	45
Selecting and Configuring Measurements	45
Viewing Switching Loss Results	50
Generating Reports	52
Troubleshooting Switching Loss Results.....	53
RDS(ON).....	55
Selecting and Configuring Measurements-Dynamic Resistance	55
Viewing Results-RDS(on)	57
Generating Reports-RDS(on).....	58
DI/DT	58
Selecting and Configuring Measurements-di/dt.....	58
Viewing Results-di/dt	60
Generating Reports-di/dt.....	61
DV/DT.....	62
Selecting and Configuring Measurements-dv/dt.....	62
Viewing Results-dv/dt	63
Generating Reports-dv/dt.....	65
PULSE WIDTH	65
Selecting and Configuring Measurements-Pulse Width	65
Viewing Results-Pulse Width	67
Generating Reports-Pulse Width	68
DUTY CYCLE	68
Selecting and Configuring Measurements-Duty Cycle.....	68
Viewing Results-Duty Cycle	70
Generating Reports-Duty Cycle.....	70
PERIOD.....	71
Selecting and Configuring Measurements-Period.....	71
Viewing Results-Period	72
Generating Reports-Period.....	73
FREQUENCY	74
Selecting and Configuring Measurements-Frequency	74
Viewing Results-Frequency.....	76
Generating Reports-Frequency	76
SAFE OPERATING AREA	77
Selecting and Configuring Measurements-Safe Operating Area	77
Viewing Results-Safe Operating Area.....	79
SOA Overlay	83
Generating Reports-Safe Operating Area	85

MAGNETICS.....	86
Selecting and Configuring Measurements-Magnetics.....	86
MAGNETIC LOSS	87
INDUCTANCE.....	88
Ivs $\int V$	89
MAGNETIC PROPERTY	89
Viewing Results-Magnetics	93
Generating Reports-Magnetics.....	99
INPUT/OUPUT ANALYSIS.....	99
POWER QUALITY	99
Selecting and Configuring Measurements-Power Quality.....	99
Viewing Results-Power Quality.....	101
Generating Reports-Power Quality	102
CURRENT HARMONICS	103
Selecting and Configuring Measurements-Current Harmonics.....	103
Viewing Results-Current Harmonics	108
Generating Reports-Current Harmonics.....	113
TOTAL POWER QUALITY	114
Selecting and Configuring Measurements-Total Power Quality.....	114
Viewing Results-Total Power Quality	119
Generating Reports-Total Power Quality.....	121
LINE RIPPLE	121
Selecting and Configuring Measurements-Line Ripple	121
Viewing Results-Line Ripple.....	122
Generating Reports-Line Ripple	123
SWITCHING RIPPLE.....	124
Selecting and Configuring Measurements-Switching Ripple	124
Viewing Results-Switching Ripple	125
Generating Reports-Switching Ripple.....	125
TURN-ON TIME	126
Selecting and Configuring Measurements-Turn-On Time.....	126
Viewing Results-Turn-On Time.....	128
Generating Reports-Turn-On Time	129
SPECTRAL ANALYSIS	129
Selecting and Configuring Measurements-Spectral Analysis	129
Viewing Results-Spectral Analysis.....	131
Generating Reports-Spectral Analysis	133
Troubleshooting Spectral Analysis Error Messages.....	133
SAVING AND RECALLING SETUPS.....	133
Saving a Setup.....	133
Recalling a Saved Setup.....	134
Recalling the Default Setup.....	134
TUTORIAL	135
INTRODUCTION TO THE TUTORIAL	135
STARTING THE APPLICATION.....	135
SETTING UP THE OSCILLOSCOPE	135
MEASURING SWITCHING LOSS	135
MEASURING MAGNETICS	137
STOPPING THE TUTORIAL	140

RETURNING TO THE TUTORIAL.....	140
APPLICATION EXAMPLES	141
TO INCREASE EFFICIENCY IN A SWITCHING POWER SUPPLY	141
Measure Switching Loss	141
Measure Hi-Power Finder.....	144
Measure Magnetic Loss	146
REFERENCE	149
ERROR CODES FOR DPOPWR	149
PARAMETERS	161
About Application Parameters.....	161
Sequencer Parameters	161
Measurement Menu Parameters.....	161
Soruce Configuration Parameters	162
Configure Hi-Power Finder and Switching Loss Parameters	163
Configure Magnetics Parameters.....	165
Configure SOA Parameters.....	166
Configure RDS(on) Parameters	166
SOA Mask	166
Configure di/dt Parameters	167
Configure dv/dt Parameters	167
Configure Power Quality Parameters	167
Configure Current Harmonics Parameters	167
Configure Total Power Quality Parameters	168
Configure Line Ripple Parameters.....	169
Configure Switching Ripple Parameters.....	169
Configure Turn-On Time Parameters	169
Configure Pulse Width Parameters.....	170
Configure Period Parameters	170
Configure Duty Cycle Parameters	170
Configure Frequency Parameters.....	171
Configure Spectral Analysis	171
Deskew-Static Deskew	171
Real Time Deskew.....	172
MEASUREMENT ALGORITHMS.....	173
About Measurement Algorithms.....	173
Oscilloscope Setup Guidelines	173
Switching Loss.....	173
Magnetics.....	177
Safe Operating Area.....	179
Dynamic Resistance (RDS(on)).....	181
di/dt.....	181
dv/dt.....	182
Power Quality	183
Current Harmonics.....	185
Total Power Quality.....	186
Line Ripple and Switching Ripple	186
Turn-On Time	187
Modulation Analysis.....	187

Period and Frequency Modulation	188
Spectral Analysis.....	189
Selecting RBW value	191
Hysteresis	191
APPENDIX.....	193
APPENDIX A	193
For P5205 probe	193
For TCP202 probe.....	193
APPENDIX B	194
GLOSSARY	195

General Safety Summary

Review the following safety precautions to avoid injury and prevent damage to the measurement instrument or any products connected to it. To avoid potential hazards, use the software and measurement instrument only as specified.

While using this software, you may need to access other parts of the system. Read the General Safety Summary and specification sections in other equipment manuals for warnings, cautions, and ratings related to operating the system with this software.

To avoid Fire and Personal Injury

Connect and Disconnect Properly. Connect the probe output to the measurement instrument before connecting the probe to the circuit under test. Disconnect the probe input and the probe ground from the circuit under test before disconnecting the probe from the measurement instrument.

Observe All Terminal Ratings. To avoid fire or shock hazard, observe all ratings and markings on the measurement instrument and other equipment used with this software. Consult the individual product manuals for further ratings information before making connections to the circuit under test.

Do Not Operate With Suspected Failures. If you suspect there is damage to the measurement instrument or other equipment being used with this software, have it inspected by qualified service personnel.

Terms in this Manual

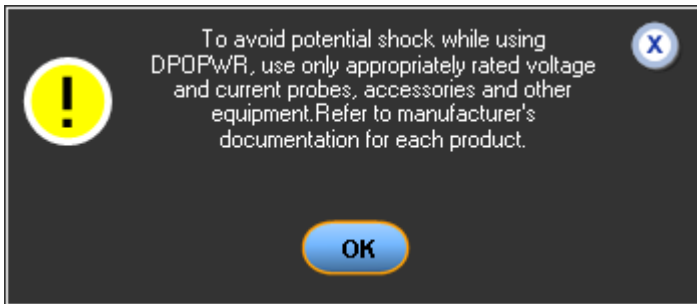
These terms may appear in this manual:

WARNING. Warning statements identify conditions or practices that could result in injury or loss of life.

CAUTION. Caution statements identify conditions or practices that could result in damage to the measurement instrument or other property.

Terms in the Application

When launched from the oscilloscope menu, the application displays a caution dialog box, warning you to use only appropriately rated voltage and current probes, accessories and other equipments. Press OK.



Introduction

Welcome

DPOPWR is a power measurement and analysis software that runs on DPO7000 series oscilloscopes. DPOPWR allows you to acquire, measure and analyze various switching power supply signals at different test points. The application provides a simple and direct means of obtaining results about Switching devices, Magnetic components and Precipitance tests to EN 61000-3-2 Std for the Switch Mode Power Supply. The features of DPOPWR are:

- Hi-Power Finder and Magnetics pinpoint the source of reliability problems
- Power Loss Measurements at the Switching Device and Magnetic Component improve switching power supply efficiency
- Sophisticated Report Generation saves time
- Customizable Safe Operating Area Mask Testing with Linear and Log Scale for reliability testing
- In-circuit Bpeak, Permeability, and Coercive Force Measurements determine the reliability of the magnetic component
- Automatic Ripple Measurement Setup eliminates manual processes
- Spectral Analysis locates ripple and noise on the Output Voltage and EMI Issues with the switching power supply
- Pre-compliance Testing to the EN61000-3-2, EN61000-3-2 AM14 and MIL 1399 (400 Hz) Standards Reduces compliance test time and risk

To return to the Table of Contents, select Help Topics from the Help Topic menu.

Online Help and Related Documentation

You can access the information on how to operate the application along with the oscilloscope through the following related documents and online help.

Using Online Help

Select Help on the right side of the oscilloscope menu bar to bring up the help file.

Tables of Contents (TOC) tab — organizes the Help into book-like sections. Select a book icon to open a section; select any of the topics listed under the book.

Index tab — enables you to scroll a list of alphabetical keywords. Select the topic of interest to bring up the appropriate help page.

Find tab — allows a text-based search. Follow these steps:

1. Type the word or phrase you want to find in the search box.
2. Select some matching words in the next box to narrow your search.
3. Choose a topic in the lower box, and then select the Display button.

To print a topic, select the Print button from the Help Topics menu bar.

Select Options from the menu bar for other commands, such as Refresh.


Select the Back button to return to the previous help window. Use the Hyperlink to jump from one topic to another. If the Back button is grayed out or a jump is not available, choose the Help Topics button to return to the originating help folder.


Sometimes you will see the word, **Note**, in the topic text. This indicates important information.

Note:

Certain aspects of the online help are unique to applications that run on the oscilloscope.

Blue or green underlined text indicates a jump (hyperlink) to another topic. Select the underlined text to jump to the related topic. For example, select the blue text to jump to the topic on Online Help and Related Documentation and the Back button to return to the topic's page.

 **Tip** You can tell when the cursor is over an active hyperlink (button, jump, or pop-up), because the cursor arrow changes to a small hand.

*Note: The light bulb and word **Tip** in the graphic  indicates additional information to help you function more efficiently.*

Printing from Online Help

Some online help topics have color in the examples of the displayed application. If you want to print this type of topic on a monochrome printer, some information may not print because of certain colors. Instead, you should print the topic from the PDF (portable document format) file that corresponds to the Online Help. You can find the file in the Documents directory on the Optional Applications Software on Windows-Based Oscilloscopes CD-ROM.

Related Documentation

In addition to the Online Help, you can access other information on how to operate the oscilloscope through the following related documents:

- Oscilloscope Information: The user manual and user online help for your oscilloscope provides general information on how to operate the oscilloscope.
- You can download PDF versions of many user manuals from the Tektronix Web site.
- Programmer Information: The online programmer guide for your oscilloscope provides details on how to use GPIB commands to control the oscilloscope.
- You can download programmer information and examples from the Tektronix Web site.

Refer to the *Optional Applications Software on Windows-Based Oscilloscopes Installation Manual* for the following information:

- Software warranty
- List of all available applications, compatible oscilloscopes, and relevant software and firmware version numbers
- Applying a new label
- Installing an application
- Enabling an application
- Downloading updates from the Tektronix Web site

You can find a PDF (portable document format) file for this document in the Documents directory on the Optional Applications Software on Windows-Based Oscilloscopes CD-ROM. The CD booklet only contains information on installing the application from the CD and on how to apply a new label.

Conventions

Online help uses the following conventions:

- Refers to the DPOPWR Power Measurements solution as DPOPWR Application.
- When steps require a sequence of selections using the application interface, the ">" delimiter marks each transition between a menu and an option. For example, one of the steps to save a setup file would appear as File> Save.
- The Source Configuration Panel is common to many measurements.
- GP knob refers to the general-purpose knob.

Feedback

Tektronix values your feedback on our products. To help us serve you better, please send us your suggestions, ideas, or comments on your oscilloscope.

Direct your feedback via e-mail to

techsupport@tektronix.com or FAX at (503) 627-5695

and include the following information. Please be as specific as possible.

General information:

- Oscilloscope model number and hardware options, if any
- Probes used
- Your name, company, mailing address, phone number, FAX number
- Please indicate if you would like to be contacted by Tektronix about your suggestion or comments

Application specific information:

- Software version number
- Description of the problem such that technical support can duplicate the problem
- If possible, save the oscilloscope and application setup files as .set files

- If possible, save the waveform on which you are performing the measurement as a .wfm file

Once you have gathered this information, you can contact technical support by phone or through e-mail. If using e-mail, be sure to enter in the subject line "DPOPWR Problem," and attach the .set and .wfm files.

Getting Started

Introduction and Product Description

DPOPWR is a power measurement and analysis software that runs on DPO7000 series oscilloscopes. It allows you to acquire, measure and analyze various switching power supply signals at different test points. The application provides a simple and direct means of obtaining results about Switching devices, Magnetic Components and Precipitance tests to EN 61000-3-2 Std for the Switch mode Power supply. The features of DPOPWR are:

- **Power Device Analysis** analyzes the switching devices in the power supply such as FET (Field Effect Transistor), BJT (Bi Polar Junction Transistor) and IGBT (Integrated Gated Bi Junction Transistor). The measurements include Switching Loss, Hi-Power Finder, Safe Operating Area (SOA), SOA Mask test, Dynamic Resistance, di/dt, dv/dt, Pulse Width Modulation, Period Modulation, Duty Cycle Modulation and Frequency Modulation.
- **Magnetics** measures Inductance, Magnetic Property, Magnetic Loss, and I vs Integral V.
- **Input/Output Analysis** measures Power Quality, Current Harmonics and Total Power Quality, Line Ripple, Switching Ripple, Turn-On Time, and Spectral Analysis. Spectral Analysis analyses the frequency component that contributes to the electromagnetic interference and measures the noise/ripple at the output DC voltage frequency range.
- A **Report Generator** tool helps you create and print custom reports.
- **Deskew and SOA Overlay** utilities.

Compatibility

The DPOPWR application is compatible with the following DPO7000 series oscilloscopes:

- DPO7254
- DPO7104
- DPO7054

Measurement dependencies on the number of channels and oscilloscope options:

To view measurement dependencies on the number of channels:

Select Help> About TekScope. The oscilloscope displays the channel dependencies in the option panel.

Table 1: Measurements and channel dependencies

Measurement Name	Number of Channels
Switching Loss	3
Magnetics	3
Safe Operating Area	2
RDS(on)	2
di/dt	1
dv/dt	1
Power Quality	2
Current Harmonics	2
Total Power Quality	2
Line Ripple	1
Switching Ripple	1
Turn-On time	2 to 4
Pulse Width	1
Duty Cycle	1
Period	1
Frequency	1

Requirements and Restrictions

Microsoft .NET Framework Version 1.1 must be installed on the oscilloscope to operate the DPOPWR application.

Accessories

The standard accessories of the application are:

- Optional Application Software on Windows-Based Oscilloscope CD-ROM
- CD booklet

Current Probes

The application supports the following probes:

- TCP0030
- TCP202
- TCPA300 with TCP305
- TCPA300 with TCP303
- TCPA300 with TCP312
- TCPA400 with TCP404XL
- AM503 Series with A6312
- AM503 Series with A6303
- AM503 Series with A6302
- AM503B with A6302XL
- AM503B with A6303XL
- AM503B with A6304XL
- AM502B with A6312
- AM502B with A6302
- AM502B with A6303

Differential Probes

- P5200
- P5205
- P5210
- ADA400A

The application also supports the Deskew Fixture (Tektronix part number: 067-1686-xx).

Note: *If you are using external fixtures, to measure Turn-On Time and Ripple, enter the external attenuation factor values in Vertical> Attenuation menu of the oscilloscope for correct results.*

Updates Through the Web Site

You can find information about this and other applications at the Tektronix Inc. Web site, www.tektronix.com. Check this site for application updates and other free applications.

To install an application update, you will need to download it from the Tektronix Web site to the oscilloscope hard disk.

Note: *More information about changes to the application or installation is in a [Readme.txt](#) file on the Web site. You should read it before you continue.*

Installation Procedures

Installing the Application

Refer to the *Optional Applications Software on Windows-Based Oscilloscopes Installation Manual* for the following information:

- Software warranty
- List of all available applications, compatible oscilloscopes, and relevant software and firmware version numbers
- Applying a new label
- Installing an application
- Enabling an application
- Downloading updates from the Tektronix Web site.

You can find a PDF (portable document format) file for this document in the Documents directory on the *Optional Applications Software on Windows-Based Oscilloscopes CD-ROM*. The CD booklet contains information on how to install the application from the CD and on how to apply a new label.

Note: *Refer the Probe manual for probe specifications such as bandwidth, maximum voltage, maximum current etc.*

Utilities

Deskewing Probes and Channels

To ensure accurate results, deskew the probes before taking measurements from your device under test. Refer to the Appendix for procedural steps to degauss and clear the offset before you deskew.

The application provides you two deskew options. They are:

- Static Deskew
- Real time deskew using the internal source and an external source.

Note: Connect the probes to the fastest transition while using the external signal mode.

Static Deskew

Static deskew automatically sets the deskew based on the probe type supported. The propagation values of the probes are fixed. You do not need an external deskew fixture to deskew the channels.

The application supports the following combination between any two-channel probes for DPO7000:

Table 2: Probes and skew support

Probe Model	Skew in ns
P6139A	6.3
P6131 (1.3 mts)	6.3
P6131 (2 mts)	9
P6138A	6.3
P5100	14.9
P5050	6.3
P6158	5
P6015A (10 ft)	14.7
P6015A (25 ft)	33.3
AM503 Series with A6312	30
AM503 Series with A6303	55
AM503 Series with A6302	30
AM503B with A6302XL	60
AM503B with A6303XL	105

Table 2: Probes and skew support (Contd.)

Probe Model	Skew in ns
TCPA300 with TCP305	19
TCPA300 with TCP303	53
TCPA300 with TCP312	17
TCPA400 with TCP404XL	103
P5205+TPA-BNC	17.25
P5210+TPA-BNC	20.25
TCP202+TPA-BNC	17.25
TekVPI TCP0030	17
P6246	7.65
P6247	7.25
P6248	6.75
Custom	0

Follow these steps to perform the Static Deskew:

1. Select Analyze> Power Analysis> Deskew.
2. Click the Static tab.
3. In the From panel, select the Source and the corresponding Probe type.
4. In the To panel, select the Source and the corresponding Probe type.

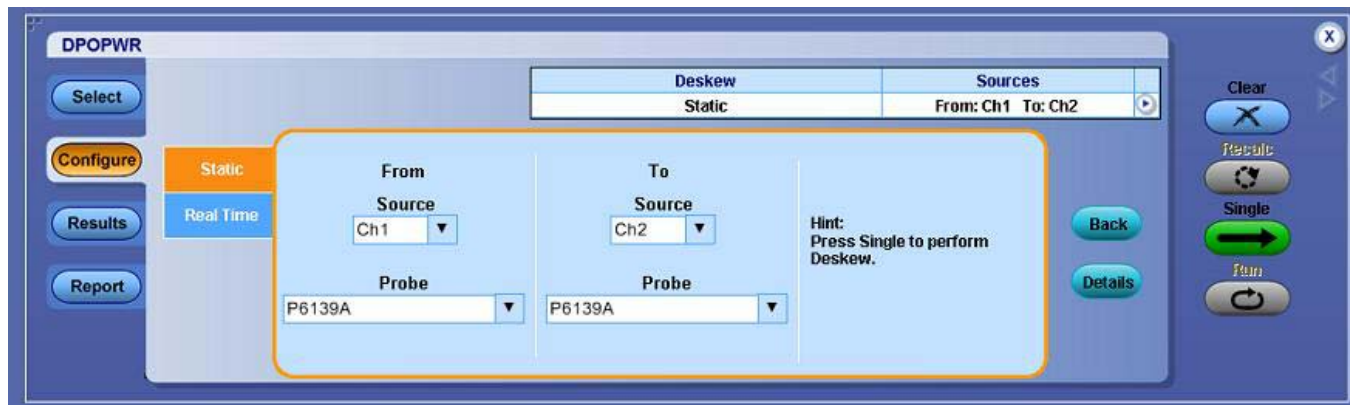


Figure 1: Static Deskew

5. Click Run to deskew the selected probe.

Note: Static deskew can be performed between any two channels of your choice. Retain the From panel parameters and change the To panel parameters to perform deskew for more than two channels.

If you are using a Custom probe, the application adjusts the deskew and sets the value when you perform static deskew.

Deskewing in DPO7000 series Oscilloscopes

Use the Aux Output signal (DPO7000) and the Deskew fixture (Tektronix part number: 067-1686-xx) to deskew the probes and channels.

Follow these steps to deskew with an internal source:

1. Connect the AUX OUT of the oscilloscope to the B side input of the deskew fixture to deskew the voltage probe and current probe.
2. Follow the instructions for the Probe Calibration and Deskew fixture to make the connections.
3. Set up the oscilloscope as follows:
 - Use the Horizontal Scale knob to set the oscilloscope to an acquisition rate so that there are two or more samples on the deskew edge.
 - Use the Vertical Scale and Position knobs to adjust the signals and display them on the screen.
 - Set the Record Length so that there are more samples for the edges in the acquisition. We recommend that you set the record length to 25000 points.
4. To start the application from a DPO7000 series oscilloscopes, select Analyze> Power Analysis> Deskew.
5. Select the Real Time tab. Click the Internal Source button in the Source panel.
6. In the From panel, set the Source to Ch1. The remaining channels will be deskewed to the Source waveform, which is the reference point.
7. In the To panel, set the Source to Ch2, the channel to be deskewed.
8. To start the deskew utility, click the Run button and confirm the operation.
9. Use Ch1 as the reference point and deskew the remaining channels.



Figure 2: Before deskew

Figure 3 displays the results after deskewing.

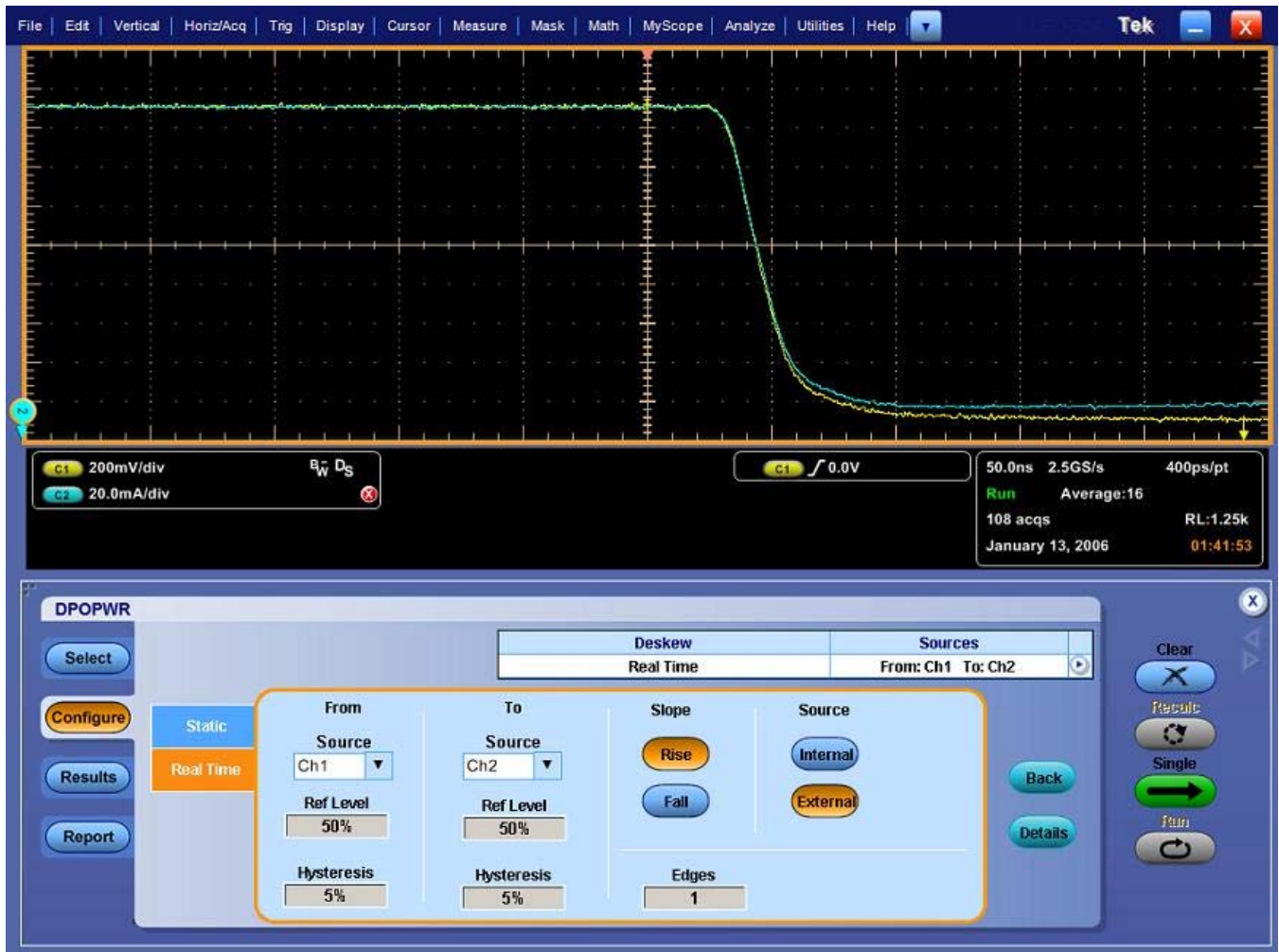


Figure 3: After deskew

- The zoomed section of the waveform available on the lower part of the screen displays the results of the deskew operation in detail. You will see that after performing the deskew operation, the skew is removed.

Note: The oscilloscope setup is saved before deskew and recalled after deskew. If the acquisition setup is in Stop condition before saving the setup, the setup with the stop condition is recalled.

When deskew recalls this setup, the application does not display the waveform. To view the waveform, press Run or Stop button in the oscilloscope.

Deskewing Using a Power Deskew Fixture

The Power Measurements Deskew Fixture with Tektronix part number (067-1686-xx) supports up to 150 A AC/DC current probes.

Follow these steps to deskew:

- Connect the AUX OUT of the oscilloscope to the J1, J2 (Cal signal input) input of the deskew fixture to deskew the voltage probe and current probe.
- Follow the steps 2-11 in the topic Deskewing in DPO7000 series oscilloscope with an internal source to complete the procedure.

SOA Overlay

Analyze> Power Analysis> SOA Overlay

Note: Select the Save button in the SOA results window to save the plot as a .plt file before using this utility.

Purpose: Plot, view and compare SOA Plots of the device operating at different load conditions. The SOA Overlay utility helps you to overlay multiple SOA plots simultaneously. You can use this feature to view multiple plots on the same grid.

Initially, the application adjusts the SOA scale according to the selected plot(s). But you can also observe the plot for the voltage and current range of operation under different conditions.

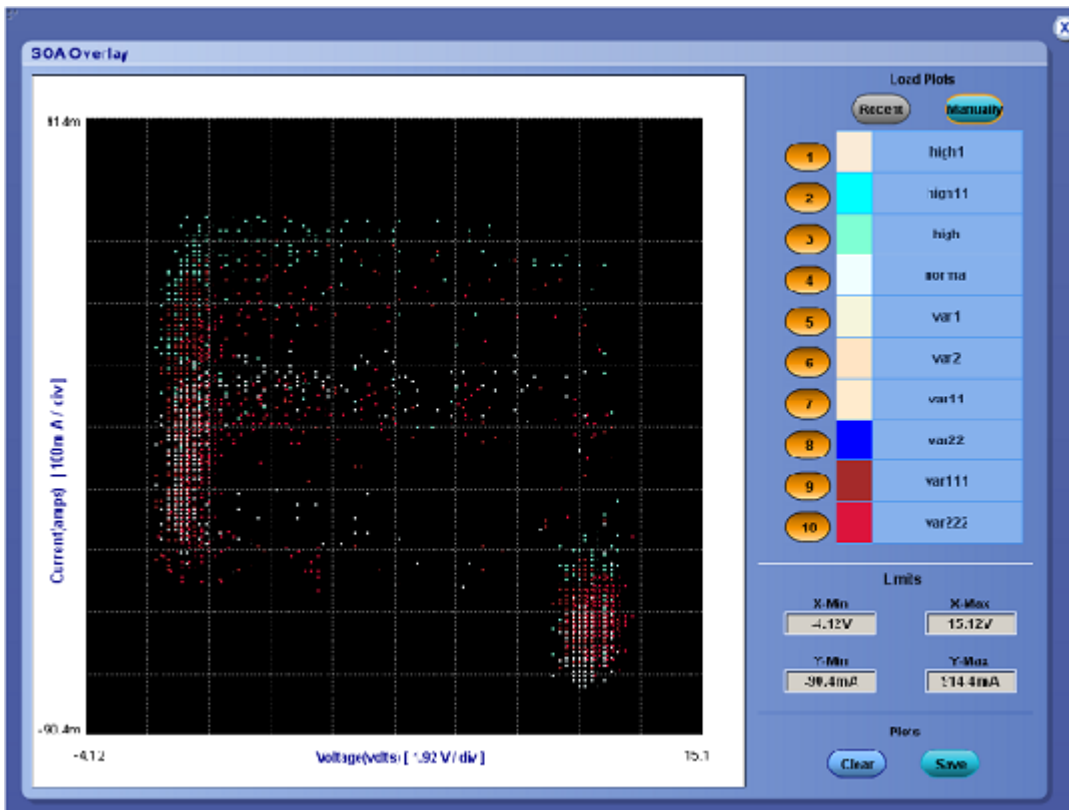




Figure 4: SOA Overlay

The Y Max, Y Min, X Max and X Min give you the flexibility to set up different operating voltage and current range to view the plot(s). The X axis represents the voltage values in volts and the Y axis represents the current in amperes.

Follow these steps to perform SOA Overlay:

1. Click the Recent button in Load Plots to automatically load the SOA plots that are saved in the current run of the application. You can load only the last ten plots. Select the Load Manually button to manually load the SOA plots from any folder or from the default folder
C:\TekApplications\DPOPWR\SOA\Data directory. You can load any of the ten plots. You can use the Shift or the Control key to select multiple plots.
2. Click the selection button (For example: ) next to the plot name to display the SOA plot on the grid area of the screen. When the utility displays the first plot, it automatically scales the plot. Continue this procedure to overlay and display all the plots. SOA Overlay displays the plot based on the color set in the Plot Name pane. The most recently selected plot is drawn over the previously selected plots. You can simultaneously view and compare different SOA plots under multiple conditions. You can change the XY limits to view the plot at a different scale.
3. Click the Plot selection button (For example: ) to remove the plot from the grid area.
4. Click the Clear button to display the message: "Do you want to clear the Plots? Press the Yes button to clear." Select Yes to clear the plots from the grid area.
5. Select the Save Plot and Legend button to save the plot as a .jpg file to any folder of your choice or to the default folder
C:\TekApplications\DPOPWR\Images.
6. Click OK to close the display window.

Operating Basics

Application Interface

The application uses a Microsoft Windows interface.

Note: The oscilloscope application shrinks to half size and appears in the top half of the display when the application is running.

Application Interface Menu Controls

The following table lists the application interface menu controls.

Table 3: Application Interface menu controls

Item	Description
Menu bar	Located at the top of the application that displays and contains application menus
Tab	Labeled group of options containing similar items
Area	Enclosed visual frame with a set of related options
Option button	Button that defines a particular command or task
Drop Down List box	Box that contains a list of items from which you can select one item
Field	Box that you can use to type in text, or to enter a value with the Keypad or a Multipurpose knob
Check Boxes	Square box that you can use to select or clear preferences
Scroll bar	Vertical or horizontal bar at the side or bottom of a display area that can be used for moving around in that area
Browse	Displays a window where you can look through a list of directories and files
Command button	Button that initiates an immediate action
Keypad	Appears when you select the box and enter a value
MP/GP knob	Select the multipurpose (MP or GP) knob to display a line between the knob and the box; turn the knob on the oscilloscope to select a value

Basic Application Functions

Using Online Help

You can use the application Help menu to access information about the oscilloscope. You can also use other oscilloscope functions and easily return to the application.

The application includes Online Help about the menus and controls.

To display the Online Help, follow these steps:

1. Choose the Help> Help Topics menu from the menu bar.
2. Use the Contents, TOC, or Index tabs to navigate through the help.
3. In the DPO7000 series oscilloscopes, the touch screen mode of operation is enabled by default. You can disable this by using the Touch Screen OFF button in the front panel of the oscilloscope.

Application Directories and File Names

The application uses directories to save and recall setup files and uses file name extensions to identify the file type.

The following table lists default directory names.

Table 4: Application directories and file names

Directory	Used For
C:\TekApplications\DPOPWR\setup	Stores the application setup files
C:\TekApplications\DPOPWR\images	Stores the images of the plot in .jpg format
C:\TekApplications\DPOPWR\Waveforms	Stores the waveforms used for the tutorial
C:\TekApplications\DPOPWR\Currentharmonics	Stores the impedance value tables and the limit tables in Current Harmonics and Total Power Quality Measurements
C:\TekApplications\DPOPWR\ReportGenerator\Templates	Stores the default templates
C:\TekApplications\DPOPWR\ReportGenerator\Layouts	Stores the default report layouts
C:\TekApplications\DPOPWR\ReportGenerator\Reports	Stores the default reports
C:\TekApplications\DPOPWR\Images\SOA.jpg	Stores the image file of the plots
C:\TekApplications\DPOPWR\soa\data*.plt	Stores the SOA data as a plot file
C:\TekApplications\DPOPWR\currentharmonics\log	Stores the results of the Current Harmonics and Total Power Quality in .csv format
C:\TekApplications\DPOPWR\soa\masks	Stores the mask file in .msk format

File Name Extensions

The following table lists the file name extensions.

Table 5: File name extensions

Extension	Description
.csv	Is a file that uses a "comma separated variable" format
.ini	Is a application setup file
.set	Is a oscilloscope setup file saved and recalled with an .ini file; both the files will have the same file name
.wfm	Is a waveform file that can be recalled into a reference memory
.rgt	Is a customized Report Generation file format, which indicates the template files
.rpl	Is a customized Report Generation file format, which indicates the defined layout files
.plt	Is a SOA data file format
.rpt	Is a customized Report Generation file format, which indicates the generated report
.jpg	Is the format of the image file of the various plots
.smk	Is the format of the SOA mask file

Returning to the Application

When you access oscilloscope functions, the oscilloscope fills the display. You can access oscilloscope functions in the following ways:

- Choose the Menu on the oscilloscope and access the application.
- Press front-panel buttons.

Setting Up DPOPWR to Take Measurements

You can set up the application to take measurements. In addition, you can view the results as a plot wherever applicable or use the report generator to generate a detailed report.

There are twenty-two measurements in three tabs.

Table of Options-Source Configuration

Table 6: Table of options-Source Configuration

Area	Option	Description
Source Configuration Panel	Voltage (V), Current (I), Gate (G)	<p>You can assign the voltage and the current channel for the selected measurement. Select the Current Source Ch1-Ch4 for the Live selection, Ref1- Ref4 for the Ref Selection.</p> <p>Selection of Ch3 and Ch4 and Ref3 and Ref4 depends on the number of channels of the oscilloscope.</p> <p>Select the Voltage Source Ch1-Ch4 and Math1-Math4 for the Live selection and Ref1-Ref4 for the Ref selection.</p> <p>Gate selection is applicable for the selected measurements. It will be in sync with gate or edge source selection under the configuration of those selected measurements.</p>
I-Probe Settings	Probe Settings	<p>Configures the selected current probe to:</p> <ul style="list-style-type: none"> • TCP Series • VPI Series • Current Shunt • Custom (Third party probes) • AM503S Settings
I-Probe Settings	AM503S Settings-Probe Type	<p>Sets the probe type and range if the probe belongs to AM503S. The probe types are:</p> <ul style="list-style-type: none"> • A63x2 • A6303/XL • A6304/XL
AM503S Settings-Range	AM503S Settings-Range	<p>Sets the range for the probe selected:</p> <ul style="list-style-type: none"> • A63x2-1 mA to 5 Amp Range:1 ma, 2 mA, 5 mA, 50 mA,100 mA, 200 mA,500 mA,1 A,2 A,5 A • A6303-5 mA to 50 A Range:10 mA, 50 mA, 100 mA, 200 mA, 500 mA, 1 A, 2 A, 5 A, 10 A, 20 A, 50 A • A6304-500 mA to200 Amp Range:500 mA, 1 A, 2 A, 5 A, 10 A, 20 A, 50 A, 100 A, 200 A
I-Probe Settings	Current Shunt	Sets the transfer impedance value in ohms to configure current shunt. The acceptable current shunt range values are 1 mohm to 10 ohms
	Custom	Sets the scale factor and propagation delay of non Tektronix probes. The acceptable range of the scale factor values are 100 p to 10 G. The acceptable range of the propagation delay factor values are 0 s to 150 ns.
	TCP Series	Sets a direct connection to the TCP series (TCP 202, TCPA with TCP series) of probes. The application does not provide any configuration parameters for this series.
	VPI Series	Sets a direct connection to the TCP series (TCP 0030) of probes. The application does not provide any configuration parameters for this series.

Table of Measurements and Configurations-Power Device Analysis

Table 7: Table of Measurements and Configurations-Switching Analysis

Measurement	SubType/Configurations	Description
Select Power Device Analysis		
Safe Operating Area (SOA)		Measures and plots the voltage versus the current plot of a power semiconductor device and indicates the operating region of the device.
SOA	SOA	Plots the Voltage and Current waveform in a single record in XY mode. The Enable Mask check box is enabled only for SOA Normal. You can use the Mask Editor to apply the mask to the SOA Plot only for SOA Normal option.
	SOA X-Y	Plots the voltage and current waveforms using the XY mode display of the oscilloscope. Use this option to measure multiple acquisitions of the same waveform in real time testing.
Switching Analysis		Analyses the power dissipated in the switching cycles across a switching device.
	Switching Loss	Measures the instantaneous minimum and maximum average energy consumed across the switching device. The loss is measured on each cycle of the acquired waveform. The results include minimum, maximum and the average loss and energy for the complete cycles in a single record.
	Hi Power Finder	Measures the Peak value of Power dissipated at the device for each switching cycle. This includes the switch finder that enables you to traverse to the switching portion of the waveform in the entire record.
Switching Loss	Ref Level	Defines the level at which the edge on a selected waveform is identified to measure the instantaneous peak power value.
	Hysteresis	Defines the tolerance band to identify the correct edge at the specified ref level.
	Percentage, Absolute	Sets the Ref Level, Hysteresis, High and Low values either in percentage of the pk-pk level or the absolute value of the waveform selected in the edge source.
	Math Destination	Defines the creation of a Power Waveform using the specified math waveform (Math1-Math4) on the oscilloscope.
RDS(on)	RDS(on)	Measures the resistance offered by the switching device during its ON condition.
	Real Time Math Destination	Sets the destination of the math waveform with a function of V/I (selected channel for V and selected channel of I).
	Average Math Destination	Sets the average math destination of Math1.

Table 7: Table of Measurements and Configurations-Switching Analysis (Contd.)

Measurement	SubType/Configurations	Description
di/dt	Measures the differentiation of the current di/dt	
dv/dt	Measures the differentiation of the voltage dv/dt	
di/dt,dv/dt	Math Destination	Sets the destination of the math waveform with the differential function of the current waveform.
di/dt, dv/dt	Ref Level	Defines the level at which the edge on a voltage waveform for dv/dt and current waveform for di/dt is identified.
	Hysteresis	Defines the tolerance band to identify the correct edge at the specified ref level.
	Percentage, Absolute	Sets the Ref and Hysteresis level either in percentage of the pk-pk level or the absolute value of the pk-pk value of the waveform selected as the edge source.

Table 8: Table of Measurements and Configurations-Modulation Analysis

Measurement	Sub Type/Configurations	Description
Pulse Width		Is a trend plot of the pulse width variation for the acquired waveform. It is the time difference between the leading and the trailing edge of a pulse.
	Mid	Defines the required edge level in percentage or volts.
	Hysteresis	Defines the tolerance band used to identify the correct edge at the specific Ref level.
	Positive	Defines the positive pulse width.
	Negative	Defines the negative pulse width.
Period		Is a trend plot of the variation of the period measured from a leading edge to a leading edge or a trailing edge to a trailing edge.
	Mid	Measures the required edge level in percentage or volts.
	Hysteresis	Defines the tolerance band used to identify the correct edge at the specific Ref level.
	Rise	Measures the time from the leading edge of the waveform to the next leading edge.
	Fall	Measures the edges from the trailing edge of the waveform to the next trailing edge.
Duty Cycle		Measures the duty cycle of the waveform and is a trend plot. The duty cycle can be a positive or a negative duty cycle.
	Mid	Defines the required edge level in percentage or volts.
	Hysteresis	Defines the tolerance band used to identify the correct edge at the specific Ref level.
	Positive	Measures the positive pulse width.
	Negative	Measures the negative pulse width.
	Rise	Measures the edges from the leading edge of the waveform.
	Fall	Measures the edges from the trailing edge of the waveform.
Frequency		Is a trend plot of the variation of the frequency of the signal.
	Mid	Defines the required edge level in percentage or volts.
	Hysteresis	Defines the tolerance band used to identify the correct edge at the specific Ref level.
	Rise	Measures the edges from the leading edge of the waveform.
	Fall	Measures the edges from the trailing edge of the waveform.

Table of Measurements and Configurations-Magnetics

Table 9: Table of Measurements and Configurations-Magnetics

Measurement	SubType/Configurations	Description
Magnetics		Monitors and measures the parameter associated with a magnetic component. The available options are: Magnetic Loss, Inductance, I vs Integral V Plot and Magnetic Property.
Magnetics	I vs Integral V	Plots the integral of the voltage waveform and the current waveform in a XY plot. Integral of V is proportional to B and the current waveform I is proportional to H.
	Magnetic Loss	Measures the power loss at the magnetic component.
	Magnetic Property	Measures the parameters related to the magnetic material properties in the magnetic component used in an in-circuit operation.
	Inductance	Measures the inductance of the DUT (inductor) and displays the proportion (B-H) curve plot with the inductance value.

Table of Measurements and Configurations- Input/Output Analysis

Table 10: Table of Measurements and Configurations-Input/Output Analysis

Measurement	SubType/ Configurations	Description
Select Input/Output Analysis		
Power Quality		Measures and displays the: <ul style="list-style-type: none"> • Root mean square value for Voltage • Root mean square value for Current • True Power • Apparent Power • Power Factor • Crest Factor for Voltage • Crest Factor for Current
Power Quality	Energy Waveform Destination	Enables the energy waveform based on the math option available on the oscilloscope.
	Power Waveform Destination	Enables the power waveform and creates the math waveform.
Current Harmonics	Harmonic Table	Captures the predefined limit values for the harmonics. This will be used to compare the measured value of the harmonics.
	I Probe Table	The impedance of the current probe at the frequency can be edited in this table. This is used to calculate the current harmonics value. In the default state, I-probe is not enabled.
	Standard> Type> 61000-3-2	Sets the limit table based on the class categories of the unit under test and updates the selected table with the standard values. Configures to the different classes of equipment like: <ul style="list-style-type: none"> • Class A-Balanced three phase equipment and all other equipment except that stated in one of the following classes • Class B-Portable tools, arc welding equipment • Class C-Lighting equipment including dimming devices • Class D-Equipment with an input current of a special wave shape and an active input power of P600W
	Standard> Type> AM - 14	Sets the line frequency and the limit table based on the Class categories. The Controls option is applicable only for Class C and D. These values will be verified by the application if the calculated Power Factor, Input Power and the Fundamental Current are within ten percent range of the manufacturer's rating entered in the Control field.
	Standard> Type> MIL 1399	Sets the line frequency to 60 Hz or 400 Hz.
	Controls	Displays the Input Power, Power Factor and Fundamental Current values for Class C. Displays the Input Power values for Class D.
	Filter	Smoothens the result of current harmonics.
	Line Frequency	Measures the frequency of the line operation. They can be of two types: 50 Hz or 60 Hz for IEC 61000-3-2 Standard and 60 Hz Line or 400 Hz line for MIL 1399 standard.

Table 10: Table of Measurements and Configurations-Input/Output Analysis (Contd.)

Measurement	SubType/ Configurations	Description
	Harmonics	Displays the user configurable features like harmonic table and group classification.
Total Power Quality	Total Power Quality	Measures and displays: <ul style="list-style-type: none"> • Root mean square value of the current and voltage • True power • Apparent power • Crest factor of current and voltage, Current harmonics • Power factor • Total harmonic distortion • It is a combination of Power Quality and Current Harmonics measurement.
Total Power Quality	Line Frequency	Measures the operating line frequency. They can be of two types: 50 Hz or 60 Hz for IEC 61000-3-2 Standard and 60 Hz Line or 400 Hz line for MIL 1399 standard.
	Harmonics	Displays the user configurable features like Harmonic Table and Group Classification.
	Harmonic Table	Captures the limit values for the harmonics which are used to compare the measured value of the harmonics.
	Standard> Type> 61000-3-2	Allows you to select the limit table based on the Class categories of the unit under test. The selected table is updated to the standard values. For Class C and D, the application automatically calculates and the updates the limits table before comparing the harmonic results.
Total Power Quality	I Probe	The impedance of the current probe and the frequency can be edited in this table and is used to calculate the current harmonics value. In the default state, I-probe is not enabled.
	I Probe	Impedance Table Measures the frequency response of the entire current probe system on the current harmonics measured value.
	Math Destination	Creates the power waveform and energy at the selected math destination.

Table 10: Table of Measurements and Configurations-Input/Output Analysis (Contd.)

Measurement	Sub Type/Configurations	Description
Select Input/Output Analysis		
Line Ripple		Measures the: <ul style="list-style-type: none"> AC component voltage present on a DC Low frequency AC voltage in the peak to peak voltage. Sets the time base to measure the Ripple at line Frequency to 50 Hz, 60 Hz or 400 Hz
	Coupling	Selects the in built coupling on the oscilloscope based on two options: AC Coupling and DC Coupling.
	Bandwidth	Removes the high frequency component on the signal. There are three bandwidth options: 20 MHz, 150 MHz and Full. Full is the maximum bandwidth available in the oscilloscope.
	Acquisition Mode	Types of acquisition modes are: <ul style="list-style-type: none"> Peak Detect Hi Resolution Sample
	High Resolution	Measures the average sample taken during an acquisition interval to create a record point. This results in a higher-resolution, lower-bandwidth waveform with a real-time, non-interpolated sampling.
	Pk Detect	Works only with real-time, noninterpolated sampling. The acquisition alternates between saving the highest sample in one acquisition interval and lowest sample in the next acquisition interval.
Switching Ripple		Measures the maximum AC voltage present on a DC
	Coupling	Sets the in built coupling on the oscilloscope to: AC Coupling or DC Coupling.
Switching Ripple	Bandwidth	There are three bandwidth options: 20 Mhz, 250 Mhz and Full. Full is the maximum bandwidth available in the oscilloscope.
	Switching Frequency	Uses the GP knob to enter the switching frequency values. Min: 50 Kz, Max: 1 MHz, Default: 10 KHz.
	Acquisition Mode	Sets the oscilloscope acquisition mode which are set to: <ul style="list-style-type: none"> Pk Detect Hi Resolution Sample
	Peak Detect	Works only with real-time, noninterpolated sampling. The acquisition alternates between saving the highest sample in one acquisition interval and lowest sample in the next acquisition interval.

Table 10: Table of Measurements and Configurations-Input/Output Analysis (Contd.)

Measurement	SubType/ Configurations	Description
Turn On Time		Measures the time delay difference between the input voltage applied to the system and the time to develop the steady state output voltage.
	V Source	Selects source V channel for the input line voltage source.
	Maximum Voltage	Measures the input voltage as a RMS value of AC or DC.
	Trigger Level	Sets the trigger level for the line voltage source.
	Output Voltages	Configures the parameters to select the oscilloscope channel, which is to be connected to the output voltage of the unit under test. The Output 2 and 3 panes are enabled for four channel oscilloscopes.
	Maximum Value	Measures the maximum output voltage of the unit under test connected to the selected channel.
	Max Turn on Time	Measures the time taken by the unit under test to develop a stable output voltage after applying input voltage.
	Converter Selection	Measures the operating input line frequency AC-DC or DC-DC.
Select Spectral Analysis		
Spectral Analysis		Analyzes, measures and displays the frequency component in a signal based on the selected Start, Stop and RBW values.
Spectral Analysis	Source	Connects the input source for spectral analysis.
	Frequency Start-Stop	Defines the frequency range for which the input signal is analyzed.
	Window Type	Specifies the window type to reduce the spectral leakage when using the spectral analysis tool. The available window types are: Rectangular, Hamming, Hanning, Black-Harris, Gaussian, Flattop2, Kaiser-Bessel and TekExp.
	Auto Setup	Makes the horizontal setup to capture all the frequency component in a signal based on Start, Stop and RBW values.
	DC Block	Removes the DC component by adding the offset and adjusting the vertical sensitivity.
	Res BW	Displays the resolution and bandwidth values based on the Start, Stop and Window Type inputs defined by you.

Taking a New Measurement

To take measurements:

1. Connect the required voltage and the current probe to the unit under test.
2. Set the Current source and the Voltage Configuration source on the Source Configuration Panel.

Note: When you run any measurement, Equivalent time and Sampling mode in the oscilloscope should be set to OFF state.



Warning: *When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.*

Analyzing the Results

You can select and run the measurements to display the results after analyzing the acquired waveform. You can also refer to Viewing Results for the different measurements and use the Report Generator tool to create customized reports.

Selecting a Measurement

To take a measurement, select Analyze> Power Analysis> Select, which is the default opening screen in the application.

There are three categories of measurements: Power Device, Magnetics, and Input/Output Analysis.

The measurements for the different categories are shown in Table 11.

Table 11: Table of Measurements and Configurations-Line Power Analysis

Power Device		Magnetics	Input Output Analysis	
Switching Analysis	Modulation Analysis		Input Analysis	Output Analysis
Switching Loss	Pulse Width	Inductance	Power Quality	Line Ripple
Hi-Power Finder	Period	Magnetic Loss	Current Harmonics	Switching Ripple
SOA	Duty Cycle	Magnetic Property	Total Power Quality	Spectral Analysis
SOA X-Y	Frequency	I vs J V		Turn On Time
dv/dt				
di/dt				
RDS(on)				

Select the measurements and configure the application based on waveform source, voltage, current and probe settings.

Source Configuration Panel

The source configuration panel on the left side of the screen has the common configuration options: Source, Voltage, Current, Gate, and I-Probe Settings.

Follow the steps to configure the options in the Source Configuration panel:

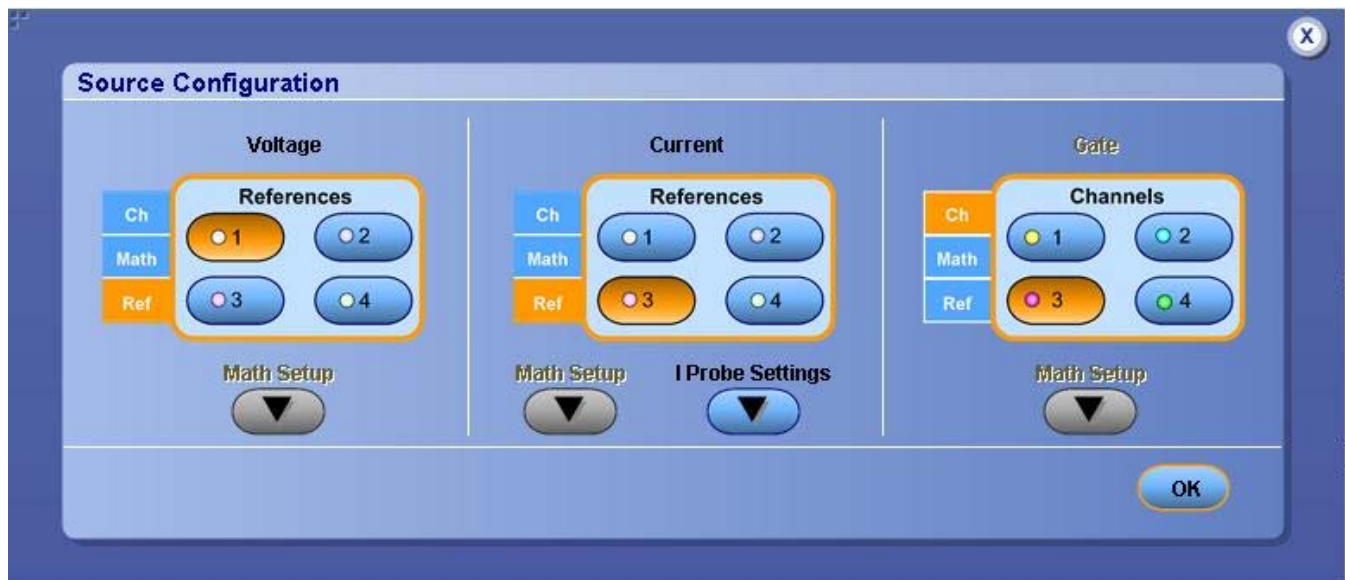


Figure 5: Source Configuration Panel



Warning: When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.

1. Select the Ref option to use the reference waveform of the oscilloscope. Select the Live option to acquire the waveform from the unit under test. When you select the Ref option, deskew and I-Probe Settings are disabled. The default selection is Live.
 - If you select the Live source, the default Voltage source is Ch1 and the Current source is Ch2. The available selections are: Ch1, Ch2, Ch3, Ch4, Math1, Math2, Math3 and Math4. The Math selections are available only if you select Live and not Ref. Click here for information on what the application does if you select the Math option. Use the drop down arrow in the Voltage and the Current field to select specific channels. In Line Ripple and Switching Ripple measurement, you can select only Live source. The available selections are: Ch1, Ch2, Ch3 and Ch4. You do not have the Math selections here. When the Current field is disabled, the I-Probe settings button is also disabled.
 - If you select the Ref option, the default Voltage source is Ref1 and the Current source is Ref2. You can also use the drop down arrow in the Voltage and the Current field to select specific channels.

Note: When Math is chosen as voltage source, it should be independent of the selected math destination. The math destination will be defined with a predefined function only if the selected math destination is independent of any selected measurement or any math definition.

You cannot select the same channel source in the Voltage and Current field.
Example: If you select Ch1 in the Voltage field and Ch1 in the Current field and select Run, the application displays an error message "Conflict in selection of voltage source and current source." The Voltage and Current Source Configuration is dependent on the measurement selected.

2. Click I-Probe Settings to display TCP, Shunt, Custom and AM503S Settings. Click the TCP button if you are using TCP series probes.

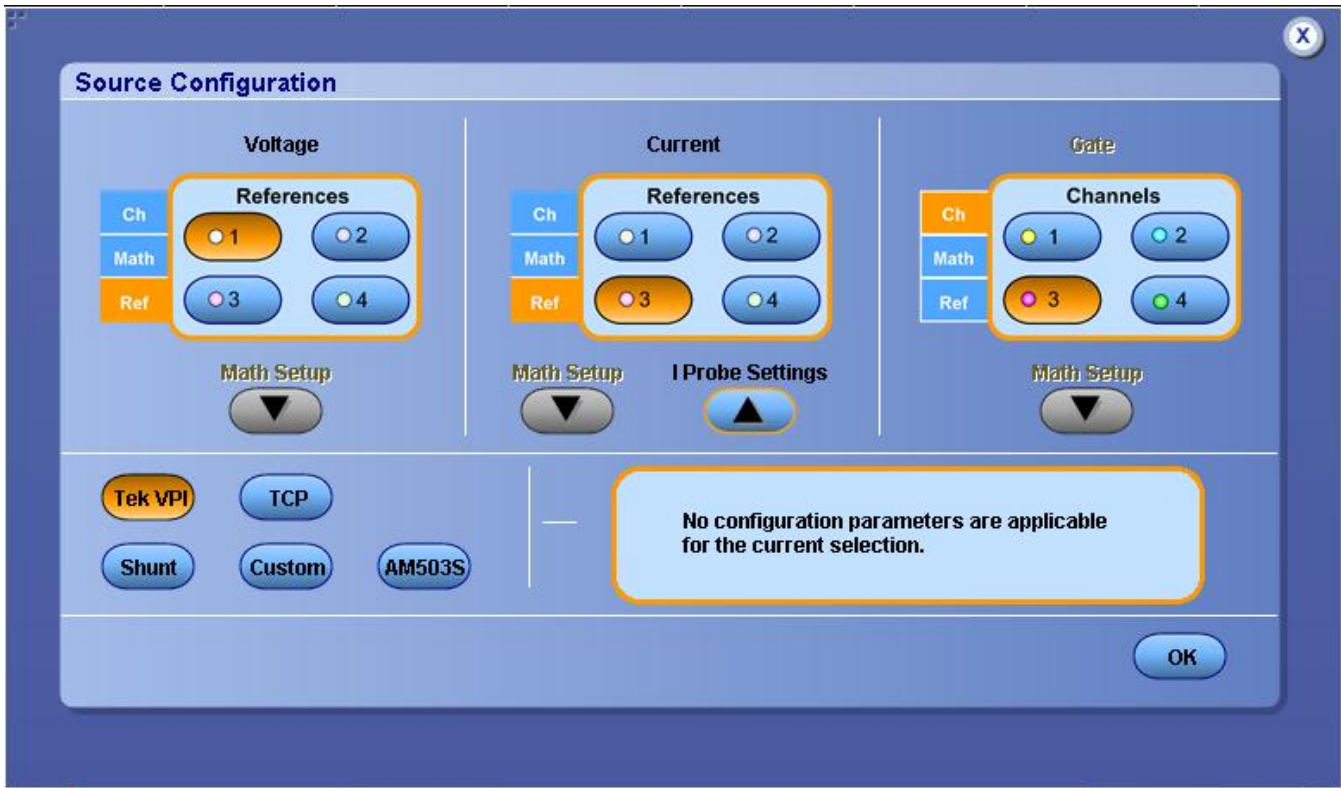


Figure 6: I-Probe settings

Note: If you are using a TCA-1Meg probe, terminate the input side with 50 Ohms for the current probes that does not automatically change the termination. If you are using a TCA to BNC probe, you do not have to terminate the input side.

Math Setup

Follow these steps to configure the Math setup:

1. Select the Math tab in the Voltage pane to set Math as the voltage source. The application replaces the measurement configuration screen and displays the Math Setup pane.

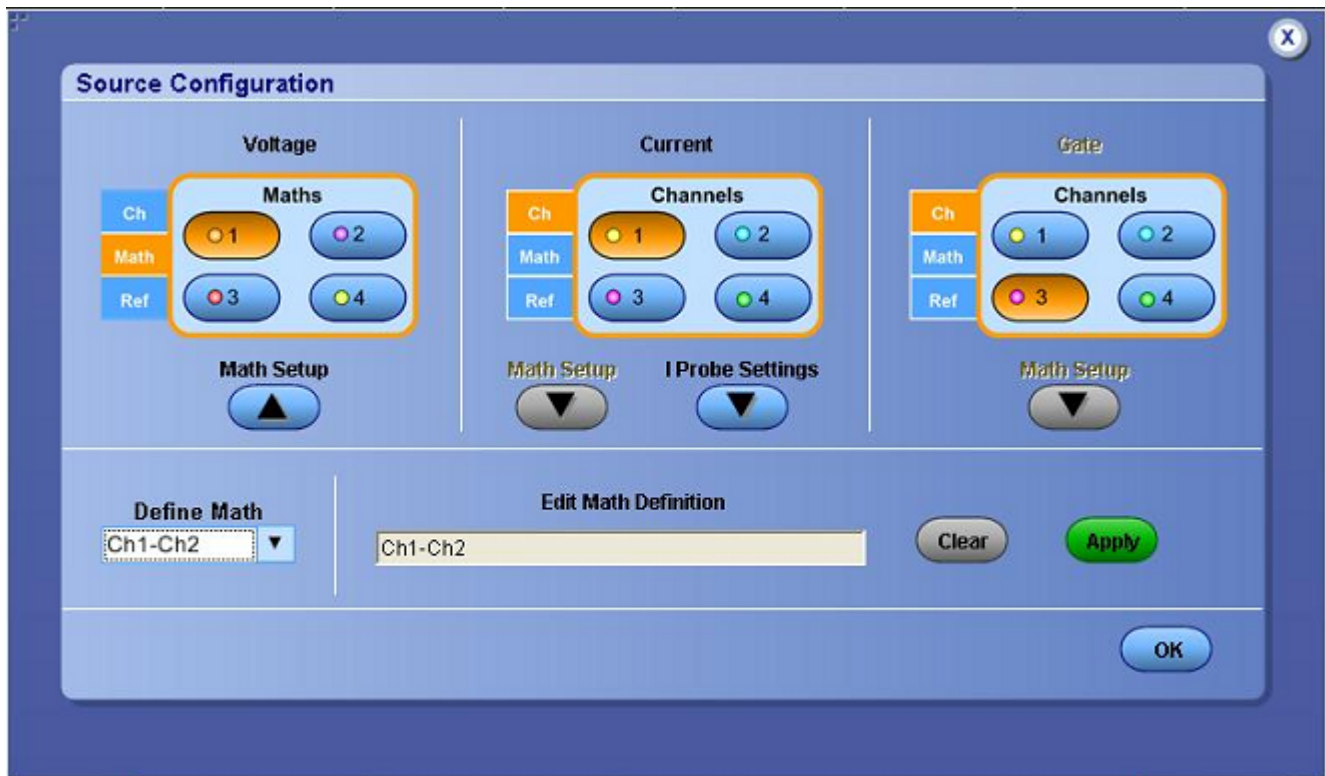


Figure 7: Math setup

2. In the Define Math field, use the drop down arrow to define the Math setup. The available options are:
 - Scope user is the default setup
 - Ch1-Ch2
 - Ch1*Ch2
 - Ch3-Ch4
 - Ch3*Ch4
 - User defined

If you select Scope User, the application displays the existing Math defined in the oscilloscope in the Edit Math Destination field. If you select the User defined option, you can use the popup keyboard in Edit Math Destination field to edit the Math values.

3. Select the Apply button to apply the selected expression (- or *) on the selected math channel in the oscilloscope.
4. Select the Clear button to clear the value from the Edit Math Destination.
5. Select the OK button to apply the math definition (if you have defined it correctly) and revert back to the measurement configuration screen.

Troubleshooting Math Setup error messages

If you select the User defined option and specify the wrong definition in the Define/Edit Expression field, the application displays the error message "Invalid Math definition. Redefine Math". A wrong definition can be a wrong syntax, empty definition string, invalid characters, invalid channel selection or circular definition.

SOA Mask Editor

Selecting and Configuring Measurements

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar to display the default screen.
2. Click the Power Device tab. Click SOA in the Switching Analysis panel.

Follow the steps to use the SOA Mask Editor:

1. Select the Enable button in the Mask pane if you want to apply the mask to the SOA plot.

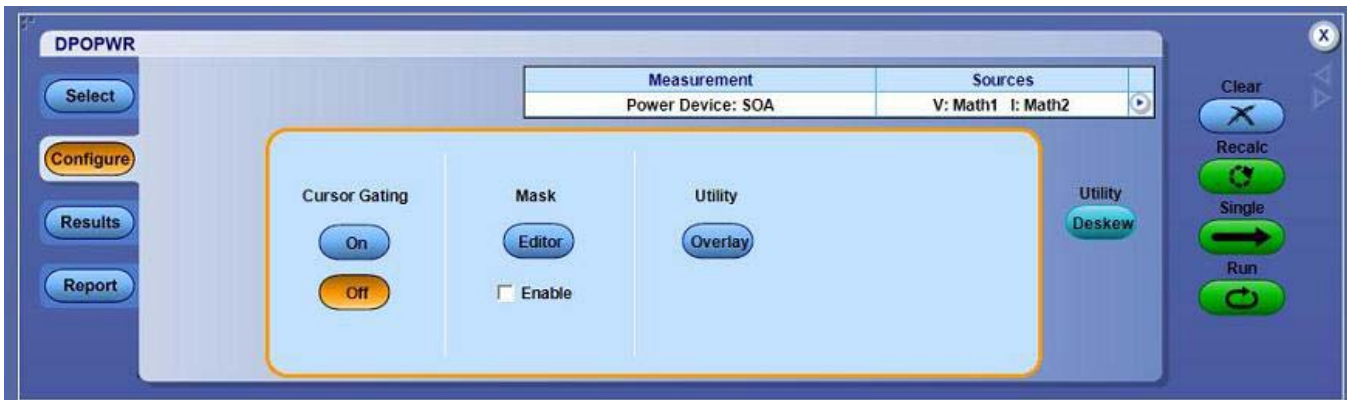


Figure 8: SOA Mask editor configuration

2. Click the Editor button to display the SOA Mask Editor as shown in the next screen.

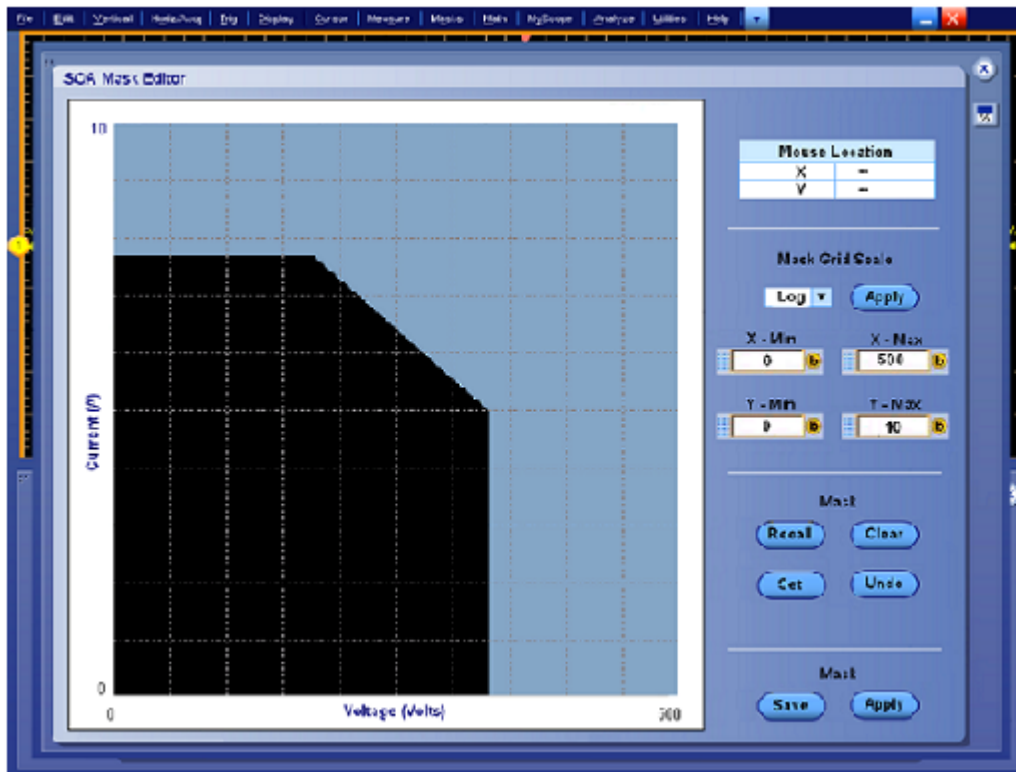


Figure 9: SOA mask editor

3. Use the mouse to draw an open-ended mask on the grid and click the Set button. Click the mouse and select the exact points that identify the mask polygon and complete the mask by double clicking at the last point. Click the Set button to set the mask and identify the Pass or Fail zones.
4. Select the Log or Linear option in the Mask Grid Scale combo box to set the Log or Linear scale for the mask. Click Apply.
5. Double-click the X-Max, X-Min, Y-Max and Y-Min, fields and use the keypad that appears to set the XY coordinates of mask values. The default values of X Min is 0 V, X Max is 250 V, Y Min is 0 A and Y Max is 10 A.

You can use the Recall option to retrieve the saved mask from the default directory or the directory where you have saved the mask.

6. Configure SOA and click Run to display the SOA Plot with the mask in the background.
7. Click the Save button to save the mask as an .smk file in the default directory C:\TekApplications\DPOPWR\SOA\Masks or in any other folder. The Mask editor identifies the Pass/Fail regions on the SOA plot and displays them in different colors. You can also set the Mask Values and define masks. You can also change the mask dynamically and view different masks on the same SOA plot. Select the Clear button to clear the mask and the Undo button to undo the previous action.

If the data points fall inside the mask zone, they show in green to indicate pass and if they fall outside the mask zone, they show in red to indicate fail.

***Note:** If the x2 or y2 mask value is less than x1 or y1, the application displays an error message "Max should be greater than Min." You can only draw open-ended masks. You cannot switch the scale after you have drawn the mask. If you do, the application clears the mask.*

Power Device Analysis

Hi-Power Finder

Selecting and Configuring Measurements

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar to display the default screen.
2. Click Power Device to display the power device screen.
3. Click Hi-Power Finder in the Switching Analysis pane. Click Configure.

***Note:** The waveform height should be at least 2-4 divisions to take the measurements. If the voltage waveform is less than a division, the application displays an error message "Unable to find the correct edges". This is due to the high variation in points in a single division and an insufficient hysteresis band to find the correct edges. The vertical range of the voltage waveform should be greater than one division.*

We recommend that you use the Hi-Res acquisition mode to analyze switching loss.

4. Click Hi-Power Finder. HiPower Finder calculates the instantaneous peak power on the switching waveform. The results vary according to the selected measurement.
5. To calculate HiPower Finder, click the Configure button. You can configure the parameters by clicking the Type, On-Off Level and Options buttons.

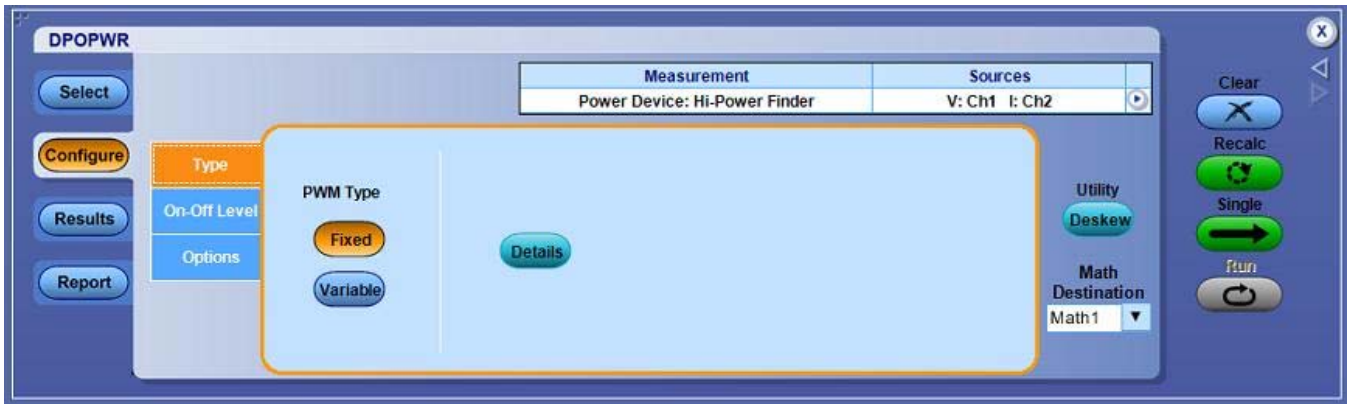


Figure 10: Hi-Power Finder

Hi-Power Finder: Type

Follow the steps given below to configure the Hi-Power Finder parameters in the Type tab:

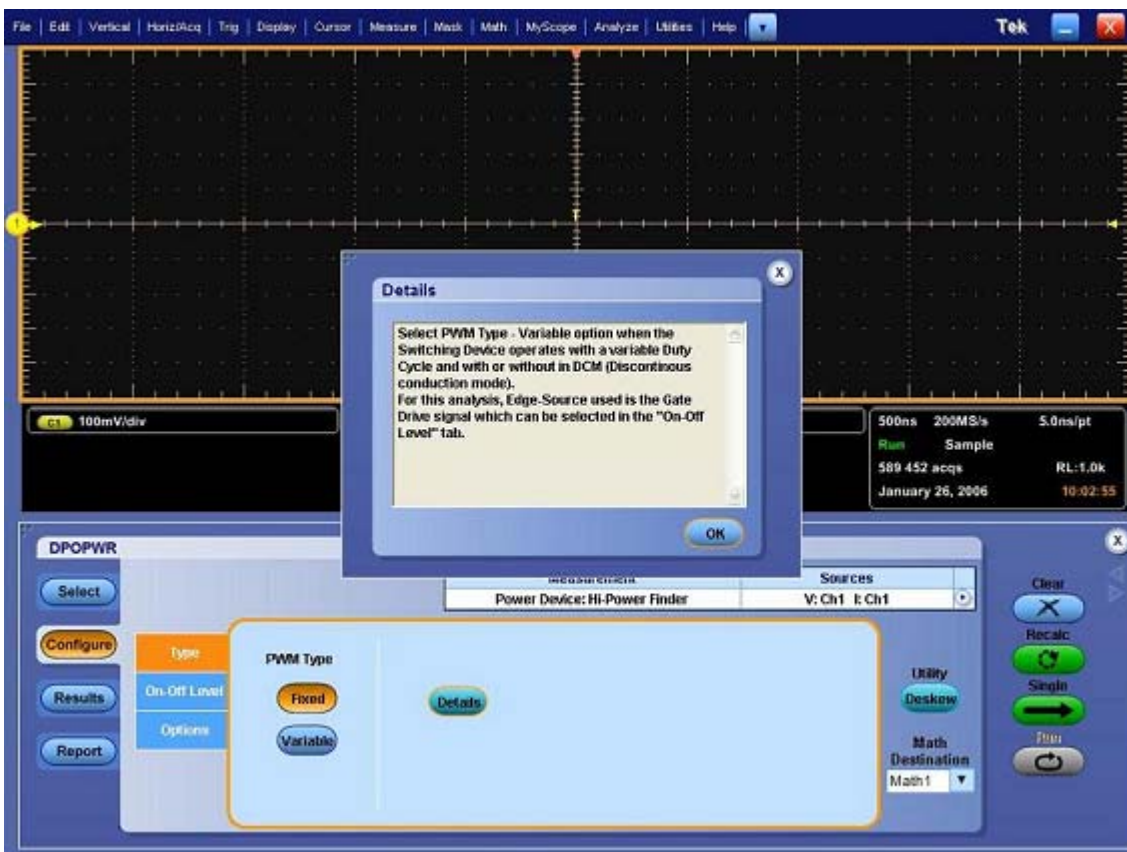


Figure 11: Hi-Power Finder: Type

1. Click the Variable button if a switching device operates with a variable Duty Cycle with or without Discontinuous Conduction Mode. If the Variable option is selected, the voltage edge source selection in the On-Off Level tab switches to Vg (gate drive signal). If the Fixed option is selected, the application measures the switching loss for all the topologies with a constant duty cycle.
2. Select the math destination from the drop down menu, Math Destination. The default destination is Math1.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

High Power Finder: Edge Source

Follow the steps given below to configure the Hi-Power Finder parameters in the Edge Source pane:

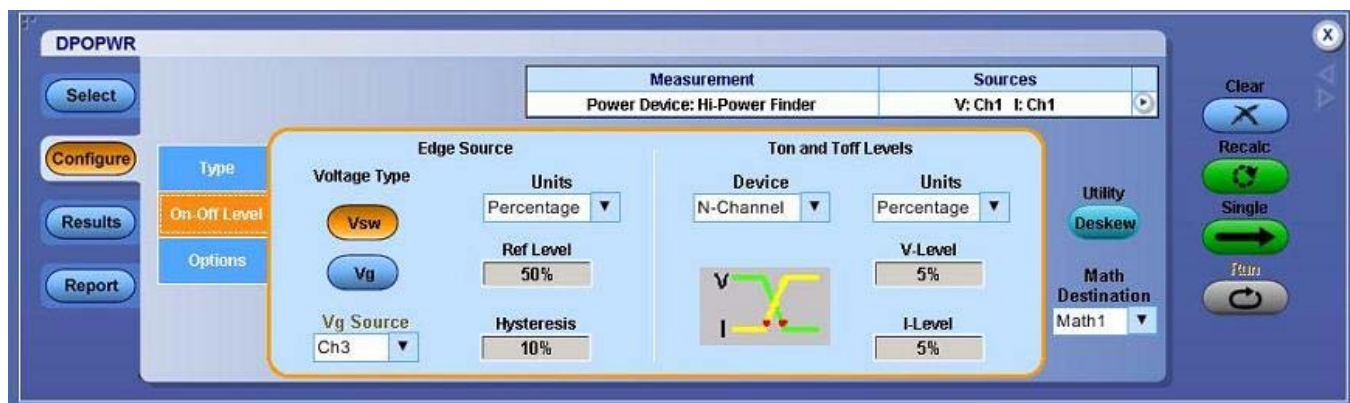


Figure 12: Hi-Power Finder: Edge Source

1. In the Voltage Type pane, select the Vsw or the Vg option button to set the edge source. The default option is Vsw. If you have selected Vsw, the Edge Source is the Voltage source configured in the Source Configuration Panel. If you have selected Vg from the Vg Source drop down list, select the Vg Source. The default is Ch1.

If you select Vsw, the application identifies:

- The edges on the switch voltage
- The Switch On and Switch Off portions using the switch voltage and switch current

If you select Vg, the oscilloscope requires at least three channels to acquire the data and the application identifies:

- The edges using the Vg (gate voltage)

- The Switch On and Switch Off portions using the Gate Voltage, Switch Voltage and Switch Current
2. Select the Percentage option in the Units drop down box to set the Ref Level and Hysteresis values as a percentage or the Absolute option to set the absolute value of the peak to peak signal.
 3. Double-click the Ref Level field and use the keypad that appears to set the absolute or the percentage of the ref level value or double-click the Hysteresis field and use the keypad that appears to set the absolute or the percentage of the hysteresis value.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

HiPower Finder: Ton and Toff Levels

Follow the steps given below to configure the HiPower Finder parameters in the in the Ton and Toff Levels pane:

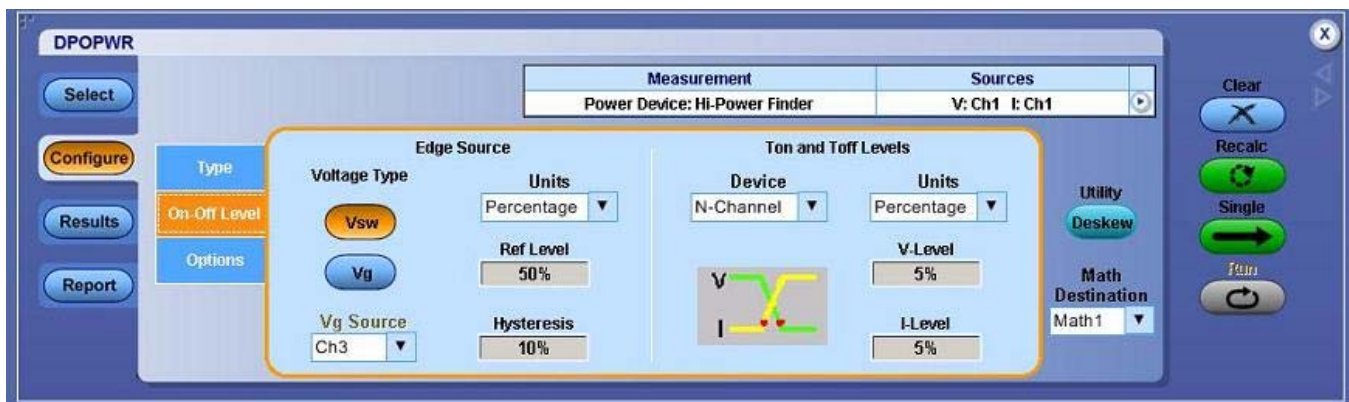


Figure 13: Hi-Power Finder: Ton and Toff Levels

1. Select the semiconductor type (DUT) from the Device drop down menu. The available options are: N-Channel and P-Channel. In an N-Channel device, the Switch Voltage, Switch Current and the Gate Voltage are positive. In a P-Channel device, the Gate Voltage can either be positive and negative or only negative.
2. Select the Percentage option in the Units drop down box to set the V-Level and I-Level values in percentage of the peak to peak signal or select the Absolute option to set the absolute value of V-Level and I-Level in amperes. The V-level and I-level values should be less than the sum of the Ref Level and Hysteresis divided by two.
3. Double-click the V-Level field and use the keypad that appears to set the absolute or the percentage of the v-level value. The V-Level value for the Start of Ton is five percent of the switch current and for Stop of Ton is five percent of the maximum switch voltage.

4. Double-click the I-Level field and use the keypad that appears to set the absolute or the percentage of the I-level value.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

HiPower Finder Options

Follow the steps given below to configure the HiPower Finder parameters in the Option pane:

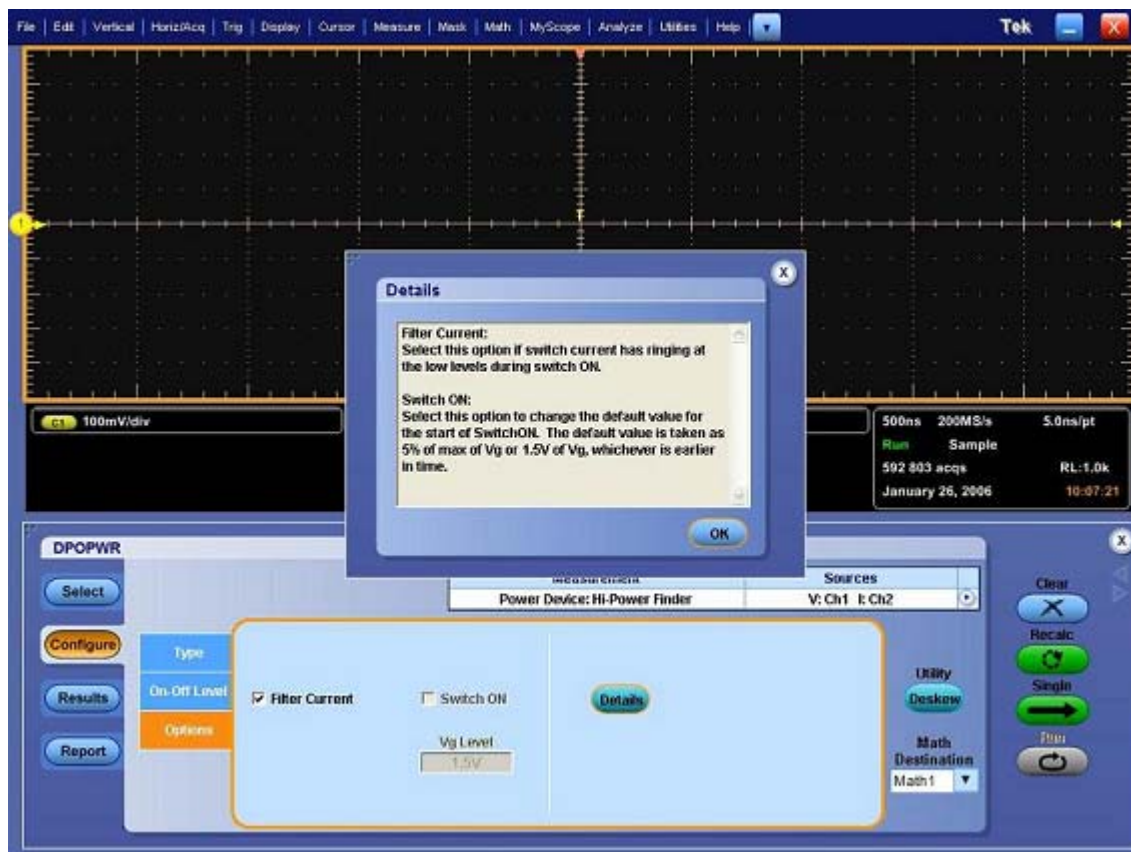


Figure 14: Hi-Power Finder-Option

1. Click Option button to enable the Signal Condition, Filter Current and Switch On options. If you select the Filter Current option, the switch current has ringing at the low levels during switch ON. If you select the Switch ON option, you can change the default value of the start of switch ON. The default value is 5% of the maximum of V_g or 1.5 V of V_g , whichever is lower.
2. Double-click the V_g Level field and use the keypad that appears to enter the values. The acceptable range of V_g values are 0 V to 50 V and the default value is 1.5 V. If you have selected the V_g option in the Control Signal tab as the Edge Source, the V_g level is used to calculate the Start of Ton.

Note: If you select the Switch ON option, the Vg Level field is enabled.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Viewing Hi-Power Finder Results

Note: Do not change or touch the oscilloscope settings during the post analysis of the results. This could lead to incorrect interpretation of results.

Follow these steps to view the results:

3. Select Analyze> Power Analysis> Results from the oscilloscope menu bar.
4. The application displays the results for Hi-Power Finder measurement as shown below.



Figure 15: HiPower Finder results: Peak finder tab

5. You can view the results of the Peak Finder and Switching Finder options.

In the Peak Finder tab options:

- The Summary displays maximum and minimum value of the power waveform peaks. It also displays the occurrences or positions of the On type and Off type peaks in a single waveform.
- In the Range panel, End and Start fields display the peaks between a range. Double-click these fields and use the keypad that appears to enter the numeric inputs. Click Update.
- The Peak Values panel displays the peaks between End range and Start range in descending order. Click Link to identify the peak portion in the voltage and current waveform. Use the Prev button to move the cursor position to the previous peak from the linked position. Use the Next button to move the cursor position to the next peak from the linked position. By default, the application positions the cursors on the first peak.
- Click the Zoom to zoom in at the cursor location.

In the Switch Finder tab:

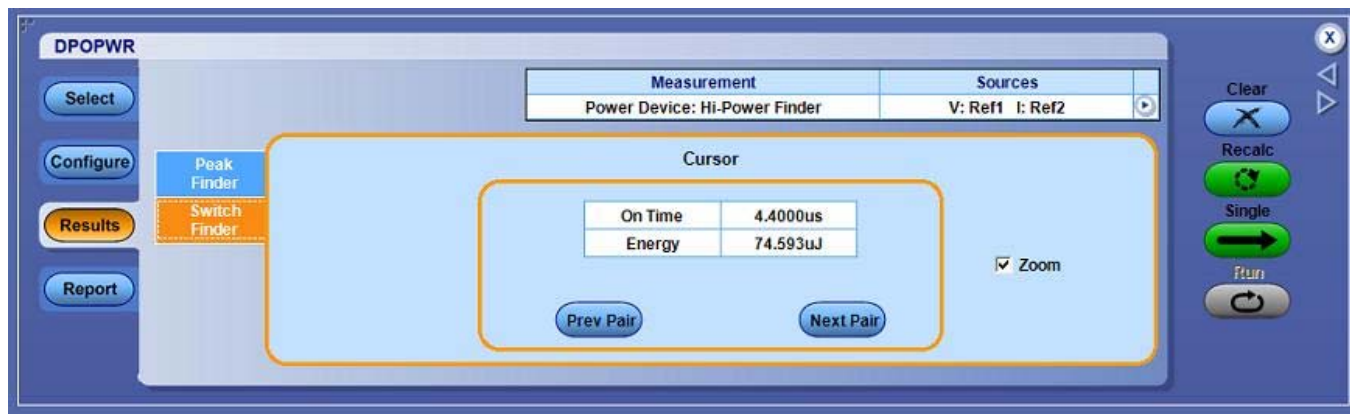


Figure 16: Switch Finder tab

- The application displays On Time and Energy results between the cursors. Energy is the amount of energy dissipated from On to Off of the waveform.
- On Time is the conduction time for any switching device. You can move the Peak Pair Cursors depending on the order of their placement.
- Use the Prev button to move the cursor to the previous cursors and the Next to the next pair of cursors.

Note: *If the current waveform is less than a division, the application displays an error message: "Unable to find the correct edges". This is because the variation of data points with in single division is very high and the hysteresis band is insufficient to find the correct edges. Therefore, for the voltage waveform, the vertical range should be greater than one division.*

Note: Negative Switching loss results may be due to the DC offset in the voltage and current probes used. To avoid this, compensate the probes for DC offset and also run the Signal Path Compensation option (SPC) on the oscilloscope before starting the application.

Generating Reports-Hi-Power Finder

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar, to display Report Generator screen.
2. The application provides default factory templates for this measurement. The template file name is HiPower_Finder.rgt for High Power Finder measurement and the layout file name is Power_Device.rpl. When you click Generate Report, this template file name is displayed by default. If you want to select or modify another template, follow the procedure given here.
3. Select the New Template button in Define Test Template tab to define and create a template for Power Device Analysis. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Switching Loss

Selecting and Configuring Measurements

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar to display the default screen.
2. Click Power Device to display the power device screen.
3. Click Switching Loss in the Switching Analysis pane. Click Configure.
4. The Switching Loss option calculates the amount of power dissipated across the switching device under the steady state of operation. The results vary according to the selected measurement.
5. Click the Configure button. You can configure the parameters by clicking the Type, On-Off Level and Options buttons.

Switching Loss Type

In the Type tab, use the following steps to configure the Switching Loss parameters:

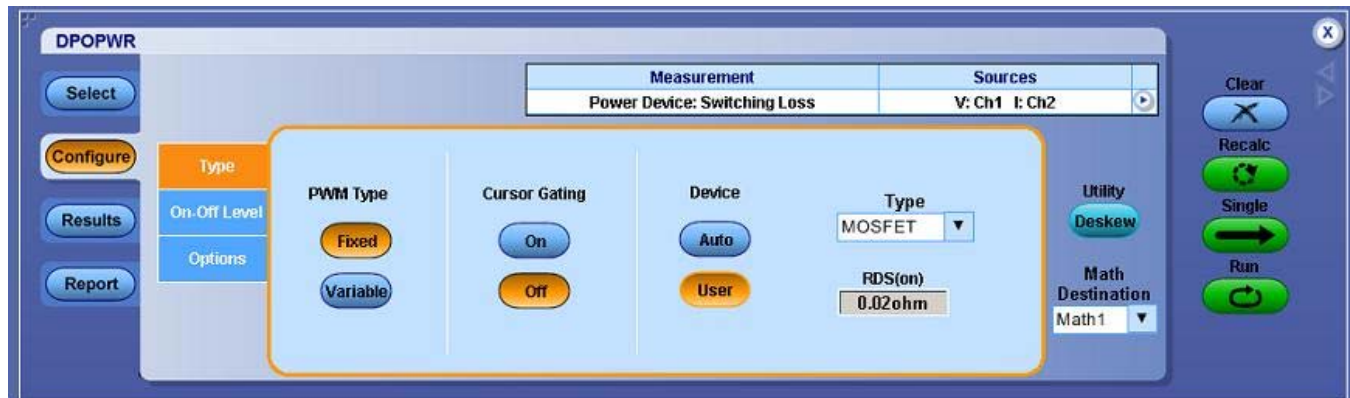


Figure 17: Switching Loss: Type

1. Select the Variable option if a switching device operates with a variable Duty Cycle with or without a Discontinuous Conduction Mode. If the Variable option is selected, the voltage edge source selection in the Control Signal tab switches to Vg (gate drive signal). If the Fixed option is selected, the application measures the switching loss for all the topologies with a constant duty cycle.

Use Vg as the Edge Source in the Edge Source pane if:

- The switching voltage or current switches between the Discontinuous Mode and the Continuous Mode.
- You use a ringing switch voltage to measure edge analysis.

If you select the Variable option, you can:

- Set the cursor gating to On or Off.
- Configure the Edge Source as Vg or Vsw in the Edge Source pane.

2. Select the Cursor Gating ON option to place the cursor at the specified portion of the waveform. In a Variable duty cycle, you can place the cursor at the half cycle of the input main signal.

Note: If you select Variable, the Cursor Gating option is enabled and the voltage Edge Source in the Control Signal tab is Vg.

3. In the Device pane, click the User button to enable the switching device type options. Select either the MOSFET or the BJT/GBT switching device. If you select MOSFET, double-click the RDS field and use the keypad that appears to enter the RDS value that is available in the data sheet, for the specified temperature of operation. If you select the Auto button, the application calculates the total power loss and energy using the switch voltage and

current and checks for the vertical scale of the switch voltage. If the switch voltage scale is greater than ten volts per division, the application displays a warning message: "Vertical Scale of Switch Voltage > 10 V and the user defined RDSON is not selected. So the calculated energy and loss may not be accurate."

4. Select the math destination from the Math Destination drop down menu, PWR Wfm. The default destination is Math1.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Switching Loss Edge Source

In the Edge Source pane, follow the steps given to configure the Switching Loss Control Signal parameters:

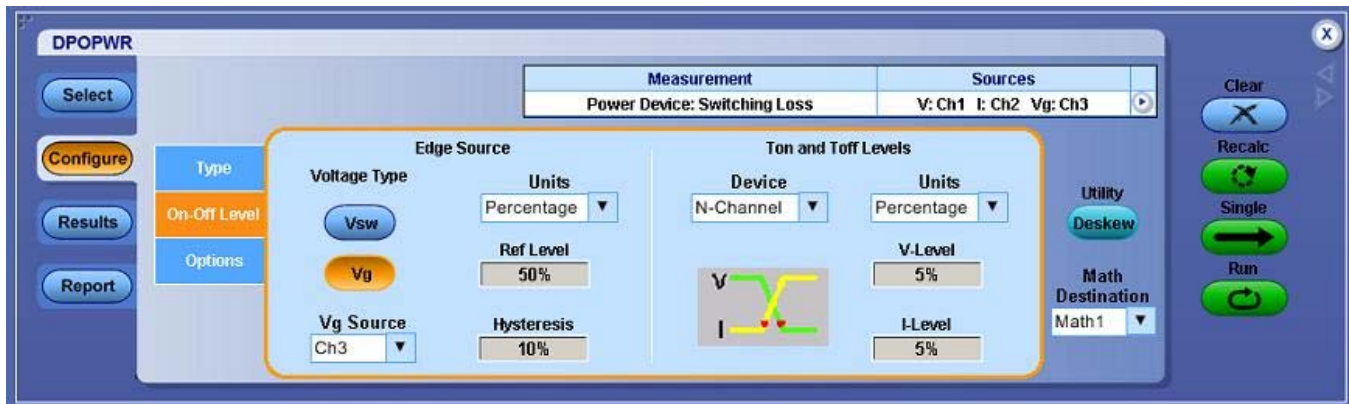


Figure 18: Switching Loss: Edge Source

1. In the Voltage Type pane, select the Vsw or the Vg option to set the edge source. The default option is Vsw. If you have selected Vsw, the Edge Source is the Voltage source configured in the Source Configuration Panel.

If you select Vsw, the application identifies:

- The edges on the switch voltage
- The Switch On and Switch Off portions using the switch voltage and switch current

If you select Vg, the oscilloscope requires at least three channels to acquire data and the application identifies:

- The edges using the Vg (gate voltage)
- The Switch On and Switch Off portions using the Gate Voltage, Switch Voltage and Switch Current

2. Select the Percentage option in the Units drop-down menu to set the Ref Level and Hysteresis values as a percentage or the Absolute option to set the absolute value of the peak to peak signal.
3. Double-click the Ref Level field and use the keypad that appears to set the absolute or the percentage of the ref level value or Double-click the Hysteresis field and use the keypad that appears to set the absolute or the percentage of the hysteresis value.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Switching Loss Ton and Toff Levels

In the Ton and Toff Levels pane, follow the steps to configure the Switching Loss Ton and Toff RefLevel parameters:

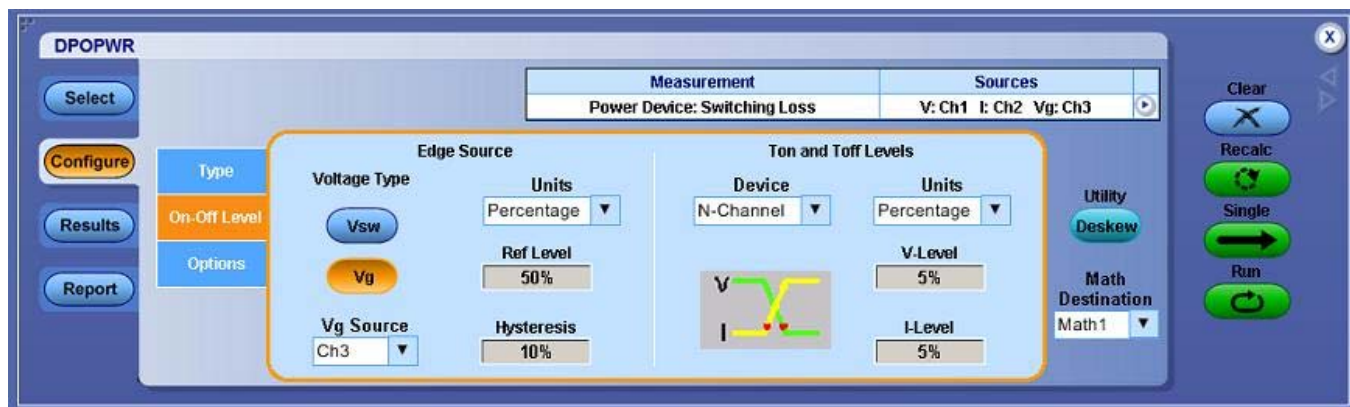


Figure 19: Switching Loss: Ton and Toff Ref Levels

1. Select the semiconductor type (DUT) from the Device drop down menu. The available options are: N-Channel and P-Channel. In an N-Channel device, the Switch Voltage, Switch Current and the Gate Voltage are positive. In a P-Channel device, the Gate Voltage can either be positive and negative or only negative.
2. Select the Percentage option in the Units drop-down menu to set the Ref Level and Hysteresis values as a percentage or the Absolute option to set the absolute value of the peak to peak signal.
3. Double-click the V-Level field and use the keypad that appears to set the absolute or the percentage of the v-level value. The V-Level value for the Start of Ton is five percent of the switch current and for Stop of Ton is five percent of the maximum switch voltage.
4. Double-click the I-Level field and use the keypad that appears to set the absolute value or the percentage of the I-level value.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze>Power Analysis>Results from the oscilloscope menu bar.

Note: When switching current or voltage amplitude varies in an envelope shape. The TON and TOFF levels should be in Absolute Units for accurate results.

Switching Loss Options

In the Options tab, follow the steps to configure the Switching Loss Options parameters:

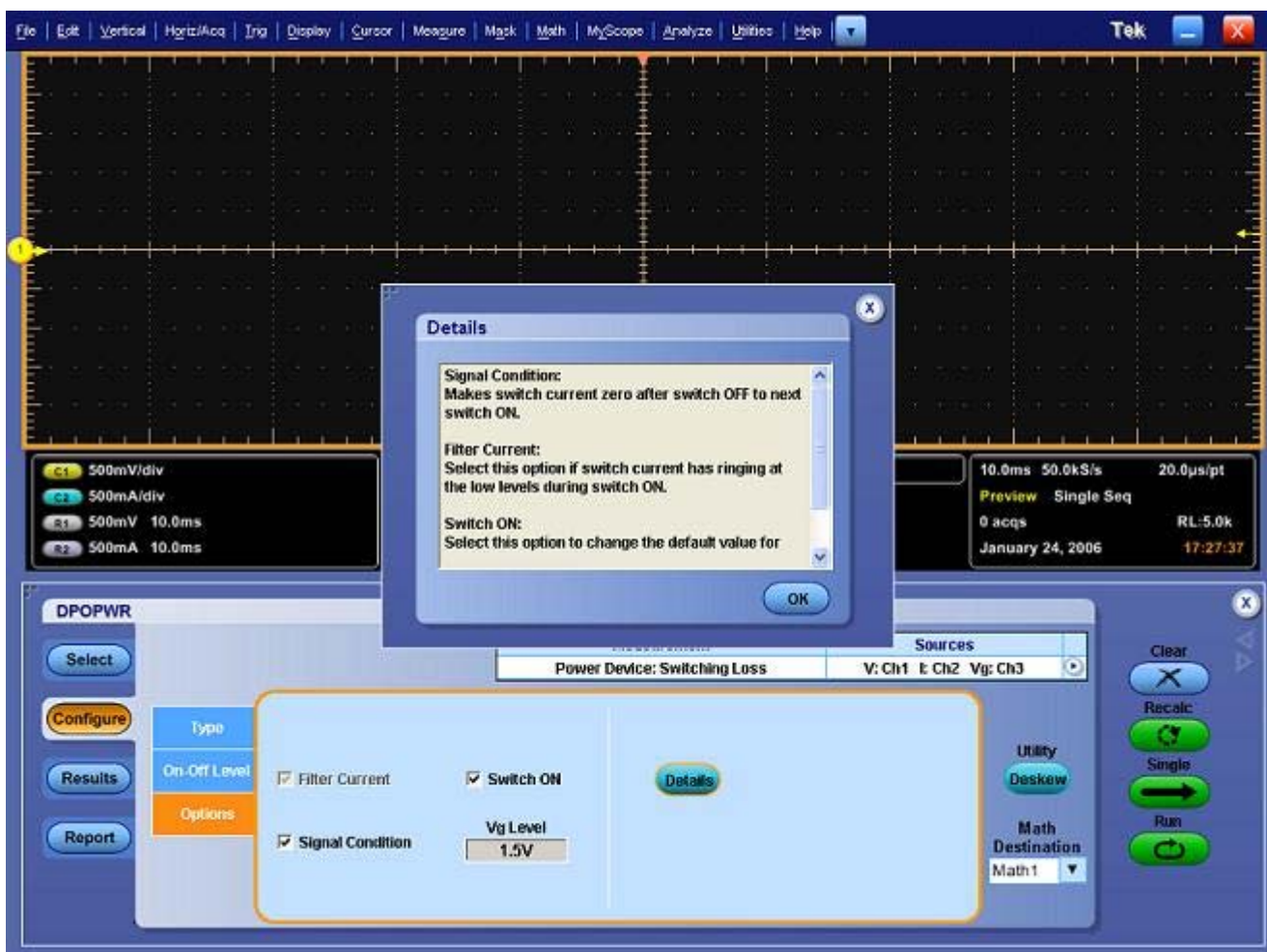


Figure 20: Switching Loss Options

1. Enable the Signal Condition, Filter Current and Switch On options. If you select the Signal Condition option, the application resets the switch current to zero after the switch OFF to the next switch ON. If you select the Filter Current option, the switch current has ringing at the low levels during switch ON. If you select the Switch ON option, you can change the default value of

the start of switch ON. The default value is 5% of the maximum of Vg or 1.5 V of Vg, whichever is lower.

2. Double-click the VG Level field and use the keypad that appears to enter the values. The acceptable range of Vg values are 0 V to 50 V and the default value is 1.5 V, if you have selected the Vg option in the On-Off Level tab as the Edge Source.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Note: *If you select the Vg option in the On-Off Level tab as the Edge Source, the Vg Level field is enabled and you can use this value to calculate the Start of Ton.*

Viewing Switching Loss Results

Note: *Do not disturb the oscilloscope settings during the post analysis of the results. This could lead to incorrect interpretation of results.*

Follow these steps to view the results:

1. Select Analyze> Power Analysis> Results from the oscilloscope menu bar.
2. The application displays the results for the Switching Loss measurement in two panes: Power Loss and Energy.



Figure 21: Switching Loss results

- In the Power Loss pane, the Min field displays the minimum loss in the acquisition in Watts. The Max displays the maximum loss in the acquisition in Watts. The Average Loss is the sum of Ton average, Toff average and the Conduction Loss.
- In the Energy pane, the Min field displays the minimum of all the energy values calculated for each cycle. The Max field displays the maximum of all the energy values calculated for each cycle in Watts. The Average energy displays the sum of Ton energy, Toff energy and the Conduction energy in Joules.
- The Type pane pictorially displays the On, Off and the Average portion of a switching cycle.

Note: Negative switching loss results may be due to the DC offset in the voltage and current probes used. To avoid this, compensate the probes for DC offset and also run Signal Path Compensation option (SPC) on the oscilloscope before starting the application. Make sure to use only DC coupling.

Generating Reports

Follow these steps to generate a report:

- 1.** Select Analyze> Power Analysis> Report from the oscilloscope menu bar, to display Report Generator screen.
- 2.** The application provides default factory templates for this measurement. The template file name is Switching_Loss.rgt for switching loss measurement and the layout file name is Power_Device.rpl. When you click Generate Report, this template file name is displayed by default. If you want to select or modify another template, follow the procedure given here.
- 3.** Select the New Template button in Define Test Template tab to define and create a template for Power Device Analysis. You can configure all the parameters in the user interface to create a template.
- 4.** Select the Define Report Layout tab to define a layout for this template.
- 5.** Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Troubleshooting Switching Loss Results

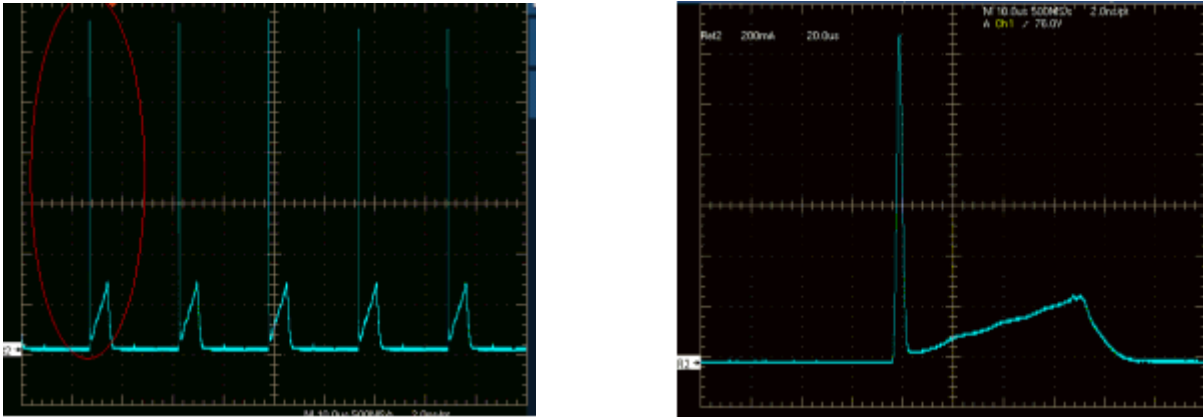


Figure 22: Current waveform of a switching device

The figure on the left shows the current waveform of a switching device. The figure on the right shows the zoomed signal. During Turn On of the switch current, a spike occurs as shown in the previous figures. If you configure I value in the Ton and Toff tab to 5%, during analysis, the application converts the 5% value to the absolute as the product of (5/100) and maximum of the switch current. The maximum of the switch current indicates the spike in the signal. This is valid only during Ton. The calculated maximum switch current value is used to identify the stop of Turn Off. Since you are using the maximum spike values to calculate Ton and Toff, the stop of Ton value is inaccurate. To overcome this, enter the reference value only in Absolute units.

What do you do when you get zero values in the switching loss results?

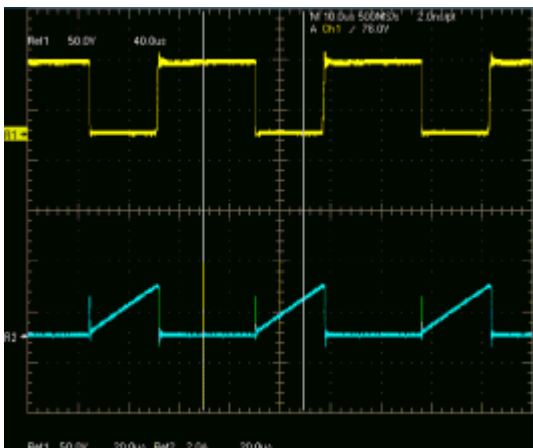


Figure 23: Switching waveform

- When you clear the Filter Current check box in the Options tab, and the switching current waveform is similar to the one in the previous figure, the application finds the start of Ton from the cursor 2 position. The algorithm searches from the cursor 2 position for the configured level

and logs on to the first level. In this case, the start of Ton is calculated after the stop of Ton and the energy and loss is zero.

- When the configured I-level is out of the signal range, the Ton and Toff energy and the loss is zero, the application displays a warning message.



Figure 24: Gate signal without a glitch

- If you select the Vg option button in the Edge Finder tab, the gate voltage should be a clean pulse without any glitch or ringing as shown in the previous figure. The gate voltage is used to identify the edges, Ton or Toff portions. When a glitch occurs, the application displays incorrect results. This type of signal can result in multiple edges (wrong edges) within a switching cycle.

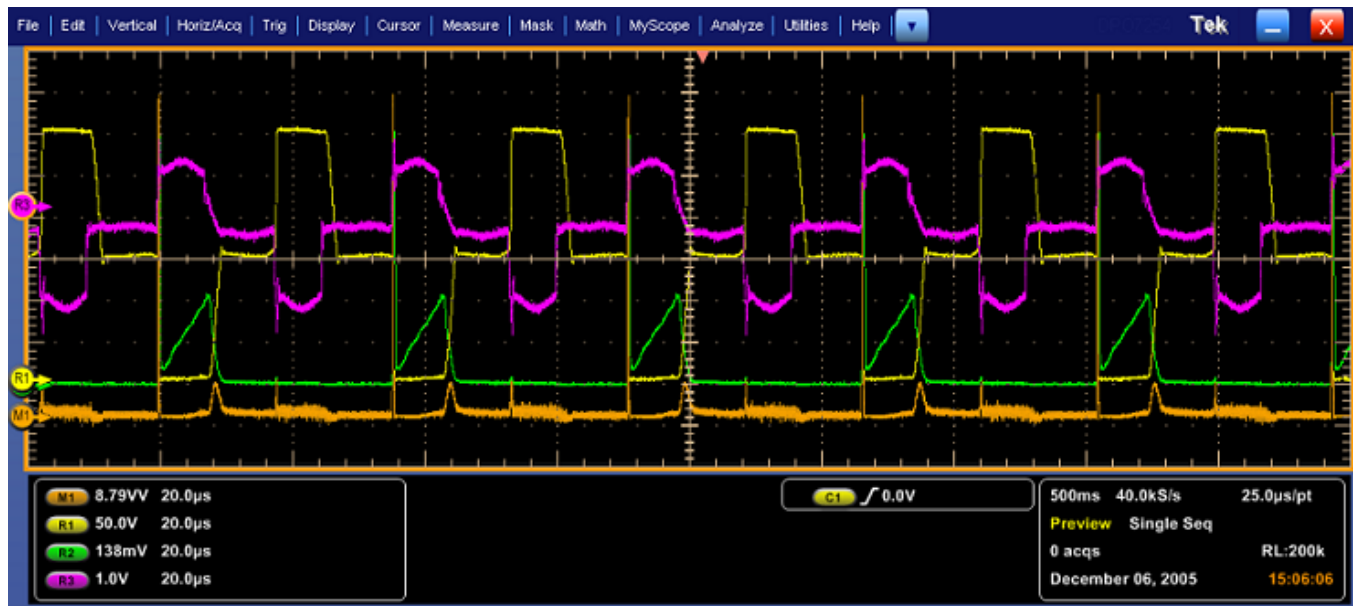


Figure 25: Push Pull device signal

If the signal (Push Pull device) is similar to the one shown in the previous figures, configure the Ref level only at the switching portion of the voltage. The

switching occurs only at the lower portion of the switch voltage. Enter the Ref value as 30%, which falls midway between the lower switching portion.

If you have selected the P-channel device, the switching voltage and current is negative and the gate voltage is negative or positive and negative. The application inverts the waveform and analyzes it like the one on an N-channel device. Select the Vertical Setup > Invert option in the oscilloscope menu to invert the switch voltage, current, gate, and set the device to N-channel.

RDS (on)

Selecting and Configuring Measurements-Dynamic Resistance

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar.
2. Click Power Device to display Power Device screen.
3. Click RDS(on) in the Switching Analysis pane. Now, click the Configure button.
4. The next figure shows a typical equipment setup for RDS(on).

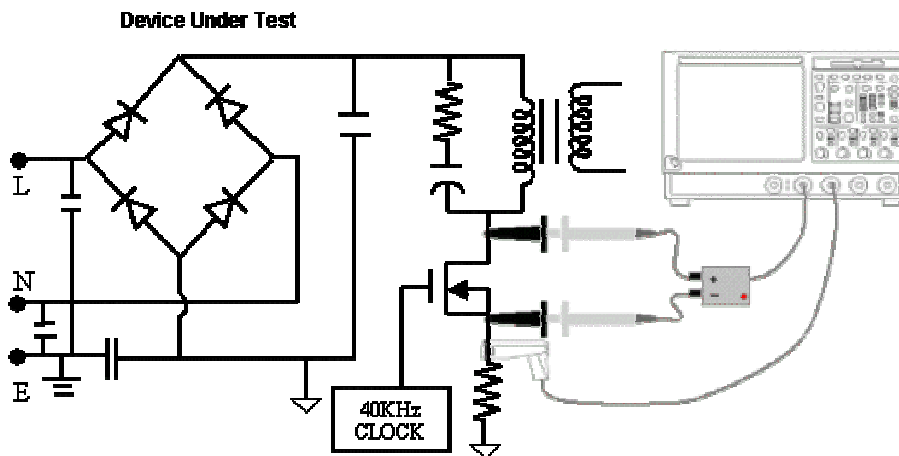


Figure 26: RDS(on) setup



Warning: When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.

Configuring Measurements

Follow the steps to configure the selected measurement:

Note: The Ref and Deskew options are disabled in the source configuration panel as dynamic resistance measures live signals.

1. Configure the options in the Source Configuration Panel.

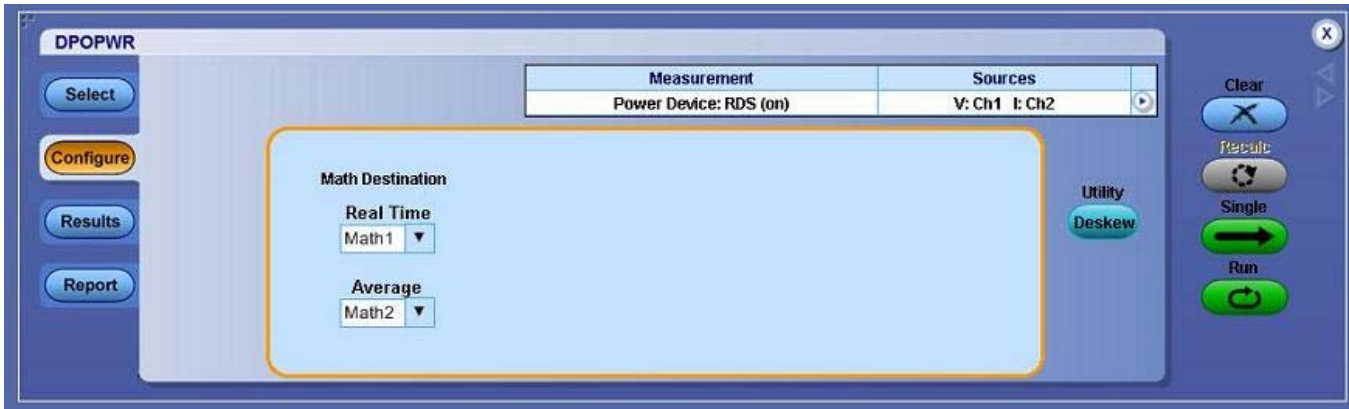


Figure 27: RDS(on) configuration

2. In the Math Destination pane, click the Real Time combo box and the Average combo box to select the math channel. You cannot select the same math channel for both the destinations.

Note: If you select the same Math destination and press Run to acquire the data, the application displays an error message "Conflict in selection of Math destinations."

To measure the low voltage in a high voltage switching (Example 400 Volts peak) in P5205 probe, set the attenuation factor X 50. This will clip any voltage above 130 Volts. Adjust the Volts/division to have the signal with in the dynamic range of the oscilloscope.

3. Press Run to acquire the data.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Viewing Results-RDS(on)

Follow these steps to view the results:

1. Select Analyze> Power Analysis> Results from the oscilloscope menu bar, to view the results for the Instantaneous RDS(on) at Math 1 and the Average RDS(on) at Math 2. Use the cursor button in the oscilloscope panel to place the cursor.

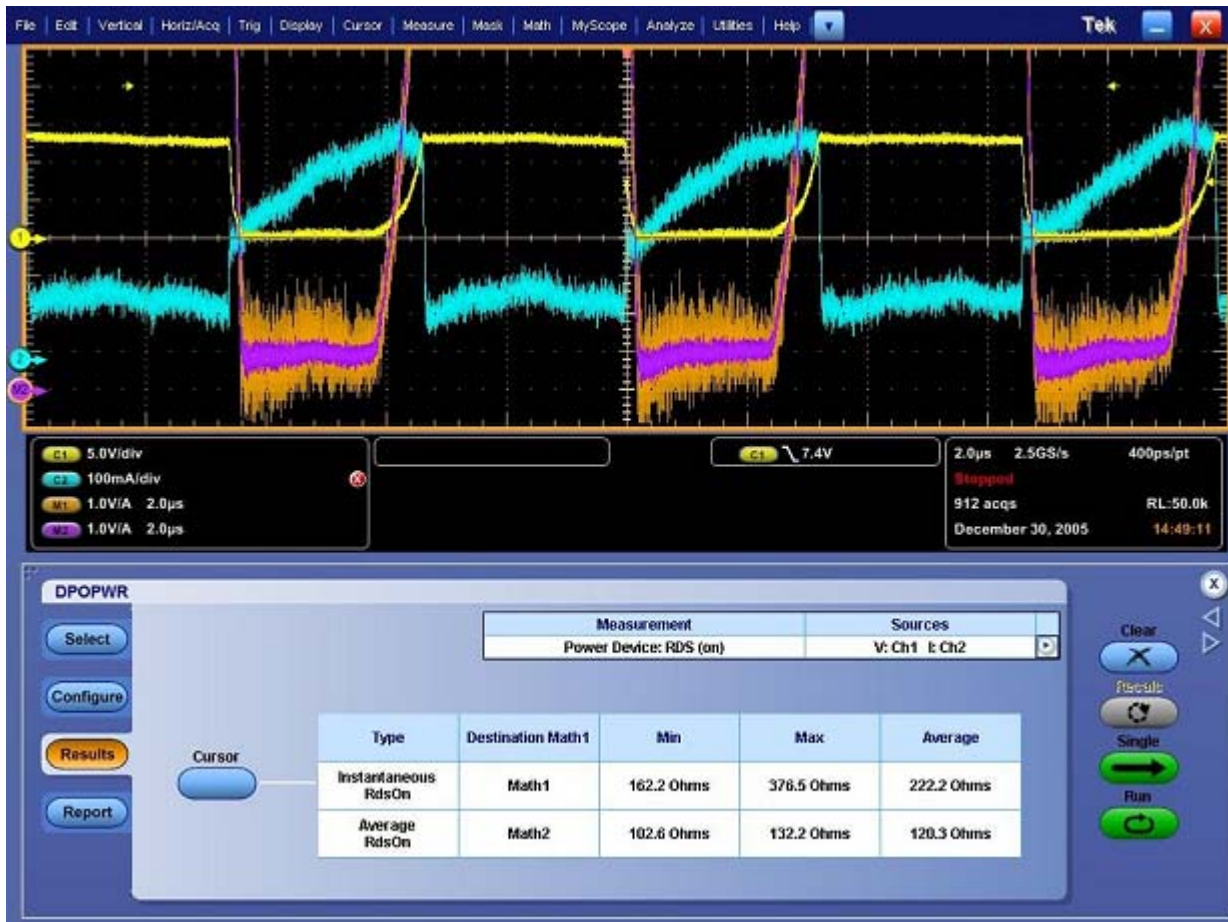


Figure 28: RDS (on) results

Note: In the first instance, the application displays the result screen with no values. Click the Cursor button to display the values.

2. Click the Cursor button to display a pop-up message.
3. Manually place the cursor in the conduction region and click OK. This displays the resistance between the selected portions of the waveform.
4. The application displays the Destination, the Minimum, the Maximum and the Average values of the Instantaneous RDS(on) and Average RDS(on). If

you select the Run mode, the application automatically updates the resistance values in the table.

Generating Reports-RDS(on)

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar, to display the Report Generator screen.
2. The application provides default factory templates for this measurement. The template file name is Dynamic_resistance.rgt and the layout file name is Power_Device.rpl. When you click Generate Report, the application displays the default template file name. However, if you want to select or modify another template, follow the procedure given here.
3. Select the New Template button in Define Test Template tab to define and create a template for Dynamic Resistance. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

di/dt

Selecting and Configuring Measurements-di/dt

To select and configure Power Quality Measurement, follow these steps:

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar.
2. Click Power Device to display Power Device screen.
3. Click di/dt in the Switching Analysis pane. Now, click the Configure button.
4. The next figure shows a typical equipment setup for di/dt.

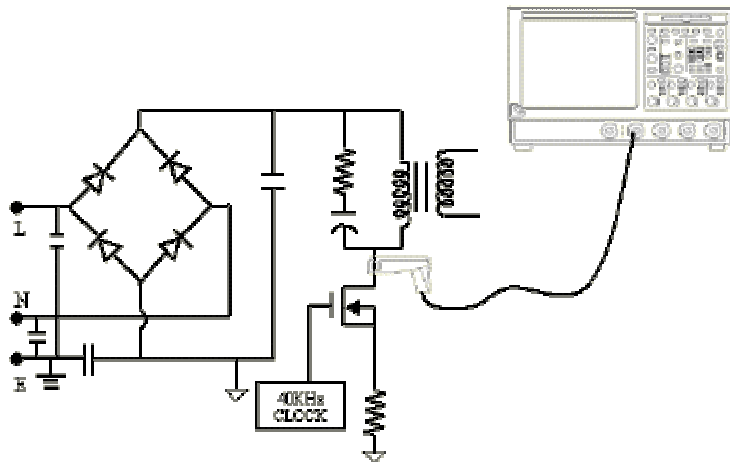


Figure 29: di/dt setup



Warning: When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.

Configuring the Measurement

Follow the steps to configure the selected measurement:

Note: The Ref source, Deskew and Voltage in the Source Configuration Panel is disabled. You can configure only the Current options.

1. Configure the Current options in the Source Configuration Panel.

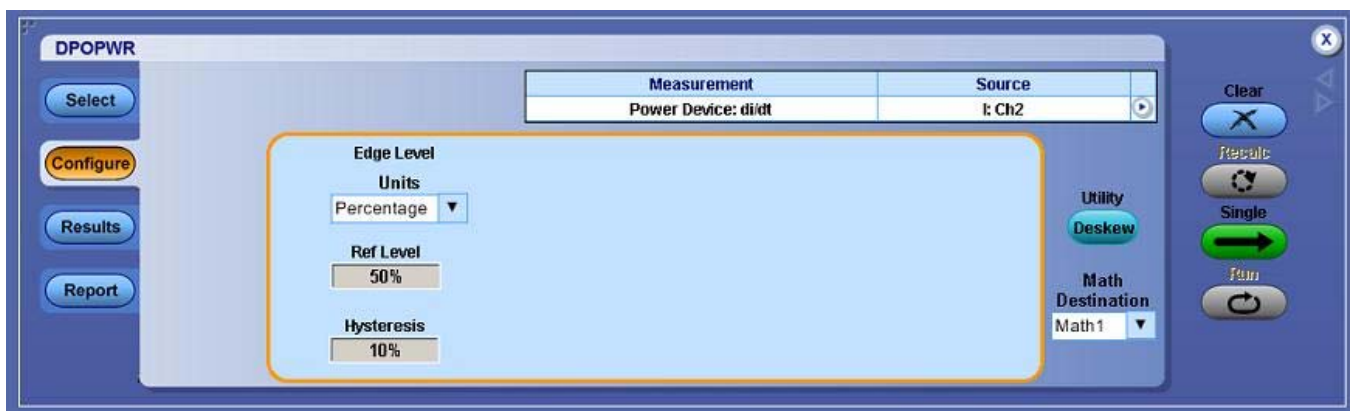


Figure 30: di/dt configuration

2. Select the Percentage option in the Units combo box to set the Ref Level and Hysteresis values in percentage of the peak to peak signal. Select the Absolute option in the Units combo box to set the absolute value of Ref Level and Hysteresis values in amperes. The Ref Level is set to 50% and Hysteresis is set to 10% as default.

3. Select the Math destination from 1 to 4. The selection depends on the number of channels in the oscilloscope from the Math Destination field. The Mode is Single Run only.
4. Press Single to measure.
5. If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Viewing Results- di/dt

To view the results, follow these steps:

1. Select Analyze> Power Analysis> Results from the oscilloscope menu bar.

***Note:** In the first instance, the application displays the result calculated for the first edge.*

2. Click the Edges Number numeric keypad to enter the number of edges for the measurement to calculate the di/dt for the region of interest. If Units is set to percentage the High field is 90%, Low Level is 10% and the Edge Number is 1. If Units is set to absolute enter the absolute values in amperes for Low and High values. The vertical cursor will be at 90% of the first edge and the other vertical cursor at 10% of the first edge. To perform di/dt analysis for any other edge, enter the edge number in the Edge Number field.
3. Use the GP knob adjacent to the High Level and Low Level field to enter the numeric values. This helps you to calculate the di/dt for the range of voltage between the High and Low level on the selected edge. Select the Update button to position the oscilloscope cursors on the High and Low levels of the waveform and on the edge you have set. This displays the updated results for the selected edge.

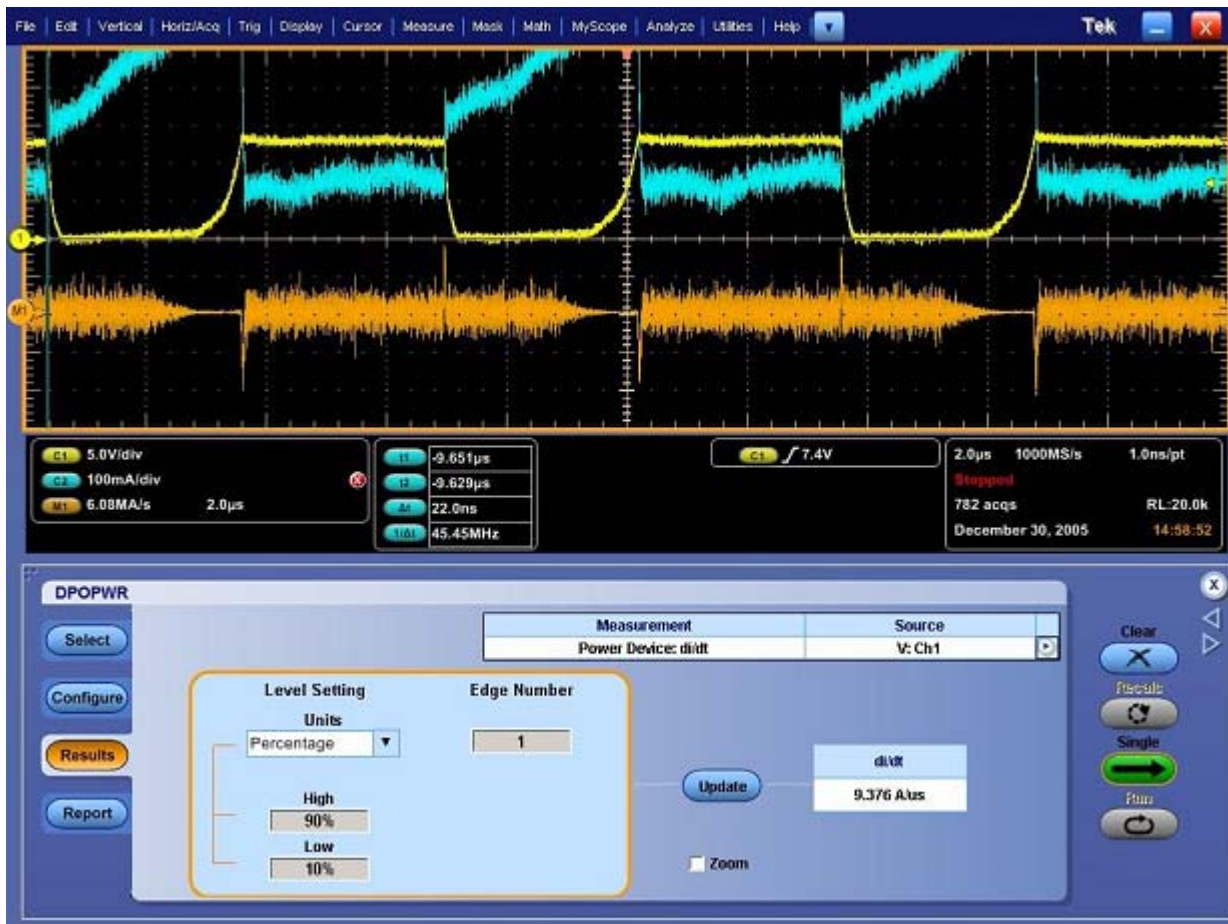


Figure 31: di/dt results

4. The application displays the results as a differential of the current waveform with respect to time in units of amperes per microsecond. Select the Zoom to zoom the area of interest.

Generating Reports-di/dt

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar to display the Report Generator screen.
2. The application provides default Factory templates are available for this measurement. The template file name is di_dt.rgt and the layout file name is Power_Device.rpl. When you click Generate Report, the application displays the default template file name. However, if you want to select or modify another template, follow the procedure given here.
3. Select the New Template button in Define Test Template tab to define and create a template for di/dt. You can configure all the parameters in the user interface to create a template.

4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

dv/dt

Selecting and Configuring Measurements-dv/dt

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar.
2. Click Power Device to display Power Device screen.
3. Click dv/dt in the Switching Analysis pane. Now, click the Configure button.
4. The next figure shows a typical equipment setup for dv/dt.

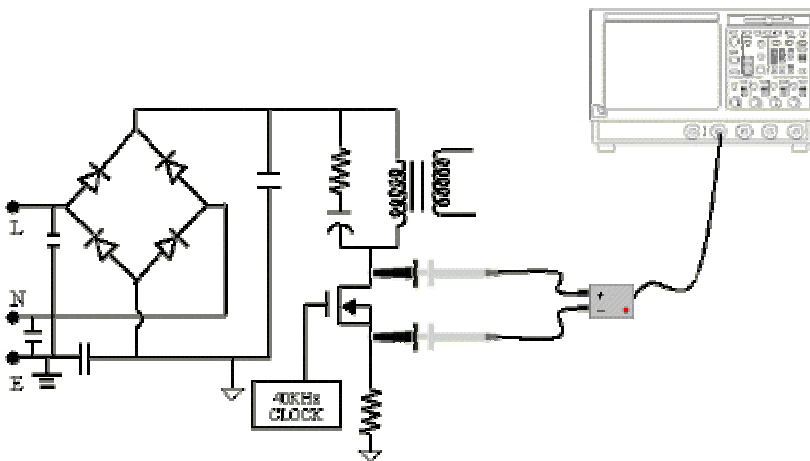


Figure 32: dv/dt setup



Warning: When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.

Configuring the Measurement

Follow the steps to configure the selected measurement:

Note: Ref, Current, Deskew and I-Probe Settings in the Source Configuration Panel is disabled. You can configure only the Voltage options.

Configure the Voltage options in the Source Configuration Panel.

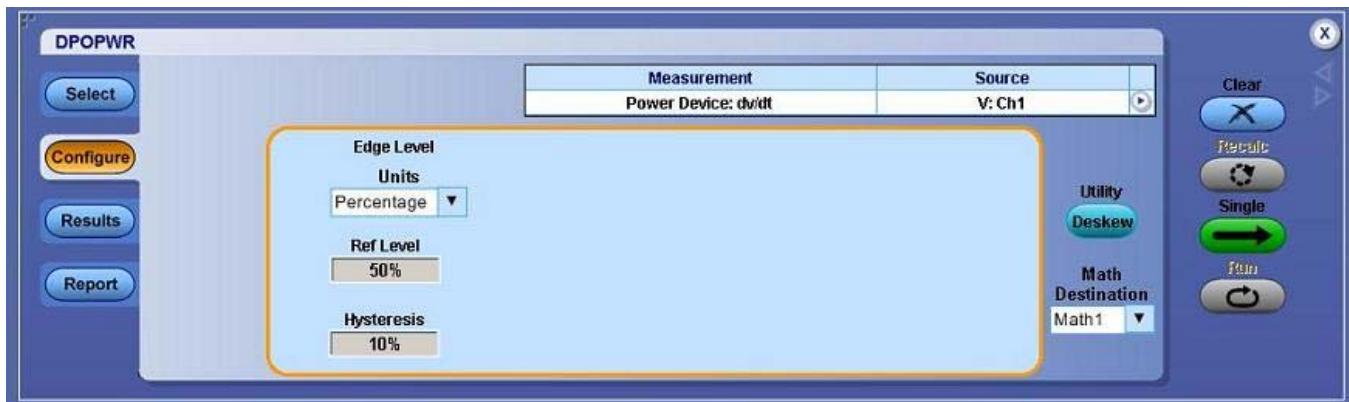


Figure 33: dv/dt configuration

1. Select Percentage from the Units combo box to set the Ref Level and Hysterisis values in percentage of the peak to peak signal. Select Absolute from the Units combo box to set the absolute value of Ref Level and Hysterisis values in volts.
2. Click the Math Destination combo box and select the Math destination from 1 to 4, depending on the number of math channels available in the oscilloscope.
3. Press Single to measure.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Viewing Results-dv/dt

To view the results, follow these steps:

1. From the oscilloscope menu bar, select Analyze> Power Analysis> Results

Note: In the first instance, the application displays the result calculated for the first edge. Click the cursor button to display the values.

2. Click the Edges button to display the Edge Analysis screen. If Units is set to Percentage, the High field is 90%, Low Level is 10% and the Edge Number is 1. If the Units is set to Absolute enter the absolute values in amperes for Low and High values. The vertical cursor will be at 90% of the first edge and the other vertical cursor at 10% of the first edge. To perform dv/dt analysis for any other edge, enter the edge number in the Edge Number field. Select the Percentage or Absolute radio button.
3. Use the GP knob adjacent to the High Level and Low Level field to enter the numeric values. This helps you to calculate the dv/dt for the range of voltage between the High and Low level on the selected edge. Select the

Update button to position the oscilloscope cursors on the High and Low levels of the waveform and on the edge you have set. This displays the updated results for the selected edge.

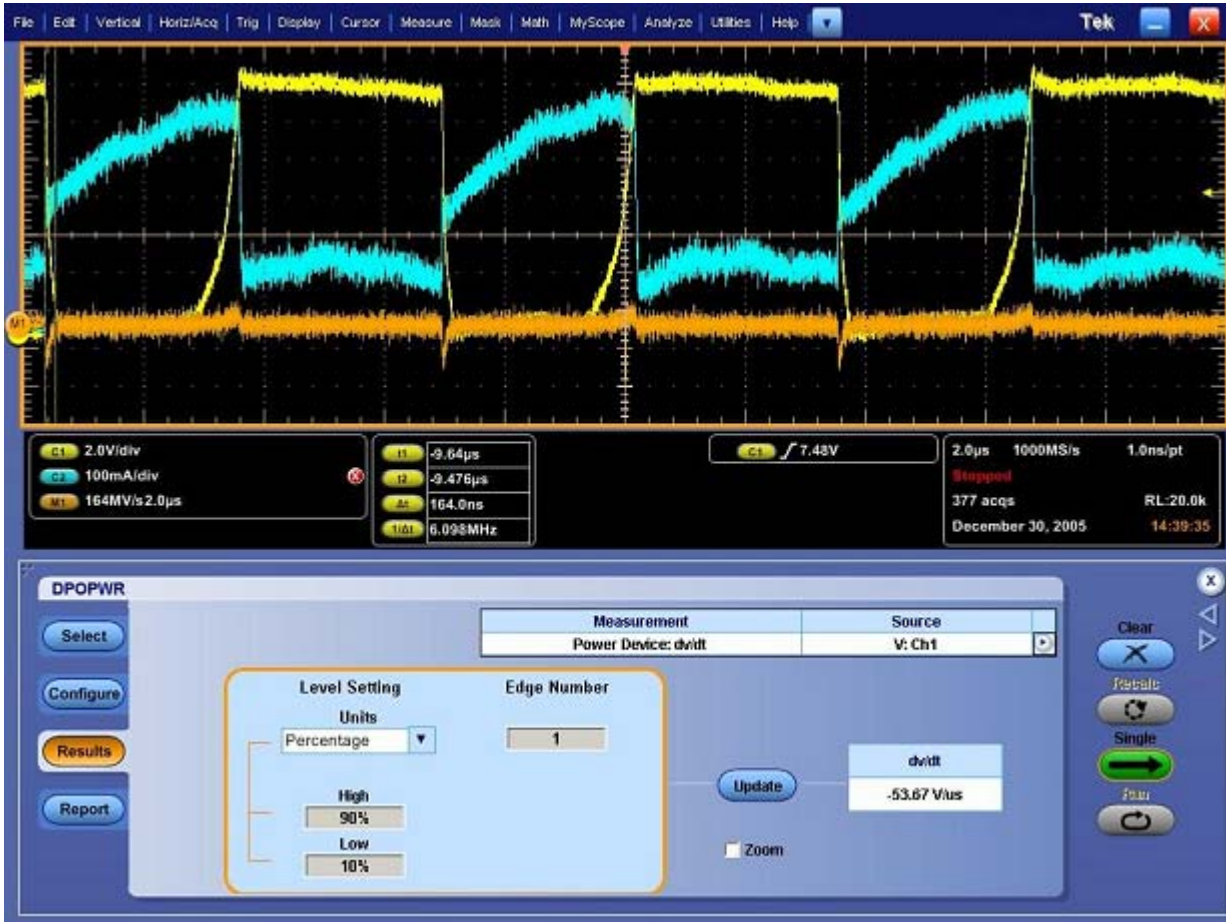


Figure 34: dv/dt results

- The application displays the results, which is a differential of the voltage waveform with respect to time in units of volts per microsecond. Select the Zoom to zoom the area of interest.

Note: If you observe the cursor positions on the oscilloscope screen in the figure below, the absolute values of the signal level read by the cursor does not match with the entered high and low levels in the result panel. This is because the values entered in Low and High levels may lie in between the samples of the waveform. This happens for di/dt and dv/dt measurements.

Generating Reports-dv/dt

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar to display the Report Generator screen.
2. The application provides default factory templates for this measurement. The template file name is dv_dt.rgt and the layout file name is Power_Device.rpl. When you click Generate Report, the application displays the default template file name. However, if you want to select or modify another template, follow the procedure given here.
3. Select the New Template button in Define Test Template tab to define and create a template for dv/dt. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Pulse Width

Selecting and Configuring Measurements-Pulse Width

1. From the oscilloscope menu bar, select Analyze> Power Analysis> Select.
2. Click Power Device to display the power device screen.
3. Click Pulse Width in the Modulation Analysis pane. Click Configure.
4. The next figure shows a typical equipment setup for Pulse Width Analysis.

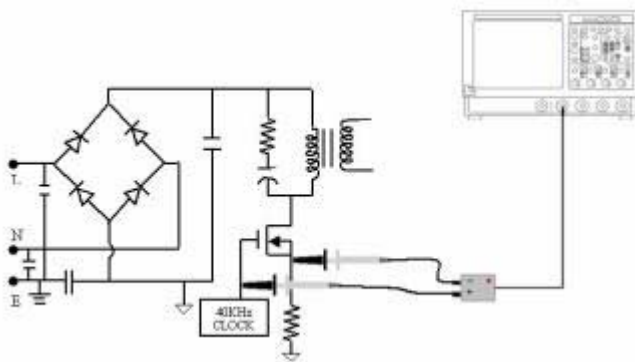


Figure 35: Pulse Width setup



Warning: When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.

Configuring the Measurement

Follow the steps to configure the selected measurement:

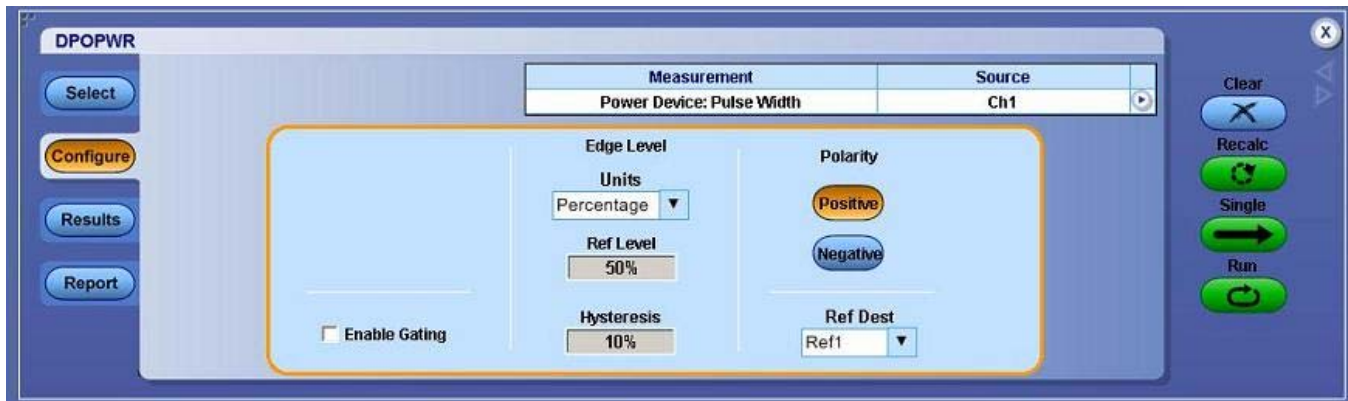


Figure 36: Pulse Width configuration

1. Select the Source for the measurement from the Source Configuration screen that appears by clicking Source field. The options available for channel sources are: 1-4.
2. Click the Ref Dest combo box. The options available for the Ref destinations are: 1-4.
3. In the Edge Level panel, you can set the Ref level and Hysteresis as an absolute value or Percentage of Pk-Pk value. If you select the Level option, the application displays the Ref level and Hysteresis in volts. We recommend you to set the mid value to 50% of the peak to peak of the source voltage.
4. Double-click the Ref Level field and Hysteresis field and use the keypad that appears to enter the Ref level and Hysteresis values.
5. Select the pulse width Polarity by clicking on either Positive or Negative.
6. Click Enable Gating to enable Gated Measurement option to analyze a specific section of the waveform. When you click the Enable Gating button, the application displays a message “Place the cursors at the appropriate region of the waveform.” Position the cursor as required. The application analyzes and plots the time trend of the selected section of the waveform between the cursors.
7. Press Run to measure.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Note: The application replaces any data in the selected ref destination when you analyze and measure modulation. You do not get a warning message here.

Viewing Results-Pulse Width

To view the results, follow these steps:

1. Select Analyze> Power Analysis> Results from the oscilloscope menu bar.
2. The application displays the results according to the selected pulse width as shown below. Check the results on the oscilloscope with reference to the plot to display the negative or positive pulse width and time.

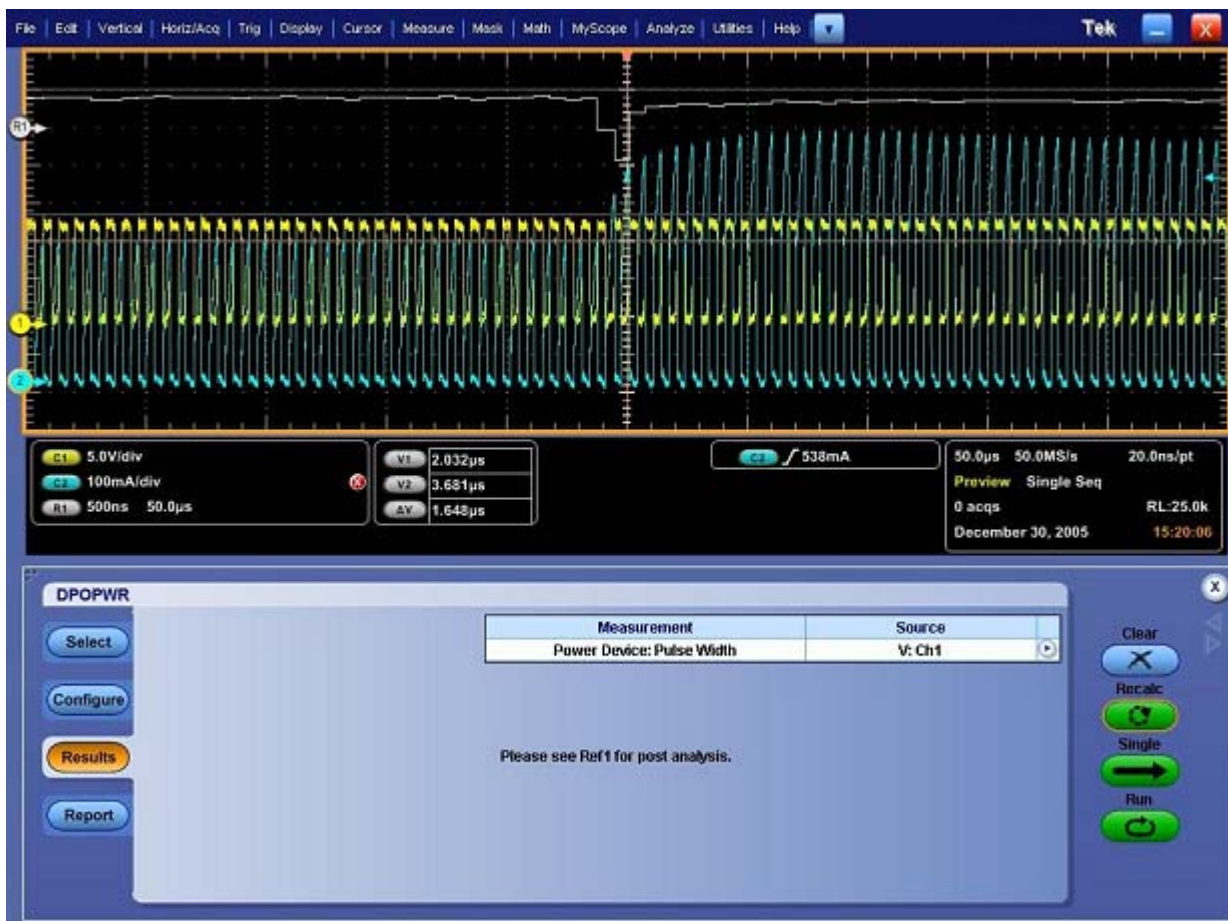


Figure 37: Pulse width results

Generating Reports-Pulse Width

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar to display the Report Generator screen.
2. The application provides default factory templates for this measurement. The template file name is Pulse_Width.rgt.and the layout file name is Modulation_Analysis.rpl. When you click Generate Report, the application displays the default template file name. However, if you want to select or modify another template, follow the procedure given here.
3. Select the New Template button in Define Test Template tab to define and create a template for Pulse Width. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Duty Cycle

Selecting and Configuring Measurements-Duty Cycle

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar.
2. Click Power Device to display the Power device screen.
3. Click Duty Cycle in the Modulation Analysis pane. Click Configure.

Configuring the Measurement

Follow the steps to configure the selected measurement:

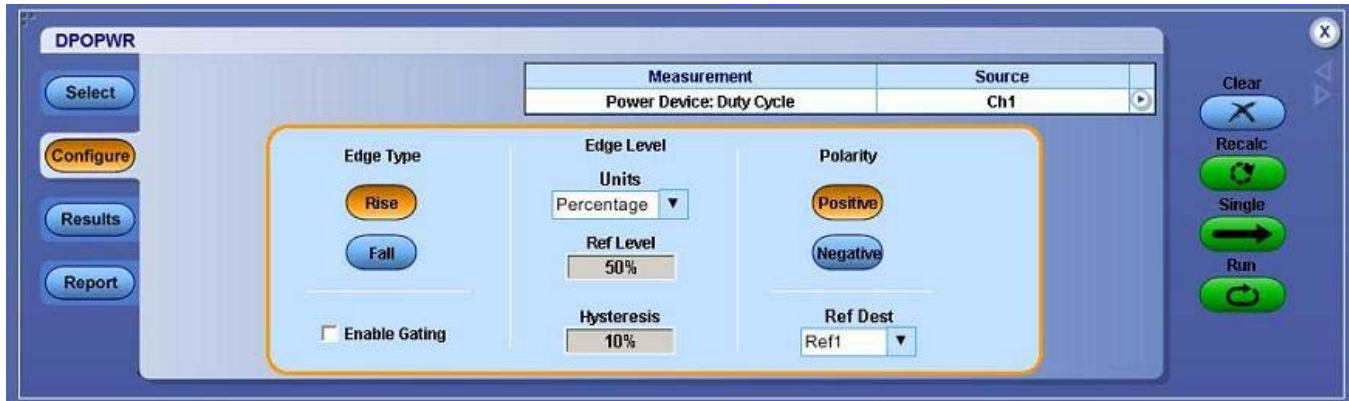


Figure 38: Duty cycle modulation configuration

1. Select the Source for the measurement from the Source Configuration screen that appears by clicking Source field. The available channel sources are: 1-4.
2. Select the Ref Destination from the Ref Destination combo box. The available ref destinations are: 1-4.
3. In the Ref Level panel, set the Mid and Hysteresis to absolute or Percentage. If you select the Level option, the application displays the Mid and Hysteresis in volts. We recommend you to set the mid value to 50% of the peak to peak of the source voltage.
4. Double-click the Ref Level field and Hysteresis field and use the keypad that appears to enter the Ref level and Hysteresis values.
5. Select the pulse width Polarity by clicking on either Positive or Negative.
6. Select the Edge Type for the pulse width by clicking either Rise or Fall.
7. Click Enable Gating to enable Gated Measurement option to analyze a specific section of the waveform. When you click the Enable Gating button, the application displays a message “Place the cursors at the appropriate region of the waveform”. Position the cursor as required. If you select Yes, the application analyzes and plots the time trend of the selected section of the waveform between the cursors.
8. Press Run to acquire the data.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Note: The application replaces any data in the selected ref destination when you analyze and measure modulation. You do not get a warning message here.

Viewing Results-Duty Cycle

To view the results, follow these steps:

1. Select Analyze> Power Analysis> Results from the oscilloscope menu bar.
2. The application displays the results according to the selected duty cycle. Check the results on the oscilloscope for the trend plot that displays the positive or negative duty cycle and time.

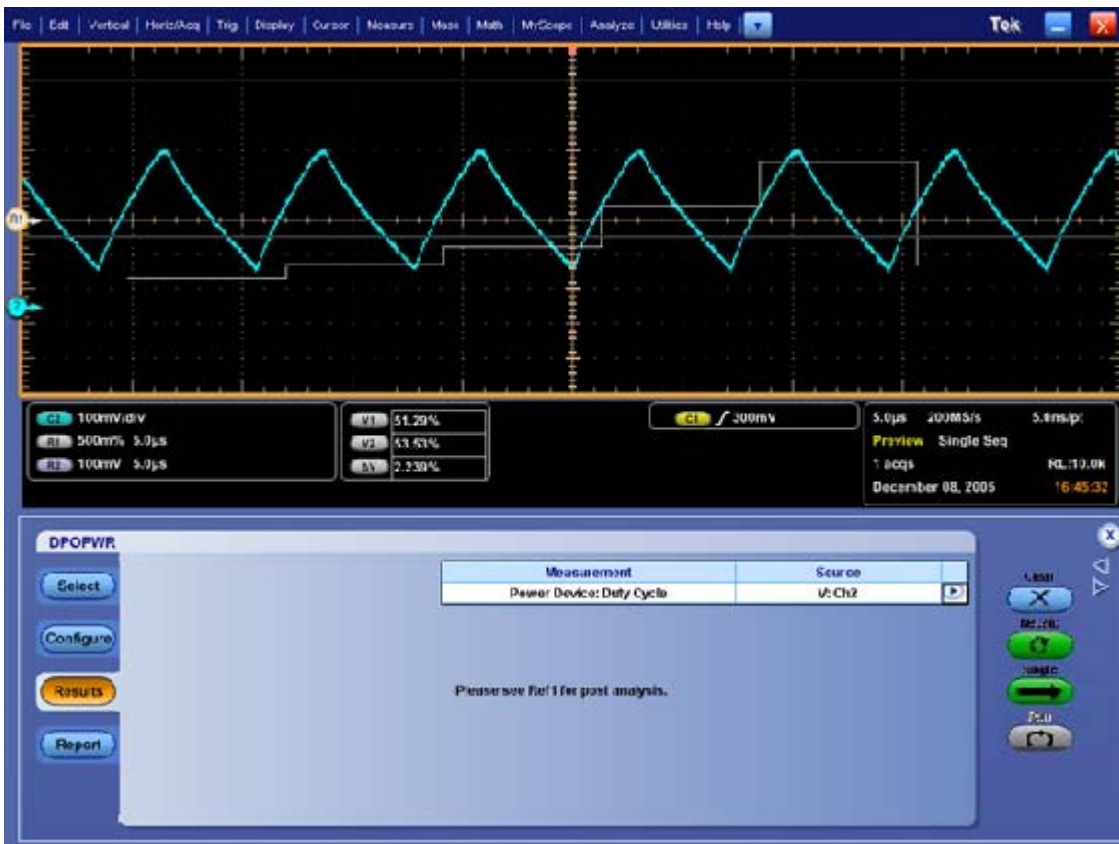


Figure 39: Duty cycle modulation results

Generating Reports-Duty Cycle

Follow these steps to generate a report:

1. From the oscilloscope menu bar, select Analyze> Power Analysis> Report to display the Report Generator screen.
2. The application provides default factory templates for this measurement. The template file name is Duty_Cycle.rgt and the layout file name is Modulation_Analysis.rpl. When you click Generate Report, the application

displays the default template file name. However, if you want to select or modify another template, follow the procedure given here.

3. Select the New Template button in Define Test Template tab to define and create a template for Duty Cycle. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Period

Selecting and Configuring Measurements-Period

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar.
2. Click Power Device to display the Power device screen.
3. Click Period in the Modulation Analysis pane. Click Configure.

Configuring the Measurement

Follow the steps to configure the selected measurement:

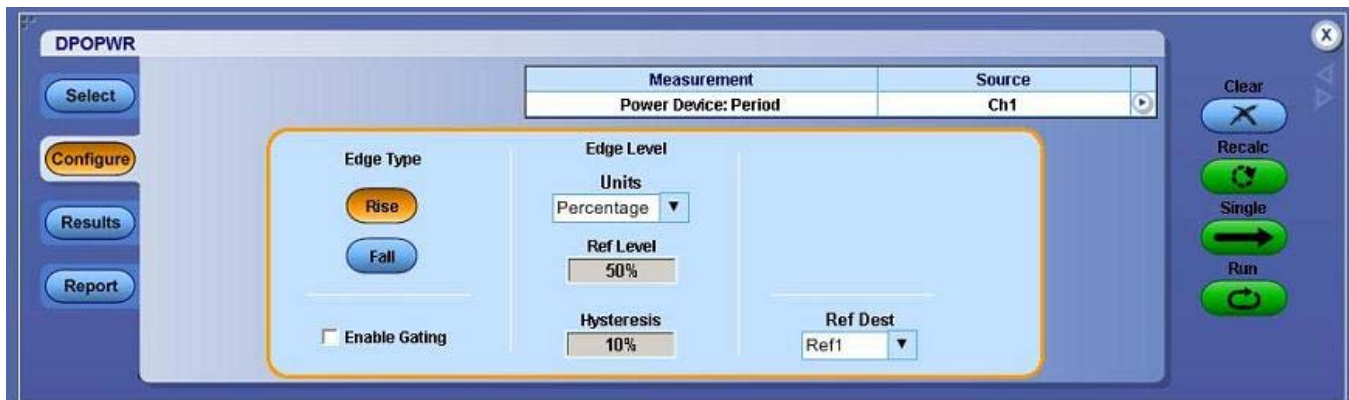


Figure 40: Period modulation configuration

1. Select the Source for the measurement from the Source Configuration screen that appears by clicking Source field. The available channel source is from 1-4.
2. Select the Ref Destination from the Ref Destination combo box. The available ref destination is from 1-4.
3. In the Edge Level panel, set the Mid and Hysteresis to Absolute or Percentage. Select the Level option to display the Mid and Hysteresis value

in volts. We recommend you to set the mid value to 50% of the peak to peak of the source voltage.

4. Double-click the Ref Level and Hysteresis field and use the keypad that appears to enter the Ref level and Hysteresis values.
5. Select the Edge Type for the pulse width by clicking either Rise or Fall.
6. Click Enable Gating to enable Gated Measurement option to analyze a specific section of the waveform. When you click the Enable Gating button, the application displays a message "Place the cursors at the appropriate region of the waveform". Position the cursor as required. If you select Yes, the application analyzes and plots the time trend of the selected section of the waveform between the cursors.
7. Press Run to acquire the data.
8. If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Note: The application replaces any data in the selected Ref destination when you analyze and measure modulation. The application does not display a warning message here.

Viewing Results-Period

To view the results, follow these steps:

1. Select Analyze> Power Analysis> Results from the oscilloscope menu bar.
2. The application displays the results for the selected clock period. Check the results on the oscilloscope that displays the trend plot of period and time.

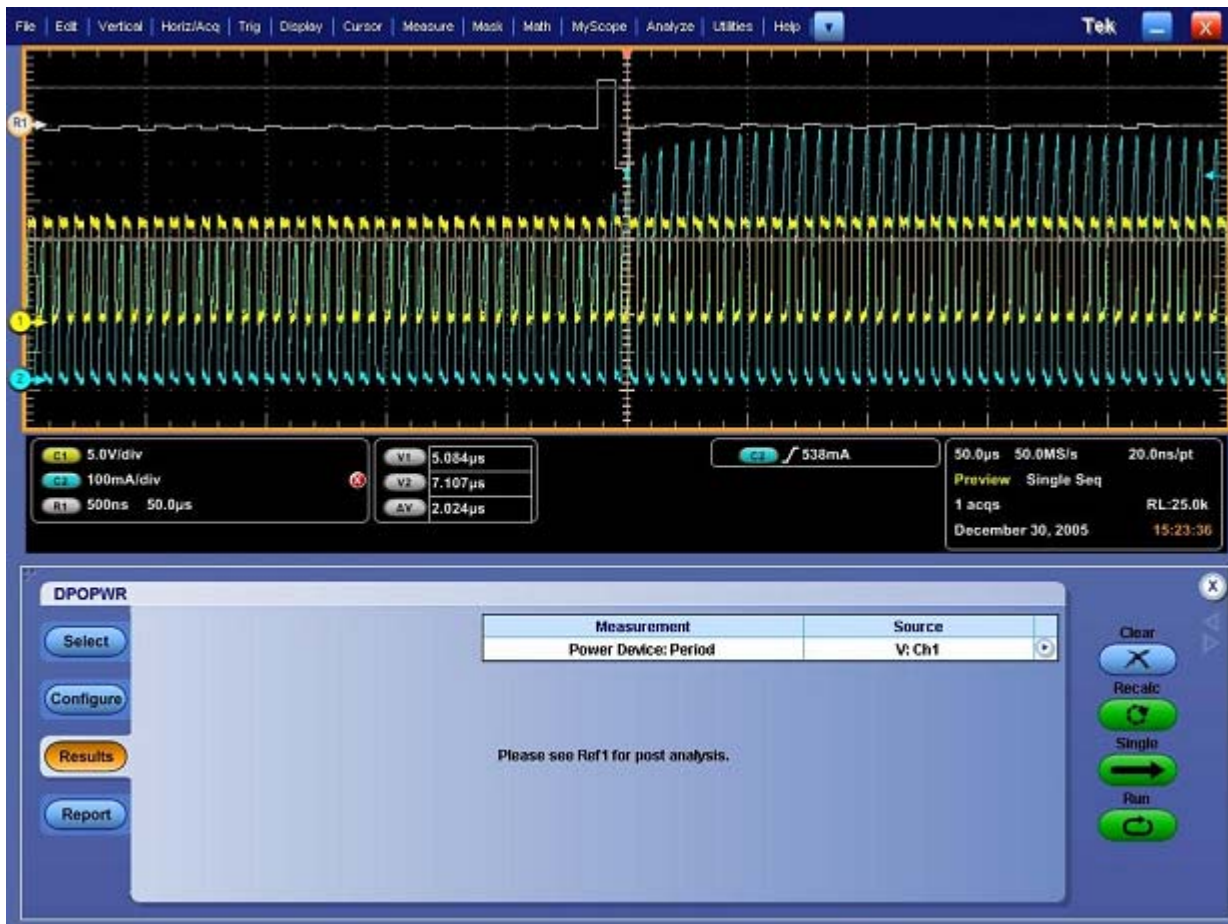


Figure 41: Period modulation results

Generating Reports-Period

Follow these steps to generate a report:

1. From the oscilloscope menu bar, select Analyze> Power Analysis> Report to display the Report Generator screen.
2. The application provides default factory templates for this measurement. The template file name is Clock_Period.rgt and the layout file name is Modulation_Analysis.rpl. When you click Generate Report, the application displays the default template file name. However, if you want to select or modify another template, follow the procedure given here.
3. Select the New Template button in Define Test Template tab to define and create a template for Clock Period. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Frequency

Selecting and Configuring Measurements-Frequency

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar.
2. Click Power Device to display the Power device screen.
3. Click Frequency in the Modulation Analysis pane. Click Configure.

Configuring the Measurement

Follow the steps to configure the selected measurement:

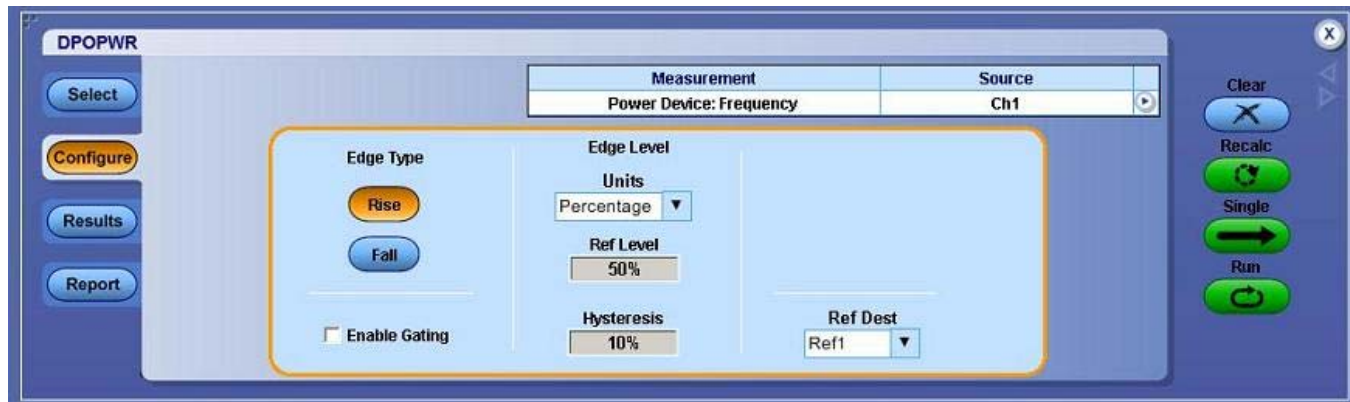


Figure 42: Frequency modulation configuration

1. Select the Source for the measurement from the Source Configuration screen that appears by clicking Source field. The available channel source is from 1-4.
2. Select the Ref Destination from the Ref Destination combo box. The available ref destination is from 1-4.
3. In the Edge Level panel, set the Mid and Hysteresis to Absolute or Percentage. If you select the Level option, the application displays the Mid and Hysteresis in volts. We recommend you to set the mid value to 50% of the peak to peak of the source voltage.
4. Double-click the Ref Level and Hysteresis field and use the keypad that appears to enter the Ref level and Hysteresis values.
5. Select the Edge Type for the pulse width by clicking either Rise or Fall.
6. Click Enable Gating to enable Gated Measurement option to analyze a specific section of the waveform. When you click the Enable Gating button,

the application displays a message "Place the cursors at the appropriate region of the waveform." Position the cursor as required. If you select Yes, the application analyzes and plots the time trend of the selected section of the waveform between the cursors.

7. Click Run to acquire the data.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Note: The application replaces any data in the selected ref destination when you analyze and measure modulation. You do not get a warning message here.

Viewing Results-Frequency

To view the results, follow these steps:

1. Select Analyze> Power Analysis> Results from the oscilloscope menu bar.
2. The application displays the results according to the selected frequency. Check the results on the oscilloscope with reference to the trend plot that displays the frequency and time.



Figure 43: Frequency modulation results

Generating Reports-Frequency

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar to display the Report Generator screen.
2. The application provides default factory templates for this measurement. The template file name is Frequency.rgt and the layout file name is Modulation_Analysis.rpl. When you click Generate Report, the application

displays the default template file name. However, if you want to select or modify another template, follow the procedure given here.

3. Select the New Template button in Define Test Template tab to define and create a template for Frequency. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Safe Operating Area

Selecting and Configuring Measurements-Safe Operating Area

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar.
2. From the Power Device tab, select the Safe Operating Area (SOA) measurement. Select the Configure option.

The next figure shows a typical equipment setup for Safe Operating Area.

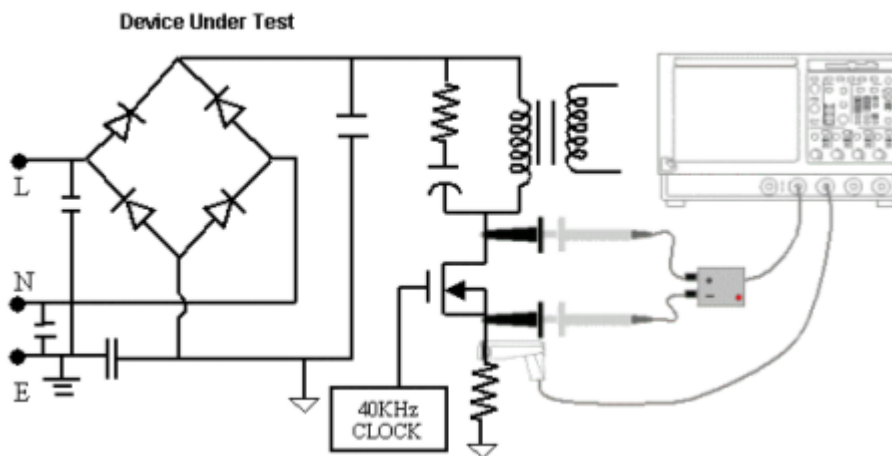


Figure 44: SOA setup



Warning: When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.

Configuring Measurements

Follow the steps to configure the selected measurement:

1. Select the Plot Type: SOA or SOA X-Y.

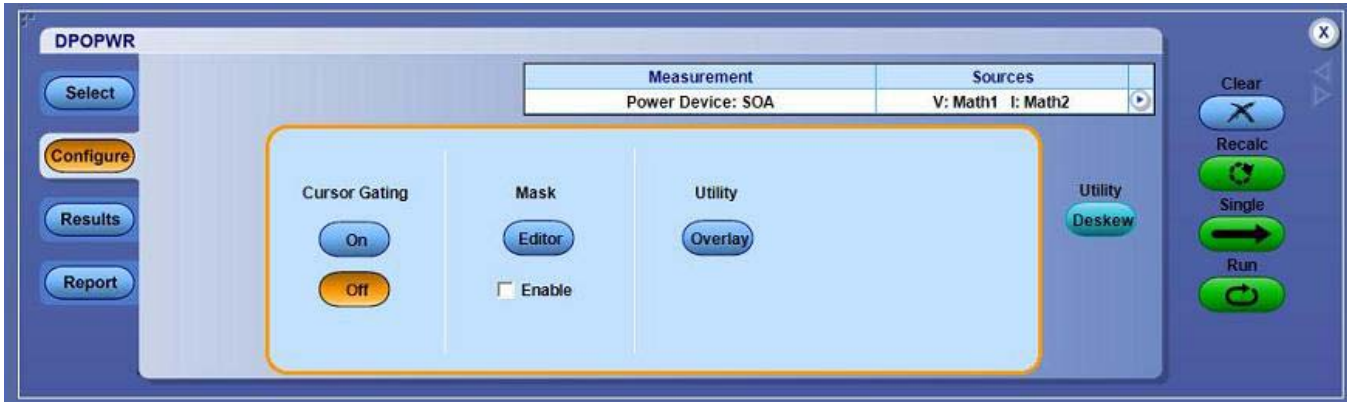


Figure 45: SOA configure

Note: The Ref option in the Source Configuration panel is disabled since the SOA X-Y option works only on live signals. You can also select the Free Run in the Mode field if you select SOA option.

2. Configure the options in the Source Configuration Panel.

Note: Make sure that there are waveforms on the oscilloscope or the application displays an error message.

Table 12: SOA measurements

Plot Type	Description
SOA	Plots the Voltage and Current waveform in a single record in XY mode
SOA X-Y	Measures and plots the voltage and current waveforms using the XY mode display of the oscilloscope. Use this option to measure multiple acquisitions of the same waveform in real time testing

3. The result varies according to the selected plot type. Select the Enable check box to define and apply masks on the SOA plot. If the Enable is selected and there is no mask set in the SOA mask editor, selecting the Run button displays a warning message: "SOA Mask is enabled, but not defined. Do you wish to continue?." Select Yes to continue without the mask. The Editor and Enable options are applicable only for SOA.

4. Select the Editor button to display the SOA Mask Editor.
5. Press Run to display the SOA plot.
6. Place the cursor on the plot to display the following results:
 - Voltage value
 - Current value
 - Power value
 - Mean power
 - Standard deviation

Viewing Results-Safe Operating Area

The SOA Measurement plots the current and waveform results in the SOA Plot. Follow these steps to view the results.

1. Select Analyze> Power Analysis> Results from the oscilloscope menu bar.
2. The application displays the SOA results as shown:

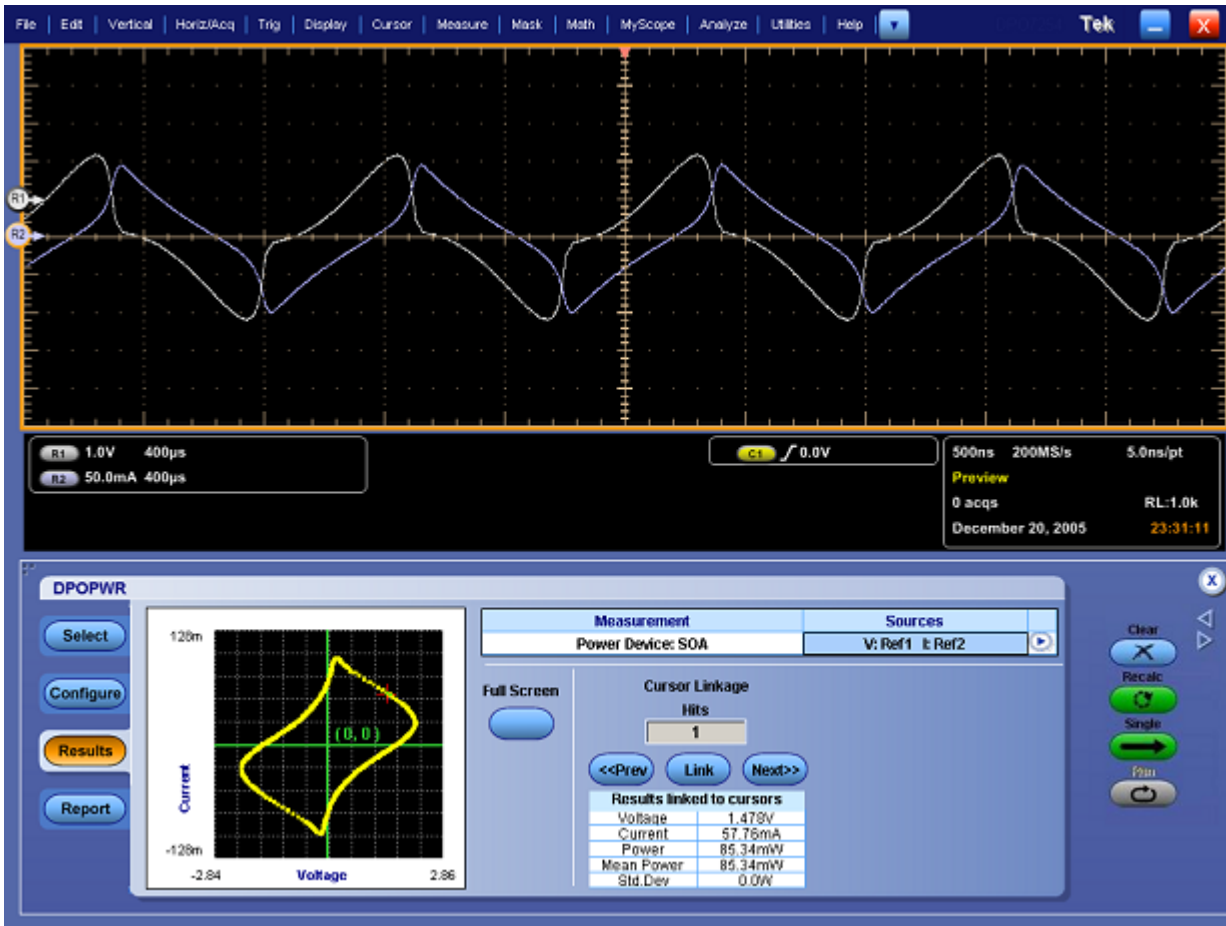


Figure 46: SOA results

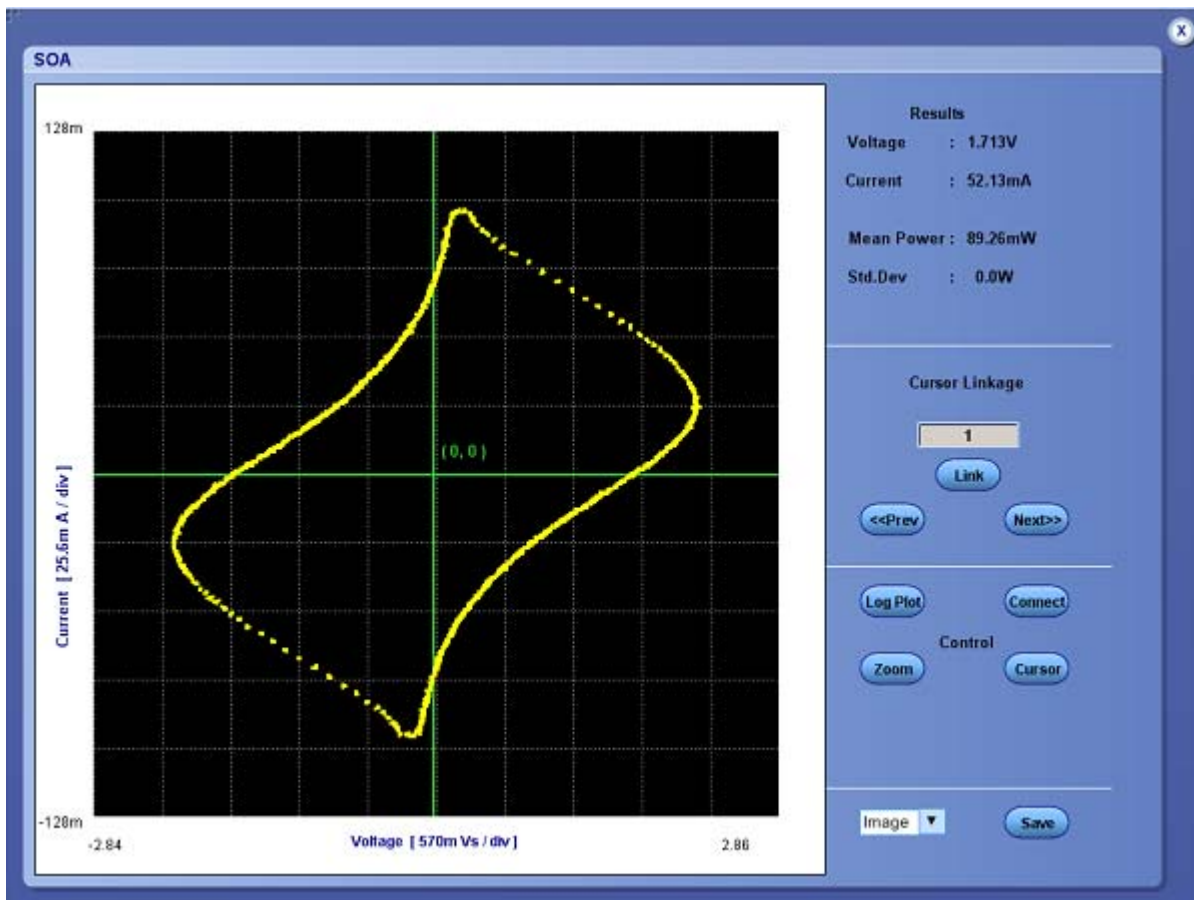


Figure 47: SOA results

3. Click On or Off in the Cursor Gating panel to enable cursor gating. Press the Run button to display the message: "Have you acquired the waveform?"
4. Select Yes to run the application and display a pop-up message: "Place the cursors at the appropriate region of the waveform." Select OK to display the results for the selected portion of the waveform. The Cursor Linkage allows you to locate the point in a SOA plot to the time domain waveform and link the data index of the waveform. Place the cursor at the region of the waveform you want to plot SOA and display the plotted area.
5. Select the Dot connect option in the SOA Plot to connect all the data points and display the stray point. This ensures that all the data points display the visible transition.
6. Select the Save option and save the image in the .jpg format. The default directory and the file name is C:/TekApplications/DPOPWR/Images.
7. Select the Save Data option and save the result plot in the .plt format.

Note: If the Cursor control button is enabled, the zoom buttons are disabled.

8. Place the Cursor to select any data point in the plot.
9. Select the Zoom buttons to display the Zoom options.
10. Select the Reset option to restore the plot to its original display after zooming.
11. Click the In button to zoom in. You can zoom up to five times the normal view. Drag the mouse to define the area of interest. Click the Out button to zoom out. You can use the Zoom In and Zoom Out buttons until the application reaches the maximum and the minimum zoom limit. You can use the Zoom buttons in three different ways:
 - Dragged zoom-in: where the area is selected by dragging on the displayed window
 - Single-clicked zoom-in: when a point is clicked on the displayed window, the resulting window is equal to 1/4th of the displayed window.
 - Single-clicked zoom-out: when a point is clicked on the displayed window, the resulting plot is the previous zoom state of the displayed window.
12. Use the cursor in the oscilloscope and the Link button to link the waveform from the SOA plot. When you select the Link button, the plot automatically resizes to the half screen mode. The Link Cursor Position field displays the number of occurrences of the particular data points. Here minimum is 1 and maximum is the number of occurrences of data points.
13. Place the cross hair cursor at the point of interest on the plot and press the Link button. This enables the cursor on the oscilloscope and places it on the waveform corresponding to the selected data point in the plot panel.
14. In case of multiple occurrences, select the Prev button to select the previous data point and the Next button to select the next data point.
15. You can view the plot and the time domain waveform simultaneously in the half screen. Select the Link check box to select the data point to link the waveform in the oscilloscope to the SOA Plot. Place the cross hair cursor on the plot and enter the values in the Link Occurrences field. Use the Zoom button to zoom the area of interest in the plot and display in the oscilloscope.

Note: You can view the result either in the half screen mode or in the full screen mode.

16. The application displays the result for Voltage, Current, Power, Mean Power and Standard Deviation at the cursor location.

Viewing Results for SOA X-Y

You can view the result in the oscilloscope as a real time plot.

Note: Select the channel as Ch1, Ch2 or Ch3, Ch4 for the SOA Plot.

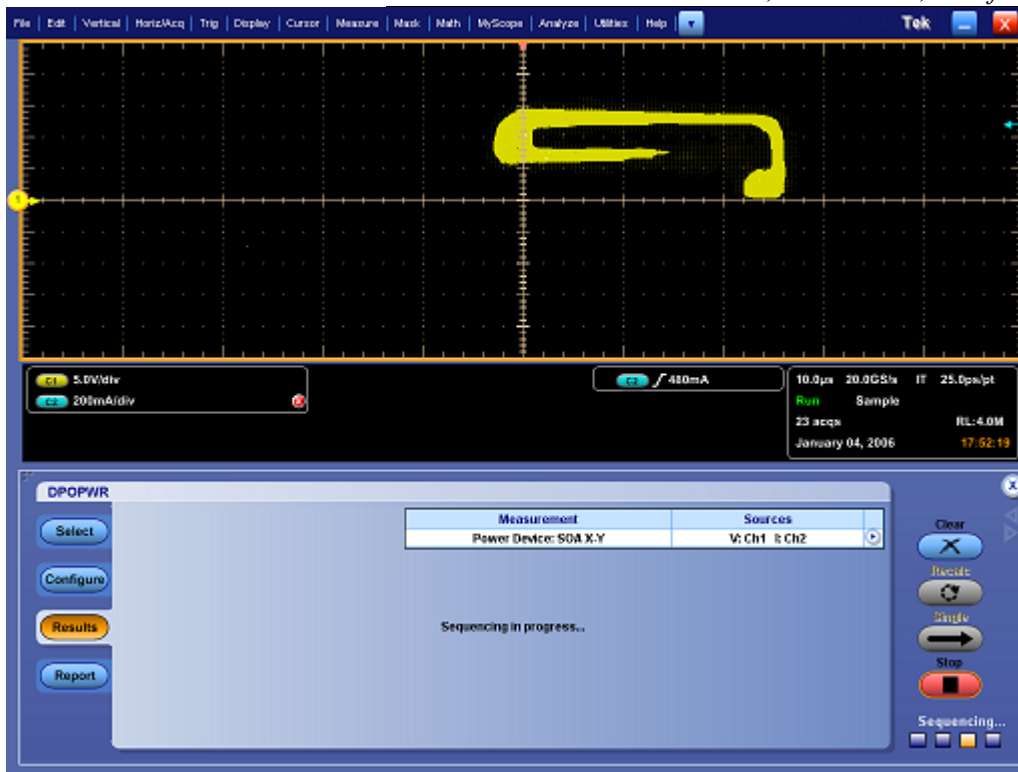


Figure 48: SOA results

SOA Overlay

Analyze> Power Analysis> SOA Overlay

Note: Select the Save Data button in the SOA results window to save the plot as a .plt file before using this utility.

Purpose: Plot, view and compare SOA Plots of the device operating at various load conditions. The SOA Overlay utility helps you to overlay multiple SOA plots simultaneously. You can use this feature to view multiple plots on the same grid.

Initially, the application adjusts the SOA scale according to the selected plot(s), but you can also observe the plot for the voltage and current range of operation under various conditions.

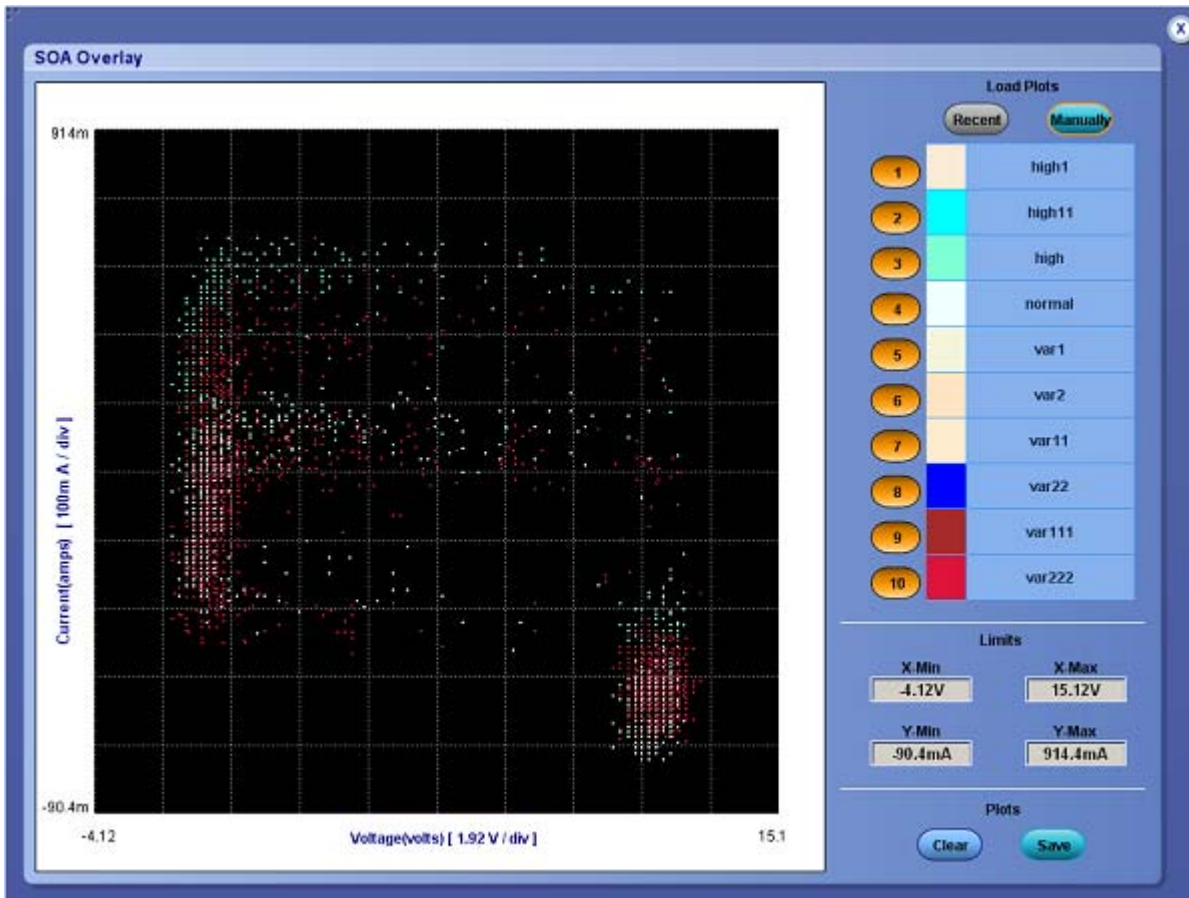




Figure 49: SOA Overlay

The Y Max, Y Min, X Max and X Min give you the flexibility to set up different operating voltage and current range to view the plot(s). The X axis represents the voltage values in volts and the Y axis represents the current in amperes.

Follow these steps to perform SOA Overlay,

1. Click the Recent button to automatically load the SOA plots saved in the current run of the application. You can load only the last ten plots.
2. Select the Manually button to manually load the SOA plots from any folder or from the default folder C:\TekApplications\DPOPWR\SOA\Data directory. You can load any of the ten plots. You can use the Shift or the Control key to select multiple plots.
3. Click the selection button (for example: ) next to the plot name to display the SOA plot on the grid area of the screen. When the utility displays the first plot, it automatically scales the plot. Continue this procedure to overlay and display all the plots. SOA Overlay displays the plot based on the color set in the Plot Name pane. The most recently selected plot is drawn over the previously selected plots. You can simultaneously view and compare different SOA plots under multiple

conditions. You can change the XY limits to view the plot at a different scale.

4. Click the Plot selection button (for example: ) to remove the plot from the grid area.
5. Click the Clear button to display the message: "Do you want to clear the Plots? Press the Yes button to clear." Select Yes to clear the plots from the grid area.
6. Select the Save button to save the plot as a .jpg file to any folder of your choice or to the default folder C:\TekApplications\DPOPWR/Images.
7. Select OK to close the display window.

Generating Reports-Safe Operating Area

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar, to display the Report Generator screen.
2. The application provides default factory templates for this measurement. The template file name is SOA.rgt and the layout file name is Power_Device.rpl. When you click Generate Report, the application displays the default template file name. However, if you want to select or modify another template, follow the procedure given here.
3. Select the New Template button in Define Test Template tab to define and create a template for Safe Operating Area. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Magnetics

Selecting and Configuring Measurements-Magnetics

Note: If the current waveform is less than a division, the application displays an error message: "Unable to find the correct edges". This is because of the high variation of data points in a single division and the insufficient hysteresis band to find the correct edges. For the current waveform, the vertical range should be greater than one division.

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar.
2. Select Magnetics option. Press the Configure button.

The next figure shows a typical equipment setup for Magnetics.

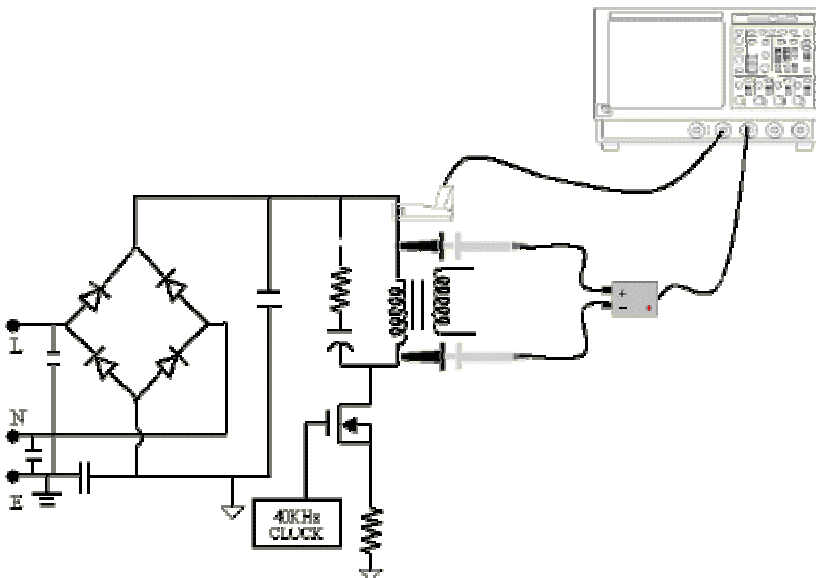


Figure 50: Equipment setup-Magnetics



Warning: When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.

Configuring Measurements

Follow the steps to configure the selected measurement:

1. Configure the options in the Source Configuration Panel.

2. Click a measurement button: Inductance, Magnetic Property, Magnetic Loss, or Ivs V Integral V.
3. Click Configure to view that measurement's configuration options.

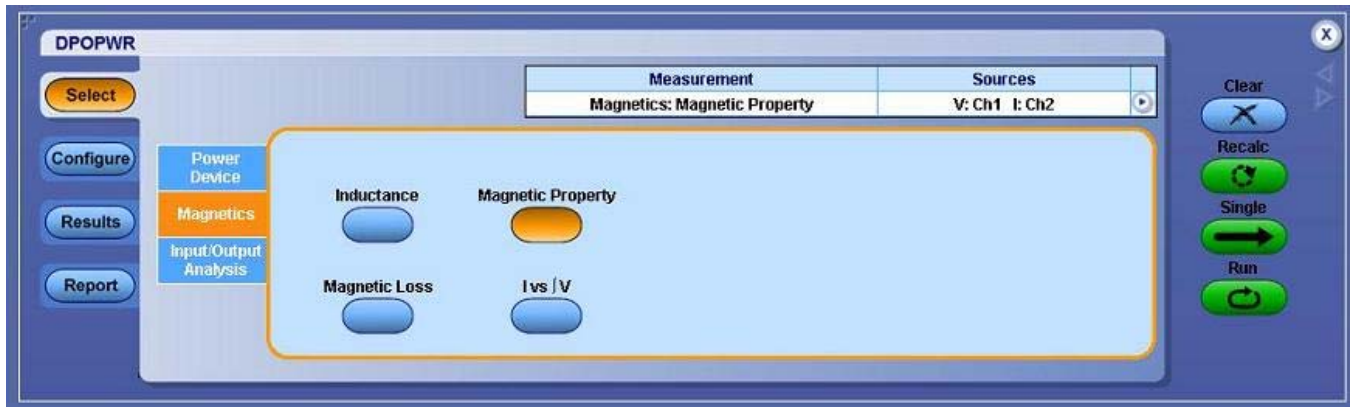


Figure 51: Magnetics configuration

4. Configure the settings. Click Single to measure Magnetics.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Magnetic Loss



Figure 52: Magnetic Loss configuration

To use the Cursor Gating option to measure Magnetic Loss for varying frequency signals, follow these steps:

1. Click the Cursor Gating On button to enable the cursors in the oscilloscope.
2. Place the vertical cursors between the specified area of the waveform to calculate the magnetic loss.
3. Press the Run button to measure magnetic loss.

Inductance

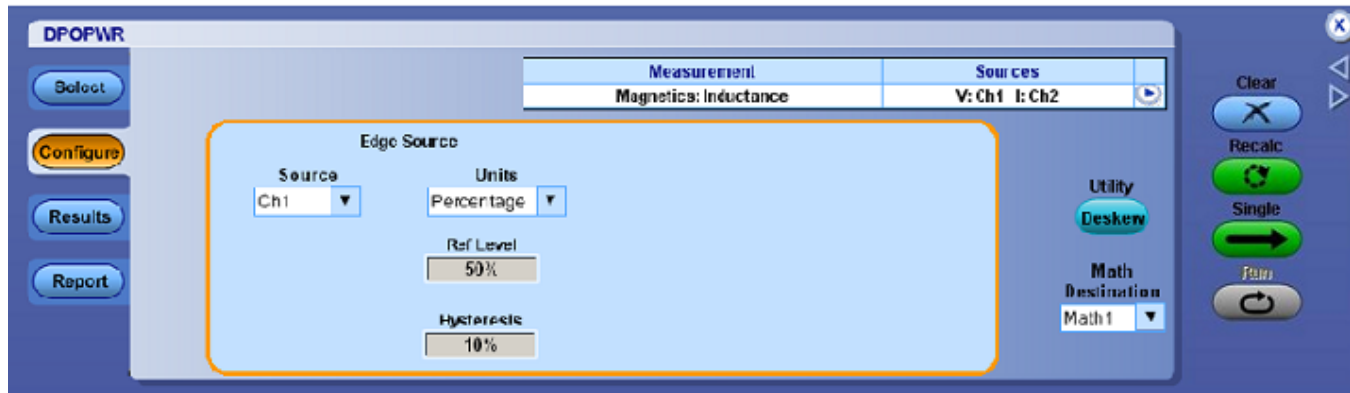


Figure 53: Inductance configuration

Note: When you are measuring the inductance of a transformer, do not load the secondary winding. The measurement of the inductance at the primary winding under no load condition is as good as measuring the inductance for a single winding.

When you are measuring the inductance of the coupled inductor with multiple winding on the same core, the measured value of the inductance will deviate from the actual value due to the influence of the current on other winding. You can use this measured value to calculate the Ripple current.

To measure Inductance:

1. Use the drop down arrow in the Source field to set the edge source. The available options are:
 - Ch1-Ch4
 - Math1-Math4
 - Ref1-Ref4
2. In the Units combo box, select the Percentage or the Absolute option to set the Ref Level and Hysteresis values in percentage or the absolute value of the peak to peak signal.
3. Select the Math Waveform Destination from the drop down list. The default destination is Math1.
4. Press the Run button to measure inductance.

Ivs $\int V$

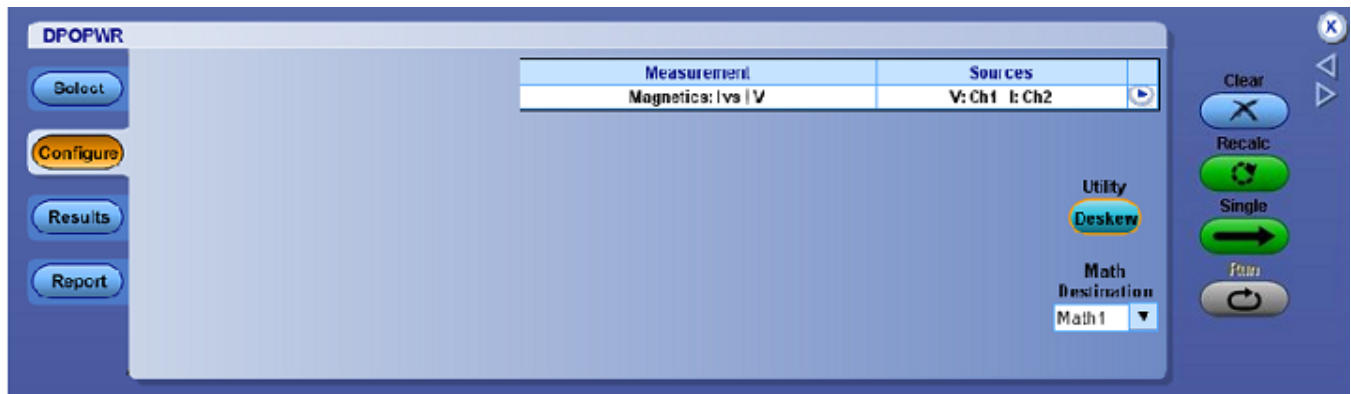


Figure 54: Ivs $\int V$ configuration

To measure Ivs Integral V select the Math Waveform Destination from the drop down list. The default destination is Math1. Click the Run button to measure Ivs Integral V.

Magnetic Property

The Magnetic Property measures the parameters related to the magnetic material in a magnetic component used in an in-circuit operation.

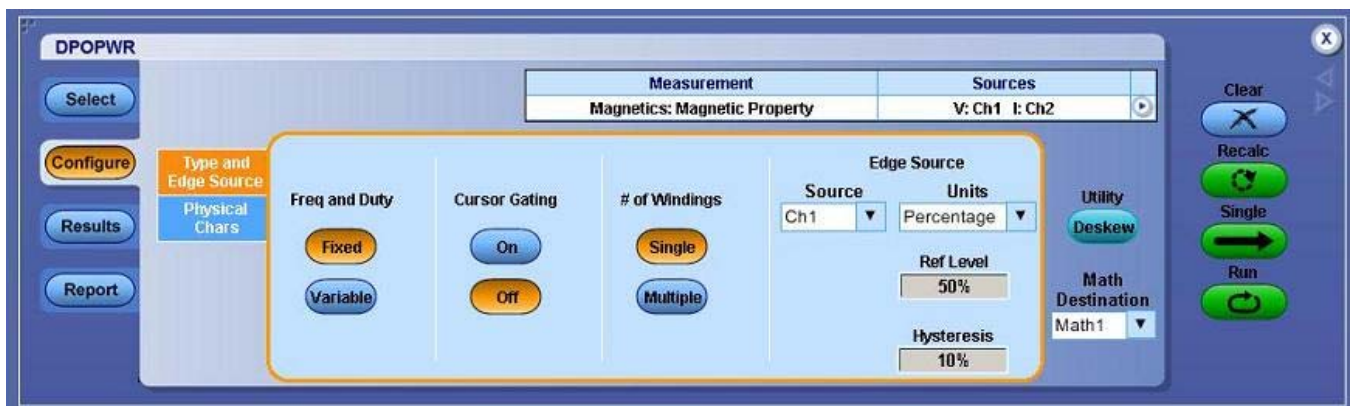


Figure 55: Magnetic Property configuration

The Type and Edge Source tab enables you to define the dimensions of the magnetic core.

1. Select the types of in-circuit operations like Variable or Fixed Freq and Duty cycle.
2. Select the Cursor Gating On button to measure the waveform data between the cursors

Note: Selection of Variable duty cycle enables Cursor Gating even if the Cursor Gating selection is Off.

3. Click Single or Multiple to select the no of windings from the # of Windings pane.
4. Use the drop down arrow in the Edge Source field to set the edge source. The data available in the selected channel is used to identify the complete integral cycle of an acquired waveform. This enables you to analyze the complex waveform across the inductor. The available options are:
 - Ch1-Ch4
 - Math1-Math4
 - Ref1-Ref4

Note: For the Variable operation, connect the gated drive signal to the edge source because the amplitude of the voltage acquired across the inductor will vary with time.

5. Select the Percentage or the Absolute option button from the Units drop down to set the Ref Level and Hysteresis values in percentage or the absolute value of the peak to peak signal.
6. Select the math destination from the Math Destination drop down list. The default destination is Math1.

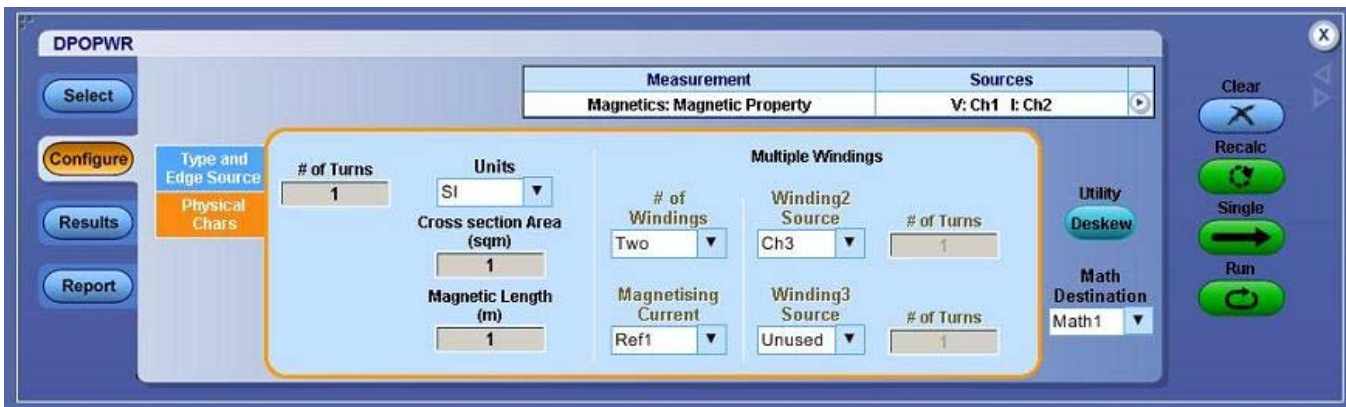


Figure 56: Magnetic Property configuration

In the Physical Characteristics tab, you can use the available parameters to specify the physical dimensions of the magnetic component. To do so, follow these steps:

1. Select the Multiple Windings option for a transformer or an inductor that has multiple windings on the same core
2. In the Units combo box, select the SI or CGS option buttons to set the system units. If you select SI, the dimensions of the component are measured in meters and units of Magnetics in Tesla and Amperes or Turns per meter. If you select CGS, the unit of measurement is centimeter and the result units are in Gauss and Oersted.
3. Double-click the Number of Turns field and use the keypad that appears to set the turns in the magnetic component.
4. Double-click the Cross SectionArea field and use the keypad that appears to set the cross section dimensions of the magnetic component. The acceptable ranges of the Cross Section Area values are:
 - 1nm² to 1Mm² if you have selected SI
 - 1cm² to 1Mcm² if you have selected CGS
5. Use the GP knob next to the Magnetic Length field to set the magnetic length of the component. The acceptable range of lengths are:
 - 1 mm if you have selected SI
 - 0.1 cm if you have selected CGS

In the Multiple Windings pane, you can measure the magnetic property of the components that has multiple windings like a coupled inductor. To do so, follow these steps:

1. Use the drop down arrow in the Winding2 Source and Winding3 Source combo boxes if you have connected to the other windings of the component.
2. Connect the voltage/current probe selected in the Source Configuration Panel to the:
 - Primary winding in a transformer winding
 - One winding in a couple inductor that is considered as the main winding
3. In the Type and Edge Source tab, use either the main winding voltage or the current channel as the edge source
4. In the Physical Chars tab, enter the number of turns in the # of Turns field.
5. In the Multiple Windings pane, connect the current probe to the other winding or the secondary winding and enter the number of turns of the

windings. If the number of windings is one, select the Unused option from the in the Winding3 Source combo box.

6. Follow the connection combinations of the voltage and current probe at the primary or the main winding at the secondary or the other windings to the dot as shown in the next figure. The voltage probe should be connected in such a way that the voltage is read as positive when the current rises.
7. In the Magnetising Current combo box, select the ref channel to view the magnetising current.

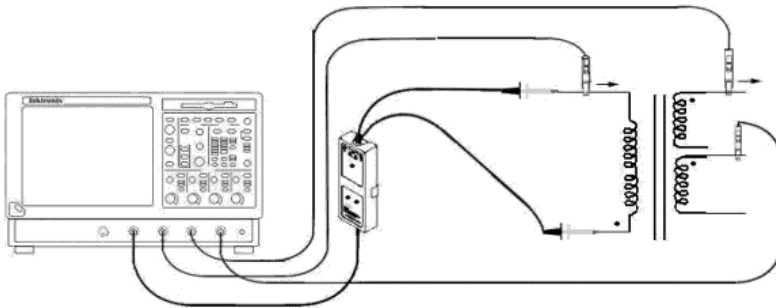


Figure 57: Windings setup



Warning: *When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.*

If you do not have a current probe to acquire the waveform simultaneously on two windings, select the Multiple Windings pane and follow the procedures given below:

1. In the Type and Edge Source tab, use either the main winding voltage or the current channel as the edge source.
2. Connect the voltage/current probe selected in the Source Configuration Panel to the:
 - Main winding in a transformer winding
 - One winding in a couple inductor that is considered as the main winding
3. Press the Run button in the application. The application acquires the voltage and current waveform of the main winding and prompts you to connect a current probe to the other winding. Select Yes if you have the other winding and No to proceed to the next step.

4. Set the current channel in the pop up message box and connect the current probe to the other winding. Enter the number of turns in the Num of Turns field for the selected winding.

Viewing Results-Magnetics

The Magnetics Measurement has different results based on the selected measurement. Follow these steps to view the results:

1. Select Analyze> Power Analysis> Results from the oscilloscope menu bar.
2. The application displays the results for the Magnetics (Magnetic Property) measurement in a tabular format.

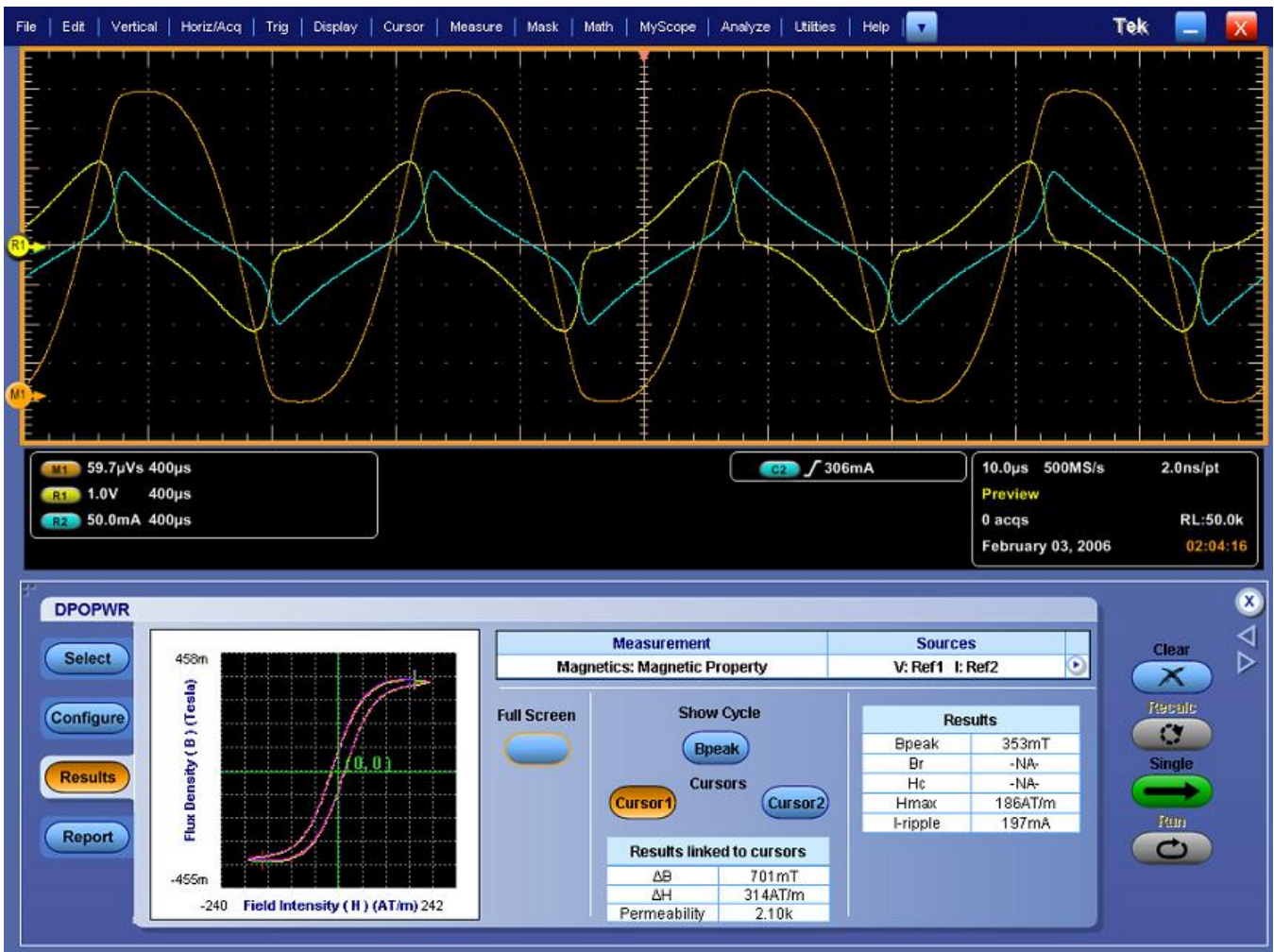


Figure 58: Magnetic Property results

Table 13: Magnetic property result fields

Field	Description
-------	-------------

Table 13: Magnetic property result fields

Field	Description
Bpeak	The maximum magnetic flux density induced in a magnetic component
Br	The induced magnetic flux density (B) that remains in the material after the externally applied magnetic field (H) is returned to zero while generating the hysteresis loop. Here the maximum value of the Br on the Bpeak cycle is measured.
Hc	The value of H at the intersection of the H-axis and the hysteresis loop. This represents the external field required to cause the induced flux density (B) to reach zero during while measuring a hysteresis loop. Hc is symmetrical with the positive and the negative Haxis.
Hmax	The maximum value of H at the intersection of the H-axis and the hysteresis loop.
I-ripple	The peak to peak value of the current

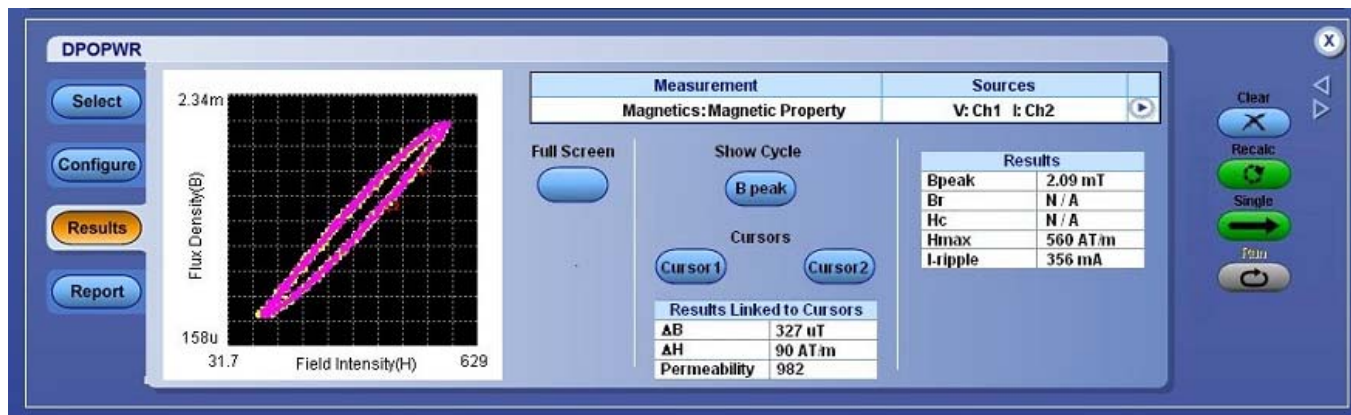


Figure 59: Magnetic Property half screen results

- Use the Cursor buttons to place the cursor on the maximum flux density curve. The Cursor1 and Cursor2 helps you to calculate the permeability, delta B, delta H on the region of interest on the maximum flux density curve.
- The Bpeak button toggles between Bpeak Cycles and All Cycles. When the Bpeak is selected, you can see only the two Bpeak cycles that have the maximum flux density. If the Bpeak is not selected, you can see all the cycles for the component.
- Click Full Screen to view the plot.
- Select the Dot connect option in the SOA Plot to connect all the data points and display the stray point. This ensures that all the data points display the visible transition.

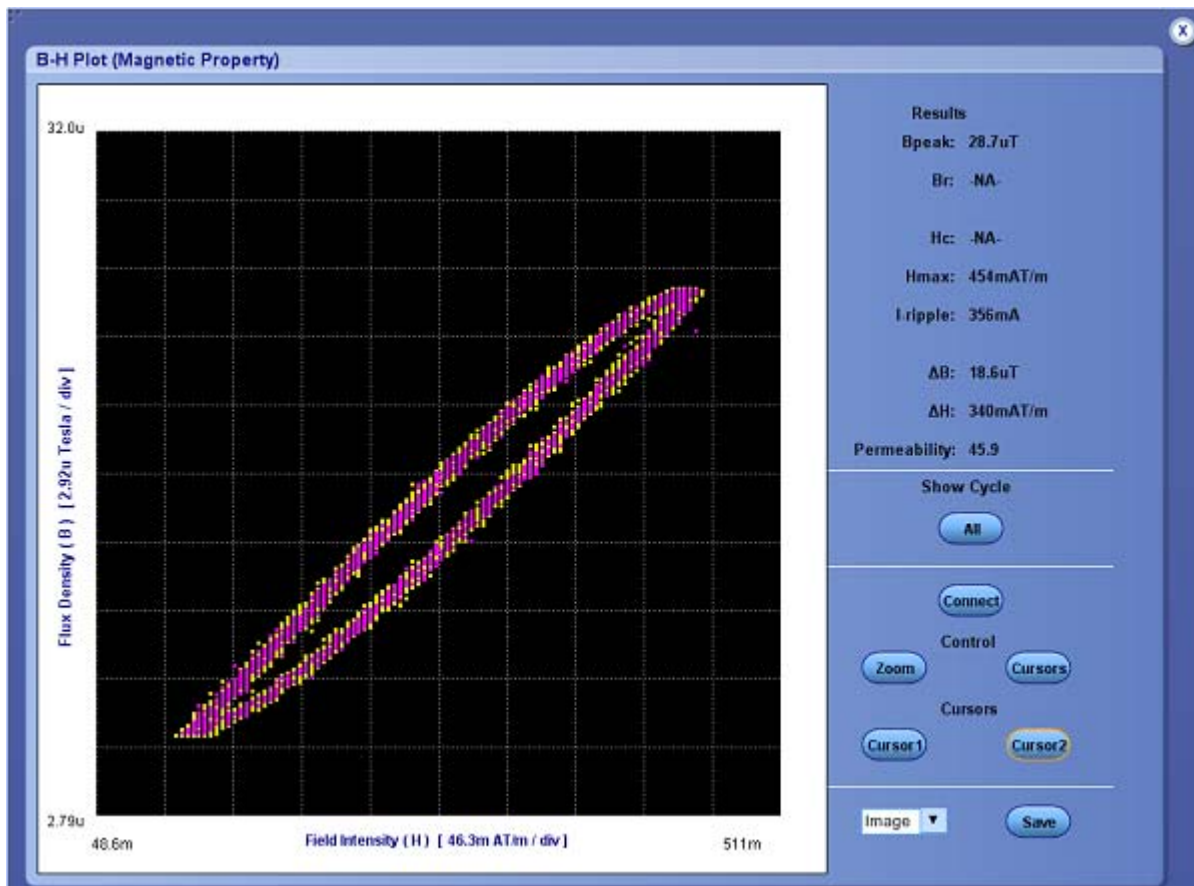


Figure 60: Magnetic Property full screen results

7. Click Save to save the plot as an image in the .jpg format. The default directory and the file name is C:/TekApplications/DPOPWR/Images.
8. Place the Cursor to select any data point in the plot.
9. Click Zoom to display the Zoom options.

Note: If the Cursor control button is enabled, the zoom buttons are disabled.

10. Click the In button to zoom in. You can zoom up to five times the normal view. Drag the mouse to define the area of interest. Click the Out button to zoom out. You can use the Zoom In and Zoom Out buttons until the application reaches the maximum and the minimum zoom limit. You can use the Zoom buttons in three different ways:
 - Dragged zoom-in: where the area is selected by dragging on the displayed window
 - Single-clicked zoom-in: when a point is clicked on the displayed window, the resulting window is equal to 1/4th of the displayed window.

- Single-clicked zoom-out: when a point is clicked on the displayed window, the resulting plot is the previous zoom state of the displayed window.

Note: If the current waveform is less than a division, the application displays an error message: "Unable to find the correct edges". This is because of the variation of data points within single division is very high and hysteresis band is insufficient to find the correct edges. Therefore, for the current waveform, the vertical range should be greater than one division..

11. Select the Reset option to restore the plot to its original display after zooming.

I vs Integral V Results:

1. Select Analyze> Power Analysis> Results from the oscilloscope menu bar.
2. The application displays the I vs Integral V results for the Magnetics measurement.
3. Click the View Plot option to view the Magnetics curve plot of the Current and Integral Voltage.

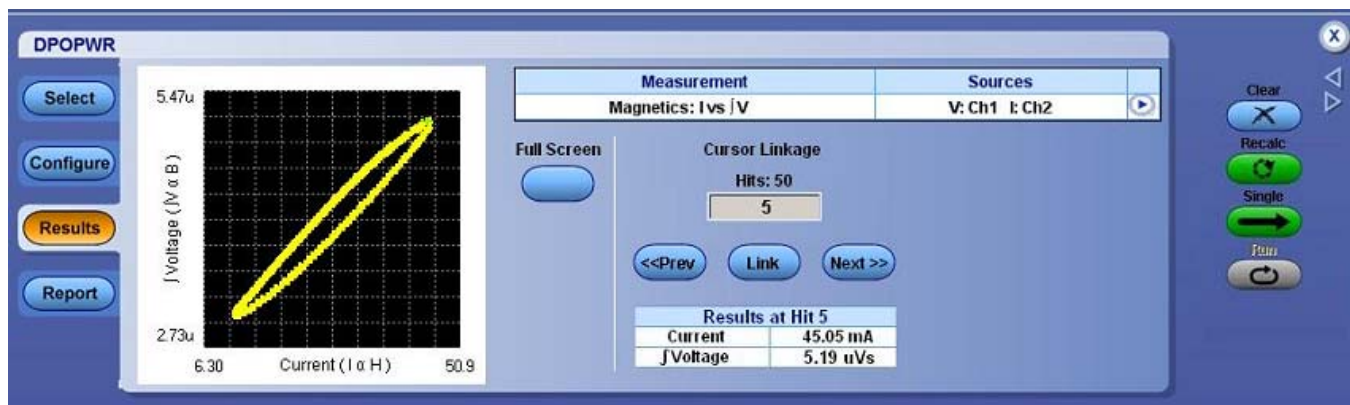


Figure 61: I vs Integral V results

4. Select the Dot Connect option to connect all the data points. You can view all the data points here.
5. Select the Save Image option to save the plot in the .jpg format. The default directory is C:/TekApplications/DPOPOWER/Images.
6. Select the Zoom controls to activate the Zoom options.

Note: If the Cursor button is enabled, the zoom buttons are disabled.

7. Select the Cursor option to select any data point from the plot.

8. Click the In button to zoom in. You can zoom up to five times the normal view. Drag the mouse to define the area of interest. Click the Out button to zoom out. You can use the Zoom In and Zoom Out buttons until the application reaches the maximum and the minimum zoom limit. You can use the Zoom buttons in three different ways:
9. Use the cursor in the Hits field to display the number of cursor hits at selected data points. When you place the cross hair icon on the I vs Integral V plot and if multiple occurrences of the data points exist at the same position, the values are displayed in the Hits field.
10. Select the Reset option to restore the plot to its original display.
11. The Link field displays the number of times a particular data point occurs. The minimum is 1 and the maximum is the number of times a data point occurs.
12. Select the Link button to link the data point you have selected to the waveform in the oscilloscope.
13. In case of multiple occurrences greater than one, select the Prev button to select the previous data point and the Next button to select the next data point.
14. Use the Half Screen option to view the B-H Plot and the time domain waveform simultaneously. Select the Cursor Linkage check box to select the data point to link the waveform in the oscilloscope to the I Vs V Integral Plot. Place the cross hair cursor on the plot and enter the values in the Cursor Linkage Hits field. Use the Zoom button to zoom the area of interest in the plot and display in the oscilloscope.

Inductance Results:

1. Select Analyze> Power Analysis> Results from the oscilloscope menu bar.

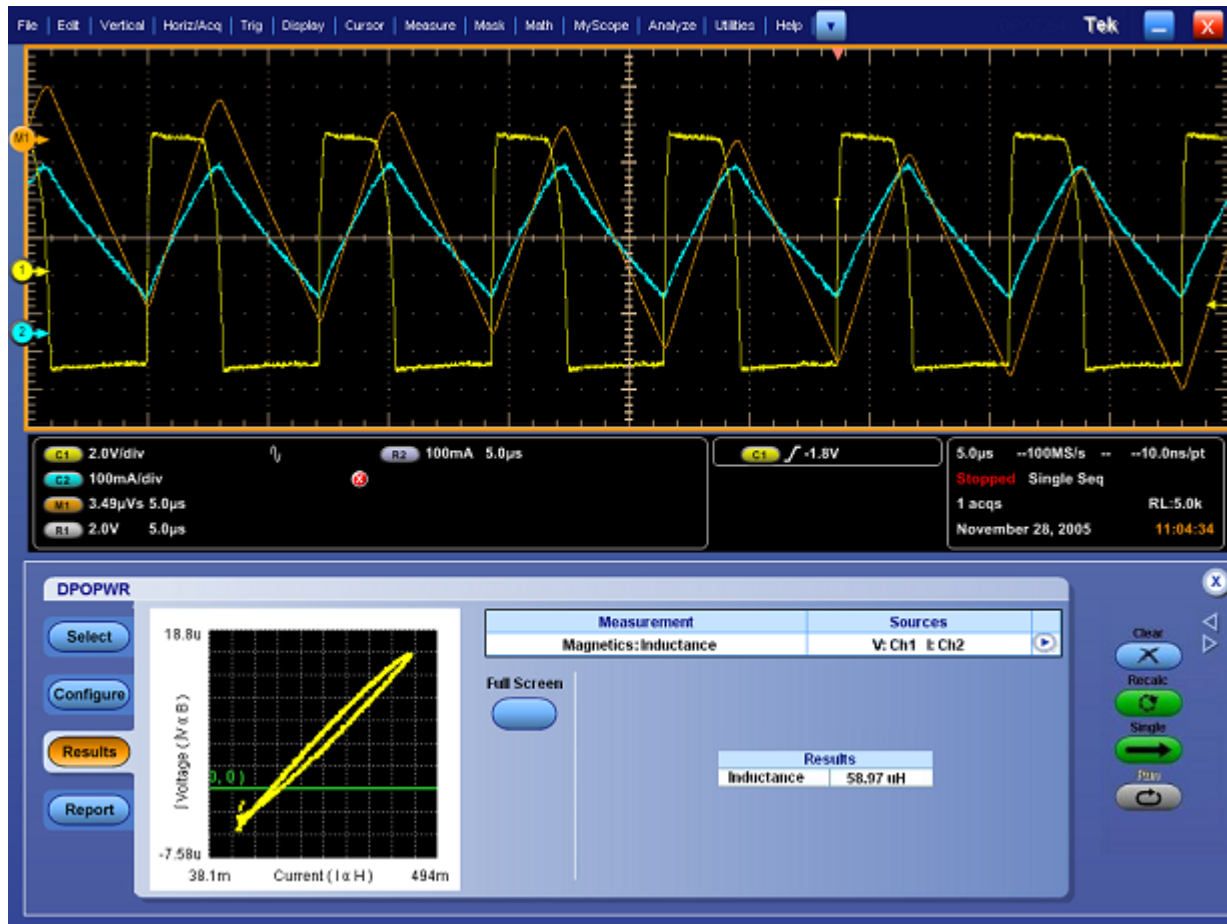


Figure 62: Inductance results

2. The application displays the B-H Curve plot with the Inductance value.

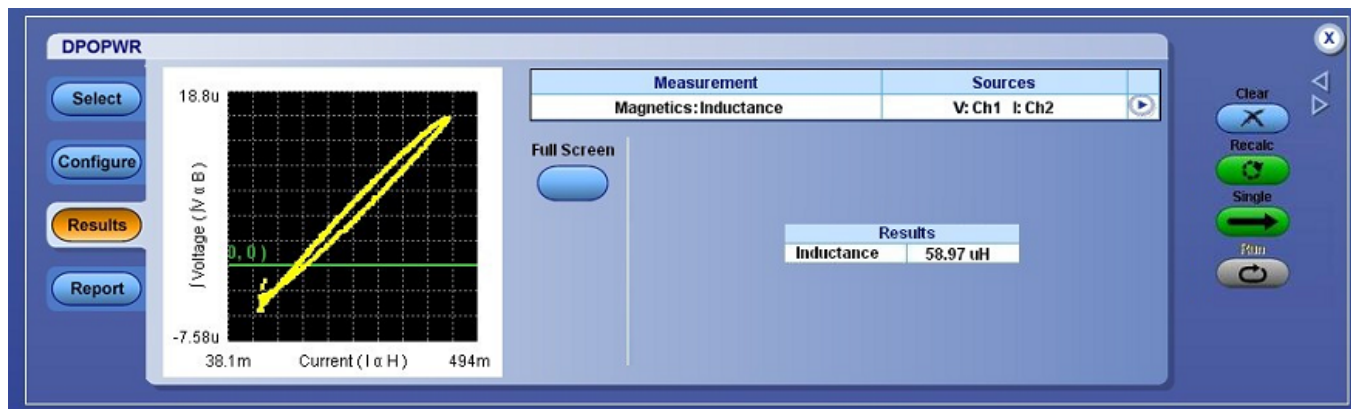


Figure 63: Inductance half screen results

You can select the Full Screen button to view the plot in full screen mode.

Generating Reports-Magnetics

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar, to display the Report Generator screen.
2. The application provides default factory templates for this measurement. The template file name is IvsIntegralV.rgt, Magnetic_Loss.rgt, Inductance.rgt, Magnetic_Property.rgt and the layout file name is Power_Device.rpl. When you click Generate Report, the application displays the default template file name. However, if you want to select or modify another template, follow the procedure given here.
3. Select the New Template button in Define Test Template tab to define and create a template for Magnetics. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Input/Ouput Analysis

Power Quality

Selecting and Configuring Measurements-Power Quality

To select and configure Power Quality Measurement, follow these steps:

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar.
2. Click Input/Output Analysis button to display Input/Output Analysis screen.
3. Click Power Quality in the Input Analysis pane. Now, click the Configure button.
4. The next figure shows a typical equipment setup for Power Quality.

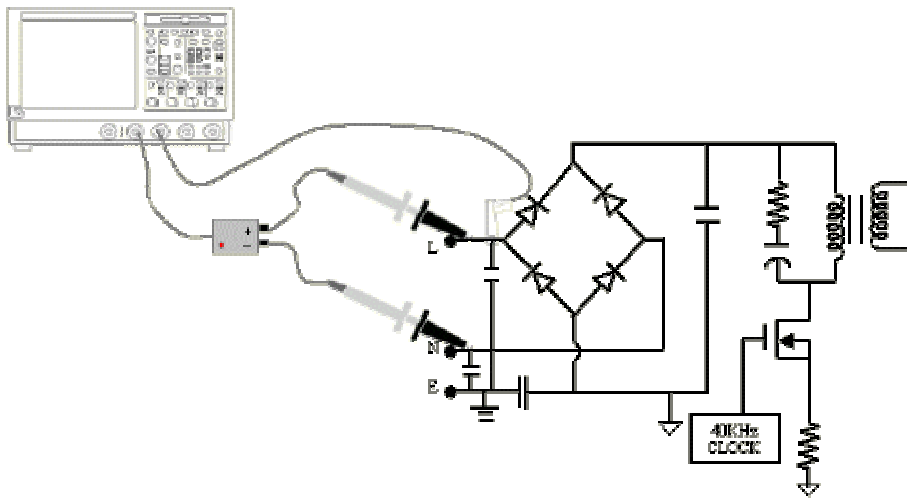


Figure 64: Power Quality setup



Warning: When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.

Configuring the Measurement

Follow the steps to configure the selected measurement:

1. Configure the options in the Source Configuration Panel.

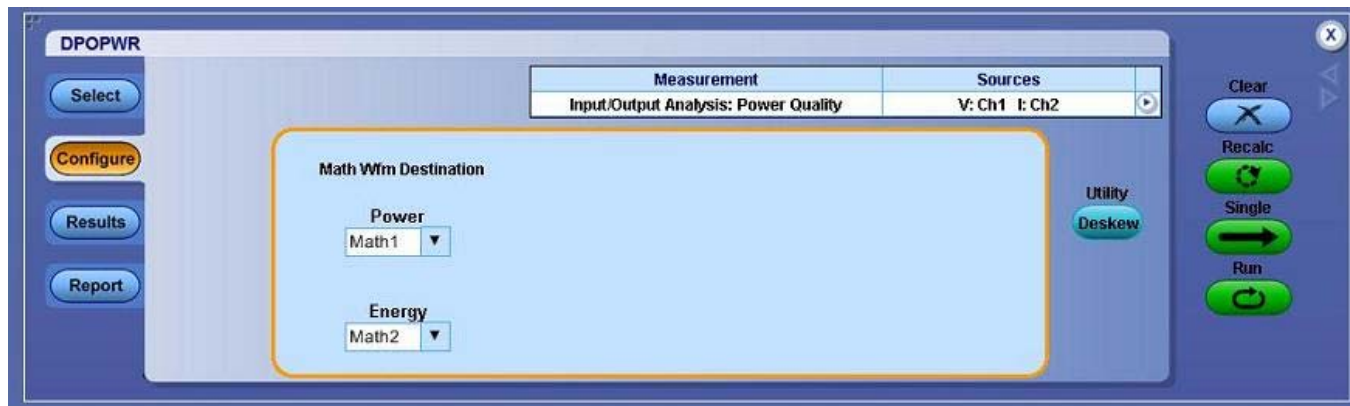


Figure 65: Power Quality configuration

2. In the Math Wfm Destination pane, set the Math option by clicking the Power and Energy combo boxes. The Math destination depends on the two or four channel math of the oscilloscope. The application automatically detects the availability of the Math channel in the oscilloscope.

Note: You cannot set the same Math channel in both the fields. If you do, the application prompts a conflict analysis message when you select the Run button. The Energy Waveform destination depends on the Math option available in the oscilloscope.

3. Press Run to measure.
4. If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Viewing Results-Power Quality

To view the results, follow these steps:

1. From the oscilloscope menu bar, select Analyze> Power Analysis> Results.
2. The application displays the results for the Power Quality measurement with the following data:
 - V Rms
 - True Power
 - Voltage Crest Factor
 - Frequency
 - I Rms
 - Apparent Power
 - Current Crest Factor
 - Power Factor

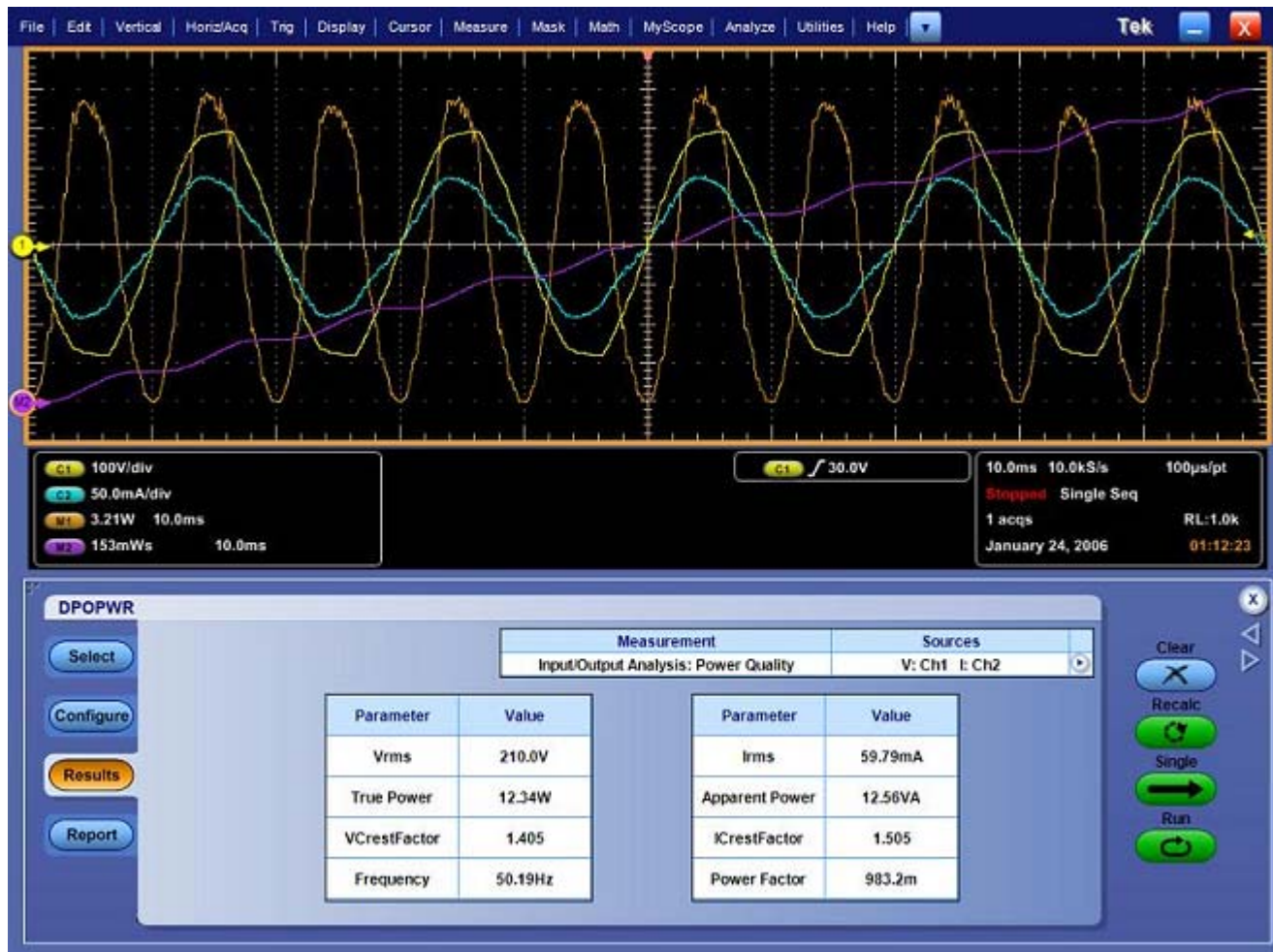


Figure 66: Power Quality results

Generating Reports-Power Quality

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar to display the Report Generator screen.
2. The application provides default factory templates for this measurement. The template file name is Power_Quality.rgt and the layout file name is Line_Power.rpl. When you click Generate Report, the application displays the default template file name. However, if you want to select or modify another template, follow the procedure given here.
3. Select the New Template button in Define Test Template tab to define and create a template for Power Quality. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.

5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Current Harmonics

Selecting and Configuring Measurements-Current Harmonics

To select and configure Current Harmonics Measurement, follow these steps:

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar.
2. Click Input/Output Analysis button to display Input/Output Analysis screen.
3. Click Current Harmonics in the Input Analysis pane. Now, click the Configure button.

Note: If you are using the same voltage source as used for any previous measurement, you can skip the procedure for configuring the common configuration options.



Warning: When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.

Configuring the Measurement

Follow these steps to configure the selected measurement:

1. Configure the options in the Source Configuration panel.
2. Click the Standard button to display the next screen. The pane displays the international EMC (Electro Magnetic Compatibility) standards 61000-3-2, AM 14 and MIL 1399 for which you can perform compliance testing.

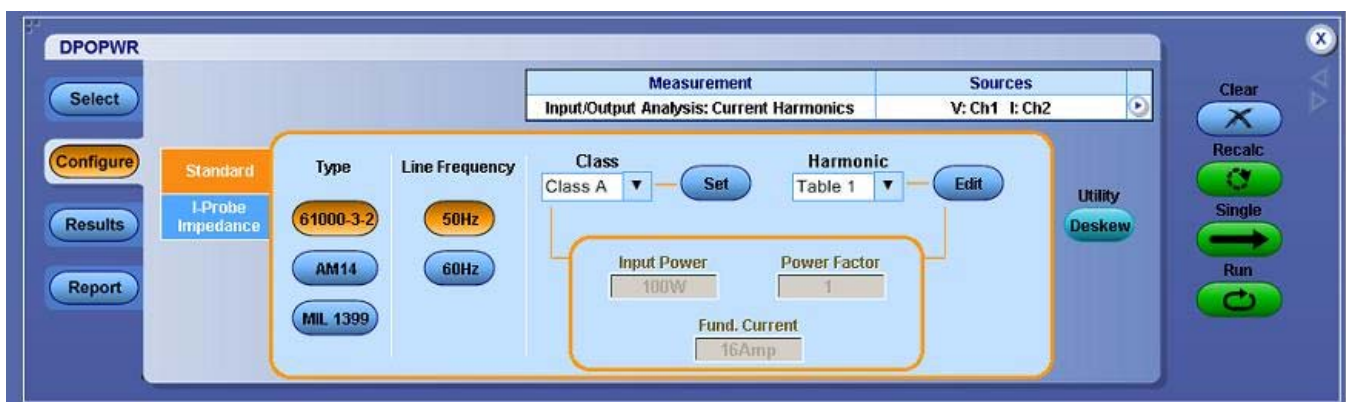


Figure 67: Current Harmonics configuration

3. Follow these steps to configure the IEC Standard configurations:
 - a. Click the IEC button in the Type pane. Then click either 50 Hz button or the 60 Hz button in the Line Frequency pane.
 - b. Use the Harmonics drop down box to select the table and click the Edit button to edit the table. The values in these tables are used as Limit values.
 - c. Select any one of the ten available Tables. Table1 is the default table. If you set the harmonic table, the application displays the class associated with in the Select Class combo box. If you run the application for the first time, the application associates Class A with all the harmonic tables. If you change the Class settings, the application retains the change after you exit and rerun the application.
 - d. Click the Edit button to display a table editor. The table editor displays the Harmonic Number from 1-40 for IEC standard and 1-50 for MIL standard, Harmonic Limits in milliamperes and the Harmonic Limits in decibel microamperes. Click OK to update the values in the table.

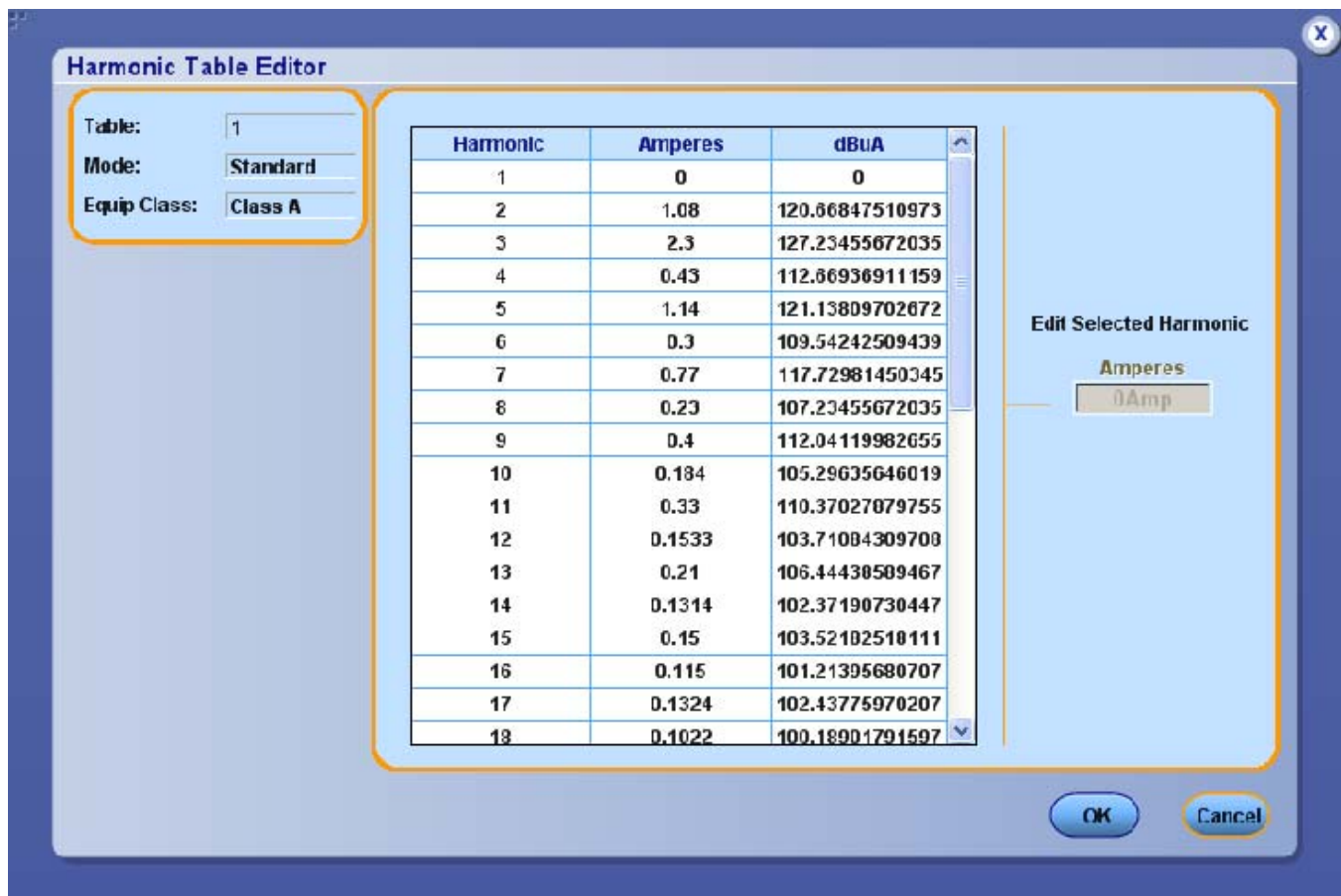


Figure 68: Harmonic Table editor

Note: You can edit only the Harmonic Limits values mA table. The same change is updated in the dbua column.

- e. Select a Class using the drop down arrow in the Class field. The available classes are: A to D.

Note: You can edit the harmonic table for Class A and B but not Class C and D.

If you select Class C, the application calculates the power factor limit and updates the limit table. If you select Class D, the application calculates the limit value of the harmonic from the true power of the unit under test.

- f. Click the Set button to automatically set the limit values of the Harmonic table to the selected standard values. These tables are applicable only to IEC-61000-3-2 standards and the Amd 14 standards.

IEC issued an amendment (amendment 14) to the IEC 61000-3-2 harmonics standard. This is documented as the IEC 61000-3-2 amendment 14.

This amendment changes the class definitions, reclassifies many products to Class A and clearly defines what Class-D products are (Class D products are limited to personal computers + monitors < 600 W, television receivers <600W) and affects the newly defined Class-D. The Class C and D limits are based on "Manufacturer's Ratings" which must be verified by measurement.

4. Follow these steps to configure the Amd 14 configurations:
 - a. Set the Line Frequency to either 50 Hz or 60 Hz.

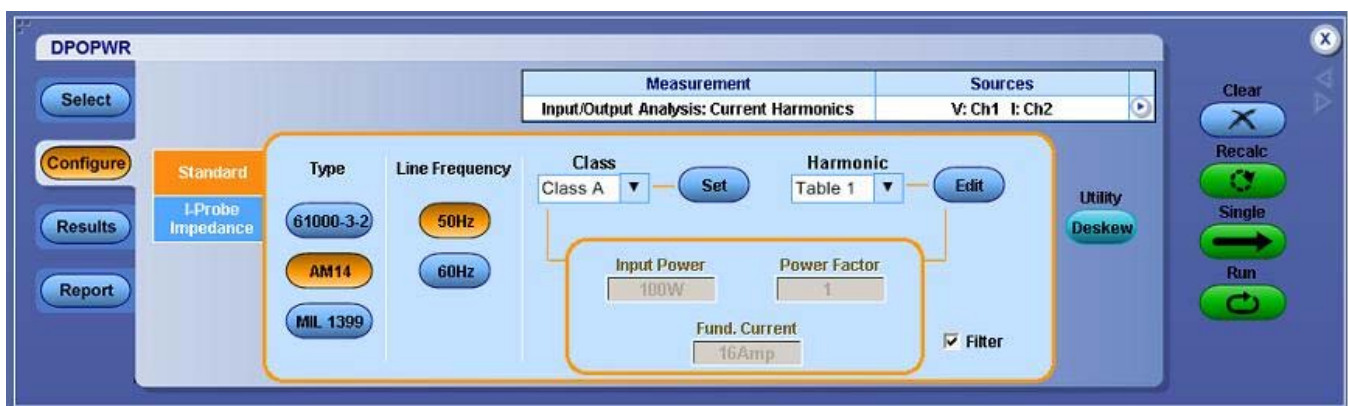


Figure 69: Amd 14 configuration

- b. Select the Table using the Harmonics drop down box and click the Edit button to edit the table. Select any one of the ten available Tables. Table 1 is the default table. If you set the harmonic table, the application displays the Class associated with in the Class combo box.

If you run the application for the first time, the application associates Class A with all the harmonic tables. If you change the class settings, the application retains the change after you exit and rerun the application.

- c. Click the Edit button to display a table editor. The table editor displays the Harmonic Number from 1-40 for IEC standard, Harmonic Limits in milliamperes and the Harmonic Limits in decibel microamperes. Click OK to update the values in the table.
- d. Select the Class using the Class drop down box. The available classes are: A to D. The Controls option provides additional inputs only to Class C or D.

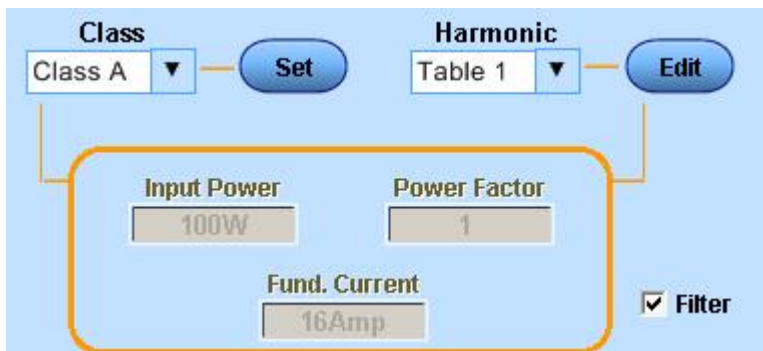


Figure 70: Controls screen

- e. Double-click the Input Power, Power Factor or Fundamental Current field and use the keypad that appears to enter the values and select OK. The acceptable ranges for:
 - Input Power is 0 W to 600 W. The default value is 100 W.
 - Power Factor is 0 to 1. The default value is 0.9.
- f. Fundamental Current is 0 A to 16 A. The default value is 16 A.
- g. Select the Filter checkbox to obtain filtered harmonic values.

Note: The signal is expected to be in steady state during the T_{obs} period where T_{obs} is the acquisition period for this measurement. This measurement has been set for very short cycle $T_{sv} = 3$ seconds.

Follow this procedure to configure the MIL standard parameters:

1. Set the Line Frequency to either 50 Hz or 400 Hz.

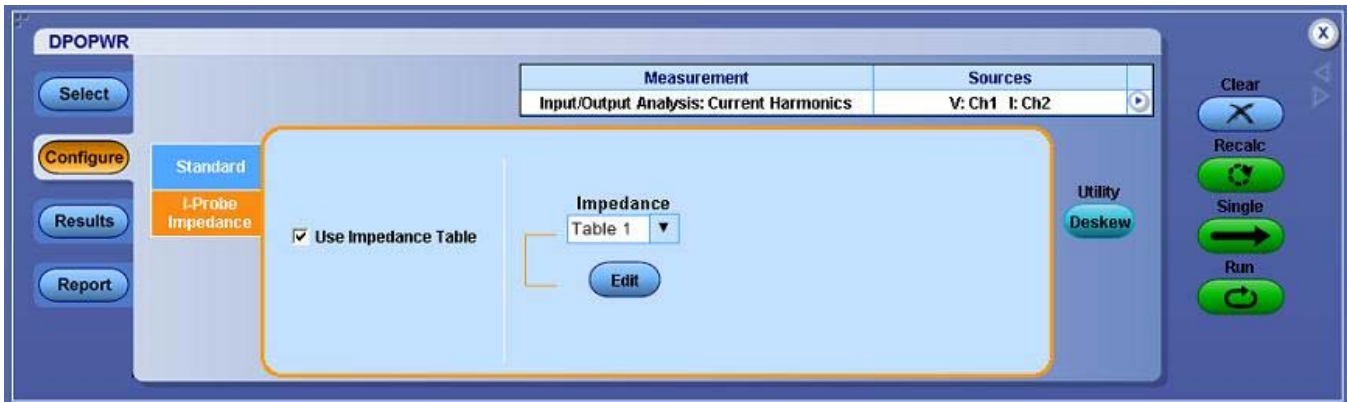


Figure 71: Current Harmonics configuration

2. You can enter the frequency deratings of any custom probe that you are using for the current measurement. I-Probe is applicable for the selected current source where the frequency derating is compensated in measurements such as 61000-3-2, Amd 14 and MIL1399 standards. Use the drop down arrow in the Impedance Table to select any one of the ten available tables. The Impedance table enables to enter frequency response of the probe. This helps you to measure the frequency component altered by the frequency response of the probe. Table 1 is the default table. Use the Edit button to edit the impedance table.

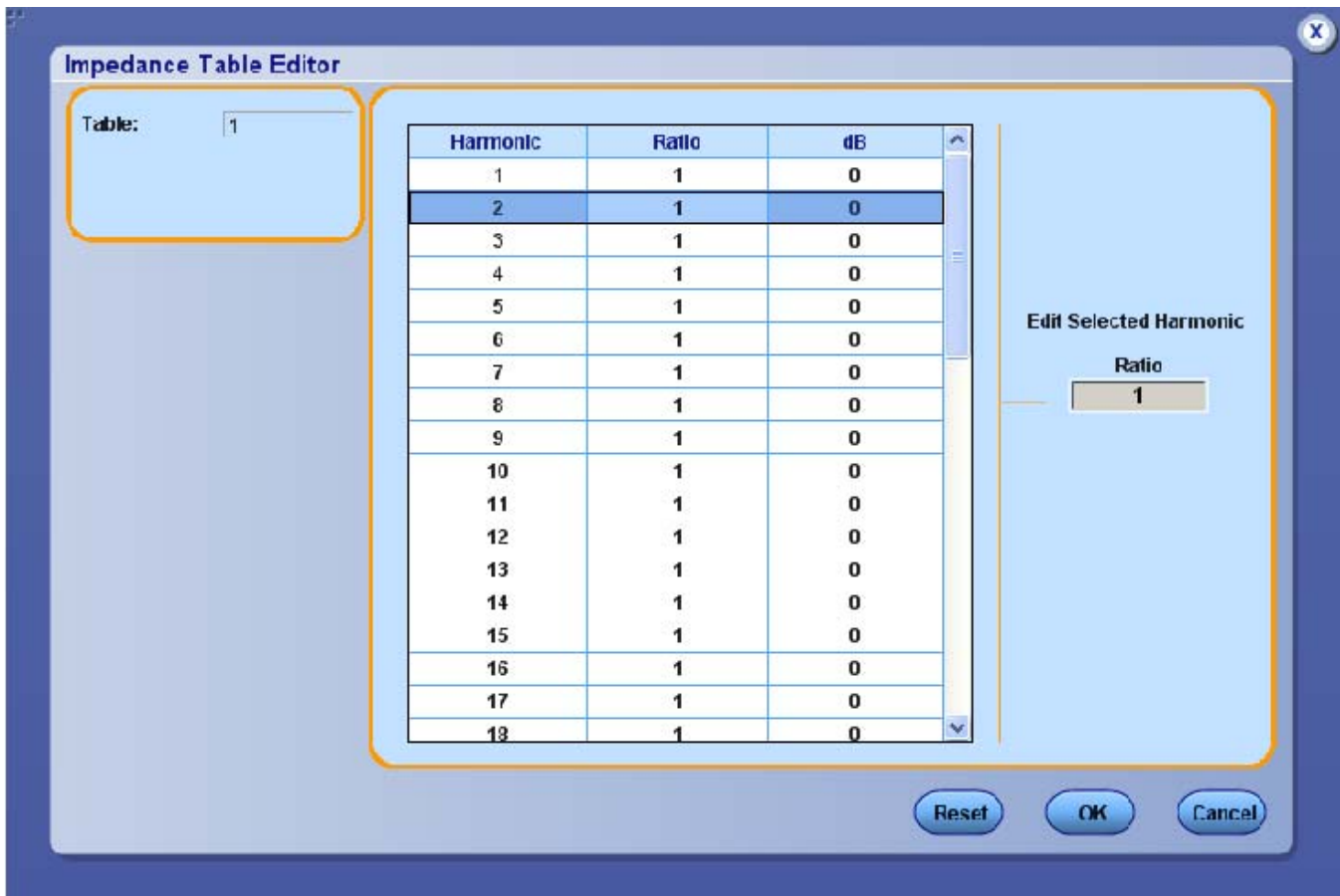


Figure 72: Impedance table editor

You can use the Impedance table to set the transfer impedance of the probe at each harmonic frequency. The table editor displays the Harmonic Number from 1-40 for IEC standard and 1-50 for MIL standard, the Impedance ratio and the Impedance ratio in decibels.

3. To edit the harmonic value, select the value. Double-click the Edit Selected Harmonic Ratio field and use the keypad that appears to edit the Impedance ratio values. Select OK.
4. Click Run to measure.

If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Viewing Results-Current Harmonics

To view the results, follow these steps:

1. From the oscilloscope menu bar, select Analyze> Power Analysis> Results or click the View Results button in the Current Harmonics Results panel.

IEC 61000-3-2 results

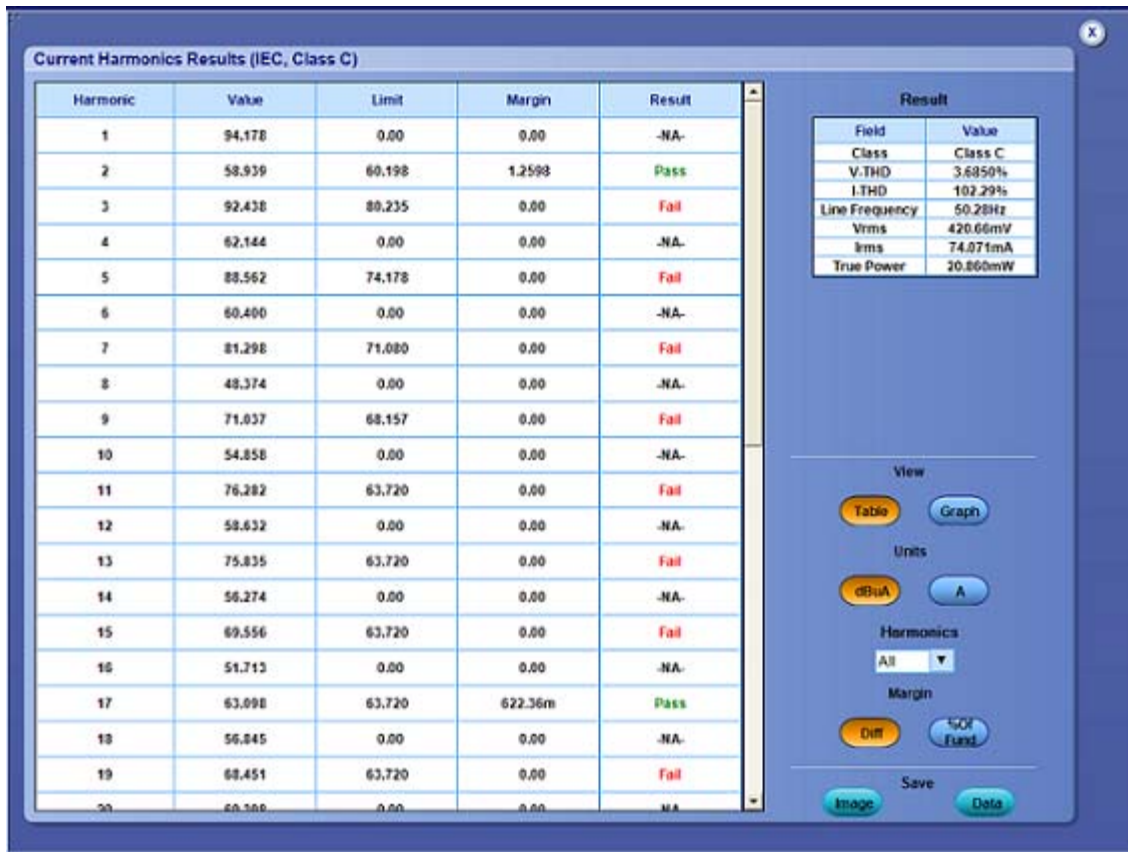


Figure 73: Current Harmonics IEC Standards results

If you have selected the IEC standard, the Current Harmonic Results displays the following information:

Table 14: IEC Standard result fields

Option	Description
Harmonic	Displays the harmonic number
Value	Displays the measured value in dBuA
Limit	Displays the harmonic limits in dBuA
Margin	Displays the difference between the Value and Limit in dBuA. The Status depends on the positive or negative margin value
Result	Displays the status of the measurement-Pass or Fail

2. Select Table or Graph button from the View pane to view the results in the table or a Graph format. If you have set the MIL Standard, the mask option is not available in the View field. You can set the units to dBuA or A in the Units pane. The Harmonics field displays harmonic number as Odd, Even or All. The All option displays the complete set of harmonic values. Even displays the even numbered values from 2-40. Odd displays the odd numbered values from 1-39.
3. In the Field and Value table, the Class value field displays the selected class, Line Frequency value in Hz, Total Harmonic Distortion in percentage (voltage and current), Line Voltage in Volts, Line Current in Amperes, and Total Power in Watts.
4. Select the Data button to save the data to a appropriate location.
5. Select the drop down arrow in the View field to select the format: Table, Bar graph or Mask. In the bar graph, red indicates the Fail status, green for Pass, and gray to set limits. The Mask option is available only if the harmonic table has Class A or D device. In the Mask results, Red indicates the mask boundary, yellow indicates one half cycle of the input current waveform. If the input waveform is completely inside the mask boundary, the percentage in window is 100%. The percentage window indicates how much waveform is inside the mask. X axis displays the waveform angle in degrees. Y axis indicates the i/i peak where i is the current and i peak is the peak of the current in one half cycle.
6. Select dBuA or the A button from the Units pane to change the units to decibel microamperes or amperes.

Note: Any change in the Units field is reflected in the Value, Limit and Margin fields of the table and the bar graph.

You can export the .csv file only if you select the table format.

7. Select the Image button to save the data as image in the appropriate location.

Amd 14 results

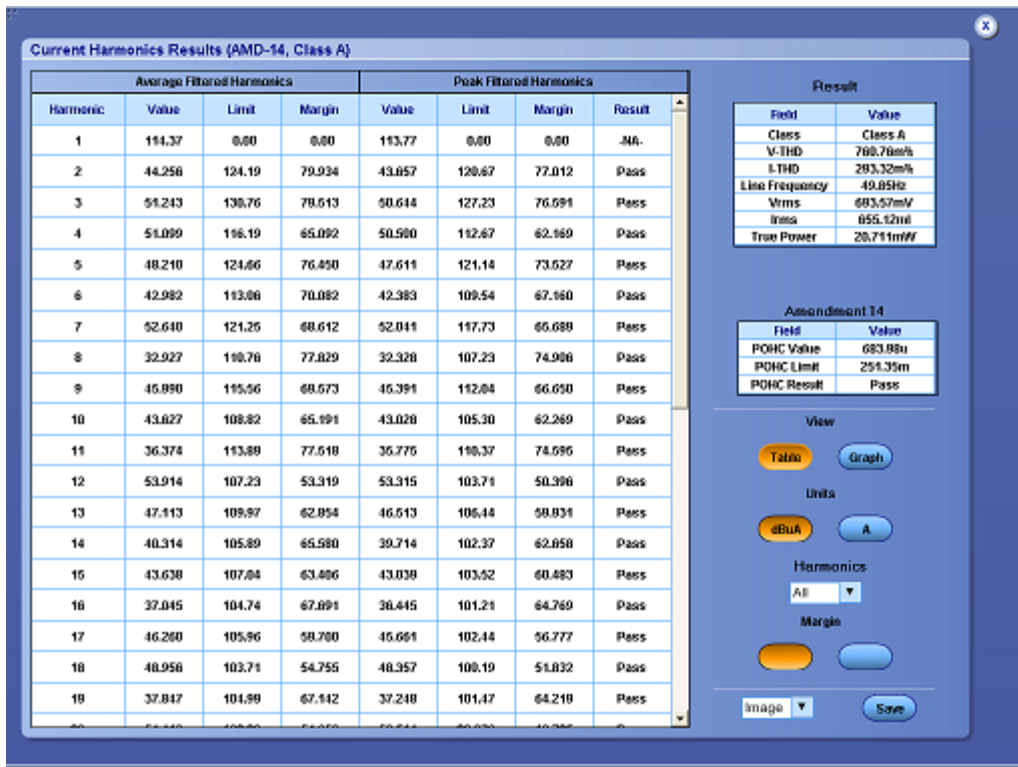


Figure 74: Current Harmonics Amd 14 results

If you have selected the Amd 14 standard, the Current Harmonic Results displays 40 harmonic values with the following information:

Table 15: Amd 14 results

Option	Description
Harmonic	Displays the harmonic number
Value	Displays the measured value in dBuA
Limit	Displays the harmonic limits in dBuA
Margin	Displays the difference between the Value and Limit in dBuA. The Status depends on the positive or negative margin value
Result	Displays the status of the measurement-Pass or Fail

Follow these steps to view the results in the Bar Graph format:

1. Click the Graph button in the View pane to view the results in the Bar Graph format.

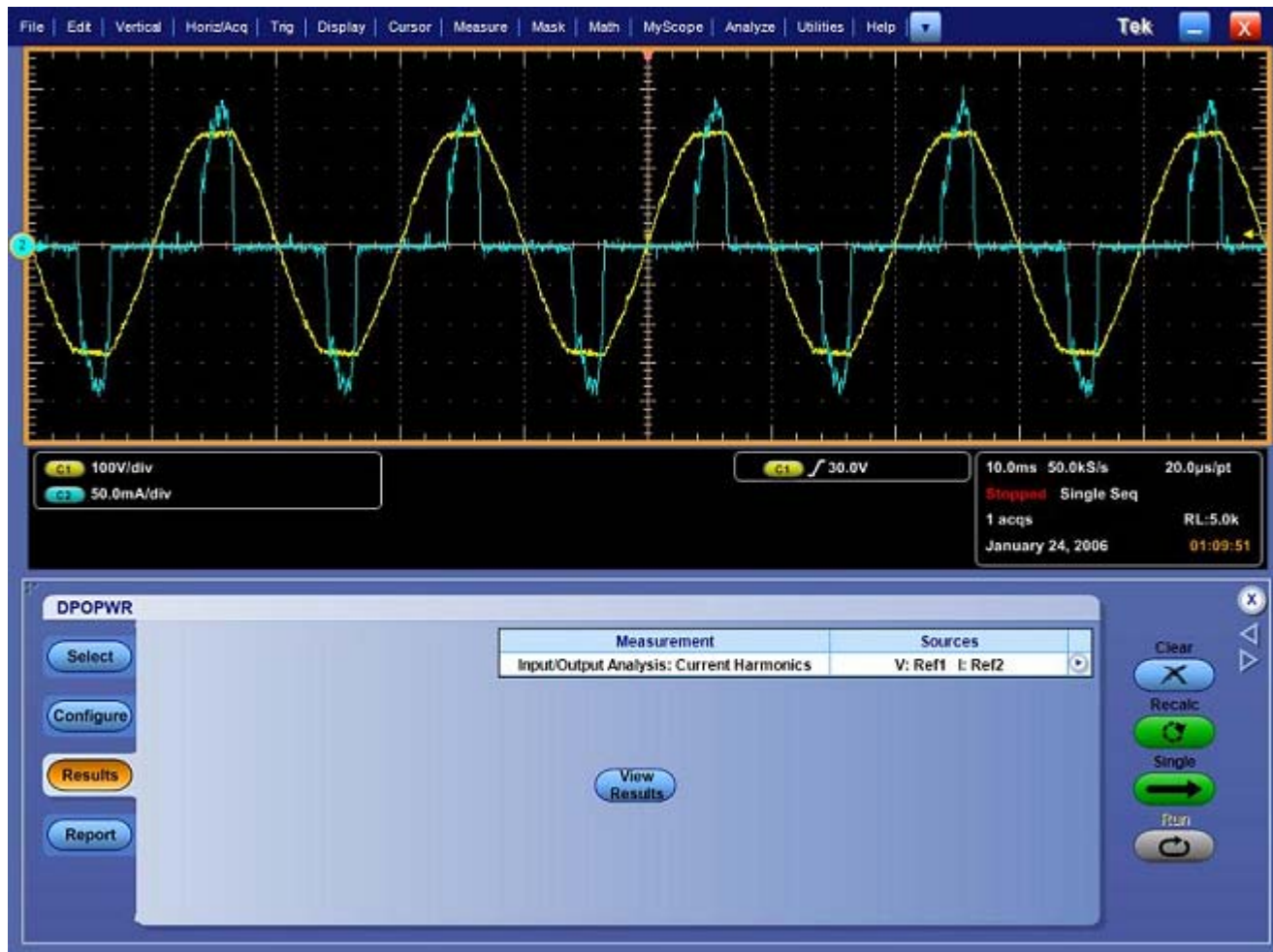


Figure 75: Current Harmonics results

2. Select the dBuA or A button from the Units pane to set the units to dBuA or A.
3. Select the drop down arrow in the Harmonics field to displays harmonic number as Odd, Even or All. The All option displays the complete set of harmonic values. Even displays the even numbered values from 2-40. Odd displays the odd numbered values from 1-39.

In the Field and Value table, the Class value field displays the selected class, Line Frequency value in Hz, Total Harmonic Distortion in percentage (current and voltage), Line Voltage in Volts, Line Current in Amperes, and Total Power in Watts.

The Field and Value table in the Amendment 14 pane displays the Partial Odd Harmonic Current value and Limit in amperes and the Partial Odd Harmonic Current status as pass or fail. The Partial Odd Harmonic Current is defined as the total rms value of the odd harmonic components of orders 21 to 39.

4. Click the Save button to save the result as an image in the appropriate location.

If you have selected the Amd 14 standard with Filter, the Current Harmonic Results displays 50 harmonic values with the following information:

Table 3-16: Amd results with Filter

Option	Description
Average Filtered Harmonics	Displays the average value of the individual harmonic currents taken over the entire test period. (measured average harmonics, limit average harmonics and margin (difference between limit and measured value))
Peak Filtered Harmonics	Displays the maximum value for each filtered harmonics.
Value	Displays the measured value in dBuA
Limit	Displays the harmonic limits in dBuA
Margin	Displays the difference between the Value and Limit in dBuA. The Status depends on the positive or negative margin value
Result	Displays the status of the measurement-Pass or Fail

Generating Reports-Current Harmonics

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar, to display the Report Generator screen.

Note: The current harmonics report displays the table with the data and not the bar graph. You have to generate separate reports for decibel microamperes, amperes, even values, and odd values.

2. The application provides default factory templates for this measurement. We recommend you to use the file layout only. The template file name is Current_Harmonics_Page1.rgt. The layout file name is Current_Harmonics.rpl. If you have selected Amd 14> Filter option, the layout file name is Current_Harmonics_Amd14.rpl. However, if you want to select or modify another layout, follow the procedure given here.
3. Select the New Template button in Define Test Template tab to define and create a template for Current Harmonics. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Total Power Quality

Selecting and Configuring Measurements-Total Power Quality

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar.
2. Click Input/Output Analysis button to display Input/Output Analysis screen.
3. Click Total Power Quality in the Input Analysis pane. Now, click the Configure button.



Warning: When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.

Configuring the Measurement

Note: If you use the same voltage source as used for an earlier measurement, you can skip the procedure for configuring the Source Configuration options.

Follow these steps to configure the selected measurement:

1. Click Standard button. The Standard pane displays the international EMC (Electro Magnetic Compatibility) standards IEC 61000-3-2, Amd 14 and MIL1399 for which you can perform compliance testing.

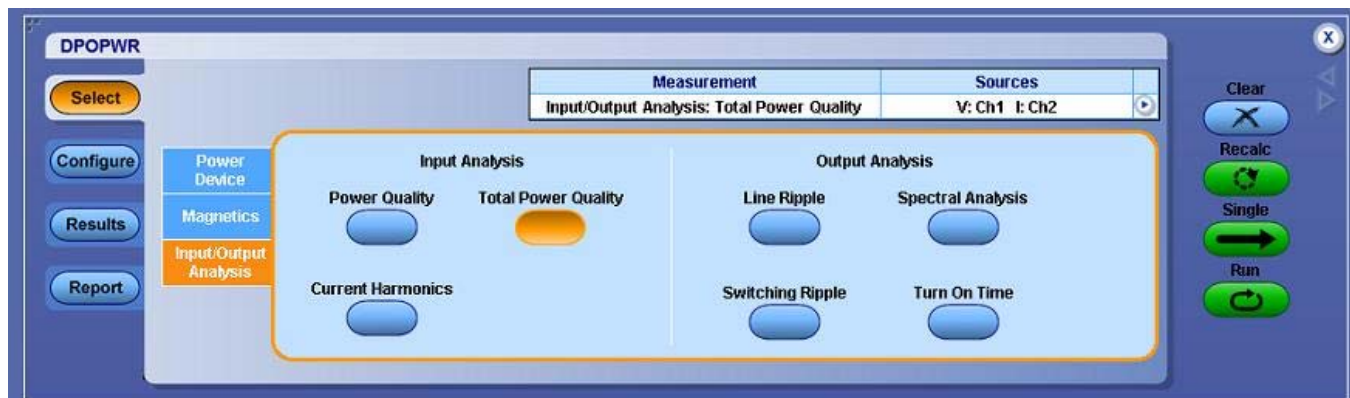


Figure 76: Total Power Quality configuration

Follow these steps to configure the IEC Standard configurations:

- a. In the Standard tab, select the IEC 61000-3-2 button. Set the Line Frequency to either 50 Hz or the 60 Hz.

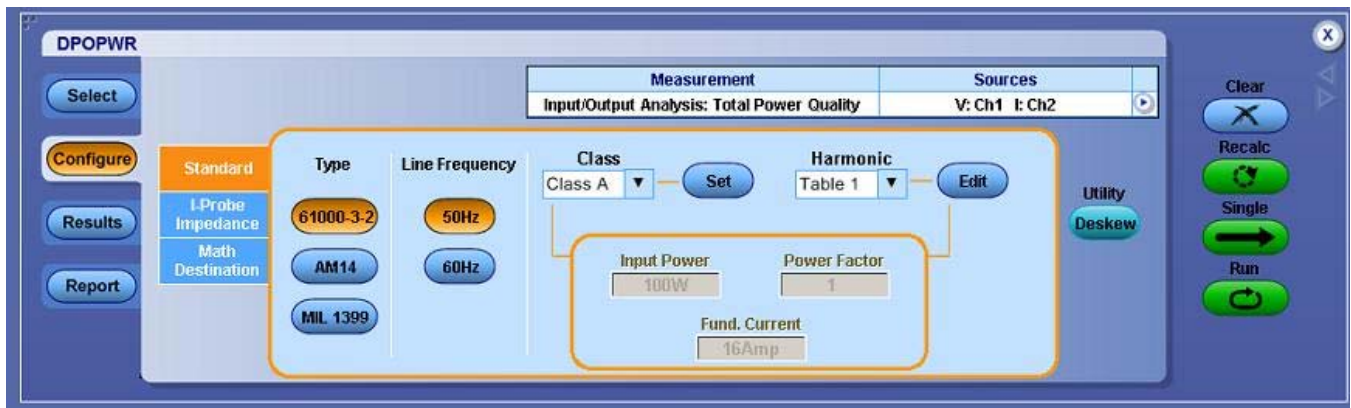


Figure 77: Total Power Quality configuration

- b. Click the Harmonic combo box to select the table and click the Edit button to edit the table.
- c. Select any one of the ten available Tables. Table 1 is the default table. If you set the harmonic table, application displays the Class associated with in the Class combo box. If you run the application for the first time, the application associates Class A with all the harmonic tables. If you change the Class settings, the application retains the change after you exit and re-run the application.
- d. Click Edit button to display a table editor. The table editor displays the Harmonic Number from 1-40 for IEC standard and 1-50 for MIL standard, Harmonic Limits in milliamperes and the Harmonic Limits in decibel microamperes. Click OK to update the values in the table.

Note: You can edit only the Harmonic Limits values mA table.

- e. Click the Class combo box to select a Class. The available Classes are: A to D.

Note: You can edit the harmonic table for Class A and B but not Class C and D.

- f. If you select the Class C, the application calculates the power factor limit and updates the limit table. If you select Class D, the application calculates the limit value of the harmonic from the true power of the unit under test.
- g. Click the Set button to set the Harmonic table to the selected standard.

IEC issued an amendment (amendment 14) to the IEC 61000-3-2 harmonics standard. This is documented as the IEC 61000-3-2 amendment 14.

This amendment changes the class definitions, reclassifies many products to Class A and clearly defines what Class-D products are (Class D products are limited to personal computers + monitors < 600 W , television receivers <600W).

Follow these steps to configure the Amd 14 Standard configurations:

1. Set the Line Frequency to either 50 Hz or 60 Hz.

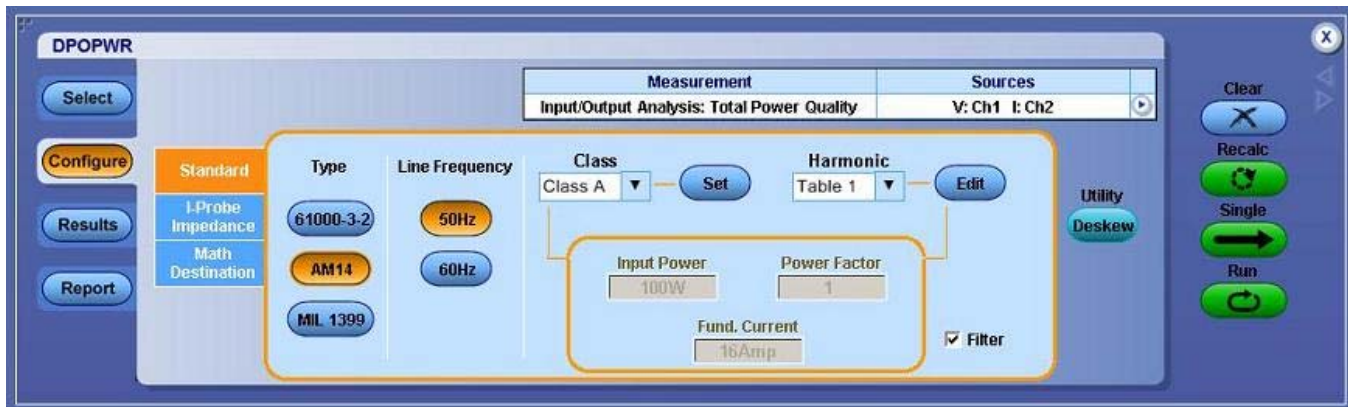


Figure 78: Total Power Quality configuration

2. Click the Harmonic combo box to select the table and use the Edit button to edit the table.
3. Select any one of the ten available Tables. Table 1 is the default table. If you set the harmonic table, application displays the Class associated with in the Class combo box. If you run the application for the first time, the application associates Class A with all the harmonic tables. If you change the Class settings, the application retains the change after you exit and re-run the application.
4. Click the Edit button to display a table editor. The table editor displays the Harmonic Number from 1-40 for IEC standard, Harmonic Limits in milliamperes and the Harmonic Limits in decibel microamperes. Click OK to update the values in the table.
5. Use the Class combo box to select a Class. The available Classes are: A to D. The Controls option provides additional inputs only to Class C or D.
6. Select the Controls button to display the controls screen.
7. Double-click the Input Power, Power Factor or Fundamental Current field and use the keypad that appears to enter the values and select OK. The acceptable ranges for:
 - Input Power is 0 W to 2 Kw. The default value is 100 w
 - Power Factor is 0 to 1. The default value is 1
 - Fundamental Current is 0 A to 16 A. The default value is 16 A
8. You can set the Filter Off if you do not want to obtain filtered harmonic values.

You can configure the MIL Standard configurations by setting the Line Frequency to either 50 Hz or 60 Hz.

Follow these steps to configure the I-Probe Impedance:

1. Click the Impedance combo box to select any one of the ten available tables. The Impedance table enables the frequency response of the probe. This helps you to measure the frequency component altered by the frequency response of the probe. Table1 is the default table. Use the Edit button to edit the impedance table. The default table is Table1.

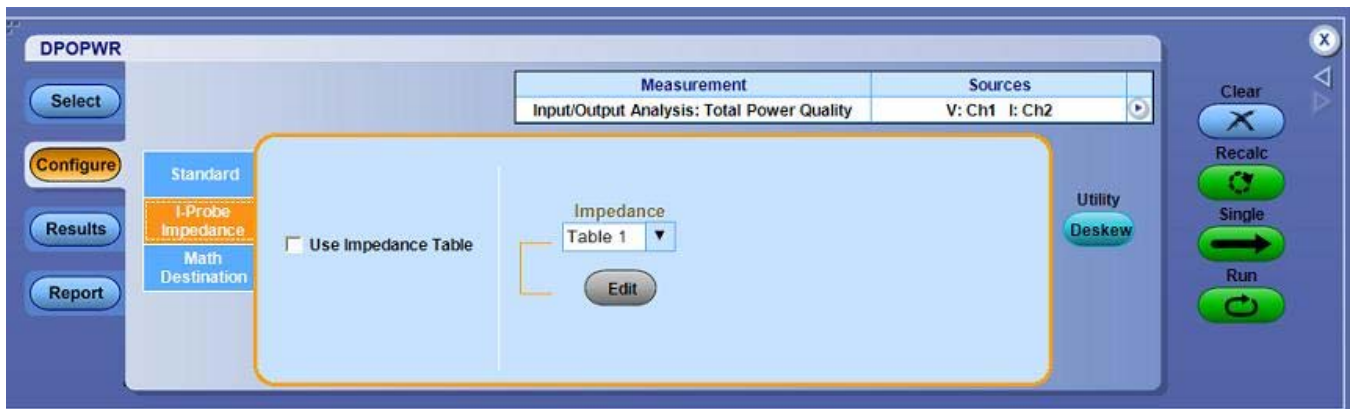


Figure 79: Total Power Quality configuration

2. Click the Edit button to display a table editor. The table editor displays the Harmonic Number from 1-40 for IEC standard and 1-50 for MIL standard, the Impedance ratio in decibel units. You can edit only the Impedance ratio column in the table. Select OK.

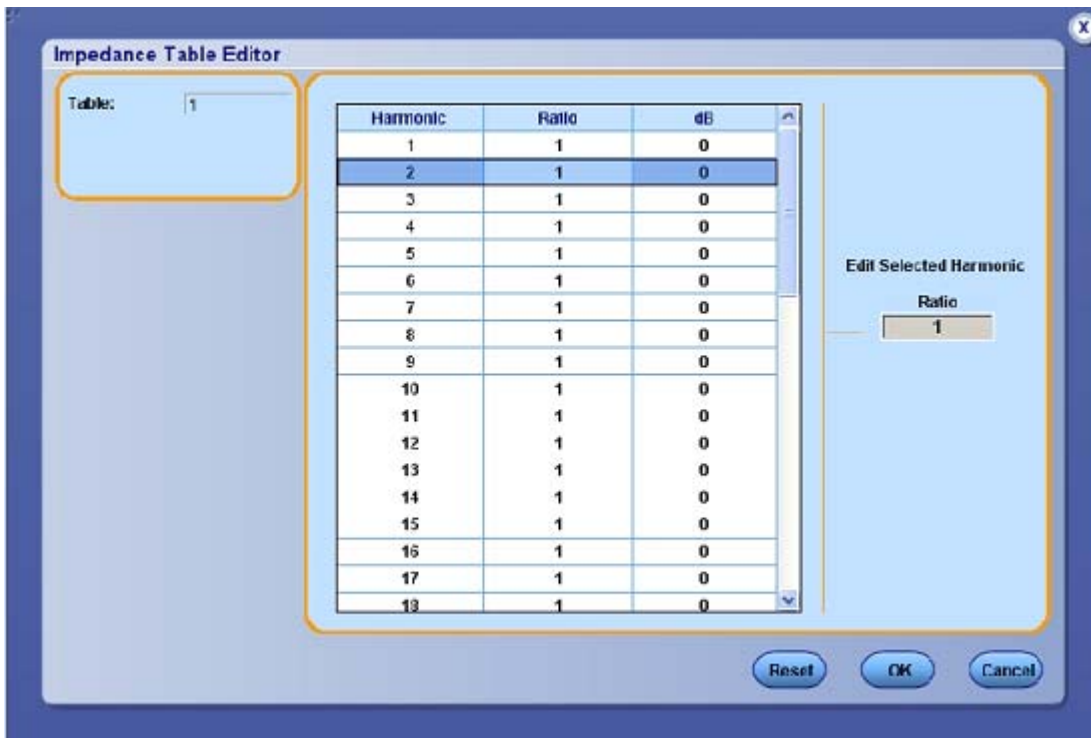


Figure 80: Impedance Table Editor

3. Click the Reset button in the Impedance Table Editor to reset the ratio and dB values to the default values. You can view the Harmonic Limits, the Harmonic Number and the Harmonic Limits in decibel microamperes. You can use the Impedance table to set the transfer impedance of the probe at each harmonic frequency. The table editor displays the Harmonic Number from 1-40 for IEC standard and 1-50 for MIL standard, the Impedance ratio and the Impedance ratio in decibel.

Note: You cannot edit the harmonic table for Class C and D.

4. To edit the harmonic value, select the value. Double-click the Edit Selected Harmonic Ratio combo box and use the keypad that appears to edit the Impedance ratio values. Select OK.
5. In the Math Wfm Destination tab, select the Math destination in the Power Waveform Destination and Energy Waveform Destination combo boxes.
6. Click Run to measure.
7. If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Viewing Results-Total Power Quality

To view the results, follow these steps:

1. Select Analyze> Power Analysis> Results to display the following screen. The application displays the results if the measurement is successful or from the oscilloscope menu bar.

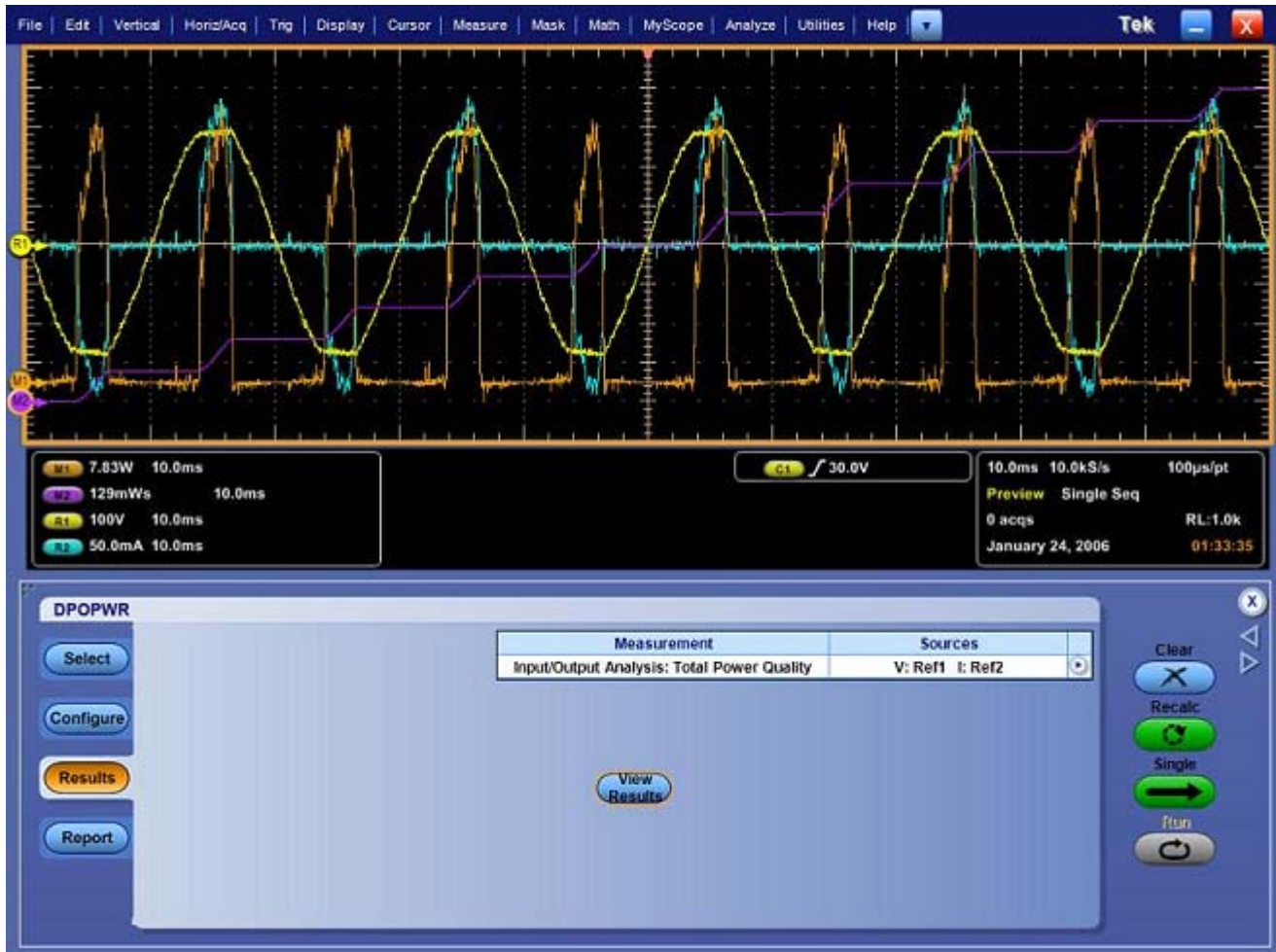


Figure 81: Total Power Quality results

Note: Since the measurement is a combination of Power Quality and Current Harmonics measurements, the results also reflects the same combination.

2. Click the View Results button to display the following results:

Table 3-17: Total power quality results

Option	Description
Harmonic	Displays the harmonic number
Value	Displays the measured value
Limit	Displays the IEC standard limits or the user defined limits set by you
Margin	Displays the difference between the Value and Limit
Result	Displays the status of the measurement- Pass or Fail

If you have selected MIL 1399 standard, the application displays 50 harmonic values. In the Field and Value table, the Class value is displayed as NA, Mode displays MIL Standard, Line Frequency value in Hz and Total Harmonic Distortion in percentage.

3. Select the drop down arrow in the View field to set the format: Table, Bar graph or Mask. If you have set the MIL Standard, the mask option is not available in the View field. The Mask option is available only if the harmonic table has Class A or D device. In the Mask results, Red indicates the mask boundary, yellow indicates one half cycle of the input current waveform. X axis displays the waveform angle in degrees. Y axis indicates the i/i peak where i is the current and i peak is the peak of the current in one half cycle.
4. In the bar graph, the red indicates Fail status, green- Pass and yellow is for the set limits. The mask option displays the True Power and the class of the device detected.
5. Select the drop down arrow in the Harmonics field to display the harmonics in three modes -All, Even and Odd. All harmonics displays the complete set of harmonic values. Even harmonics display the even numbered values from 2-40. Odd harmonics display the odd numbered values from 1-39.
6. Use the drop down arrow in the Units field to change the units to decibel microamperes or amperes. The default units is dbmicroA.

Note: Any change in the units reflects in the Value, Limit and Margin fields of the table and the bar graph.

You can export the .csv file only if you select the table format.

7. Browse the destination directory, and click the export button to export the result in the .csv format to `c:\TekApplications\DPOPWR\currentharmonics\log`. Click OK.

Generating Reports-Total Power Quality

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar, to display the Report Generator screen.
2. The application provides default factory templates for this measurement. We recommend you to use the file layout only. The template file name is Total_Power_Quality_Page1.rgt. The layout file name is Total_Power_Quality.rpl. If you have selected Amd 14> Filter> Total Power Quality option is selected, the layout file name is Total_Power_Quality_Amd14.rpl. However, if you want to select or modify another layout, follow the procedure given here

Note: The current harmonics report displays the table with the data and not the bar graph. You have to generate separate reports for decibel microamperes, amperes, even values, and odd values.

3. Select the New Template button in Define Test Template tab to define and create a template for Total Power Quality. You can configure all the parameters in the user interface to create a template. The application displays the values corresponding to the selected units. If you change the unit from decibel microamperes to amperes, the generated report displays the updated unit.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Line Ripple

Selecting and Configuring Measurements-Line Ripple

To select and configure Ripple Measurement, follow these steps:

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar.
2. Click Input/Output Analysis button to display Input/Output Analysis screen.
3. Click Line Ripple in the Output Analysis pane. Now, click the Configure button.

Configuring the Measurement

Follow the steps to configure the selected measurement:

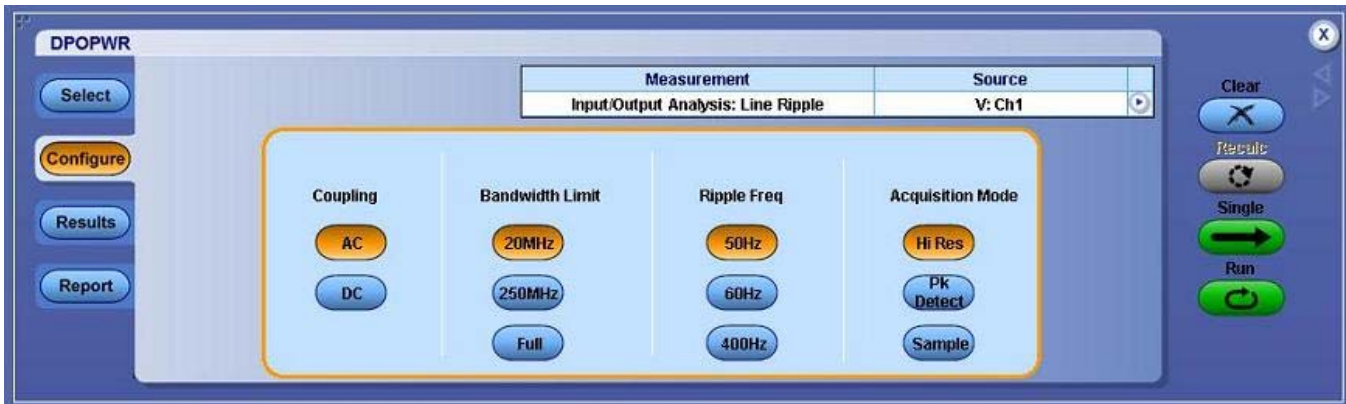


Figure 82: Line Ripple configuration

1. In the Coupling pane select AC Coupling or DC Coupling.
2. In the Bandwidth Limit pane, select the Bandwidth option: 20 MHz, 250 MHz or Full.
3. In the Ripple Frequency pane, set the line frequency to 50 Hz, or 60 Hz or 400 Hz.
4. In the Acquisition Mode pane, select the acquisition mode of the oscilloscope to Hi Res, Pk Detect, or Sample mode.
5. If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Viewing Results-Line Ripple

To view the results, follow these steps:

1. Select Analyze> Power Analysis> Results from the oscilloscope menu bar.
2. The application displays the results for the Line Ripple measurement with the following data as shown.

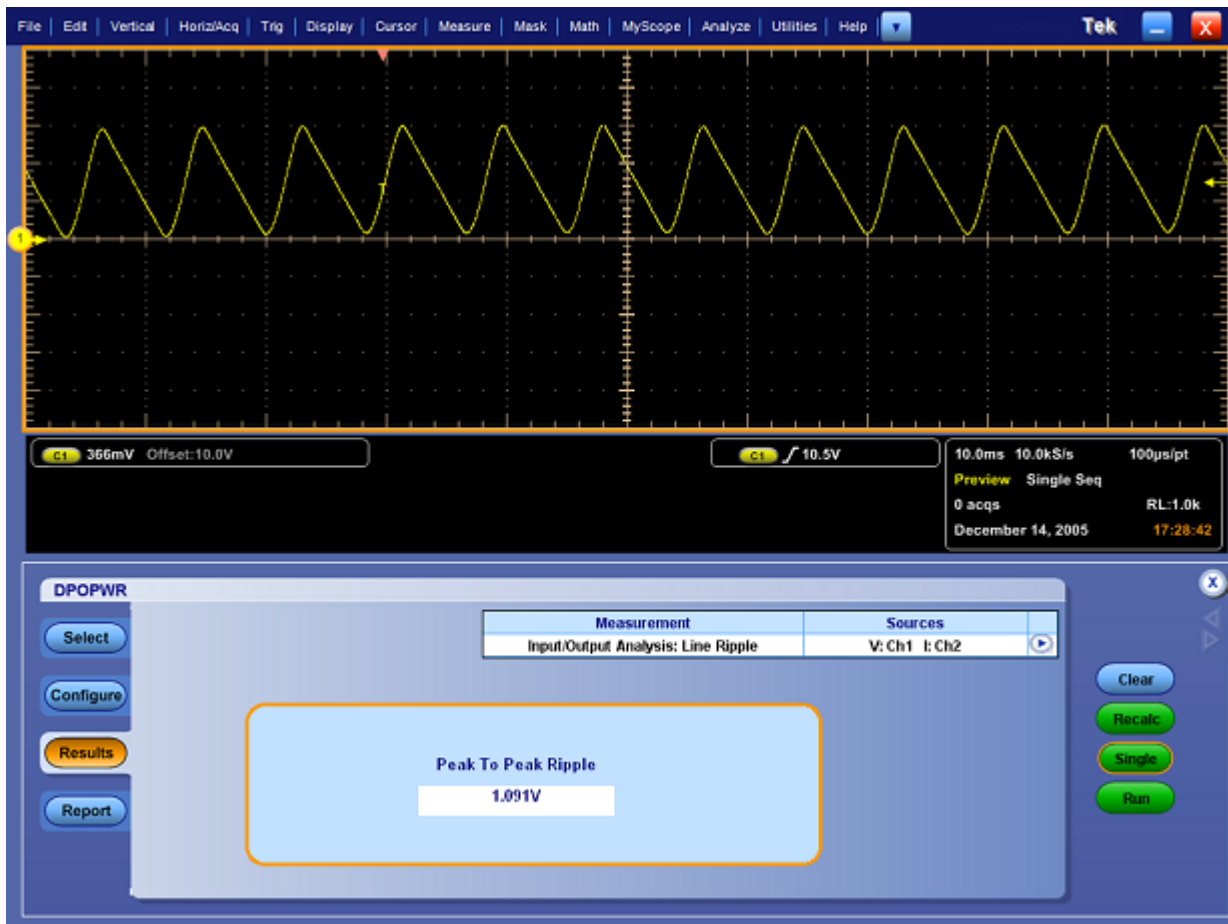


Figure 83: Line Ripple results

Generating Reports-Line Ripple

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar to display the Report Generator screen.
2. The application provides default factory templates for this measurement. The template file name is Ripple_Line.rgt and the layout file name is Output_Analysis.rpl. When you click Generate Report, the application displays the default template file name. However, if you want to select or modify another template, follow the procedure given here.
3. Select the New Template button in Define Test Template tab to define and create a template for Line Ripple. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Switching Ripple

Selecting and Configuring Measurements-Switching Ripple

To select and configure Ripple Measurement, follow these steps:

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar.
2. Click Input/Output Analysis button to display Input/Output Analysis screen.
3. Click Switching Ripple in the Output Analysis pane. Now, click the Configure button.

Configuring the Measurement

Follow the steps to configure the selected measurement:

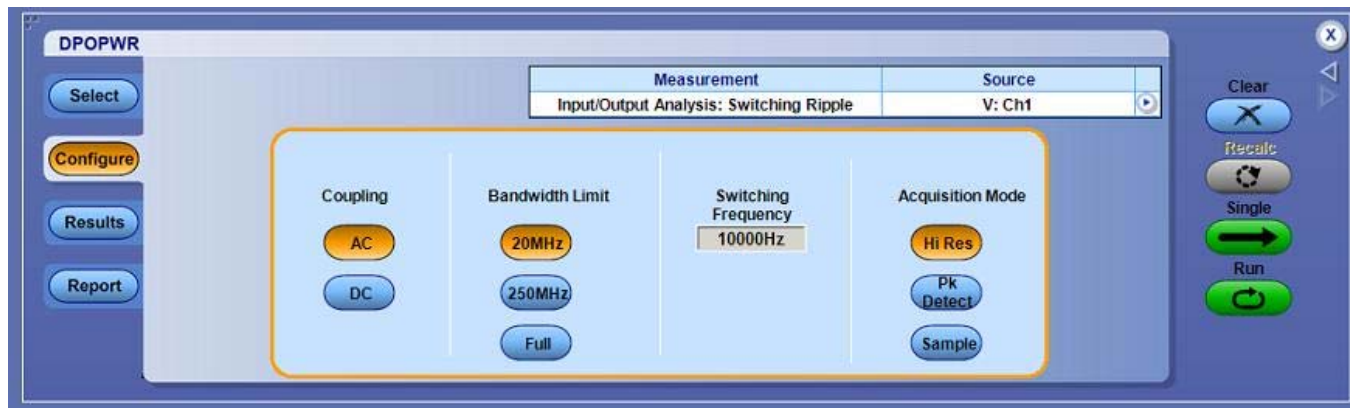


Figure 84: Switching Ripple configuration

1. Configure the options in the Source Configuration Panel.
2. Select the Coupling option from the Coupling pane: AC or DC.
3. Select the Bandwidth option from the Bandwidth Limit pane: 20 MHz, 250 MHz or the Full option. The Full option refers to the maximum bandwidth available in an oscilloscope.
4. In the Switching Frequency pane, double-click the switching frequency filed and use the keypad that appears to select a switching frequency. The default value is 10 KHz.
5. From the Acquisition Mode pane, set the acquisition mode of the oscilloscope to High Resolution, Pk Detect, or Sample.

6. Select Run to acquire the data.
7. If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Viewing Results-Switching Ripple

To view the results, follow these steps:

1. Select Analyze> Power Analysis> Results from the oscilloscope menu bar.
2. The application displays the results for the Ripple measurement with the following data.

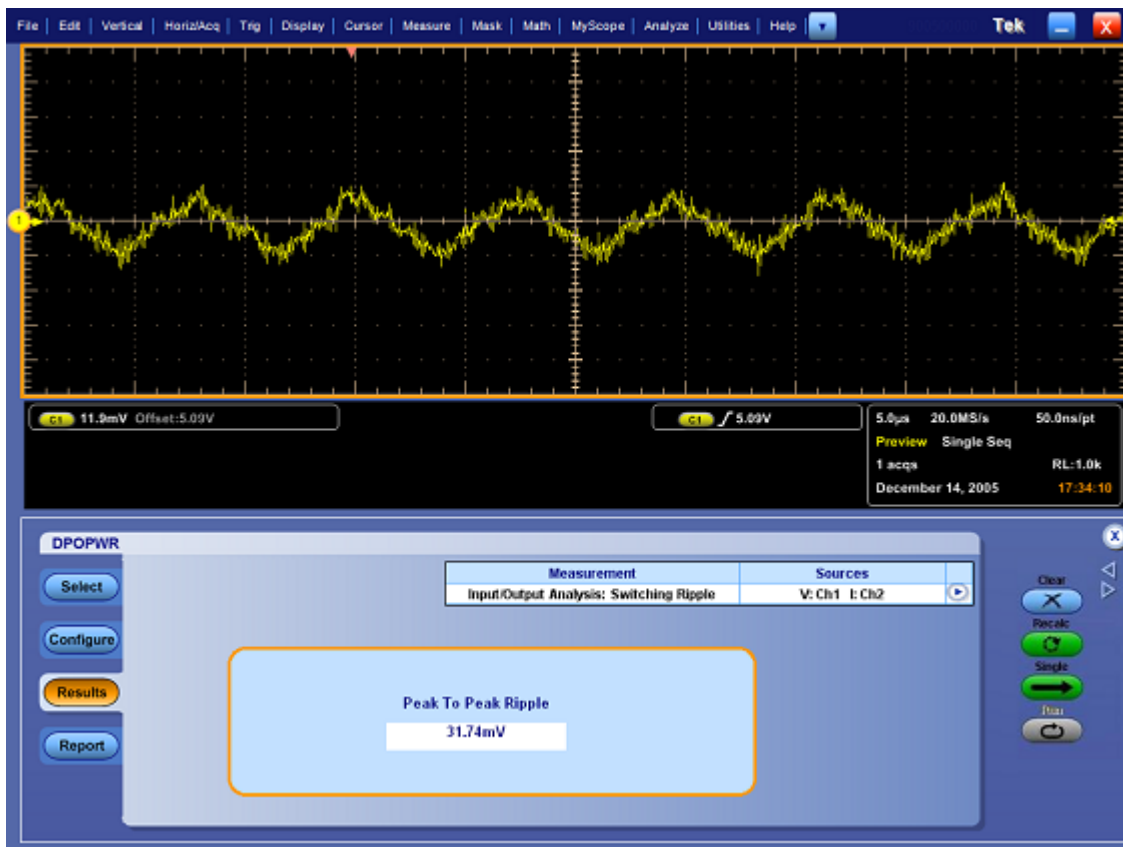


Figure 85: Switching Ripple results

Generating Reports-Switching Ripple

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar to display the Report Generator screen.

2. The application provides default factory templates for this measurement. The template file name is Ripple_Switching.rgt and the layout file name is Output_Analysis.rpl. When you click Generate Report, the application displays the default template file name. However, if you want to select or modify another template, follow the procedure given here.
3. Select the New Template button in Define Test Template tab to define and create a template for Switching Ripple. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Turn-On Time

Selecting and Configuring Measurements-Turn-On Time

To select and configure Turn-On Time Measurement, follow these steps:

1. Select Analyze> Power Analysis> Select from the oscilloscope menu bar.
2. Click Select. Click Input/Output Analysis button to display Input/Output Analysis screen.
3. Click Turn On Time in the Output Analysis pane. Now, click the Configure button.



Warning: *When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.*

Configuring the Measurement

Follow the steps to configure the selected measurement:

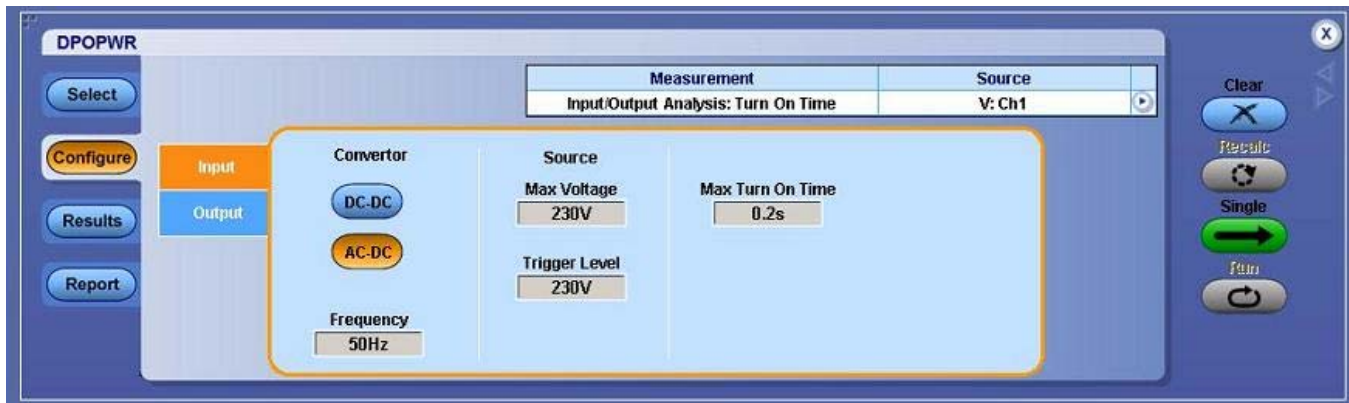


Figure 86: Turn-On Time configuration

1. In the Converter pane, select the type of converter used: DC-DC or AC-DC. Select the AC-DC option to enable the Frequency option. Double-click the Frequency field and use the keypad that appears to set the Line input frequency.
 - If you have selected AC-DC converter in the Converter pane: In the Source Configuration panel set the Voltage Source by clicking on the Sources field and selecting Voltage Channels 1 to 4. To enter Max Voltage and Trigger Level (1 V to 500 V), double-click the respective field and use the keypad that appears.
 - If you have selected DC-DC converter in the Converter pane: In the Source Configuration panel set the Line Voltage Source by clicking on the Sources field and selecting Voltage Channels 1 to 4. To enter Max Voltage and Trigger Level (1 V to 500 V) double-click the respective field and use the keypad that appears
2. Double-click the Max Turn-On Time field and use the keypad that appears to enter the expected turn on time. This enables you to set the timing window of the oscilloscope.
3. In the Output Voltage pane, click either the OFF button or ON button. Click the Source1 combo box to select the channel source. The source can be from Ch1-Ch4. The Output depends on the number of channels available in the oscilloscope.
4. Double-click the Max Voltage field and use the keypad that appears to enter the maximum voltage value.
5. Repeat steps 2-3 for Source 2 and Source 3 selections.
6. Switch On the device under test.
7. Press Run to measure.

8. If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze>Power Analysis>Results from the oscilloscope menu bar.

Viewing Results-Turn-On Time

To view the results, follow these steps:

Note: To view the results when the application is running, turn off the power supply and turn on to display the results.

1. Select Analyze> Power Analysis> Results from the oscilloscope menu bar.
2. The application displays the results for the Turn-On Time measurement with the following data.

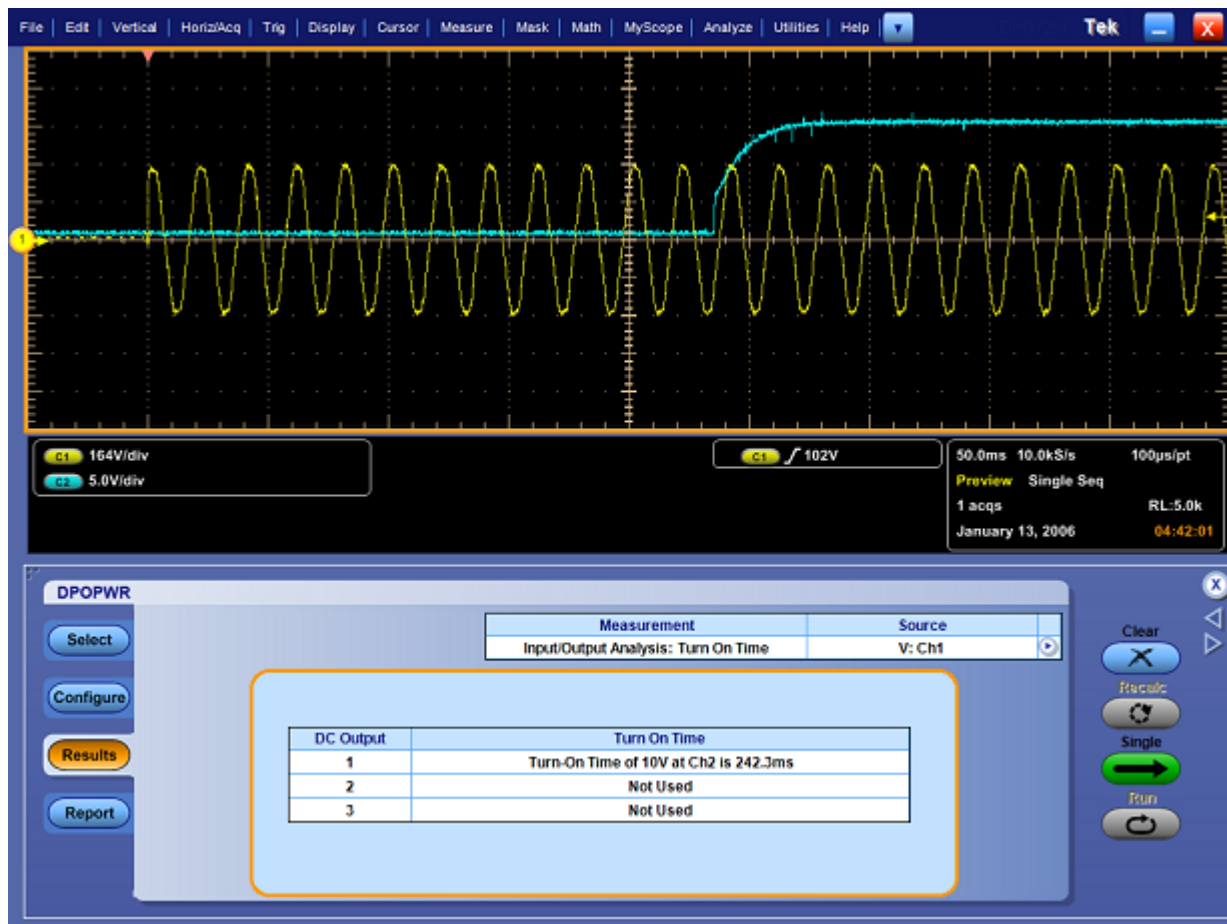


Figure 87: Turn-on Time results

Generating Reports-Turn-On Time

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar to display the Report Generator screen.
2. The application provides default factory templates for this measurement. The template file name is Turnon_time.rgt.and the layout file name is Output_Analysis.rpl. When you click Generate Report, the application displays the default template file name. However, if you want to select or modify another template, follow the procedure given here.
3. Select the New Template button in Define Test Template tab to define and create a template for Turn-On Time. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Spectral Analysis

Selecting and Configuring Measurements-Spectral Analysis

Purpose: To analyze the frequency component that contributes to the electromagnetic interference and measures the noise/ripple at the output DC voltage frequency range. The result plot component plots the spectrum amplitude on Y-axis and Frequency (units: Hz) on X-axis. Follow these steps to use the Spectral Analysis tool:

1. Click Input/Output Analysis button to display Input/Output Analysis screen.
2. Click Spectral Analysis in the Output Analysis pane.
3. Now, click the Configure button to get the next screen.

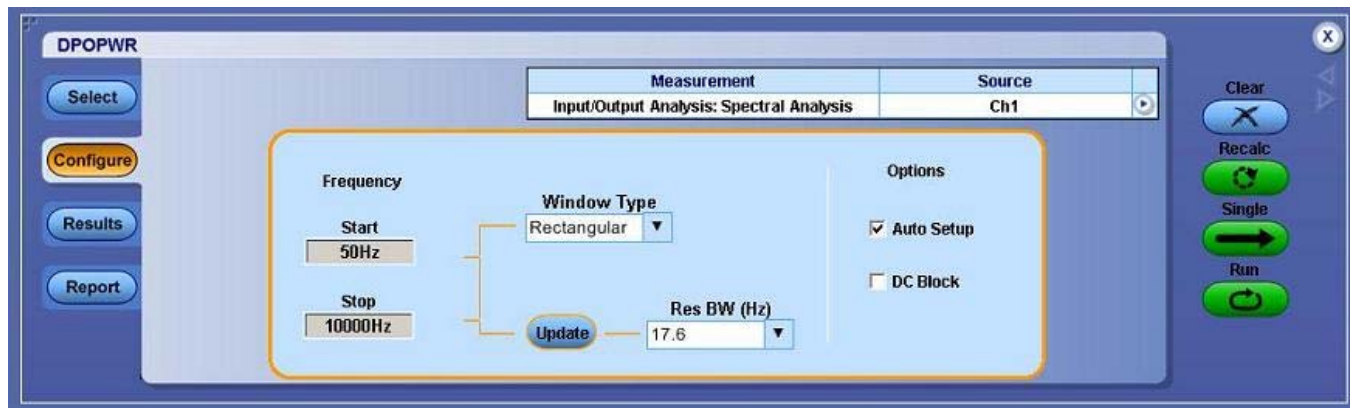


Figure 88: Spectral analysis

4. Click the drop down arrow in the Source field to select the input source. The Source field specifies the input source to which the DUT is connected. The available options are: Ch1-Ch4, Math1-Math4 and Ref1-Ref4. These selections depend on the number of channels in the oscilloscope.
5. Double-click the Start and Stop field and use the keypad that appears to set the frequency values. The Frequency Start and Stop options define the frequency range values of the input signal to be analyzed. In the Start field, you can set the value between the range of 0 Hz to 499 MHz. The default value is 50 Hz. In the Stop field, you can set the value between the range of 50 Hz to 500 MHz. The default value is 10 KHz.
6. Click the Window Type combo box to set the window type. The window type reduces the spectral leakage when using the spectral analysis. The available window types are: Rectangular, Hamming, Hanning, Black-Harris, Gaussian, Flattop2, Kaiser-Bessel and TekExp.

In the Options pane, the Auto Setup is selected by default. The application automatically sets the record length and the time base for the given Start, Stop and Window Type inputs. In the Frequency Values pane, click the drop down arrow to set the RBW (Resolution and Bandwidth) value.

The application calculates the RBW values depending on the Start, Stop and Window Type inputs. Click the Update button to update the RBW values. If the Auto Setup checkbox is not selected, the Update button is disabled and the application runs with the existing record length and time base available in the oscilloscope.

The Options pane is enabled:

- If you have selected a Ref source in the Source field
- If you have selected the Single option in the Mode field

If you have selected a live source in the Source field, enable the DC Block check box. Clicking the DC Block adjusts the vertical sensitivity

of the signal and measures the frequency component of the noise at the output DC voltage.

7. Select the Run button to acquire the data. If you have entered the Start and Stop frequency values, the application calculates the selected RBW value, sets it to the highest RBW and runs the measurement. If the measurement is successful, the application automatically displays the results. You can also view the results by selecting Analyze> Power Analysis> Results from the oscilloscope menu bar.

Refer to the Spectral Analysis Algorithm for more information on RBW calculation.

Viewing Results-Spectral Analysis

To view the results, follow these steps:

1. Click Analyze> Power Analysis> Results from the oscilloscope menu bar or click the View Plot button to view the spectrum plot.

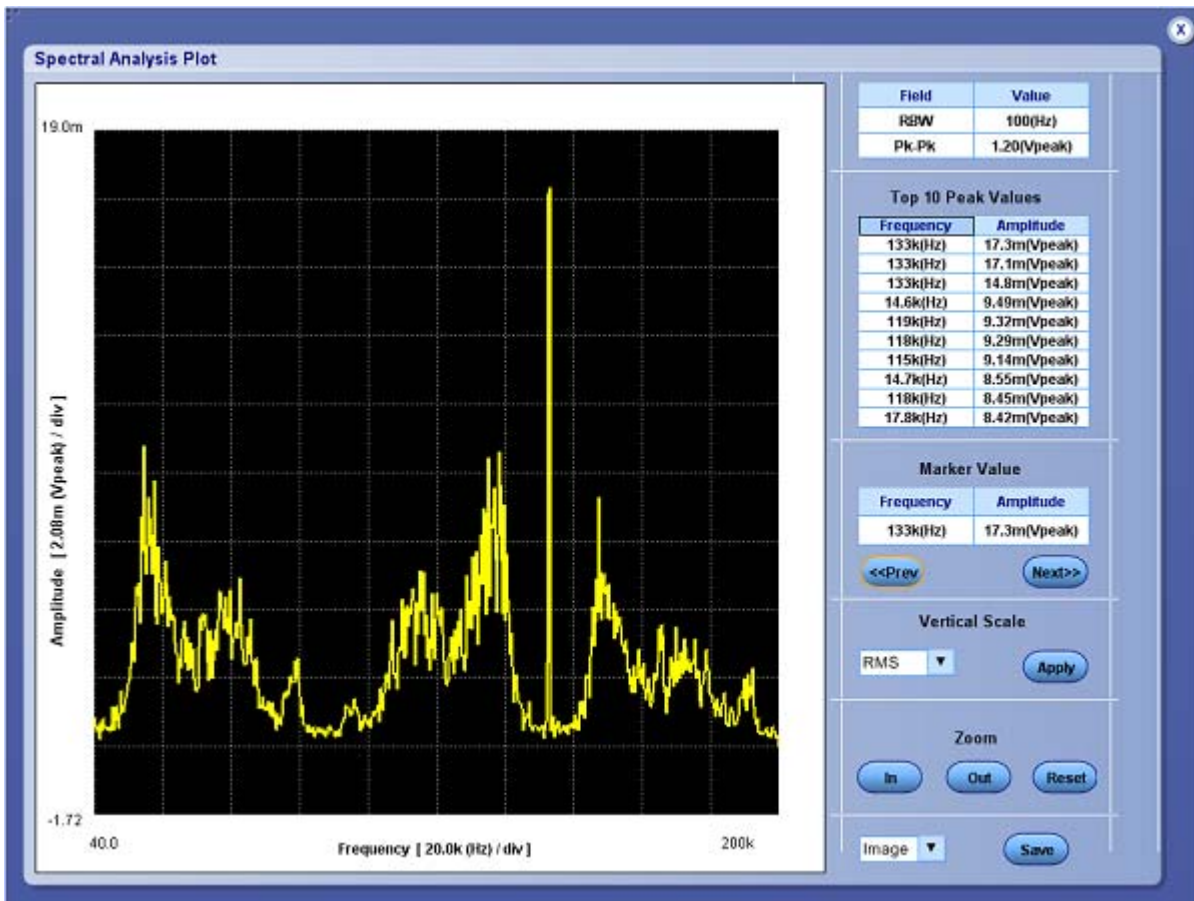


Figure 89: Spectral analysis plot

The application plots the Frequency values on the X axis and the Amplitude on the Y axis and displays the Start and Stop range values. The Y axis shows the maximum and minimum range of the voltage peaks in each division.

In the Vertical Scale pane, you can switch the plot between the different vertical scale values like dB, dBm, RMS and Peak.

2. Click the Zoom In button and place the cursor on the specific region to zoom the plot. You can also use the Zoom Out button to zoom out.
3. Click the Save button to save the plot in .jpg format in the default directory C:\TekApplications\DPOPWR\Images.
4. Click the Next button to place the marker on the next highest peak. You can also use the Prev button to position the marker on the previous peak. The Frequency and Amplitude field displays the values at the specific peak in KHz and mV.

The Marker positions itself on the highest peak value. The Marker Value field displays the marker details. The Top Peak Values field displays the first ten peaks and the frequencies at which they occur.

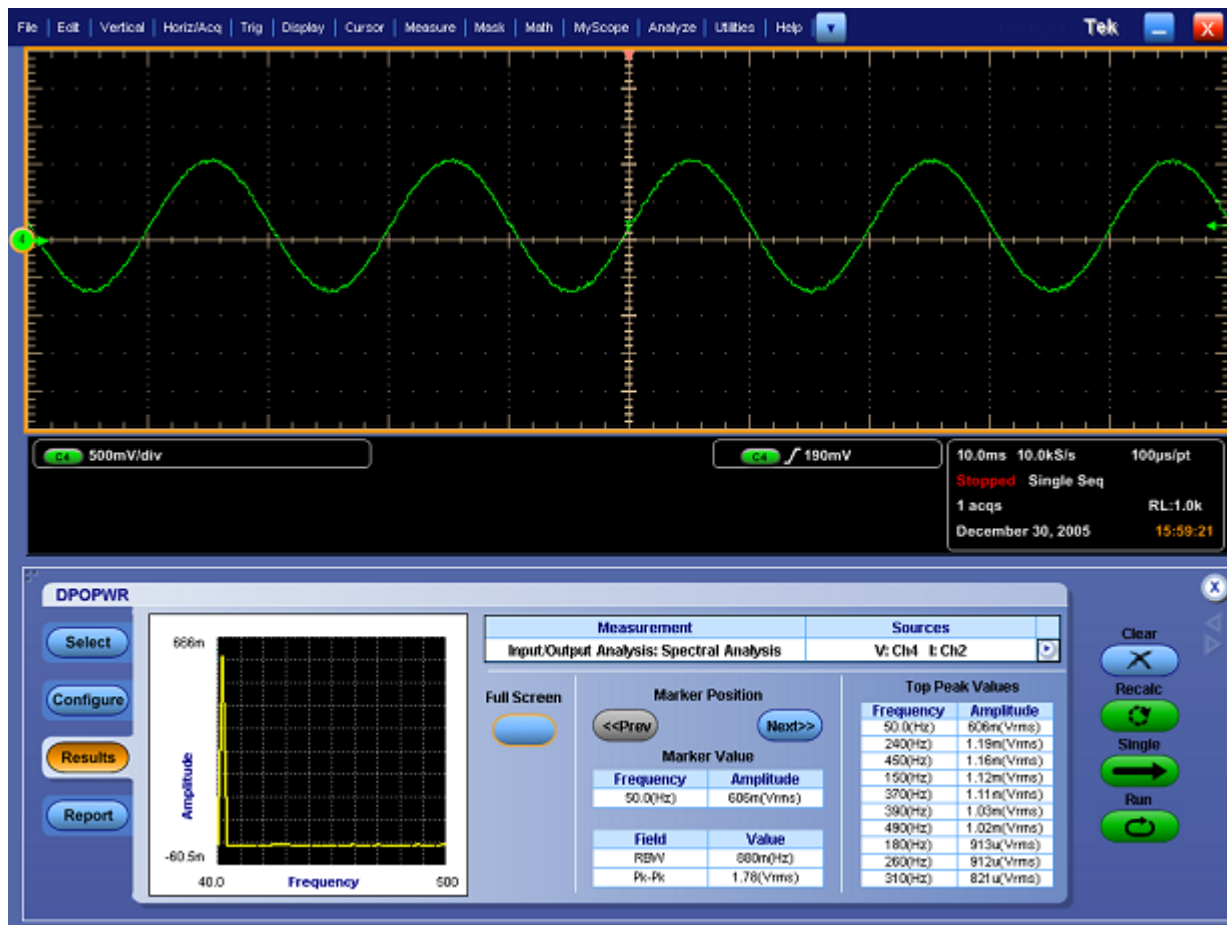


Figure 90: Spectral analysis results

Generating Reports-Spectral Analysis

Follow these steps to generate a report:

1. Select Analyze> Power Analysis> Report from the oscilloscope menu bar to display the Report Generator screen.
2. The application provides default factory templates for this measurement. The template file name is Spectral_Analysis.rgt and the layout file name is Analysis_Tools.rpl. When you click Generate Report, the application displays the default template file name. However, if you want to select or modify another template, follow the procedure given here.
3. Select the New Template button in Define Test Template tab to define and create a template for Frequency. You can configure all the parameters in the user interface to create a template.
4. Select the Define Report Layout tab to define a layout for this template.
5. Select the Generate Report tab to generate, view and print the report.

[For more information on Report Generator, click here to view the Report Generator Printed Help Document.](#)

Troubleshooting Spectral Analysis Error Messages

If the minimum record length required to configure the Start and Stop values is greater than the maximum record length supported by the application, the application displays the error message "Frequency Range higher than range supported."

If the sampling rate value is lesser than twice the Stop frequency rate in the oscilloscope, the application displays the error message "Sampling rate lesser than twice stop frequency."

Saving and Recalling Setups

Saving a Setup

To save the application state and the oscilloscope settings to a setup file, follow these steps:

1. Select File> Save As> Setup.
2. Select the Save menu.
3. Select the directory to save the setup file.

4. Select or enter a file name. The application appends an ".xml" extension to the name of setup files.
5. Choose Save.

***Note:** While saving a Report, the application does not validate for the available size in the drive.*

Recalling a Saved Setup

To recall the application setup and the oscilloscope settings from a saved setup file, follow these steps:

1. Select File> Recall.
2. Select the directory from which you can recall the application settings.
3. Select or enter a file name.

***Note:** The application also recalls the oscilloscope setup from a ".xml" file when you recall an application setup.*

Recalling the Default Setup

To recall the application settings from the Default setup file Select File> Recall Default Setup.

Tutorial

Introduction to the Tutorial

This tutorial teaches you how to set up the application, take two types of measurements, and view the results.

Before you begin the tutorial, you must do the following tasks:

- Set up the oscilloscope
- Start the application

Starting the Application

Go to the File menu on the menu bar and select Run Application>Power Measurements menu to start the application.

Setting Up the Oscilloscope

To set up the oscilloscope, go to the File menu in the oscilloscope menu bar and select Recall Default Setup to set the oscilloscope to the default factory settings.

Measuring Switching Loss

This section discusses how to measure switching loss, view the results and generate report. To perform these tasks, install and enable the option key for the application in DPO7000 series oscilloscopes.

Follow these steps to measure switching loss:

1. From the oscilloscope menu, select File> Reference Wfm> Recall Waveform.
2. In the Ref 1 tab, select Recall Ref 1 button to recall the switchvoltage.wfm waveform file from C:\TekApplications\DPOPWR\waveforms folder to Ref 1.
3. In the Ref 2 tab, select Recall Ref 2 button to recall the switchcurrent.wfm waveform file from C:\TekApplications\DPOPWR\waveforms folder to Ref 2.
4. In DPOPWR application, click the Power Device tab.

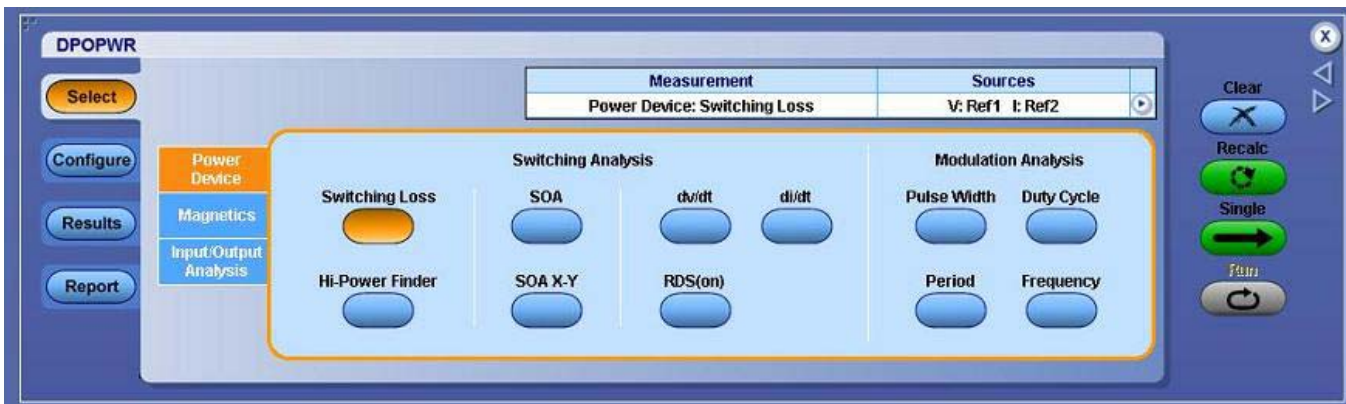


Figure 91: Switching Loss configuration

5. Click Switching Loss and then click Configure.
6. In the Source Configuration panel, set the Source to Ref, Voltage to Ref 1 and Current to Ref 2.

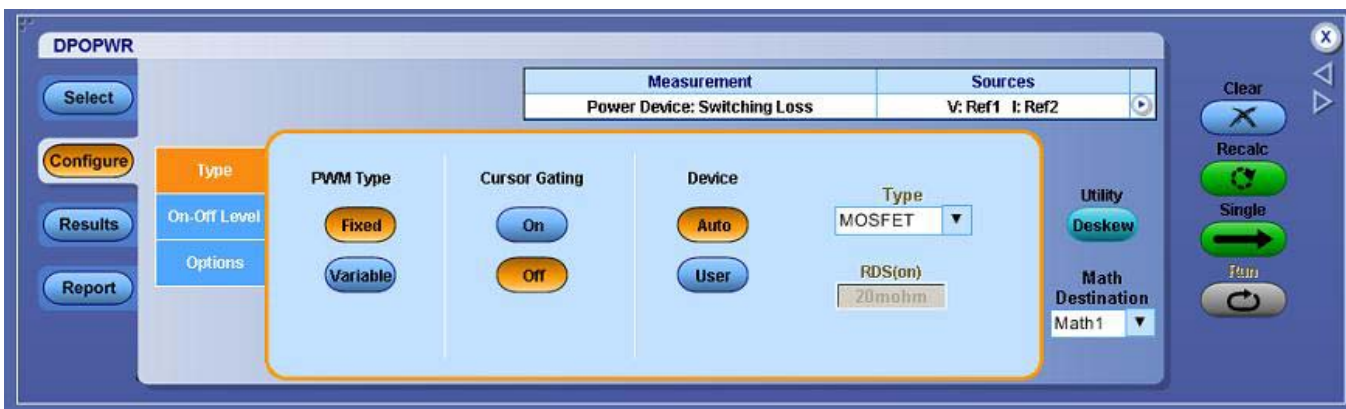


Figure 92: Switching Loss configurations

7. Select the Switching Loss measurement. Configure the options in the Type, On-Off Level, and the Options tabs. Press the Run button.
8. The application displays the results as Turn-on Power Loss, Turn-off Power Loss, T-Average Loss and Energy.

Figure 4-3 shows the results in a DPO7000 series of oscilloscopes.

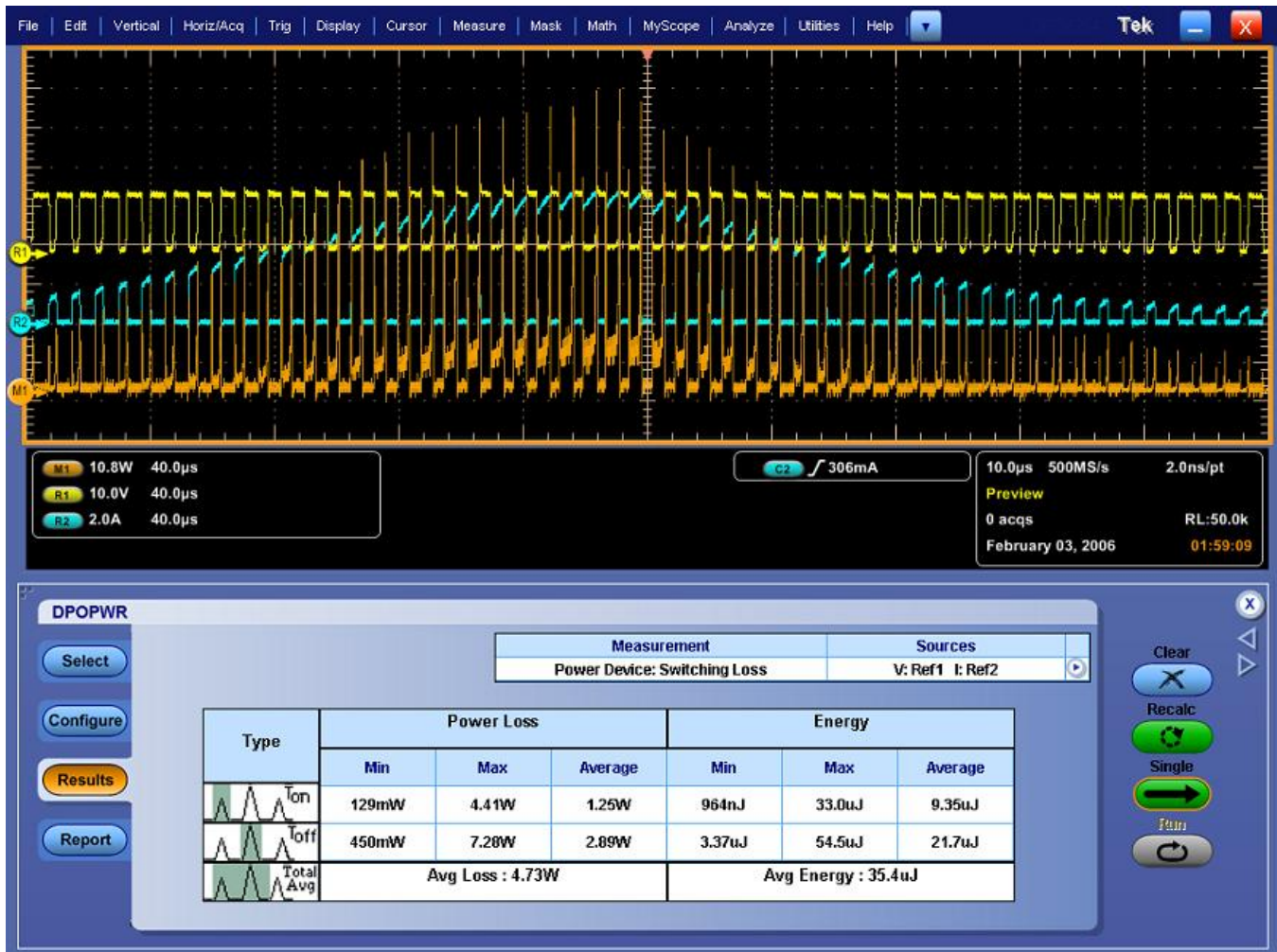


Figure 93: Switching Loss results

9. Select Analyze> Power Analysis> Report to generate, view and print reports.

Measuring Magnetics

This section discusses how to measure Magnetics, view the results and generate report. To perform these tasks, install and enable the application in DPO7000 series oscilloscopes.

Follow these steps to measure Magnetics:

1. From the oscilloscope menu, select File> Reference Wfm> Recall Waveform.

2. In the Ref 1 tab, select Recall Ref 1 button to recall the ref1vol.wfm waveform file from C:\TekApplications\DPOPWR\waveforms folder to Ref 1.
3. In the Ref 2 tab, select Recall Ref 2 button to recall the ref2cur.wfm waveform file from C:\TekApplications\DPOPWR\waveforms folder to Ref 2.
4. In DPOPWR application, select Magnetics tab.

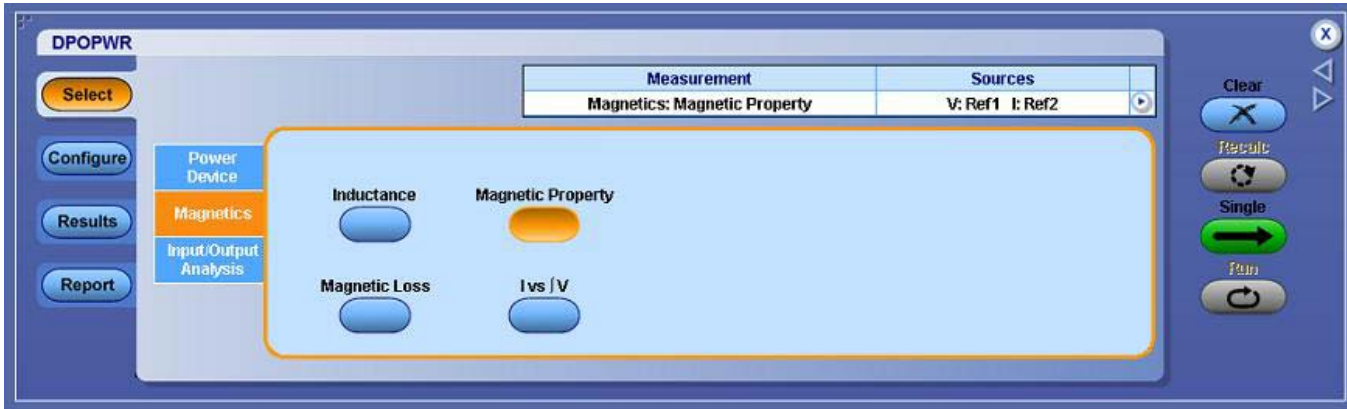


Figure 94: Magnetics configurations

5. In the Source Configuration Panel, set the Source to Ref, Voltage to Ref 1 and Current to Ref 2.

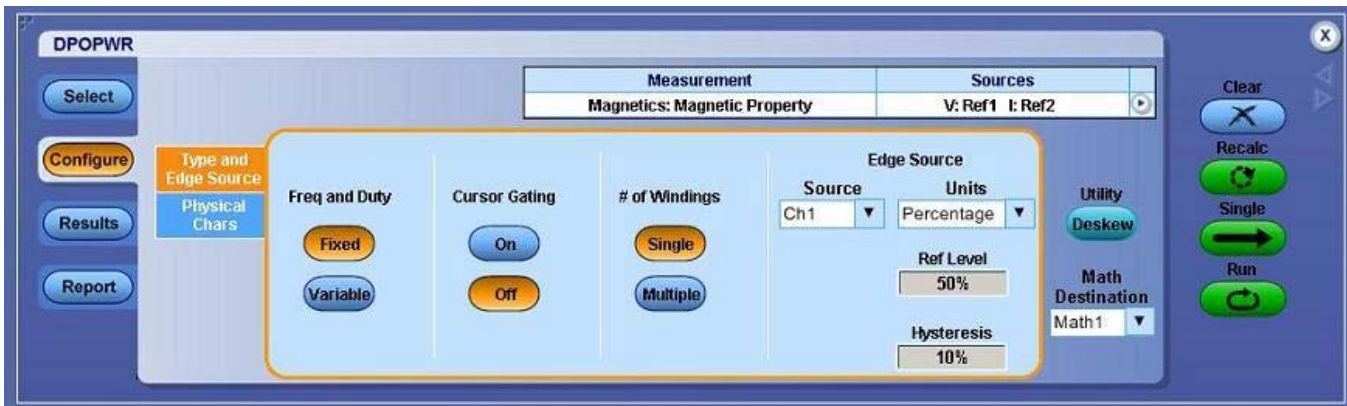


Figure 95: Magnetic Property configurations

6. Click Magnetic Property button. Select the Freq and Duty cycle option, Cursor Gating Off button and Single Winding button. In the Phy Chars tab, select the SI Unit, set the Number of Turns to 50, XsectionArea to .00001358m², Magnetic Length to 0.0265m.

- Click Single to display the results in a B-H Curve format. The application displays the results for the Bpeak, Br, Hc, HMax and I-ripple.

Figure 96 shows the results in a DPO7000 series of oscilloscopes.

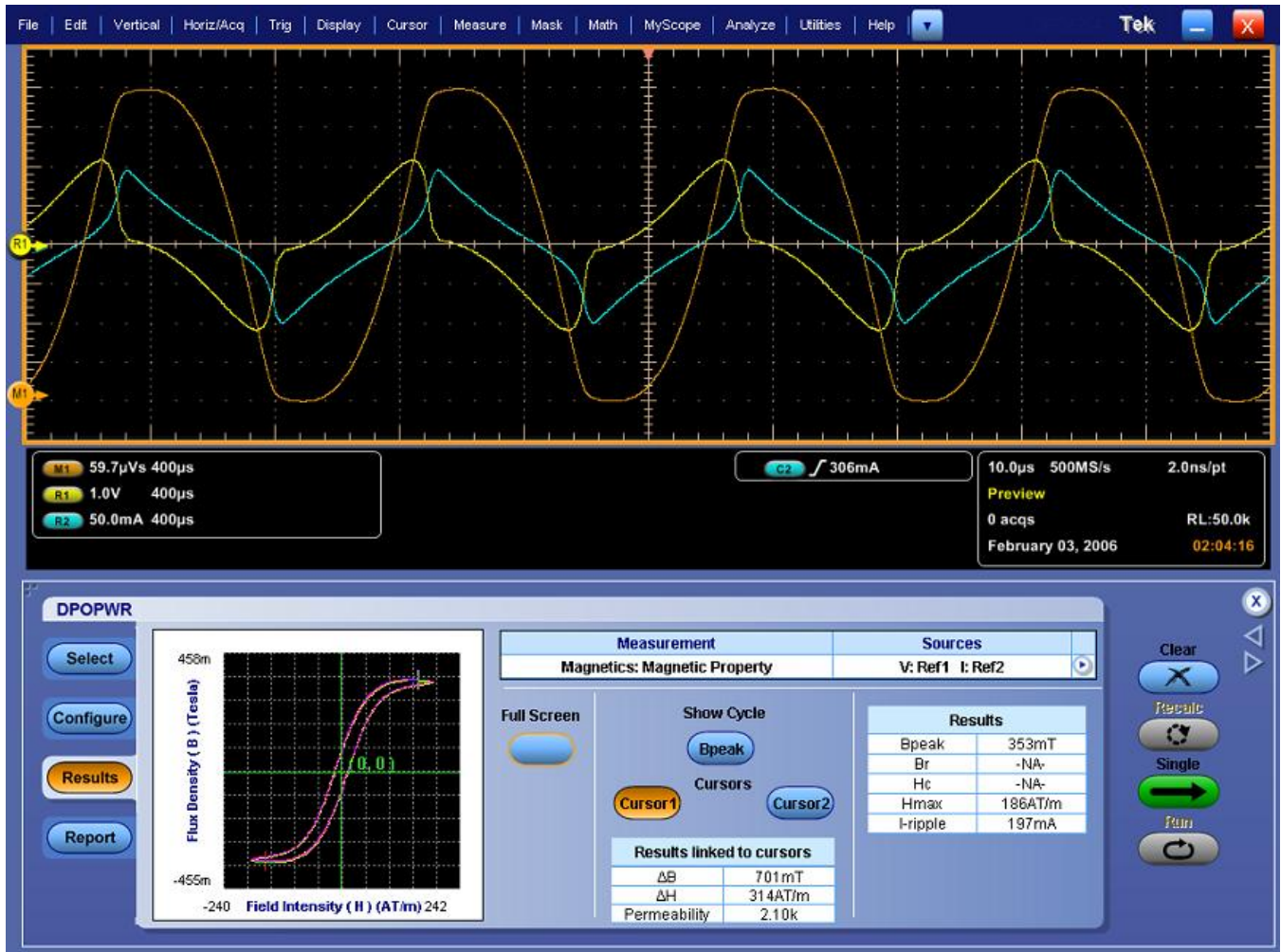


Figure 96: Magnetic Property results

- Select Analyze> Power Analysis> Report to generate, view and print reports.

Stopping the Tutorial

If you need more than one session to complete the tutorial lessons, you can stop the tutorial and return later.

To save the application setup and stop your session, refer to [Saving a Setup and Exiting the Application](#).

Returning to the Tutorial

To return to the tutorial, you can start the application and then recall the saved setup. To recall the application setup, refer to [Recalling a Saved Setup](#).

Application Examples

This section presents many application examples. The simplified AC/DC circuit diagram example highlights the application measurements and gives you ideas on how to use the application to solve your own test problems.

To run the application examples, install and enable the DPOPWR application on the oscilloscope, connect the probes to the device under test and configure the device. To install the application, refer to the section Installation procedures. For more information on compatible probes, refer to the section Compatibility and Accessories. To configure the application, refer to the section, Setting Up the Application.

To Increase Efficiency in a Switching Power Supply

Measure Switching Loss

Aim: To optimize the power loss at switching device and improve the efficiency of power supply

Specifying the equipment:

- Tektronix oscilloscope as mentioned in the Compatibility section
- DPOPWR application
- Appropriate current probe and differential probe

Equipment Setup:

Refer to the General Safety Summary before connecting to a circuit

1. To measure the voltage across the drain, connect the voltage probe and the current probe to the switching source of the MOSFET as shown:

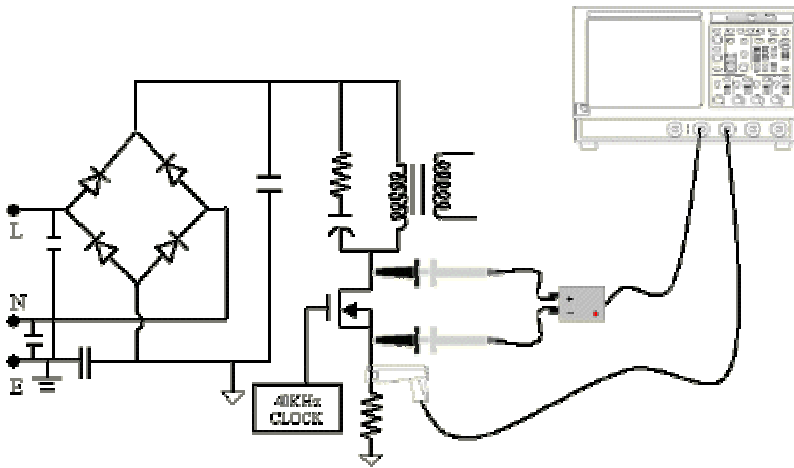


Figure 97: Equipment setup



Warning: When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.

2. Run the DPOPR application.
3. Select the Power Device tab.
4. Select Switching Loss option and press the Configure button. In the Source Configuration Panel, set the appropriate I-Probe Settings. Select the Deskew button to deskew probes and channels.
5. Select Switching Loss measurement.
6. Select Run to acquire data and display the following results.



Figure 98: Switching Loss Results

Table 18: Switching Loss results

Result	Description
Minimum	Measures the minimum power loss or cycle in the acquired data
Maximum	Measures the maximum power loss or cycle in the acquired data
Average	Measures the variation in the power loss for multiple acquisitions

Measure Hi-Power Finder

Aim: To analyze the Power Device Analysis in a switching device in a dynamic environment

Specifying the equipment:

- Tektronix oscilloscope as mentioned in the Compatibility section
- DPOPWR application
- Appropriate current probe and differential probe

Equipment Setup:

Refer to the General Safety Summary before connecting to a circuit

1. To measure the voltage across drain, connect the Tektronix differential and current probe to the switching source of the MOSFET as shown below:

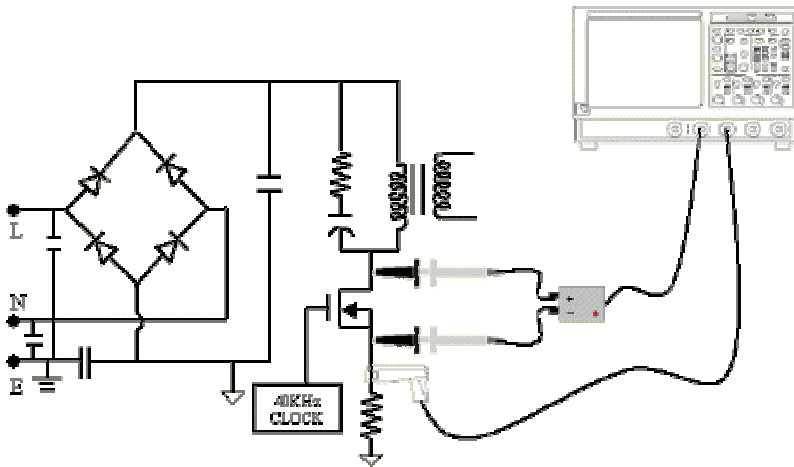


Figure 99: Equipment setup



Warning: When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.

2. Run the DPOPWR application.
3. Select the Power Device tab.
4. Select Hi-Power Finder option and press the Configure button.
5. In the Source Configuration Panel, set the appropriate i-Probe Settings. Select the Deskew button to deskew probes and channels.
6. Select the Hi-Power Finder measurement.

7. Select the Deskew button to deskew. Select Run by setting the appropriate trigger related to the dynamic environment and display the results.

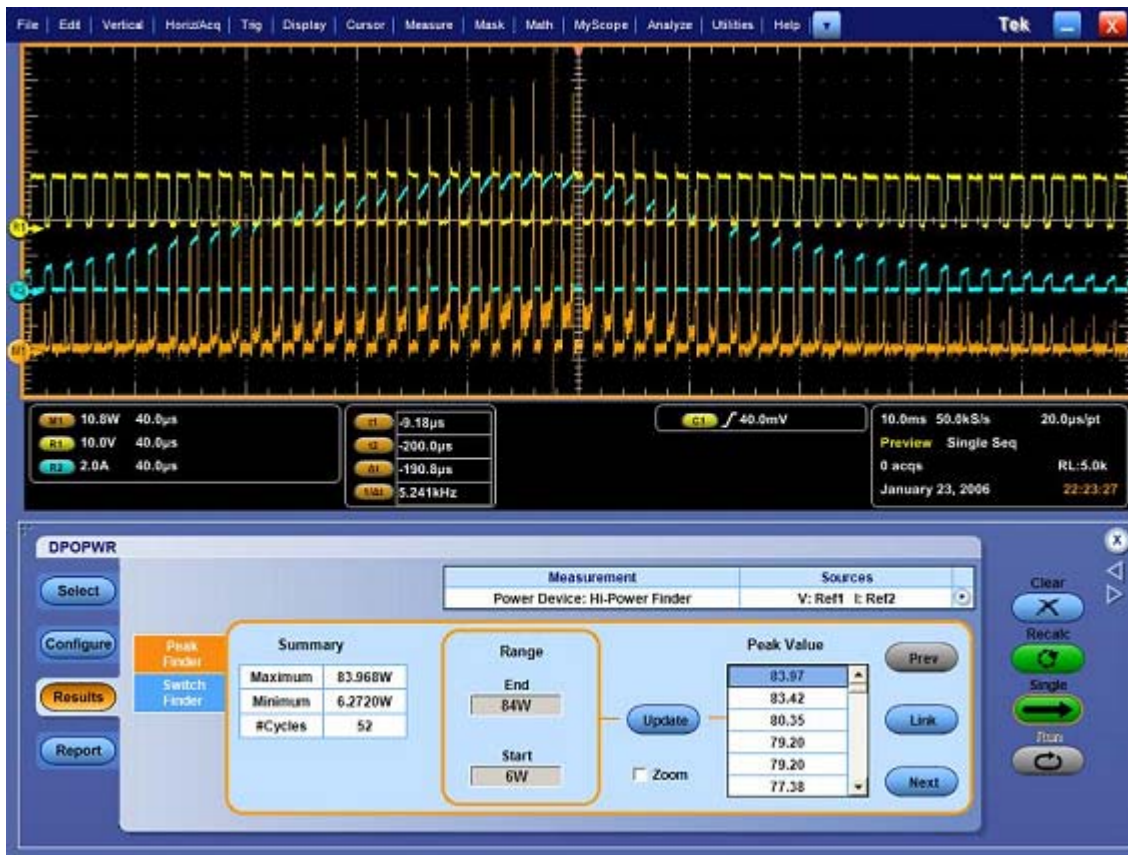


Figure 100: Hi-Power Finder Results

8. The results display the instantaneous peak power in the switching and the number of peaks in the acquired data. Select the range of interest to view the area of interest in the instantaneous peak power.
9. You can link the cursor by selecting the instantaneous peak power from the table.
10. Select the zoom button to view the detailed information around the cursor.

Measure Magnetic Loss

Aim: To measure Magnetic Loss

Specifying the equipment:

- Tektronix oscilloscope as mentioned in the Compatibility section
- DPOPWR application
- Appropriate current probe and differential probe

Equipment Setup:

Refer to the General Safety Summary before connecting to a circuit

1. Connect the Tektronix differential and current probe to the switching to measure the voltage across the inductor as shown below:

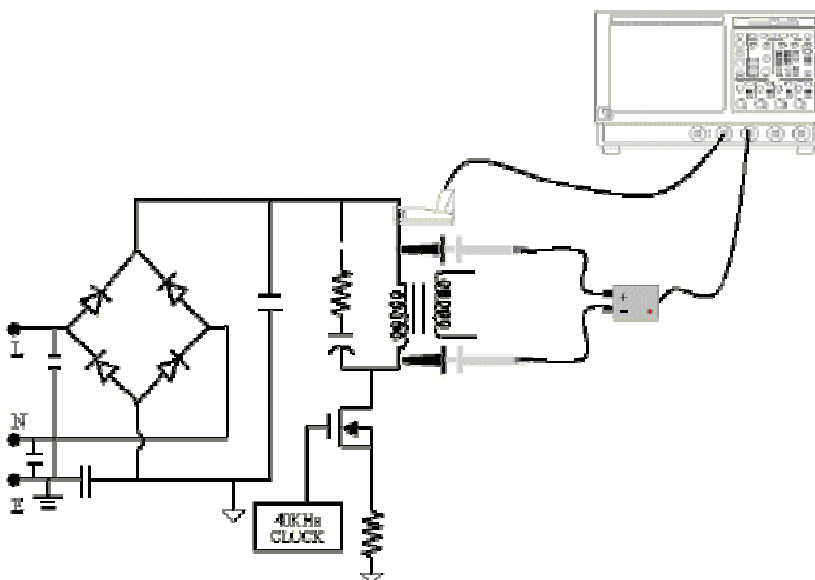


Figure 101: Equipment setup



Warning: When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components are used within their ratings. Refer to the General Safety Summary.

2. Run the DPOPWR application.
3. Select the Magnetics tab.
4. Select Magnetic Loss measurement and select the Cursor Gating option.
5. In the Source Configuration Panel, set the appropriate I-Probe Settings. Select the Deskew button to deskew the voltage and current signal.

6. Press Single to measure and display the result shown below:



Figure 102: Magnetic Loss results

7. The result displays the power loss of the magnetic material.

Result:

The results of the Switching Loss, Hi-Power Finder, and Magnetic Loss help locate the Power Device Analysis, reduce power loss and improve efficiency.

Reference

Error Codes for DPOPWR

Tables 19 through 23 show the error codes for DPOPWR.

Table 19: Error Codes 101 through 175

Error Codes	Error Message	Description	Possible Solutions
101	Failed to acquire the signal. Unable to trigger. Press F1 for more details.	The oscilloscope is not able to acquire the signal because the trigger is not available.	Select the proper trigger source, set the trigger level appropriately, and then run the measurement again.
102	Failed to import the waveform. Press F1 for more details.	The operation of importing the waveform from the acquisition memory failed because there is no valid waveform in the acquisition memory.	Reacquire the new waveform.
103	Failed to find the required edges on the waveform. Press F1 for more details.	The waveform does not have enough edges for the selected measurement to run successfully.	Decrease the horizontal scale of the oscilloscope or increase the record length value. Set proper ref level and hysteresis level to find the correct edges.
104	Set the record length value to less than XXXXXXX data points. Press F1 for more details.	The maximum record length value supported by the application depends on the oscilloscope that you use.	On the oscilloscope menu bar, click Horiz/Acq > Horizontal/Acquisition Setup. In the Horizontal tab, decrease the record length value. The maximum record length value supported by the application depends on the oscilloscope model that you use and the available memory option.
105	Set the record length value to less than 4M data points. Press F1 for more details.	The maximum record length value supported by the application depends on the oscilloscope that you use.	On the oscilloscope menu bar, click Horiz/Acq > Horizontal/Acquisition Setup. In the Horizontal tab, decrease the record length value.
106	Record length of one cycle is greater than 2M. Press F1 for more details.	The record length value of one cycle should be less than 2M data points.	Increase the horizontal scale of the oscilloscope or decrease the record length value to modify the data points in one cycle to be less than 2M.
107	Failed to export the waveform. Press F1 for more details.	Application failed to export the time trend data to the selected reference memory.	Run the measurement again.
108	No waveform data is available at the selected cursor position. Press F1 for more details.	Cursor is positioned beyond the waveform horizontal range on the oscilloscope display.	Position the cursor inside the displayed waveform range. Run the measurement again.

Table 19: Error Codes 101 through 175 (Contd.)

Error Codes	Error Message	Description	Possible Solutions
110	The oscilloscope was unable to wait during the acquisition 'RUN' state. Press F1 for more details.	The application will set the oscilloscope to single sequence mode and trigger will be in the ready state. This error occurs under three conditions: 1) The trigger condition is not met 2) Acquiring a very long record length value 3) The acquisition is not completed within a stipulated time due to the average mode Selection	The oscilloscope will wait for the acquisition to complete for one minute. 1) Acquire the waveform on the oscilloscope and use the single no-acq as the mode to run the measurement. 3) Select the proper trigger source and adjust the trigger level.
111	No waveform is available. Press F1 for more details.	The data available on the selected source is not valid for the computation.	On the oscilloscope menu bar, click File > Reference Waveforms > Recall Wfm. In the 'recall ref1 from file' pane, click Recall to browse for the required Ref waveform and recall it. Next, in the required Ref tab, click Display to set it 'On.' If it is used for both ref and live, then reacquire the waveform and run the measurement again for live.
112	Ensure that data is present in the Ref waveform. Press F1 for more details.	The Ref waveform is not present in the oscilloscope.	On the oscilloscope menu bar, click File > Reference Waveforms > Recall Wfm. In the 'recall ref1 from file' pane, click Recall to browse for the required Ref waveform and recall it. Next, in the required Ref tab, click Display to set it 'On.'
113	The data source associated with the cursor is not correct. Press F1 for more details.	This error has occurred as the data is not valid for the cursor association.	Run the measurement again.
114	Math waveform cannot be turned on. It may have an empty, an invalid or a circular definition string. Press F1 for more details.	Math definition is an empty string or the definition is invalid.	Define math properly before running the measurement using math as voltage source.
115	Math should be defined only as a function of live channels. Press F1 for more details.	Math selected for voltage source should not be defined using a ref channel because the Math selection is available only for Live source type.	Define the math using live channels only
174	Unable to process Ref wfm. Record Length is more than <####>. Press F1 for more details.	Record Length of the reference waveform exceeds the maximum record length supported by the application.	On the oscilloscope menu bar, click Horiz/Acq > Horizontal/Acquisition Setup. In the Horizontal tab, decrease the record length value. Reacquire the Live waveforms. Save waveform to ref.

Table 19: Error Codes 101 through 175 (Contd.)

Error Codes	Error Message	Description	Possible Solutions
175	Unable to process the Ref waveform. Record length value is more than 4M data points. Press F1 for more details.	The record length value of the selected source waveform on the oscilloscope is more than the maximum record length value supported by the application to measure Spectral Analysis.	On the oscilloscope menu bar, click Horiz/Acq > Horizontal/Acquisition Setup. In the Horizontal tab, decrease the record length value. Reacquire the Live waveforms. Save waveform to ref.

Table 20: Error Codes 201 through 351

Error Codes	Error Message	Description	Possible Solutions
201	Unable to perform computation on the waveform data. Source waveforms have different time/div settings. Press F1 for more details.	The voltage and current waveforms have different time/div settings.	Try to run the selected measurement with the voltage and current waveforms acquired simultaneously.
202	Unable to perform calculation. Source waveforms have different record lengths. Press F1 for more details.	The voltage and current waveforms have different record lengths.	Try to run the selected measurement with the voltage and current waveforms acquired simultaneously having the same record length.
203	Higher sample rate is required for accurate measurement. Press F1 for more details.	The time/div settings for the selected waveform is more. Alternatively, sampling rate is low.	On the oscilloscope menu bar, click Horiz/Acq > Horizontal/Acquisition Setup. In the Horizontal tab, decrease the time base or increase the record length value.
204	Ensure that at least one complete cycle is available for analysis. Press F1 for more details.	The signal has less than one cycle in the specified record length.	Decrease the time/div settings of the oscilloscope to have more cycles.
205	Ensure that the cycle has more than 20 sample points. Press F1 for more details.	The number of sample points between the two edges is less than 20. The edge can be a rising or a falling edge.	Increase the record length value and/or decrease the time/div settings of the oscilloscope.
206	Unable to get the frequency value, when queried from the oscilloscope. Press F1 for more details.	Unable to measure Frequency due to the non availability of the complete cycle.	Run the measurement again.
251	Insufficient edges for the Modulation Analysis measurement. Press F1 for more details.	Insufficient edges found on the source waveform. At least two edges are required to run the measurement.	1) Decrease the horizontal scale to get complete cycles of the waveform. 2) Select the source that has signal cyclic in nature. 3) Adjust the Ref and Hysteresis Levels of the signal to find the edge at the required level.
252	The Ref Level value is outside the voltage range of the waveform. Enter a proper Ref Level value. Press F1 for more details.	Unable to locate the edges on the waveform at the transition because the voltage level does not fall in the transition of the waveform.	Enter the Ref Level value where the transition occurs on the waveform. Enter 50 percent of the level of the peak-to-peak transition level.

Table 20: Error Codes 201 through 351 (Contd.)

Error Codes	Error Message	Description	Possible Solutions
273	Unable to process the Ref waveform. High sampling rate is required for accurate measurement. Press F1 for more details.	The time/div settings for the selected waveform is more. Alternatively, sampling rate is low.	On the oscilloscope menu bar, click Horiz/Acq > Horizontal/Acquisition Setup. In the Horizontal tab, increase the record length value. Reacquire the Live waveforms. Save waveform to ref.
274	Unable to process the Ref waveform. Not enough signal cycles in a record. At least one cycle is required. Press F1 for more details.	The signal has less than one cycle in the specified record length.	Increase the record length value or decrease the time/div settings of the waveform. Reacquire the Live waveforms. Save waveform to ref. If you have selected gating, ensure that the gating period has more than one cycle.
301	Conflict in selection of sources. Press F1 for more details.	The sources selected for voltage and current are same.	Select different channel sources for voltage and current.
302	Conflict in selection of math destinations. Press F1 for more details.	The Math sources selected as destinations may be same.	Select different math sources as destinations.
303	Invalid combination of voltage and current sources. Press F1 for more details.	FastAcq mode works only for paired channels.	Select only paired channels like Ch1/Ch2, Ch2/Ch1, Ch3/Ch4, or Ch4/Ch3 to run the measurement.
304	Conflict in selection of voltage source and math destination. Press F1 for more details.	The voltage math source and math destination are the same.	Select different channel sources for voltage source and math destination.
305	Ref destination used in the definition of math voltage source. Press F1 for more details.	Math is defined as a function of Ref.	Define math only as a function of live source.
306	Conflict in selection of source and destination. Press F1 for more details.	The voltage/edge source is same as the math destination.	Select different channel sources for voltage source/edge source and math destination.
351	Unable to plot SOA because of insufficient data points. Press F1 for more details.	There are insufficient data points to plot.	Place the cursors on the waveform where at least two data points are present between the cursors.

Table 21: Error Codes 401 through 498

Error Codes	Error Message	Description	Possible Solutions
401	Change the time/div setting to 10 ms/div. Press F1 for more details.	The current harmonics algorithm (IEC standard) requires the horizontal scale to be 10 ms.	Change the horizontal scale to 10 ms using the knobs on the oscilloscope front panel.
402	Incorrect record length value. Press F1 for more details.	The current harmonics algorithm (IEC Standard) requires the record length value of the waveform to be 2500.	The record length availability for the set time base depends upon the number of active channels. Turn off the channels which are not required for the measurement. On the oscilloscope menu bar, click Horiz/Acq > Horizontal/Acquisition Setup. In the Horizontal tab, change the record length value to 2500.
403	Calculated true power is negative. Press F1 for more details.	This happens if: 1) The direction of current and voltage polarity is reversed. 2) The current has phase shift more than voltage by 90 degrees. 3) The frequency of the signal is not equal to the selected input frequency for the harmonic measurement.	Ensure that the direction of current and voltage polarity is right. Ensure that the frequency of the signal is equal to the selected frequency.
404	Class D detected; harmonic limits will be set to Class D. Press F1 for more details.	Equipment Class A was selected; Class D is detected.	Configure Class D and run the measurement again.
405	Class A detected; harmonic limits will be set to Class A. Press F1 for more details.	Equipment Class D was selected; Class D is not detected.	Configure Class A and run the measurement again.
406	Set the horizontal scale to 200 ms and the record length value to 1M. Press F1 for more details.	The current harmonics algorithm (MIL standard) requires the horizontal scale to be 200 ms and the record length value to be 1M.	1) Unable to set the required length, time base combination because the record length availability for the set time base depends upon the number of active channels. 2) Turn off the channels which are not required for the measurement. 3) Change the horizontal scale to 200 ms and the record length value to 1M.
407	Set the horizontal scale to 400 ms and the record length value to 2M. Press F1 for more details.	The current harmonics algorithm (MIL standard) requires the horizontal scale to be 400 ms and the record length value to be 2M.	1) Unable to set the required length, time base combination because the record length availability for the set time base depends upon the number of active channels. 2) Turn off the channels which are not required for the measurement. 3) Change the horizontal scale to 400 ms using the oscilloscope front panel. Set the record length value to 2M.

Table 21: Error Codes 401 through 498 (Contd.)

Error Codes	Error Message	Description	Possible Solutions
408	Set the horizontal scale to 200 ms and the record length value to 125000. Press F1 for more details.	The current harmonics algorithm (MIL standard) requires the horizontal scale to be 200 ms and the record length value to be 125000.	Change the horizontal scale to 200 ms and the record length value to 125000.
409	Set the horizontal scale to 400 ms and the record length value to 1M. Press F1 for more details.	The current harmonics algorithm (IEC A14 standard) requires the horizontal scale to be 400 ms and the record length value to be 1M.	Change the horizontal scale to 400 ms and the record length value to 1M.
410	Set the horizontal scale to 400 ms and the record length value to 125000. Press F1 for more details.	The current harmonics algorithm (MIL standard) requires the horizontal scale to be 400 ms and the record length value to be 125000.	Change the horizontal scale to 400 ms and the record length value to 125000.
411	Unable to calculate harmonics for the record length and the sampling rate combination. Press F1 for more details.	Current harmonics could not be run for the record length and the sample rate combination. (Refer to error codes 401 - 410.)	Change the horizontal scale and the record length value.
412	Calculated power is more than 25 W. Set the value greater than 25 W and enter the power factor and input current. Press F1 for more details.	IEC Amendment 14 requires that the power factor and input current are entered by the user for Class C equipment type when power is more than 25 W.	Enter power greater than 25 W and enter the current and power factor values.
451	Not enough samples available for analysis. Press F1 for more details.	The slope calculation requires a minimum of three samples.	On the oscilloscope menu bar, click Horiz/Acq > Horizontal/Acquisition Setup. In the Horizontal tab, increase the record length value.
452	Low level cannot be greater than high level. Press F1 for more details.	The dv/dt low level cannot be greater than high level.	Enter levels such that low is lesser than high.
453	Levels entered are not within the range of the waveform. Press F1 for more details.	Unable to locate the edges on the waveform at the transition because the voltage level does not fall in the transition of the waveform.	Enter the Ref Level voltage where the transition occurs on the waveform. Enter the 50 percent of the level of the peak-to-peak transition level.
454	Oscilloscope settings changed. Run the measurements di/dt and dv/dt again. Press F1 for more details.	If you change any of the vertical or horizontal settings, then the oscilloscope goes into preview mode after running di/dt and dv/dt for edge analysis.	Do not change the settings after the results are obtained for di/dt and dv/dt.
455	Unable to perform the di/dt measurement for the configured levels. Press F1 for more details.	The configured levels are not found on the edge. The cursors are placed for maximum di/dt.	Reconfigure the levels. Run the measurement again.
456	Unable to associate the cursor to the selected data. Press F1 for more details.	Unable to associate the cursor to the identified data of the power.	The oscilloscope settings have been altered. Run the measurement again.
457	Unable to calculate dv/dt for the configured levels. Cursors will be placed for maximum dv/dt.	Configured levels not found on the edge .	Reconfigure the levels.

Table 21: Error Codes 401 through 498 (Contd.)

Error Codes	Error Message	Description	Possible Solutions
471	Unable to process ref wfm. Horizontal Scale is not equal to 10ms.	The current harmonics algorithm for IEC standard requires the time/div settings to be 10 ms/div.	Change the time/div settings to 10 ms/div using the oscilloscope front panel. Reacquire the live waveforms. Save waveform to ref.
472	Unable to process the Ref waveform. The record length value is not equal to 2500. Press F1 for more details.	The current harmonics algorithm for IEC standard requires the record length value of the waveform to be 2500.	Set the record length value to 2500. Reacquire the live waveforms. Save waveform to ref.
476	Unable to process the Ref waveform. Horizontal scale is not equal to 200 ms. The record length value is not equal to 1M. Press F1 for more details.	The current harmonics algorithm for MIL standard requires horizontal scale to be 200 ms and the record length value to be 1M.	Set the record length value to 1M and the horizontal scale to 200 ms. Save waveform to ref.
477	Unable to process the Ref waveform. Horizontal scale is not equal to 400 ms. The record length value is not equal to 2M. Press F1 for more details.	The current harmonics algorithm for MIL standard requires the horizontal scale to be 400 ms and the record length value to be 2M.	Set the record length value to 2M and the horizontal scale to 400 ms. Save waveform to ref. To get the appropriate record length and scale, set the source for the measurement either as Ch1 and Ch2 or Ch3 and Ch4. Switch off the unused channels.
478	Unable to process the Ref waveform. Horizontal scale is not equal to 200 ms. The record length value is not equal to 125000. Press F1 for more details.	The current harmonics algorithm for MIL standard requires horizontal scale to be 200 ms and the record length value to be 125000.	Set the record length value to 125000 and the horizontal scale to 200 ms. Save waveform to ref.
479	Unable to process the Ref waveform. Horizontal scale is not equal to 400 ms. The record length value is not equal to 1M. Press F1 for more details.	The current harmonics algorithm for IEC 61000-3-2 A14 standard requires the horizontal scale to be 400 ms and the record length value to be 1M.	Set the record length value to 1M and the horizontal scale to 400 ms. Save waveform to ref.
480	Unable to process the Ref waveform. Horizontal scale is not equal to 400 ms. The record length value is not equal to 125000. Press F1 for more details.	The current harmonics algorithm for IEC 61000-3-2 A14 standard requires the horizontal scale to be 400 ms and the record length value to be 125000.	Set the record length value to 125000 and the horizontal scale to 400 ms. Save waveform to ref.
481	Unable to process the Ref waveform. Failed to calculate harmonics for the available record length and sample rate. Press F1 for more details.	Current harmonics requires a particular record length and sample rate for each standard.	Save the Ref waveform with a valid record length value and sample rate for the particular standard.
482	RDS ON or Vce(Sat) is not configured. The calculated energy and loss may not be accurate. Press F1 for more details.	This warning message appears if the vertical scale of the switch voltage is more than 10 V per division and device parameter RDS On or Vce(SAT) is not selected.	Select device type MOSFET or IGBT/BJT and enter the respective parameter.
484	Multiple edges found in a switching cycle or DC offset present in the waveform. Press F1 for more details.	This error appears in two scenarios: 1) When multiple edges are found within a switching cycle. This is because of improper ref and hysteresis levels. 2) If voltage and current probes are not properly compensated for DC offset, then the configured Ton and Toff levels are not met.	Set the ref hysteresis level for the voltage edge source. Remove the DC offset in the probes and run the measurement again.

Table 21: Error Codes 401 through 498 (Contd.)

Error Codes	Error Message	Description	Possible Solutions
485	The calculated edge at (Ref Level +/- Hysteresis/2) is less than Ton and Toff voltage level. Press F1 for more details.	The error appears when the configured Ton and Toff levels are less than the edge (Ref Level + Hysteresis /2).	Set the ref and hysteresis level/2 more than Ton and Toff level on the voltage waveform and run the measurement again.
486	The calculated edge at (Ref Level +/- Hysteresis/2) is less than Ton and Toff current level. Press F1 for more details.	The error appears when the configured Ton and Toff levels are less than the edge (Ref Level + Hysteresis /2).	Set the ref and hysteresis level/2 more than Ton and Toff level for the current waveform and run the measurement again.
487	The calculated edge at (Ref Level +/- Hysteresis/2) is more than 80 percent of the gate voltage. Press F1 for more details.	The error appears when the configured edge (Ref Level + Hysteresis/2) is more than 80 percent of the gate voltage.	Set the ref and hysteresis level/2 less than 80 percent of the gate voltage and run the measurement again.
488	(Ref Level +/- Hysteresis/2) is more than 100 percent. Press F1 for more details.	The error appears when the edge Ref Level + Hysteresis/2) is more than 100 percent.	Set the ref and hysteresis level/2 less than 100 percent and run the measurement again.
489	Failed to acquire and process the waveform. Press F1 for more details.	When the oscilloscope is unable to return the maximum and minimum levels, this error appears.	Run the measurement again.
491	Switching loss and energies in the result can be zero. Press F1 for more details.	The probable reason is that there can be a sharp spike in the switch current during turn ON. The amplitude of the spike is more than the remaining portion of the switch current cycle. This affects the configured I-level (percent) and is converted to absolute as product of the maximum spike current and percentage level.	Check the I and V levels and run the measurement again.
492	The calculated switching loss and energy is zero. Press F1 for more details.	There may be a sharp spike in the switch current during turn ON or the amplitude of the spike may be more than the remaining portion of the switch current cycle.	Check the I and V levels and run the measurement again. The spike can be because of the parasitic effect of the associated components and extended current loop. Use small loop to measure current and small lead to measure the voltage.
493	The number of samples between the cursors is less than two. Press F1 for more details.	This is caused by improper placing of cursors such that there are no sample points between the cursors.	Reposition the cursors and run the measurement again.
498	The number of samples between the consecutive edges is less than ten. Increase the sample rate and use the HiRes acquisition mode. Press F1 for more details.	This warning appears when multiple edges are found within a switching cycle. The probable reason for multiple edges is ringing that is present in the switch voltage.	Configure the edge level to avoid ringing portions and increase the hysteresis band. Set Hi Res as an acquisition mode.

Table 22: Error Codes 403 through 606

Error Codes	Error Message	Description	Possible Solutions
503	Set the coupling to 1 M Ω . Press F1 for more details.	Selected channel input coupling is set to 50 Ω .	Use TCS-1M convertor to run the measurement with DC block enabled. Then, run the measurement with DC block disabled. Change the coupling to 1 M on the oscilloscope.
504	Start frequency cannot be more than or equal to stop frequency. Press F1 for more details.	The entered value of start frequency is higher than stop frequency.	Enter a start frequency value that is lesser than the stop frequency value.
505	The frequency resolution (sample rate/record length) is more than the stop frequency. Press F1 for more details.	The frequency resolution for the signal for the record length and sampling rate combination is more than the stop frequency component to be analyzed.	Increase the record length value or decrease the sampling rate such that stop frequency is more than the frequency resolution (frequency resolution = sampling rate/record length) and run the measurement again.
506	Configured sample rate is less than twice the stop frequency. Press F1 for more details.	Sampling rate should be more than or equal to twice the stop frequency for accurate analysis to prevent aliasing.	Increase the sampling rate such that sampling rate is more than or equal to twice the stop frequency.
507	Frequency resolution (sample rate/record length) is larger than the range of frequency. Press F1 for more details.	The frequency resolution for the signal for the record length and sampling rate combination is more than the range of frequencies to be analyzed.	Decrease the stop frequency value to run or use the autoset option for automatic record length and time base settings.
509	1M Coupling required; DC block cannot be performed. Press F1 for more details.	50 ohms coupling is required for the DC block operation.	Set coupling to 1 Meg.
510	File path not available to save the Spectral Analysis Plot. Press F1 for more details.	Path entered for saving the plot is not valid.	Enter a valid path name.
511	The RBW value was not updated. It will be set to the maximum value. Press F1 for more details.	RBW values are not updated for the selected start, stop frequency, and window type.	Update RBW values after selecting start/stop frequency and window type before you run the measurement.
512	The oscilloscope settings have changed. Cannot set the selected RBW value. Press F1 for more details.	You might have changed the settings on the oscilloscope after updating the RBW values for the selected start/stop frequency window type.	Update RBW values after selecting start/stop frequency/window. Do not change the scope settings after updating RBW values.
513	The configured frequency range is not supported by the spectral analysis measurement. Press F1 for more details.	For the range selected, there are no RBW values available for the spectral analysis measurement.	Increase the start frequency value or decrease the stop frequency value. To run without autoset, uncheck autosetup in the options pane.
550	Timing synchronization of the previously acquired waveform is not matching with the current acquisition. Press F1 for more details.	Time domain synchronization is essential for the calculation of results even though the waveforms were not acquired simultaneously. The synchronization is checked for the given trigger level, trigger position, and cursor position for the present waveform with the previously acquired waveform.	Do not alter these settings, which results in loss of synchronization. Run the Magnetic Property measurement again.

Table 22: Error Codes 403 through 606 (Contd.)

Error Codes	Error Message	Description	Possible Solutions
551	Number of data points in a cycle is less than 10. Press F1 for more details.	The number of data points in the identified cycle is less than 10. Increase hysteresis and Ref Level for the correct identification of edge source.	Use the switching device gate waveform as edge source if the ringing is more on the voltage waveform across the magnetic component.
552	Not enough samples for correct calculation of the DC flux density to position the Hysteresis curve on the plot. Press F1 for more details.	The number of data points available for the calculation of DC value of flux density is insufficient in: 1) Number of data points in identified cycle 2) Lesser number of data points in identified integral number of cycles	1) If you are using the current source as edge source, then increase the number of divisions of the current waveform. 2) If the current waveform is noisy, use Hires mode to remove noise. 3) Increase hysteresis and adjust the Ref Level for the correct identification of edge and integral number of cycles.
553	Amplitude of each cycle is improper for computation. Press F1 for more details.	The amplitude of the signal is very low.	Decrease the vertical scale of the voltage and current channel.
602	The calculated skew value is more than the oscilloscope deskew range. Press F1 for more details.	The skew computed is more than the oscilloscope deskew range. So, the deskew will be done to the maximum range value.	The deskew will be done to the maximum deskew value applicable to the oscilloscope.
603	Edges on the waveform are lesser than the number of edges configured. Number of edges set by the user to deskew is not available on the waveform. Press F1 for more details.	Configured value of the number of edges is not found on the waveform.	Ensure that the number of edges in the waveform and the display in the numeric input are same. Adjust the horizontal scale to increase the number of transitions that are required for adjusting the deskew.
604	Edges on the waveform are more than the required number of edges for internal deskew. Press F1 for more details.	Internal deskew in DPO 5000 series oscilloscopes expect only one edge in the waveform for deskew.	The waveform or the setup may be wrong. To make accurate connections, follow the procedure in the section 'Deskewing Probes and Channels' of the online help.
605	(Hysteresis Level/2 + Ref Level) cannot be more than 100 percent of the maximum signal level. Press F1 for more details.	Level for finding the edge is more than the maximum level of the signal.	Adjust the Ref Level and Hysteresis Level such that it should not exceed the maximum level of the signal in percentage.
606	Error in copying the delays.txt file. Press F1 for more details.	File is corrupted to run the static deskew measurement.	Restart the application with the correct file to deskew.

Table 23: Error Codes 801 through 975

Error Codes	Error Message	Description	Possible Solution
801	Unable to find the required edges. Press F1 for more details.	The waveform may not be able to find the required edges due to the following reasons: 1) Noisy signal 2) Low hysteresis band level 3) Oscilloscope is in the preview mode 4) The same error reflects low record length leading to less number of samples	If the signal is too noisy, enable the 'Average' mode and acquire the signal instead of sample mode. This reduces the jitter on the signal and smoothes the signal. Increase the hysteresis band level from the application to compensate for noise in the signal.
803	Number of sample points available for analysis during Ton or Toff is zero. Press F1 for more details.	Due to low record length, the number of samples present during Ton or Toff is zero, which leads to infinity when calculating the switching frequency that is calculated for every switching.	Increase the record length value and run the measurement again. Increase the record length value to have enough samples during each switching.
804	Number of samples points available for analysis during Ton or Toff should be more than two. Press F1 for more details.	If the number of samples between 10 percent and 90 percent of switching ON or OFF portion of the voltage waveform is less than two, then the application displays a warning message.	Increase the record length value to have enough samples during each switching.
805	Invalid data for HiPower finder results. Press F1 for more details.	The waveform does not have enough edges or the waveform is not proper for the selected measurement to run successfully.	Decrease the horizontal scale of the waveform, or increase the record length value.
806	The maximum/minimum plot value is lesser than 1e-15 or more than 1e+15. The plot cannot be displayed. Press F1 for more details.	The application is not able to calculate proper plot co-ordinates if the signal level is low.	Connect the proper signal to obtain the correct value.
851	File name has invalid character(s). Press F1 for more details.	This error appears when the save/recall file name has any invalid character.	Ensure that the save/recall file name does not have any invalid character.
852	File does not exist. Press F1 for more details.	Recall the present file.	Recall the present file.
853	Settings of the recalled file are corrupted. Press F1 for more details.	This error appears when the recalled file has its data corrupted or modified.	Recall the file in which the data remains unchanged.
854	File name should not exceed 50 character(s). Press F1 for more details.	File name is limited to 50 characters because long file name will take more time to save the settings.	Ensure that the save file name does not exceed 50 characters.
871	Unable to process the Ref waveform. Not able to find the required edges. Press F1 for more details.	Waveform Analysis Module may not be able to find the required edges because of the following reasons: 1) Noisy signal 2) Low hysteresis band level selected. The same error also reflects low record length leading to less number of samples.	If the signal is too noisy, make the acquisition with 'Average' mode enabled instead of 'Sample' mode. This reduces the wavy on the signal and smoothes the signal. Increase the hysteresis band level from the application to compensate for noise in the signal. Reacquire the live waveforms. Save waveform to ref.

Table 23: Error Codes 801 through 975 (Contd.)

Error Codes	Error Message	Description	Possible Solution
873	Unable to process the Ref waveform. The number of samples available for analysis during Ton and Toff is zero. Press F1 for more details.	Due to the low record length, the number of samples present during switching ON or OFF is zero, which leads to infinity when calculating the switching frequency that is calculated for every switching.	Increase the record length value to have enough samples during each switching. Acquire the waveforms and save waveform to ref.
901	All the three outputs are off. Ensure that at least one output is on. Press F1 for more details.	Before you run the measurement, set the state of the used output to ON state.	Ensure that at least one output is set to ON.
902	The trigger voltage cannot be set more than the maximum line voltage. Press F1 for more details.	The trigger voltage should be less than the maximum line voltage.	Set the trigger voltage level lower than the maximum line voltage level.
905	Calculated power loss value is zero. One complete switching ON to ON cycle may not be available or the conduction portion of the power signal may be negative. Reacquire with at least two complete cycles. Press F1 for more details.	Power loss minimum or maximum and average value can become zero under the following conditions: 1) When coupling used is AC, then signal can vary from negative to positive and power loss calculated can become zero. 2) AC-coupled current and voltage signals are multiplied to get power-signal. To calculate the power, loss integration of the power signal is done which varies from negative to positive where probability of calculating zero value is possible. 3) When probe has DC offset.	Ensure that coupling used is DC. DC offset on the probes used should be compensated before you run the measurement again.
975	Unable to process the Ref waveform. One complete switching ON to ON cycle is not available. At least two complete cycles are required. Press F1 for more details.	One cycle in switching loss will be considered from start of ON switching to stop before next ON switching. If one complete switching cycle is not present, then the error appears.	Decrease the time base to accommodate at least two complete cycles. Acquire the waveforms and save waveform to ref.

Parameters

About Application Parameters

This section describes the DPOPWR application parameters, and includes the default menu settings. You should refer to the user manual for your oscilloscope for operating details of other controls, such as front-panel buttons.

The parameters for the menus and options list the selections available for each and include the default values.

Sequencer Parameters

Table 24: Sequencer Parameters

Parameters	Selections
Clear	None
Recalc	None
Single	None
Run	None

Measurement Menu Parameters

Table 25: Measurement Menu Parameters

Parameter	Selection	Default Setting
Power Device	Switching Analysis Switching Loss, Hi-Power Finder, SOA, SOA X-Y, di/dt, dv/dt Modulation Analysis Pulse Width, Period, Duty Cycle, Frequency	Switching Loss
Magnetics	Inductance, Magnetic Loss, Magnetic Property, I vs Integral V	None
Input/Output Analysis	Input Analysis Power Quality, Current Harmonics, Total Power Quality Output Analysis Line Ripple, Switching Ripple, Turn on Time, Spectral Analysis	None

Source Configuration Parameters

Table 26: Source Configuration Parameters

Parameter	Selection	Default Setting
Source	Ref, Live	Live
Voltage	Ch1, Ch2, Ch3, Ch4, Math1, Math2, Math3, Math4, Ref1, Ref2, Ref3, Ref4	Ch1
Current	Ch1, Ch2, Ch3, Ch4, Math1, Math2, Math3, Math4, Ref1, Ref2, Ref3, Ref4	Ch2
i-Probe Settings	TekVPI, TCP Series, Shunt, Custom, Am503S	TekVPI
AM503S	Probe Type	A63X2
AM503S Settings	Range	1 mA
Transfer Impedance	-	500 mohm
Custom	Scale Factor	1
Custom	Propagation delay	0 s
Sequencer Panel	Recalc, Single, Run	NA

Configure Hi-Power Finder and Switching Loss Parameters

Table 27: Hi-Power Finder and Switching Loss Parameters

Parameter	Selection	Default Setting
Hi-Power Finder tab		
Type tab	Fixed, Variable, Details	Fixed
On-Off Level tab	Vsw, Vg	Vsw
Vg Source	Ch1-Ch4, Math1-Math4	Ch3
Units	Absolute, Percentage	Percentage
Hysteresis %	-	10%
Ref Level	-	50%
Reference Absolute		50V
Hysteresis Absolute		5V
Device	N-Channel, P-Channel	N-Channel
V-level percentage	-	5%
I-level percentage	-	5%
V-level absolute	-	5 V
I-level absolute	-	1 A
Options tab		
Vg Level	-	1.5 V
Filter Current	-	Checked
Switch On	-	Disabled
Details		

Table 27: Hi-Power Finder and Switching Loss Parameters (Contd.)

Parameter	Selection	Default Setting
Switching Loss		
Type tab		
PWM Type	Fixed, Variable	Fixed
Cursor gating	On, Off	Off
Device	Auto, User	Auto
RDS On	-	20 mohm
On-Off Level tab	Vsw, Vg	Vg if PFC is selected else Vsw
Vg Source	Ch1-Ch4, Math1-Math4	Ch3
Units	Absolute, Percentage	Percentage
Hysteresis %	-	10%
Ref Level	-	50%
Reference Absolute		50V
Hysteresis Absolute		5V
Device	N-Channel, P-Channel	N-Channel
Units-V-level percentage	-	5%
I-level percentage	-	5%
V-level absolute	-	5 V
I-level absolute		1 A
Option tab		
Vg Level	-	1.5 V
Signal Condition	-	Checked
Filter Current	-	Checked
Switch On	-	Disabled
Details		

Configure Magnetics Parameters

Table 28: Magnetics Parameters

Parameter	Selection	Default Setting
Magnetic Loss		
Cursor Gating Details	On, Off	Off
Inductance-Edge Source	Ch1-Ch4, Math1-Math4	Ch1
Ref Level	Absolute, Percentage	Percentage
Ref Level Absolute	-	5 V
Hysteresis Absolute	-	150 mV
Ref Level Percentage	-	50%
Hysteresis Percentage	-	10%
Math Destination	Math1-Math4	Math1
# of Windings	Single, Multiple	Single
Magnetic Property tab-Type and Edge Source		
Freq/Duty	Fixed, Variable	Fixed
Cursor Gating	On, Off	Off
# of Windings	Single, Multiple	Single
Edge Source	Ch1-Ch4, Math1-Math4	Ch1
Units	Absolute, Percentage	Percentage
Ref Level Percentage	10-90%	50%
Hysteresis Percentage	5-40%	10%
Magnetic Property tab-Physical Chars		
Units	SI, CGS	SI
Number of Turns	1-1 M	1
Cross Section Area-SI		1 m ²
Cross Section Area-CGS		1 cm ²
Magnetic Length-SI		1 m
Magnetic Length-CGS		1 CM

Table 29: Magnetics Parameters (Contd.)

Parameter	Selection	Default Setting
# of Windings	Two, More	Two
Windings 2 Source	Ch1-Ch4	Ch3
Magnetizing Current	Ref1-Ref4	Ref1
Windings 3 Source	Ch1-Ch4, Unused	Unused

Configure SOA Parameters**Table 30: SOA Parameters**

Parameter	Selection	Default Setting
Cursor Gating	On, Off	Off
Mask	Editor	
Utility	Overlay	

Configure RDS(on) Parameters**Table 31: Dynamic Resistance Parameters**

Parameter	Selection	Default Setting
Real Time Math Destination	Math1, Math2, Math3, Math4	Math1
Average Math Destination	Math1, Math2, Math3, Math4	Math2

SOA Mask**Table 32: SOA Mask Parameters**

Parameter	Selection	Default Setting
Mask Grid		
Scale	Log, Linear	Linear
Linear	Mask Values	X-Min, X-Max -40 KV to 40 KV Y-Min, Y-Max -40 KA to 5 KA
Log	Mask Values	X-Min, X-Max -40 KV to 40 KV Y-Min, Y-Max -40 KA to 5 KA

Configure di/dt Parameters

Table 33: di/dt Parameters

Parameter	Selection	Default Setting
Units	Percentage, Absolute	Percentage
Math Destination	Math1, Math2, Math3, Math4	Math1
Ref Level Percentage		50%
Hysteresis Percentage		10%
Ref Level Absolute		5 A
Hysteresis Absolute		150 mA

Configure dv/dt Parameters

Table 34: dv/dt Parameters

Parameter	Selection	Default Setting
Units	Percentage, Absolute	Percentage
Math Destination	Math1, Math2, Math3, Math4	Math1
Ref Level Percentage		50%
Hysteresis Percentage		10%
Ref Level Absolute		5 V
Hysteresis Absolute		150 mV

Configure Power Quality Parameters

Table 35: Power Quality Parameters

Parameter	Selection	Default Setting
Math Wfm Destination: Power	Math1, Math2, Math3, Math4	Math1
Math Wfm Destination: Energy	Math1, Math2, Math3, Math4	Math2

Configure Current Harmonics Parameters

Table 36: Current Harmonics Parameters

Parameter	Selection	Default Setting
Standards	61000-3-2, AM 14, MIL 1399	61000-3-2
61000-3-2		
Line Frequency	50 Hz, 60 Hz	50 Hz
Harmonic Table	Table 1-10	Table 1
Class	Class A-Class D	Class C
AM 14		
Line Frequency	50 Hz, 60 Hz	50 Hz

Harmonic Table	Table 1-10	Table 1
Class	Class A-Class D	Class C
	Controls> Input Power	100 W
	Controls> Power Factor	1
	Controls> Fundamental Current	16 A
MIL 1399		
Line Frequency	60 Hz, 400 Hz	400 Hz
i-Probe Impedance		
Transfer Impedance Table	Table 1-10	Table 1
Use Impedance Table Checkbox	Unchecked, Checked	Unchecked

Configure Total Power Quality Parameters

Table 37: Total Power Quality Parameters

Parameter	Selection	Default Setting
Standards	61000-3-2, AM 14, MIL 1399	61000-3-2
61000-3-2		
Line Frequency	50 Hz, 60 Hz	50 Hz
Harmonic Table	Table 1-10	Table 1
Class	Class A-Class D	Class C
AM 14		
Line Frequency	50 Hz, 60 Hz	50 Hz
Harmonic Table	Table 1-10	Table 1
Class	Class A-Class D	Class C
	Controls> Input Power	100 W
	Controls> Power Factor	1
	Controls> Fundamental Current	16 A
MIL 1399		
Line Frequency	60 Hz, 400 Hz	400 Hz
i-Probe Impedance		
Transfer Impedance Table	Table 1-10	Table 1
Use Impedance Table Checkbox	Unchecked, Checked	Unchecked
Math Destination		
Power	Math 1- Math 4	Math 1
Energy	Math 1- Math 4	Math 2

Configure Line Ripple Parameters

Table 38: Line Ripple Parameters

Parameter	Selection	Default Setting
Coupling	AC, DC	AC
Bandwidth Limit	20Mhz, 250Mhz, Full	20 Mhz
Acquisition Mode	Hi Res, Pk Detect, Sample	Hi Res
Ripple Freq	50Hz, 60Hz, 400Hz	50 Hz

Configure Switching Ripple Parameters

Table 39: Switching Ripple Parameters

Parameter	Selection	Default Setting
Coupling	AC, DC	AC
Bandwidth Limit	20Mhz, 250Mhz, Full	20 Mhz
Switching Freq	50Hz to 1MHz	10 KHz
Acquisition Mode	Hi Res, Sample, Pk Detect	Hi Res

Configure Turn-On Time Parameters

Table 40: Turn-on-Time Parameters

Parameter	Selection	Default Setting
Input		
Source	Ch1, Ch2, Ch3, Ch4	Ch1
Converter	AC-DC, DC-DC	AC-DC
Max Voltage	1V-500V	230 V
Trigger Level	1V-500V	230 V
Max Turn-On-Time	1us to 5 us	200 ms
Output		
Source 1	On, Off	On
Source 2	On, Off	Off
Source 3	On, Off	Off
Source 1	Ch1-Ch4	Ch2
Max Voltage	-5.9 KV to 5.9 KV	5.0 V

Configure Pulse Width Parameters

Table 41: Pulse Width Parameters

Parameter	Selection	Default Setting
Source	Ch1, Ch2, Ch3, Ch4,	Ch1
Ref Dest	Ref1, Ref2, Ref3, Ref4	Ref1
Units	Level, Percentage	Percentage
Percentage	Ref Level Hysteresis	Ref Level: 50 % Hysteresis: 3 %
Absolute	Ref Level Hysteresis	Ref Level: 0 V Hysteresis: 6 V
Edge	Rise, Fall	Rise
Enable Gating	-	Unchecked
Polarity	Positive, Negative	Positive

Configure Period Parameters

Table 42: Period Parameters

Parameter	Selection	Default Setting
Source	Ch1, Ch2, Ch3, Ch4,	Ch1
Ref Dest	Ref1, Ref2, Ref3, Ref4	Ref1
Ref Level Mode	Level, Percentage	Percentage
Percentage	Ref Level Hysteresis	Ref Level: 50 % Hysteresis: 3 %
Absolute	Ref Level Hysteresis	Ref Level: 0 V Hysteresis: 6 V
Edge	Rise, Fall	Rise
Enable Gating	-	Unchecked

Configure Duty Cycle Parameters

Table 43: Duty Cycle Parameters

Parameter	Selection	Default Setting
Source	Ch1, Ch2, Ch3, Ch4,	Ch1
Ref Dest	Ref1, Ref2, Ref3, Ref4	Ref1
Edge Level: Units	Level, Percentage	Percentage
Percentage	Ref Level Hysteresis	Ref Level: 50 % Hysteresis: 3 %
Absolute	Ref Level Hysteresis	Ref Level: 0 V Hysteresis: 6 V
Edge	Rise, Fall	Rise
Enable Gating	-	Unchecked
Polarity	Positive, Negative	Negative

Configure Frequency Parameters

Table 44: Frequency Parameters

Parameter	Selection	Default Setting
Source	Ch1, Ch2, Ch3, Ch4,	Ch1
Edge Type	Rise, Fall	Rise
Edge Level: Units	Absolute, Percentage	Percentage
Ref Level: Percentage	1% to 99%	1%
Ref Level : Absolute	0 V to 6 V	0 V
Hysteresis	0 V to 3 KV	6 V
Ref Dest	Ref 1-Ref 4	Ref 1

Configure Spectral Analysis

Table 45: Spectral Analysis Parameters

Parameter	Selection	Default Setting
Source	Ch1, Ch2, Ch3, Ch4 Math1, Math2, Math3, Math4 Ref1, Ref2, Ref3, Ref4	Ch1
Frequency-Start	Min-0 Hz, Max-499 MHz	50 Hz
Frequency-Stop	Min-50 Hz, Max-500 MHz	10 KHz
Window Type	Rectangular, Hamming, Hanning, Black-Harris, Gaussian, Flattop2, Kaiser-Bessel, TekExp	Rectangular
Res RBW (Hz)	Dynamic	Depends on Start, Stop and Window values
Options	Auto Setup	Checked
Options	DC Block	Unchecked

Deskew-Static Deskew

Table 46: Static Deskew Parameters

Parameters	Selections	Default Setting
From	Source Ch1, Ch2, Ch3, Ch4	Ch1
Probe	TCP202, P5205, P5210, AM503B with A6302, AM503B with A6312, AM503B with A6303, P5100, P5050, AM503B w/ A6302X, AM503B w/ A6303XL, AM503B w/ A6304XL, P6158, P6015A (10ft), P6015A (25ft), TCPA300 w/ TCP305, TCPA300 w/ TCP303, TCPA400 w/ TCP404XL, TCPA300 w/ TCP312	P5205
To Source	Ch1, Ch2, Ch3, Ch4,	Ch2
Probe	TCP202, P5205, P5210, AM503B with A6302, AM503B with A6312, AM503B with A6303, P5100, P5050, AM503B w/ A6302X, AM503B w/ A6303XL, AM503B w/ A6304XL, P6158, P6015A (10ft), P6015A (25ft), TCPA300 w/ TCP305, TCPA300 w/ TCP303, TCPA400 w/ TCP404XL, TCPA300 w/ TCP312	TCP202

Real Time Deskew**Table 47: Real Time Deskew Parameters**

Parameters	Selections	Default Setting
From Source	Ch1, Ch2, Ch3, Ch4	Ch1
To Source	Ch1, Ch2 Ch3, Ch4,	Ch2
Slope	Rise, Fall	Rise
Edges	Min: 1,Max: 50%	1%
Ref Level	Min:0, Max: 100%	50%
Hysteresis	Min:0, Max: 25%	5%
Source	Internal, External	Internal
Edges	-	1

Measurement Algorithms

About Measurement Algorithms

This section contains information about the algorithms used by the each measurement. The application uses voltage or current waveforms for the algorithms.

Oscilloscope Setup Guidelines

DPOPWR sets the oscilloscope parameters appropriately for each measurement before running the measurements. We recommend that you follow the instructions for best results.

1. The voltage or current waveforms are expected in any Ch1-Ch4 or Ref1-Ref4. In the case of two channel oscilloscopes, they are expected in Ch1-2 or Ref1-2.
2. For best results, the vertical scale for the waveform must be set such that the waveform does not exceed the vertical range of the oscilloscope. We also recommend that you to set the vertical scale in such a way that the waveform occupies the full screen.
3. The sample rate must be set to capture sufficient waveform detail and avoid "aliasing" here.

Switching Loss

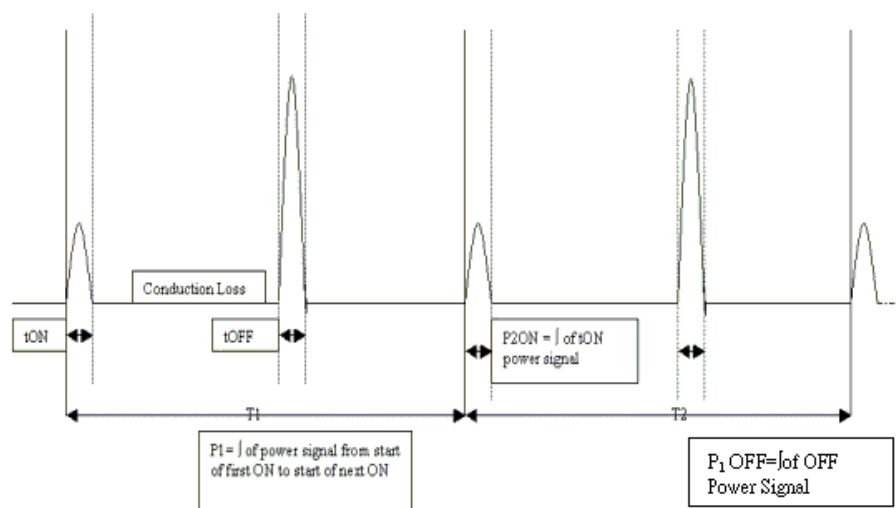


Figure 103: Switching Loss

Where:

T1 is the First switching cycle

T2 is the Second switching cycle

Tn is the nth switching cycle

Use the previous figure as a reference and calculate Total Loss using the following equation:

$$TotalLoss = 1/(T1 + T2 \dots Tn) * \int P(t) dt$$

$$Where \int P(t) dt = \sum \int P1(t) + \int P2(2) \dots \int Pn(t)$$

$$TONLoss = 1/(T1 + T2 \dots Tn) * \int PON(t) dt$$

$$Where \int PON(t) dt = \sum \int PON1(t) + \int PON2(2) \dots \int PON(t)$$

$$TOFFLoss = 1/(T1 + T2 \dots Tn) * \int POFF(t) dt$$

$$Where \int POFF(t) dt = \sum \int POFF1(t) + \int POFF2(2) \dots \int POFFn(t)$$

When the RDS On value for the MOSFET is used to calculate Total Loss in the application, use the following equation to calculate Total Loss:

Take Rds ON as the input for the switching device. Conduction loss is calculated using Rds ON as the input.

$$Conduction\ loss = (I_{rms})^2 \cdot R_{ds\ ON}$$

I_{rms} is the rms value of the switching current (I_d).

$$Total\ loss = TON\ loss\ (average) + TOFF\ loss\ (average) + Conduction\ loss$$

$$Total\ energy = TON\ energy\ (average) + TOFF\ energy\ (average) + Conduction\ energy$$

To measure conduction loss in a MOSFET, use the following equation:

$$ConductionLoss = I_{rms}^2 * RDSON$$

where:

I_{rms} is the root mean square of the switching current

To measure conduction loss in a BJT/IGBT, use the following equation:

$$ConductionLoss = V_{ce(SAT)} * I_{mean}$$

where:

I_{mean} is the mean of the switch current

To measure conduction energy, use the following equation:

Conduction Energy= Conduction Loss* Acquisition duration

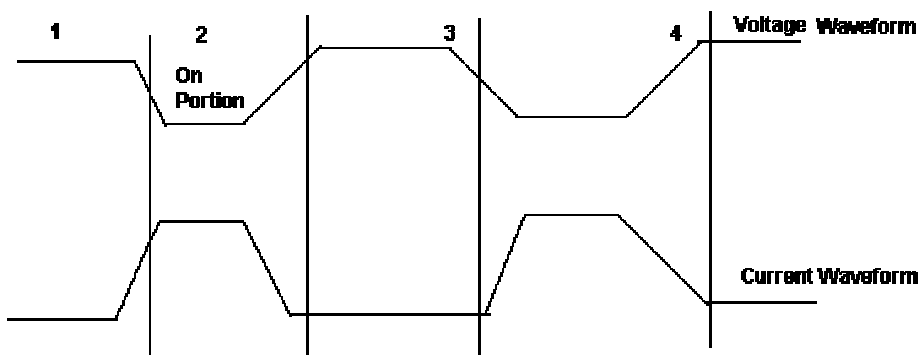


Figure 104: Calculating Ton and Toff energy

From the previous figure, the application uses the following equation to calculate TON and TOFF energy.

$$TONenergy = 1/N * \sum \int PON1(t) + \int PON2(2)..... \int PONn(t)$$

$$TOFFenergy = 1/N * \sum \int POFF1(t) + \int POFF2(2)..... \int POFFn(t)$$

Total Switching Loss=Ton Loss+Toff Loss+Conduction Loss

Basic Algorithm: For the calculation of the TON, TOFF, and average Total loss.

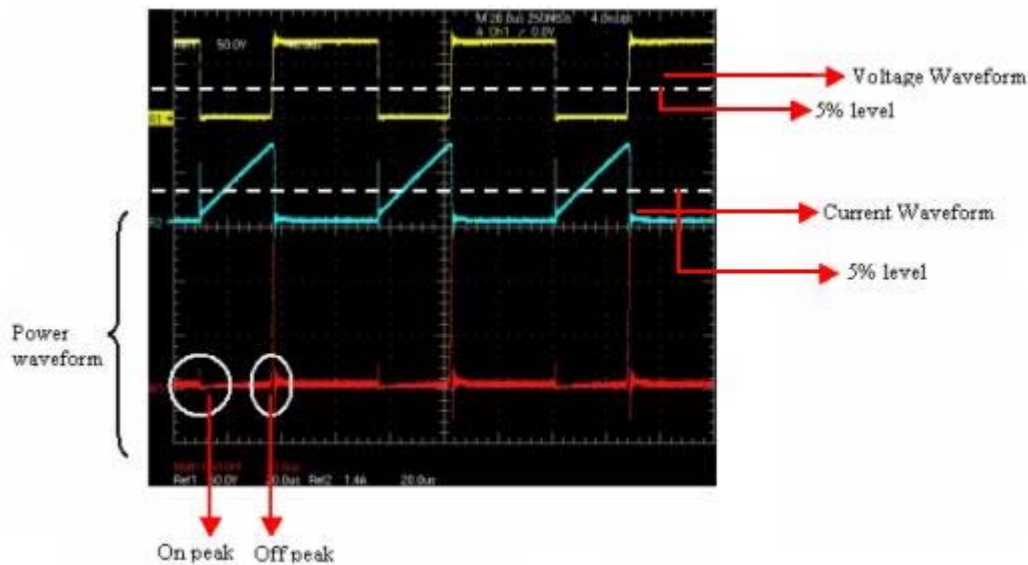


Figure 105: Calculating Ton and Toff

Concept to identify TON and TOFF using gate voltage for edge analysis:

Use gate voltage for edge analysis with default 50% edge level and hysteresis 10%.

- To find the start of TON: The start of the TON is 5% or 1.5V whichever is lower on the rise slope of the gate voltage.
- To find the stop of TON: The start index on the switch voltage is 5% or 1.5V of the rise slope gate voltage. Move forward on the switch voltage from the start index until 5% or the configured level is met.
- To find the start of TOFF: The start index is 80% of the gate voltage. From the Stop index, search for 5% of the switch voltage (on rise slope).
- To find the stop of TOFF: The 80% of the gate voltage is start index. From this start index on switch current (fall slope) move forward until 5% of the max (switch current) is met.

If you select the Filter check box in the application, use the following equation to calculate moving average filter.

$$y(n) = \frac{1}{M} \sum_{k=0}^{M-1} x(n-k)$$

y(n)= is the output

x(n)= is the input

M is four sample points that are averaged

Magnetics

The Magnetics measurement displays I vs Integral V Plot and measures Inductance, Magnetic Property and Magnetic Loss. When using the integral of voltage, take care to check that the voltage 'V' does not have any DC components. Use the oscilloscope AC coupling to avoid any DC shifts on the integral of the voltage waveform. The application creates a single cycle by averaging multiple cycles of current and integrated voltage.

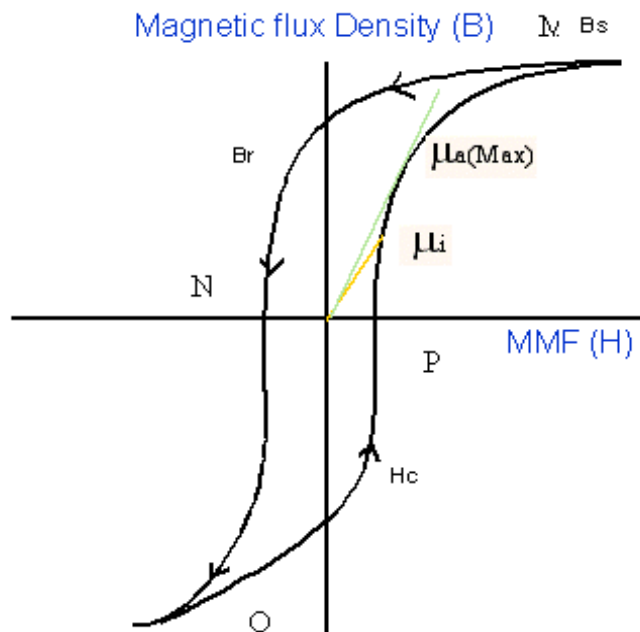


Figure 106: Calculating Magnetics

In the previous figure:

Bs is the Saturation Flux Density

Br is the Remanence Flux Density

Hc is the Coercive Force (Hc)

Ui is the Initial Permeability

Ua is the Max Amplitude permeability

Note: The data waveform starts from the Max value of H and Decreases and increases again (M-N-O-P).

Magnetic Field Strength (H): The previous figure shows the hysteresis in a typical magnetic material. The magnetic field induces a magnetic flux in the

DUT. The units of measurement are Ampere/mtr in SI unit and Oerstead in CGS unit.

Saturation Flux Density (Bs): The maximum magnetic flux density induced in the material regardless of the magnitude of the externally applied field H.

This is calculated on the maximum flux density cycle on the acquired waveform and let this be the kth cycle on the acquired waveform.

$$B_s = \text{Max}(B_k)$$

The Magnetic Field Intensity H is also calculated on the Maximum flux density cycle Bk.

$$H_s = \text{Max}(H)$$

Index I where H is maximum

$$I = \text{Index of}(\text{Max}(H))$$

$$B_s = B(I) \text{ (2)}$$

Remanence (Br): Is the Induced magnetic flux density that remains in the material after the externally applied magnetic field (H) is returned to zero during the generation of the Hysteresis loop.

Remanence is also calculated on the kth cycle (Bk) where the maximum flux density occurs in the entire acquired waveform.

Find the index at Zero value of H on the H waveform and calculate the maximum value of B from these indexes.

Let 'q' be the index at Zero value of H on H waveform. Let the q1 and q2 be the index of the waveform. Calculate the value of B at the Index q1 and q2 on the Kth cycle. The maximum magnitude value of B is the remanence Flux density.

Coercive Force (Hc): The value of H found at the intersection of the H-axis with the Hysteresis loop is the coercive force. This represents the external field required to cause the induced flux density (B) to reach zero during the measurement cycle of a Hysteresis loop. HC is symmetrical with positive and negative axis.

Coercivity Hc is also calculated on the kth cycle where the maximum flux density occurs in the entire acquired waveform.

Finding the index at Zero B value on the B waveform: Let 'q' be the index at Zero value of B on B waveform. Let q1 and q2 be the index on the B waveform where B is Zero. The maximum magnitude of the H data at the index of q1 and q2 is the coercivity.

Permeability: It is the ratio of B and H calculated on the Bk cycle. You can select the points on the BH plot using the cursor and calculate the slope of the BH curve

using the data selected between the cursor. You can choose the portion of the plot using the cursor to obtain the results.

Slope Calculation

There are N data points between the Cursor

Find $H_{av} = (H_1 + H_2 + \dots + H_n) / N$

$B_{av} = (B_1 + B_2 + \dots + B_n) / N$

$H_{norm_i} = H_i - H_{av}, i=1..N$

$B_{norm_i} = B_i - B_{av}, i=1..N$

$$B/H = \frac{\text{SUM}(H_{norm1} * B_{norm1} + H_{norm2} * B_{norm2} + \dots + H_{normN} * B_{normN})}{\text{SUM}(H_{norm1} * H_{norm1} + H_{norm2} * H_{norm2} + \dots + H_{normN} * H_{normN})}$$

Safe Operating Area

This measurement has three options:

- SOA: Plots the graph of the voltage and current waveform.
- SOA X-Y: Invokes the oscilloscope's built in XY mode. This is applicable only for live channels.

Results:

The voltage vs current graph (also known as SOA Plot) displays the value of voltage, current and power for a selected portion on the graph.

The occurrence field displays:

- The number of occurrence of the selected voltage and current position in the acquired waveforms
- Average power
- Standard deviation for the selected data point

The application calculates the Power Value using the following equation:

$$P_n = V_n I_n$$

Where:

P_n is the instantaneous power value

V_n is the voltage value

I_n is the current value

n is the sample at the particular point

The application calculates the Average Power using the following equation:

$$P_{Avg} = \frac{1}{N} \sum_{n=0}^{n=N} V_n I_n$$

Where:

N is the number of samples which has the same value in the plot

The application calculates the Standard Deviation using the following equation:

$$S_d = \frac{1}{N-1} \sqrt{\sum (x_i - \bar{x})^2}$$

where:

X_1 is the Power Value

X is the Mean Power

Dynamic Resistance (RDS(on))

Dynamic resistance (RDS(on)) is the resistance offered by a switching device when it is in the ON condition. DPOPWR helps to monitor the dynamic resistance through V/I curve using oscilloscope built-in Math capability.

Place the cursor in the area of interest on resistance curve to get the resistance value.

di/dt

di/dt measurement represents the rate at which the current changes during switching. The application uses the oscilloscope's built in math feature to provide a differentiation waveform of the current input.

When you run the measurement, the application calculates di/dt for the first edge by taking the default levels as 10% and 90% and displays the results.

You can select a specific section of the waveform on the live signal by providing inputs for high and low levels in terms of percentage and absolute value of voltage and current. You can select the edge of interest by viewing it visually on

the oscilloscope. You can also enter the edge number on the results panel. The application displays the results for the selected edge and levels on the results panel.

The application calculates di/dt using the following equation:

$$\frac{di}{dt} = \frac{\sum(x_i - \bar{x}_i) \times (y_j - \bar{y}_j)}{\sum(x_i - \bar{x}_i)^2}$$

Where:

X = timing values

Y = Vertical values of the waveform data

dv/dt

dv/dt represents the rate at which the voltage changes during switching. The application uses the math feature to provide a differentiation waveform of the voltage input.

When you run the measurement, the application calculates dv/dt for the first edge by taking the default levels as 10% and 90%, and displays the results.

You can select a specific section of the waveform on the live signal by providing inputs for high and low levels in terms of percentage and absolute value of voltage and current. You can select the edge of interest by viewing it visually on the oscilloscope. You can also enter the edge number on the results panel. The application displays the results for the selected edge and levels on the results panel

The application calculates dv/dt using the following equation:

$$\frac{dv}{dt} = \frac{\sum(x_i - \bar{x}_i) \times (y_j - \bar{y}_j)}{\sum(x_i - \bar{x}_i)^2}$$

where:

X = timing values

Y = Vertical (Voltage) values of the waveform data between the cursors

Power Quality

The Power Quality measurement calculates the root mean square value for voltage, root mean square value for current, true power, apparent power, power factor, crest factor for voltage and crest factor for current.

The application calculates the RMS Voltage using the following equation:

$$RMS(v) = \sqrt{\frac{1}{N} \sum_{n=0}^{N-1} v^2(n)}$$

Where:

RMS(v) is voltage

N is the number of samples

n is the data point

v(n) is the absolute value of the voltage at the particular data point

The application calculates the RMS Current using the following equation:

$$RMS(i) = \sqrt{\frac{1}{N} \sum_{n=0}^{N-1} i^2(n)}$$

Where:

i is current

N is the number of samples

n is the data point

i(n) is the absolute value of the current at the particular data point

The application calculates the True Power using the following equation:

$$True/RealPower = P_{Real} = \sum_{n=0}^{N-1} i(n)v(n)$$

Where:

N is the number of samples

n is the data point

The application calculates the Apparent Power using the following equation:

$$\text{ApparentPower} = P_{\text{Appar}} = \text{RMS}(v) * \text{RMS}(i)$$

Where:

RMS(v) is the root mean square of the voltage

RMS(i) is the root mean square of the current

The application calculates the Power Factor using the following equation:

$$\text{PowerFactor} = \frac{P_{\text{Real}}}{P_{\text{Appar}}}$$

Where:

P real is the real power

P appar is the apparent power

Crest factor is the ratio of the PK value of the signal to the RMS value of the signal. Use the following equation to calculate the crest factor for the voltage and current.

$$Cv = \frac{V_{Pk}}{V_{RMS}}$$

Where:

Vpk is the peak value of the voltage

Vrms is the Rms value of the voltage

$$C_i = \frac{I_{PK}}{I_{RMS}}$$

Where:

I_{pk} is the peak value of the current

I_{rms} is the RMS value of the current

Current Harmonics

The Current Harmonics measurement calculates the harmonic emissions at different harmonic levels and the total harmonic distortion.

The application calculates the harmonic emissions using the following equation:

$$X(k) = \sum_{n=0}^{N-1} x(n) e^{-j2\pi nk/N}$$

Where:

$X(k)$ is the set of Fourier coefficients obtained by the discrete Fourier transform of $x(n)$

k is the index for the computed set of discrete frequency components

N is the number of samples considered

$X(n)$ is the discrete set of time samples that defines the transformed waveform

This value of the frequency domain component is calculated using the following equation:

$$\text{Re}(k) + j \text{Im}(k)$$

Where:

Re is the real part

Im is the imaginary part

The magnitude of the Harmonics is calculated using the following equation:

$$\mathbf{Harmonics}(k) = \sqrt{\mathbf{Re}(k)^2 + \mathbf{Im}(k)^2}$$

The magnitude of the harmonics is available in Amps and decibels microamp. Use the following equation to convert Amps to decibels microamp.

$$db\mu A = 20 \log \left[\frac{\mathbf{Signal}(\mu A)}{1\mu A} \right]$$

The application displays the results for the harmonic distortion for current and the harmonic measurement. Harmonic distortion is calculated using the following equation, where the variables X_1 and x_n represent voltage or current.

$$THD_x = \frac{\sqrt{\sum_{n=2}^{\infty} x_n^2}}{x_1}$$

where:

X_1 is the fundamental value of the current or voltage

x_n is the n^{th} harmonic value of current or voltage

Total Power Quality

The algorithm for this measurement is a combination of the Current Harmonics and Power Quality algorithms.

Line Ripple and Switching Ripple

The Ripple measurements display the ripple content in the signal. In Line Ripple, the time base is set to have three cycles of 50 Hz or 60 Hz in the input waveform. In Switching Ripple, depending on the switching frequency input sets the time base to display four cycles.

Depending upon the coupling type you select, the application sets the required offset and adjusts the vertical scale to the appropriate sensitivity. The application measures the peak to peak value and displays the result.

Turn-On Time

Turn-On Time is the time taken to get the output voltage of the power supply after the input voltage is applied. DPOPWR provides an option of measuring three output signals concurrently in case of four channel oscilloscopes.

The application measures Turn-On Time with the following inputs.

You have to have the following inputs to measure Turn-On Time:

- Levels of all the channels voltages
- Maximum Turn-On Time

The application sets the vertical settings for the waveform to occupy a minimum of two divisions. The horizontal settings are set according to the configuration inputs mentioned.

Modulation Analysis

This section explains the algorithm details of the following measurements which are part of Modulation Analysis:

- Pulse Width
- Period
- Frequency
- Duty Cycle

The application measures the signals on the complete cycles in the voltage waveform. All these measurements are related with the time locations of the edges in an acquisition. The result is displayed in graphical format in one of the reference memory selected as part of the setup of the measurement.

Period and Frequency Modulation

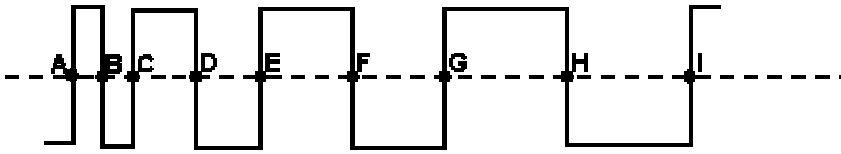


Figure 107: Calculating period and frequency modulation

The above diagram is a sample period-modulated signal that explains period and frequency modulation. The annotations A to I in the above diagram represent the V ref mid crossing in an acquired waveform.

The application calculates the period and frequency using the following equation:

$$Frequency = \frac{1}{Period}$$

$$Period = (C - A) \text{ or } (E - C) \text{ or } (I - G)$$

Where:

A, B, C, D, E, F, G, H, I are the data points

Pulse Width and Duty Cycle

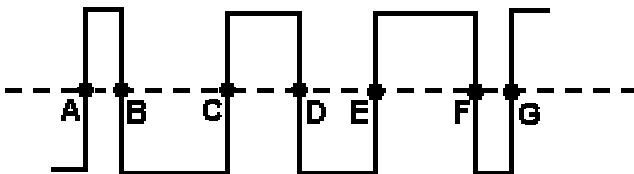


Figure 108: Calculating pulse width and duty cycle

The above diagram is a sample pulse width-modulated signal that calculates the pulse width and duty cycle modulation. The annotations A to I in the above diagram represent the V ref Mid crossing in an acquired waveform.

You can also configure the positive or negative pulse width or duty cycle.

The application calculates the pulse width and duty cycle using the following equation:

$$\text{PositivePulseWidth} = (B - A) \text{ or } (D - C) \text{ or } (F - E)$$

$$\text{PositiveDutyCycle} = \frac{B - A}{C - A}$$

$$\text{NegativeDutyCycle} = \frac{C - B}{C - A}$$

Where:

A, B, C, D, E, F, G are the data points.

Spectral Analysis

Spectral Analysis measures the magnitude of the signal over the specified frequency range by computing the Fast Fourier Transform.

The application calculates the two-sided amplitude spectrum using the following equation:

$$\text{Amplitude spectrum in quantity peak} = \text{Magnitude [FFT (A)]}/N$$

Where:

N = number of sampling points

A is the signal

The application calculates the two sided amplitude spectrum using the following equation:

$$\text{Single sided Amplitude spectrum in volts rms} = 1.141 * \text{Magnitude [FFT (A)]}/N$$

for i = 1 to N/2 - 1

= Magnitude [FFT (A)]/N for i = 0(DC)

Where:

i is the frequency line number of the FFT of A

Each frequency other than the DC is multiplied by 2 and the second half of the array is discarded.

Amplitude spectrum is closely related to the power spectrum. Single sided power spectrum is computed by squaring the single-sided rms amplitude spectrum.

The application calculates the single-sided amplitude spectrum using the following equation:

$$DB = 10\text{Log}10P/Pr$$

Where:

P is the measured power

Pr is the reference power

$$Db = 20\log10A/Ar$$

Where:

A is the measured amplitude measured in Decibel

Ar is the reference amplitude measured in Decibel

The application calculates the FFT using the following equation:

$$X(k) = \sum_{n=0}^{N-1} x(n)e^{-j2\pi nk/N}$$

Where:

X (k) is the discrete Fourier transform of x computed using a Fast Fourier Transform algorithm.

x(n) is the discrete set of time samples

k is the index for the computed set of discrete frequency components

N is the number of samples considered

Selecting RBW value

Here is an example of selecting RBW value from the calculated list. For an input square waveform of 50 Hz, 1 Volt (p-p), configure the start frequency as 50 Hz and the stop frequency as 10 KHz. When you press the update button, you will see a list of RBW values. The highest value displayed is 17.6 Hz when the window type is Rectangular.

The value 17.6 Hz is calculated as a product of 20 Hz (sample rate (100KS/s) / record length (5K)) RBW and the Rectangular window constant. The fundamental frequency 50 Hz is 2.5 times the 20 Hz RBW.

The algorithm uses an integral multiple of the RBW and the fundamental frequency. So 2.5 is rounded (Floor) to 2.0. Therefore, the 20 Hz RBW is unable to calculate the fundamental frequency and does not show 50Hz. In such cases, use an RBW value other than 17.6 and which ensures that the number of cycles is an integral value. This is observed in some start and stop frequencies and for some combinations of the RBW where the RBW is not an integral of the fundamental frequency.

Hysteresis

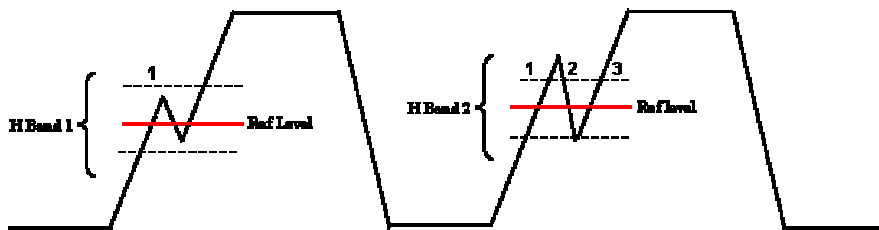


Figure 109: Hysteresis

The hysteresis input helps you to identify the cross over time on the waveform at specified reference levels as shown in the above figure. If there is any noise at the Reference level, the hysteresis band (H Band 1) helps to identify the single cross over.

If the hysteresis band (H Band 2) is less than the signal noise at the Reference level, this shows as three edges or transitions instead of one transition. Hysteresis band helps you to identify the correct transition.

Appendix

Appendix A

Use differential voltage and current probes to probe voltage and current signals in a switching power supply. Since these probes have active circuits, correct the DC offset errors. The DC voltage error adds to the test signal to display wrong results. The different propagation in differential voltage probes and current probes causes errors in time-correlated measurements and displays wrong power waveform measurements.

Follow the procedure below to eliminate these errors:

For P5205 probe

1. Connect P5205 to Channel 1 of the oscilloscope.
2. Set the probe at 50X position.
3. Connect the + and - probe inputs to the same test point.
4. Set the coupling to DC.
5. Set the vertical sensitivity to maximum.
6. Check for no DC voltage offset on screen.
7. If there is a DC offset on the oscilloscope screen, use a small screwdriver and set "adjust offset" potentiometer at the BNC end of P5205, so that the DC voltage is zero.

For TCP202 probe

1. Connect TCP202 to Channel 2 of the oscilloscope.
2. Close the TCP202 clamp.
3. Press degauss and release degauss button at the BNC connector of TCP202.
4. If DC offset is present on screen, use "Balance" at the BNC connector of TCP202 to adjust the DC current level to zero.

Note: P5205 and TCP202 are matched probes. You do not have to deskew these probes because they have the same propagation delay.

5. The test setup is ready to take power measurements.

Appendix B

1. Do not disturb any oscilloscope settings during measurement analysis. Any disturbance may result in unexpected behavior of the application.
2. The application expects the data waveform in a nonequivalent time mode (ET) for its analysis. If the equivalent time mode is turned AUTO, the application switches off the equivalent time mode and measures the signal. The application does not display a warning message and the record length may vary according to the horizontal settings.
3. The application uses the reference memory and/or the math based on the measurement selected. The application overwrites the data in the reference memory and/or math. The application does not display a warning message.
4. The application runs with a record length of 20M or less. If you select a greater record length, the application limits itself to 20M.
5. While measuring Power Device Analysis, SOA and Magnetics, do not disturb the oscilloscope settings during the post analysis of the results. This leads to wrong interpretation of results.
6. Connect the current probe in the correct direction. If not, the application displays negative results for the inductance value.
7. Compensate the DC offset in the current probe and voltage probe. If not, the core loss result becomes negative or Peak Finder measurement displays improper cursor association.
8. Select the appropriate voltage and current channels according to the voltage and current probes connected to the channels. If not, the B-H curve will be reversed and other measurements may provide wrong results.

Glossary

AC Signal

A time-varying signal whose polarity varies with a period of time T, and whose average value is zero.

Crest Factor: cf

The ratio of the peak value to the rms value of an ac waveform measured under steady-state conditions. It is unit-less, and the ratio for a pure sine wave is equal to $\sqrt{2}$.

$$cf = \frac{V_{in, peak}}{V_{in, rms}}$$

where V_{in} is the voltage at the user input terminals.

dB

The abbreviation for decibels.

Current Probe(s)

A current probe is used to measure dc, ac, or composite currents. DC current probes should measure dc and composite currents to within $\pm 1\%$ with a probe calibrator and $\pm 3\%$ without the calibrator. AC current probes should measure ac currents to within $\pm 5\%$. This accuracy should be maintained up to the worst case expected peak current. Proper bandwidth should also be ensured.

DC Signal

A signal whose polarity and amplitude do not vary with time.

Electrical Power Device Analysis

The difference between the electrical input power to the UUT and its electrical output power, expressed in watts.

$$P_{diss} = P_{in} - \sum_i P_{o,i}$$

Alternatively, Power Device Analysis is expressed as:

$$P_{diss} = P_{in} - \sum_i P_{o,i} = (1 - \eta)P_{in}$$

Harmonics

Sinusoidal voltage or current components (distortion) of a periodic waveform which occur at a frequency that is an integer multiple of the fundamental frequency.

Most non-linear loads generate odd-numbered harmonics, for example, as a result of full wave rectification of the input power. The frequencies at which these "characteristic harmonics" are produced by a user with an input rectifier can be determined by the equation:

$$f_H = (k \cdot q \pm 1) \cdot f_1$$

where

f_H = the characteristic harmonic (e.g. the "third harmonic" when $H = 3$);

H = the number of the harmonic;

k = an integer, beginning with 1;

q = an integer, representing the number of rectifier commutations per cycle

f_1 = the fundamental frequency.

Half wave rectification produces even-numbered harmonics that cause very undesirable results (e.g., dc content) in the ac power system. Full wave rectification at the input of single-phase power loads results in 'triplen' harmonics at odd multiples of three times the fundamental frequency. These are also very undesirable, considering the potential quantity of single-phase loads and the fact that these harmonics interact with the distribution system's normally high zero sequence impedance. User distortion current requirements are intentionally restrictive therefore for even and triplen harmonics.

Power Factor (Displacement)

For user equipment, the displacement power factor is equal to the cosine of the angle, φ , between the input current and the input voltage at the fundamental frequency.

$$PF_{\Delta\varphi} = \cos \varphi$$

This definition of power factor does not include the effect of distortion in the input current (and/or voltage) waveform.

Power, Total or Apparent

The total or apparent power (S) is the product of rms voltage and current (VA)

Power Factor (Displacement)

For user equipment, the displacement power factor is equal to the cosine of the angle, φ , between the input current and the input voltage at the fundamental frequency.

$$PF_{\Delta\varphi} = \cos \varphi$$

This definition of power factor does not include the effect of distortion in the input current (and/or voltage) waveform.

Power Factor (Distortion)

The distortion power factor is defined as:

$$PF_{dt} = \frac{I}{\sqrt{I + THD^2}}$$

where THD (Total Harmonic Distortion) is defined in 1.22

This definition of power factor does not include effect of displacement.

Power Factor (True)

For user equipment, the true power factor is the ratio of the active, or real, power (P) consumed in watts to the apparent power (S) drawn in volt-amperes, with

$$PF = P / S \quad \text{and}$$
$$S = \sqrt{P^2 + Q^2}$$

where

PF = power factor;

P = active power in watts;

Q = reactive power in vars;

S = total power in volt-amperes.

This definition of power factor includes the effect of both displacement and distortion in the input current (and/or voltage) waveform.

Alternatively, if there are no inter-harmonics, equation 7 can be simplified to

$$PF = PF_{dp} \cdot PF_{dt}$$

RMS Value (Voltage)

The square root of the average of the square of the value of the function taken throughout the period. For instance, the rms voltage value for a sinewave may be computed as:

$$V_{rms} = \sqrt{\frac{1}{T} \int_0^T v^2(t) dt}$$

where

T = waveform time period;

$v(t)$ = instantaneous voltage at time t ;

V_{rms} = rms voltage value.

Total Harmonic Distortion

The ratio, expressed as a percent, of the rms value of the ac signal after the fundamental component is removed and inter-harmonic components are ignored, to the rms value of the fundamental. The formula defining total harmonic distortion (THD) is provided in equation (7). The variables 'X₁' and x_n may represent either voltage or current, and may be expressed either as rms or peak values, so long as all are expressed in the same fashion.

$$THD_x = \frac{\sqrt{\sum_{n=2}^{\infty} x_n^2}}{X_1} \cdot 100\%$$

Where:

X₁ = Fundamental value of current or voltage;

x_n = nth harmonic value of current or voltage.

Transfer Impedance

The output voltage divided by the input current while the output is delivering rated current.

Hysteresis

The difference between the threshold voltage for the UVLO and the threshold voltage for turn-on.

Vmax Input

The maximum allowable input voltage rating at which UUT can operate to specifications.

Vmin Input

The minimum allowable input voltage rating at which UUT can operate to specifications.

Vnom Input

The stated or objective value of the input voltage, which may not be the actual value measured. The value should be between the minimum and maximum input value.

Voltage Probe

A connecting device, usually consisting of a two-conductor shielded cable and frequency-compensating network, with a hand-held tip, for use with an oscilloscope to measure the amplitude and waveshape of a DC, AC or a composite signal. It should include a ground reference. The measurement bandwidth should be at least 10 times greater than the frequency of interest. The impedance should be at least 50 times greater than the node impedance under measurement. A low impedance probe should be used for measurement.