

Setting Up Load Pull with X-Parameters Using the Agilent NVNA

Introduction

This application note provides step-by-step instructions for setting up a test bench for load pull with x-parameters measurement using two Maury MT98x Series automated tuners, Maury ATS software version 5.1 (build 5 or newer), and the Agilent PNA-X test set, to test a low-power DUT of a type suitable for direct connection to the PNA-X.

Step 1. Software Installation

- 1.1 Required Software Versions:
 - 1.1.1 PNA-X firmware revision A.08.50.01 or later
 - 1.1.2 NVNA firmware revision A.01.01.00 or later
 - 1.1.3 ATS Software version 5.1.00 Build 5 or later
- 1.2 Boot up the PNA-X. It will start the PNA-X app automatically.
- 1.3 In the PNA-X app, select File->Exit to close the PNA-X app.
- 1.4 Put a memory stick with the ATS installation file into a USB port and use the explorer to find and run it. When the installation is complete, reboot the PNA-X.
- 1.5 Use the start menu to find the icon to start the ATS software. Click and hold the right button, and drag it to the desktop. Release the mouse, and select copy the shortcut.

Step 2. Characterizing The Tuners

- 2.1 The tuner planes should be setup so both tuners are 3.5mm-female to 3.5mm-male.
- 2.2 If the tuners need to be characterized, setup the system to go between the PNA-X ports 1 and 2, and run the standard PNA software, not the NVNA software. Setup a config with just the tuners, a VNA, and RF switches, using VnaPnaxCom.exe and SwPnaxCom.exe for the drivers.
- 2.3 Make sure that the connection links are all in place on the PNA-X.
- 2.4 Do an in-situ cal, and uncheck both noise and power. It will prompt to cal 2-port at the DUT planes, 1-port after the load tuner, and 1-port before the source tuner (if used). Then it will cal both tuners together. Adaptive Modeling with .08 separation, 45° max phase step, and 5 harmonics with speed enhancement will take about 20 minutes per tuner.

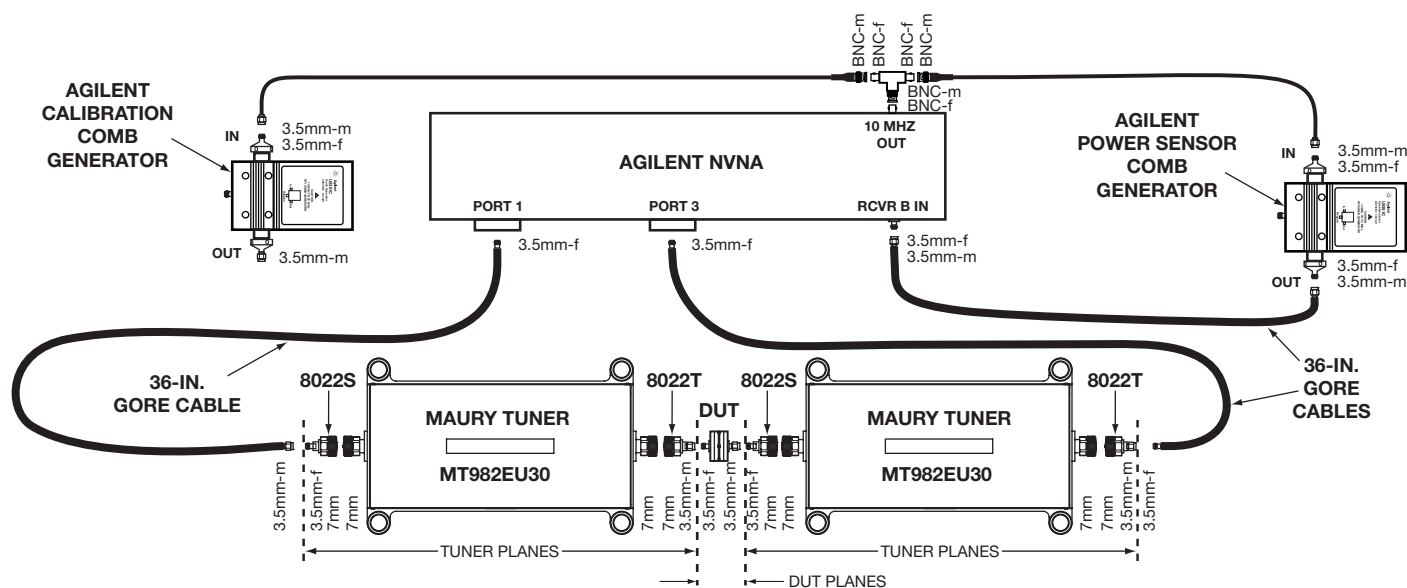


Figure 1. Setup for the Standard Demo



Step 3. Setting Up The System

- 3.1 See Figure 1 for the standard demo setup. Every link on the front panel must remain connected except the top link on the right, which must be removed to allow the Comb Generator to be connected to the RCVR B IN.
- 3.2 Use a BNC Tee on the 10 MHz reference output (rear panel), and use 2 BNC to SMA cables to connect the 10 MHz to the input of both Comb Generators.
- 3.3 Note that both the comb generator to be used for calibration and the power sensor have 3.5mm male connectors, so they can both connect directly to the DUT output reference plane (PNA-X port 3).
- 3.4 Set up a 12 Volt supply capable of about 1 amp. Connect the bias cables from both Comb Generators in parallel to the 12 Volts. On the Comb Generator bias cables:
 - Red = +12 Volts
 - Green = ground
 - Black = not used
- 3.5 RF Connections (See Figure 2 for connection diagram).
 - 3.5.1 Remove the top link on the right side of the front panel and connect the output of the measurement Comb Generator to the "RCVR B IN" connector.
 - 3.5.2 Arrange the second (Calibration) Comb Generator so that the cables will reach the DUT port. It will be used during the NVNA cal only.

- 3.5.3 Connect the PNA-X port 1 to the input of the source tuner.
 - 3.5.4 Connect the PNA-X port 3 to the output of the load tuner.
- **Note:** If external instruments are to be controlled by GPIB, use the top GPIB connector on the back of the NVNA (labeled "Controller"). See Step 6 for setting up NI-488.2 compatibility.

Step 4. Starting Up The NVNA

- 4.1 Boot up the PNA-X. It will start the PNA-X app automatically.
- 4.2 In the PNA-X app, select File->Exit to close the PNA-X app.
- 4.3 Click on the NVNA icon on the desktop to start it up.
- 4.4 Setup the 10 MHz reference signal as the Comb Generator source.
 - 4.4.1 In the NVNA app, select Utilities->External Instruments->Phase Reference->Source Setup (see figure 2).
 - 4.4.2 In the top drop-down box, select 10 MHz.
 - 4.4.3 Select Divide ratio to be 1.
- 4.5 For this demo, we will only make measurements using the General Domain without envelop domain, doing CW measurements.
- 4.6 Turn off leveling, if desired.
- 4.7 Turn on high power mode, if desired.

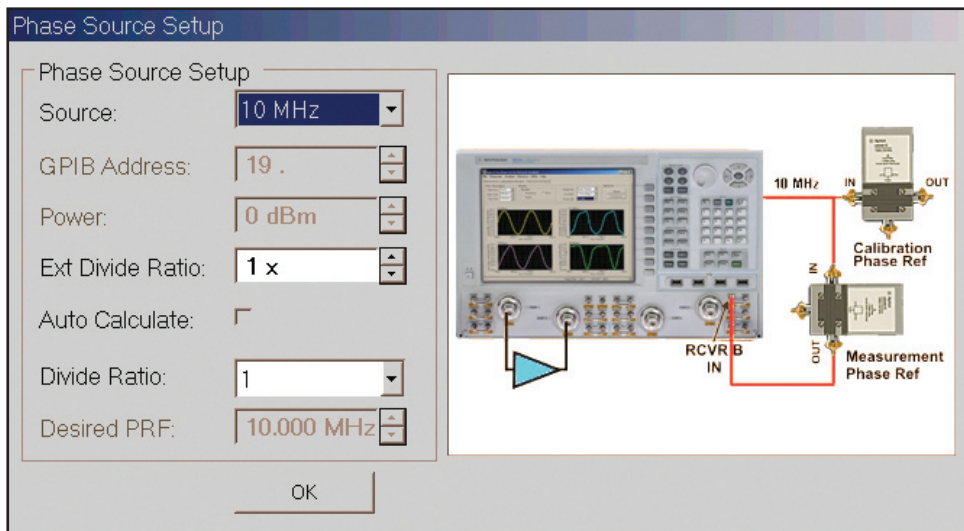


Figure 2. Connection Diagram and Setup Using the 10 MHz Source to Drive the Comb Generators.



Step 5. Setting Up The Bias

- 5.1 The ATS software should control and set the bias, and also read back the bias to get PAE into the ATS data.
- 5.2 When taking x-parameter measurements, the NVNA should also read the bias, so that PAE is modeled correctly in the x-parameter file. To set this up:
 - 5.2.1 From the NVNA menu, select Utility->External Instruments->GPIB Instrument Control.
 - 5.2.2 Click on the “Add Instrument” button.
 - 5.2.3 In the list of Instrument Properties, highlight a property one at a time to enter the data for that property. The main properties to setup for reading bias are usually:
 - a) **Name**
 - b) **GPIB Address**
 - c) **Measured Variables** – typically I2 for output current. Could also add I1 for input current, V2 for output voltage, or V1 for input voltage. Typically input bias variables are not needed because they don’t change or are not significant.
 - d) **Measurement Commands** – need all commands to measure I2 on one line. If additional measured variables are to be used, use one line for each.
 - e) **Timeout** – this is entered in mSec.
 - 5.2.4 Do not include the following, as doing so will interfere with the ATS settings:
 - a) **Initialization Commands**
 - b) **Outer Loop Commands**
 - c) **Sweep Variables**

Step 6. Setting Up NI-488.2 Compatibility

- 6.1 Required to run ATS GPIB drivers.
- 6.2 Run the Agilent connection expert.
 - 6.2.1 Run it from the computer start menu->Agilent IO Libraries Suite.
 - 6.2.2 Select Tools->Agilent 488...:
 - a) Check Enable Agilent GPIB cards for 488 programs
 - 6.2.3 Right-click on “GPIB0” to get the popup menu:
 - a) From the popup menu, select “Change Properties”.
 - b) Click the “Agilent 488 Properties...” button, and note the GPIB board number. This is the address of the GPIB board to use in the ATS software (NVNA default is 10).
 - 6.2.4 In the ATS software, select “Setup Instruments” from the main block diagram menu, and click the

“GPIB Board button. Enable the GPIB and set the address of the GPIB board.

Step 7. Calibrating

- 7.1 Initialize the tuners and set them to the Z0 position.
 - 7.1.1 Start the ATS software by clicking on the ATS icon on the desktop.
 - 7.1.2 Select nvna.cfg as the starting configuration. Change the tuner model and serial number as required.
 - 7.1.3 At least initialize the tuners and set them to Z0 before calibrating the NVNA
- 7.2 To calibrate the the NVNA:
 - 7.2.1 In the NVNA app, select Utilities->FW Visible. This is an optional step that will start up the normal PNA-X firmware and make it visible. There is a delay while it starts, but once started, the cal wizard will go more quickly.
 - **Note:** Press the Alt-Tab keys together to switch back and forth between the NVNA app and the ATS software.
 - 7.2.2 In the NVNA “Measurement Configuration/Calibration” tab, enter the start and stop frequency to be the same as the fundamental frequency that you want to measure, and click “Apply”. This will make the calibrate menu item available.
 - 7.2.3 In the NVNA app, select Response->Cal-> Calibrate. This will start the calibration wizard.
 - a) **Vector cal** (click on the red button) – this will start the calibration wizard from the normal PNA software. There may be a significant delay until the PNA software loads for the first time. Do a 2-port cal between ports 1 and 3.
 - b) **Phase cal** – Connect a comb generator to port 1 or 3 at the DUT plane (In my setup, port 3 is used because the test port is a female connector.) Be sure to select the correct comb generator.
 - c) **Power Cal** – Connect the power sensor to the same DUT plane as the phase cal. In my setup, that would again be port 3.
- 7.3 Prepare the ATS software for measurement with the NVNA.
 - 7.3.1 In the ATS software, double-click on the LSNA icon to bring up the properties dialog. Then select the LSNA Calibration Reference planes to specify where the NVNA cal planes are:
 - a) **DUT:** means that the NVNA cal planes were exactly the same as the DUT planes.



- b) **Tuner:** means that the NVNA cal planes are next to the tuner, on the side of the test fixture away from the DUT. If there is no fixture specified in the block diagram, the DUT planes and the Tuner planes are identical.
 - c) **Test Port:** means the NVNA cal planes are on the back side of the tuner, away from the DUT. This may be required if the power sensor and comb generator used for the NVNA cal cannot be connected to the DUT or Tuner planes (e.g., when making on-wafer measurements).
- 7.3.2 Measure the load termination behind the load tuner. In the ATS software, select Calibrate->LSNA->Load Termination. Enter a label and filename, and an .s1p file will be saved. Then to use the file, select Setup->Options->Power tab, and set the Load Gamma Calibration to .s1p file. Select Setup->Default Files/Directories->Power tab, and enter the filename for the Power Load Gamma.
- 7.4 Calibrate for power in the ATS software
- 7.4.1 In the ATS software, select Setup->Options, power tab, make sure that:
- a) the Power Measuring Instrument is the LSNA.
 - b) Interpolated Tuning is checked (required to export x-parameters to ADS)
 - c) The gamma-state Separation is reasonable (~0.2).
- 7.4.2 Run the CW power cal.
- 7.5 Measuring with X-Parameters in the ATS Software.
- 7.5.1 In the ATS software, select Measure->CW to enter the interactive measurement screen.
- 7.5.2 Select Setup->Options and Configuration..., and in the tuning box of the power tab, set interpolated tuning to Yes, and enter a suitable Gamma-State Separation, such as 0.2. This is important, so that you can select a uniform grid of gamma points, as required by ADS. Note that the PHD model in ADS is said to be valid for roughly 2:1 vswr around each measured point.
- 7.5.3 Select Setup->Select Measurement Parameters..., and double-click on X-Params to select them.
- 7.5.4 Clear any previously selected gamma points. Then right click on the load chart, and Select points Density. Then drag the mouse over the desired portion of the Smith chart. With interpolation set to yes, this should give a uniform grid of gamma points.
- 7.5.5 A sweep plan can sweep frequency, input power, and bias.
- a) If frequency is a variable, all frequencies must be on the same regular grid. To set this up, select the gamma points on a regular grid at one freq, then in the sweep plan setup dialog, click on details for the gamma variables. In the gamma details dialog, click the button "Select these gammas for all frequencies"
 - 1) **Note:** this is required for ADS to interpolate between freqs.
 - 2) If the desired gamma needs to rotate much on the Smith chart vs freq, select a range covering all desired freqs. Then to save time, un-select portions of the Smith chart that are not needed at each freq. The data should then be valid in ADS at each freq, but interpolation between freqs may not work.
- 7.5.6 Run a measurement. If X-Parameters were selected for measurement, then when the normal ATS file is saved, two additional files will be saved. If Filename is the file name for the normal ATS data, the extra files will use names derived from it as follows:
- a) Select a Filename with no spaces, otherwise ADS will not accept it.
 - b) Select a Filename starting with alpha character, otherwise ADS will not accept it.
 - c) Filename_PHD.mdf – this file contains the normalized x-parameters that are used in the PHD model in ADS.
 - d) Filename_XP.mdf – this file contains the un-normalized x-parameters.
- 7.6 Loading the X-Parameters into ADS for simulation
- 7.6.1 Copy or move the Filename_PHD.mdf file into the ADS project directory. Typically this will be C:\users\default\PHD_Demo_prj\PHD_Files.
- 7.6.2 Run ADS, and install the design kit that comes with the NVNA option.
- 7.6.3 Select PHD (X-params)->Create Single PHD Component, and select the .mdf file from the load pull or sweep plan measurement. This will create a .dsn file.
- 7.6.4 Open the ADS schematic, and drag and drop the new .dsn file into the place for the active device.
- 7.6.5 Click the Simulate icon to start circuit simulation.