ECE 451

Automated Microwave Measurements Laboratory

# ADS tutorial (Reading CITIfile)

An engineer frequently will need to load measured data into his or her simulation software of choice. This tutorial shows you how to do so for Keysight Advanced Design System (ADS) with CITIfile format data.

Launch ADS in Citrix. Browse to your prefered working directory (i.e. where you will store your ADS workspaces, in this tutorial, I will put my ADS workspace in "ADS_tut" folder). From the window in Figure 1, click on ![icon] to create a new workspace.
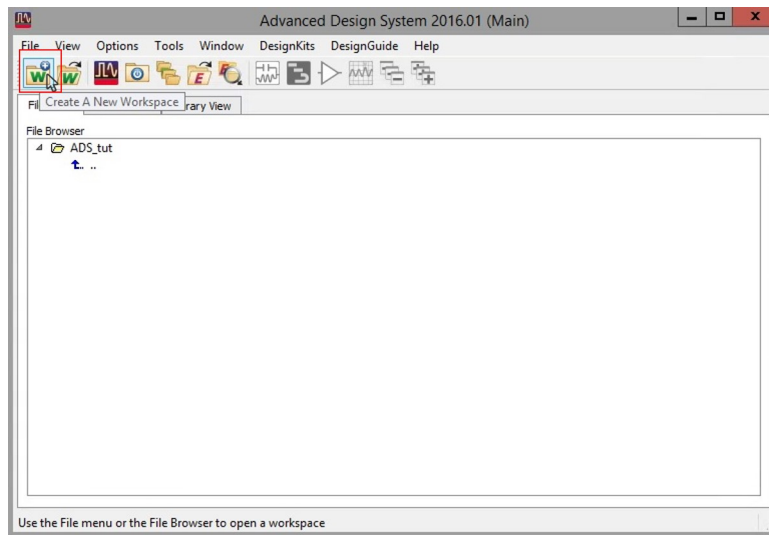


Figure 1: ADS main window

Name your workspace as you want, but keep in mind that ADS does **not** allow any whitespace in the name. Your working directory should **not** have any whitespace as well. After doing so, click "Finish". Once you do so, the workspace window will appear automatically.
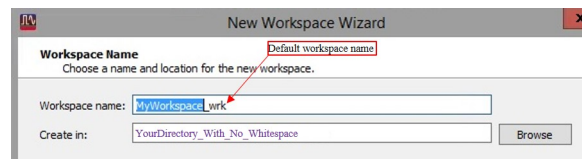


Figure 2: Naming a new workspace

Now, when you're in a workspace, ADS will allow you to create a new schematic where you can draw circuits, or a new display data window where you can work with data and visualization as all the features in Toolbar will be activated (earlier most of them were greyed out).



Figure 3: ADS Workspace Toolbar

Click "*Window>New Data Display*" or the icon  on Toolbar (see Figure 3). A new data display will pop up as shown in Figure 4
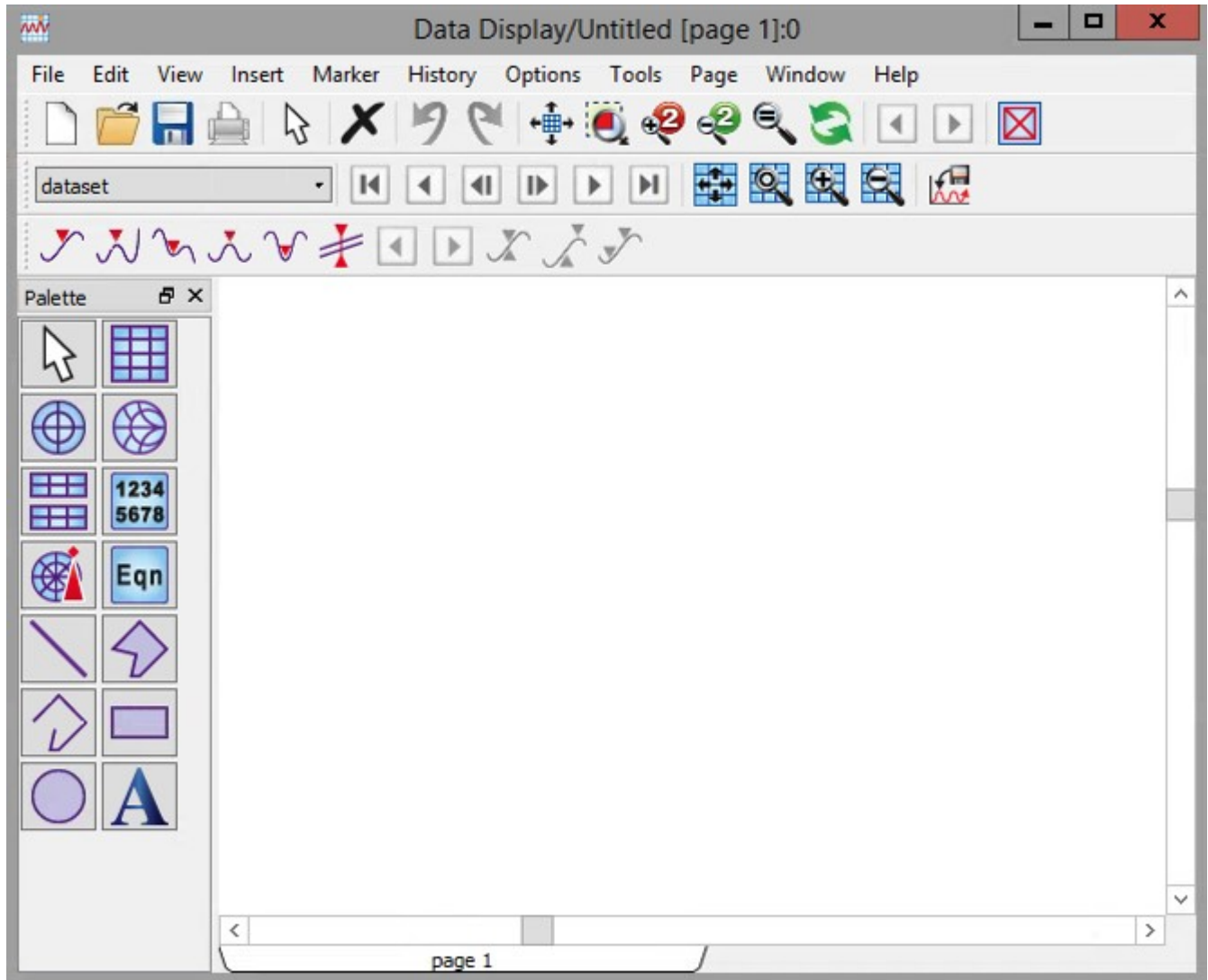


Figure 4: Data display window

On this new display, click "Tools" and select "Data File Tool...". The Data File Tool window will pop up as well as the simulation status screen (the window with the menu bar includes: File, Simulation, Text, Window). Ignore the simulation status window for now and click on the data file window, shown in Figure 5.
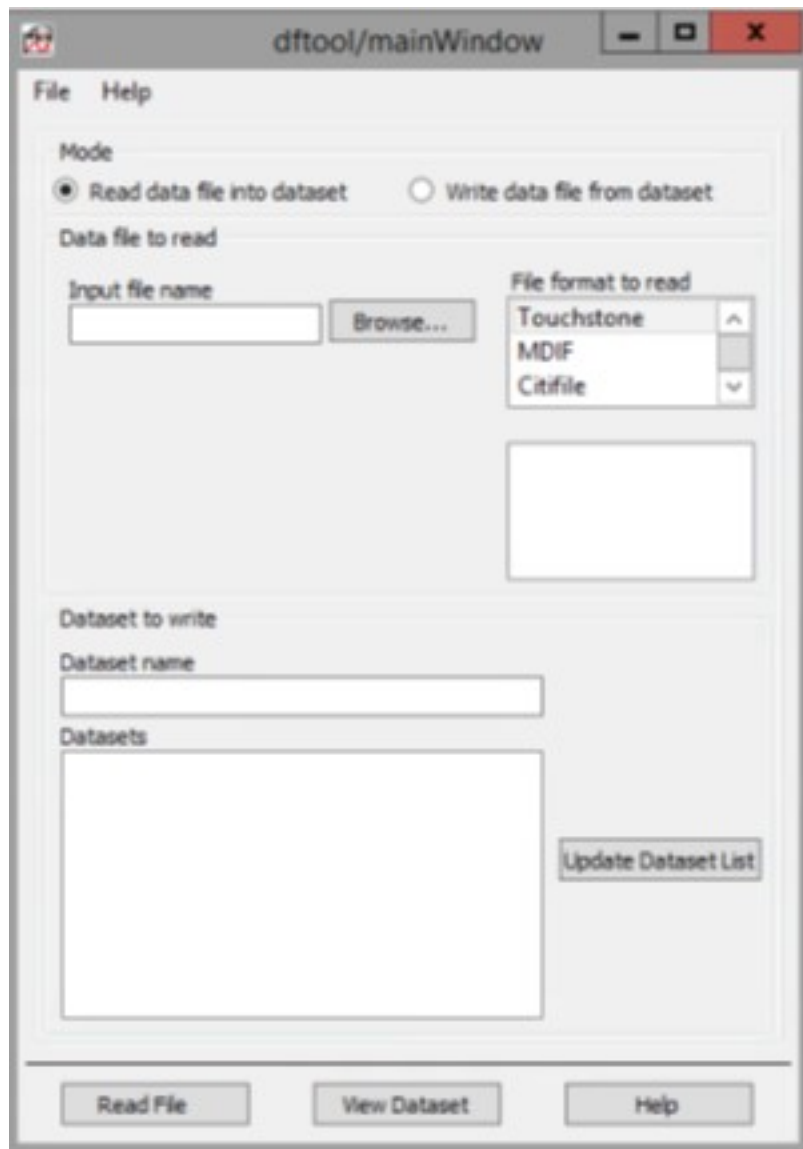
Figure 5: Data File Tool Window

*Browse* to the CITIfile you generated in Labview Tutorial 1, make sure *Citifile* was the "File format to read". Give a name to the data you're about to read into ADS in "Dataset name" box. Again, no whitespace is allowed. Press *Read File*. The simulation status window should pop to the front and says "File read was successful". You can now close the Data file toll window.

Back on the Data Display Window, click on the Rectangular Window icon,  , and place it on the white space. After you do so, a dialog box will pop up. Select "logvolt" and then press "Add Vs." and select "pwrsweep." Press "OK" twice and you will see plot of logvolt vs. pwrsweep. Do the same to generate another plot with volt vs. pwrsweep.

Next try pressing  and then clicking on the logvolt vs. pwrsweep graph to place a marker. Place a marker at -10dBm, as shown in Figure 6.
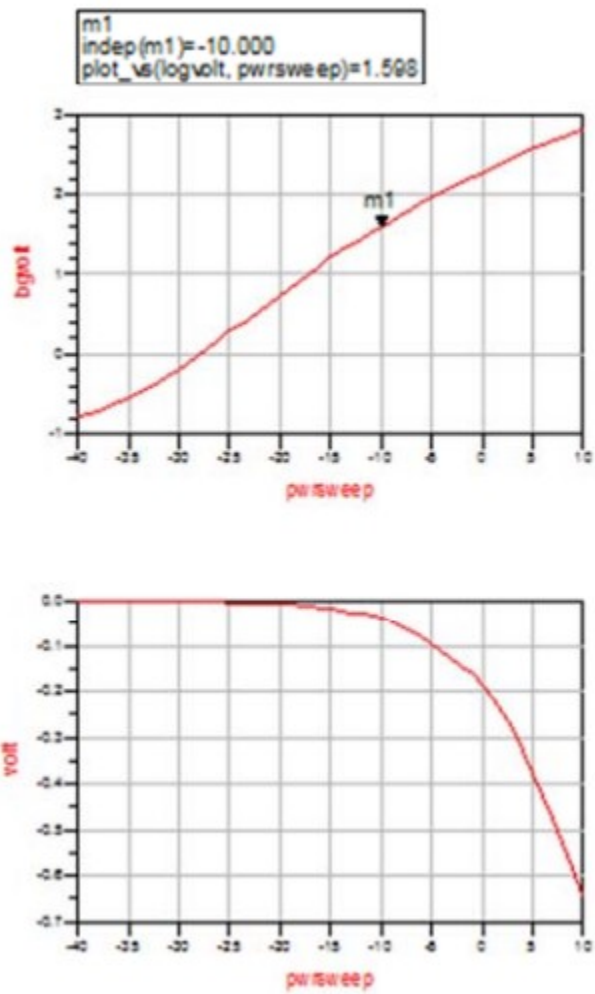
Figure 6: Final result