

PSPICE for TI

A short overview

2/22/2022

Art Kay

Thanks for support on this

JC Zhu

Sean Cashin

Carolina Gomez

Cynthia Sosa

Keith Nicholas

Collin Wells

István Eperjesi (Design Soft)

Dr Michael Koltai (Design Soft)

And many others...

Thanks!

Table of contents (click link to jump to section)

[Getting and Installing PSPICE for TI](#)

[New Project](#)

[Adding components](#)

[Adding simulation profile & Running a Simulation](#)

[Adding plots and curves](#)

[Post processing](#)

[Using Cursors](#)

[DC Sweep](#)

[AC Sweep and Noise analysis](#)

[Parameter Stepping](#)

[Monte Carlo Analysis](#)

[Using multiple schematics and simulation profiles](#)

[Getting professional PSPICE](#)

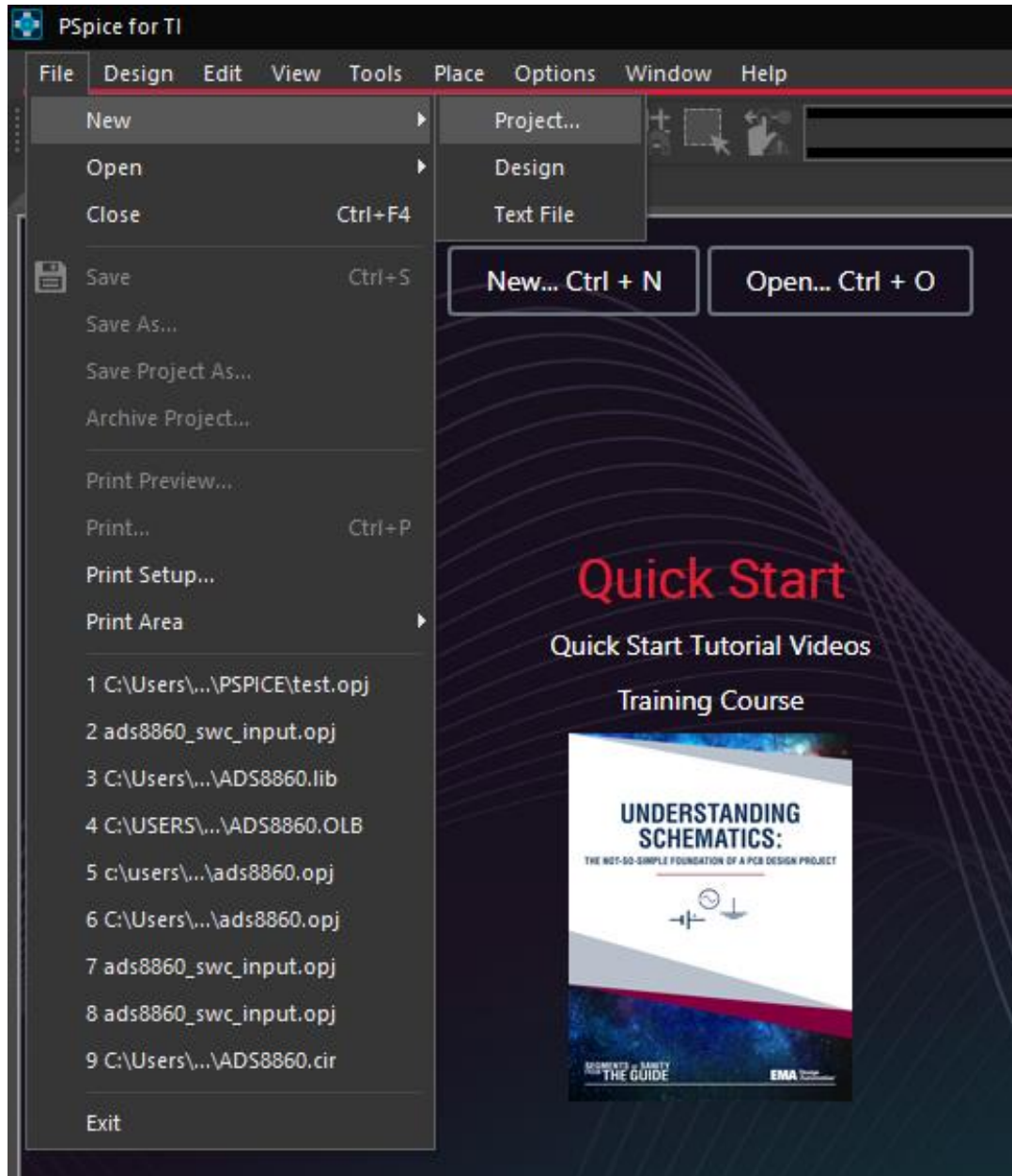
[Uploading a Model to the web and PSPICE TI library](#)

[Back to Table of Contents](#)

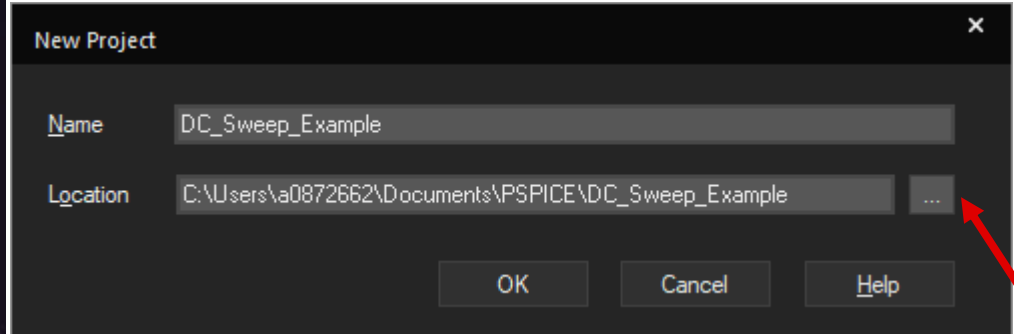
Getting and Installing PSPICE for TI

- Visit <https://www.ti.com/tool/PSPICE-FOR-TI> to make the request.
- Make sure your myTI information is complete, all required fields (marked with “*”) are filled in.
- An email will be sent to the registered email address containing a download link and an access key.
- Click the download link to get the installer. Run the installer, enter the registered email and the access key when prompted.

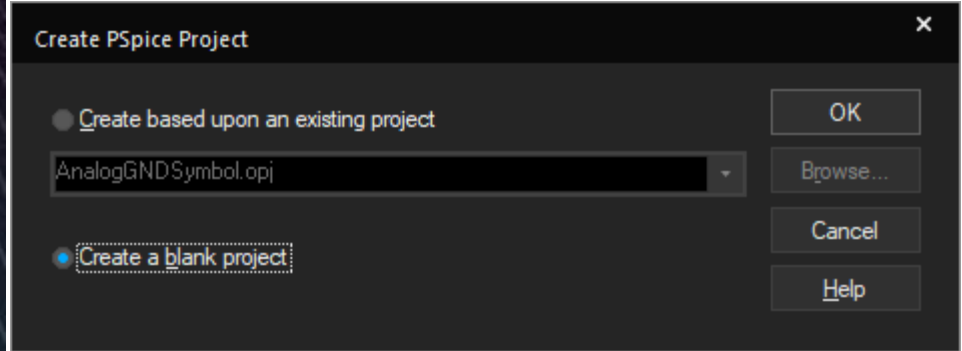
New project



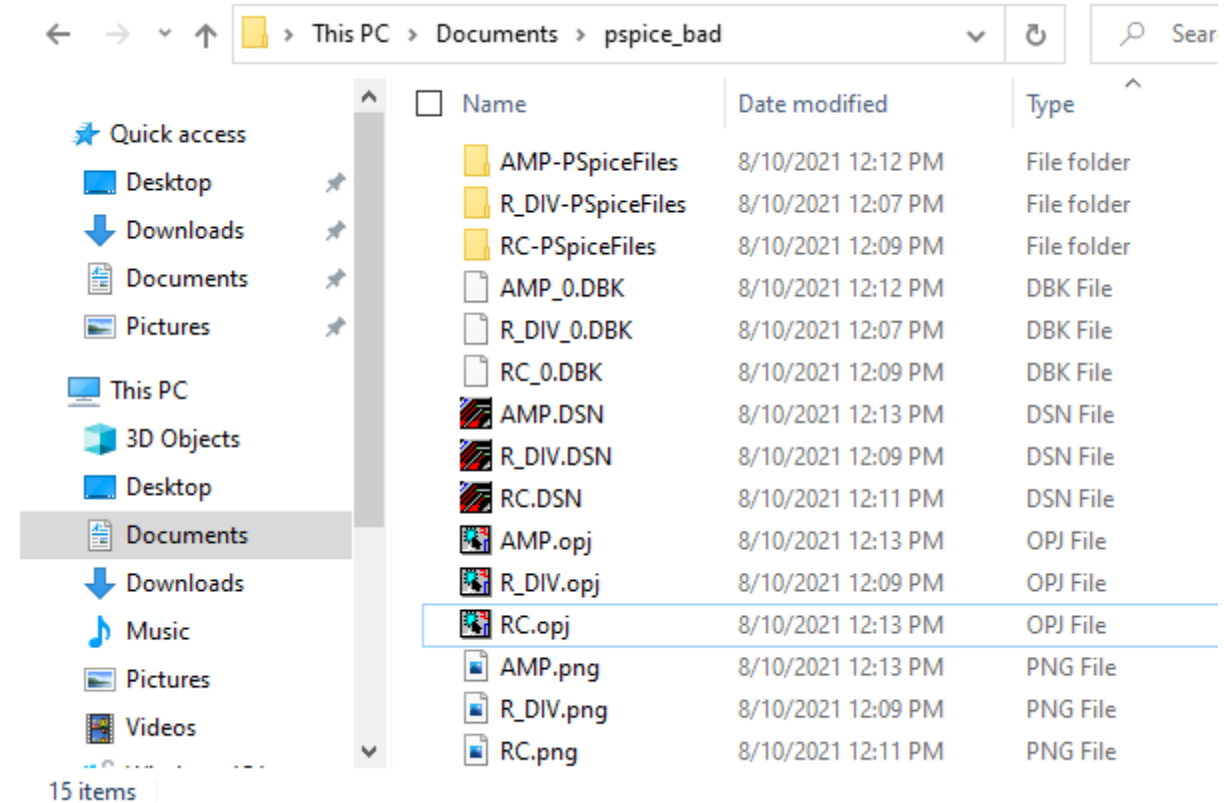
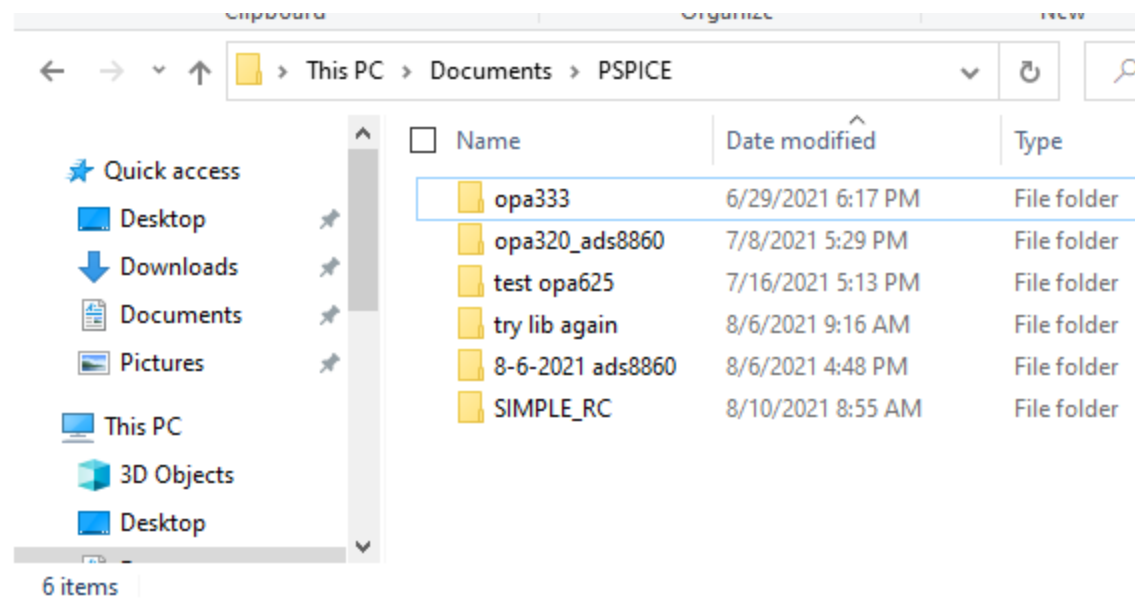
1. Select File>New>Project
2. Choose project name and location. You may want to create a unique folder for each project. The project contains a *.opj, *.png, *.dsn, and a separate folder with the net lists, simulation profiles, and other support files. If you put everything in one folder you will “mix-up” files for all your projects”
3. Select “create a blank project”



Create a unique folder for each example. Press three dots and press “New Folder” to create a folder then select it.



Bad vs Good structure



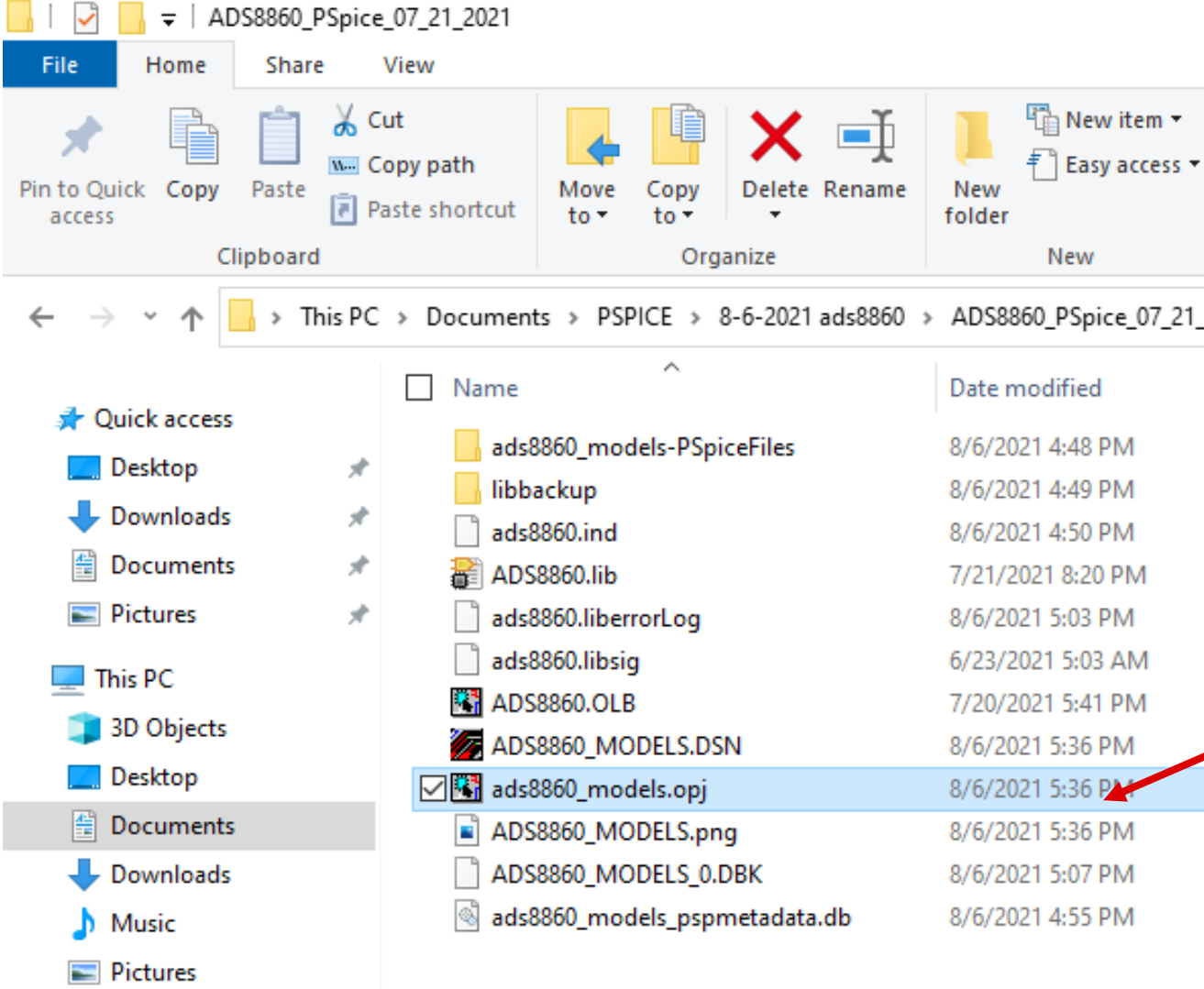
Good Structure

- Manually created new folder for each project.
- Each project folder contained in PSPICE main folder
- Files don't get mixed together

Bad Structure

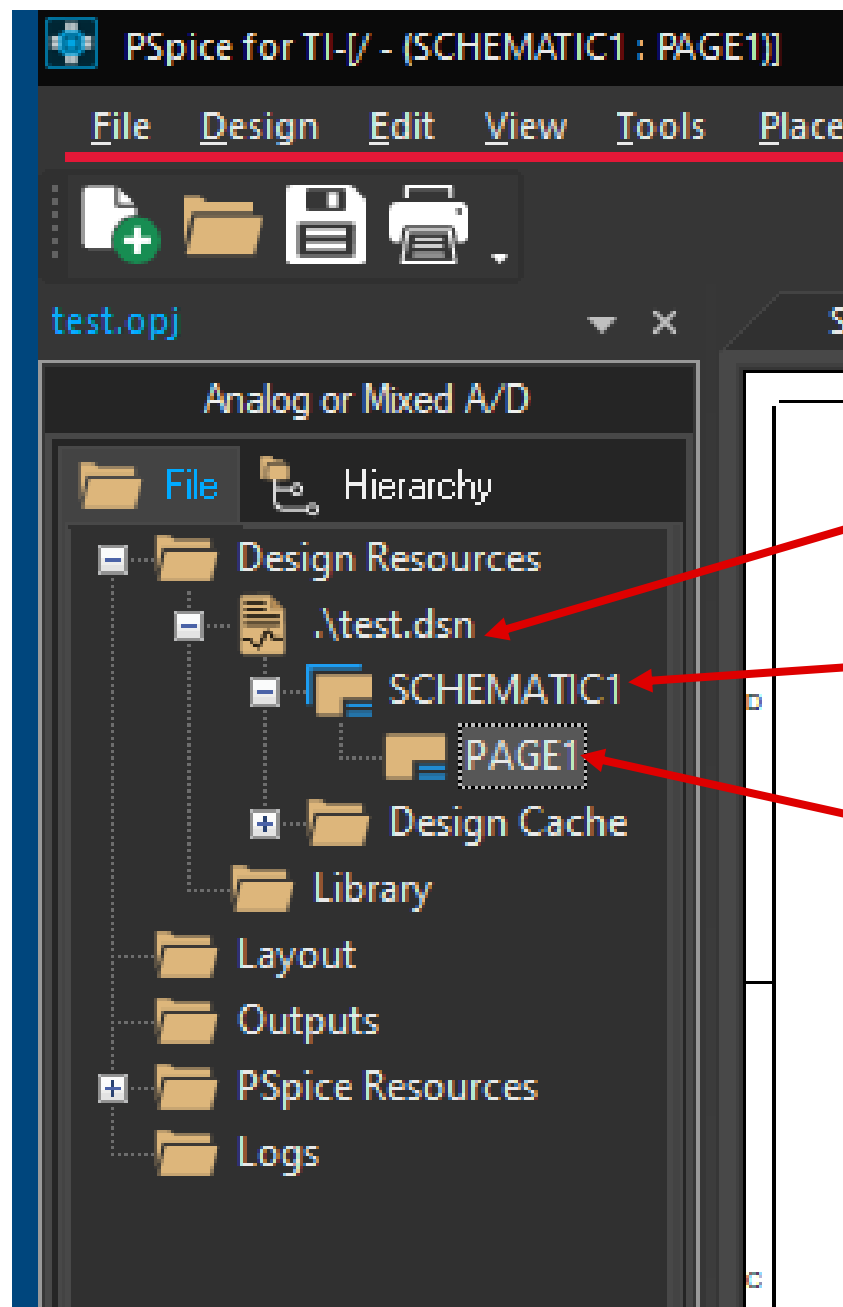
- All files mixed together
- All the PSPICE files in one folder.
- No project specific folder created automatically

Launch PSPICE from file explorer



If you already have a project and want to just launch from windows explorer, click on the OPJ file. This is the project file.

Find the Schematic

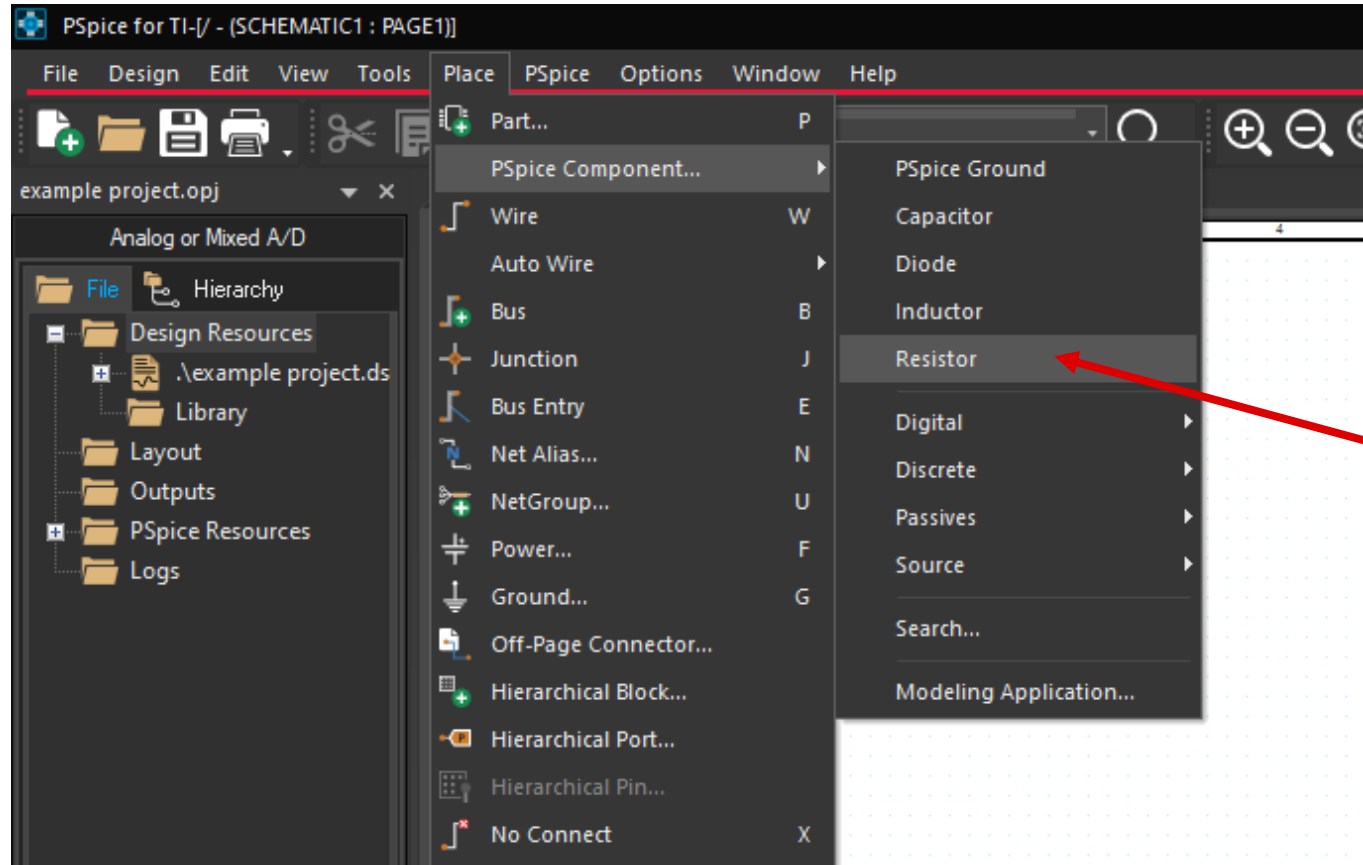


Project name

Folder containing all schematic pages. You can rename this

"PAGE1" is the editable first page of the schematic

Adding passive components



- Select resistor, capacitor inductor.

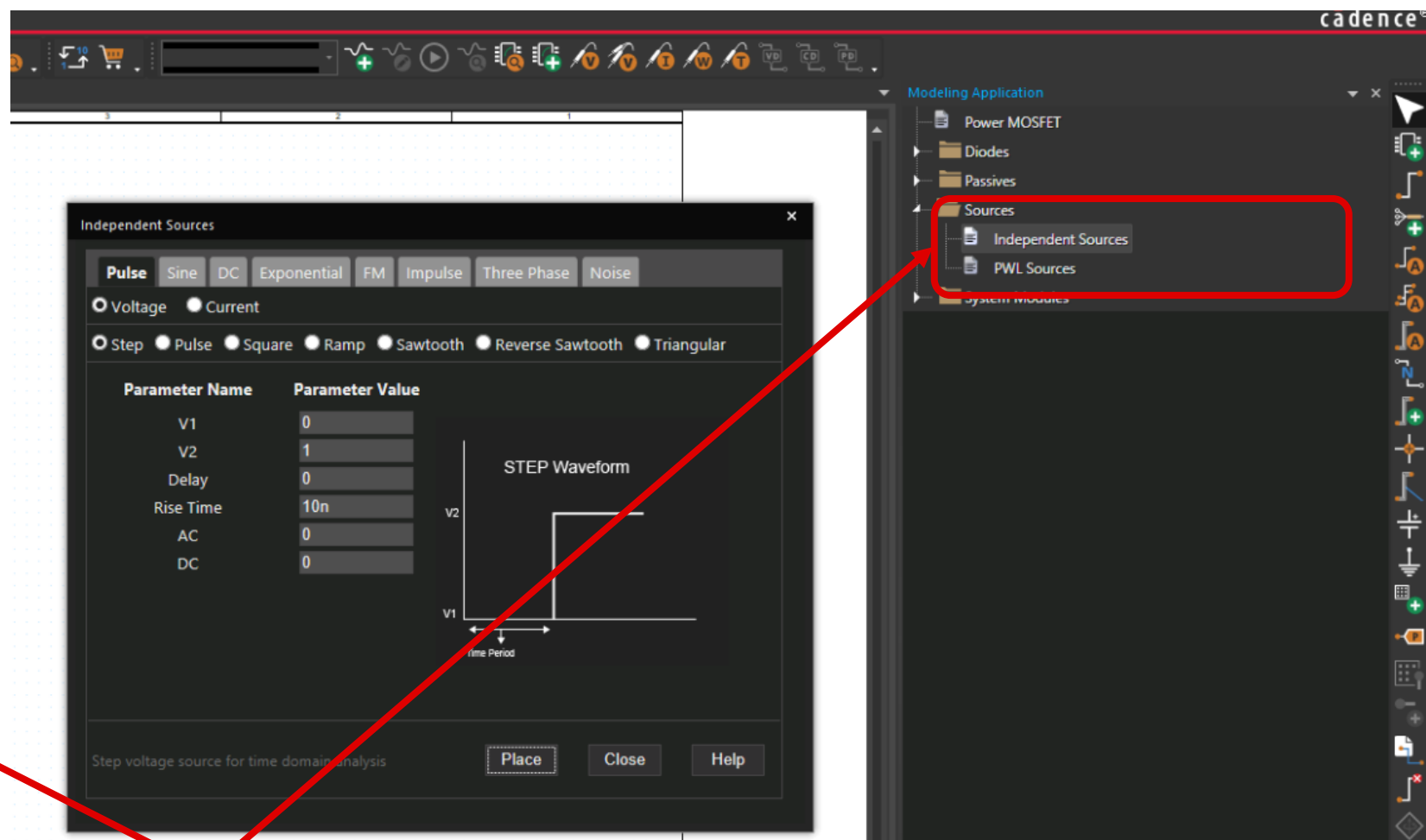
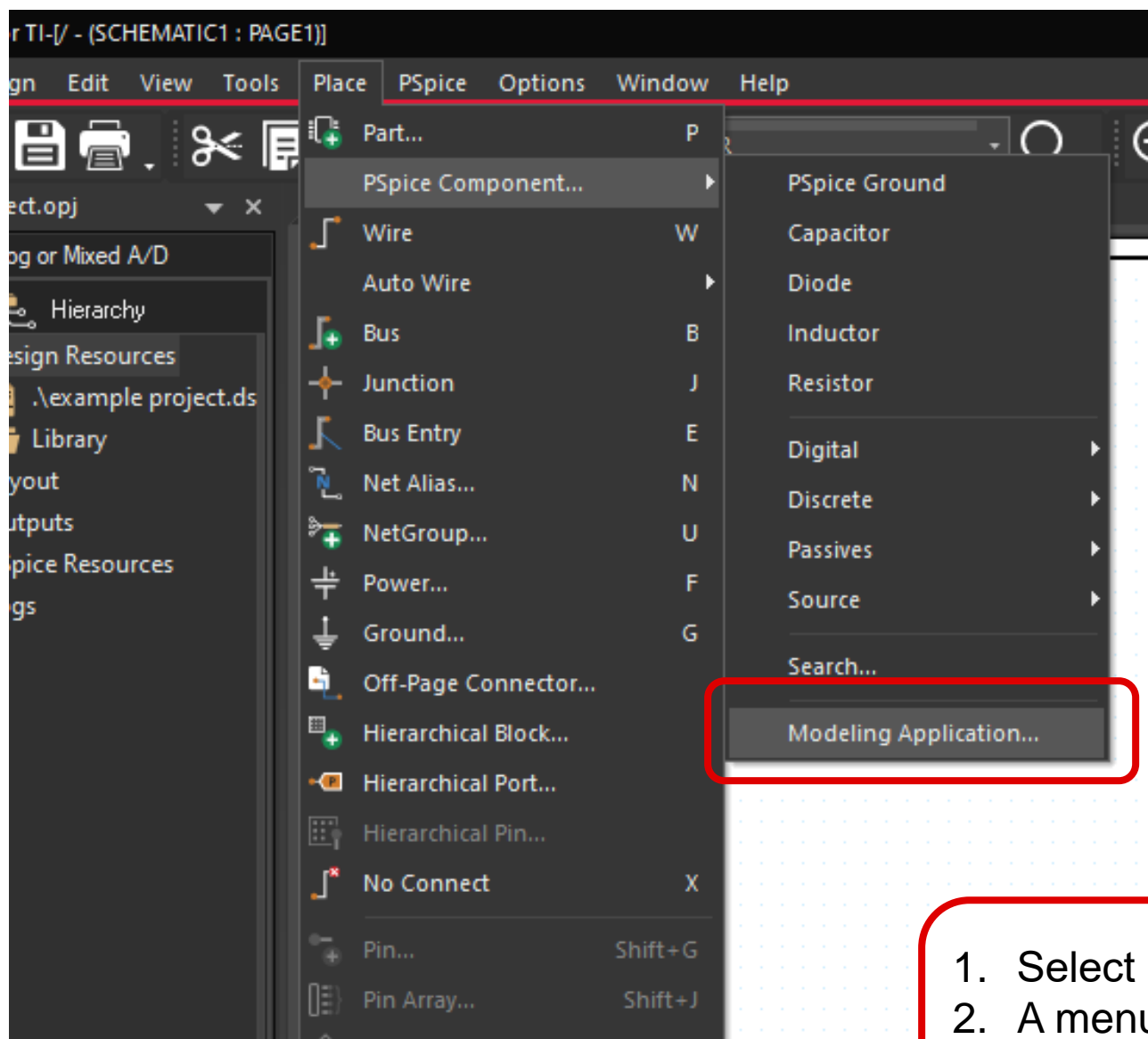
Adding GND symbol

One way to get access to a GND symbol. There are a few different locations to access this.

It is very important that you use the GND symbol with the "0" on it. Other GND symbols are really just reference points and not the 0V system reference.

Title	<Title>
Size	Document Number
A	<Doc>
Date:	Monday, August 16, 2021
Sheet	

Adding sources for transient analysis



1. Select "Modeling Applications"
2. A menu will appear at the side. Choose "independent sources" for typical sources needed for transient analysis.

Adding sources continued

The screenshot shows the 'Independent Sources' dialog box with the 'Pulse' tab selected. The 'Voltage' radio button is chosen, and the 'Square' waveform type is selected. The parameter table is as follows:

Parameter Name	Parameter Value
V1	0
V2	5
Delay	0
Rise Time	10n
Fall Time	10n
Time Period	10u
AC	0
DC	0

The waveform graph shows a square wave between levels V1 and V2 with a 'Time Period' indicated. At the bottom, it says 'Periodic square-wave voltage source for time domain analysis' and has 'Place', 'Close', and 'Help' buttons.

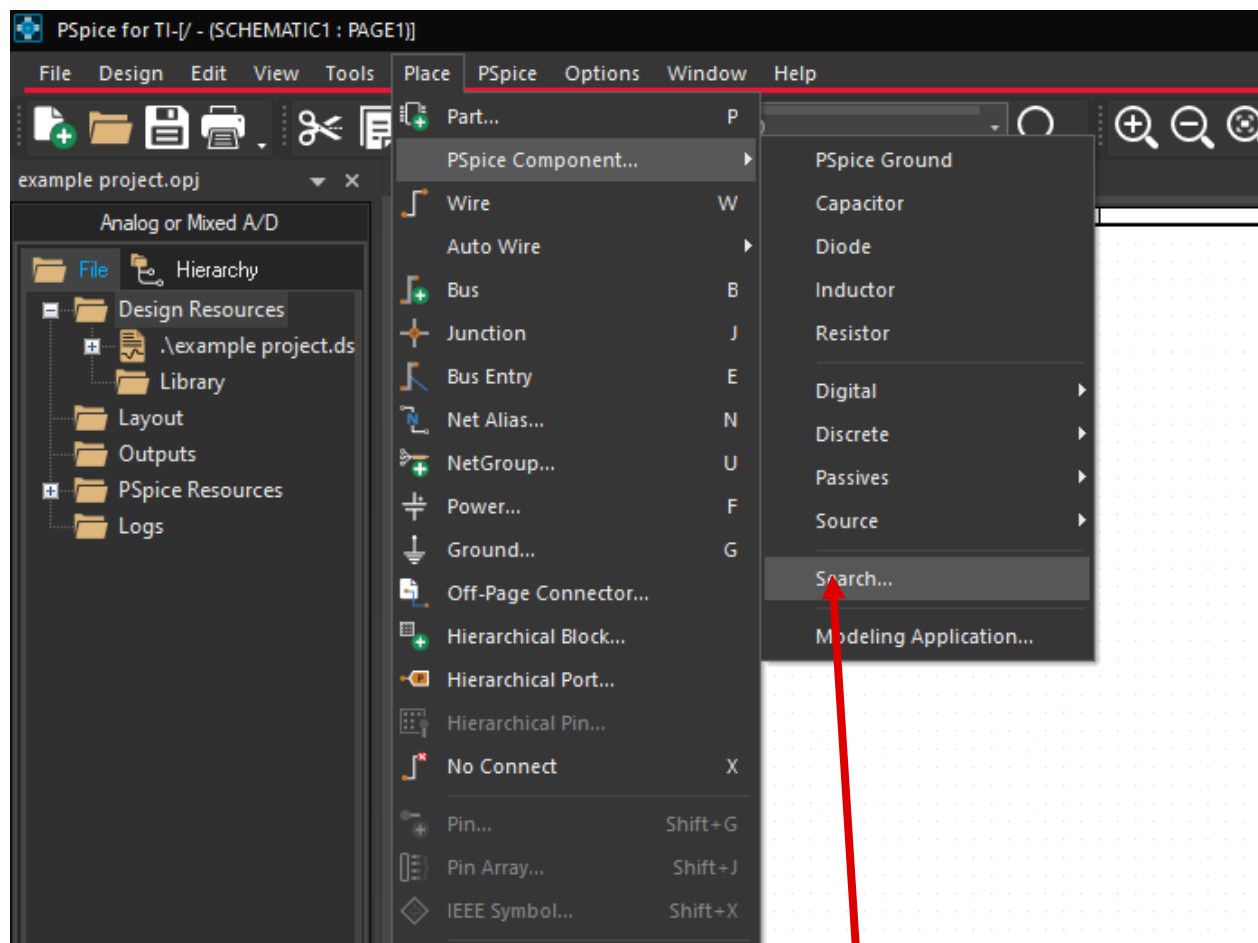
The screenshot shows the 'Independent Sources' dialog box with the 'Sine' tab selected. The 'Voltage' radio button is chosen, and the 'Sine' waveform type is selected. The parameter table is as follows:

Parameter Name	Parameter Value
Offset	0
VAMPL	154
Frequency	60
Delay	0
Phase	0
Damping Factor	0
AC	0
DC	0

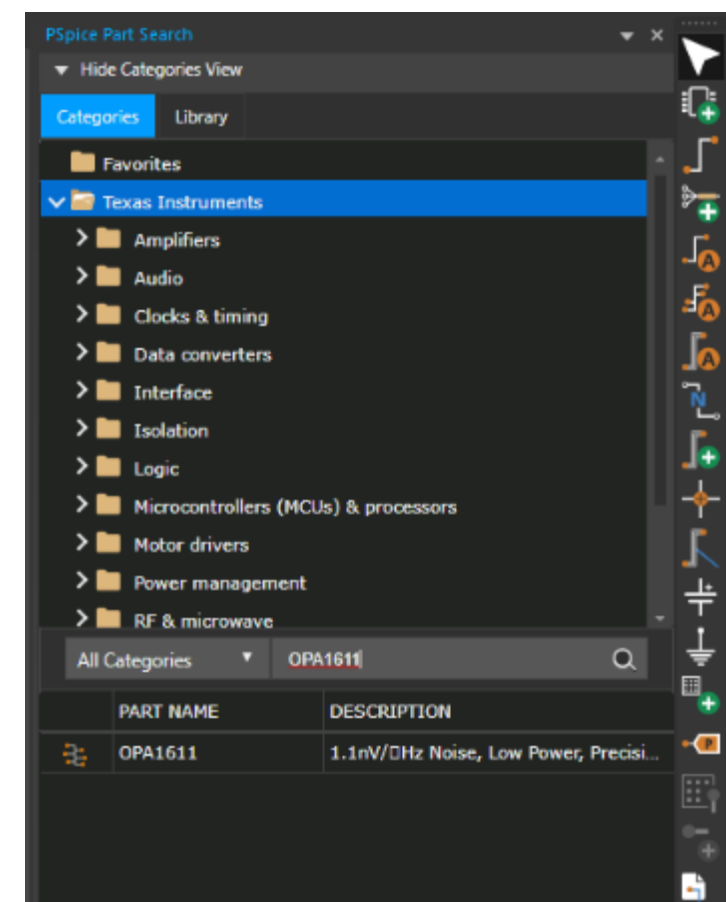
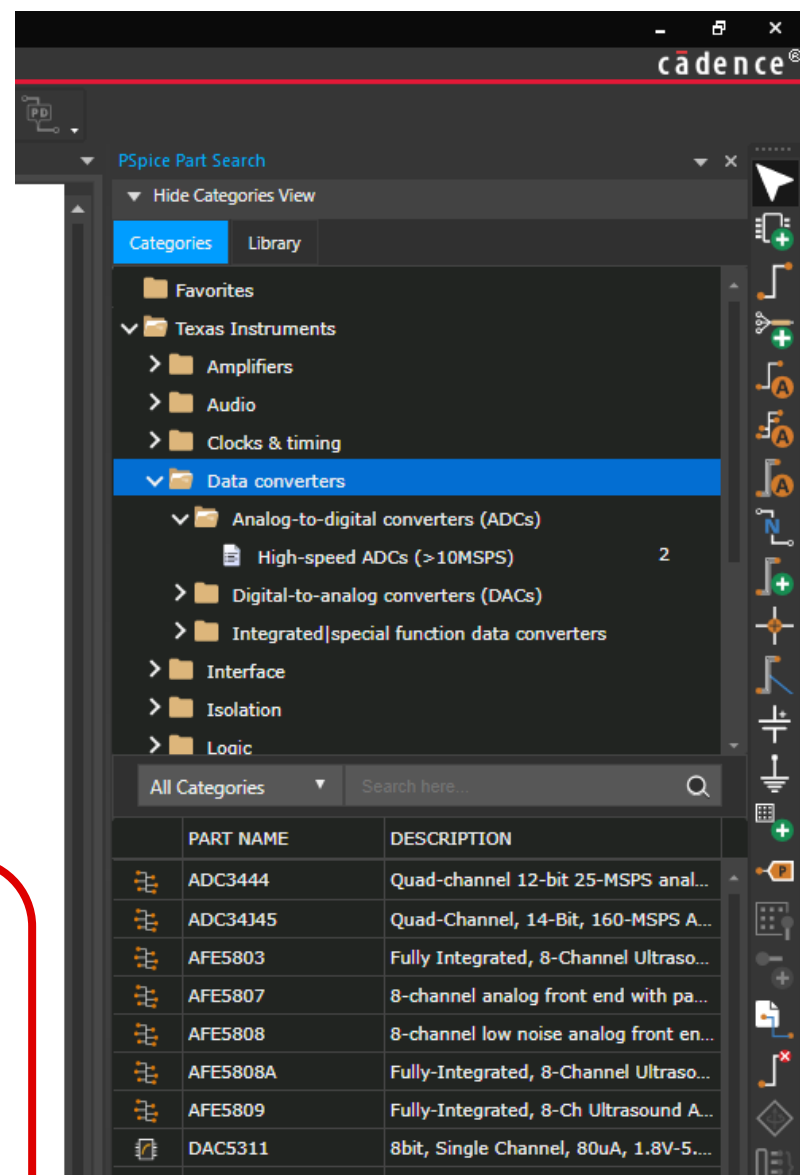
The waveform graph shows a sine wave with labels for 'VAMPL', 'Offset', and 'Phase'. At the bottom, it says 'Sine-wave voltage source for time domain analysis' and has 'Place', 'Close', and 'Help' buttons.

- Some common signal types
- Note that “pulse” includes square, ramp and other types

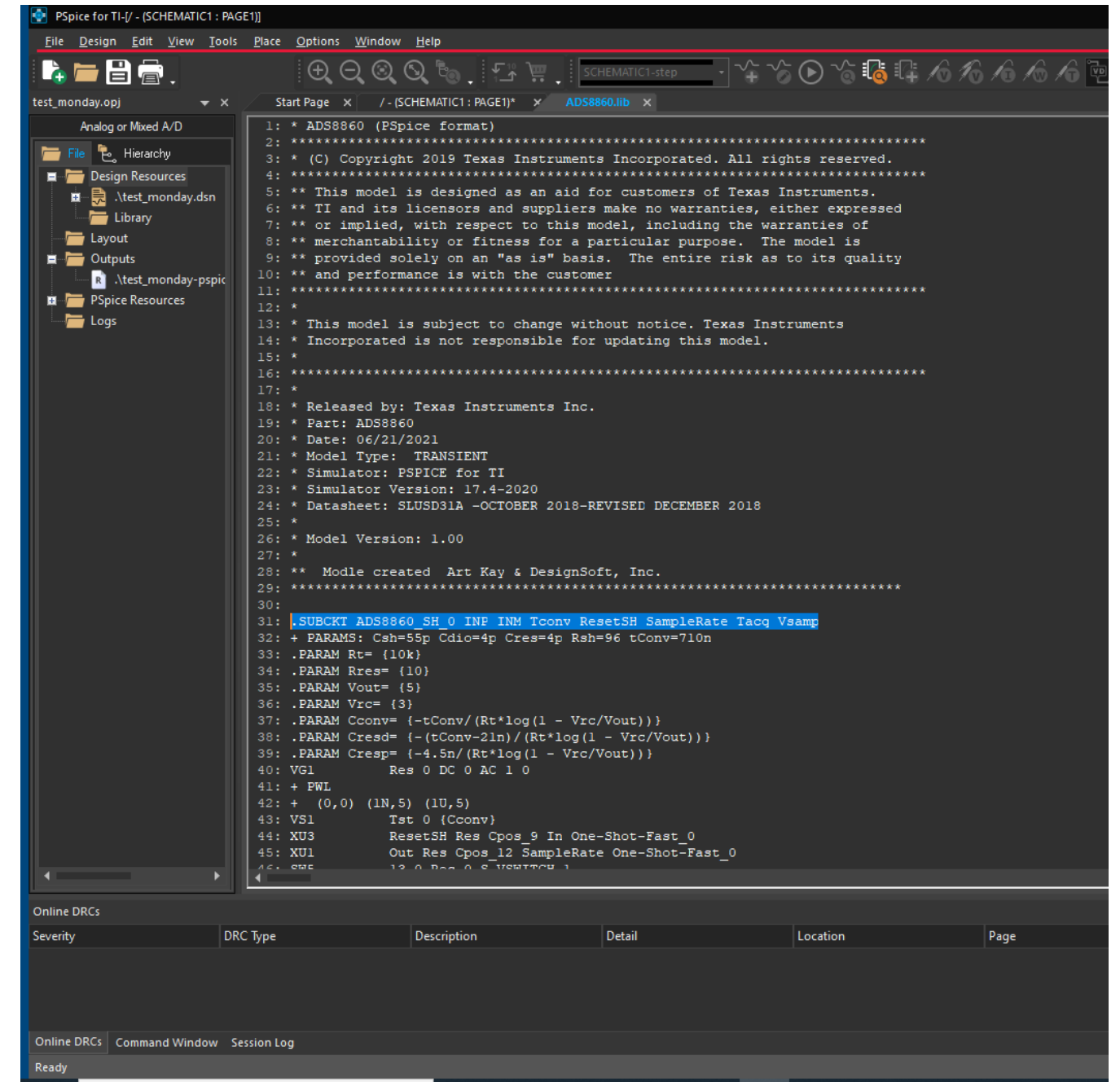
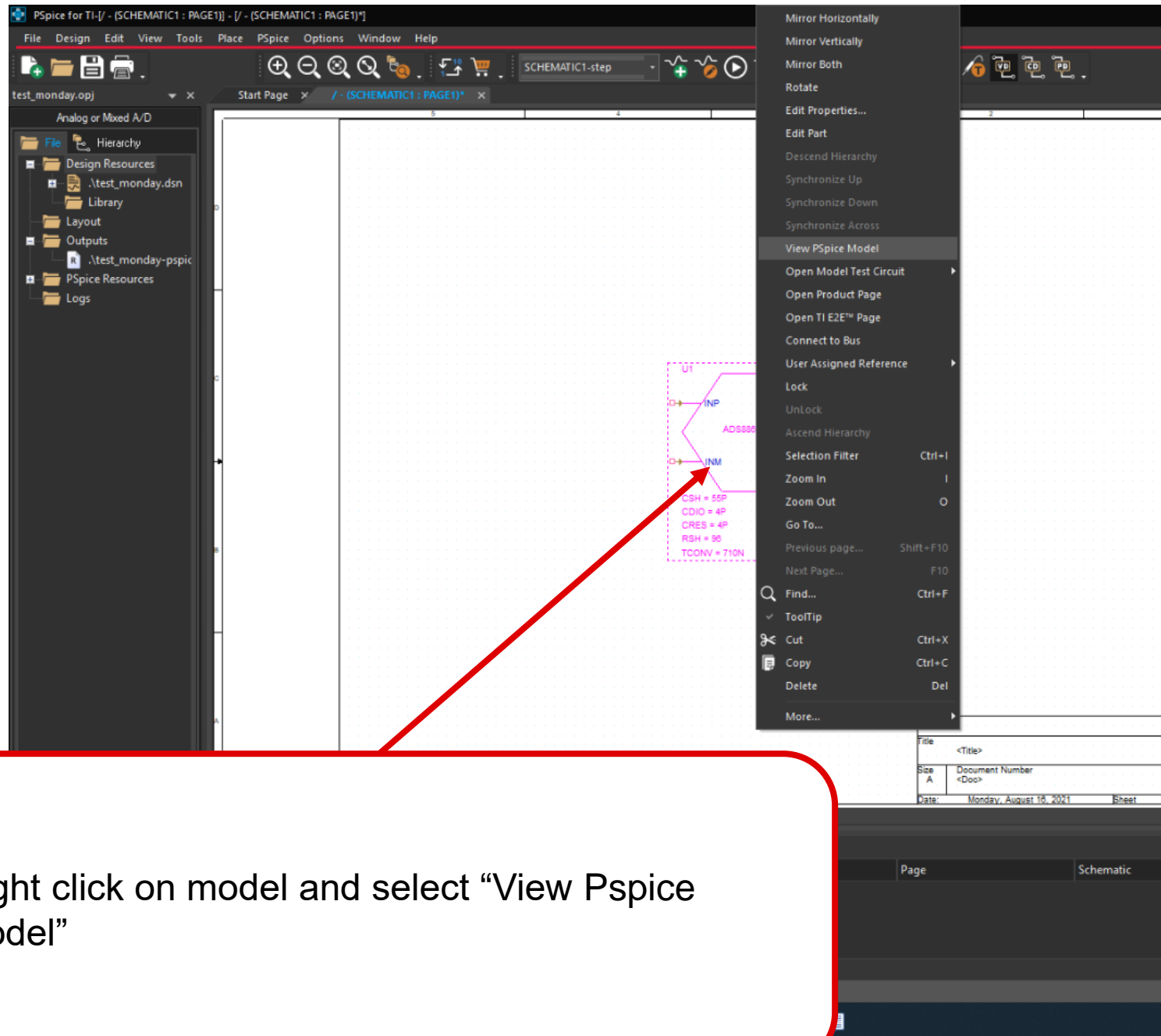
Find TI components



1. Select "Place>Pspice Component > Search"
2. A menu will appear at the side. Find "Texas instruments" and navigate to subcategory "Data converters"
3. Alternatively, search for a part in the search window.

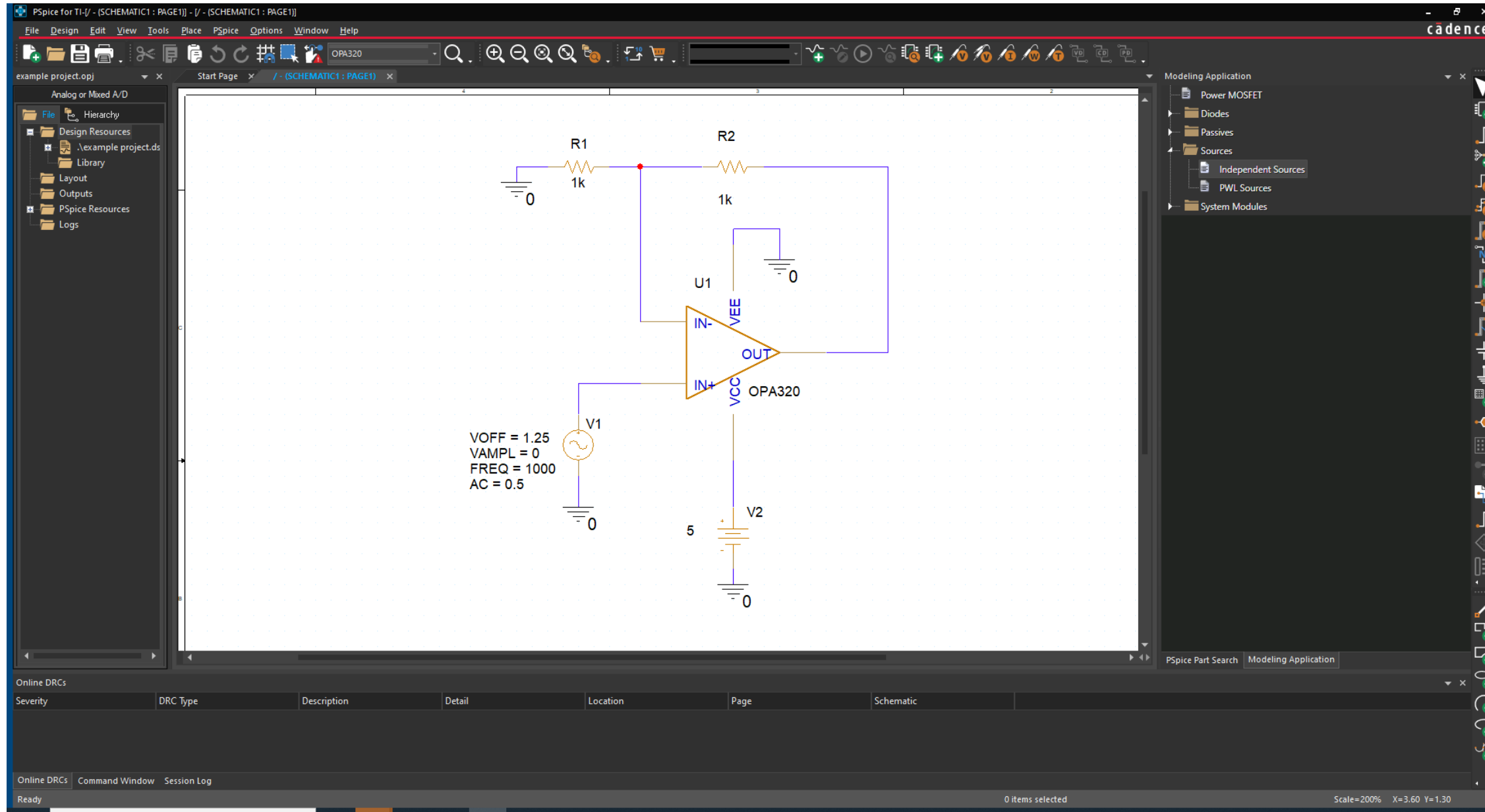


View PSPICE model

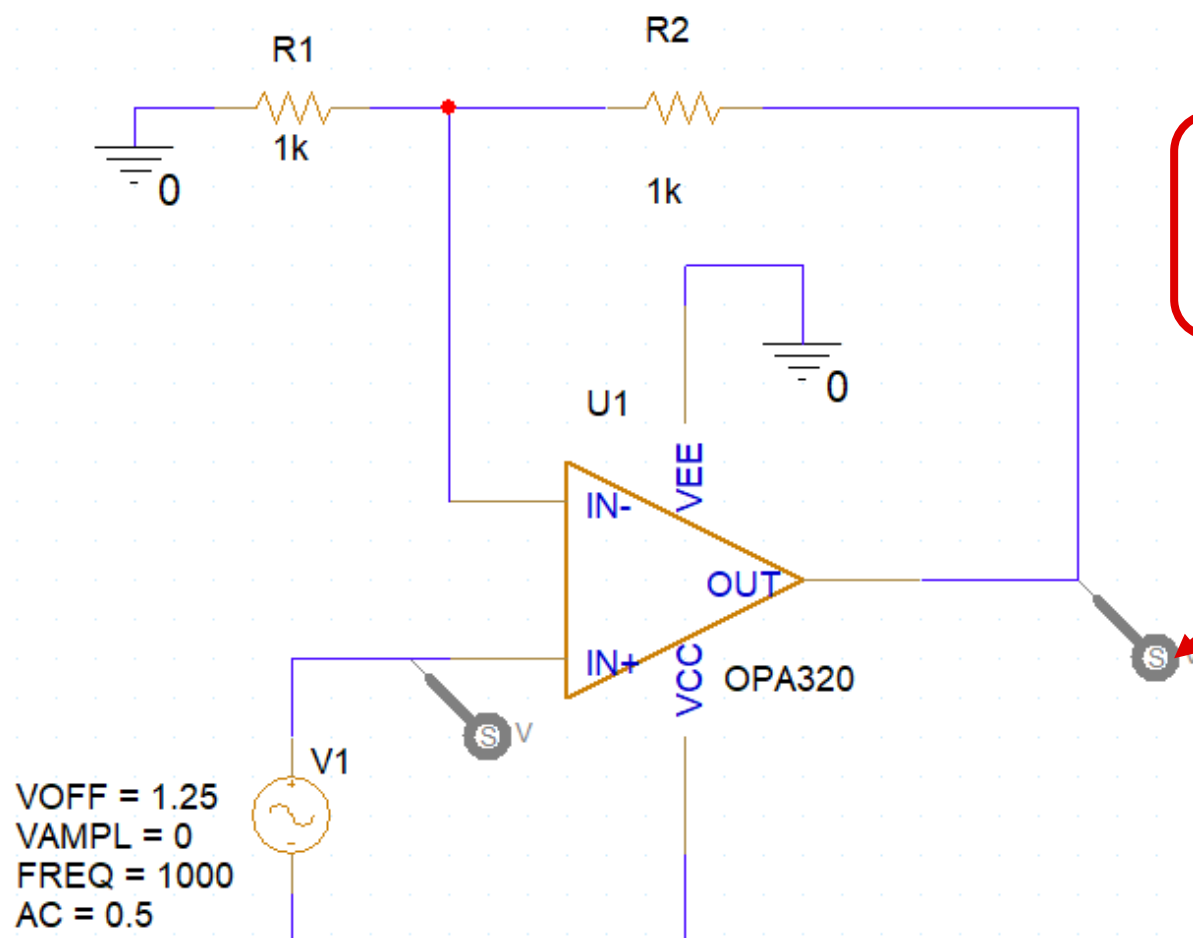
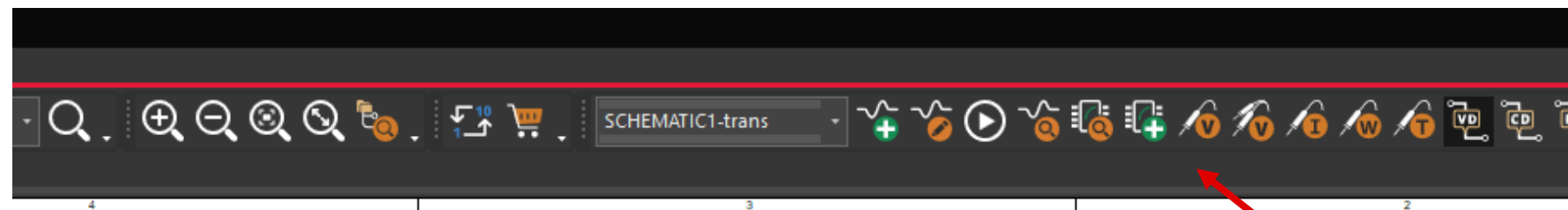


[Back to Table of Contents](#)

Use shortcut “w” to wire

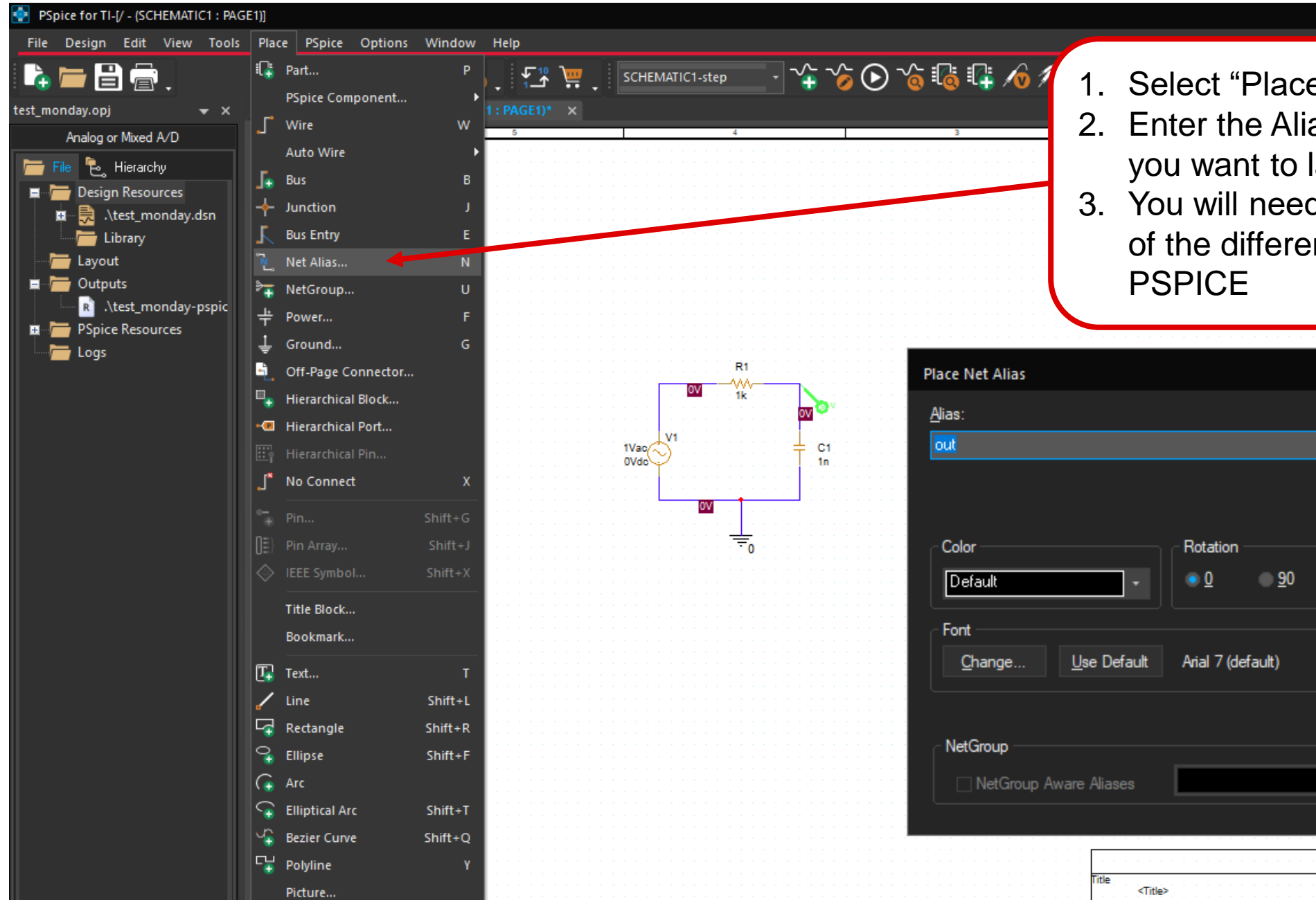


Add probes (voltage markers) to input and output

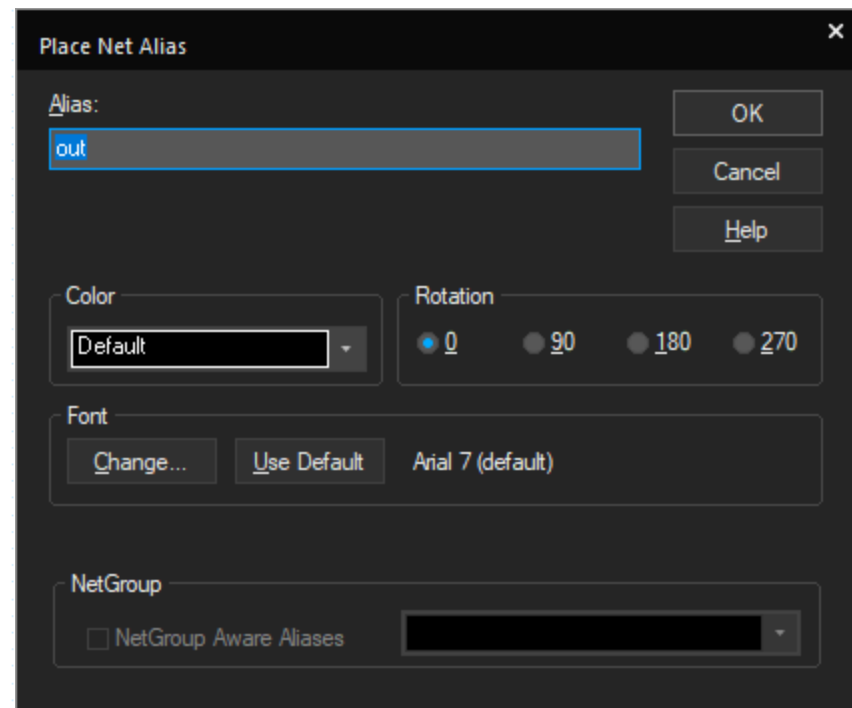


A probe will automatically generate an output plot at the selected node.

Add net alias label

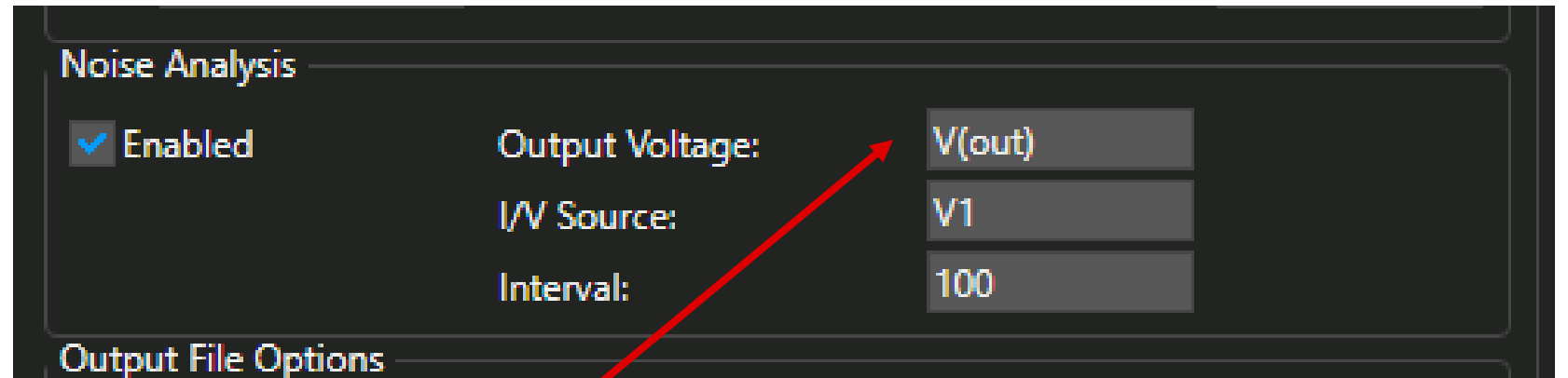
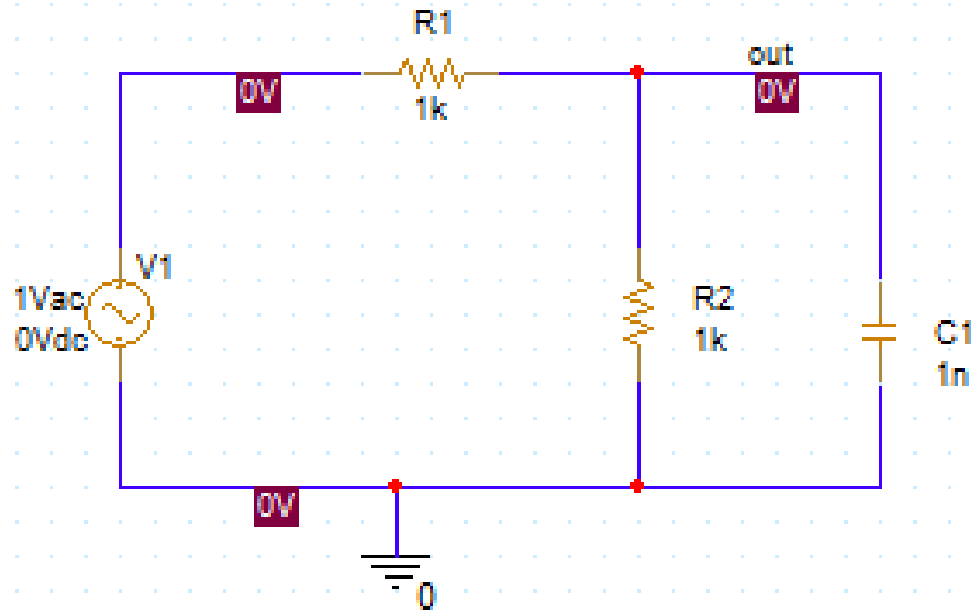


1. Select "Place > Net Alias".
2. Enter the Alias and click on the net you want to label
3. You will need a "net alias" to do many of the different simulation types in PSPICE



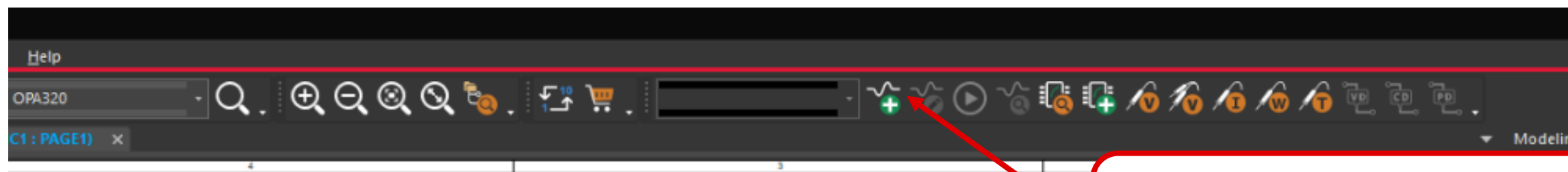
Title <Title>

The net alias is very important.

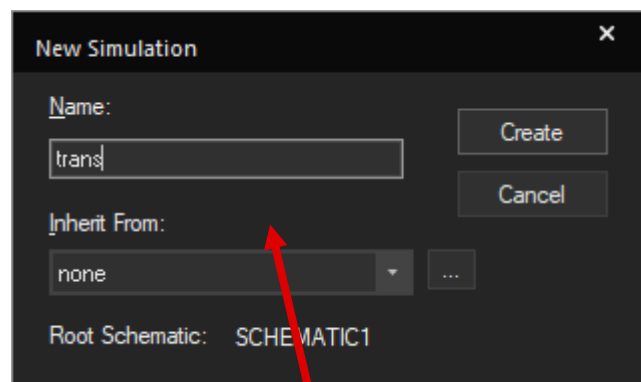


When referring to the “out” node you will use V(out), or I(out) to indicate that you want to plot voltage or current of this node.

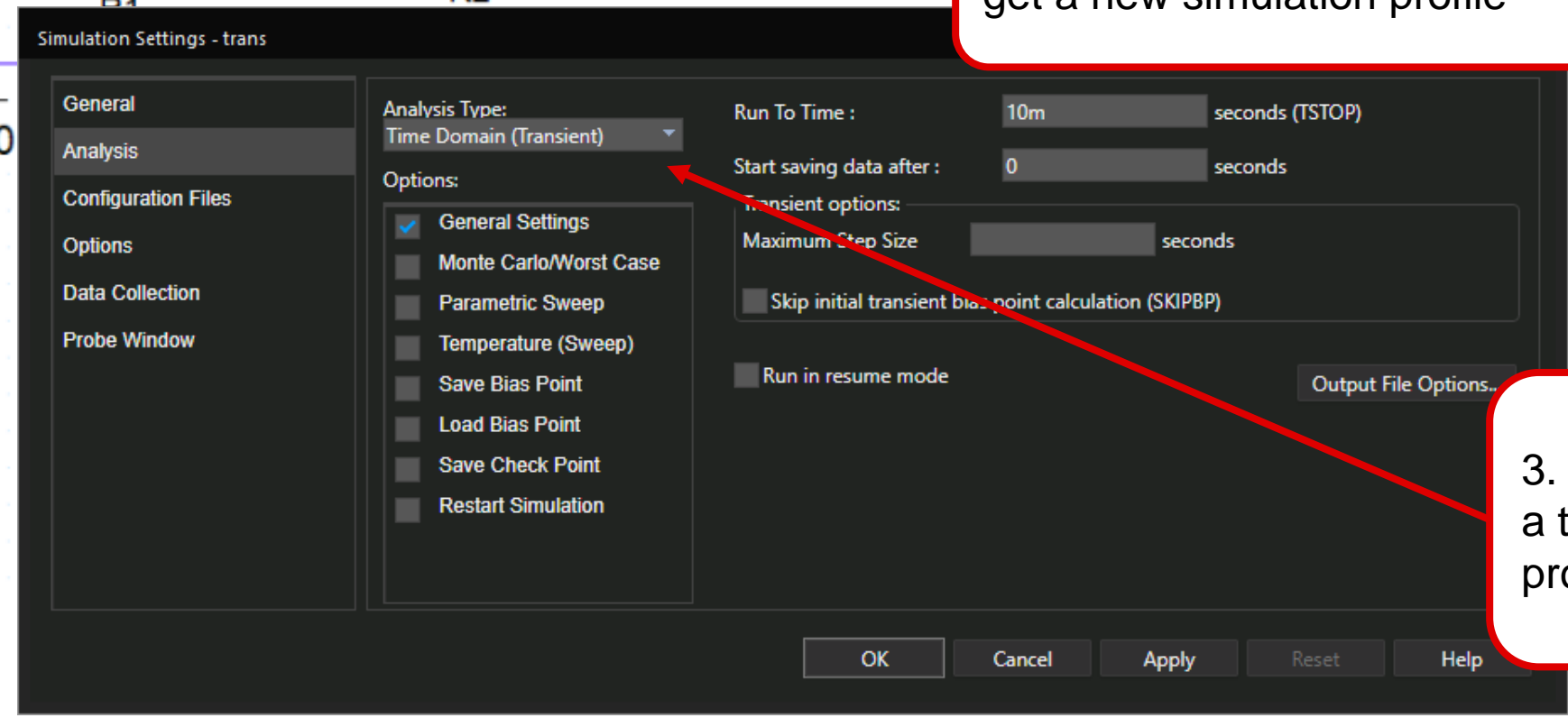
Add “simulation profile” - transient



1. Click on the green “+” sign to get a new simulation profile



2. Choose a name for the simulation profile. You can save multiple simulation profiles if you want to quickly re-do your simulations.



3. This example shows a transient simulation profile.

Simulation profiles

The image shows a screenshot of the PSpice for TI software interface. The top menu bar includes File, Design, Edit, View, Tools, Place, PSpice, Options, Window, and Help. Below the menu bar is a toolbar with various icons. A red box highlights a specific section of the toolbar containing icons for creating a new profile (a green plus sign), editing a profile (an orange pencil), running a profile (a white play button), and other simulation-related functions. A red arrow points from this highlighted section to a larger, detailed view of the same icons, which is also enclosed in a red box. Below this detailed view are three callout boxes with blue borders and white text: 'New simulation profile' (pointing to the green plus icon), 'Edit currently selected existing profile.' (pointing to the orange pencil icon), and 'Run currently selected profile.' (pointing to the white play button icon). On the left side of the interface, a file tree is visible under the 'Simulation Profiles' folder, listing profiles such as 'AC_Noise-AC_N', 'Transient-Transi', and 'DC_Sweep-DC_S'. A blue arrow points from the 'Simulation profiles' callout box to the 'Simulation Profiles' folder in the file tree.

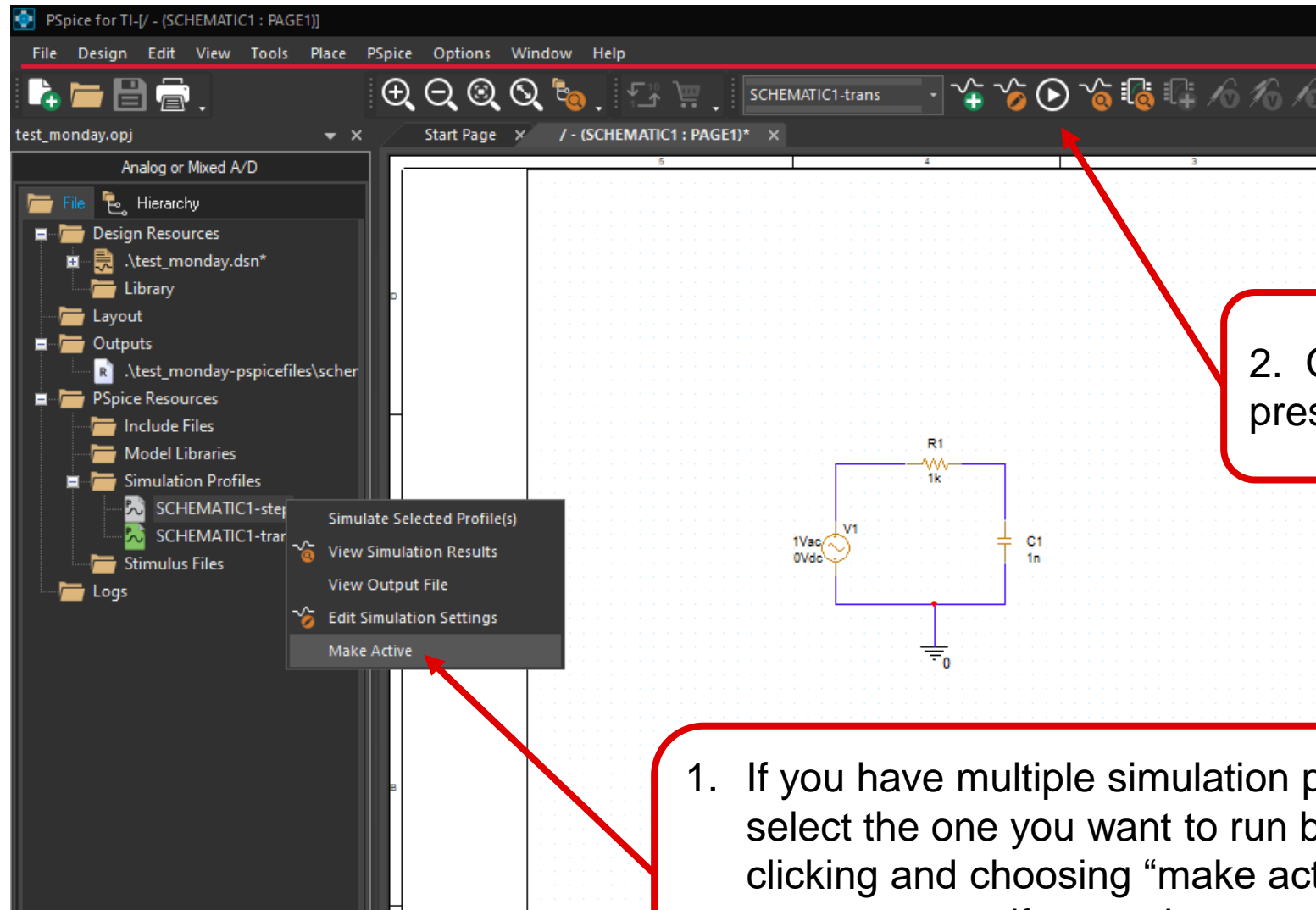
New simulation profile

Edit currently selected existing profile.

Run currently selected profile.

Simulation profiles

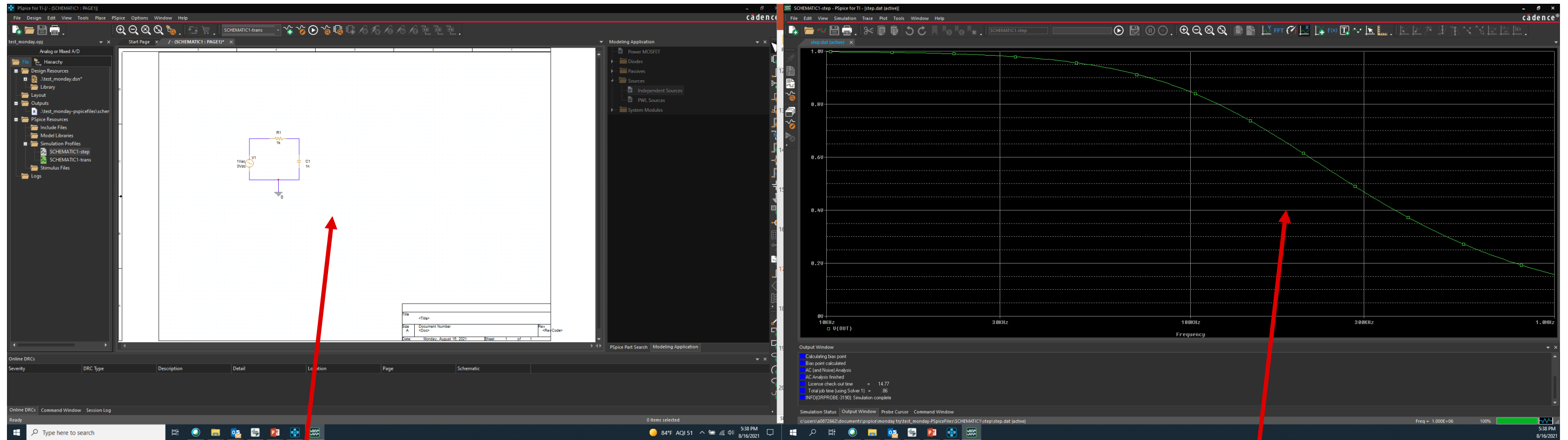
Running simulation



2. Once your simulation profile is selected press the run button 

1. If you have multiple simulation profiles, select the one you want to run by right clicking and choosing "make active". This is not necessary if you only use one simulation profile.

Results



Schematic entry and project control on one screen



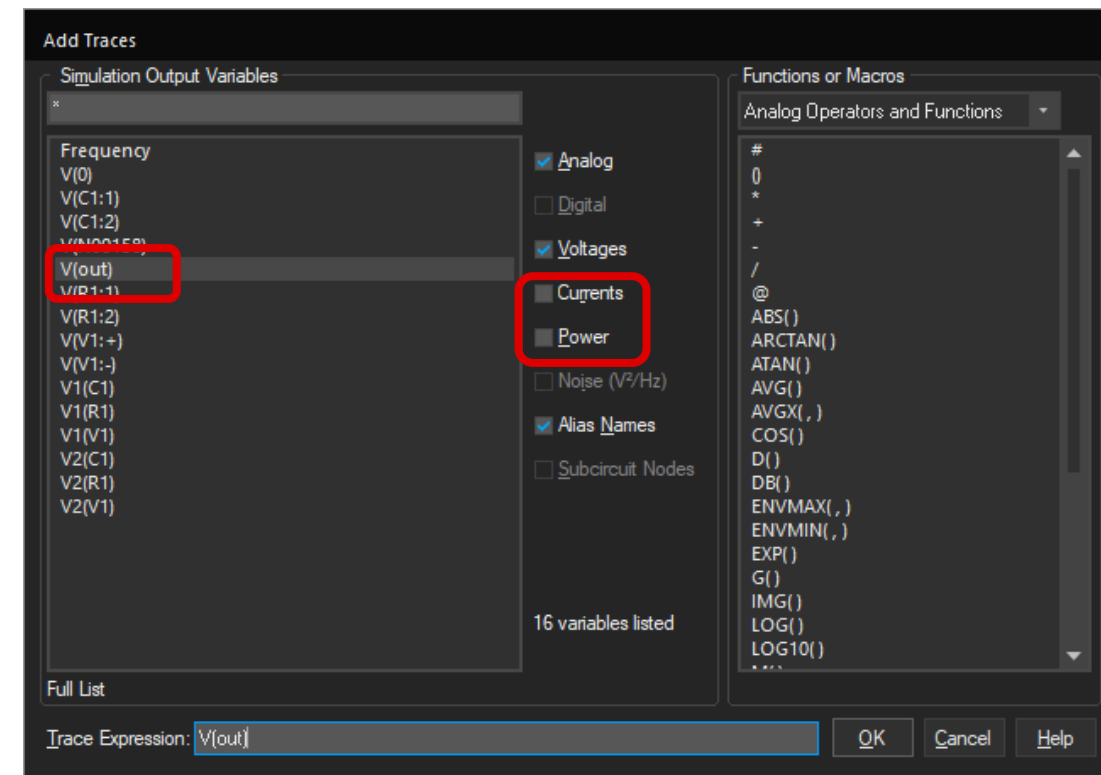
Simulation results on the other screen.

Adding plots and curves

1. Right click on the blank area to left of y-axis and select "add plot"

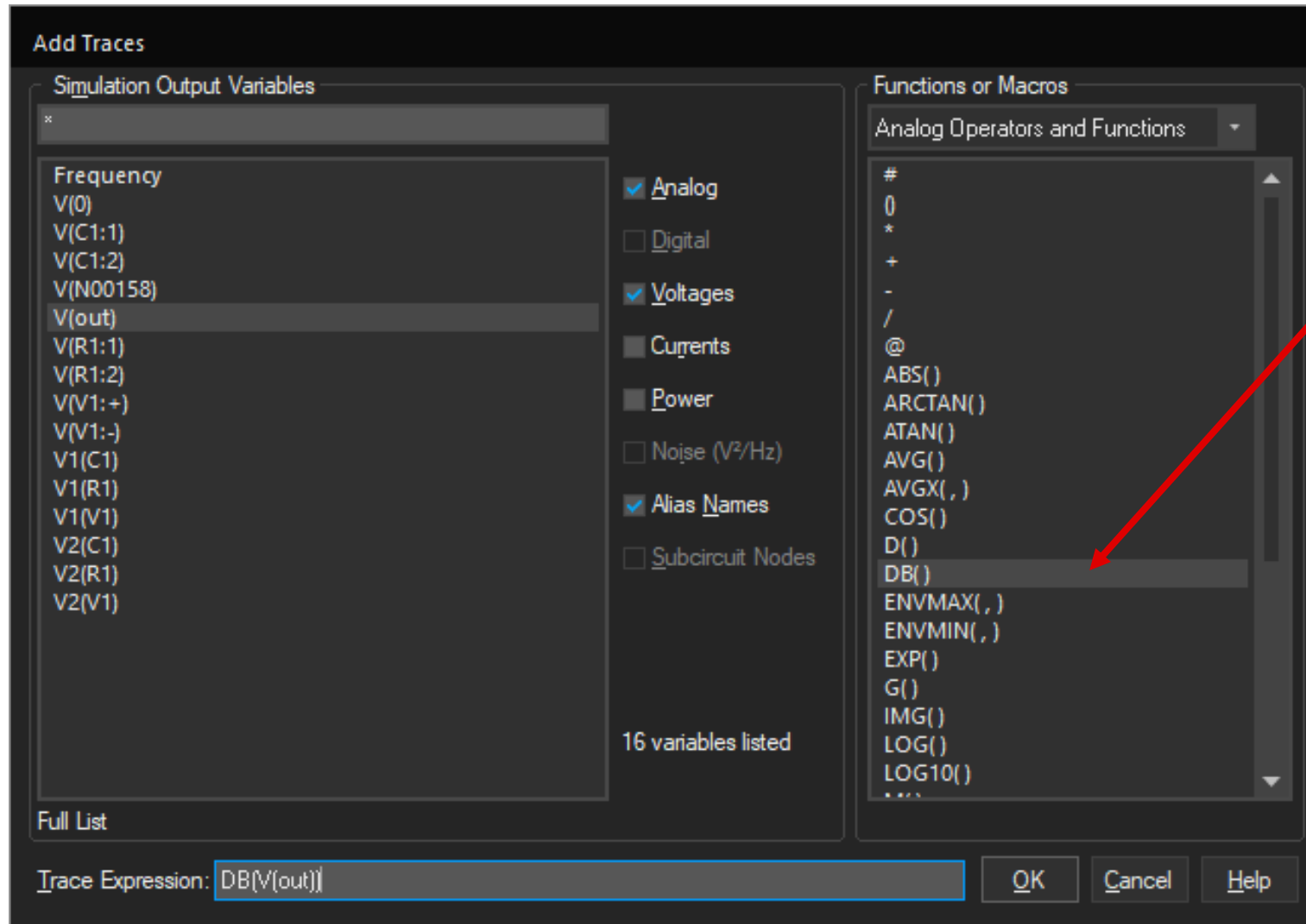
2. Right click on the blank plot area and select "Add trace".

3. Select the trace you want to plot. You may want to uncheck some categories like current and power to make the list easier to sort through.



4. Press "ok" and the trace will be added to the selected plot.

Post processing



You may want to perform a mathematical operation on the waveform. You can select the math function here. DB() converts the voltage plot to decibels.

1. First press the math operation
2. Second select the signal
3. In this example: DB(V(out)) will plot V(out) in decibels.

Post processing functions

ABS(x) $|x|$
SGN(x) +1 (if $x > 0$), 0 (if $x = 0$), -1 (if $x < 0$)
SQRT(x) $x^{1/2}$
EXP(x) e^x
LOG(x) $\ln(x)$ (log base e)
LOG10(x) $\log(x)$ (log base 10)
M(x) magnitude of x
P(x) phase of x (result of degrees)
R(x) real part of x
IMG(x) imaginary part of x
G(x) group delay of x (result in seconds)
PWR(x,y) x^y
SIN(x) $\sin(x)$ (x in radians)
COS(x) $\cos(x)$ (x in radians)
TAN(x) $\tan(x)$ (x in radians)
ATAN(x) $\tan^{-1}(x)$ (result in radians)
ARCTAN(x) $\tan^{-1}(x)$ (result in radians)
d(x) derivative of x with respect to the X axis variable.
s(x) integral of x over the range of the X axis variable.
AVG(x) running average of x over the range of the X axis variable.
AVGX(x,d) running average of X (from $x-d$ to x) over the range of the X axis variable.
RMS(x) running RMS average of x over the range of the X axis variable.
DB(x) magnitude in decibels of x .
MIN(x) minimum of the real part of x .
MAX(x) maximum of the real part of x .

Using cursors

1. Press "toggle cursor"

2. Right click mouse for green cursor and left click for red cursor.

3. Cursor x/y values and delta cursor given here.

Trace Color	Trace Name	Y1	Y2	Y1 - Y2	Y1(Cursor1) - Y2(Cursor2)	Max Y	Min Y	Avg Y
	X Values	505.049K	147.156K	357.893K				
	V(OUT)	300.569m	734.233m	-433.664m	10.742	734.233m	300.569m	517.401m
	CURSOR 1,2	-10.442	-2.6834	-7.7581	0.000	-2.6834	-10.442	-6.5625

Adding Measurements

1. Press "toggle measurement result window"

3. Choose measurement (e.g. Bandwidth, Max, Min, rise time...)
4. Select output variable

3. Results displayed here.

Evaluate	Measurement	Value
<input checked="" type="checkbox"/>	Max(V(ONoise))	4.07133n
<input checked="" type="checkbox"/>	Min(V(ONoise))	3.44733n

Remember plot window setup in simulation profile

The screenshot shows the PSpice interface with a plot window displaying a signal. The plot has a y-axis ranging from 2.7nV to 2.9nV. A red box highlights the 'Probe' icon in the left sidebar. A red arrow points from this icon to a text box containing instructions. Another red arrow points from the text box to the 'Probe Window' tab in the 'Simulation Settings' dialog. The 'Probe Window' tab is selected, and the 'Last Plot' option is chosen under the 'Show' section.

Simulation Settings - AC_Noise

- General
- Analysis
- Configuration Files
- Options
- Data Collection
- Probe Window

Display Probe window when profile is opened

Display Probe window:

- during Simulation.
- after simulation has been completed.

Show

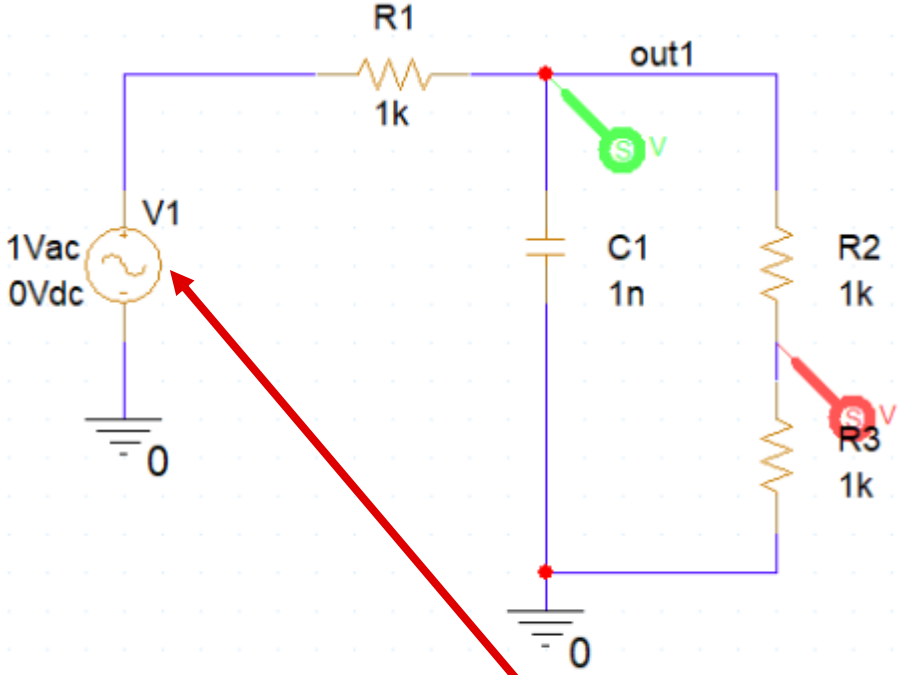
- All Markers on open schematics.
- Last Plot
- Nothing.

OK Cancel Apply Reset Help

This is very helpful to retain all the different plots, and calculations in a simulation profile.

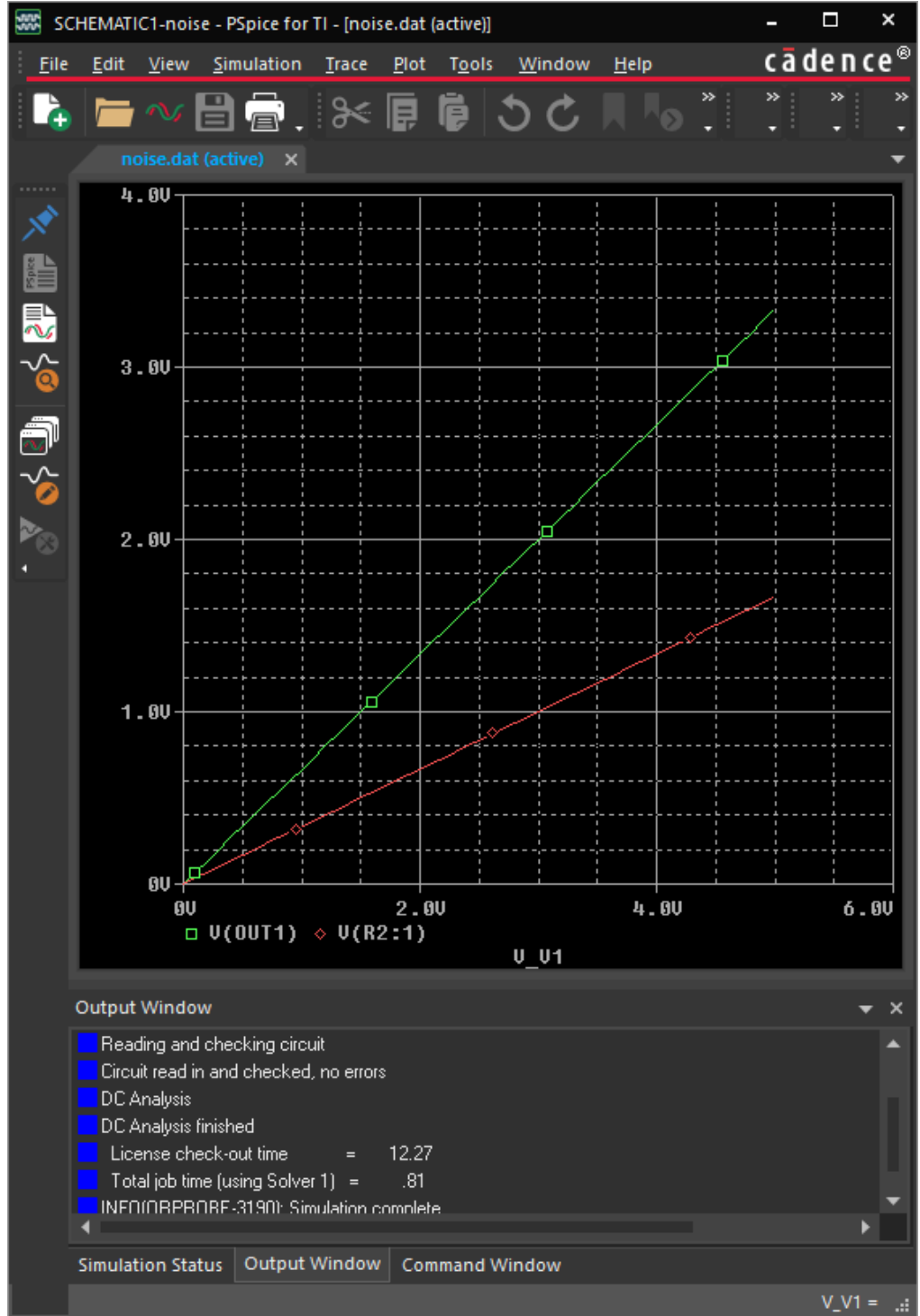
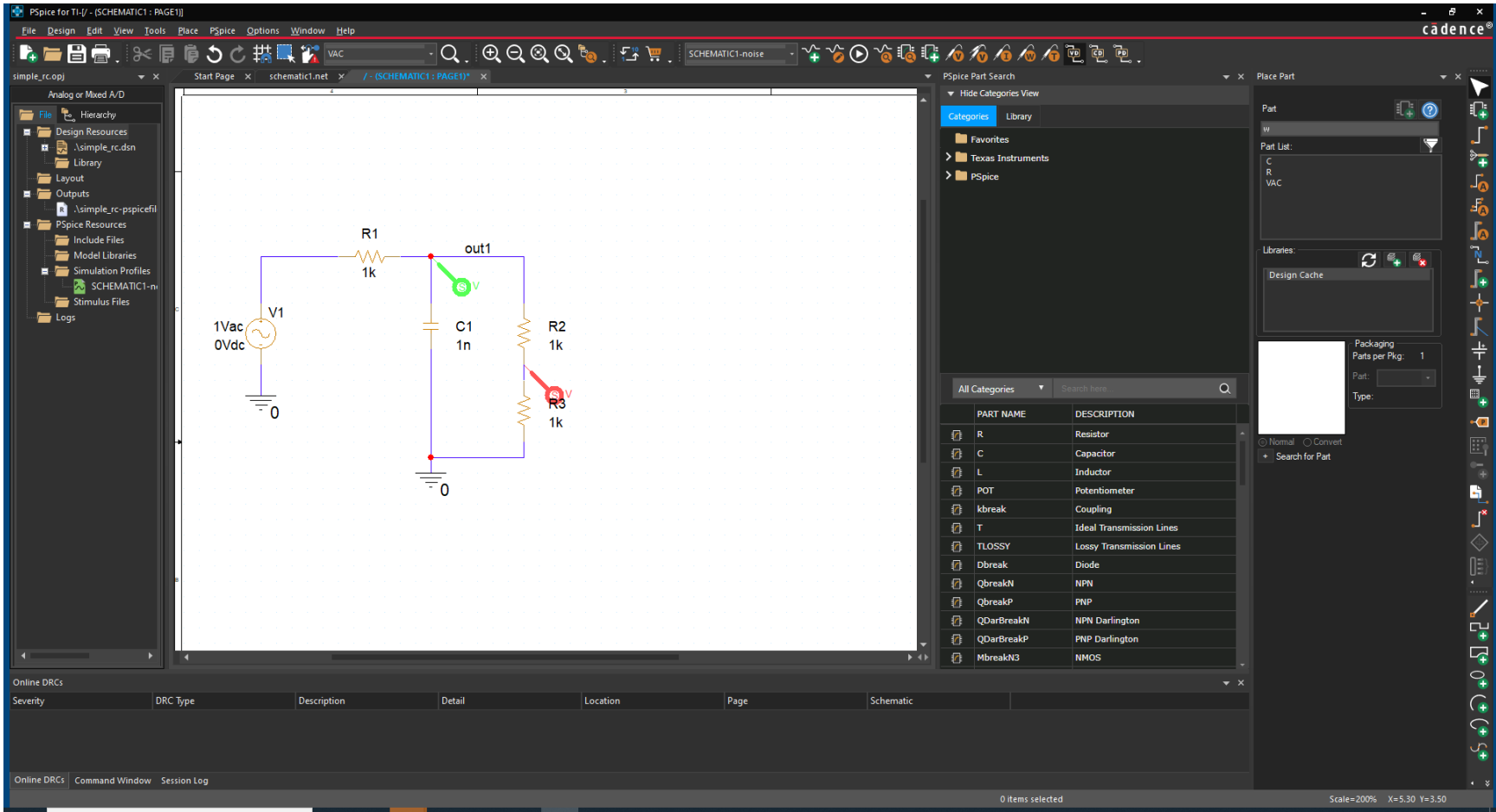
1. Click on edit simulation profile button.
2. Select probe window
3. Choose "last plot"

DC sweep



For DC sweep: specify the voltage source you want to sweep (V1 in this example). Specify the sweep type and range.

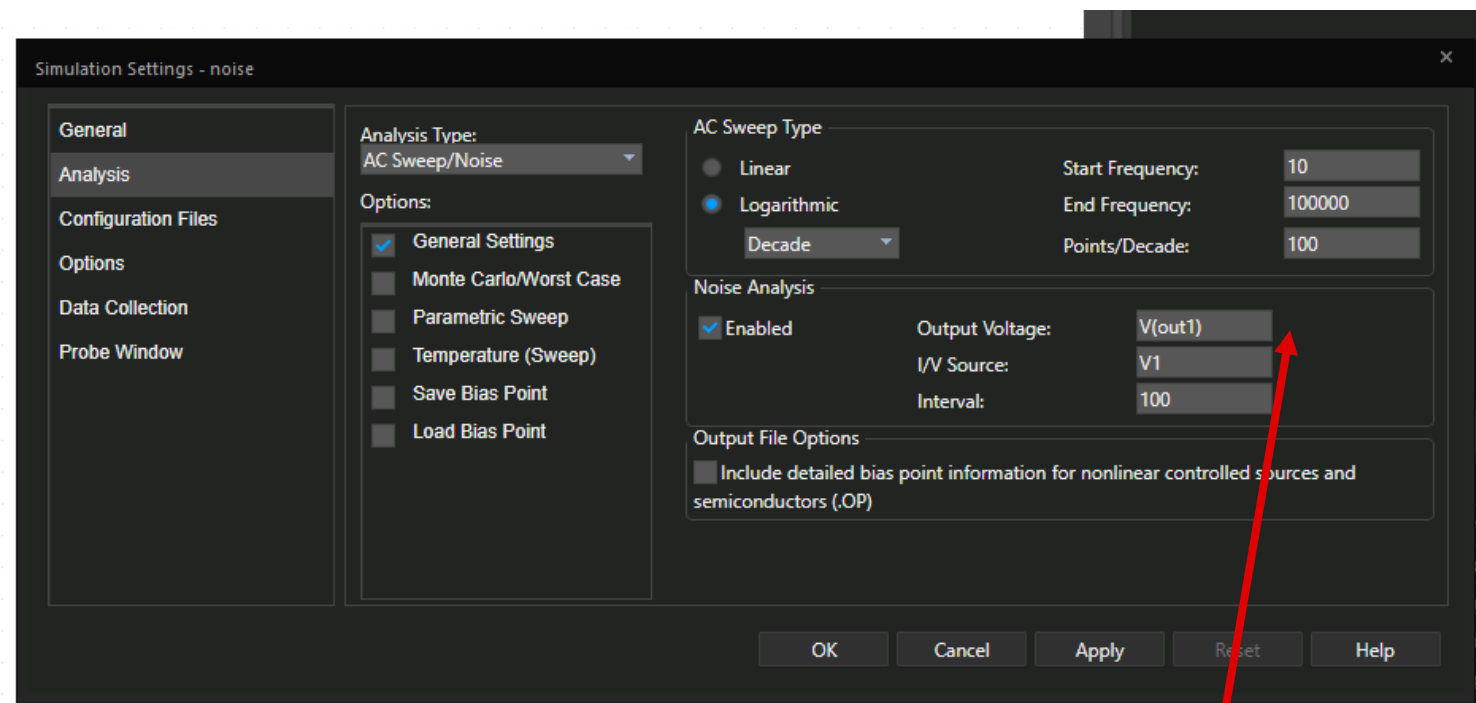
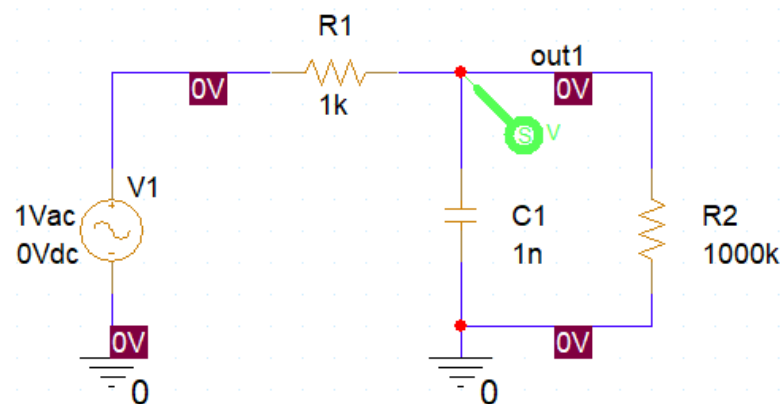
DC Sweep



This shows the example results for DC sweep where two probes measure the outputs.

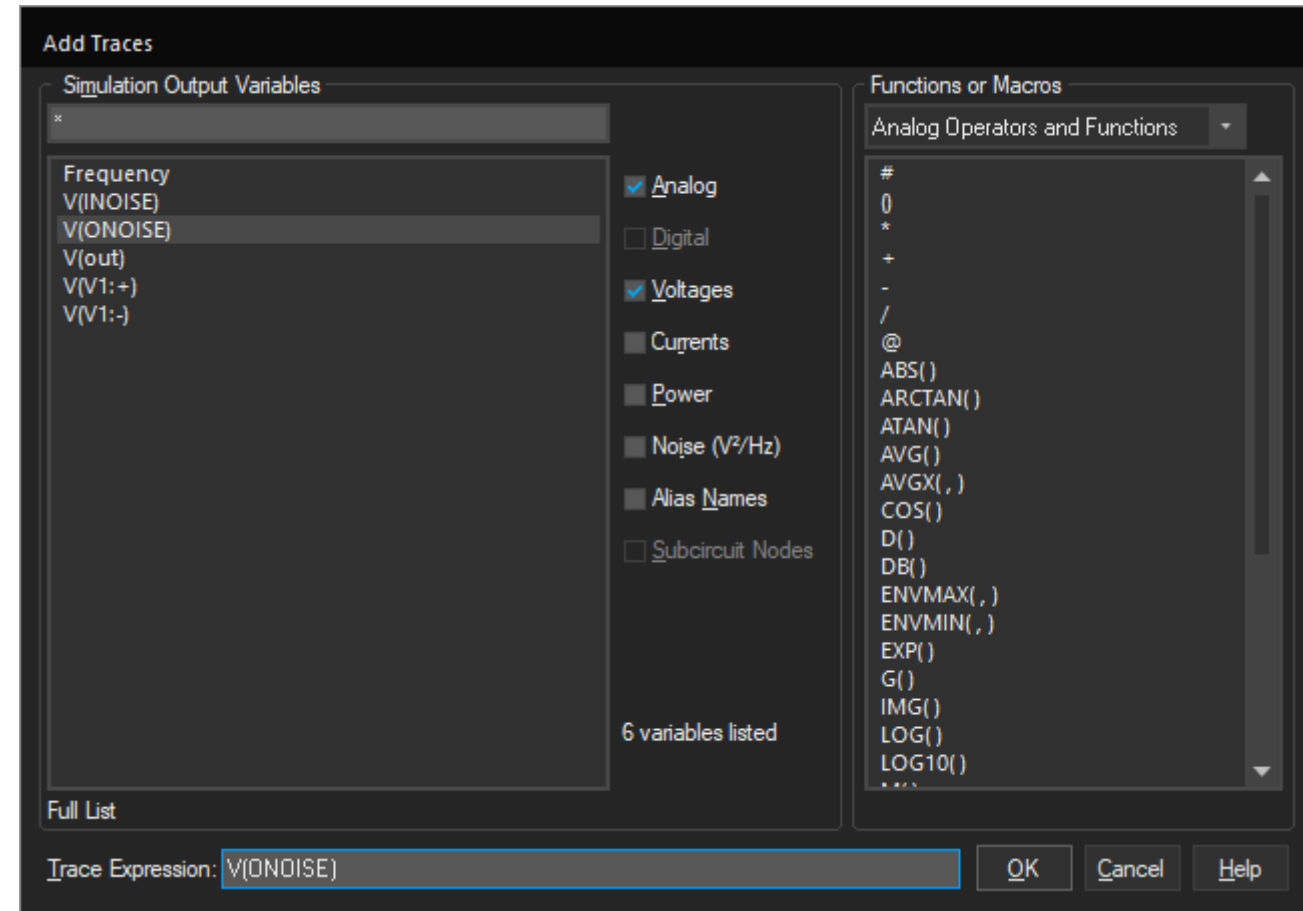
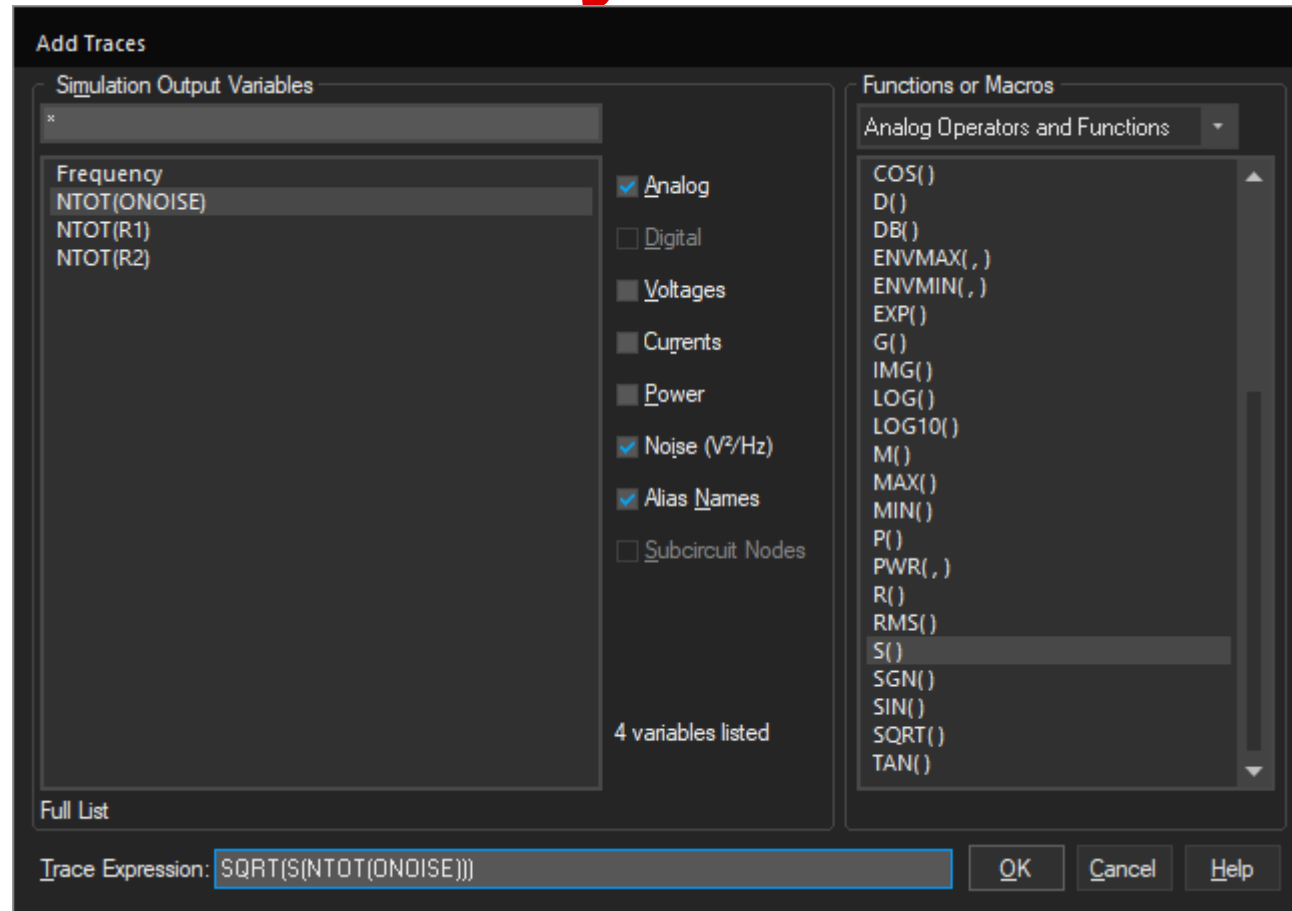
[Back to Table of Contents](#)

AC Sweep and Noise analysis



Where **Output voltage: V(net_name)** is used to plot voltage noise for **net_name**. In this case the net name is **out1**
V/I Source: source_name – In this example the source name is **V1**

Noise analysis



Integrated Total RMS noise

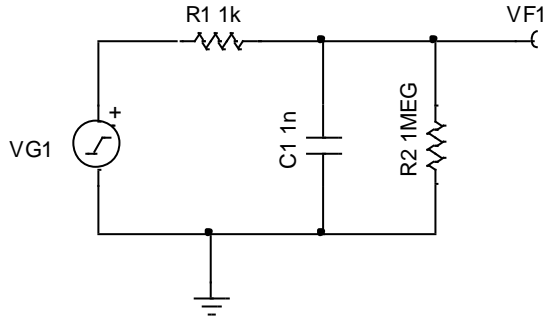
- “S” is the integral function
- SQRT is the square root function
- NTOT(ONoise) = Noise power in V²/Hz

$$E_n = \sqrt{\int P_n}$$

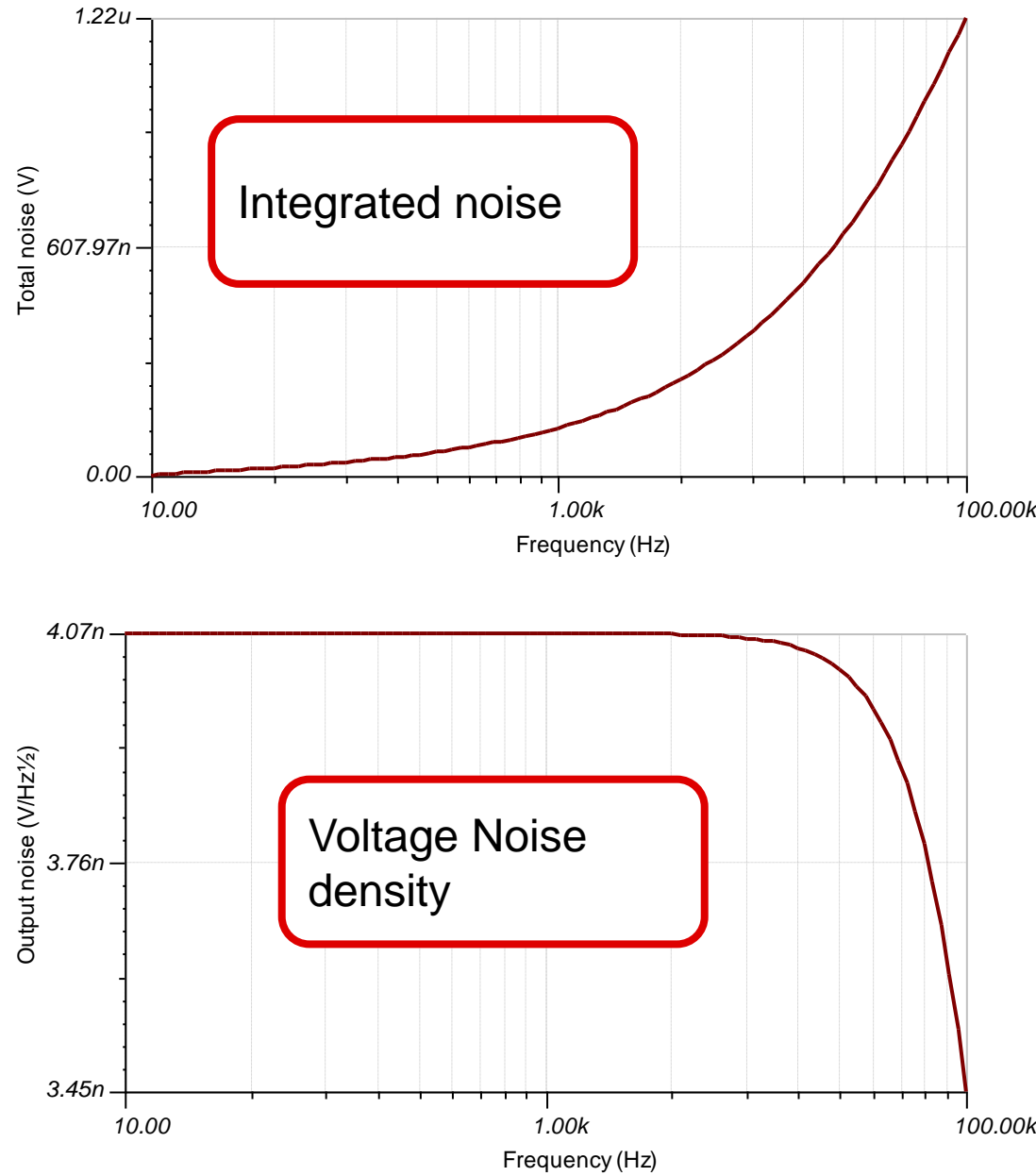
Voltage Noise Spectral Density

- V(ONoise) is the noise density in V/rtHz at the output
- V(INoise) is the input noise density
- Note: the “Noise” check box will give noise power in V²/Hz

Noise analysis



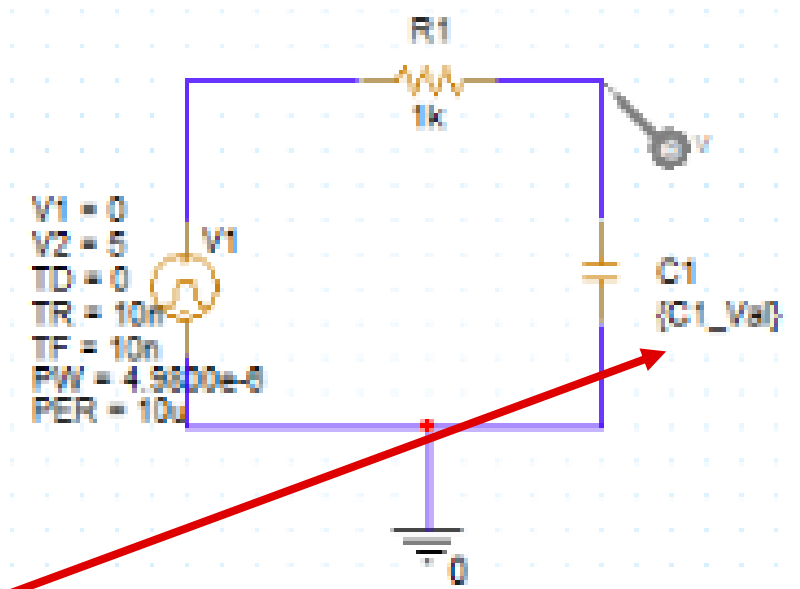
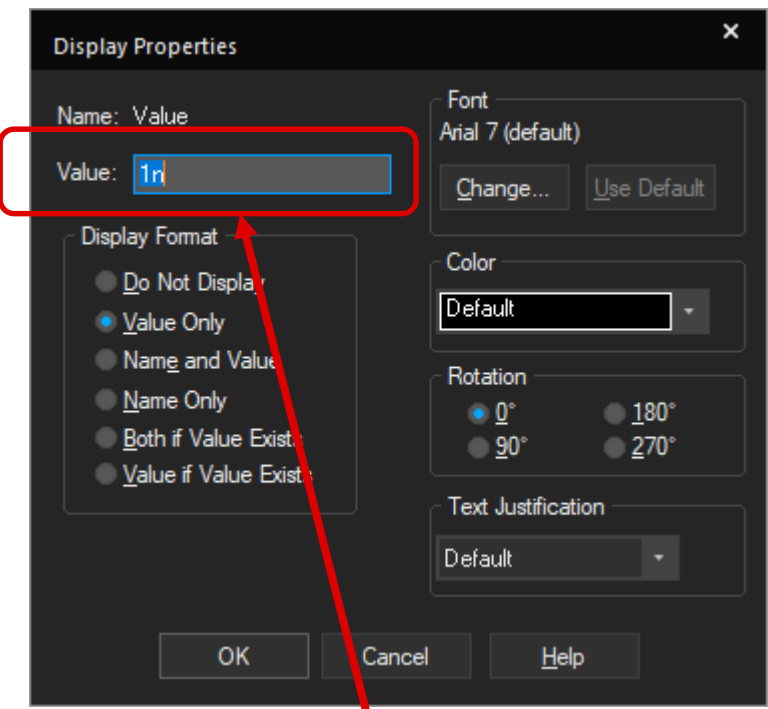
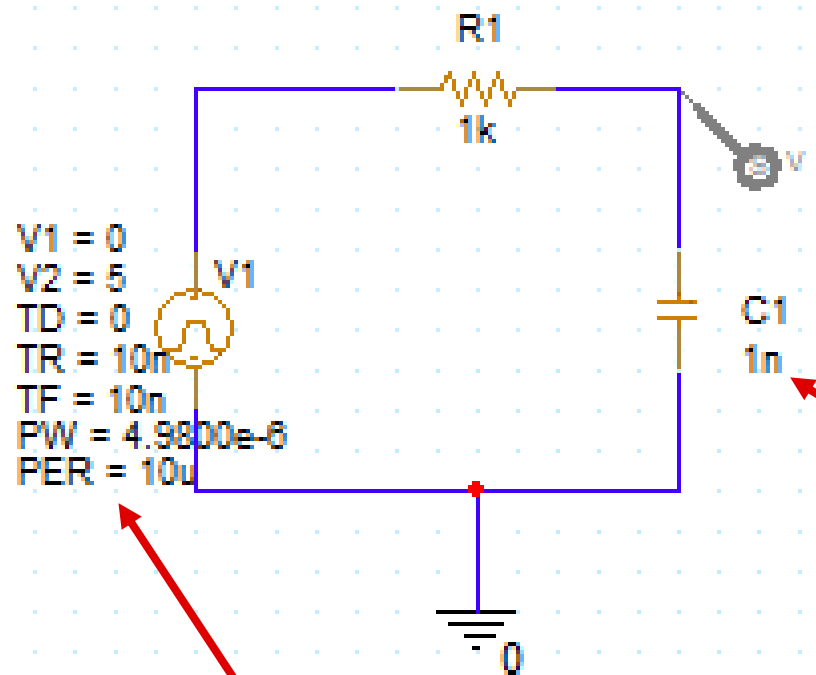
This slide shows the two traces for noise density and integrated noise match TINA. By default PSPICE plots noise power which isn't very useful



Output Window

- Calculating bias point
- Bias point calculated
- AC (and Noise) Analysis
- AC Analysis finished
- License check-out time = 9.93
- Total job time (using Solver 1) = .78
- INFINBRPDRF-3190: Simulation complete

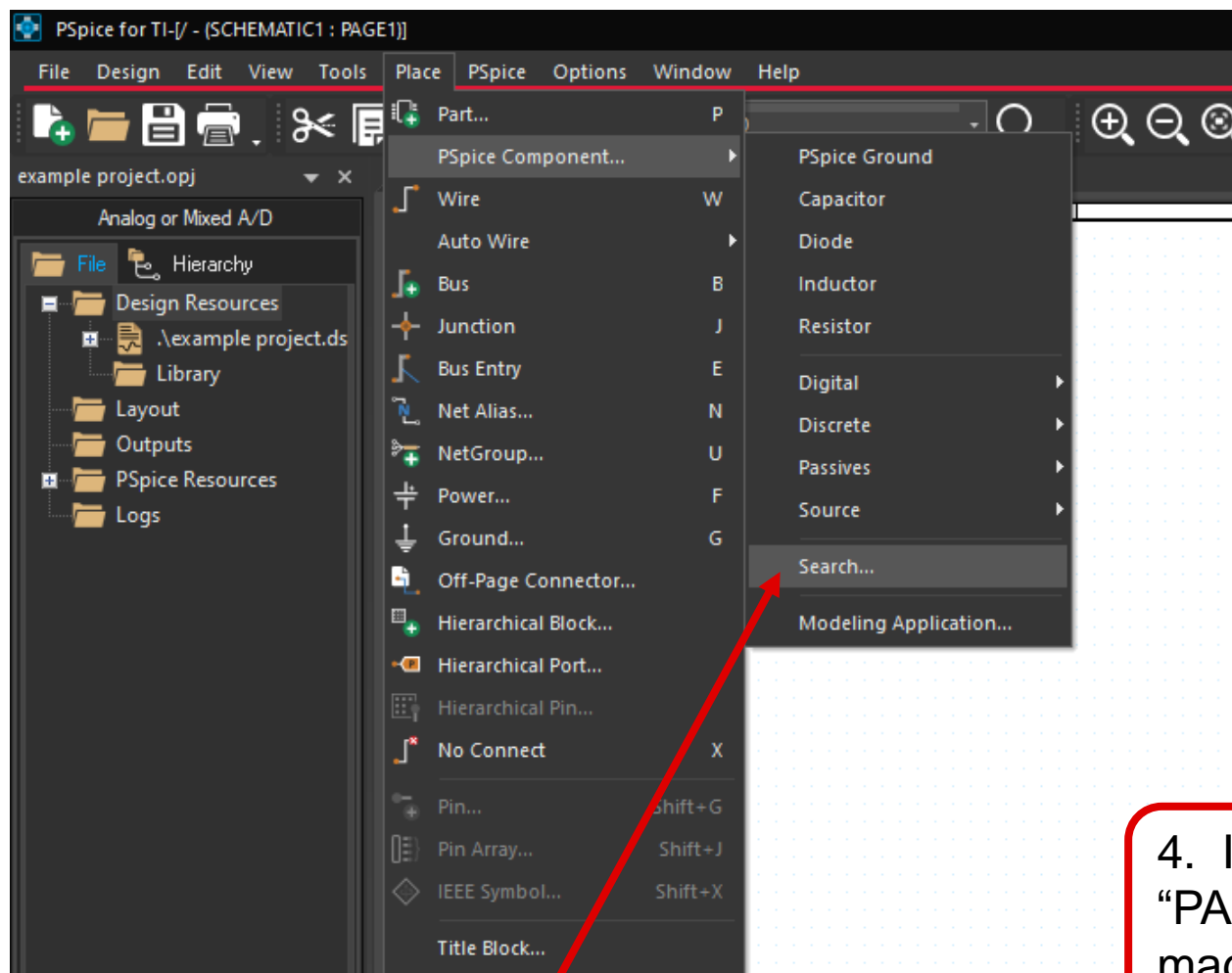
Parameter Stepping: Edit schematic value



1. Draw circuit schematic. For this example, use 100kHz (10us period) square wave.

2. Double click on the value "1n". Change it to {C1_Val}

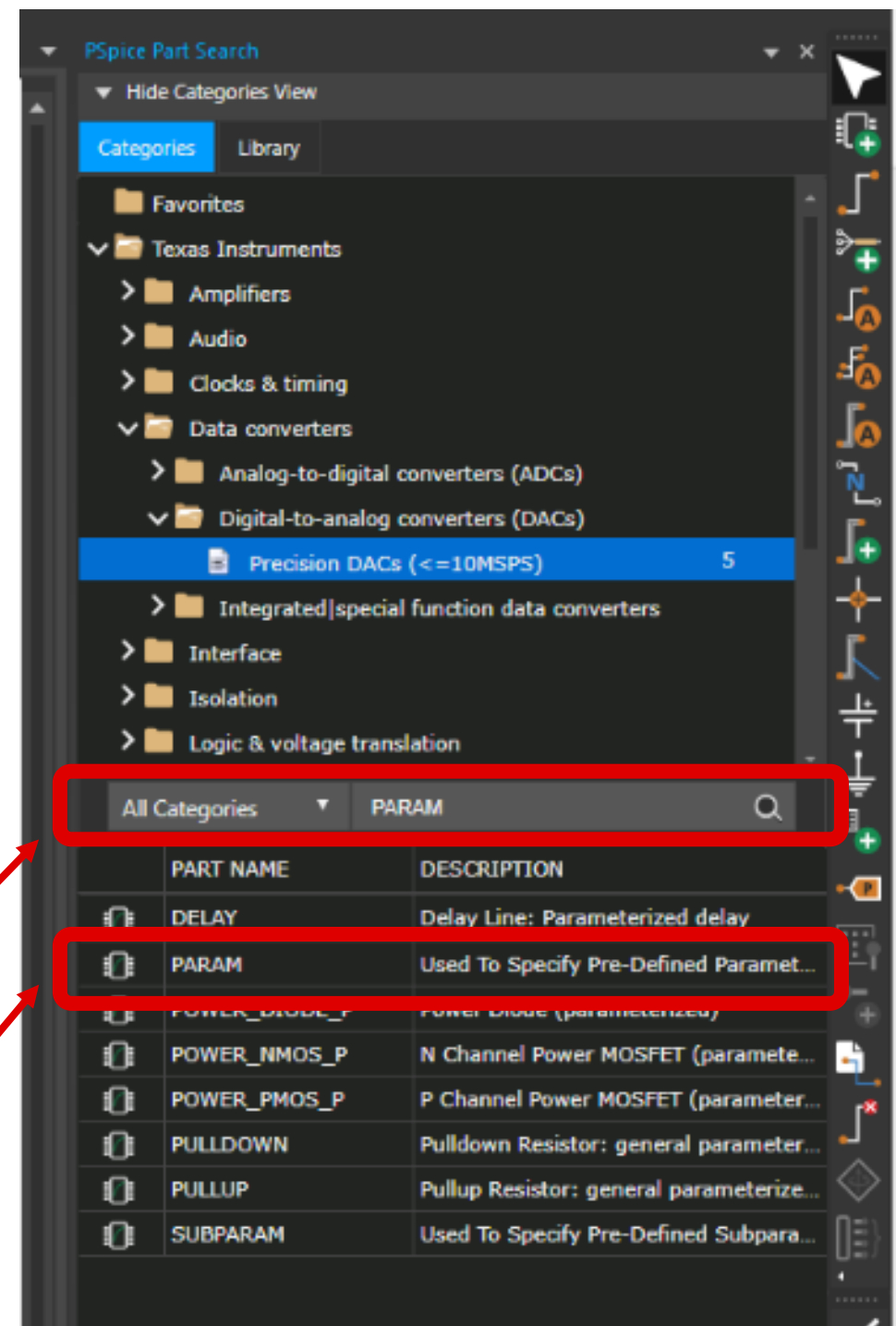
Parameter Stepping: Find PARAM



3. Select "Place > Pspice Component > Search"
This will bring up the PSPICE search utility that you typically use to find TI components.

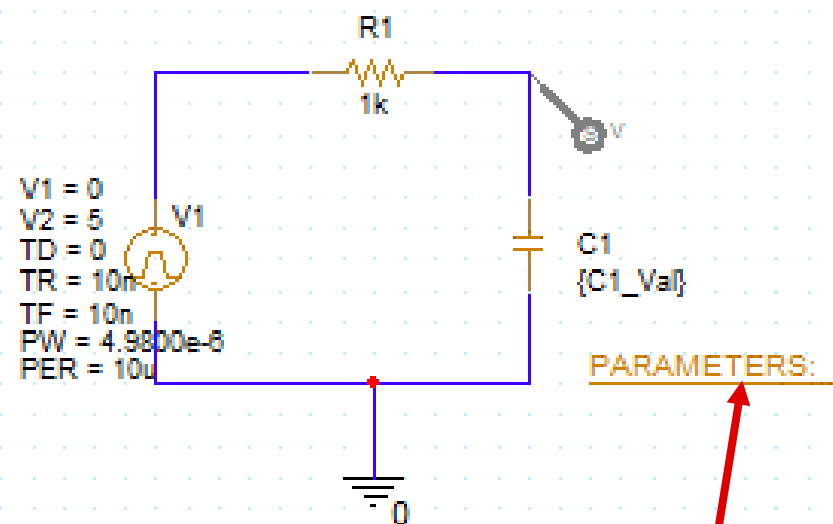
4. In the search window type "PARAM" and press the search magnifying glass.

5. Double click on "PARAM" and drag it onto your schematic.

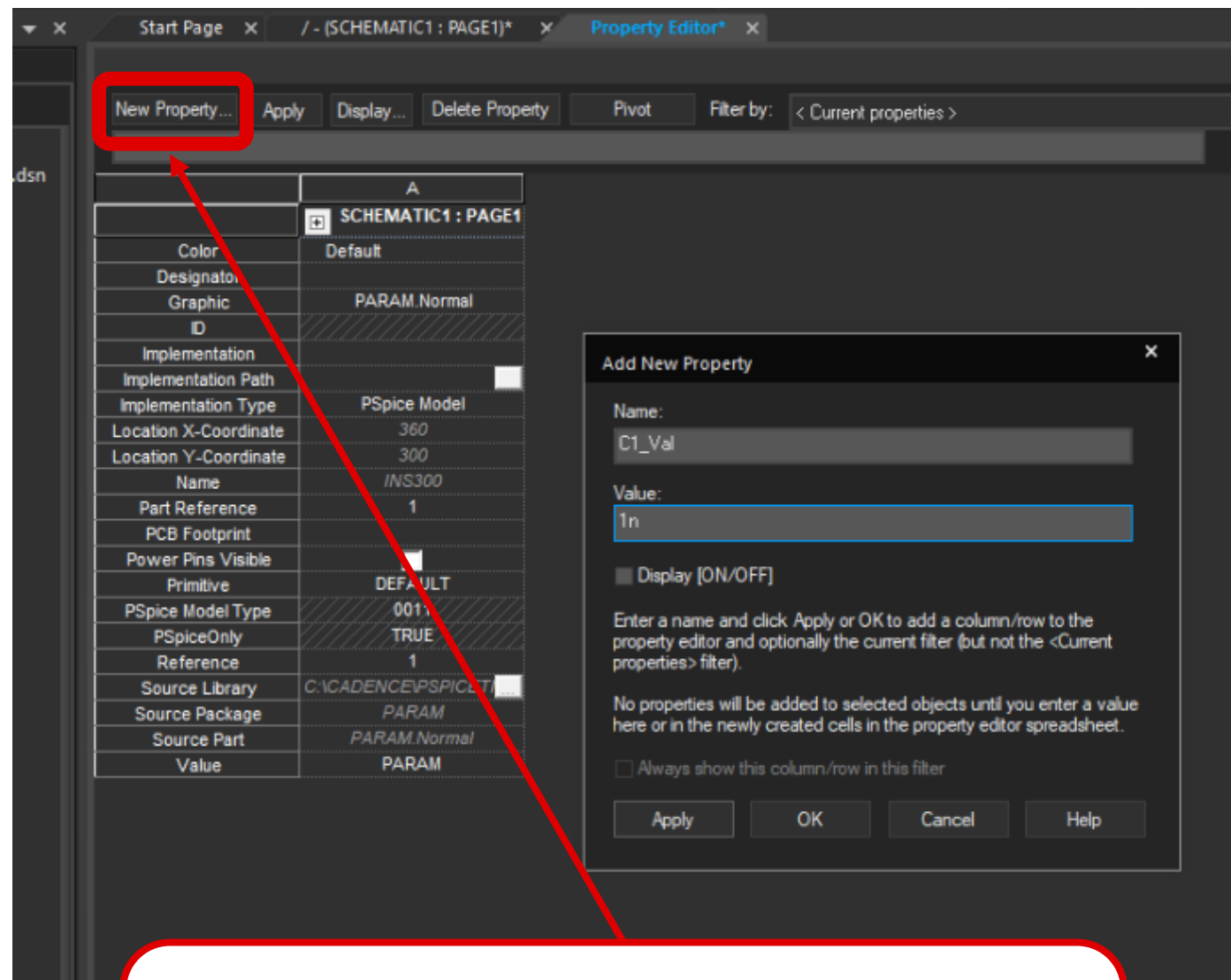


	PART NAME	DESCRIPTION
	DELAY	Delay Line: Parameterized delay
	PARAM	Used To Specify Pre-Defined Paramet...
	POWER_DIODE_P	POWER DIODE (parameterized)
	POWER_NMOS_P	N Channel Power MOSFET (paramete...
	POWER_PMOS_P	P Channel Power MOSFET (parameter...
	PULLDOWN	Pulldown Resistor: general parameter...
	PULLUP	Pullup Resistor: general parameterize...
	SUBPARAM	Used To Specify Pre-Defined Subpara...

Parameter Stepping: Edit PARAM

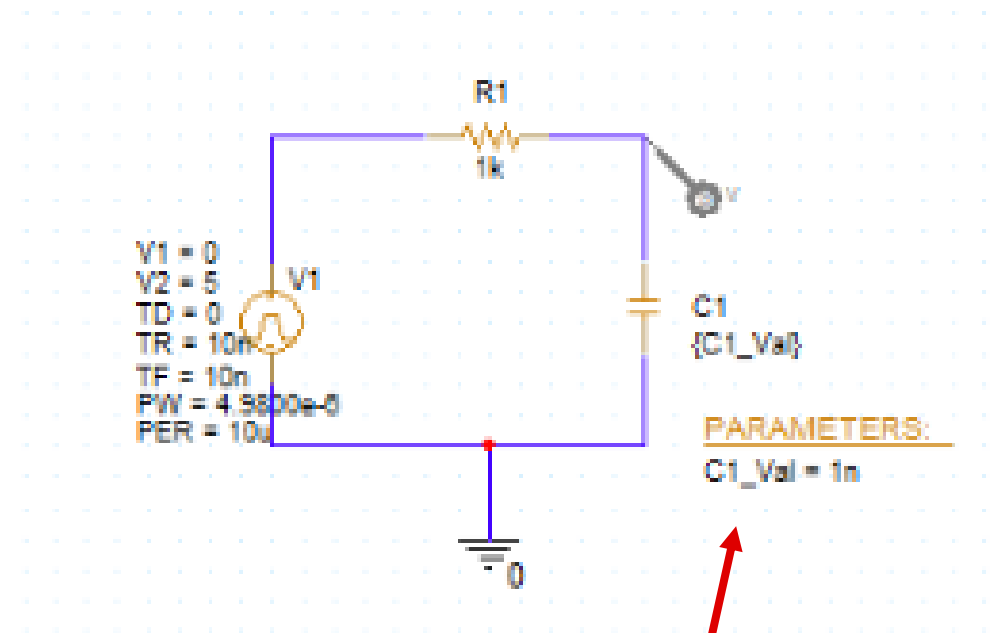
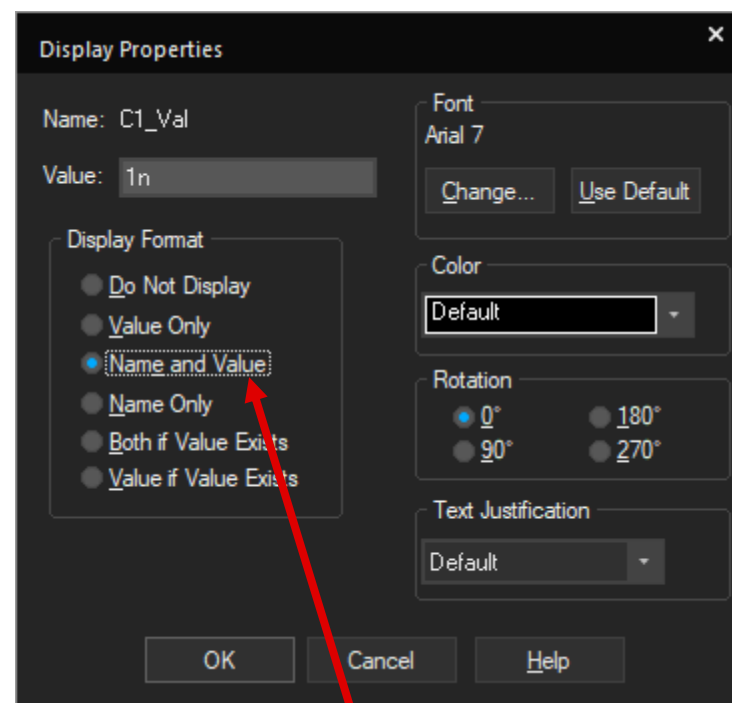
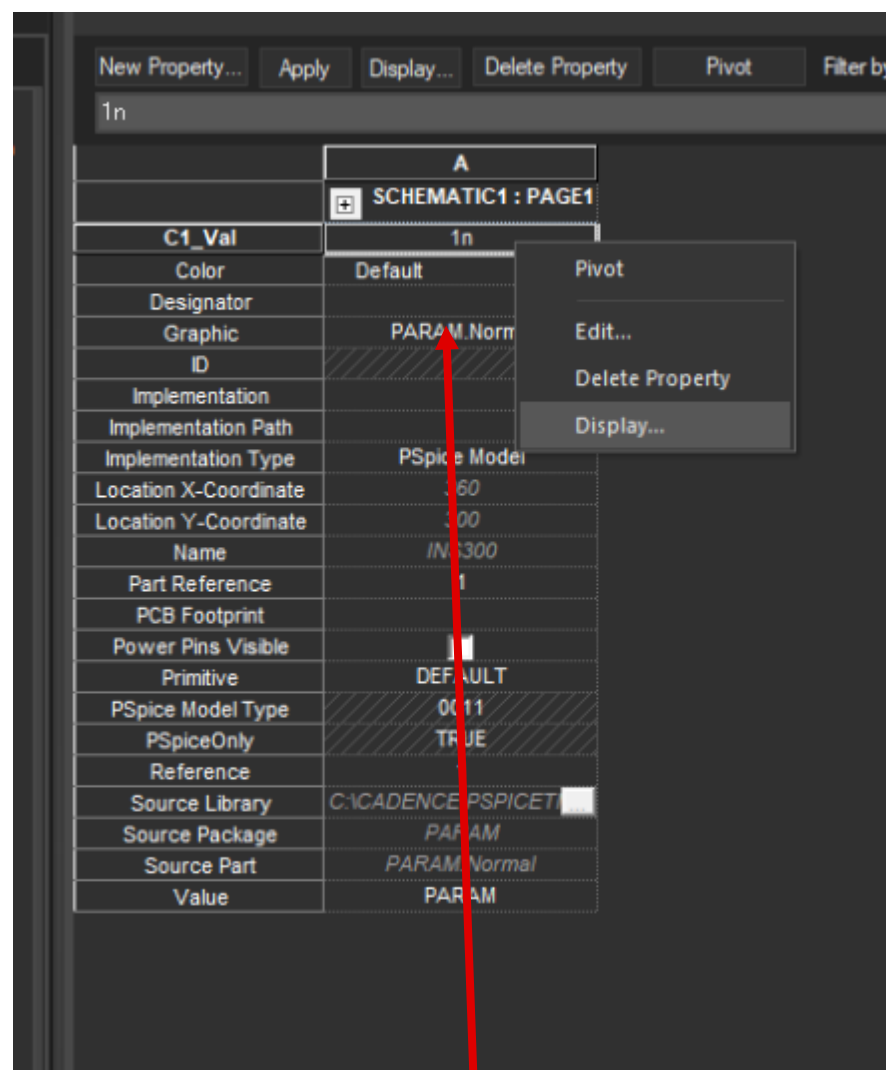


6. Double click the parameter to edit.



7. Press the "New Property" button.
8. Enter the "Name" to match the name in the braces: {C1_Val}
9. Enter a good value (e.g. 1n) and press ok.

Parameter Stepping: Display value

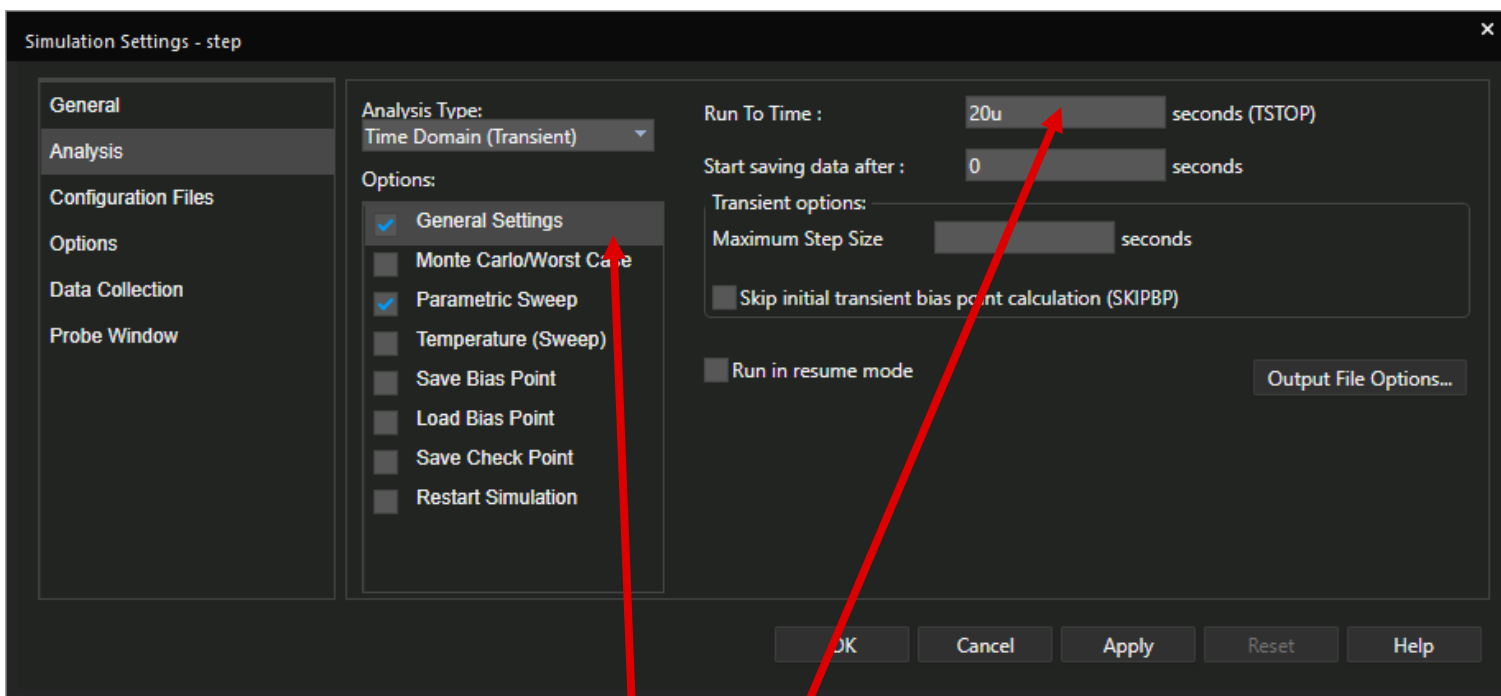


10. Right click and select "Display" to display the value on the schematic.

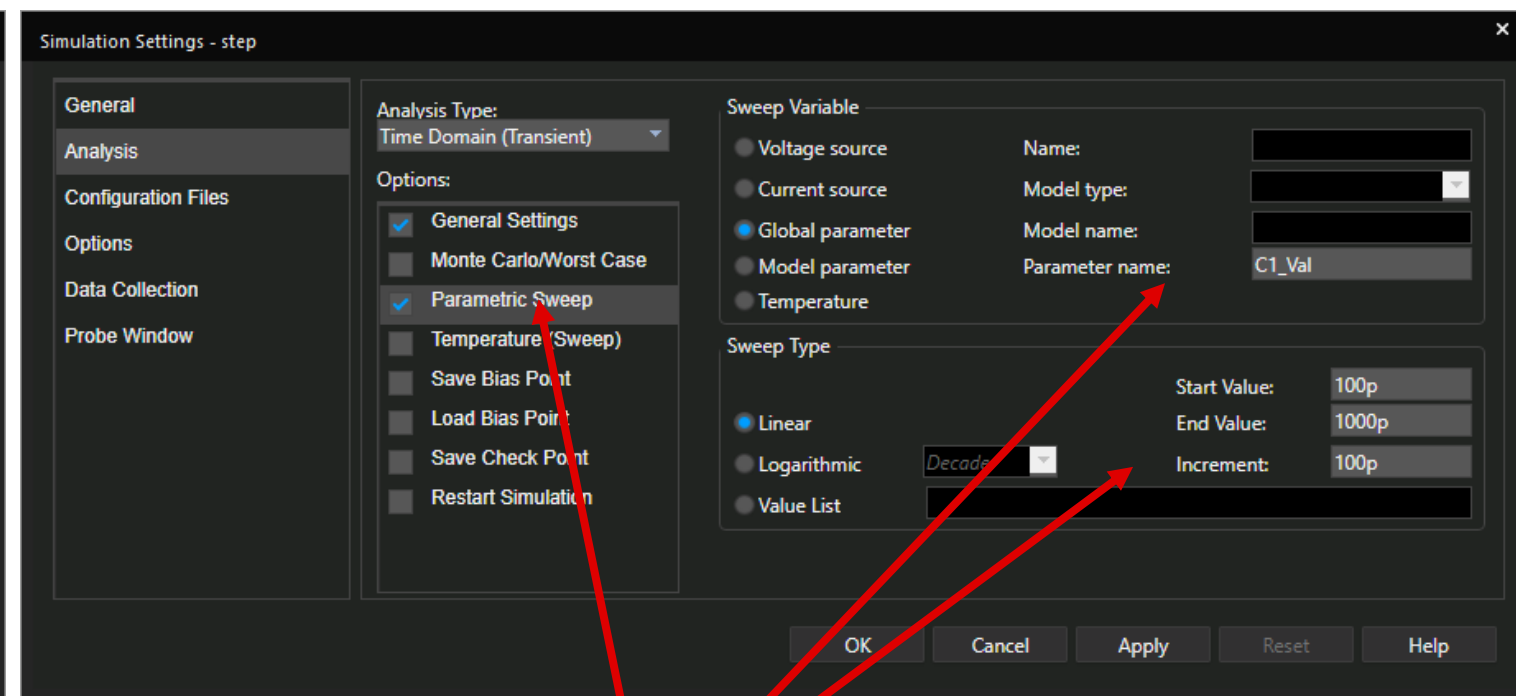
11. Select "Name and Value" and press "ok"

12. Schematic after displaying the value.

Parameter Stepping: Set up analysis profile

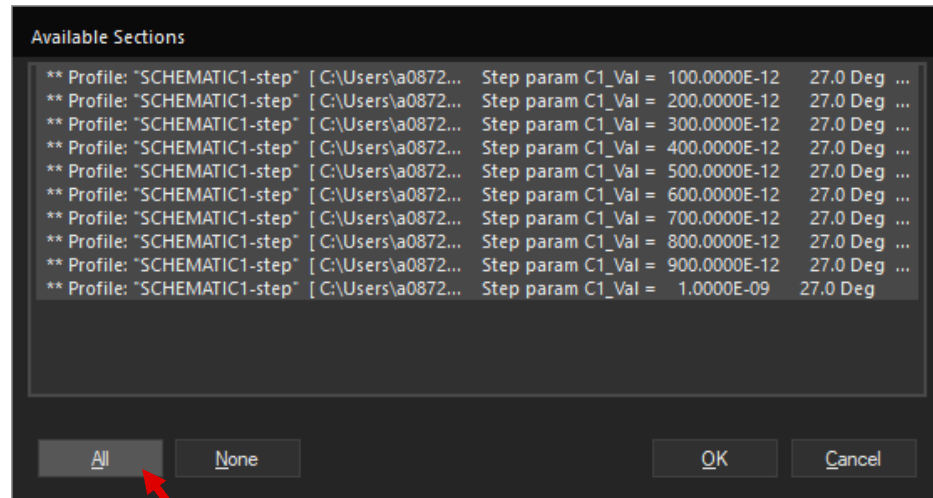


13. Choose a transient analysis for 20us under “general settings”

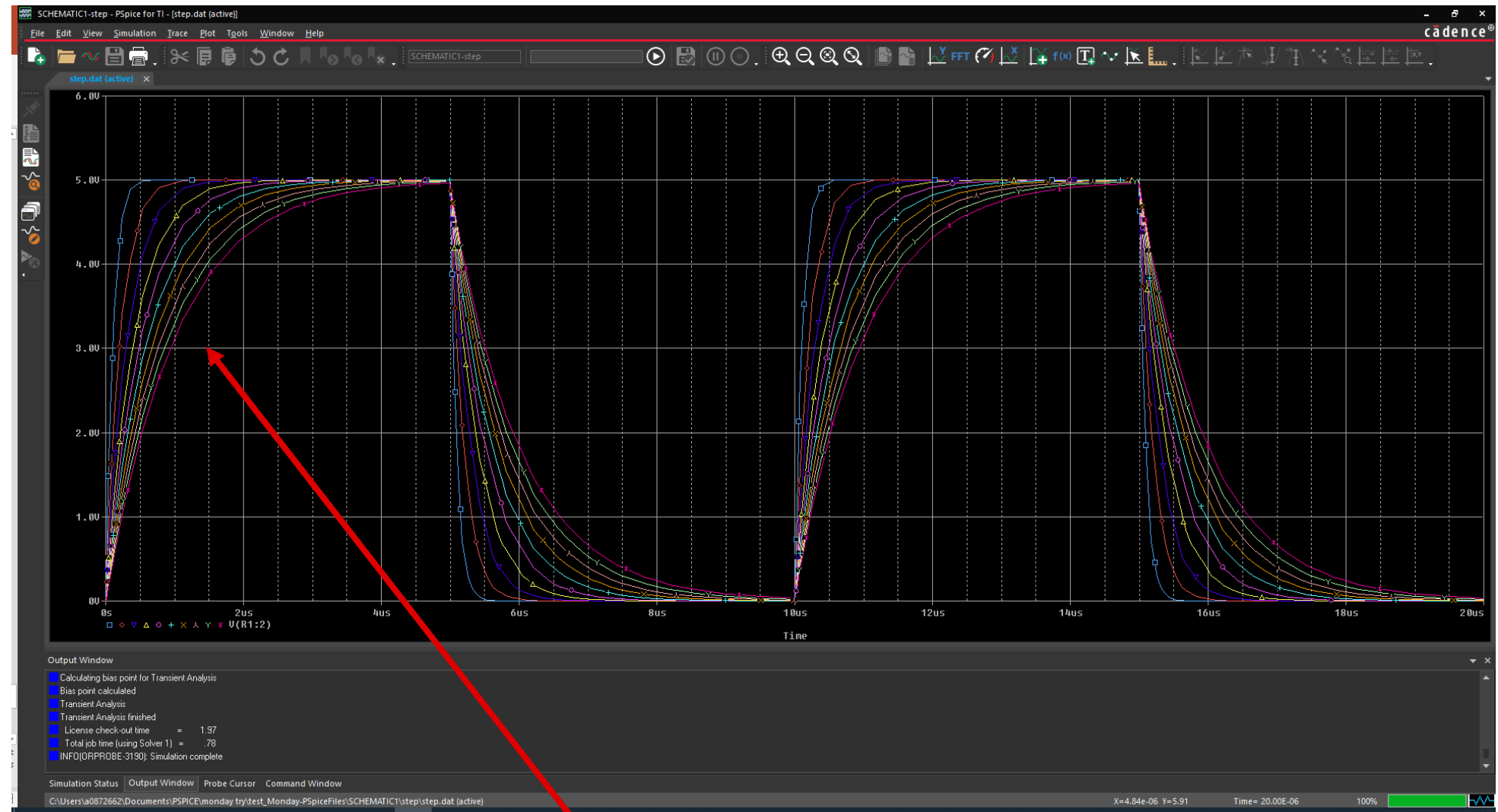


14. Under “Parametric Sweep”, select:
- Parameter name
 - Sweep type (Linear in this example)
 - Start value, end value, and increment

Parameter Stepping: Example Results

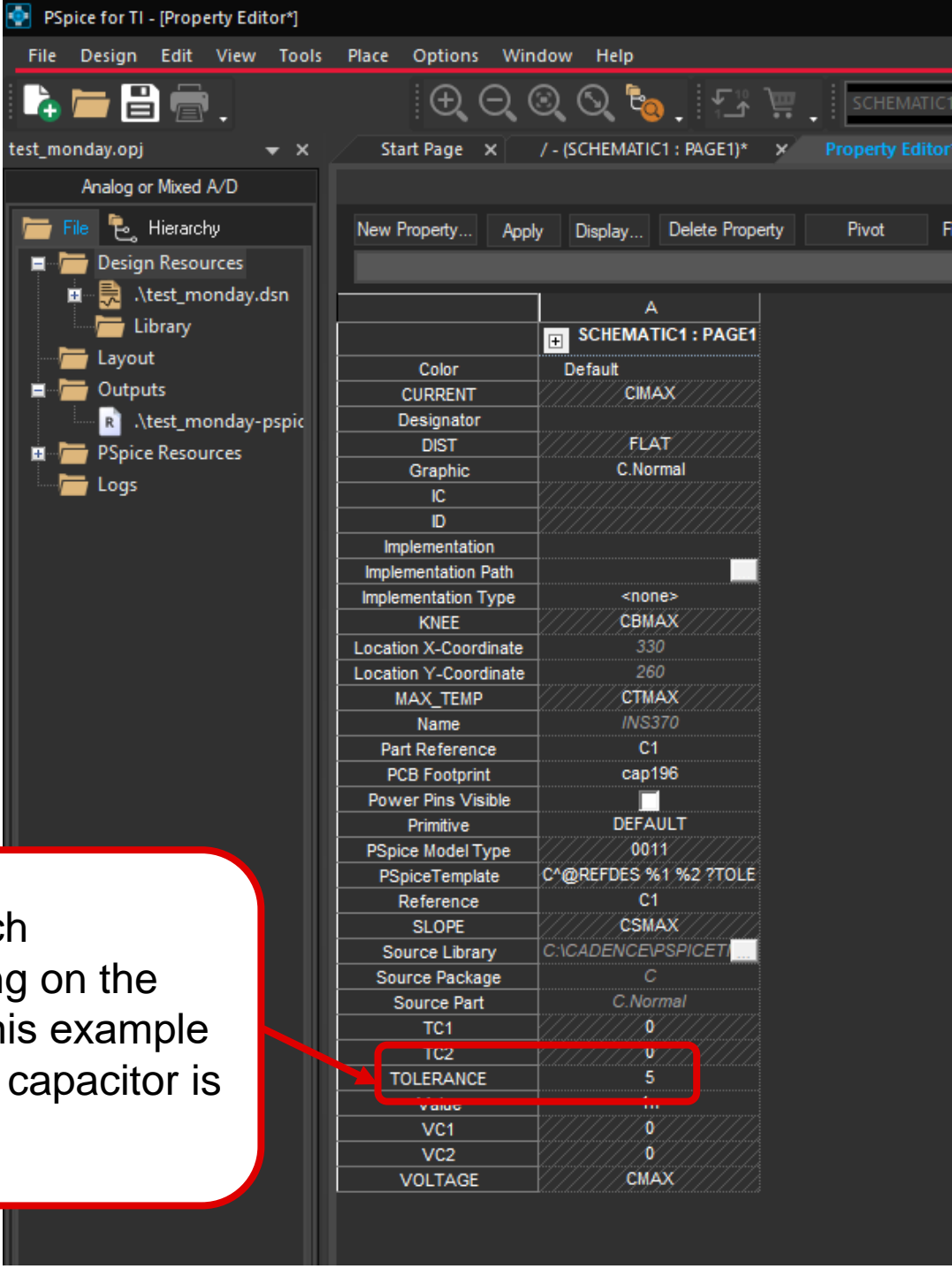
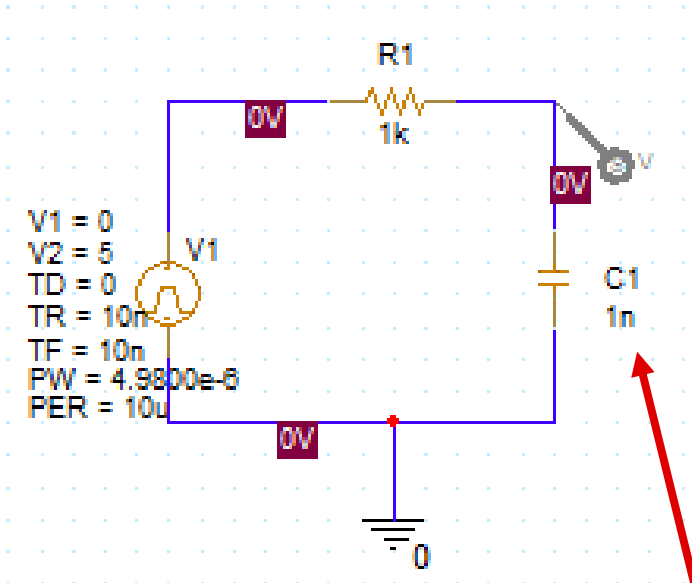
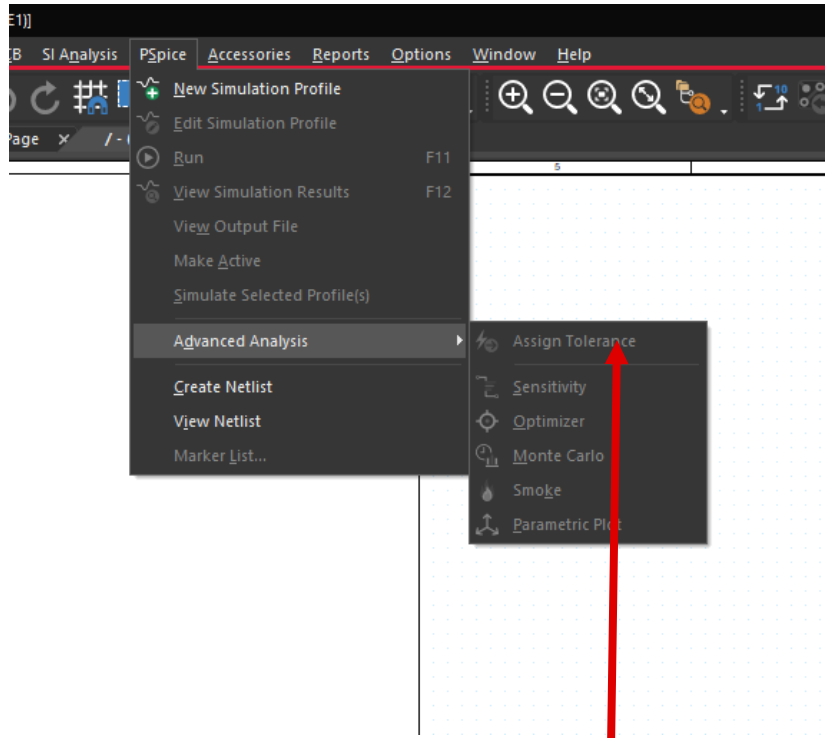


15. Select "all" and press "ok"



16. Parameter step results.

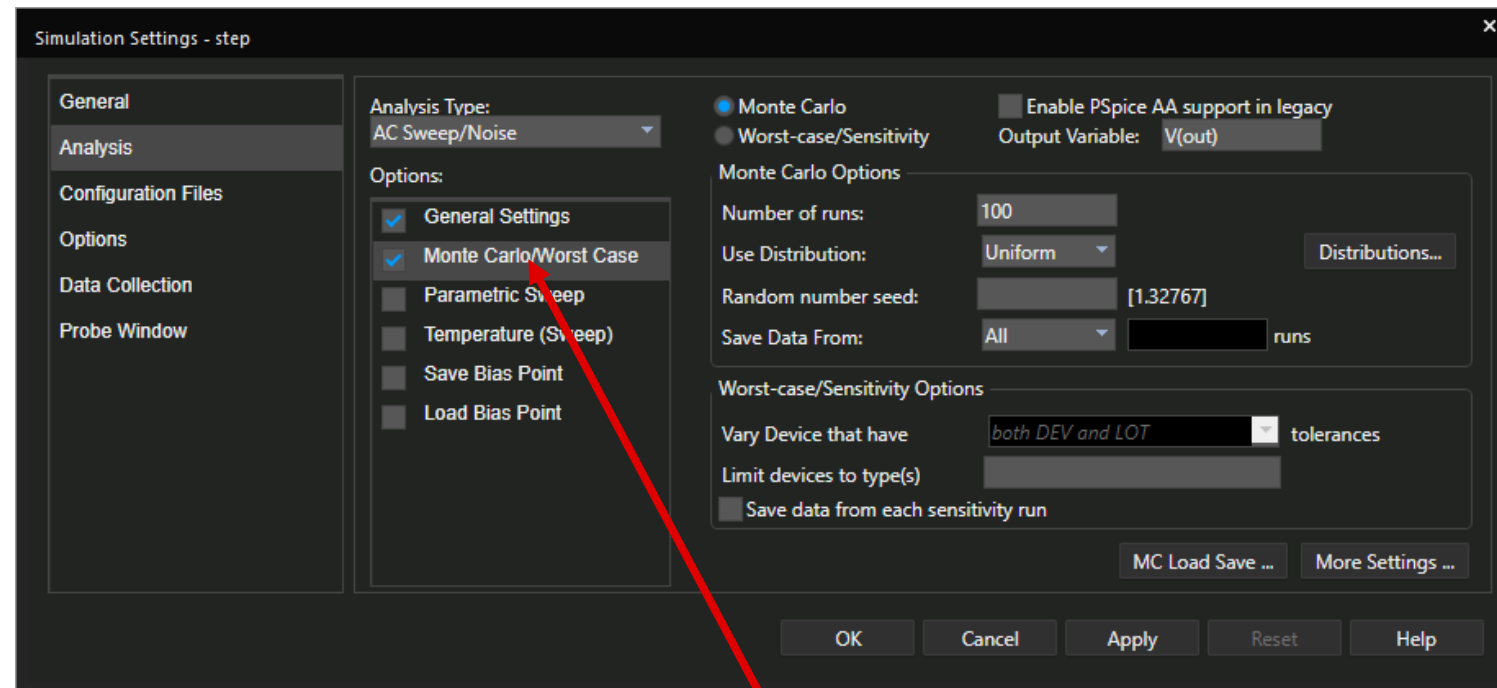
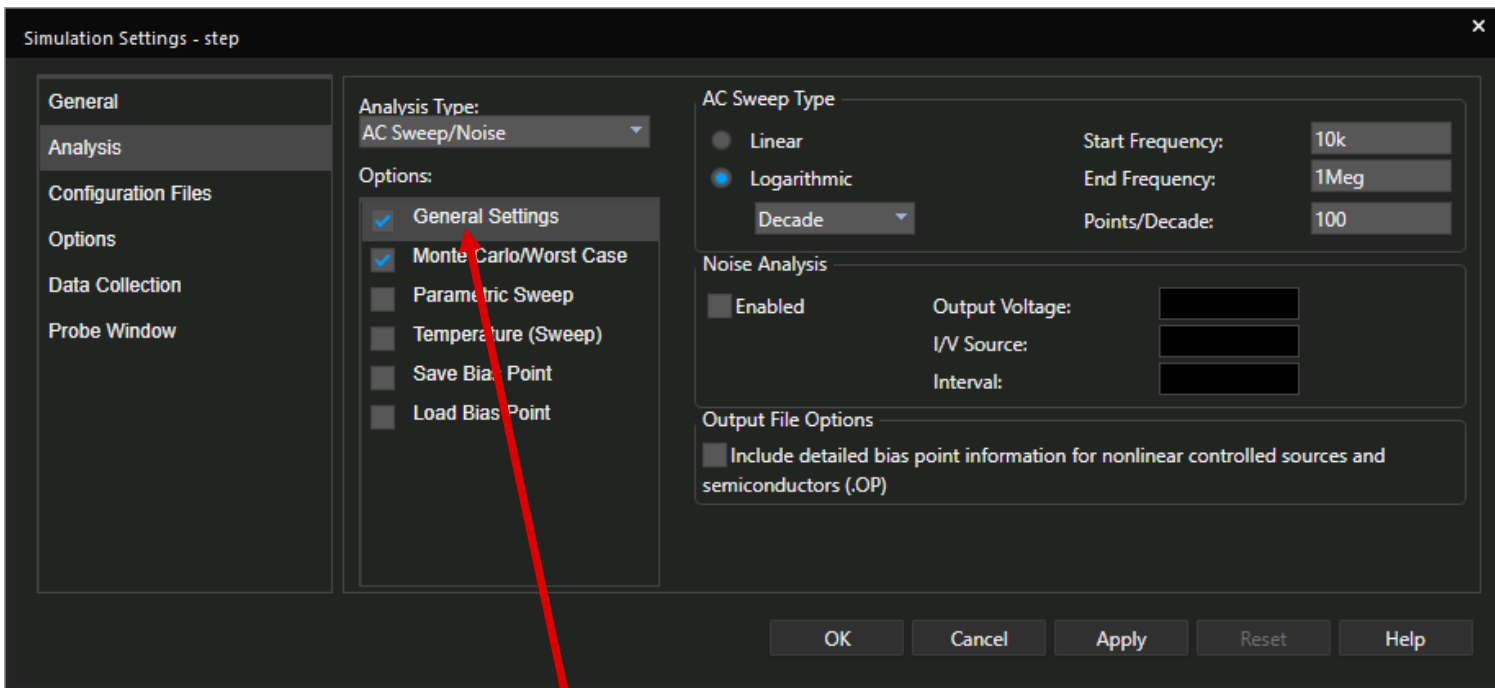
Monte Carlo Analysis



1. Note: the full version of Cadence allows assigning tolerances to all components using "assign tolerance" option. If you did a complex design and wanted to compare 0.1% tolerance to 1% tolerance results this would be a useful option. This is not available on PSPICE for TI.

2. Choose tolerance for each component by double clicking on the "tolerance" parameter. In this example the resistor is set to 1% and capacitor is set to 5%.

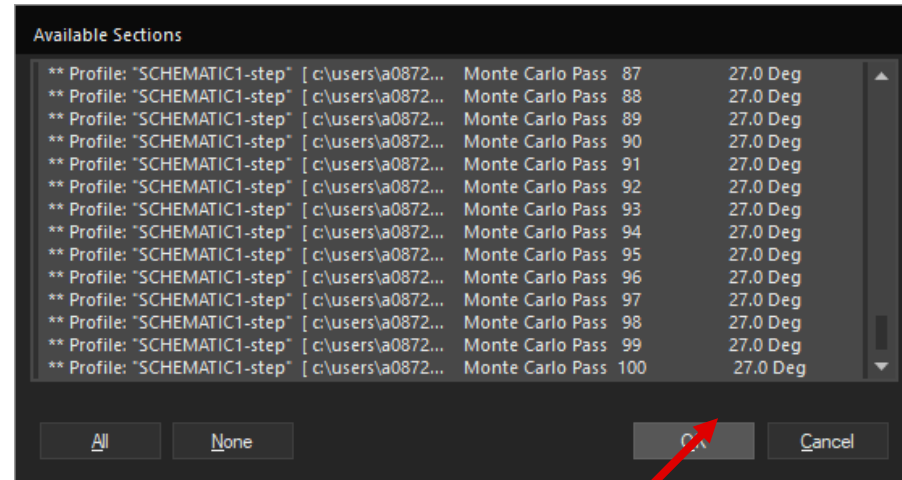
Monte Carlo Analysis



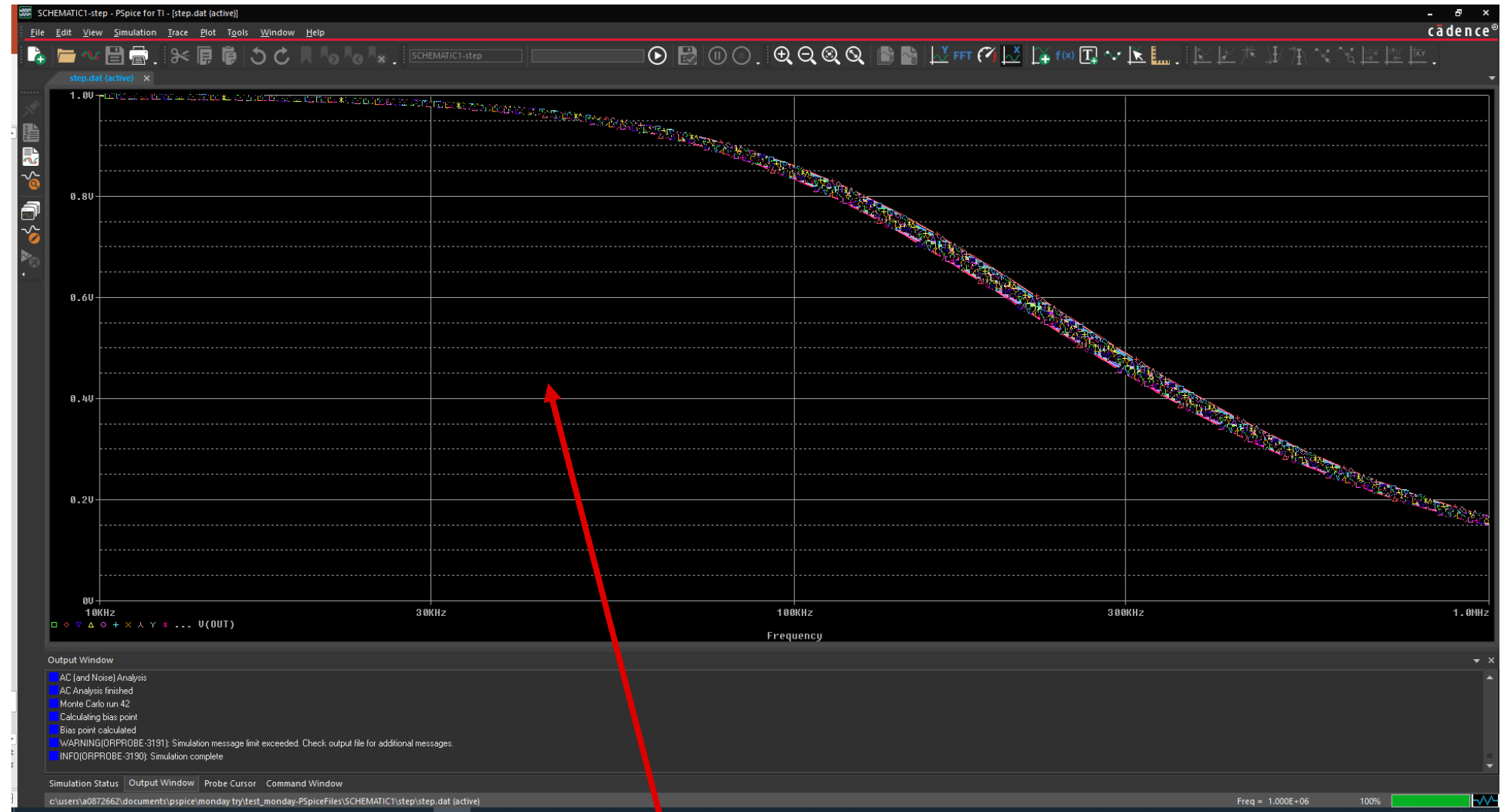
3. Choose analysis type for “General Settings”. In this example we use an AC sweep type.

4. Select Monte Carlo Worst Case
5. Choose the output variable (use the net name). In this case the net name is “out”, so V(out) plots the voltage for the “out” net.
6. Enter the “number of runs” (100 used in this example)

Monte Carlo Analysis

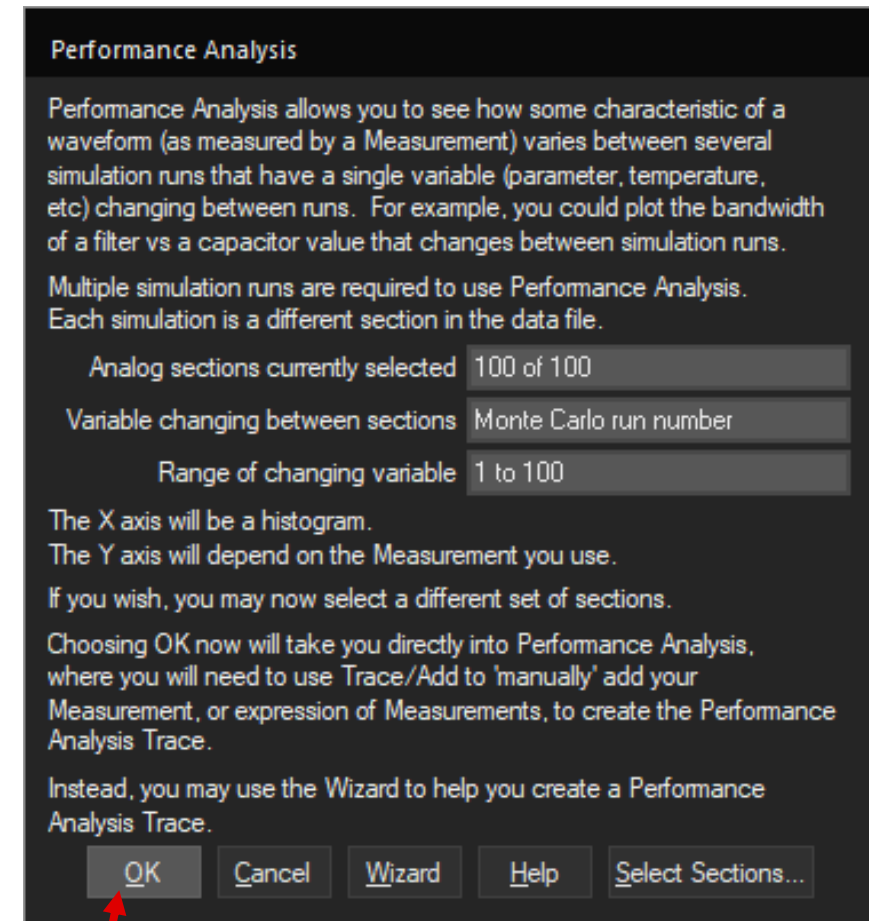
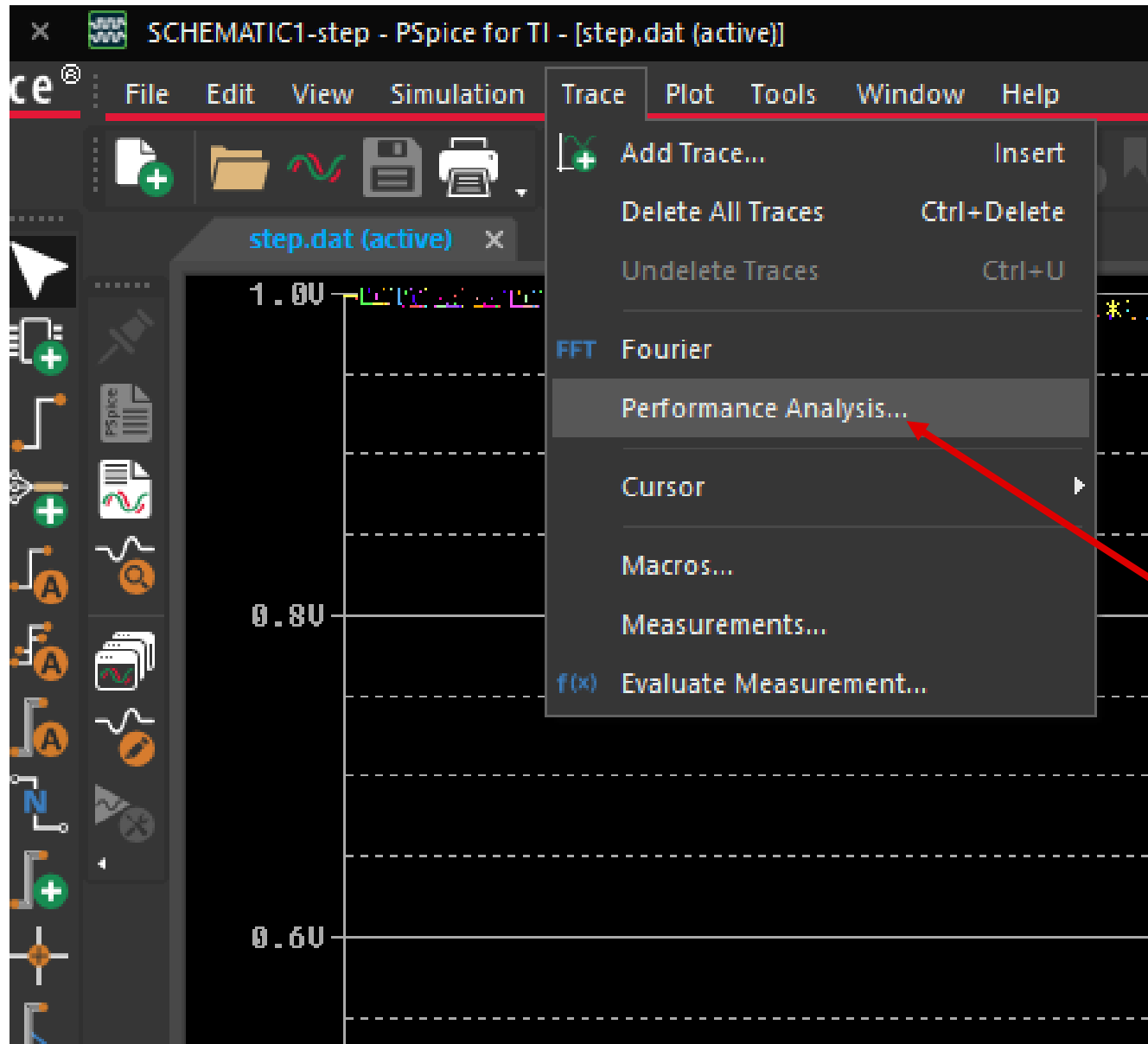


7. Run the simulation. When the simulation is complete you will get a list of available results. Select all and press ok.



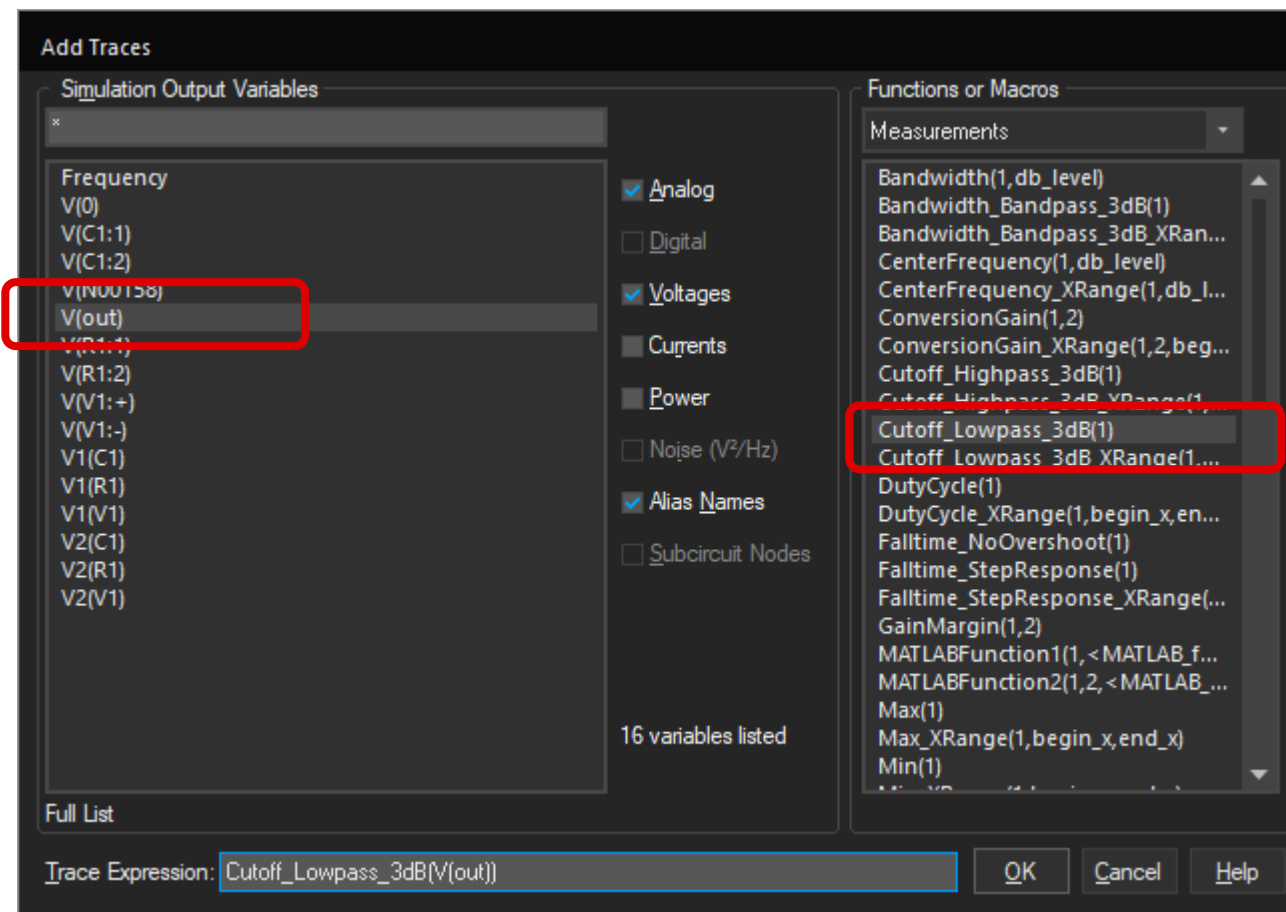
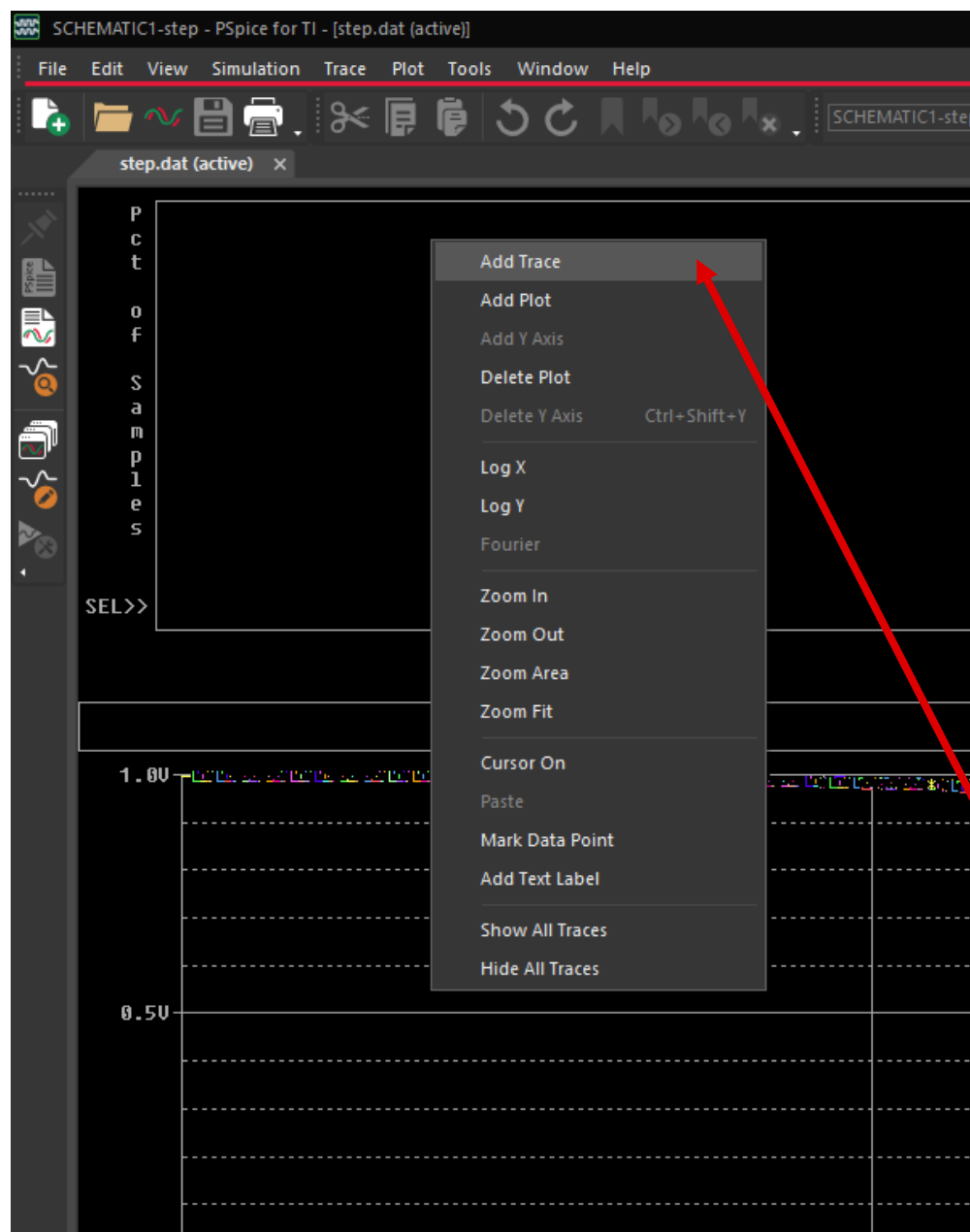
8. This shows a typical Monte Carlo result. 100 different simulation curves are displayed.

Monte Carlo Analysis



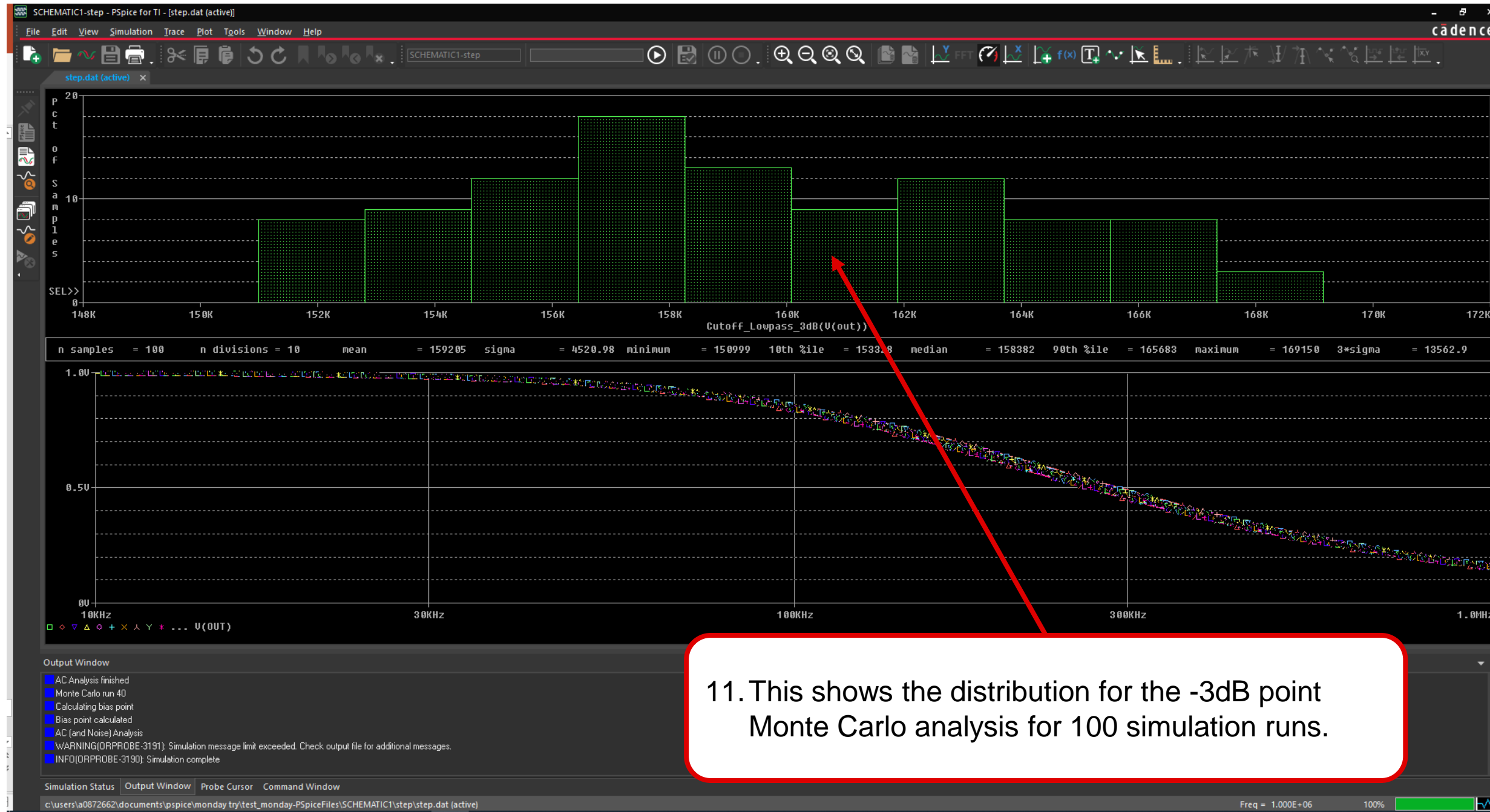
8. Choose performance analysis then press ok for the pop-up.

Monte Carlo Analysis



9. Press “add trace” on the new plot.
10. Select the function you want to apply then the output variable. In this case the function is “Cutoff_lowpass_3dB” and the variable is V(out).

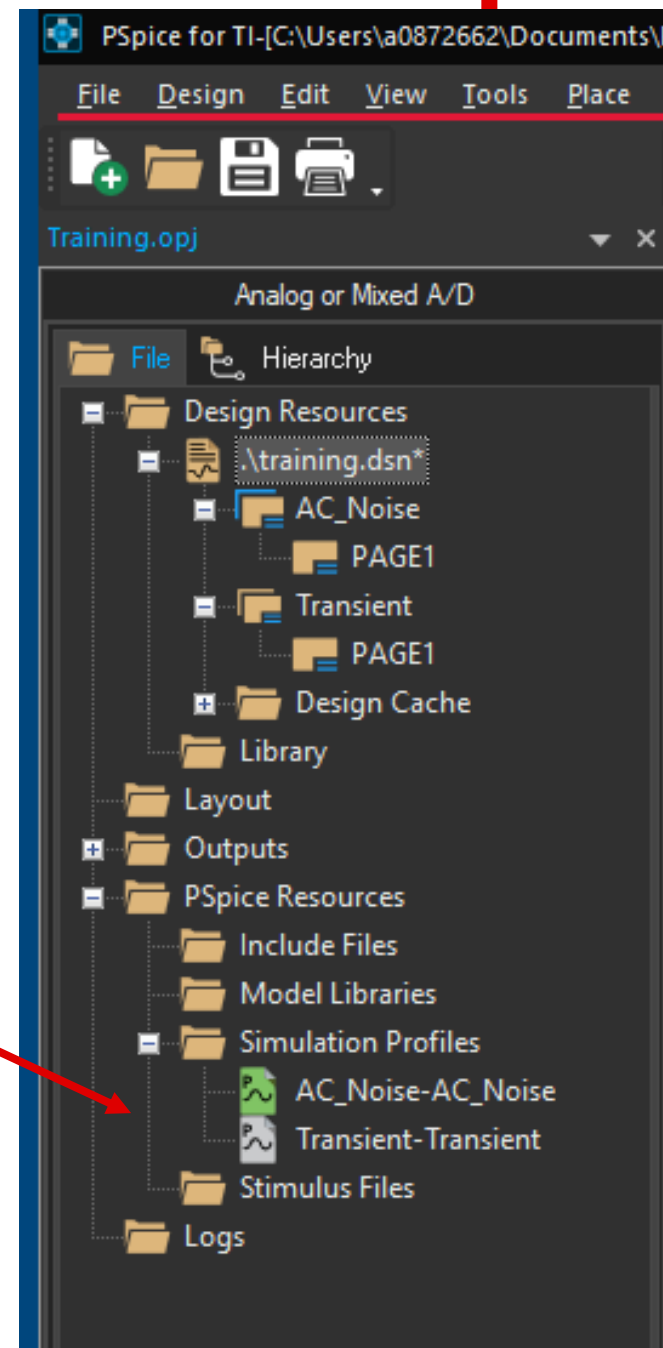
Monte Carlo Analysis



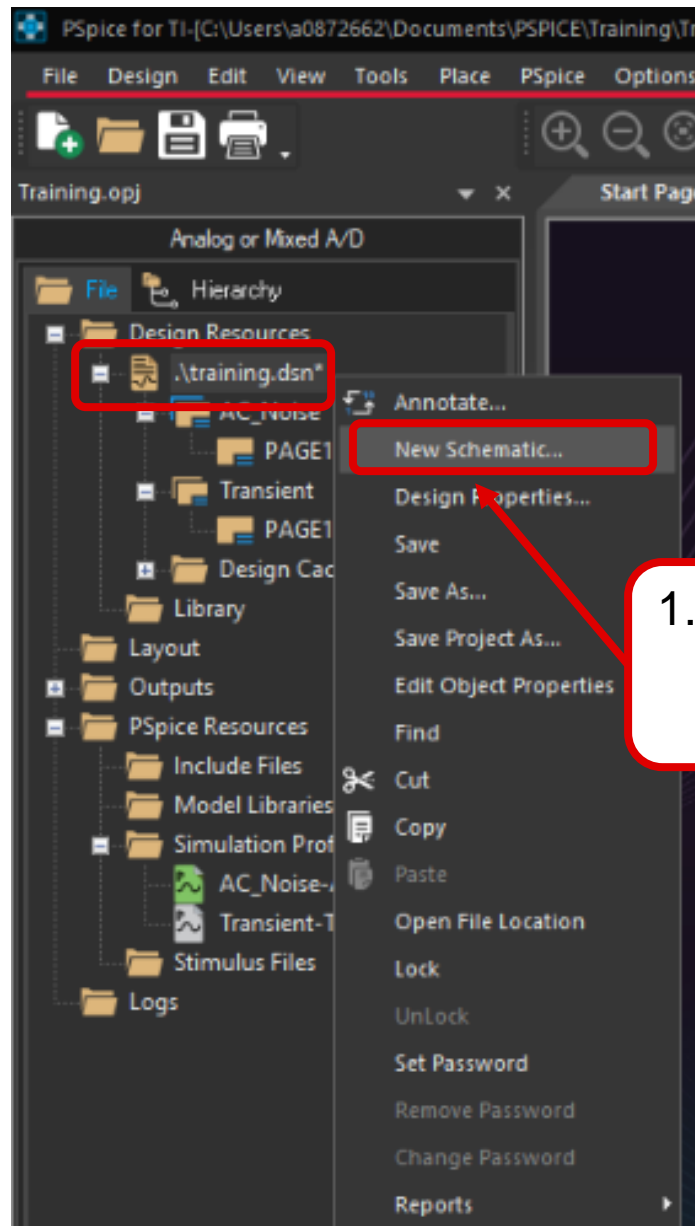
11. This shows the distribution for the -3dB point Monte Carlo analysis for 100 simulation runs.

Using multiple schematics and simulation profiles

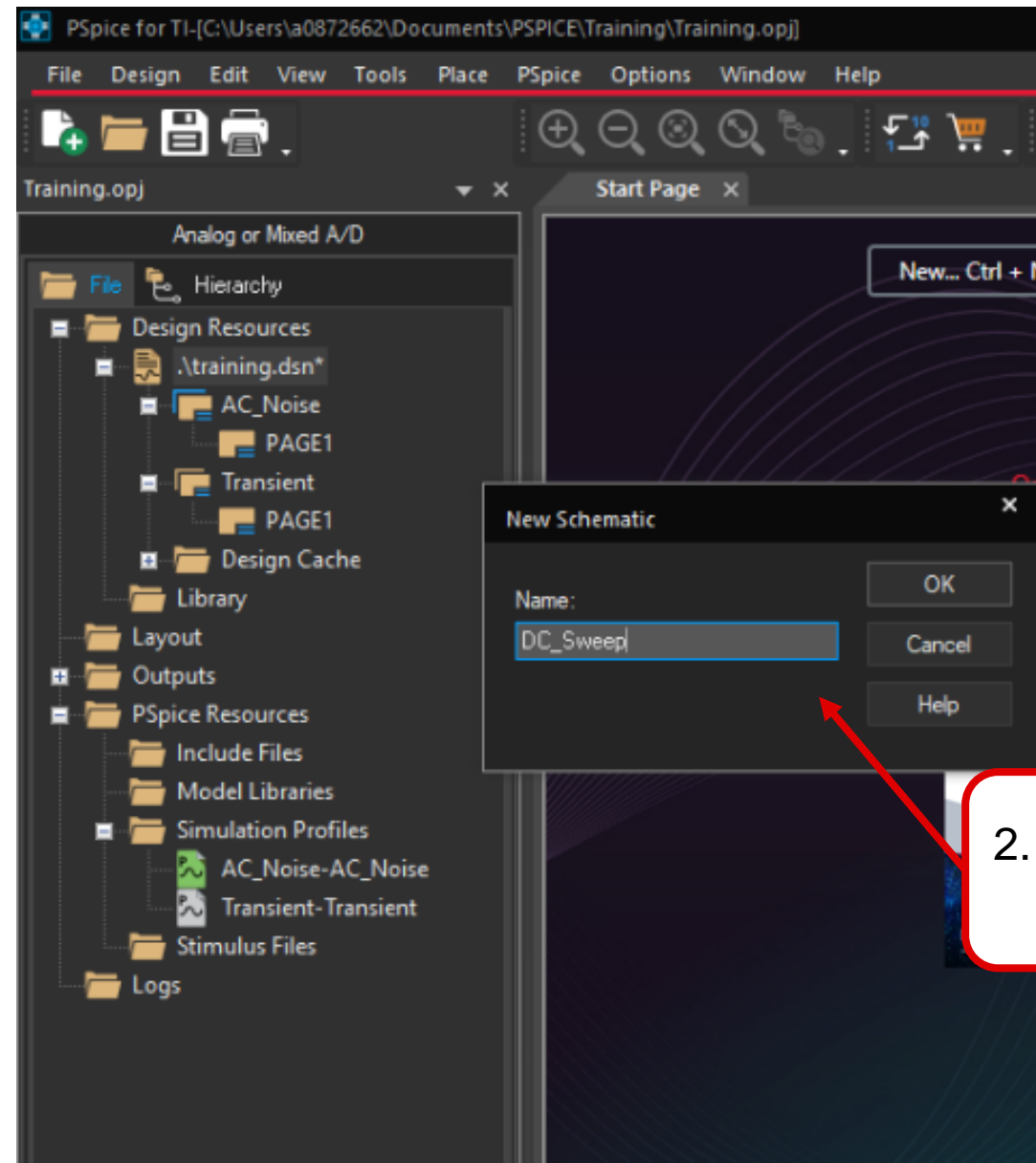
This shows a project with two schematics added and two simulation profiles. This approach can be useful if you want to do variations on different types of simulations. For example, a transient simulation requires a square wave source, and an AC simulation requires an AC source. You can build two slightly different schematics and develop simulation profiles for each schematic.



Using multiple schematics and simulation profiles

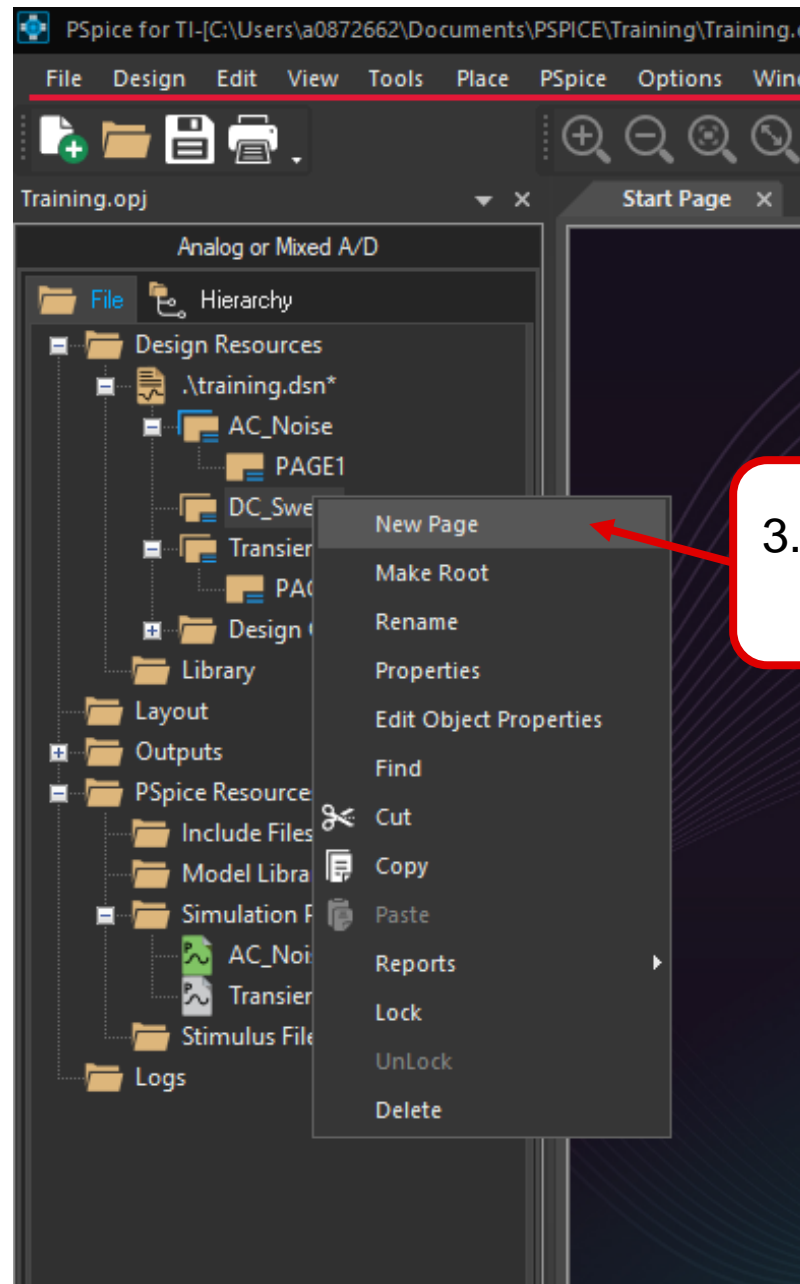


1. Right click on project and select "New Schematic"

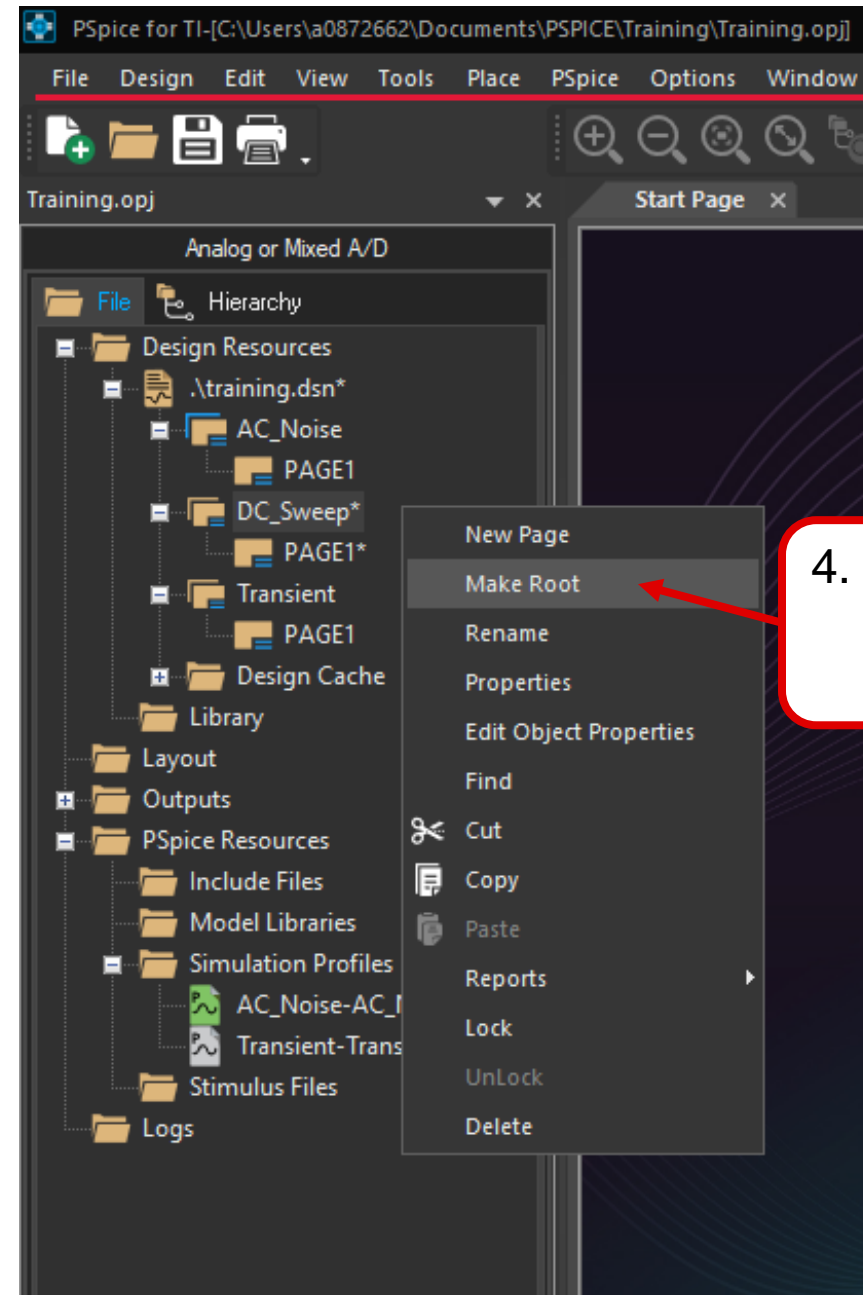


2. Choose a name for the schematic

Using multiple schematics and simulation profiles



3. Add a page to the schematic



4. To create a simulation profile for this schematic select "Make Root"

Using multiple schematics and simulation profiles

PSpice for TI - (DC_Sweep : PAGE1)

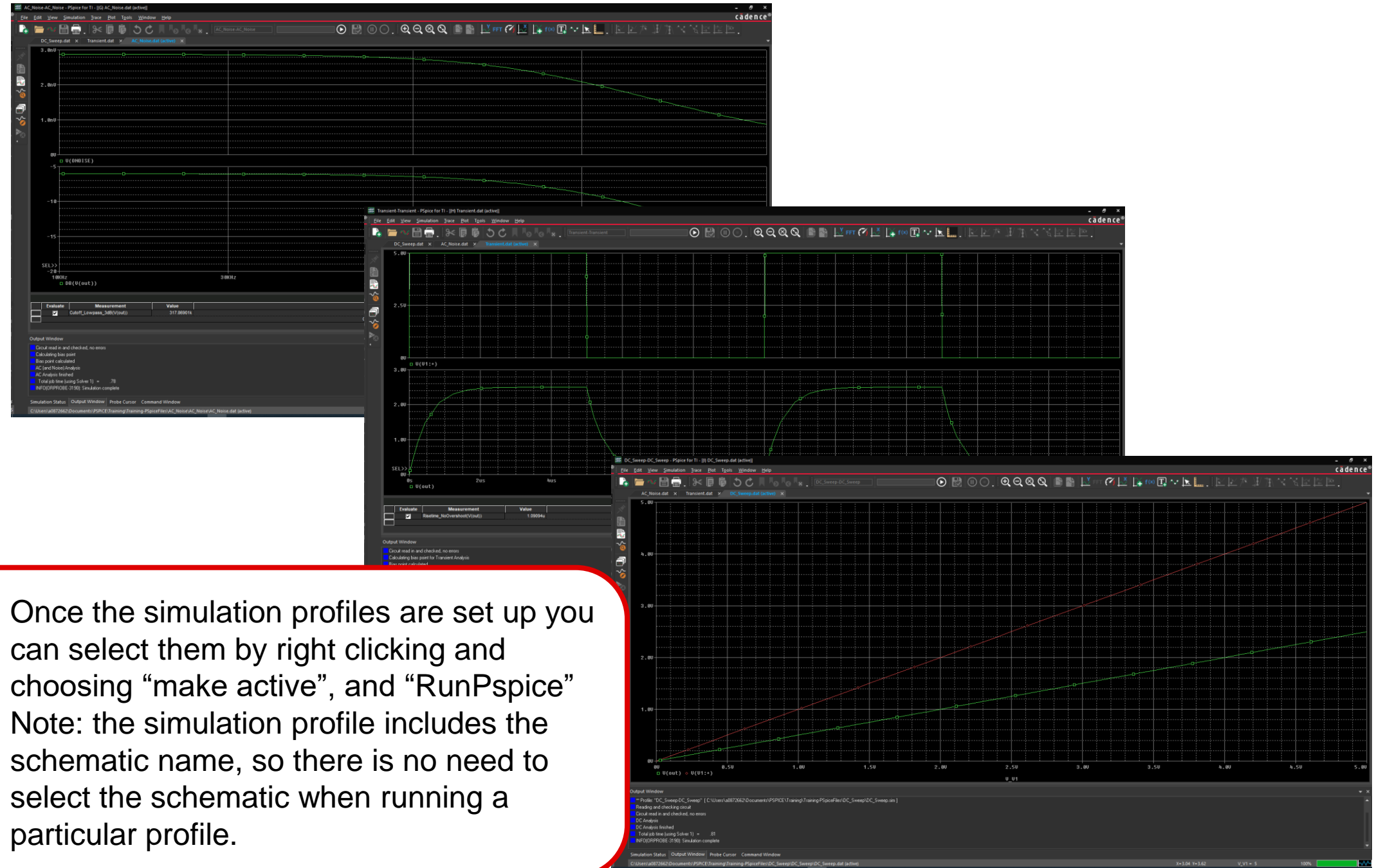
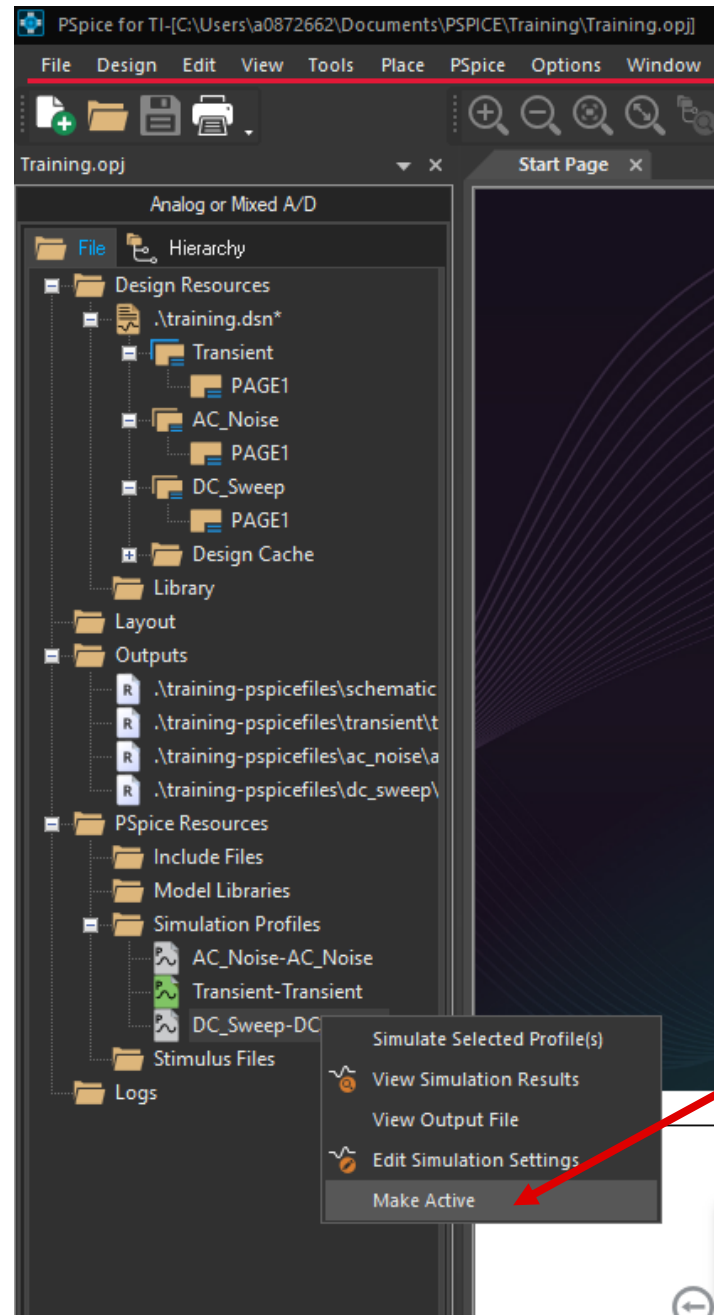
File Design Edit View Tools Place PSpice Options Window Help

Training.opj Start Page / - (DC_Sweep : PAGE1) AC_Noise : PAGE1

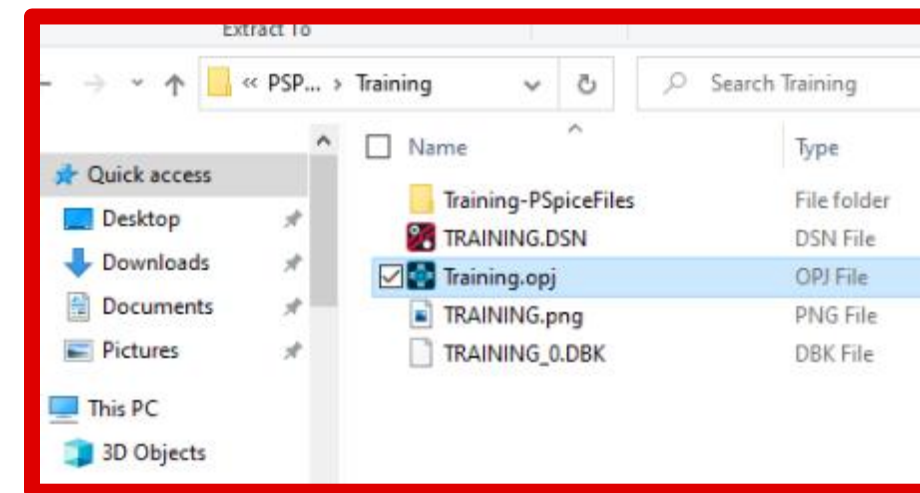
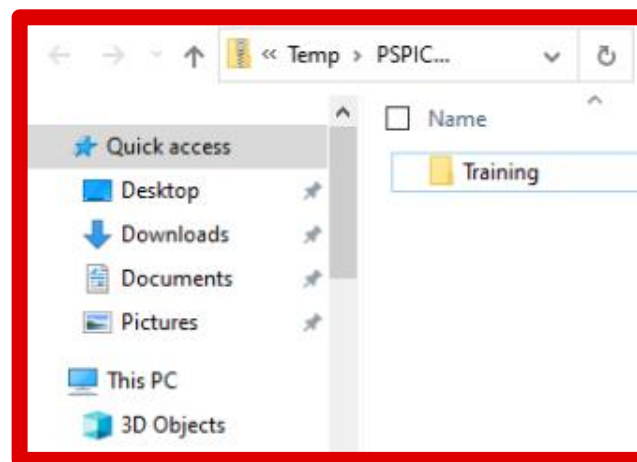
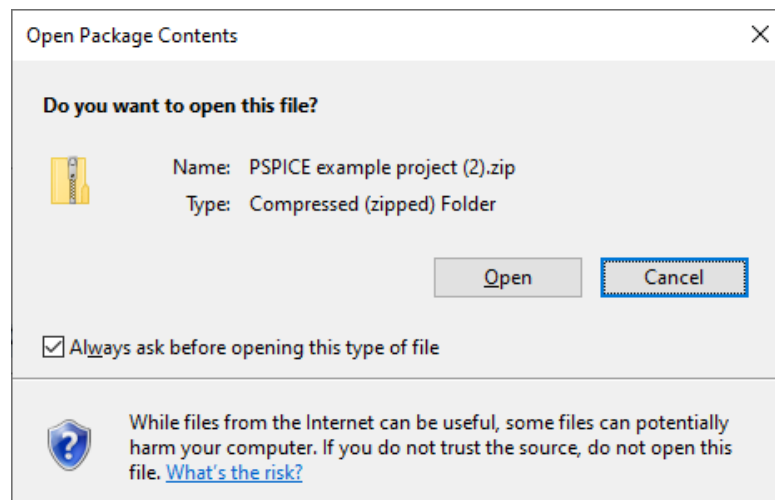
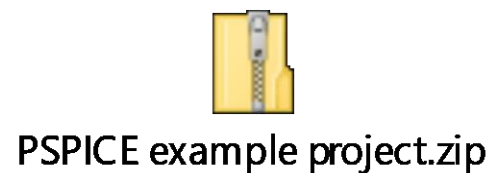
5. Select the page, and enter the schematic.

6. Create a simulation profile for the current root schematic. Not that in this example it is "DC Sweep"

Using multiple schematics and simulation profiles

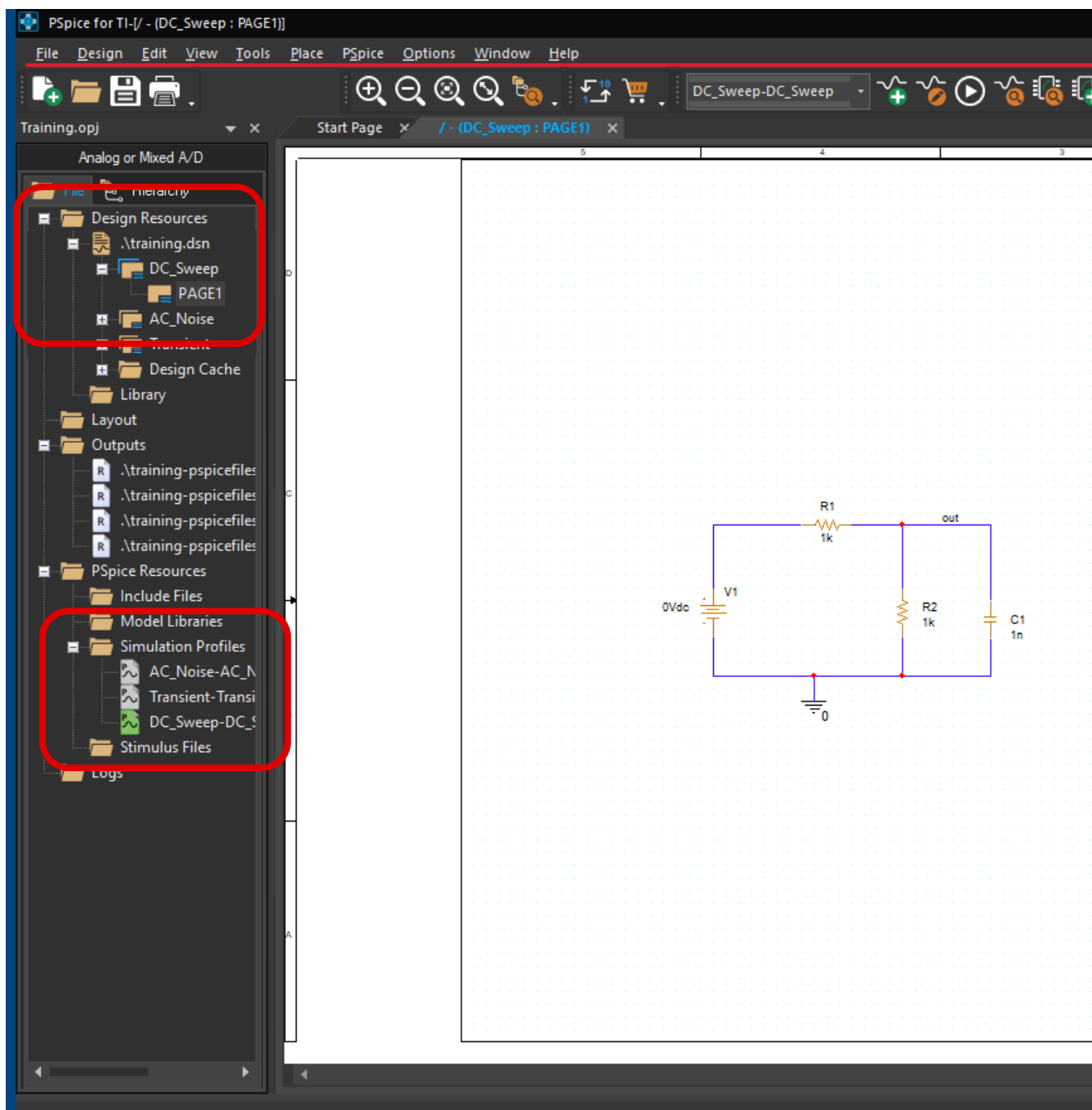


Using multiple schematics and simulation profiles



1. Click on the imbedded file above.
2. Open the file.
3. Drag the training folder to the location where you keep you PSPICE projects.
4. Click on "training.opj" to start the project

Using multiple schematics and simulation profiles



1. This is what the example project should look like once opened.
2. Select the different schematics under “training.dsn”
3. Choose the different simulation profiles and run them under “simulation profiles”

Getting professional PSPICE

[Confluence instructions for installing full PSPICE from JC Zhu](#)

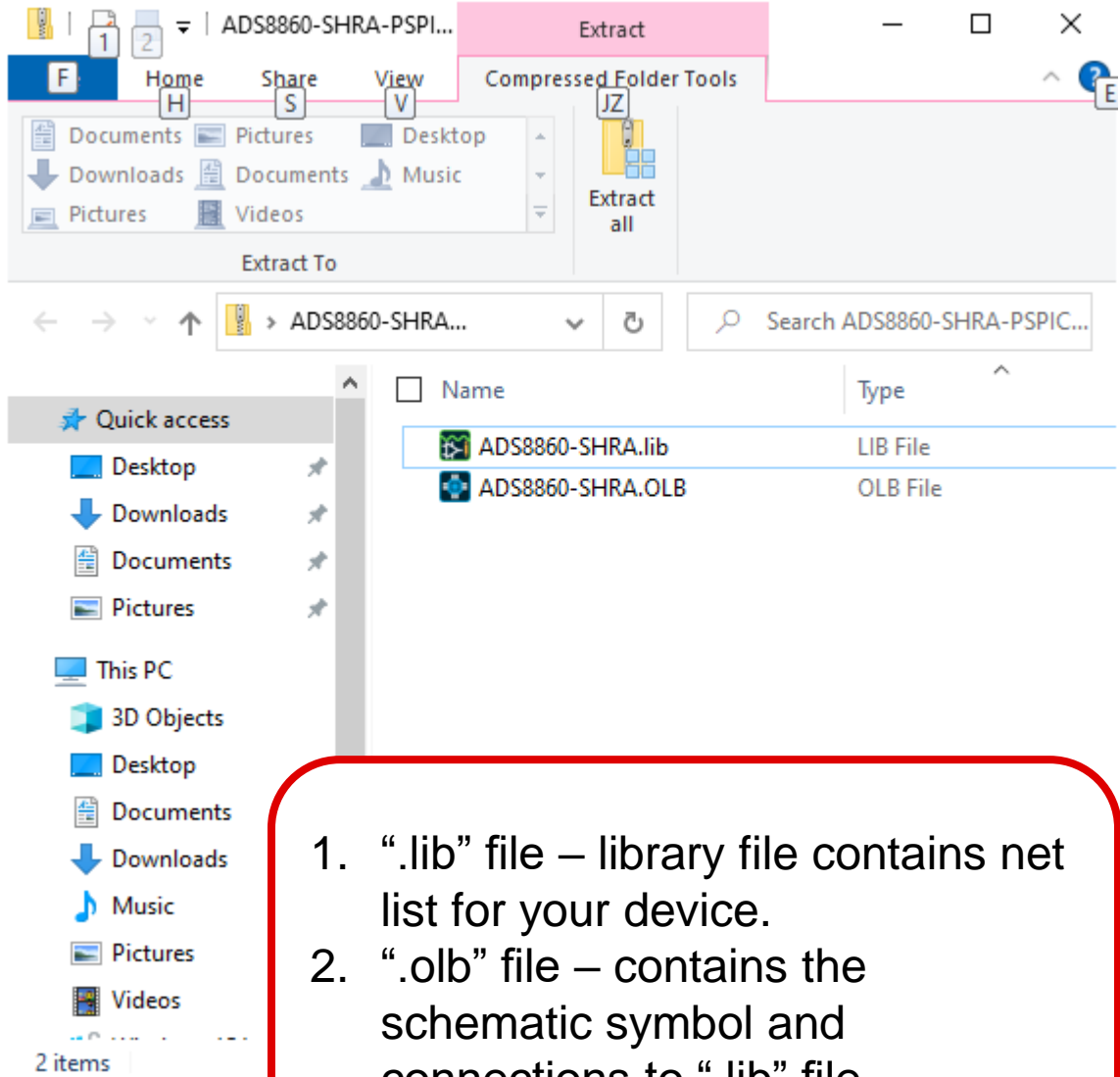
Why you may want the full version

- You need to run Non-TI models. If this is the case, the PSPICE for TI is very limited (three probes)
- If you are developing your own models, or if you are working with a developer (e.g. design soft).
- If you want access to the PCB development tools. Since we use Altium this may not be too useful but I suppose if you had a customer design this may be helpful.

Why you may not want the full version

- It is very large and takes a long time to install
- PSPICE for TI works well if you aren't using non-ti models. You can do all analysis types (e.g. montecarlo). The only limit happens when you use non-TI models. In this case you can only monitor three signals.

Uploading a Model to the web and PSPICE TI library

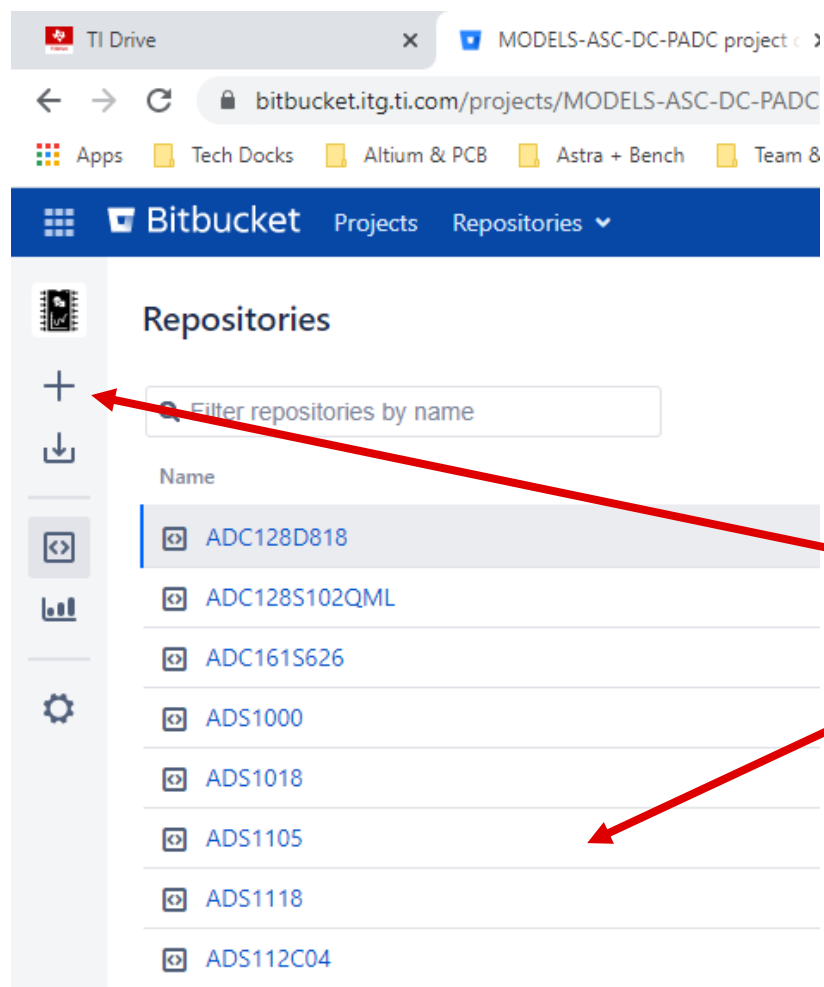


```
C:\Users\ao872662\Desktop\ADS8860_SHRA - Copy.lib - Notepad++
File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?
RegSpecTool.exe.config ADS8860.lib ADS8860.lib ADS8860_SHRA.lib ADS8860.lib ADS8860_SHRA - Copy.lib
16 *****
17 *
18 * Released by: Texas Instruments Inc.
19 * Part: ADS8860
20 * Date: 07/16/2021
21 * Model Type: TRANSIENT
22 * Simulator: PSPICE for TI
23 * Simulator Version: 17.4-2020
24 * Datasheet: SLUSD31A -OCTOBER 2018-REVISED DECEMBER 2018
25 *
26 * Model Version: 1.10
27 *
28 ** Modle created Art Kay & DesignSoft, Inc.
29 *****
30
31 .SUBCKT ADS8860-SHRA INP INM Tconv ResetSH SampleRate Tacq Vref_err Vsamp
32 + Vsettling_err Vref_ideal Vref
33 + PARAMS:
34 + Csh=55p Cdio=4n Cres=4n Rsh=96 tConv=710n Vref=5 fSampl=1MEG Iref=300u N=16 Iref_c
35 .PARAM Rt= {1
36 .PARAM Rres=
37 .PARAM Krcr=
38 .PARAM Crcr=
39 .PARAM Rd1=
40 .PARAM Rd2=
```

3. Note that the .lib file has the SUBCKT netlist. Also note that the name of the SUBCKT matches the name of the .lib file and the .olb file.

Uploading a Model to the web and PSpICE TI library

[bitbucket repository for ADC model files](#)
[Confluence Bit Bucket Tutorial](#)



4. Select the appropriate repository. These have been pre-populated for all device names. If you need to make a custom model name press the “+” sign to add a new repository.

Uploading a Model to the web and PSPICE TI library

TI Drive | bitbucket.itg.ti.com/projects/MODELS-ASC-DC-PADC/repos/ads8860/browse

Bitbucket Projects Repositories

MODELS-ASC-DC-PADC / ADS8860

Source

master ADS8860 / + New File Upload Browse Filter

19 commits 1 branch 0 releases 4 contributors

Source	Description	Last Modified
Legacy-Publish		
ads8860_PSPICE_6_22_2022.zip	MDL-7024	23 Jun 2021
ads8860_PSPICE_8-10-2021.zip	MDL-7345	5 days ago
ads8860_PSPICE_8-6-2021.zip	Uploaded files	06 Aug 2021
Readme.md	MDL-7345	5 days ago

Readme.md Edit README

REPOSITORY EXAMPLES: <https://bitbucket.itg.ti.com/projects/MODELS>

Example_Pspice /

Source

Development-Pspice

LM4030_NA2P5_PSPICE_TRANS.zip

Example_TINA /

Source

Development-TINA

LM4030_NA2P5_TINA_TRAN.zip

For individual GPN repository, the published file to the web can be put in the repository root.

For "Simulator" folder within the GPN folder.

TI Drive | bitbucket.itg.ti.com/plugins/servlet/uploadFile/projects/MODELS-ASC-DC-PADC/repos/ads8860?at=refs%2Fheads%2Fmaster

Bitbucket Projects Repositories

MODELS-ASC-DC-PADC / ADS8860

File upload to /

Drop files here to add to the repository
Or choose your files

Uploaded files

Commit directly to the repository
Branch and start a pull request

Commit

Open

This PC > Desktop >

can not place PSPICE model.jpg
hot fix.jpg
ADS8860-SHRA-PSPICE-8-11-2021.zip
BOOSTXL-ADS1219.pdf
ADS8860_SHRA - Copy.lib
ADS8860_SHRA.lib
latest_pspice_adc_model-8-6-2021.zip
53622_ADS8555EVM.pdf
opa625vsOPA333.zip
try lib again.zip
Camtasia - Shortcut
Team
designs
TIPL

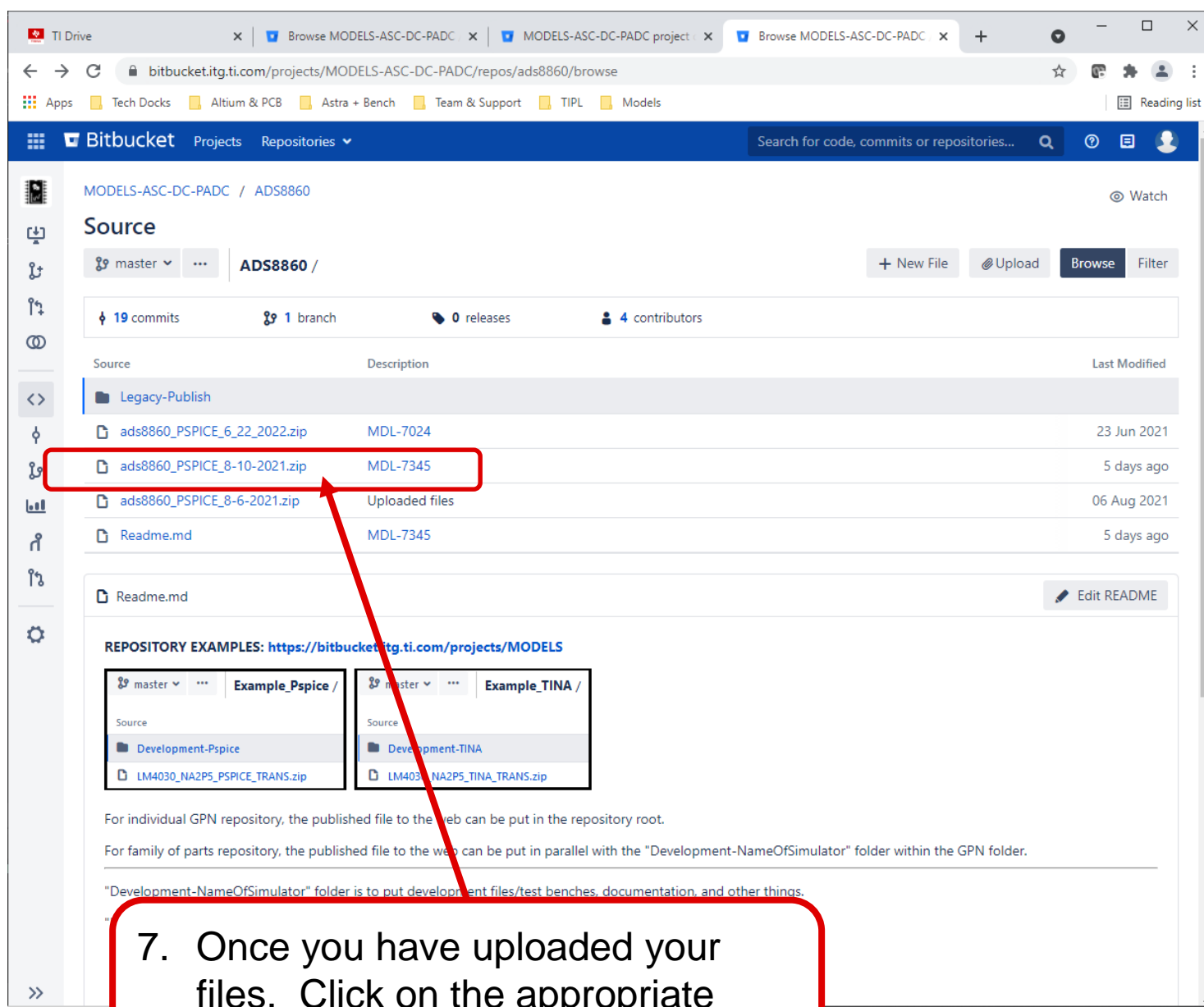
File name: All Files (*.*)

Open Cancel

5. Press upload to upload your ZIP file repository.

6. Click on "choose your files" and select your ZIP file to upload.

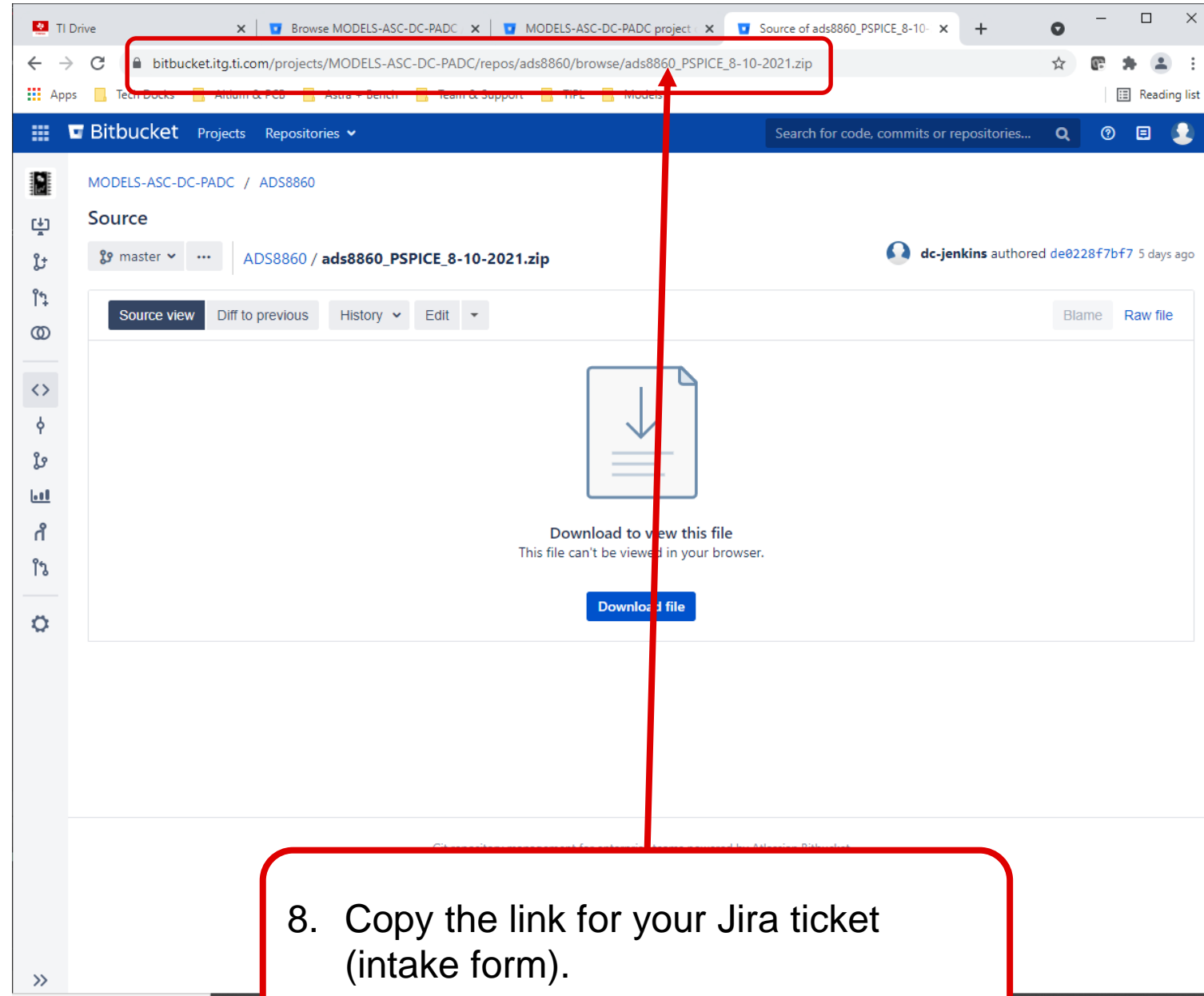
Uploading a Model to the web and PSPICE TI library



The screenshot shows a Bitbucket repository page for 'MODELS-ASC-DC-PADC / ADS8860'. A table lists files with columns for Source, Description, and Last Modified. The file 'ads8860_PSPICE_8-10-2021.zip' is highlighted with a red box. A red arrow points from this box to a callout box below.

Source	Description	Last Modified
Legacy-Publish		
ads8860_PSPICE_6_22_2022.zip	MDL-7024	23 Jun 2021
ads8860_PSPICE_8-10-2021.zip	MDL-7345	5 days ago
ads8860_PSPICE_8-6-2021.zip	Uploaded files	06 Aug 2021
Readme.md	MDL-7345	5 days ago

7. Once you have uploaded your files. Click on the appropriate reparatory



The screenshot shows the Bitbucket file view for 'ads8860_PSPICE_8-10-2021.zip'. The URL in the browser's address bar is highlighted with a red box. A red arrow points from this box to a callout box below.

8. Copy the link for your Jira ticket (intake form).

Uploading a Model to the web and PSPICE TI library

Confluence Model Intake Form makes Jira ticket

Example of completed Jira Ticket

Model Request Intake Form

Created by JC Zhu, last modified on Jul 25, 2021

Choose from below options:

Select Engineering Need *

- Register Product Model Development**
 - New model
 - Publish existing model
 - Revise existing model (bug)
 - Translate Model to another simulator (LTSpice is not supported)
- Get General Help**
 - Request access to unpublished model (e.g. unencrypted, restricted)
 - Generate training plan
 - See tickets assigned to me (JIRA query shortcut)
 - List all open tickets by SBE2 (JIRA query shortcut)

Modeling Needs *

- Develop a new model
- Publish / release a model that has been completed
- Revision to existing model due to bug OR change in model requirement / product specification
- Translation (simulation platforms with limited support)
- Translation (supported simulation platforms)

Reason for Publishing *

- This is a new model (never published to ti.com)
- This is a revision to an already published model due to change in encryption status, CIP level, or access classification

Select the Product Line *

SBE

ASC

SBE1

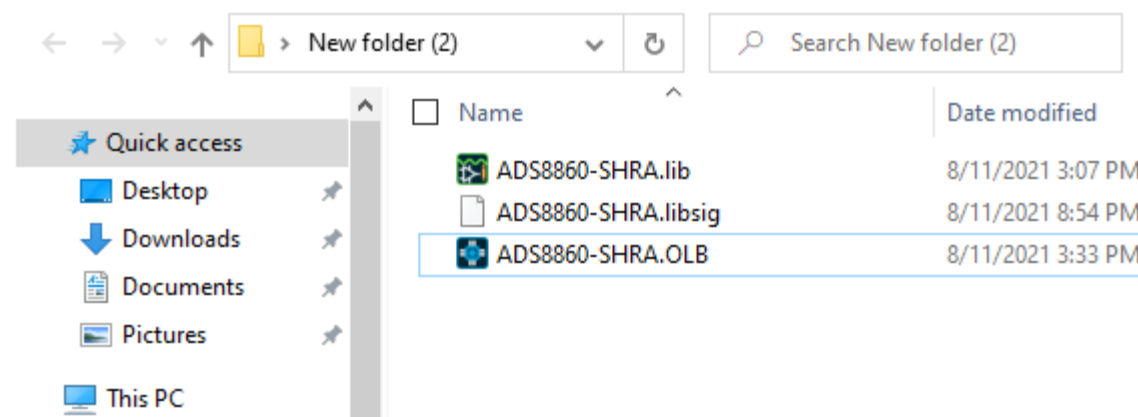
DC

SBE2

DC-PADC

Visit [SC Marketing Automated Services](#) to look up the product line (listed as DesignWIN Hierarchy) owning the underlying TI product by searching the GPN.
Product line modeling appointees are listed on the [modeling appointee page](#).

9. Click on the link to start your Jira ticket for model creation.
10. The form questions are mostly self explanatory. At one point in the form you will need to past in the bit bucket repository link from the previous slide.
11. A few hours after the files are uploaded, a signed copy of your files will automatically be uploaded back to the bit bucket repository. You can copy these to your computer and this will enable you to use PSPICE for TI without limitations. If the .lib is not signed, PSPICE for TI considers the model to be non-TI and limits you to three probes. See signed library below:



Uploading a Model to the web and PSPICE TI library

The screenshot shows a Jira issue page for 'ADS8860 PSPICE (Publish New Model)' with the following details:

- Issue ID:** MDL-7365
- Type:** General Support
- Priority:** Green
- Component/s:** PSPICE
- Status:** CLOSED (View Workflow)
- Resolution:** Published
- Security Level:** CAE Security Level
- Labels:** form-submission, release-pf-model
- Product:** ADS8860
- SBE:** ASC
- SBE1:** DC
- SBE2:** DC-PADC
- Analysis Type:** Generic
- Requesting Source:** Customer Request - Non CSC
- Development Team:** Product Line
- Deliverable Access:** Open
- Deliverable CIP:** Not Applicable
- Deliverable Encryption:** Unencrypted
- Deliverable Repository:** <https://bitbucket.itg.ti.com/projects/MODELS-ASC-DC-PADC/repos/ads8860-shra/browse/ADS8860-SHRA-PSPICE-8-11-2021.zip>
- Datasheet Information:** UNKNOWN
- Reason:** New model
- Release Notes:** curated
- Model Name:** ADS8860_SHRA
- Symbol Name:** ADS8860_SHRA
- OLB File Name:** ADS8860_SHRA
- CIP Classification:** TI Information - Selective Disclosure
- GPN:** ADS8860
- Deliverable Lit Number:** SBAM457

At the top of the issue page, there are buttons for 'Comment', 'Assign', 'More', 'Re-package', 'Re-New', and 'Un-Publish'. Red arrows point from a text box to the 'Re-package' and 'Published' buttons.

12. You will get Jira emails regarding the status. Press the buttons at the top to change the status.

Uploading a Model to the web and PSPICE TI library

The screenshot shows a Jira issue page for 'MDL-7365' in the 'Modeling' project. The issue is filtered by 'Reported by me'. The left sidebar shows navigation options like 'Copy of DSS Modeling Priorit...', 'Backlog', 'Kanban board', 'Releases', 'Reports', 'Issues', 'Components', 'Timesheets', and 'Add-ons'. The main content area shows a list of comments:

- Mindy Mitchell added a comment - 3 days ago: "Hi Art Kay and Collin Wells, Thanks for the explanation. I'll ask about adding this PSpice type to the guidelines for consistent naming. But for now... The file has been loaded and should be active on the web within the next 24-48 hours. Thx, Mindy"
- DCIT Jenkins added a comment - 3 days ago: "Publishing successful : <https://bitbucket.itg.ti.com/projects/MODELS-ASC-DC-PADC/repos/ads8860-shra/browse/ADS8860-SHRA-PSPICE-8-11-2021.zip>"
- DCIT Jenkins added a comment - 3 days ago: "PSpice library curation successful: <https://bitbucket.itg.ti.com/projects/MODELS-ASC-DC-PADC/repos/ads8860-shra/browse/ADS8860-SHRA-PSPICE-8-11-2021.zip>"
- Art Kay added a comment - 2 days ago: "JC Zhu, I see the new device in the model library, but I cannot place it into a schematic (ADS8860-SHRA). I can place a previously created model (ADS8860)."

Below the comments is a table with the following data:

Literature Number	Document Type	View Doc	Concise Description
SBAM457	PSpice Model	View	ADS8860 PSpice Model (SHRA)

Below the table is a screenshot of a Cadence PSpice schematic showing an error message: "PSpice cannot place ADS8860-SHRA because the component is not available in the library. To use, see 'Project Setup' page 'PSpice ADS8860-SHRA.lib'. Search for the component to present in the library." Below the schematic is a 'Comment' input field.

13. Use comments to send messages to key people involved in model release. Note that "DCIT Jenkins" is an automated response. JC Zhu's team is a good contact for issues. Also launch marketing (e.g. Mindy Mitchel) is also a good contact.

Thanks for your time!

Choose from below options:

Select Engineering Need *

- Register Product Model Development**
 - New model
 - Publish existing model
 - Revise existing model (bug)
 - Translate Model to another simulator (LTSpice is not supported)
- Get General Help**
 - Request access to unpublished model (e.g. unencrypted, restricted)
 - Generate training plan
 - See tickets assigned to me (JIRA query shortcut)
 - List all open tickets by SBE2 (JIRA query shortcut)

Modeling Needs *

- Develop a new model
- Publish / release a model that has been completed
- Revision to existing model due to bug OR change in model requirement / product specification
- Translation (simulation platforms with limited support)
- Translation (supported simulation platforms)

Reason for Publishing *

- This is a new model (never published to ti.com)
- This is a revision to an already published model due to change in encryption status, CIP level, or access classification

Select the Product Line *

SBE

ASC

SBE1

DC

SBE2

DC-PADC

Visit [SC Marketing Automated Services](#) to look up the product line (listed as DesignWIN Hierarchy) owning the underlying TI product by searching the GPN.
Product line modeling appointees are listed on the [modeling appointee page](#).

Digital Marketing Contact *

Mindy Mitchell (a0208213)

Visit this [page](#) and enter the launch manager (for new model) or the web manager (for revising a model) here.

Product (Recorded in Galileo) *

ADS9110

!! Value MUST equal Pink Folder Name as found in Galileo!!
If you do not have access to Galileo, enter GPN

Visit this [page](#) and enter the launch manager (for new model) or the web manager (for revising a model) here.

Product (Recorded in Galileo) *

!! Value MUST equal Pink Folder Name as found in Galileo!!

If you do not have access to Galileo, enter GPN

Literature Concise Description *

Enter the description of the model. This will be displayed on ti.com.

Product Folder(s) *

Enter the name of the product folder where models should be displayed. This may not be identical to the part number. If the model should show up in multiple product folders, list the names comma-separated.

BitBucket URL *

Enter the URL to the file to be released (must be checked into the **modeling BitBucket**).
The repository and file names are case-sensitive so please copy and paste from your web browser.
Spaces in the folder hierarchy and file names are not allowed.
User "DC Jenkins Build" must have read/write permission to the repository.
The only file formats supported are: *.zip, *.tsc and *.sxcmp.
For more details, refer to [the BitBucket tutorial](#).

Requesting Source *

Simulator *

OLB File Name *

Please enter the name of the PSpice OLB file (without the .olb extension). This is required for adding model into PSpice library for TI parts.

Symbol Name *

Please enter the name of the PSPICE symbol (must be in the OLB file above). This is required for adding model into PSpice library for TI parts.

Model Name *

Please enter the name of the PSPICE model (usually the top-level subckt). This is required for adding model into the PSpice library of TI parts.

Analysis Type *

- Average
- Transient
- Generic

Please select the analysis supported by the model (average, transient ...).

OPJ (Project) File



for TI parts.

Symbol Name *

Please enter the name of the PSPICE symbol (must be in the OLB file above). This is required for adding model into PSpice library for TI parts.

Model Name *

Please enter the name of the PSPICE model (usually the top-level subckt). This is required for adding model into the PSpice library of TI parts.

Analysis Type *

Average
 Transient
 Generic

Please select the analysis supported by the model (average, transient ...).

OPJ (Project) File

If exists, enter the path to the project file (OPJ) inside the PSpice project hierarchy. E.g. "ADC3444_PSPICE_TRANS/ADC3444.opj".

Model CIP *

NDA Restriction
 Selective Disclosure
 Not Applicable


Model Access *

Open
 Restricted

Model Encryption *

Encrypted
 Unencrypted

Description *

Request on Behalf of 

If you are requesting a service for someone else, type in his/her name and select a user from the drop down that appears as you type.

Product Classification *

Submit