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!

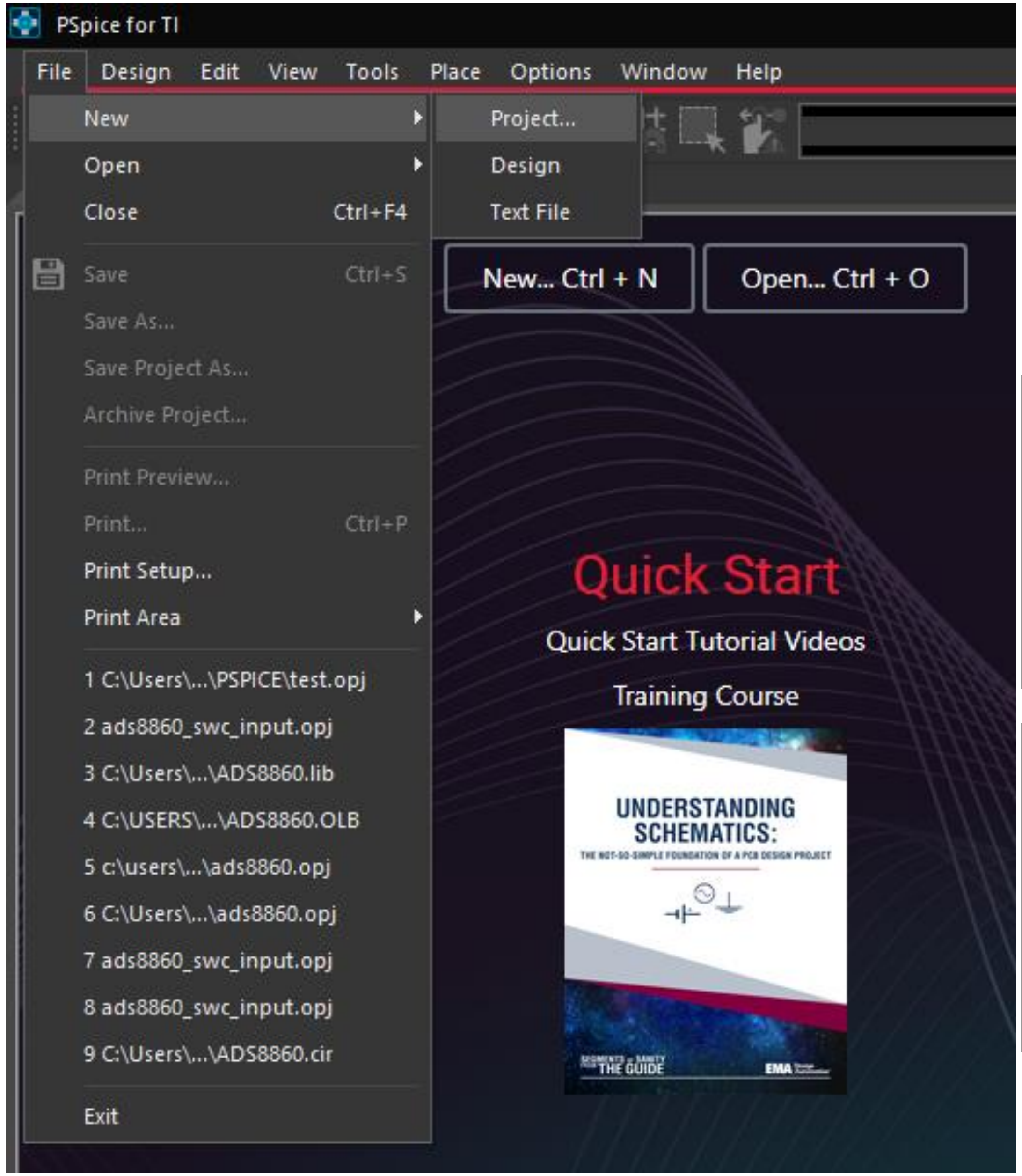
Table of contents (click link to jump to section)

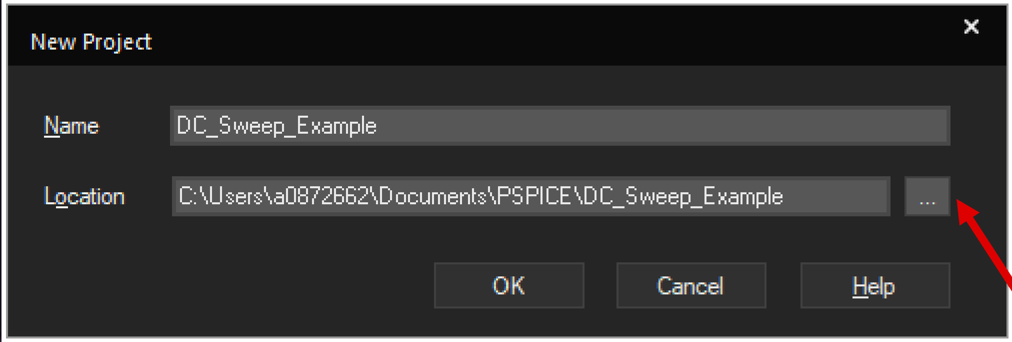
Getting and Installing PSPICE for TI

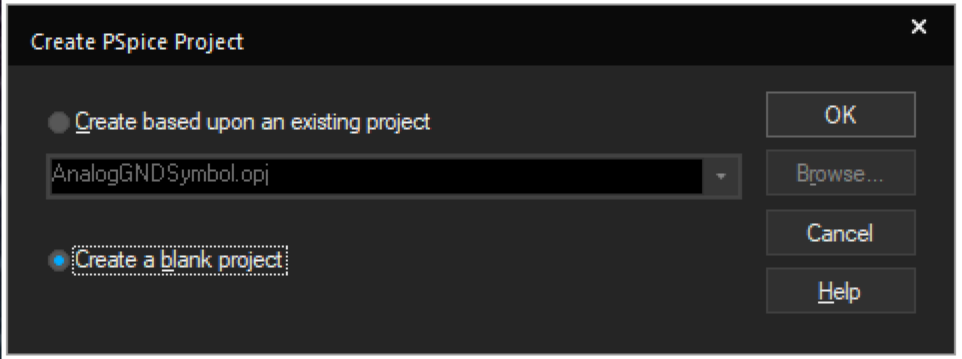
- Visit https://www.ti.com/tool/PSPICE-FOR-TI to make the request.
- Make sure your myTI information is complete, all required fields (marked with "*") are filled in.
- An email will be sent to the registered email address containing a download link and an access key.
- Click the download link to get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



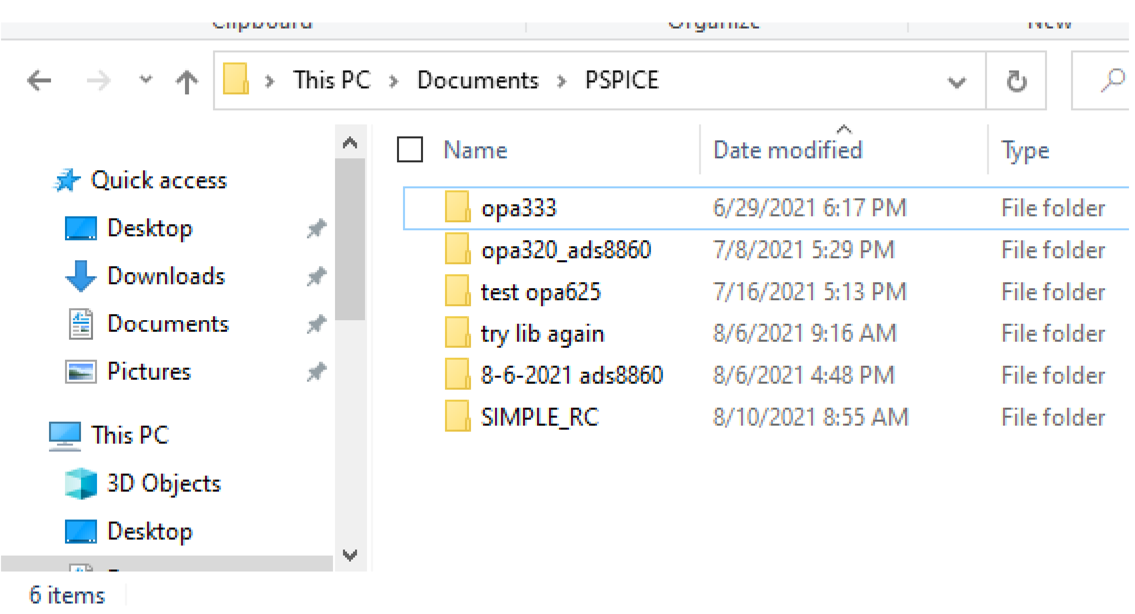


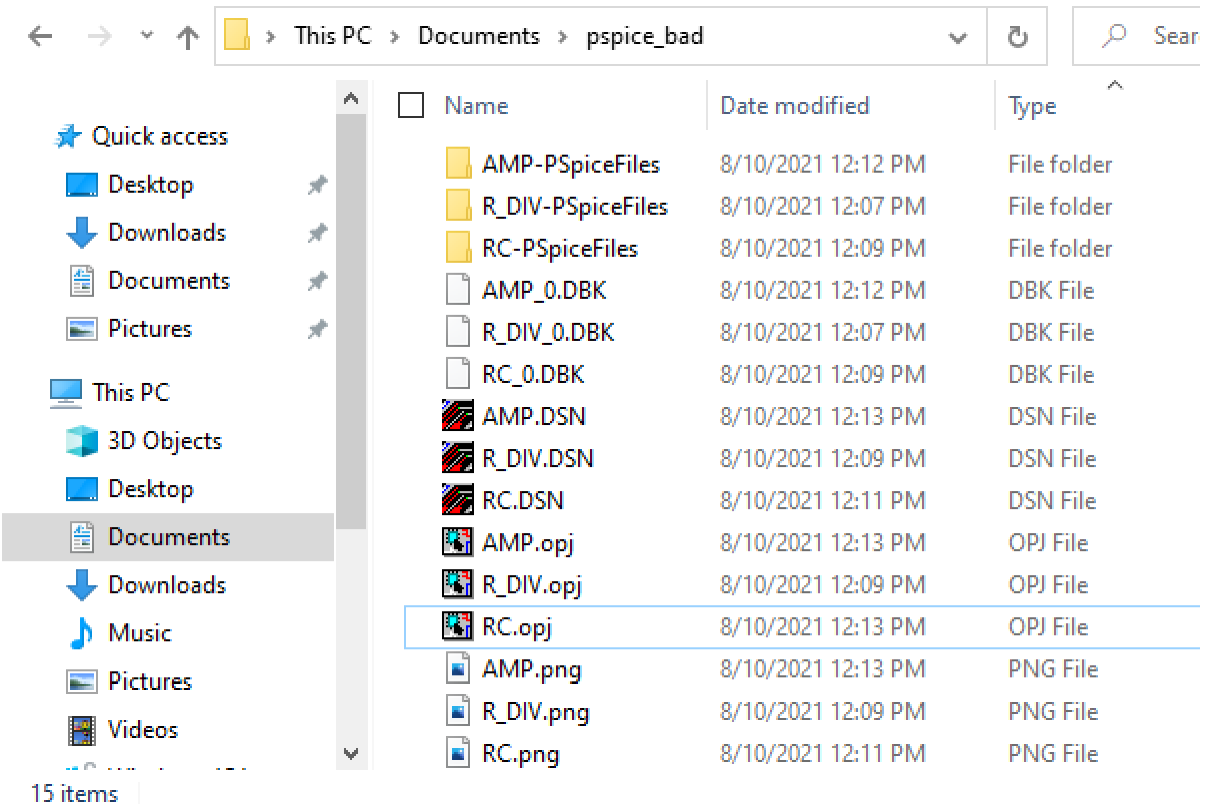


Create a unique folder for each example. Press three dots and press “New Folder” to create a folder then select it.

Back to Table of Contents

Bad vs Good structure





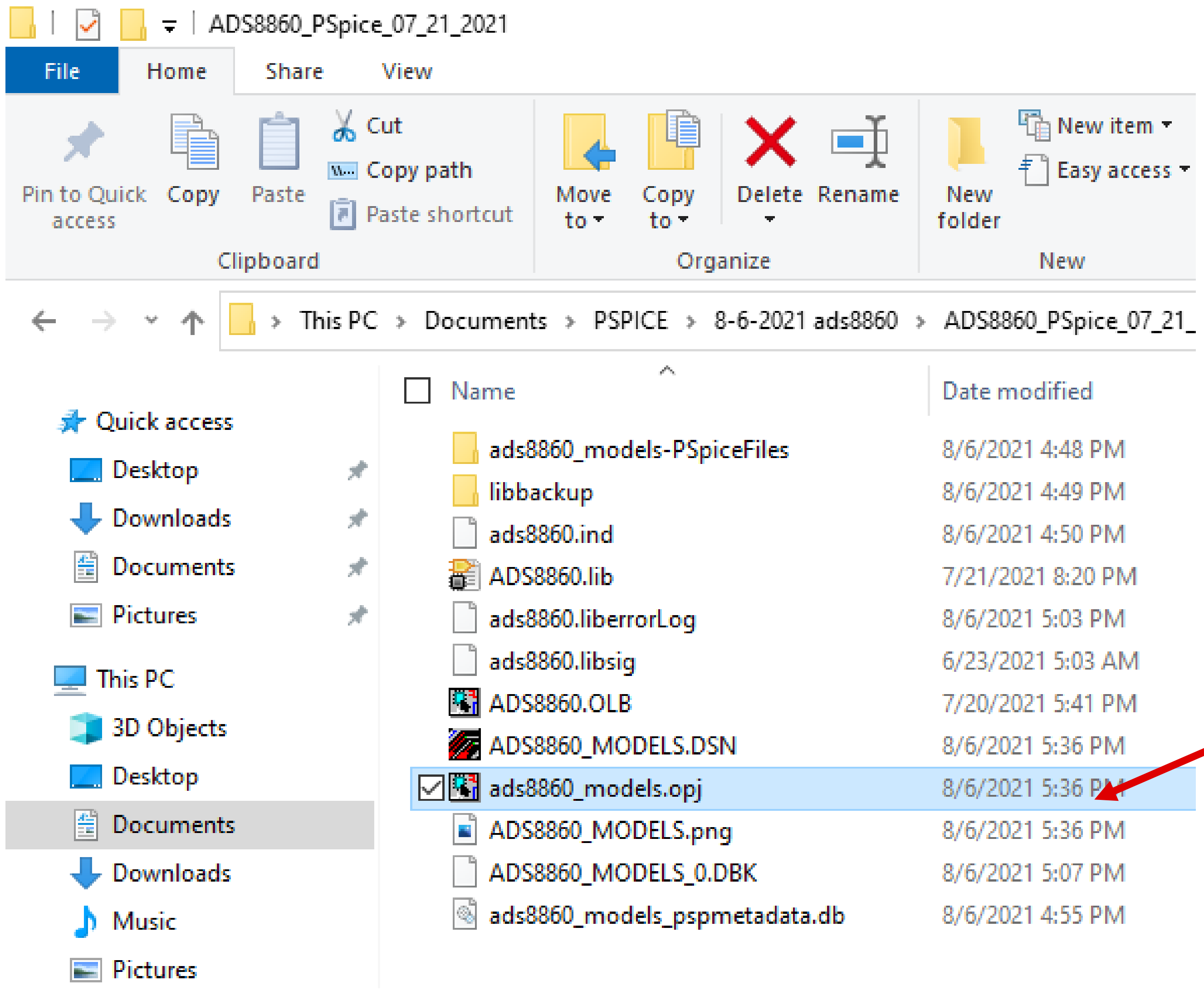
Good Structure

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

Bad Structure

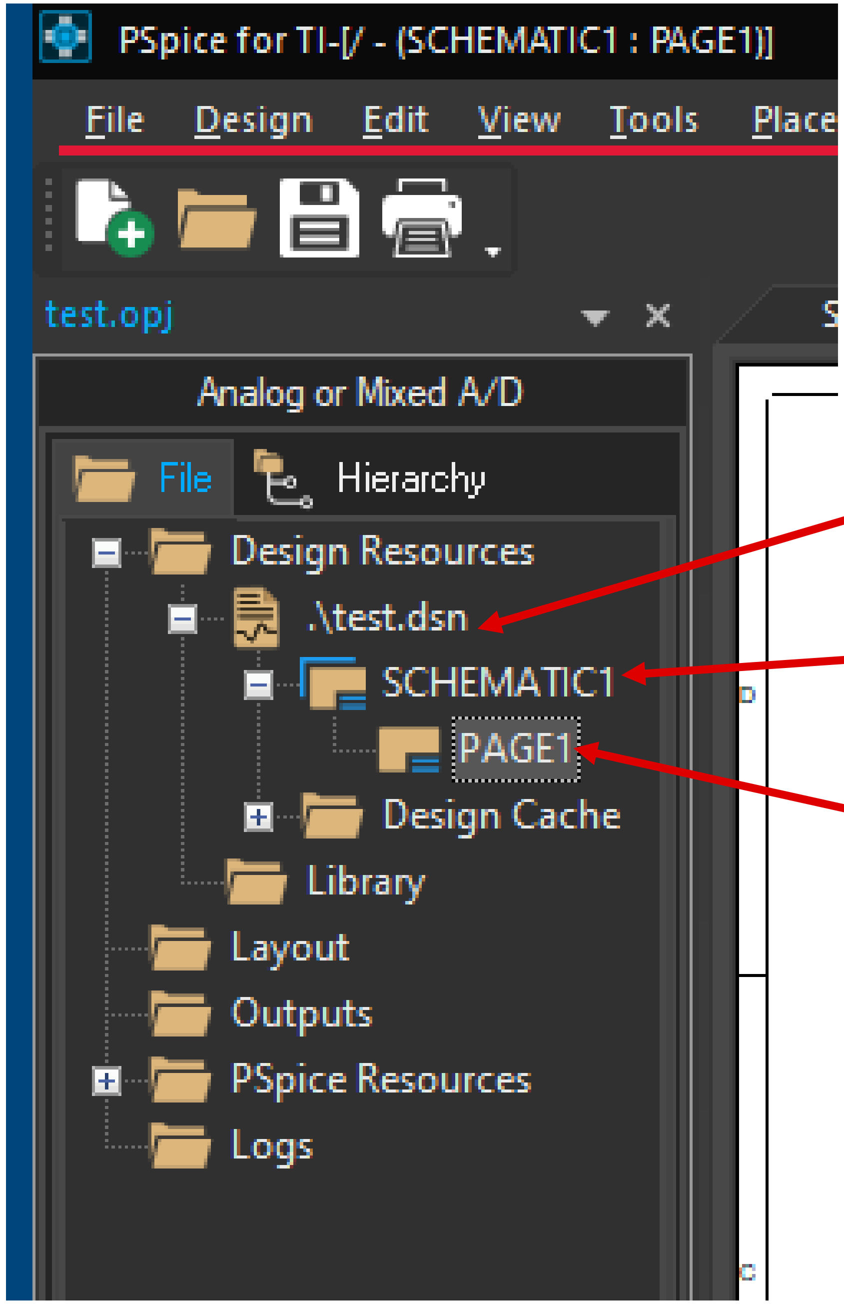
- All files mixed together
- All the PSPICE files in one folder.
- No project specific folder created automatically

Launch PSPICE from file explorer



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

Find the Schematic

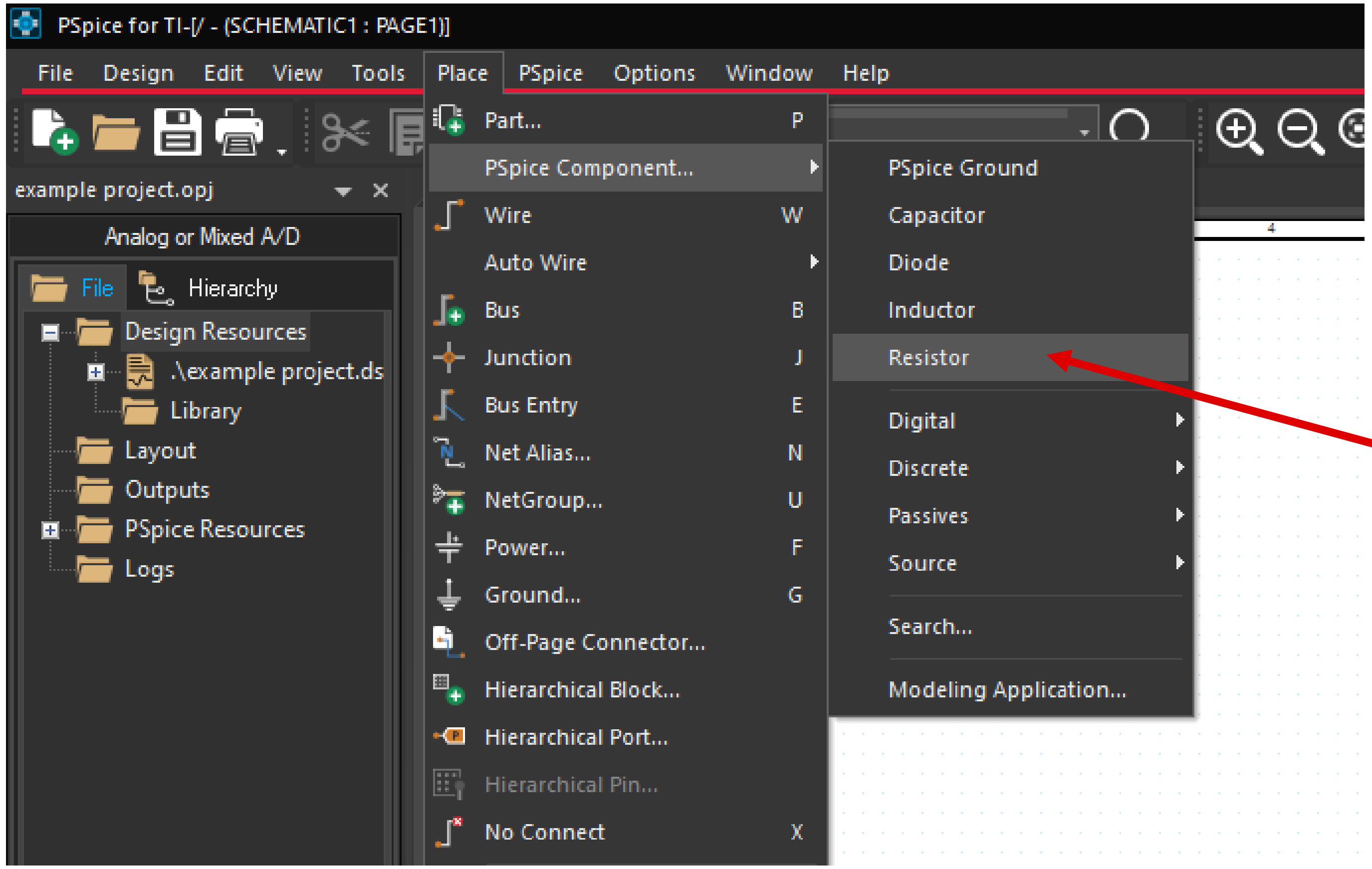


Project name

Folder containing all schematic pages. You can rename this

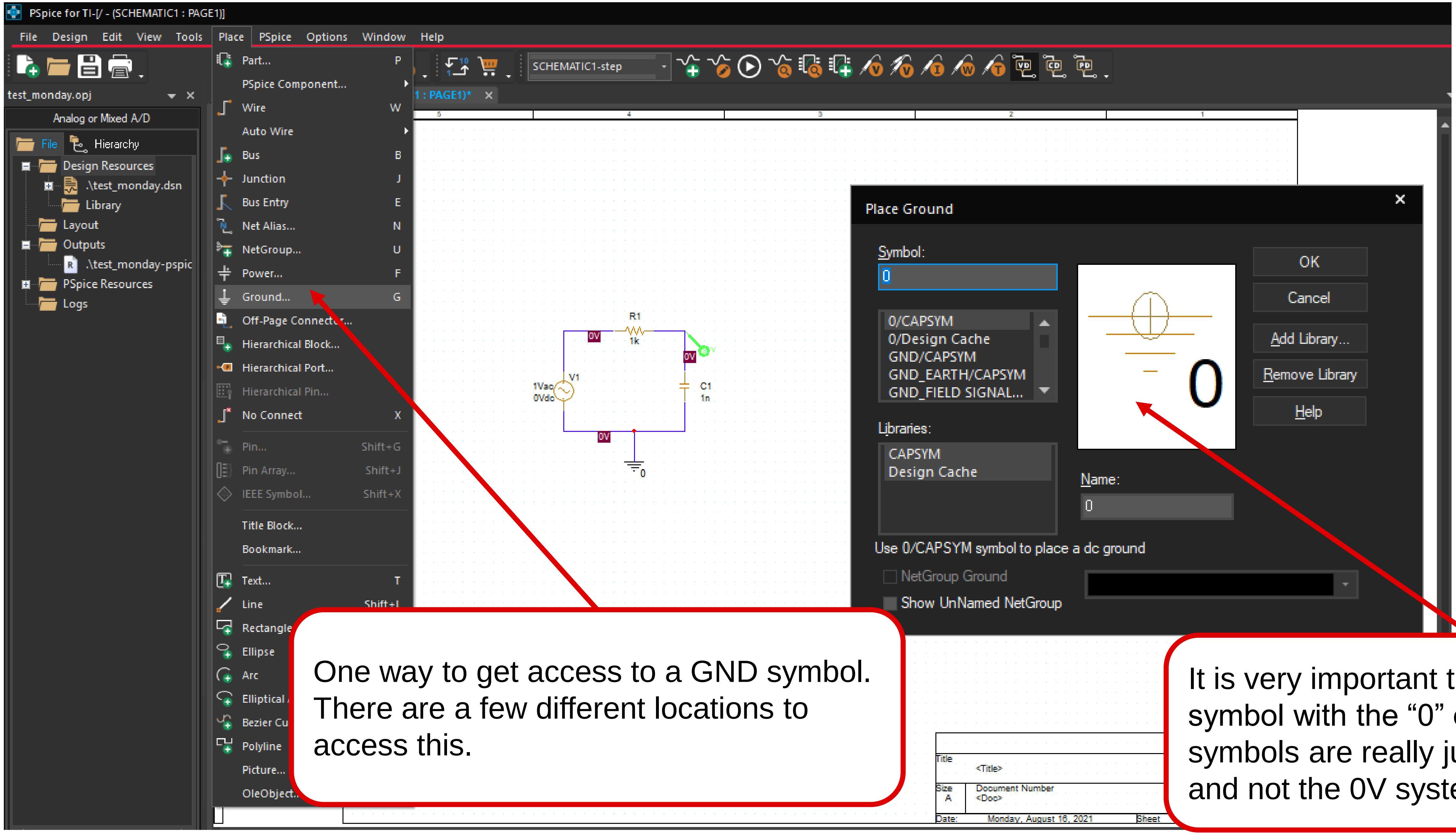
"PAGE1" is the editable first page of the schematic

Adding passive components



- Select resistor, capacitor inductor.

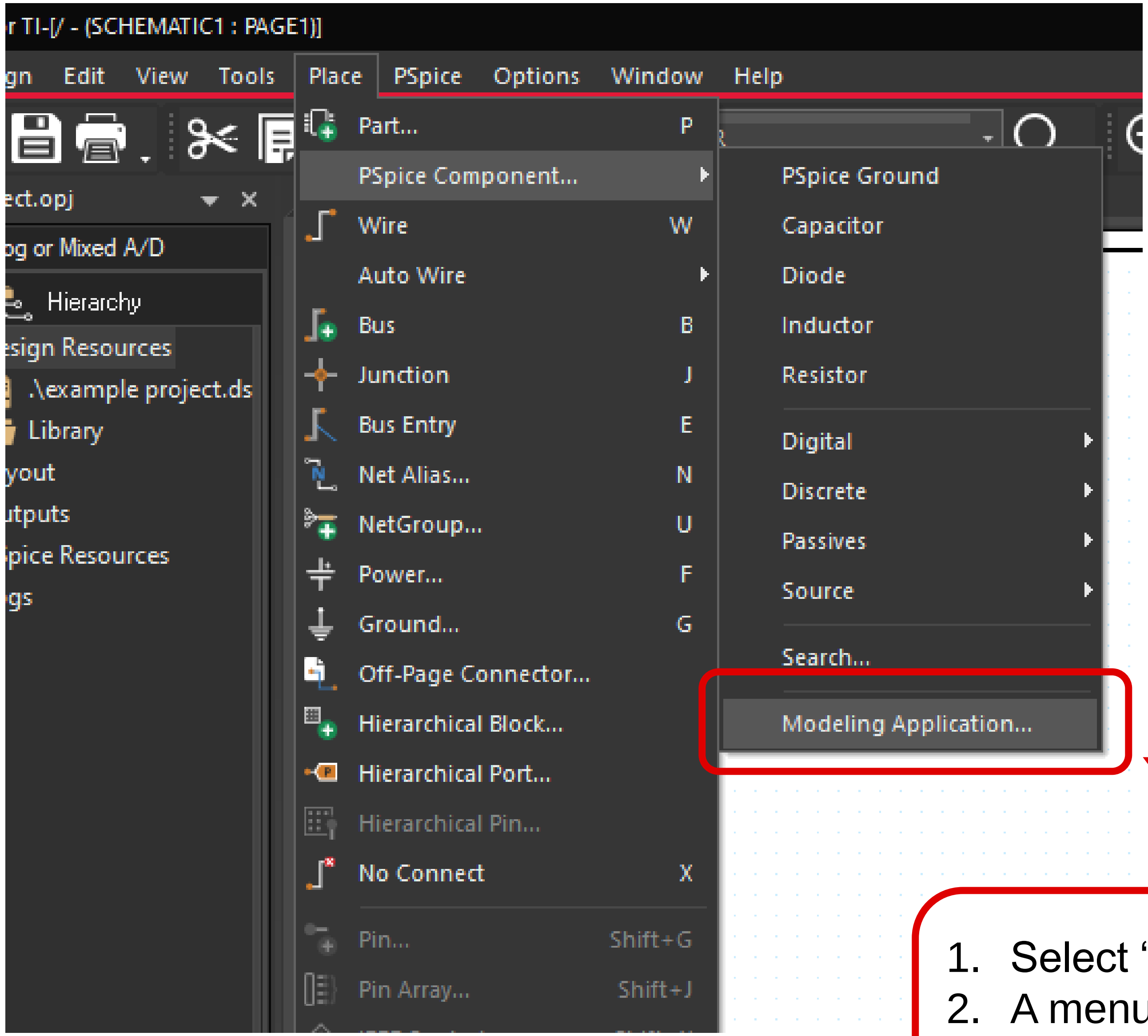
Adding GND symbol

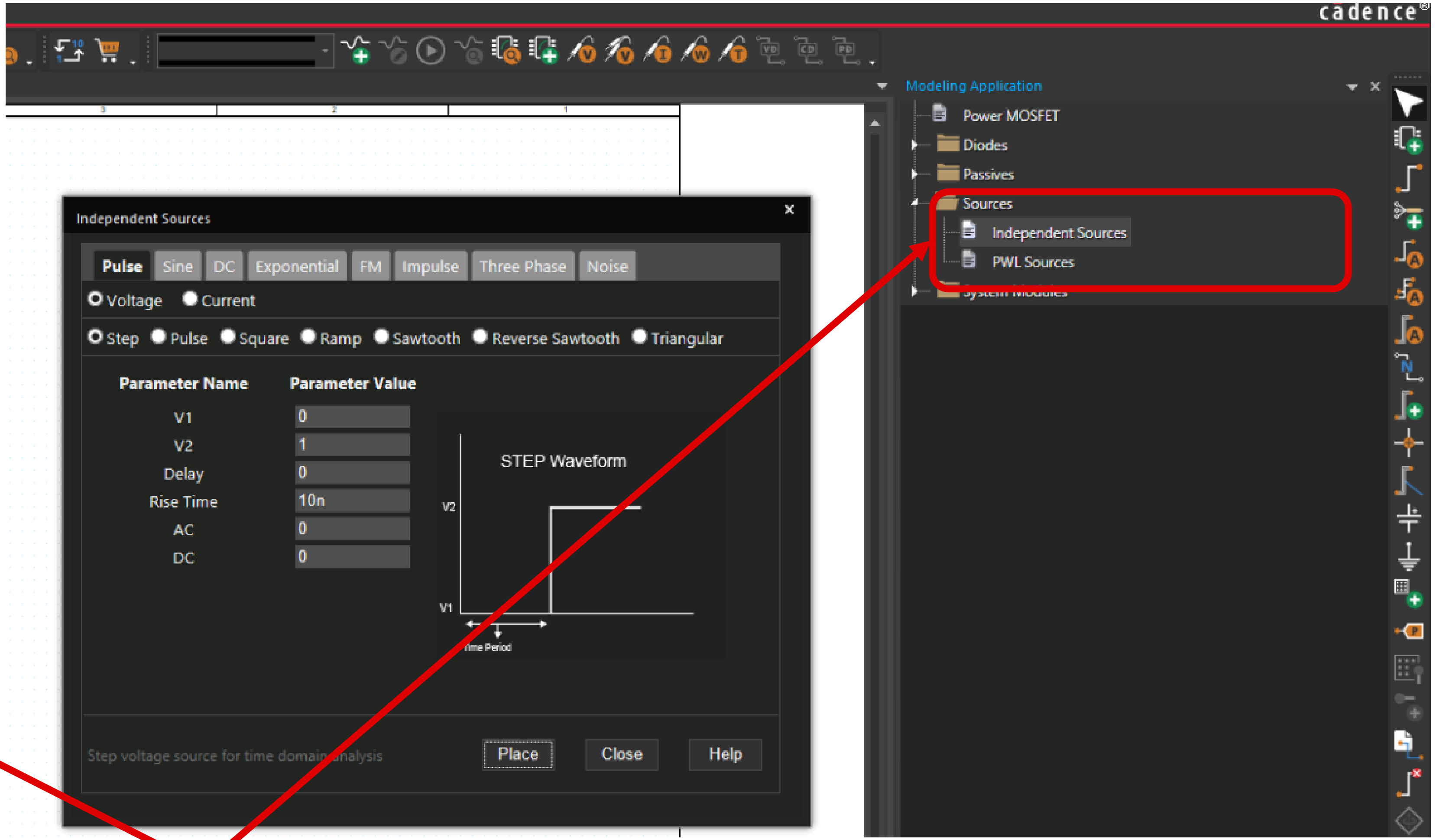


One way to get access to a GND symbol. There are a few different locations to access this.

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.

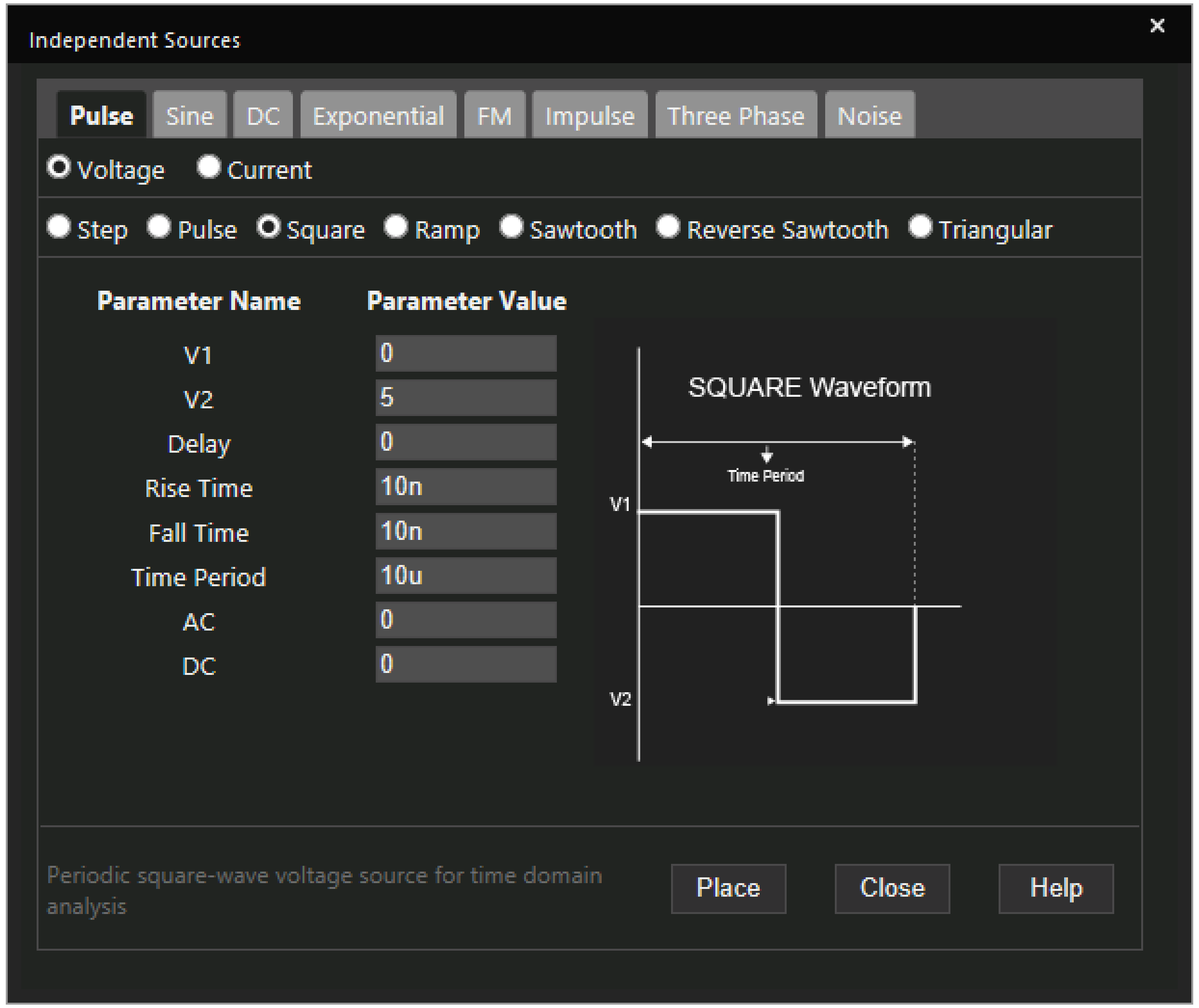
Adding sources for transient analysis

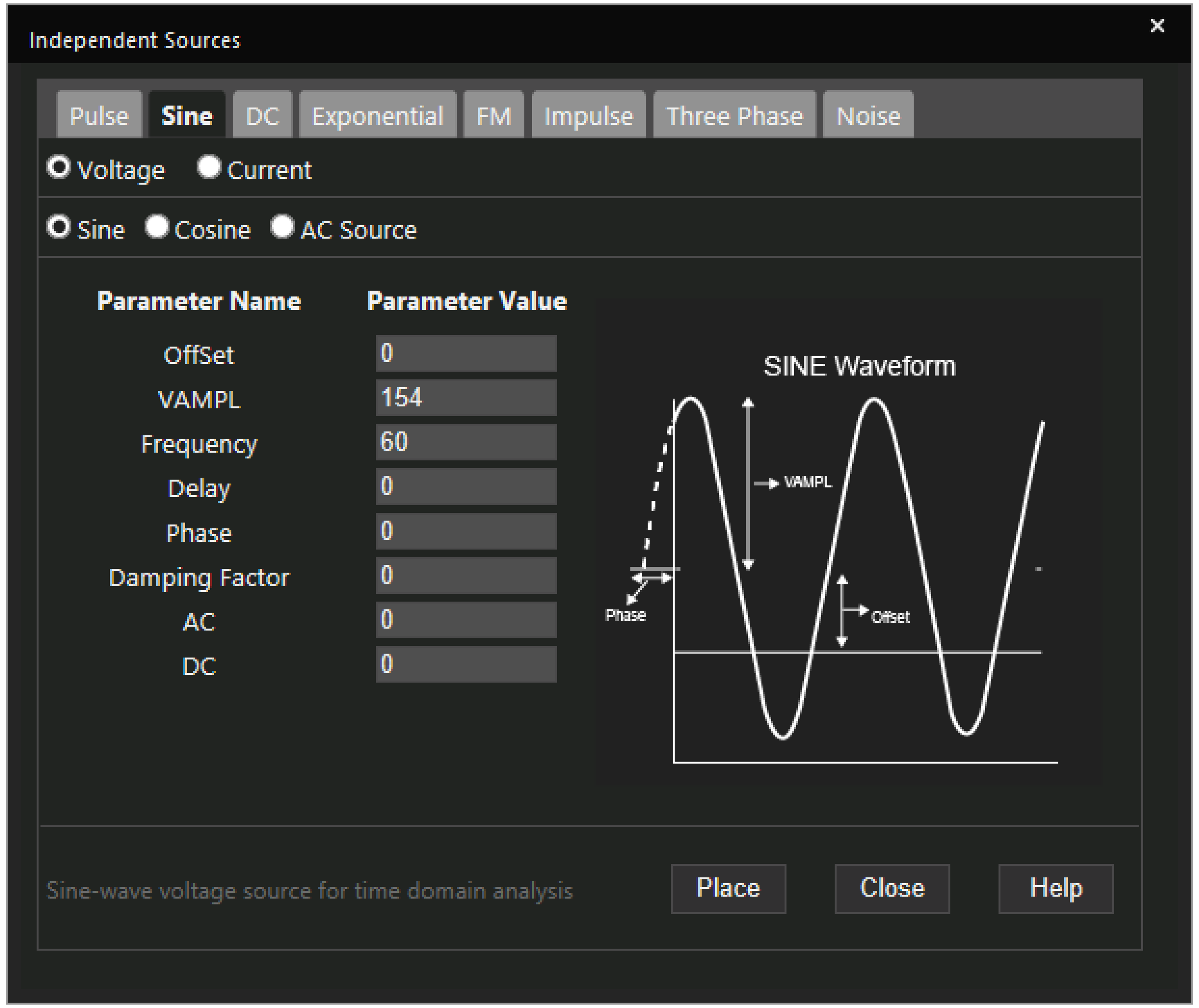




1. Select “Modeling Applications”
2. A menu will appear at the side. Choose “independent sources” for typical sources needed for transient analysis.

Adding sources continued

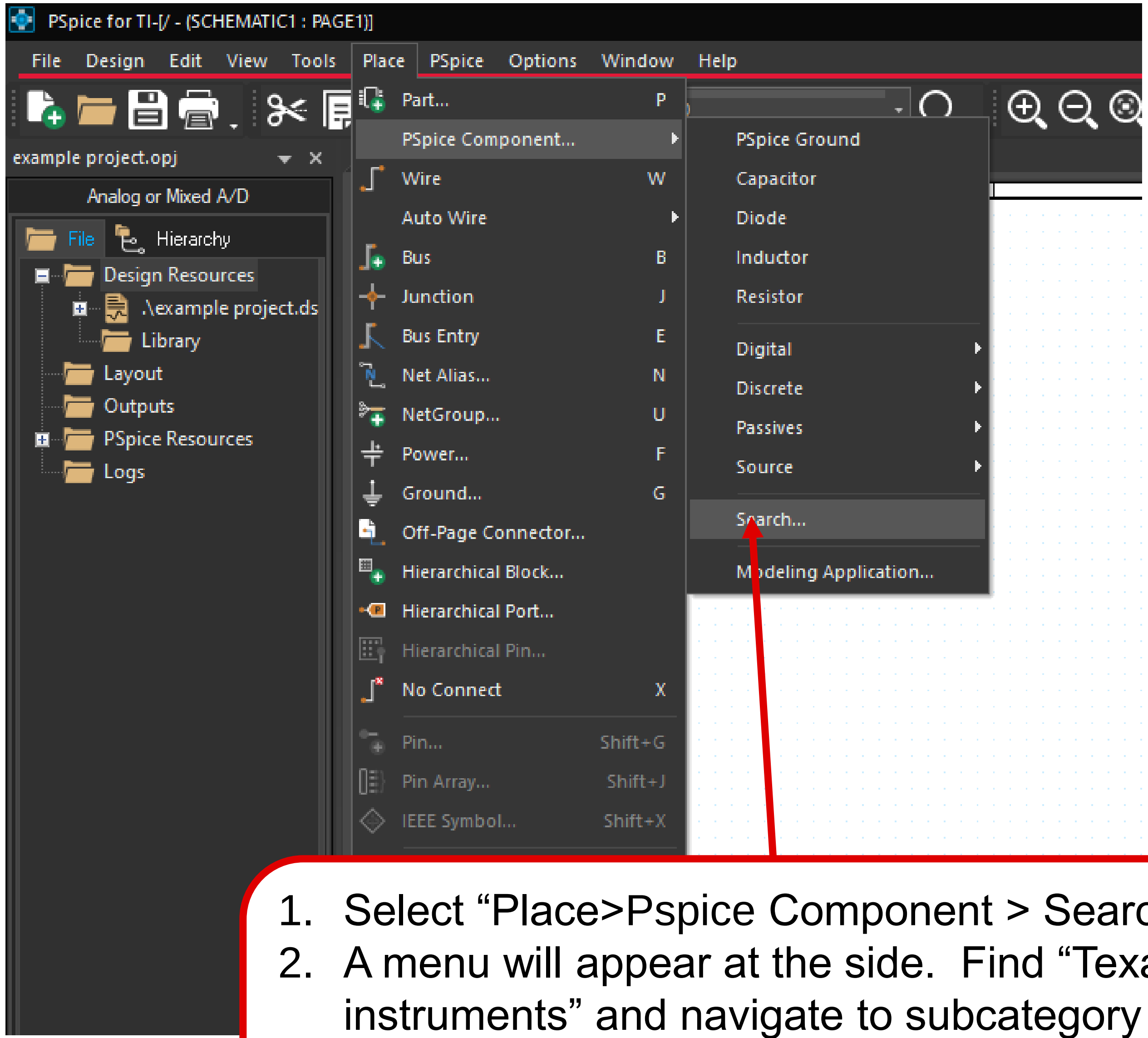


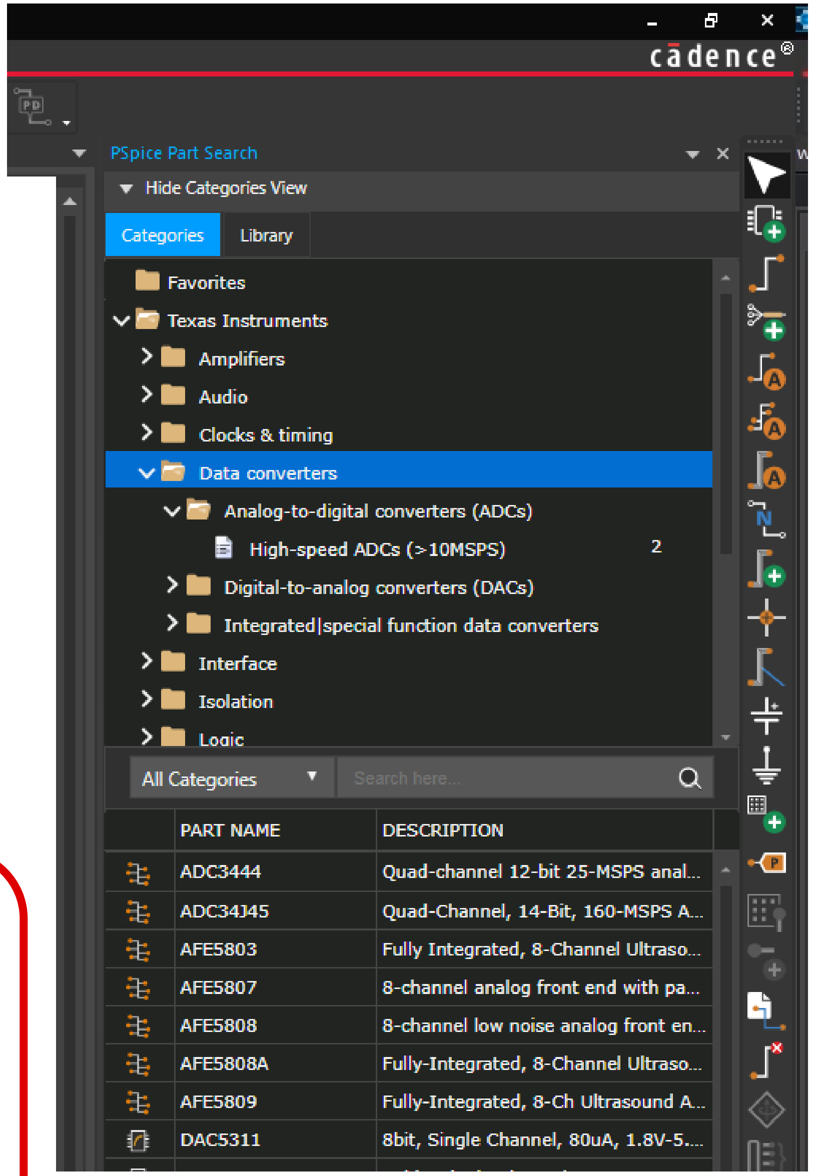


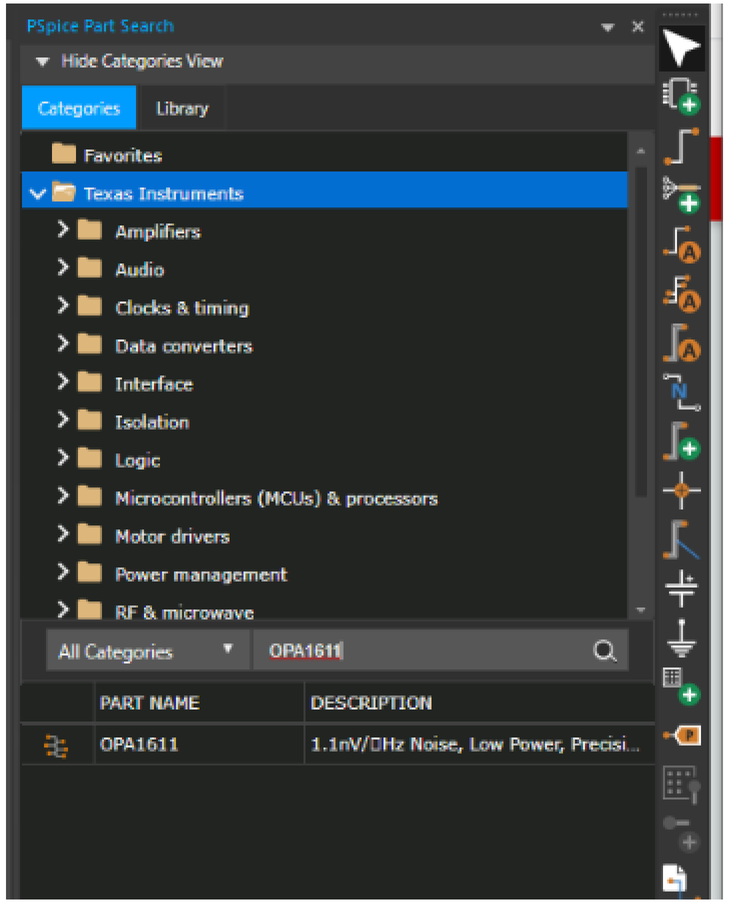
- Some common signal types
- Note that “pulse” includes square, ramp and other types

Back to Table of Contents

Find TI components

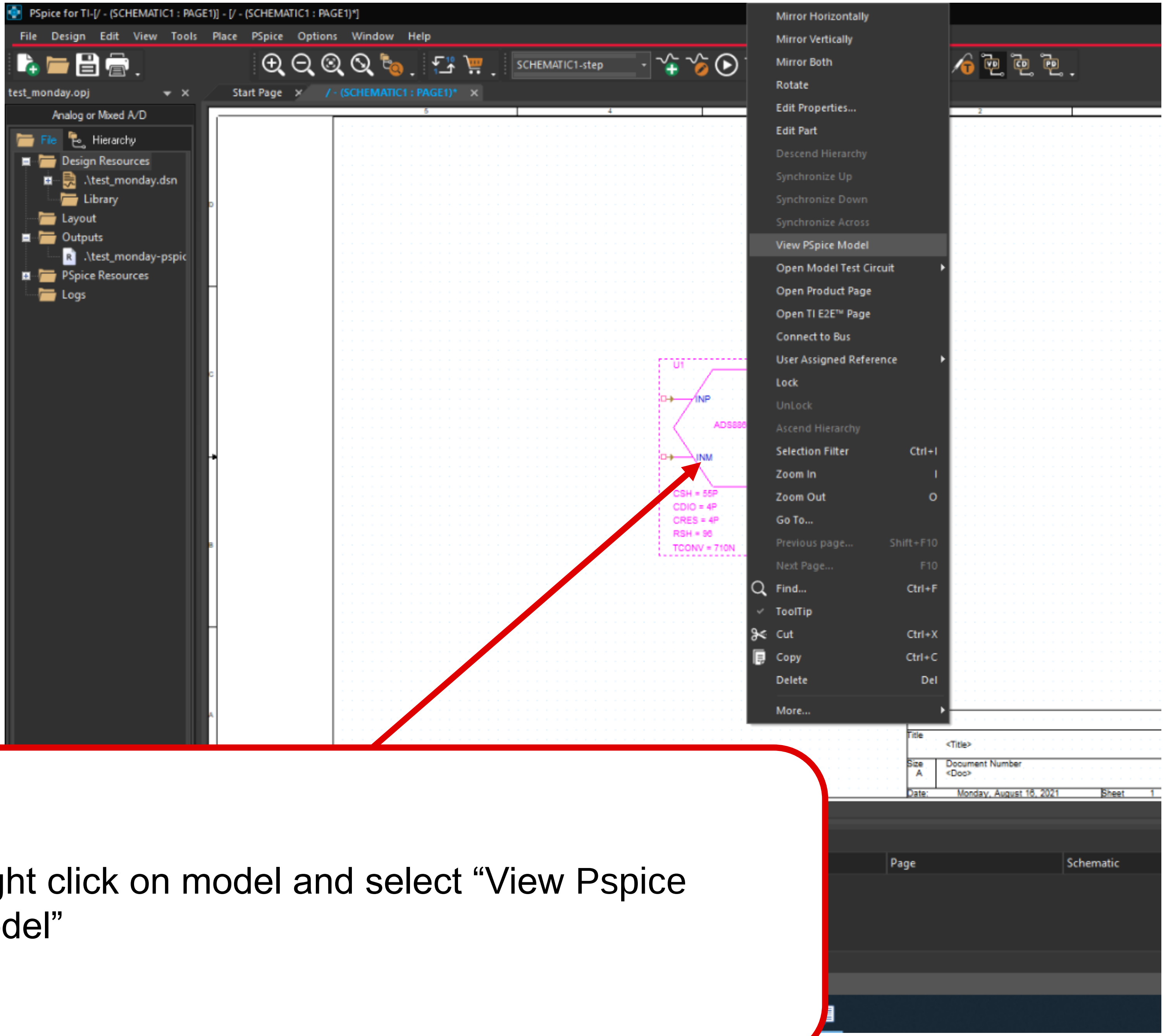


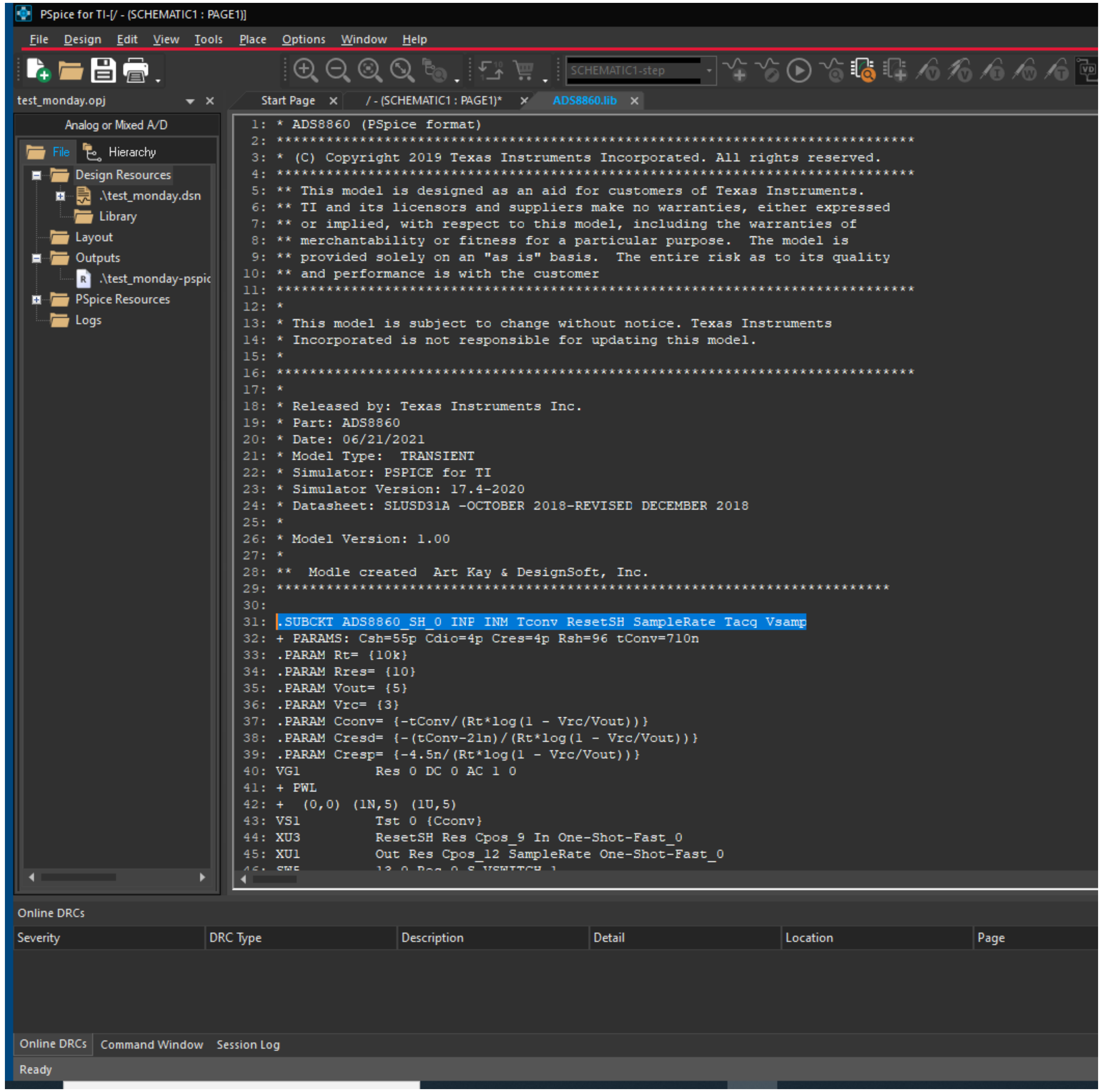




1. Select “Place>Pspice Component > Search”
2. A menu will appear at the side. Find “Texas instruments” and navigate to subcategory “Data converters”
3. Alternatively, search for a part in the search window.

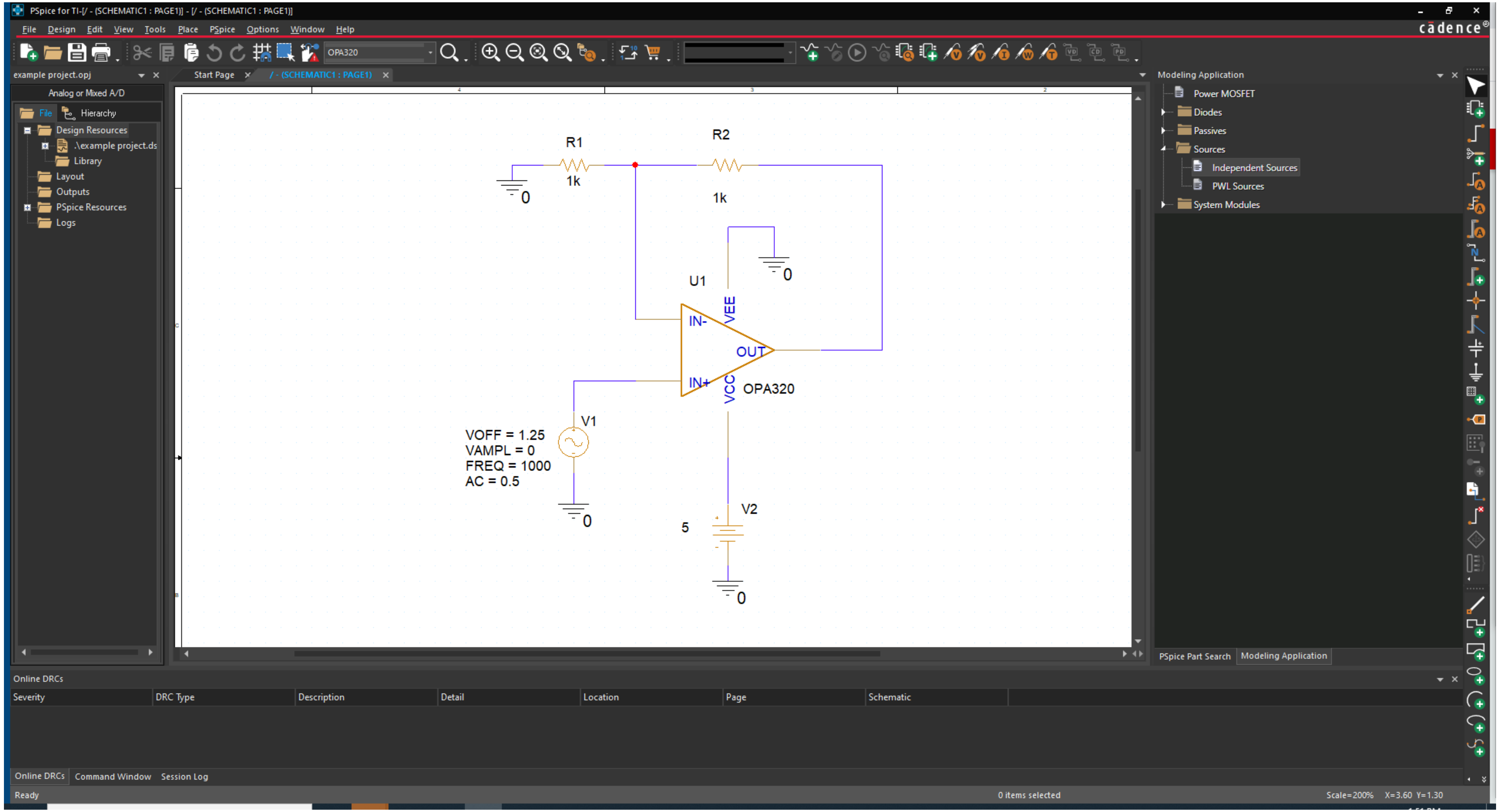
View PSPICE model



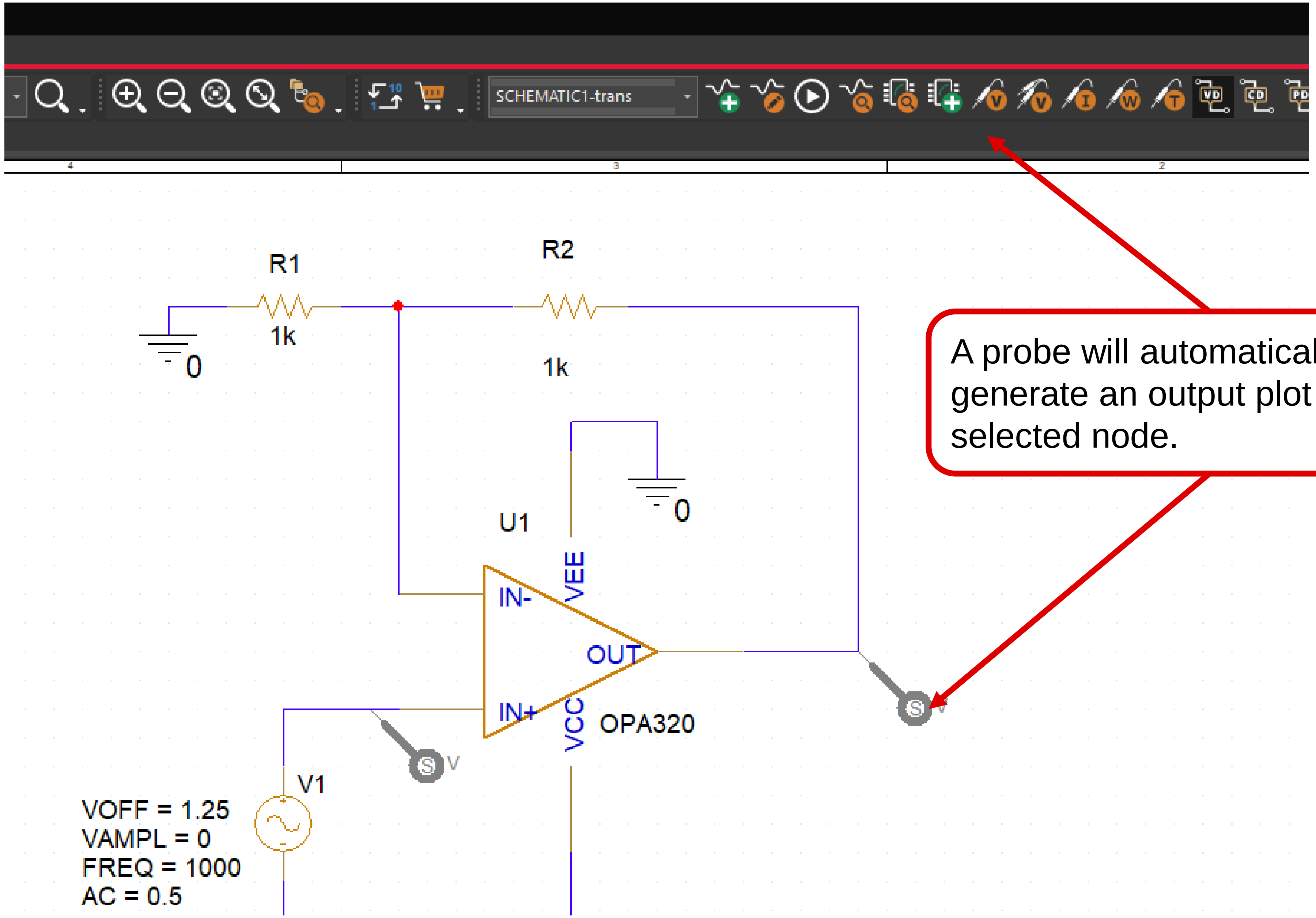


Right click on model and select "View Pspice model"

Use shortcut "w" to wire

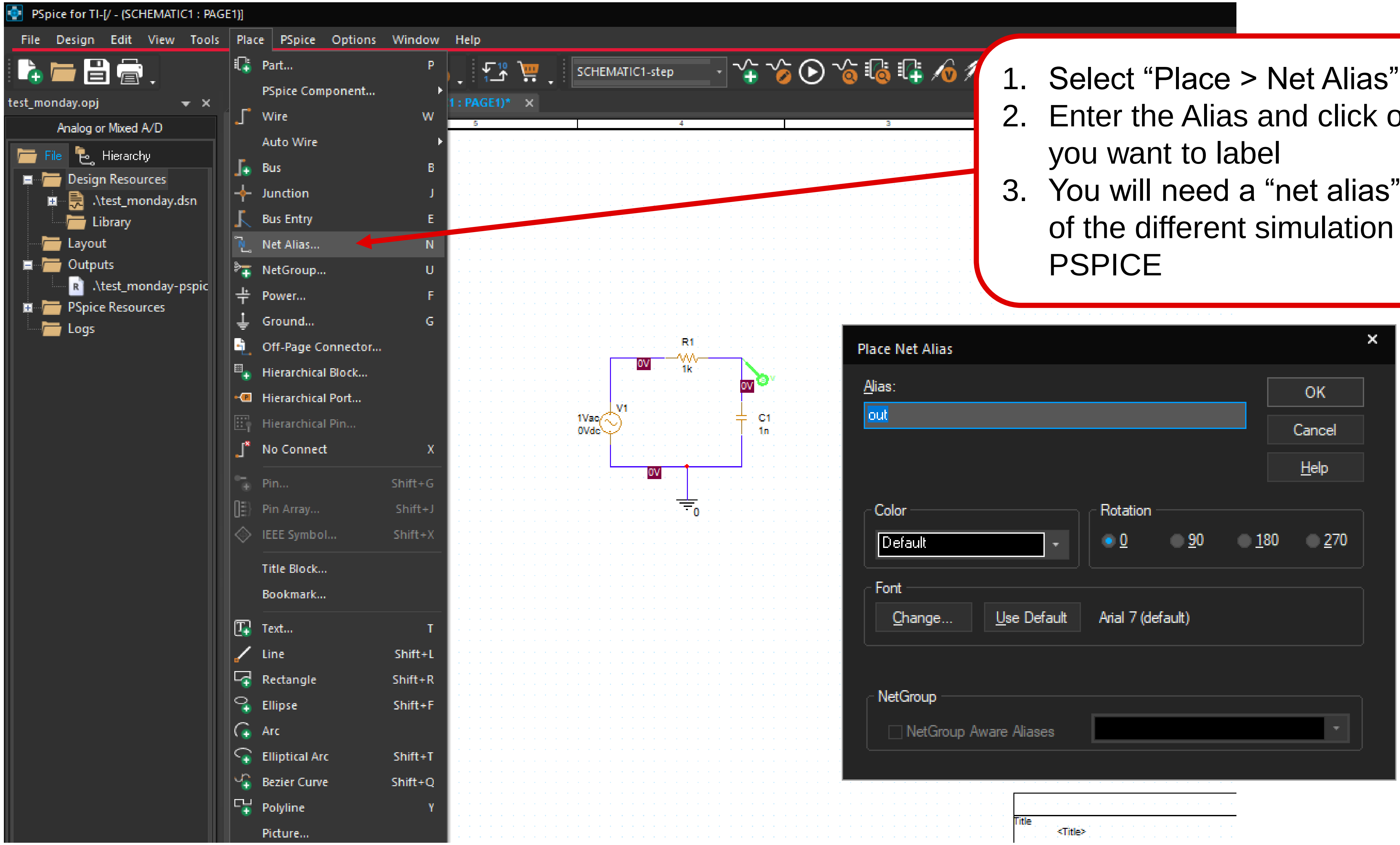


Add probes (voltage markers) to input and output



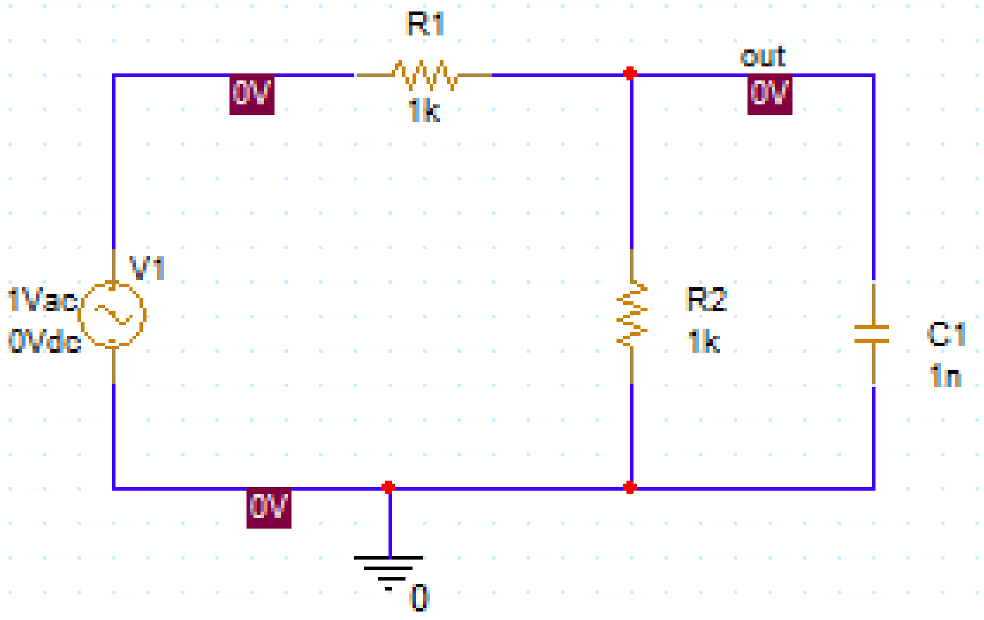
A probe will automatically generate an output plot at the selected node.

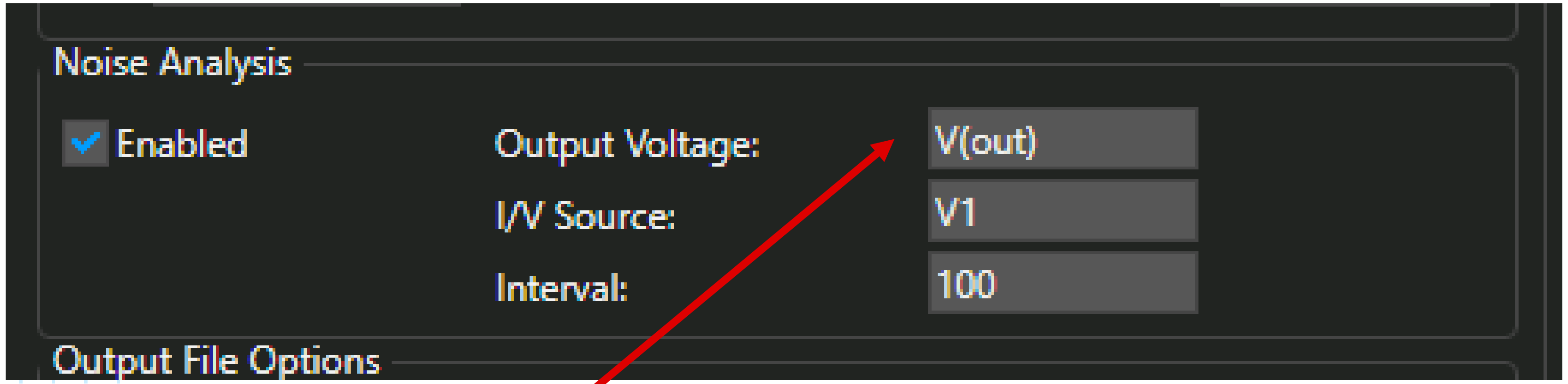
Add net alias label



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

The net alias is very important.

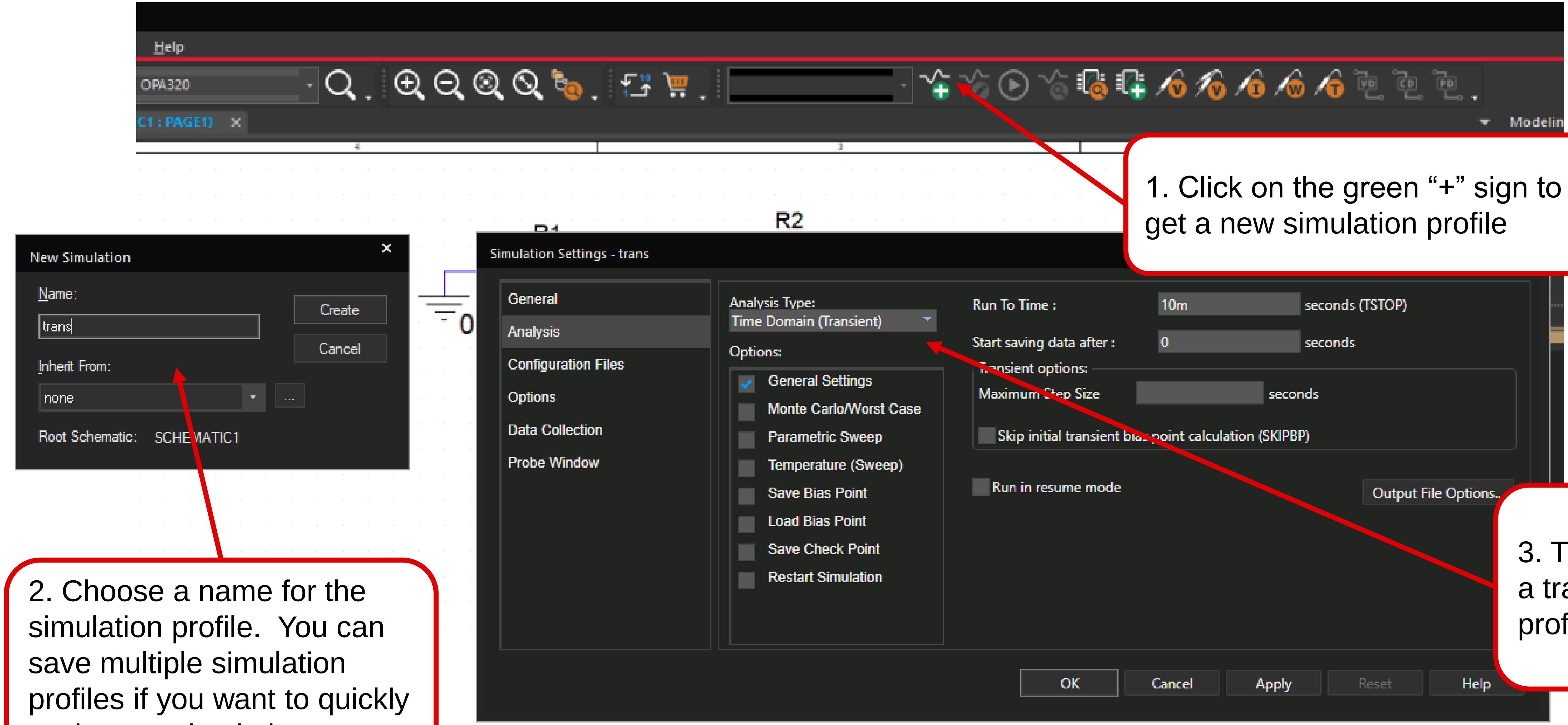




When referring to the “out” node you will use V(out), or I(out) to indicate that you want to plot voltage or current of this node.

Back to Table of Contents

Add “simulation profile” - transient



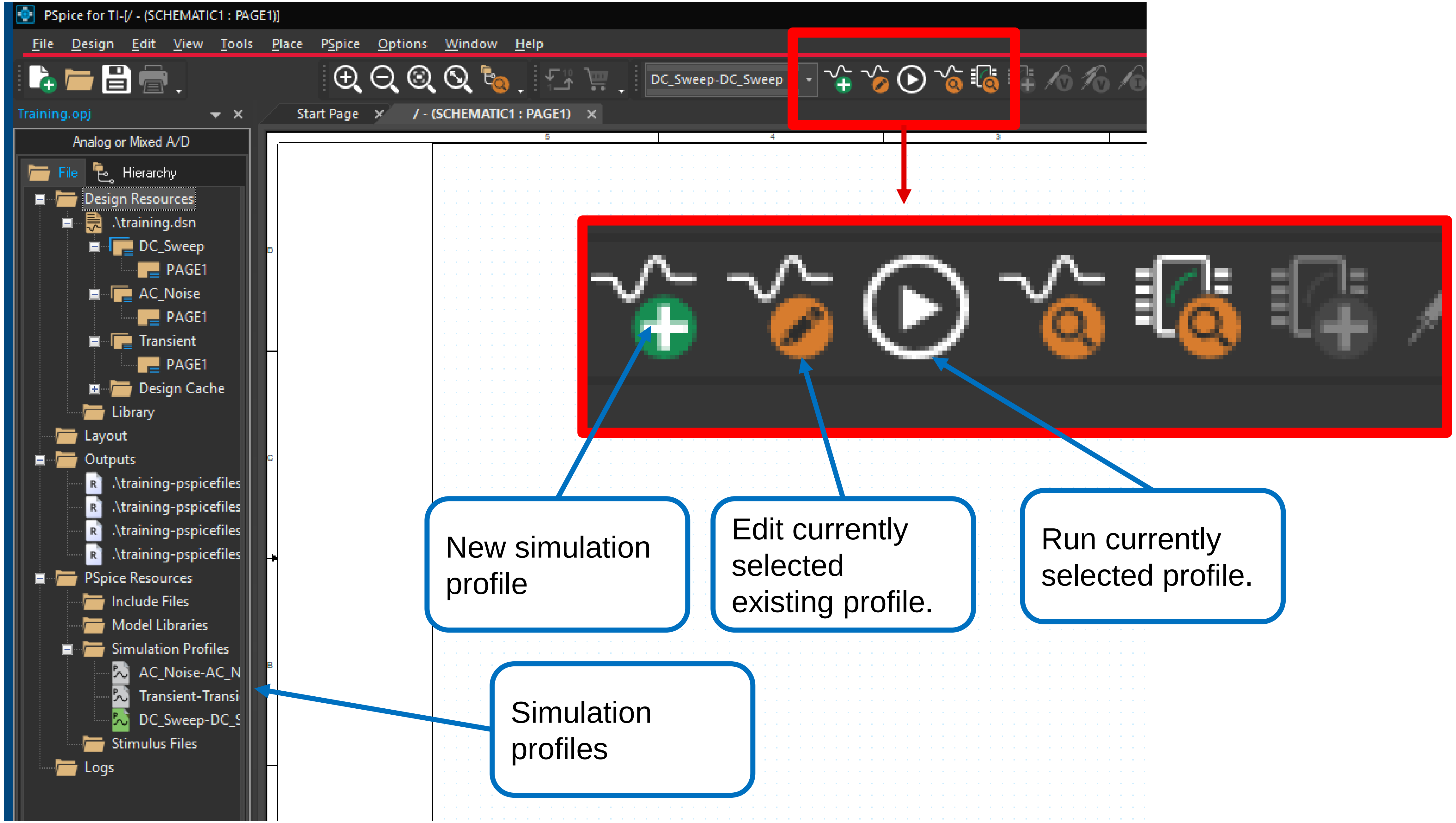
1. Click on the green “+” sign to get a new simulation profile

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.

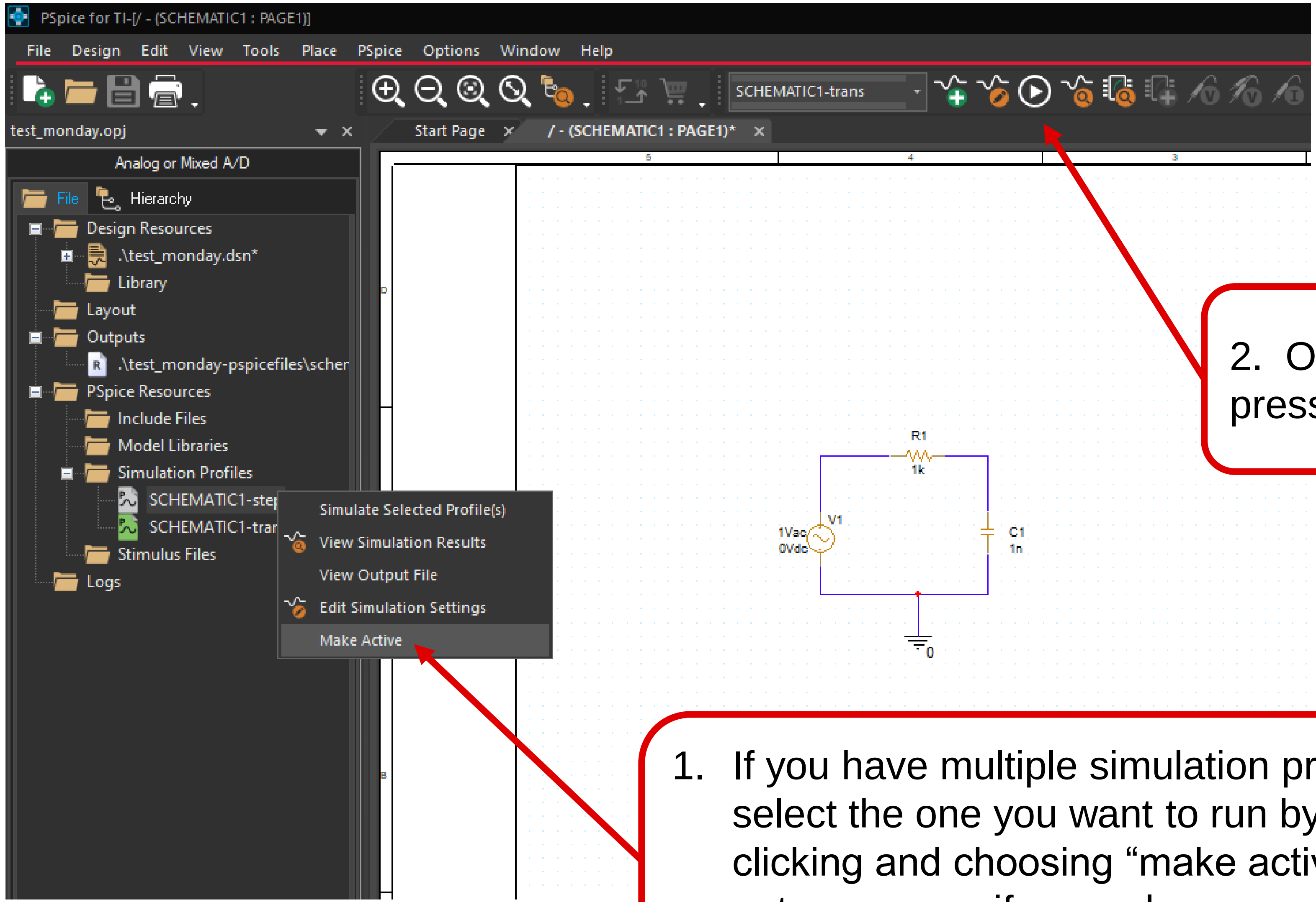
Back to Table of Contents

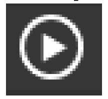
Simulation profiles



[Back to Table of Contents]

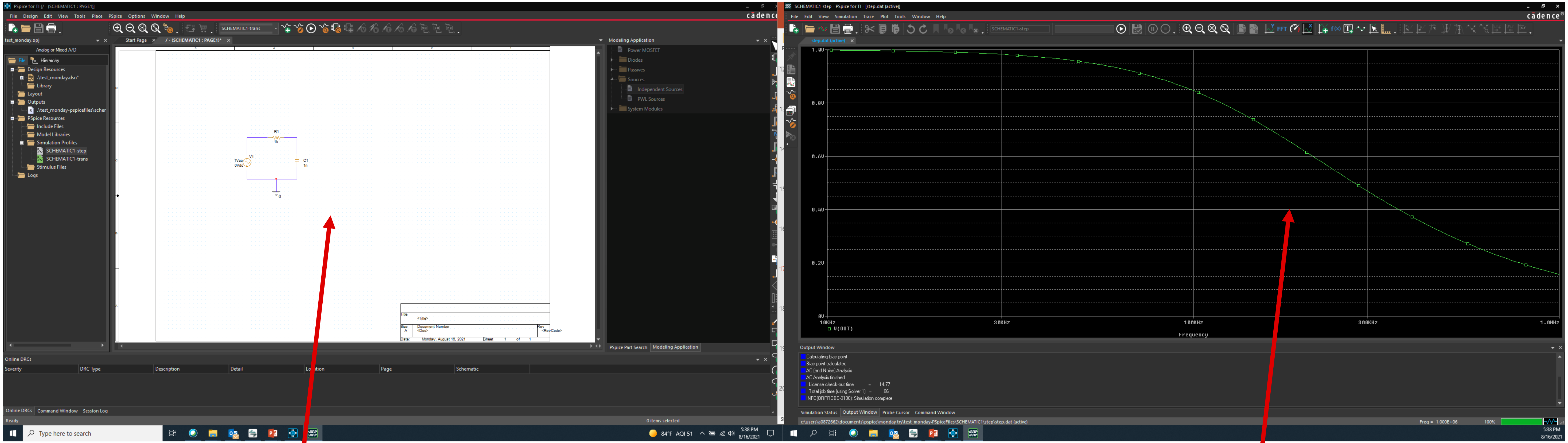
Running simulation



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.

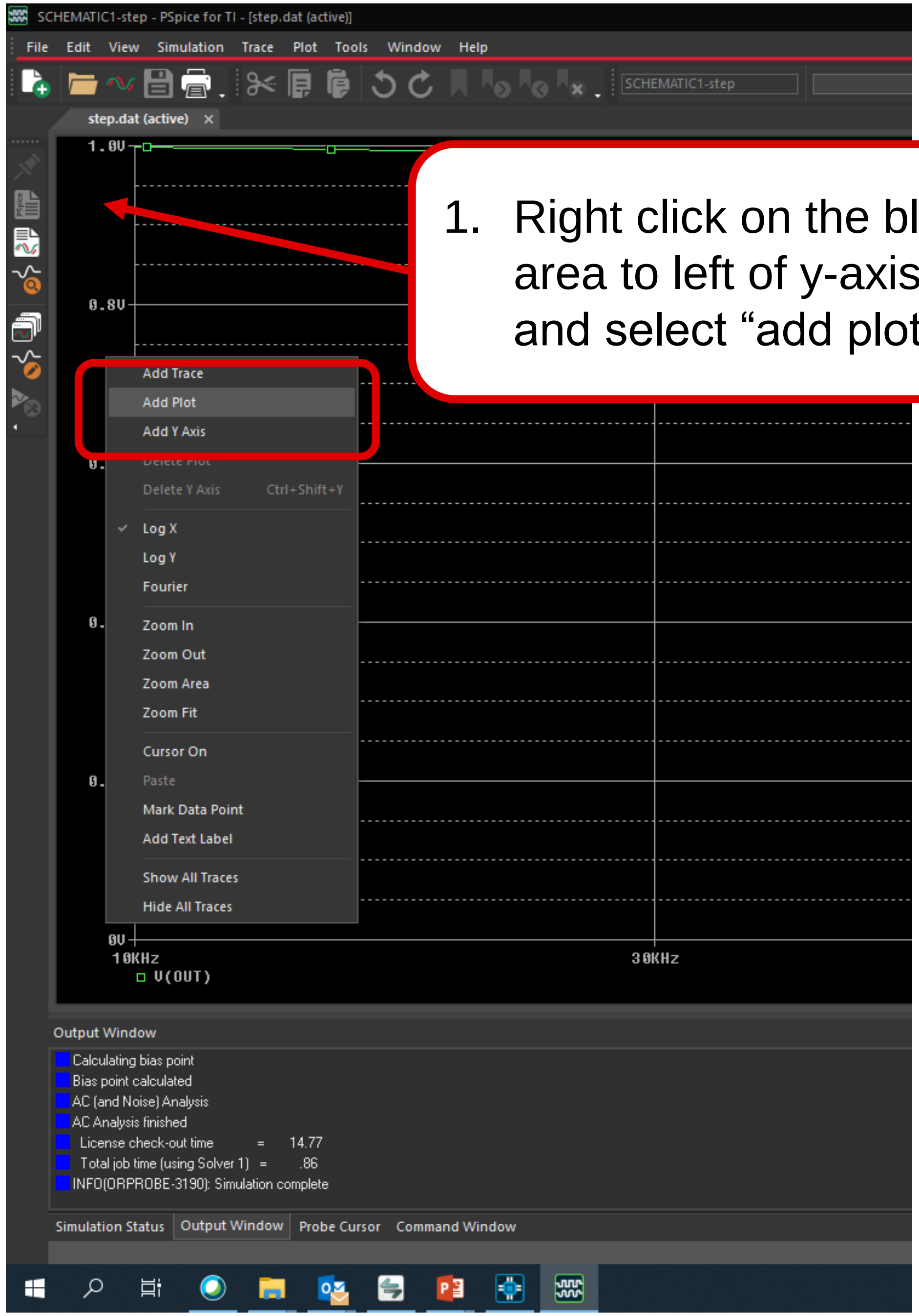
Results



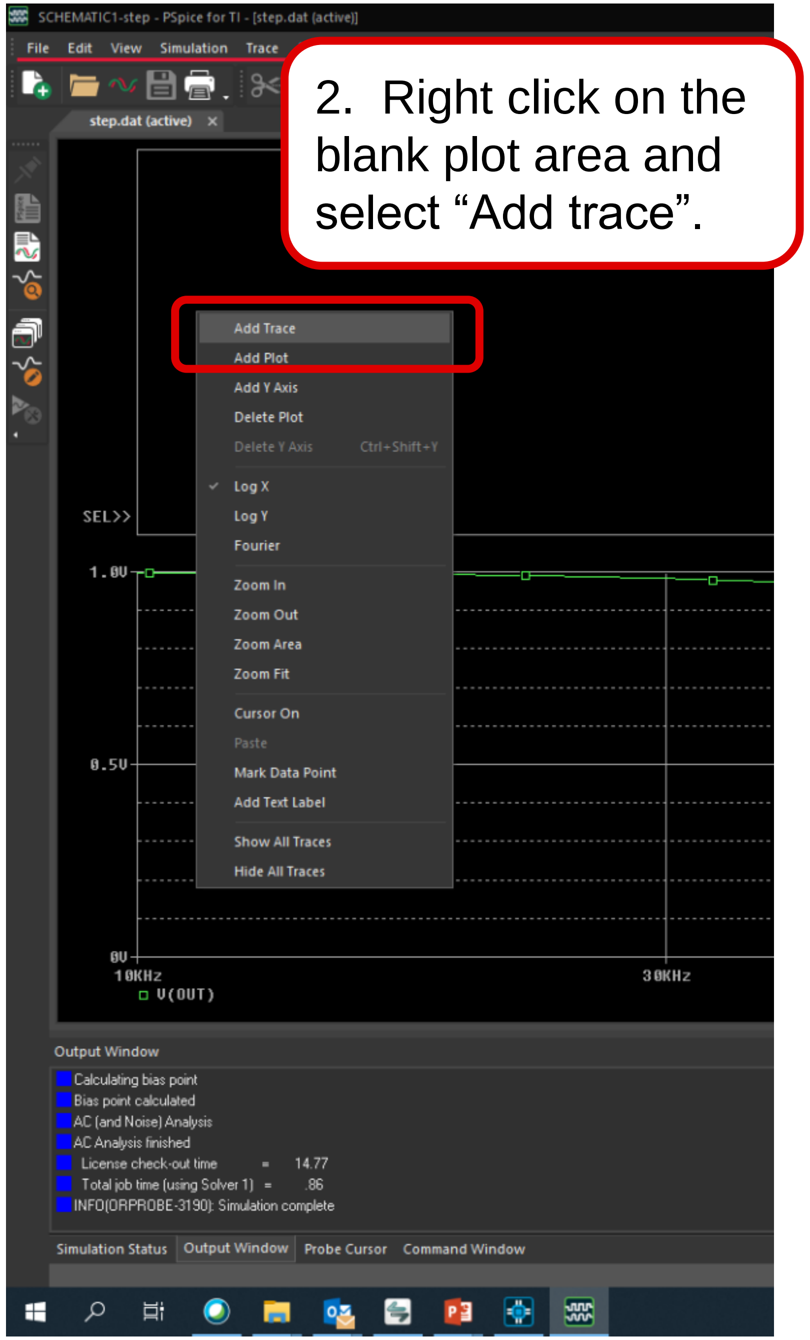
Schematic entry and project control on one screen

Simulation results on the other screen.

Adding plots and curves

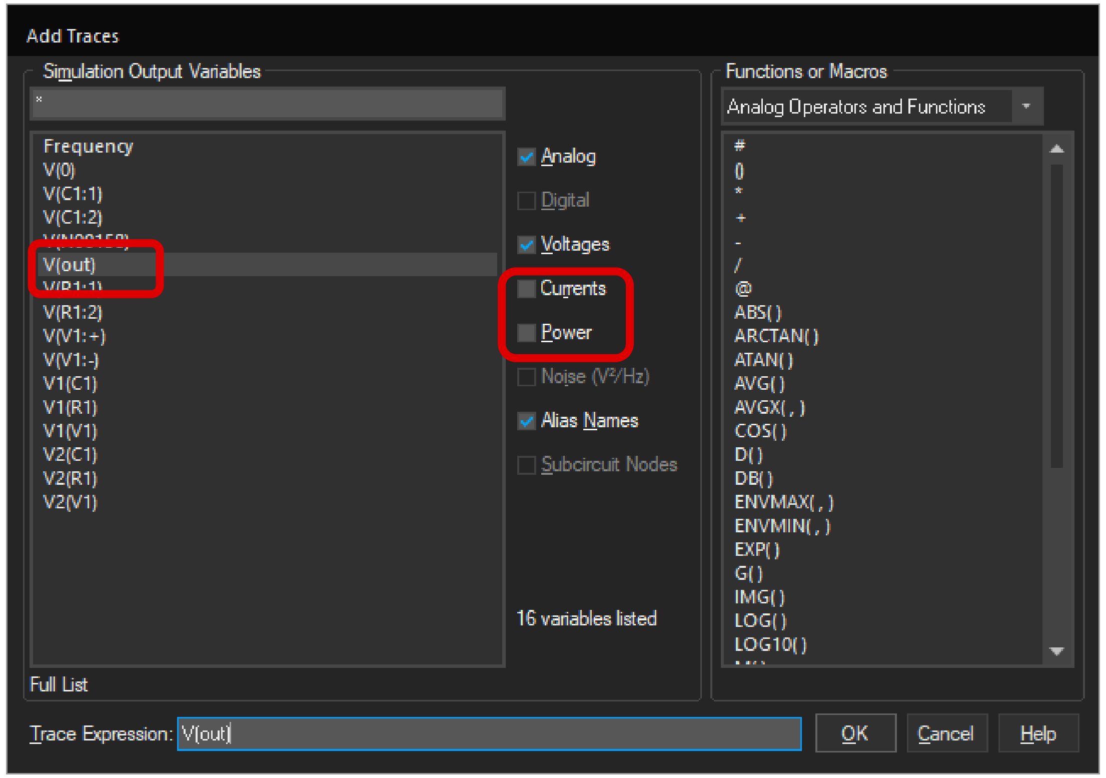


1. Right click on the blank area to left of y-axis and select "add plot"



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

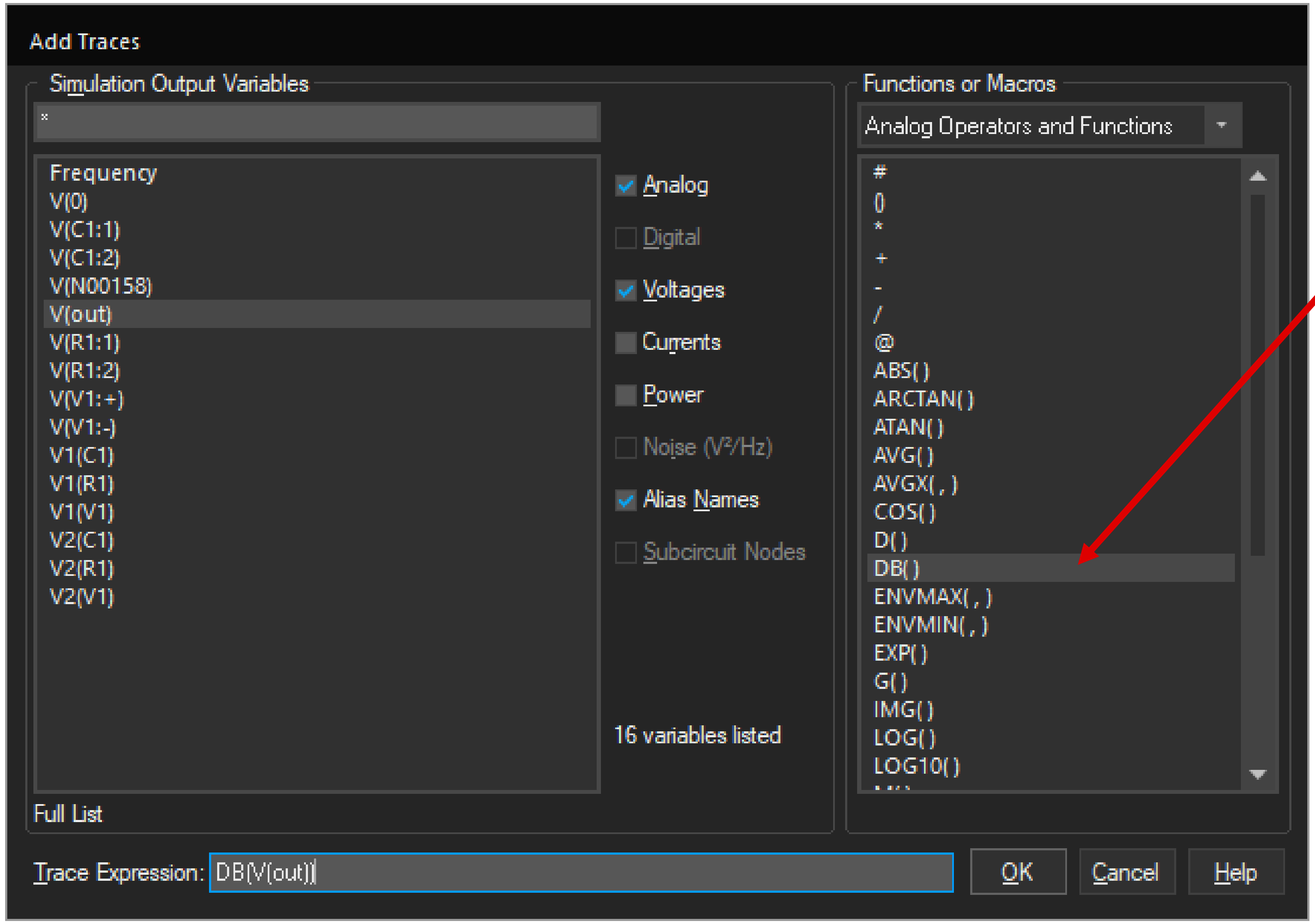
3. Select the trace you want to plot. You may want to uncheck some categories like current and power to make the list easier to sort through.



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

Back to Table of Contents

Post processing



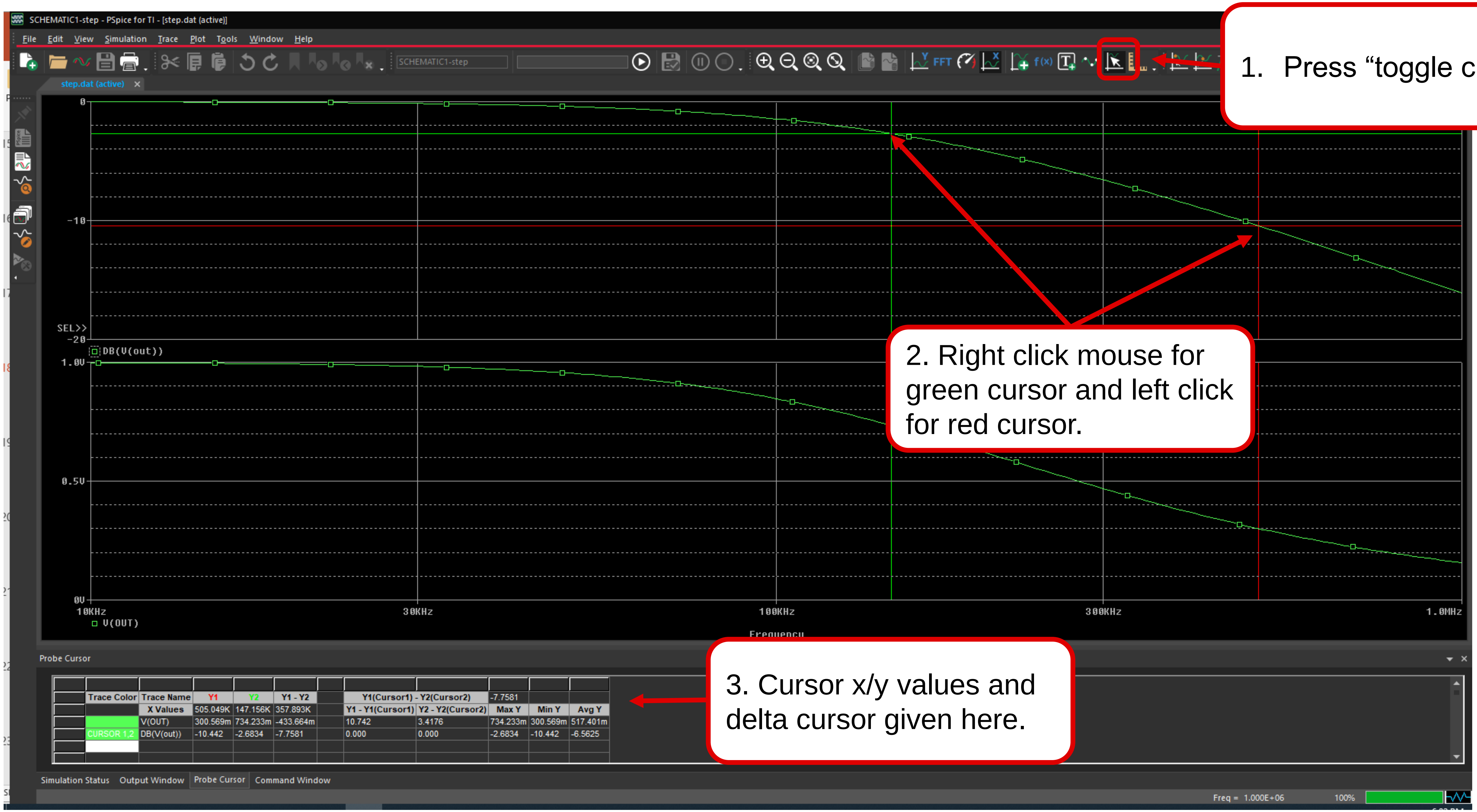
You may want to perform a mathematical operation on the waveform. You can select the math function here. DB() converts the voltage plot to decibels.

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) $x^{½}$
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



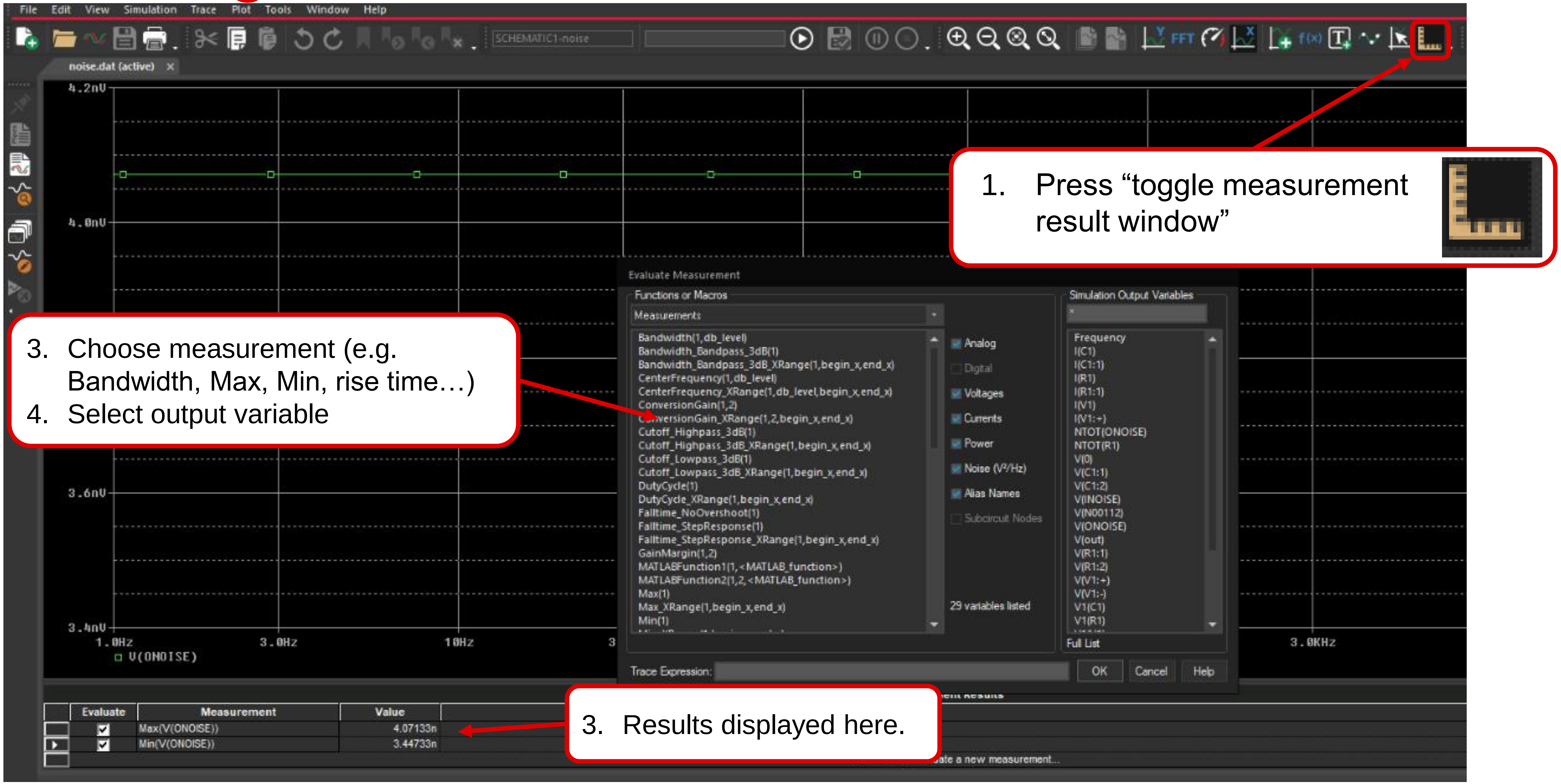
1. Press "toggle cursor"

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

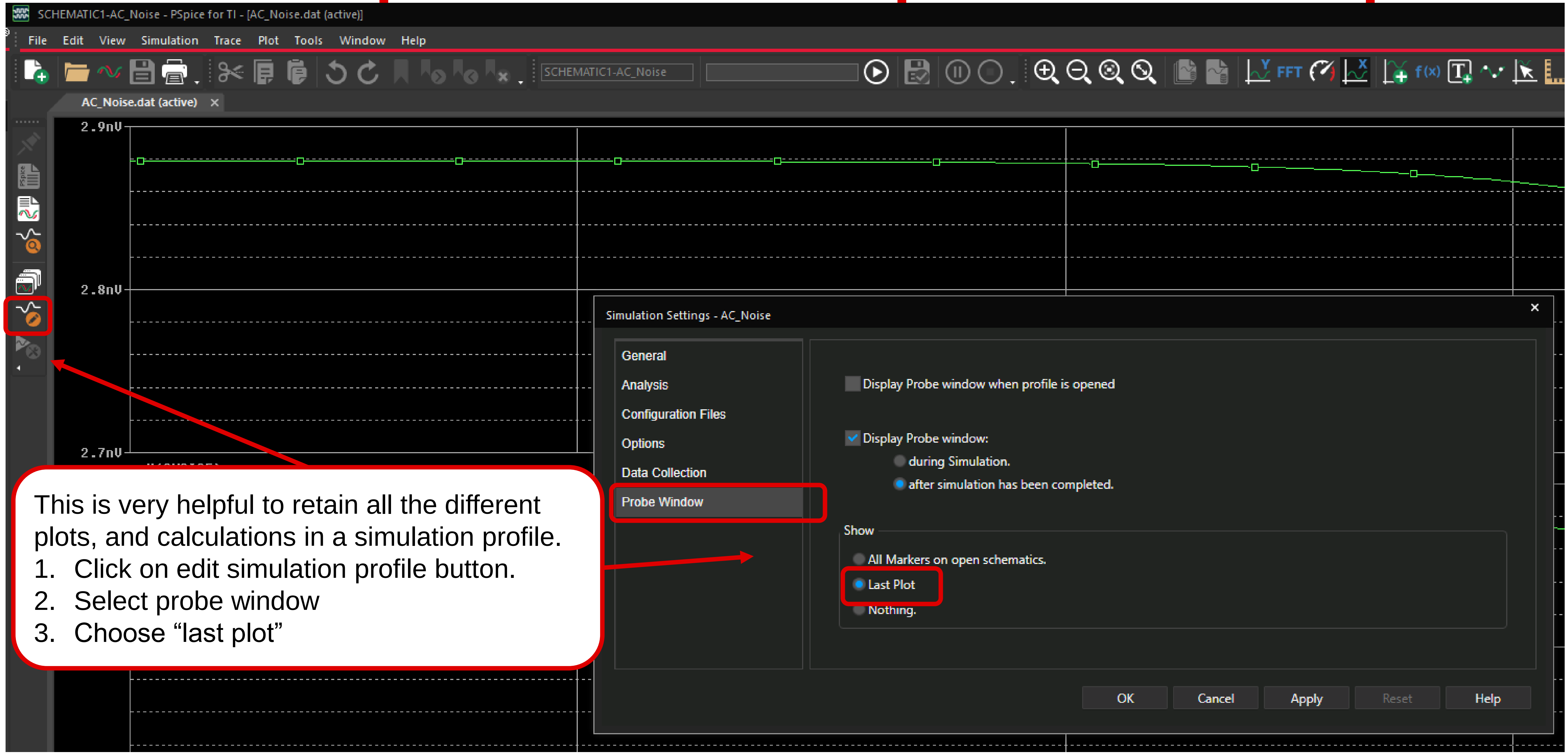
3. Cursor x/y values and delta cursor given here.

[Back to Table of Contents](#)

Adding Measurements



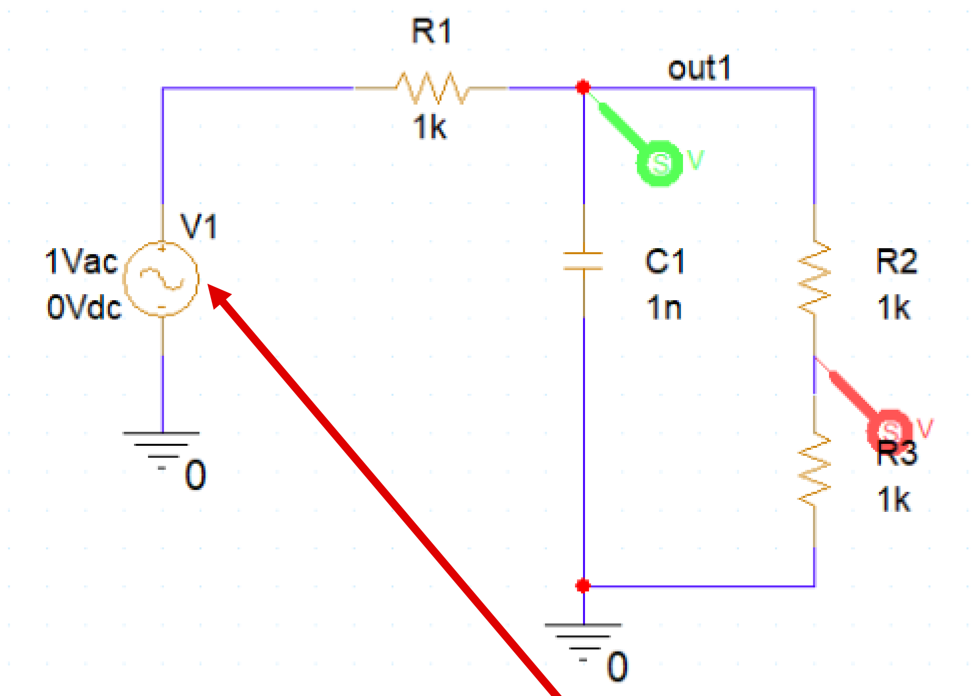
Remember plot window setup in simulation profile

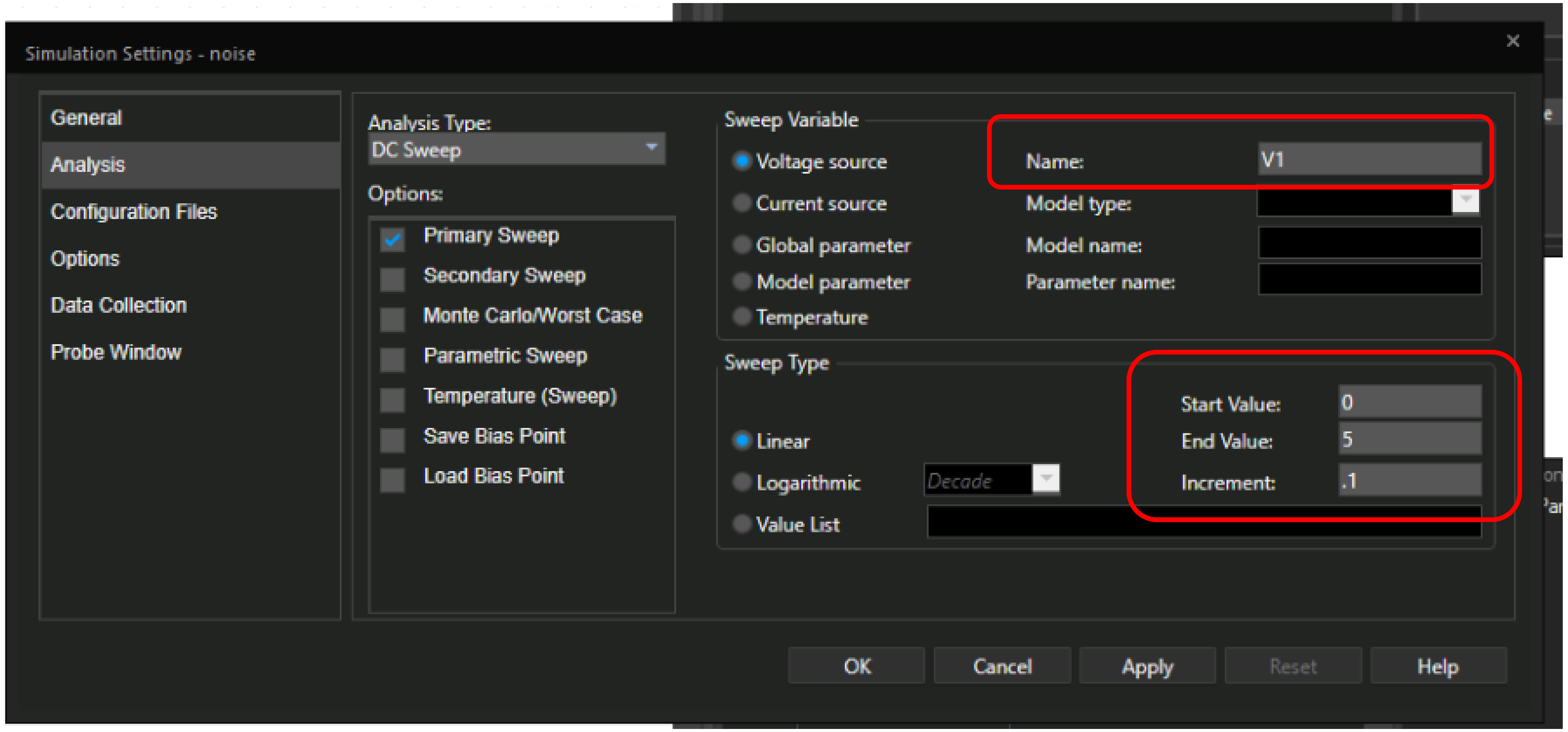


This is very helpful to retain all the different plots, and calculations in a simulation profile.

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

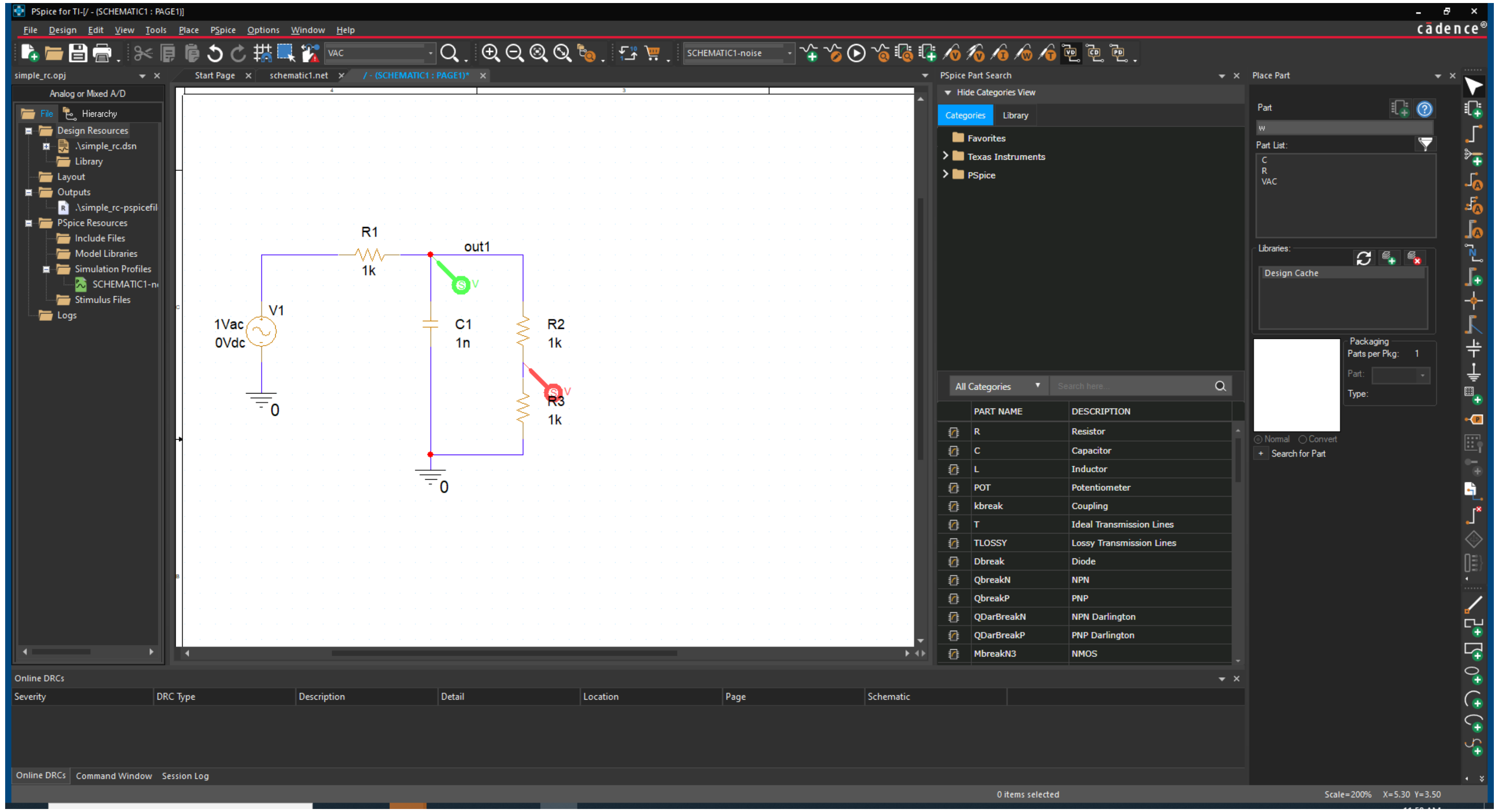
DC sweep

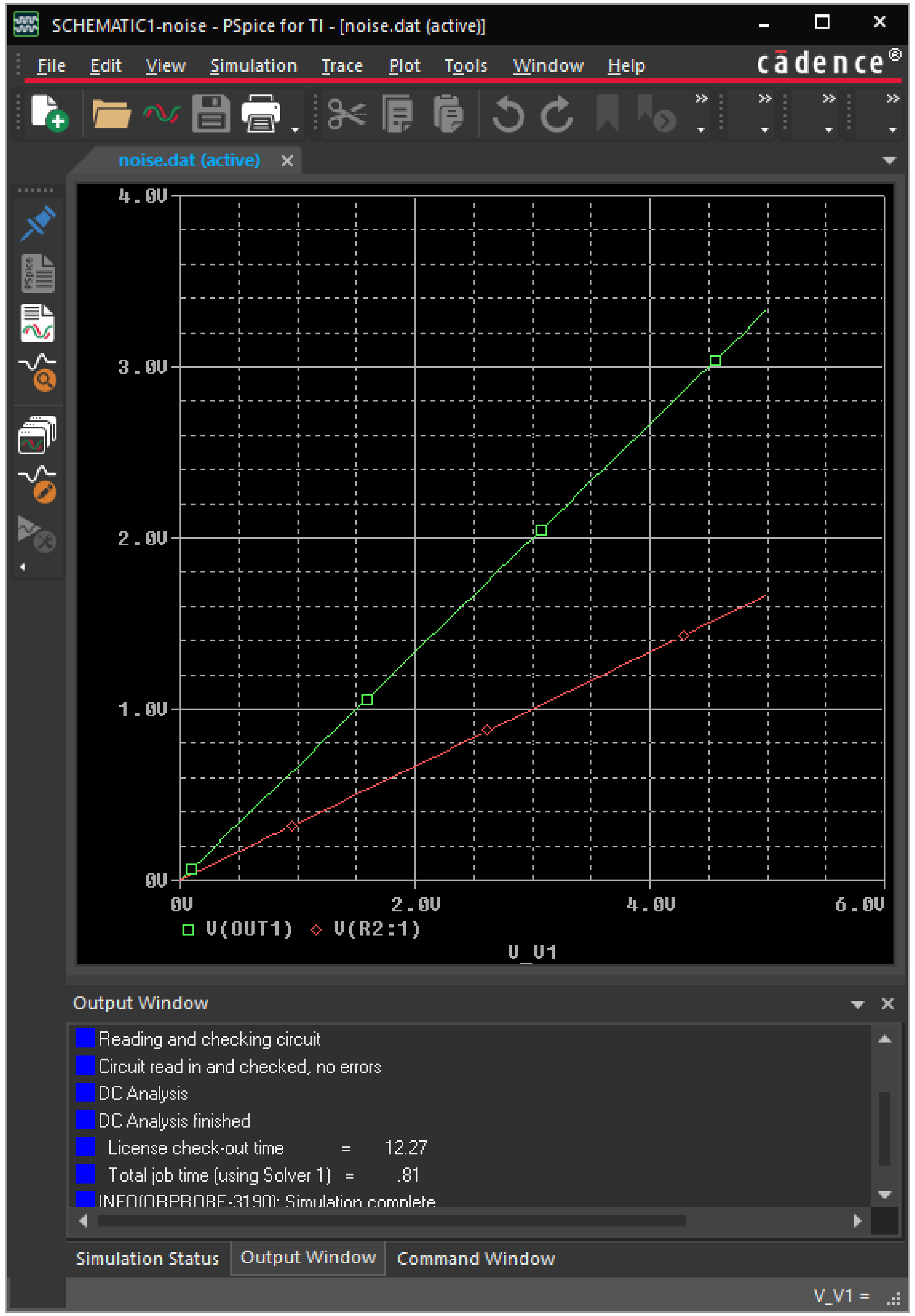




For DC sweep: specify the voltage source you want to sweep (V1 in this example). Specify the sweep type and range.

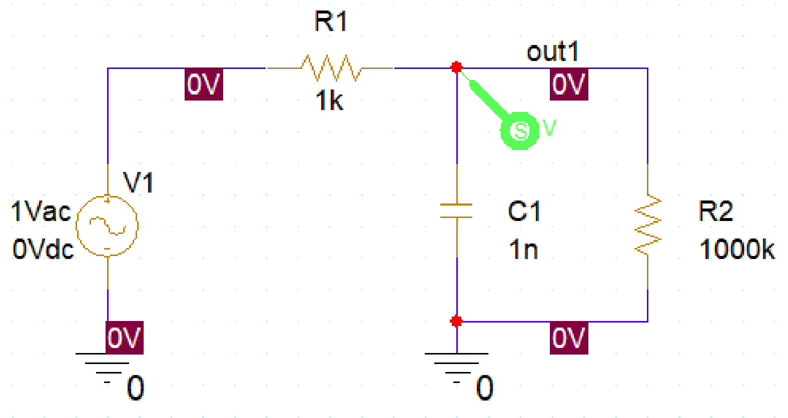
DC Sweep

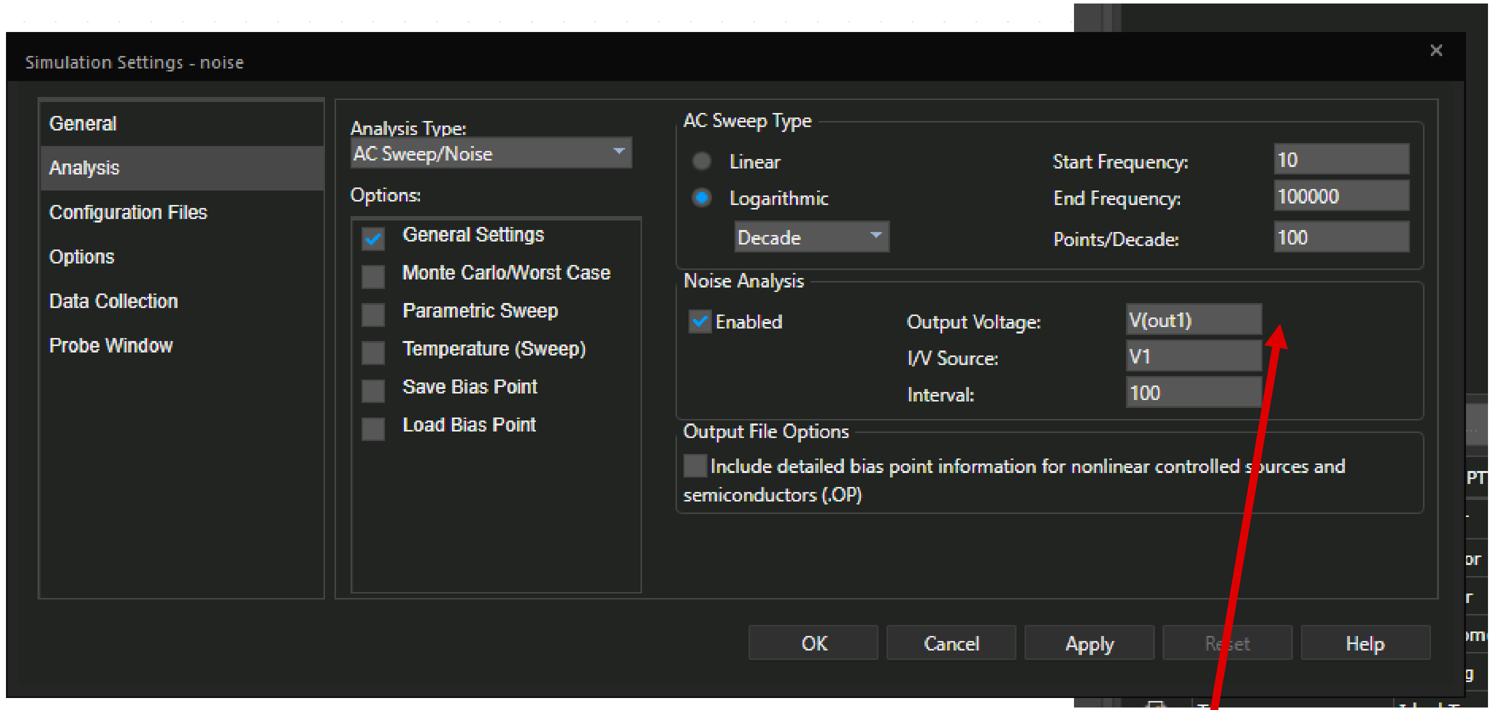




This shows the example results for DC sweep where two probes measure the outputs.

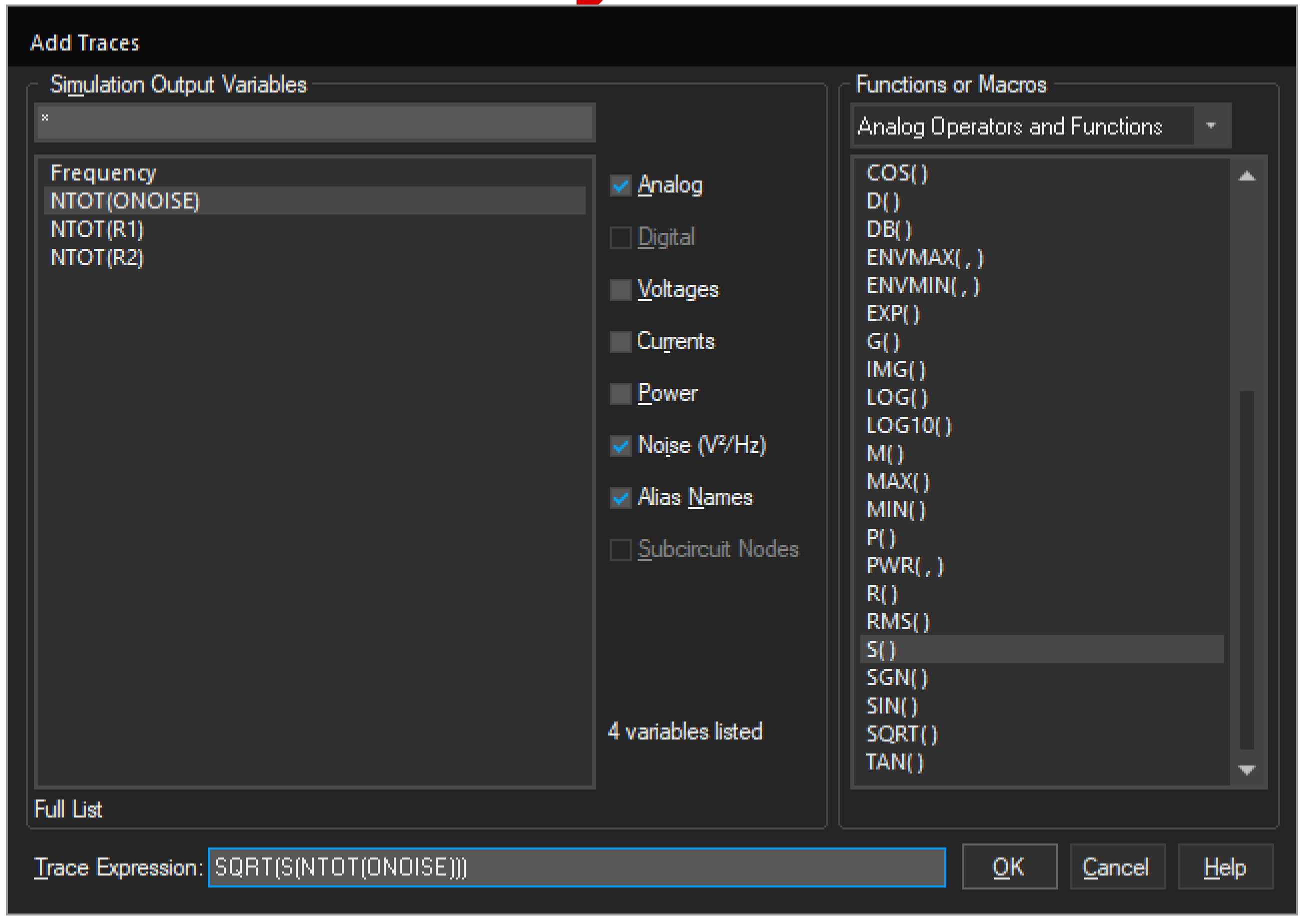
AC Sweep and Noise analysis

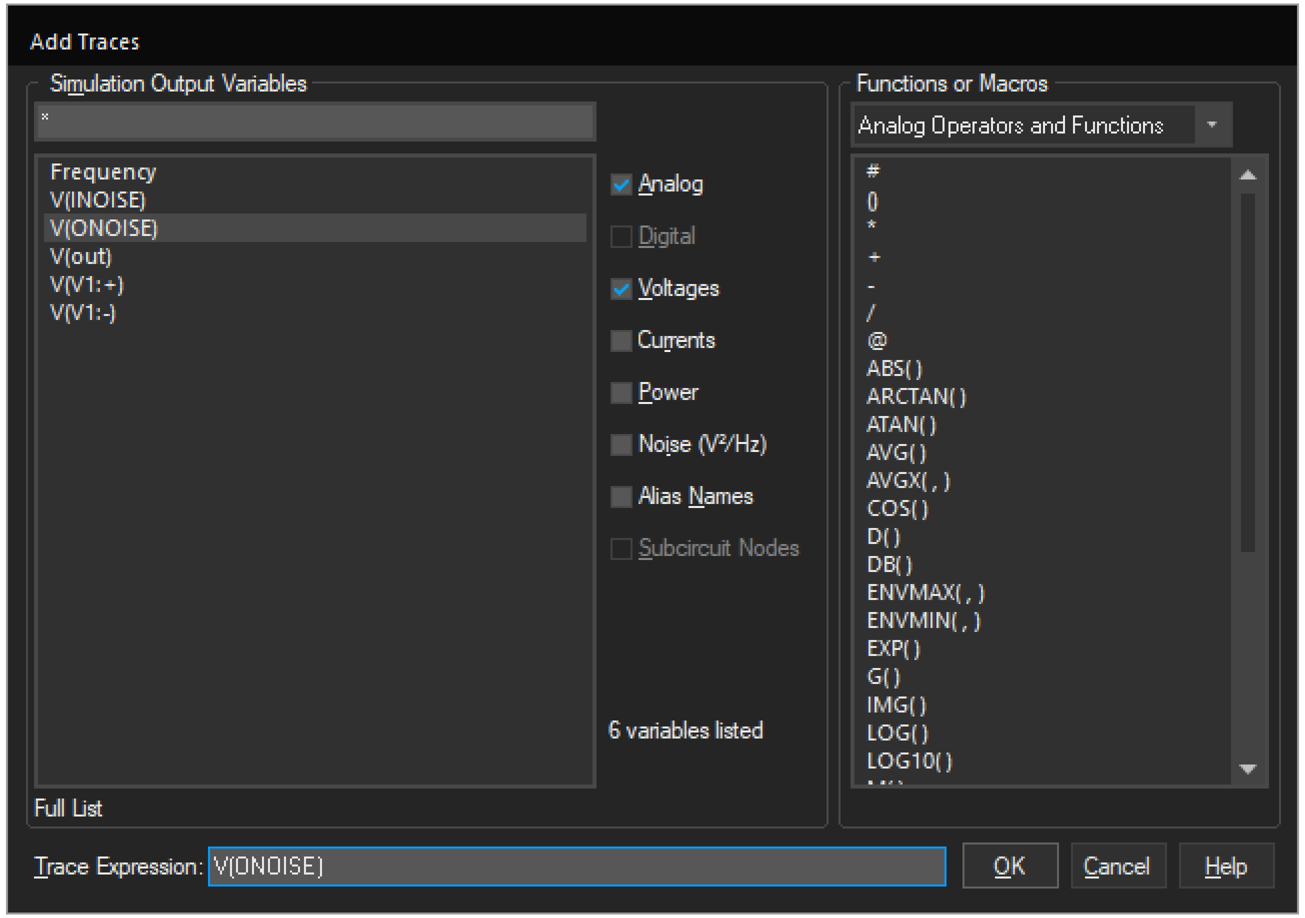




Where ***Output voltage: V(net_name)*** is used to plot voltage noise for ***net_name.*** In this case the net name is ***out1***
V/I Source: source_name – In this example the source name is ***V1***

Noise analysis





Integrated Total RMS noise

- “S” is the integral function
- SQRT is the square root function
- NTOT(ONOISE) = Noise power in V^2/Hz

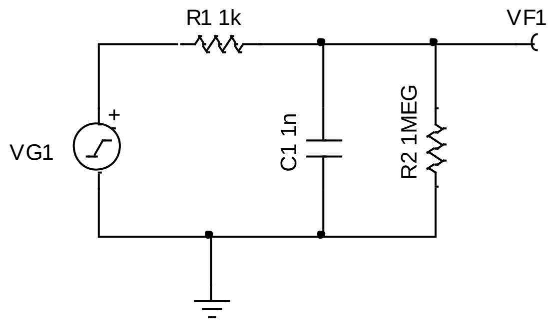
$$E_n = \sqrt{\int P_n}$$

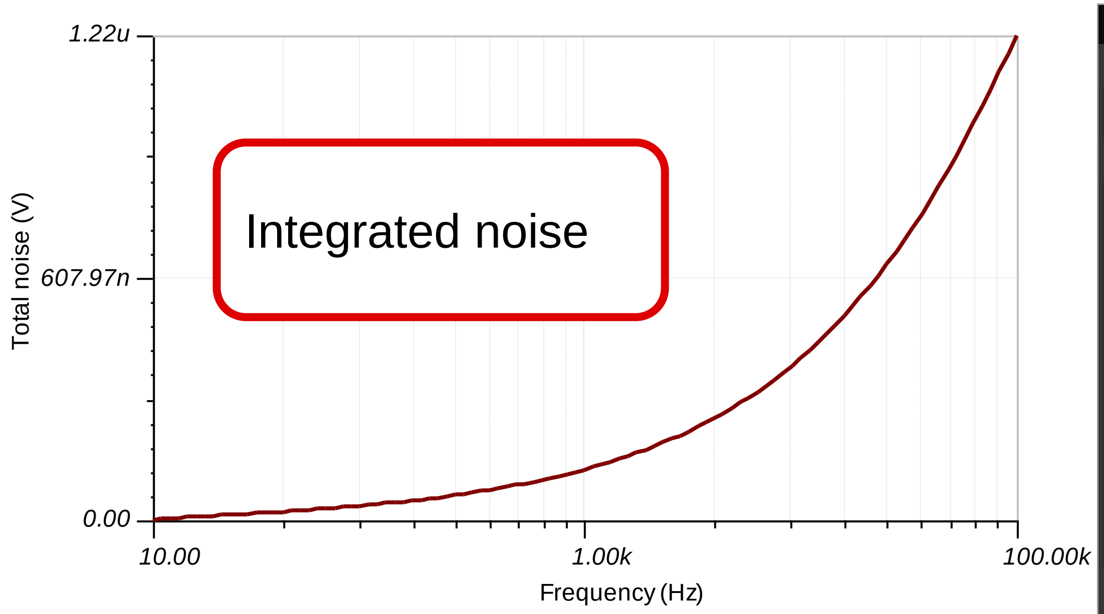
Voltage Noise Spectral Density

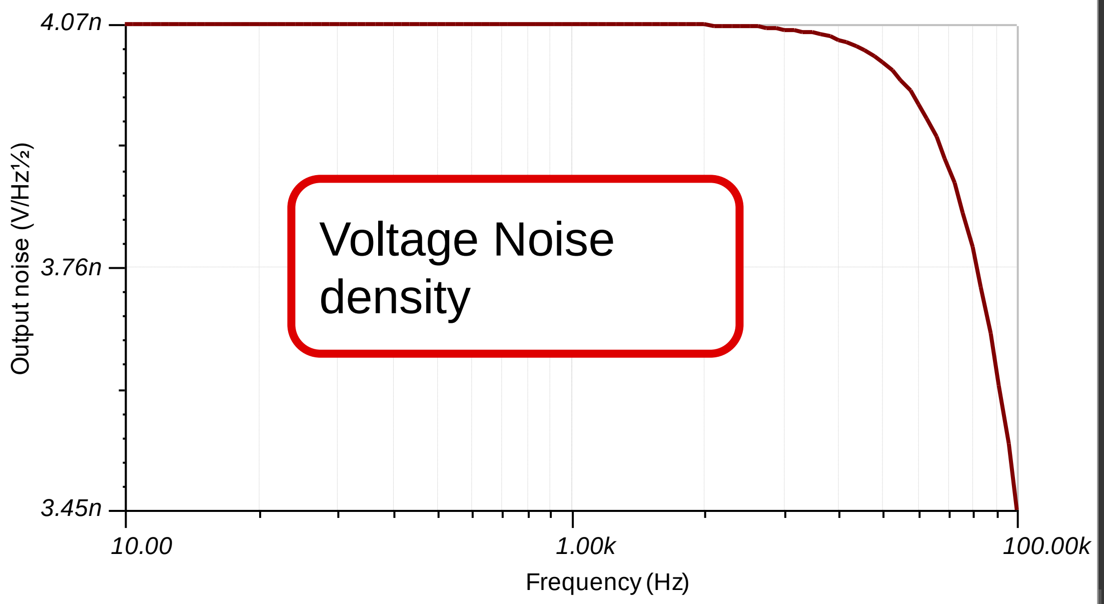
- V(ONOISE) is the noise density in V/rtHz at the output
- V(INOISE) is the input noise density
- Note: the “Noise” check box will give noise power in V^2/Hz

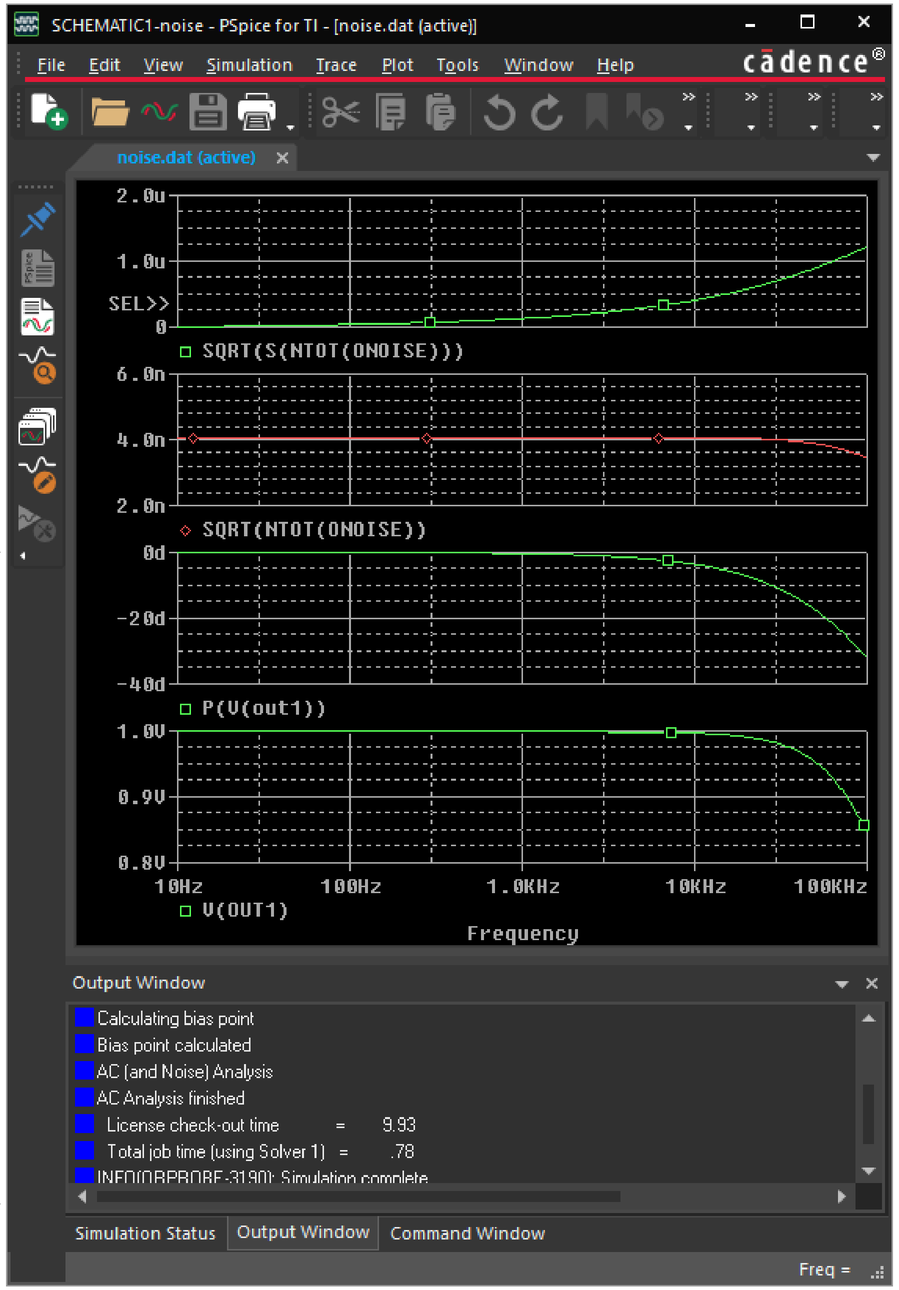
Back to Table of Contents

Noise analysis





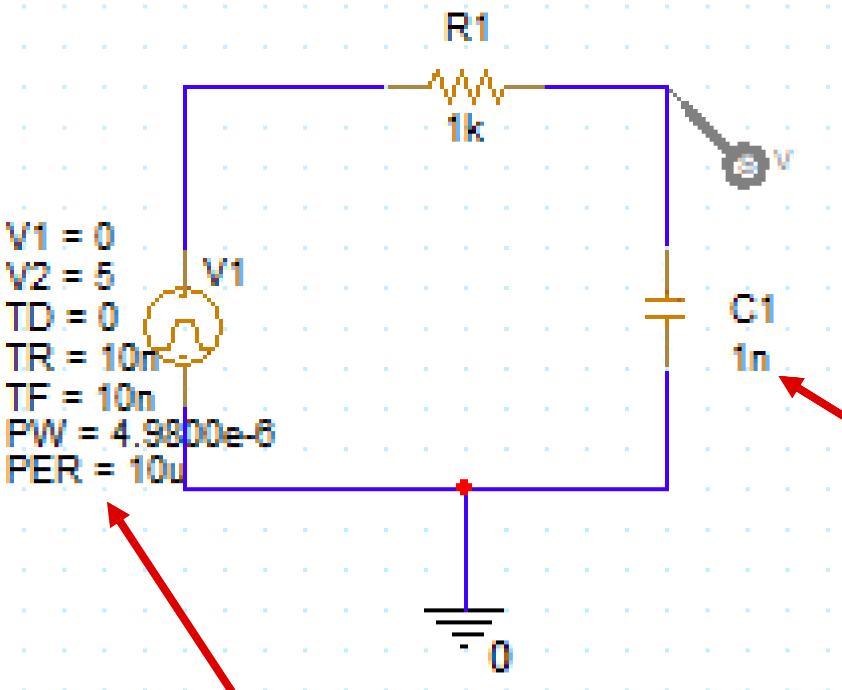


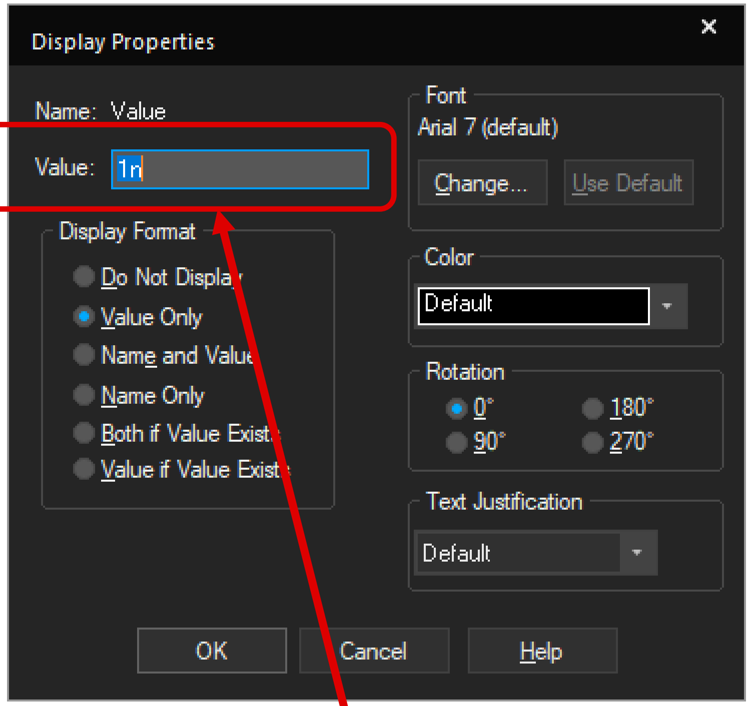


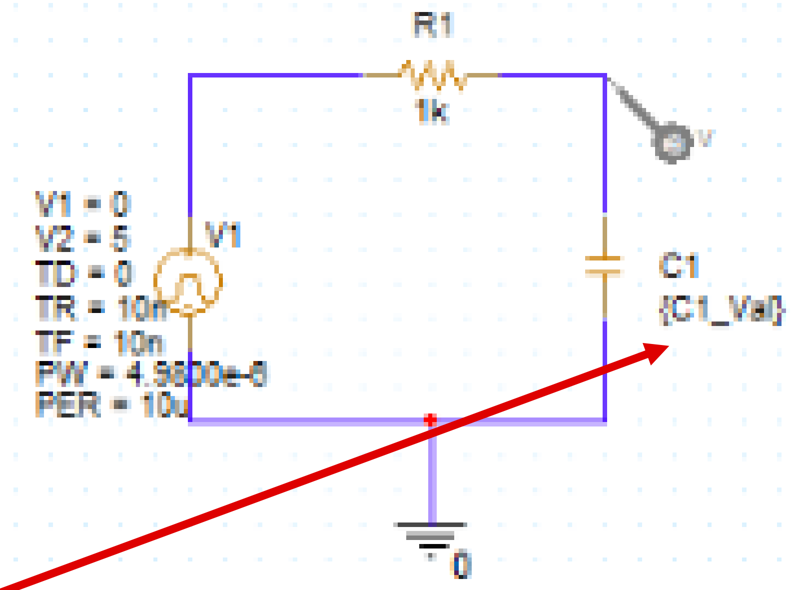
This slide shows the two traces for noise density and integrated noise match TINA. By default PSPICE plots noise power which isn't very useful

Back to Table of Contents

Parameter Stepping: Edit schematic value



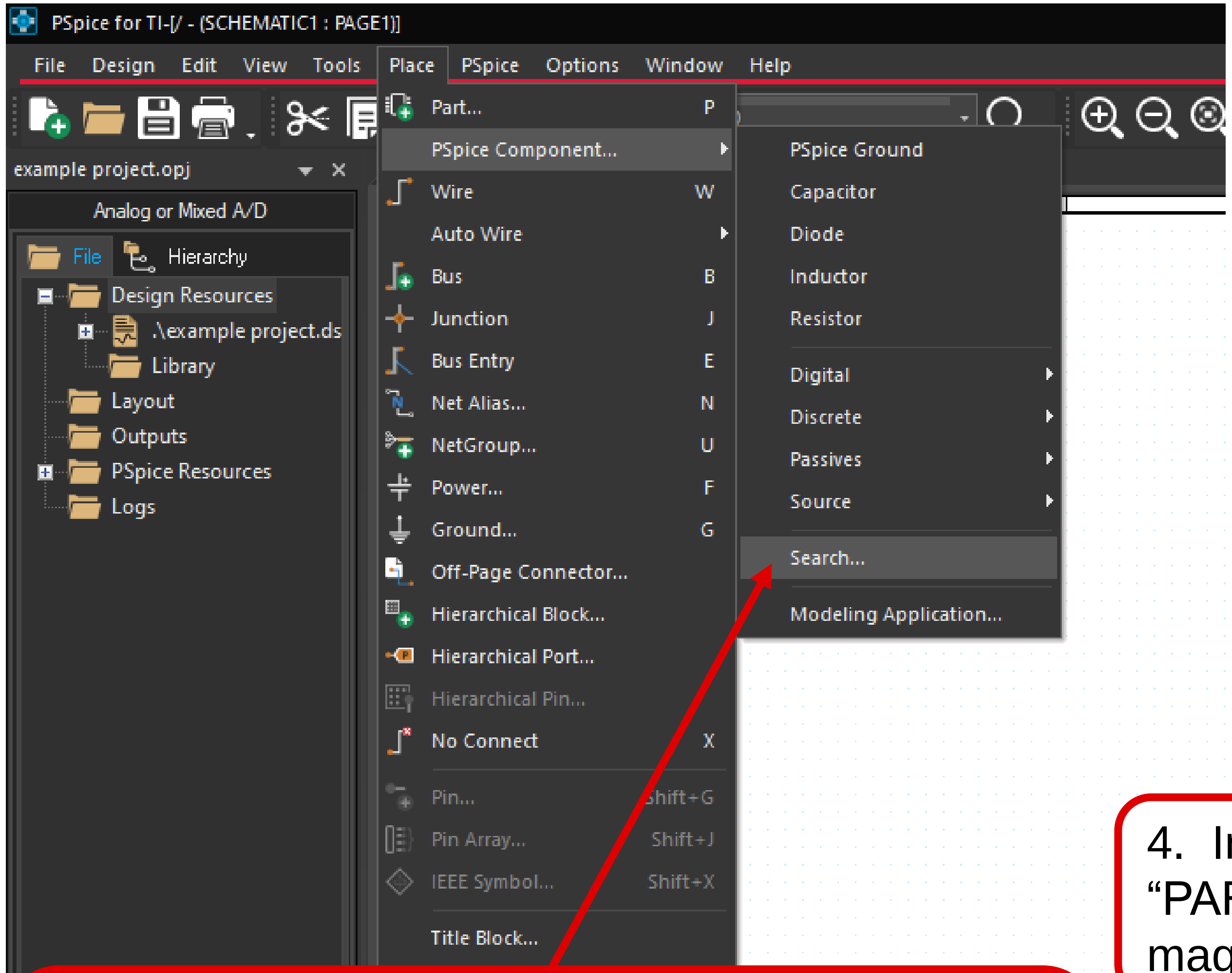


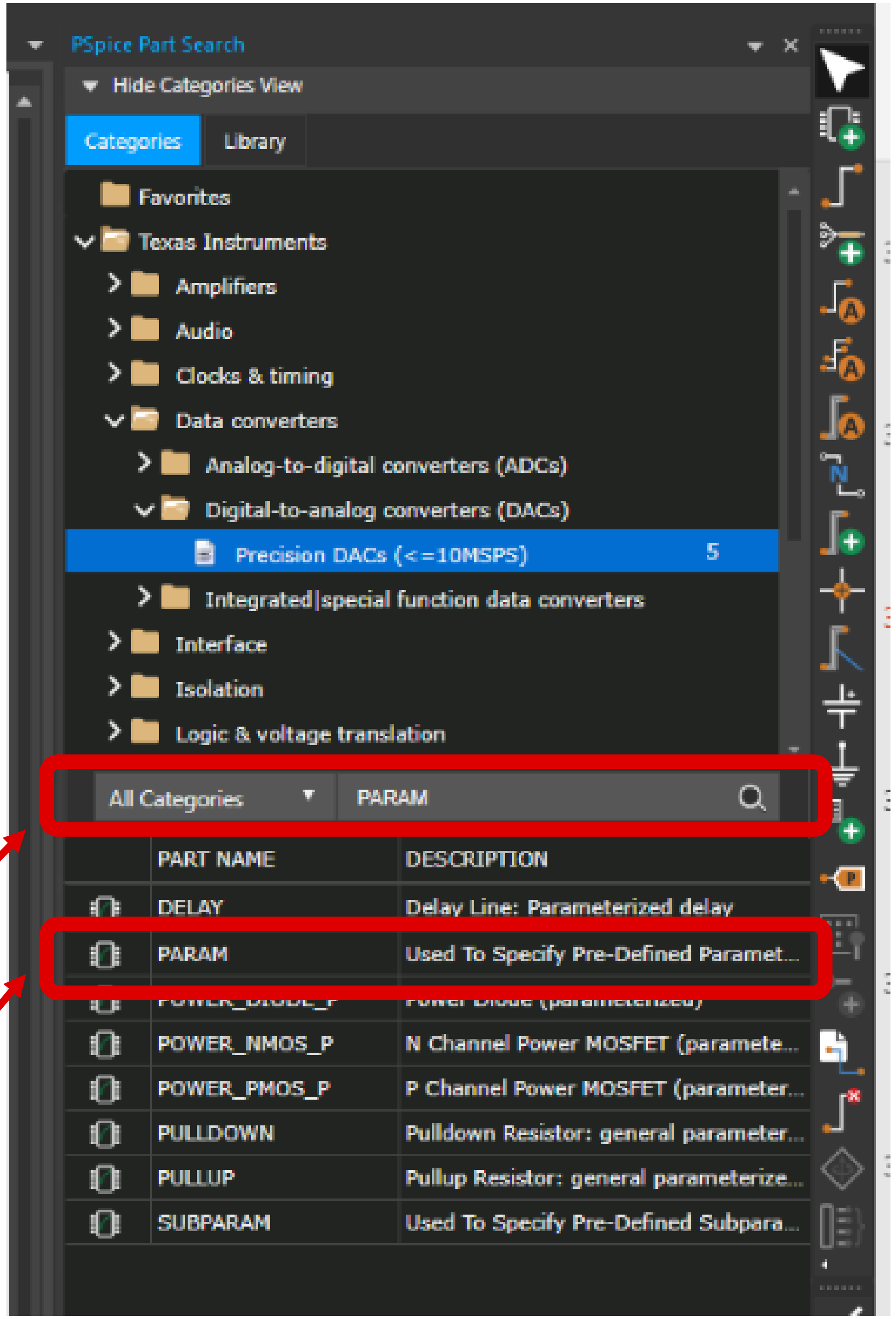


1. Draw circuit schematic. For this example, use 100kHz (10us period) square wave.

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

Parameter Stepping: Find PARAM





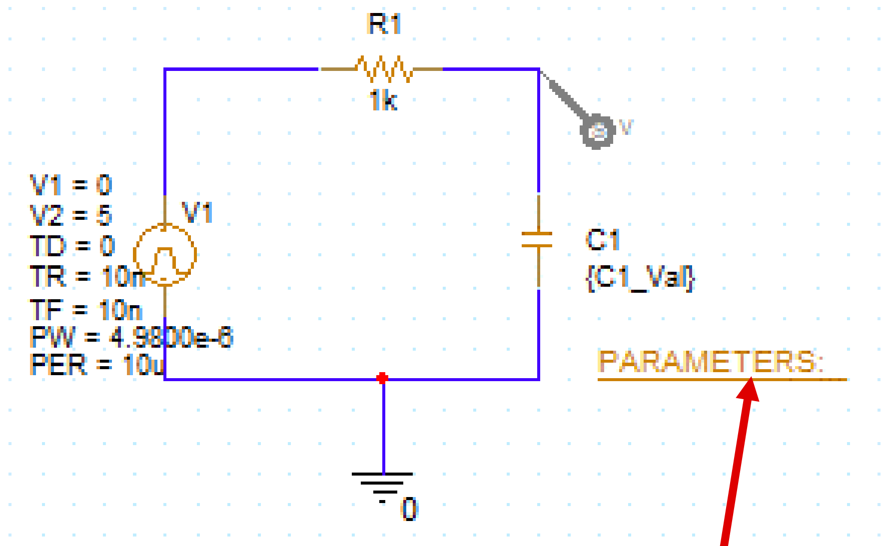
3. Select “Place > Pspice Component > Search” This will bring up the PSPICE search utility that you typically use to find TI components.

4. In the search window type “PARAM” and press the seach magnifying glass.

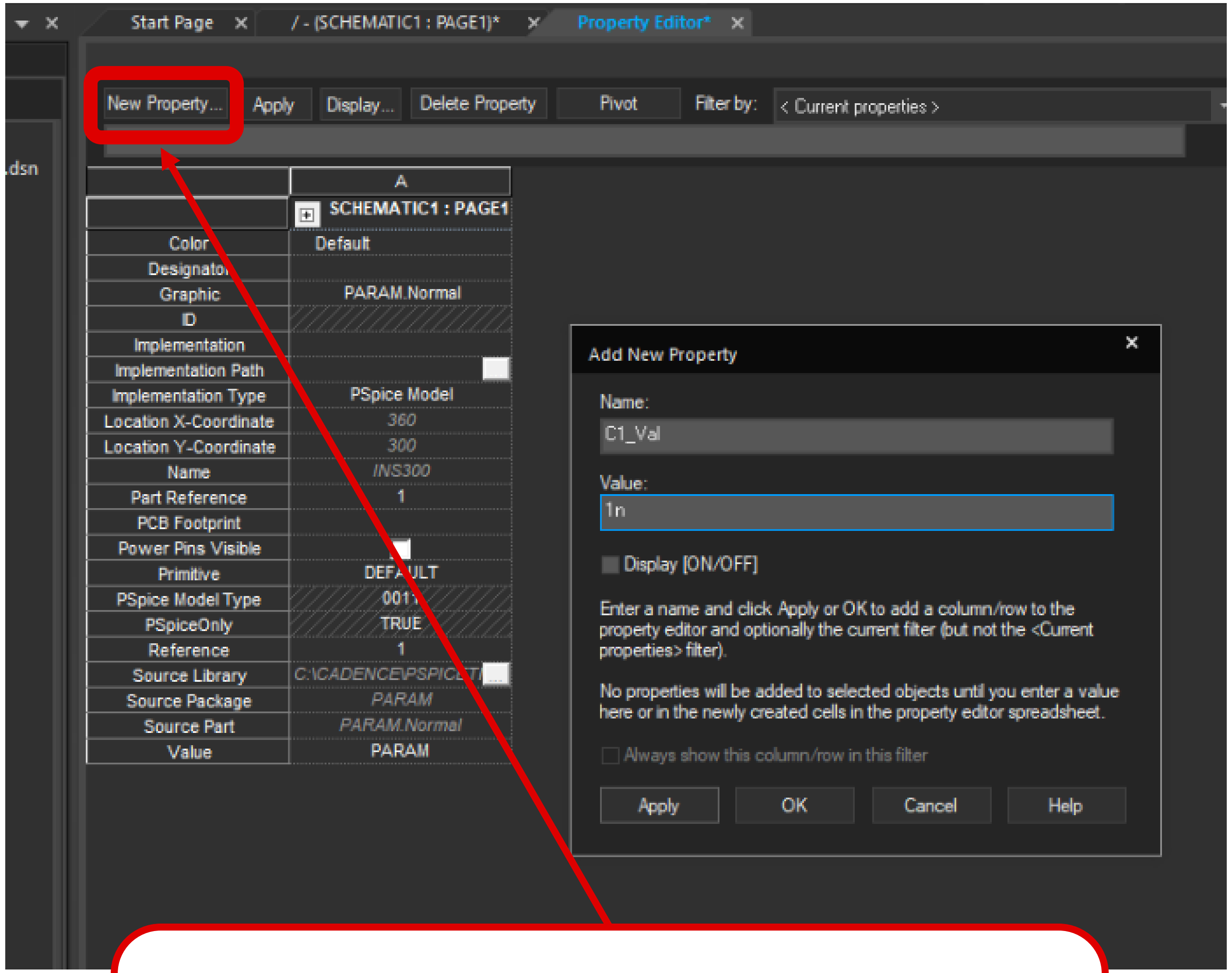
5. Double click on “PARAM” and drag it onto your schematic.

Back to Table of Contents

Parameter Stepping: Edit PARAM



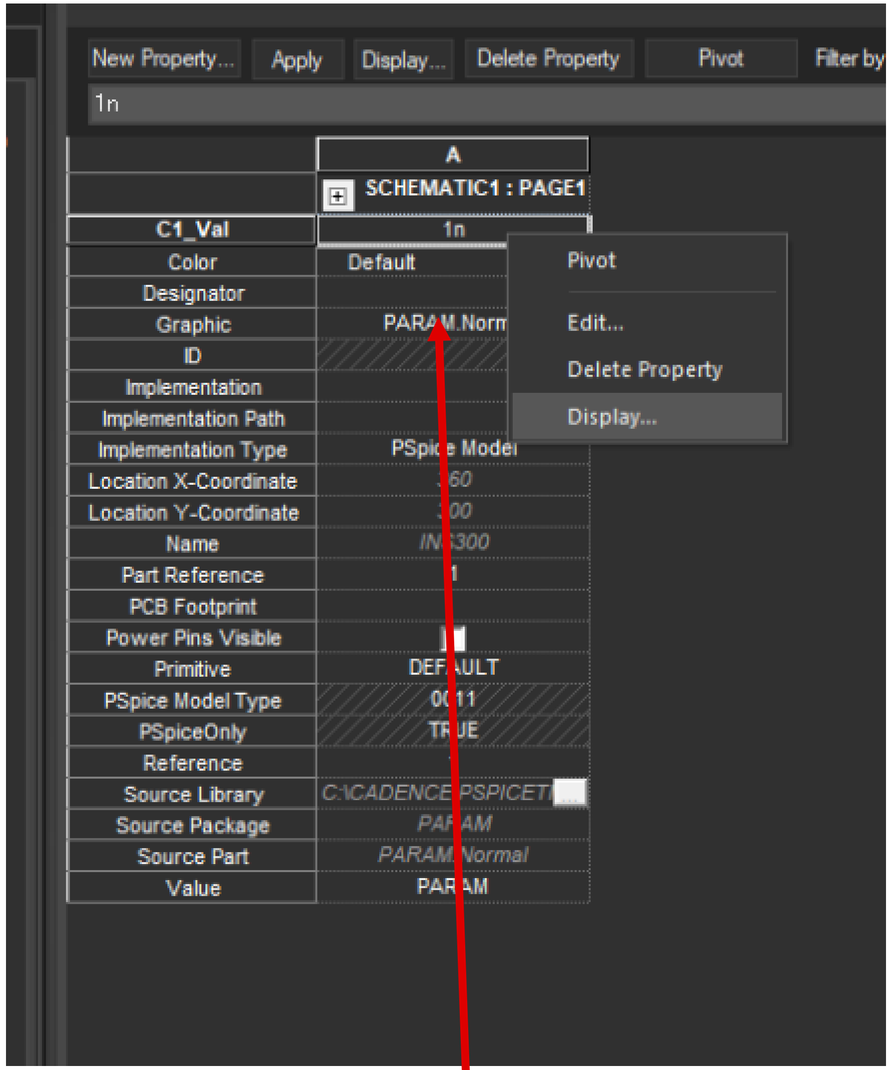
6. Double click the parameter to edit.

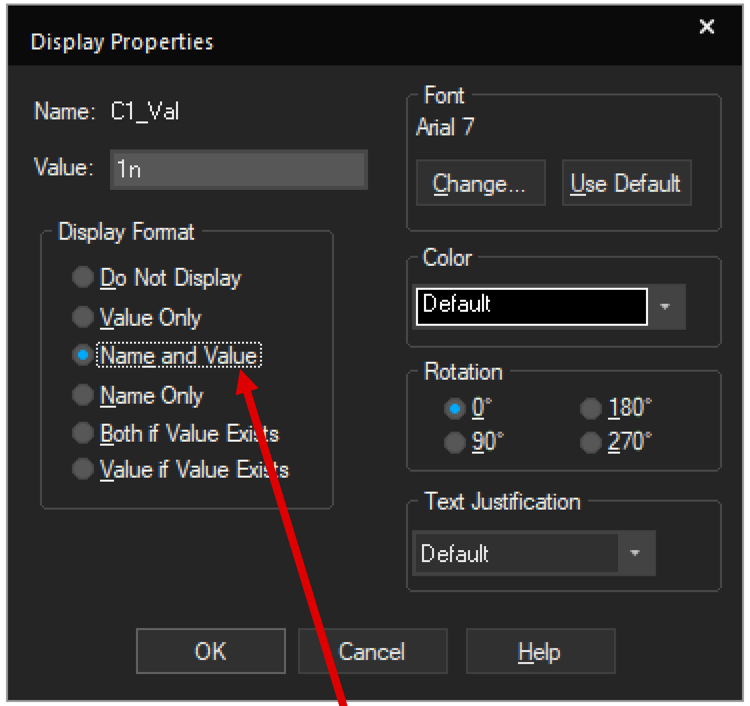


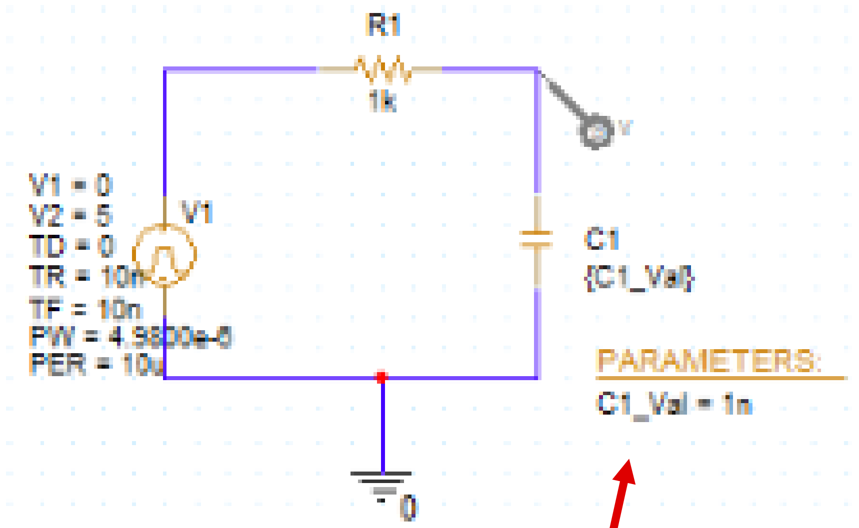
7. Press the “New Property” button.
8. Enter the “Name” to match the name in the braces: {C1_Val}
9. Enter a good value (e.g. 1n) and press ok.

Back to Table of Contents

Parameter Stepping: Display value





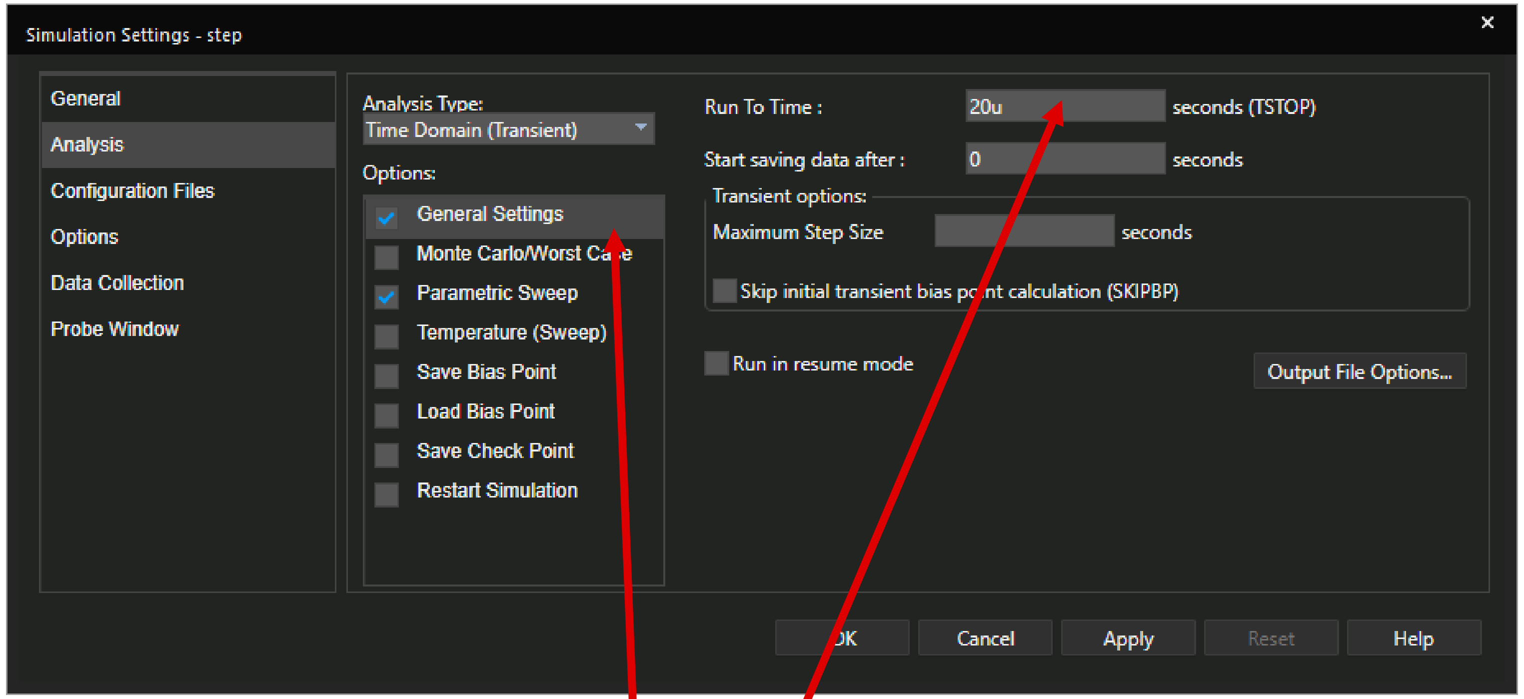


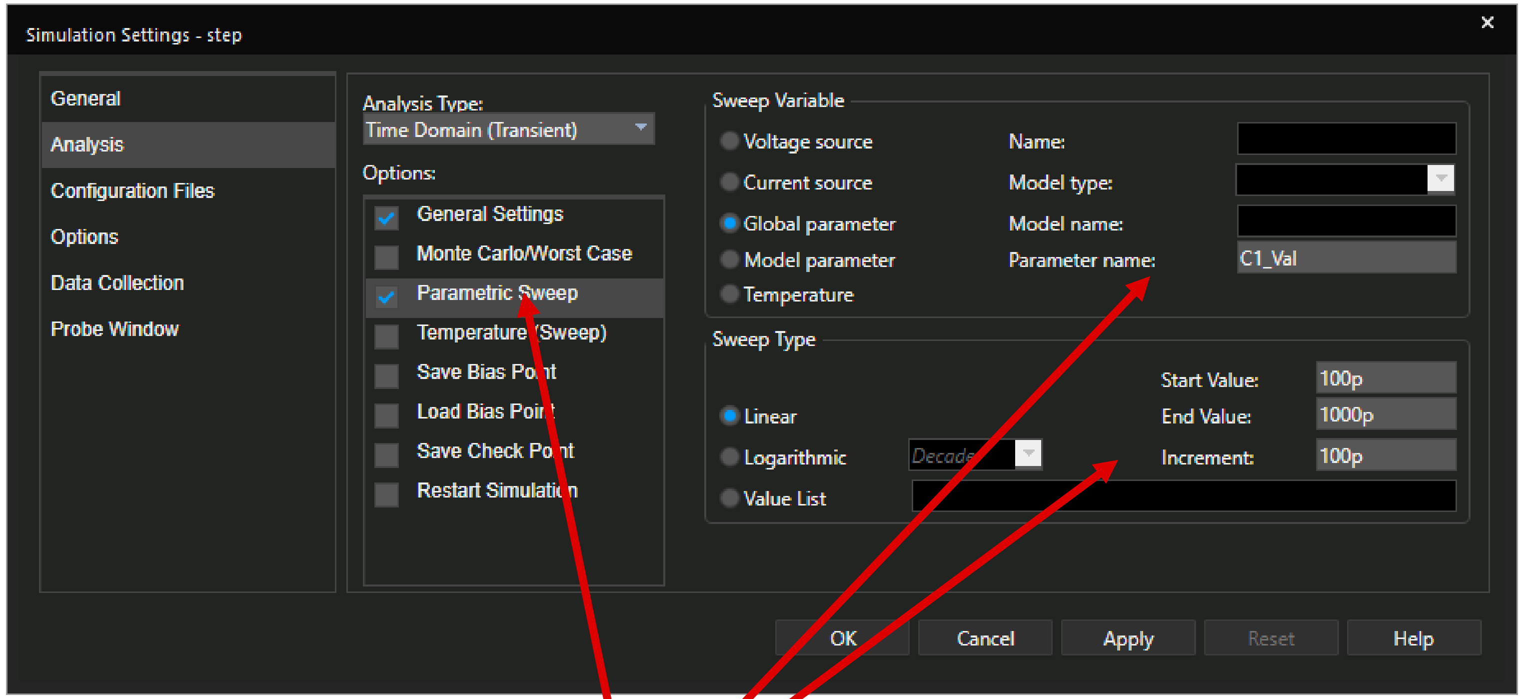
10. Right click and select “Display” to display the value on the schematic.

11. Select “Name and Value” and press “ok”

12. Schematic after displaying the value.

Parameter Stepping: Set up analysis profile

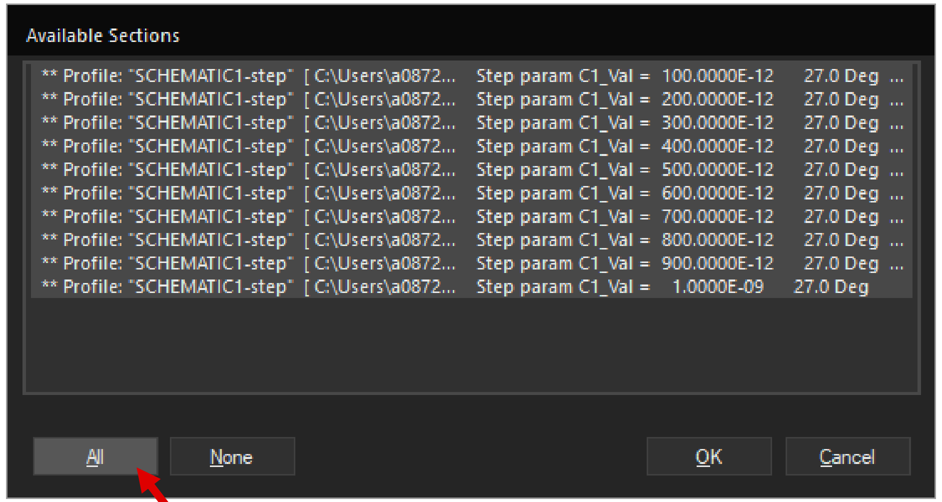


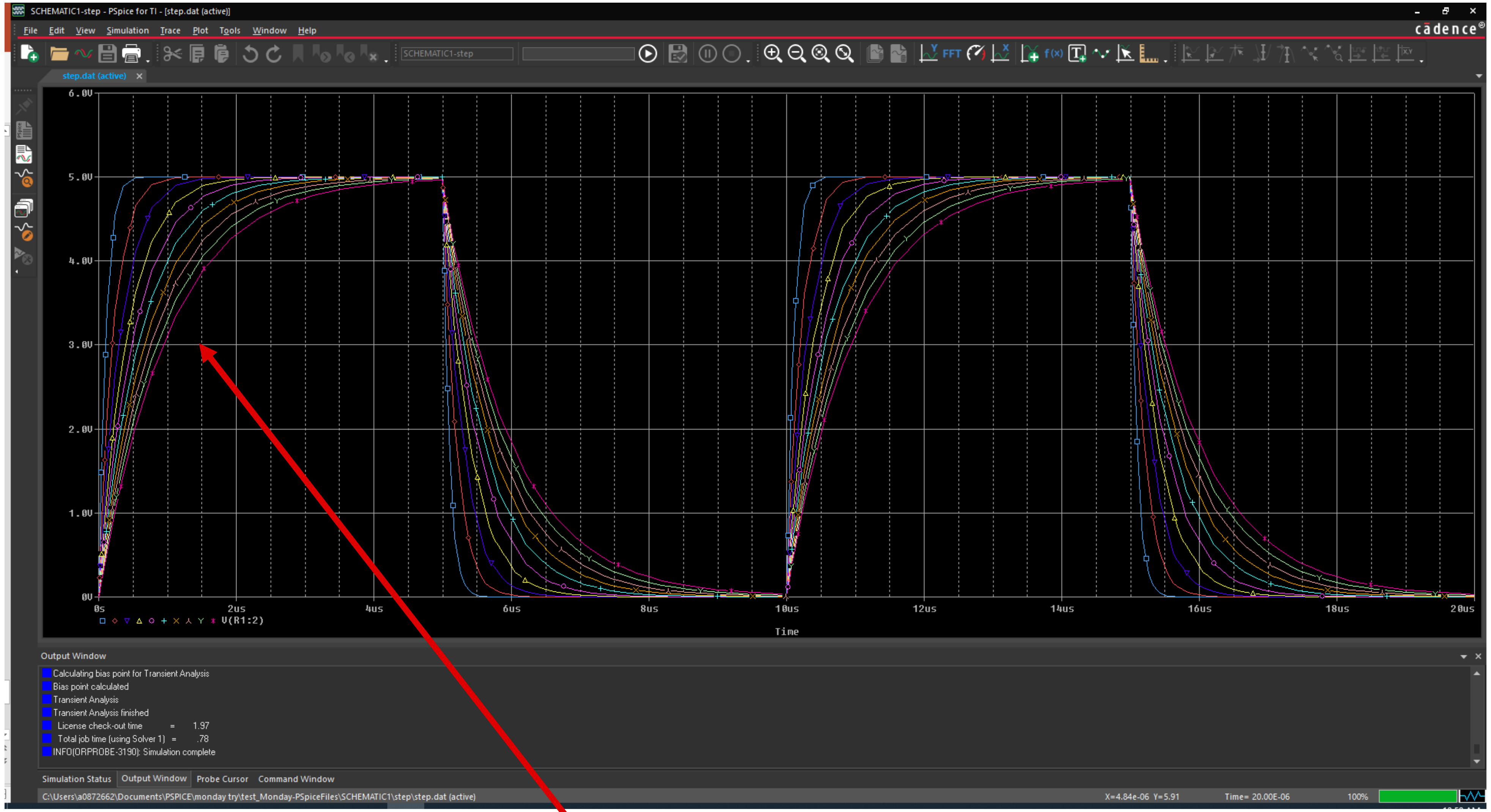


14. Under “Parametric Sweep”, select:
    a. Parameter name
    b. Sweep type (Linear in this example)
    c. Start value, end value, and increment

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

Parameter Stepping: Example Results

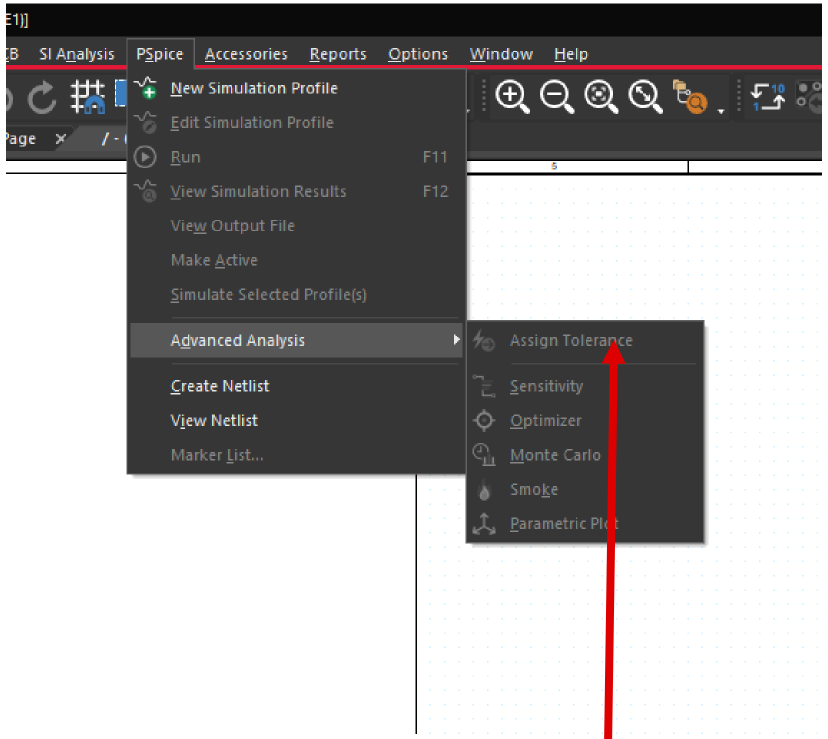


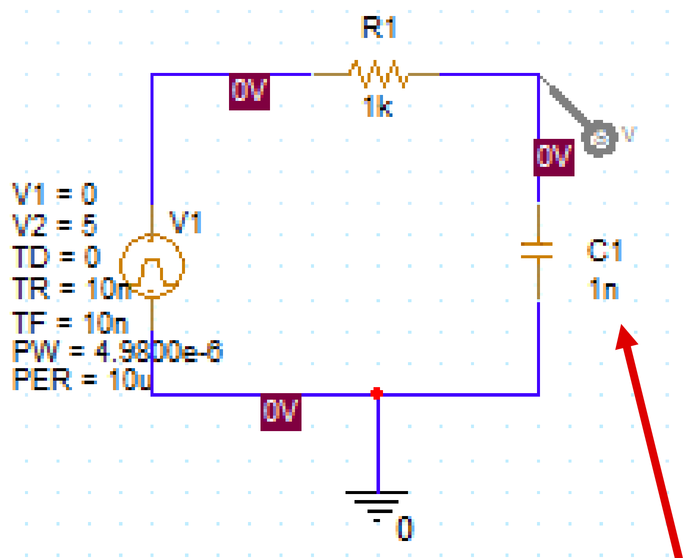


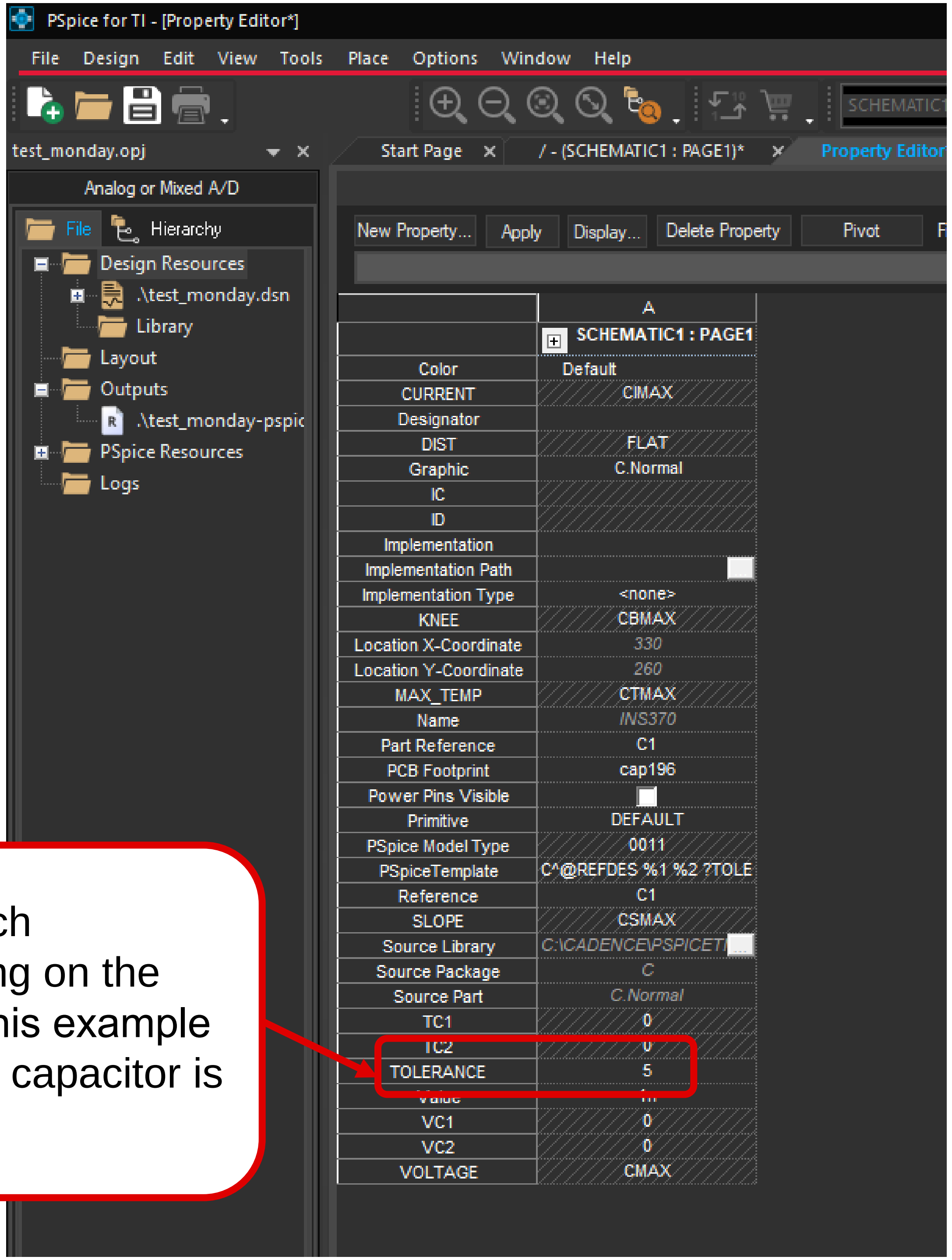
15. Select "all" and press "ok"

16. Parameter step results.

Monte Carlo Analysis





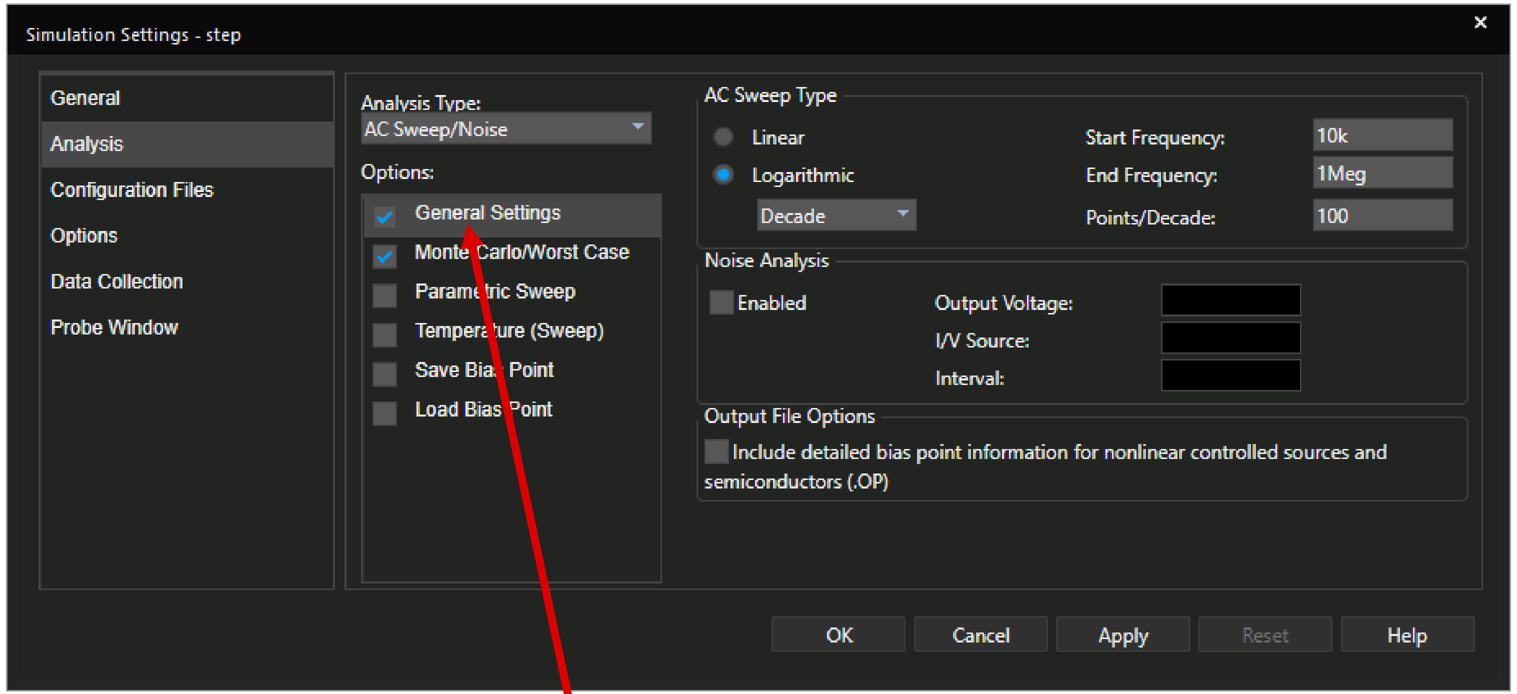


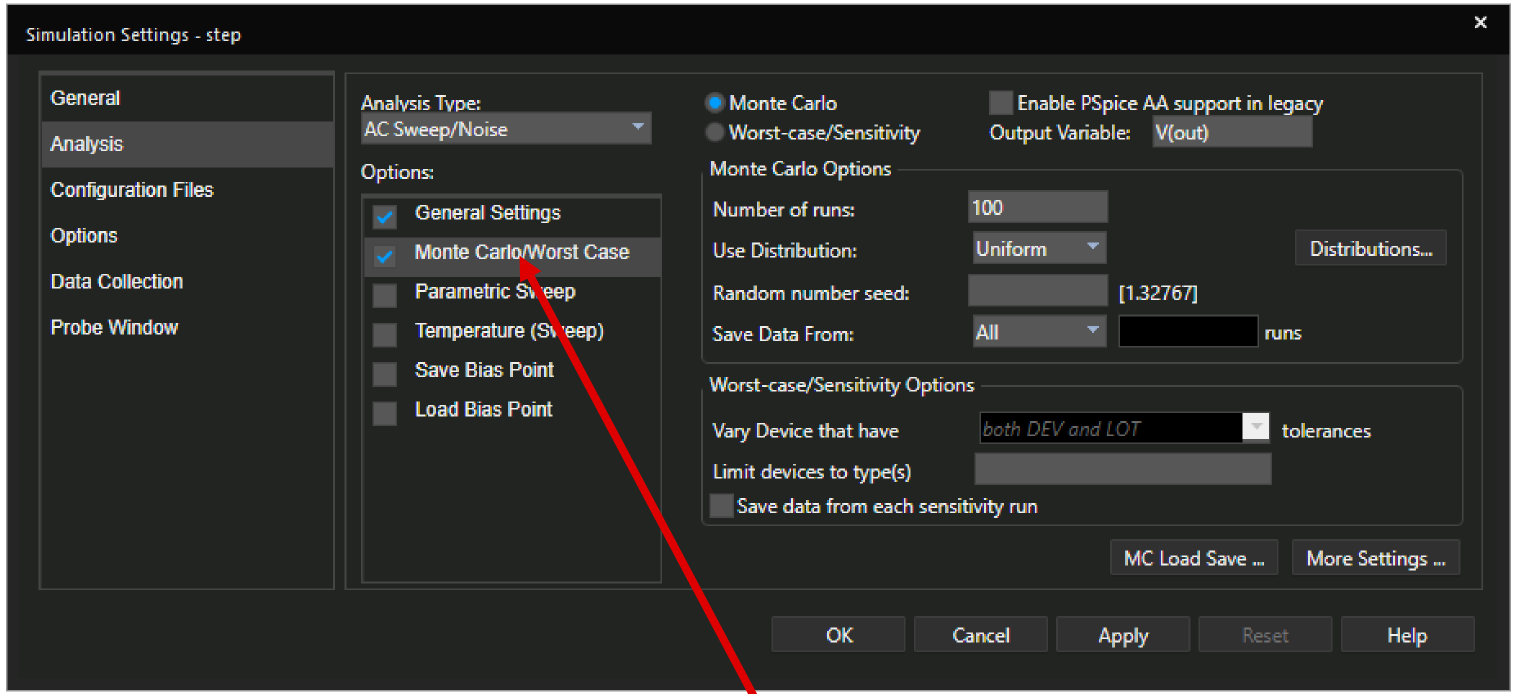
1. Note: the full version of Cadence allows assigning tolerances to all components using “assign tolerance” option. If you did a complex design and wanted to compare 0.1% tolerance to 1% tolerance results this would be a useful option. This is not available on PSPICE for TI.

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%.

Back to Table of Contents

Monte Carlo Analysis

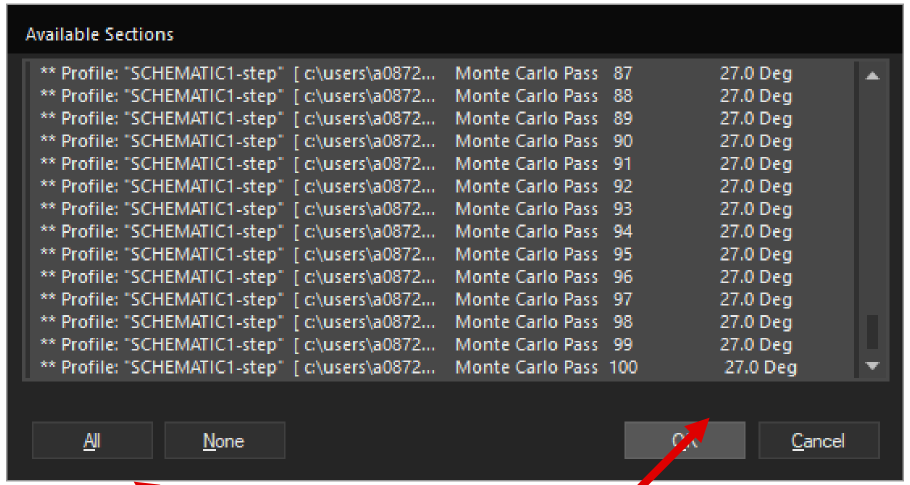


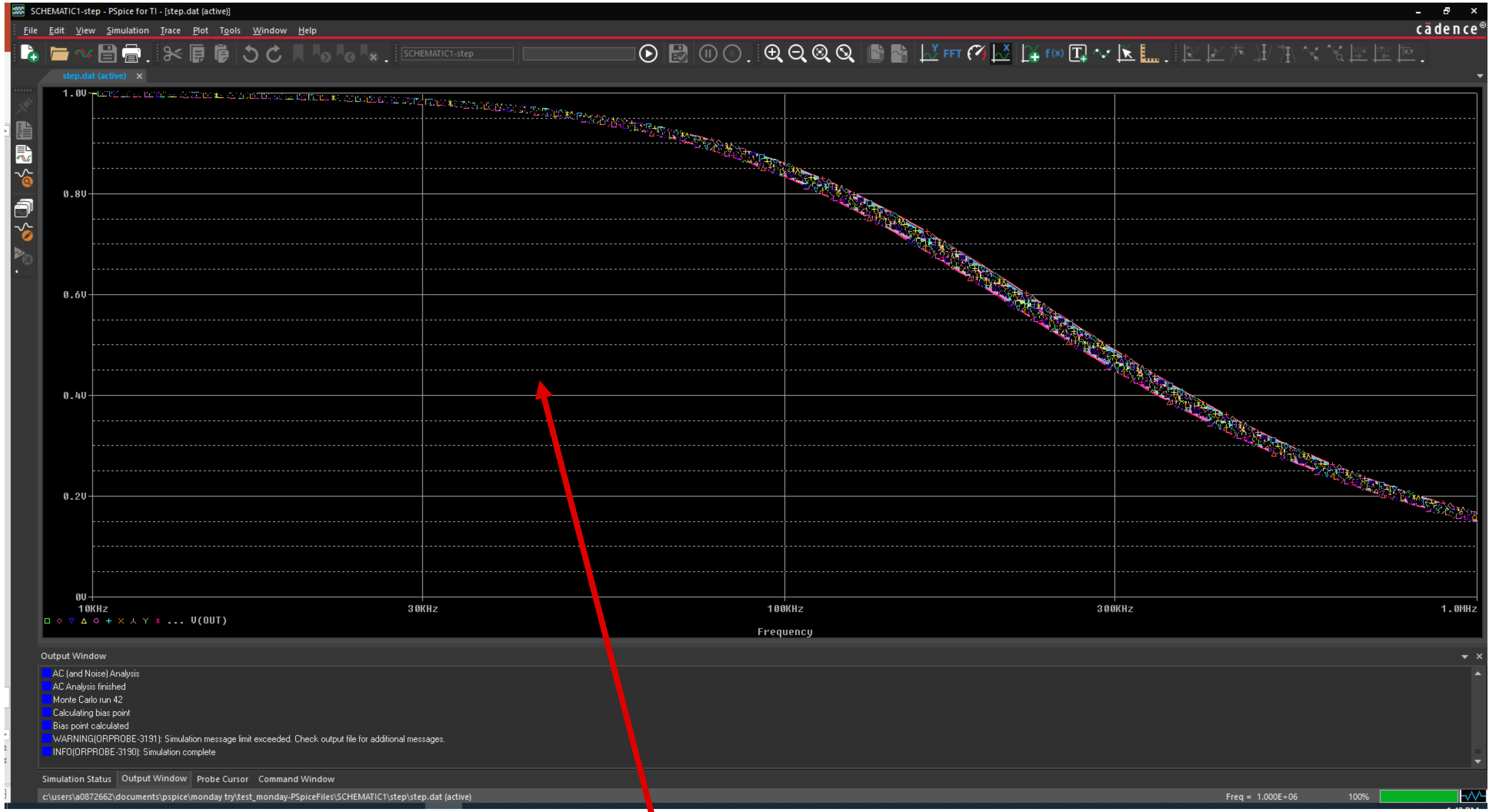


3. Choose analysis type for "General Settings". In this example we use an AC sweep type.

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



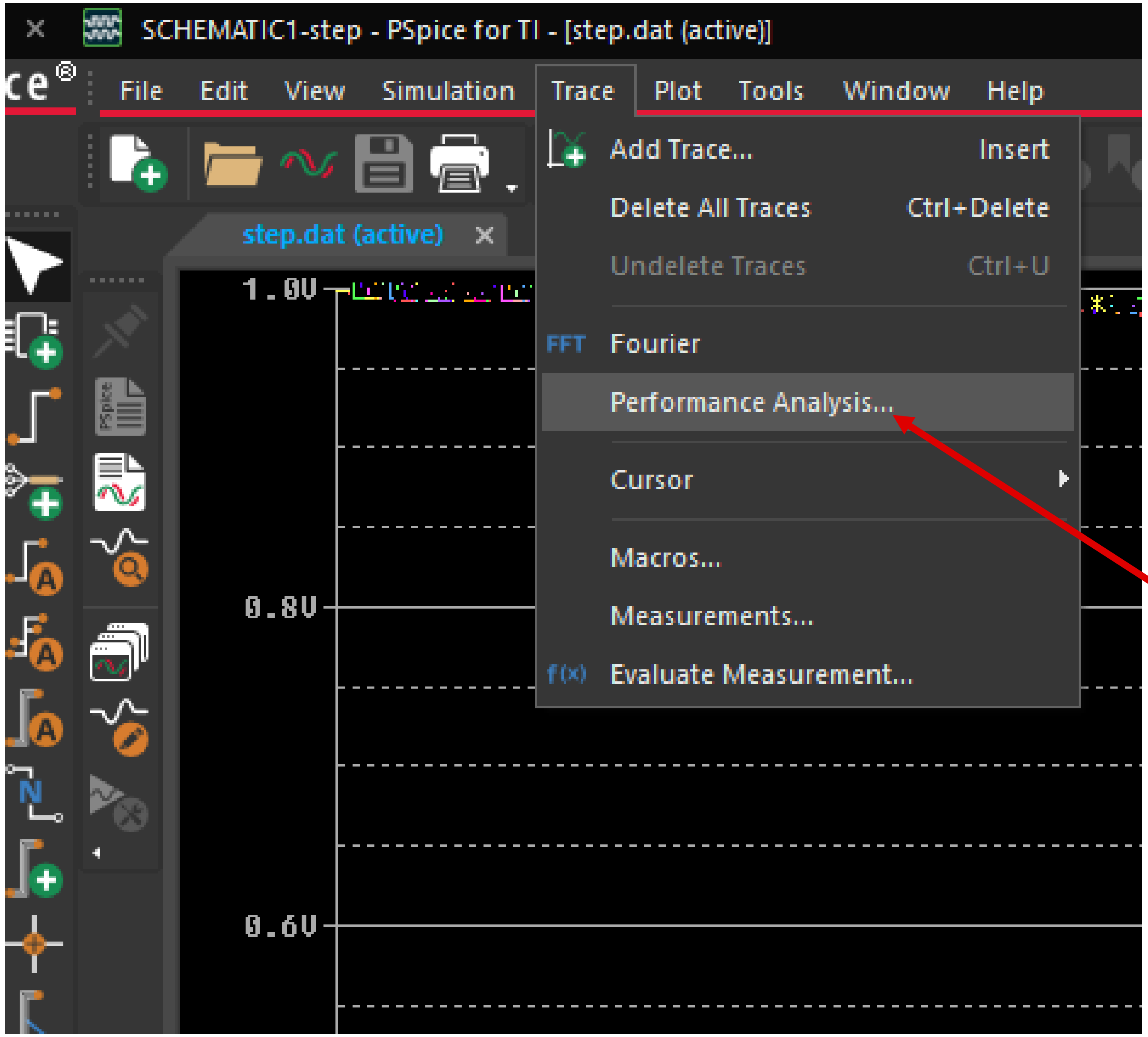


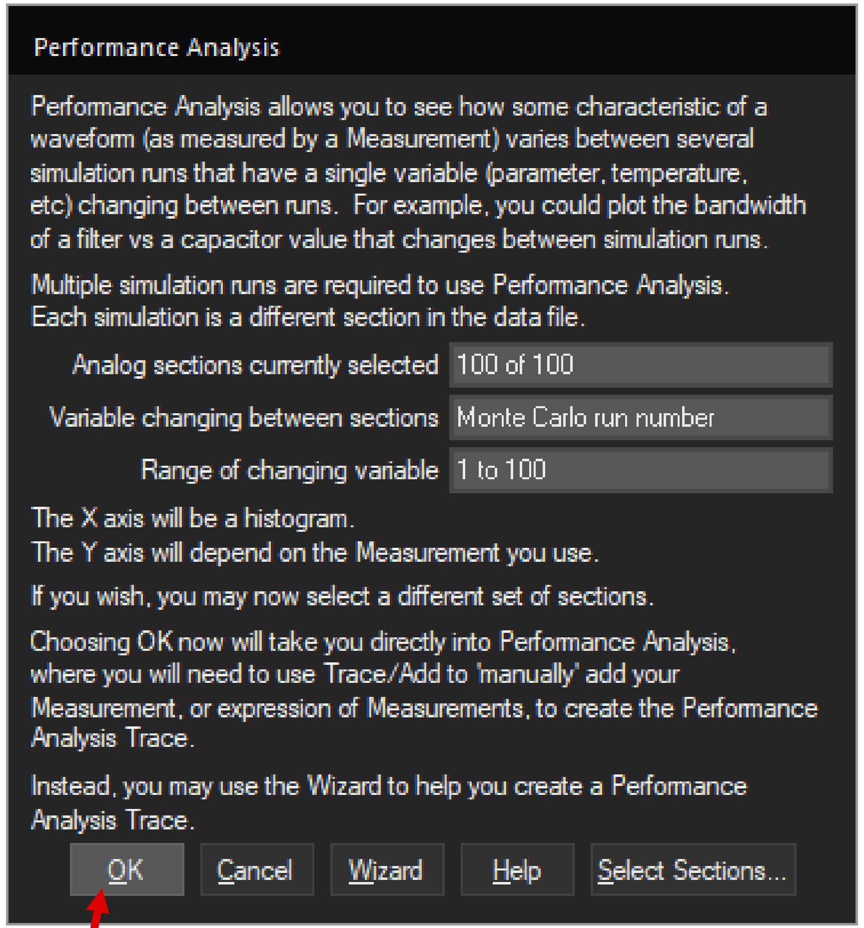
7. Run the simulation. When the simulation is complete you will get a list of available results. Select all and press ok.

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

Back to Table of Contents

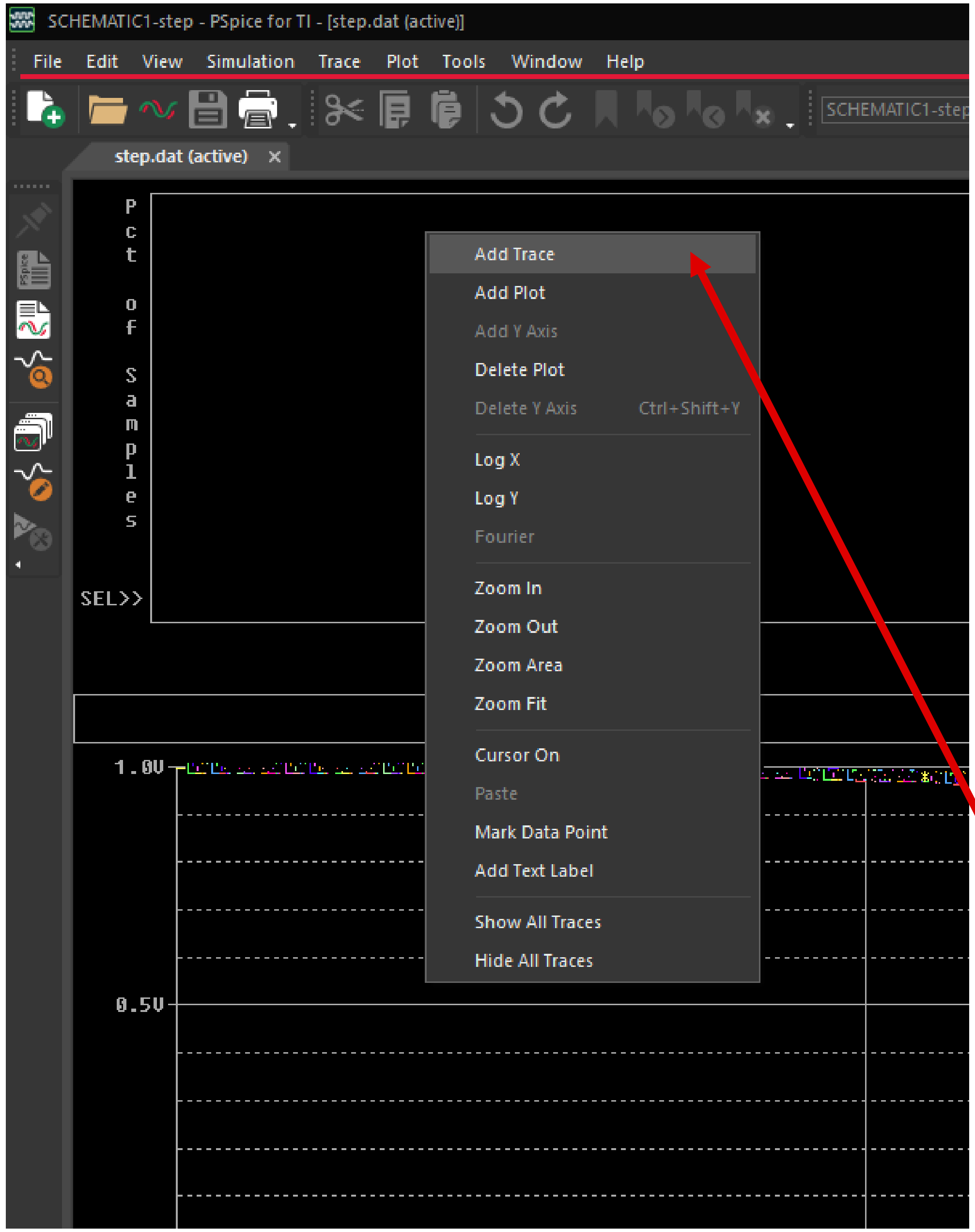
Monte Carlo Analysis

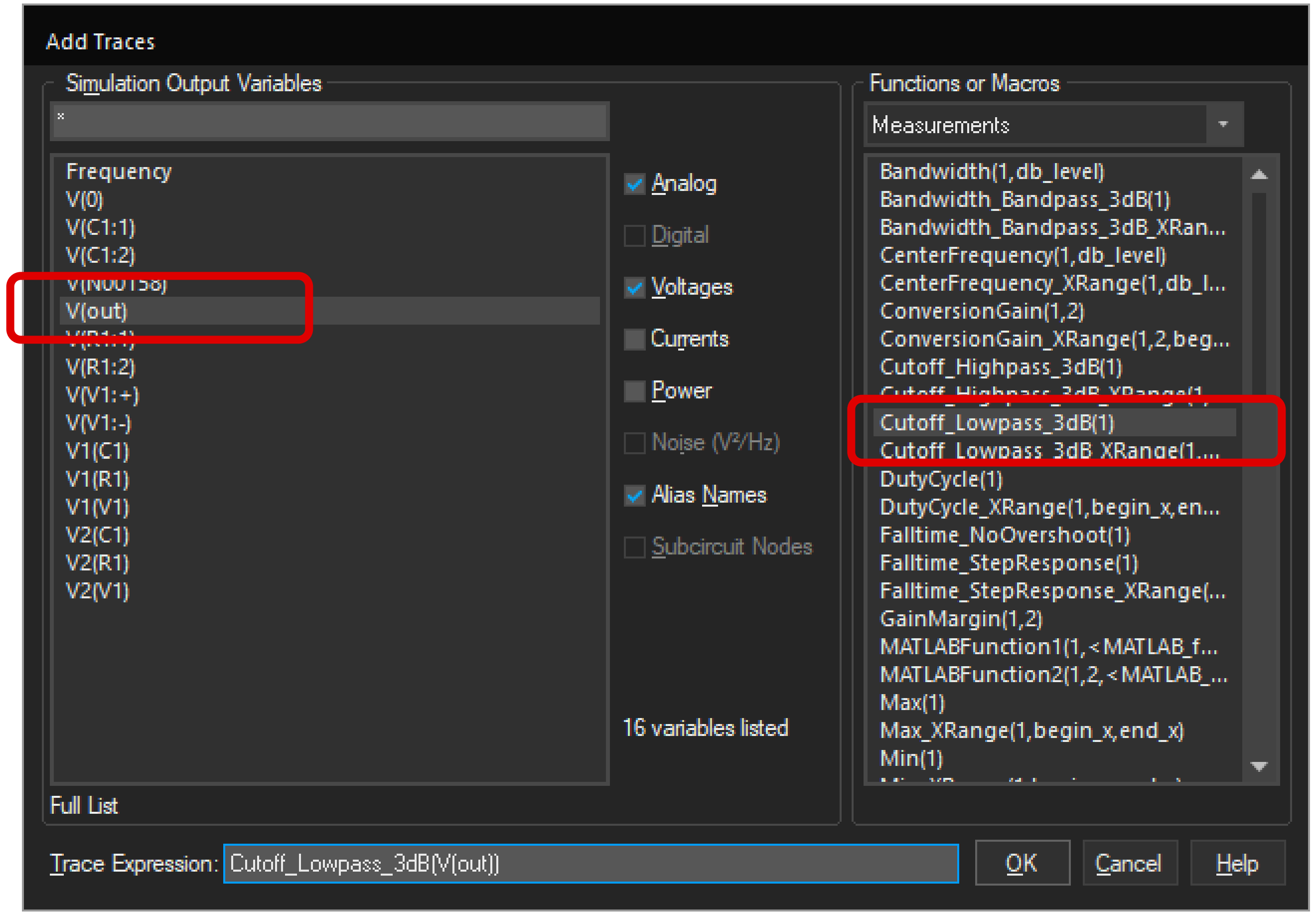




8. Choose performance analysis then press ok for the pop-up.

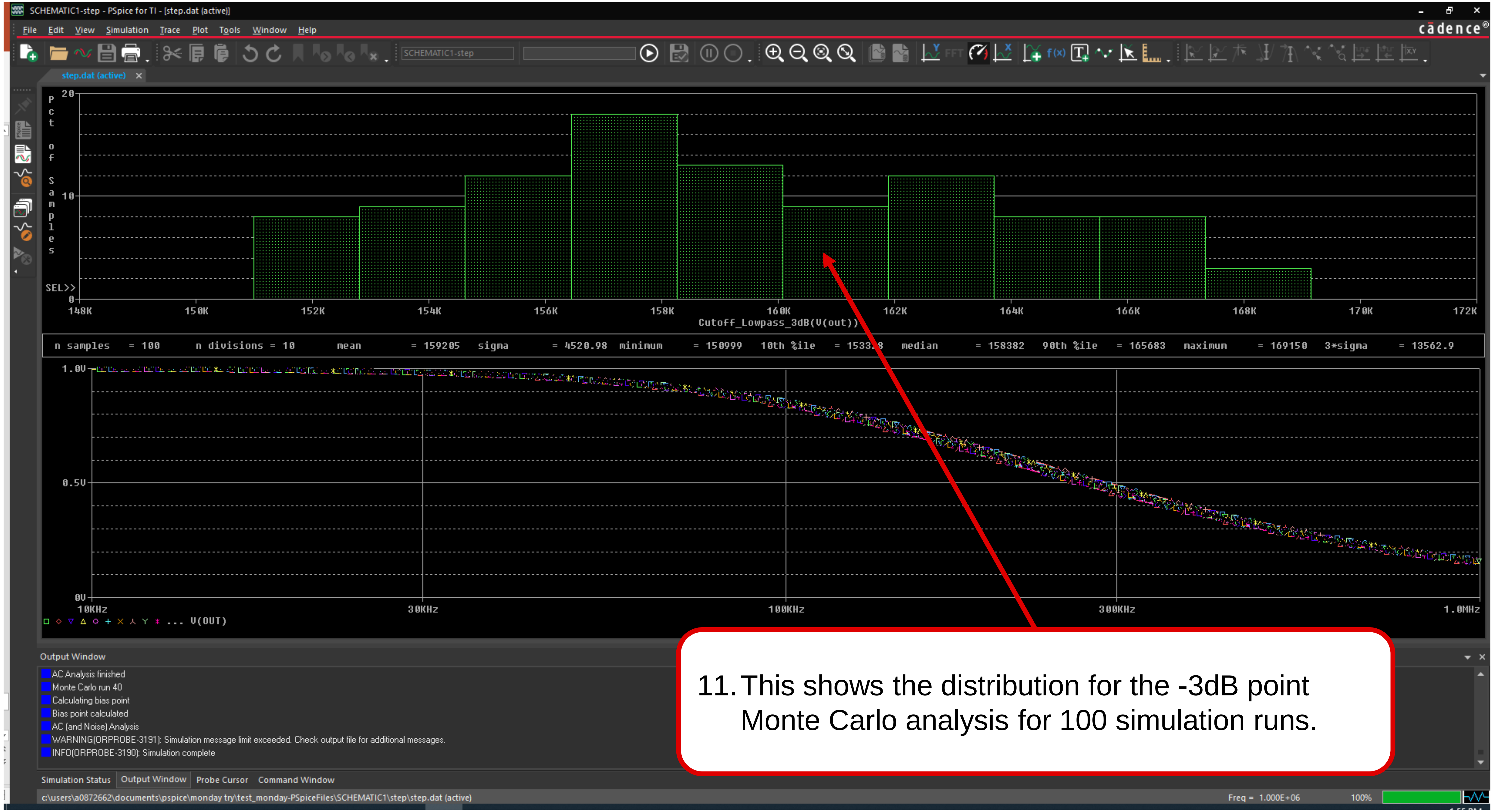
Monte Carlo Analysis





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).

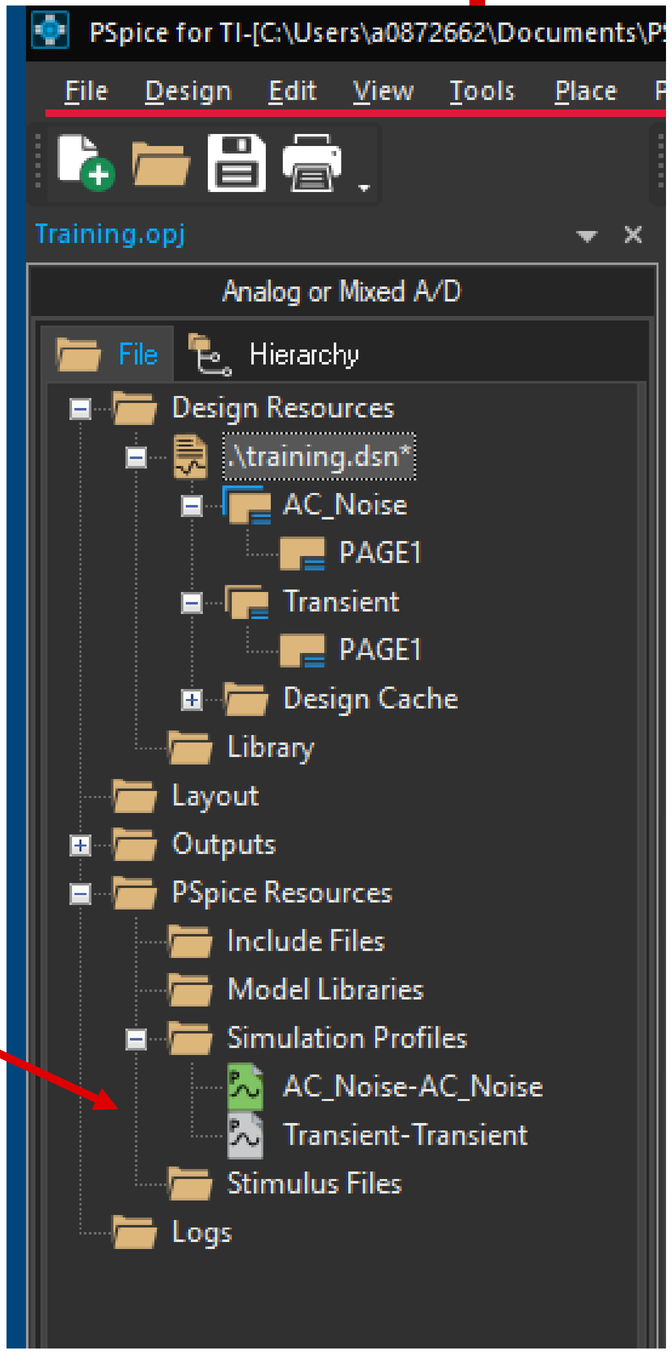
Monte Carlo Analysis



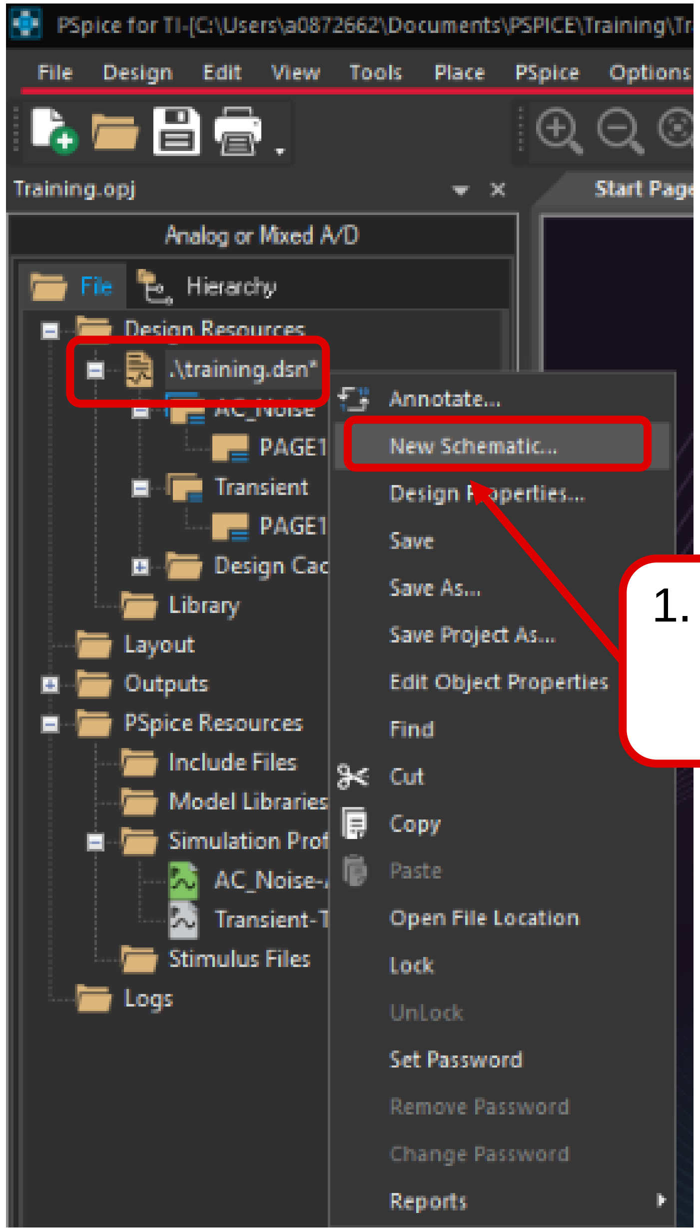
11. This shows the distribution for the -3dB point Monte Carlo analysis for 100 simulation runs.

Using multiple schematics and simulation profiles

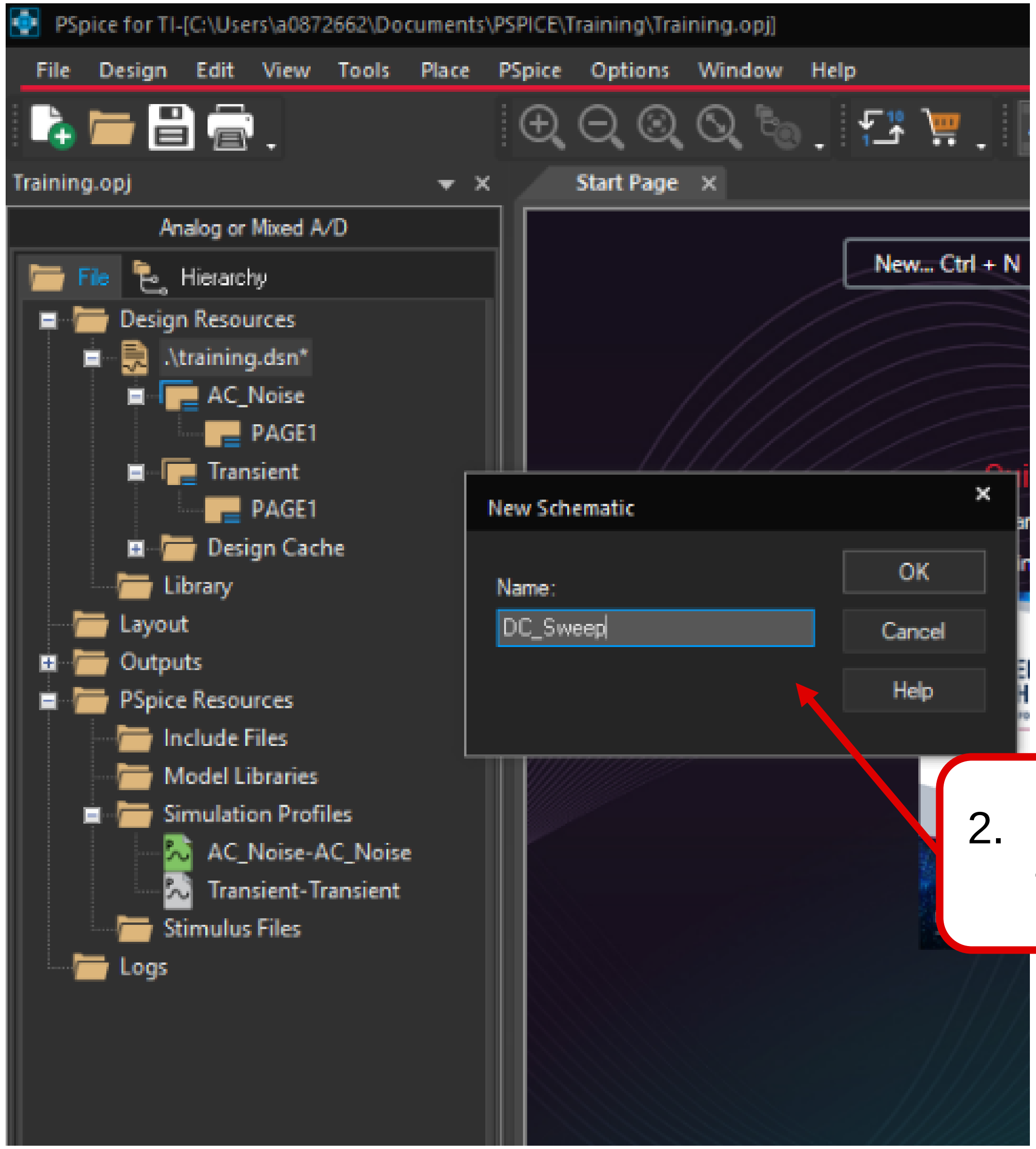
This shows a project with two schematics added and two simulation profiles. This approach can be useful if you want to do variations on different types of simulations. For example, a transient simulation requires a square wave source, and an AC simulation requires an AC source. You can build two slightly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



Using multiple schematics and simulation profiles

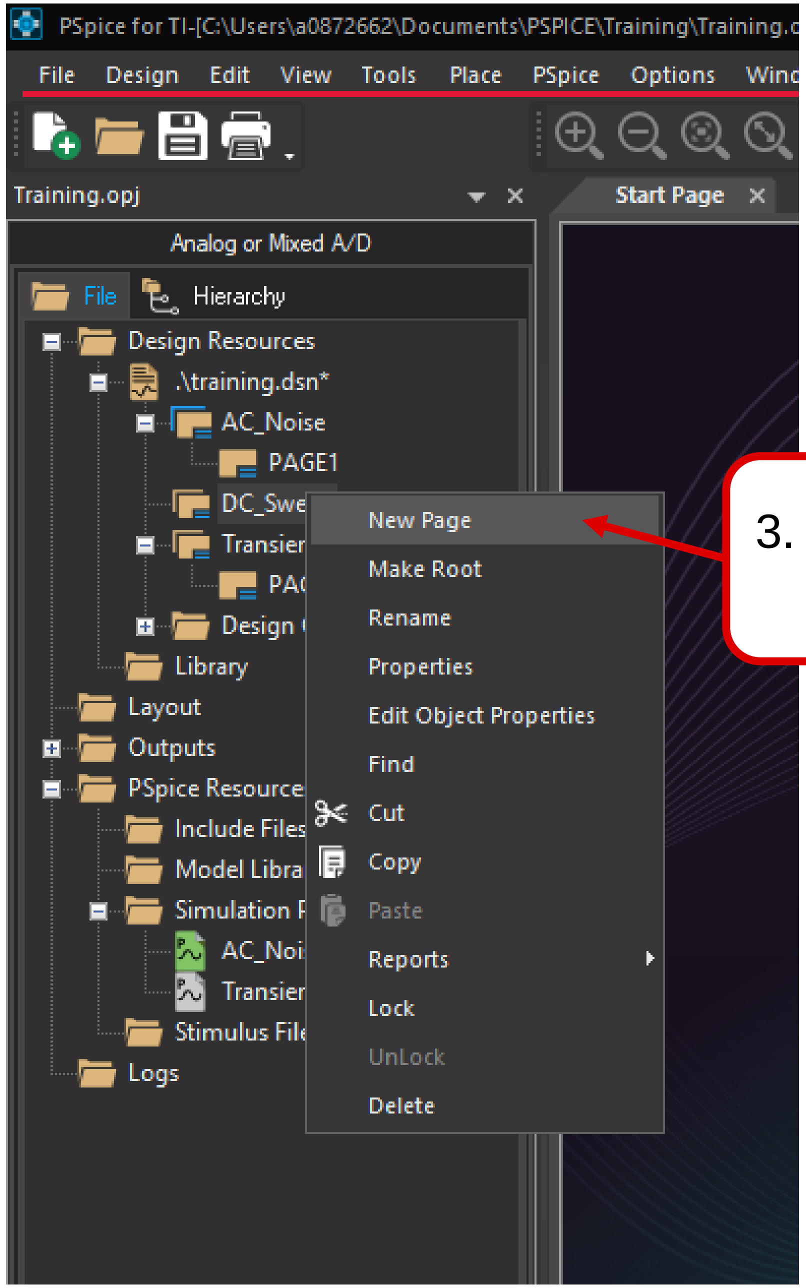


1. Right click on project and select “New Schematic”

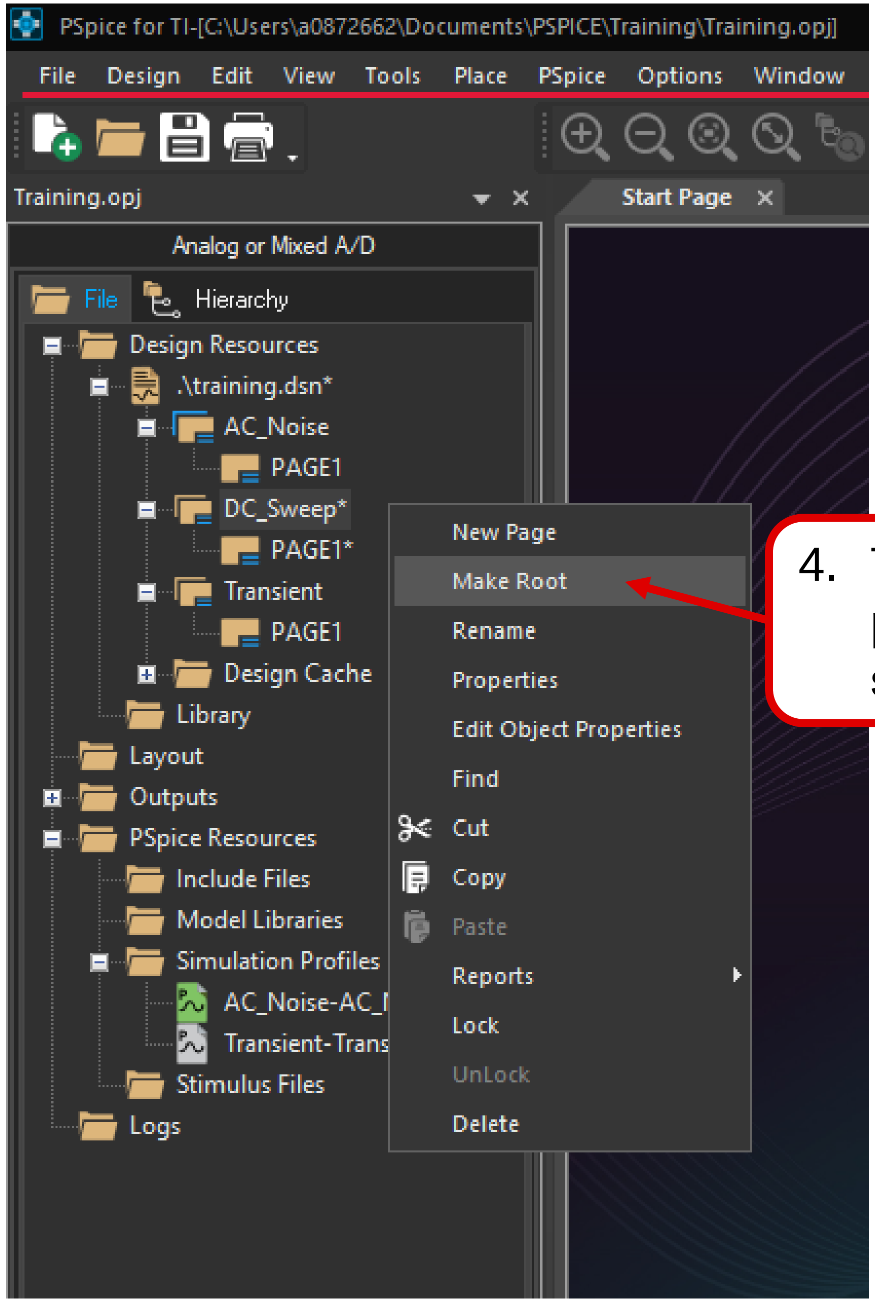


2. Choose a name for the schematic

Using multiple schematics and simulation profiles

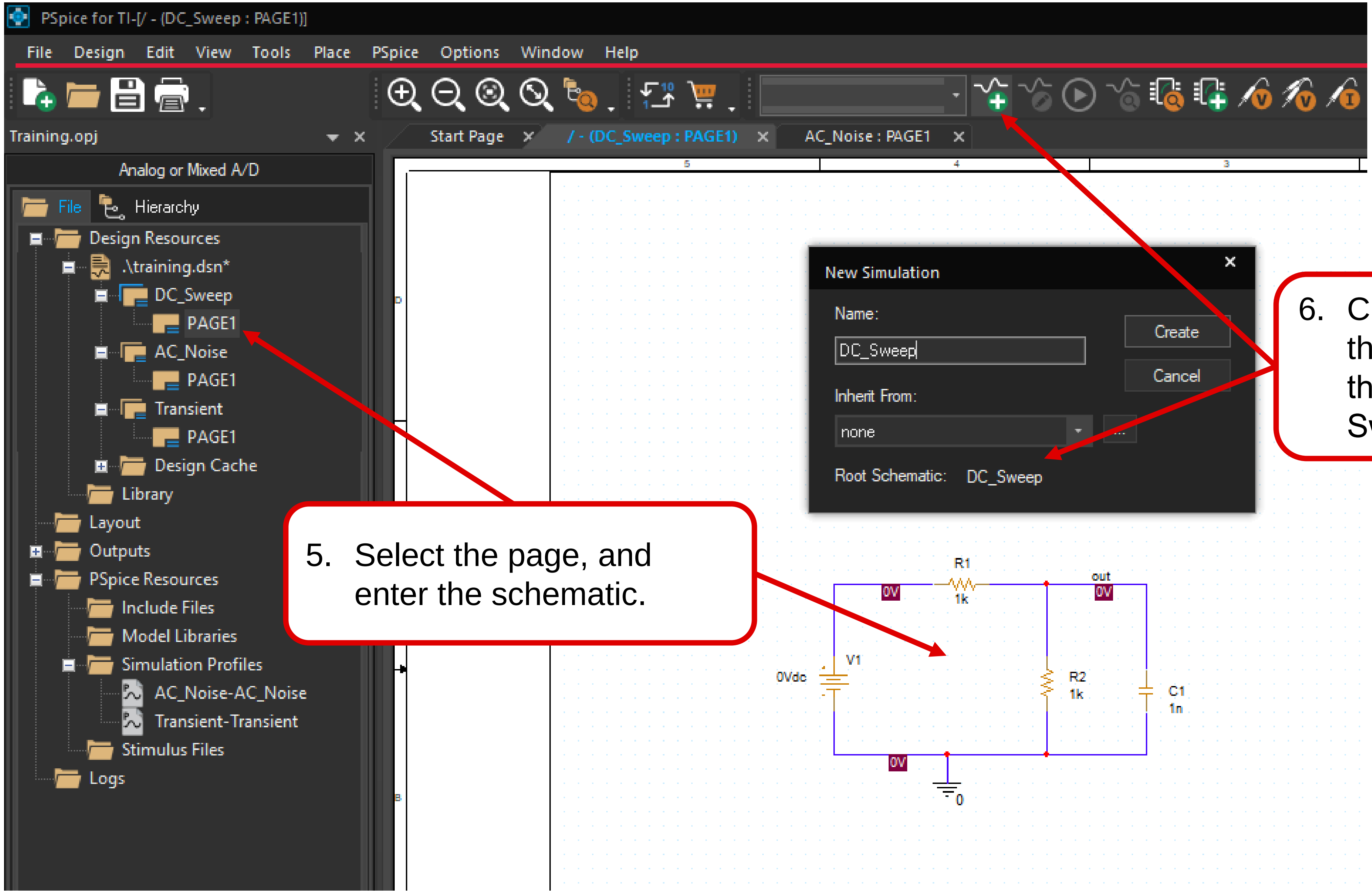


3. Add a page to the schematic



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

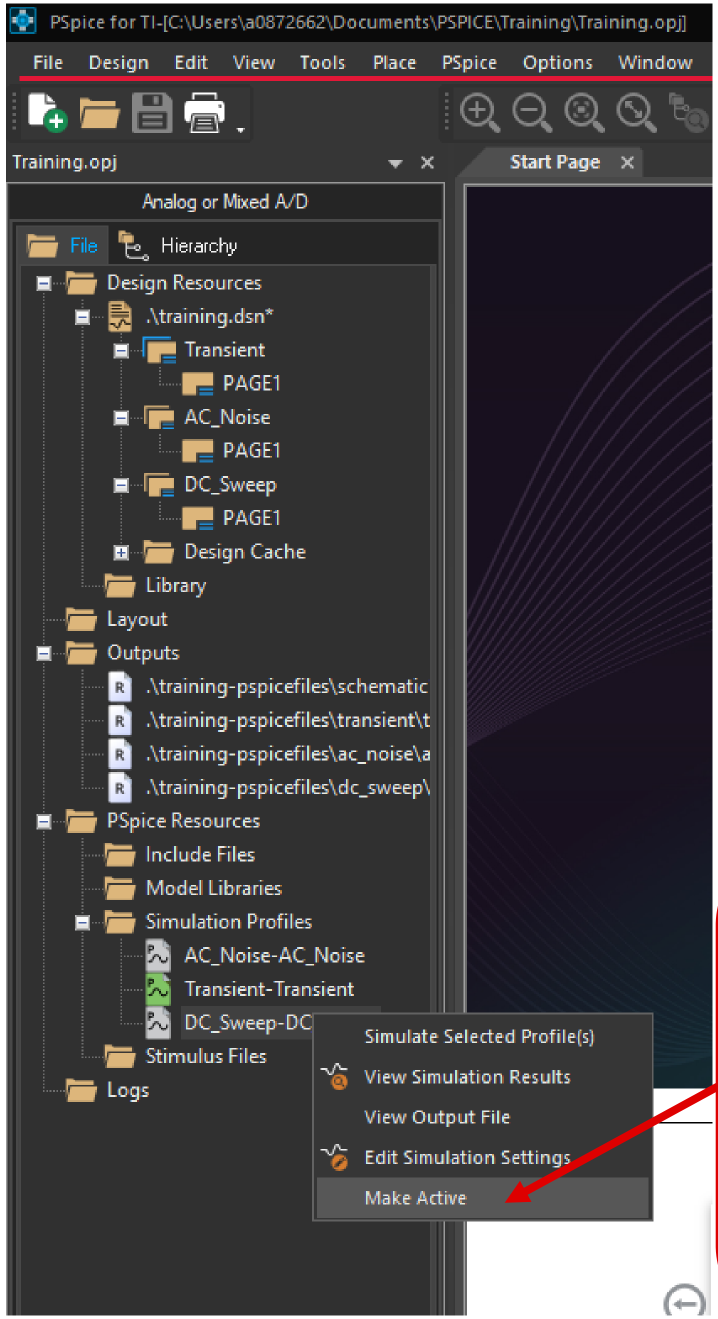
Using multiple schematics and simulation profiles

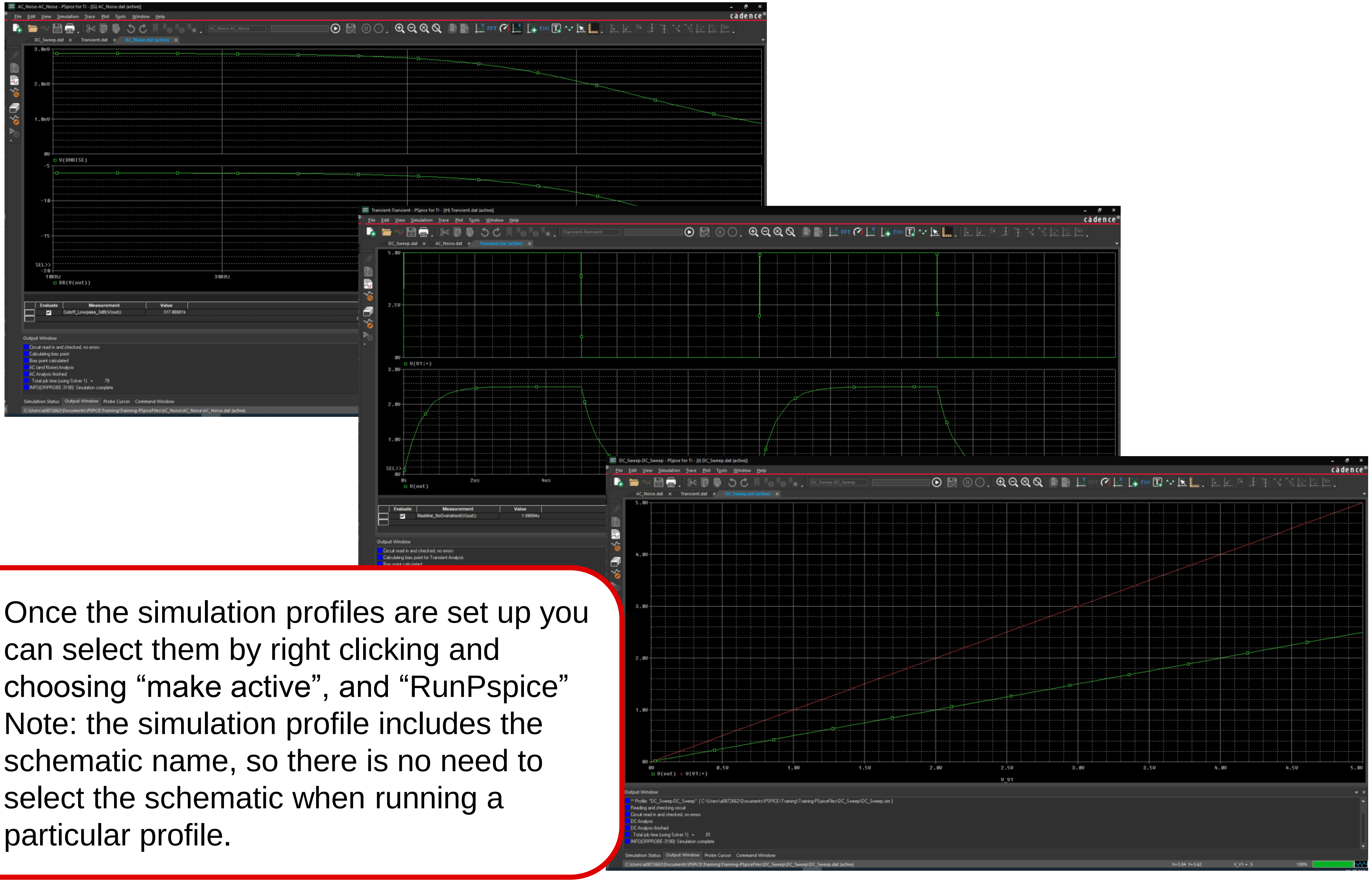


5. Select the page, and enter the schematic.

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

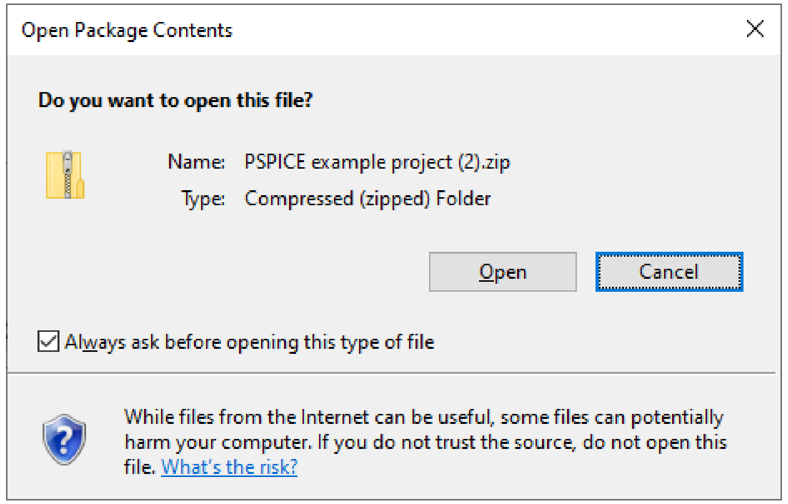


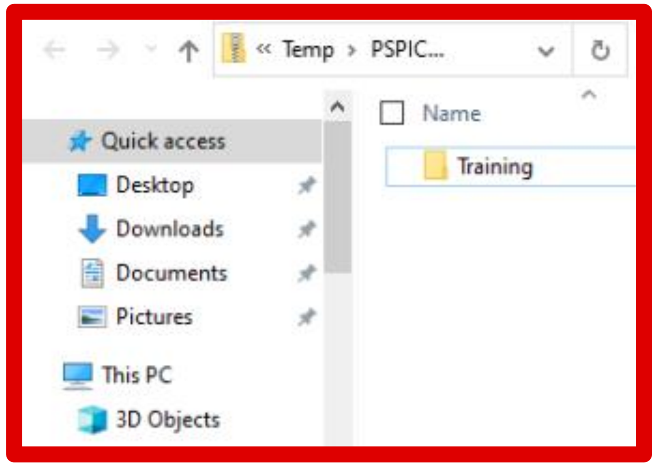


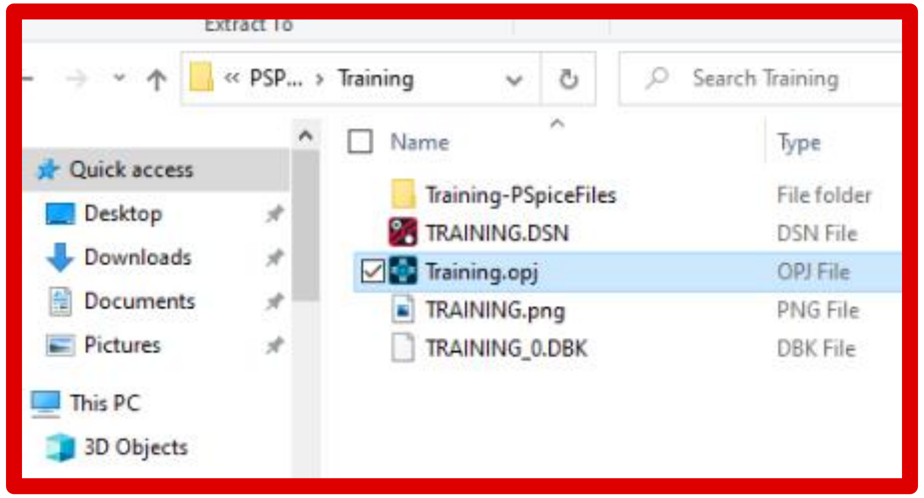
5. Once the simulation profiles are set up you can select them by right clicking and choosing “make active”, and “RunPspice”
6. Note: the simulation profile includes the schematic name, so there is no need to select the schematic when running a particular profile.

Using multiple schematics and simulation profiles

PSPICE example project.zip

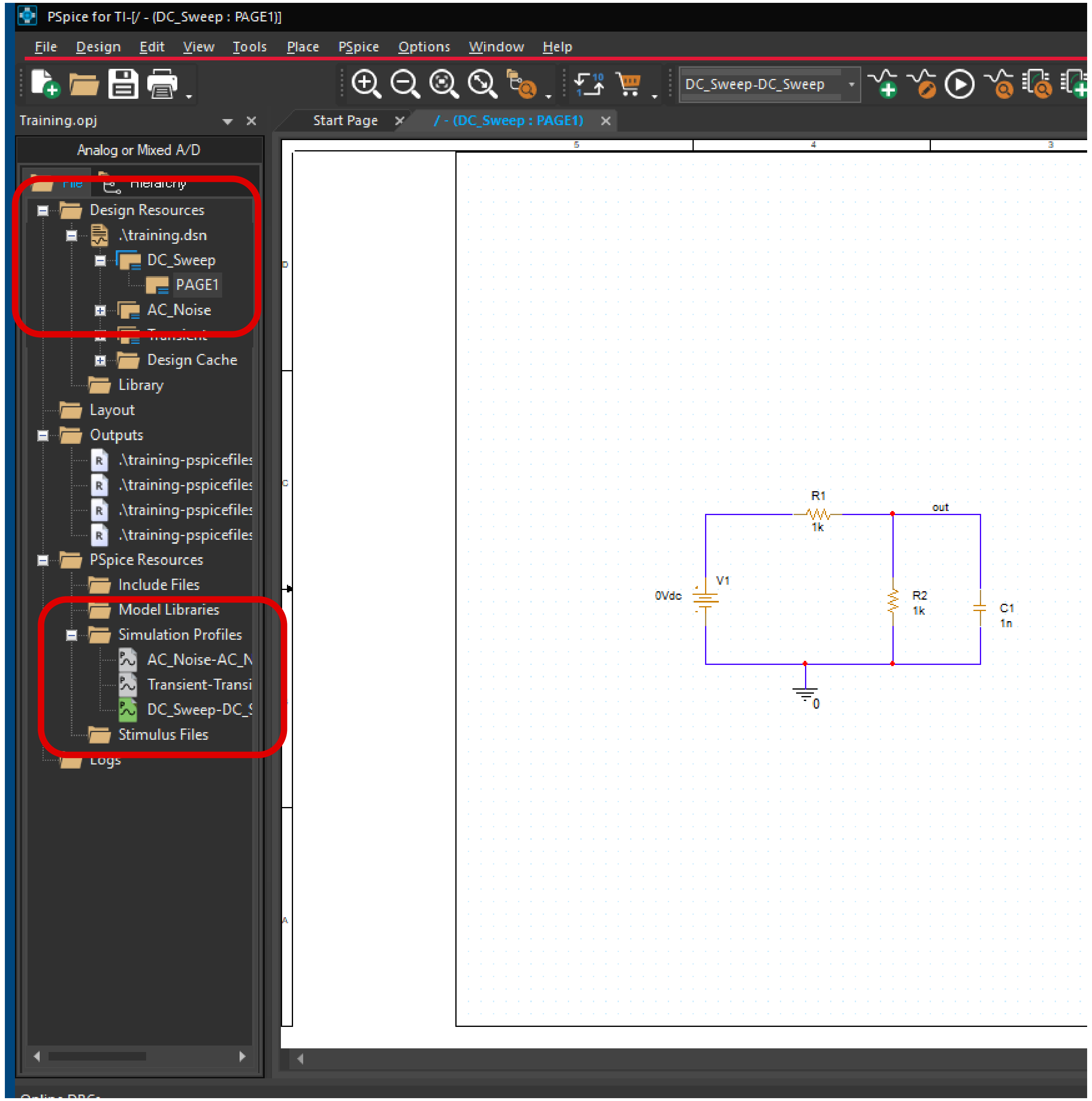






1. Click on the imbedded file above.
2. Open the file.
3. Drag the training folder to the location where you keep you PSPICE projects.
4. Click on “training.opj” to start the project

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

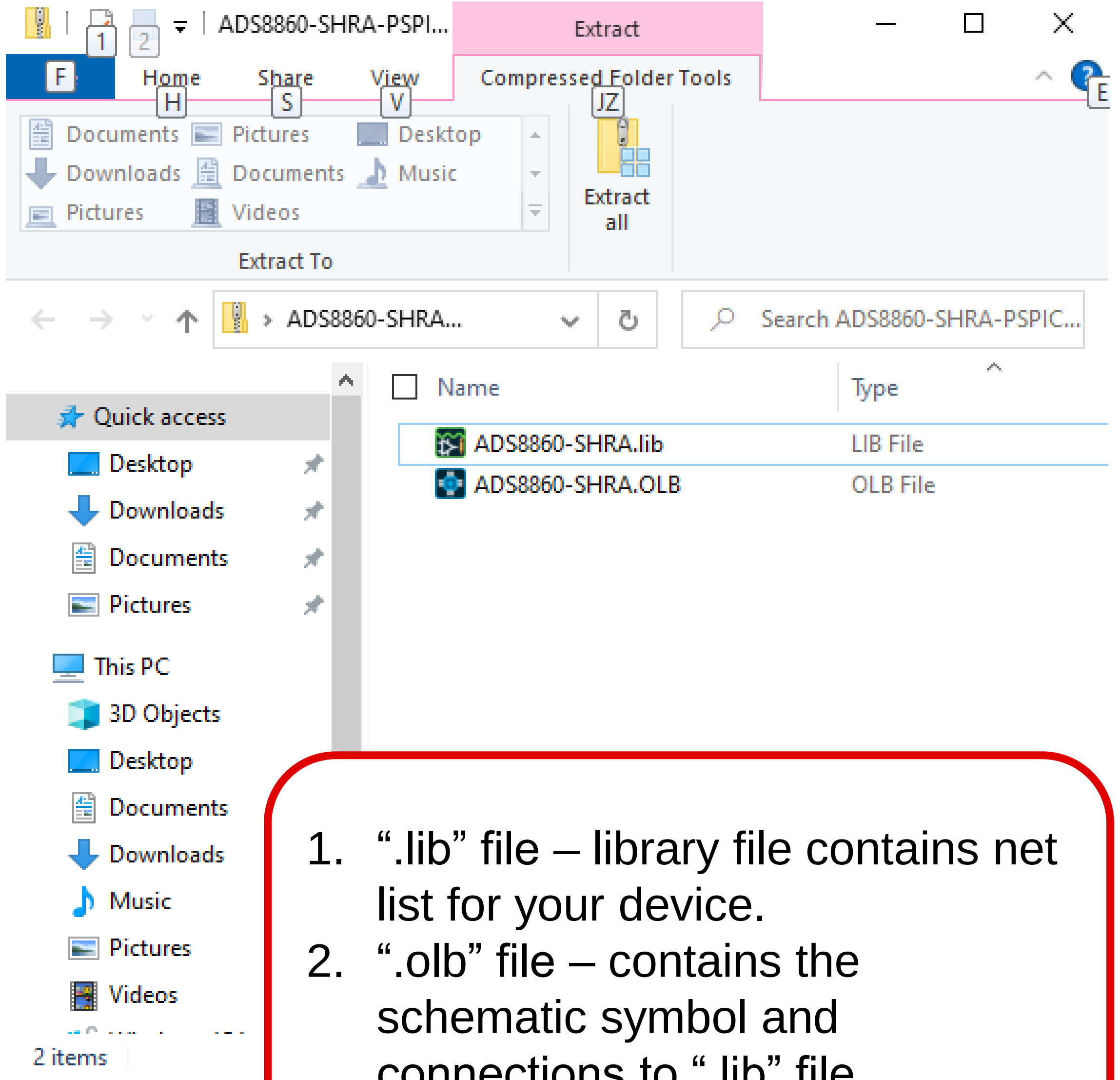
- You need to run Non-TI models. If this is the case, the PSPICE for TI is very limited (three probes)
- If you are developing your own models, or if you are working with a developer (e.g. design soft).
- If you want access to the PCB development tools. Since we use Altium this may not be too useful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

Why you may not want the full version

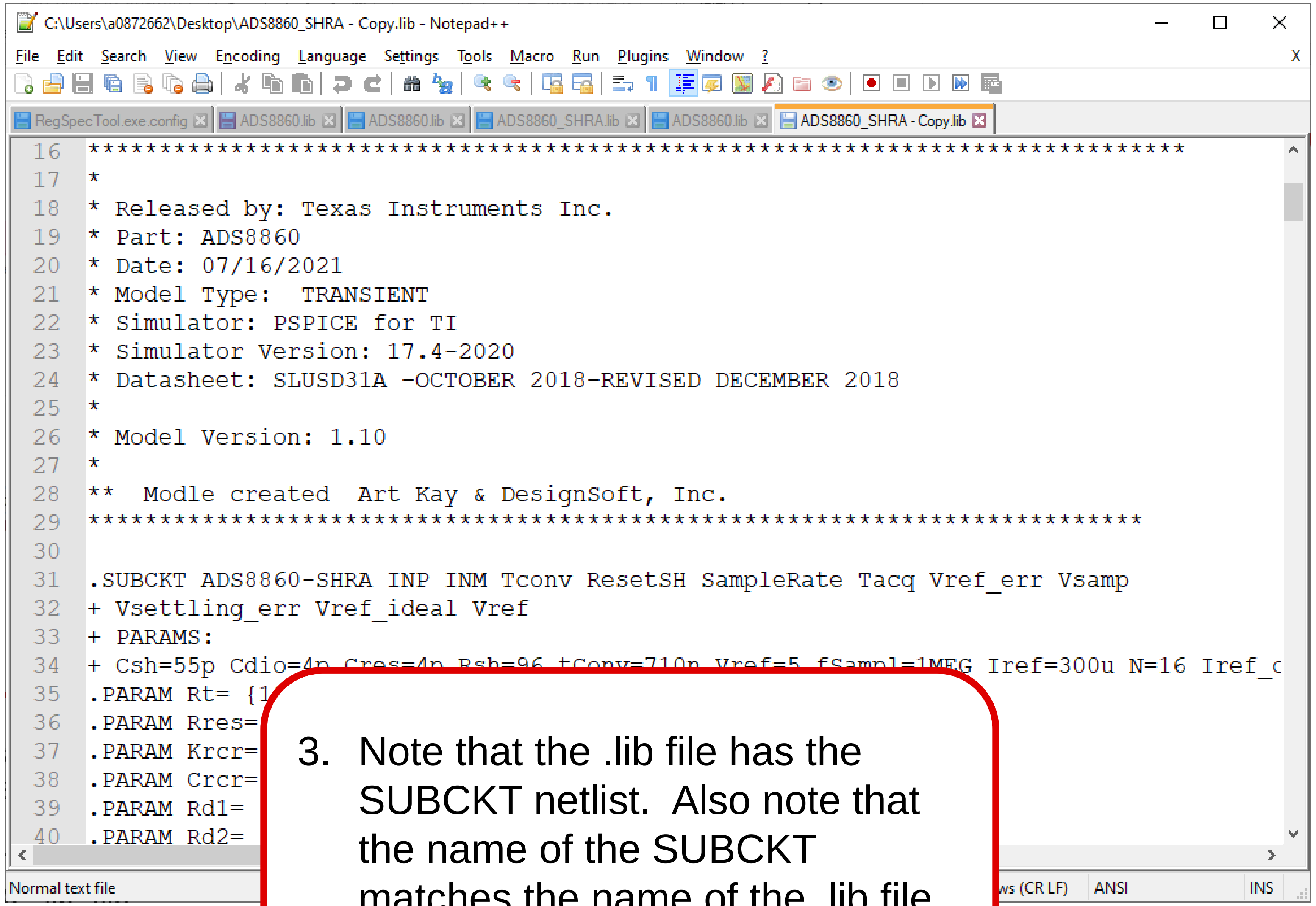
- It is very large and takes a long time to install
- PSPICE for TI works well if you aren't using non-ti models. You can do all analysis types (e.g. montecarlo). The only limit happens when you us non-TI models. In this case you can only monitor three signals.

Back to Table of Contents

Uploading a Model to the web and PSPICE TI library



1. “.lib” file – library file contains net list for your device.
2. “.olb” file – contains the schematic symbol and connections to “.lib” file.

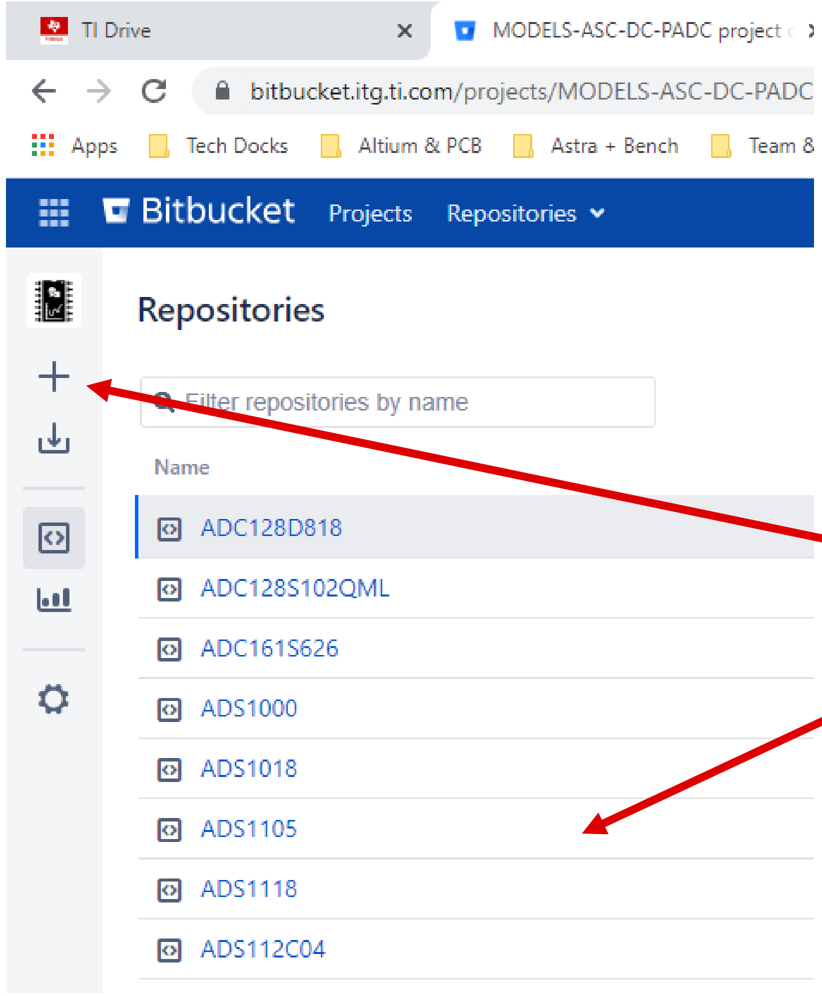


3. Note that the .lib file has the SUBCKT netlist. Also note that the name of the SUBCKT matches the name of the .lib file and the .olb file.

Uploading a Model to the web and PSPICE TI library

[bitbucket repository for ADC model files]

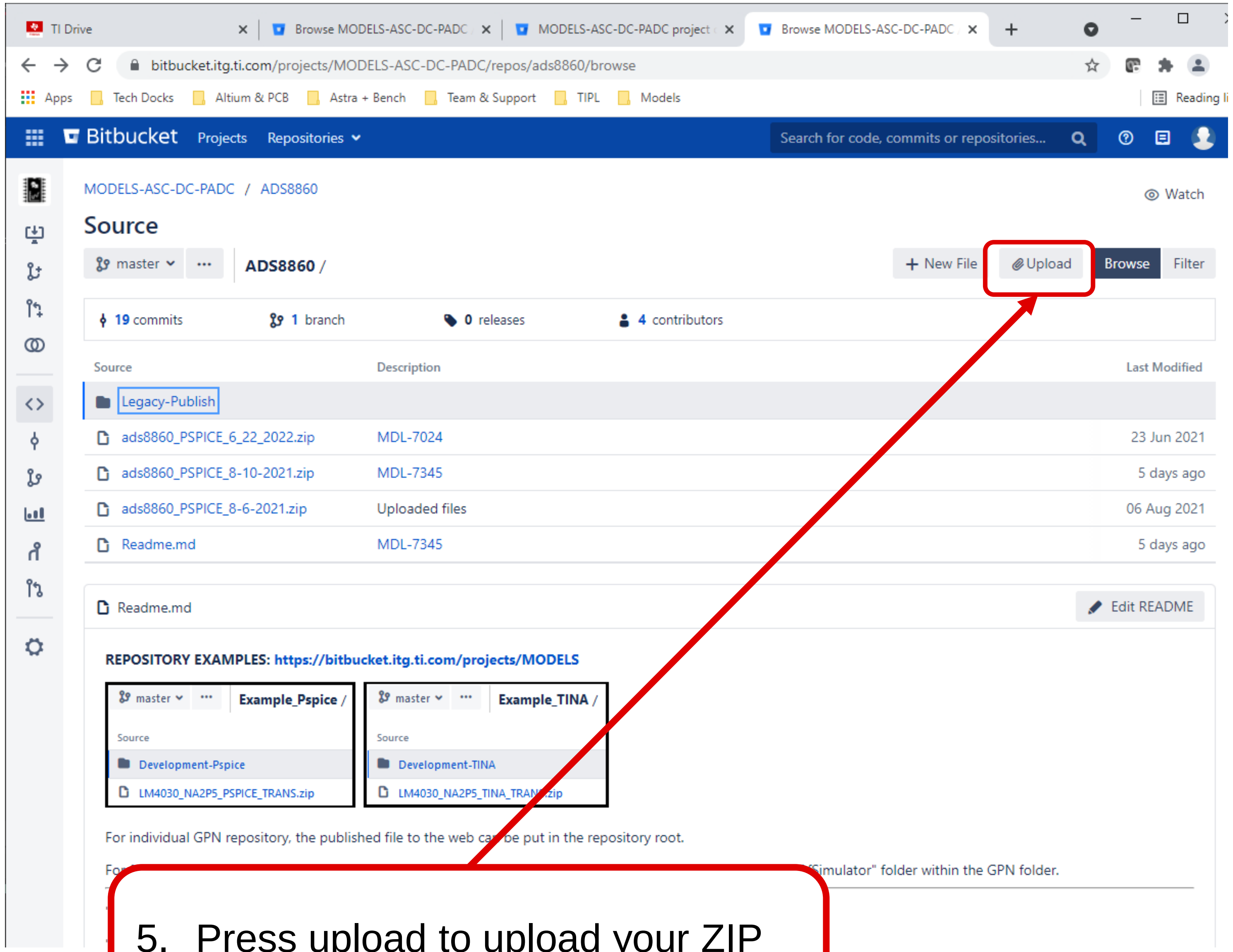
[Confluance Bit Bucket Tutorial]

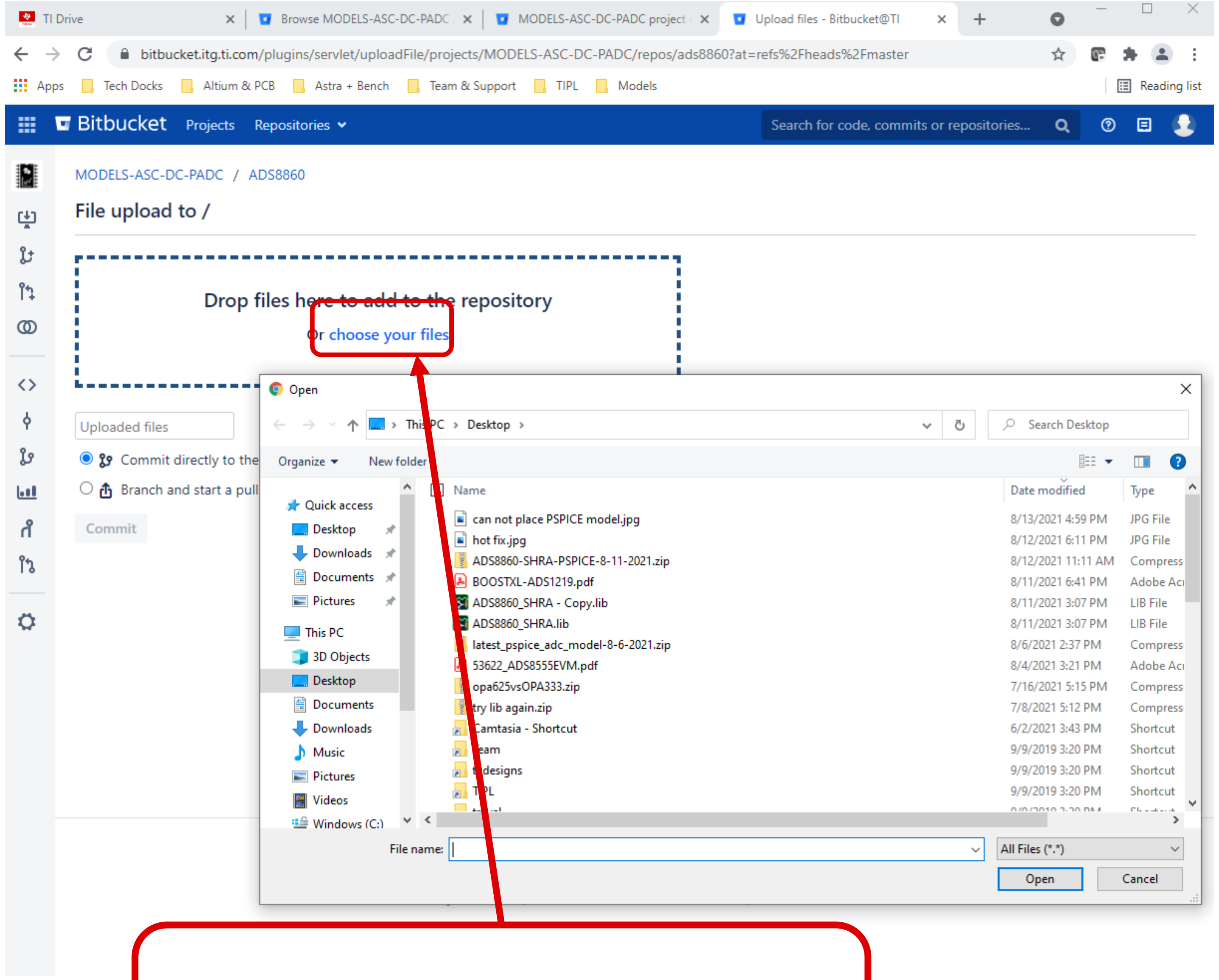


4. Select the appropriate repository. These have been pre-populated for all device names. If you need to make a custom model name press the “+” sign to add a new reparatory.

[Back to Table of Contents]

Uploading a Model to the web and PSPICE TI library



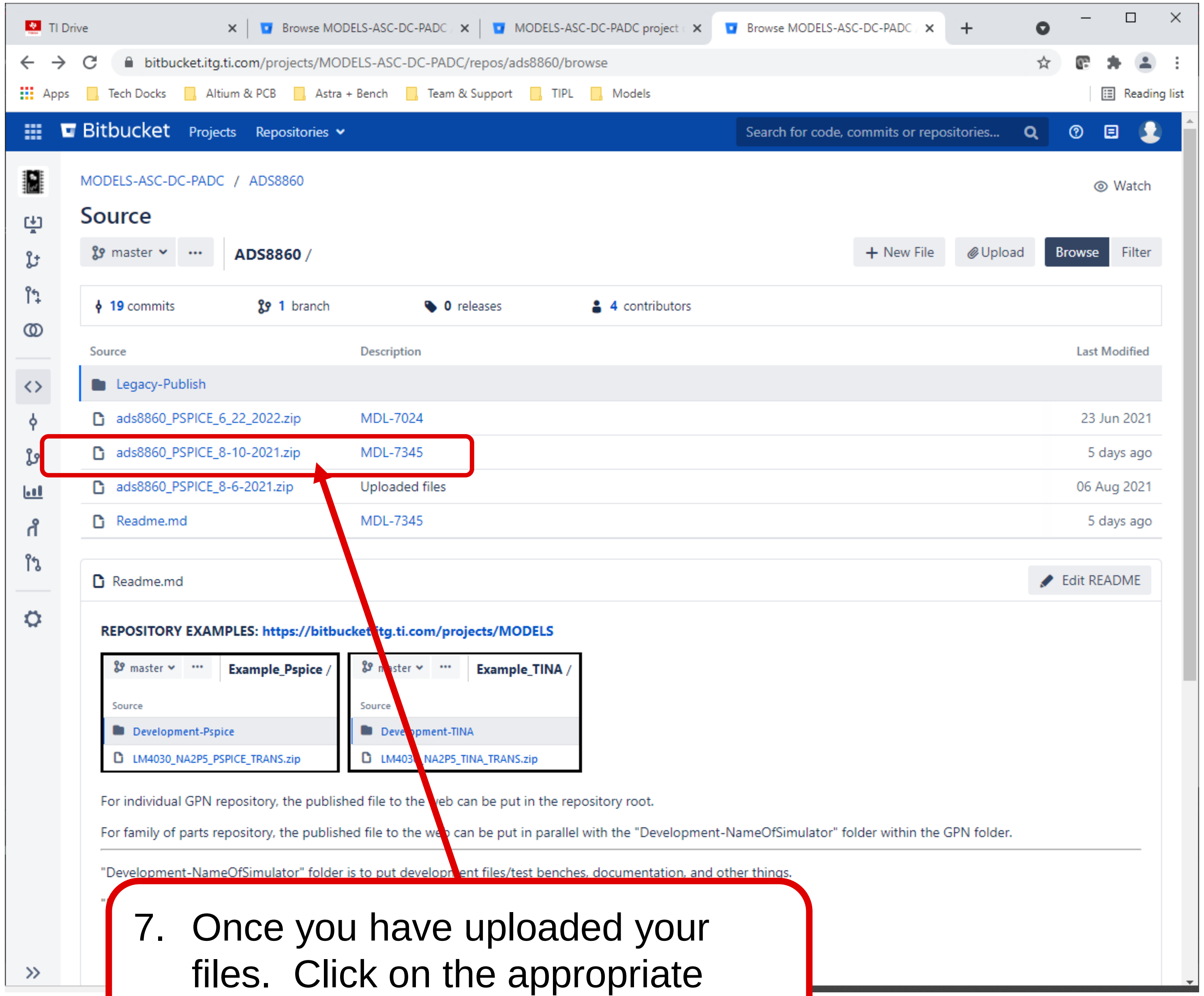


5. Press upload to upload your ZIP file repository.

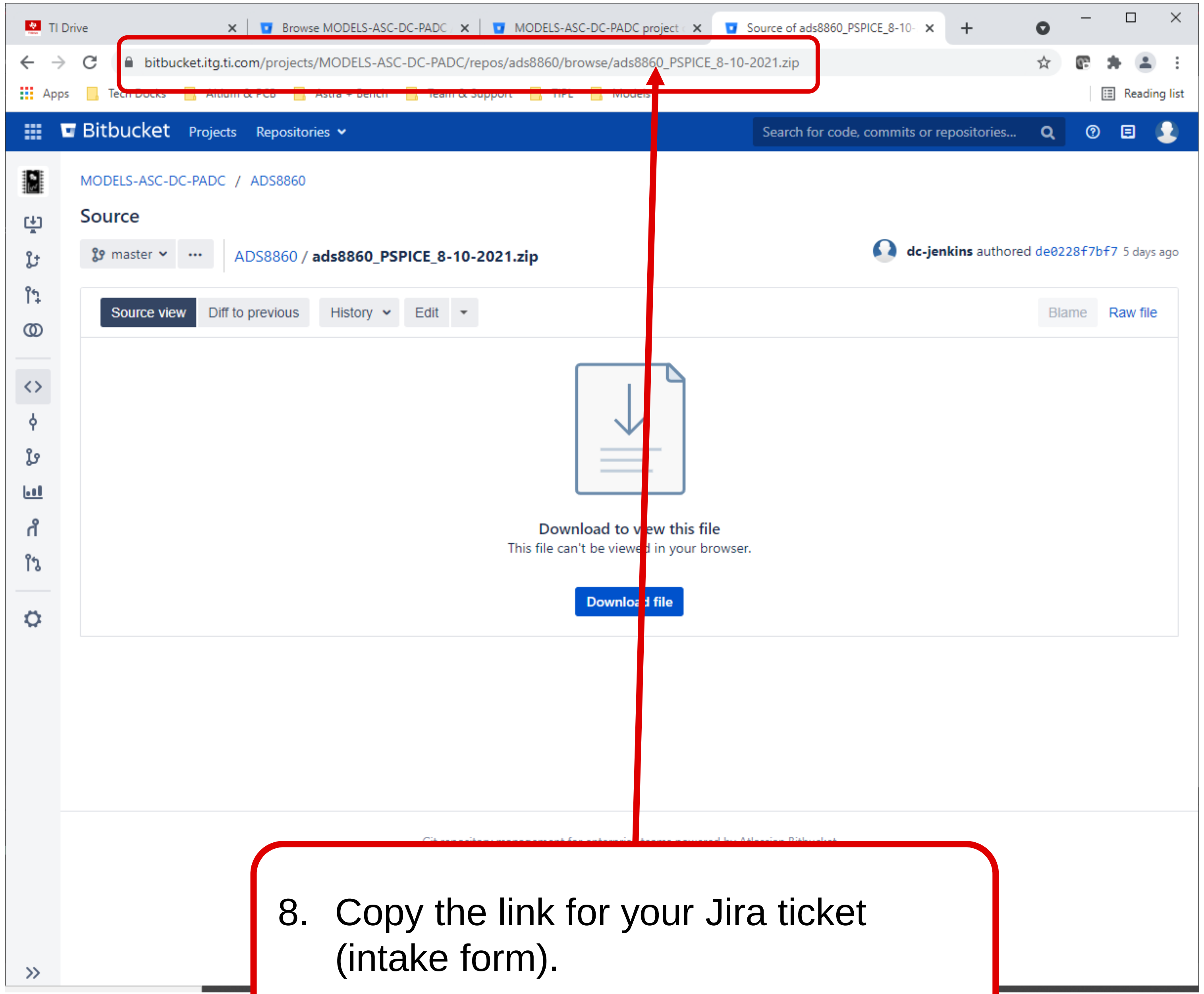
6. Click on “choose your files” and select your ZIP file to upload.

TEXAS INSTRUMENTS

Uploading a Model to the web and PSPICE TI library



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

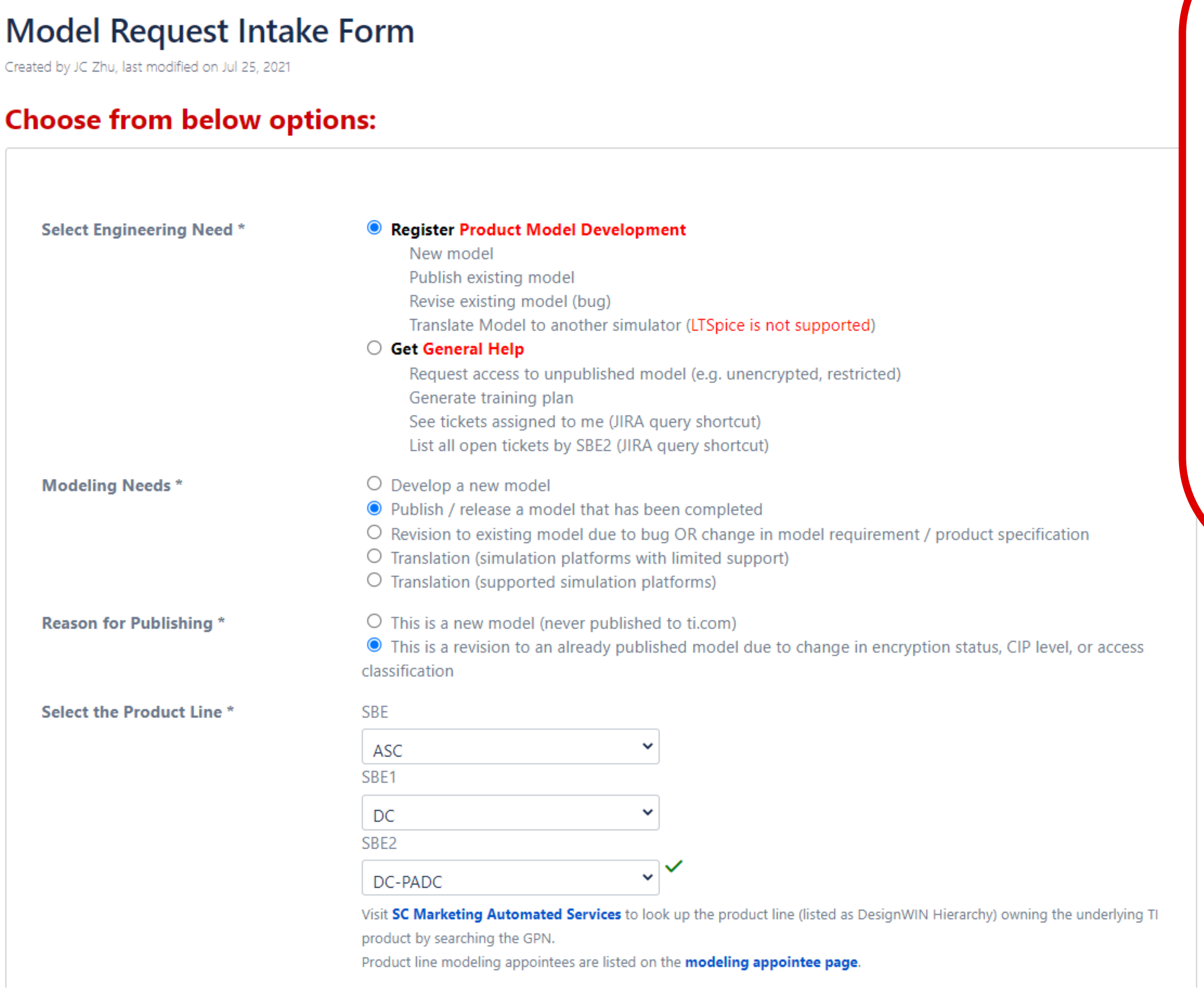


8. Copy the link for your Jira ticket (intake form).

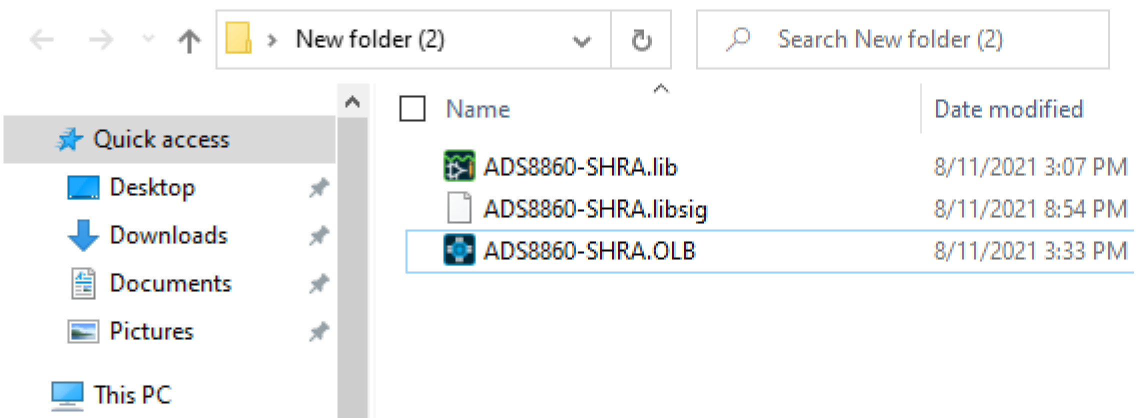
Uploading a Model to the web and PSPICE TI library

Confluence Model Intake Form makes Jira ticket

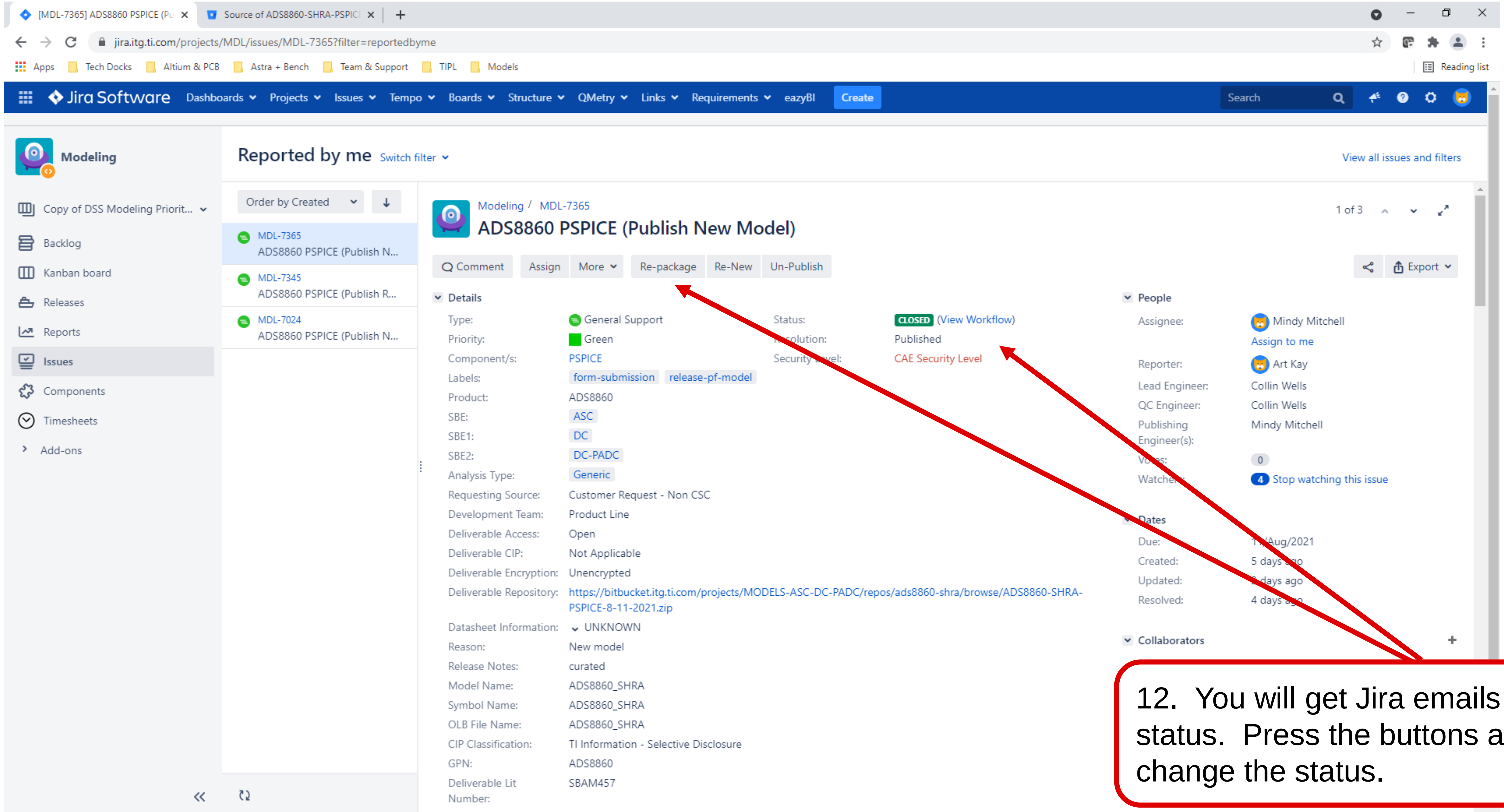
Example of completed Jira Ticket



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:

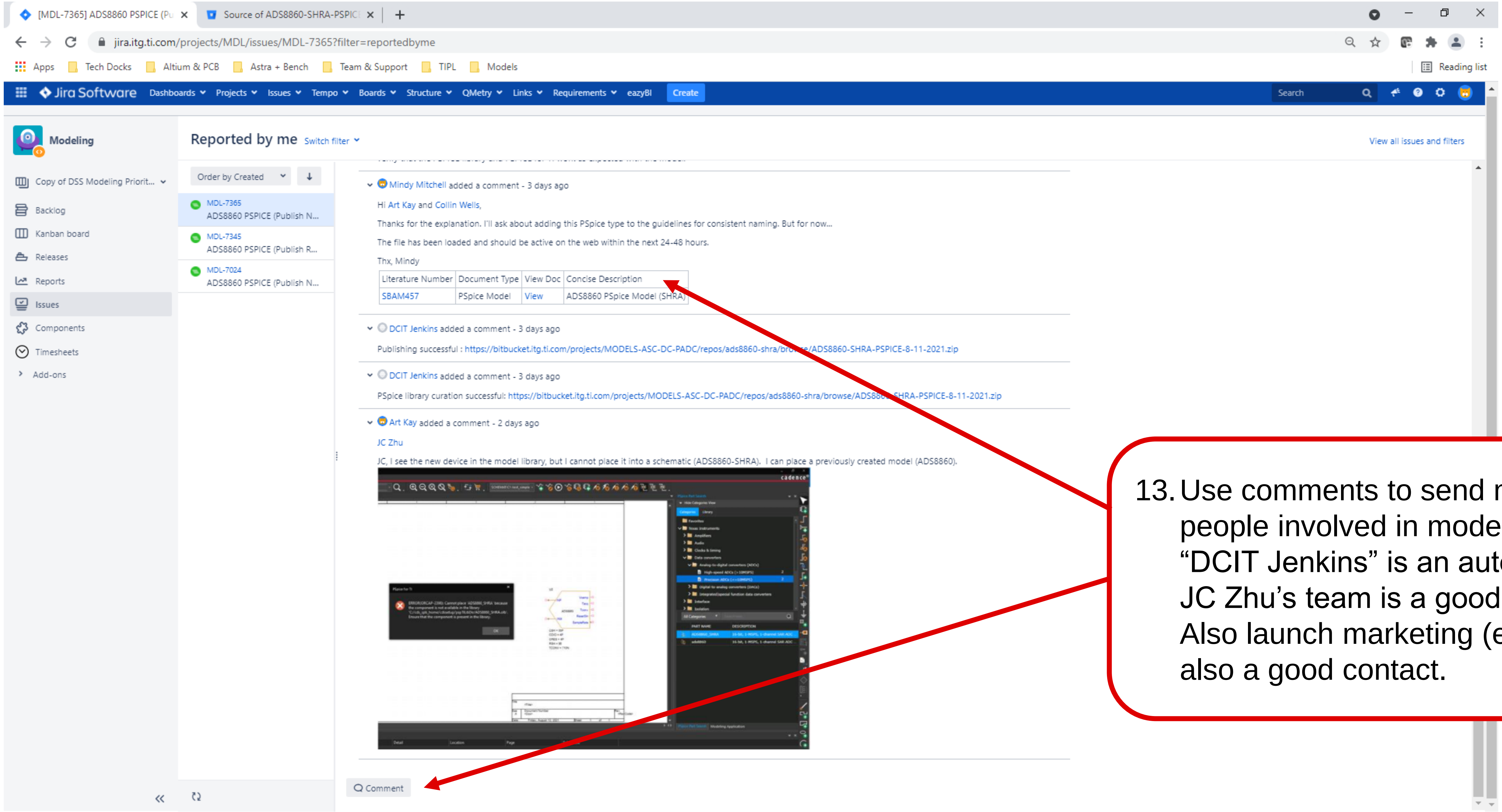


Uploading a Model to the web and PSPICE TI library



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



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.

Back to Table of Contents

Thanks for your time!

Source of ADS9110-PSPICE-10-6 | Infolink | TI Web SSO Login | Model Request Intake Form - Mc | [MDL-7365] ADS8860 PSPICE (Pu

confluence.itg.ti.com/pages/viewpage.action?spaceKey=MODEL&title=Model+Request+Intake+Form

Apps | Tech Docks | Altium & PCB | Astra + Bench | Team & Support | TIPL | Models | Reading list

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) *

ADS9110

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

If you do not have access to Galileo, enter GPN

Literature Concise Description *

ADS9110 PSPICE Model

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

Product Folder(s) *

ADS9110

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 *

https://bitbucket.itg.ti.com/projects/MODELS-ASC-DC-PAD(

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 *

Active on ti.com - FAE, E2E, Legacy

Simulator *

PSPICE: Cadence

OLB File Name *

ADS9110

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 *

ADS9110

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 *

ADS9110

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 *

ADS9110

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 *

ADS9110

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 *

PSPICE Model for ADS9110

Request on Behalf of

Arthur Kay

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 *

Core

Submit