# Simulating with PSpice

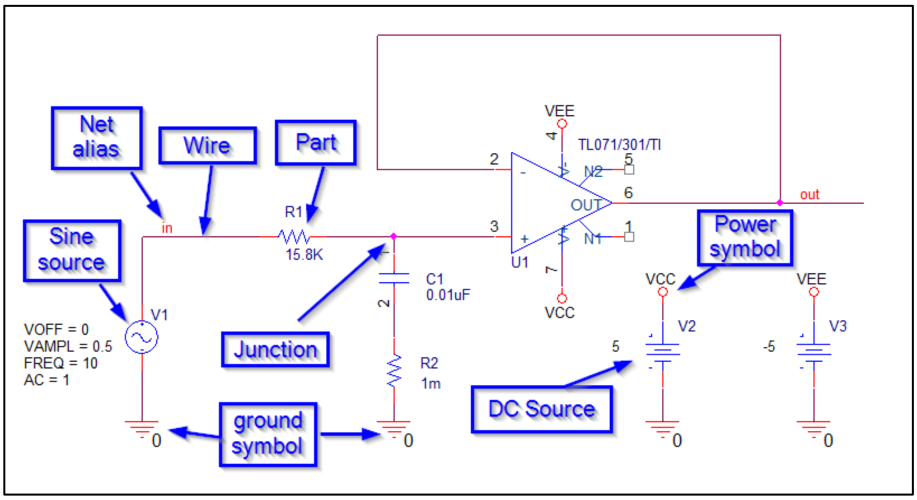
The focus of this guide is to learn to simulate a circuit using PSpice.
The circuit is a simple lowpass first order filter where we will be looking at the cutoff frequency and the gain of the filter.
Finally, at the end of this guide a stress analysis ("Smoke") is run on the circuit.

## Table of Contents

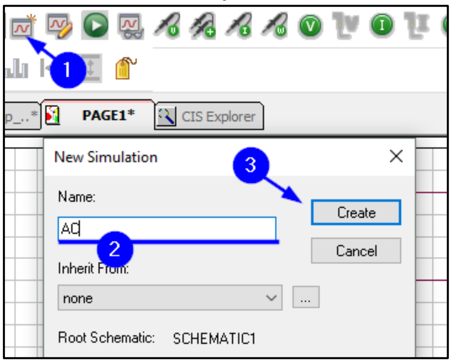


## Creating a simulation profile

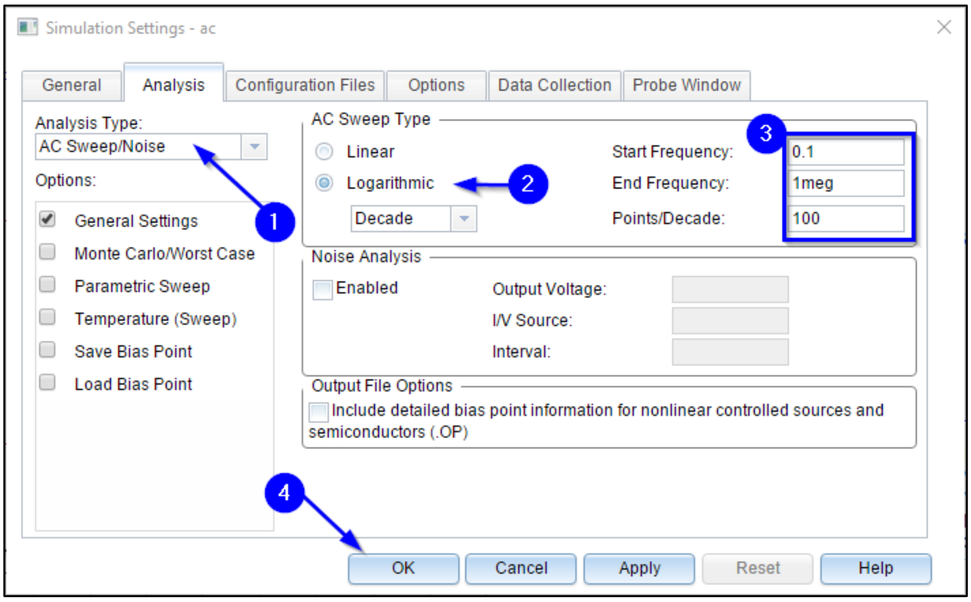
Continue with the circuit previously drawn or alternatively create a new project and draw the schematic below




1. Create a new simulation profile, follow the steps below.

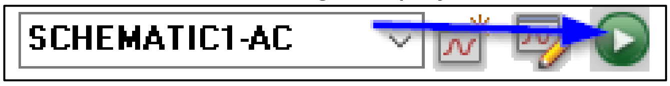


2. Select ***'AC Sweep/Noise'*** in the analysis type drop down and specify settings as shown below. Notice that to specify Mega Hertz in the End Frequency the notification 'Meg' is used.

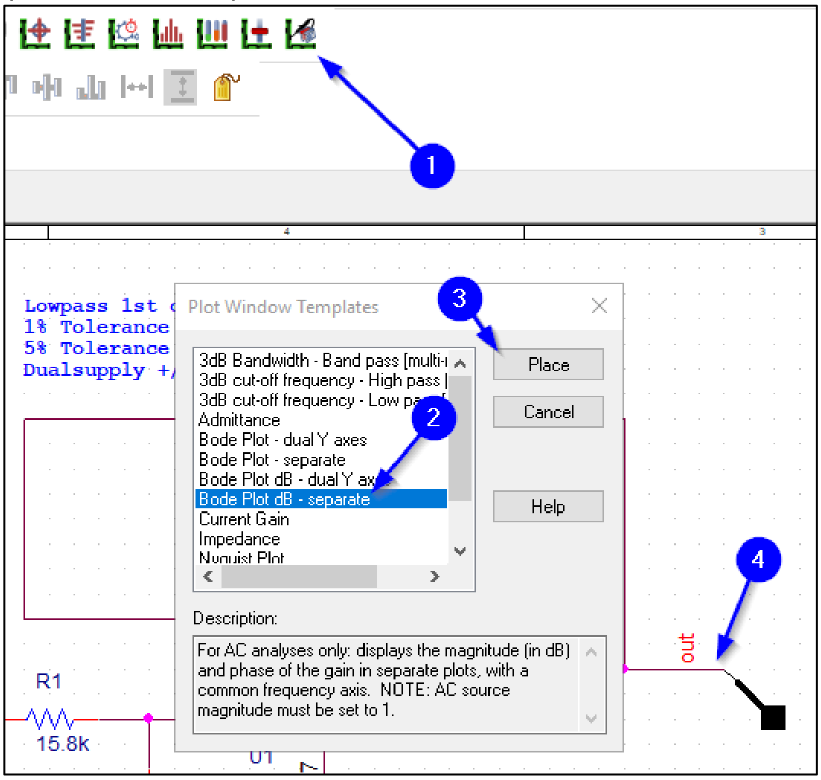



## Simulating the circuit

3. Run the AC simulation with the green play button in the toolbars.

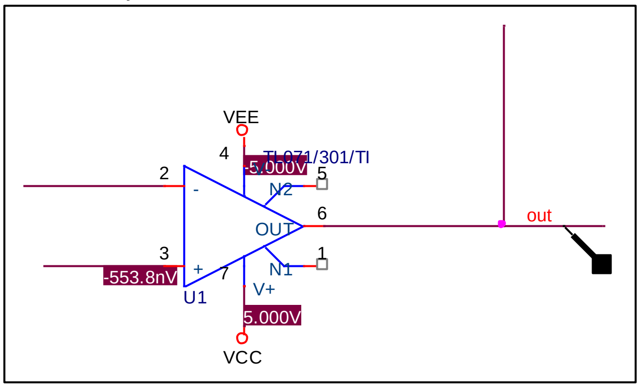



4. Now, to measure performance of the filter we will be looking at a Bode Plot. PSpice has a built-in template to set this up.

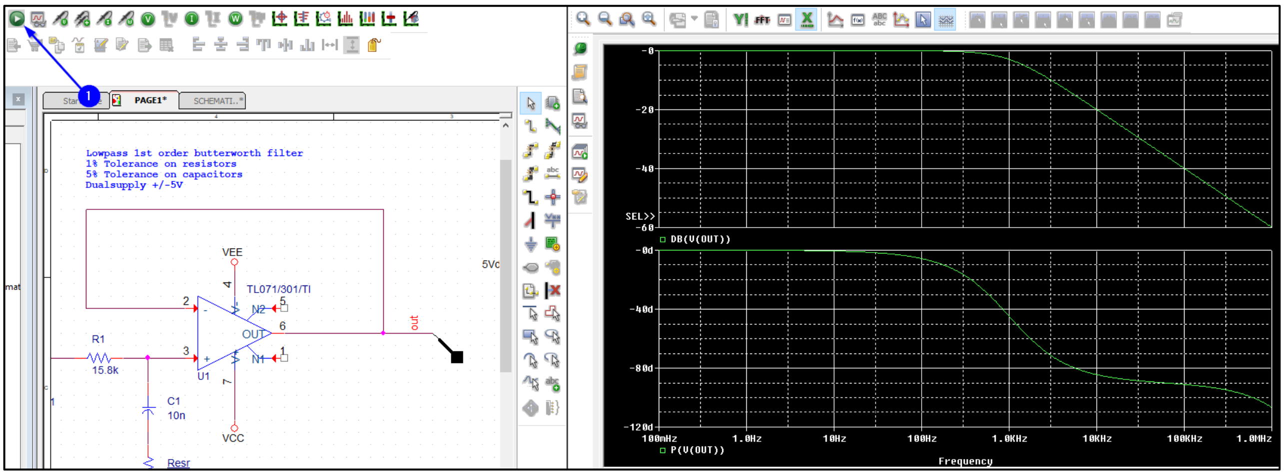



5. Select ***'Bode Plot dB – separate'*** and click on ***Place***

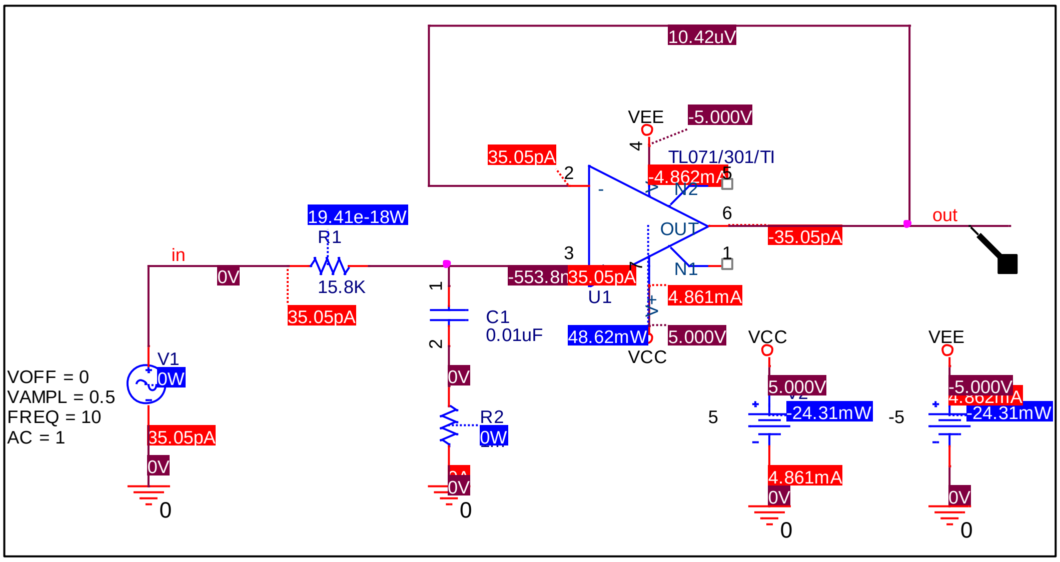
6. Place the marker at the output of the filter:




7. After placing the Bode Plot marker you can open the Probe Window to see the results.



8. It is possible to see the Bias points directly on the circuit. You can add these by clicking the green V (Volt), I (Current) or W (Watts) icons.

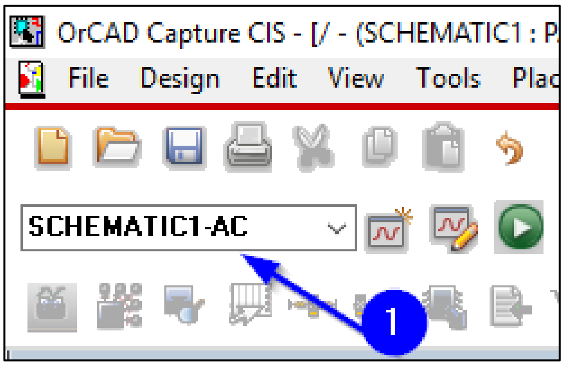


## Measuring performance of the circuit

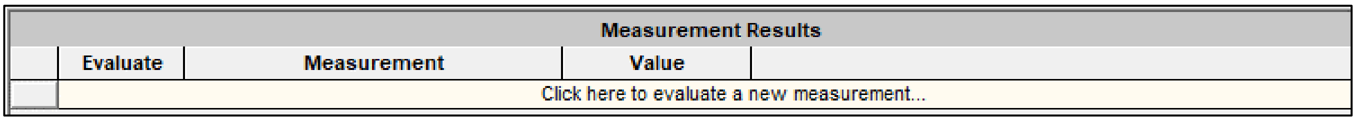
Since we've designed an active lowpass filter, it would be relevant to verify the cutoff frequency and the gain as well.

To do this, we will open the AC analysis simulation results and the PSpice Measurements.

1. Select the AC analysis ***'SCHEMATIC1-AC'*** if it is not already shown




2. Press ***F12*** to open up the simulation result from earlier
3. The Bode plot is shown inside the Probe Window
    a. If not, perform step 4 from "Simulating the circuit" section
4. Select ***View*** → ***Measurement Results***
    a. A yellow line will appear below the Bode Plot




5. Click the line with ***'Click here to evaluate a new measurement'***
6. Now a measurement can be added
    ***a. Select 'Cutoff_Lowpass_3dB(1), then V(out) and finally click 'Ok'***

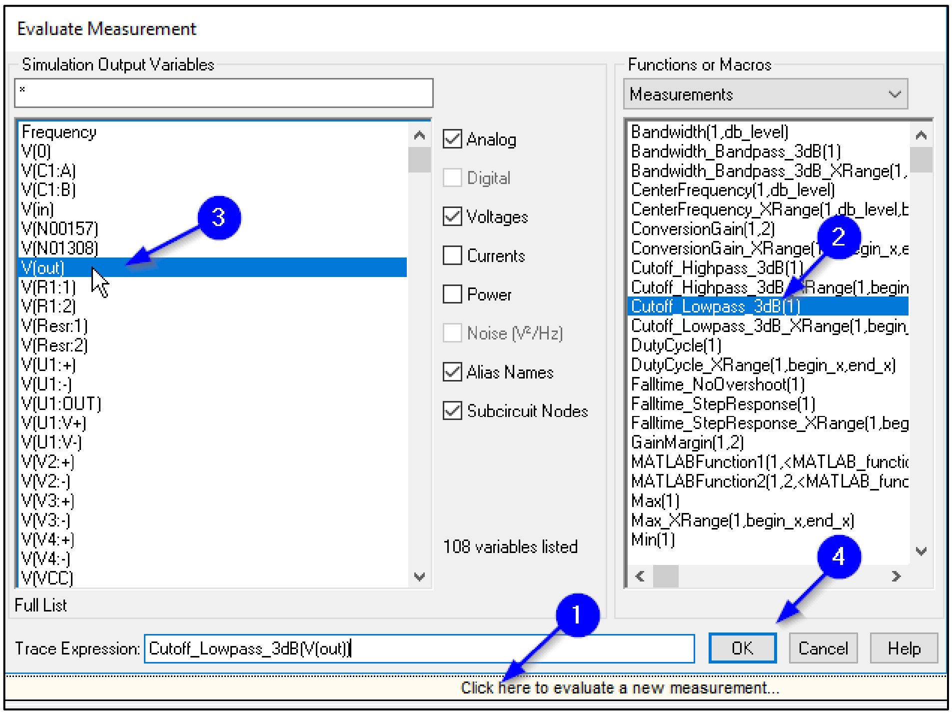
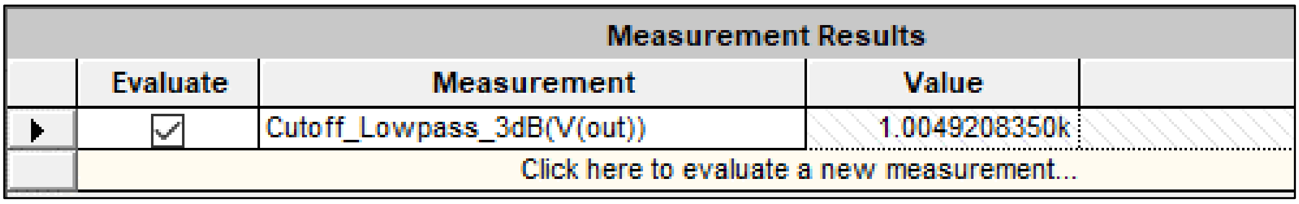
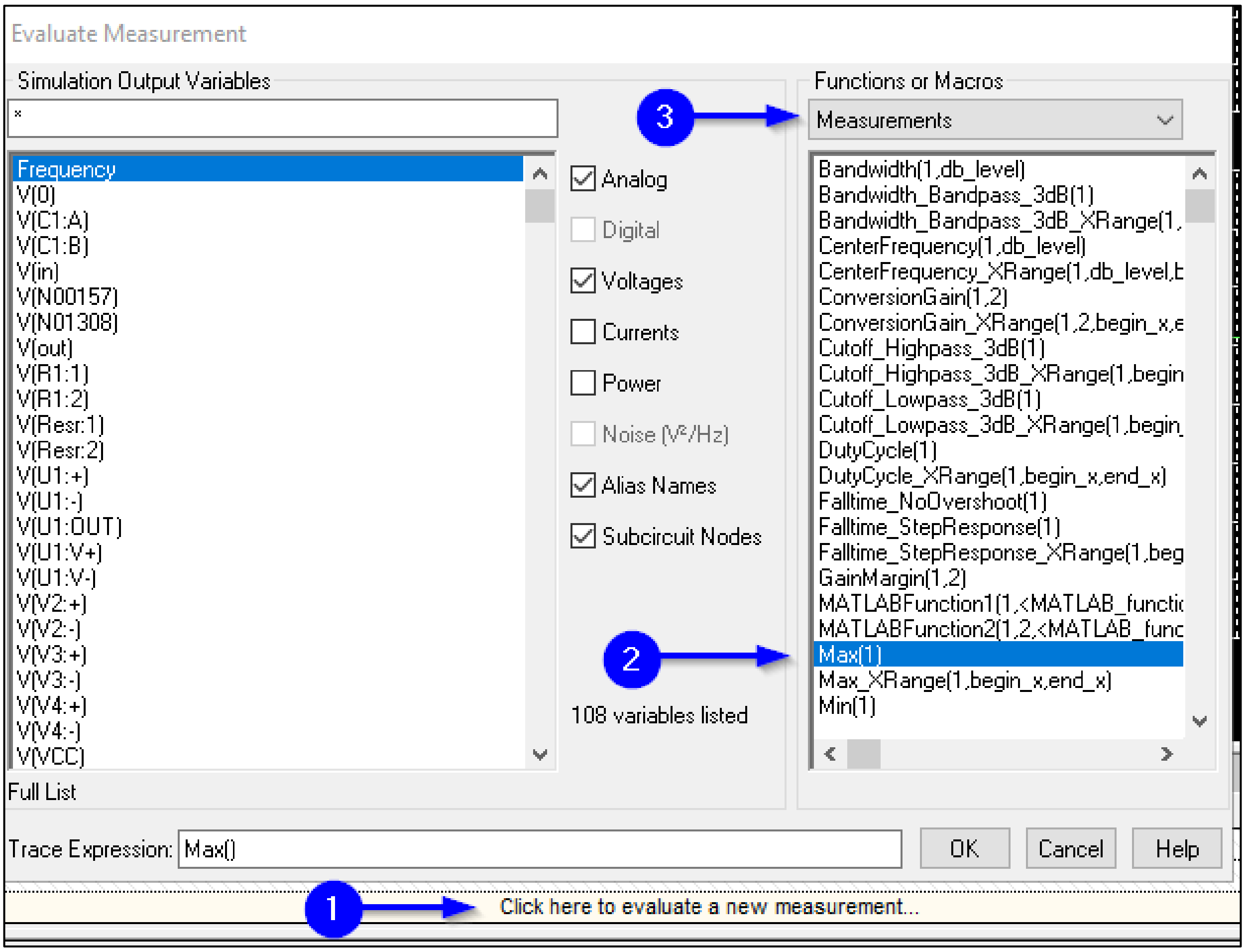



*Figure 1: Righthand side shows available measurement templates. Lefthand side shows available sections to measure.*

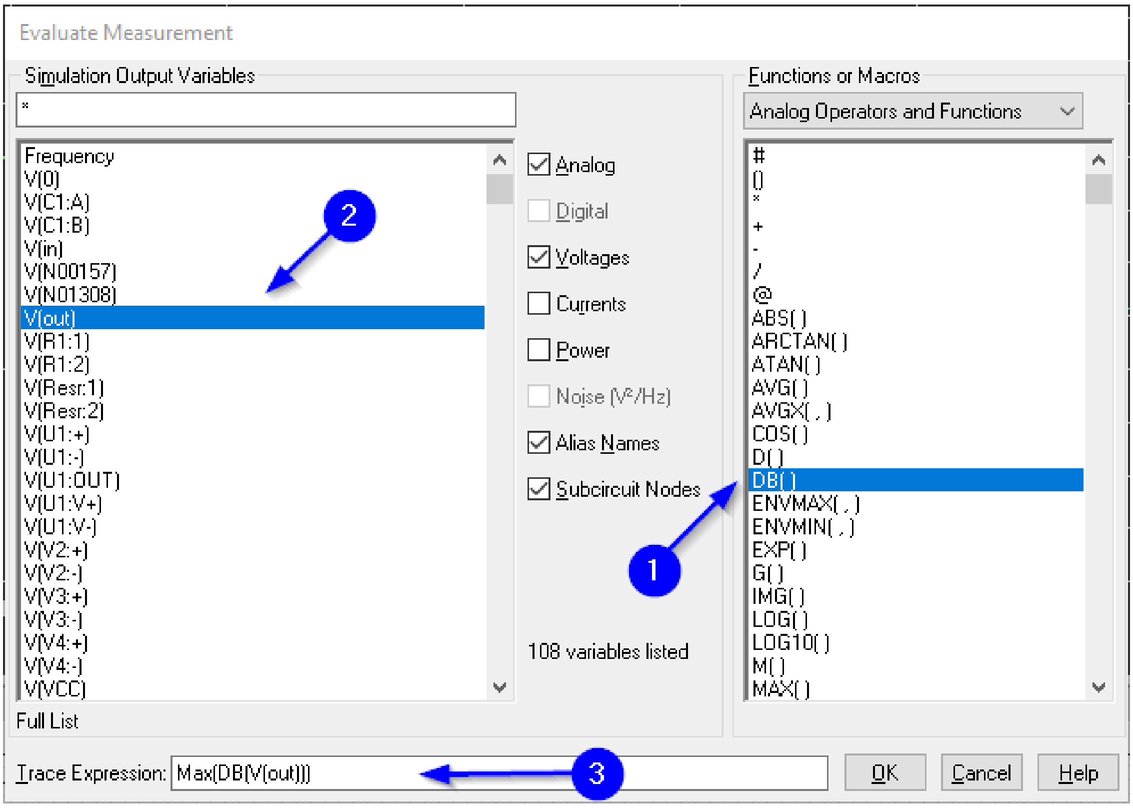
7. Note that a line has been added in the bottom of the Probe Window with the result:




8. To add the maximum dB at V(out) click on ***'Click here to evaluate a new measurement'*** again:




9. ***Select 'Analog Operators and Functions'*** in the dropdown shown in step 3 above
   a. Now ***Select 'DB()' and then V(out) and click 'OK'***




10. Verify that Max(DB(V(out))) is shown in the bottom of the window
    a. *Tip: This can be written by hand in "Trace Expression" instead of clicking on the operators in this window*

| | Evaluate | Measurement | Value |
|---|---|---|---|
| | ☑ | Max(DB(V(out))) | -295.76169u |
| ▶ | ☑ | Cutoff_Lowpass_3dB(V(out)) | 1.00492k |

11. It is very relevant to compare these results with the calculations that are typically done during design creation. In this example

$$fc = \frac{1}{2 * \pi * 10nF * 15.8k\Omega} = 1007Hz$$

This concludes the first part of the simulation exercise. We have created an AC simulation profile and set PSpice up to show the frequency response of this filter. Measurements in Probe Window are a powerful tool to calculate eg. the cutoff frequency and verify it against your own calculations. Remember that when using a real operational amplifier like in this guide, measurements will not match up exactly since PSpice takes real parameters into account.

# Advanced Analysis – Smoke

Smoke analysis is a tool used to do stress analysis on a circuit. Smoke predicts the stress applied to the components, such as voltage, current, wattage and temperature. It is possible to enter a derating which is useful in predicting how much component derating will influence a circuit's performance over time.

Smoke analysis will require the following:

- A circuit that is simulated in the time domain
- Components with Smoke information
- Optionally user deratings
    - This will give the user SOL (Safe Operating Limit) information.

First thing we'll do is to add a couple of parameters to globally define values for the Smoke analysis.

1. Open ***Place*** → ***PSpice Component*** → ***Search*** and search for ***'Variables'***
2. Place it next to the lowpass filter. The variables specify default parameters for tolerances, ratings etc. for the analysis.

**Advanced Analysis Properties**

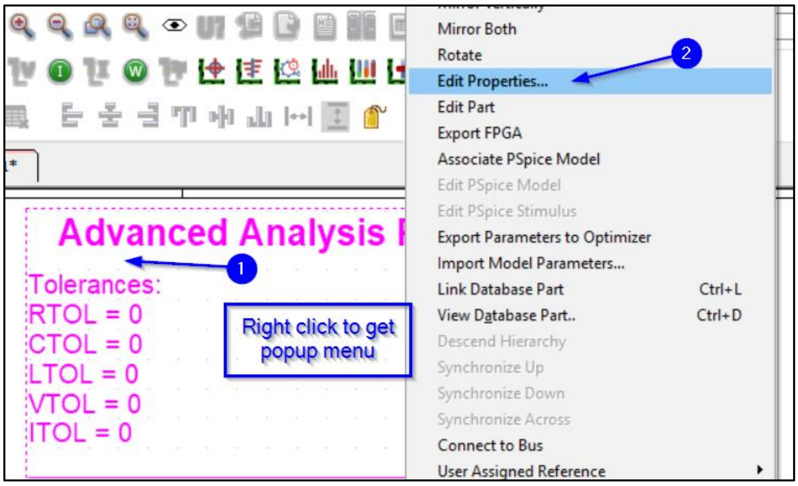
Tolerances:
RTOL = 0
CTOL = 0
LTOL = 0
VTOL = 0
ITOL = 0

Smoke Limits:

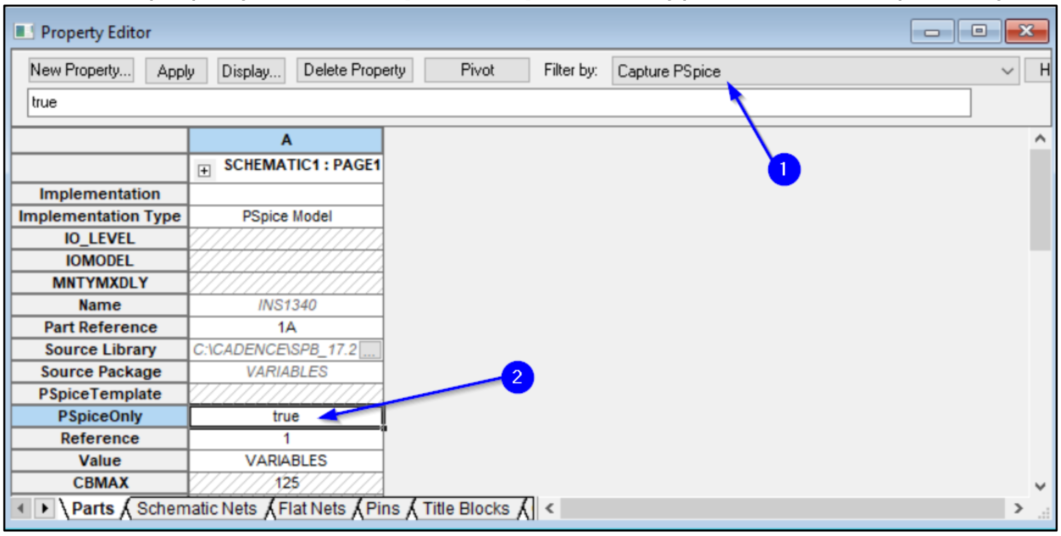
| | |
|---|---|
| RMAX = 0.25 | ESR = 0.001 |
| RSMAX = 0.0125 | CPMAX = 0.1 |
| RTMAX = 200 | CVN = 10 |
| RVMAX = 100 | LPMAX = 0.25 |
| CMAX = 50 | DC = 0.1 |
| CBMAX = 125 | RTH = 1 |
| CSMAX = 0.005 | |
| CTMAX = 125 | |
| CIMAX = 1 | |
| LMAX = 5 | |
| DSMAX = 300 | |
| IMAX = 1 | |
| VMAX = 12 | |

User Variables:

3. When the ‘Variables’ block is placed then select it using left click and do a ***Right click*** → ***Edit Properties***



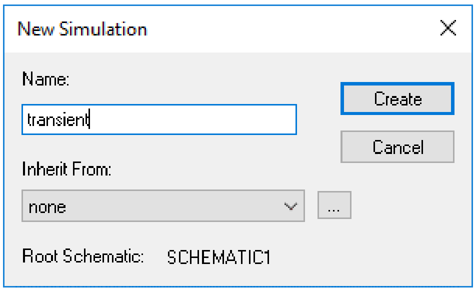

4. Now set the property filter to ***‘Capture PSpice’*** and type ***‘true’*** into PSpiceOnly.

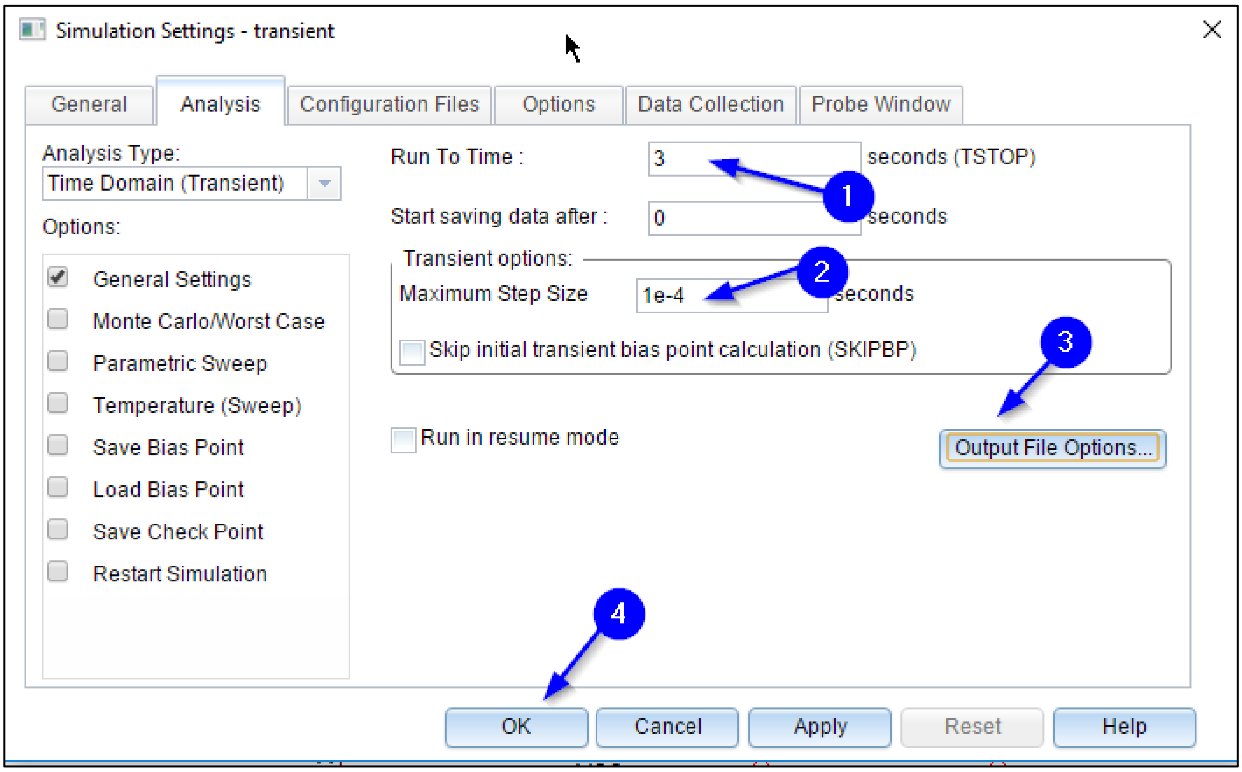



The result of setting PSpiceOnly=True is that this “part” won’t be transferred into PCB Design and won’t be part of Bill of Materials management.

5. Close the property editor using ***Ctrl+F4***
6. Create a new simulation profile using ***PSpice*** → ***New Simulation Profile***
   a. Or use the PSpice toolbar icon: 

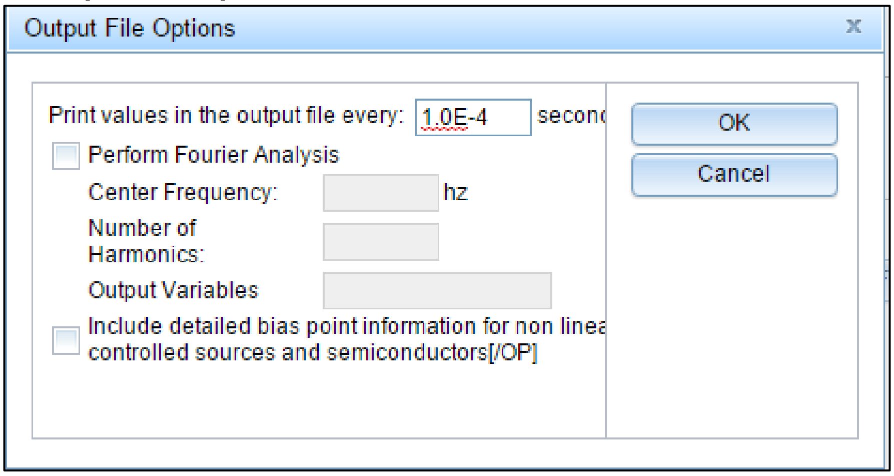
7. Specify the simulation profile name as ***‘Transient’*** – click ***Ok***



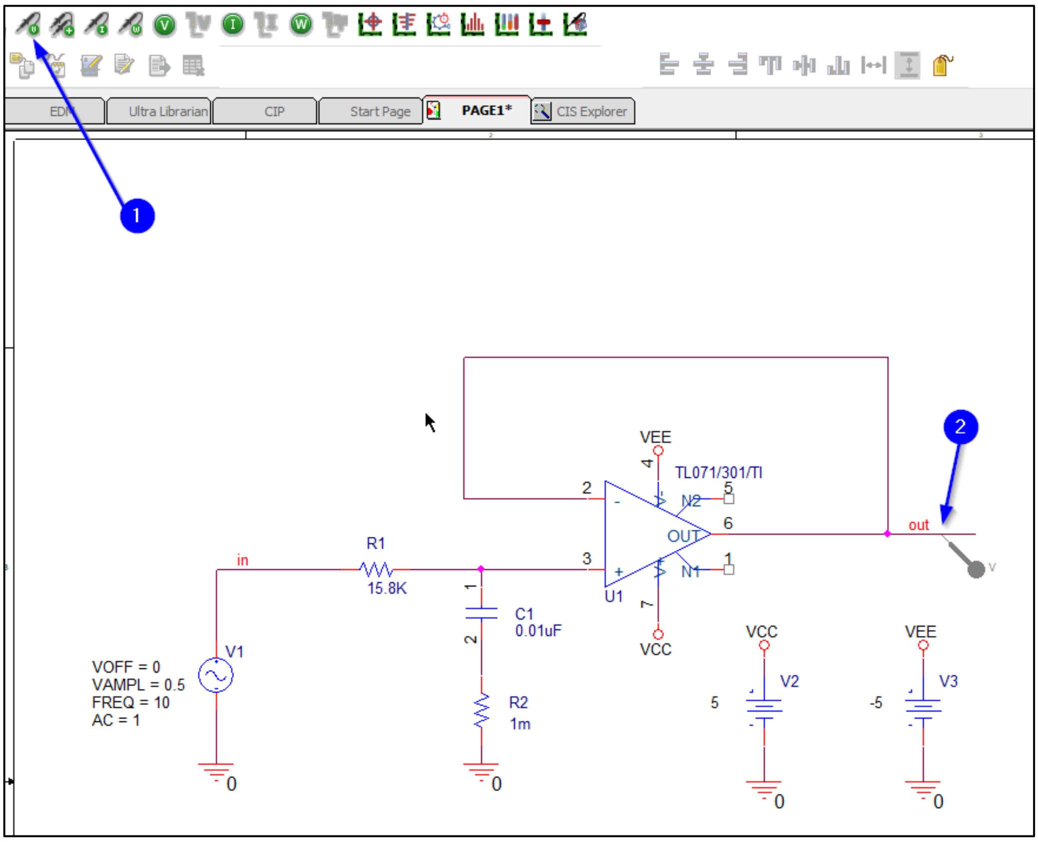



a. Run to Time: 3.0
b. Maximum Step Size: 1.0E-4

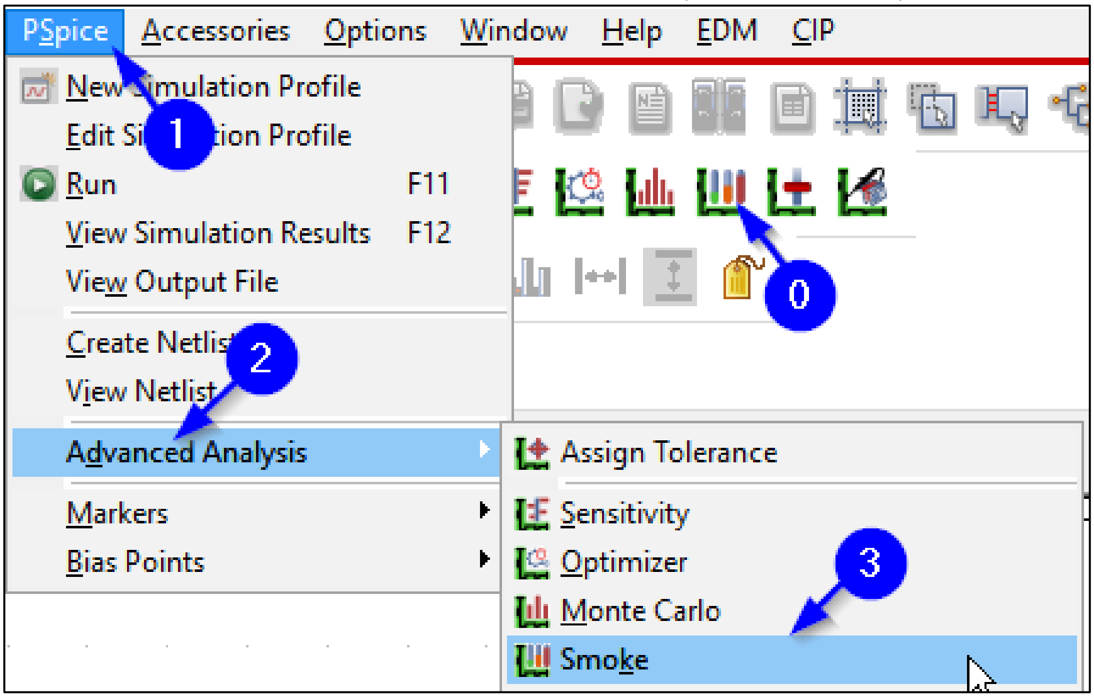
8. ***Click on 'Output File Options'*** and enter 1.0E-4 seconds:



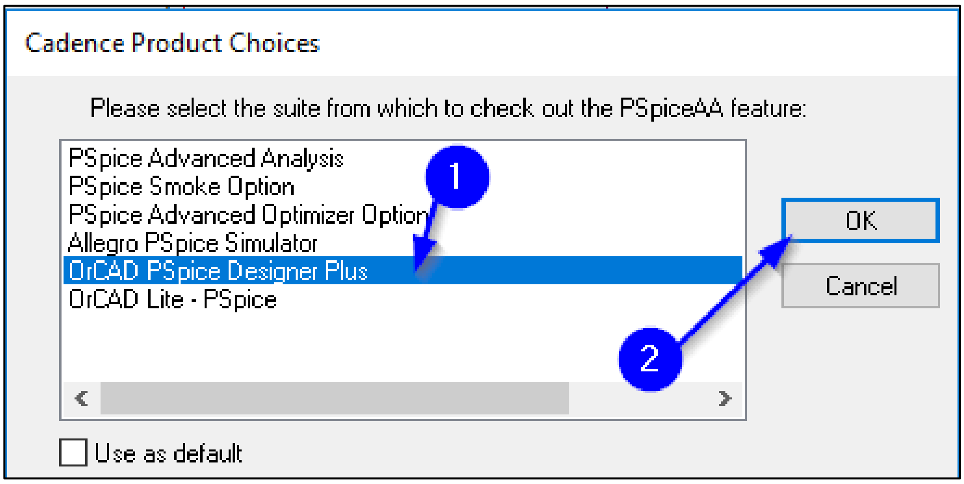

9. Click ***OK*** until the simulation settings dialog is closed.
10. Place a Voltage Probe on the output of the circuit:



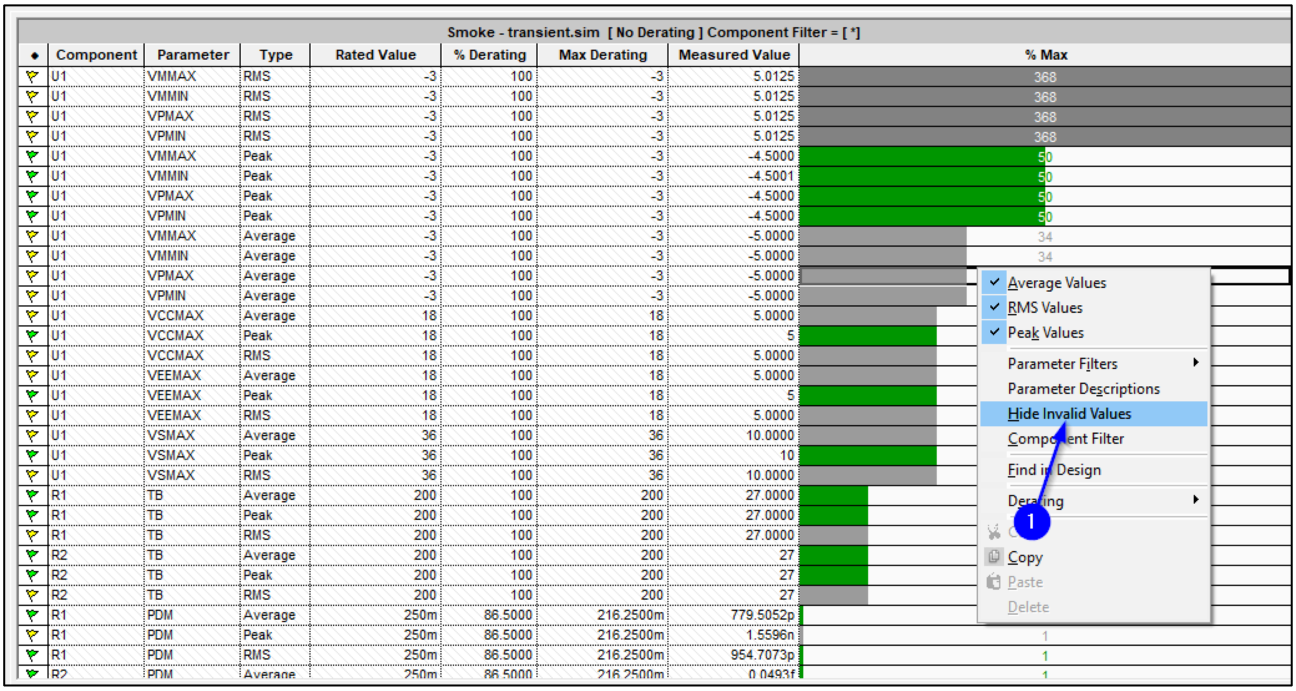

11. Run the simulation ***PSpice* → *Run*** or use the shortcut ***F11***
12. The reason for setting up and running the transient simulation is that the more Advanced Analysis types like Smoke/stress requires a transient solution
13. Next, start Smoke analysis using ***PSpice* → *Advanced Analysis* → *Smoke***.
    a. Step '0' below is the toolbar button access directly to Smoke Analysis




Depending on your previous choice of license, you might be asked what license you would like to use. Select OrCAD PSpice Designer Plus




14. With Advanced Analysis open you’ll be shown a row for each component parameter. The results show if any component parameters are exceeding their recommended values. The data also shows if the component is being stressed in case the values are close to the maximum.
15. Right click in the parameter list and select Hide Invalid Values


Smoke - transient.sim [ No Derating ] Component Filter = [ * ]

| Component | Parameter | Type | Rated Value | % Derating | Max Derating | Measured Value | % Max |
|---|---|---|---|---|---|---|---|
| U1 | VMMAX | RMS | -3 | 100 | -3 | 5.0125 | 368 |
| U1 | VMMIN | RMS | -3 | 100 | -3 | 5.0125 | 368 |
| U1 | VPMAX | RMS | -3 | 100 | -3 | 5.0125 | 368 |
| U1 | VPMIN | RMS | -3 | 100 | -3 | 5.0125 | 368 |
| U1 | VMMAX | Peak | -3 | 100 | -3 | -4.5000 | 50 |
| U1 | VMMIN | Peak | -3 | 100 | -3 | -4.5001 | 50 |
| U1 | VPMAX | Peak | -3 | 100 | -3 | -4.5000 | 50 |
| U1 | VPMIN | Peak | -3 | 100 | -3 | -4.5000 | 50 |
| U1 | VMMAX | Average | -3 | 100 | -3 | -5.0000 | 34 |
| U1 | VMMIN | Average | -3 | 100 | -3 | -5.0000 | 34 |
| U1 | VPMAX | Average | -3 | 100 | -3 | -5.0000 | |
| U1 | VPMIN | Average | -3 | 100 | -3 | -5.0000 | |
| U1 | VCCMAX | Average | 18 | 100 | 18 | 5.0000 | |
| U1 | VCCMAX | Peak | 18 | 100 | 18 | 5 | |
| U1 | VCCMAX | RMS | 18 | 100 | 18 | 5.0000 | |
| U1 | VEEMAX | Average | 18 | 100 | 18 | 5.0000 | |
| U1 | VEEMAX | Peak | 18 | 100 | 18 | 5 | |
| U1 | VEEMAX | RMS | 18 | 100 | 18 | 5.0000 | |
| U1 | VSMAX | Average | 36 | 100 | 36 | 10.0000 | |
| U1 | VSMAX | Peak | 36 | 100 | 36 | 10 | |
| U1 | VSMAX | RMS | 36 | 100 | 36 | 10.0000 | |
| R1 | TB | Average | 200 | 100 | 200 | 27.0000 | |
| R1 | TB | Peak | 200 | 100 | 200 | 27.0000 | |
| R1 | TB | RMS | 200 | 100 | 200 | 27.0000 | |
| R2 | TB | Average | 200 | 100 | 200 | 27 | |
| R2 | TB | Peak | 200 | 100 | 200 | 27 | |
| R2 | TB | RMS | 200 | 100 | 200 | 27 | |
| R1 | PDM | Average | 250m | 86.5000 | 216.2500m | 779.5052p | |
| R1 | PDM | Peak | 250m | 86.5000 | 216.2500m | 1.5596n | 1 |
| R1 | PDM | RMS | 250m | 86.5000 | 216.2500m | 954.7073p | 1 |
| R2 | PDM | Average | 250m | 86.5000 | 216.2500m | 0.0493f | 1 |



16. Next, look at the right side of the window and verify that all values are green

**How to interpret the colors in PSpice Smoke analysis:**

- RED: SOL[1] exceeded (a lot of smoke)
- YELLOW: between 90% and 100% of SOL
- GRØN: under 90% of SOL

[1] SOL = Safe Operating Limits

- GRÅ: No data available, or “typically not used”

| % Max |
|---|
| 257 |
| 148 |
| 96 |
| 42 |
| 41 |
| 39 |

*Figure 2: Example of how colors are used in PSpice AA Smoke analysis*

It is possible to setup derating tables in order to implement how the different component parameters can be stressed. This is very important since component parameters that are stressed to their maximum can result in lower circuit lifetime as a result of component lifetime.

## What did you learn?

- ✓ Create a PSpice simulation profile and simulate with PSpice
- ✓ Use the Smoke in the Advanced Analysis to stress your circuit
- ✓ Use Smoke to identify components which might fail early
- ✓ Add measurements to calculate performance of a filter
- ✓ Setup Bode Plot in PSpice Probe Window

# Keyboard shortcuts