Filippo Quadri
November 12, 2025

Cadence Virtuoso - CheatSheet

Important:
This document serves as a quick reference for performing the most basic operations in `Cadence Virtuoso`. It is not intended as a step-by-step tutorial for designing a complete circuit from start to finish. The scope is deliberately limited to the schematic and simulation stages; layout creation and post-layout analysis are not covered here.

For a comprehensive guide that walks through the entire design process—from schematic to layout and post-layout simulations (both digital and analog)—please refer to the following resource:

`/education/tutorials/FullCustom_Design`

Pro Tip

In a Linux terminal, you can navigate to a directory using the `cd` command:

```
cd path_to_directory
```

To open a PDF file in a specific directory, use the `evince` command:

```
evince name_of_pdf_file
```

Contents

1 Library and Schematic Creation

1.1 Open the Library Manager

All operations (such as creating a new library, viewing, or modifying a schematic) begin from the `Library Manager`, which functions similarly to a file explorer. When you start `Virtuoso`, the `Virtuoso Terminal` opens by default. To access the `Library Manager`, navigate to `Tools` → `Library Manager` (Figure 1).

Figure 1: Open the Library Manager

1.2 Creating a New Library

To create a `New Library`, first open the `Library Manager` (Section 1.1). Next, open the `Library Creation Helper` by navigating to `File` → `New` → `Library` (Figure 2). Enter the name of the library (e.g., `EE426_circuits`) and click `OK`.

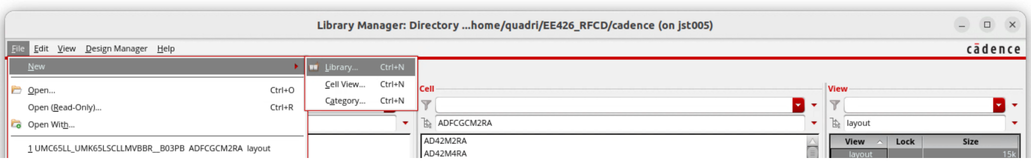


Figure 2: Opening the `Library Creation Helper`

A pop-up window like the one in Figure 3a will appear. Select the option `Attach to an existing technology library` and click `OK`. Another pop-up will then appear (Figure 3b); select `umc65ll` (as shown) and press `OK`. You should now see the newly created library listed in the `Library Manager`.

(a) Attach to a `Technology Library`

(b) Select the `Technology Library`

Figure 3: Attaching the new library to an existing `Technology Library`

1.3 Create a New Schematic

To create a New Schematic, first select the library in which you want to create the schematic. Then navigate to File → New → Cell View. Configure the pop-up window as shown in Figure 4. Be sure to select the correct Library and choose Schematic XL as the application for the view. Click OK, and the schematic editor will open.

You can now begin creating your circuit by adding instances (Create → Instance, hotkey: i), modifying their properties (hotkey: q) and connecting them with wires (Create → Wire (narrow), hotkey: w) and allow a connection with some external sources and signals by adding pins (hotkey: p). A complete list of useful hotkeys is provided in Section 1.3.1.

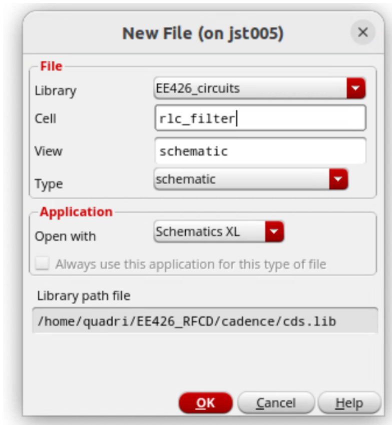


Figure 4: Create a New Circuit

When creating a new schematic, you primarily import and connect instances from different libraries (e.g., basic electrical components such as resistors, capacitors, and transistors). In addition, you can also import custom components that you have created in the current or other libraries. However, before these circuits can be reused as components, a symbol must first be created for them (see Section 1.4).

By default, a number of libraries are available: some are provided by the tool itself, while others come from the foundry design kit. The most important libraries in our case are briefly described below:

- basic: contains mostly graphical elements for circuit schematics.
- analogLib: includes many elements useful for simulation, such as voltage and current sources, ideal resistors, capacitors, inductors, and switches. These cells are primarily used to create simulation testbenches and simple analog circuits.
- umc65ll (Primitives Library): provides all the primitive devices (MOSFETs, resistors, capacitors, inductors, etc.) from the UMC foundry design kit. When you design a circuit that, in a later stage, will be printed, you should use the components inside this library
- UMC65LL_UMK65LSCLLMVBBR__B03PB (Standard Cell Library): contains the layout views from the UMC foundry design kit. You do not need to use these cells, but they can serve as examples of how standard cells are typically designed.

1.3.1 Useful Keyboard Shortcuts (Hotkeys)

A complete list of all the keyboard shortcut of the Virtuoso Schematic Editor can be found here. Figure 5 summarizes them.

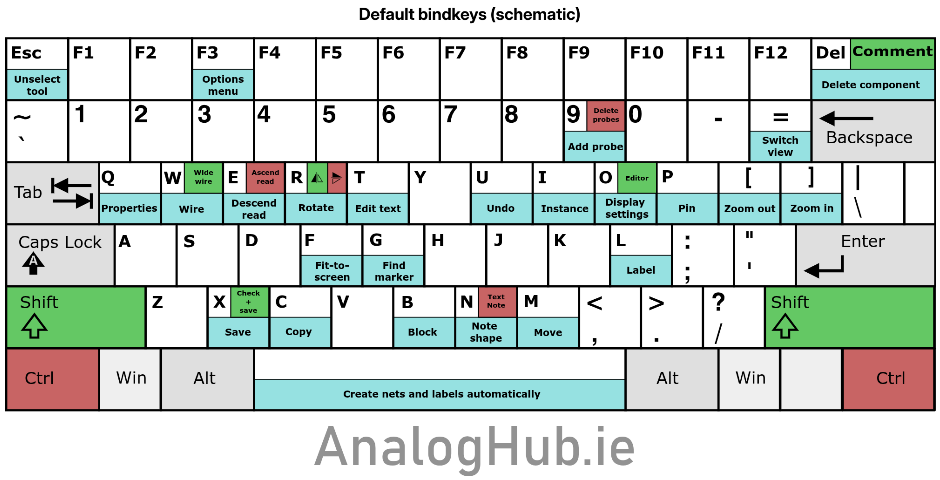


Figure 5: Virtuoso Schematic Editor Hotkeys

1.4 Create the symbol associated to a schematic

To create hierarchical circuits and testbenches, you need to associate a symbol with each schematic. A symbol is essentially a simplified view of the circuit in which only the pins are visible. When a circuit is instantiated inside another circuit, its symbol is used as the graphical representation, allowing you to connect it to sources and signals.

To create a symbol, open the `Schematic Editor` and select: `Create` → `Cellview` → `From Cellview`.

A first pop-up window will appear asking you to specify some options. The default values are usually sufficient, so you can simply confirm by clicking `OK`. A second pop-up window then allows you to define the location of the pins on your symbol. It is standard practice to place input pins on the left, output pins on the right, and power/ground pins at the top and bottom. Once you are satisfied, click `OK`.

The `Symbol Editor` will then open, where you can shape and customize the appearance of the symbol. When finished, perform a check, save your changes, and close the `Symbol View`.

2 Simulation Setup and Execution

2.1 Testbench preparation

The `Testbench` is basically the top level circuit where you are going to test the behavior of your circuit under certain conditions. Normally, it contains the `top-level module` of the block you are designing (basically the block itself) and some sources (to generate the input signals and to power the circuit) and some loads (to simulate the interaction with the environment). Except for the blocks you designed, most of the components present in the `Testbenches` are from the `analogLib`.

Pro Tip 1

Have a look at the Multiple-Bit Wire Naming Conventions to easily define multiple times the same instance (multiple instances in parallel) and see how to connect wires to those instances. To create busses, you can use the hotkey: `Shift + w`

Pro Tip 2

You can use labels as tunnel, so you do not need to route every connection! To insert a label, you can use the hotkey: `l` (L)

2.2 Create a maestro View

The maestro view is the Cadence Virtuoso view to run and handle simulations. In order to create the maestro view, proceed with the following steps

- Choose Launch → ADE Assembler in the top-left corner of the schematic editor. A small window will appear—select Create New View and click OK. If a Create New ADE Assembler View window appears, simply click OK. Make sure that the view being created is of type maestro.
- The tool may notify you that it will obtain a different license than expected. This is normal, as Cadence periodically updates and rebrands its licenses. Simply click Always to proceed.

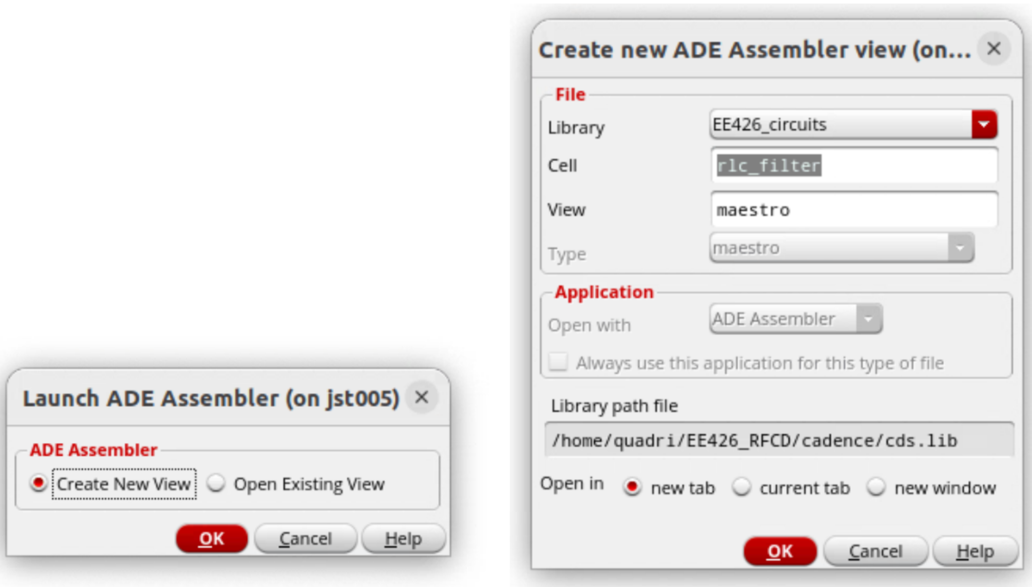


Figure 6: Create a new maestro view - Steps

2.2.1 Variables (Parameters) Definition

Similar to programming, it is often useful to define variables instead of hardcoding values repeatedly in a circuit. Using variables makes your design cleaner and simplifies simulation, as it provides a centralized overview of all component sizes and factors. Additionally, it allows you to run multiple simulations with different values without modifying the schematic itself.

Attention

The variables only work in the simulation domain! If you continue and start doing the layout you have to specify the exact dimension of each component inside the schematic!

Defining a variable is straightforward. Let's go through an example where we parametrize the value (in Farads) of a capacitor inside a circuit:

- Select the instance for which you want to create a variable (in this example, select the capacitor you want to parametrize).
- Open the instance properties (hotkey: q) and assign a variable name to the desired parameter. Use clear and valid names (avoid starting with numbers or using special characters). For example, enter

c_in in the Value field. This associates the capacitor's value (in Farads) with the variable c_in, meaning the capacitor will take whatever value is assigned to that variable.

- In the maestro view, expand the Global Variables menu and click on Click to add variable. In the pop-up window, define the variable name and its value. For example, set c_in = 1p to assign a capacitance of 1 pF.

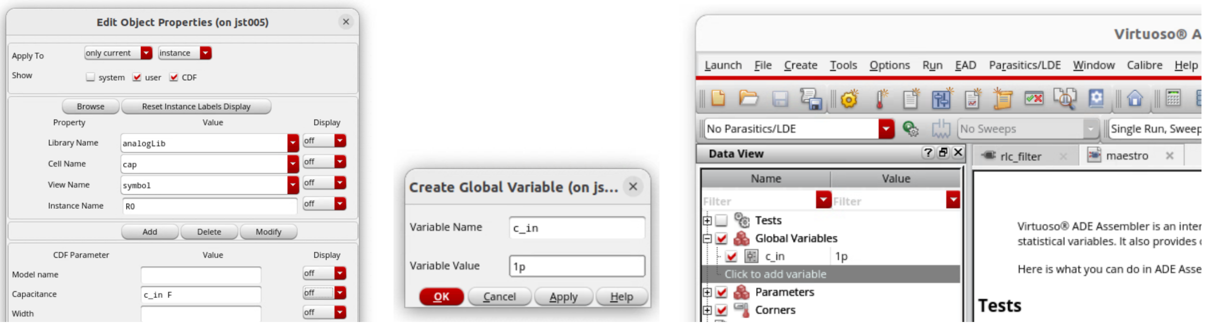
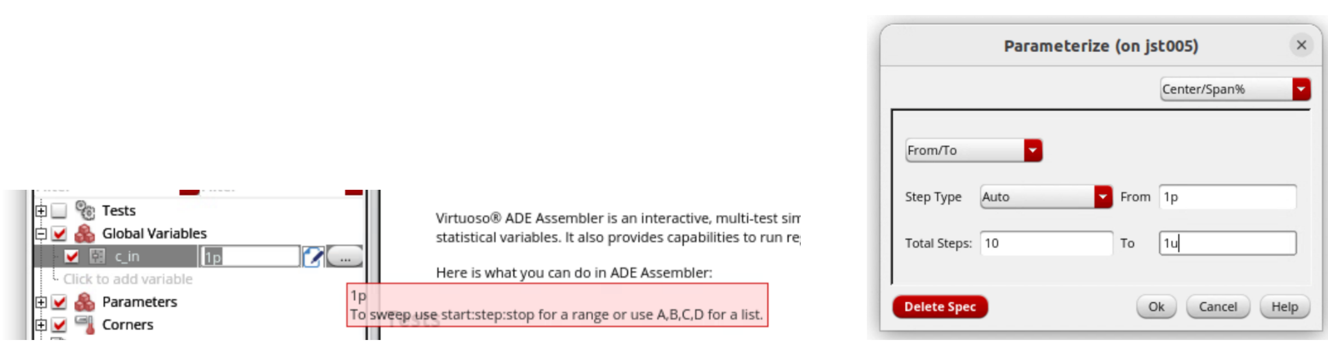


Figure 7: Setup variables in Cadence Virtuoso

2.2.2 Variable (Parameter) Sweeps

In many cases, you may want to run multiple simulations with different parameter values (e.g., varying a capacitor size) and compare the results. Virtuoso provides an easy way to set up such parameter sweeps:

- In the maestro view, double-click on the value of the variable you want to sweep. A detailed view of the variable settings will open (Figure 8a).
- Click on ... to open the sweep configuration pop-up.
- Remove any existing specifications by pressing Delete Spec until the window is empty.
- Using the selector in the top-right corner of the pop-up, choose the type of sweep you want to perform. A simple From-To sweep configuration is shown in Figure 8b. For more details on the available sweep options, click the Help button.



(a) Variable configuration window

(b) Example of a From-To sweep setup

Figure 8: Defining parameter sweeps in Cadence Virtuoso

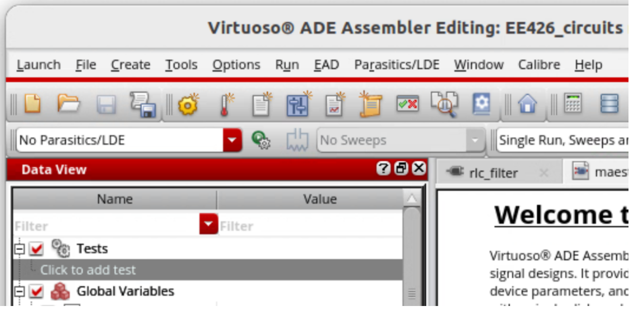


Figure 9: Adding a new Test in Virtuoso

2.3 Configuring a Simulation

To run a simulation, you first need to create a `Test` within the `maestro` view and add it to the workspace. `Cadence Virtuoso` supports multiple types of simulations (e.g., DC, AC, and Transient) that can be configured within a test. To create a new `Test`, go to `Test` → `Click to add test`, as shown in Figure 9. A pop-up window will appear: select the circuit you want to simulate (typically your testbench) and click `OK`. This will open the `Test View`.

Design Variables vs. Global Variables

In the `maestro` view, two types of variables are available: `Global Variables` and `Design Variables` (defined inside a test).

- **Global Variables:** Shared across all tests within the view.
- **Design Variables:** Apply only to the specific test in which they are defined.

A useful feature is the option to right-click in the `Design Variable` section (inside the Test View) and select `Copy from CellView`. This automatically imports all variables defined in the corresponding cell view into the test. Note that this option is not available for `Global Variables`.

Once inside the `Test View`, you can return to the `maestro` view by clicking the blue arrow near the test name (top-left corner). To rename a test, triple-click on its title (next to the blue arrow) and type the new name.

Test View vs. maestro View

- **Test View:** Focused on a single test. It allows you to configure, run, and visualize simulations related to that specific test.
- **maestro View:** Provides an overview of all defined tests. From here, you can compare results across different simulations, although some test-specific features are not available.

Choosing the appropriate view depends on whether you want to work on a single test or analyze multiple simulations together.

Once in the Test view, you should add an analysis. to do so choose `Analyses` → `Clock to add analysis` (Figure 10).

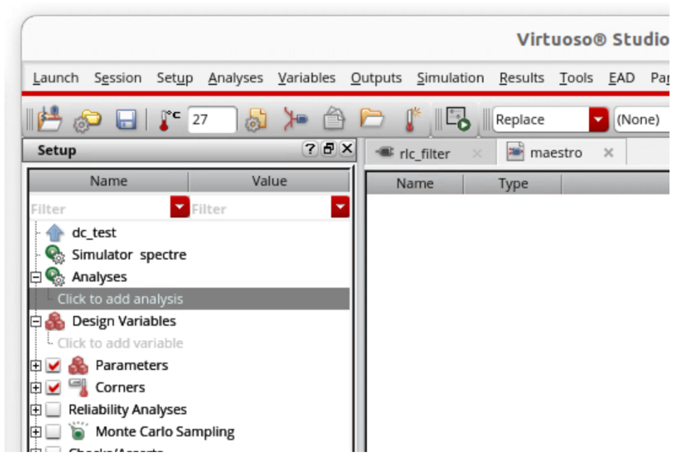


Figure 10: Add Analysis

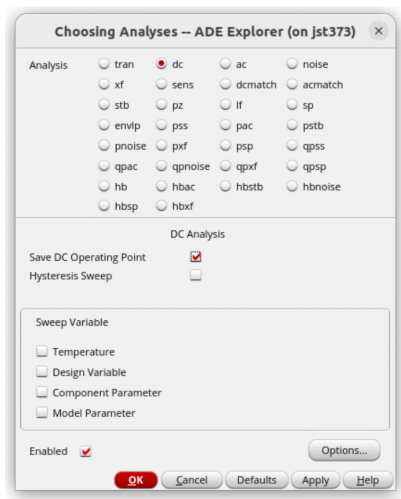


Figure 11: DC Analysis Setup

2.3.1 DC Simulation Setup

DC simulations compute the steady-state voltages and currents in the circuit, providing a baseline for further analysis (e.g. check if the polarization of the transistors in the circuit is correct).

To perform a DC simulation, first add an analysis (Figure 10). Then setup the pop-up as shown in Figure 11. It is important to select the `dc` analysis and to check the `Save DC Operating Point` option. Click `OK`. If you run the simulation from the `Test` view, you need to define all the variable again as the `Test` view has no access to the `Global Variables`.

If you run the simulation from the `Test` view, you can now annotate the DC operating point (voltages and currents) directly on the schematic by choosing `Results` → `Annotate` → `DC node voltages`. If you want to see the internal parameters of some components (like the g_m of a transistor) you can choose `Results` → `Print` → `DC Operating Point` and select the component you want to inspect.

You can also run the simulation directly from the `maestro` view. Here you cannot directly plot on the schematic, but you can check the value of some specific signal or expression (see Section 2.3.5 for further details).

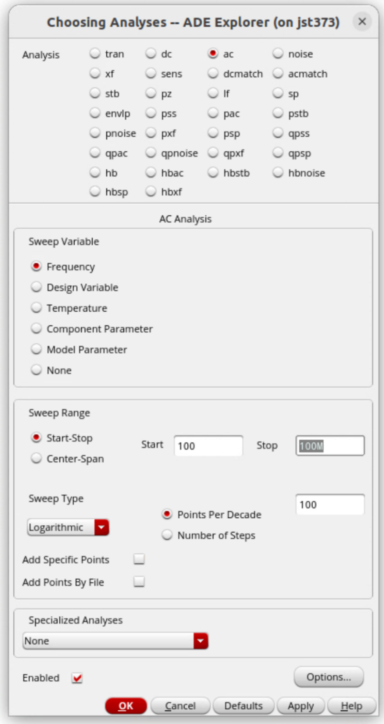


Figure 12: AC Analysis Setup

2.3.2 AC Simulation Setup

AC simulations are used to analyze the frequency response of a circuit, including parameters such as gain, phase, and impedance.

To set up an AC simulation, first add a new analysis (Figure 10). In the pop-up window (Figure 12), configure the following:

- Set `Frequency` as the `Sweep Variable`.
- Choose the desired `Sweep Range`.
- Select the appropriate `Sweep Type`.

Figure 12 shows one example configuration. Once the parameters are defined, click `OK`.

You can now run the simulation directly from the `Test View`. To plot the results, go to `Results` → `Direct Plot` → `Main Form`, complete the form as shown in Figure 13, and then click on the node of interest in the schematic. The AC magnitude response will be displayed. Note that this plot shows the absolute magnitude of the node voltage and is **not automatically normalized with respect to the input signal**.

Alternatively, in the `maestro` view, you can define a custom expression such as:

```
dB20((VF("/Vout")/VF("/Vin")))
```

to plot the transfer gain. Be careful to match the variable and node names exactly.

2.3.3 Transient Simulation Setup

Transient simulations analyze the circuit's response to time-varying signals, capturing dynamic effects such as switching behavior, propagation delays, and waveform shapes.

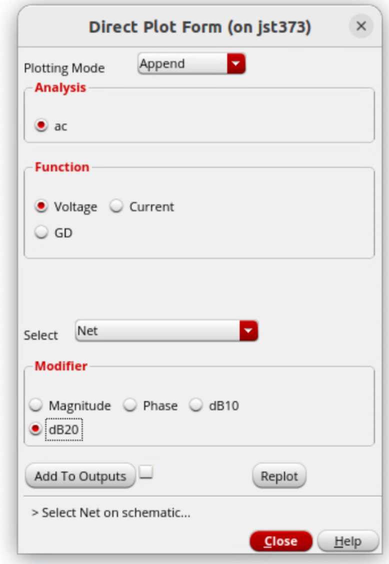


Figure 13: AC Plot Setup

To configure a transient simulation, first add a new analysis (Figure 10). In the pop-up window (Figure 14), set the following parameters:

- **Stop Time:** Defines when the simulation ends.
- **Accuracy:** Controls the precision of the analysis (in most cases, `moderate` is sufficient).

An example configuration is shown in Figure 14. Once the parameters are set, click `OK`.

For transient simulations, it is often more convenient to plot signals directly from the `maestro` view (click the blue arrow to return to maestro). See Section 2.3.5 for further details.

2.3.4 Noise Simulation Setup

Noise simulations are used to evaluate how different circuit components contribute to the overall noise performance of a design. They provide key insights into parameters such as the total output noise, input-referred noise, and the Noise Figure (NF) across frequency. This type of analysis is essential for validating low-noise designs, such as LNAs or sensor front-ends, and for identifying the dominant noise sources.

To set up a noise simulation, add a new **Noise Analysis** (Figure 10). In the configuration window (Figure 16), specify the following parameters:

- Set **Frequency** as the *Sweep Variable*.
- Define the desired **Frequency Sweep Range**.
- Choose the appropriate **Sweep Type** (e.g., linear or logarithmic).
- Select the **Output** and **Input Noise Nodes**. For single-ended configurations, set the negative terminal to ground.

Once these parameters are configured, the noise simulation is ready to run. This analysis enables the plotting of several important quantities, including the *Noise Figure*, input-referred and output noise spectra, and total integrated noise over a given bandwidth. Refer to Section 2.3.5 for detailed instructions on plotting simulation results in the Maestro environment.

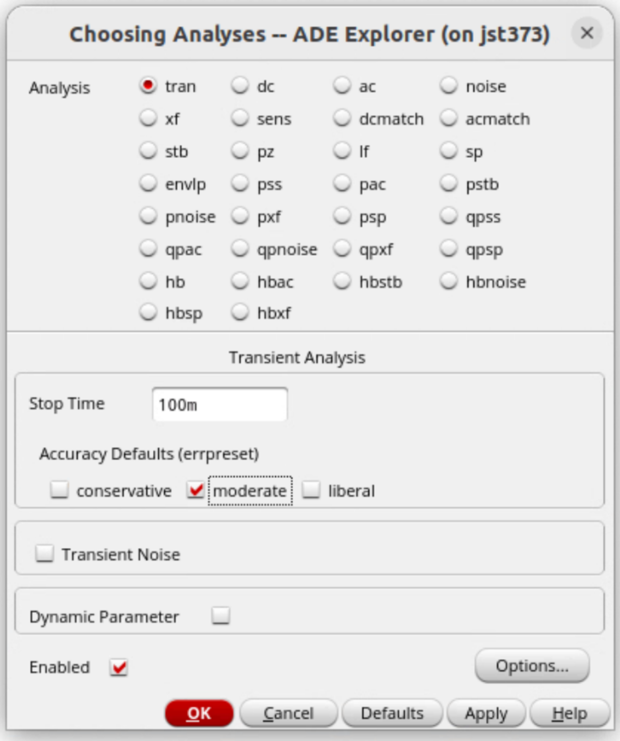


Figure 14: Example setup for a Transient Analysis

In addition to graphical plots, you can also generate a **Noise Summary Report**, which provides a detailed breakdown of the main noise contributors at a specific frequency. To obtain this report, navigate to:

`Results` → `Print` → `Noise Summary`

Then, select the instances to include (`Include All Types` is recommended) and specify how many top contributors to display in the `Truncate` section. This feature helps identify which devices or circuit sections dominate the noise performance. An example of the resulting summary output is shown in Figure 15.

2.3.5 Plotting from the maestro View

Working in the `maestro` view can be very useful, as it allows you to plot signals and evaluate expressions across different simulations. Figure 17 illustrates how to add a new signal for plotting.

The procedure is as follows:

1. Select the test of interest.
2. Choose whether to plot a signal or an expression (to simplify expression creation, you can use the calculator via `Tools` → `Calculator`).
3. Once a new signal has been added to the view, double-click in the `Details` field and click the three dots (...). This opens the schematic, allowing you to select the node or signal to plot.
4. When ready, click the green button at the top center of the window to run the simulation and display results.

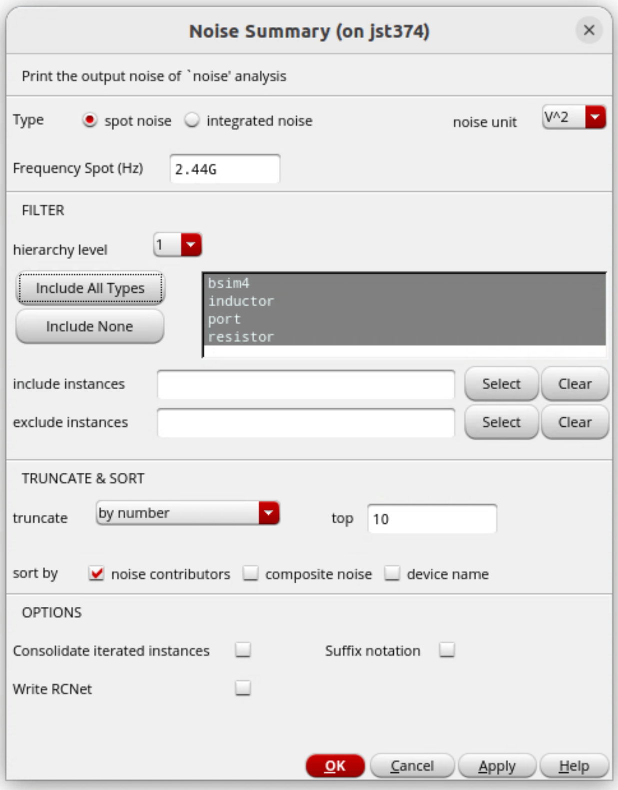


Figure 15: Example setup for printing the Noise Summary

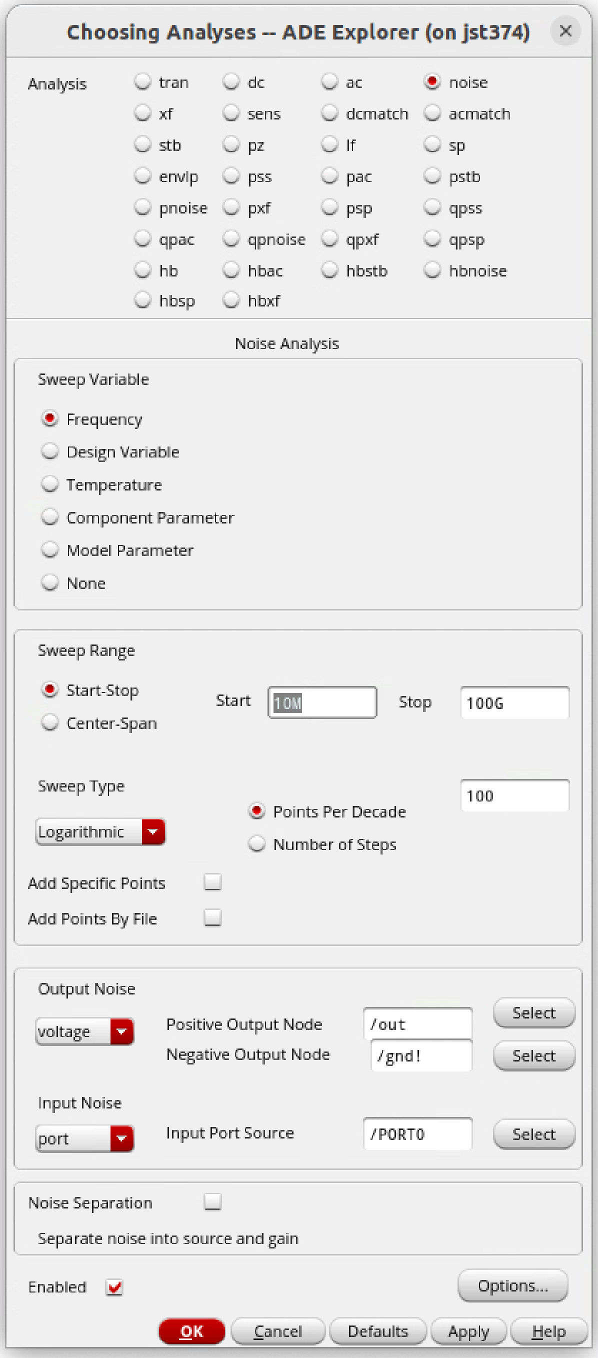


Figure 16: Example setup for a Noise Analysis

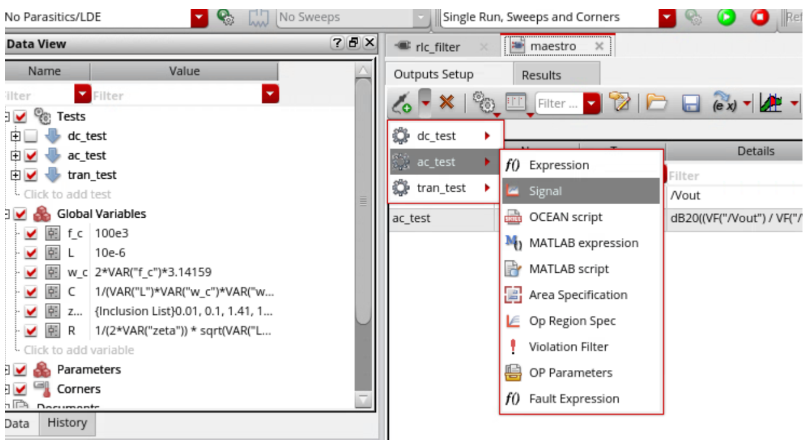


Figure 17: Adding and plotting signals from the maestro view