

Revolution EDA Preview Release First Steps

This document is written to help the users to get acquainted with Revolution EDA Preview Release. **Revolution EDA** is a complete custom IC design that currently includes:

1. Glade schematic and layout editors.
2. Xyce circuit simulator including direct access to AC, TRAN, HB, NOISE and DC analyses. SP analyses will be added soon.
3. Revolution EDA Waveform plotter with a notebook interface.
4. GCC and ADMS for inclusion of Verilog-a behavioural models.
5. Revolution EDA Verilog-A model importer and symbol generator.
6. Example Sky130 process symbols with callback functions.
7. Example Sky130 process layout parametric cells.
8. Example Sky130 process substrate taps and vias.
9. Revolution EDA simulation GUI including:
 1. Point-and-click drawing of waveforms.
 2. More than 20 performance parameters related to AC, TRAN, NOISE and HB analyses.
 3. Sweep of circuit parameters in steps or by choosing particular values.
 4. The order of sweeps can be changed.
 5. Process corners in the model libraries can be chosen.
 6. Node voltages, currents, some element values and small-signal parameters can be saved to be plotted after each simulation run.
 7. Simulation options such as transient integration methods, etc can be changed within GUI.

Revolution EDA preview release includes a few example schematics, behavioural models and layout parametric cells. In this document, we will concentrate on simulating the example transimpedance amplifier circuit that uses Sky130 process models.

We assume that the user has already installed Revolution EDA container image on his or her computer and has started the image.

Introduction

The user will be greeted with the following window once the container image is started.

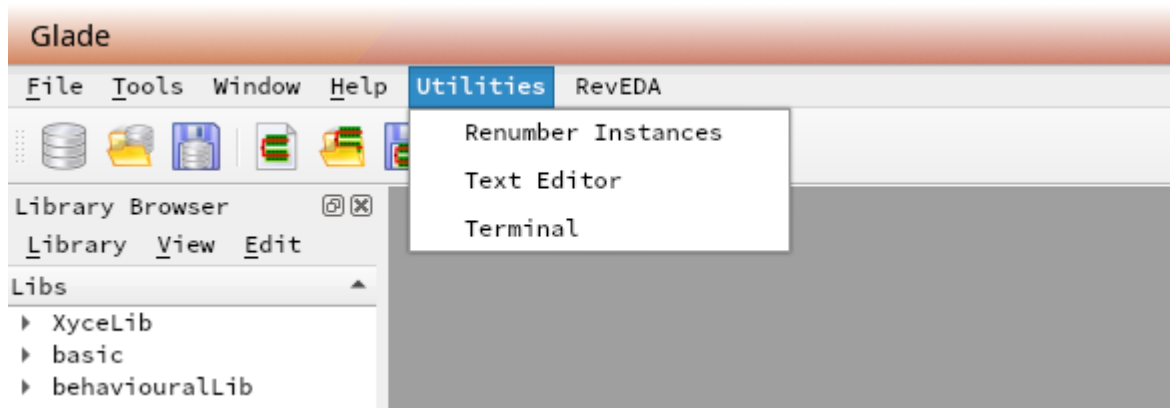


As a start, we would like to review two last menu items on the menu bar.

1. Utilities

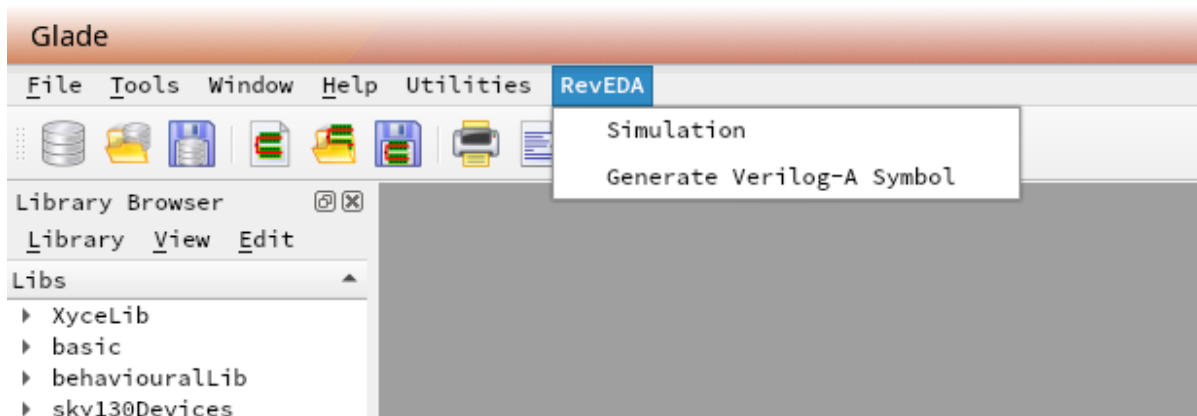
Utilities menu has three items:

1. *Renumber instances*: This is used to renumber instances in the schematic.
2. *Text Editor*: It opens “Featherpad” editor to allow easy editing or viewing of text files.
3. *Terminal*: This starts “QTerminal” to start a terminal in the container image.



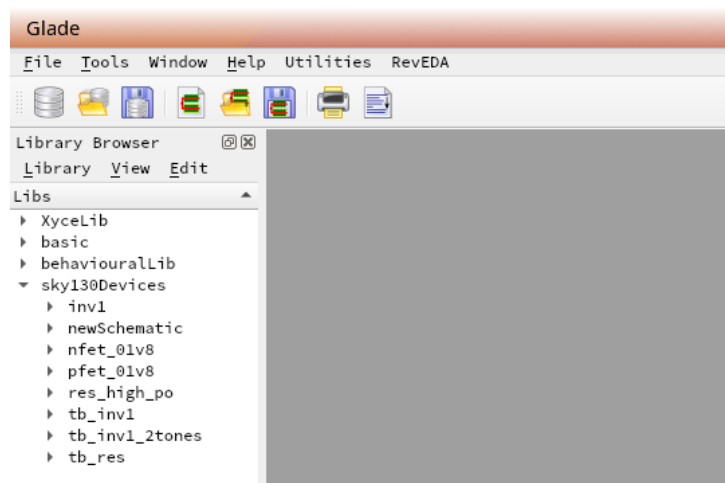
2. RevEDA

1. *Simulation*: It starts the main simulation GUI for Revolution EDA environment.
2. *Generate Verilog-A model*: It starts a dialogue for easy integration of Verilog-A behavioural models in Xyce simulations.

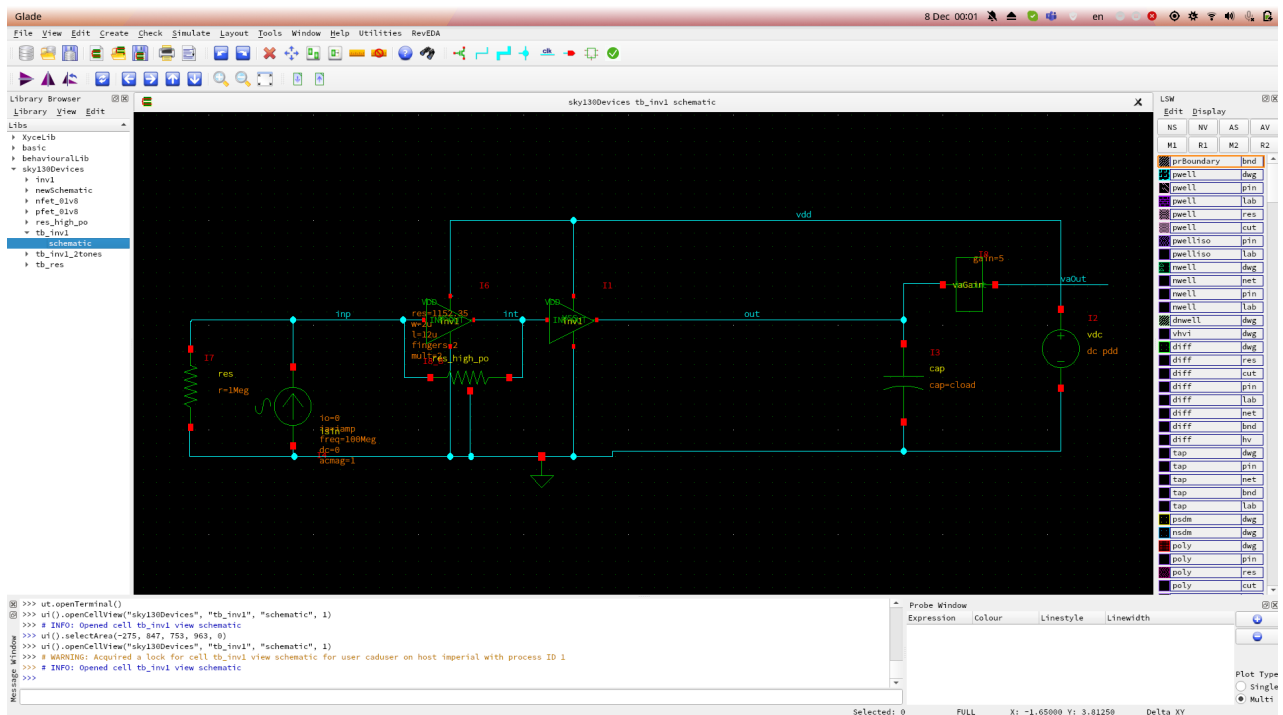


An example simulation testbench

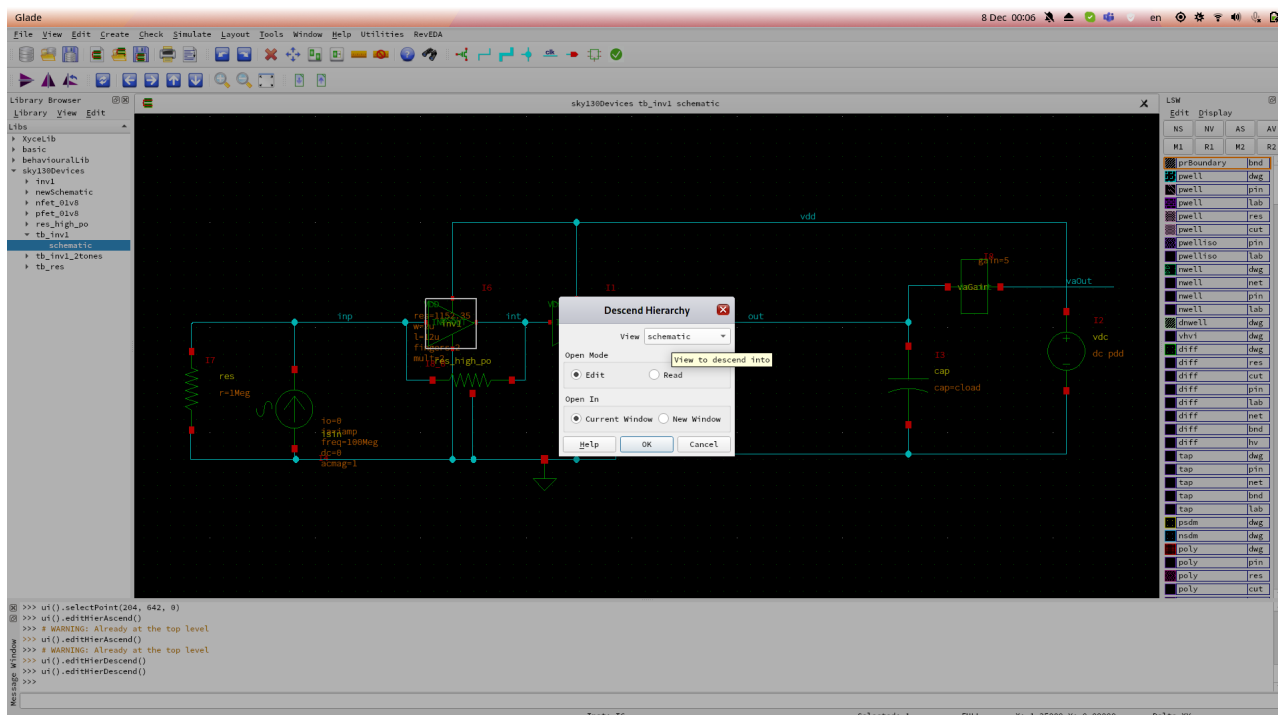
Now, click on the triangle left of the *sky130Devices* library to reveal the cells in this library:



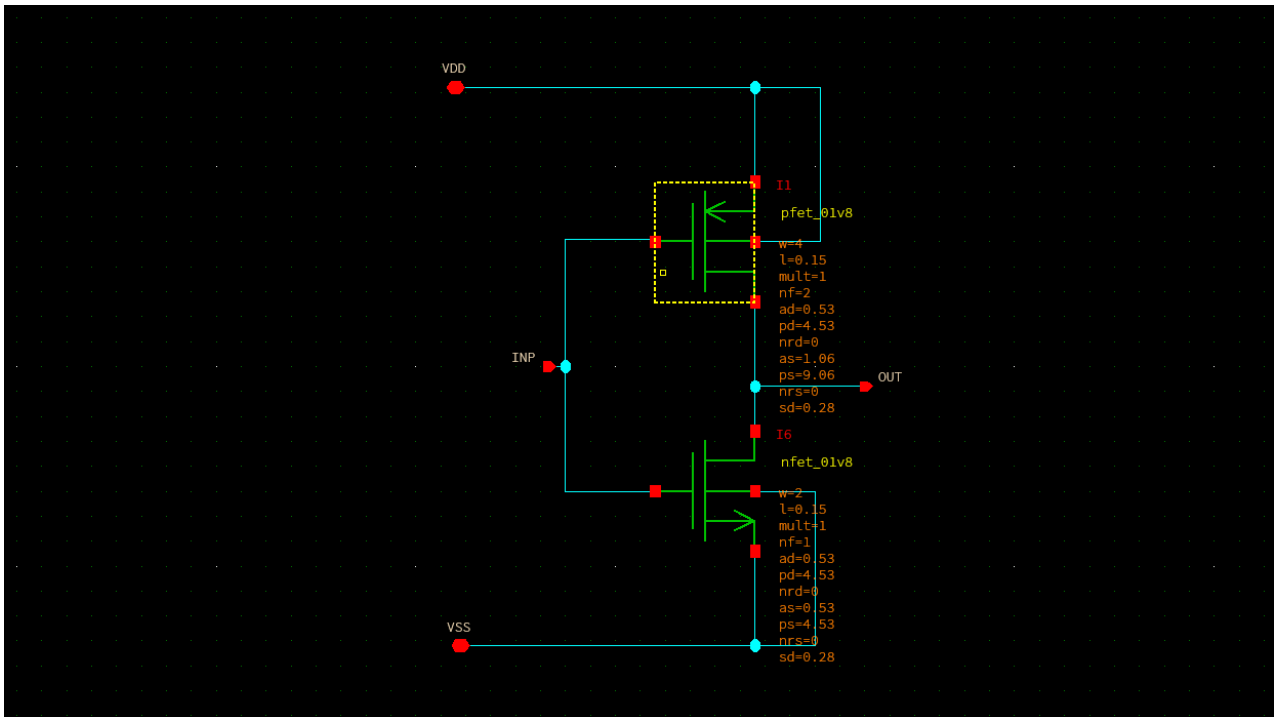
We will start with *tb_inv1* cell. Once again click on the triangle left of *tb_inv1* cell to reveal available schematic views. In this case, there is only *schematic* view. Double-click on schematic item, to open the schematic view.



Select one of the inverters denoted by *inv1* cellname in the schematic and then descend in the hierarchy. Either you could try *shift-X* key combination or click on down-arrow key on the second-row of toolbar.



We can now descend in the hierarchy to *inv1* cell. This is a simple digital inverter, used as a class-AB amplifier in this example. Note that *nfet* and *pfet* devices are annotated with not only with customary width (W), length (L), number of fingers (nf) and multiplication factor (mult), but also second-order geometry dependent parameters such as *nrd*, *nrs*, *pd*, *ps*, etc. This is one of the most important advantages of Glade schematic editor in Revolution EDA that allows the use of full Python-based **callback** functions to determine such parameters.



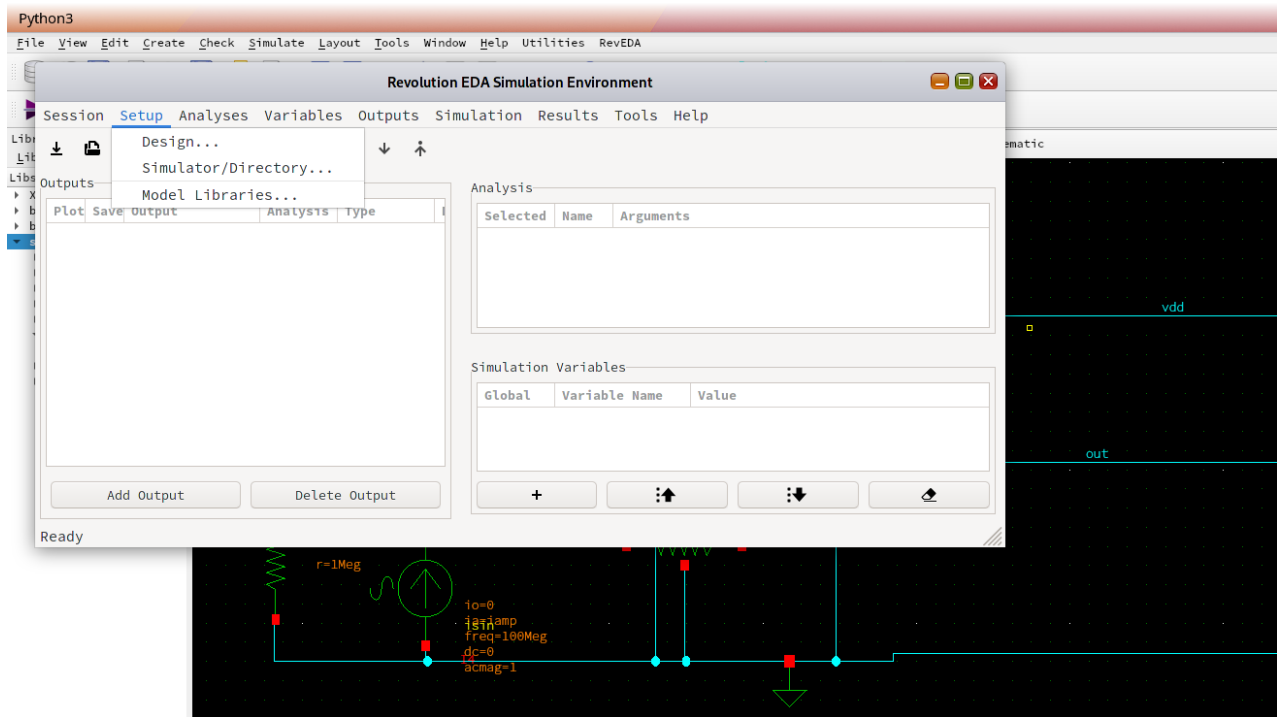
Simulation GUI

Now let's go back to the top-level test schematic and try few simulations. You could just use *Shift-B* key combination or click up-arrow on toolbar to do that.

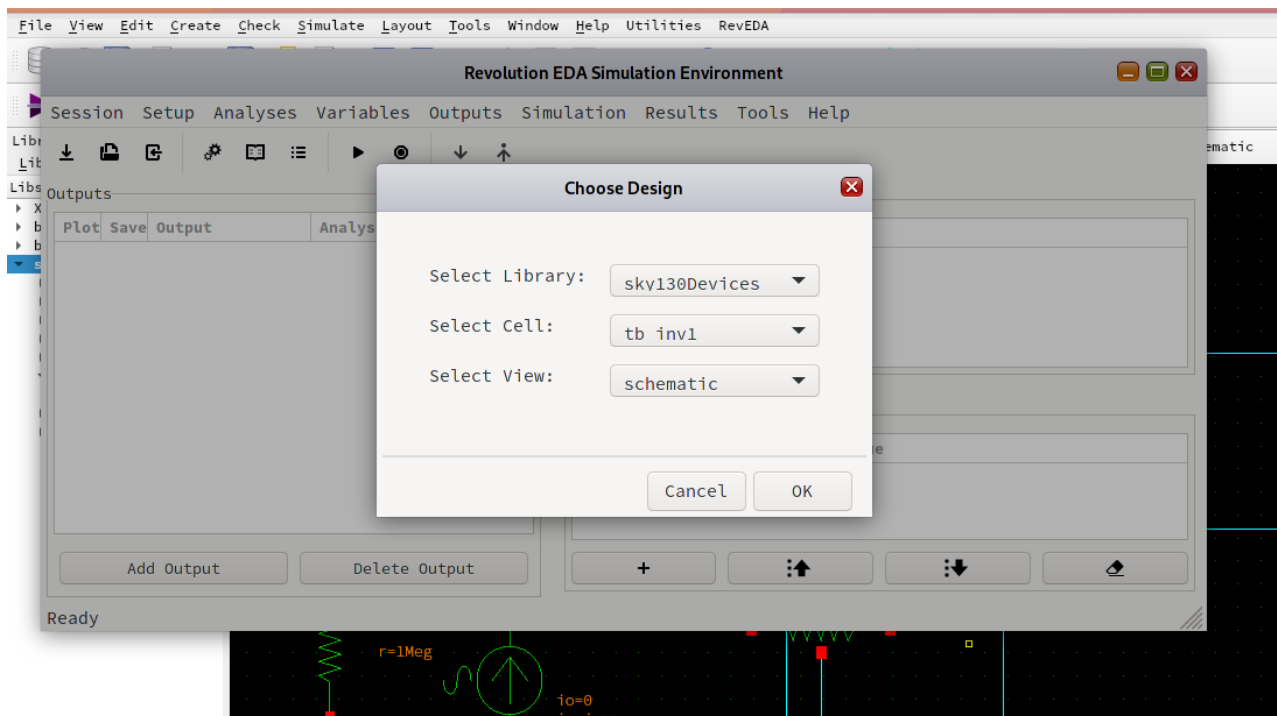
Click on RevEDA menu on the menubar and select *Simulation* menu item. You will be presented with a familiar analogue simulation environment. For this example, we will create our simulation setup from scratch. Once a simulation setup is created, it can be easily saved and restored in a YAML formatted text file. YAML files are easy to read and modify helping the reuse of such simulation setup files for other testbenches. Compare this to the legacy solutions from other EDA vendors which require lock-in to their tools.

Setup

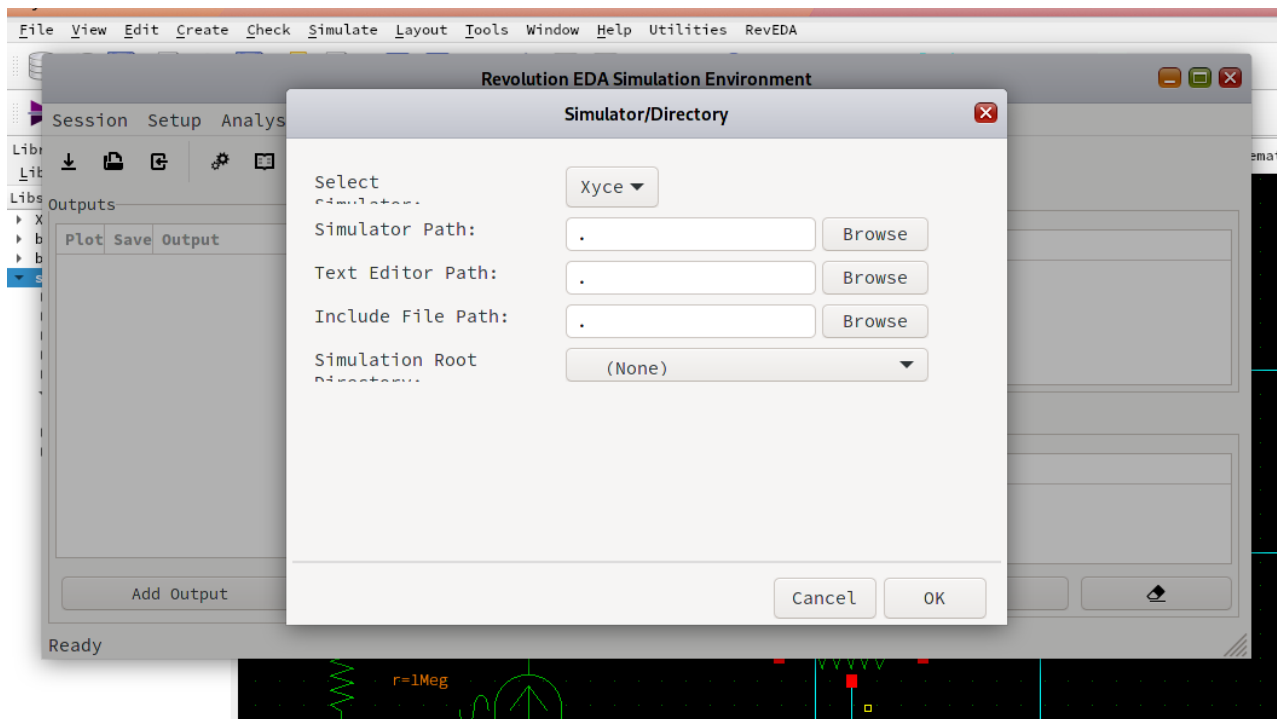
The setup steps are logically laid out in the menubar. The first stop is *Setup* menu. Click on *Design* menu item to check if the right design is selected. This is important when more than one design is opened in Glade schematic editor.



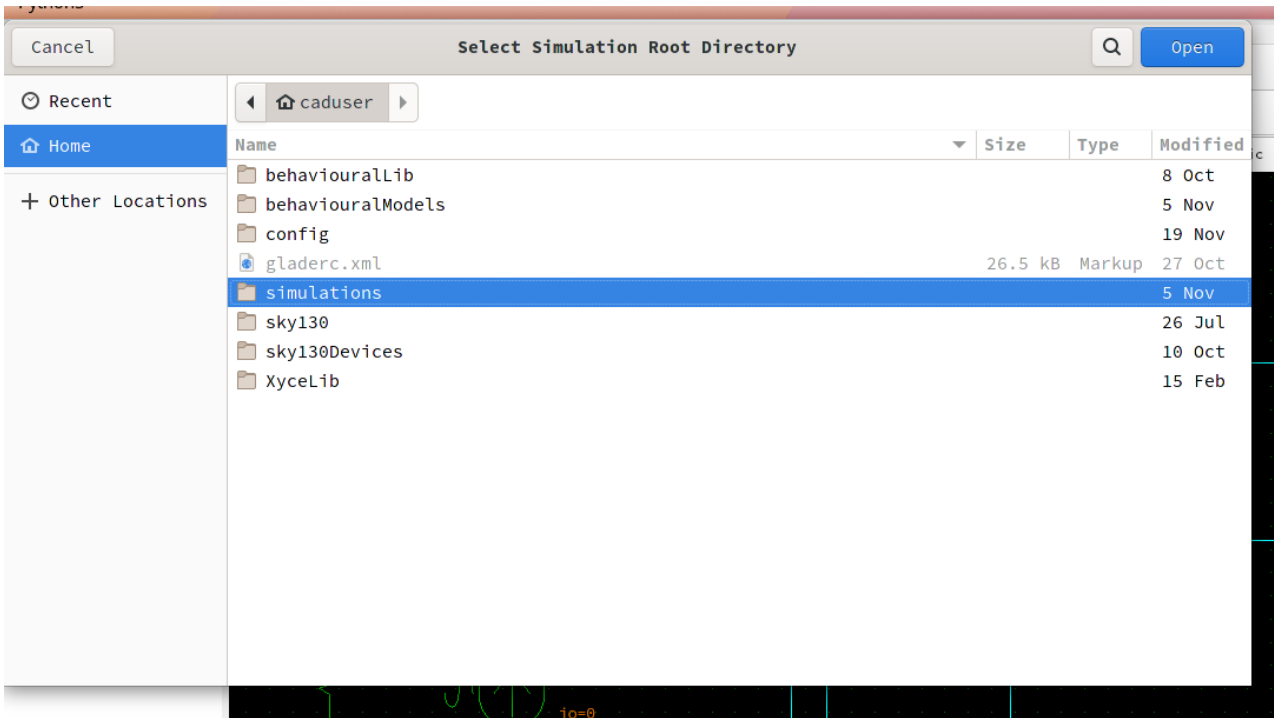
Default action is to use the schematic view when the *Revolution EDA* simulation environment is started. You could always select another library, cell and cellview in *Choose Design* dialogue.



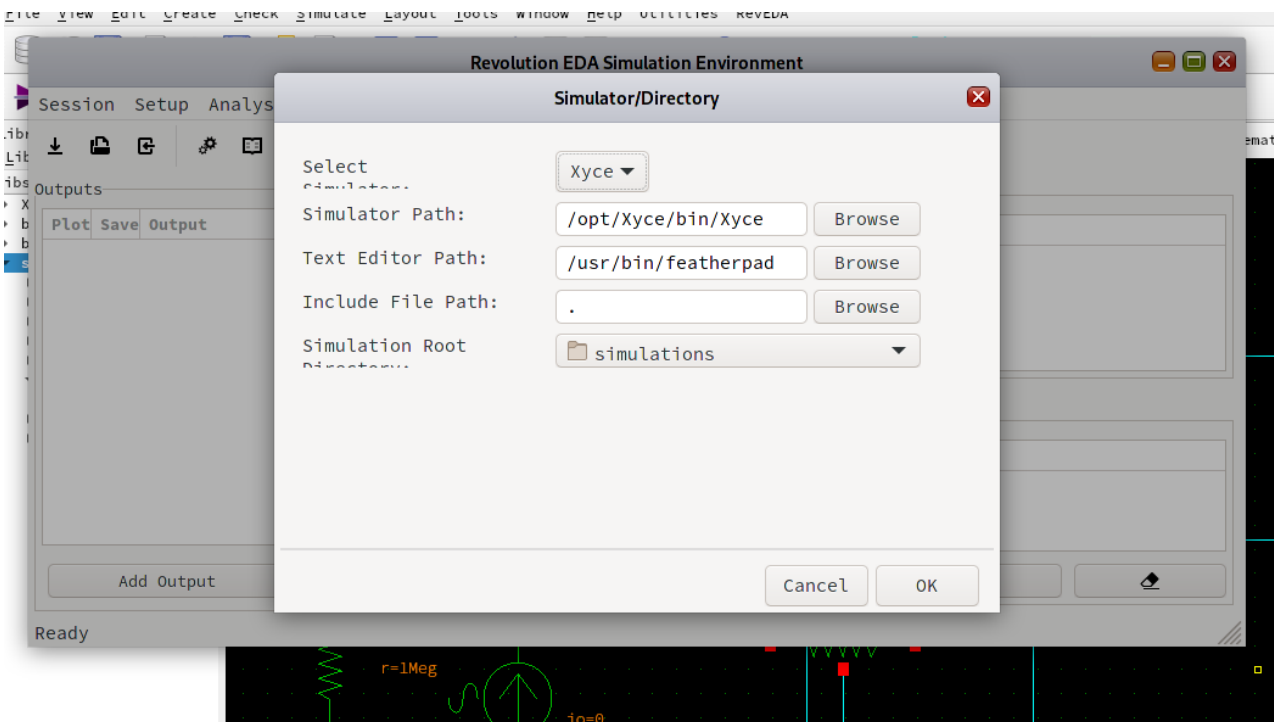
After selecting the right testbench, click OK and let's move to *Simulator/Directory...* dialogue. For the moment, only Xyce simulator can be used within Revolution EDA. In the future, we might add other circuit simulators as needed. The next step is the selection of the simulator path. In the container image, it is at */opt/Xyce/bin/Xyce* path. The next step is not essential for the operation of Simulation GUI is useful to be able to view simulation log and output files. Here, we are using *featherpad* editor. *Include File Path* is there when the user needs to use Xyce functions that are not yet implemented in the simulation GUI. We will leave empty for the moment. The next step is the selection of simulation output root directory. Revolution EDA will create a separate directory for each library and under that directory another directory with the name of testbench to store the simulation log, output and data files.



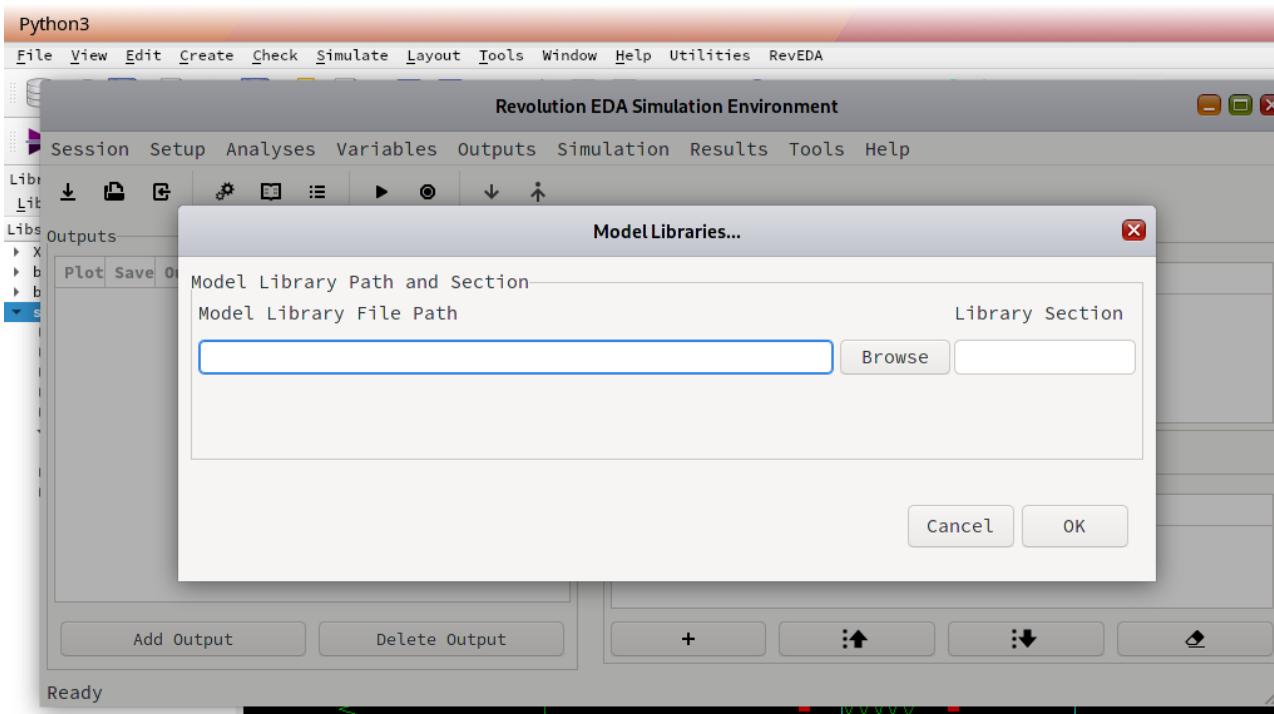
There is already a *simulations* directory under */home/caduser* path. Choose it as shown below and click *Open*.



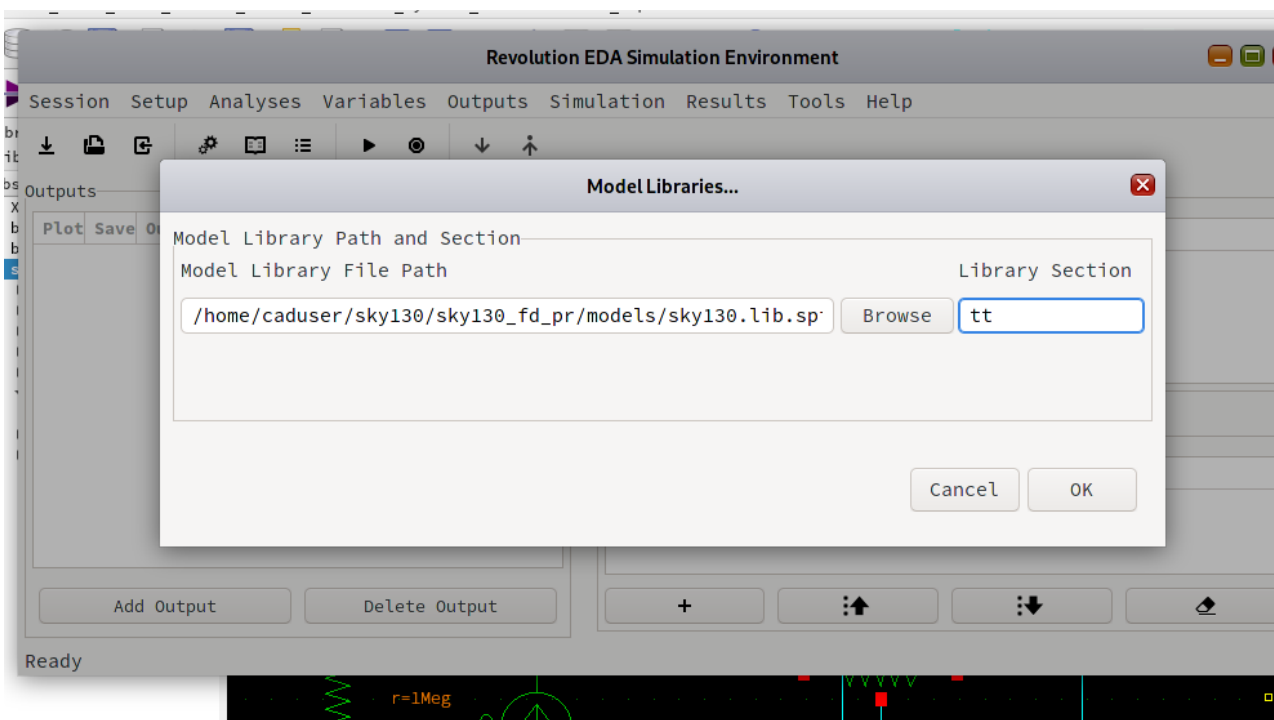
You should now the following have the following settings for your testbench:



Click OK to close this dialogue. The next step is the selection of simulation models using *Model Libraries...* dialogue. In this example, we will be using modified **Sky130** process models. Click on *Browse* to open file chooser dialogue.



Find and choose *sky130.lib.spice* file under */home/caduser/sky130/sky130_fd_pr/models/* path and click *Open* to return to *Model Libraries...* dialogue. Enter *tt* in the *Library Section* line and click *OK*.



Library section *tt* signifies that we selected so-called *typical-typical* models. We could also have used other process corners such as **fast-fast ** by entering *ff* or *slow-slow* by entering *ss*. You need to check Sky130 model documentation to see which process corners are available. Now click *OK* to finish with the setup phase.

Analyses

The next step is setting up the simulation analyses. Revolution EDA currently can handle five analyses types:

1. TRAN
2. AC
3. DC
4. NOISE
5. HB

A sixth analysis type, SP, will be added soon.

Click on *Analyses* menu and select *Analyses...* menu item. This dialogue has a tabbed interface to allow setting up the parameters for one or more analyses. Xyce can normally do only one simulation analysis at each run. Revolution EDA sets a run for each selected analysis type and runs them as needed.

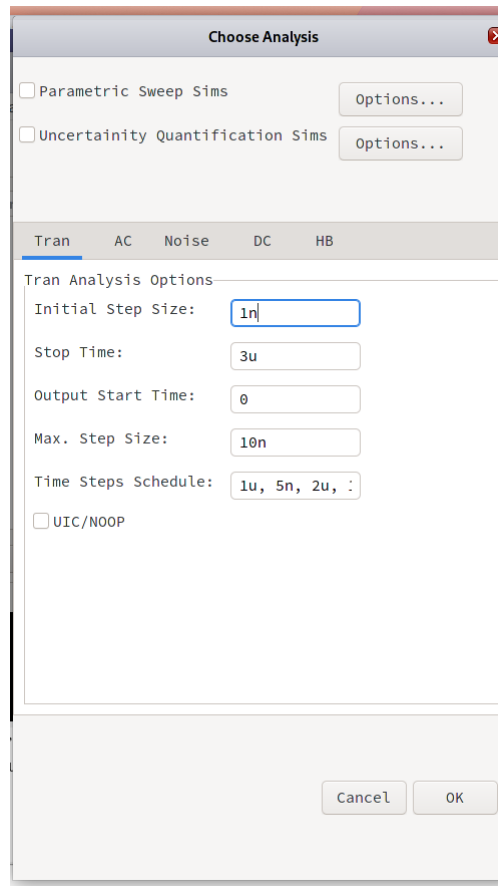
TRAN Analysis

TRAN analysis options follow Xyce transient analysis parameters. You can set here most important parameters related to Xyce transient analysis:

1. **Initial Step Size:** This is the value of first time step.
2. **Stop Time:** Duration of the transient analyses.
3. **Output Start Time:** The output will be saved at the output file starting at this time.
4. **Max. Step Time:** Sets the maximum time step. Xyce defaults to

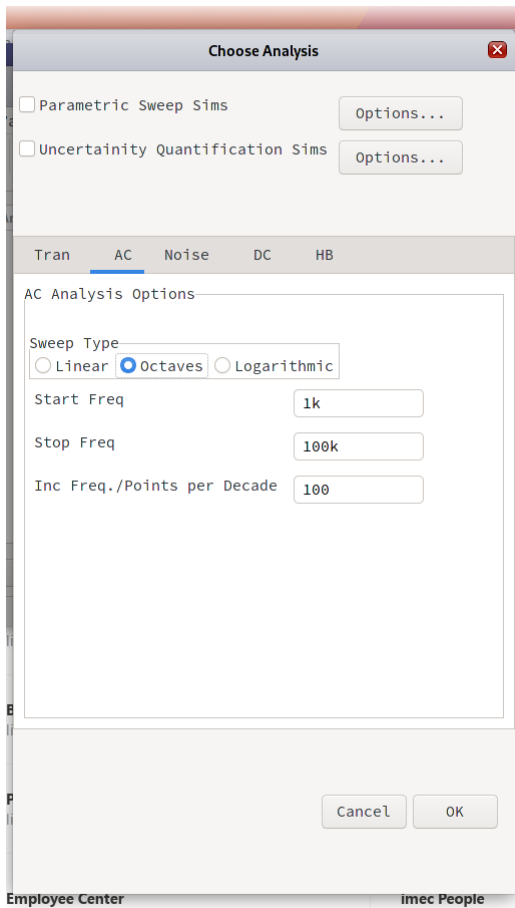
$$\text{Max. Step Time} = \frac{\text{Stop Time} - \text{Initial Step Size}}{10} \quad (1)$$

5. **Time Steps Schedule:** Defines a schedule of time steps for transient simulations. More information can be found in Xyce Reference Guide.
6. **UIC/NOOP:** If you prefer Xyce to skip the operating point calculation, then you can check this box. This is especially useful for circuits with more than one stable operating points or more oscillators. It is possible to set individual node voltage initial conditions inside Revolution EDA.



AC Analysis

In AC analyses, the frequency is swept. Frequency sweep type can be linear, or logarithmic in octaves or decades. For linear sweep, the number of frequency points is entered. For logarithmic sweeps number of frequency points per octave or decade is determined.

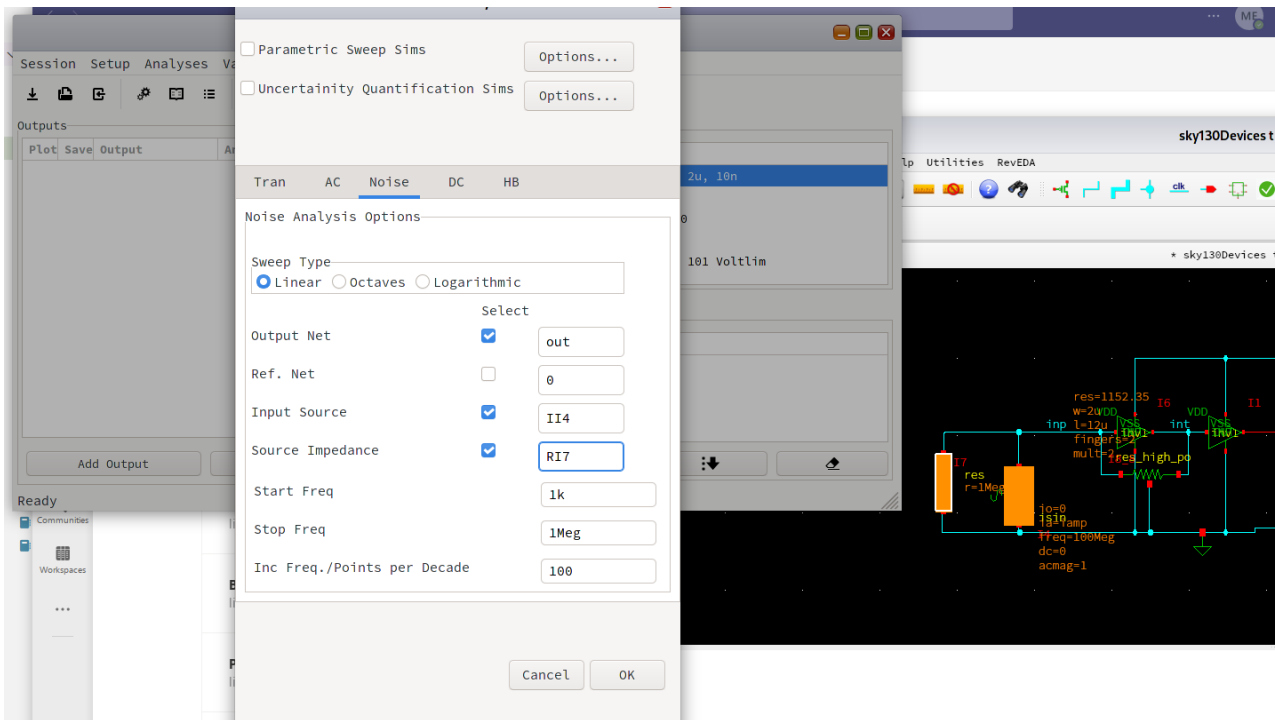


Noise Analysis

Noise analysis is similar to AC analysis. However it requires the input of some extra parameters. For more detailed information, please refer to Xyce Reference and User Guides.

Output Net: The circuit output node. To select a node on the schematic, just click checkbox under *Select* column, and then select a node on the schematic. Make sure that cursor is now in node name entry field and press *ESC* (escape key). Node name should be correctly displayed in the relevant field. The other fields can be also filled similarly. If *source impedance* field is filled, Revolution EDA can also calculate *noise figure* and *noise factor* over the simulation frequencies.

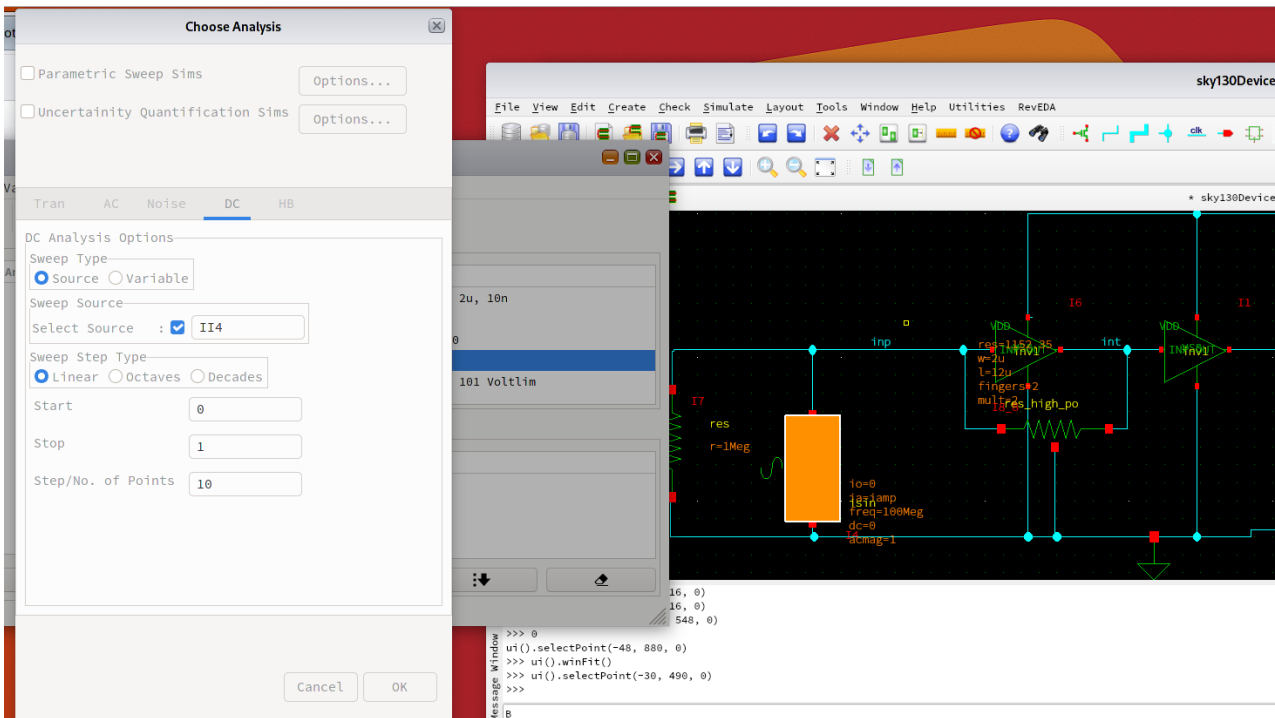
1. **Output Net:** Output net for the noise simulations.
2. **Ref Net:** Normally 0, i.e. ground, but could be another node.
3. **Input Source:** Signal source to which output noise will be referred to.
4. **Source impedance:** Impedance of the signal source, important for *noise figure* and *noise factor* calculations.
5. **Start Freq, Stop Freq and Inc Freq./Points Per Decade:** Frequency specifications.



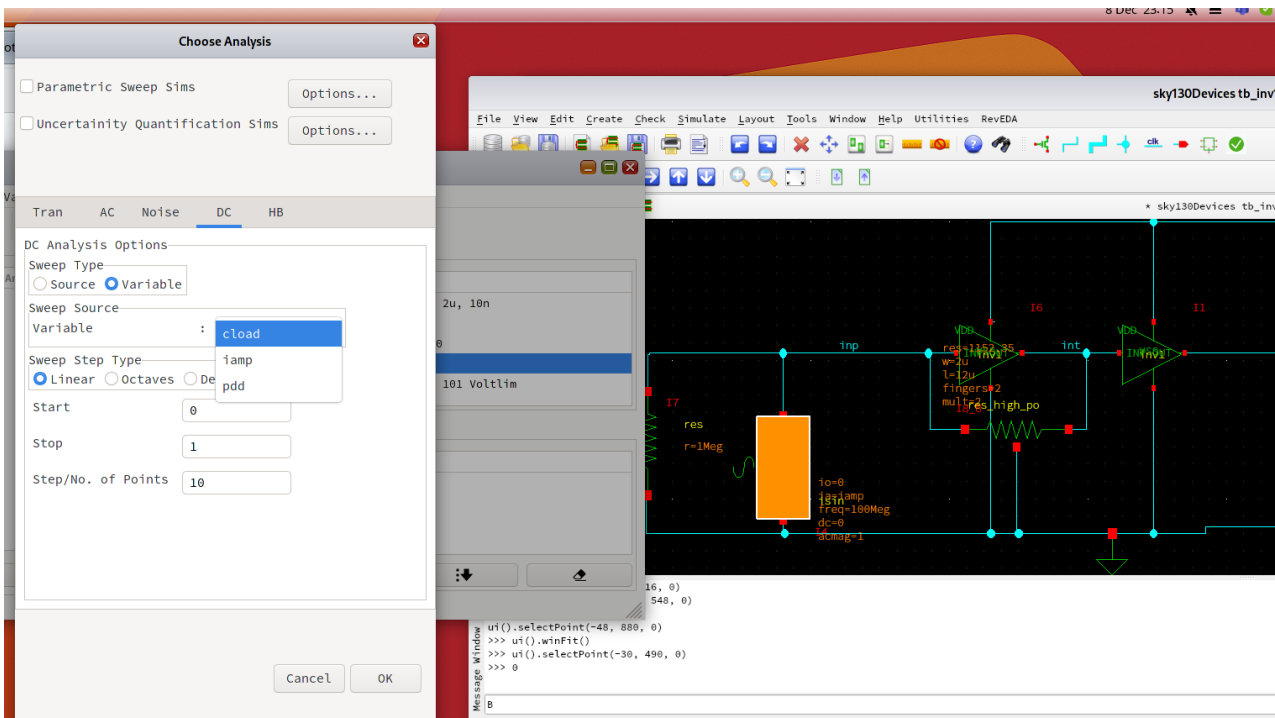
When an instance is selected for *input source* and *source impedance* fields, the selected instances will be marked by an orange rectangle over the instances. Once the dialogue is closed, this marking rectangles will be cleared.

DC Analysis

DC analysis tab allows user to sweep either a source or design variable. If a source is to be swept, then select *Source* radio button in *Sweep Type*. To select the source, click the checkbox before source name field, select an instance in the schematic and then place the cursor in the field, and press *ESC* key.



Alternatively, a global parameter can be swept by choosing *Variable* button as Sweep Type. In this case, the user will be presented with a combo box listing all the *global variables* defined in the GUI.



Similar to AC and Noise analysis, the sweep variable can also be swept linearly or logarithmically per octave or decade.

HB analysis

Harmonic balance analysis requires one or more sinusoidal current or voltage sources. It is important that frequencies in HB analysis tab correspond to frequencies defined for the sources on the schematic. At the moment, Xyce does not accept parameters for frequencies of the sources. If the frequencies used in HB analysis tab do not match the frequencies of the sources, the results of the simulations may not be meaningful.

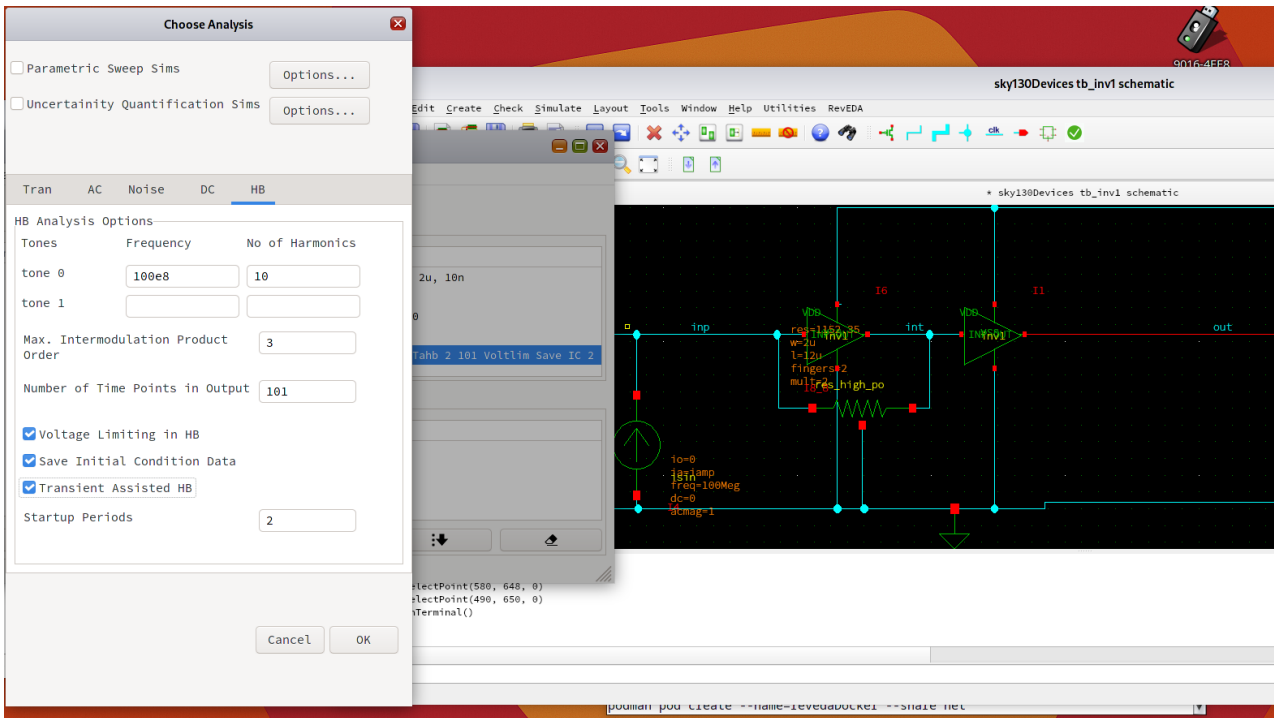
If there are more than two harmonics frequencies to be simulated, just press *ENTER* after filling *number of Harmonics* field for *tone 1*, and a new line called *tone 2* will appear. There are no limits on the number of tones that can be entered. Note that it is advised to use the tone that produces most nonlinear response by the circuit as the first tone in Xyce Reference Guide.

By default, the frequency and number of harmonics for one tone is filled as a guidance. Feel free to change to fit your circuit. Once frequencies and number of harmonics entered, the user can select some other pertinent options for HB analysis such as:

1. **Max. Intermodulation Product Order:** This settings is important for strongly non-linear circuits.
2. **Voltage limiting in HB:** Sets the voltage limiting for HB analysis. Equivalent to `.OPTION HBINT VOLTLM=1`
3. **Save Initial Condition Data:** Saves the initial conditions in a file. The file has an extension of `hb_ic.prn`.
4. **Transient Assisted HB:** This option activates transient assisted HB option to help the convergence. If the checkbox is checked, a new field will appear titled *Startup Periods*.

Equivalent to:

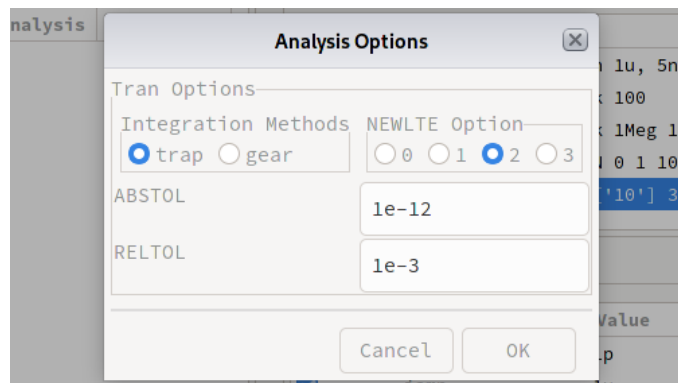
```
1 | .OPTION HBINT TAHB=1
2 | .OPTION HBINT STARTUPPERIODS=2
```



Options

Analysis Options dialogue allows the user to setup a few important simulation related options such as:

1. Integration Method: trap or gear. Equivalent to: `.OPTIONS TIMEINT METHOD=TRAP` or `.OPTIONS TIMEINT METHOD=GEAR`
2. NEWLTE: According Xyce Reference Guide, "This parameter determines the reference value for relative convergence criterion in the local truncation error based time step control". The possible values are 0,1,2 and 3 and can be selected using radio button controls.
3. ABSTOL: Absolute tolerance for the solver.
4. RELTOL: Relative tolerance for the solver to determine whether the solution has converged.



Variables

The next menu, we will cover is *Variables* menu. It has currently only one menu item, titled *Variables....* Invoking *Design Variables* dialogue, we can specify the design parameters. The design parameters can be global or local. The global parameters can be used in sweeps.

There are two methods to change parameter values in sweeps.

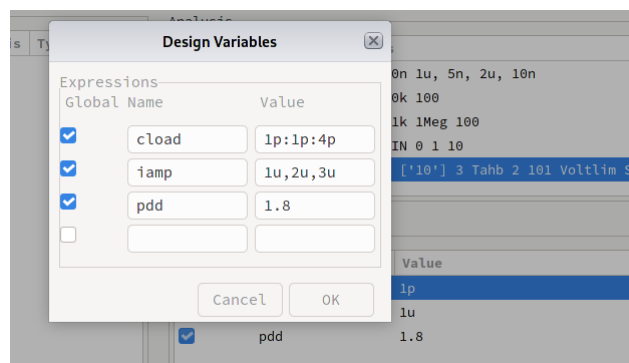
1. The user can enter the parameter values in step increments:

```
start value: step value: stop value
```

2. The user can select specific parameter values to be used in the sweeps:

```
value 1, value 2, value 3
```

In the screenshot below, an example where both types of parameter sweeps are used is shown:

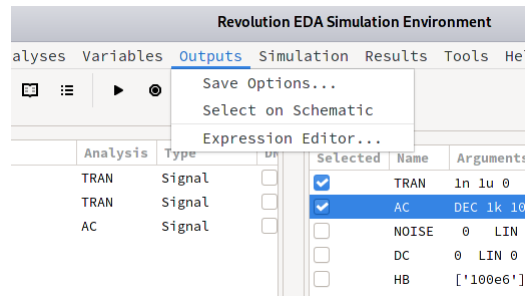


`cload` parameter will be swept from 1p to 4p in 1p increments, while `iamp` parameter will take 1u, 2u and 3u values. Note that there will be a total of 12 simulations for each selected analysis. For each value of `iamp` parameter, `cload` parameter will be swept between 1p and 4p. In other words, `cload` sweep will be the *inner sweep*, while `iamp` will be the *outer sweep*. There is no limit to how many parameters can be swept simultaneously. Using *Simulation variables* panel on the main window, the order of parameter sweeps can be changed. This will be covered later in this document.

Outputs

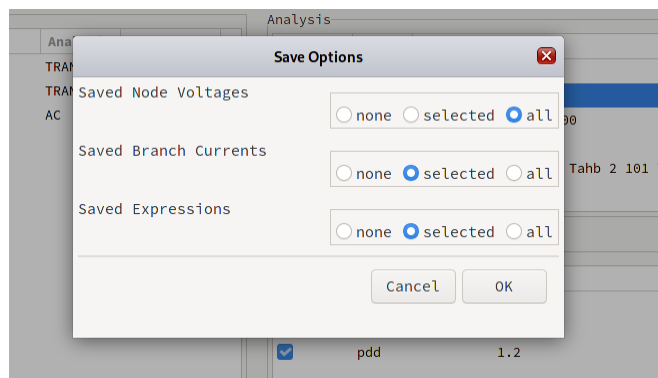
Outputs menu has three menu items:

1. Save Options...
2. Select on Schematic
3. Expression Editor



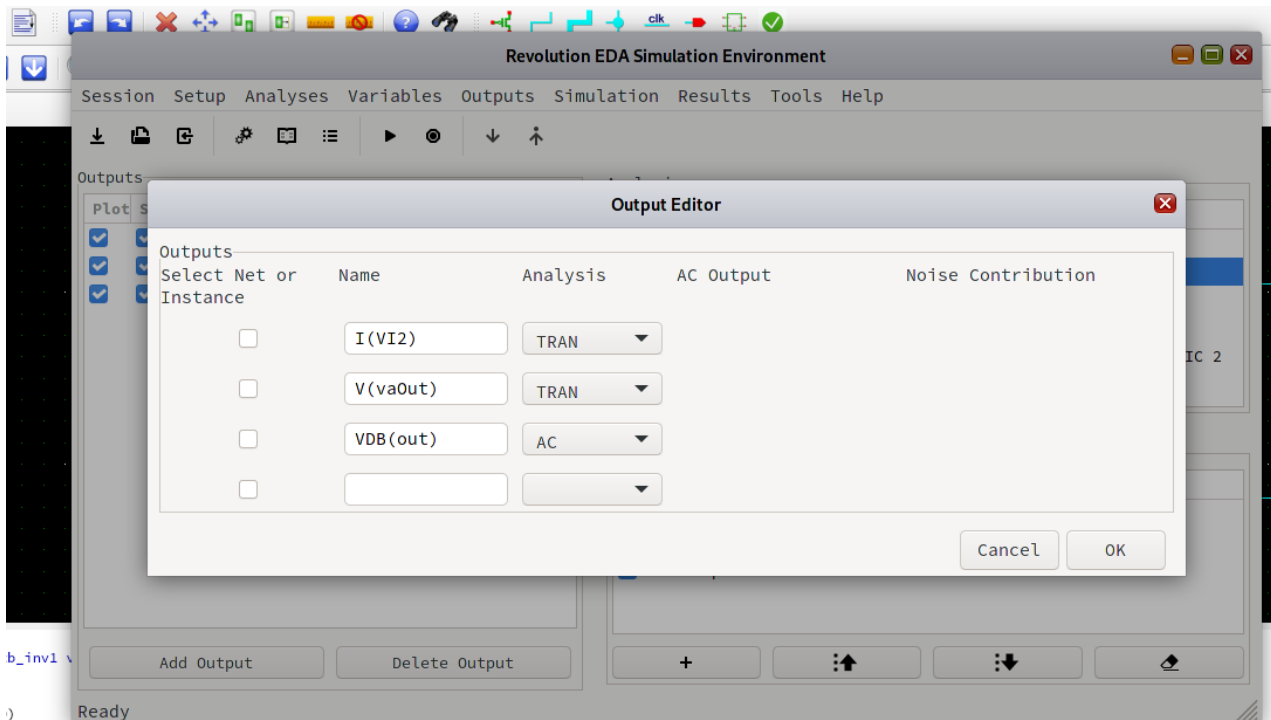
Save Options

Selecting this menu item opens *Save Options* dialogue. In this dialogue, the user can determine which outputs to select. For example, you could choose select no node voltages, selected node voltages, i.e. the node voltages shown on the *Outputs* panel, or all node voltages. The same is true for branch currents and expressions selections.

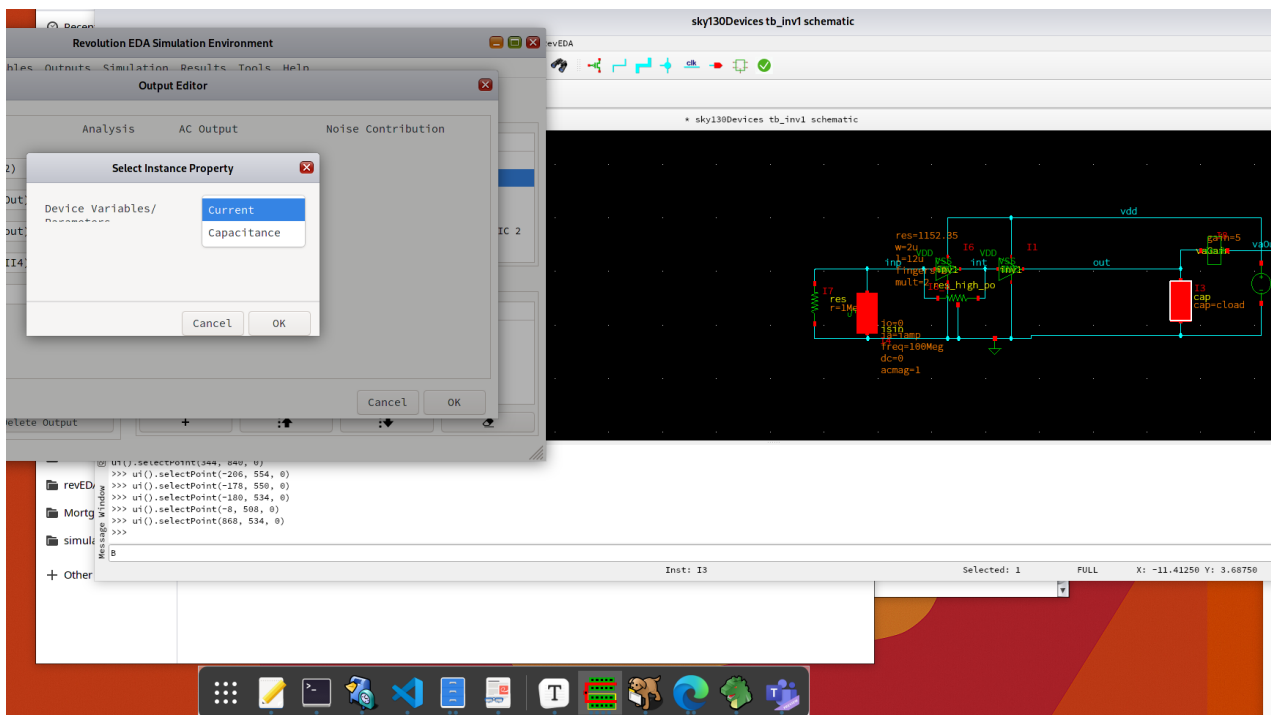


Select On Schematic

This menu item leads to *Outputs Editor* dialogue. Using this dialogue, you could choose node voltages, voltage source branch currents, and some element parameters.



In the *Output Editor*, it is easy to select a node or an instance on the schematic. Just check the checkbox under *Select Net or Instance Column*, then select a node or instance in the schematic and then bring cursor back on GUI window and press `ESC` key. If an instance is selected another dialogue box can pop out, asking which device variable or parameter you would like to add to outputs.



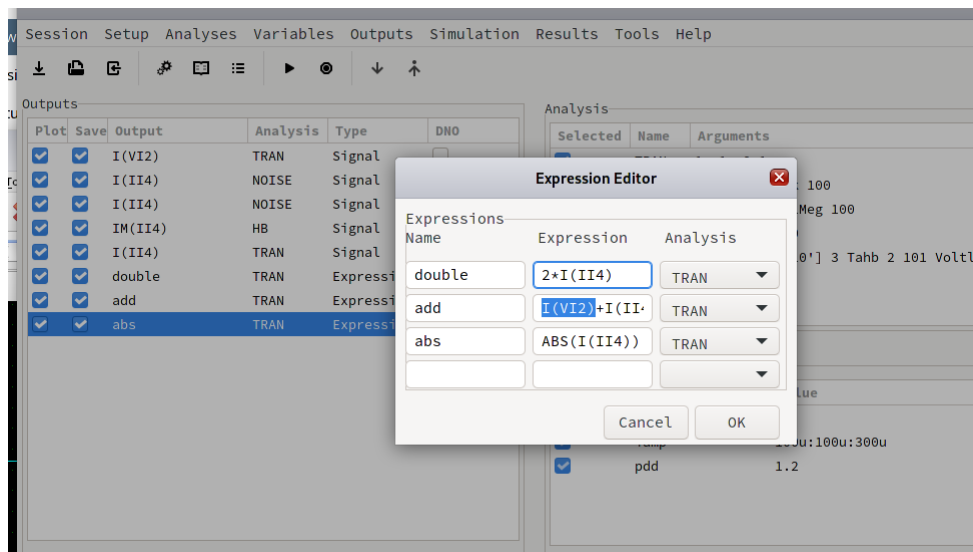
The next step is to select for which analysis this output will be plotted. If an AC or HB analysis is selected, another combo box appears to select whether the output is dB, magnitude or phase. On the other hand, if NOISE analysis is selected, the noise plot could be for input referred noise `DNI(device name)` or output referred noise `DNO(device name)`.

Once all the wanted outputs are configured, press `OK` button. You will see a summary of outputs in the `Outputs` panel. When an analysis is completed, if there are any outputs selected for that analysis, it will be plotted.

Expression Editor

Expression Editor dialogue allows you to enter a formula involving the outputs. For example, two outputs can be added, multiplied, or can be arguments for any Xyce function. In the dialogue below:

1. `I(II4)` is multiplied by 2.
2. `I(VI2)` and `I(II4)` are added.
3. Absolute value of `I(II4)` is calculated using Xyce `ABS` function.

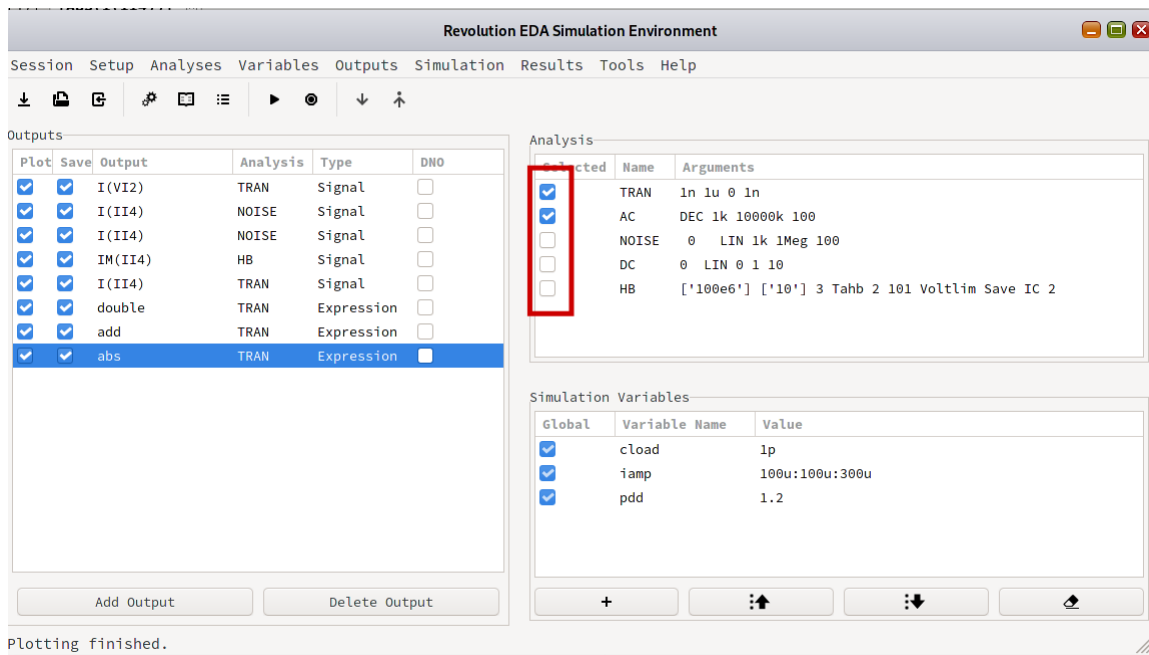


Note that it is not yet possible to use expressions to build more complex expression. In other words, expression names *can not be used* in other expressions. It is expected that will be possible in a later release.

Simulation

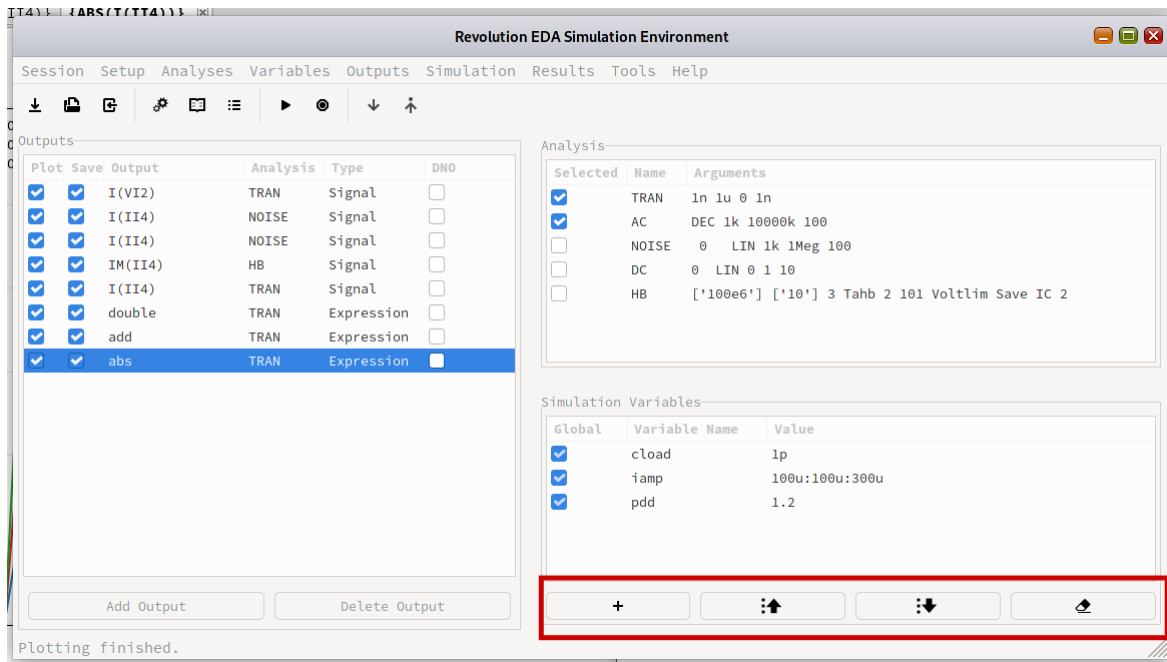
Simulation menu collects functions related to simulation runs as well as convergence aids such as *initial conditions* and *nodesets*.

Before starting simulation run(s), the user has to decide which analyses to run. The analyses can be selected to be run by checking the checkbox at the leftmost column of *Analysis* panel.

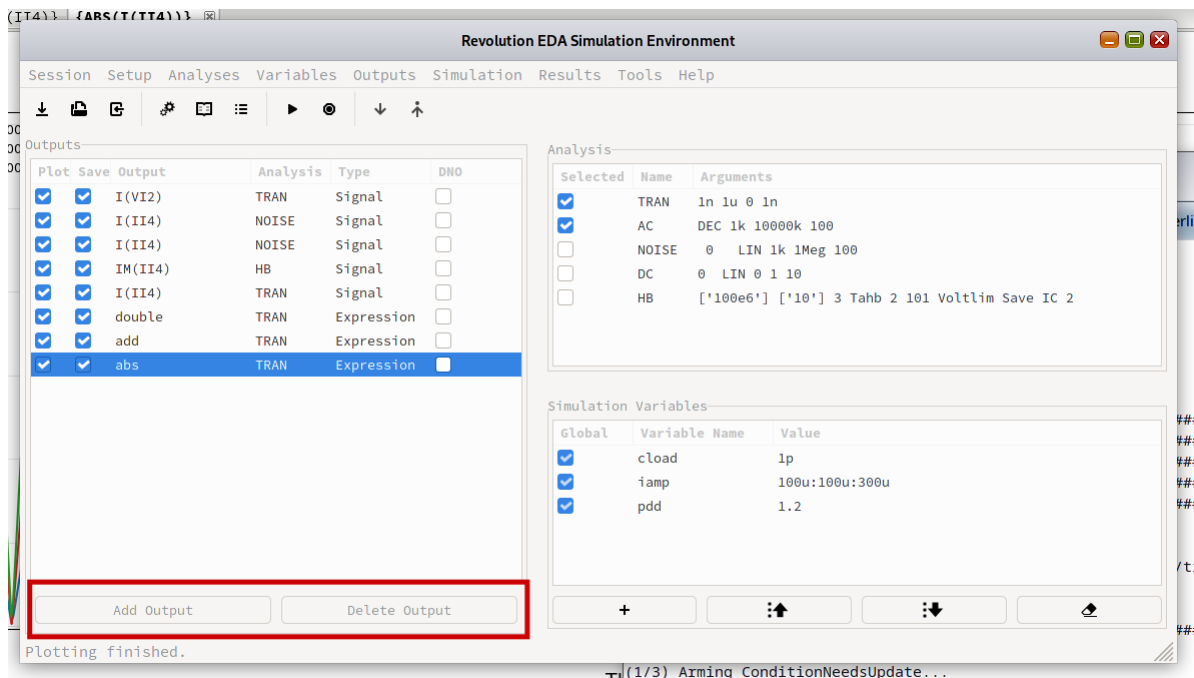


Analysis panel also summarises the most important the analysis parameters. If you would like to change any analysis parameter, you could go through *Analyses->Choose...* menu sequence, or just simply double click on the *Analysis* panel to bring up the *Choose Analysis* dialogue.

The next step is to check whether simulation parameters are set up correctly. *Simulation variables* panel summarises the parameter values, and if they are swept, the sweep method. If you would like to change the order of parameter sweeps, add or delete parameters, one could use the toolbar at the bottom of the panel.



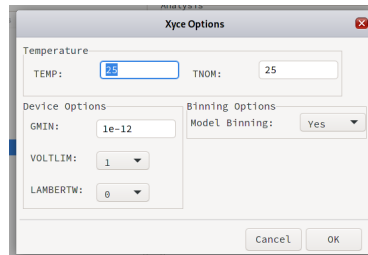
Finally, *Outputs* panel summarises the selected outputs and expression that are to be plotted at the end of each analysis, only if the relevant analyses is selected to be run. For the simulation setup shown above, *I (II4)* will not be plotted as *NOISE* analysis is not selected in the analysis panel. Output panels also have two buttons at the bottom as a shortcut to add and delete outputs.



There are seven menu items under *Simulation* menu:

1. *Netlist and Run*: Select this menu item to netlist the circuit and then run the selected analyses.

2. *Run*: If the circuit is not changed, this menu item can be used to save time to run the simulation(s) without going through netlisting.
3. *Stop*: If the simulation is not converging or you changed your mind, you could use this menu item to try to stop the simulation.
4. *Options*: This menu item has actually three sub-menus, but only one of them, i.e. Analog, is implemented so far. It starts *Xyce Options* dialogue where some important simulation parameters not-specific to any one analysis can be set.

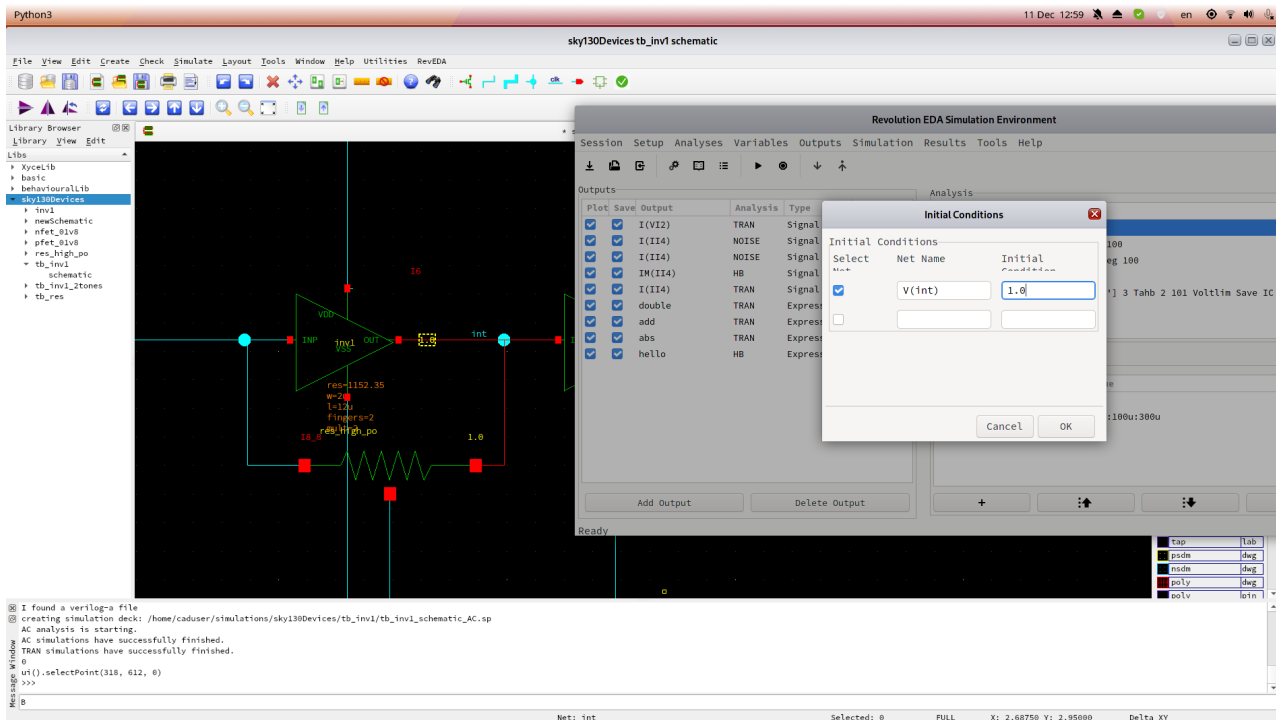


- 1 | 5. **Netlist** submenu has two items:
- 2 | 1. **Create**: The circuit is netlisted and the resulting netlist is displayed in a text editor.
- 3 | 2. *Display*: Existing netlist is displayed in a text editor.

6. *Output Log*: The output log of the the first completed analysis is shown. In a later release, this menu will allow the viewing of all analyses logs.
7. *Convergence Aids*: This submenu has two items:
 1. *Initial Conditions*
 2. *Nodeset*

Initial Conditions

Initial conditions dialogue works very similar to outputs dialogue. Once the *Select Net* checkbox is checked, the user can switch to schematic view and select a node. Switching back to the initial conditions dialogue and pressing **ESC** key carries the select net's name to *Net Name* field. The initial condition voltage can be entered in *Initial Condition* field. When **ENTER** key is pressed, the relevant node is labelled with the initial condition value. Moreover, a new row is added to the dialogue. If you cannot see it, just resize the dialogue window to make that row visible. Note that it is not yet possible to set branch currents as initial conditions in Xyce. Once the dialogue is dismissed by clicking **OK** or **CANCEL** buttons, the initial condition label on the net(s) in the schematic will also disappear.



Nodeset

The *Nodeset Voltages* dialogue works very similarly to initial *Initial Conditions* dialogue. The difference is that it is more like a suggestion to the simulator to nudge it into the correct operation point.

Results

Results menu is where the simulation outputs are handled. Revolution EDA uses *RevPlot* plotting utility to plot the node voltages, branch currents, element values, expressions, and other derived performance parameters. *RevPlot* has a notebook interface and usual zoom/unzoom features. It is based on *wxmplot* library and *numpy*. It is still work-in-progress and thus have a few functions not yet fully implemented.

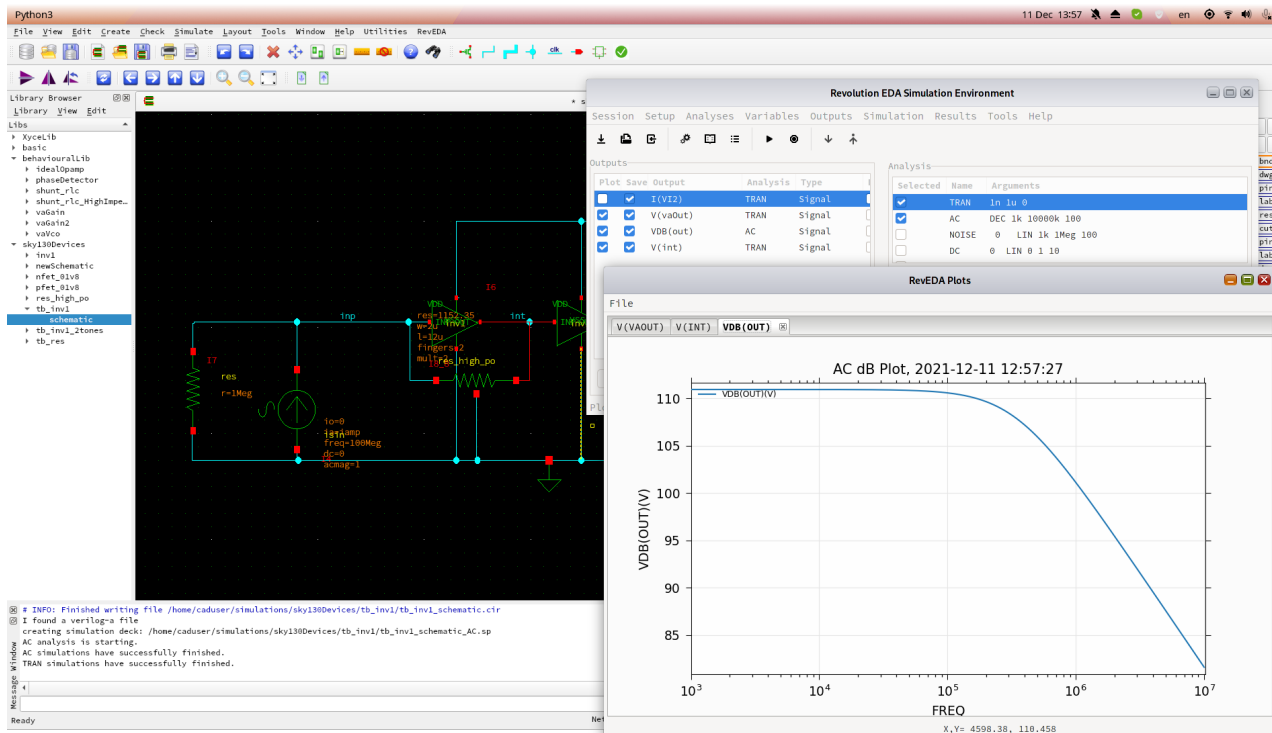
Results menu has three menu items:

1. *Plot Outputs*
2. *Direct Plot*
3. *Annotate*

Plot Outputs

This menu item is used to replot all the selected outputs. For example, the user might have decided not to plot all the outputs on *Outputs* panel. After unselecting relevant checkbox for the output on that panel, she or he can use *Plot Outputs* to replot all the selected outputs.

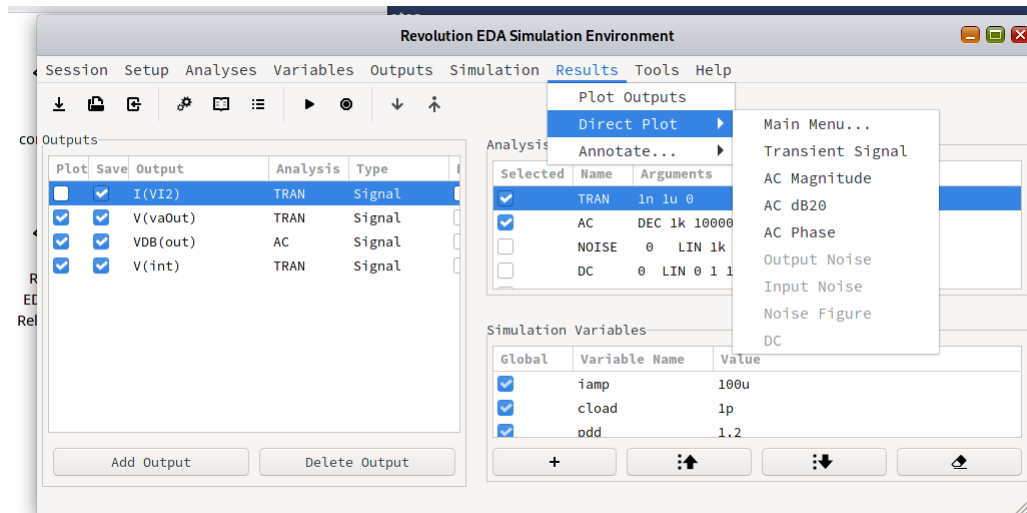
In the Revolution EDA screenshot below, there are four outputs saved for **TRAN** and **AC** analyses: **I(VI2)**, **V(vaOut)**, **VDB(out)** and **V(int)**. However the user wants to draw only the last three of them. By unchecking the checkbox for **Plot** column, he or she can plot only the wanted outputs, in this case **V(vaOut)**, **VDB(out)** and **V(int)**.



Direct Plot

Direct Plot submenu allows the user to plot node voltage or source current values for Transient, Noise, AC and DC analysis directly. If there are no results available an analysis, the corresponding submenu item will be shaded out and can not be selected.

In the screenshot below, only AC and TRAN analyses were done and thus only the results for these two analyses are selectable.



Main Menu

Main menu dialogue is where *heavy-duty* plotting of the simulation results including significant and most-used performance parameters for each analysis type are created.

There are in total six tabs in the *Main Plot Dialog*:

1. TRAN
2. AC
3. NOISE
4. DC
5. HB FD
6. HB TD

HB analysis outputs can be drawn in the frequency domain in *HB FD* tab or in the time domain in *HB TD* tab.

Both single-ended and differential signals are handled with ease in the interface.

TRAN Tab

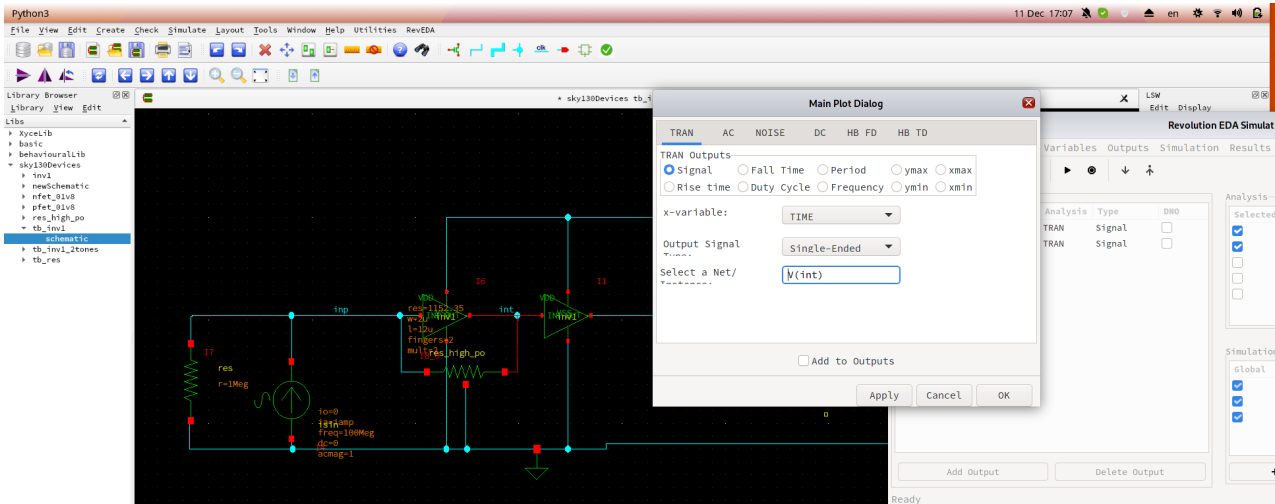
Tran tab is where the outputs and significant performance parameters related to transient signals can be plotted.

TRAN Outputs panel has eight (8) output types. We will cover each one here.

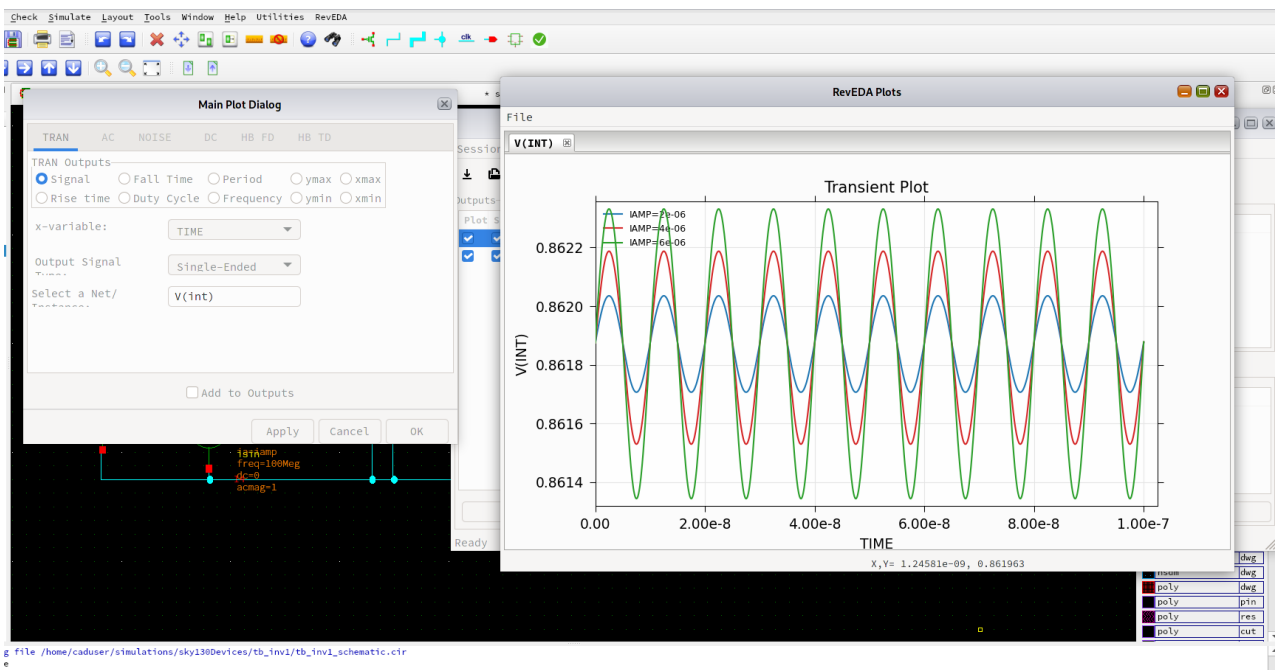
Signal: Signal tab is used to plot node voltages and branch currents against time or any other output saved in the output files. For example, a user might plot the node voltage of *int* node in the schematic against time by

1. Selecting *TIME* as the x-variable,
2. Selecting *Single-ended* as the output signal type,

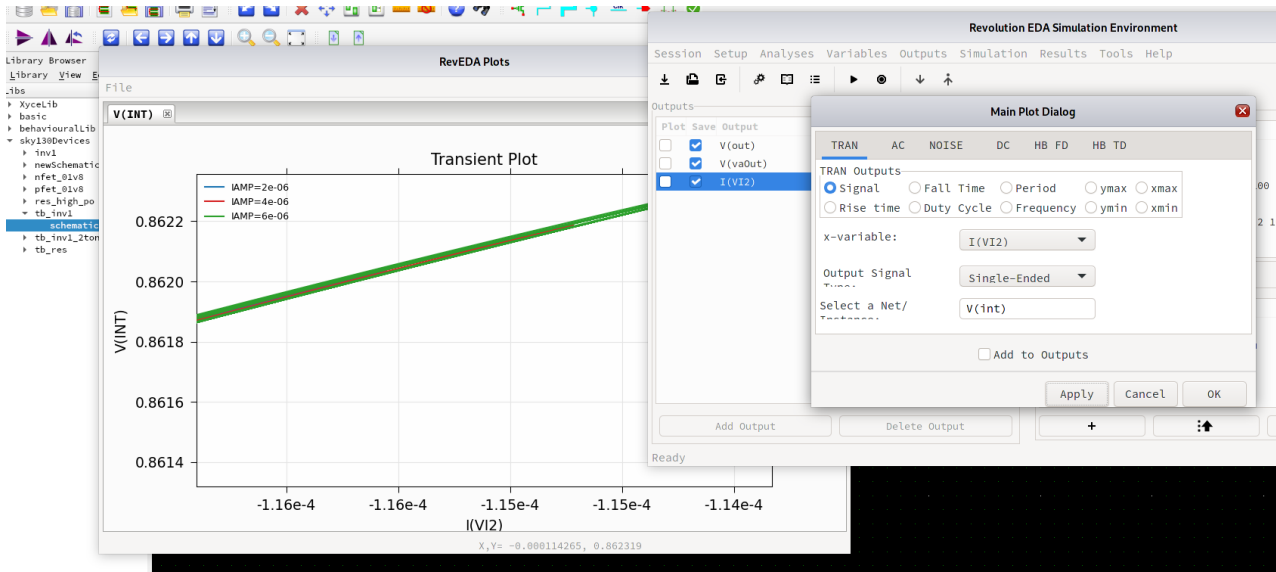
3. First double clicking in the field named “Select a Net/Instance” and selecting a node or source in the schematic and then clicking once again in the output field in the dialogue and pressing `ESC` key.
4. Finally pressing *Apply* button.



There will be a new tab opened in *RevEDA Plots* window with the selected output. If there is a sweep of variables in the plot, the plot for each parameter(s) will be clearly labeled.



By choosing an x-variable other than *TIME*, the behaviour of circuit outputs against each other can be plotted. In the plot below, the voltage of the `int` node is plotted against the supply current.



Rise-Fall Time

They are more relevant for digital signal outputs and can be used to determine both 10-90% and 20-80% rise and fall times for both single-ended and differential signals.

Period-Duty Cycle-Frequency

These functions can be used to determine the period, duty cycle and frequency of any single-ended or differential signal over time.

y_{max}-y_{min}-x_{max}-x_{min}

These functions are to be implemented in a future release of Revolution EDA.