

Chapter 5

CLAMPING CIRCUIT

AIM

To design and implement circuit for clamping waveforms

DESIGN AND CIRCUIT DIAGRAM

Inorder to plot the transient response of clamping circuits use a SINE source whose amplitude, frequency, phase etc can be fixed during simulation. The SINE source is connected across a series connection of a diode and a capacitor and the output is taken across the diode and the GND. The circuit for a clamper circuit is in Figure 5.1 .

PROCEDURE

The steps to plot the transint resonse of a clamper are explained below..

Launch eSim

Launching eSim will take you to the dialog box which asks for the default workspace. Browse the folders and set the wokspace location. It will finally end up in the eSim window.

Create a New Project

The new project is created by clicking the New icon on the menubar. The name of the project is given in the pop up window as 'clamper' for the circuit in Figure 5.1. Follow the steps explaiend below to implement the circuit.

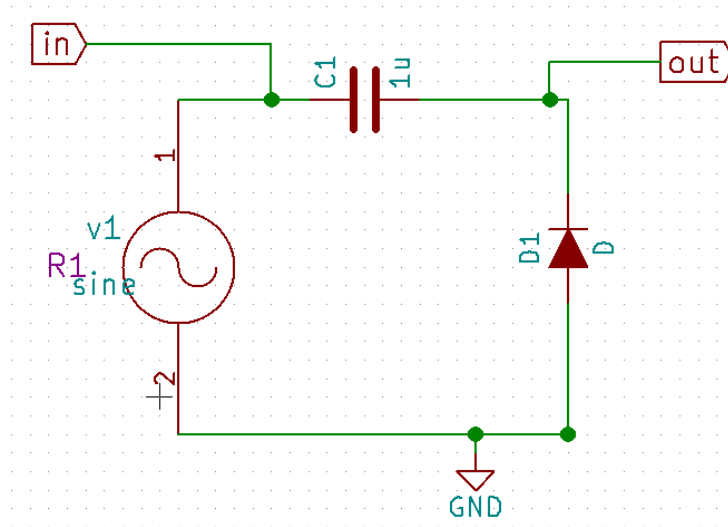


Figure 5.1: Schematic diagram for clamper circuit

Create the Schematic

To create the schematic, click the very first icon of the left toolbar. This will open KiCad Eeschema.

To create a schematic in KiCad, we need to place the required components. After all the required components of the clipper circuit are placed, wiring is done using the Place Wire option. The ‘Place Wire’ and ‘Place Component’ tools are available in the left toolbar. Scroll up and down for zooming in and out.

Placing the Components: Normally all the components available in eSim can be chosen by left mouse click in the grid. The components are listed in different libraries.

- Choose SINE source from eSim_Sources
- Choose C from eSim_Devices
- Choose D from eSim_Devices
- Choose GND from power

Select the capacitor and edit its component value to 1u.

Wire the components to get the circuit. A global label ‘in’ and ‘out’ has been added to identify the node whose voltage will be later recorded and plotted. Global label is added from the right toolbar of Eeschema.

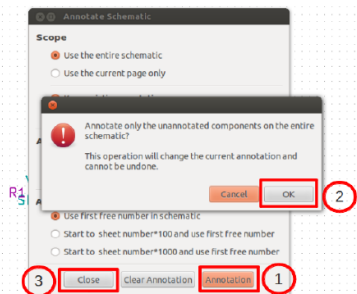


Figure 5.2: Annotation

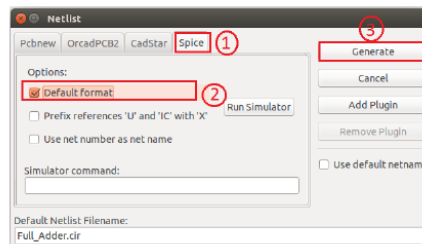


Figure 5.3: Netlist Generation

Annotating the circuit: Once the schematic diagram is completed, annotate it so that the ‘question marks’ associated with the components are converted to meaningful numbers automatically. Choose annotate button from the top toolbar and in the subsequent dialogue boxes appearing click ok and finally close. See Figure 5.2.

Now we have the circuit diagram as shown in Figure 5.1.

Note: If some libraries are found missing, you can add them from the ‘Preferences’ menu by following the procedure:

1. Choose ‘Component Libraries’ from Preferences menu.
2. Click on the Add button on the top right side of the window.
3. Choose the required libraries from ‘user/share/kicad/library’ and click OK button

Create Netlist

To simulate the circuit that has been created in the previous section, we need to generate its netlist. Netlist is a list of components in the schematic along with their connection information. To do so, click on the Generate netlist tool from the top toolbar. Click on spice from the window that opens up. Check the option Default Format. Then click on Generate. Save the netlist. This will be a .cir file. Do not change the directory while saving. See Figure 5.3. Now the netlist is ready to be simulated.

KiCad to Ngspice conversion

To convert KiCad netlist of clipper circuit to NgSpice compatible netlist click on KiCad to Ngspice icon . Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models.

Analysis: Choose analysis type as ‘Transient’. Give the values of time variables as shown in Figure 5.4. Enter the time to be varied from ‘Start time=0 ms’ to ‘Stop time=4ms’ with a ‘Step time=0.01 ms’.

Transient Analysis

Start Time	0	ms
Step Time	.01	ms
Stop Time	4	ms

Convert

Figure 5.4: Choose analysis type as ‘transient’ and enter the values

Source Details: Set the details of ‘sine’ source as shown in Figure 4.6.

- Offset value(volts): 0
- Amplitude(volts): 4
- Frequency(Hz): 1k
- Delay time(Seconds): 0
- Damping factor(1/seconds):0

Analysis Source Details NgSpice Model Device Modeling Subcircuits

Add parameters for sine source v1

Enter offset value (Volts/Amps):	0
Enter amplitude (Volts/Amps):	4
Enter frequency (Hz):	1k
Enter delay time (seconds):	0
Enter damping factor (1/seconds):	0

Convert

Figure 5.5: Enter the parameters of ‘Sine’ source

Ngspice Model: No Ngspice model to be given.

Device Model: Add the diode model available in the eSim library by browsing the folder, `/opt/eSim/src/deviceModelLibrary/Diode/D.lib`

Subcircuits: No subcircuits to be given.

Once these details are provided click on convert button. Now you are ready to see the simulation results.

Simulate

To run Ngspice simulation click the simulation icon in the left tool bar. It will open up two windows - ngspice plotting window and python plotting window. In order to plot the transient response of the clipper you can use either plotting types.

Python plotting: This provides a graphical interface for plotting. We need to plot the value of voltage across the 'SINE' source as well as the diode with respect to time. We have already labeled these nodes as **in** and **out** respectively. The nodes will be listed on the GUI. Choose 'in' and 'out' and click on 'plot' button. See Figure 5.6.

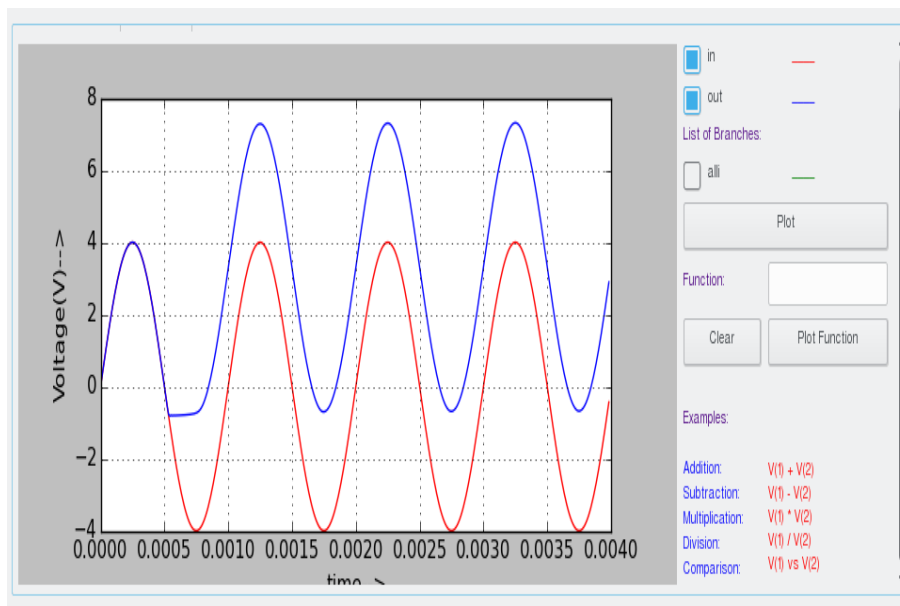


Figure 5.6: The transient response of the clamper on python plotting window

Ngspice plotting: . Time of simulation has already been set in the previous step. Use the commands in ngspice plotting window for obtaining the required plots.

```
plot v(out), v(in)
```

This would plot the transient response of input and output of the clamper.

RESULT

The circuit for plotting the transient analysis of clamper was implemented and simulated.