

Expt. No. 2 Simulation of Fully controlled rectifier on FreeEDA

Aim: To perform transient analysis of fully controlled rectifier on FreeEDA.

Components required with references :

1. Sine wave source (sine)
2. Diodes (diode)
3. Resistor (R) of 100Ω
4. SCR (scr)
5. Voltmeters (vplot1 and vplot)
6. Ground (gnd)

Procedure:

1. Create the schematic of the fully controlled rectifier as shown in Fig. 2.1.
2. Change the reference of the SCR from U to X.
3. Annotate the schematic.
4. Test Electric rules.
5. Generate the netlist.
6. Insert analysis for transient analysis from 0 to 40 ms with a step time of $40\ \mu\text{s}$.
7. Import the subcircuit of SCR.
8. Edit model for diode (set $b_v=1800$ and $I_s=2.2\text{E}-15$).
9. Convert KiCad netlist to Ngspice netlist.
10. Simulate the Ngspice netlist using Ngspice simulator.

Conclusion : Transient analysis of full wave rectifier is performed using FreeEDA.

FreeEDA schematic of fully controlled rectifier :

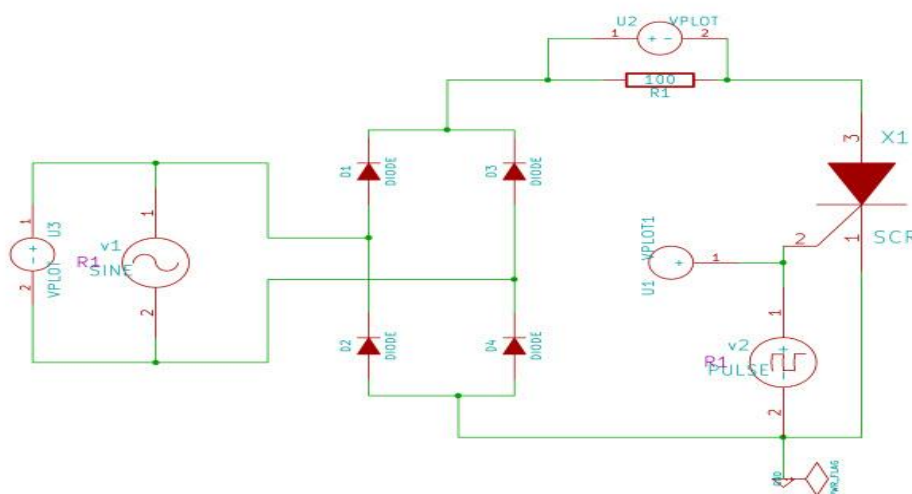


Fig. 2.1 FreeEDA schematic of full wave rectifier.

Parameters of Input :

Parameters of sine wave :

Offset : 0
Amplitude : 100
Frequency : 50
Delay : 0
Damping Factor : 0

Parameters of pulse :

Initial value : 0
Pulsed value : 5
Delay : 6m
Rise time : 0
Fall time : 0
Pulse width : 2m
Pulse period : 10m

Simulation Results :

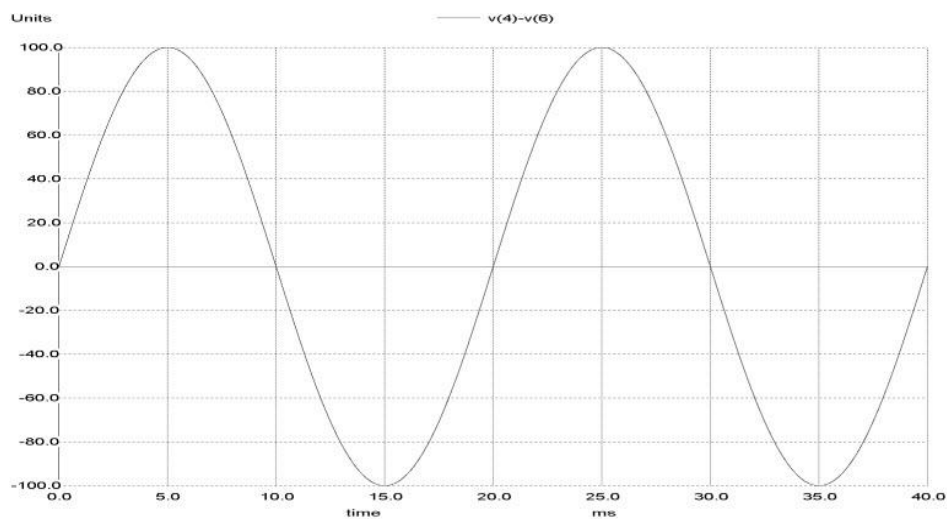


Fig. 2.2 Input waveform

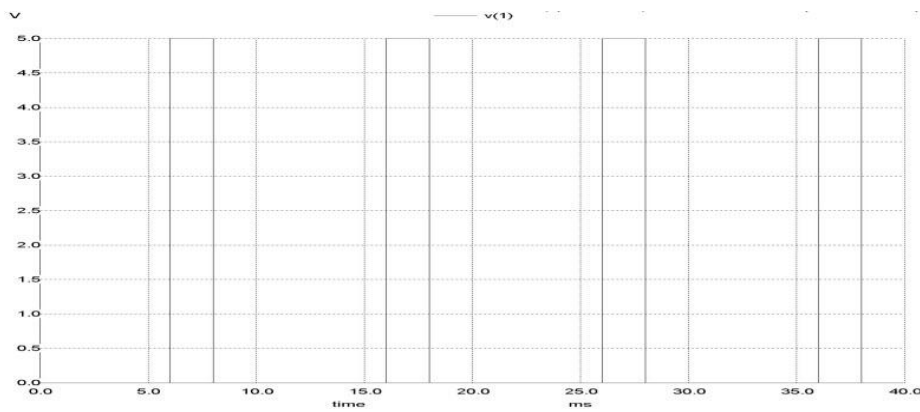


Fig. 2.3 Pulse waveform for triggering SCR

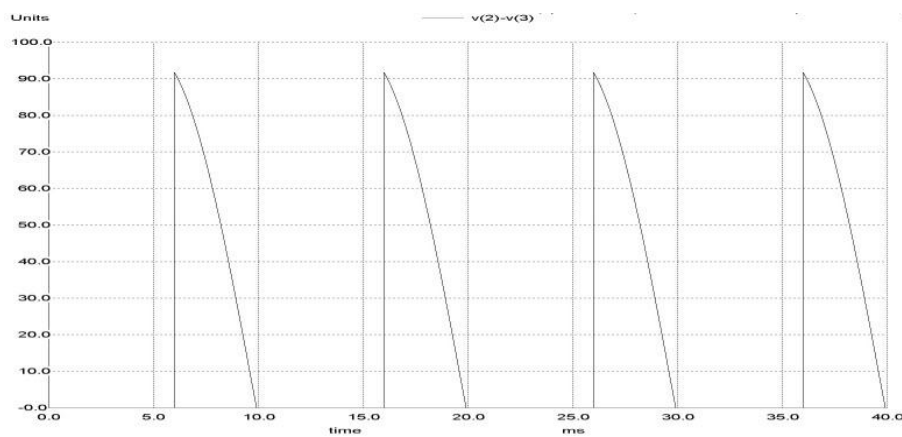


Fig. 2.4 Simulated output of fully controlled rectifier.