

Ex. No.: 5

Date:

Realization of single-phase rectifier circuit

Aim:

To Design Half wave and Full wave Rectifier Circuit in Pspice simulation.

Software /Tool required:

ORCAD / PSpice simulator -> Diode Library - D1N4002/4007,

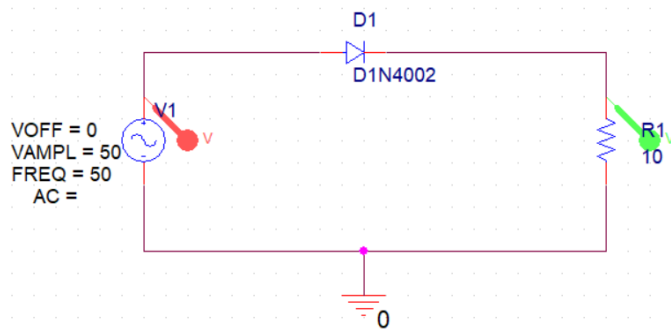
Source Library – Vsin & Ground (GND) – 0(zero) Analog Library – R

Simulation Settings: Analysis Type - Time Domain

Run to time: 40ms (for 2 cycles)

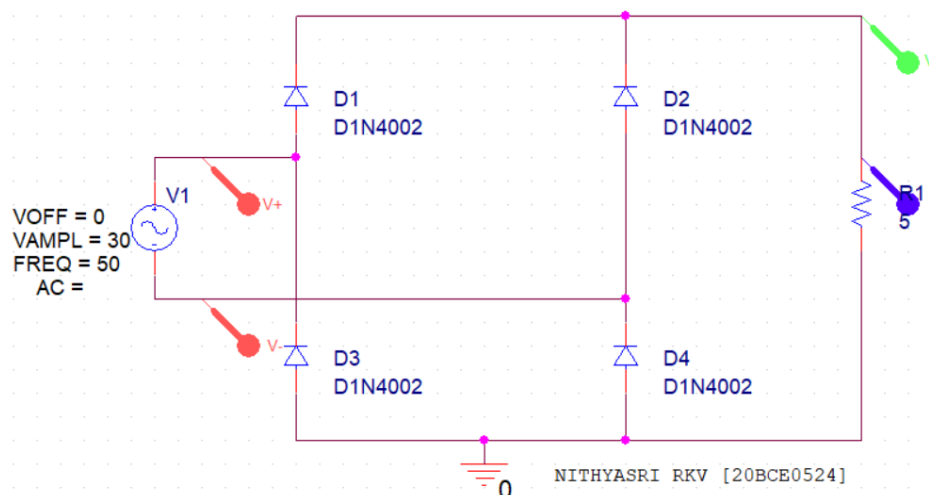
Circuit Diagram:

Single phase Half – wave Rectifier



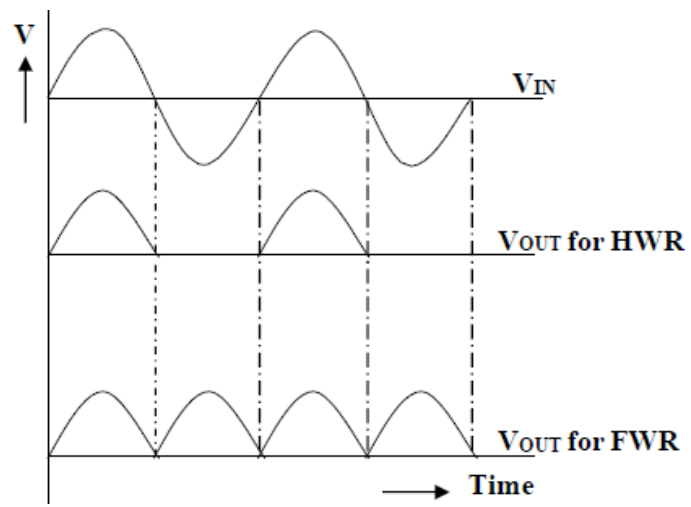
NITHYASRI RRV [20BCE0524]

Single phase Full – wave Rectifier



NITHYASRI RRV [20BCE0524]

Model Graph:



Procedure

1. Create the given circuit diagram in new project file using the general procedure.
2. Create New Simulation Profile and select Time Domain (Transient).
3. Give the run time for two cycles, (i.e. 0.04 seconds for 50Hz frequency for two cycles) and press OK.
4. Select the voltage differential marker and connect across the input voltage source and load resistor.
5. Run the simulation and observe the input and output signal variation.

Result: