Fun with Rectifiers!

**Step 1 (Creating the schematic):**

Double-click on the “Capture-Lite icon” on the desktop to launch PSpice.

Goto **File → New → project**

- Under Name type “Your Name, PSpice lab#2”
- Make sure the ‘Analog or Mixed A/D’ box is checked
- Make sure it saves in the right directory (i.e U:\EE 186\ PSpice labs)
- Click ok

Then, make sure the “Create a blank project” box is checked and Click ok

The “dotted” page will be your schematic page and here you will be able to draw your circuit.

Your next step is to Goto **Place → Part**

- Under libraries choose ‘EVAL’ (Note: if the libraries are not there you have to goto “Add Library” and add all the libraries from the PSpice subfolder as shown in the window on the right)

- Under part type ‘D1N4002’ you should see a diode as shown below.
- Click ok
Place **two** diodes as shown below in the diagram. You can right click and **end mode** once you are done placing the diodes.

Place one resistor (R) as shown below in the diagram.

Now goto **Place → Part again**
- Under libraries choose ‘**SOURCE**’
- Under part type ‘**VSIN**’ and you should see an AC voltage source
- Click ok

Place the voltage source as shown in the diagram Right click and **end mode** again.

Now choose the wiring option on the toolbar located on the right hand side of the screen and connect all the 3 components in series as shown in the diagram below. (**Note:** you could also select the wiring option from **Place → Wire** or by hitting Shift+W)
Right Click and “end wire”

Next:-
Goto  
**Place → Ground**

- Add library ‘**SOURCE**’
- Then choose the ‘0’ ground which looks like this
- Connect it to the lower part of the circuit
Note: The last step of drawing the circuit is to add the circuit ground, which must be node numbered (0). If you don’t see this part, you will need to click the “Add Library” button and add the Source library from the PSpice subdirectory) this is a very important step and easy to forget, but your circuit cannot be simulated by PSpice unless it contains a ground at node 0!

For easier analysis place two voltage markers (You can find this by PSpice Markers → Voltage level or by clicking on this icon). One on the AC source (input) and the other on the output. Your output graph will only have these two plots as this is an AC analysis. The markers are shown in blue and yellow on the figure below.

![Figure 1.1](image.png)
**Step 2 (Specifying the values for the different components):**

Here are the values (Please look at lab#1 to understand how this is done):

**AC Source:**
Double click on the “VOFF” and specify the value to 0.
Double click on the “VAMPL” and specify the value to 50V.
Double click on the “FREQ” and specify the value to 60Hz.

At this point your schematic should look like the above figure.

**Step 3 (Simulation):**

Goto  *PSpice* → *New simulation profile* 

Under Name type “*Your Name, PSpice lab#2*”

Then for the *Simulation* settings
- Analysis type: choose ‘**Time Domain (transient)**’ as shown below.
- Set the run time (TSTOP) to 0.05 seconds and start saving data from 0s
- Then click on the Options tab → Output File→ uncheck the “page break and banners for each section” box (NOPAGE)
- Then Click ok
We have completed the simulation settings and now we need to run the simulation. You can Run the Simulation in different ways

- By hitting F11.
- Or By going to PSpice → Run.
- Or by hitting the Blue triangle Button on the top toolbar.

**Step 4 (Analysis and Printing):**

You should be able to look at the Output file (An example is shown below), Print it out in landscape format. Then analyze it.

On the schematic hit the “V” button on the upper bar and you will be able to see the voltages at the different nodes.

Also print the schematic in Landscape format and all the values should be clearly seen. Please scroll down and double-click on the “<title>” and type in Your Name, PSpice lab#2.

The circuit we built above is a half wave rectifier. It only allows the positive cycle of the AC voltage to pass through.
Please feel free to ask any questions at this point. If you don’t have any, please move on.

Repeat the above step and build and analyze the following circuit:

Discuss the similarities and differences. What is happening in these circuits?

**Complete the following exercises and report on it. We will require a printout of the graph and schematic for each new circuit.**

1) Try only having the negative half of the AC cycle pass through. For this you will have to rotate the two diodes by 180 degrees and simulate it.

2) Now let’s see how the full wave rectification is done. For this we will need two more diodes and a load resistor as connected below.

This should give you a full wave rectified signal, which allows both the positive and negative cycles to flow across the resistor.
3) Let's try seeing a more constant DC voltage across the resistor. This is done by placing a capacitor of 100uF across the resistor and you will see a nice DC voltage but with a little ripple in it which is removed by a voltage regulator.

Simulate the above three circuits and print out your graph and the schematic. Please make sure you schematic and graphs are printed in the Landscape format.

**Summary:**

So by the end of this lab you should have built the following in PSpice

1) A half wave rectifier.
2) A negative half wave rectifier. (flipping the two diodes)
3) A full wave rectifier (the sum of the previous two circuits).
4) A rippled full wave rectified signal. (adding a resistor and capacitor)

Now you can see how the rectifier converts AC voltage to DC voltage.

Please feel free to ask any questions on the working of these circuits.