CECS 311 - LAB 1 – Part 2
Rectifier Diodes

NAME: POSSIBLE POINTS: 10

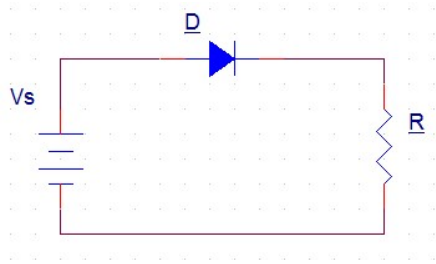STUDENT ID:

COURSE DATE & TIME:

Use LTSpice to model the following circuit and answer the questions.

For calculations assume:

        Germanium Diodes have a typical Forward Voltage - V_F = 0.3v

        Silicon Diodes have a typical V_F = 0.7v

        LEDs and other specialized diodes have V_F = 1.2 to 3.3v and even greater sometimes

1a) V_S = 5v, D is a 1n4001, R = 1k

How is the diode biased?:

Calculate V_D (show equation if it exists):

Calculate V_R (show equation if it exists):

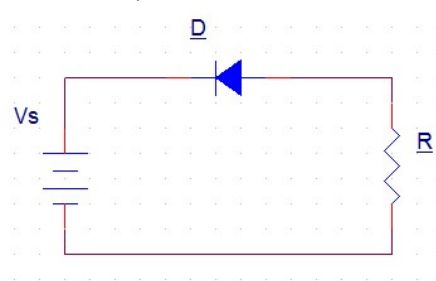Measure V_D :

Measure V_R :

Does $V_D = V_F$, Why?

Include a Screen Capture of your LTSpice model with both measured values shown on the graph

1b) V_S = 5v, D is a 1n4001, R = 1k

How is the diode biased?:

Redraw the Schematic by hand using the 1st approximation (i.e. Ideal Diode V_F = 0v)

Calculate V_D (show equation if it exists):

Calculate V_R (show equation if it exists):

LTSpice Measurement of V_D :
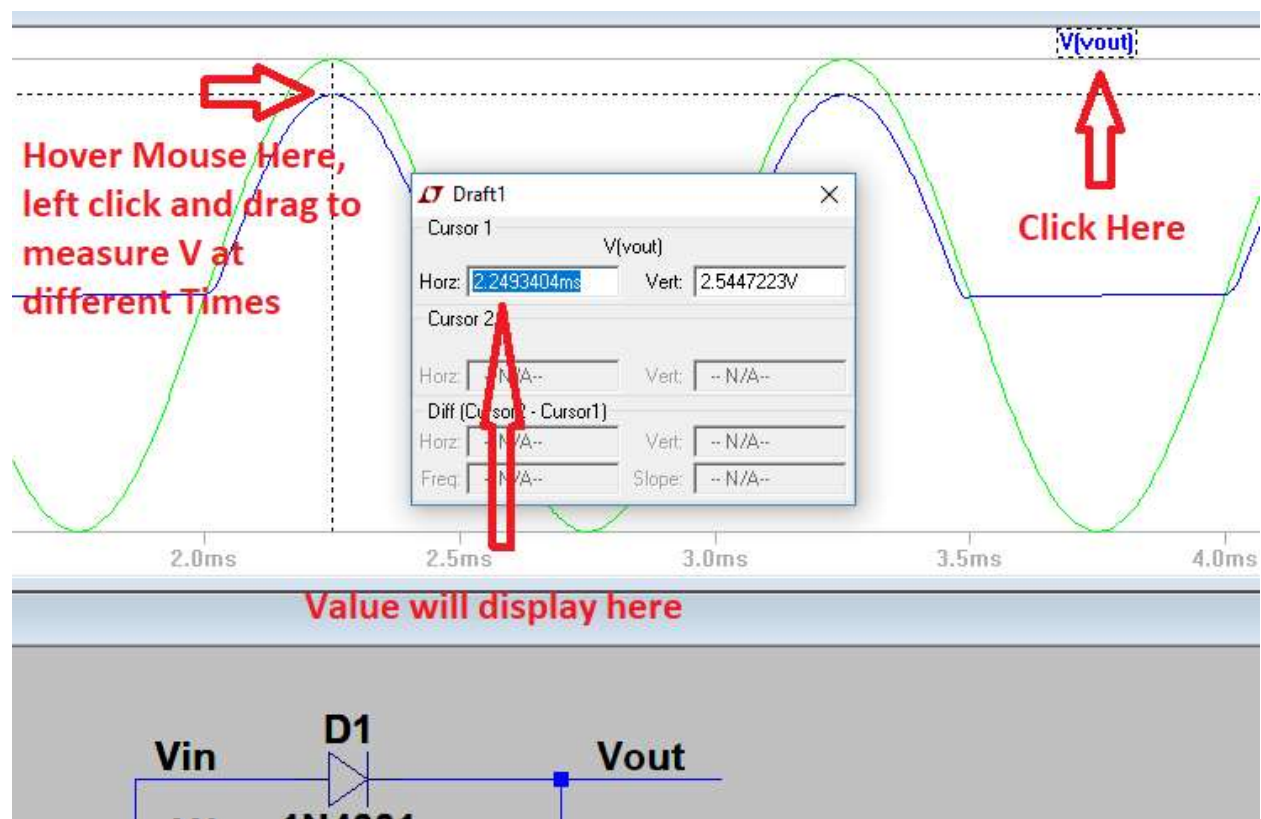
LTSpice Measurement of V_R :

Does $V_D = V_F$, Why?

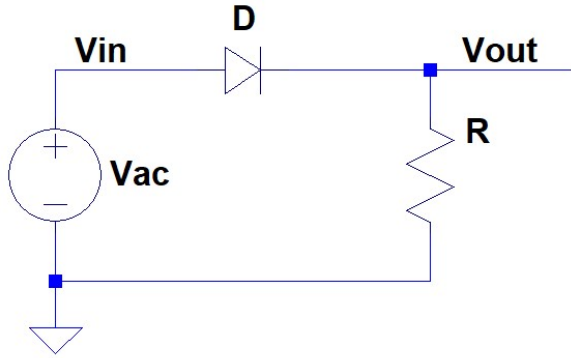Include a Screen Capture of your LTSpice model with both measured values shown on the graph

LTSpice Screenshot and Write-Up Requirements:

- For the following circuits, include a Screen Capture of your LTSpice model with all measured max and min values annotated on the waveform. You may edit the image and add them or draw them in with pencil after printing, it's your choice. In other words, you should take a screenshot of the waveform, then use the built in LTSpice waveform measurement tools to measure the min and max voltage for V_{AC}, V_R, V_D, Vout. In the following circuits, Vout will be equal to V_D or V_R, In these instances you can just draw a line and label the voltage as: max Vout = V_R = 3v, or min Vout = V_R = 3v.

- In LTSpice be sure to label the nets with Vin and Vout.

- In LTSpice, you can take measurements by clicking on the net name in the waveform. You can then hover the mouse over the intersection/crosshairs, left click and drag so you can measure the values at different time points.
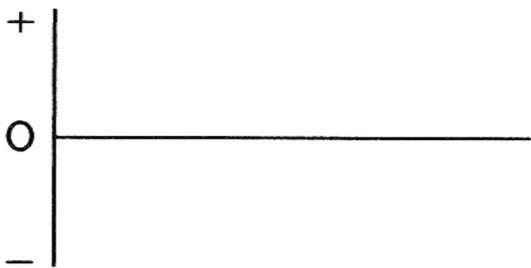
2a) Series Diode Clipper: Vac = 6V$_{pk-pk}$, Fac = 1Khz, D = 1n4001, R = 27K

D

Vin   **Vout**

R

Vac

Draw the waveform, label and include V$_{AC}$ and Vout/V$_R$

What is V$_{AC}$ in pk-pk voltage?
What is V$_R$ in pk-pk voltage?
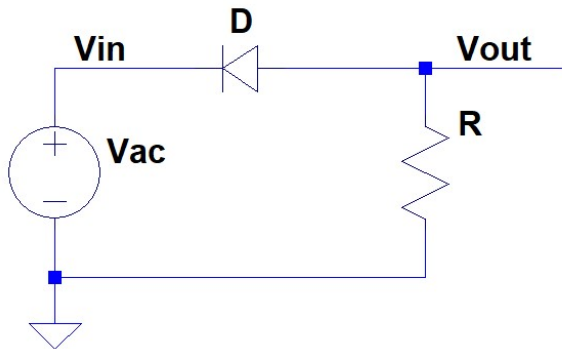What is V$_D$ in pk-pk voltage?

Is this negative or positive clipping?

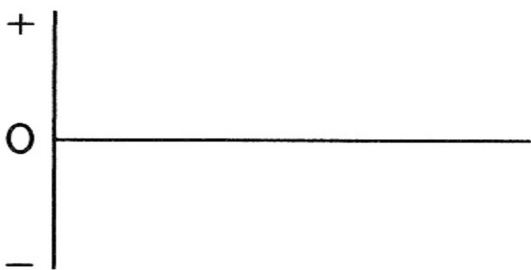LTSpice Measurements, V$_{AC\ Pk-Pk}$:   V$_{R\ Pk-Pk}$:   V$_{D\ Pk-Pk}$:
Include a Screen Capture of your LTSpice model with all measured values shown on the graph

2b) Series Diode Clipper: V$_{ac}$ = 6V$_{pk-pk}$, D = 1n4001, R = 27K

D

Vin   **Vout**

R

Vac

Draw the waveform, label and include V$_{AC}$ and Vout/V$_R$

What is V$_{AC}$ in pk-pk voltage?
What is V$_R$ in pk-pk voltage?
What is V$_D$ in pk-pk voltage?

Is this negative or positive clipping?
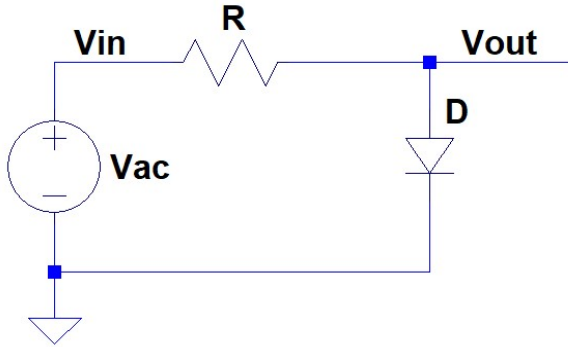
LTSpice Measurements, V$_{AC\ Pk-Pk}$:   V$_{R\ Pk-Pk}$:   V$_{D\ Pk-Pk}$:
Include a Screen Capture of your LTSpice model with all measured values shown on the graph

3a) Shunt Diode Clipper: $V_{ac} = 6V_{pk-pk}$, D = 1n4001, R = 27K

Draw the waveform, label and include V_{AC} and $Vout/V_D$

What is V_{AC} in pk-pk voltage?
What is V_R in pk-pk voltage?
What is V_D in pk-pk voltage?

Is this negative or positive clipping?

LTSpice Measurements, $V_{AC\ Pk-Pk}$: $V_{R\ Pk-Pk}$: $V_{D\ Pk-Pk}$:
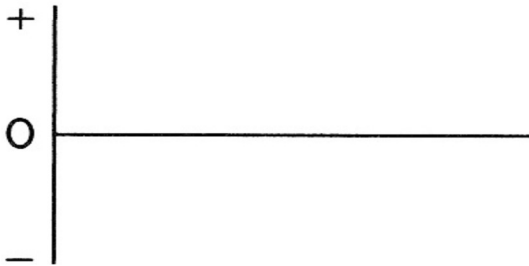Include a Screen Capture of your LTSpice model with all measured values shown on the graph

3b) Shunt Diode Clipper: $V_{ac} = 6V_{pk-pk}$, D = 1n4001, R = 27K

Draw the waveform, label and include V_{AC} and $Vout/V_D$

What is V_{AC} in pk-pk voltage?
What is V_R in pk-pk voltage?
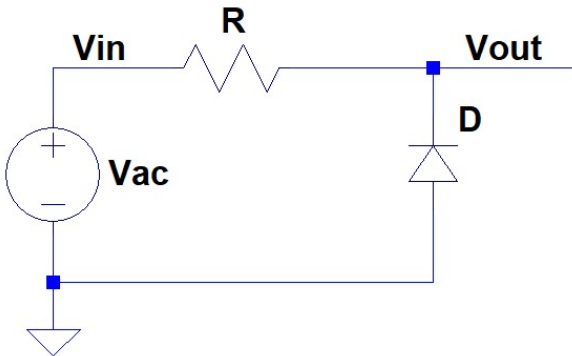What is V_D in pk-pk voltage?

Is this negative or positive clipping?

LTSpice Measurements, $V_{AC\ Pk-Pk}$: $V_{R\ Pk-Pk}$: $V_{D\ Pk-Pk}$:
Include a Screen Capture of your LTSpice model with all measured values shown on the graph.