

**CECS 211 - LAB 10 –  
Diodes in AC**

NAME:

POSSIBLE POINTS: 10

STUDENT ID:

COURSE DATE & TIME:

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

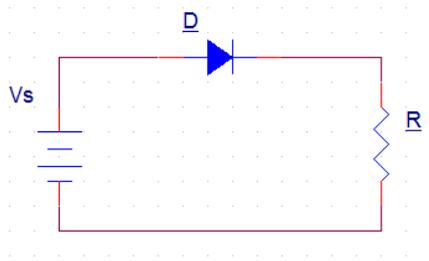
For calculations assume:

Germanium Diodes have a typical Forward Voltage -  $V_F = 0.3\text{v}$

Silicon Diodes have a typical  $V_F = 0.7\text{v}$

LEDs and other specialized diodes have  $V_F = 1.2$  to  $3.3\text{v}$  and even greater sometimes

1a)  $V_S = 5\text{v}$ , D is a 1n4001,  $R = 1\text{k}$



How is the diode biased?:

Calculate  $V_D$  (show equation if it exists):

Calculate  $V_R$  (show equation if it exists):

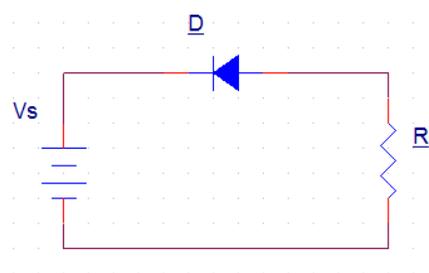
Measure  $V_D$  :

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)  $V_S = 5\text{v}$ , D is a 1n4001,  $R = 1\text{k}$



How is the diode biased?:

Redraw the Schematic by hand using the 1<sup>st</sup> approximation (i.e. Ideal Diode  $V_F = 0\text{v}$ )

Calculate  $V_D$  (show equation if it exists):

Calculate  $V_R$  (show equation if it exists):

LTSpice Measurement of  $V_D$  :

LTSpice Measurement of  $V_R$  :

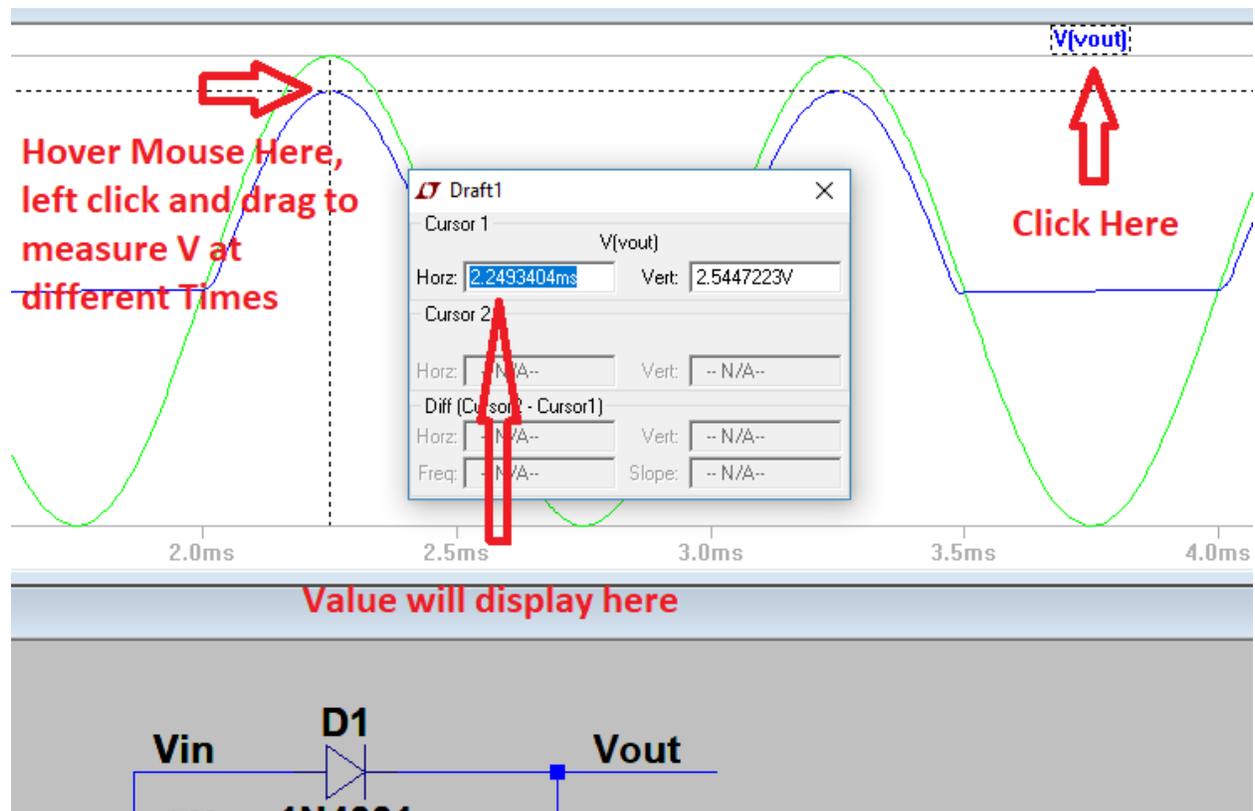
Does  $V_D = V_F$ , Why?

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

## CECS 211 - LAB 10 – Diodes in AC

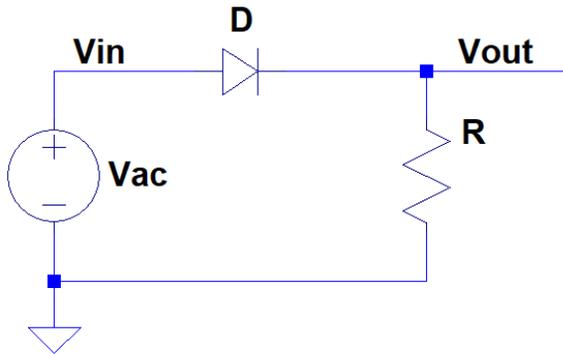
### LTSpice Screenshot and Write-Up Requirements:

- 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  $V_{AC}$ ,  $V_R$ ,  $V_D$ ,  $V_{out}$ . In the following circuits,  $V_{out}$  will be equal to  $V_D$  or  $V_R$ . In these instances you can just draw a line and label the voltage as: max  $V_{out} = V_R = 3v$ , or min  $V_{out} = V_R = 3v$ .
- In LTSpice be sure to label the nets with  $V_{in}$  and  $V_{out}$ .
- 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.

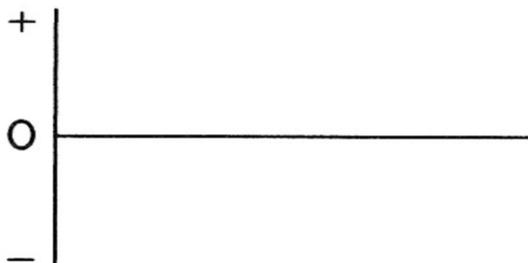


**CECS 211 - LAB 10 –  
Diodes in AC**

2a) Series Diode Clipper:  $V_{ac} = 6V_{pk-pk}$ ,  $F_{ac} = 1Khz$ ,  $D = 1n4001$ ,  $R = 27K$



Draw the waveform, label and include  $V_{AC}$  and  $V_{out}/V_R$



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?

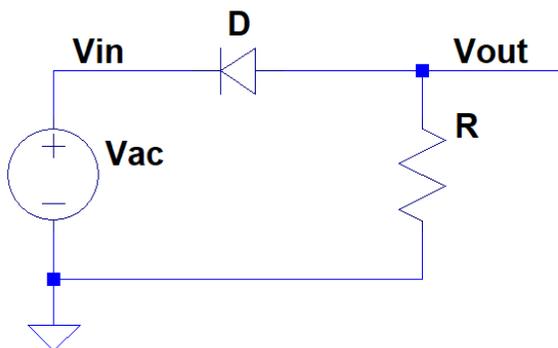
LTSpice Measurements,  $V_{AC\ pk-pk}$  :

$V_{R\ pk-pk}$  :

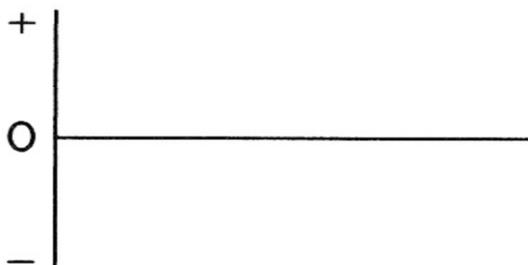
$V_{D\ pk-pk}$  :

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

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



Draw the waveform, label and include  $V_{AC}$  and  $V_{out}/V_R$



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,  $V_{AC\ pk-pk}$  :

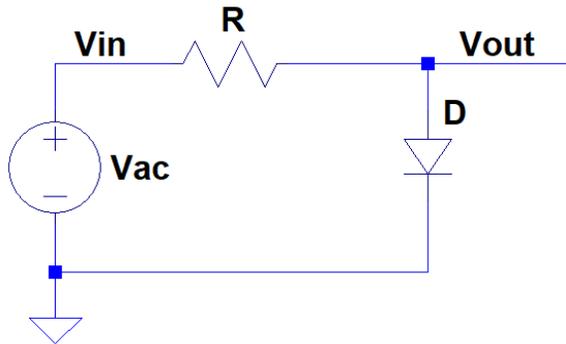
$V_{R\ pk-pk}$  :

$V_{D\ pk-pk}$  :

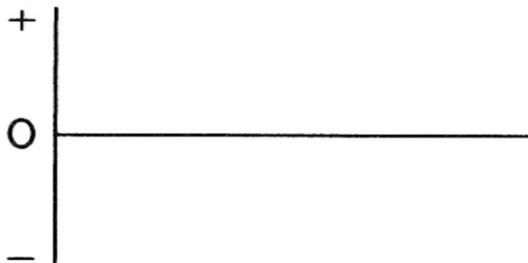
Include a Screen Capture of your LTSpice model with all measured values shown on the graph

**CECS 211 - LAB 10 –  
Diodes in AC**

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



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



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?

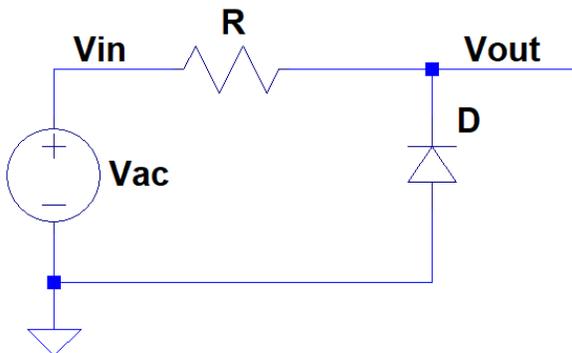
LTSpice Measurements,  $V_{AC\ pk-pk}$  :

$V_R\ pk-pk$  :

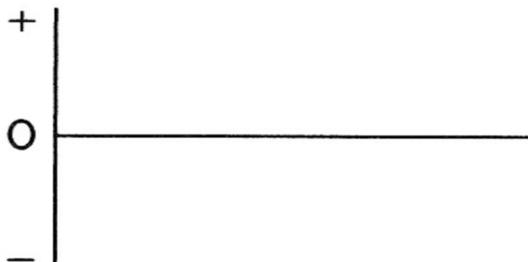
$V_D\ pk-pk$  :

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  $V_{out}/V_D$



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.