CECS 311 - LAB 1 – Diodes
Diodes: LEDs and Clipper Circuit Simulation

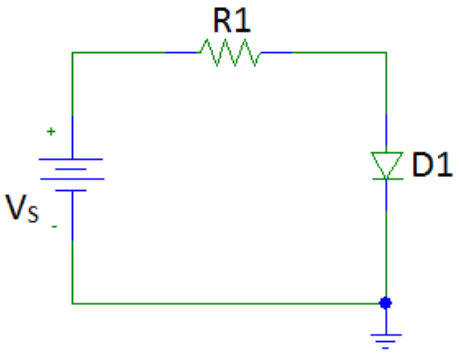
NAME: POSSIBLE POINTS: 10

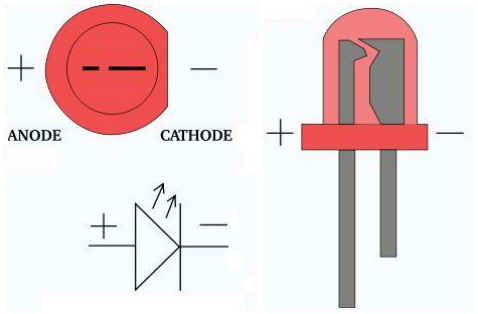
STUDENT ID:

COURSE DATE & TIME:

Part 1 – LEDs (Light Emitting Diodes)

Use the Benchtop Variable Power Supplies and the provided LED to connect the following circuit.





Turn on the Benchtop Variable Power Supply using the smallest voltage that the PSU (Power Supply Unit can produce. Start at 0v and try to go up to 5v. Fill out the following table as you increase your voltage. Use a Multimeter to double check the voltage being displayed on the bench PSU.

	From Simulation			From Lab Bench			
V_S	I_T	V_D	V_{R1}	I_T	V_D	V_{R1}	Apparent Brightness of LED
1.3v							
1.50v							
1.75v							
2.00v							
2.50v							
3.00v							
5.00v							

CECS 311 - LAB 1 – Diodes
Diodes: LEDs and Clipper Circuit Simulation

For the following questions, use the data gathered above and the datasheet found on the course website.

1.) LED's are Current or Voltage driven devices?

2.) Why did the LED get brighter when the voltage was increased?

3.) Write a simple one sentence description of what it means for an LED to be Current or Voltage driven.

4.) What is Vf according to the datasheet?

5.) What was Vf according to your test?

6.) What is the max current that the LED should ever have through it according to the datasheet?

7.) Try adjusting the voltage until you get 30ma through the LED. Look directly at the LED from the top of it, notice that not only is it really bright but that the shape is designed as a lense to focus the light. Now adjust the current back down to what you think is a usable brightness for the LED (this is 100% subjective). What is the current at the useable brightness?

8.) (True/False) It's ok to sometimes use an LED without a resistor. If you answered True, please explain…

CECS 311 - LAB 1 – Diodes
Diodes: LEDs and Clipper Circuit Simulation

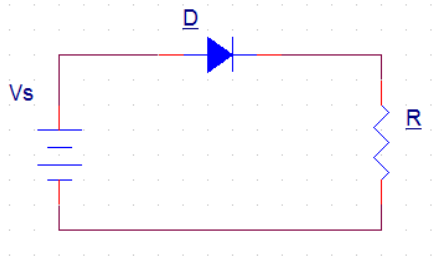
Part 2 – Diode Clipper Circuits

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

For calculations assume:

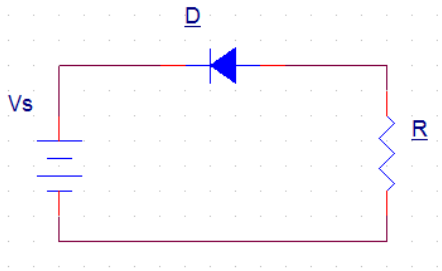
Germanium Diodes have a typical Forward Voltage - $V_F = 0.3v$
Silicon Diodes have a typical $V_F = 0.7v$
LEDs and other specialized diodes have $V_F = 1.2$ to $3.3v$ and even greater sometimes

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

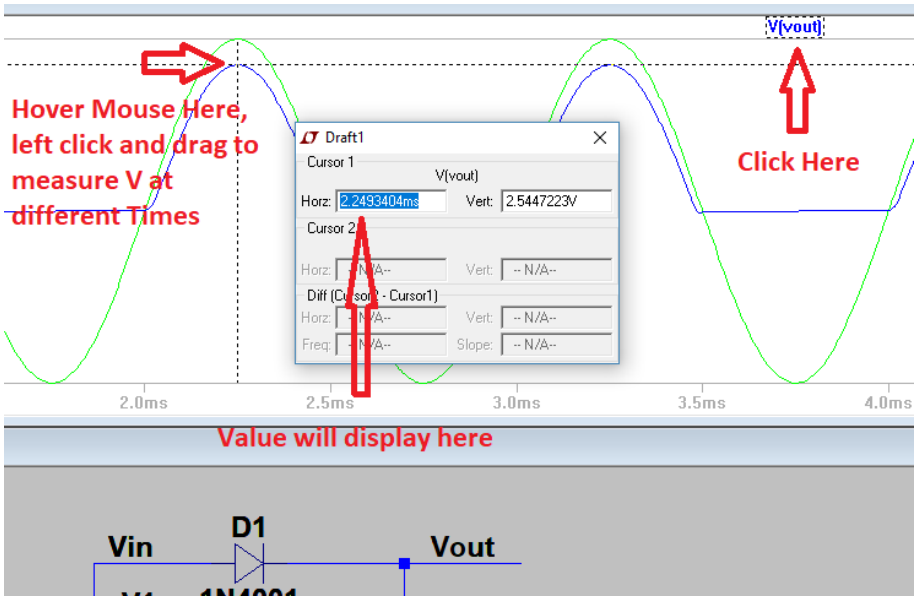
CECS 311 - LAB 1 – Diodes
Diodes: LEDs and Clipper Circuit Simulation

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 and annotate the screenshot with those values. 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.

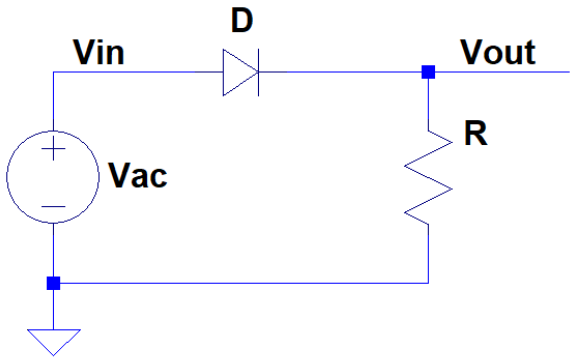
- In LTSpice be sure to label the nets with Vin and Vout.

- 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.



CECS 311 - LAB 1 – Diodes
Diodes: LEDs and Clipper Circuit Simulation

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



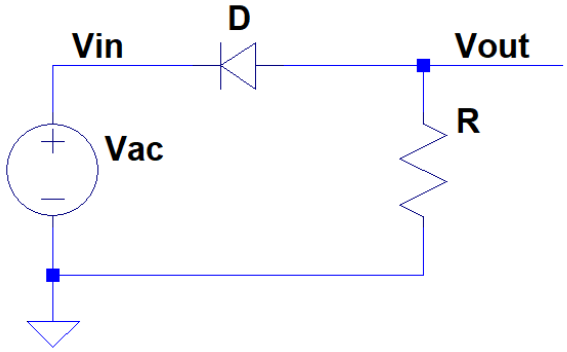
Draw the waveform, label and include V_{AC} and Vout/V_R

Calculate V_{AC} in pk-pk voltage?
Calculate V_R in pk-pk voltage?
Calculate 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 Vout/V_R

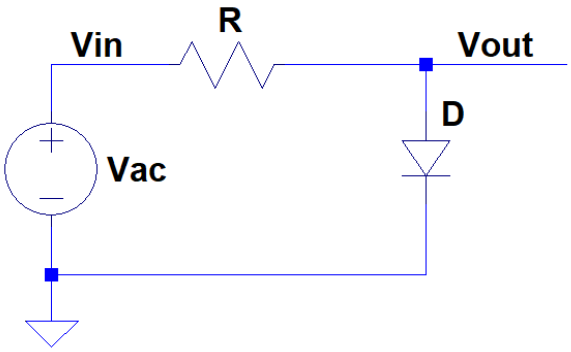
Calculate V_{AC} in pk-pk voltage?
Calculate V_R in pk-pk voltage?
Calculate V_D in pk-pk voltage?

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 311 - LAB 1 – Diodes
Diodes: LEDs and Clipper Circuit Simulation

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



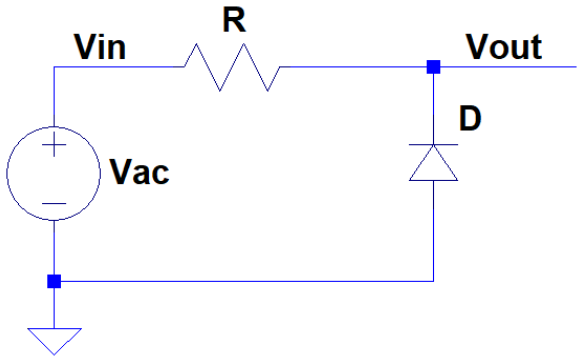
Draw the waveform, label and include V_{AC} and Vout/V_D

Calculate V_{AC} in pk-pk voltage?
Calculate V_R in pk-pk voltage?
Calculate V_D in pk-pk voltage?

Is this negative or positive clipping?

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

3b) Shunt Diode Clipper: V_{ac} = 6$V_{pk\text{-}pk}$, 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, $V_{AC\ Pk\text{-}Pk}$: $V_{R\ Pk\text{-}Pk}$: $V_{D\ Pk\text{-}Pk}$:
Include a Screen Capture of your LTSpice model with all measured values shown on the graph.