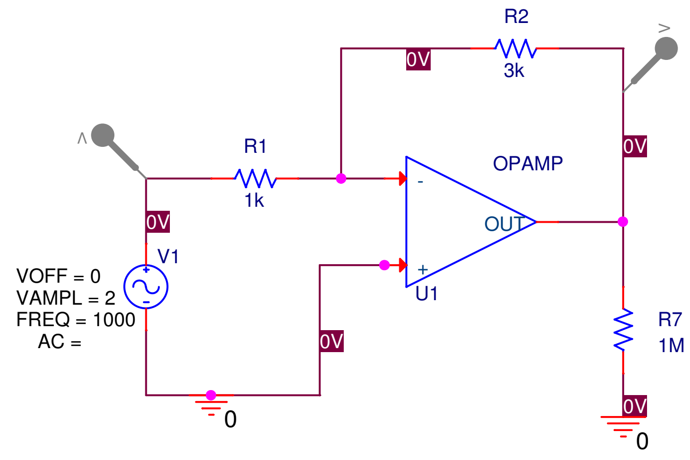
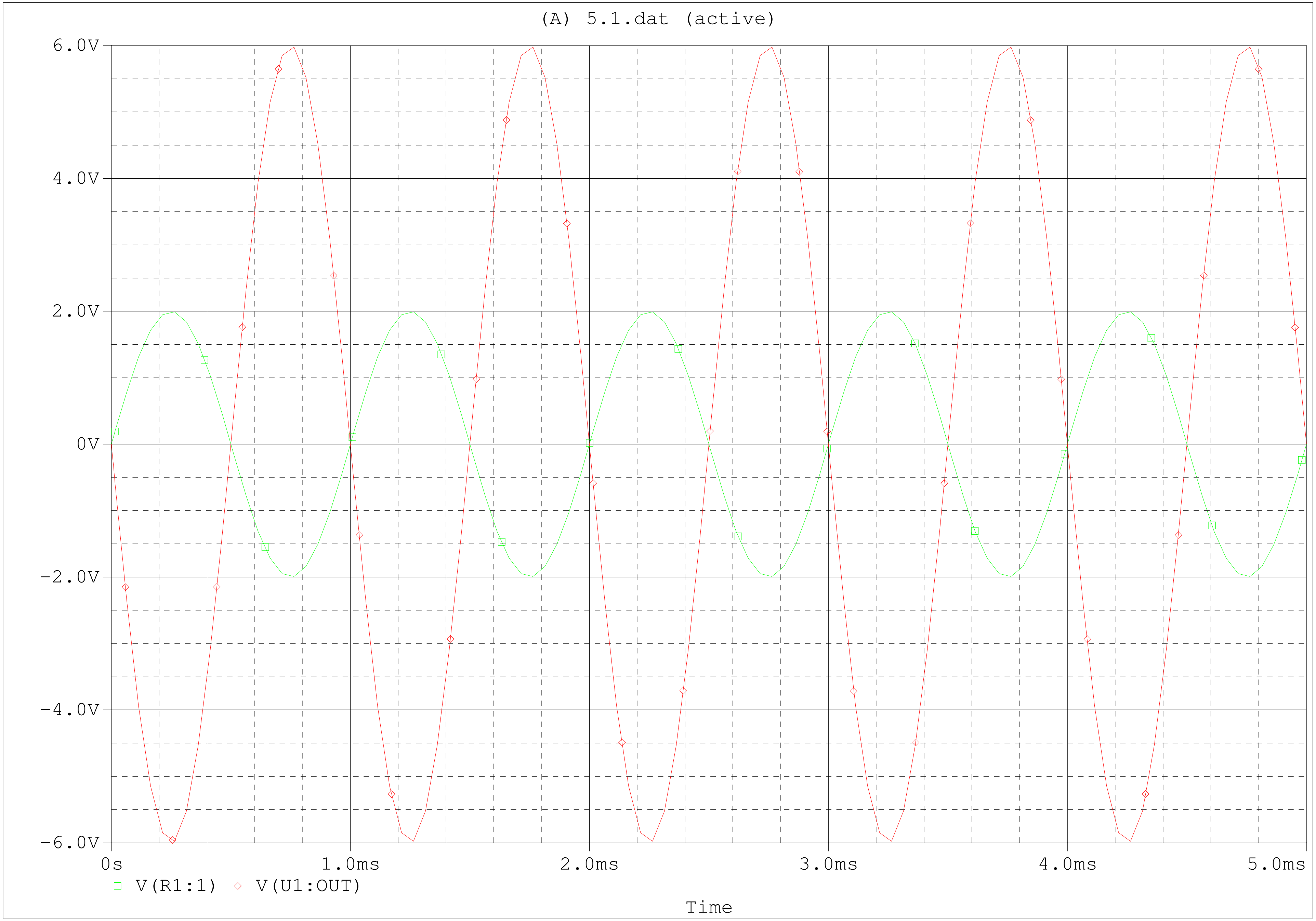


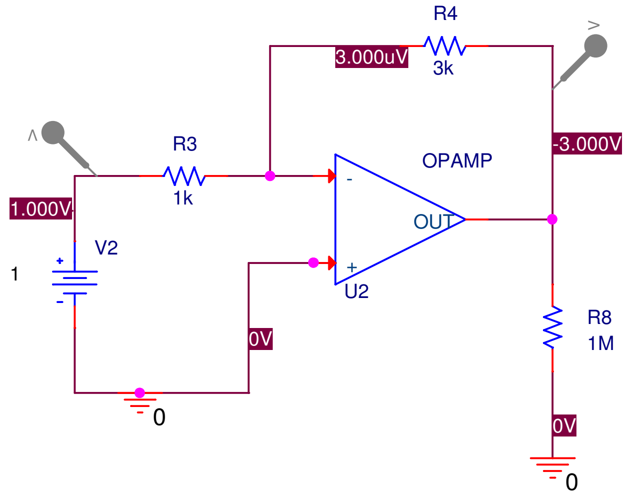
Title	PreLab1 Xi Kun Zou	
Size A	Document Number <Doc>	Rev <RevCo
Date: Friday, February 10, 2017	Sheet 1 of 1	

XiKun Zou

** Profile: "SCHEMATIC1-5.1" [C:\PExcercise\Prelab1-PSpiceFiles\SCHEMATIC1\5.1.sim]
Date/Time run: 02/10/17 05:37:13 Temperature: 27.0



Date: February 10, 2017 Page 1 Time: 05:37:20



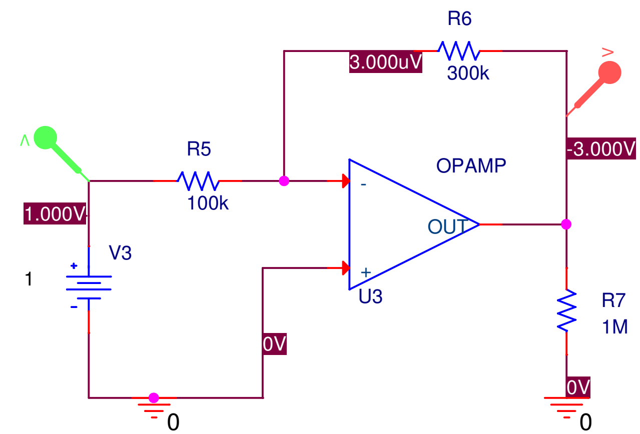
Title
PreLab 5.2 Xi Kun Zou

Size	Document Number	Rev
A	<Doc>	<RevCo

Date: Friday, February 10, 2017 Sheet 1 of 1

```
**** 02/10/17 05:45:56 ****** PSpice Lite (October 2012) ****** ID# 10813 ****

 ** Profile: "SCHEMATIC1-5.2sim"  [ C:\PExcercise\Prelab52-PSpiceFiles\SCHEMATIC1\5.2sim.sim ]


 ****     CIRCUIT DESCRIPTION


******************************************************************************




** Creating circuit file "5.2sim.cir"
** WARNING: THIS AUTOMATICALLY GENERATED FILE MAY BE OVERWRITTEN BY SUBSEQUENT SIMULATIONS

*Libraries:
* Profile Libraries :
* Local Libraries :
* From [PSPICE NETLIST] section of C:\SPB_Data\cdssetup\OrCAD_PSpice/16.6.0/PSpice.ini file:
.lib "nomd.lib"

*Analysis directives:
.TRAN  0 1000ns 0
.OPTIONS ADVCONV
.PROBE64 V(alias(*)) I(alias(*)) W(alias(*)) D(alias(*)) NOISE(alias(*))
.INC "..\SCHEMATIC1.net"



**** INCLUDING SCHEMATIC1.net ****
* source PRELAB52
V_V2         N00968 0 1
E_U2         N00916 0 VALUE {LIMIT(V(0,N00950)*1E6,-15V,+15V)}
R_R4         N00950 N00916  3k TC=0,0
R_R3         N00968 N00950  1k TC=0,0

**** RESUMING 5.2sim.cir ****
.END
.
**** 02/10/17 05:45:56 ****** PSpice Lite (October 2012) ****** ID# 10813 ****

 ** Profile: "SCHEMATIC1-5.2sim"  [ C:\PExcercise\Prelab52-PSpiceFiles\SCHEMATIC1\5.2sim.sim ]


 ****     INITIAL TRANSIENT SOLUTION       TEMPERATURE =   27.000 DEG C


******************************************************************************
```

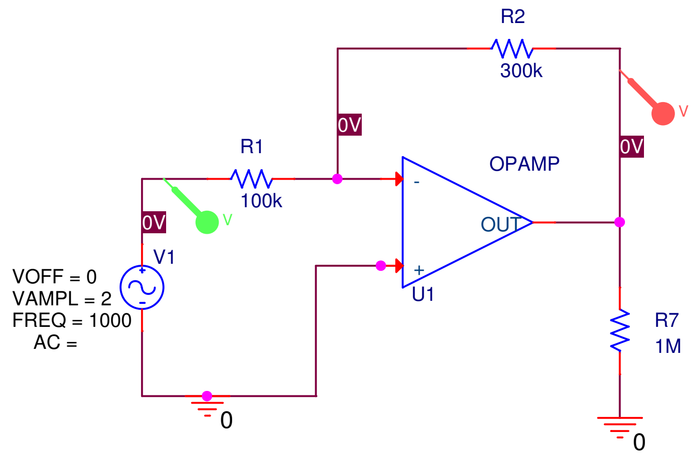
```
 NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE


(N00916)   -3.0000 (N00950) 3.000E-06 (N00968)    1.0000




    VOLTAGE SOURCE CURRENTS
    NAME         CURRENT

    V_V2        -1.000E-03

    TOTAL POWER DISSIPATION   1.00E-03  WATTS



          JOB CONCLUDED
.
**** 02/10/17 05:45:56 ****** PSpice Lite (October 2012) ****** ID# 10813 ****

 ** Profile: "SCHEMATIC1-5.2sim"  [ C:\PExcercise\Prelab52-PSpiceFiles\SCHEMATIC1\5.2sim.sim ]


 ****     JOB STATISTICS SUMMARY


******************************************************************************



  Total job time (using Solver 1)   =        .02
.
```



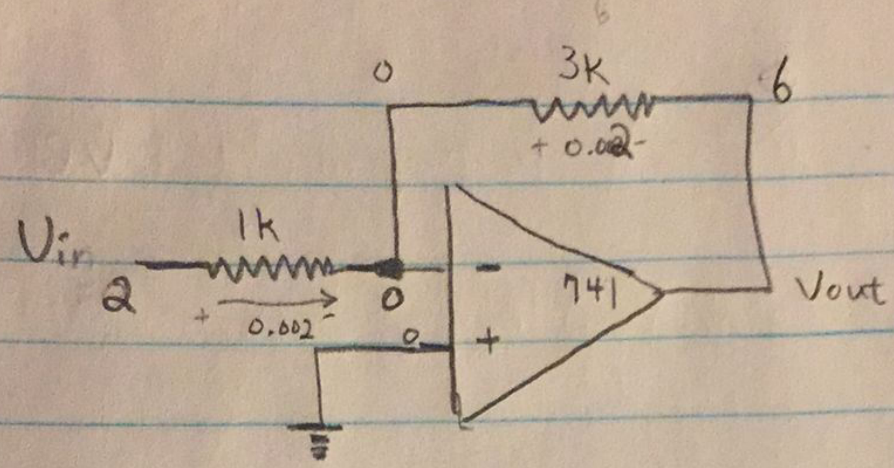
Title	Xi Kun Zou 5.3		
Size A	Document Number <Doc>		Rev <RevCo
Date:	Friday, February 10, 2017	Sheet 1 of 1	

```
•
**** 02/10/17 06:07:24 ****** PSpice Lite (October 2012) ****** ID# 10813 ****

 ** Profile: "SCHEMATIC1-5.3sim"  [ C:\PExcercise\plab53-PSpiceFiles\SCHEMATIC1\5.3sim.sim ]


 ****     CIRCUIT DESCRIPTION


******************************************************************************




** Creating circuit file "5.3sim.cir"
** WARNING: THIS AUTOMATICALLY GENERATED FILE MAY BE OVERWRITTEN BY SUBSEQUENT SIMULATIONS

*Libraries:
* Profile Libraries :
* Local Libraries :
* From [PSPICE NETLIST] section of C:\SPB_Data\cdssetup\OrCAD_PSpice/16.6.0/PSpice.ini file:
.lib "nomd.lib"

*Analysis directives:
.TRAN  0 1000ns 0
.OPTIONS ADVCONV
.PROBE64 V(alias(*)) I(alias(*)) W(alias(*)) D(alias(*)) NOISE(alias(*))
.INC "..\SCHEMATIC1.net"



**** INCLUDING SCHEMATIC1.net ****
* source PLAB53
V_V3         N01535 0 1
E_U3         N01495 0 VALUE {LIMIT(V(0,N01525)*1E6,-15V,+15V)}
R_R6         N01525 N01495  300k TC=0,0
R_R5         N01535 N01525  100k TC=0,0
R_R7         0 N01495  1M TC=0,0

**** RESUMING 5.3sim.cir ****
.END
•
**** 02/10/17 06:07:24 ****** PSpice Lite (October 2012) ****** ID# 10813 ****

 ** Profile: "SCHEMATIC1-5.3sim"  [ C:\PExcercise\plab53-PSpiceFiles\SCHEMATIC1\5.3sim.sim ]


 ****     INITIAL TRANSIENT SOLUTION       TEMPERATURE =   27.000 DEG C
```

```
******************************************************************************

 NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE


(N01495)   -3.0000 (N01525) 3.000E-06 (N01535)    1.0000




    VOLTAGE SOURCE CURRENTS
    NAME         CURRENT

    V_V3        -1.000E-05

    TOTAL POWER DISSIPATION   1.00E-05  WATTS



          JOB CONCLUDED
.
**** 02/10/17 06:07:24 ****** PSpice Lite (October 2012) ****** ID# 10813 ****

 ** Profile: "SCHEMATIC1-5.3sim"  [ C:\PExcercise\plab53-PSpiceFiles\SCHEMATIC1\5.3sim.sim ]


 ****     JOB STATISTICS SUMMARY


******************************************************************************



  Total job time (using Solver 1)   =        .02
.
```



Title	Xi Kun Zou 5.6	
Size A	Document Number <Doc>	Rev <RevCo
Date: Friday, February 10, 2017	Sheet 1 of 1	



$V_{in} = 2V$

$V_{out} = -6$

gain $= -6/2 = -3$

For 100k & 300k case

$i = \frac{2V}{100{,}000\Omega} = 0.00002A$

$V_{in} = 2V$

$V_{out} = 0.00002 \times 300{,}000 = 6V$

$\text{gain} = -\frac{6}{2} = -3$

Part Five: Analysis of an Op-Amp Circuit

1. Consider the op-amp circuit in Figure 1. Compute the ratio Vout/Vin and record your values for Vin, Vout, and Vout/Vin in the spaces provided at the bottom of the page. The ratio Vout/Vin is called the *gain*. Next, simulate this circuit in PSpice with a 2V (peak-to-peak) 1kHz source for Vin and compute the gain. For this simulation, the ideal OPAMP will suffice. (You will need to place a large resistor, say 100k or 1M, connecting Vout to ground.) Record your values below.

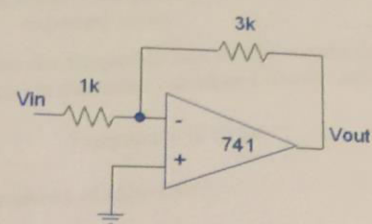


Figure 1: Op-Amp Circuit with a Fixed Gain

Construct the same circuit using a breadboard. The pin layout of the LM741 op-amp is shown in Figure 2. To wire the circuit as shown in the symbolic diagram of Figure 1, follow the wiring diagram shown in Figure 3. The 15V voltage source will come from the variable voltage source on the breadboard. The 1V (amplitude) sine wave will come from the function generator (the frequency can be set to 1kHz).

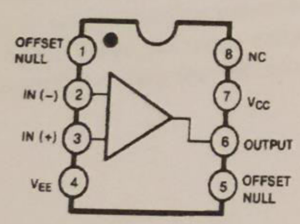


Figure 2:
Pin Layout of the LM741 Op-Amp

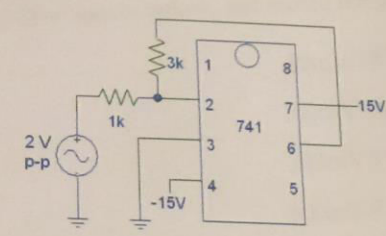


Figure 3:
Wiring Diagram for the Op-Amp Circuit

Build the circuit, but **before connecting the power, have a TA check the wiring of your circuit**.

Once the circuit is wired as shown in Figure 3, connect a probe from CH1 on the oscilloscope to the input voltage (coming from the function generator). Connect a second probe from CH2 on the oscilloscope to the output voltage (pin 6 of the op-amp). Display both readings on the oscilloscope. Record the peak-to-peak voltages below and compute the gain.

Quantity [units]	Hand Calculation	PSpice Simulation	Circuit Measurement
Vin [Vpp]	2V	2V	-1.96V
Vout [Vpp]	-6V	-6V	6V
gain (Vout/Vin)	-3	-3	-3

1. (continued)

Looking at the positive and negative peaks of both signals, why is this op-amp considered to be "inverting"? What is the phase difference between Vin and Vout?

When Vin is at minimum (−) Vout is at maximum (+)
by definition, the op-amp is inverting.
phase difference = 180°

2. Replace the 2V (pp) AC source with a 1V DC source. We do this to check the maximum power dissipated by the circuit. Run the simulation and this time check the output file (under the View menu). Near the bottom you should see the Total Power Dissipation. Record your value here.

Total Power Dissipation 0.001 W

3. Now, minimize power consumption while keeping the same gain across the amplifier, and also keeping Vin and Vout the same. Change only the two resistors, and be sure to only use resistors available in the lab for your simulation. (This is where the Electronics Lab Parts List is useful.) Design for a specification of 0.2mW. (In other words, make sure the power dissipated is less than this value.) Indicate the values you have chosen below.

R1 (which used to be 1k) 100k

R2 (which used to be 3k) 300k

4. Simulate the circuit in PSpice with the values chosen (and a 1V DC source), and verify that the gain has not changed and also that the design constraint has been met. Record the Total Power Dissipation from the output file here, and then fill out the appropriate spaces in part 6 on the next page.

Total Power Dissipation 0.01 mW

5. Before constructing the circuit, let's make sure you have the right resistors. Use the multimeter to measure each value of resistance and then compute the percent difference as compared to its labeled value. Comment on whether or not these resistors are within the labeled tolerances.

R1, tolerance _______

R1, measured _______

R1, % error _______

R2, tolerance _______

R2, measured _______

R2, % error _______

/ 2017 / Lab 1

6. Construct the circuit with the values chosen, now using the 2V (pp) AC source, and verify that the gain is the desired value. Measure Vin and Vout, and compute the gain.

Quantity [units]	Hand Calculation	PSpice Simulation	Circuit Measurement
Vin [Vpp]	2V	2V	
Vout [Vpp]	-6V	-6V	
gain (Vout/Vin)	-3	-3	

When you have completed the lab, sign and print your names below and have the TA initial next to each name.

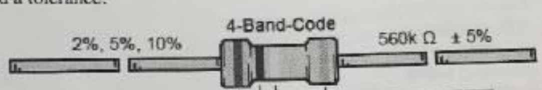
TA

______________________________ ______

______________________________ ______

Appendix: Resistor Color Code Chart

Here is a chart to show you how to convert the colored bands on a resistor to a value of resistance and a tolerance.



COLOR	1ST BAND	2ND BAND	3RD BAND	MULTIPLIER	TOLERANCE
Black	0	0	0	1Ω	
Brown	1	1	1	10Ω	± 1% (F)
Red	2	2	2	100Ω	± 2% (G)
Orange	3	3	3	1KΩ	
Yellow	4	4	4	10KΩ	
Green	5	5	5	100KΩ	± 0.5% (D)
Blue	6	6	6	1MΩ	± 0.25% (C)
Violet	7	7	7	10MΩ	± 0.10% (B)
Grey	8	8	8		± 0.05%
White	9	9	9		
Gold				0.1Ω	± 5% (J)
Silver				0.01Ω	± 10% (K)

(source: http://www.digikey.com/~/media/Images/Marketing/Resources/Calculators/resistor-color-char