```
**** 11/05/19 20:01:22 ***** PSpice 17.2.0 (March 2016) ***** ID# 0 ******
** Profile: "SCHEMATIC1-10 84" [ D:\On Editing\Sophomore\VE215, Intro to Circuits\10 84-PSpiceFiles\SCH
EMATIC1\10 84.sim ]
         CIRCUIT DESCRIPTION
***
************************************
** Creating circuit file "10_84.cir"
** WARNING: THIS AUTOMATICALLY GENERATED FILE MAY BE OVERWRITTEN BY SUBSEQUENT SIMULATIONS
*Libraries:
* Profile Libraries :
* Local Libraries :
* From [PSPICE NETLIST] section of E:\Cadence\SPB Data\cdssetup\OrCAD PSpice\17.2.0\PSpice.ini file:
.lib "nom.lib"
*Analysis directives:
.AC DEC 1 0.1592 0.1592
. OPTIONS ADVCONV
.PROBE64 V(alias(*)) I(alias(*)) W(alias(*)) D(alias(*)) NOISE(alias(*))
.INC "..\SCHEMATIC1.net"
**** INCLUDING SCHEMATIC1.net ****
* source 10 84
C C1
            N00196 N00355 0.5 TC=0,0
L L1
            N00196 N00229 4
I I1
            NOO196 0 DC OAdc AC 3Aac
            0 N00229 1 TC=0,0
R R1
            0 N00355 2 TC=0,0
RR2
.PRINT
              AC
+ VM([N00355])
+ VP([N00355])
            N00355 N00229 N00229 0 2
G G1
**** RESUMING 10 84.cir ****
. END
**** 11/05/19 20:01:22 ****** PSpice 17.2.0 (March 2016) ***** ID# 0 ******
** Profile: "SCHEMATIC1-10_84" [ D:\On Editing\Sophomore\VE215, Intro to Circuits\10_84-PSpiceFiles\SCH
EMATIC1\10 84.sim ]
         SMALL SIGNAL BIAS SOLUTION
***
                                        TEMPERATURE =
                                                       27.000 DEG C
***********************************
NODE
       VOLTAGE
                  NODE
                         VOLTAGE
                                    NODE
                                           VOLTAGE
                                                       NODE
                                                             VOLTAGE
```

Page: 1

```
(N00196)
          0.0000 (N00229)
                          0.0000 (N00355)
                                          0.0000
   VOLTAGE SOURCE CURRENTS
   NAME
              CURRENT
   TOTAL POWER DISSIPATION 0.00E+00 WATTS
**** 11/05/19 20:01:22 ****** PSpice 17.2.0 (March 2016) ***** ID# 0 ******
** Profile: "SCHEMATIC1-10_84" [ D:\On Editing\Sophomore\VE215, Intro to Circuits\10_84-PSpiceFiles\SCH
EMATIC1\10_84.sim ]
                                                  27.000 DEG C
        AC ANALYSIS
***
                                    TEMPERATURE =
*******************************
 FREQ
           VM (N00355) VP (N00355)
  1.592E-01
           1.664E+00 -1.464E+02
        JOB CONCLUDED
**** 11/05/19 20:01:22 ***** PSpice 17.2.0 (March 2016) ***** ID# 0 ******
** Profile: "SCHEMATIC1-10_84" [ D:\On Editing\Sophomore\VE215, Intro to Circuits\10_84-PSpiceFiles\SCH
EMATIC1\10_84.sim ]
        JOB STATISTICS SUMMARY
***
Total job time (using Solver 1) =
                                       . 02
```

Page: 2