**** 09/15/23 12:41:04 ****** PSpice 17.2.0 (March 2016) ****** ID# 0 ********

 ** Profile: "SCHEMATIC1-HW1_1_AC" [g:\andy\23me\hw1\hw1\hw1\hw1-pspicefiles\schematic1\hw1_1_ac.sim]

 ****   CIRCUIT DESCRIPTION

****

** Creating circuit file "HW1_1_AC.cir"
** WARNING: THIS AUTOMATICALLY GENERATED FILE MAY BE OVERWRITTEN BY SUBSEQUENT SIMULATIONS

*Libraries:
* Profile Libraries :
* Local Libraries :
.LIB "C:/Program Files (x86)/OUP/sedra_lib.lib"
* From [PSPICE NETLIST] section of C:\cadence\Cadence\SPB_Data\cdssetup\OrCAD_PSpice\17.2.0\PSpice.ini file:
.lib "nom.lib"

*Analysis directives:
.AC DEC 10 0 1G
----------$
ERROR(ORPSIM-16103): Invalid value
.OPTIONS ADVCONV
.PROBE64 V(alias(*)) I(alias(*)) W(alias(*)) D(alias(*)) NOISE(alias(*))
.INC "..\SCHEMATIC1.net"

**** INCLUDING SCHEMATIC1.net ****
* source HW1
V_VDD   N00700 0 3.3Vdc
R_RS   0 N00725 150k TC=0,0
M_M1   N00411 N01229 N00662 N00662 PMOS0P5

```
+ L=0.5u
+ W=12u
+ AD=1.72E-12
+ AS=1.72E-12
+ PD=5.25e-6
+ PS=5.25E-6
+ M=1
M_M2            VO N01180 N00662 N00662 PMOS0P5
+ L=0.5u
+ W=12u
+ AD=1.72E-12
+ AS=1.72E-12
+ PD=5.25e-6
+ PS=5.25E-6
+ M=1
M_M6            N00725 N00725 N00700 N00700 PMOS0P5
+ L=0.5u
+ W=0.75u
+ AD=1.72E-12
+ AS=1.72E-12
+ PD=5.25e-6
+ PS=5.25E-6
+ M=1
M_M5            N00662 N00725 N00700 N00700 PMOS0P5
+ L=0.5u
+ W=2.5u
+ AD=1.72E-12
+ AS=1.72E-12
+ PD=5.25e-6
+ PS=5.25E-6
+ M=1
M_M3            N00411 N00411 0 N00571 NMOS0P5
+ L=0.5u
+ W=5u
+ AD=1.72E-12
+ AS=1.72E-12
+ PD=5.25E-6
+ PS=5.25E-6
+ M=1
```

```
M_M4            VO N00411 0 N00580 NMOS0P5
+ L=0.5u
+ W=5u
+ AD=1.72E-12
+ AS=1.72E-12
+ PD=5.25E-6
+ PS=5.25E-6
+ M=1
C_CL           0 VO  1p  TC=0,0
V_Vin+          N01229 0 DC 1.65Vdc AC 0.5749020786Vac
V_Vin-          N01180 0 DC 1.65Vdc AC -0.5749020786Vac


**** RESUMING HW1_1_AC.cir ****
 .END
```