

Re: PSPICE A/D simulation problem

Source: <http://sci.tech-archive.net/Archive/sci.electronics.design/2007-04/msg02016.html>

- *From:* "Helmut Sennewald" <helmutsennewald@xxxxxxxxxxx>
 - *Date:* Tue, 10 Apr 2007 23:49:11 +0200
-

"RsK" <rizkhan7@xxxxxxxxxx> schrieb im Newsbeitrag
<news:1176237702.167172.14240@xx>

Hi anyone can see the pspice file and please tell me how to remove the error it displays while simulation?

regards,
rizwan

Hello rizwan,

I don't know what PSPICE reports, but the error report from LTspice is crystal clear if you know SPICE.

WARNING: Node 1 is floating.
WARNING: Less than two connections to node 1. This node is used by C1.
----> one pin of C1 is not connected

No AC stimulus found:
Set the value of a current or voltage source to "AC 1."
to make it behave as a signal generator for AC analysis.
----> An AC-analysis requires at least one voltage source with an "AC"
definiton.

This doesn't matter, but why do you have a transistor model when you don't use a transistor?

Best regards,
Helmut

```
*  
.MODEL QBC337-25 NPN(  
+ IS = 4.13E-14  
+ NF = 0.9822
```

Re: PSPICE A/D simulation problem

```
+ ISE = 3.534E-15
+ NE = 1.35
+ BF = 292.4
+ IKF = 0.9
+ VAF = 145.7
+ NR = 0.982
+ ISC = 1.957E-13
+ NC = 1.3
+ BR = 23.68
+ IKR = 0.1
+ VAR = 20
+ RB = 60
+ IRB = 0.0002
+ RBM = 8
+ RE = 0.1129
+ RC = 0.25
+ XTB = 0
+ EG = 1.11
+ XTI = 3
+ CJE = 3.799E-11
+ VJE = 0.6752
+ MJE = 0.3488
+ TF = 5.4E-10
+ XTF = 4
+ VTF = 4.448
+ ITF = 0.665
+ PTF = 90
+ CJC = 1.355E-11
+ VJC = 0.3523
+ MJC = 0.3831
+ XCJC = 0.455
+ TR = 3E-08
+ CJS = 0
+ VJS = 0.75
+ MJS = 0.333
+ FC = 0.643)
*
*Resistors
*
R1 2 4 56k
R2 2 0 22k
RB 2 3 180
RC 4 5 5.1k
RE 6 0 3.3k
* Load
RL 8 0 1k
CL 5 8 10u
*
*Capacitors
*
C1 1 2 330n
```

Re: PSPICE A/D simulation problem

```
CC 3 5 470p
CE 6 0 220u
VCC 4 0 15
*
.AC DEC 10 10 100k
.PROBE
.END
```