

Help using Linear's free simulator

Source: <http://sci.tech--archive.net/Archive/sci.electronics.basics/2007-01/msg00186.html>

- *From:* "fulgape" <simu@xxxxxxxxxxxxxxxxx>
 - *Date:* 4 Jan 2007 17:38:57 -0800
-

Hello ,

I am trying to simulate a simple opamp circuit with Linear's LT Spice simulator. The circuit is simply a single supply opamp, the minus input connected to 0.5V and the plus input connect to a 1V sine wave. The output is connected to ground by 1K R.

According to my understanding, I should have at the output a square wave. However, I am getting only a dc value. Can someone check my "design"

I have the LT Spice asc file:

```
Version 4
SHEET 1 880 680
WIRE -256 224 -256 32
WIRE -256 336 -256 304
WIRE -144 224 -144 112
WIRE -144 304 -256 304
WIRE 32 192 32 144
WIRE 32 304 -144 304
WIRE 32 304 32 272
WIRE 64 112 -144 112
WIRE 64 144 32 144
WIRE 96 32 -256 32
WIRE 96 96 96 32
WIRE 96 304 32 304
WIRE 96 304 96 160
WIRE 192 128 128 128
WIRE 192 176 192 128
WIRE 192 304 96 304
WIRE 192 304 192 256
FLAG -256 336 0
SYMBOL Misc\\battery -144 208 R0
WINDOW 123 0 0 Left 0
WINDOW 39 0 0 Left 0
SYMATTR InstName V1
SYMATTR Value SINE(0 1 50)
SYMBOL Misc\\battery -256 208 R0
WINDOW 123 0 0 Left 0
```

Help using Linear's free simulator

```
WINDOW 39 0 0 Left 0
SYMATTR InstName V2
SYMATTR Value 5
SYMBOL Opamps\\LT1006 96 192 M180
SYMATTR InstName U1
SYMBOL res 176 160 R0
SYMATTR InstName R1
SYMATTR Value 1000
SYMBOL Misc\\battery 32 176 R0
WINDOW 123 0 0 Left 0
WINDOW 39 0 0 Left 0
SYMATTR InstName V3
SYMATTR Value 0.5
TEXT -290 506 Left 0 !.tran 0.1
```

Thanks.