

Re: frequency response

Source: <http://sci.tech-archive.net/Archive/sci.electronics.cad/2007-05/msg00056.html>

- *From:* "Helmut Sennewald" <helmutsennewald@xxxxxxxxxxx>
 - *Date:* Tue, 15 May 2007 22:59:11 +0200
-

"DDDiID" <linuxfreak87@xxxxxxxxxxx> schrieb im Newsbeitrag
news:1179259959.208298.232300@xx

Hello,

I tried a transient analysis of your netlist with LTspice and it worked in principle but there is a small oscillation at the output. The value of Cout is undersized. Just increase its value to 10u and you will be saved.

Thanks for the suggestion.

How can you run any .AC-analysis without any source having AC specified? It's impossible. Please explain where your AC is specified. AC-analysis is a linear system analysis in SPICE! It has nothing to do with the AC-voltage applied to your rectifier diodes. Where do you want to apply the AC-voltage or current?

I thought having "v1 4 5 dc 0 sin(0 34 60)" would allow me to do an ac analysis because its an ac source. But from your reply, I see that this is not the case. The applied ac voltage should be applied to the rectifier as v1 is now. What changes should I make to the v1 to allow me to perform this ac analysis, while retaining my peak value of 34v and frequency of 60Hz.

Hello,

I still think you expect something different from the .AC simulation. The .AC simulation could be used to measure the small signal input

Re: frequency response

"noise" rejection of the regulator versus frequency or it could be used to measure the dynamic output resistance of your regulator. All these .AC simulations will require to setup the appropriated DC condition for the LM317. This means you would apply a DC voltage of e.g. 32V at the input with an additional AC definition. It would be also necessary to add the correct load device, e.g, a resistor load or a current source load.

The normal 60Hz ripple rejection has to be simulated with .TRAN because this is a really nonlinear system.

Best regards,
Helmut

PS: Nobody can exactly reproduce your circuit without knowing your LM317 model.

I'm using the following model:

http://www.industrycommunity.com/myforum/moazzam_mahmood_next1/messages/48.html

Thanks again.