

Pspice Monte Carlo analysis.

Source: <http://sci.tech-archive.net/Archive/sci.electronics.design/2004-10/7152.html>

From: learner (sonalsingh28_at_yahoo.com)

Date: 10/25/04

Date: 25 Oct 2004 13:57:29 -0700

Hi,

I am designing a telescopic opamp using MOSFETs. I made change sin the transistor model to accomodate for changes in threshold voltage for both P and N type MOS by adding dev/gauss .022 and dev/gauss 5% for change in Kp. My problem is when I run Pspice, I get an Error message saying " invalid number" for bias point calculations and " Invalid terminal abbreviation for device " when I run a AC sweep.

I am not able to debug this error message.

Will appreciate any suggestions.

Thanks

Learner