Subject: This is extraordinarily cool
Posted by wunhuanglo on Sun, 24 Apr 2005 13:53:14 GMT
View Forum Message <> Reply to Message

This a link to the free student version of CircuitMaker. I guess it's graphical Spice.I have essentially no idea of what I'm doing, but with a little fumbling I made a first-order 2-way crossover with a 2.82V p-p sine wave input and variable frequency.When you get it "right" you get an oscilloscope window, and you can move a voltage probe around the circuit to see amplitudes. Varying the frequency you can see the effect of larger caps or smaller inductors etc... You can learn a lot with this thing.
CircuitMaker

Subject: Re: This is extraordinarily cool
Posted by Damir on Sun, 24 Apr 2005 17:43:18 GMT
View Forum Message <> Reply to Message

Although PSpice ("OrCad"-for example) is almost "standard", I like "Circuit Maker" (3f4/3f5 "spice engine") better and used it every day in the last few years. Although not without its imperfections (transformer models, some "too simple" models, etc.) it`s easy to use and understand. It has some tube models, and other can be found on the net ("Duncan amp pages" - models, links, explanations, list of other free "Spice" programs...). Great tool for learning - you have virtual multimeter, oscilloscope, bode plot (frequency AC analysis), Fourier-analysis, etc., and you can see "what`s goin` on" in every part of the circuit. Of course, your analysis is that good like the models, but I found that even the simplest triode models, based on the expression:Ia=K(μ*Ugk+Uak)^1,5 are "good enough" for DC and some AC "results". Of course, you can`t expect accurate results for distortion analysis ("optimistic" results, because of "more ideal" Ua/Ia/Ug "lines").All in all, great - and free program...

Subject: Re: This is extraordinarily cool
Posted by Mike.e on Fri, 29 Apr 2005 03:38:28 GMT
View Forum Message <> Reply to Message

yeah i dont enjoy typing nodes and commands into SPICE so Ive left it unused on my pc!

Subject: Re: This is extraordinarily cool
Posted by Wayne Parham on Fri, 29 Apr 2005 10:42:03 GMT
View Forum Message <> Reply to Message

You should try PSice, the companion of OrCad. You draw a schematic using a CAD program, and it can be used to layout a circuit board and model the circuit with Spice.