

---

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:  $I_a = K(\mu U_{gk} + U_{ak})^{1,5}$  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"  $U_a/I_a/U_g$  "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 PSpice, 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.

---