For our purposes, circuitlab.com offers a great service, offering drag-and-drop design and on-line analysis. There is a student version available for only $12 for 6 months, you only have to have an ".edu" e-mail address that they can verify.

(Show start up.) You can try the demo on your own, I'm going to show just a little. You are not <u>required</u> to subscribe, it will make things easier, especially for lab reports, since you can easily produce ckts and export them as ~~pdf's~~ pdf's for inclusion in your report. You can even use it to confirm your analysis.

There are many Circuit Analysis Packages out there. PSPICE is a common one, OrCAD is another tool suite for

Electronic
Design or E.D.A.
Automation

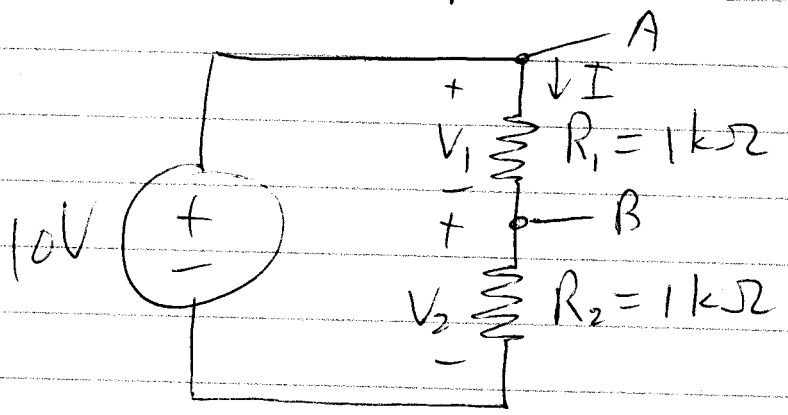
which is now owned by Cadence, another big player. OrCAD is now in its 16th version (16.6, with 16.7 coming soon.) You can go to orcad.com and find out more.

Those tools can be very complicated to learn, and ~~like mock~~ often look more like computer programming than circuit design and analysis.

(Start up, sign in, click "My Workbench", then "Create new circuit" on right.

You start with a clean page and a list of possible components on the left.

Let's do a simple voltage divider:



Predict: Symmetry ⇒ $V_1 = V_2 = \frac{1}{2} V_S = 5V$

$V_B = 5V$

$V_A = 10V$

$I = \frac{10V}{2k\Omega} = 5\ mA$

Click Simulate, DC, + Add Expression, Click on top of R_1, +Add :, Click on top of R_2, "Run DC Solver"