

ECE 340: Solving diode-containing circuits

day11 – 2026-09-23

1. First diode circuit

Circuit of Solving nonlinear equations, but don't distribute the handout yet.

Set up the circuit equations using two different techniques:

1. #1 — KCL at the anode's node,
which is **nodal analysis**.
Yielding a voltage (v_D) as the solution.

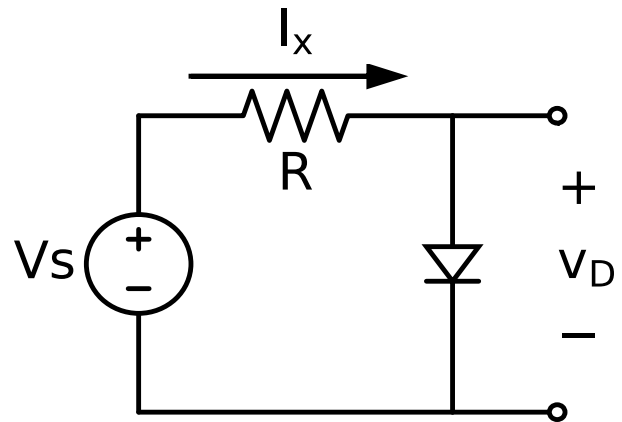


Figure 1. Simplest circuit with a diode

1. #2 — KVL around the single loop,
mesh current analysis.
Yielding a current (I_x) as the solution.

Which is better to solve for: voltage or current?

...

It doesn't matter! Having one and wanting the other is merely a substitution with one of your original equations.

Sometimes, it is less work one way; do that. Sometimes, you don't know or have enough experience with that particular form of problem to know which will be easier until you've already started. Such is life.

1.1. Load line

A **load line analysis** is a graphical method to help solve a circuit problem. It is common in analog, especially with amplifiers and for nonlinear circuits.

- [Wikipedia: Load line \(electronics\)](#)
- [AAC: How is a Load Line Used in Circuit Design?](#)

You'll notice that the first example in these references is *always* the {V_{source} — resistor — diode} series circuit. That's because it's a great example of the technique.

generate two equations,
involving the same two variables,
and plot them on two axes,
the curves' intersection is the circuit's solution.

— *Load line poem*

1.2. Guess and iterate method

See: [Solving nonlinear equations](#) webpage or handout.

- Learn about it by reading,
- then see the calculator trick demonstration (plus a setting for your TI calculator that will change your life).
- Practice by using the technique to solve a diode circuit with numbers.

2. Diode circuit simulation

2.1. CircuitLab

All ECE students^[1] have access to the full version of CircuitLab. It is great for drawing circuits that look nice (including SVG export!), but the simulation engine and plotting features are so-so. Every simulator has things to like and things to dislike, so we learn several.

CircuitLab Getting Started.pdf

- Draw the V—R—D circuit in CircuitLab.

Add a **GND** symbol to designate which node in the circuit is **declared** to be at zero volts. Essentially *all* circuit simulators use “modified nodal analysis”, and therefore they all require the user to declare one node as the zero node before the software can solve the circuit.

Modified?

Modified nodal analysis (MNA) modifies the system of equations' matrix by adding a few rows and columns that avoid issues like identifying *super-nodes* and *super-meshes*.

<http://lpsa.swarthmore.edu/Systems/Electrical/mna/MNA1.html> is a nice set of pages that describes MNA and how one might go about writing a program to set up and solve the equations.

The QUCS “Quite universal circuit simulator” documentation includes another nice description of MNA: <http://qucs.sourceforge.net/tech/node14.html>.

Once you’ve drawn the circuit in CircuitLab and connected the **GND** symbol to a node, check the far lower-right corner for a **green circle** ●. The color indicates whether it has computed the DC solution for the circuit. If so, then just hover your mouse over various nodes and devices to see the DC values as a tooltip. When the circle is **red** ●, then there is some error that prevents the simulator from finding a valid solution. This is super handy!

- Write down the values of the loop current I_x and the diode’s voltage v_D .

2.2. LTspice

LTspice is from Analog Devices (ADI). It was developed by the company Linear Technology Corporation, hence the prefix, and kept the name after ADI acquired LTC in 2017.

The simulation engine in LTspice is also based on the original Berkeley SPICE 3f5. What makes it one of my favorites is that it is **fast** and **accurate** especially for transient (time-domain) simulations. It actually generates custom assembly code for solving the circuit's MNA matrix *after* you Run the simulation. See the article SPICE Differentiation by Mike Engelhardt (author of LTspice) for more information.

LTspice is installed on the ECE lab computers.

- See the demo on how to run LTspice.

The full version is *free* and has no “student version” limits that are unlocked with \$\$\$.

- Draw the V—R—D circuit in LTspice. Select the 1N4148 diode.
- Perform a **DC op pnt**, DC operating point, or .op simulation.

If successful, the simulator will pop up a window with the node voltages and device currents. Afterwards, you can hover over devices and nodes and see the DC solution's values in the grey status bar at the bottom.

- Write down the values of the loop current I_x and the diode's voltage v_D .

2.3. Other simulators?

Do the same in Pspice, if you dare.

There are other simulators that operate in a web browser.

SIMetrix/SIMPLIS Elements was my first favorite simulator. Its simulation engine is also great.

Micro-Cap was a well-regarded commercial simulator that is now free, though no longer being developed.
<https://www.spectrum-soft.com/download/download.shtm>

NL5 Circuit Simulator is **not** based on SPICE and therefore doesn't suffer from that family's limitations.

The maker of NL5 also offers idealCircuit (free) that can simulate ... ideal circuits. The problem of solving nonlinear circuits (and equations) is wide and varied, so there is space for alternative (non-SPICE) approaches. Very interesting!

1. Technically, it's *everyone* with an @valpo.edu email address, but the license is purchased by the ECE department.