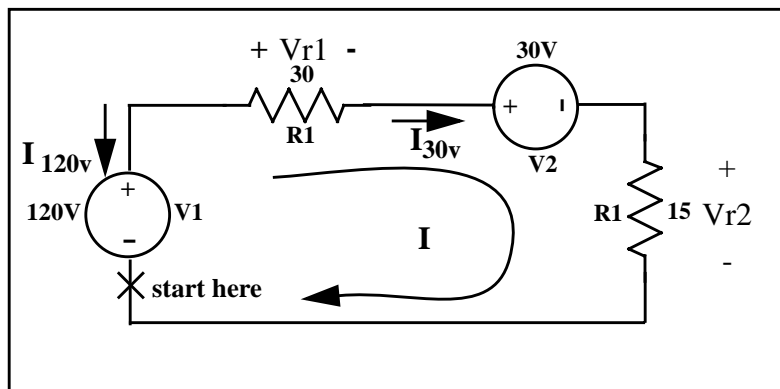


KVL Analysis with Hspice

“Spice” or its derivatives (Spice 2G.6, Hspice, Pspice) are used to simulate the behavior of electrical circuits.

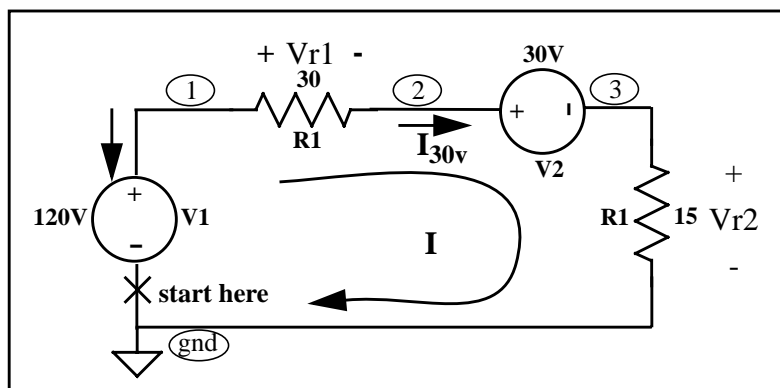
Using a circuit simulator is a good way of confirming your intuition about a circuit or confirming a hand analysis. It should basically be used as confirmation or optimization tool. It should not be used as a design tool. For this class, Hspice can serve as an easy and quick way to check almost any homework problem you can think of. Think of it as the ultimate answer key.

Below is the circuit we analyzed by hand earlier. We see that the steps of selecting a reference direction for the current I and element voltages have been done.



In Spice, circuits are described as “netlists”. Netlists are simply a listing of how the elements in the circuit are connected. We describe the connection of the elements by telling which node each terminal of an element is connected to. Therefore, we need to mark the nodes in the circuit with unique node names. These are shown below as the numbers 1,2,3 and the node name “gnd”. Note that all Spice netlists must have a ground node in the circuit. The node names are circled for clarity.

Each element in the netlist must be distinct from the others. Therefore, we need to distinguish the two voltage sources with reference designators. For this circuit, the designators are “V1” and “V2”. Note that R1 and R2 are already marked. The new annotated schematic is shown below. Now we can write the netlist.



The Spice netlist for the circuit above is placed in a normal text file with your choice of editor. It looks like this:

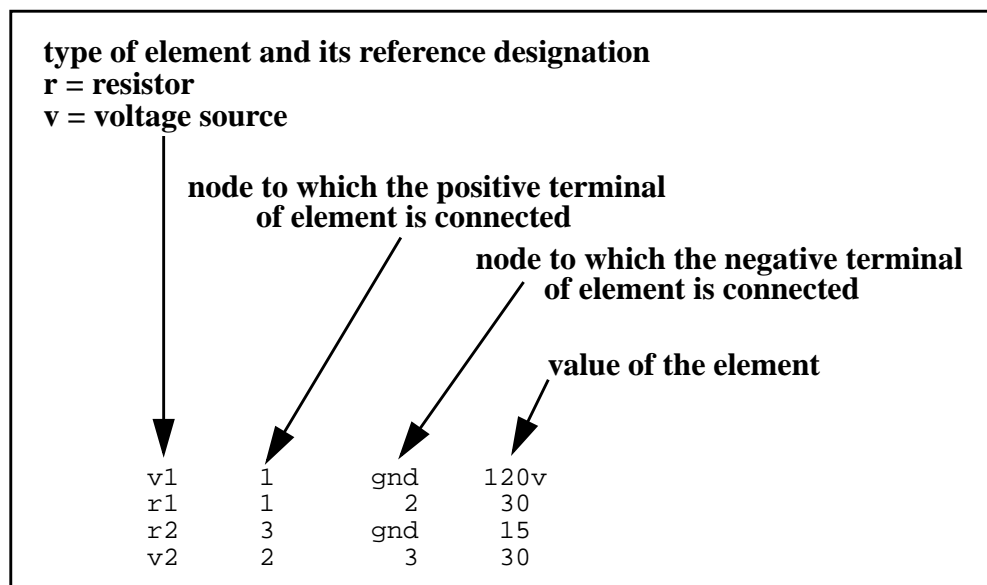
```
.title Spice example for kcl
.options
+ badchr=1 $detect bad chars
+ ingold=1 $combined exp and fixed output format
+ numdgt=4 $number of significant digits for output variables
$ netlist follows
v1 1 gnd 120v
r1 1 2 30
r2 3 gnd 15
v2 2 3 30
.op $find dc operating point
.options post $save data for post processing
.end
```

A description of the spice file is as follows:

The first line is always the title line. With or without the `.title`, the first line is still the title. **(Watch out!)**

The `.options` line sets up some convenient settings for your simulation. The “+” is the continuation character for continuing the `.options` selections.

The body of the netlist is shown below. It is the exact description of the circuit.



The `.op` statement instructs Spice to run a dc steady state analysis.

The `.options post` statement saves the results.

The `.end` statement indicates the end of a Spice file. It is mandatory.

Comments follow the “\$” delimiter

Note that the currents through the elements in the netlist are implied by the positive and negative terminal definitions. Think about this.

If we name this file *kcl.sp*, Hspice is invoked on the file at the Unix prompt by typing:

```
hspice kcl.sp > output
```

The results from the simulation are in the file *output*. Among the other info in the file we find the following:

```
***** operating point information  tnom= 25.000 temp= 25.000 *****
node    =voltage    node    =voltage    node    =voltage
0:1      =120.0000    0:2      =60.0000    0:3      =30.0000

**** voltage sources
element  0:v1        0:v2
volts    120.0000    30.0000
current  -2.0000     2.0000
power    240.0000    -60.0000

total voltage source power dissipation= 180.0000          watts

**** resistors
element  0:r1        0:r2
r value  30.0000     15.0000
v drop   60.0000     30.0000
current  2.0000      2.0000
power    120.0000    60.0000
```

We see that the loop current is identical to the hand calculated value of 2A. The dc operating point analysis also gives the voltage across, current through, and power consumption for all circuit elements.

Note the current through the 120V source. It is specified as -2A. This is referenced to the passive sign convention which has the current arrow pointing into the voltage source. From the minus sign, then, we can see that the 120V source is *sourcing* current (it is leaving the positive terminal). Note that the 30V source is *sinking* current.

Hspice calculates power supplied by elements, contrary to what it states. For example, the 120V source is definitely not dissipating 240 watts, its supplying or generating 240 watts.

Take some time examining the Hspice results. You can learn much by doing so.