LTSpice Supplemental Documentation

This document is intended to supplement the on-line documentation for LTSpice. The on-line documentation is linked off http://www-ece.eng.uab.edu/LTSpice/ .

Current Controlled Dependent Voltage Source

This model exists in the LTSpice system but is not well supported by the Graphical User Interface. Proper use relies on some knowledge of the underlying SPICE software.

From University of Exter School of Physics Spice 3 User's Manual[1],

---

3.2.2.4 Linear Current-Controlled Voltage Sources

General form:

```
HXXXXXXX N+ N- VNAM VALUE
```

Examples:

```
HX 5 17 VZ 0.5K
```

N+ and N- are the positive and negative nodes, respectively. VNAM is the name of a voltage source through which the controlling current flows. The direction of positive controlling current flow is from the positive node, through the source, to the negative node of VNAM. VALUE is the transresistance (in ohms).

The LTSpice product has the H model which matches up with the underlying SPICE3 model but alas, it is graphic representation is a voltage source not a dependent voltage source. The graphics tool does allow specifying additional parameters. By inspection, the first value (past the connection nodes, called Value in the H model's advanced menu) is the current dependency and the second value (called Value2 in the H model's advanced menu) is the gain.

A second complexity to overcome is that the underlying SPICE3 requires that the controlling current be the current through an independent voltage source. Often, one has to add a 0 Volts Voltage source in series with another element to fulfill this dependency. The note above describes the orientation of the voltage source relative to the current to be sensed.

For example, suppose one desires to simulate the simple circuit in Figure 1[2]. The actual LTSpice circuit (Figure 2) would include the additional voltage source to sense the current.

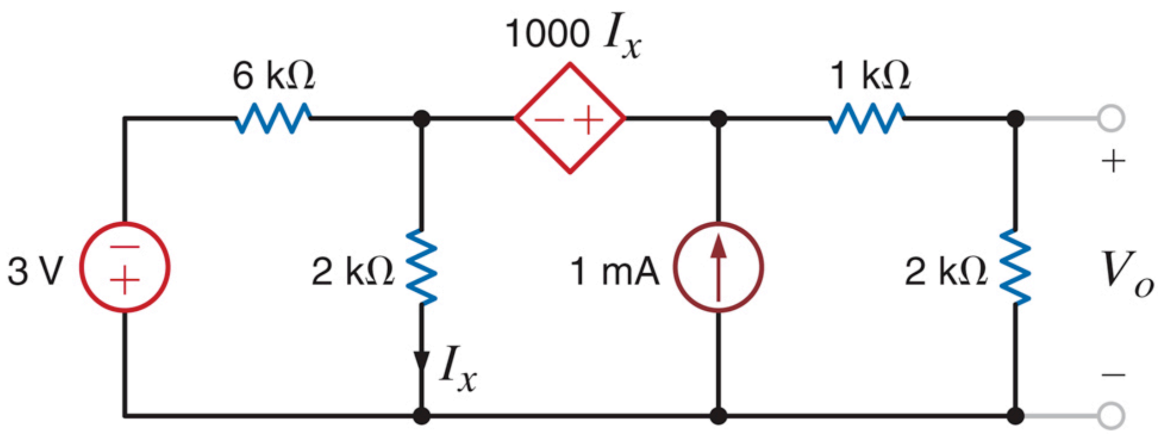

Figure 1. Original Circuit

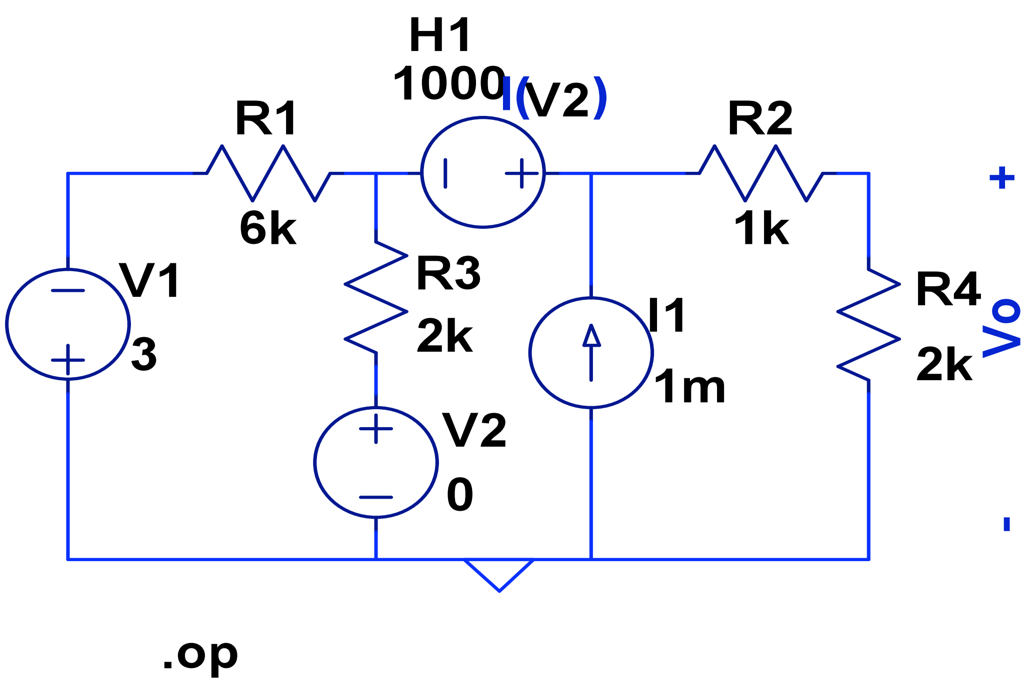

Figure 2. LTSpice Equivalent Circuit

The underlying SPICE3 commands for the circuit in Figure 2 (View | Netlist) are

```
* C:\Program Files\LTC\LTspiceIV\Draft5.asc
V1 0 N001 3
R1 N001 N002 6k
R2 N003 N004 1k
R3 N002 N005 2k
```

```
R4 N004 0 2k
I1 0 N003 1m
H1 N003 N002 V2 2000
V2 N005 0 0
* -             Vo         +
* I(
* )
.op
.backanno
.end
```

Where the * commands are merely adding comments to the graphic. Note that V_o is the same as V(N004).

The operating point for the circuit (DC solution) is given as text when run as

```
--- Operating Point ---

V(n001):      -3     voltage
V(n002):      0.428571     voltage
V(n003):      0.642857     voltage
V(n004):      0.428571     voltage
V(n005):      0      voltage
I(H1):        0.000785714       device_current
I(I1):        0.001        device_current
I(R4):        0.000214286       device_current
I(R3):        0.000214286       device_current
I(R2):        0.000214286       device_current
I(R1):        -0.000571429      device_current
I(V2):        0.000214286       device_current
I(V1):        -0.000571429      device_current
```

So the answer is 0.429 Volts.

---

[1] http://newton.ex.ac.uk/teaching/CDHW/Electronics2/userguide/sec3.html

[2] Irwin, J. David and Nelms, R. Mark, *Basic Engineering Circuit Analysis,* 9th edition, John Wiley & Sons, 2008, Problem 5.59, p238.