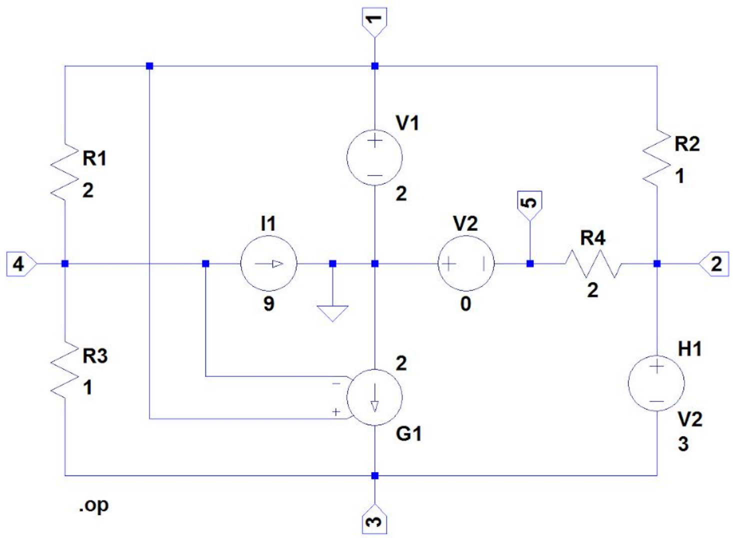
Example 48

Last updated: 15 Oct 2017

This example is from problem 4.8 in Basic Electric Circuit Analysis [1]. The schematic is drawn using LTspice [2]. The nodes are numbered using the label tool. This particular circuit uses controlled sources as well as independent sources; H and G type sources are used in this example. V2 is associated with the current flowing in R4 and the value of V2 is zero. Putting an extra voltage source in the branch allows the python code to automatically generate the equations for the current in the branch. The ipython notebook for this example is available in reference [3].



The netlist was exported from LTspice and is shown below.

```
* C:\Users\Jim\Documents\Python SciPy\NETWORK\spice test circuits\example48.asc
R1 1 4 2
R2 1 2 1
R3 4 3 1
R4 2 5 2
V1 1 0 2
I1 4 0 9
H1 2 3 V2 3
G1 0 3 1 4 2
V2 0 5 0
.op
.backanno
.end
```

LTspice solved for the DC operating point and the voltages and currents are listed below..

```
--- Operating Point ---
V(1):    2       voltage
V(4):    -2      voltage
V(2):    2       voltage
V(3):    5       voltage
V(5):    0       voltage
I(H1):   -1      device_current
I(I1):   9       device_current
I(R4):   1       device_current
I(R3):   -7      device_current
I(R2):   2.22045e-016 device_current
I(R1):   2       device_current
I(G1):   8       device_current
I(V2):   -1      device_current
I(V1):   -2      device_current
```

The Python code generated the following network equations from the netlist. EIght equations are generated. Two of the equations are trivial and probably would not be even thought of if the equations were generated by hand.

```
Matrix([[Eq(I_V1 + v1*(1/R2 + 1/R1) - v2/R2 - v4/R1, 0)], [Eq(I_H1 + v2*(1/R4 + 1/R2) - v5/R4 - v1/R2, 0)], [Eq(-I_H1 - g1*v1 + v4*(g1 - 1/R3) + v3/R3, 0)], [Eq(v4*(1/R3 + 1/R1) - v3/R3 - v1/R1, -I1)], [Eq(-I_V2 - v2/R4 + v5/R4, 0)], [Eq(v1, V1)], [Eq(-I_V2*h1 + v2 - v3, V2)], [Eq(-v5, 0)]])
```

$$
\left[\begin{array}{c}
I_{V1} + v_1 \left(\frac{1}{R_2} + \frac{1}{R_1}\right) - \frac{v_2}{R_2} - \frac{v_4}{R_1} = 0 \\
I_{H1} + v_2 \left(\frac{1}{R_4} + \frac{1}{R_2}\right) - \frac{v_5}{R_4} - \frac{v_1}{R_2} = 0 \\
-I_{H1} - g_1 v_1 + v_4 \left(g_1 - \frac{1}{R_3}\right) + \frac{v_3}{R_3} = 0 \\
v_4 \left(\frac{1}{R_3} + \frac{1}{R_1}\right) - \frac{v_3}{R_3} - \frac{v_1}{R_1} = -I_1 \\
-I_{V2} - \frac{v_2}{R_4} + \frac{v_5}{R_4} = 0 \\
v_1 = V_1 \\
-I_{V2} h_1 + v_2 - v_3 = V_2 \\
-v_5 = 0
\end{array}\right]
$$

The network equations were copied to a different notebook for solving. The reason for using a different notebook is to keep the main notebook from getting too long and there maybe other plotting and numerical techniques that are required for some particular analysis.

Substituting the values for the components with the following sympy code.

```
equ1a = equ.subs({R1:2})
equ1a = equ1a.subs({R2:1})
equ1a = equ1a.subs({R3:1})
equ1a = equ1a.subs({R4:2})
equ1a = equ1a.subs({V1:2})
equ1a = equ1a.subs({V2:0})
equ1a = equ1a.subs({I1:9})
equ1a = equ1a.subs({h1:3})
equ1a = equ1a.subs({g1:2})
```

The network equations are:

$$\left[\begin{matrix}I_{V1} + \frac{3v_1}{2} - v_2 - \frac{v_4}{2} = 0\\I_{H1} - v_1 + \frac{3v_2}{2} - \frac{v_5}{2} = 0\\- I_{H1} - 2v_1 + v_3 + v_4 = 0\\- \frac{v_1}{2} - v_3 + \frac{3v_4}{2} = -9\\- I_{V2} - \frac{v_2}{2} + \frac{v_5}{2} = 0\\v_1 = 2\\- 3I_{V2} + v_2 - v_3 = 0\\- v_5 = 0\end{matrix}\right]$$

Using the solve function from sympy the unknowns are obtained.

solve(equ1a,[v1, v2, v3, v4, v5, I_V1, I_H1, I_V2])

{I_V2: -1, v4: -2, v1: 2, v3: 5, v5: 0, v2: 2, I_H1: -1, I_V1: -2}

These answers agree with LTspice. The python code to generate network equations is still in the validation stage. This example shows that much of the code is working properly.

References

1. Basic Electric Circuit Analysis <- Fix
2. LTspice, Linear Technology Corporation, retrieved October 6, 2017
3. Example48.ipnb located in github.