

For the current mirror simulation lab, in Procedures 2 and 3 you are required to obtain the input and output currents in each of the demonstration circuits. There are two approaches for doing this.

### Method 1

The first option is to insert meter sources in series with Rref and RL; these are DC voltage sources with 0V, like this:

```
Rref nvdd n1a 1k
Vm1 n1a n1 DC 0V
XM1 n1 n1 0 0 aldn
```

When NGspice completes the simulation, it will save the currents through all independent voltage sources, so you can access the current as "Vm1#branch". In exercise 3, after you perform a .OP simulation, you can access the currents like this:

```
print Vm1#branch
```

In Exercise 4, after you do a DC simulation, you can access the currents like this:

```
plot Vm1#branch
```

**NOTE:** print and plot commands need to be placed inside a **.control** block, or else you must manually enter them into the NGspice shell.

### Method 2

**!!!CORRECTION!!! The method described here only works for the .OP simulation! Do not use this method to prove currents in DC or TRAN simulations!**

The second method is to declare that NGSpice should save all currents for every device in the system. This method is convenient because it doesn't require modifying the actual circuit schematic. To use this method, add the line ".options savecurrents" at the top of your spice file, like this:

```
.options savecurrents
Rref nvdd n1 1k
XM1 n1 n1 0 0 aldn
```

Then, after the .OP simulation, you can access the current like this:

```
print @Rref[i]
```

After the .DC simulation, you can access the current like this:

```
plot @Rref[i]
```

There is one reliable way to measure currents in SPICE simulations: **use 0V meter sources on the branches where you want to measure currents.** This method works in all SPICE simulators and all simulation types.