

To obtain a suitable spice extraction file from Magic

1. Load your magic file into a magic window
(example used below is ring.mag)
Steps: tap magic [gives the magic window]
magic ring [gives a layout window]
2. obtain the .ext file
finish drawing the layout in the layout window
Steps: put the cursor in the layout window
type and enter :ext
leave layout by type & enter :q

This gives in the same directory ring.ext

3. obtain the .spice file
Step: in the magic window type and enter
ext 2 spice ring

This gives ring.spice

4. Edit .spice as .cir
Steps: backup as .cir file, type and enter
cp ring.spice ring.cir
Edit ring.cir, type and enter
emacs ring.cir &

Make the following changes

1. add bias voltages
Vcc Vcc 0 5V
Vss Vss 0 -5V
2. tell it what to run
.tran 1ns 100ns
3. change GND to 0
4. change W, L, PD, DS by multiplying by the value of λ , 0.8 μ ; multiply AD, AS by λ^2 , 0.64 μ
5. change nfet & pefet to your model (AM12L4N & AM12L4P) and copy your models in.
6. add capacitors
Cin1 in1 Vcc 3.54p
...
Cin5 in5 Vcc 3.54p

7. save and exit emacs, from the file menu

5. Run sparc

steps: top sparc [gives a sparc menu]

cd (to where the .cis file is)

enter sparc

enter ring.cis

enter run

after sparc finishes, see what is available

enter display

there should be in1, ..., in5

enter plot V(in1) V(in2)

this should give another window with
a plot of two of the voltages

enter quit