DIC L5: Layout (2)

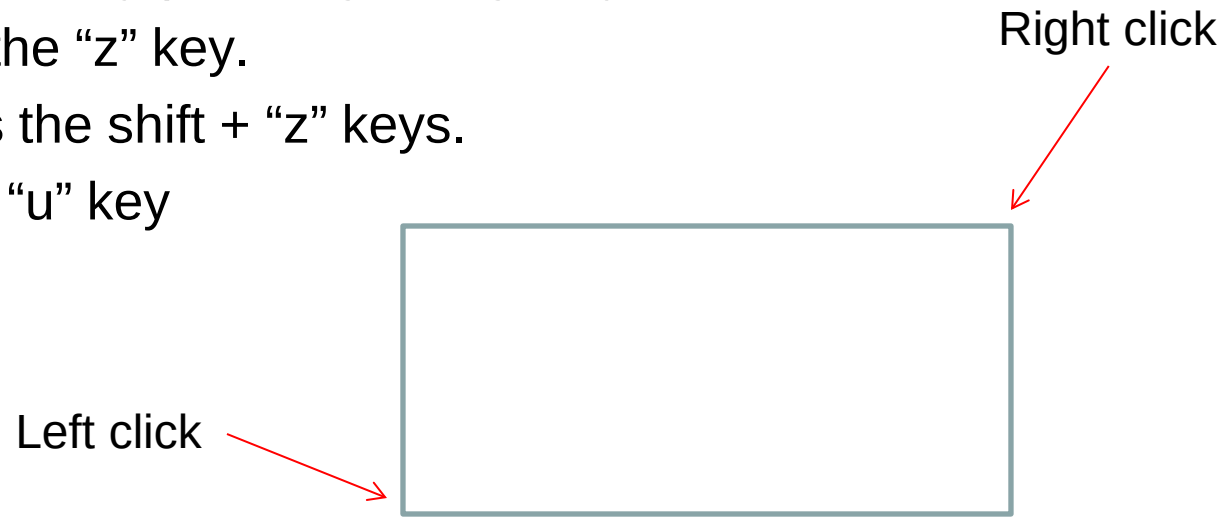
Sung-Min Hong (smhong@gist.ac.kr)

Semiconductor Device Simulation Lab.
School of Electrical Engineering and Computer Science
Gwangju Institute of Science and Technology

GIST Lecture on September 26, 2019

1.5. CMOS layout (9)

- Short summary about the Magic
  - Grid can be toggled by pressing the “g” key.
  - Zoom-in: Press the “z” key.
  - Zoom-out: Press the shift + “z” keys.
  - Undo: Press the “u” key
  - Draw a box.



  - The middle click paints the box.

1.5. CMOS layout (10)

- First layout example, NMOS
  - Type “magic -T scmos”
  - Draw a pwell.
  - Draw a polysilicon.
  - Draw a ndiffusion.
  - Draw two ndcontact’s.
  - Draw metal1 lines
  - Draw several psubstratepcontact’s.
  - Draw a polycontact.

1.5. CMOS layout (10)

- Label
  - Make a point by left-clicking and right-clicking at the same position.
  - Then, in the console window, label your nodes.
- Save
  - Save is save.
- Extract
  - extract: From *.mag file to *.ext file
  - ext2spice: From *.ext file to *.spice file

1.5. CMOS layout (10)

- Second layout example, PMOS
  - Repeat it with the opposite polarity.
  - For example,
    - Draw a nwell.
    - Draw a pdiffusion.
    - Draw two pdcontact’s.
    - Draw several nsubstratencontact’s.

Homework#1

- Layout of an inverter
  - Use the “-T scmos” option.
  - NMOS, W/L=4/2 (in terms of lambda)
  - PMOS, W/L=8/2
  - All other options are up to you.
- Extracted spice file
  - Attach the extracted spice file.

- Due: October 1, 2019 (Before the lecture starts)
  - Upload your Homework to our GitHub repository.

Spice3f5

- There are many SPICE variants.
  - For example, hspice from Synopsys
- Spice3f5
  - The last version directly from UC Berkeley
  - A GitHub repository can be found at
    github.com/hedhyw/spice3f5
  - “spice3 –b [filename]” would be needed.