



In this Project you will start layout design with Magic. But before that, you can check TinyTapeout Project:

TinyTapeout

TinyTapeout is a project for availing digital designers get their own chips with their own hardware. You can use a tiny space, which is **167 x 100uM** and put your design into it. By paying \$100 you get a chip including lots of different designs and one of them is yours. The last tapeout is Tapeout-5 and will end in 11 days.

Surely, it may be a bit late for this one but you can consider the next one if interested. It uses a preinstalled version of OpenLane framework and makes the usage of the framework much easier. You can check all details from:

<https://www.tinytapeout.com/>

Tiny Tapeout

Prepurchase space on Tiny Tapeout 05

- Each tile is approximately 167 x 100 uM and costs \$50.
- If you purchase multiple tiles, you can use them all for one design or split them across multiple designs.
- You can purchase additional tiles and PCBs at any time later (depending on availability).
- Purchases are **non-refundable**, cannot be carried over to future Tiny Tapeout shuttles, and are subject to [terms](#).

Number of tiles *
1

Number of PCBs *
1

Total: \$100 + Shipping

NEXT

Assignment

In this assignment, you will design an inverter using **Magic Layout Tool** and simulate it using **ngspice**:

1. Design your inverter having $4\lambda/2\lambda$ NMOS and PMOS transistors and implementing nwell, poly, n+, p+ and metal regions on Magic Layout Tool. Also implement the taps for p-substrate and n-well.
2. Convert that inverter into a Spice Deck that can be tested through Spice.
 - a. Test its functionality to see your design works correctly.
 - b. Test its speed for both high-to-low (H2L) and low-to-high (L2H) transitions at the output.
3. Now change the width of PMOS transistor to $8\lambda/2\lambda$.
4. Apply part 2 above and collect the results.

What happened to the functionality of the inverter, H2L and L2H delays of the inverter?
5. (Bonus) What happens when you do not put any taps?

Put all H2L and L2H delays and voltage transition plots into your report. Comment on the results.

Attention

- ☞ Next lecture you will get a tutorial that you need for solving that homework, so please attend the lecture.
- ☞ Put all your magic and spice files as well as your report into a zip file and post it to Teams Assignment Page before the due date. Each late day results in 20% degradation.
- ☞ There are multiple Magic Tutorials on YouTube. You can check them.
- ☞ Download and install Magic from <http://opencircuitdesign.com/magic/>
- ☞ Download and install Technology files from <http://opencircuitdesign.com/magic/archive/2002a.tar.gz>
- ☞ Download and install ngspice from <https://ngspice.sourceforge.io/download.html>
- ☞ We make our layout design at TSMC 0.25um process technology. Therefore, start Magic as:
`magic -T SCN5M_DEEP.12.TSMC.tech27`
- ☞ Convert Magic Layout to Spice Deck using the following commands in Magic:

```
: extract do all
: extract all
: ext2spice cthresh 0.001
: ext2spice rthresh 1
: ext2spice merge none
: ext2spice extresist off
: ext2spice
```
- ☞ But still you have to manipulate the output and include tsmc025 tech file for Spice simulation. Attend the lesson to learn. But the provided Spice Deck example file can help you learn too.
- ☞ In this assignment, the remaining files provided as `hw1_material.zip` and includes:
 - Spice tutorial (`tut_spice3_invertor.html`)
 - Spice Deck
 - tsmc025 technology file for Spice simulation

