# Table of Contents

1. Objective .................................................................................................................. 3
2. Setup & Preparation ................................................................................................. 4
3. Launching IC Studio ................................................................................................. 5
4. Opening the project ................................................................................................. 6
   1. Opening icstudio and opening the project ............................................................. 6
   2. Opening Layout Cell ............................................................................................. 6
5. Drawing the Layout ................................................................................................. 9
   1. Actual Inverter Layout ......................................................................................... 10
6. Adding Well Ties .................................................................................................... 17
7. Making Ports .......................................................................................................... 20
8. Save/Reserve Cell ................................................................................................. 21
1. Objective

This tutorial shows a step-by-step procedure for creating a custom (manual) layout of the inverter schematic that you created in Schematic entry tutorial. The final schematic is shown in the figure below.

It also helps you verify your layout by using the IC Station physical verification tools Calibre to perform simple DRC (Design Rule Check), and LVS (Layout vs Schematic) check.
2. Setup & Preparation

The set of directives listed below is applicable to users of the Engineering Design Center at Santa Clara University. If you are working in a different environment please check with your system administrator.

The steps below are necessary only for the first time to setup the Mentor Graphics environment by changing the settings in your .profile file. Add the following lines in your .profile:

```
setup mentor-2008.1
alias swd="export MGC_WD=$pwd"
```

Remember to execute

```
$ . .profile
```
3. Launching IC Studio

On the command line

- To create a directory to contain your projects type: “mkdir Tutorial”
- To change the current directory to Tutorial type: “cd Tutorial”.
- To open ICSTUDIO type: “icstudio”.

This launches the ICStudio window shown below.
4. Opening the project

To create a project follow the three steps given below:

1. Opening icstudio and opening the project

On the ICStudio Window

- Click File -> Open -> Project to create a new project.
- Enter the Project name (e.g. vlsi_tutorial) and the Project Location
- Click Open in the Open Project pop-up window

2. Opening Layout Cell

- Right click on the name of the Schematic you entered (e.g.; Inverter) and click New View.
- Select View Type as Layout.
- Click Finish.
The following window will open wherein you can draw your schematic layout.
Note: Before you make any changes to your work, make sure your work is in the 'Edit' mode.
5 Drawing the Layout

For this Click File > Enable editing > Current Context

NOTE: The layout will not be automatically drawn for you. You need to draw it on your own through scratch. The above figure is just for illustration. How to enable editing mode.
1. Actual Inverter Layout

Now that we are familiar with basic editing commands, let's start with the layout of the inverter.

Note: In the working space, one unit is equal to one Lambda (\( \lambda \)). (refer to the cursor coordination on the window frame). If you follow the Lambda rules for TSMC0.35u technology, the smallest feature size will be \( 2\lambda = 0.4\mu \) (for poly width and contact size). Therefore we have \( 1\lambda = 0.2\mu \) micron

- To Create the NMOS with 0.4 micron length and 1.2 micron width:
  - Click **ACTIVE** (43) on RHS of the window.
  - Click **Draw Rectangle** on the LHS of the window and draw a \( 6\lambda \times 6\lambda \) square of layer 43.
  - Next create **CONTACT_TO_ACTIVE** (48) \( 2\lambda \times 2\lambda \) Square.
  - Next create \( 6\lambda \times 6\lambda \) METAL1 Square for Contact.

. The result should look like in the following figure.
• Next Click **Edit> Paste**
• Place all these shapes beside the original ones with 4\(\lambda\) spacing. They are source and drain contacts.
• Create 6\(\lambda\) x 16\(\lambda\) ACTIVE (active area) to cover exactly source and drain metal contact.

Next
• **To Make the PMOS:**
  o Click **ACTIVE(43)** on RHS of the window.
  o Click **Draw Rectangle** on the LHS of the window and draw a 6\(\lambda\)X6\(\lambda\) square of layer 43.
  o Next create **CONTACT_TO_ACTIVE (48) 2\(\lambda\) x2\(\lambda\)** Square.
  o Next create 18\(\lambda\) x6\(\lambda\) **METAL1** Square for Contact.

• **Note: Distance between the NMOS and the PMOS should be at least 13\(\lambda\).**

• Select all the shapes and Click **Edit> Copy> Selected**
• Next Click **Edit> Paste**
- Place all these shapes beside the original ones with $4\lambda$ spacing. They are source and drain contacts.
- Create $18\lambda \times 16\lambda$ ACTIVE (active area) to cover exactly source and drain metal contact.

- Next create a $41\lambda \times 2\lambda$ poly between the drain and source contacts for NMOS and PMOS.
- Add $22\lambda \times 20\lambda$ P_PLUS_SELECT for PMOS and $10\lambda \times 20\lambda$ N_PLUS_SELECT for NMOS.
- The spacing between N_PLUS_SELECT and P_PLUS_SELECT should be kept at least $9\lambda$ for correct NWELL and PWELL spacing as shown below.
Note: It is always better to add as many CONTACTS as possible to reduce the contact resistance. So add 3 contacts in the Source area of PMOS (with 3\(\lambda\) distance in between the contacts) and 4 contacts in the drain area of PMOS (with 2\(\lambda\) distance in between the contacts) as shown in the adjacent figure.

*Note: As per the design rules of TSMC0.35, minimum separation between two contacts should be 3\(\lambda\).*

- Add 30\(\lambda\) x 28\(\lambda\) NWELL for PMOS and 18\(\lambda\) x 28\(\lambda\) PWELL for NMOS. The spacing between NWELL and PWELL should be at least 1\(\lambda\).
- Add METAL1: to form Vdd (power wire) and GND (ground wire). Note that METAL1 should have at least 3\(\lambda\) width and 3\(\lambda\) spacing (METAL1 to METAL1 spacing).

- Now, connect PMOS-source to Vdd and that of NMOS to GND with METAL1. (Use ‘Notch’ – For this select the shape you want to notch or connect and click Edit->Notch – Use your Left Mouse Button to alter Move the shape as desired and when you are done press Escape)
- Connect the drains of both transistors by METAL1, to get OUTPUT net of the inverter.
The figure when you have notched the Vdd, GND and POLY would look as under:
6. Adding Well Ties

- Use **Notch** to extend the left hand side edges of **WELL**, **ACTIVE** (active area), and **METAL1** of the **SOURCES** of both **NMOS** & **PMOS** by 8\( \lambda \).
- Add **N_PLUS_SELECT** inside **NWELL** and **P_PLUS_SELECT** inside **PWELL**. Also add contacts to both **N_PLUS_SELECT** and **P_PLUS_SELECT** with 3\( \lambda \) spacing.
- Notice that the PMOS's source is connected to Vdd and NMOS's source is connected to GND. By connecting the body of the wells to the sources through metal1 we are connecting the NWELL to Vdd and the PWELL to GND.

The final figure when you have notched the Vdd, GND and POLY would look shown:
7. Making Ports

We will use METAL1 for all ports, so the first thing we must do is route the POLY gate to a suitable METAL1 trace.

- Select CONTACT TO POLY (47) from the RHS layer palette.
- Create a $2\lambda \times 2\lambda$ square of CONTACT TO POLY on the left side of the POLY.
- Select the POLY trace and use the NOTCH tool to surround the contact by $2\lambda$ on each side.
- Add a square of METAL1 over the POLY/CONTACT TO POLY pad

The figure should look like the drawing below:
Labeling Ports:

- **Click** on the top Metal polygon (Drain of the PMOS) to select it.
- From **main menu bar**, select **Connectivity -> Port -> Make Port**:
- For the **Port Name**, enter **VDD**. Make sure the **Port Type** is **Signal and Direction** is in. Then click OK
- Repeat the steps above to assign GND, IN and OUT ports.

**Note:** ‘GND’ and ‘IN’ are of ‘in’ Direction while ‘OUT’ is of ‘out’ Direction.
- Select **METAL1.PORT (2)** on RHS of the window.
- Select, from **main Menu Bar, Add -> Text**
- Enter the label **VDD** and place the text object on the VDD metal line.
- Repeat for the **OUT, GROUND, and IN** metal lines.

**Note:** Be sure to label the GND port as **GROUND**. This name is required for the LVS check to work correctly.
8. Save / Reserve Cell

To save Cell:
- Select, from main Menu Bar, File -> Cell -> Save Cell

Note: After you save your work, the system will automatically set your work to 'Read-Only' mode. To set your work to 'Editable' mode, Click File> Enable Editing>Current Context

Your final cell should match the one shown below:
9. DRC/LVS

To Run DRC:

- File → Cell → Set Logic Source

Mentor Graphics window Pops up

- Select → Schematic → OK
From Menu bar:

Click Calibre → Run DRC

Calibre Interactive_nmDRC window pops up

Select Run DRC on left panel
**LVS:**

From Menu Bar:

Click on **Calibre → Run LVS**
Calibre Interactive _nmLVS eindow pops up

Select Run LVS (on left panel of window)
If your LVS pass out you get a smiley on the caliber_LVS RVE window as shown below.