I. Introduction

This document contains a step-by-step tutorial for doing the layout of an inverter in the Mentor Graphics tool IC Station.

II. Starting IC Station

1) Invoke 'Design Manager' by typing:

   $ dmgr &

   The window shown below will appear.

2) Double click on ic.

3) In the IC Station window that appears, from the session palette on the right, select 'Create' to start a new layout. A dialog box will appear.

   Enter the name of the device you will layout in the "Cell Name" box. This can be any name you like, it need not be the schematic name. In this example, enter 'inverter'. Now proceed to set up the process files as per the steps in the next section.

   For an existing cell, in the window use the Navigator to go to the directory you created and type the cell name. Then click OK.

III. Setting up the process files

   In the same window (see Figure below)
1) Enter "$ADK/technology/ic/tsmc035" into both the "Process" and "Attach Library" fields.

2) Enter "$ADK/technology/ic/tsmc035.rules" into "Rules File" field.

3) Then click OK. A new layout sheet window will appear.

Note: You will see an empty sheet in the window. This sheet is where you will draw the layout of your schematic by drawing polygons and routing the wires.

IV. Editing

Note: Before you make any changes to your work, make sure your work is in the 'Edit' mode. Please refer to section for steps in detail.

Select 'Easy Edit' on the top of the right hand side 'IC Palettes'. A new palette titled 'Easy Edit' will appear.

1) Setting up the palette

It will be convenient to show layer palette along with the 'Easy Edit' palette when adding layers. To show the layer palette select from the main menu bar:

Other -> Layers -> Show Layer Palette

A dialog box titled 'Show Layer Palette' will appear as shown in the following figure.
Select following layers from the layer palette and click OK.

- N_WELL
- P_WELL
- N_PLUS_SELECT
- P_PLUS_SELECT
- POLY
- ACTIVE
- METAL1
- METAL2
- METAL3
- CONTACT
- CONTACT_TO_ACTIVE
- CONTACT_TO_METAL
- CONTACT_TO_POLY
- VIA
- VIA2

Other layers like metal4, metal5, via3, via4 etc. may be required for the complex layout. But for the inverter layout or other simple layouts the above layers are sufficient.

2) Adding Shapes/Text

To add shapes

a. Select 'Shape' from 'Easy Edit' palette, a prompt bar will appear at the bottom of the IC Station window. Select 'nwell' from layer palette to set layer to NWELL.

b. In the work sheet, click and drag to size the rectangle you want, then release mouse button to create rectangle shape.

c. The created shape will be automatically in 'Selected' mode -- which means shapes with bold lines. Press 'F2' to 'Unselect' all selected shapes and you will see them in original style.
d. Repeat the previous steps to create shapes in every layer.

**To add text**

a. Set layer to 'text' or 'metal1.text' or 'metal2.text' by selecting from 'Layer Palette'.

b. Select 'Text' from the 'Easy Edit' palette, then key in the content in the display prompt bar and press 'Enter'. Move the cursor to locate that text and then click to place it.

3) **Moving and Copying Shapes**

While editing layouts, you may often want to move and copy existing shapes.

a. Make sure nothing is selected by pressing 'F2'. Select only the shapes you will move/copy by clicking on the edges of these shapes one at a time. The selected shape will be in bold line style.

b. Select 'Move' or 'Copy' on the 'Easy Edit' palette. Then click on the work sheet at the location where you want to place them.

4) **Modifying Shapes**

There are two convenient ways to modify existing shapes. One is 'Notch', and the other is 'Move Edge'.

**Notch**

Make sure nothing is selected by pressing 'F2' then select one that you want to edit. After that select 'Notch' on 'Easy Edit' palette; a prompt bar appears at the bottom of the IC Graph window. Select the shape you will modify by clicking on one of its edges.

Click and drag a rectangle **inside** that shape. The area covered by the rectangle inside that shape will be **notched out**.

Click and drag a rectangle **from inside** to outside that shape. The area covered by the rectangle outside that shape will be **notched in** -- merged to become part of that shape.

**Move Edge**

Make sure nothing is selected by pressing 'F2'. From main menu bar

Select -> Select -> Edge:

A prompt bar will appear in the bottom of the IC Station window. Select the edge you want to be **stretched**. Select 'Move' on the 'Easy Edit' palette. In the work sheet, click where you want to place that edge.

V. **Example: Creating an inverter layout**

The following steps show the steps required to create an inverter.

**Note:** In the working space, **one unit is equal to one Lambda (λ).** If you follow the
Lambda rules for 0.4 micron technology, the smallest feature size will be $2\lambda$ (for poly width and contact size). Therefore we have $\lambda = 0.2\text{micron}$

1) **Drawing the basic transistor**
   a. Use “CONTACT_TO_ACTIVE” layer and create one $2\lambda \times 2\lambda$ contact square.
   b. Also create $6\lambda \times 6\lambda$ METAL1 square for contact. Your layout should appear as in the figure below.

   ![Diagram of basic transistor](image1.png)

   c. Copy and place all these shapes beside the original ones with $4\lambda$ spacing. They are **source** and **drain contacts**.

   ![Diagram of copied shapes](image2.png)

   d. Create $2\lambda \times 10\lambda$ POLY between these two contact-metals with $1\lambda$ spacing.
e. Add $10\lambda \times 20\lambda$ P_PLUS_SELECT for PMOS and $10\lambda \times 20\lambda$ N_PLUS_SELECT for NMOS. The spacing between P_PLUS_SELECT and N_PLUS_SELECT should be kept at least $9\lambda$ apart for correct N_WELL and P_WELL spacing.

f. Create $6\lambda \times 16\lambda$ ACTIVE (active area) to cover exactly source and drain metal-contact.

g. Add $18\lambda \times 28\lambda$ N_WELL for PMOS and $18\lambda \times 28\lambda$ P_WELL for NMOS. The spacing between N_WELL and P_WELL should be at least $1\lambda$. 
h. Draw a PMOS transistor in a similar fashion. The P_PLUS_SELECT needs to be 10\(\lambda\)x20\(\lambda\) for the PMOS device. The spacing between P_PLUS_SELECT and N_PLUS_SELECT should be kept at least 9\(\lambda\) apart for correct N_WELL and P_WELL spacing. Add 18\(\lambda\)x28\(\lambda\) N_WELL for the PMOS device. The spacing between N_WELL and P_WELL should be at least 1\(\lambda\).

Your layout after you are done drawing the two transistors should be as in the following figure.
i. Extend and connect the **POLY** layer from both transistors. Connect the **drains** of both transistors by **metal1**, to get the output net of the inverter. Your layout will look like the following figure.
j. Add METAL1 : VCC (power wire) and GND (ground wire). Note that METAL1 should have at least 3 Lambda width and 3 Lambda spacing (metal1 to metal1 spacing).

k. Now, connect PMOS-source to VCC and that of NMOS source to GND with METAL1. Extend and connect the POLY layer from both transistors then connect the POLY layers of both transistors by METAL1.

Your layout will look like the following figure.
1. For well ties continue your layout with the following steps.

   Use 'Move Edge' to extend the WELL and ACTIVE regions.

   Add N_PLUS_SELECT inside N_WELL to provide a n-well tie to power. Add P_PLUS_SELECT inside the P_WELL to provide a substrate tie to ground.

   Add METAL1 and CONTACT_TO_ACTIVE to both N_PLUS_SELECT and P_PLUS_SELECT and connect them to VCC and GND respectively.
Notice that the **PMOS**'s source is connected to **VCC** and **NMOS**'s source is connected to the **GND**. By connecting the body of the wells to the sources through metal1 we are connecting the Nwell to VCC and Pwell to the GND.

2) **Naming the ports**

Assigning port names is to make the netlist more recognizable for Layout VS. Schematic (LVS) check and Layout Extraction. Using exactly the same names as those you gave in the schematic is highly recommended. If you don’t, the tool will try to recognize each port by categorization and even guessing, you will inevitably get warnings/errors.

To assign port name:

From 'IC Palettes', select 'Easy Edit'.

DAIC__________________________ ______________ 11 ________________________________ ______________
Reserve your work for edit by selecting from main menu bar:

**File -> Cell -> Reserve**

In your work sheet, select the metal1 you want to be 'VCC' port. In this example, the 'VCC' port should be the top-most metal1.

From main menu bar select

**Connectivity -> Port -> Make Port:**

A prompt dialog bar will appear in the bottom of window.

![Port Make Window](image)

For the 'Port Name', enter 'VCC'. Make sure the 'Direction' is 'in'. Then click 'OK'

Repeat the steps above to assign 'GND', 'IN', 'OUT' ports.

**Note:** 'GND' and 'IN' are of 'in' Direction while 'OUT' is of 'out' Direction.

3) Save your schematic

4) **You are now ready to run a design rule check.**