Tutorial

MICRO MAGIC, INC.

MAX LAYOUT ENVIRONMENT

VER. 5.7

PART 1

INTRODUCTION TO MAX

INTRODUCTION

Welcome to MAX, the full-custom layout editor from Micro Magic, Inc. MAX is more than just a simple layout editor. It is a complete layout environment.

MAX FEATURES

- Optimized for large databases
- Very Fast Redisplay for Whole Chip Viewing and Editing
- Interactive viewing and editing of hierarchical layout
- Continuous DRC feedback during layout
- Hierarchical and incremental DRC
- Built in netlisting
- Interactive connectivity tracing
- Interactive wiring tool with flylines to show connections not yet completed
- Layout generator for gates (using MAX-LS)
- Generators for large regular structures such as SRAMS, ROMs, PLA's, and DRAM's (with the optional MegaCell Compiler)
- Smart palette for easy control and feedback on layers
- Full customization and extension via Tcl/Tk scripting language and API
- Technology independence via technology description files
- Reads/writes GDSII
- True transparency through layers
- Representational 3D rotation view
- True 3D Thru-Silicon Via Wafer Level editing (using MAX-3D)
- Calibre® interface can display Calibre® DRC errors on layout view (Optional feature)
- Runs on Linux and Solaris-10 operating systems.

This tutorial will introduce you to many of these features. Consult the Micro Magic, Inc. *MAX User Manual* and on-line documentation for further information.

TUTORIAL OBJECTIVES

In this tutorial for MAX Layout Editor, you will learn how to:

- Understand MAX Layout screen
- Navigate around the layout
- Push into and Pop out of sections of layout
- Check net connectivity
- View DRC errors with MAX or check for Calibre® DRC errors
- Create layout with MAX_LS
- Create and edit custom layout
- Load GDSII layout files
- Create MAX technology files

The tutorials for MAX-3D and MAX-LS (separate licenses are required) address specific issues pertaining to these products.

INSTALLING MAX

If MAX is already installed at your site, skip this section.

If you are reading this tutorial, you should have already downloaded the MMI software. This tutorial assumes that you have already received a license file for MAX. The following information is also found in the `README` file that comes with the software.

INSTALLING MAX

Step 1 ⇨ Create an installation directory and `cd` to it.

```
cd <install_dir>
```

Step 2 ⇨ Put the release in the install directory. The release file `mmi_090831.tgz` listed below is an example. Use the file which you have downloaded.

```
cp  <where_ever_you_put_the_release>/mmi_090831.tgz .
gtar xzvf mmi_090831.tgz
```

- If you do not have access to `gtar` (or `tar` on Linux) but do have `gunzip`, then type:

```
gunzip mmi_090831.tgz
tar xvf mmi_090831.tar
```

TIP:

Note that moving the release around with "`cp -r`" is not a good idea, since `cp` can screw up symbolic links and permissions. Just `untar` it where you want it to live. If you absolutely need to copy it, use "`cp -a`".

Step 3 ⇨ Make a symbolic link from `<install_dir>/mmi` to the `<release_dir>`.

```
ln -s <release_dir> mmi
```

From the example above the `<release_dir>` is `mmi_090831`. You would type:

```
ln -s mmi_090831 mmi
```

This allows you to refer to the release simply as `mmi`.

When you wish to upgrade to a new release, all you need to do is `untar` the new release into your `install_dir` and then change the "`mmi`" relative link to point to the new release.

Step 4 ⇨ Create your initial `mmi_local` directory as follows:

```
cp -a mmi/mmi_local.sample mmi_local
```

All site specific data is kept in `<install_dir>/mmi_local`. That way you can upgrade to new releases without losing or having to move around local data.

Step 5 ⇨ Put the license file where MAX can find it.
One place MAX automatically looks for the license file is the `mmi_local` directory. The file needs to be called `mmi_license.lic`

```
cd mmi_local
cp <license file> mmi_license.lic
```

Step 6 ⇨ Start the license server for for floating license: (If you have a nodelocked license, skip this section.)

```
Nohup mmiserver -f /full_path/mmi_license.lic -l
mmi_server_log &
```

- To have the license server automatically start up when the machine is rebooted, add the following line to `/etc/rc.d/rc.local`:

```
mmiserver -f /full_path/mmi_license.lic -l mmi_server_log &
```

Step 7 ⇨ Add path and environment variable info to your `.cshrc` or `.login` file. This assumes you're using `csh` or `tcsh`. Add the following lines to the file:

```
setenv MMI_TOOLS <install_dir>/mmi
set path=(${MMI_TOOLS}/bin.i486-linux $path) #For Linux
set path=(${MMI_TOOLS}/bin.sparc-solaris5.10 $path) #For Sun
Solaris
```

Step 8 ⇨ Source your `.cshrc` file.

You are now ready to run MAX and start the tutorial.

STARTING UP MAX

Before we get started make sure someone has already installed MAX at your site.

INSTALL TUTORIAL To run the tutorials, you need to make a personal copy of the MAX tutorial example files using `mmi_tutorial`.

Step 1 ⇨ **To install the MAX tutorial**, type the following at the UNIX prompt:

```
mmi_tutorial
```

Make sure that the appropriate tool (MAX) is selected and then click on **Install Tutorial**.

The default installation directory is "`mmi_private/tutorial`" in your home directory. You can also select a different directory for installation. Don't worry if the directory doesn't exist, the `mmi_tutorial` script will make it if it can. You can also use this to reinstall a clean copy of the tutorial.

Step 2 ⇨ **To start the tutorial**, you must first "**cd**" (change directory) to the directory that you installed the tutorial into. If you used the default directory, type:

```
cd ~/mmi_private/tutorial/max
```

Otherwise, replace the above directory with the directory you selected. This directory contains files that you will need to run the tutorial.

Step 3 ⇨ **Then to start MAX**, simply type:

```
max -tech mmi25
```

A window should come up that looks something like Figure 1-a.

MAX lets you customize all of the layer colors including the background color.

If you are used to working with a black background, you can add "`-bb`" to the end of any MAX technology name. A separate palette is created for a black background with the layer colors adjusted for visibility (shown in Figure 1-b).

⇨ So for this tutorial, to use a black background you would type:

```
max -tech mmi25-bb
```

- ***NOTE:*** The bulk of this tutorial will be illustrated using MMI default colors for clarity and ease of reading.

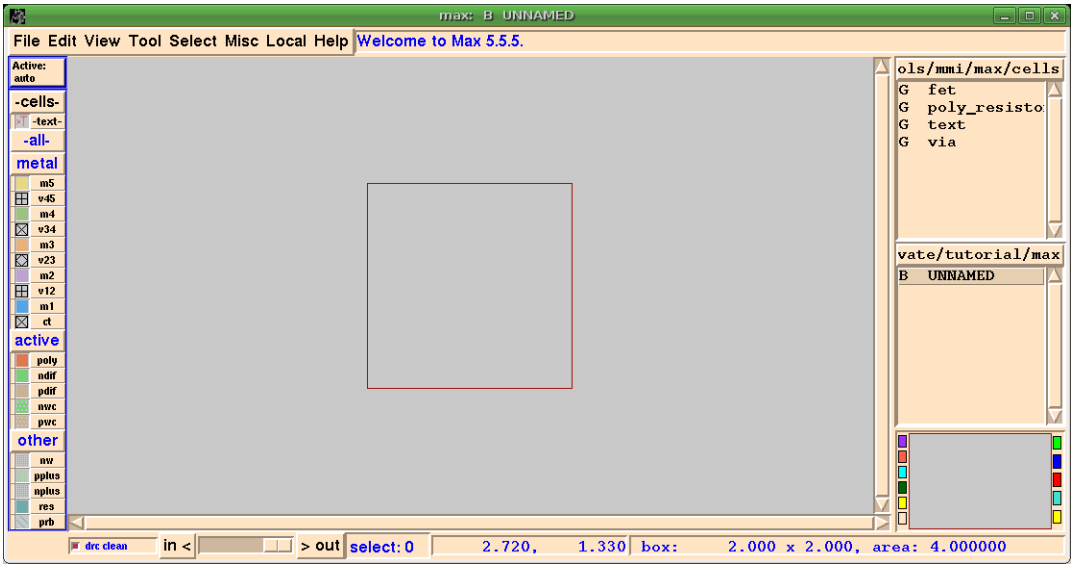
TIP:

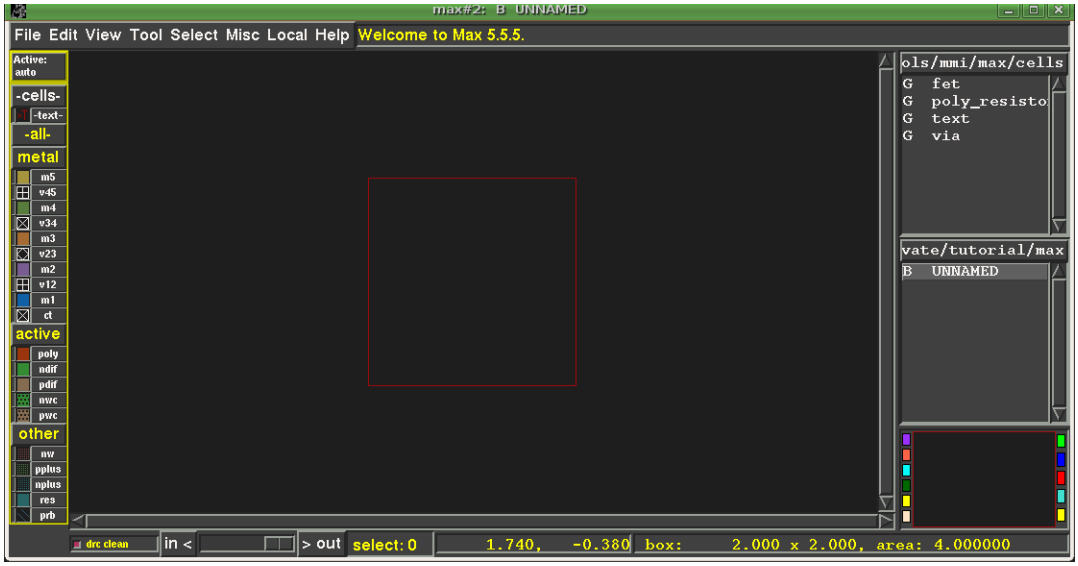
If you have just installed MAX, the the default technology is mmi25, so you would only need to type: **max**

If you want MAX to come up with a different technology by default, you can set the environment variable MAX_DEFAULT_TECH.

The rest of this section of the tutorial will go over each element of the MAX window.

Figure 1: MAX Window a) MMI Default Colors; b) Generic Black-Background Colors





MAX SCREEN LAYOUT

On the very top of the window the title bar should say "**max: B UNNAMED**".

"**UNNAMED**" means that you have *not specified the file* (cell) you wish to edit. The "**B**" tells you that the cell resides only in the memory buffer.

Next, across the top you should see the menu bar which contains the following menu items: **File**, **Edit**, **View**, **Tool**, **Select, Misc**, **Local**, and **Help**. These are pull down menus much like any other application.

Directly to the right of the **Help** menu is the **MAX Message Area**. Depending on the location of the cursor, it currently says something like "`main mode, (shift)-BUT-1 selects (more), BUT-2 pans, (Control)-BUT-3 moves (stretches) selection`". The **MAX Message Area** is a sort of mini-help feature. It tells you what each mouse button does, and when you have pulled down one of the menus from the menu bar, the message area will give you a short explanation of what the highlighted menu item does.

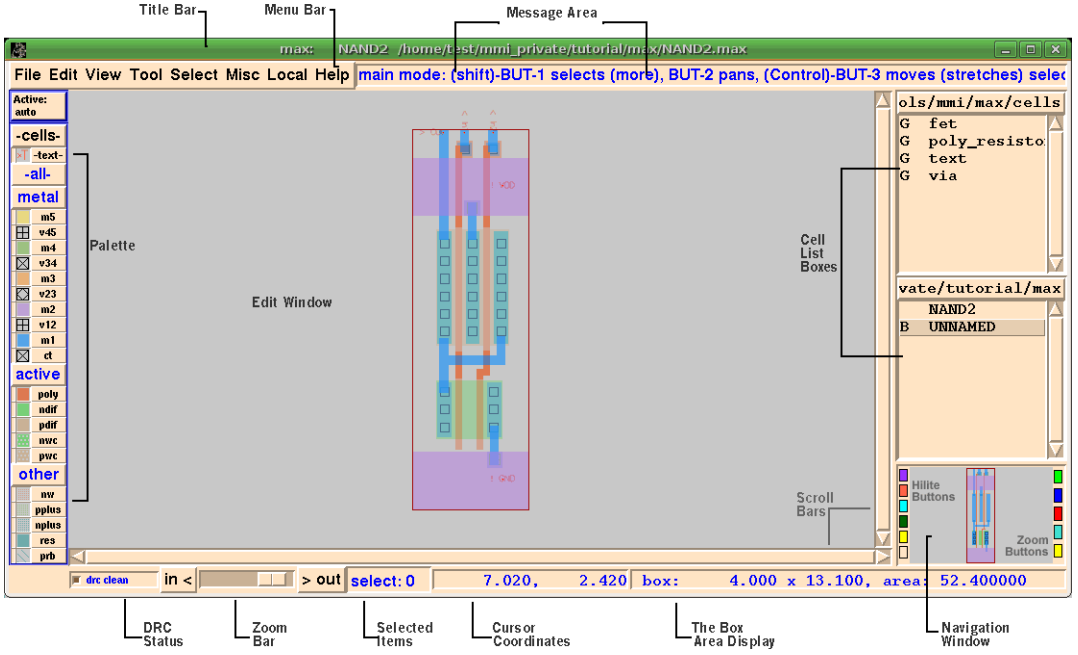
TIP:

In each of the menus, hotkeys are listed to the right for those commands that have hotkeys. You can always find out what the current hotkeys are by selecting "**Current Hot Keys**" from the **Help** menu or using the **SPACE** hotkey. Later in the tutorial we will cover how to change hotkeys.

The remainder of this section of the tutorial goes over each area of the MAX window. Refer to the *MAX User Manual* for a more detailed description of the MAX window.

Figure 2 points out the different attributes of the MAX window.

Figure 2: Items in MAX Window



Cell List Boxes

Down the right side of the window are small **List Boxes**.

Each List Box displays the cells in a given directory. That directory can be the user's “working directory” or a library.

Additional List Boxes can be added (or boxes deleted) by:

- Clicking the left mouse button (**Button-1**) over the List Box header (the horizontal bar saying: `demo/tutorial/max` in Figure 2), then,
- Clicking the left mouse button (**Button-1**) on either **Make new list box,** or **Close this list box**.

The directory the list box points to can also be changed by clicking on the desired directory. The directory list includes the directories of all cells that are currently loaded into MAX.

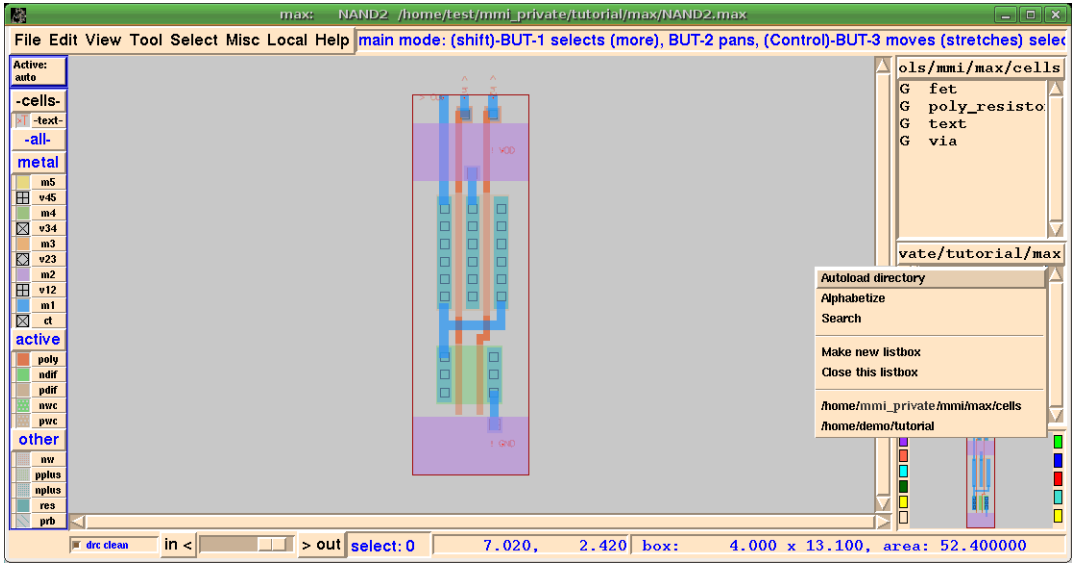
By default, no cells are loaded into MAX if not specified on the command line.

AUTOLOAD CELLS INTO LIST BOX

To load all of the cells in a paticular directory, you autoload the directory. Let’s try this:

Step 1 ⇨ Hold down the **LEFT mouse button** over the bottom list box directory and select **Autoload directory** as shown in Figure 3.

Figure 3: Autoload Directory from Cell List Box



You should now see the cells **drc_test**, **test_cell**, and **UNNAMED** in the cell list box. **INV**, **NAND2**, and **row** are in a different technology (mmi18), so you will get a warning message stating this.

Your `max.rc` file can specify library directories to be autoloaded at startup time using the `max_auto_load` command.

The `max.rc` file contains several other useful pieces of information. (See the ***Micro Magic, Inc. MAX User Manual*** for more details.)

TIP:

The commands and variables available to use in your `max.rc` file can be found by selecting **Text Commands/Variables** from the **Help** menu or using the **?** hotkey. If you click on one of the commands, a definition and syntax for that command is listed. If you select a variable, a description and the current value is listed. The syntax for setting variables is:

```
set <variable> <value>
```

You can also adjust the width of the list boxes.

Step 2 ⇨ Move the mouse pointer over the vertical bar just to the left of the list box. The mouse will change to a small horizontal line with arrows on both ends.

Step 3 ⇨ When the arrows appear, adjust the list box width by holding down and dragging with **Button-1** to the left or right.

Navigation Window

Below the list boxes is the **Navigation Window**. For large designs, you can use the **Navigation Window** to control the view you see in the main **MAX window**.

The **Navigation Window** shows a copy of the layout; the **brown outline** represents the boundary of the MAX Window. A **red outline** represents the "Box". The "Box" is used for editing, and is used to define regions used by other commands in MAX.

TIP:

Holding down the left mouse button (**Button-1**) and dragging out a region in the **Navigation Window** will cause the main window to zoom to that region.

The small colored (green, blue, red, cyan, yellow) rectangular buttons on the right of the **Navigation Window** allow you to do several things.

- The topmost button (green by default) steps through the view stack. **Button-1** zooms backwards in the view stack, **Button-3** zooms forward in view stack.
- The other buttons allow you to save (remember) and return to views — a different view for each. Position the mouse cursor over the buttons, and read their function descriptions in the **Message Area**.
  - For example, if you are at a view you know you want to return to, simply click the right mouse button (**Button-3**) on the blue button to lock in and remember that location and zoom factor.
  - If you later wish to return to that view you simply click the left mouse button (**Button-1**) on that blue button to recall that view.
- The top button always zooms back and forth in the view stack. If you desire, you can change the number of buttons and their colors. Both the number of view-save buttons, and their colors, can be changed in your `max.rc` file by setting the `ZOOM_BUTTONS` variable.

CHANGE ZOOM BUTTONS

Let's try changing the color of zoom buttons.

Step 1 ⇨ Exit MAX (hotkey: **Ctrl-d**).

- Go to the command window (the shell window where you started MAX).

Step 2 ⇨ Create a `max.rc` file in your current directory (default is `~/mmi_private/tutorial/max`).

Add the following line:

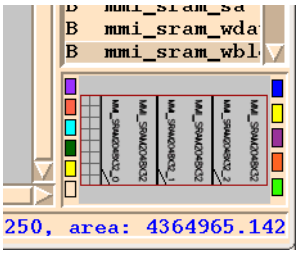
```
set ZOOM_BUTTONS {blue yellow purple orange green}
```

Step 3 ⇨ Restart MAX. From the shell window, type:

```
max -tech mmi25
```

Your Navigation Window should now look like Figure 4.

Figure 4: Navigation Window Zoom Buttons - New Colors



HILITE BUTTONS

The buttons along the left side of the Navigation Window are used to **highlight** items you have selected. Whenever you have selected objects, for example a net, you can highlight them in the layout with the specified colors.

LOAD EXAMPLE LAYOUT

To cover the other portions of the MAX window, we will load in an example GDSII file.

Step 1 ⇨ Exit out of MAX by going to the **File** menu and selecting **Exit** or use the hotkey **Ctrl-d**.

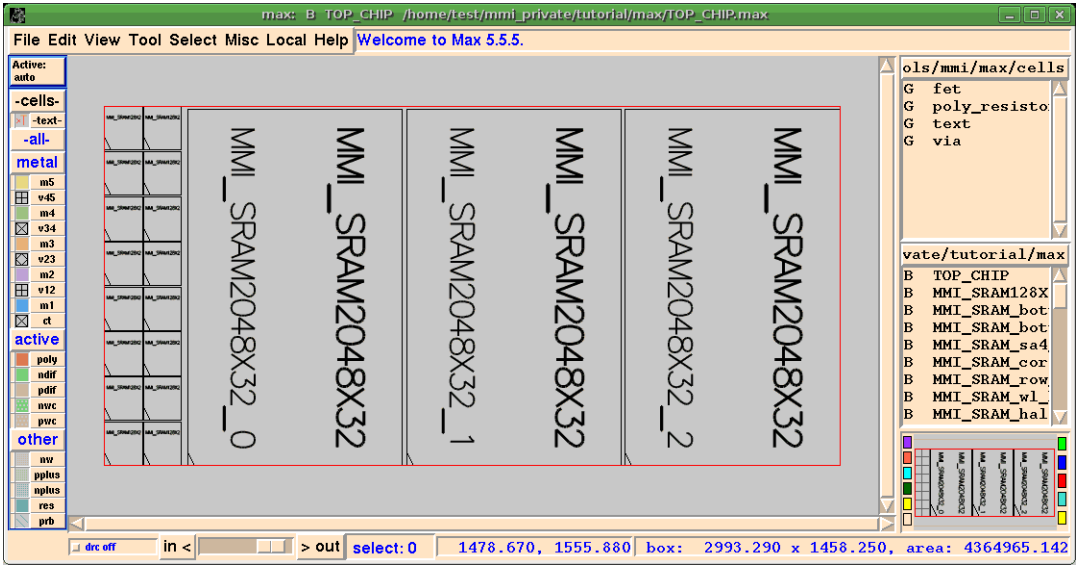
The rest of the tutorial will be using the default zoom button colors, so if you want MAX to match, you can delete the `max.rc` file which you created earlier.

Step 2 ⇨ In your shell window (make sure you're still in the correct directory), type the command:

```
max -tech mmi25 TOP_CHIP.gds
```

You should see the example layout shown in Figure 5. This block was created in MAX using the **mmi25** technology and then exported to a GDSII file.

Figure 5: Example Layout Loaded into MAX (default colors)



VIEW INTERNALS

This GDSII file is made up of a number of SRAMs. The top level cell has over **1 million** transistors. By default, MAX shows the top level of the hierarchy.

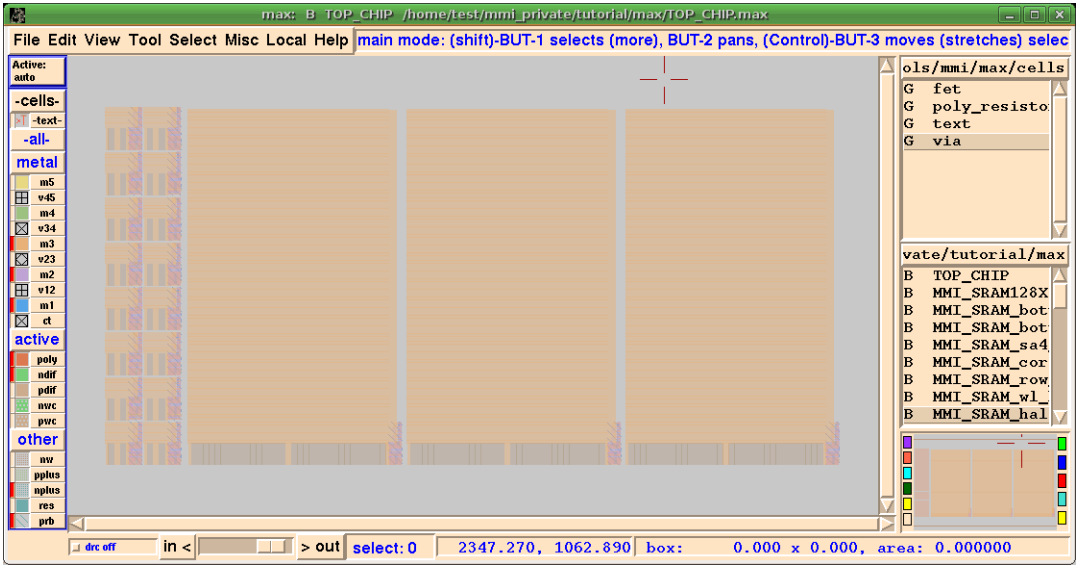
Step 3

⇨ View all levels of hierarchy by typing the hotkey "**i**" or selecting "**Internals, View All**" from the **View** menu.

Your MAX window should now look like Figure 6. All layers and all levels of hierarchy are visible.

Figure 6-b shows this same layout loaded using generic black-background colors.

Figure 6: Layout of TOP_CHIP.gds with "View Internals, All": a) Default colors; b) Black Background colors



We've already covered the **Cell List Box**. Notice now though that there are many cells listed.

USING THE LIST BOX ⇨ Load the cells into the MAX window by clicking the **right (Button-3)** mouse button on the desired cell name. This is noted in the **MAX Message Area** at the top of the window.

Step 1 ⇨ Try loading cells by clicking on cells with the **right (Button-3)** mouse button in the cell list. If you accidently use the **left (Button-1)** mouse button, **Ctrl-C** will cancel.

- To get back to the top level, click on **TOP_CHIP** at the beginning of the list with the **right (Button-3)** mouse button.

TIP:

If you click with the **left (Button-1)** mouse button on a cell, MAX adds an instance of that cell to your current cell. We'll try this later in the tutorial. Don't worry if you make a mistake. You can always undo a command with the hotkey **u**.

DRC Control and Status

In the lower left corner is the **DRC Status Box**. It tells you the current status of the continuous interactive DRC which runs in MAX. The **DRC Status Box** also has a small radio button where you can click to turn the interactive DRC on or off. If there are DRC errors, the **DRC Status Box** will report the number of DRC errors.

Step 1 ⇨ Because we read in a GDSII file, MAX defaults to **drc off**. Click on the radio button and it may report some DRC errors.

Number of Items Selected

To the right of the DRC Control and Status area is a box that shows the number of items selected. The default is to display ">100" if more than 100 items are selected. By default any item (gcells, instances, rectangles, etc.) selected is displayed. This can be changed in your `max.rc` file.

TIP:

To find commands and variables, you can select **Text Commands/Variables** from the **Help** menu or use the **?** hotkey. In the search field, if you type in "`select`", you'll find that the variables mentioned above are: `SELECT_DISPLAY` and `SELECT_MAX_DISPLAY`.

Scroll and Zoom bars

Scroll Bars are laid across the bottom and right side of the **Layout Window** (and on the right side of the List boxes). These work like the scroll bars in most other applications.

Below the bottom scroll bar is the **Zoom Bar**. Pulling the slider button to the left zooms in, and to the right zooms out.

SCROLL WHEEL ZOOM

In addition, the mouse scroll wheel can be used to zoom in and out. Scroll wheel zooming is centered around the cursor position.

Step 1 ⇨ Try zooming around using the scroll wheel on your mouse. Place the cursor in the upper left corner of the cell and move the scroll wheel forward to zoom in. Zoom out by moving the scroll wheel down. To get back to the **full view** of the cell, type the **v** hotkey.

- If you hold down the **Shift** key while while using the **scroll wheel**, MAX pans left and right. If you hold down the **Ctrl** key while using the **scroll wheel**, MAX pans up and down.

Using the Navigation Window

SAVING ZOOM VIEWS

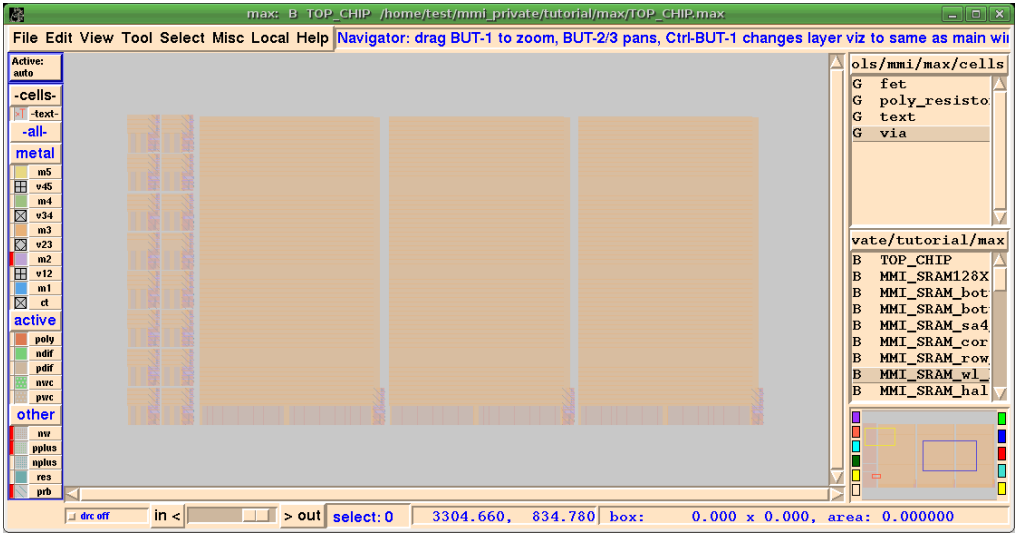
The **Navigation Window** can be used to zoom to particular areas, save views, and save highlights.

Step 1 ⇨ In the Navigation Window, drag out a box by holding down the **left (Button-1)** mouse button. Notice that the main MAX window zooms to this area.

Step 2 ⇨ Save this zoom by moving over to the colored buttons on the right edge and selecting the second button (blue by default, yellow if you changed it earlier) with the **right (Button-3)** mouse button. Notice that view area in the Navigation Window is now surrounded with a (blue or yellow) box.

Step 3 ⇨ Zoom around to some other areas and save their zoom views using the buttons on the right edge of the Navigation Window. You should see something similiar to Figure 7. Notice that the zoom areas you saved are shown in the Navigation Window. You can return to these zoom views by clicking with the **left (Button-1)** mouse button.

Figure 7: Zoom Views Saved in Navigation Window



Step 4 ⇨ Push into one of the smaller cells by clicking on it with the **right (Button-3)** mouse button in the cell list.

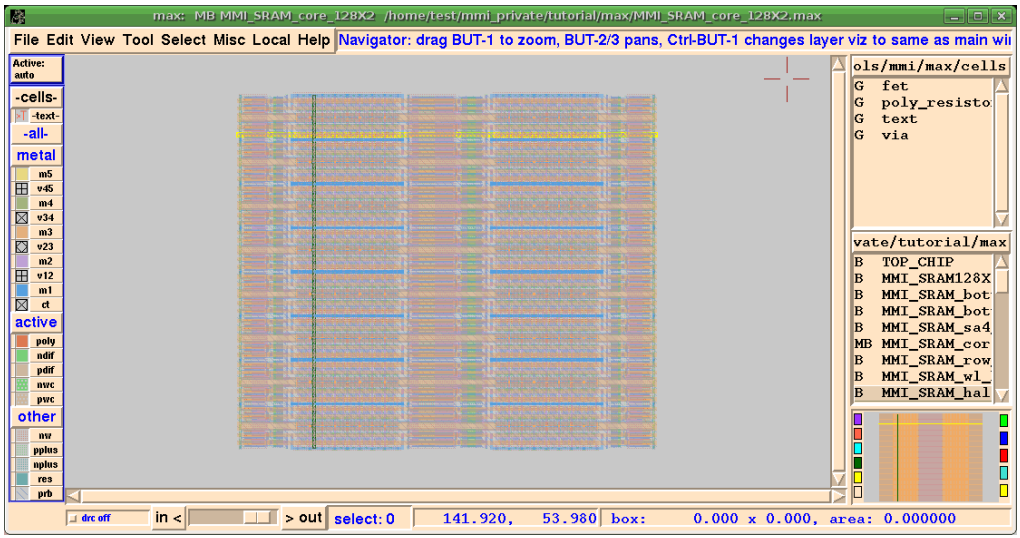
Step 5 ⇨ Select a net by moving the mouse over part of the net and typing the **s** hotkey for **Select Net**. (We'll cover this in more detail later in the tutorial.) If you select a large net (i.e. `vdd` or `gnd`) a popup will appear that will allow you to stop the selection.

Step 6 ⇨ Save this selection by clicking on the top colored button at the left edge of the navigation window with the **left (Button-1)** mouse button. Notice that the net is now highlighted in yellow (default color) both in the

Navigation Window and the main MAX Window. Repeat this for some other nets.

You should now see something similiar to Figure 8.

Figure 8: Highlighted Nets Saved in Navigation Window



Notice that if you go back to the top level of the layout by clicking with the **right (Button-3)** mouse button on TOP_CHIP in the Cell List box, the zoom view areas are still saved. The zoom view area and highlights are saved for each cell.

- To **clear a saved zoom view or highlight**, click on the colored buttons in the Navigation Window using the **middle mouse button (Button-2)**.

TIP:

Remember, if you ever forget which mouse button to use just move the mouse over the area you're interested in (Cell List, zoom buttons on right of navigation window, etc.) and read the description in the Message Area at the top of the MAX window.

Mouse Coordinates and Box Size Location

In the lower right hand corner the **Box Size** (in microns) is displayed. (The Box is used extensively in the MAX Layout Editor.) This is useful for when you select an object and want to know how big it is. It is also used for the measure command.

If you click with the **left mouse button** on the **Box Size** area, a pop-up will appear where you can specify a box or change what information is displayed in the **Box Size** area. It defaults to displaying the *size and area* of the box. You can also have MAX display the *corners* or the *origin and size*.

Just to the left of the Box Size is the **Mouse Coordinates display**. Mouse coordinates are oriented with respect to the top level cell. As you move the mouse notice that this info changes.

The Palette

Along the left side of the window is the **Smart Palette**. The Smart Palette provides many features:

- It controls which layers are visible.
- It gives feedback on what is currently under the cursor, and what is currently selected.
- It allows you to control which layers can be selected.
- And finally, it allows you change the color or stipple patterns of any of the layers.

The Smart Palette features are described in detail in the *MAX User Manual*.

WORKING WITH THE LAYOUT VIEW

Now that you have some familiarity with the MAX window and its' particulars, let's experiment with moving around and viewing the TOP_CHIP layout in detail.

ZOOMING IN

We're going to zoom in on the layout, and change the layer visibility.

Step 1 ⇨ Make sure that TOP_CHIP is the edit cell. Remember, you click on it with the **right (Button-3)** mouse button in the cell list.

Step 2 ⇨ To clear any highlights you might have, go to the **View** menu and select **Clear**.

Step 3 ⇨ Zoom in on the upper left corner of TOP_CHIP. You can zoom in by placing the cursor in the upper left corner of the cell and moving the scroll wheel up on your mouse.

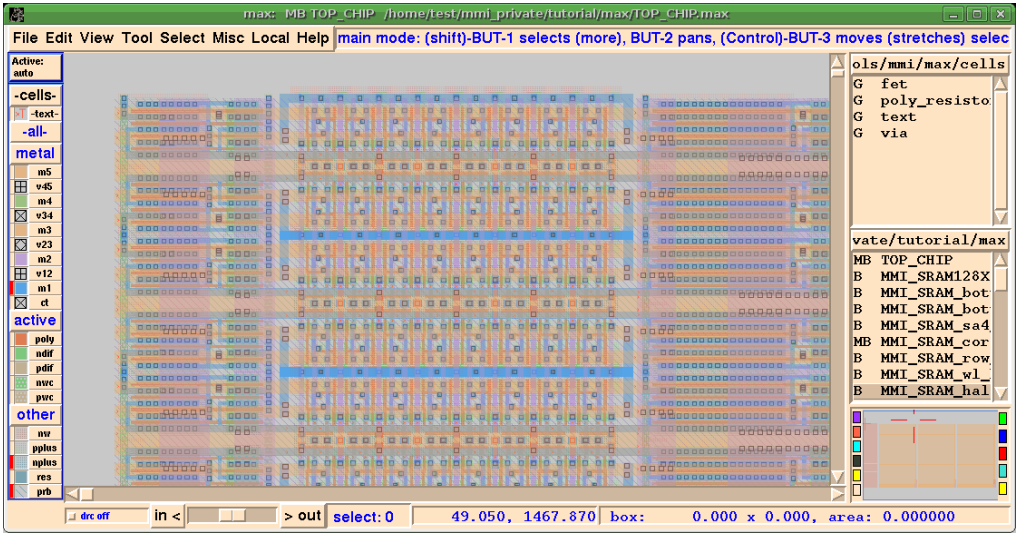
Step 4 ⇨ A faster way is to use the **z** hotkey or select **Zoom to Area** on the **View** menu. After you type the **z** hotkey, drag out a box around the upper left portion of TOP_CHIP.

⇨ You can also zoom in by placing the cursor in the top left of the layout, and typing the hotkey **j**, or selecting **Zoom In on Cursor** from the **View** menu.

⇨ Or, you can use the Navigation Window to zoom to this area.

Zoom in until you see something similiar to Figure 9.

Figure 9: Zoomed In View with All Layers Visible

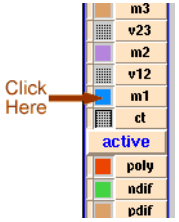


CHANGING LAYER VISIBILITY

Currently, all layers are on and visible. Let's change that.

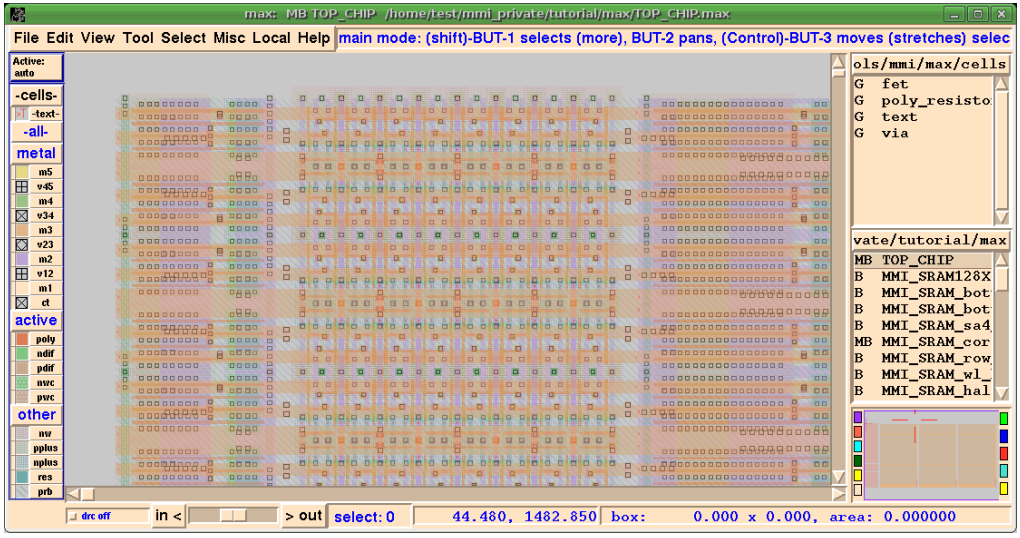
Step 1 ⇨ Turn off ***visibility*** of `m1` by clicking with the **LEFT (Button-1)** mouse button on the blue square to the left of "`m1`", as shown here.

Figure 10: Turn Off Visibility of "m1"



You should now see something similiar to Figure 11. The `metal 1` layer is no longer visible (the bright blue has disappeared).

Figure 11: Metal 1 Layer Visibility Turned OFF - No Blue Showing



- If you click with the **LEFT (Button-1)** mouse button on the box with the word "`-metal-`", the ***visibility*** of all metal layers will be turned on.

- If you click on the label again, the ***visibility*** for all metal layers will be turned off. We will discuss how the palette can be used to control ***selectability*** later in this tutorial.

⇨ Click on the "empty" `m1` palette square to turn visibility for `m1` back on.

CHANGING LAYER COLOR

Now, let's try changing the color of the layer `m1`. First make sure all layers are **visible**.

Step 1 ⇨ To access the palette:

- Click with the **MIDDLE mouse button** (**Button-2**) on the blue square to the left of `m1` on the palette,

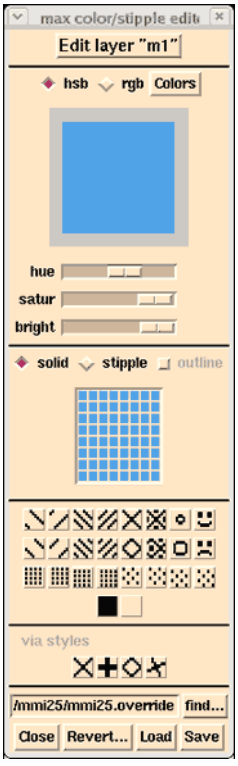
OR:

- Go to the **File** menu, select **User Preferences** and then **Color Editor...**

You should now see the pop-up in Figure 12.

⇨ At the top of the pop-up, hold down the **LEFT** (**Button-1**) mouse button and select "m1."

Figure 12: Editing Layer "m1"



Step 2 ⇨ To change the color of a layer, you can use the hue, saturation and brightness scroll bars or select a color from a list by holding down the **LEFT** (**Button-1**) mouse button on **Colors**.

Change the color of **metal 1** and notice that it changes interactively in the layout.

Step 3 ⇨ To get back to the original color, click on the **Revert...** button and then click on **Done**.

Step 4 ⇨ To exit the color editor, click on **Close**.

Refer to the *MAX User Manual* for more details on editing layer colors and styles.

Layers

Just above the palette are buttons labeled **Active:**, **-cells-**, **-text-** and **-all-**.

Active lets you select the *active* layer for wiring or drawing polygons.

Cells controls whether or not sub-cells and gcells can be selected.

Text controls the visibility and selectability of text.

All allows you to toggle the visibility or selectability of all layers at the same time.

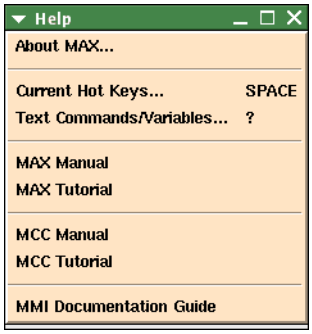
Getting Help and the Help Menu

Before we go any further, here is how to get help if you ever need it.

- There are several levels of help available on-line to MAX users. You already know about the quick **Help** feature listed in the **MAX Message Area**.
- In addition, you can access the complete on-line manual, this tutorial, a list of MAX commands and variables and the complete list of active hotkeys.

Step 1 ⇨ Click and hold down the **left mouse button** (**Button-1**) on the **Help** menu. This should pop up a sub menu as shown in Figure 13.

Figure 13: MAX Help Menu



Help Menu

About MAX tells you the version of MAX you are running, the technology you are using, etc.

> Unless you have the technology specified in a `max.rc` file, or you used the `-tech` option when starting MAX, it defaults to the **mmi25** technology.

Current Hot Keys gives you a list of all available hot keys. You can also hit the Space bar to bring up this list.

Text Commands/Variables brings up documentation for the MAX text commands and variables. Text commands are useful for writing scripts, generators, and other helpful programs. Text commands and the Tcl/Tk interface are the MAX API. They allow the user to add their own features to the **Local** and **Tool** menus.

MAX Manual brings up the MAX manual in your browser. The default browser is ***Firefox***. This can be changed in your `max.rc` file.

MAX Tutorial brings up this tutorial.

MMI Documentation Guide brings up all of the documentation for all of the Micro Magic, Inc. software. This document has links to items like the *Micro Magic, Inc. SUE User Manual*, the *Micro Magic, Inc MAX User Manual*, other Micro Magic, Inc. software programs, and even to this tutorial.

Before proceeding you should bring up the Micro Magic, Inc *MAX User Manual* and see what's there. The User Manual is the reference manual to MAX and contains lots of information that is not found in this tutorial.

You might also want to look at the menu items just to familiarize yourself with what's there.

Tear Off Menus

Sometimes it is convenient to make a menu its own window.

For example:

Step 1 ⇨ Going back to the **Help** menu: when you hold down the **left mouse button** (**Button-1**) over the **Help** menu notice that there is a dotted line just before the **About MAX** menu item (not shown in the above figure).

Step 2 ⇨ If you release the button while over the dotted line the menu will "tear off". The menu then becomes a full-fledged window, like any other window on your desktop. The exact behavior will depend upon which OS and window environment you are running.

If you are running another window manager, your windows might behave a little differently. Consult your window system documentation for more details.

The chosen menu will stay pulled-down until you select either a menu item, or another menu.

PART 2

VIEWING AND SELECTING LAYOUT

This part of the tutorial covers viewing, selecting and editing an existing layout. The next part will cover creating new layout.

SIMPLE VIEWING

In the previous section of this tutorial, we covered some basic commands for moving around the layout and changing layer color and fill patterns. This section will cover how you move around the layout, view the hierarchy, view connectivity, edit wires, and measure things.

- You should still have the GDSII file `TOP_ CHIP.gds` loaded in the MAX window. If not, re-load the GDSII file (refer to the previous section).

OK, let's get started.

Zooming

Once loaded, there are several methods for zooming into/out of the layout. Zoom commands are found in the **View** menu, and also have hotkeys.

ZOOM IN ⇨ Use the zoom hotkey to zoom to an exact region.

Step 1 ⇨ Type z. This puts you in *zoom mode*. Note that the cursor has changed. Next, hold down the left mouse button (**Button-1**) and "drag" out a box surrounding the region to which you wish to zoom. When you release the mouse, MAX will zoom to this region.

- If you mess up don't panic. Remember, **u** will *undo any edits you have made*, and **v** will always *zoom to fit*.

Zoom Out

Step 1 ⇨ Zoom out by typing **Shift-z**, or use **Zoom Out** in the **View** menu, or use the **scroll wheel** on your mouse (if you have one). You can also use the **Zoom Bar** at the bottom of the MAX window, just below the scroll bars.

Zoom to Fit

Step 2 ⇨ Type **v.** The layout will re-center to fit within the size of the layout window.

Zoom In/Out by 2X ⇨ One often-used technique for zooming is to use the **j** hotkey to *zoom in* by a factor of two, centered on the cursor. The hotkey **Shift-z** can be used to *zoom out* by a factor of two.

Zoom with Scroll Wheel ⇨ If you are using a scroll wheel mouse, you can zoom in/out by rolling forward or backward on the scroll wheel while in the MAX window. Zooming will be centered around the point of the cursor.

Zoom Using Navigation Window ⇨ Using **Button-1** within the Navigation Window itself (lower right corner), drag out a box enclosing the desired zoom area. The advantage of using the Navigation Window is that it always displays the entire edit cell, so you can zoom to a particular area of the layout without having to first zoom out.

Using **Button-3** within the Navigation Window lets you pan around the design.

Pan with Scroll Wheel ⇨ If you hold down the **Shift** key while using the mouse **scroll wheel**, MAX will pan to the left as you scroll 'away, the to the right as you scroll 'towards' yourself.

If you hold down the **Ctrl** key while using the **scroll wheel**, MAX will pan up and down.

Selecting Layout

After your layout is loaded, MAX offers several different selection options. Using these options for an object (cell) or rectangle, you can:

- Select individual rectangles of the layout
- Select cells or layers within the layout
- Select nets and trace connectivity

When MAX is in main mode (the default), dragging out a region while holding down **Button-1** will select whatever is contained within that box when you release the button, highlighted in white.

Loading a Smaller Cell

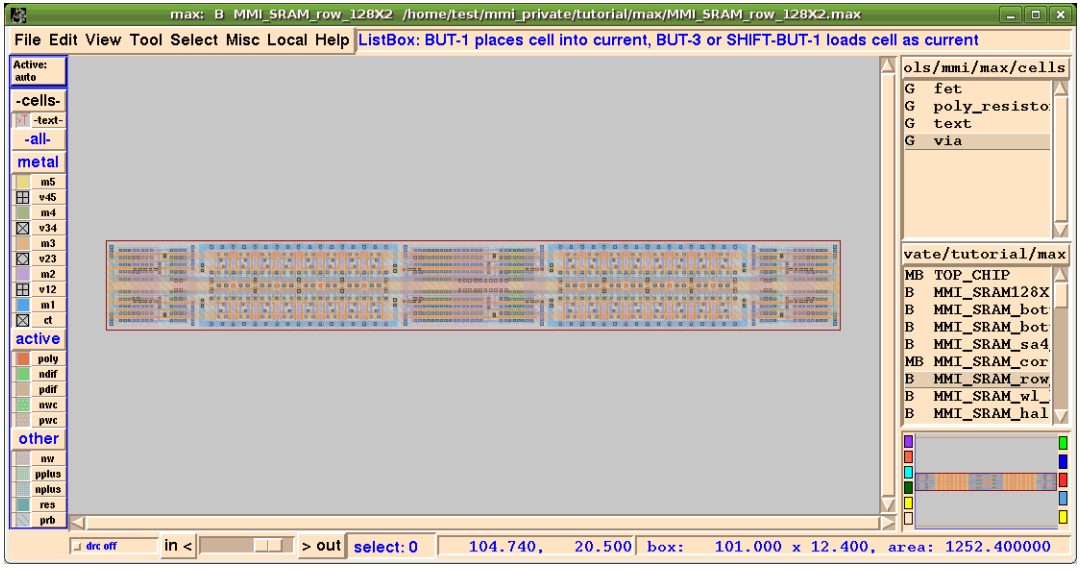
First, to simplify things, we will load and view a smaller cell.

Step 1 ⇨ Go over to the Cell List Box on the right side of the MAX window and click with **BUTTON-3** on `MMI_SRAM_row_128x2`.

> **TIP:**
>
> Remember, if you click with **Button-1**, MAX will **instantiate** `MMI_SRAM_row_128x2` in your current layout.
>
> Dont' worry, you can always use the **undo** (hotkey: **u**) command. If the cell has not been placed yet, **Ctrl-c** will abort the command.

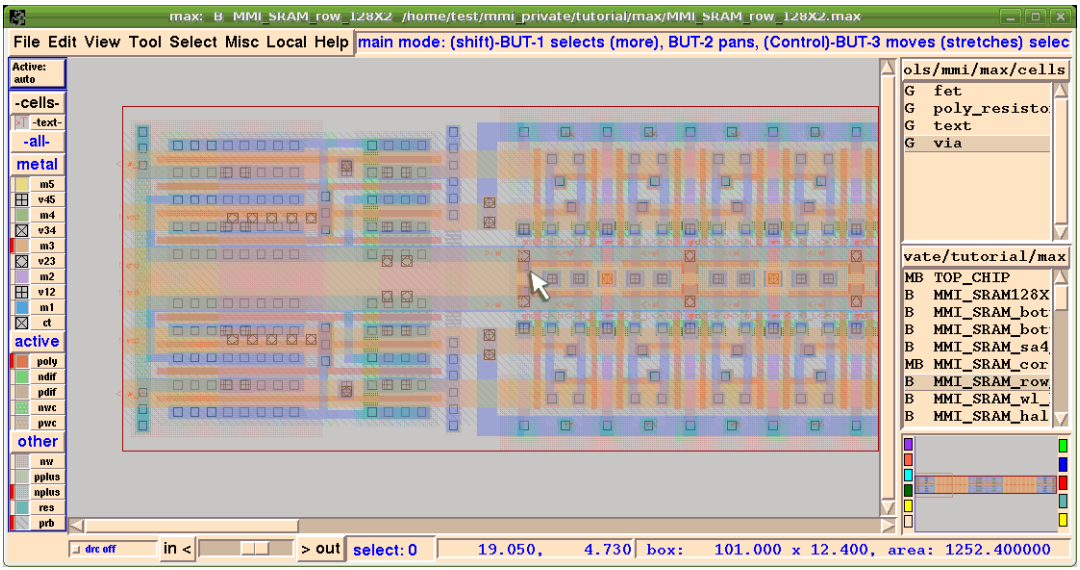
You should now see something similar to Figure 14.

Figure 14: Cell MMI_SRAM_row_128x2 Loaded



Step 2 ⇨ Zoom in on the left half of this cell. Type the **z** hotkey and drag out a box using **Button-1**. You should be looking at something similar to Figure 15.

Figure 15: Zoomed In on MMI_SRAM_row_128x2



SELECT USING HOTKEY Now we will select cells through the hierarchy of the layout.

Step 3 ⇨ Move your mouse to the center of the MAX window as shown in Figure 15 (above). Type the hotkey **f** to **select a cell**.

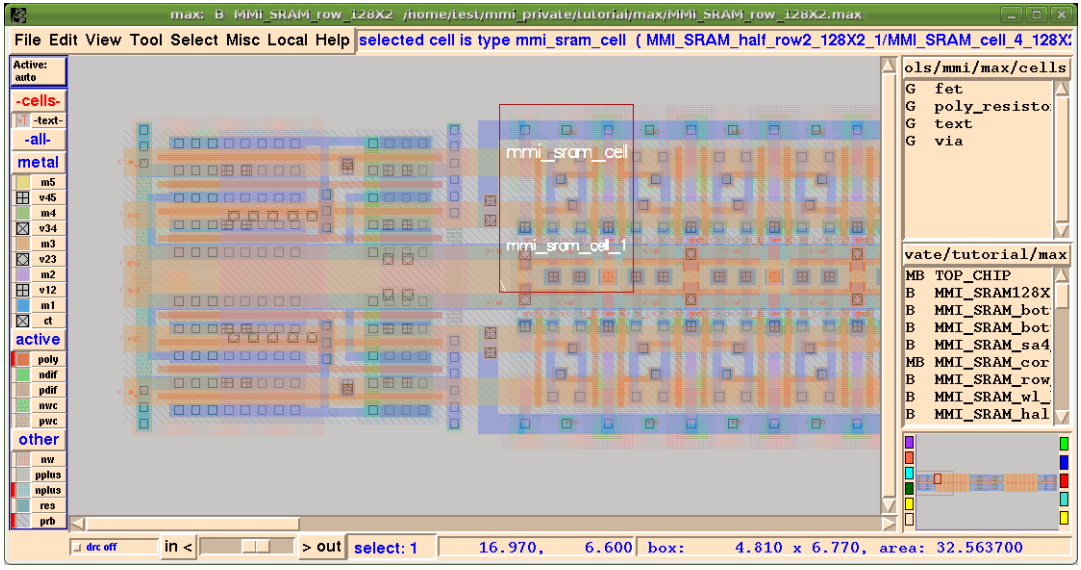
⇨ You should now have the cell `mmi_sram_cell` selected.

- If you look in the Message Area at the top of the MAX window, you should see a message like "`selected cell is type mmi_sram_cell (MMI_SRAM...)`".

  This tells you which cell has been selected and shows the hierarchicah path to the cell inside the parentheses.

You should see something similar to Figure 16. It doesn't matter if you have a different `mmi_sram_cell` selected than is shown in the figure.

Figure 16: 'mmi_sram_cell' Selected in MMI_SRAM_row_128x2



Step 4 ⇨ Without moving the mouse, type **hotkey f** once more. You should now have the cell `MMI_SRAM_CELL_4_128x2` selected.

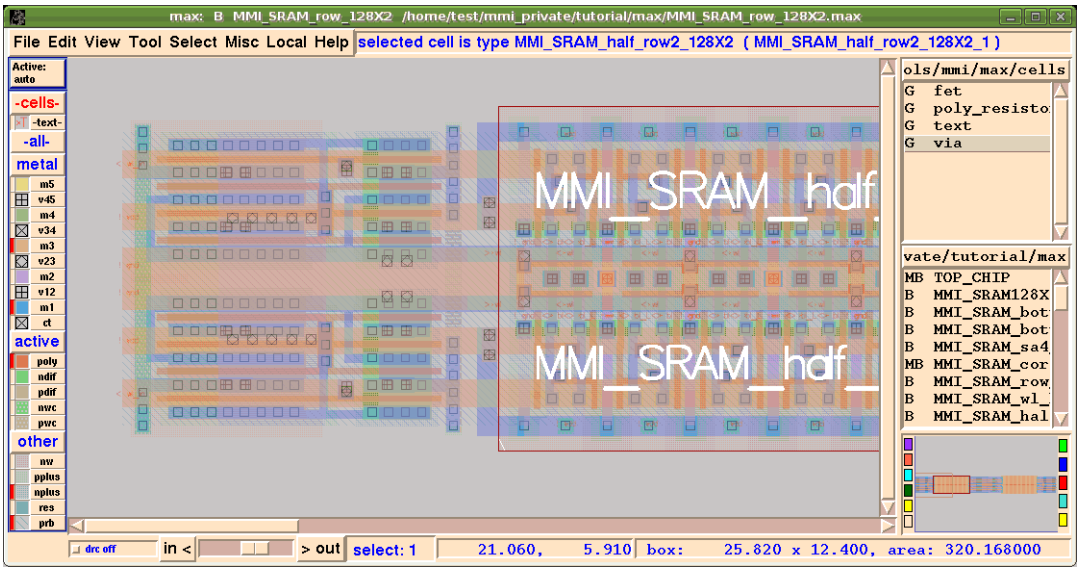
⇨ If you have moved the mouse, you will see `mmi_sram_cell` selected again. That's OK, just type the **f hotkey** a second time without moving the mouse.

⇨ Depending on where you placed the cursor initially, you might have selected the `mmi_sram_cell` adjacent to the first one. Simply type **f** for a 3rd time and `MMI_SRAM_cell_4_128x2` will be selected.

⇨ Notice the Naviagtion Window shows you where this particular cell (dark red outline) is located in relation to the current cell and the zoomed area of the cell (light brown rectangle).

Step 5 ⇨ Type **f** (*select cell*) another time without moving the mouse to select `MMI_SRAM_half_row2_128x2`. You should see something similar to Figure 17.

Figure 17: MMI_SRAM_half_row2_128x2 Selected



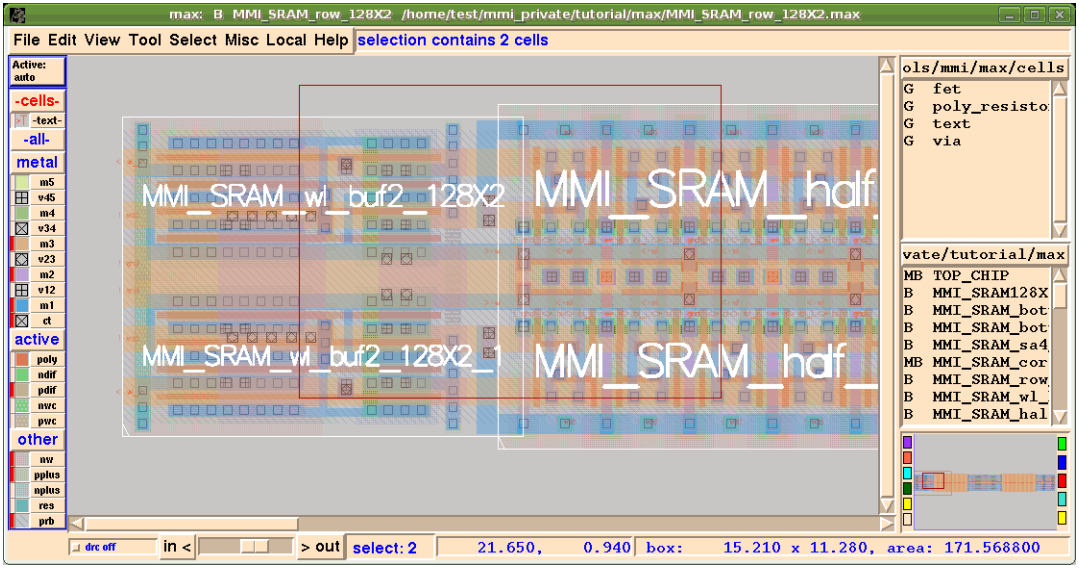
The first time you use the **Select Cell** command (hotkey: **f**) it finds the cell at the bottom of the hierarchy. The next time it moves up to the next level of hierarchy. If two cells overlap at the same level of the hierarchy and you position the mouse over the region that overlaps, **Select Cell** will highlight the first cell, then the second cell, and then move up the hierarchy. You may have seen this happen when selecting the cells.

SELECT USING THE BOX

You can also select cells by dragging out a box with **Button-1** that includes some part of the cell.

Step 6 ⇨ Try selecting a cell by dragging a box with **Button-1** around part of the layout. You should see something similar to Figure 18.

Figure 18: Selecting by Dragging Out a Box



Notice that the Message Area at the top of the MAX window says "`selection contains 2 cells`" and the Select Box at the bottom of the window says "`select:2`". If only one cell has been selected, then the name of this cell is listed in the Message Area. Dragging out a box only selects cells at the current level of hierarchy.

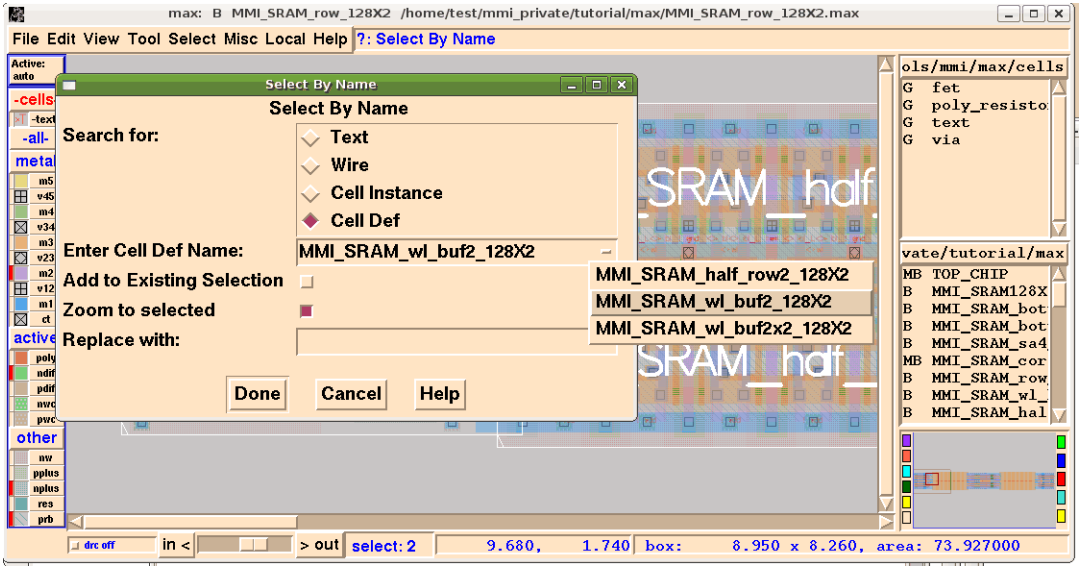
⇨ Another way to select a cell is to click over it with **Button-1**. This method only selects cells at the current level of hierarchy. If you want to select cells throughout the hierarchy, you need to use the **Select Cell** (hotkey: **f**) command.

SELECT BY NAME

Additionally, you can select cells by **name**.

Step 7 ⇨ In the **Select** menu, go to **Select by Name...** or use the **n** hotkey. You'll see a pop-up similar to Figure 19.

Figure 19: Select By Name Pop-up



TIP:

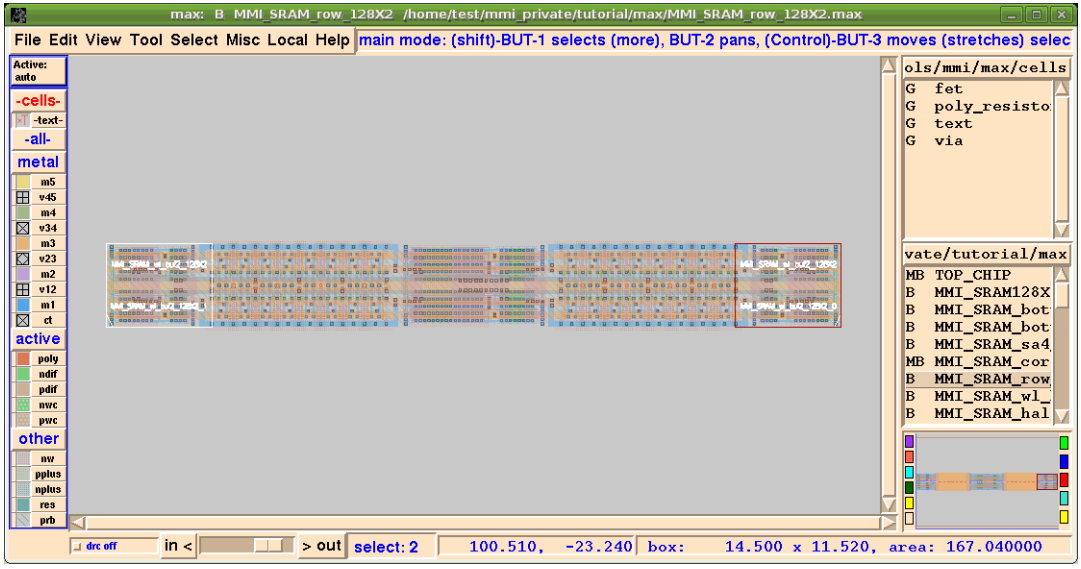
Most pop-ups in MAX have a **Help** button. Clicking on the **Help** button opens an item-description pop-up.

Step 8 ⇨ Click on the **Cell Def** toggle button. We're going to be selecting cells by the name of the cell definition (e.g.: `mmi_sram_cell`) and not by the instance name.

Step 9 ⇨ Hold down **Button-1** on the pull down list to the right of **Enter Cell Def Name:**. Select `MMI_SRAM_wl_buf2_128x2`.

⇨ Click on the **Zoom to selected** toggle button. Now click on **Done** and you should see something similar to Figure 20.

Figure 20: MMI_SRAM_wl_buf2_128x2 Selected by Name; Zoomed out



There are two cells selected. MAX zoomed out so that both cells are visible.

Selecting Polygons and Controlling What Is Selected

In this section we will now look at how to select rectangles or polygons. We will also look at how you know which layers have been ***selected*** and how to control what layers are ***selectable***. If you are trying to select a rectangle on `m1` and there are other layers on top of it, you can *turn off the selectability of the other layers*. That way you can select just the `m1` rectangle.

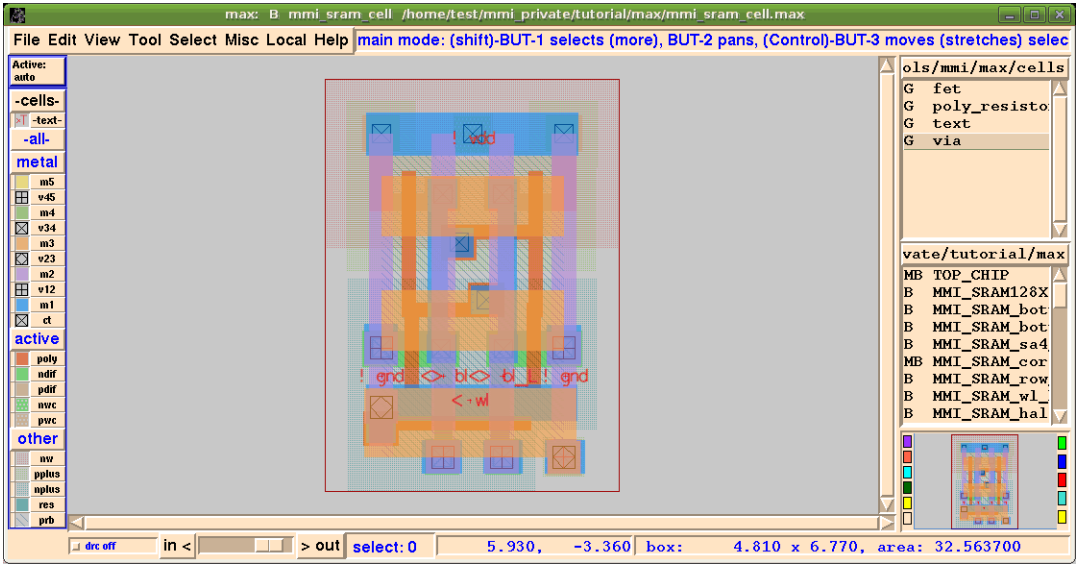
Selecting Polygons

First, we need to push into a cell containing layout, and not just instances of cells. We will push into the bottom level cell `mmi_sram_cell`.

Step 1 ⇨ Go back to the left side of the layout and use the **Select Cell** command (hotkey: **f**) to select the same `mmi_sram_cell` as we did earlier.

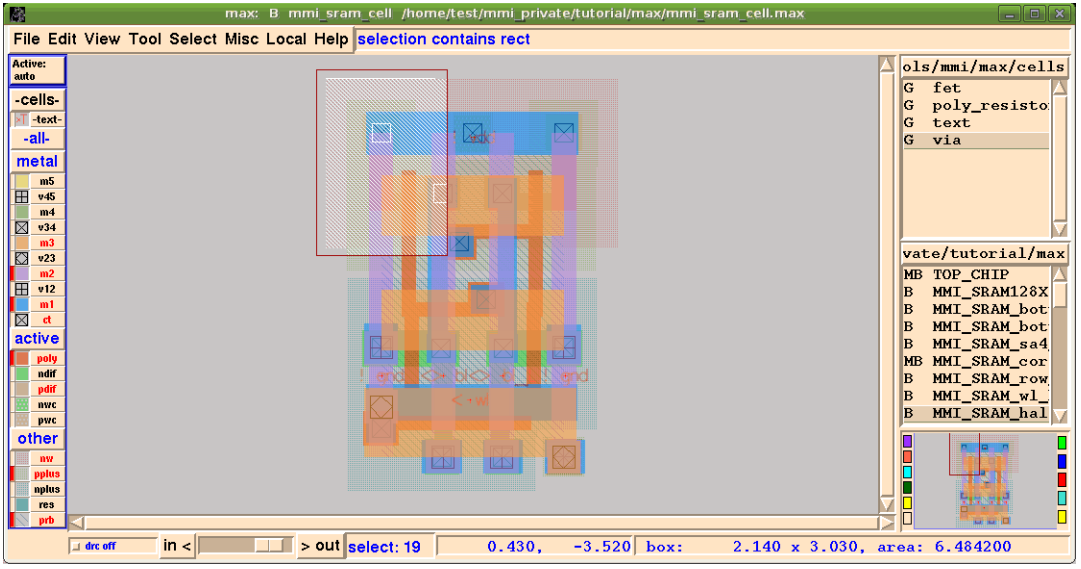
Step 2 ⇨ Push into the cell by using the **Push into Cell** command (hotkey: **e**) in the **View** menu. You should now see something like Figure 21.

Figure 21: Push Into View of MMI_SRAM_cell



Step 3 ⇨ Drag out a box using **Button-1** tha encloses the upper left corner of the cell. You should see something similar to Figure 22.

Figure 22: Selecting Rectangles by Dragging Out a Box



The number of rectangles selected will most likely differ from the example above. The Message Area should say "`selection contains rect.`" Notice that the Select Box at the bottom middle of the MAX window (to the right of the Zoom sliders) lists how many things are selected. In Figure 22 above, 19 rectangles have been selected.

If your box included the `vdd` text, then the Message Area will say "`selection contains rect. text.`"

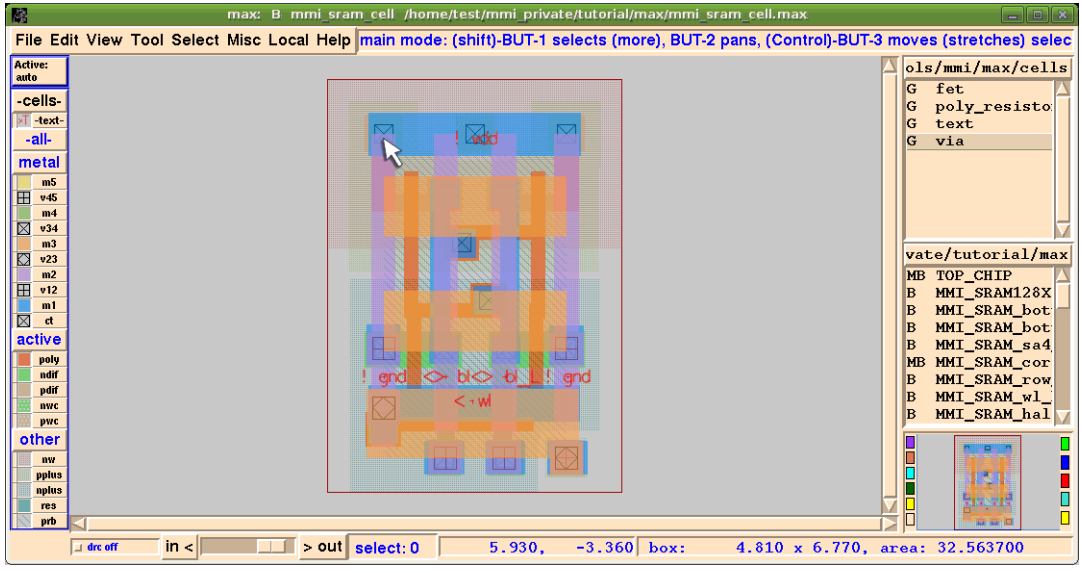
Step 4 ⇨ Look at the palette along the left side of the MAX window. Notice that some of the layer names are now red. This indicates that rectangles on that layer have been selected.

On the far left of the palette, notice the red indicator rectangles ("LEDs") next to some of the layers. These show whenever the cursor is currently hovering over that layer.

⇨ You can also select objects by clicking once over them with **Button-1**.

Step 5 ⇨ Move the mouse so that the cursor is over the upper left contact, as shown in Figure 23.

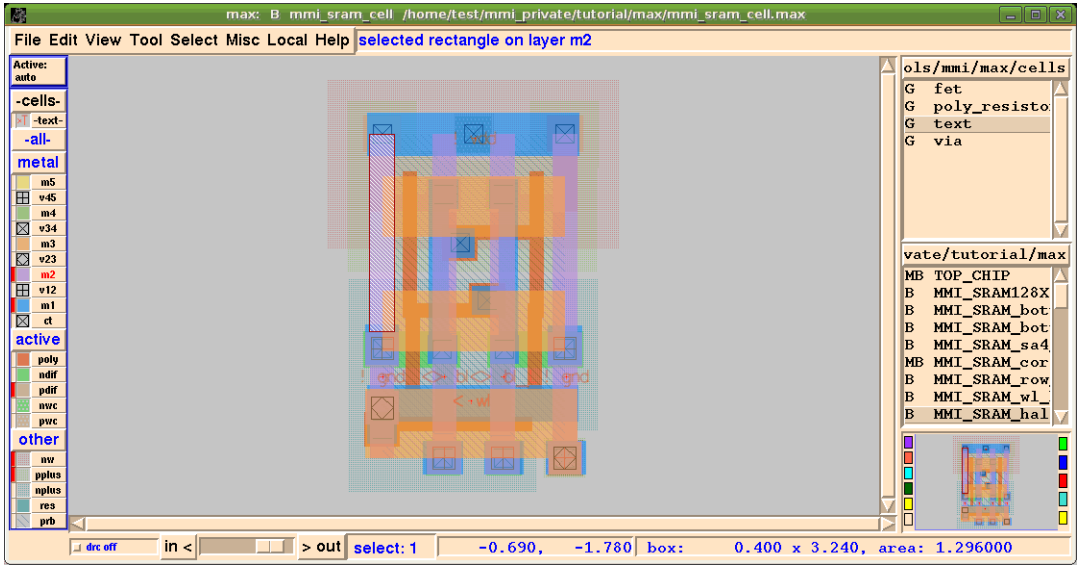
Figure 23: Mouse Cursor Positioned Over Upper Left Contact



Clicking **Button-1** on a rectangle selects that particular rectangle.

⇨ Click once with **Button-1** at the location shown in the above figure to see something similar to Figure 24.

Figure 24: 'm2' Rectangle Selected



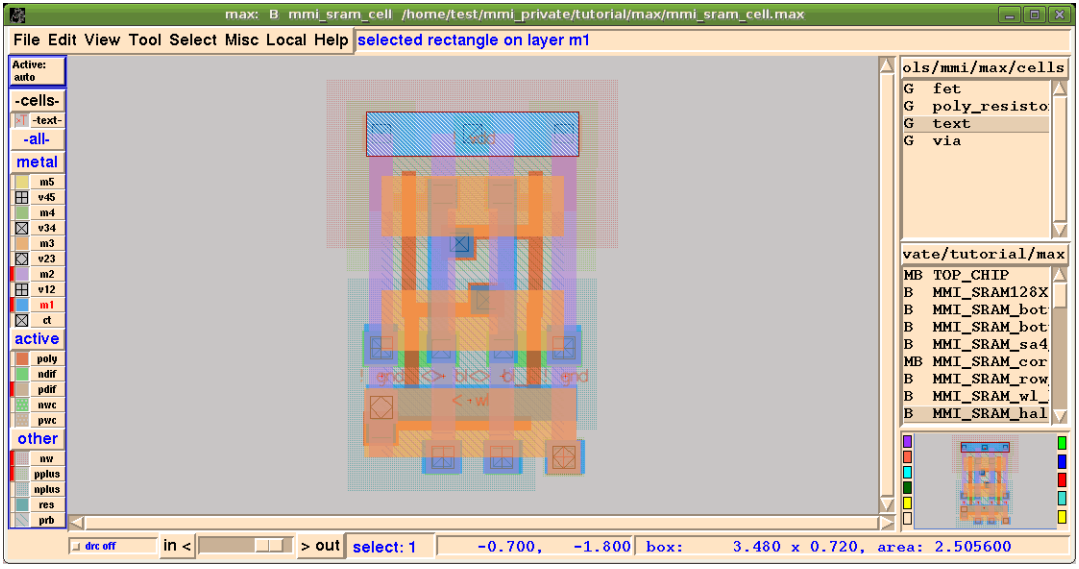
You should have selected an m2 rectangle. The Message Area should say "`selected rectangle on layer m2`".

When there are multiple layers under the mouse cursor, clicking more than once without moving the mouse will select each layer in turn. You can keep clicking until the one you want is selected.

MAX starts with the "highest" layer. For example, it would select `m3`, then `v23`, then `m2`, etc.

Step 6 ⇨ Without moving the mouse, click **Button-1 two more times**. You should now see something similar to Figure .

Figure 25: "m1" Rectangle Selected



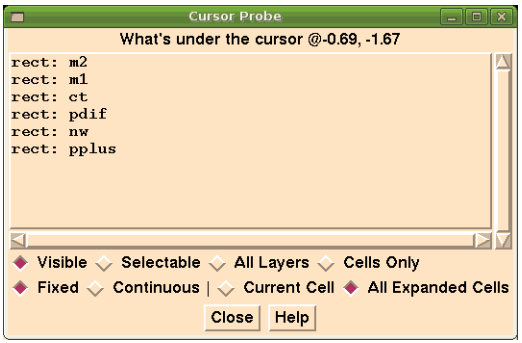
SELECTING USING CURSOR PROBE

Another way to select a rectangle on a particular layer is to use the **Cursor Probe** command (hotkey: **l** - ell; lower-case L). This opens a pop-up window listing everything underneath the cursor.

Step 1 ⇨ Clear everything that is selected using the **Clear Selection** command (hotkey: **c**) from the **Select** menu.

Step 2 ⇨ Move the mouse back to the same location shown in Figure 25 above. Type the **Cursor Probe** hotkey **l** (ell) and a pop-up like Figure 26 should appear.

Figure 26: Cursor Probe Results for Layers Under Selection



Step 3 ⇨ As you click on each of the layers in the list, that layer will be selected. Click on the layer a second time to turn off the selection. Refer to the *MAX User Manual* for more information on the **Cursor Probe**.

Selecting Nets

MAX can trace connectivity through vias and contacts to interactively show everythig connected to a net. We'll first see this using the `mmi_sram_cell`.

Selecting a Net

Step 1 ⇨ Place the cursor in the same location shown in Figure 23 and type hotkey **s**, or choose **Select Net** from the **Select** menu. You should now see something like Figure 27.

Figure 27: Select Net on 'mmi_sram_cell'

MAX will automatically select the entire net that is connected to the m2 rectangle that was under the cursor, and will also display the net name, if known, in the MAX Message Area. The net name(s) are taken from any Text (labels) attached to the net. For the above net, the net name is gnd.

Notice in the palette on the left that some of the layer names are now in red. This indicates that the selected net contains these indicated layers.

Step 2 ⇨ **Clear the selection** by typing the hotkey **c**.

⇨ Now we will use the **Cursor Probe** to select the net. Move the cursor once more to that same location and type **l** (ell - lower-case "L"). In the pop-up, double click with **Button-1** on "`rect: m2`" and you should see the same net highlighted.

Step 3 ⇨ **Close** the **Cursor Probe** pop-up.

Controlling Layer Selectability

Now we will load a simple test cell that will make it easier to see how selecting works.

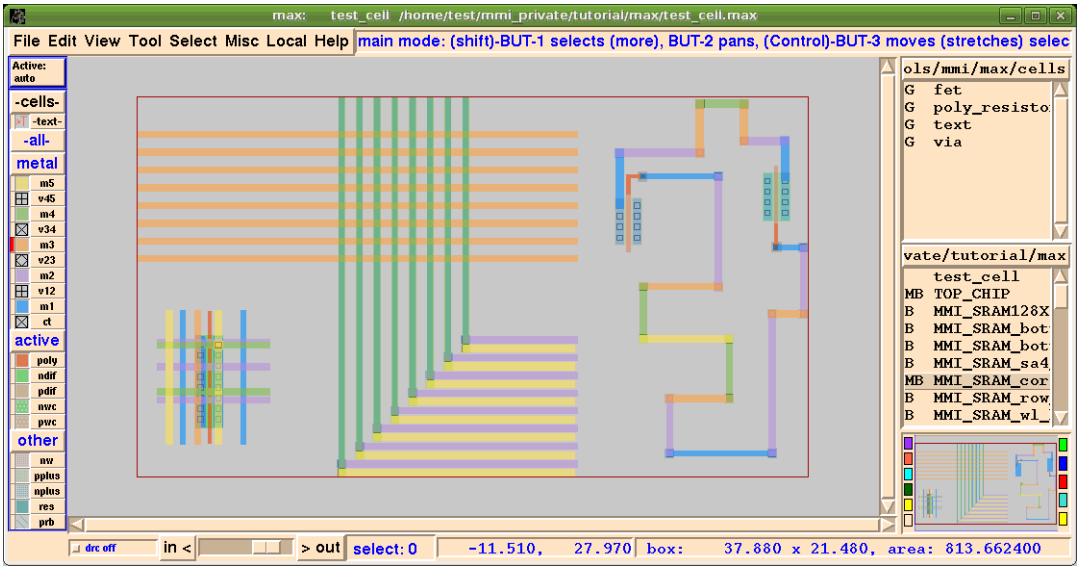
Step 1

⇨ Go to the **File** menu and select **Open**.

⇨ Once the pop-up form appears, click to select "`test_cell.max`".

⇨ Click **OK**. You should now see Figure 28.

Figure 28: Loaded MAX File 'test_cell'

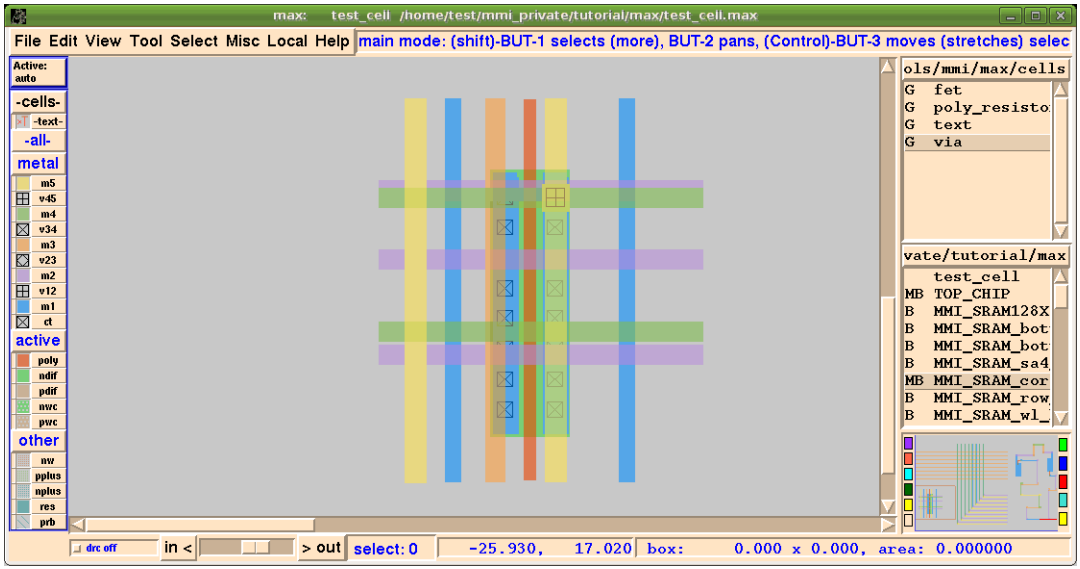


Step 2

⇨ Zoom in on the lower left corner of the cell so that you see something similar to Figure 29.

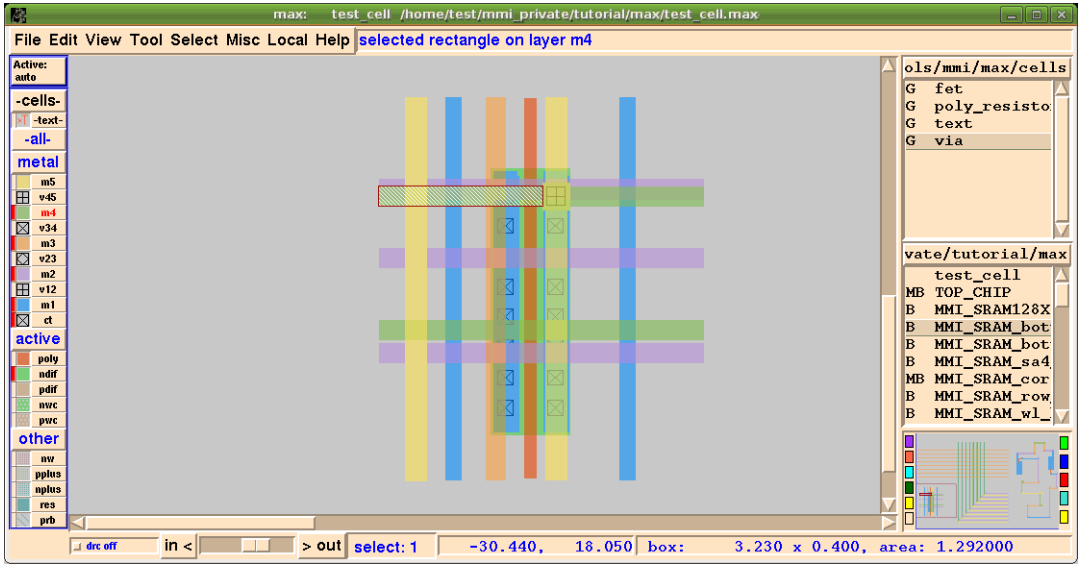
⇨ Remember, to zoom in you can use the **z** hotkey and then drag a box around the desired area.

Figure 29: Zoomed In on 'test_cell'



Step 3 ⇨ Place the cursor so that it is over both the left side `diffusion` and `m4`, as shown in Figure 30. Click **Button-1** once and notice that an `m4` rectangle has been selected.

Figure 30: Selected 'm4' Rectangle in "test_cell.gds"



For this example, the nfet has been glattened and is no longer a gcell. (Gcells are discussed later in this tutorial.)

TURNING OFF SELECTABILITY

To select the `ndif` rectangle, you could continue clicking at the same location until the piece of `ndif` is selected.

Another way is to ***turn off the selectability*** of some of the layers.

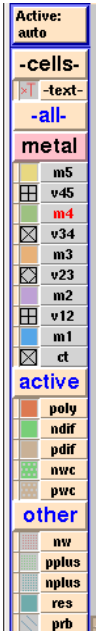
Step 4 ⇨ Move over to the palette along the left side of the MAX window. Click with **Button-1** on the "`metal`" button to ***turn off visibility*** of the metal layers. It may still show you where the piece of `m4` is that you previously selected.

Step 5 ⇨ Turn ***visibility*** of all the metal layers ***back on*** by clicking again with **Button-1** over the "`metal`" toggle button.

Step 6 ⇨ Now click with **Button-3** on the "`metal`" toggle. The palette should look like Figure 31.

Notice that all of the metal layer names are greyed out. This means that you **cannot select** anything on that layer. if the piece of `m4` is still selected, the name "`m4`" will show in red on the palette.

Figure 31: Palette Showing Metal Layers Selectability Turned Off



-metal- toggle button: Click with **Button-3** to ***turn off selectability*** of all metal layers (buttons turn grey)

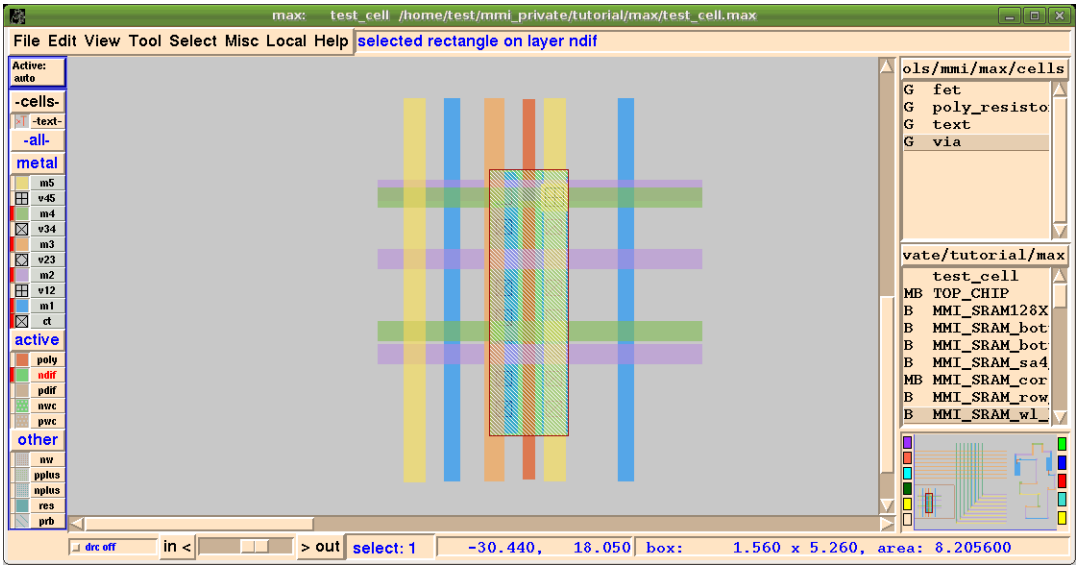
Red text showing **m4** layer ***selected***

Grey buttons indicate **selectability** of all metal layers is ***off***

Step 7 ⇨ Now click **Button-1** in that same previous location (refer to Figure 30).

A rectangle of `ndif` should now be selected, as shown in Figure 32. Because the ***selectability*** of all metal layers has been **turned off**, MAX finds the next layer under all of the metal layers, That layer name is shown in red in the palette. Remember that the red "LED" rectangles on the far left of the palette indicate which layers are underneath the cursor.

Figure 32: Rectangle of 'ndif' Selected



Step 8 ⇨ ***Turn back on the selectability*** of the metal layers by clicking again with **BUTTON-3** on the "`metal`" toggle button. The layer names will no longer be greyed out.

Selecting Nets; Layer Selectability; Editing Wires

We have selected single and multiple rectangles. Now we will look at how to select nets to help you investigate connectivity.

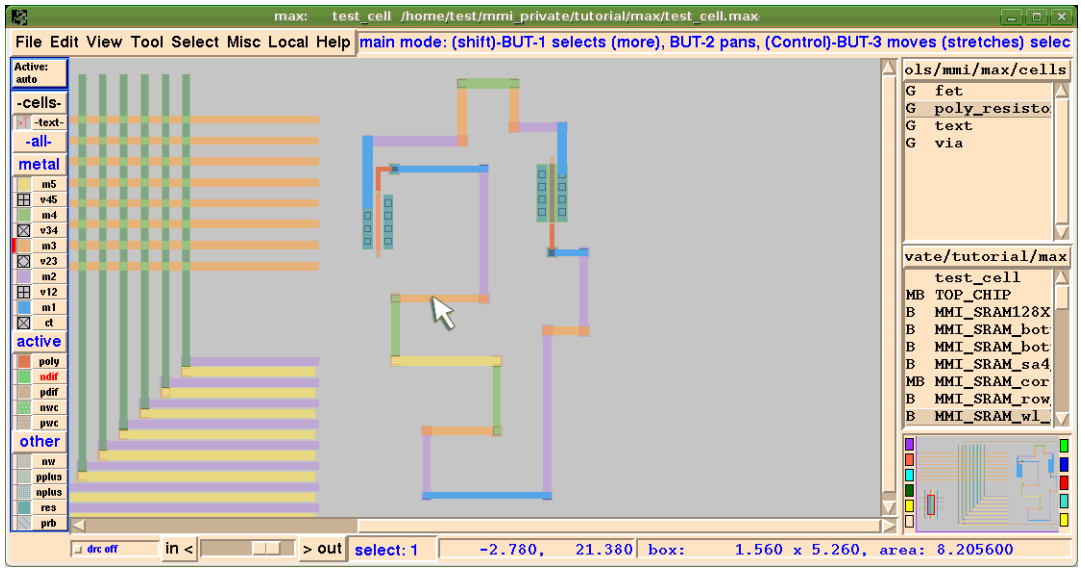
Zooming in on Selection

Step 1 ⇨ **Zoom back out** to view the entire `test_cell`. Remember, you can use the **v** hotkey (centered zoom-to-fit).

Step 2 ⇨ Zoom in on the right side of `test_cell`. You should now see something similar to Figure 33.

Remember that you can also use the Navigation Window to zoom in to the same area.

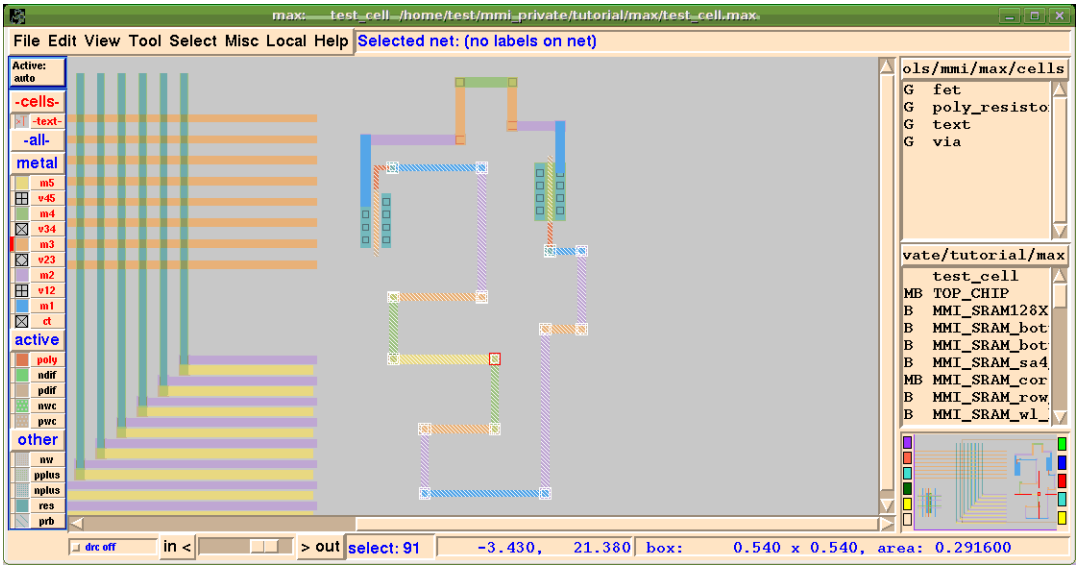
Figure 33: Cursor Positioned Over 'm3' in "test_cell"



Step 3 ⇨ Place the cursor over the middle left `m3` rectangle as shown above. Now type the **Select Net** hotkey: **s**.

You should now see something like Figure 34.

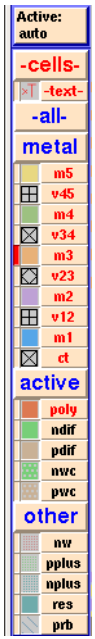
Figure 34: Select Net at Cursor Position Over ‘m3’



Notice that the entire net is selected from gate to gate. The information on how layers connect is specified in the `MAX technology file`. The technology file is also where the layer names, colors, fill patterns, and DRC rules are specified. We’ll cover a couple of ways to generate a technology file in a later section.

Step 4 ⇨ Look at the palette, as shown in Figure 35. Notice that there is a red LED rectangle to the left of `m3`. This tells you that the cursor is currently over a piece of `m3`. All of the metal names, as well as `poly`, `nfet`, and `pfet` names are highlighted in red. This indicates that rectangles on those layers have also been selected.

Figure 35: Highlighted Layer Names in Palette Indicate Selected Layers



Red LED and text showing cursor over **m3** layer ***and*** **m3** ***selected***

Names of Layers in red indicates there are rectangles on these layers selected, but cursor not over these layers.

TIP:

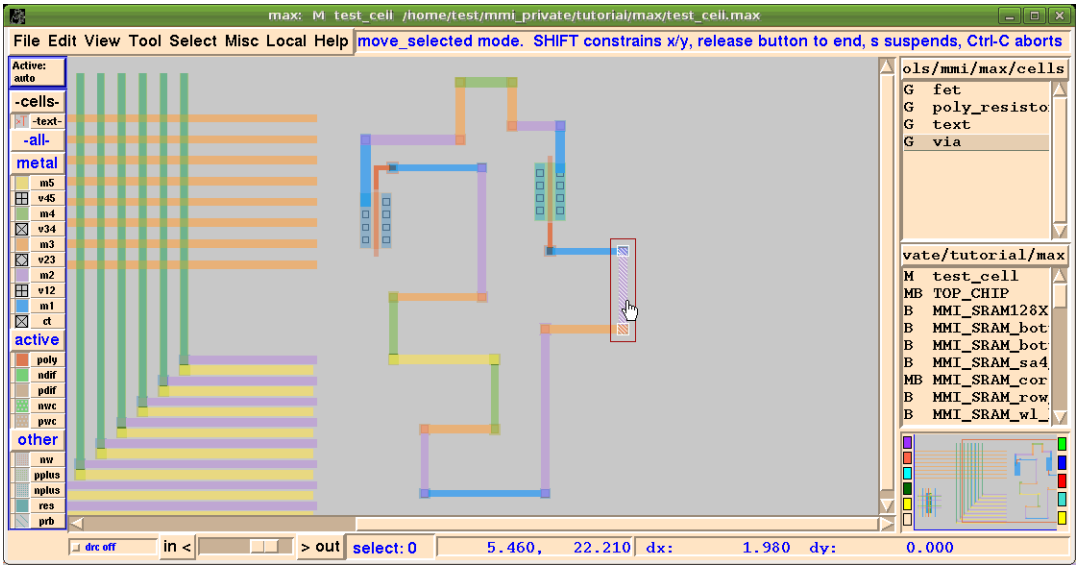
A nice feature often used for debugging is that when MAX selects a net, the name of all the text labels on that net are displayed in the MAX Message Area at the top. For example, if you select a VDD net and the names VDD and GND both appear, you know you have a problem.

EDITING WIRES

One of the features of MAX is the ability to easily edit wires while ensuring that the appropriate vias are moved along with the wire segment.

Step 5 ⇨ Type the **Ctrl-w** hotkey or select **Edit Wire** from the **Edit** menu. Hold down **Button-1** over the right middle segment of m2, as shown in Figure 36. Drag the mouse to the right and notice that the associated vias move along with the segment of m2.

Figure 36: Editing 'm2' Wire in "test_cell.max"



USING LAYER SELECTABILITY

We will now look at the ways you can use layer selectability to control which nets are selected.

Step 1 ⇨ Use the **Pan** hotkey "**->**" (**KP_RIGHT**) to pan the layout over until you see something similar to Figure 37.

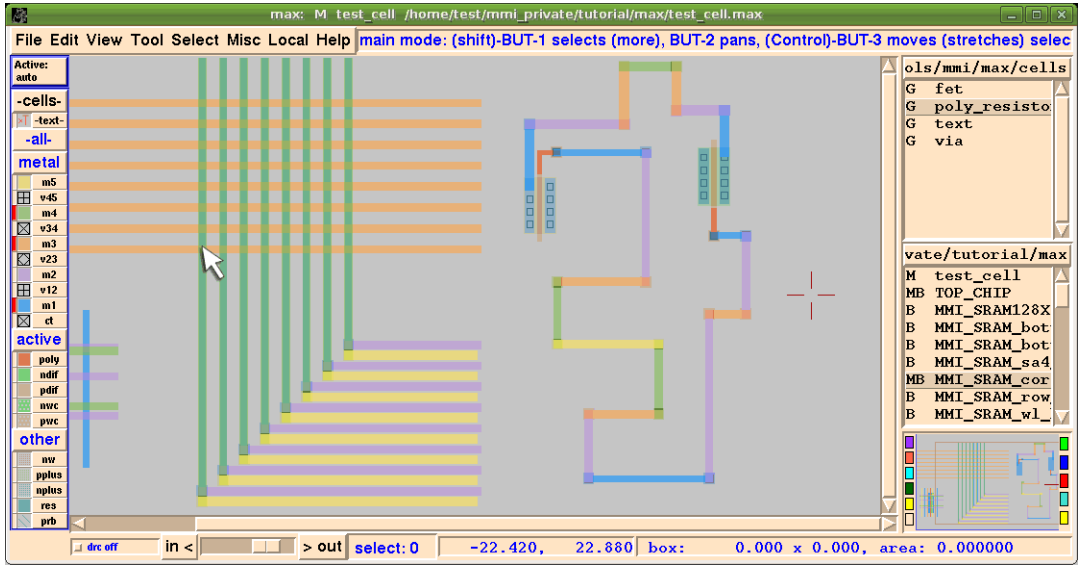
KP_RIGHT is the right arrow key on the number key pad to the right of the main key pad. If your key pad is mapped differently, you can always use the **Pan scroll bar** at the bottom of the MAX window, or mouse **Button-3** in the Navigation Window.

TIP:

You can also pan left and right by holding down the **Shift** key while using the **scroll wheel** on your mouse. If you hold down the **Ctrl** key, MAX will pan up and down.

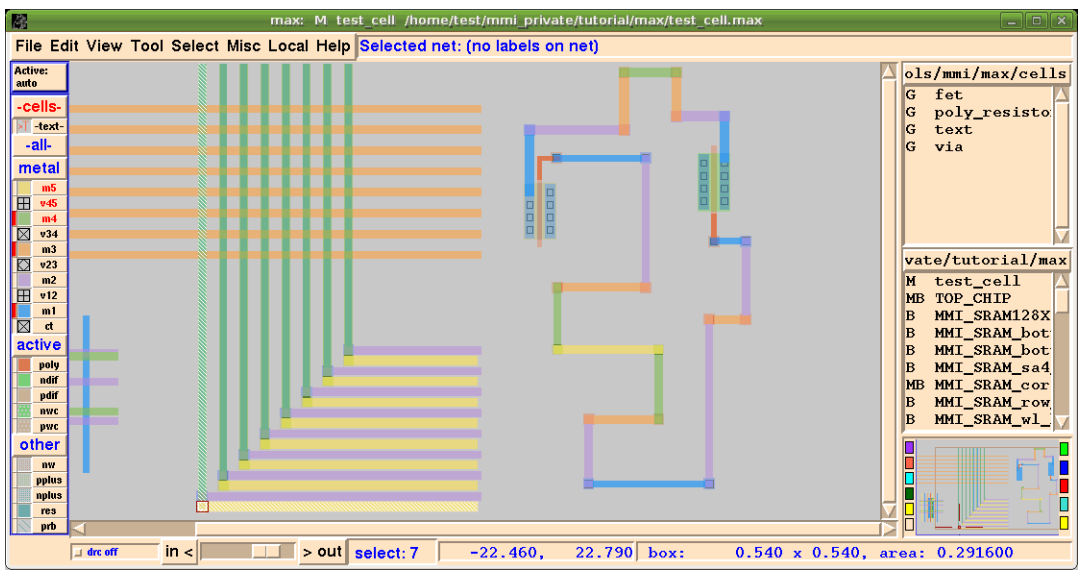
Step 2 ⇨ Move the cursor over an area where `m4`, `m3` and `m1` overlap, as shown in Figure 37 below.

Figure 37: Cursor on Overlapping Layers 'm4', 'm3' and 'm1'



Step 3 ⇨ Click on the **Select Net** hotkey: **s**. You should see something like Figure 38.

Figure 38: Select Net at Cursor Position



⇨ MAX selected the net containing m4.

⇨ Typing the **s** hotkey a second time at the same location selects the net containing m3 below.

⇨ Typing the **s** hotkey a third time selects the net containing m1.

TURNING OFF SELECTABILITY

If you know that you want to select the net containing m1, you can ***turn off the selectability*** of the other metal layers. This way you only have to **Select Net** one time.

Step 4

⇨ On the palette, click with **Button-3** on the layer names m4 and m3. These layer names should now be greyed out.

⇨ Move the cursor over the same location as shown in Figure 38 above. Type the **s** hotkey and notice that the net cotaining m1 was selected. Remember, you can always look to see which layer names in the palette are displayed in red to find selected layers.

BIGGER DESIGNS AND HIERARCHY

Show/Hide Internals

In this section, we will look at ways to view the layout through the hierarchy.

HIDING INTERNALS

Step 1

⇨ Load TOP_CHIP again into the MAX window by clicking at the beginning of the Cell List Box with **Button-3**.

TIP:

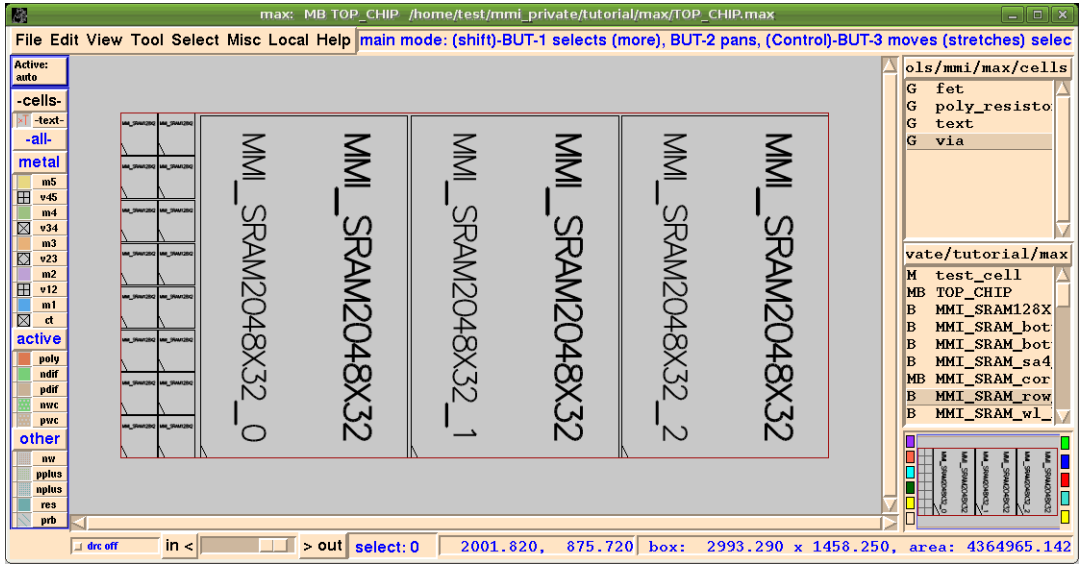
Remember that if you click with **Button-1**, MAX will ***instantiate*** TOP_CHIP in your current cell. Don't worry, you can always use the **undo** (hotkey: **u**) command.

If you have not restarted MAX, then all layers and all levels of hierarchy are visible.

Step 2 ⇨ Type the **Internals, Hide All** hotkey: **h**. This hides all of the internals of all the cells so you are viewing only the top level instantiations of cells.

You should now see something similar to Figure 39.

Figure 39: Hiding Internals of 'TOP_CHIP.gds'



One way of looking at the hierarchy of the layout is to view internals of the entire cell or selected subcells. At the beginning of this tutorial we already viewed internals of the entire cell using the hotkey **i** (**Internals, View All** from the **View** menu).

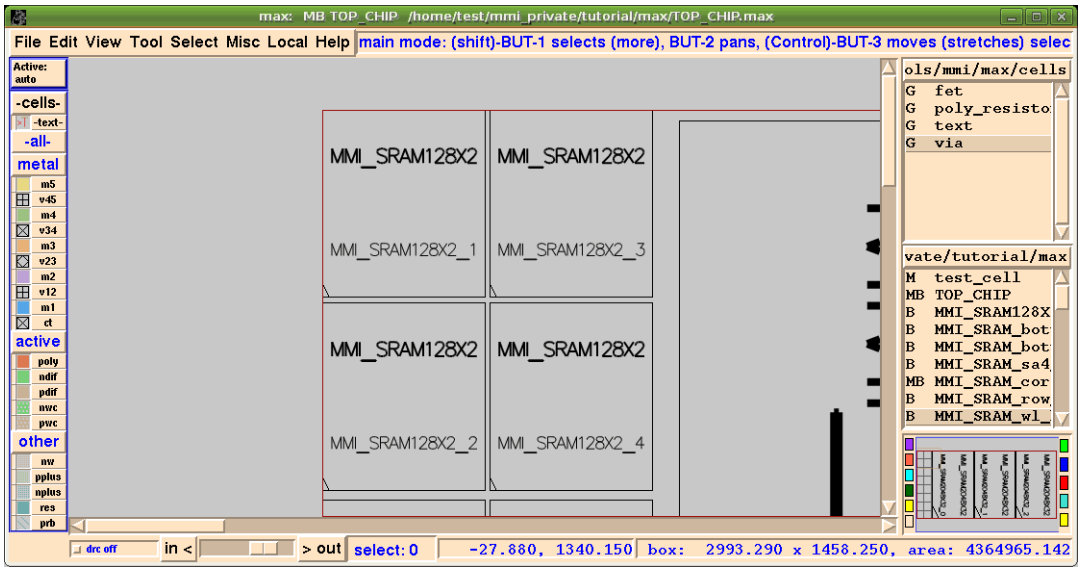
Now we'll look at viewing the internals of selected cells.

VIEWING SELECTED CELL INTERNALS

First we will look down at the next level of hierarchy of a cell.

Step 1 Zoom in on the upper left corner of `TOP_CHIP` as shown in Figure 40.

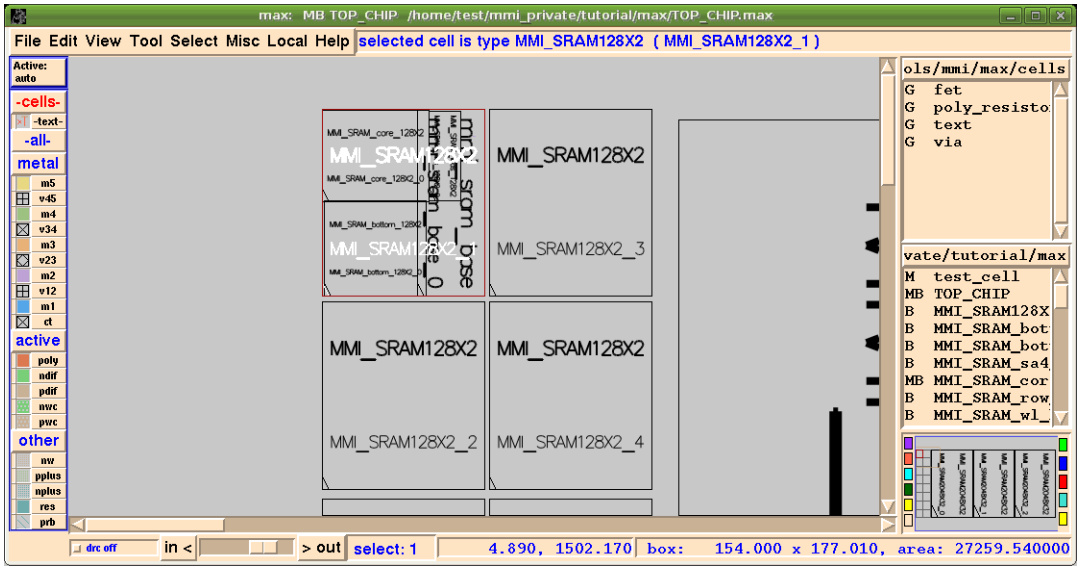
Figure 40: Zoom In On Upper Left of 'TOP_CHIP.gds'



Step 2

- Select the upper left `MMI_SRAM128x2` cell by clicking on it with **Button-1**.
- Type **Shift-i**, or select **Internals, Show Cell** from the **View > Internals, Misc.** menu.
- You should now see something similar to Figure 41. (Typing **Shift-h** will **Hide Internals** for the cell.)

Figure 41: Viewing Internals fo 'MMI_SRAM128x2' Cell



You are now looking at the top level cells for the upper left MMI_SRAM128x2 cell.

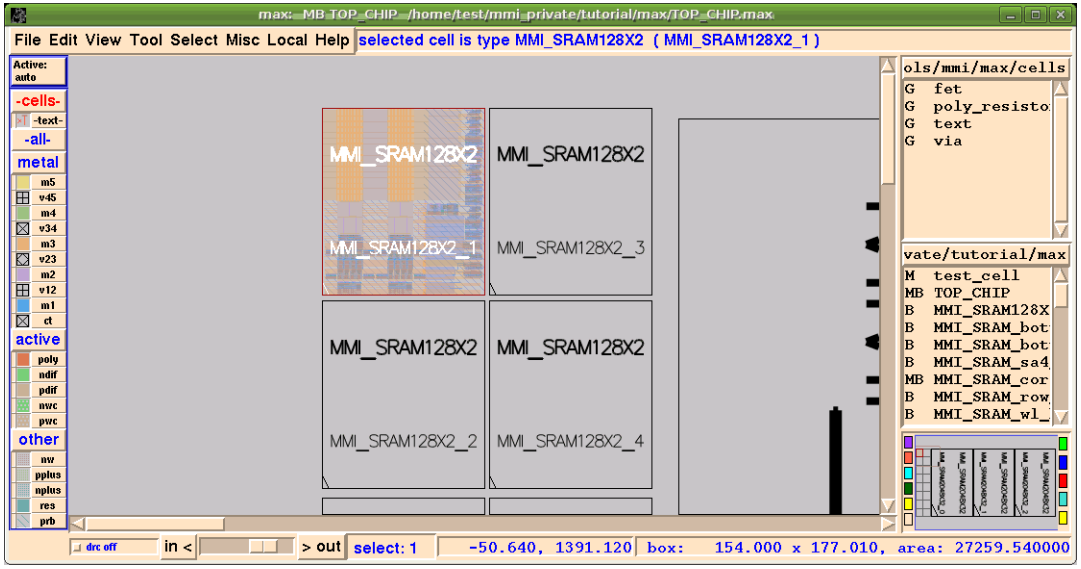
Step 3 ⇨ To veiw all internals for the MMI_SRAM128x2 cell, select **Internals, View Area** frim the **View > Internals, Misc.** menu.

TIP:

If you find that you are using a command often that doesn't currently have a hotkey, you can edit the hotkeys and add a hotkey for this command.

You should now see something similar to Figure 42. All levels of hierarchy for the upper left MMI_SRAM128x2 are visible.

Figure 42: Viewing All Area Internals of Cell 'MMI_SRAM128x2'



VIEW INTERNALS OF HIERARCHY

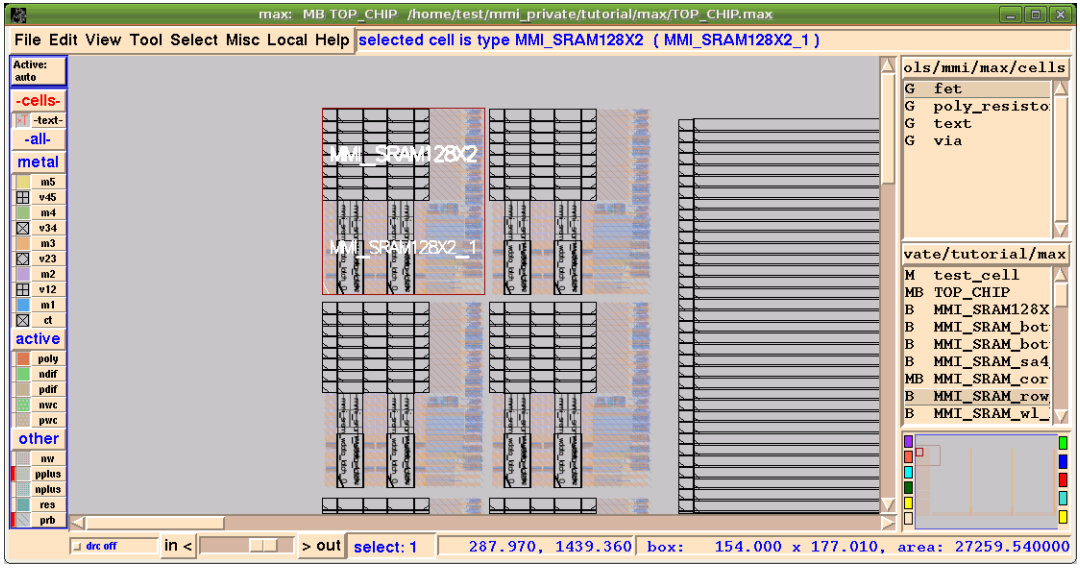
If you want to continue viewing down another level of hierarchy, you can use the **Internals, View More** command.

Step 4

⇨ First **hide** all of the internals by typing the **h** hotkey.

⇨ Type hotkey **Ctrl-i** 3 times. You should now see something similar to Figure 43.

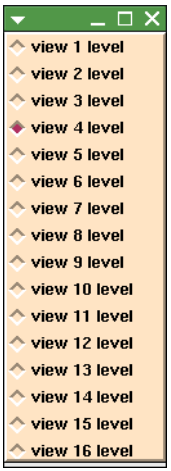
Figure 43: Internals, View More of Cell 'MMI_SRAM128x2



Step 5 ⇨ Another way to view levels of hierarchy is to select **Internals, View Levels** from the **View** menu.

⇨ The easiest way to use this feature is to tear off the sub-menu. Do this by clicking with **Button-1** on the dotted line at the top of the sub-menu. You should now see the pop-up in Figure 44.

Figure 44: Internals, View Levels of Cell 'MMI_SRAM128x2



Step 6 ⇨ Click on the different levels and notice the results.

Push & Pop

In addition to the **zoom** and **internals** features for really getting to know your layout, you can **Push into** or **Pop out of** cells. ***Pushing into*** goes deeper into the hierarchy one level at a time; ***popping out*** moves back up through the hierarchy.

Pushing Into/Popping Out Of Cells

TO PUSH IN

Step 1 ⇨ You should still have `TOP_CHIP` loaded in MAX. **View all levels** of hierarchy using the **i** hotkey.

Step 2 ⇨ Select a cell using **f**, or **Select Cell** from the **Select** menu.

Step 3 ⇨ Now type **e**, or select **Push into cell** from the **View** menu.

TO POP OUT You will now be viewing just the cell you selected. To get back to the top level of hierarchy, you can pop up to that level.

Step 4 ⇨ Type **Ctrl-e** or select **View > Pop out of Cell** to "pop up" a level.

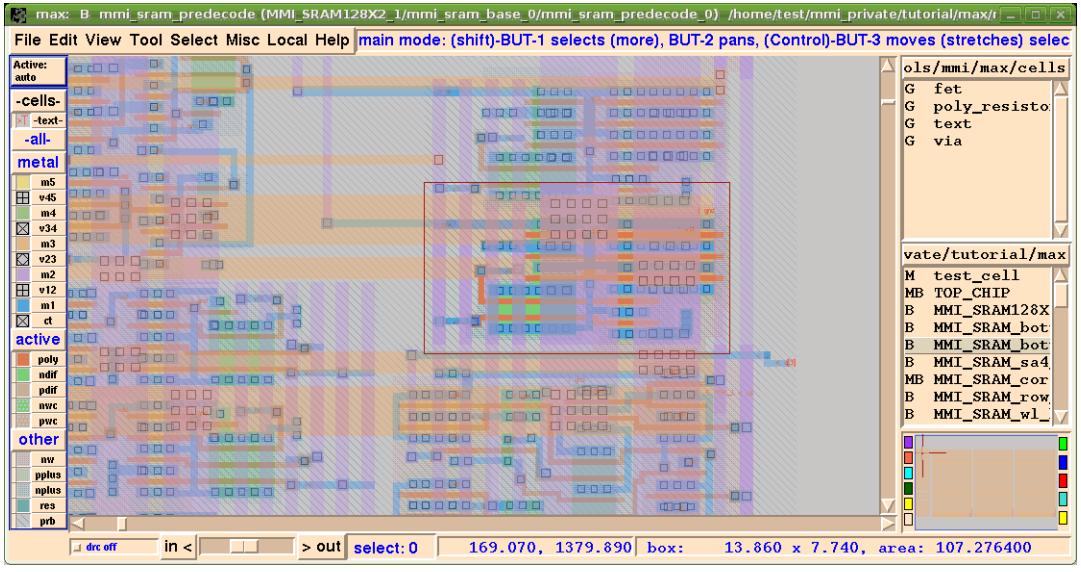
⇨ You can continue using **e** or **Ctrl-e** to go further into or out of levels.

EDIT IN PLACE If you want to edit a cell and also see all of the layout around it, you can ***edit a cell in place.***

Step 5 ⇨ From the top level again select a cell using the **f** hotkey. Now type **Shift-e** or select **View > Edit Cell or Object in Place**.

The cell you are editing in place will show darker than the surrounding layout, as demonstrated in Figure 45.

Figure 45: Edit Cell in Place



Cell Hierarchy Browser

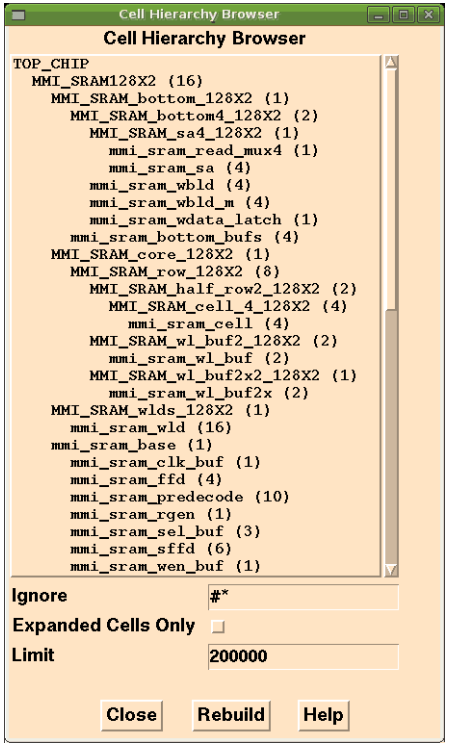
Yet another way to explore the hierarchy of your chip or block is to use the **Cell Hierarchy Browser**.

BROWSING CELL HIERARCHY

Make sure you are back at the top level of TOP_CHIP. You can always **RIGHT click** (**Button-3**) on TOP_CHIP in the Cell List at the right.

Step 1 ⇨ Select **Display Hierarchy...** from the **View** menu, or type **Shift-I** (ell). You should now see a pop-up like Figure 46.

Figure 46: Cell Hierarchy Pop-up



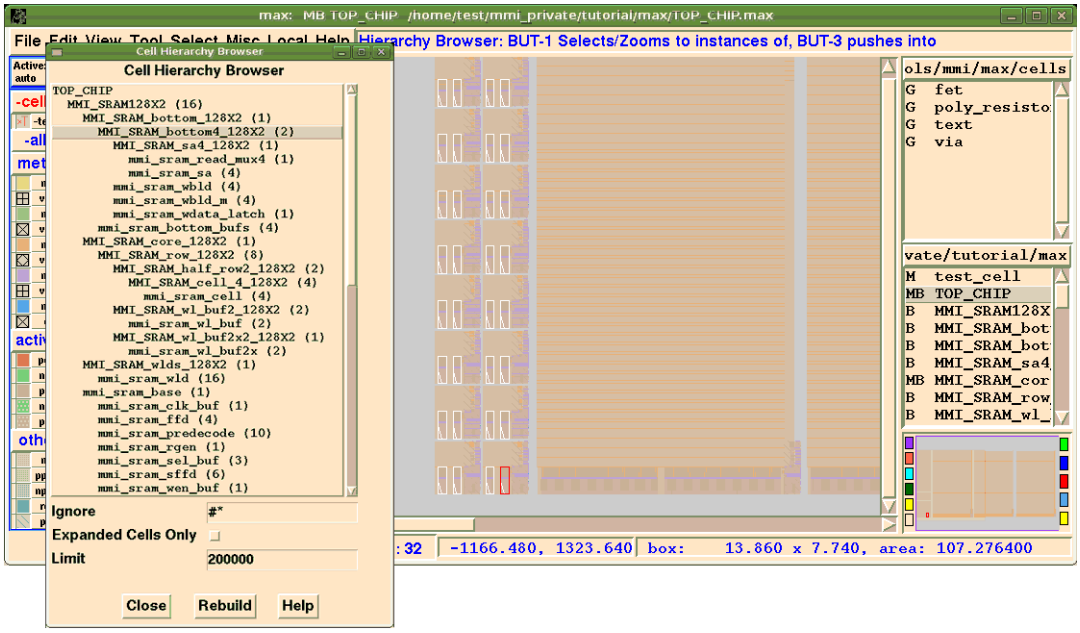
⇨ If you click with **Button-1** on any of the cells in the hierarchy browser, all instances of the cell at that level of hierarchy will be selected in the layout and you will be zoomed out to fit all of those instances.

Step 2 ⇨ Click (**Button-1**) on MMI_SRAM_bottom4_128x2 toward the top of the **Cell Hierarchy Browser** list as shown in Figure 47.

The '2' in brackets, to the right of MMI_SRAM_bottom4_128x2, indicates there are 2 of these cells at this level of hierarchy. Because there are

16 copies of MMI_SRAM128x2, there will be 32 MMI_SRAM_bottom4_128x2 cells selected, as shown in Figure 47.

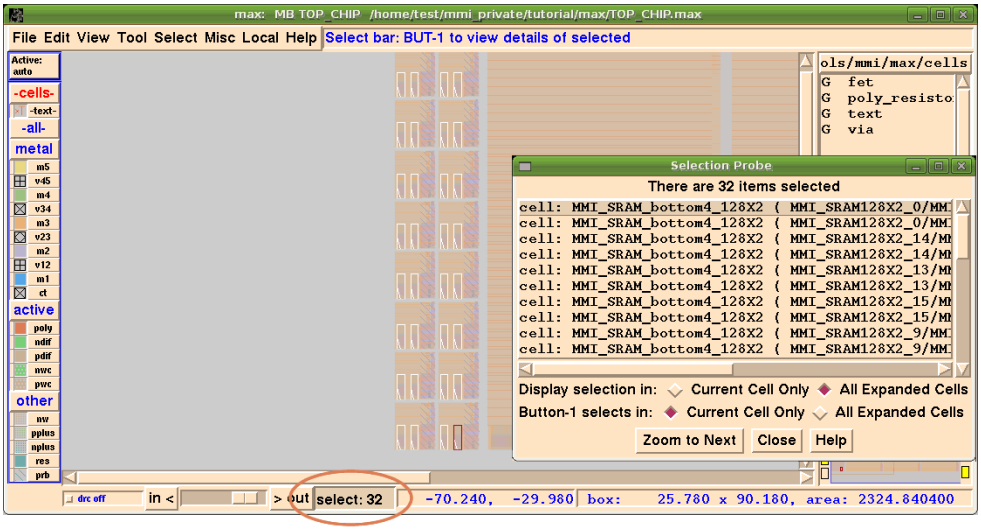
Figure 47: Selecting 'MMI_SRAM_bottom4_128x2' in Cell Hierarchy Browser Shows 32 Selected Cells in Layout



If you want to zoom to each of them, you can use the **Selection Probe**.

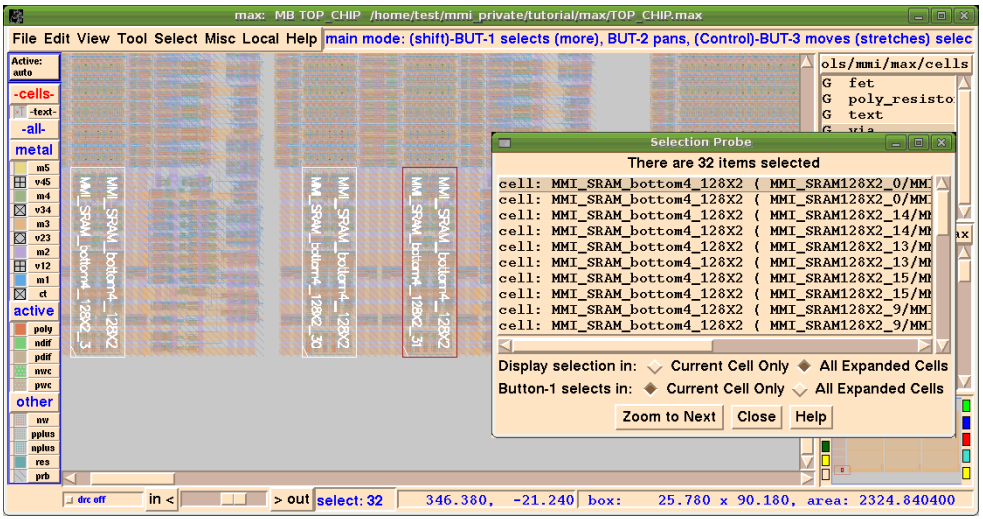
Step 3 ⇨ With the cells still highlighted, click on the **Select Box** at the bottom middle of the MAX window as shown in FXX. It should say "`select: 32`".

Figure 48: Click on Select Box to Open Selection Probe Pop-up



Step 4 ⇨ Click on one of the cells (**Button-1**) or click on the **Zoom to Next** button to step through each of the `mmi_sram_clk_buf` cells. You should see something similar to Figure 49. When you are done, click on **Close**.

Figure 49: Zoom to Next Cell in Selection Probe

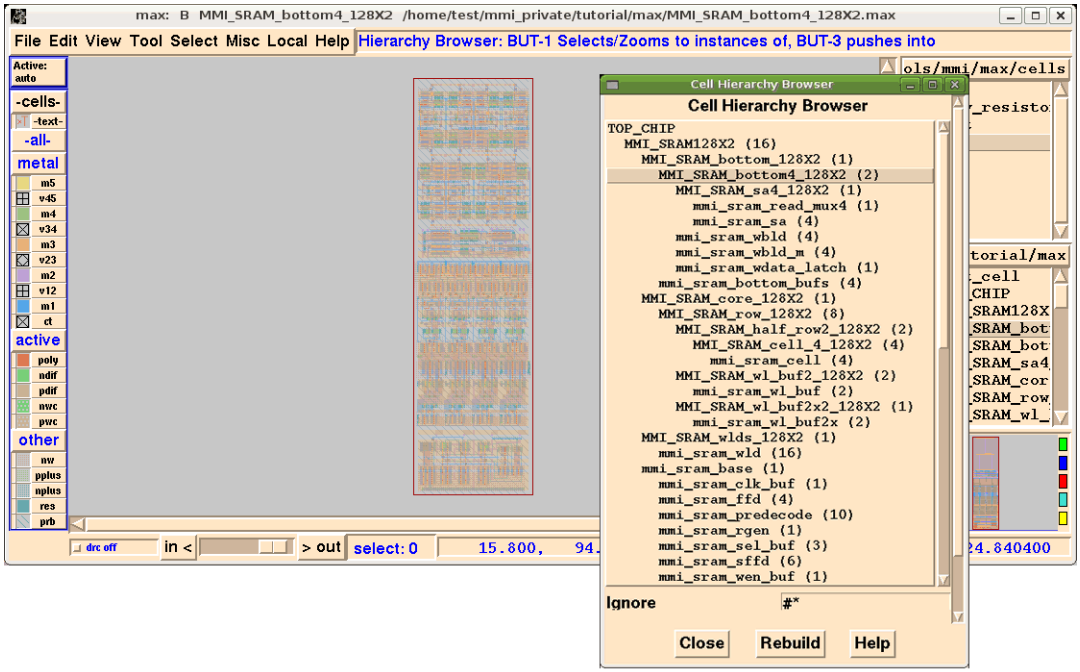


Step 5 ⇨ On the Cell Hierarchy Browser, **RIGHT-click** (**Button-3**) on `MMI_SRAM_bottom4_128x2`.

⇨ If you can't find the **Cell Hierarchy Browser**, you can always select **View > Display Hierarchy**, or type the hotkey **Ctrl-I** (ell). You should see something similar to Figure 50.

⇨ When you are done, click on **Close**.

Figure 50: 'MMI_SRAM_bottom4_128x2' Cell Loaded Into MAX from Cell Hierarchy Browser



Next, we'll see how MAX handles interactive real-time DRC, a useful and powerful feature.

VIEWING DRC ERRORS IN MAX AND WITH THE CALIBRE® INTERFACE

MAX includes interactive DRC (design rule) checking. You can step through DRC errors and fix them on the fly. MAX also includes an interface to the Mentor Graphics' *Calibre*® DRC checker where MAX is used to step through and view each of the DRC errors. You can interactively correct the DRC errors as you go. MAX reads in the `<cell_name>.drc_maskdb` results from a Calibre® run.

MAX can also interface with Calibre® RVE. For information on how to use this interface, refer to the *MAX User Manual*.

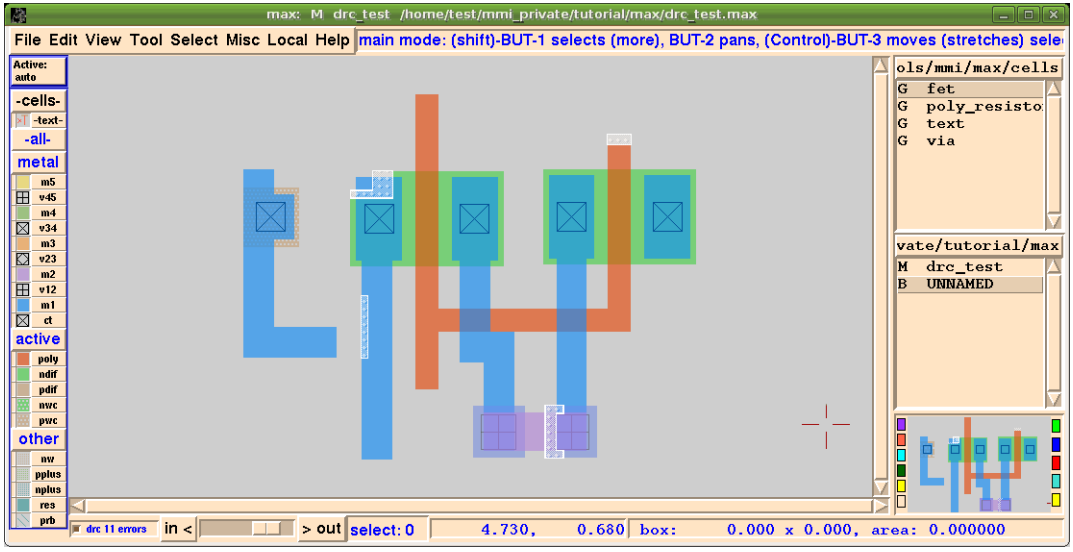
VIEW MAX DRC ERRORS

We'll load a small text cell which has some DRC errors and for which we've already run Calibre®.

Step 1 ⇨ Select **Open...** from the **File** menu. This will open a layout file that is in the MAX format.

Step 2 ⇨ Select `drc_text.max` and click on **OK**. You should now see something like Figure 51.

Figure 51: Loading 'drc_text.max' File



Notice that MAX displays DRC errors as dotted white rectangles. (DRC colors can be easily modified; we'll cover this in the section **Modifying Palette/Layer Styles within MAX** (page -148).)

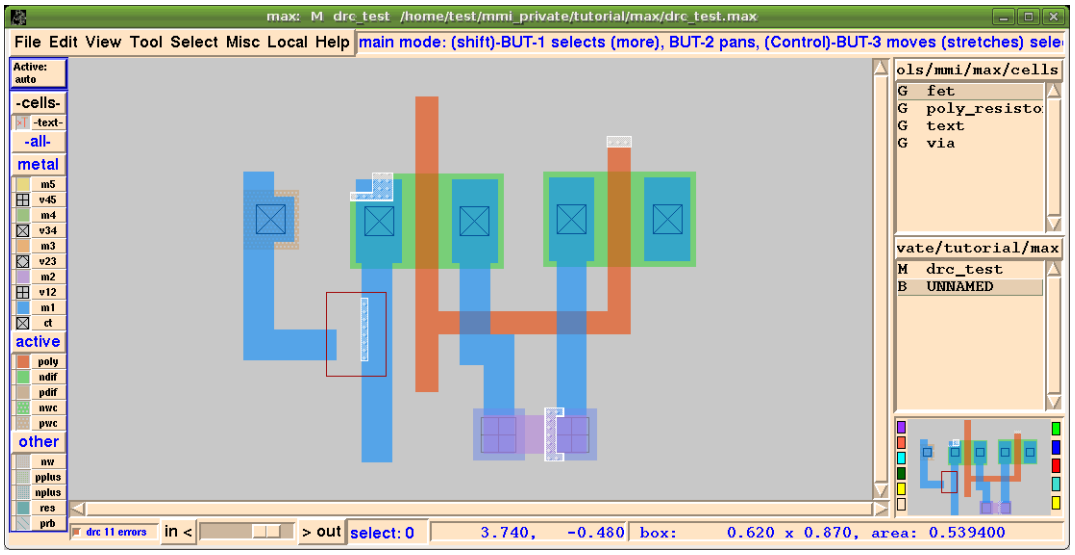
Step 3 ⇨ Look at the box at the lower left corner of the MAX window and notice that it says "`drc 11 errors`". This means there are 11 DRC errors in this cell.

There are a number of ways to look at the DRC errors. Let's say you're interested in finding out what a specific DRC rectangle

represents. You can daw a box around that error and ask MAX what it represents.

Step 4 ⇨ Type the hotkey **b** for menu select **Misc. > Make/move Box**. Hold down **Button-1** and drag a box around the lower left DRC error, as shown in Figure 52.

Figure 52: Dragging Box Around Lower Left DRC Error



TIP:

You could also have simply dragged out a box around the DRC error without using the **Make/move Box** command, but this would also have ***selected the layout*** under the box.

Step 5 ⇨ Type hotkey **Shift-y**, or menu select **Misc. > Explain DRC under Box**. In the Message Area at the top of the MAX window you should see:

```
DRC: m1 minimum spacing - 0.32 um.
```

⇨ If you drag a box around multiple DRC errors, the MAX Message error will display the number of DRC errors under the box and tell you to look at the MAX command window (the shell window from which you started MAX) for details.

Step 6 Another way to find out what the DRC errors are is to step through them.

⇨ Type hotkey **Shift-n** or select **DRC Find Next Error** from the **Misc.** menu. MAX will zoom in on a DRC error and display the text of the error in the MAX Message Area.

⇨ Continue to type **Shift-n** to step through the remaining DRC errors.

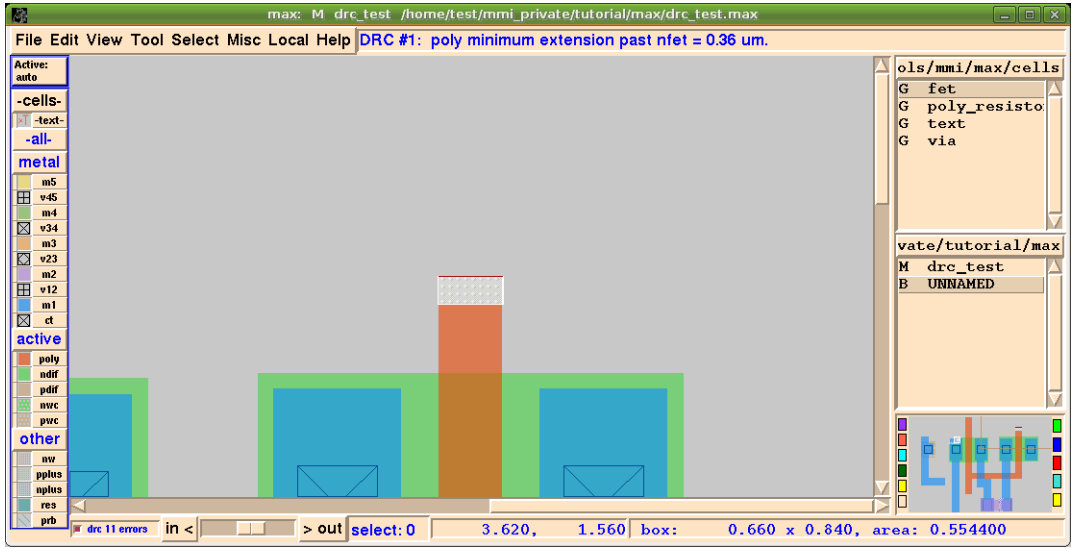
The **Shift-n** hotkey can be very useful if you're looking at a reasonably large block and MAX informs you of a DRC error (bottom left corner of the MAX window). Sometimes you can't see where the error is if it's small, and you can use **Shift-n** to find the DRC error.

FIXING DRC ERRORS

As you are stepping through the DRC errors, you can fix them on the fly.

Step 1 ⇨ Step through the DRC errors using the **Shift-n** command until you see "DRC #1" in the Message Area, and you see something similar to Figure 53.

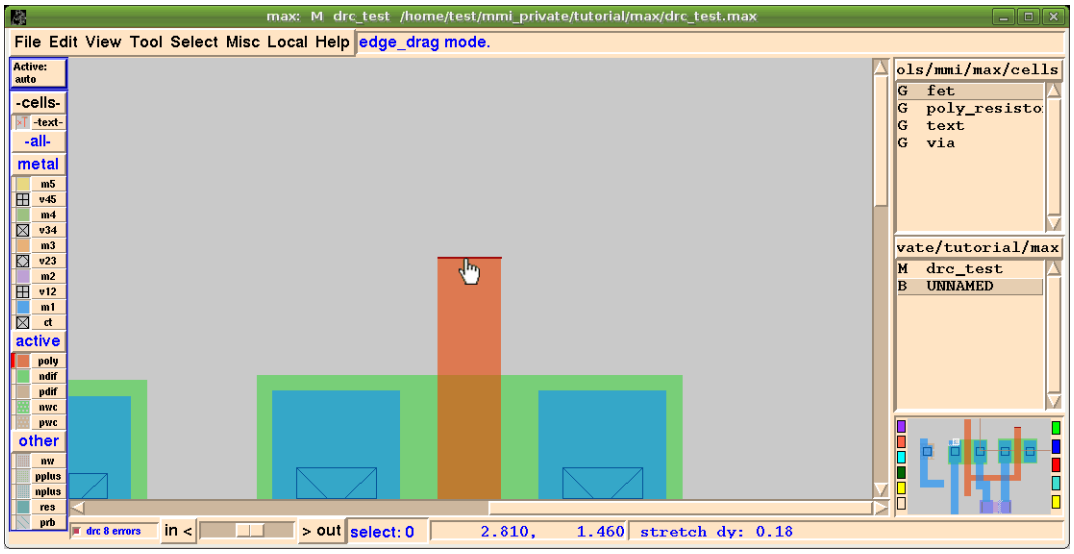
Figure 53: Using 'Shift-n' to Find Error 'DRC #1'



Step 2 ⇨ Type hotkey **a** or menu select **Edit > Edit Edge**. Notice the cursor has now changed to a hand with a pointing finger, as shown in Figure 54.

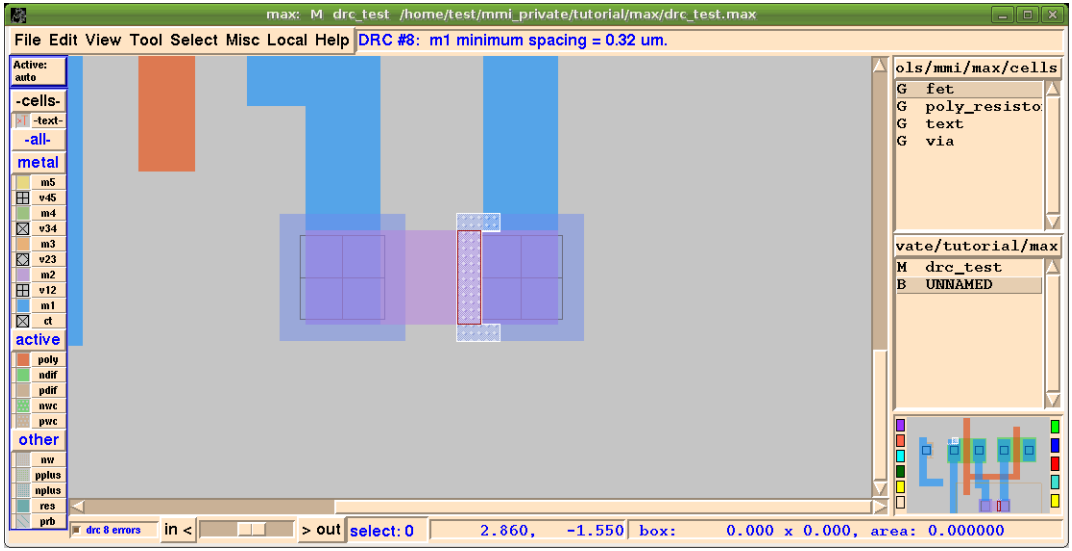
⇨ Move the cursor near the upper edge of the poly. Hold down **Button-1** and drag the mouse upwards (thus moving the poly) until the DRC error disappears.

Figure 54: Editing Edge To Remove DRC Error



Step 3 ⇨ Let's fix another DRC error. Type **Shift-n** until you see "DRC #8" in the Message Area. You should see something similar to Figure 55.

Figure 55: Using 'Shift-n' to View DRC Error #8

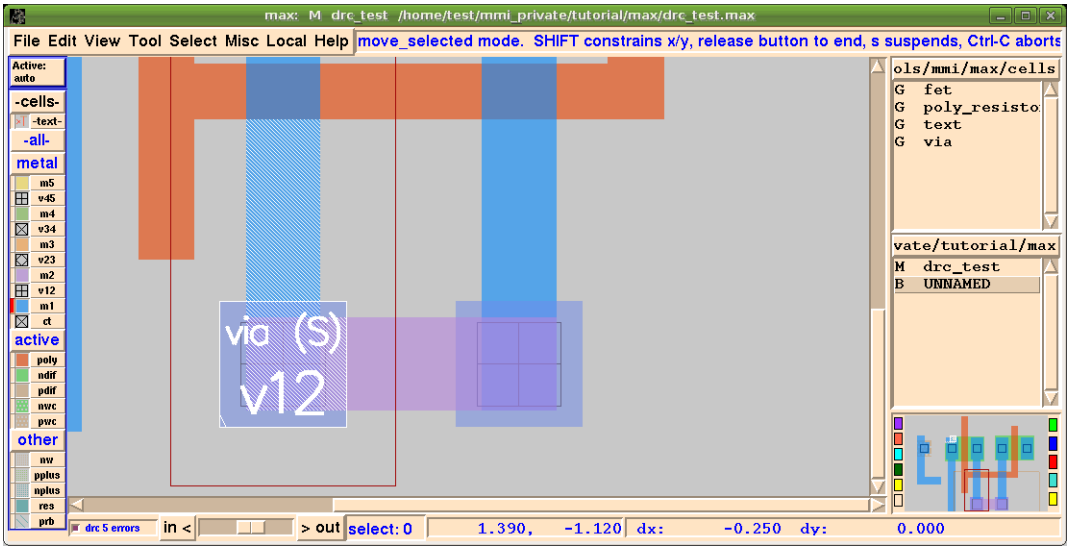


Step 4 ⇨ We'll use the **Edit Wire** command to fix this DRC error.

⇨ Type the hotkey **Ctrl-w** or menu select **Edit > Edit Wire.**

⇨ Move the cursor over the lower left `m1` segment as shown in Figure 55, ***above***. Hold down **Button-1** and slide the mouse to the left until the two vertical pieces of `m1` are aligned, as seen in Figure 56 ***below***.

Figure 56: Using the 'Edit Wire' Command to Fix DRC Errors



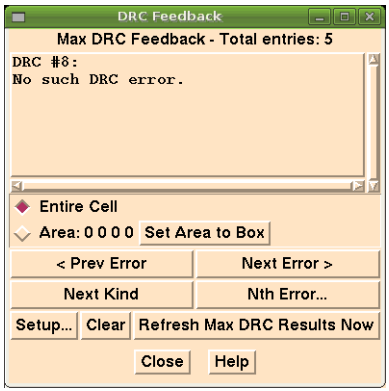
You should now only have 5 DRC errors left. Look at the DRC status box at the lower left corner of the MAX window.

DRC FEEDBACK WINDOW

We'll now look at a third method to step through your DRC errors using the DRC Feedback window.

Step 1 ⇨ Go to the **Misc**. menu and select **DRC Results**, or type hotkey **Ctrl-y.** A pop-up like Figure 57 will open.

Figure 57: DRC Feedback Pop-up



The **DRC Info** section of the pop-up may say:

```
DRC #8
No such DRC error.
```

Remember, we just fixed `DRC error #8`, so it doesn't exist any more. The **DRC Feedback** window comes up showing the most recent DRC error you were viewing.

Step 2

⇨ Click on the **Next Error** button to step through each of the remaining DRC errors and fix them as we did earlier.

⇨ Close the **DRC Feedback** window.

VIEWING CALIBRE® DRC ERRORS

MAX does not check all DRC rules: for example, antenna rules or wide metal rules. To check for these errors, you need to use an external DRC checking tool. MAX includes an interface to Mentor Graphics' Calibre® tool.

For this tutorial we assume you don't have the Calibre® interface available, so we've already run Calibre® on this cell.

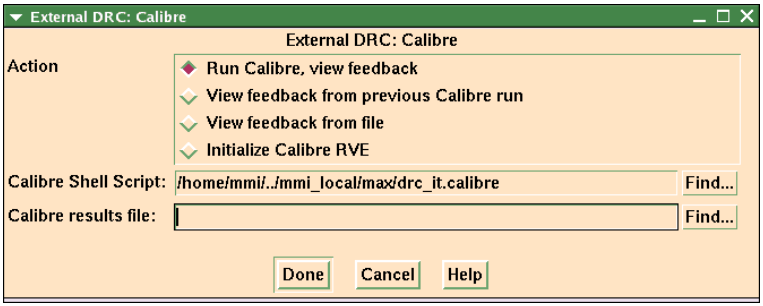
Step 1

⇨ Since Calibre® was run on the cell with all of the DRC errors, we need to first get back to the original cell. Either type the **undo** hotkey **u** multiple times to step back through all DRC errors, or go to the **File** menu and select **Revert to Last Saved**.

⇨ Click **Yes** when asked:
"Really throw away changes to cell 'drc_test'?"

Step 2 ⇨ Now go to the **Tool** menu and select **Calibre DRC...** to open a form like Figure 58.

Figure 58: Calibre DRC... Pop-up Menu



If the Calibre® interface **is** set up at your site, you can run Calibre® directly from MAX and then view the results. For this example, Calibre® has already been run.

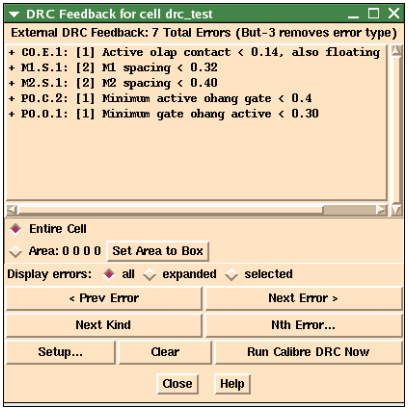
Step 3 ⇨ Select **View feedback from file** and then click on **Done**. You should now see a pop-up form like Figure 59.

⇨ By default, MAX looks in the same directory as your cell to find the Calibre® results file. If it is named `<cell_name>.drc_maskdb`, then it will be found automatically. Otherwise, you can specify the file using the **Find...** button.

CAUTION:

If you forget to select **View feedback from file**, the Calibre® output file could get overwritten. If that happens, you can click on the **Find** button and specify the Calibre® results file as: `drc_text.drc_maskdb.save`.

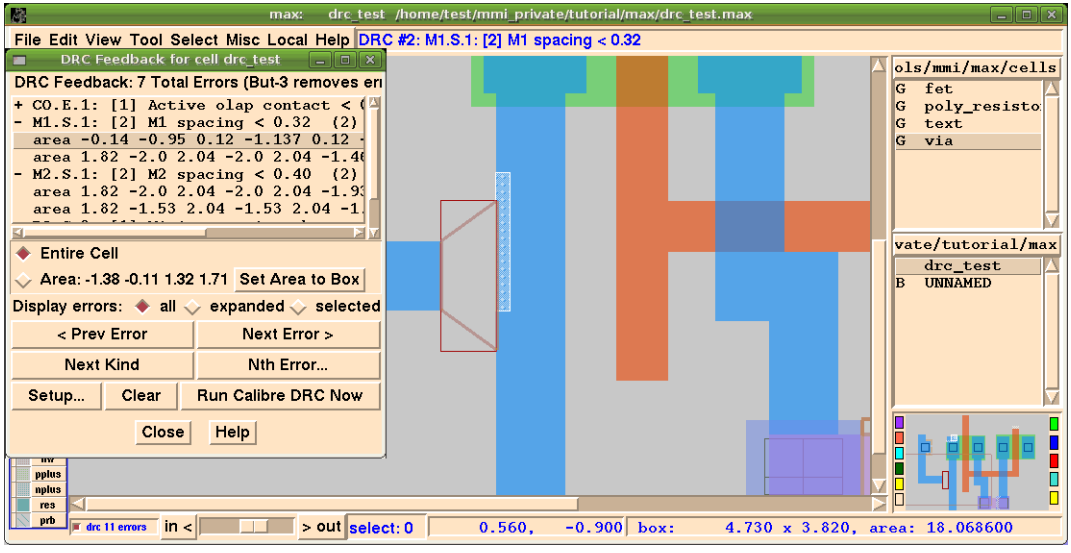
Figure 59: Viewing Calibre® Feedback Pop-up Form



The form lists each of the DRC errors reported by Calibre®. Clicking on the error types expands or collapses that error.

Step 4 ⇨ Click on **Next Error**. The MAX window should now look similar to Figure 60, with the DRC error polygons shown in brown.

Figure 60: DRC Pop-up Showing Error Listed, and MAX Window Zoomed In on Error



MAX automatically zooms you in on the first DRC error. You can also click on a specific DRC error in the list and MAX will zoom to that error location. In the above example, `error #2` has been selected.

Refer to the *MAX User Manual* for more information on the Calibre® DRC interface.

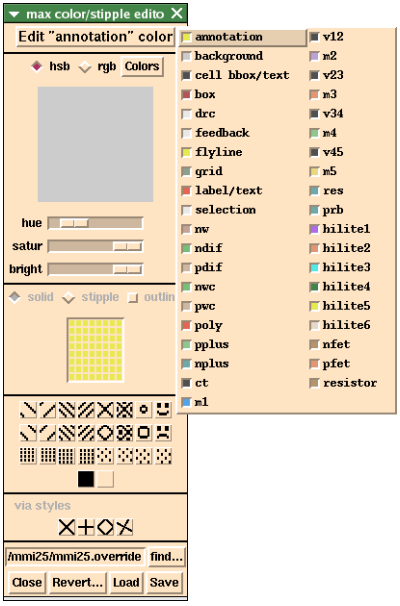
CHANGING COLOR OF DRC ERRORS

If you don't like the color used to display the location of DRC errors, you can always change it in the **Color Editor.**

Step 5

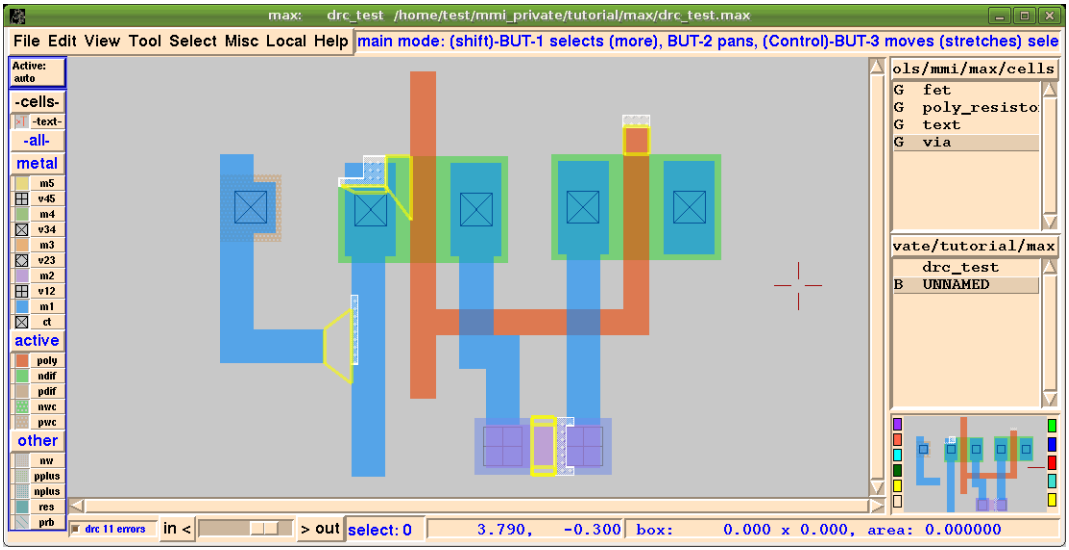
- From the **File** menu select **User Preferences > Color Editor...**
- At the top of the editor panel, click and hold down the button displaying **Edit layer "nw"** and then select **annotation** from the list, as shown in Figure 61.
- Hold down the **Colors** button and select **yellow**, the last color in the list.
- Click on **Close** in the **Color Editor**.

Figure 61: Color Editor: "annotation" Layer Changed to Yellow



You should now see a display similar to Figure 62, with the DRC error polygons showing yellow.

Figure 62: DRC Error Color Changed to Yellow



In this fashion, you can customize the MAX display colors as desired, and see the changes instantly.

Now we'll go on to try something really fun.

VIEWING LAYOUT IN 3D

Yes, it's true! MAX (and MAX-LS) provide a symbolic **three-dimensional view** of your layout, generated real-time, on the fly. While there is no true commercial use for a three-dimensional view yet, it can be quite informative to see how layers of a chip actually interact. (Please note this is a VIEW only. True 3D Layout and Editing are part of the MAX-3D Layout Editor.)

ISSUE:

Please keep in mind that this is a symbolic or representational 3D view, not an actual one, and meant more for informative purposes.

In MAX and MAX-LS, Z-axis dimensions are estimated and can be specified in your tech file if needed. Distances between layers are also estimated in the 3D view. Refer to the *MAX User Manual* for information about setting Z-axis dimensions and coordinates.

The 3-D View is not as fast as the standard layout view. Smaller cells work best and are easiest to see, so we'll be selecting one of the smaller cells from the Cell List Box.

Viewing in 3D

If you have exited MAX and do not still have `TOP_CHIP` loaded, you first need to load it.

Step 1 ⇨ Got to **File > Import File**.

⇨ Select **Load GDS File using current mmi25 max technology file**.

⇨ Click on **Find...** and select `TOP_CHIP.gds`, then click on **Done**.

First we'll put the cells in alphabetical order to make it easier to find cells

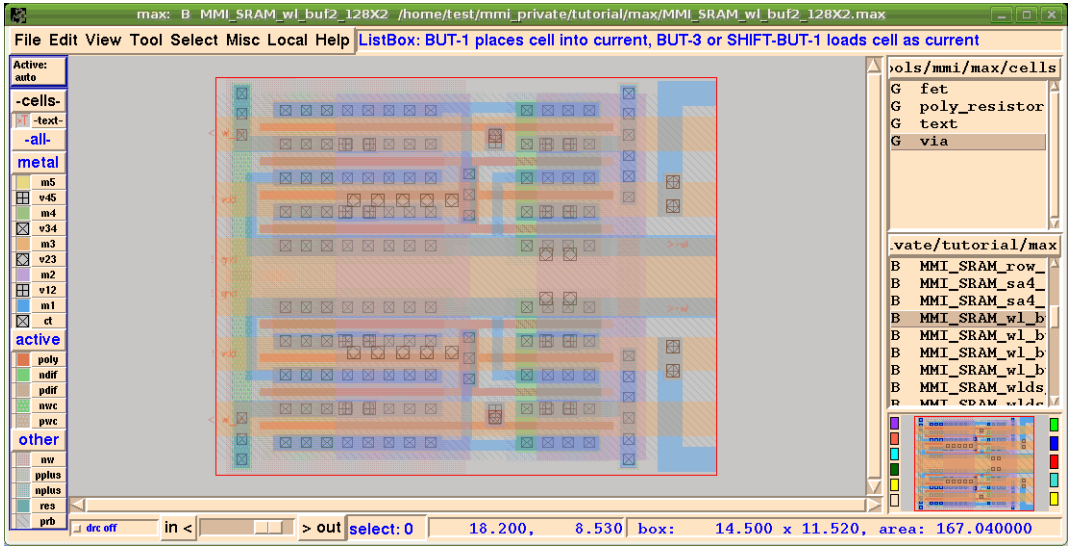
Step 2 ⇨ Hold down the title bar at the top of the Cell List Box (in the illustration below, it says "`/home/text/mmi_private/tutorial/max/Samples`") and select **Alphabetize** from the menu options.

Step 3 ⇨ Click on **MMI_SRAM_wl_buf2_128x2.max** with the **RIGHT (Button-3) mouse button**. It's the first `MMI_SRAM_wl...` shown highlighted in Figure 63 below.

You should see this cell (notice that the cell name is displayed along the top).

Step 4 ⇨ If you can't see the internals of the cells, type the hotkey **i** (**View Internals**).

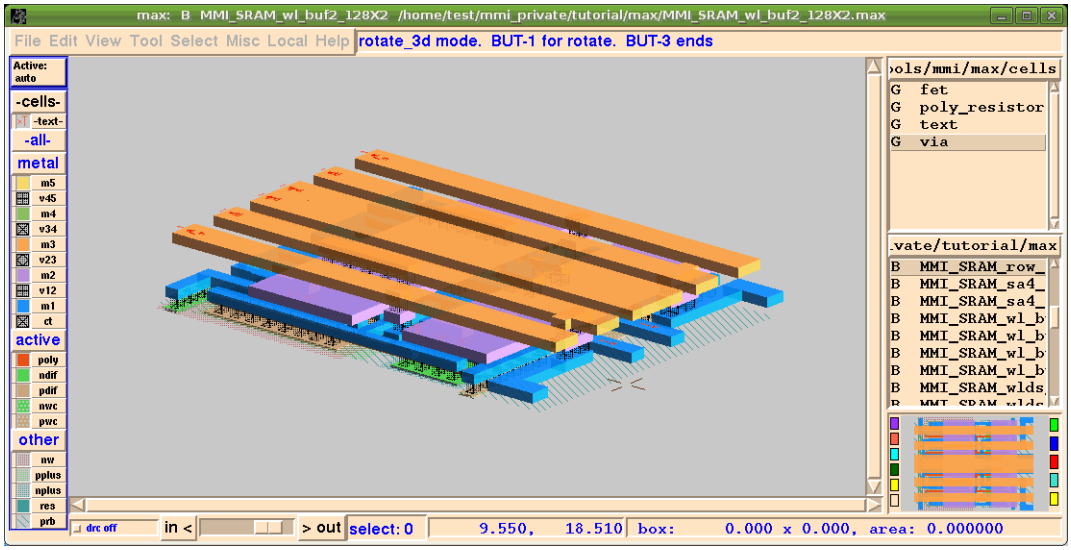
Figure 63: MMI_SRAM_wl_buf2_128x2.max



Step 5 ⇨ Now, from the **View** menu click on **Display 3D View,** or use the hotkey **Alt-z**. The cell display immediately changes to a standard orthogonal 3-D view, shown in Figure 64.

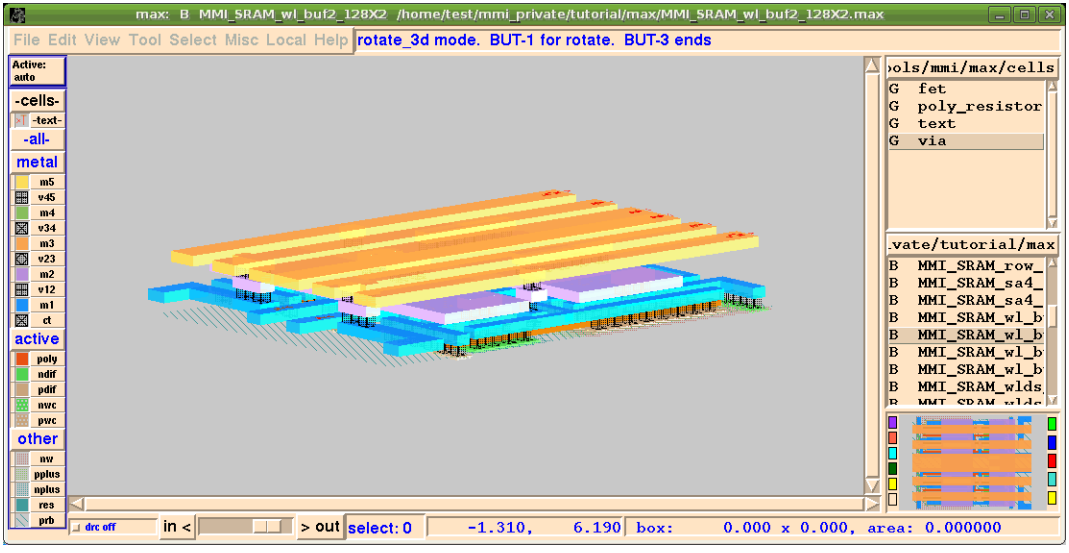
⇨ From here, you will be able to rotate and zoom in/out of the view using the mouse buttons. The button functions are described in the MAX Message Area.

Figure 64: 3-D View of MMI_SRAM_wl_buf2_128x2



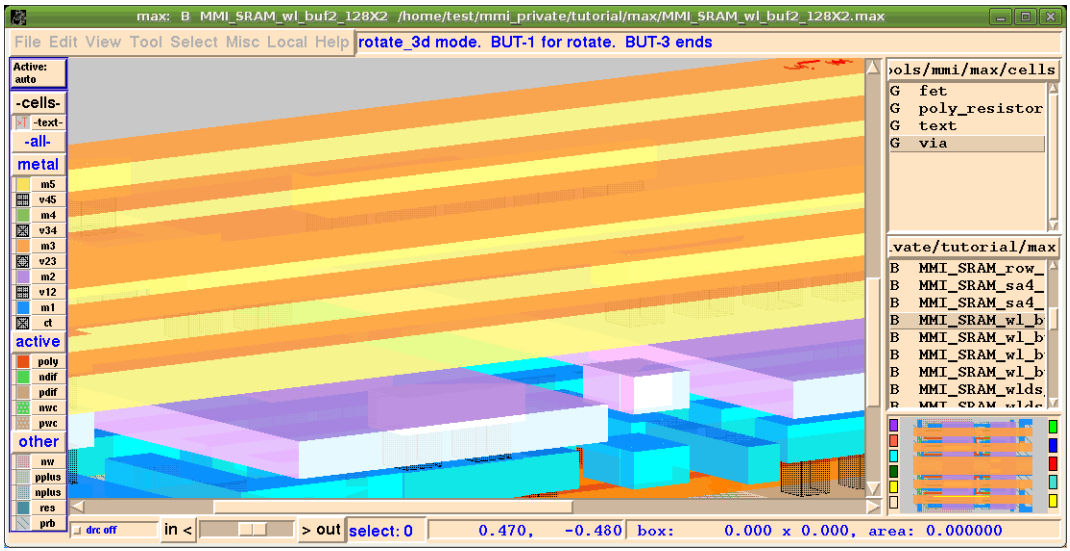
Step 6 ⇨ Rotate the view using **Button-1**. Press and hold down **Button-1** and drag around the window to see the view rotate, as demonstrated in Figure 65.

Figure 65: Rotated 3-D View of the Cell



⇨ Zoom into or out of the view using the scroll-wheel or zoom keys.

Figure 66: Zoomed In on 3-D View



⇨ Pressing the hotkey **v** will re-center the view in the window

⇨ As MAX supports true transparency, layers underneath show slightly through the ones on top. Transparency values can be set in the **Display Options** section of the **User Preferences** menu.

TIP:

NOTE: Editing is not possible during the 3-D viewing mode. You must revert to 2-D to continue editing your cells.

RETURN TO STANDARD VIEW

⇨ Clicking **Button-3** will restore the original layout view (2-D).

⇨ Remember that the Message Area displays the mouse button functions available during 3-D View.

By now you should have a good grasp of MAX's potential. In the next section, we will start creating layout.

If you have purchased MAX-LS Layout System jump now to the "MAX-LS" tutorial.

When you have completed the MAX-LS section return to Part 3 of this tutorial.

Otherwise, continue on to Part 3.

PART 3

SIMPLE LAYOUT

OK, let's get started on some layout. If you have MAX-LS and have just completed the MAX-LS tutorial, some of that information will be repeated in this part of the MAX tutorial.

For this section, we will be using the `mmi18` technology.

Step 1 ⇨ If you still have MAX running, exit MAX (ctrl-d). Start MAX again with the command:

```
max -tech mmi18
```

Step 2 ⇨ To find out what technology you are using, go to the **Help** menu and select **About Max...** It should say that the technology is `mmi18`.

Step 3 ⇨ Go to the **File** menu and select New... This will bring up the **File Select** box.

Step 4 ⇨ Type in the cell name `foo` where it says **Filename:**. Then click **OK**, or hit **Return**.

The **UNNAMED** at the top of yor MAX window should now say `foo`. This means that you are now editing the cell `foo`. When you save it out it will be saved as `foo.max`.

The top of your MAX window also has the path to your cell foo.max. In this example the path is:
"`/home/text/mmi_private/tutorial/max/foo.max`."

Also, notice that `foo` now appears in the **Cell List Box** on the right side of the MAX window. The '`B`' to the left of foo means that this cell only exists in memory (Buffer) and has never been saved.

- Other letters you will find to the left of cell names are:
  "`G`" shows it is a **G**cell (explained later) or Generator cell;
  "`M`" indicating the cell has been **M**odified; and
  "`W`" meaning the cell has been **W**ritten, or saved.

Using Gcells

While MAX can do very low level full custom layout, it also has a complete Tcl/Tk interpreter built in. Tcl is a language developed by John K Ousterhout at UC Berkeley. Tk is the windowing tool kit. Books on both are easily available. The Tcl/Tk interface in MAX allows full access to MAX lsyout functionality, providing unprecedented power and flexibility.

A **gcell** is a ***cell generator*** written in the Tcl/Tk language. The gcell generates layout for a cell automatically from "properties" that you specify using a simple menu interface. MAX comes with gcells to generate FETs, Vias, Text, and a poly_resistor.

CGELL PROPERTIES

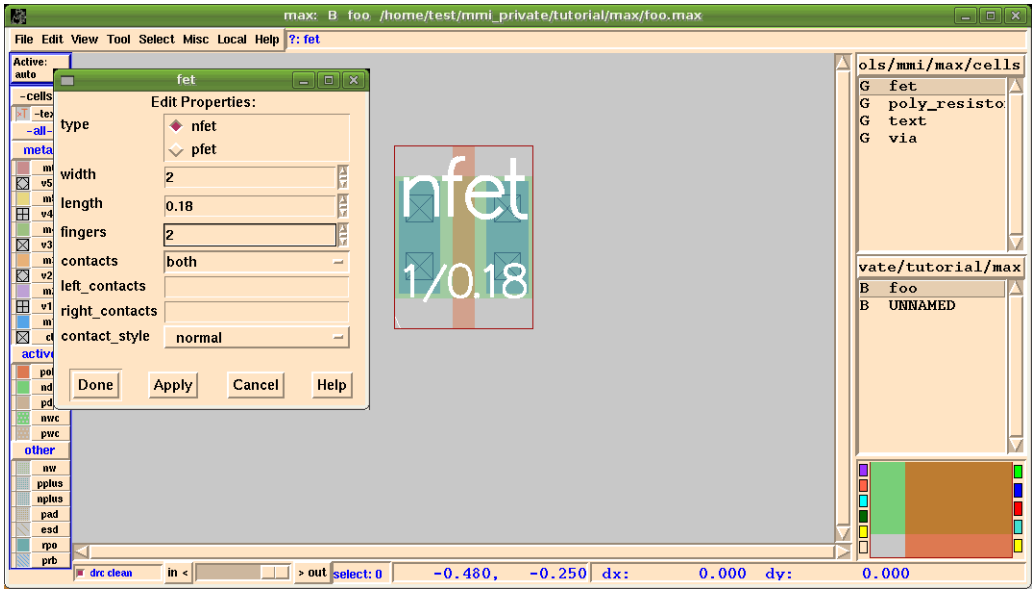
Step 1 ⇨ Go to the **List Box** along the right side and select `fet` from the list by clicking on it with **Button-1**. The `G` next to the fet indicates that this cell is a `gcell`, or *generator*.

⇨ When you click the left mouse button (Button-1) on fet in the List Box, the fet generator window will pop up. It should look something like Figure 67.

TIP:

If you do not see `fet` in the List Box, click and hold **Button-1** over the List Box header to display the directory list, then select the directory ending in `cells`. This is the direcotry containing the fet and via gcells provided with the Micro Magic, Inc. software.

Figure 67: FET Gcell



Step 2 ⇨ Now, change the `width` to *2.0*. This will make the nfet 2.0 µm wide.

Step 3 ⇨ Since we eventually want to use this device to make a 2 input NAND gate, change `fingers` to *2*, and `contacts` to *both*.

Contacts allows you to put contacts on the left side, the right side, both sides, between the devices (all), or no contacts at all.

(Click on **Help** or refer to the Micro Magic, Inc. *MAX User Guide* for more details.)

If you click the **left mouse button** (**Button-1**) on the word **both** it will change to the next option. Clicking **Button-1** on the little bar to the right of this option will bring up all the possible variations.

Step 4 ⇨ Now click **Done**.

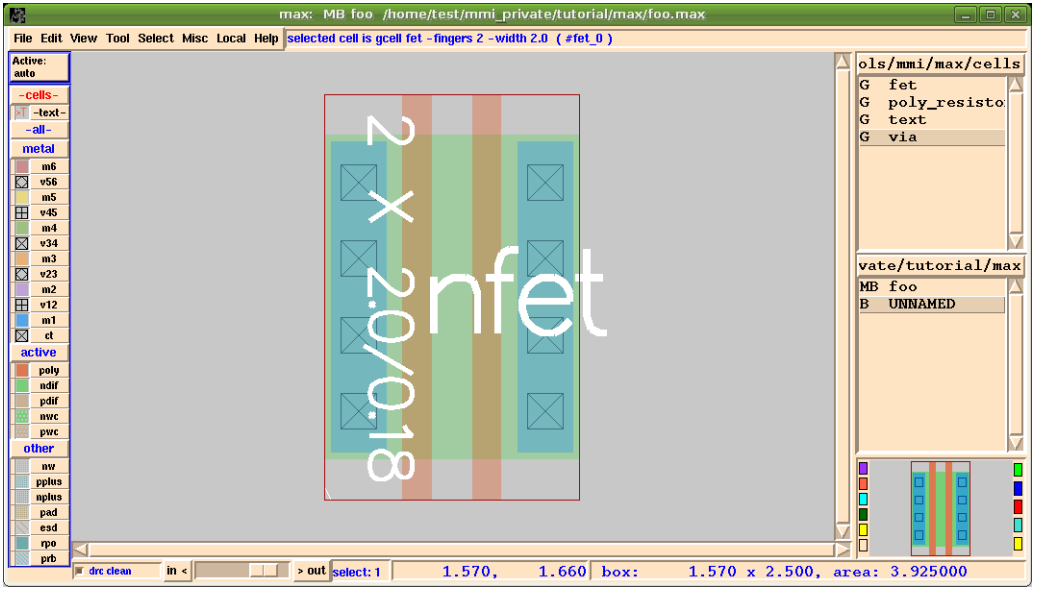
Two nfets merged (stacked) together should appear, with the cursor attached to the lower left corner.

Step 5 ⇨ Click **Button-1** to place the nfets in the window.

Step 6 ⇨ Type **v** (the hotkey for **Zoom To Fit Edit Cell** under the **View** menu) and the layout will center in the window.

Your screen should now look like Figure 68.

Figure 68: Generated nfet Gcell



The white text on the cell informs you that you are editing an nfet, and that it is 2 gates x 2 microns wide, at a .18 micron channel length.

Notice that the letter M appears to the left of foo in the Cell List Box. This tells you that foo has been **Modified**.

Step 7 ⇨ Zoom out by typing **Shift-z** or using **Zoom Out** from the **View** menu; or use the scroll wheel on your mouse (if you have one). You can also use the **Zoom Bar** at the bottom of the window just below the scroll bars.

TIP:

You can undo whatever you ahve just done in MAX. Use the **u** hotkey or select **Undo** from the **Edit** menu. You can also **Redo** what you have just undone using **Shift-u**. There are 99 levels of undo in MAX. If you can remember back more than 99 items please call us, we would love to meet you!

Selecting and Moving Layout

There are two types of layout you can select and move in MAX: **objects** and **rectangles**.

Objects may be *sub-cells* (cells), *generated cells* (gcells) or *polygons*.

Rectangles are basically *flat layout*.

TIP:

Be aware that rectangles may sometimes be refered to as **paint** in the MAX documentation. To create a rectangle, you draw out a box and fill it with "paint" of a certain layer.

There are many methods for selecting objects and rectangles.

- When MAX is in **main mode** (the default mode), if you simply drag out a region while holding down **Button-1**, whatever is within that box will be selected when you release the button, and will be highlighted in white.

  For ***rectangles***, only the portion of the rectangle that is enclosed by the box is selected.

  For ***objects*** (cells, gcells, or polygons) if any portion of the object is enclosed by the box, that object is selected.

SELECTING LAYOUT

Step 1 ⇨ Drag a box over the stacked nfets we just drew. When you release the mouse button whatever is "selected", in this case the fet gcell, will be highlighted.

MANIPULATING LAYOUT

Once you have selected an object or rectangle, you can do lots of things with it.

Step 2 ⇨ For example, you can *rotate clockwise* by typing the **r** hotkey. The **R** (**shift-r**) hotkey will *rotate counter-clockwise*. Try it.

⇨ Type **r** three more times to rotate the object back to its original position.

Step 3 ⇨ You can *move* layout up/down/left/right one grid at a time by using the arrow keys.

Step 4 ⇨ You can also *move* any layout by first selecting it, then clicking and holding down the **right mouse button** (**Button-3**) and dragging it around the screen.

MAX also supports Cut, Copy and Paste in the same format you would see on any good Mac- or PC-based drawing program.

TIP:

All of the zoom, move, cut and paste, rotate, etc. commands specified with hotkeys also have menu equivalents. These can be found in the **Edit**, **View**, and **Select** menus.

Duplicating Gcells

We now wish to create a couple of pfets. We could go back to the List Box and select the generator again, but there is another way.

SELECT AND DUPLICATE GCELL

Step 1 ⇨ Simply select your nfet by either clicking over it with **Button-1**, or by dragging a selecting box around it with **Button-1**.

Step 2 ⇨ Now select **Duplicate** from the **Edit** menu, or type **d**. This creates a duplicate of the selected objects (or rectangles), slightly offset from the original.

Step 3 ⇨ Hold down **Button-3** and move the new nfet above the existing nfet.

⇨ If you hold down the **Shift** key with **Button-3**, the move will align to the existing fet as you move horizontally or vertically.

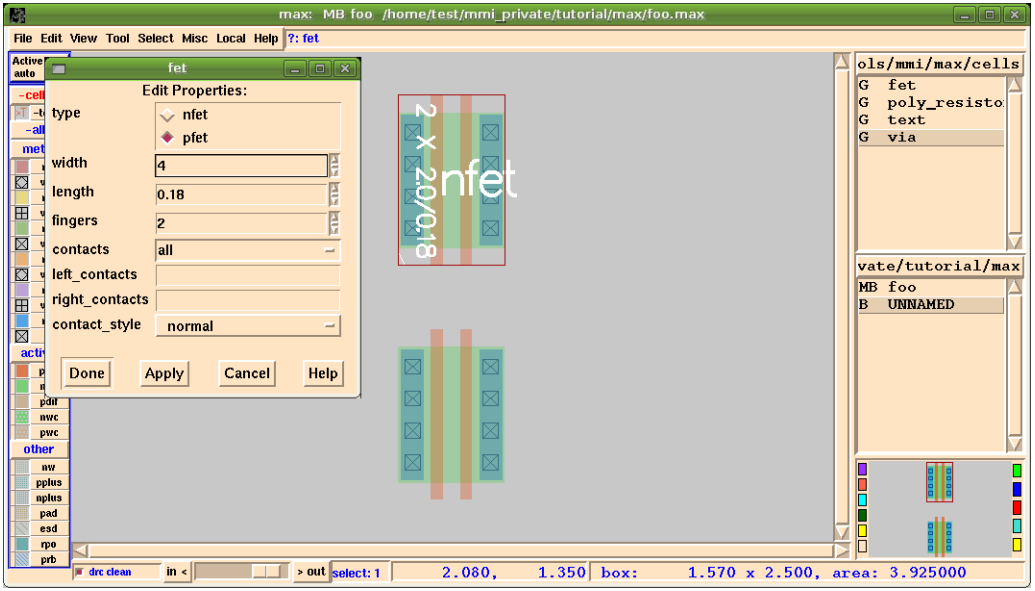
Notice that when you selected the fet and held down **Button-3**, the cursor changed to a **hand**. This change in the cursor tells you that you are in a mode other than ***main mode***. In this case you are in ***move mode***. Remember to look up at the Message Area to see what the functions of the mouse buttons are in your current mode.

EDIT GCELL PROPERTIES

Now you will begin to see the true power of gcells.

Step 4 ⇨ Select the top nfets and then type **e**, or select **Push Into Cell** from the **View** menu. The pop-up menu shown in Figure 69 appears.

Figure 69: Edit Properties Form for fet Gcell



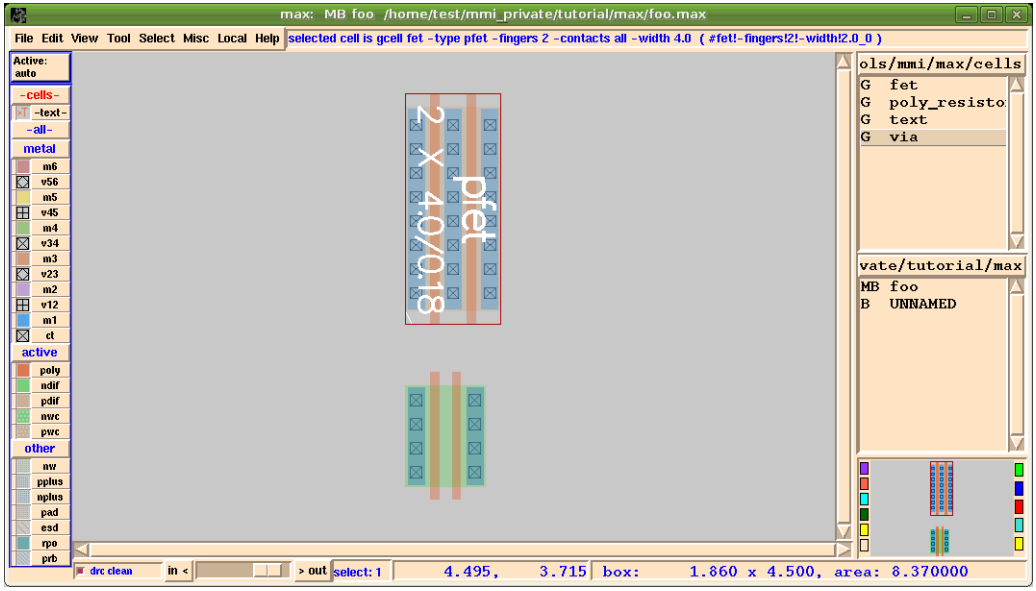
Step 5 ⇨ Click on **pfet** the change the selected nfet to a **pfet**.

⇨ Change the **width** to **4**, to make the pfet 4 µm wide.

⇨ Change contacts to **all** by either clicking several times on the box next to the word contacts, or hold down **Button-1** on the little bar on the right side of the box next to **contacts** and then select "**all**".

Step 6 ⇨ Once you are finished, click **Button-1** on **Done** to invoke the generator. This will change the old nfets to magically become new 4 µm wide pfets with contacts in the middle and on both sides. **Zoom to fit** (hotkey: **v**) and you should now see something similar to Figure 70.

Figure 70: Two fets Generated



Stretching Gcells

As you have seen, gcells can be used to create fets and vias. The wiring tool can also automatically drop gcell vias. You can change the size of a gcell fet by:

- Using the **e** hotkey, and accessing the **Edit Properties** box, or
- Doing a **Stretch Selected** from the **Edit** menu (hotkey: **Ctrl-Button-3**).

STRETCH PFET Try this on your pfet.

Step 1 ⇨ ***Select*** the pfet by clicking on it with **Button-1**

⇨ Move the mouse near the top border of the pfet.

Step 2 ⇨ Hold down **Ctrl-Button-3** (the cursor will change into a double-headed 'stretch arrow') and drag the mouse up. Release the mouse button.

Notice that as you stretch the pfet, additional contacts are added automatically.

⇨ To return to the original width, you can **Undo** the new width with the **u** hotkey or edit it again and change the width back to 4.

You can stretch via gcells in the same way, except the outcome is placing multiple vias.

TIP:

You can stretch a well contact to help you with things like quickly building guard rings.

Aligning Objects

It would be nice if our new fets were lined up. One way to align objects is to draw a ruler (hotkey: **Ctrl-r**), zoom way in, and carefully move the objects until they are aligned with the ruler. But a much easier way is to use the **Align Objects** command.

Step 1 ⇨ If you aligned the two fets when you duplicated the nfet, move one of them with **Button-3**.

SELECT OBJECTS TO ALIGN

First, select all the objects to be aligned.

⇨ One easy way to select multiple objects is to hold down the **shift** key while clicking on them with **Button-1** (**Shift-Button-1**)

⇨ Using **Shift-Button-1** ***adds*** something under the mouse to the selection.

⇨ **Alt-Button-1** ***subtracts*** what's under the mouse from the selection.

You can tell when the gcell is selected because its outline is highlighted, and its name and size are also displayed in white. If there are multiple objects under the cursor, use the **Selection Probe** (demonstrated earlier in this tutorial) to select exaclty what you want.

Step 2 ⇨ Click with **Shift-Button-1** over the two gcells until they are both selected.

Now you will use "**the Box**" to tell MAX how the gcells should be aligned. By selecting the gcells using **Shift-Button-1**, the "Box" has automatically been placed over the last gcell you selected.

- The gcells will be moved so they align with the Box: in this case, it means the first gcell you selected will be moved so it aligns with the second gcell selected.
- You could instead draw the Box somewhere else (**box mode**, hotkey: **b**) and have all of the objects aligned to the Box (as shown in Figure 71.)

ALIGN THE OBJECTS

Finally, use the **Align Objects** command.

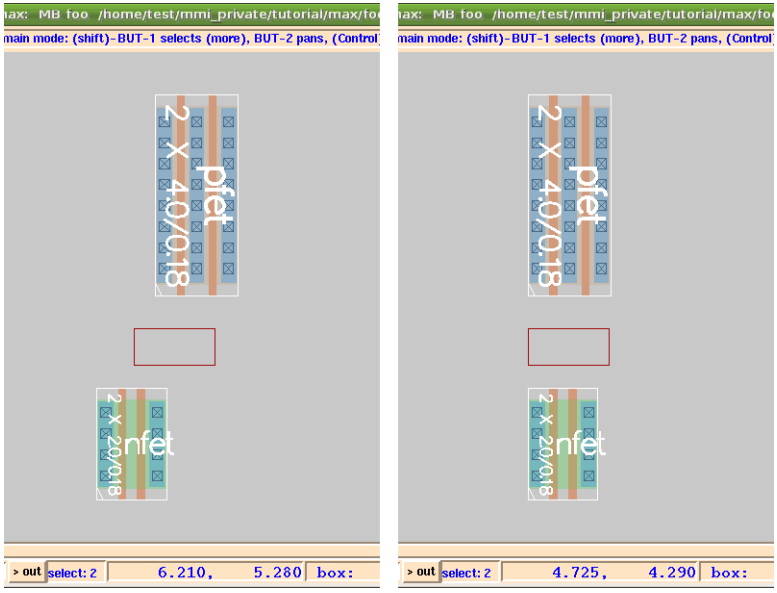
Step 3 ⇨ Choose **Align Objects** from the **Edit** menu (hotkey: **Ctrl-a**).

⇨ Make sure that **left sides** (the default) is selected (red dot showing).

Step 4 ⇨ Click **Done**. The gcells are now aligned vertically on their left sides.

Figure 71 shows both fets being aligned to the Box, which in this case was drawn separately (hotkey: **b**).

Figure 71: Aligning fets Using The Box: a) Drawing the Box; b) Aligning "Left"



Continuous DRC

We are now going to see a really powerful feature in MAX, real-time DRC.

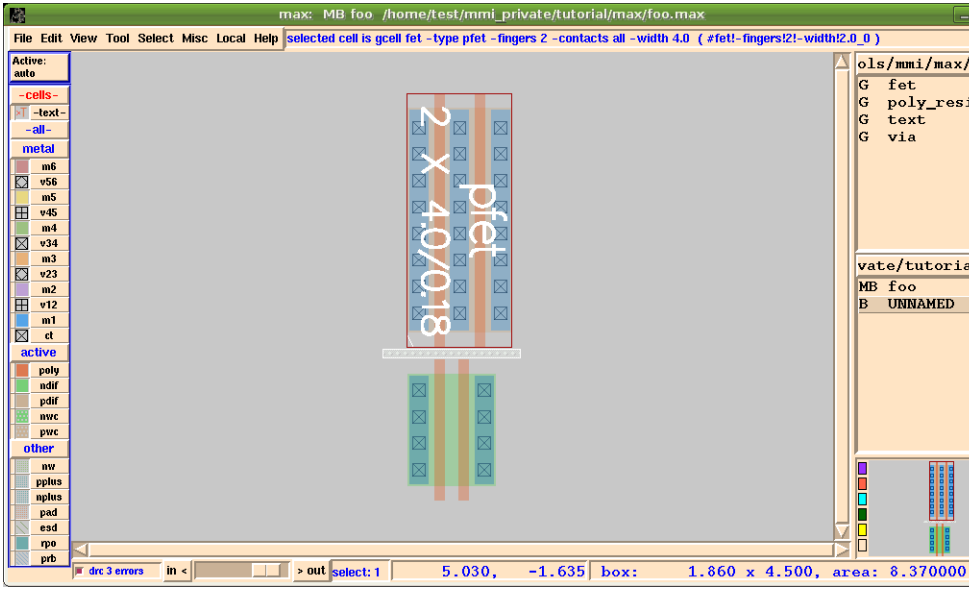
Step 1 ⇨ With the new pfets on top of the nfets, select the pfets.

⇨ You can move the fet using the **arrow keys** (best for precise placement) or by holding down the **Shift** key and **Button-3**.

⇨ Using the **Shift** along with **Button-3** *locks the move* in the vertical or horizontal direction.

Step 2 ⇨ Use the arrow keys to move the pfet just above the nfet, as shown in Figure 72.

Figure 72: Real-time DRC



⇨ If you move the pfet too close to the nfet, little white dots appear. These little white dots tell you there is a ***Design Rule Check error*** (DRC error.)

⇨ Notice that as you move the pfet closer or further away the DRC errors show up in “real” time! This is one of the power features in MAX.

The Continuous DRC means you don't even have to learn the DRC rules, or count grids. Just move things closer together until you see the white dots, and then back off a little. The small **DRC error box**, to the left of the zoom bar at the bottom of the MAX window, displays the number of errors.

VIEWING DRC ON LAYOUT

If you are curious about what the DRC error is, you may view the results on the layout.

Step 3 ⇨ Drag a box around the white dots and type **Shift-y** or choose **Explain DRC Under Box** from the **Misc** menu.

> **TIP:**
>
> Remember that if you drag out a box using just **Button-1**, the box is drawn *and* everything underneath it is selected.
>
> To draw a box only, type the **Make/move box** hotkey **b** and then drag out with box with **Button-1**.

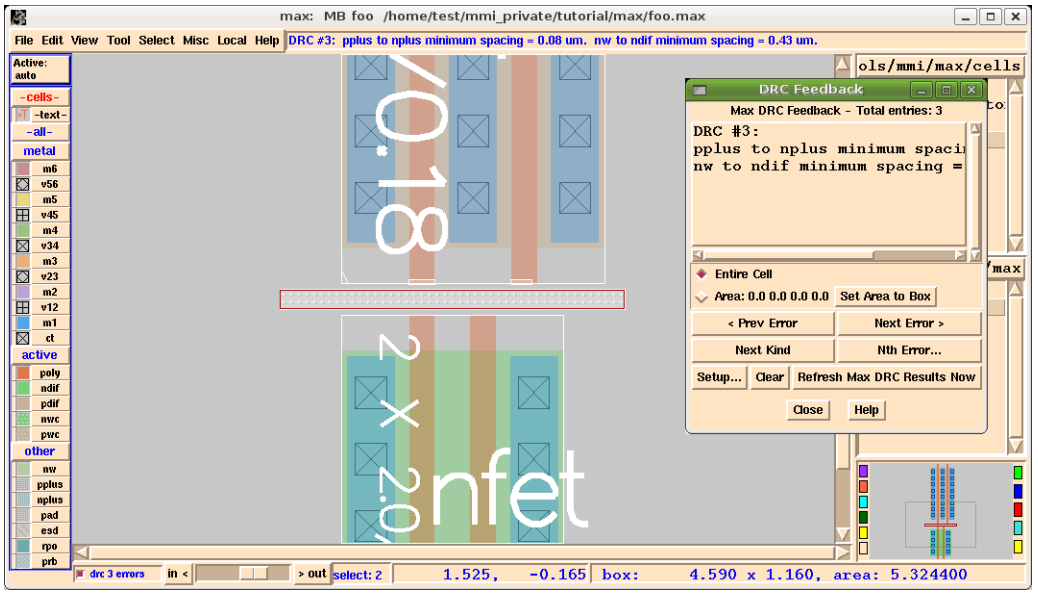
- If the box is over only one DRC error, that DRC error will be displayed in the **Message Area**.
- If more than one DRC error lies under the box, the number of DRC errors is listed in the **Message Area**, and the list of all DRC errors under the box is displayed in the MAX command window (the shell window from which you started MAX).

VIEWING DRC RESULTS ON FORM

Another method of viewing DRC errors is to use the **DRC Results** form.

Step 4 ⇨ Move the pfet close to the nfet, as shown in Figure 73, so that DRC errors occur.

Figure 73: DRC Results Pop-up Window



Step 5

⇨ Click on **Next Error** to step through all of the DRC errors. Notice that each DRC error is highlighted in the layout, and the text of that DRC error is displayed in the **DRC Feedback** window as well as the **MAX Message Area**.

⇨ When you are finished looking at the DRC errors, click on **Close** in the **DRC Feedback** pop-up.

Refer to the MAX User Guide for more details on the DRC Results pop-up window. This window can also be used to view results of DRC runs with other tools, such as Mentor Graphics' Calibre.

Step 6

⇨ Move the pfet above the left nfet so that the gates of the pfet and left nfet line up (this should already be the case after you aligned them earlier).

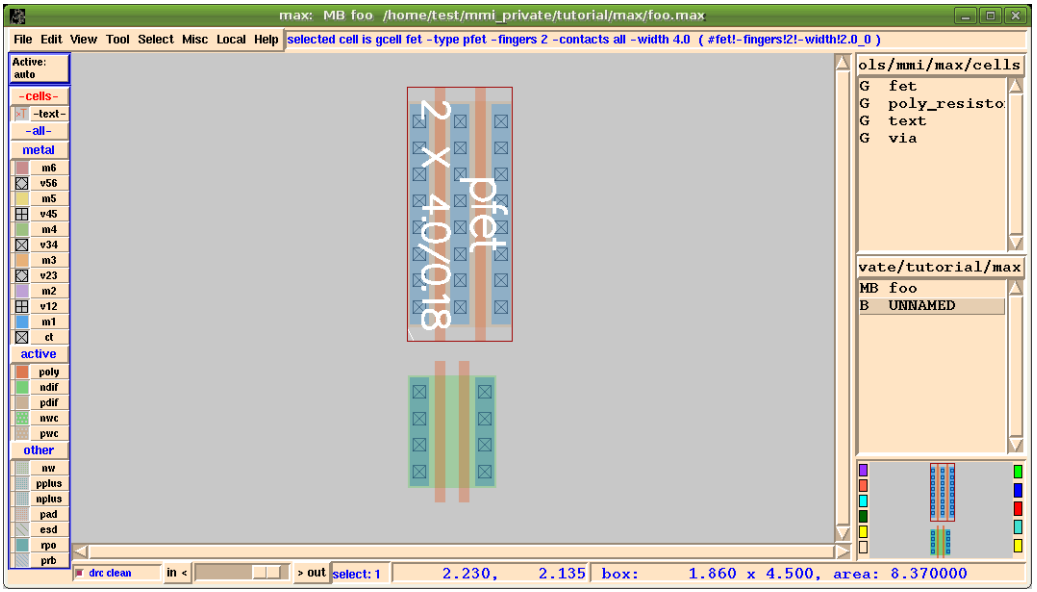
⇨ Then move the pfet up (**arrow keys**) until it is DRC correct as shown in Figure 74.

⇨ to get it DRC correct, move up until there are no white dots showing, and the DRC status box on the lower left of the MAX window says `drc clean`. Sometimes there can be DRC errors even if you can't see any white dots, expecially if you are zoomed out.

TIP:

If the DRC status box says there are still DRC errors, but you cannot see any white dots, you can use the **DRC Find Next Error** command in the **Misc** menu or type the hotkey **Shift-n**. MAX will soom to the DRC error and describe the error in the MAX Message Area.

Figure 74: FETs Lined Up



The Continuous DRC feature used design rules defined in the `MAX Technology file`. This single plain text file defines all the design rules needed by MAX's extensive DRC. The technology file uses a simple table format for most rules, resulting in a compact, easy to understand file format that can be as short as a single page.

Micro Magic, Inc. provides common CMOS technology files as a starting point to build your own. In addition, you can build a prototype technology file automatically from an existing GDS file by choosing **Import File** from the **File** menu.

IMPORT PDKs Technology information from a PDK, generally provided by the foundry, can be converted to MAX format using **cds_convert**. This will be covered further in Part 5 of this tutorial.

IMPORT OPENACCESS Technology information from a PDF for the process may also be imported using an OpenAccess database. We will address OpenAccess further in Part 5.

Wiring Mode

Now you will route up the poly. You could route the poly by drawing rectangles and then filling them with poly (red) paint, but that's a lot of work. There's a much easier way — MAX's built-in wiring tool.

WIRE TOOL

Step 1 ⇨ To use the Wiring Tool, type **w** or select **add wire** from the **Edit** menu.

Notice the cursor has changed. Remember, that's to let you know you are in a different mode. In this case, you are in ***wire mode***.

Step 2 ⇨ Place the cursor over the poly gate of the right hand nfet (fet on the bottom) and click **Button-1**.

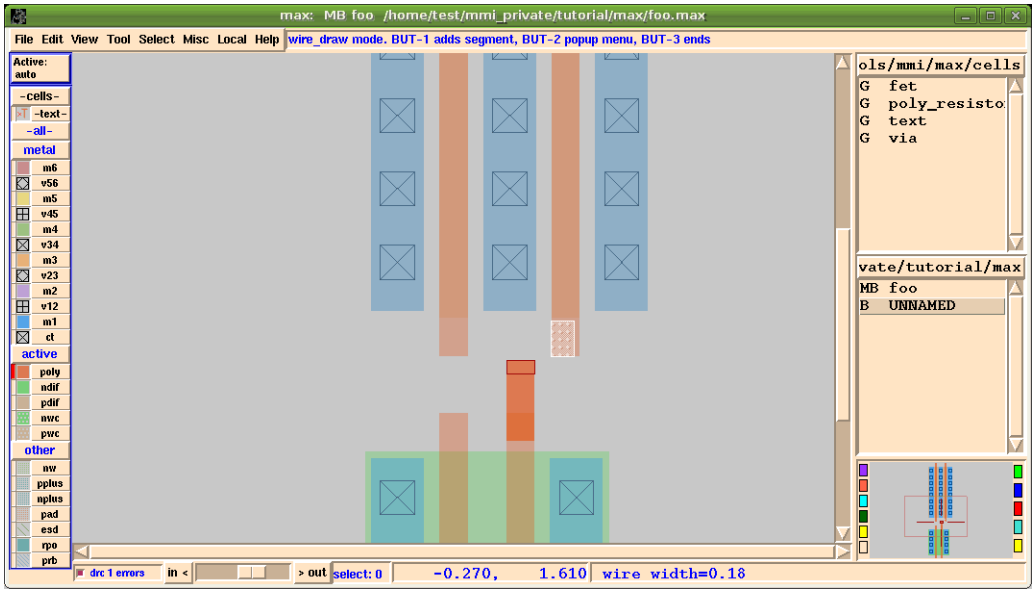
⇨ Now move the cursor up. A poly wire should follow the cursor, as shown in Figure 75.

If the wire doesn't start in poly, check to make sure the **Active** layer is set to `auto` (in the upper left corner of the MAX window).

TIP:

The wiring starts the wire in the active layer (very top of the palette). IF the **Active** layer is set to `auto`, the wire starts in whichever layer is under the cursor. If there are multiple layers under the cursor, it picks the highest layer. If you set **Active** to a specific layer (i.e. `m1`) then the wire will start in that layer no matter what is under the cursor.

Figure 75: Drawing a Wire



- If at any time when in wire mode you wish to bail out and start over, you can do so by typing **Ctrl-c**.

Adding Corners in Wires

As the pfet is not directly above the nfet we need a “kink” or corner in the wire. You do this by simply clicking with **Button-1** at the place you want the kink.

Step 3 ⇨ Move the cursor up and then click with **Button-1** to add a kink/corner. Notice that as you move the mouse, you also move the corner up and

down. When you click **Button-1** again, the horizontal portion of the poly wire will lock in place.

⇨ Now move the cursor to the right until it's directly under the pfet poly and click **Button-1** again to add another kink/corner.

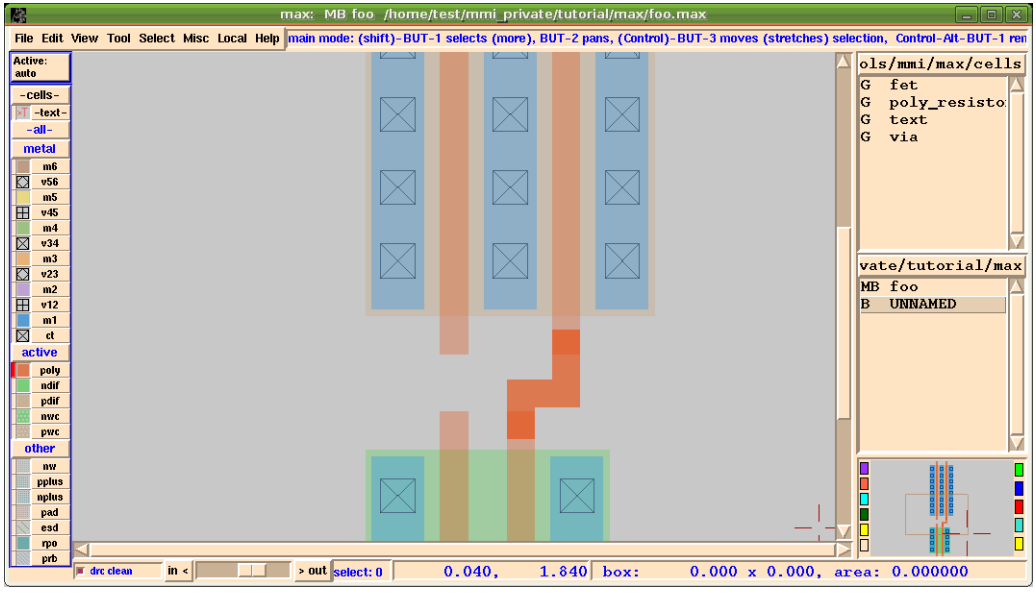
⇨ Finally, move the mouse up until it's over the right pfet poly gate. If you glance up to the **MAX Info Bar**, notice that it says "`BUT-1 adds new segment BUT-3 ends`". This means that when you click **Button-3** (**right mouse button**) it ends the wire.

⇨ In fact, MAX does even better than that. If you click **Button-3** over the poly but are not exactly in the middle of the wire, MAX will actually snap the wire into place for you and end it. How about that?

If you didn't put the wire exactly where you wanted it don't worry. You can always **undo**.

Your layout should now look like Figure 76.

Figure 76: Drawing Poly Wire



TIP:

If you want to back up one step in the wire, use the **Undo** hotkey **u**. You can undo (**u**) back to the beginning of the wire.

If you cant to **cancel** the current wire, use **Ctrl-c**. **Ctrl-c** always aborts any sub-mode and returns you to main mode.

You might have noticed that the little piece of poly you just added is a brighter color than the poly that was part of the gcell fets. This is a feature in MAX designed to help you know what you are editing. Rectangles or objects drawn in the cell you are currently editing show up brighter than the cells you are not editing. This includes gcells. Therefore, since you are not editing the gcell fets, they show dimmer than the cell you are editing.

⇨ If you wish to have all of thelayers show up with the same brightness (even in the gcells or subcells) then go to **File > User Preferences**, select **Display Options...**, then select **Dim Non-Edit Cells** and click **Done**. The radio button should go "off" with the non-edit cells no longer dimmed.

Editing a Placed Wire

Once a wire has been placed, you can edit that wire. The Edit menu contains commands to Move, Stretch, and Fill any rectangles, including wires. There is also the Edit Wrie command as we saw earlier in this tutorial.

You many find that it is sometimes even faster to simply redo a wire if you need to make a number of edits.

⇨ If you just want to move the wire a little you can select the wire segment you wish to move and then nudge it using the arrow keys. The arrow keys will do a "stretchy" move.

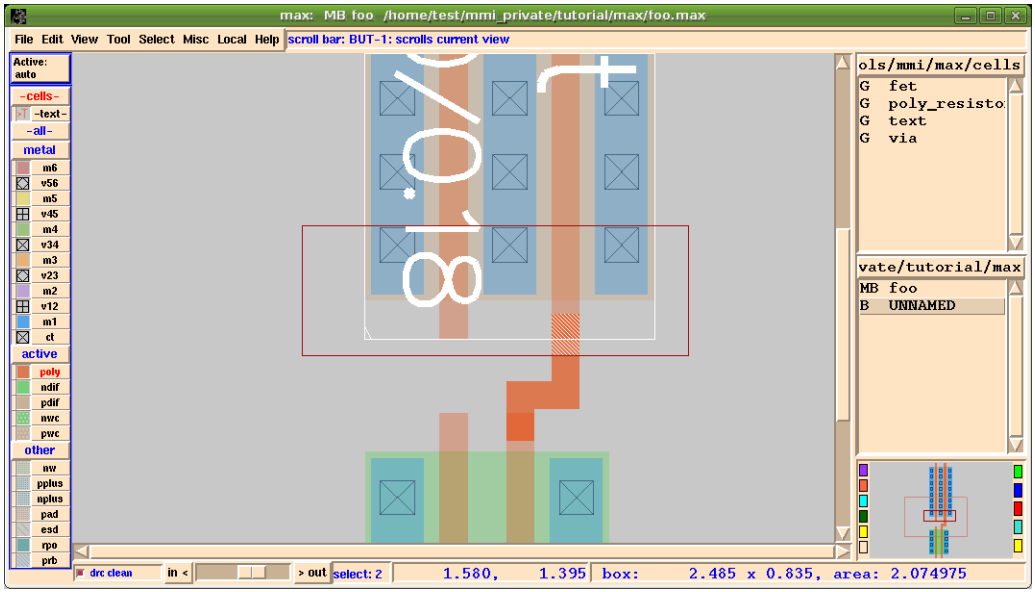
- If your segment has a via at either end you need to use the **Edit Wire** command.

STRETCHING WIRES

Let's try a stretchy move with our connected fets.

Step 1 ⇨ Draw a box by holding down and dragging Button-1 so that it encloses port of all of the pfet, and part of the vertical section fo the poly wire as shown in Figure 77.

Figure 77: Stretch Move



Step 2 ⇨ Press the **up arrow** key a few times and notice that the poly wire stays connected.

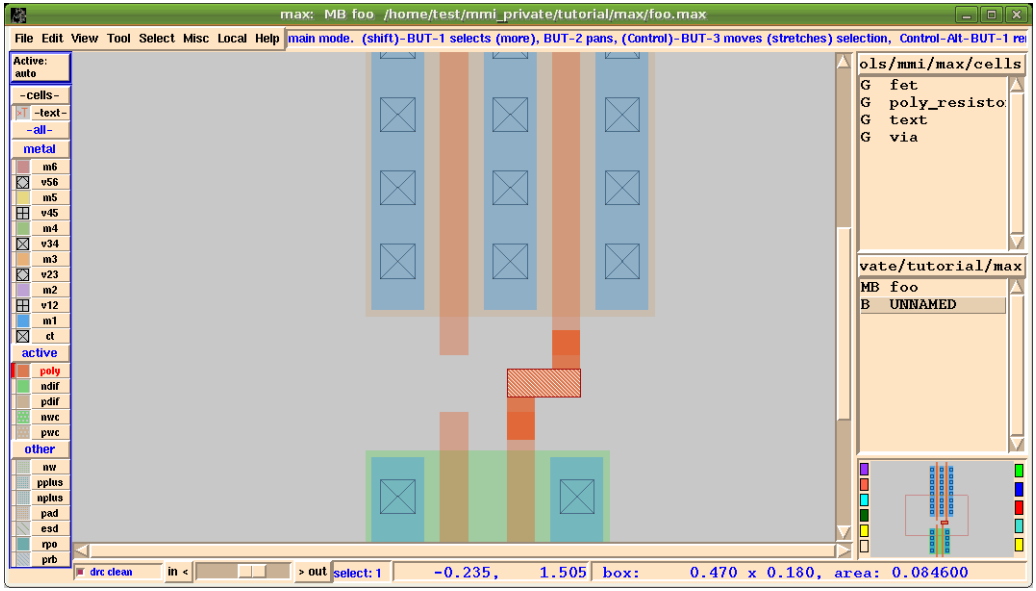
Step 3 ⇨ **Undo** your stretchy move with the **u** hotkey.

MOVING A WIRE

Now let's move the middle segment of the poly wire.

Step 4 ⇨ Select the horizontal section of the poly wire by clicking on it with **Button-1**, as shown in Figure 78. You may need to click multiple times to select the entire horizontal section of the wire.

Figure 78: Moving Wire Segment



⇨ Use the **up and down arrow keys** to move the segment. Notice that the wire stays connected.

TIP:

If you want to move (and keep the net connected) the segment a long distance, you can use **Ctrl-Button-3.** Using **Button-3** without **Ctrl** will move the segment but will not maintain connectivity.

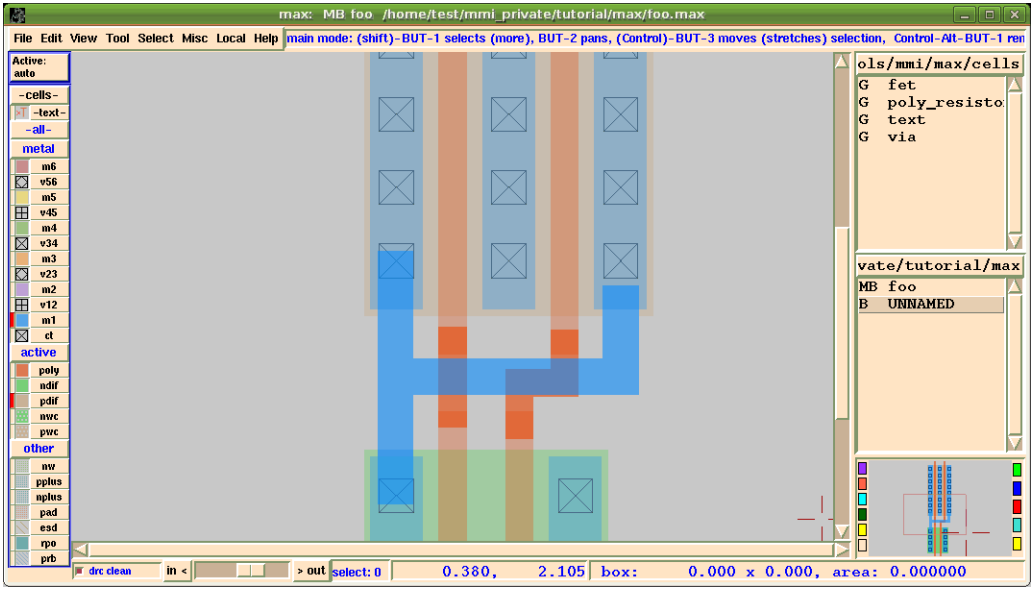
If you are using the arrow keys for the move and do NOT want the wires to stretch, hold down the **Shift** key when moving the selected layout.

⇨ Finish wiring up the fets gates by drawing the second poly wire as shown in Figure 79 using the **wire** (hotkey: **w**) command.

Now wire up the metal just as you did the poly.

⇨ Enter **wire mode** (hotkey: **w**) and then click on one of the contacts and draw the metal so it looks like Figure 79. First wire the two left contact regions and then add the connection to the right pfet contacts.

Figure 79: Wired 2 Left Poly Gates



When starting a new wire, the **Active Layer** box (remember it's on the top left just under the **File** menu) tells you in which layer the wire (or circle or polygon) is going to be drawn. If it is set to auto it picks the layer under the mouse.

If you want to tell it exactly which layer to start drawing on, then hold down **Button-1** over the **Active Layer** box. When the pop-up appears simply place the mouse over the layer you want and then release the button. All wires will now start in this layer.

TIP:

To ***change the layer of the wire*** while you are in **wire mode** (before you start drawing the wire or before you drop a via), type the hotkey: **l** ("ell" - lower case L).

If you type the **hotkey m** while in **wire mode** (**w**), the **Wire** menu will appear. The **Wire** menu changes based on whether or not you have already started the wire.

Also, remember that if you type the **Space** hotkey, the list of hotkeys active in the current mode will appear.

SAVE EDITS Since you have done a fair amount of work you should now save your layout.

Step 5 ⇨ Type **Ctrl-s** or select **Save** from the **File** menu.

Notice that the M next to the foo in the **Cell List** went away. There is now a W next to foo, meaning that this cell has been ***written***.

Adding the Power Rails

We are ready to put in the power rails. Let's say we were layout out this NAND gate for a standard cell library. Furthermore, assume that the power and ground rails are run in metal 1 (m1) and that they are 2µm wide.

Before adding the power rails, you need some more information about the wire mode, the box and the palette.

We could use the wire tool to draw the power rails, by drawing wires with width set to 2µm.

⇨ To do this, you would start drawing a normal wire (hotkey: **w**) in layer M1, positioned above or below the fets.

⇨ Then, press and hold **Button-2** to access a pop-up menu for wiring options including the current wire width. Here you can set the wire width of the current wire segment(s) to 2µm, or the width desired.

⇨ You can also set the default width, spacing and snap grid for each wire layer in the **Wiring Parameters** menu. It is found under **File > User Preferences**, then choose **Wire Setup...** (or just use the **Shift-w** hotkey) then choose **Wiring Parameters**.

However, this is a good opportunity to use rectangles directly. MAX has excellent capabilities for low-level layout -"rectangle hacking", and this is an ideal way to put in the power rails. To use rectangles effectively in MAX we will first have to learn to manipulate *the Box* and *the Palette*.

The Box

The Box is important in MAX because it is used by many commands to determine where you are going to place, or do, something. For *painting* (creating rectangles of a specific layer) the Box marks the region where paint is going to be drawn or erased. We have already seen the Box on the screen while doing the commands above; for example, the **Align Objects** command.

⇨ You can create a Box by simply holding down **Button-1** and dragging the outline around a region.

TIP:

Drawing a Box by holding down **Button-1** also automatically selects what is under the Box, so don't drag a box around an entire chip! Use **Box Mode** (below) for that.

Assuming you were in **Main Mode** (the default), any paint within the Box is selected. This is the way we have done things thus far.

⇨ You can also position the Box exactly by typing **Shift-b** or selecting **Box Dimensions** from the **Misc** menu, or by clicking on the **Box Size** window in the lower right corner of the display. Any of these will open a menu that allows you to precisely set the Box size.

Box Mode

Box Mode is useful when you need to do detailed layout. You can resize the box to the exact size you want, and place it exactly where you want it, without having to worry about corrupting any existing layout that you might have inadvertently selected.

TIP:

Dragging out a box in **Box Mode** does NOT select anything!

After typing **b** (or **Make Box** in the **Misc** menu) you can do any of the following:

- ***You can specify a new box position*** by holding down **Button-1** as you drag out a region, then releasing **Button-1**. The cursor shows as a hand with a pointing finger.
- ***Resize the existing box*** by moving the mouse over the sides or corners of the existing box until the cursor icon changes (to either a 'stretch-arrow' or a 'corner'), then drag that side or corner using **Button-1**.
- ***Move the box*** by holding down **Button-3** and moving the mouse (a hand cursor)

Remember, the above commands function while you are in **Box Mode.** You can always look at the **MAX Message Area** (to the right of the menus) to determine your current working mode.

The Palette

A few more words about the Smart Palette (along the left side of the MAX window).

The Smart Palette provides many features. It allows you to control what you can see, what you can select, what's under the cursor, what can be drawn as rectangles, and what the layers look like.

- Remember, the **MAX Message Area** will tell you what the available options are if you place the mouse over the palette.

  Here's a reminder of how to use the palette:

  - To control ***which layers are visible***, click **Button-1** over the little colored square (to the left of the layer labels `-m5` ... `v45` ... `m4`, etc.)
  - To control v***isibility by group***, click **Button-1** on the **group** button: `metal`, `active` or `other`. This is a very useful feature for reducing clutter when you are working on a subset of layers.
  - To control ***which layers are selectable***, click **Button-1** on the **layer name** (such as `m2`, `v12`, `pfet`, and so on). The **layer name** button will turn grey, and that layer will be *disabled* (NOT selectable).

    Or, click **Button-3** over the **group** button to toggle selectability for that *entire group*.
  - Layers which are ***under the mouse*** are indicated by a small rectangular red LED just to the left of the colored boxes.
  - Layers which are ***currently selected*** are indicated by the layer name(s) changing from black to red.
  - Finally, you can ***change the color or stipple patterns*** of any of the layers by clicking **Button-2** (**middle mouse button**) over either the colored box or the layername. This brigns up the **MAX Color/Stipple Editor**. Using the editor, you can even change the colors of things like the background (for example, make it black), the grid, labels, etc.

    While the grid color is changed in the **MAX Color/Stipple Editor**, the ***grid size and style*** are changed in **Grid Setup...** from the **File>User Preferences** menu, or by typing **Shift-g** hotkey.

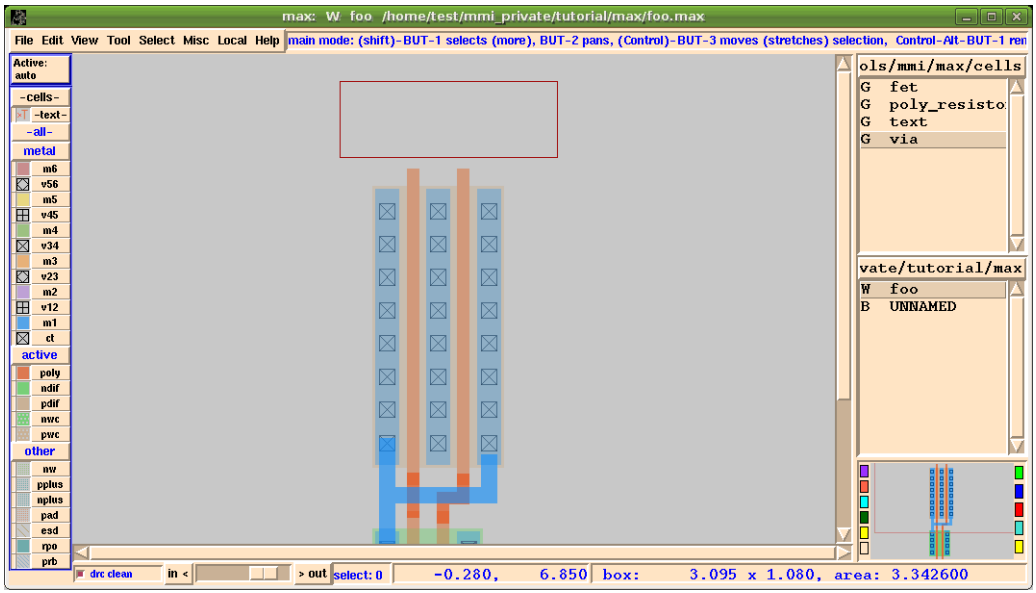
Painting the Power Rails

Now we are ready to add the power rails as paint rectangles.

Step 1 ⇨ Type the **Add Box** hotkey, **b**.

Step 2 ⇨ Hold down **Button-1** and drag out a region where you want to paint the new `Vdd` power rail, as shown in FIG.

Figure 80: Drawing a Box for Painting Power Rails



Step 3 ⇨ Place the cursor over the **M1** button in the palette (either over the name `m1` or the bright blue square to the left of the name), then click the **paint** hotkey **p**.

> **TIP:**
>
> You can also type the **p** hotkey over any piece of m1 in the layout. Be sure there are no other layers under or over m1. MAX will fill the box with all of thelayers under the cursor.
>
> **If you click the p hotkey over a place with NO layout, then the layout of all layers in the area under the box will be erased.**

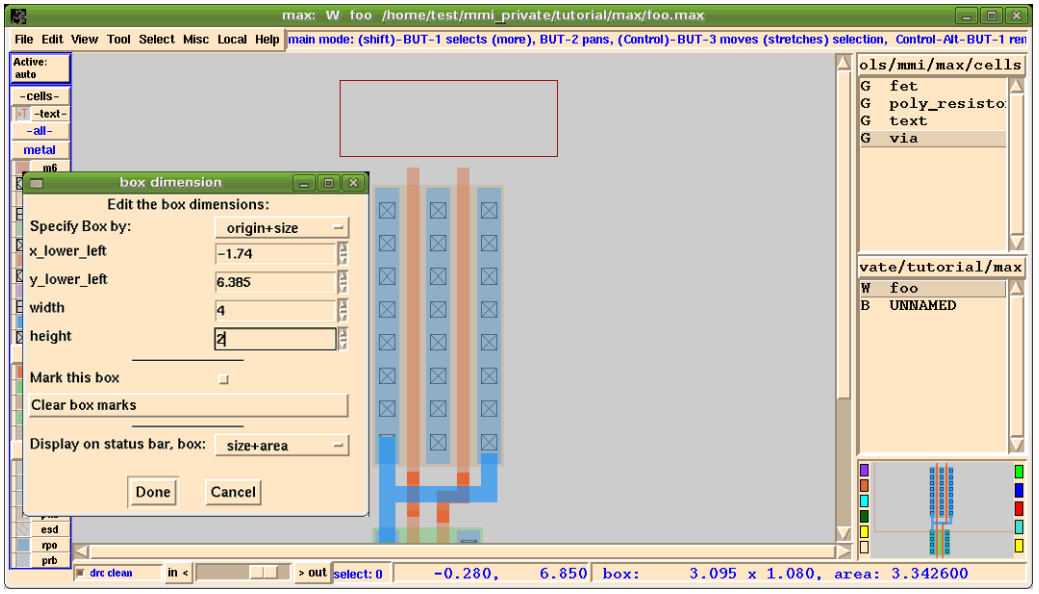
RESIZING RECTANGLE

If the box isn't the size you want (see the lower right corner of the window for the box size) you can make the box exactly 2µm tall or you can edit the rectangle. Notice in the figure above that the size of the box is 3.65 x 1.355.

Step 4 ⇨ **Undo** (**u**) the m1 paint before we resize the box.

Step 5 ⇨ To make the box an exact size type **Shift-b** or select **Box Dimensions** from the **Misc** menu. A pop-up menu appears (see Figure 81).

Figure 81: Box Dimensions Pop-up Form



⇨ Make sure the **Specify Box By:** is set to **origin + size**. Then type in `4.0` for the width and `2.0` for the height. Click **Done**.

Step 6 ⇨ Fill the box with `m1` again by clicking **p** (paint) on '`m1`' in the palette or on any `m1` in the layout.

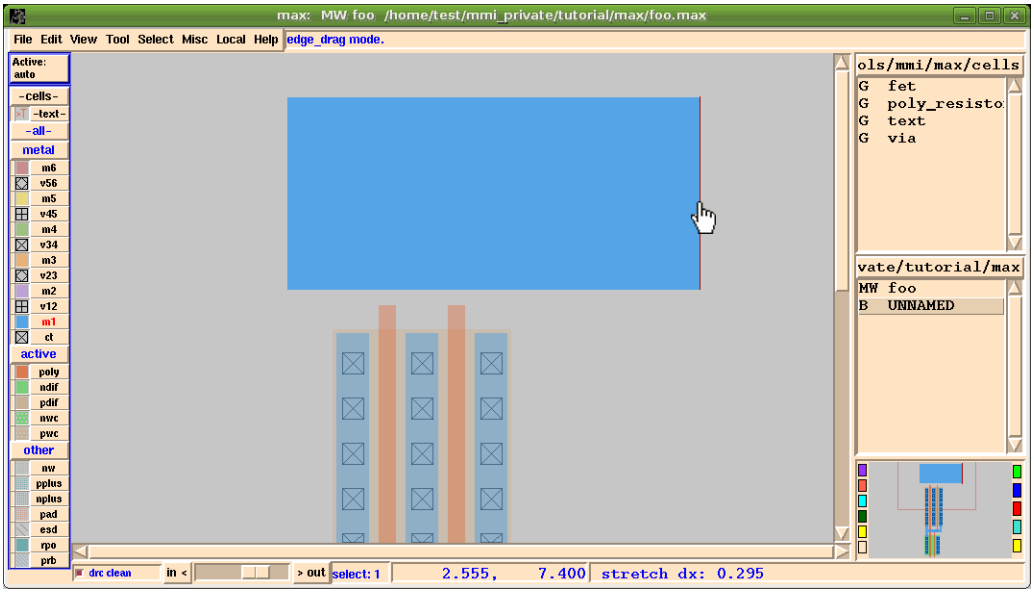
STRETCHING RECTANGLE

You can change the size of a rectangle by *stretching an edge* using the **Edit Edge** command (hotkey: **a**) found in the **Edit** menu.

Step 7 ⇨ Type the **a** hotkey. The cursor changes to let you know you are in **Edit Edge** mode.

⇨ Move the cursor (pointing finger) over the right edge/side of the m1 rectangle that we just drew. A select line will show you which edge you are editing, as shown in Figure 82.

Figure 82: Editing Edge of Power Rail Rectangle



Step 8 ⇨ Hold down Button-1 and move the mouse to the right to stretch the edge. Release the button when the edge is where you want it.

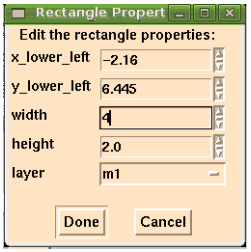
EDIT PROPERTIES OF RECTANGLE

You can also change the size of a paint rectangle by first selecting it, andthen selecting **Edit Properties** from the **Edit** menu (hotkey: **Ctrl-p**).

Step 9 ⇨ Let's change the width of the rectanble back to 4 by editing it's properties. Select the m1 rectangle by clicking on it with **Button-1** and type **Ctrl-p**.

You should now see a pop-up like Figure 83.

Figure 83: Edit Rectangle Properties pop-up



⇨ Change the *width* back to 4 and click **Done**.

PAINT EDIT MODE

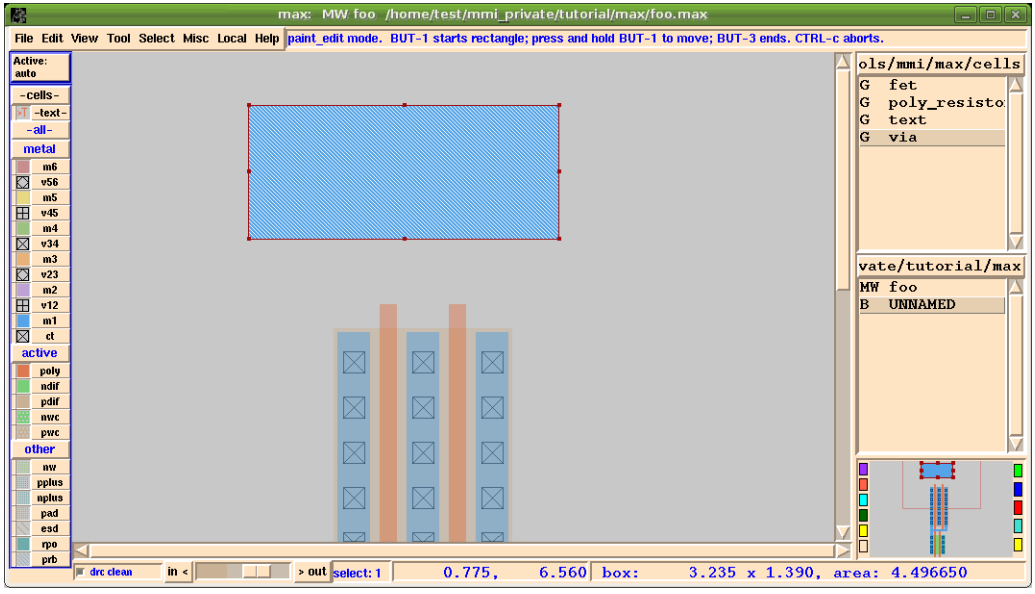
One other way you can edit paint rectangles is to use the **Paint Edit Mode**.

Step 10

⇨ Select the power rail again, and type the **e** hotkey. (**View** menu> **Push into Cell.** MAX is smart enough to know that because a paint rectangle has been selected it should go into **Paint Edit Mode**.)

You will now see the paint rectangle outlined with dots at the 4 corners and in the middle of each edge, as shown here in Figure 84.

Figure 84: Paint Edit Mode for m1 Rectangle



As always, the **MAX Message Area** tells you what mode you are in and the function of each mouse button while in that mode.

- Holding down **Button-1** in the middle of the rectangle ***moves*** the rectangle.
- Clicking-and-dragging with **Button-1** on one of the dots ***resizes*** the rectangle.
- **Button-3** ***exits out*** of **Paint Edit Mode**.

⇨ Try resizing and moving the rectangle. Click **Button-3** to exit **Paint Edit Mode**. Type the **undo** command (hotkey: **u**) to move the power rail back to it's original position and correct size.

DUPLICATING RECTANGLE

Once you have painted the top power rail you can simply duplicate it and move it down for the bottom rail.

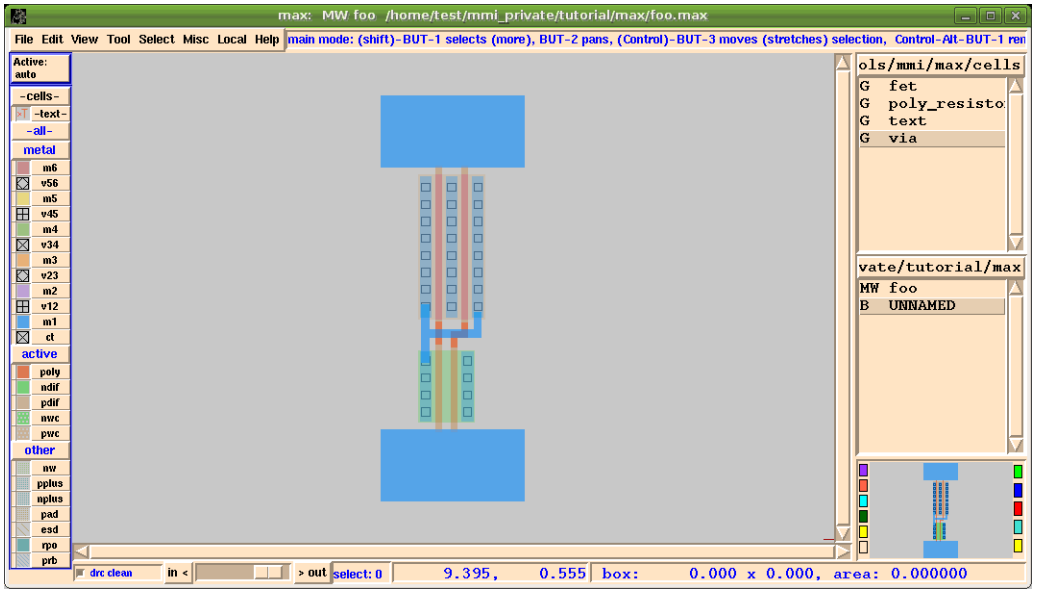
Step 11 ⇨ Select the power rail with **Button-1**.

Step 12 ⇨ Duplicate the top power rail (hotkey: **d**) and move it down using **Shift-Button-3**, as shown in Figure 85.

TIP:

If you ***duplicate*** something and hold the **Shift** key down while moving it, the duplicated item will stay lined up with the original.

Figure 85: Power Rails Placed



Step 13 ⇨ You can move both of the power rails as close as possible to the fets, as shown in Figure 85 (above) by selecting each power rail and using the up and down arrow keys while watching the **DRC Status Box** to make sure there are no DRC errors.

WIRING UP RECTANGLES

Now that you have painted in your power and ground rails you want to wire them up using the **Wiring Mode** as before.

Step 14 ⇨ After you have entered the **Wire Mode** (**w**), hold downthe **Shift** key and click **Button-1** (**Shift-Button-1**) over the middle of the pfet contact region. This will start an m1 wire using the width of the m1 underneath the cursor.

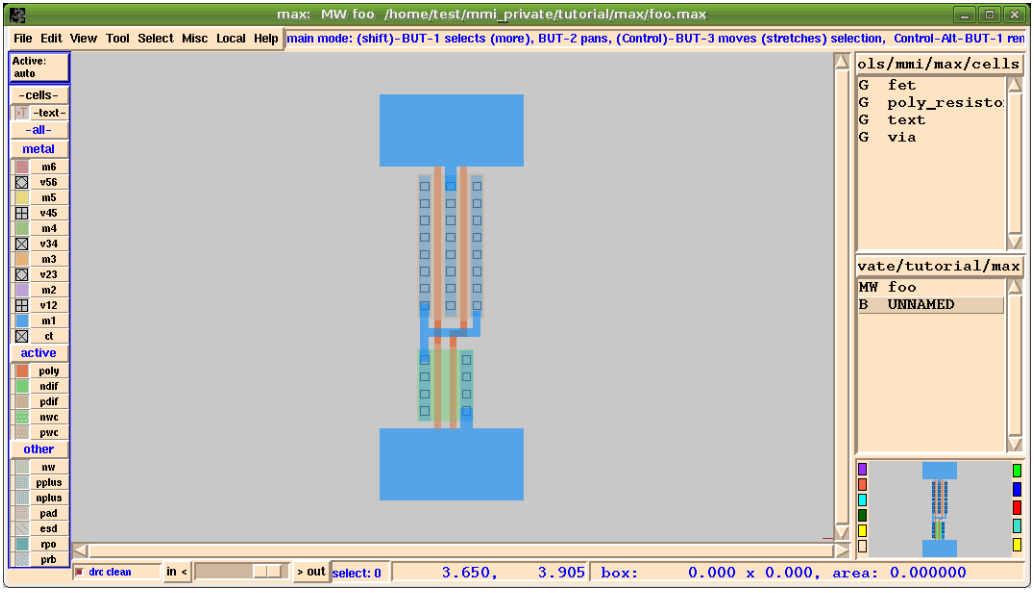
⇨ Click with **Button-3** on the m1 power rail to terminate.

By holding down the **Shift** key you are telling the wire tool to use the width of the wire under the cursor and NOT the minimum wire width.

Step 15 ⇨ Now connect the right nfet contact region to the ground rail.

Your cell should now look like Figure 86.

Figure 86: NAND Wired to Power Rail



Wiring Mode - Changing Layers

Notice that the inputs and outputs are all found within the power straps. What if they needed to be brought out so a router can get to them?

Let's assume the router requires all signals to be at the top of the cell and in m1. You need to route the poly wires over the m1 power straps and then change to m1.

Wire Mode in MAX can make this job easier since it can change layers for you.

CHANGING WIRE LAYER

Step 1 ⇨ To route the wire, type **w** (for **Wire Mode**) and click **Button-1** over the upper end of one of the poly gates.

Step 2 ⇨ Drag the mouse up and over them1 power rail, drawing the poly wire.

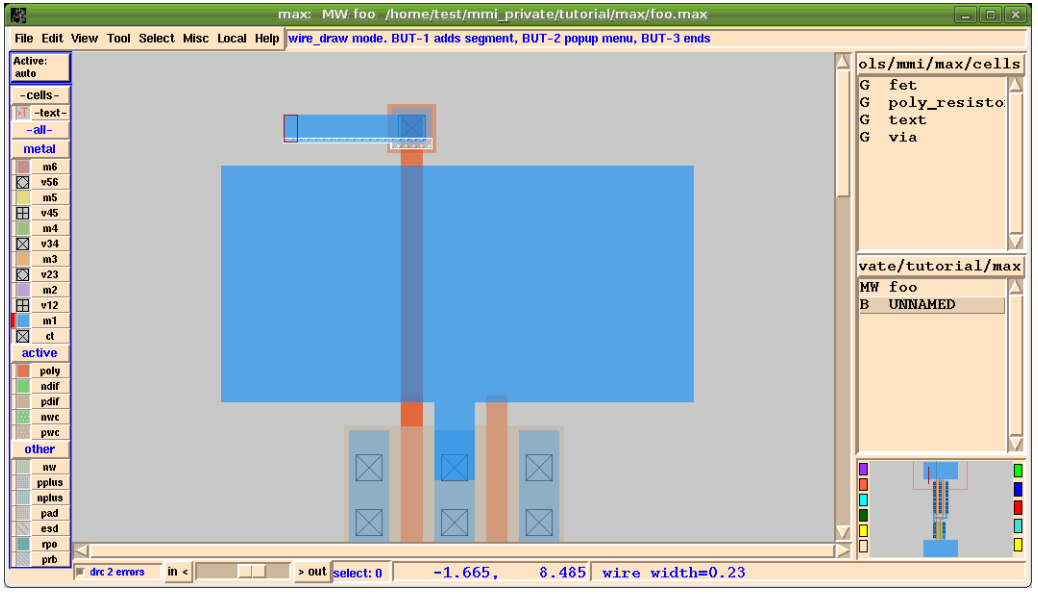
DROPPING A CONTACT/VIA

Step 3 ⇨ Type **d**. This will ***drop a poly to m1 contact*** for you, which you can then move around.

⇨ Now try moving the mouse to the left. This draws an m1 line left of the contact. While holding themouse to the left try moving it up and down.

Notice that the contact tracks the verticla location of the mouse as you move it. Also, notice that the DRC is working while you are moving the contact. Whenever you get too close to the Vdd line, the little white DRC dots appear, as you can see in Figure 87.

Figure 87: Active DRC Showing Errors As Wire Is Drawn



Step 4 ⇨ Move the mouse around to get the contact where you want it (with no DRC errors showing) and type hotkey **a** to 'freeze' or ***anchor the contact*** in place. Remember, you may need to refer to the **DRC Status Box** to make sure there are no DRC errors.

Then drag the mouse upward.

TIP:

To access the commands in **Wire Mode**, press and hold **Button-2**. Next to each command is the hotkey for that command. Another way to find out the hotkeys while in **Wire Mode** is to press **Space**.

So, to freeze the via in place, you could also have selected **anchor vertex** from the **Wire Mode** pop-up.

As you continue dragging the mouse upward, notice that you are now in m1. If you type **d** (***drop a contact***) again, you will add a `via1` and automatically change to layer `m2`. This will continue until your technology runs out of metal layers.

The generic technology file we have included (`mmi18`) has six layers of metal.

TIP:

You can zoom in on an area while in wire mode. Click hotkey **z** and ***zoom in*** on the desired area using **Button-1**. You are then returned to wire mode. You can also ***zoom out*** (**Shift-z**) or ***zoom to fit*** (**v**) while in wire mode.

Step 5 ⇨ ***End the wire*** by clicking with **Button-3** at the desired location, shown in Fig.

TIP:

Typing **d** in **Wire Mode** ***drops a contact*** and ***raises you to the next higher layer.***

If you want to ***go down a layer*** simply type **Shift-d** and the wiring tool will drop a contact or via, and bring you down to the next lower level.

There are LOTS of other options in the wiring tool, such as snap to grid, wire in 45's, change wire layer, and more.

- To see the **Wire Mode** menu simply hold down **Button-2** while in wire mode. This opens the **Wire Mode pop-up** menu.

- One of the **Wire Mode** options is **Shift-w**, which brings up a more extensive wire menu (also found as **Wire Setup...** in **File > User Preferences.**) This menu gives you such options as **Active Layer** (which can also be set via the **Active Layer Box** in the upper left corner of the MAX Window), the **Default Layer**, **Snap options**, and you can even override any or all of the wiring parameters.

TIP:

The same hotkey can be used for different functions in different modes.

For example, while in **Wire Mode** the **Shift-w** hotkey brings up the ***Wiring Menu***. In the **Main Mode** the **Shift-w** hotkey does an ***Add Wire Bus***.

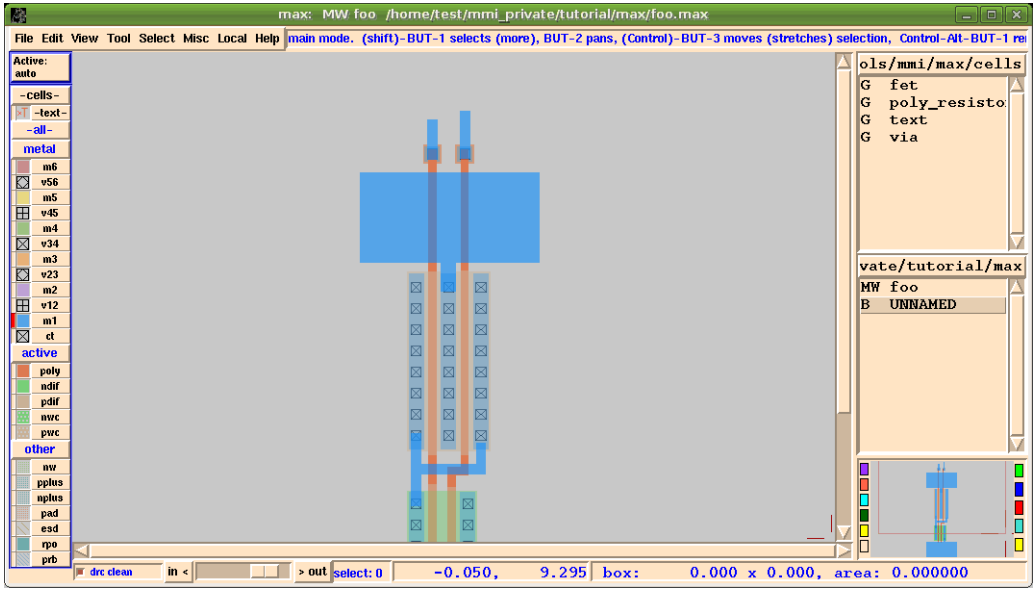
Remember, for a list of all the options (hotkeys) available in wiring mode, hit the **Space bar**, or select **Help > hotkeys**.

The **Space bar** hotkey option works for ALL modes in MAX.

Step 6 ⇨ Wire up the other poly input.

Your cell should now look like Figure 88.

Figure 88: Inputs of NAND Brought Outside



ALIGNING WIRES

If the top of the m1 wires are not aligned as shown in Figure 88, there are a number of ways to align them.

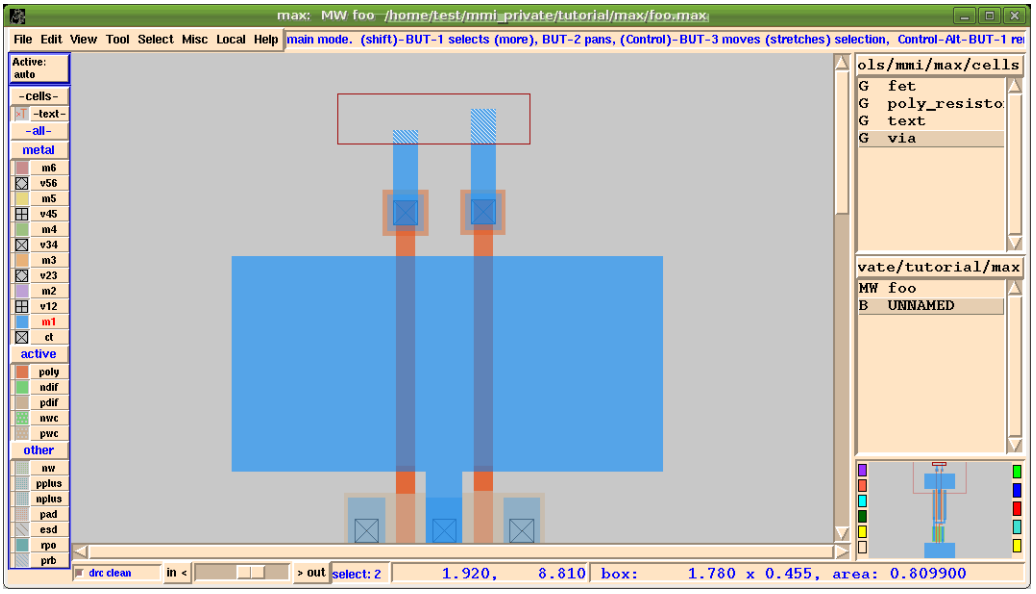
Step 7

⇨ The easiest way is to draw a box and select the top edges of both wires, as shown in Figure 89.

Then click the ***delete*** hotkey **q** and the selected paint will be erased.

⇨ If there are multiple layers selected and you only want to delete a single layer (m1) move the mouse over m1 in the palette or over a piece of m1 in the layout and type the ***erase*** hotkey **o**.

Figure 89: Aligning Wire Ends



⇨ You can also use the **Fill** command in the **Edit** menu to extend both top edges of the m1 wires up to the top edge of the box. (Refer to the Micro Magic, Inc. *MAX User Guide* for more details.)

Changing Layers

We are getting ready to finish this cell and your boss has just informed you that the power rails have been changed from m1 to m2. We'll show you two ways of changing the power rails to m2.

1st Method of Changing Layers

You'll see later that there is an easier way to change layers for this rectangle, but this method can be useful for other types of edits.

Step 1 ⇨ To change the first power rail, simply click **Button-1** over the upper (Vdd) rail to select it. You may have to click **Button-1** twice to get the entire rectangle. Notice that the Box encloses the last thing you selected.

Step 2 ⇨ Now if you type the ***Erase*** command (hotkey: **o**) over blank space (background) on the screen it will erase everything within the box. But wait! You also have some poly within the box. If you erase everything inside the box your poly will also go away.

⇨ ***Undo*** (**u**) your last change.

MAX allows you to ***selectively erase*** layers.

Step 3 ⇨ The power rail should still be selected. If not, select it again with **Button-1**.

Step 4 ⇨ Use the ***Erase*** command (hotkey: **o**) over a piece of m1 in the layout, or over 'm1' in the palette. The ***Erase*** command (**o**) tells MAX to erase those layer(s) under the cursor.

Once you have erased the m1 layer, notice that your Box is exactly the size it needs to be for the m2.

Step 5 ⇨ To repaint the erased m1 with m2, simply click the **p** hotkey (***fill with paint***) over 'm2' in the palette.

⇨ Instead of using the ***Erase*** command, you could also have used the ***Delete*** or **q** hotkey to delete the selected item.

CAUTION:

Delete will delete everything you currently have selected, including paint, polygons, gcells, and sub-cells.

As a reminder, there are several options for mouse selection.

- Clicking **Button-1** will ***select the object or rectangle of paint*** under the cursor.
- Holding down **Shift** with **Button-1** will ***add to the selection***.
- Holding down **Button-1** while dragging out a region will ***select objects and paint within that region***.

2ND METHOD OF CHANGING LAYERS

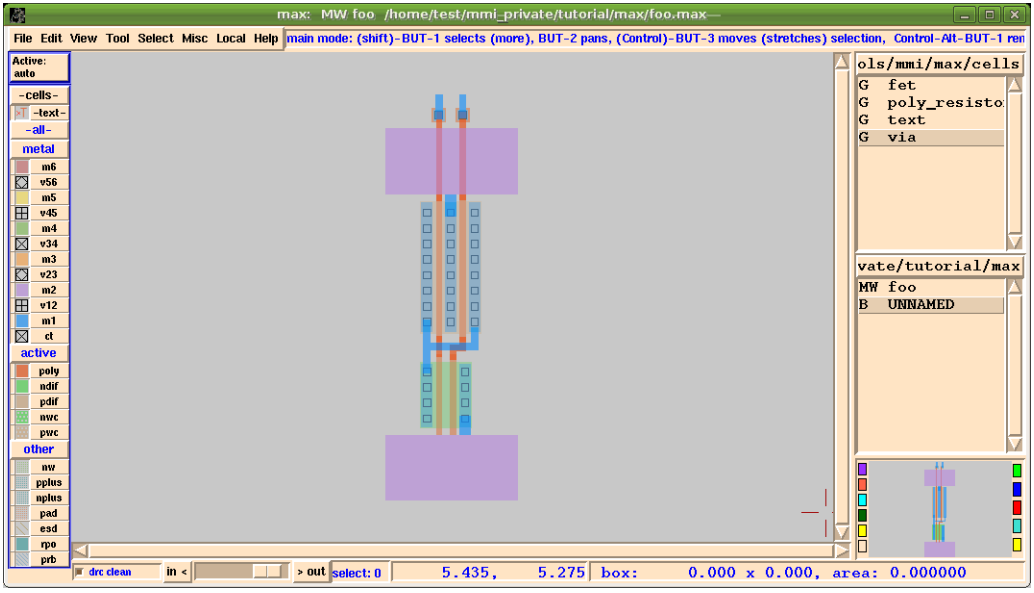
An easier way to change the layer of a rectangle is once you have selected the layer, use the niftly little **Change Layer** feature found in **Edit Properties...** under the **Edit** menu.

Let's change the lower (GND) layer m1 and replace it with m2.

Step 1 ⇨ To do this, simply select the m1 rectangle, then choose **Edit > Edit Properties...**, or type the **Ctrl-p** hotkey.

Step 2 ⇨ Change the layer to **m2** and click **Done**. You should now have something similar to Figure 90.

Figure 90: Edit Properties... Layer 'm1' Changed to 'm2'



Finish Wiring the NAND Gate

You now need to attach the power and ground lines to the new m2 straps.

To do this, use the **Wire Mode**.

Step 1 ⇨ ***Start the wire*** (**w**) from the m1 wires you drew earlier using **Shift-Button-1**, so that the wire width with match.

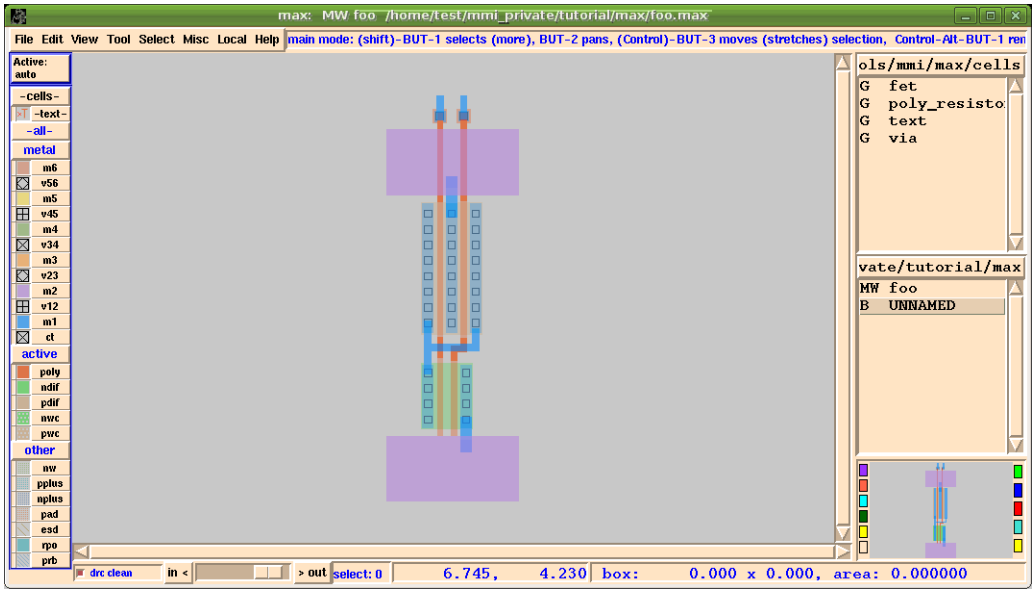
Step 2 ⇨ When you are over the m2 straps type **d** to ***add a via*** to m2.

Step 3 ⇨ Click **Button-3** to ***end the wire***.

⇨ Repeat for the second power rail.

You should now have something similar to Figure 91.

Figure 91: Connected Power Rails

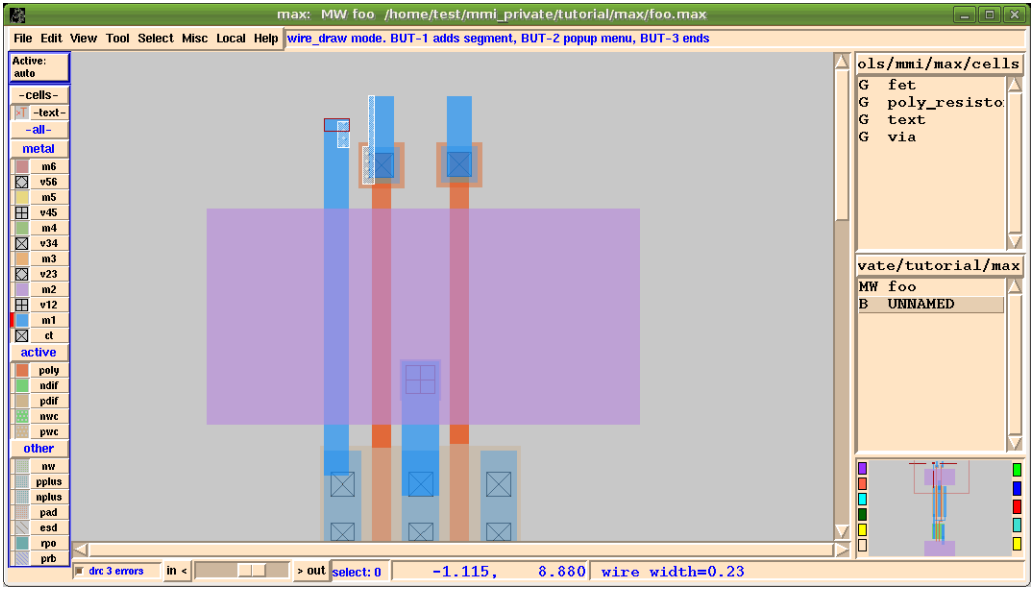


Next, we need to route the output of the top over the m2 next to the inputs so the router can get to it as well.

Step 4 ⇨ ***Start a wire*** (**w**) over the left pfet contact region (**Button-1**). Move the mouse up.

Notice that if you go straight up, you create DRC errors as shown in Figure 92.

Figure 92: Poly Gcells and M1 Wires Showing DRC Errors

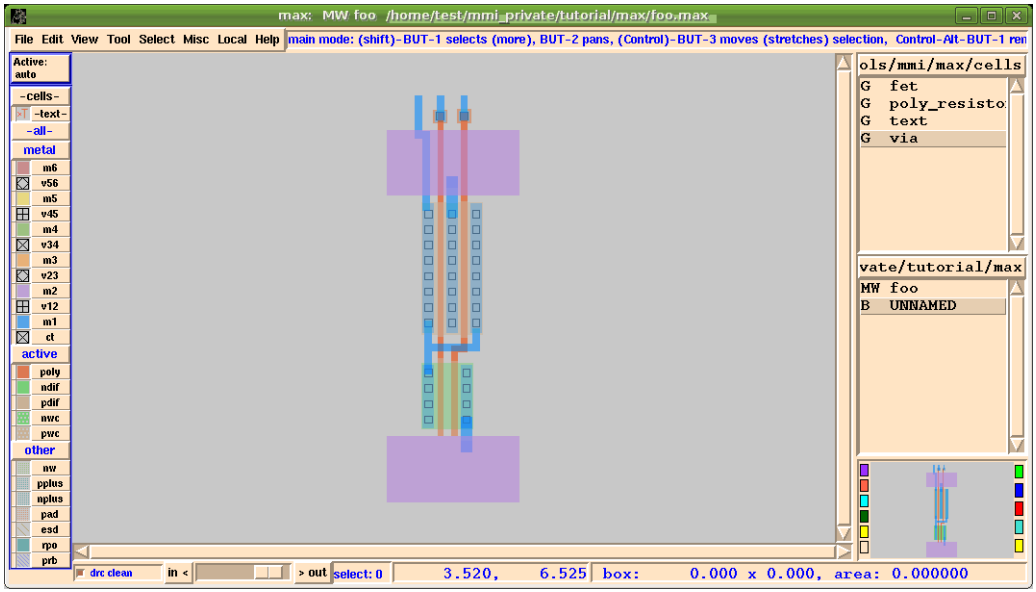


Step 5 ⇨ So that there are no DRC errors, add two corners such that the m1 wire makes a job to the left, and end the wire.

Your design should now be DRC correct and look similar to Figure 93

⇨ You've done a lot of work so ***save*** it by typing **Ctrl-s**.

Figure 93: NAND Gate Wired Up



There is one last thing before we are finished with our NAND gate.

The input and output nets of the NAND gate should be marked with unique names. This is accomplished by adding **Text** to the nets.

- To ***add Text***, you ***first select the rectangle*** where you want to put the text. Simply click **Button-1** over the rectangle, which will select the rectangle and place the Box around it, and **Add Text** (hotkey: **t**).
- To ***edit text*** that has been placed, first select the text (it will turn white) and press **Shift-t** to bring up the **MAX Text Edit** dialog box.

PLACE TEXT

Step 1

- Select the ground (lower) rail using **Button-1**.
- Type **t** for ***text***, or choose **Edit > Add Text**. The **MAX Text Edit** dialog box will appear as shown in FIG.
- Enter the name **GND** into the Text field.

The **MAX Text Edit** dialog box permits you to set the type of Text label: ***input***, ***output***, and so forth. Marking labels as inputs, outputs, global, etc. can be useful for use with other extraction or routing tools.

You can also specify the Text "position", such as **n** for north or **s** for south. Specifying the orientation of the Text, relative to the crosshair that marks the Text location, places it on one side or another for easier viewing.

TIP:

North and south orient vertically, above or below the crosshair. All other directions orient horizontally, to the left (nw, w, sw) or right (ne, e, se) sides of the crosshair.

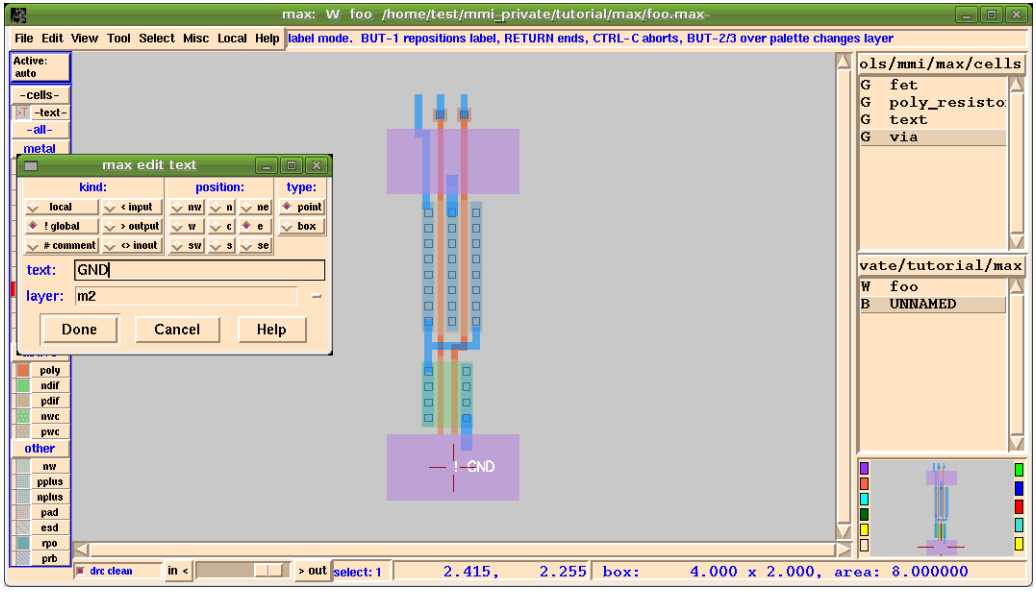
The Text "type" can be a ***point*** (the default) or a ***box***. The"box" option produces text which marks a specified area, but is rarely used.

Step 2 ⇨ Specify the GND text direction to be **e** (***east***) and specify that it is a **global** signal.

Step 3 ⇨ Now move the mouse to the point on the layout you wish the text to be and click **Button-1**.

⇨ Click on **Done** or hit **Return**.

Figure 94: Adding Text “GND” to Bottom Power Rail



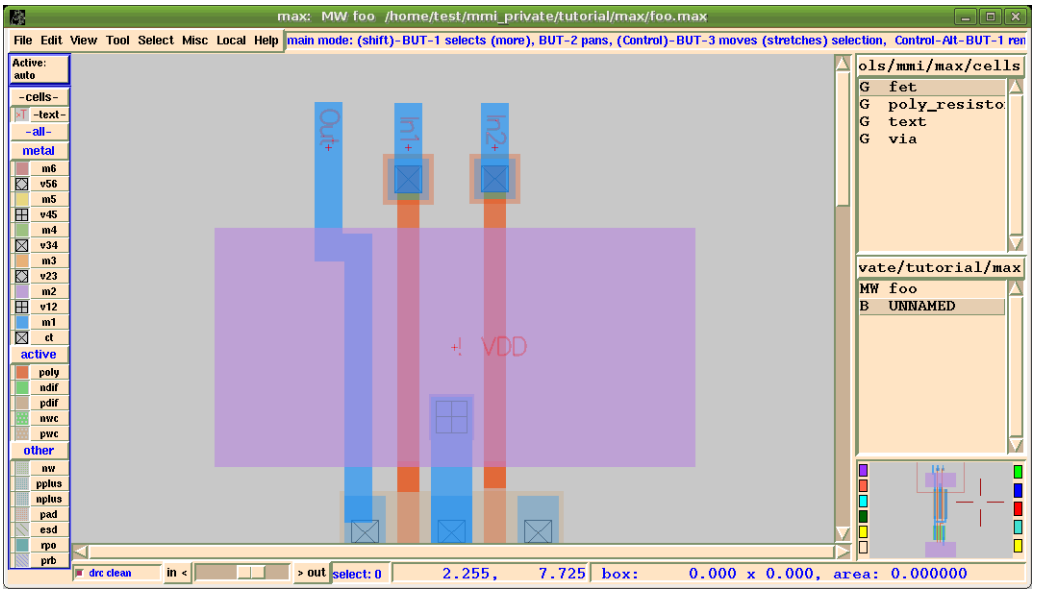
In this tutorial:

- VDD and GND are marked as ***global***, and are positioned ***east*** (**e**). The default location for the text point of origin is in the center of the selected rectangle. Text was positioned at the far right side of the rectangle.

- The inputs (In1, In2) and output (Out) were marked as ***local***, and are positioned ***north*** (**n**).

Step 4 ⇨ Add labels for VDD, In1, In1, and Out as shown in Figure 95.

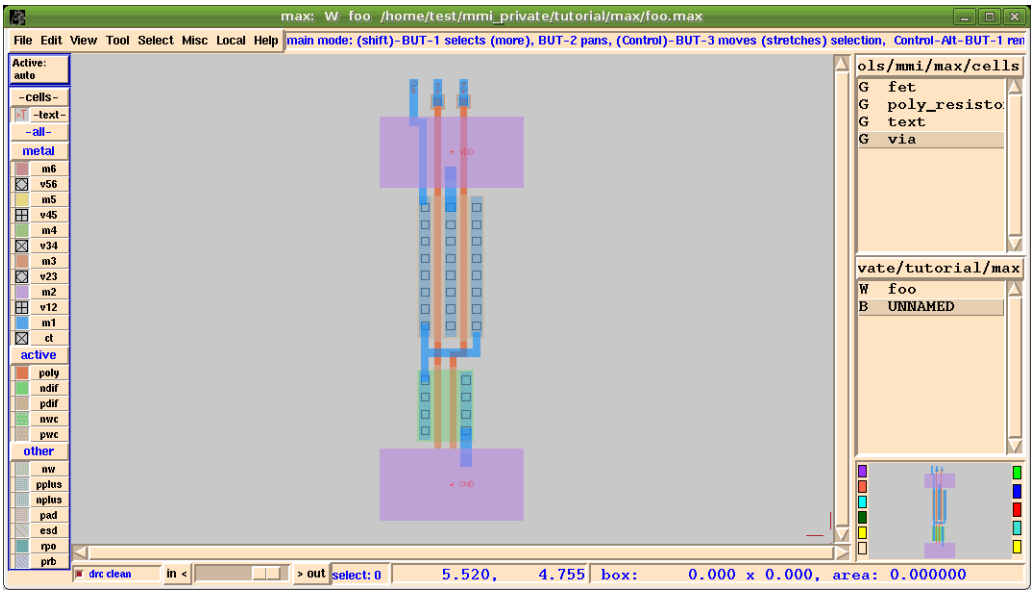
Figure 95: Adding Text Labels



⇨ Remember to save your cell (**Ctrl-s**).

At this point our little NAND gate example should be done, and yours should look very similar to Figure 96.

Figure 96: Finished NAND Gate



COPY CELL TO NEW NAME

You can copy the cell into a new name. Let's call it my_nand.

Step 5

- ⇨ Select **Save As...** from the **File** menu.
- ⇨ Type in "`my_nand`" in the **Filename** field and click **OK**.
- ⇨ You will now see both `foo` and `my_nand` in the List Box. You are currently editing `my_nand`.

Selecting Nets

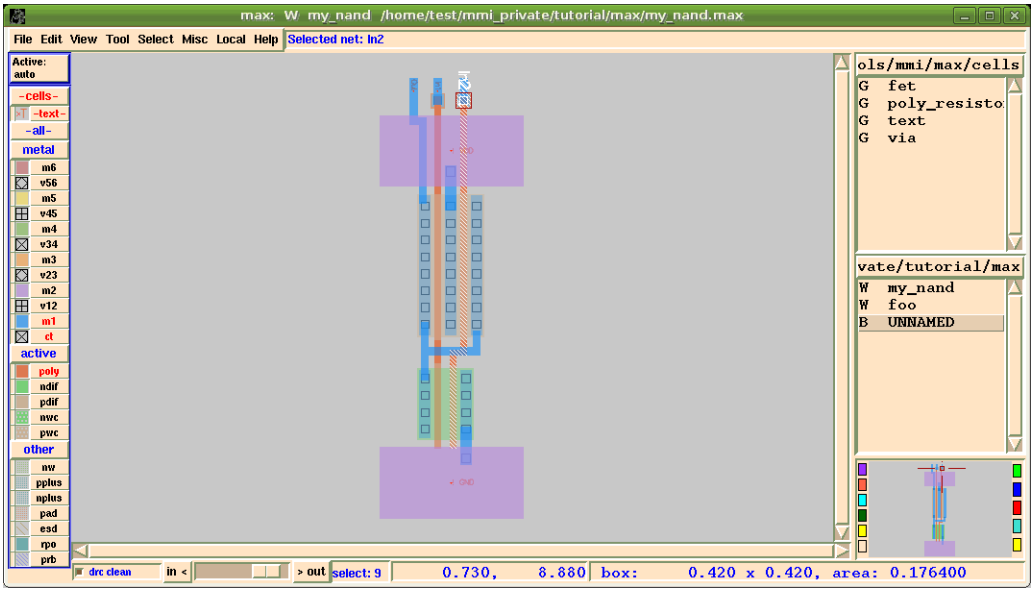
MAX can trace connectivity through vias and contacts to interactively show everything connected to a net.

- ⇨ Place the mouse over any piece of metal, poly or diffusion and type **s**, or choose **Select > Select Net**. MAX will automatically select the entire net that is connected to that piece of layout, and will also display thenet name, if known, in the **MAX Message Area**. The net name(s) are taken from any Text (labels) attached to the net.

⇨ Move the mouse over the m1 rectangle for 'In2' in the upper right corner.

⇨ Type the **s** hotkey. You should now see the net highlighted as in Figure 97. Also, in the Message Area, you should see "Selected Net: In2".

Figure 97: Highlighted Net



If there are multiple layers under the mouse cursor, you type **s** multiple times to cycle through the nets.

TIP:

One nice feature often used for debugging is that when MAX selects a net the names of all of the text labels on that net are displayed in the **MAX Message Area**. For example, if you select VDD and the names **VDD** and **GND** both appear, you know you have a problem.

Additional Commands

Some of this is a repeat of earlier information. It's here again simply as a reminder.

- The **Cursor Probe** command is another way to see which layers and objects are under a particular point.
  - To ***Probe***, put the mouse over some layout geometry in your cell and select the **Cursor Probe** hotkey **I** ("ell", lower case L).
  - You could also choose **Select > Cursor Probe**, then point at the location you want to probe. The **Cursor Probe** menu that appears contains a list of all the layers and objects under that point.
  - To ***select a rectangle*** or other object, click on it in the pop-up with **Button-1**. To deselect it, click again.
  - by default, only items in the current edit cell are displayed, since those are the only ones you can effectively manipulate. But if you choose the **All Expanded Cells** radio-button, the list will contain everything in any cell under the point where you probed.
  - You can also ***clear the selection*** by choosing **Select > Clear Selection**.
- Another way of ***duplicating layout*** is to use the **Copy Cell Buffer** command in the **File** menu. This command will copy the current cell buffer to a new cell buffer in memory. The new cell will be saved to disk when you **Save** the cell.
- The **Save As** command (also in the **File** menu) is similar to **Copy Cell Buffer**, but also immediately saves the cell to disk.
- If you are working on a cell and want to ***discard its contents***, perhaps to completely replace it with another cell, you can use the **Delete Cell Buffer** command (**File** menu) to delete the in-memory copy of a cell. This command does not affect the file on disk.

PART 4

BIGGER DESIGNS AND HIERARCHY

Now let's use our NAND gate and an inverter cell to build something a little bigger.

LAUNCH MAX USING MMI18

Step 1 ⇨ If you have quit out of MAX, start it using

```
max -tech mmi18
```

Step 2 ⇨ Go to the **File** menu and select **New...** Name the new cell "`row`".

- You should now be editing `row.max`.
- Look up at the title bar. It should say: `max: B row`.

MAX does not automatically load all the cells in a library. Auto-loading all the cells can be done from the Cell List if all MAX cells in the directory use the same technology, or by adding it to your `max.rc` file.

Because the tutorial directory contains both cells in the `mmi25` and `mmi18` technologies, we'll load the cells manually.

Step 3 ⇨ Select **File > Open**, then `INV.max`, and click **OK**. Do the same for `NAND2.max`.

You should now see `INV` and `NAND2` listed in the Cell List box.

⇨ **RIGHT-click** (**Button-3**) on "`row`" in the Cell List to return to your new cell.

DROP A NAND2

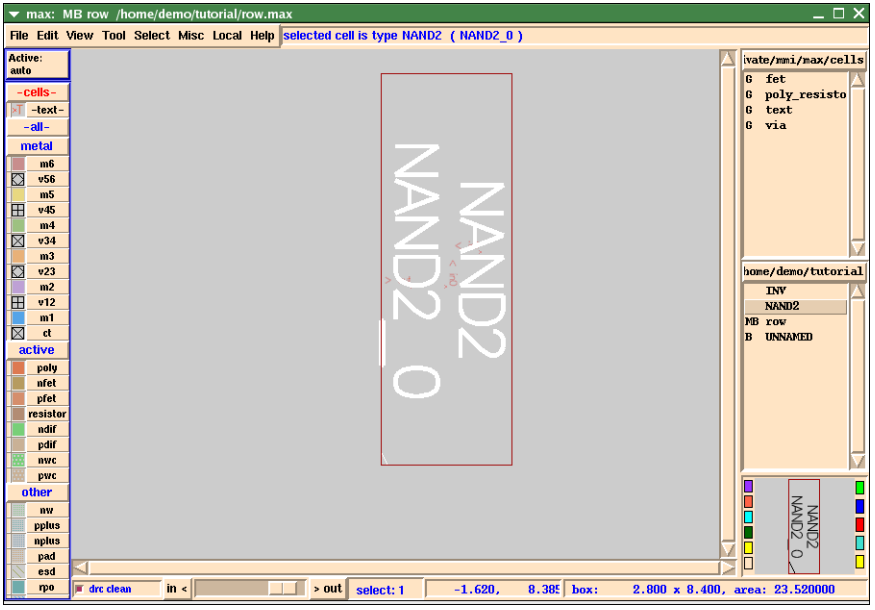
Step 4 ⇨ In the Cell List box, click on `NAND2` with the **LEFT mouse button** (**Button-1**).

This ***drops an instance*** of NAND2 into your row cell. Now you should see something like Figure 98.

You only see the ***instantiation*** of the NAND2 cell and not the internals. We'll cover that later.

(Remember, clicking with the **RIGHT mouse button** (**Button-3**) ***loads*** that cell into MAX.)

Figure 98: Instantiation of NAND2 Cell



Step 5 ⇨ Thie new cell should already be selected. If not, select it now by clicking with **Button-1**.

SET UP ARRAY

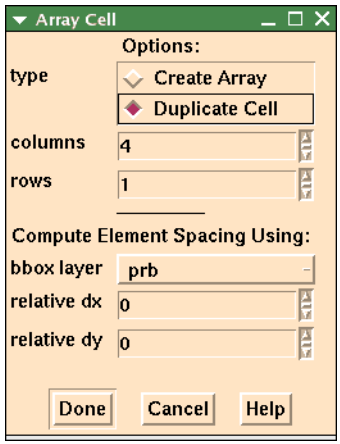
Step 6 ⇨ Select **Array Cell...** from the **Edit** menu.

⇨ Enter 4 for the **columns** field and Duplicate Cell for the **type**.

- Look at **bbox layer**. By default it says **_bbox_** which is the rectangle containing all of the layout of the cell. For this cell the **nw** layer extends beyond the power rails, and we need to use the **prb** layer so that the wells overlap.
- Click on the rectangle to the right of **_bbox_**, scroll down and select `prb`.

  The array cell pop-up should now look like Figure 99.
- Click on **Done**.

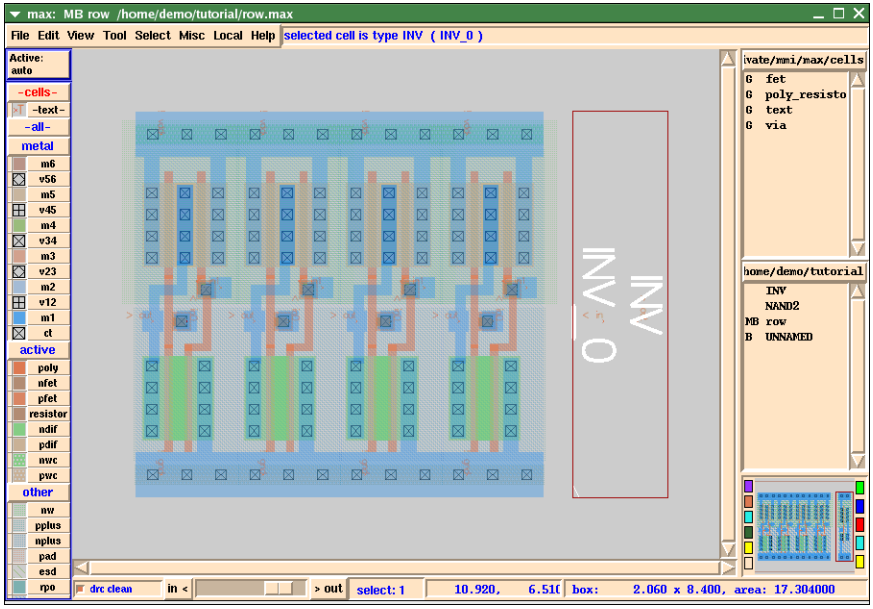
Figure 99: Array Cell Pop-up



Step 7
- ***Zoom out*** (hotkey: **Shift-z**) so that you can see all of the `NAND2` instances.

Step 8
- Return to the Cell List box and click on `INV` with **Button-1**.
- Place it to the right of the `NAND2` cells as shown in Figure 100.

Figure 100:'INV' placed to the right of 'NAND2' Cell



POSITION INV CELL We now need to move the INV so that it abuts the NAND2 gates.

Step 9 ⇨ The INV should still be selected. If not, click with **Button-1**.

Step 10 ⇨ Select one of the NAND2 gates by holding down the **Shift** key as you click with **Button-1**. You should now have the INV and one of the NAND2 cells selected.

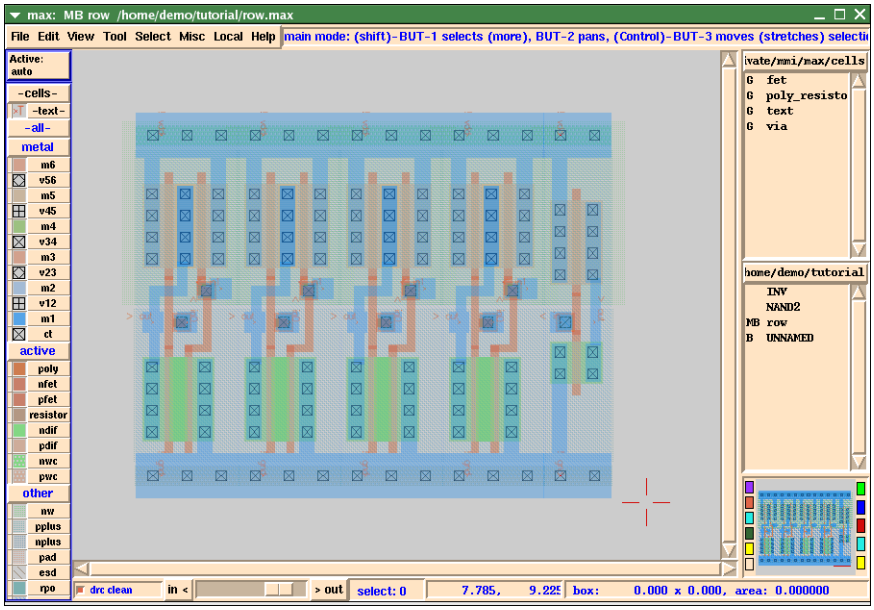
⇨ The **align** command aligns all cells to the ***last*** cell selected. Menu select **Edit > Align Objects**, or type **Ctrl-a**. When the pop-up opens, align the objects to the **bottoms**, then click **Done**.

Step 11 ⇨ Type the hotkey **i** to ***view the internals*** of all of the cells.

Step 12 ⇨ Select just the INV cell. We want to move it over so that the power rails abut. Hold down **Shift-Button-3** (***lock move either horizontal or vertical***) and move INV over until the power rails abut as shown in Figure 101. It will be easier to abut the cells if you zoom in on either the upper or lower power rails.

➪ You can also use the **left arrow key** to move the INV over one grid unit at a time.

Figure 101:MAX Hierarchy - INV Abuts NAND2



While moving the inverter, white dots may appear indicating that the DRC check is working. DRC checking runs automatically as you go, and the dots disappear once the inverter is correctly positioned. The DRC status area in the lower left corner of the MAX window should say “drc clean”.

Push, Pop, and Edit in Place

Let’s look at editing hierarchical layout. At this point let’s say our boss comes over again and says:

> “Sorry, but you need to bring the outputs to the bottom of the cells.”

There are two ways to do this. We will use one method on the INV cell and the other (just to be different) on the NAND2 cell.

First Method: Editing by Pushing Into Cell

EDIT BY PUSHING IN

Step 1 ⇨ First, select the INV cell by moving the cursor over INV and then using **f**, or **Select > Select Cell**. If you get a fet or row selected, type the **f** hotkey multiple times until INV is selected.

Step 2 ⇨ Now type **e** to ***edit the cell***, or menu select **View > Push into Cell**.

Now that you're inside the cell, use the wire mode to:

Step 3 ⇨ ***start the wire*** (**w**) at the right nfet contact region, and click **Button-1** while moving the mouse down,

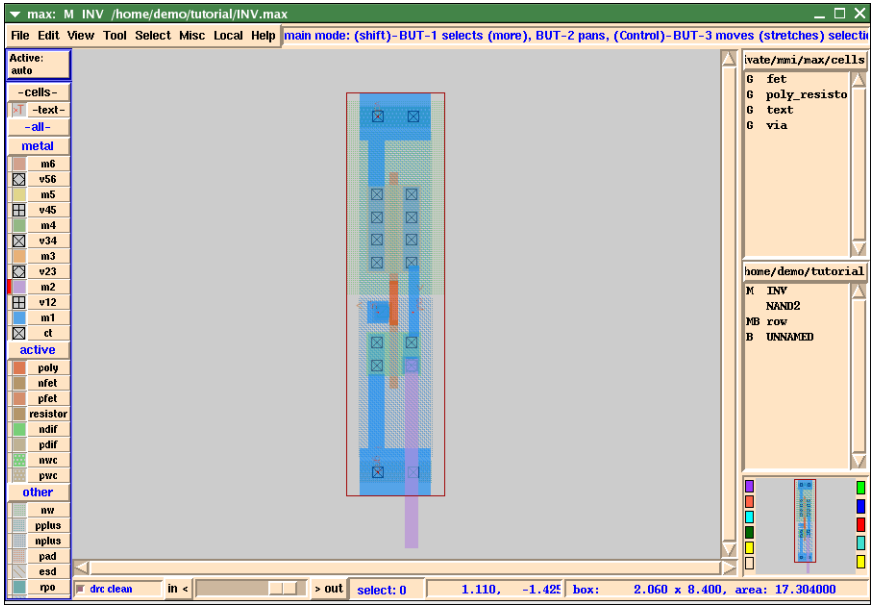
⇨ ***drop*** (**d**) a contact to m2,

⇨ ***anchor*** (**a**) the via, move the mouse downward past the GND power rail

and

⇨ ***end the wire*** using **Button-3**. Refer to Figure 102.

Figure 102:Editing 'INV' Cell - Wiring Output



Step 4 ⇨ Now type **Ctrl-e** or menu select **View > Pop out of Cell** to "pop" up a level and return back to `row.max`.

You should now see that the change you made in the `INV` cell within `row.max`.

Second Method: Editing Cell or Object in Place

EDIT IN PLACE Now let's edit the `NAND2` cell. To do this one we are going to use the **Edit Cellor Object in Place** option from the **View** menu.

Step 5 ⇨ Select one of the `NAND2` cells, then type **Shift-e** to ***Edit in Place***.

Notice that the `NAND2` cell you selected appears darker than the other cells. This indicates that you are now editing that cell.

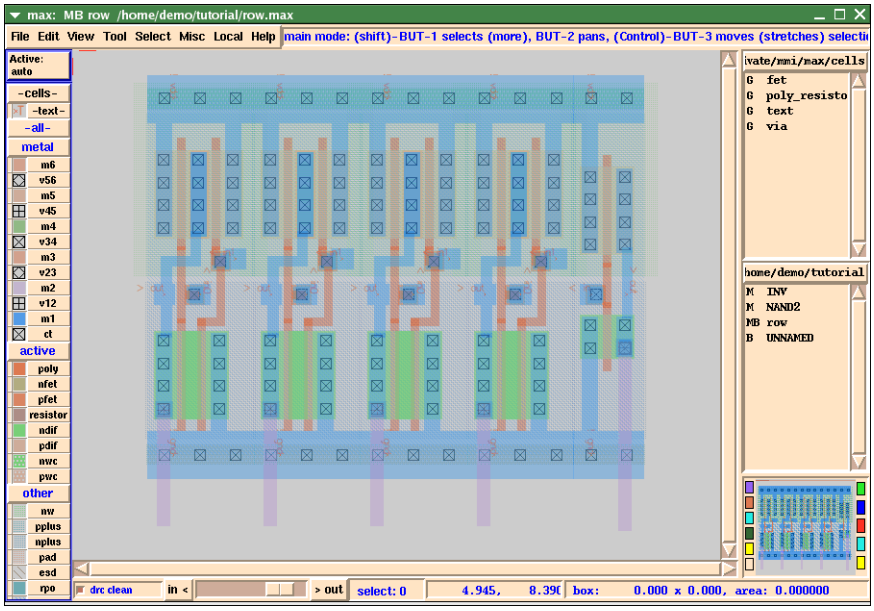
Step 6 ⇨ Go ahead and add your new wire just as you did for the `INV` cell. You will need to put the `m1` to `m2` via directly over the nfet contact region.

⇨ Observe that the wire appears in ALL INSTANCES of NAND2, not just the one your are editing. Look down at the DRC status in the lower left corner ofthe MAX window and it should say drc clean.

Step 7 ⇨ Type **Ctrl-e** to ***pop back up*** to the top level (row).

⇨ Your row cell should now look like Figure 103.

Figure 103:Outputs Brought Out to Bottom



Now, just for grins, let's wire up two of the gates.

- Since we want to wire them up in row.max and NOT in NAND2.max, we must first make sure row.max is our edit cell.

  NOTE: You can always tell which cell you are editing by looking at the description in the **MAX Title Bar**.

To get back to row.max (if you're not there already) there are different methods:

Step 8 ⇨ Type **Ctrl-e** a few times or until you are sure you have popped out of any cells you entered when using **Shift-e** or **View > Edit Cell or Object in Place**. Watch the MAX Title Bar.

⇨ You can also click with **Button-3** on `row` from the Cell List box.

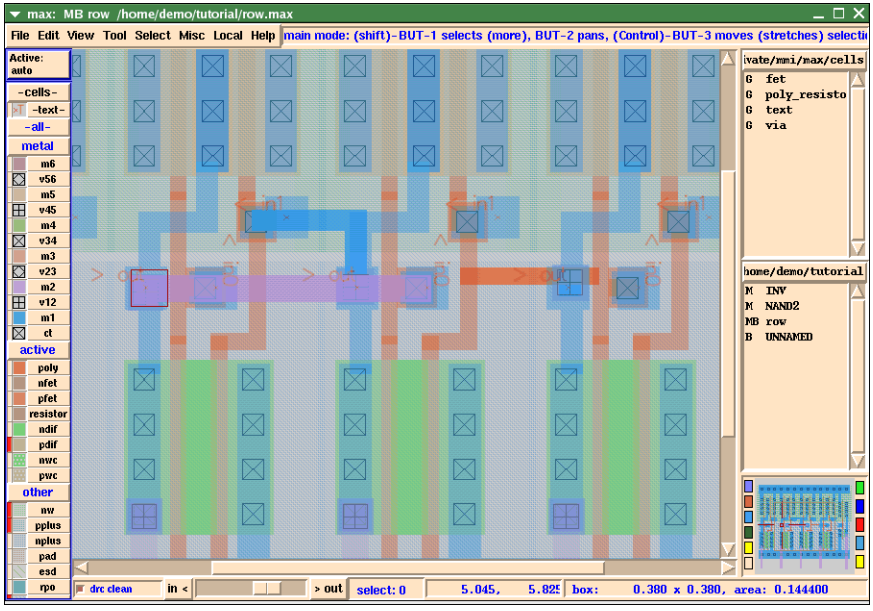
In general, ***push***, ***pop***, and ***edit in place*** work well for cells in the same hierarchy, while using the Cell List boxes is the preferred method for traversing larger, less connected databases.

Step 9 ⇨ Now that `row.max` is your edit cell, use the **wire mode** to hook up a couple of wires.

⇨ Notice that these wires are once again darker than the wires in the cells.

⇨ Figure 104 shows a zoomed in view of a couple of gates wired up within `row.max`. You can see that the new wires in `row.max` (the current edit cell) are darker than the wires contained within `NAND2.max` and `INV.max`.

Figure 104:Cells Wired Up



Let's save our work.

Step 10 ⇨ Menu select **File > Save Multiple**, then **Save Edit Cell and Descendents**. This saves the current edit cell row, as well as any modified cells that are instantiated in that row.

Step 11 ⇨ ***Exit*** out of MAX by typing **Ctrl-d** or selecting **File > Exit**.

⇨ If you have made changes to cells since your last save, you'll get a warning pop-up message. You can either **Exit and Lose Changes** or **Cancel** and save the cells.

PART 5

IMPORTING GDSII AND CUSTOMIZING MAX

READING IN GDSII FILES AND MAX TECHNOLOGY FILES

Each GDSII or MAX layout file needs an associated **technology file** to describe the layer names, how layers connect to each other, and layer colors and fill patterns. This tech file contains the color (HSV or RGB), the transparency, and the dot/stipple pattern and/or outline settings, DRC rules and other information concerning each layer. It also contains information on how to map GDSII layer numbers to layer names.

The technology file we used for `TOP_CHIP` near the beginning of this tutorial is a generic technology created using Micro Magic, Inc. standard layer definitions and colors. When you read in your own GDSII file, MAX will need the information for that technology. We will cover two different ways for obtaining that information.

Creating Technology Files from GDSII Files

One way to get the technology information is to let MAX figure out what the layers are. You'll need to have a list showing you which GDSII layer numbers map to which layer names. This is used to verify that MAX picked the layers correctly. MAX generally figures out metal, via, and poly layers correctly for digital CMOS processes.

MAX will default to the Micro Magic layer color and fill patterns. You can modify these settings as needed, and as often as needed. Black background options are available for MAX technology files by adding the "`-bb`" modifier (example: `-tech mmi18-bb`).

IMPORTING GDSII FILES

For the purposes of this tutorial, we will be using an example GDS file included in the tutorial installation.

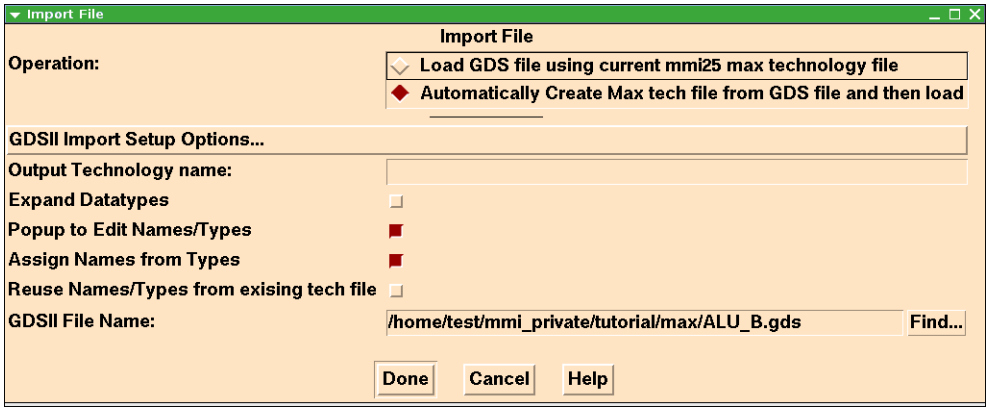
Step 1 ⇨ If MAX is still running, quit MAX (**Ctrl-d**).

If you do a "pwd" you should see the directory: ~/mmi_private/tutorial/max.

Step 2 ⇨ Start MAX by typing "max" on the command line.

Step 3 ⇨ Select **Import File...** under the **File** menu. You should see a pop-up like Figure 105.

Figure 105:Import GDSII File



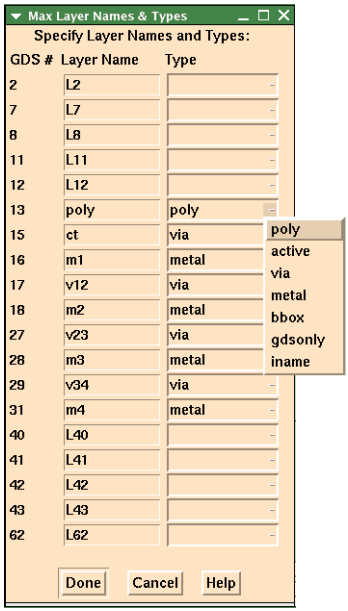
Step 4 ⇨ Click on "**Automatically Create MAX tech file from GDS file and then load.**"

⇨ Click on the **Expand Datatypes** toggle button so that it is turned **off** as shown in the above figure.

⇨ Click on **Find...** (at the lower right) and select ALU_B.gds.

⇨ Click on **Done** from the **Import File** pop-up. You should now see the **Layer Names** pop-up as shown in Figure 106.

Figure 106:Layer Names and Type Pop-up



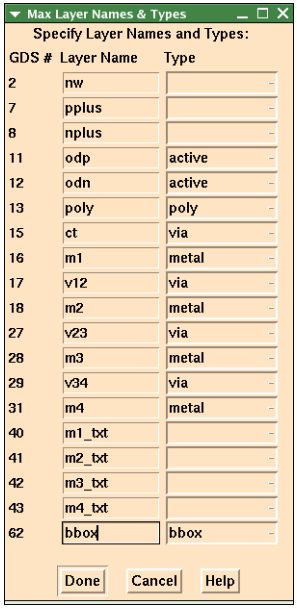
If the layer is not metal, via or poly, then the **Layer Name** is simply a generic name. We will change some of these names to make more sense.

The **Type** is a category MAX uses to determine how layers are connected.

TIP:

To simply have a quick look at the layout, you don't need to specify any of the layer names or types. The layers for poly, metal, and vias need to be defined as the correct type in order for **Select Net** to work. In order to create fet and via gcells, MAX need to have the active layer specified as well as some of the DRC rules. How to specify DRC rules is covered in the *MAX User Manual*.

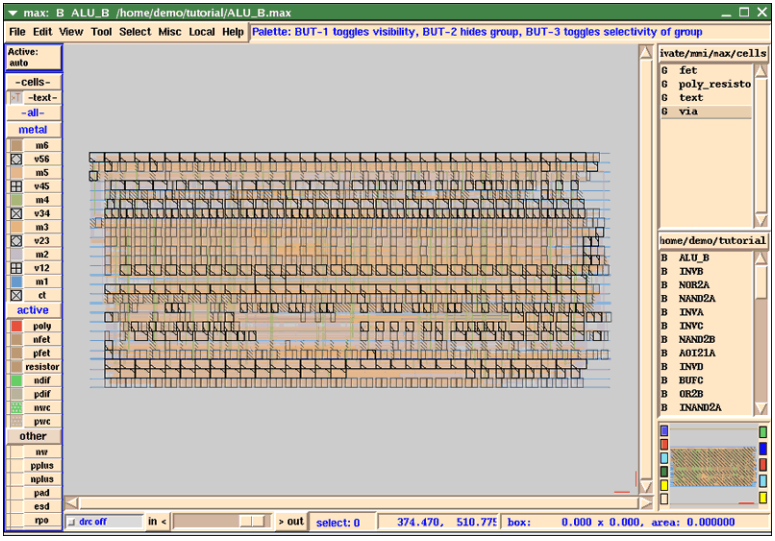
Figure 107:Changed Layer Names



Step 5 ⇨ When prompted, click on **Restart Max Automatically Now with File**.

You should now see something like Figure 108.

Figure 108:Results of Changing Layer Name(s) - “other” layers now turned OFF

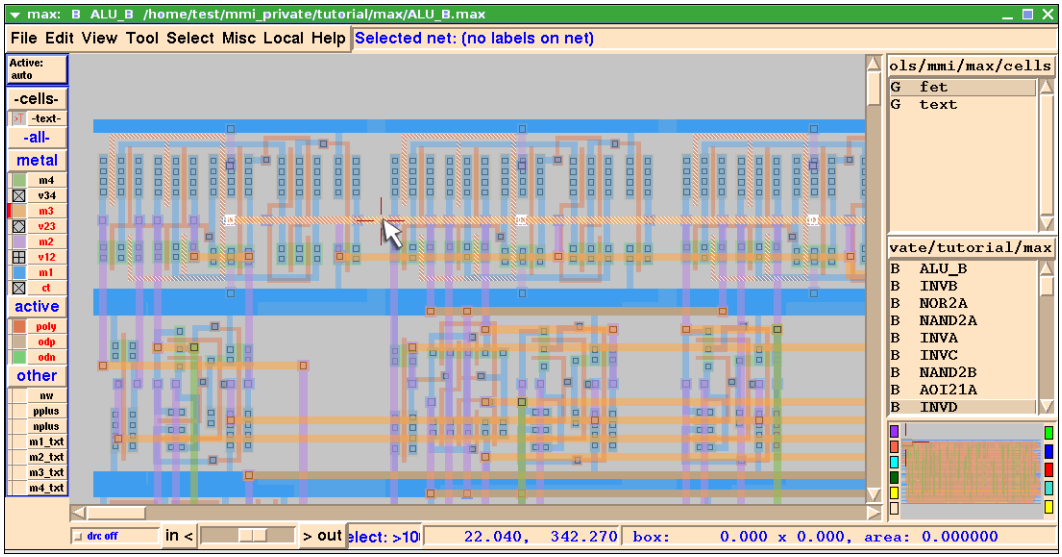


Step 6 ⇨ Zoom in on the upper left corner of the layout, and type the hotkey **i** for **Internals, View All** to display internals for the layout.

(Note that the “**other**” layers in the palette have been turned off for greater clarity in this tutorial.)

Step 7 ⇨ Select a net. Type the hotkey **s** (for **Select Net**, in the **Select** menu) to see layer connectivity tracing working. In Figure 109 the net attached to the m3 layer has been selected.

Figure 109:Tracing Net Connectivity



⇨ To view the extent of the selected net, use the hotkey **Shift-v**, or click on **Zoom to Fit Selected** from the **View** menu. The display will zoom out enough to show the entirety of the selected net.

Because this layout was read in from GDSII and the technology file was created from this GDSII, there are no DRC rules in this particular technology file. Those rules can be added manually, as described in the MAX Technology Files chapter of the *MAX User Manual*.

Modifying Palette/Layer Styles within MAX

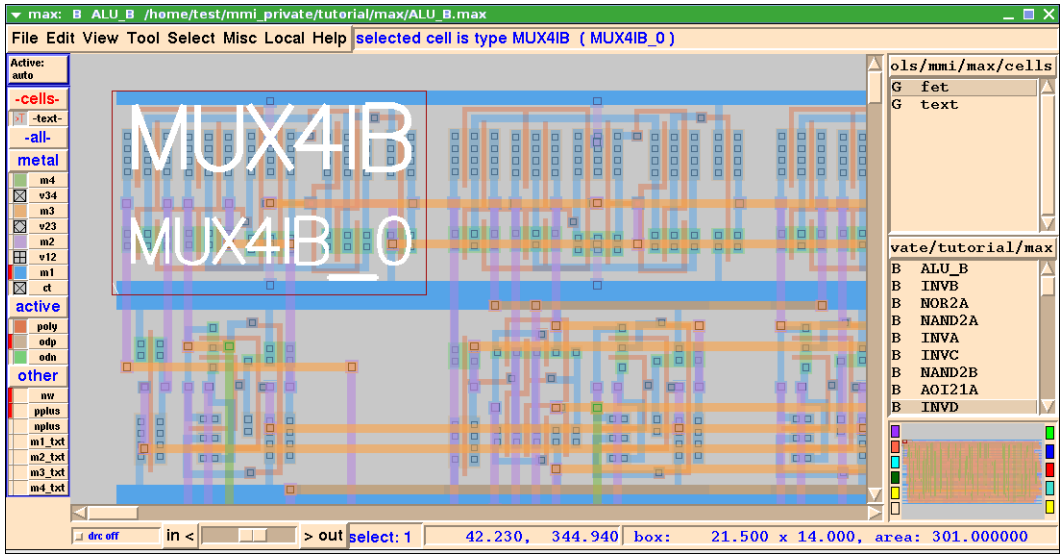
MODIFY LAYER COLORS

Let's change some of the layer colors and stipple patterns for the cell ALU_B.

Step 1

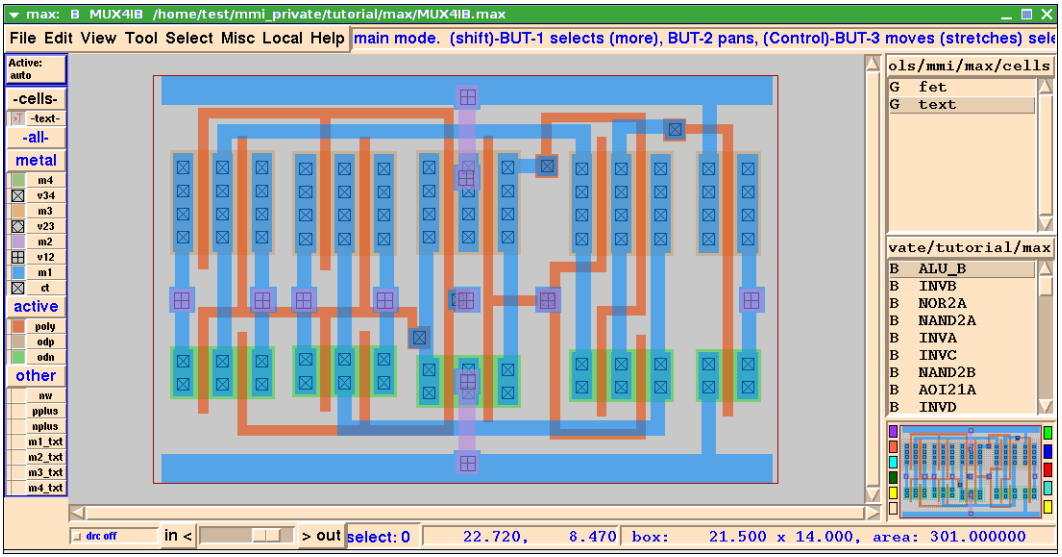
⇨ Zoom in on the very upper left corner of ALU_B. Click on the corner cell with the **Button-1** or use the select cell command (hotkey: **f**). You should now see MUX41B_0 selected as shown in Figure 110

Figure 110:Cell MUX41B Selected in Upper Left Corner of ALU_B



Step 2 ⇨ ***Push into the cell*** `MUX41B` using the **e** hotkey, as shown in Figure 111. This will make any changes to layer colors in the palette much easier to see.

Figure 111:Editing MUX41B Cell

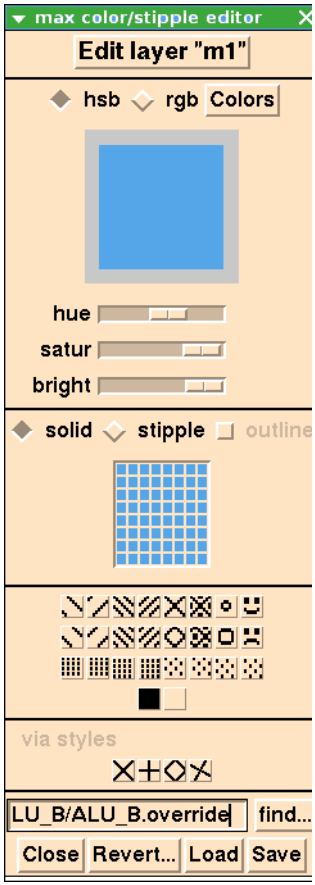


CHANGING LAYER COLOR

First, we need to open the **Color/Stipple Editor**.

Step 1 ⇨ Click with the **MIDDLE mouse button** (**Button-2**) on **m1** in the palette or, from the **File** menu, select **User Prefs** and then **Color Editor**. Figure 112 shows the **Color/Stipple Editor** and its features.

Figure 112:Color/Stipple Editor



Layer Selection pull-down menu - Click and hold to select layer to edit.

Select HSB or RGB values, or choose from a list of pre-set colors.

This box displays the solid fill color.

HSB (or RGB) slider bars. Move these left or right to change the color.

Fill pattern buttons: Solid for solid fill; Stipple for patterned fill; Outline draws an outline around that fill.

Stipple edit area

Pre-set Stipple patterns. Use as-is, or modify further as needed.

Solid fill or "nothing/empty" fill buttons.

Pre-set via styles.

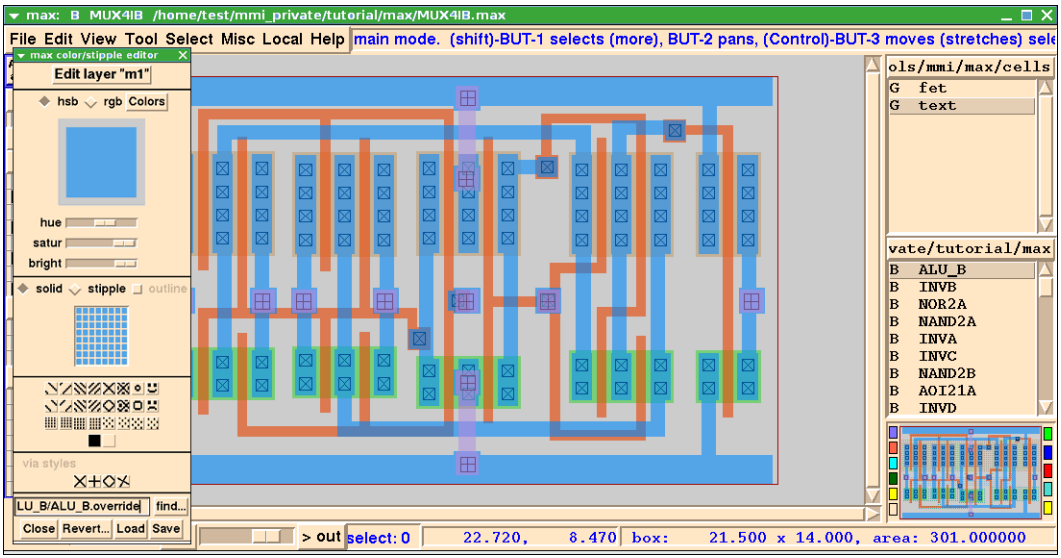
Find... palette file button

Buttons for: Close color/stipple editor (without saving), Revert... (to last saved version), Load (palette file) Save (this palette as a new file)

Step 2 ⇨ Select the layer you want to edit from the pull-down menu at the top.

In this example we'll be editing layer "m1". If you brought up the **Color/Stipple Editor** using the **MIDDLE mouse button** on the palette, then m1 is already the edit layer. Otherwise, select m1 from the list.

Figure 113:Selecting Layer “m1” to Modify

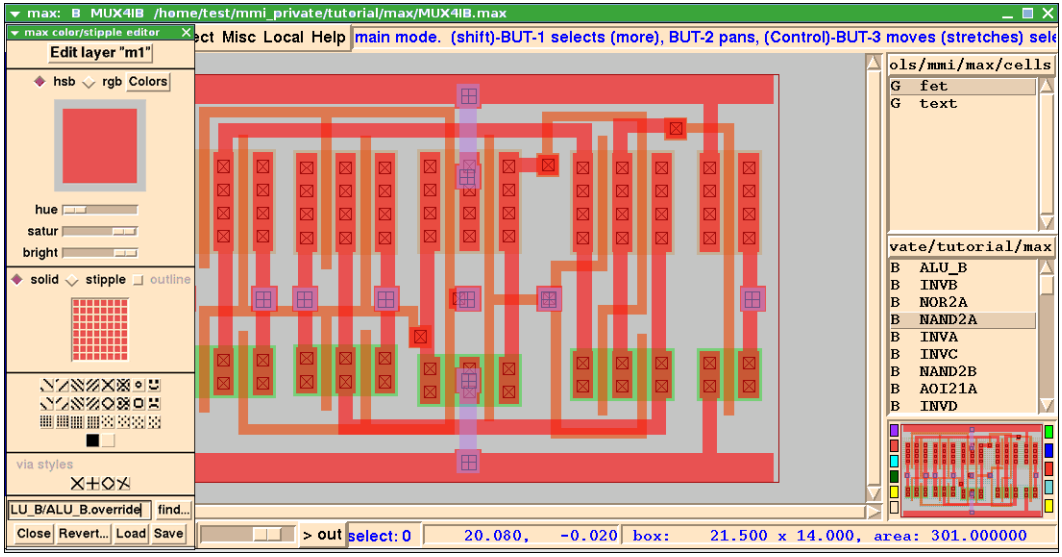


Step 3 ⇨ Now that you’ve chosen the layer to edit (“m1”), it’s time to choose the color and fill pattern. Click on the radio buttons at the top to work in either HSV or RGB values, and then modify the color itself using the slider bars underneath.

Colors will change dynamically as you work.

In this next illustration (Figure 114), the bright blue color has been changed to bright red by sliding the HSV bars. The H S and V bars were pushed all the way right. You can see an immediate difference in the layout.

Figure 114:Color Changed to Bright Red in Editor and Layout



⇨ If you prefer, select a pre-set color from the list available by clicking the **Colors** button. This list presents both color names and their RGB triplets.

CHANGING STIPPLE PATTERN

Step 4

⇨ Click on the **stipple** toggle button. Now experiment with stipple patterns by clicking on the grid of squares in the stipple-pattern area to toggle the color on/off.

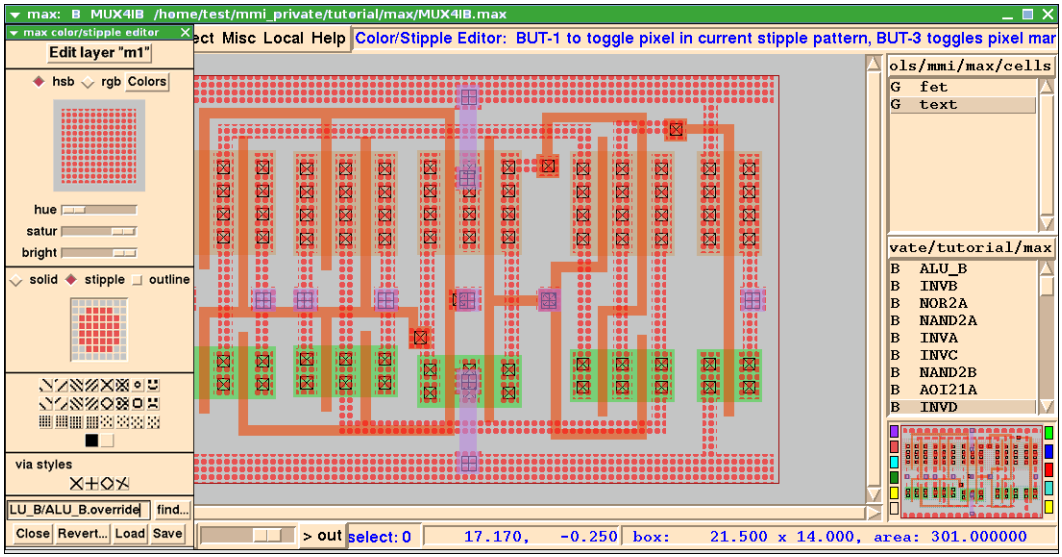
As you make changes, these are reflected immediately in the display window. Figure 115 shows effects of this stipple pattern on the layout.

⇨ For speedier editing, you can select the '**solid**' square button under the stipple patterns and remove colors for a heavy stipple such as this example. And the reverse is also true — select the '**empty**' square button and click on squares to set up a light stipple, or to clear a pattern and start fresh.

⇨ With the immediate feedback on the layout view, it's very easy to see when a stipple pattern is "right".

In this fashion, you can generate stipple/fill patterns to mimic those from any other layout editor.

Figure 115:Large Dot Stipple Pattern in Editor and Layout



Step 5 ⇨ Clear the edits you've just made by clicking on the **Revert...** button at the bottom of the color/stipple editor. Then click on **Done** to revert to the **Start-up Defaults**. This reverts the palette settings to the last saved version.

⇨ If you would prefer to save the changes you've made to the palette, click on **Save**. The default location for the palette override file is: `~/mmi_private/max/tech/ALU_B/ALU_B.override`.

Since we didn't specify a technology name we we read in the GDSII, the technology name defaulted to the name of the GDSII file. When the technology file was created from the GDSII, MAX automatically made an `ALU_B.palette` file with the default colors. The `ALU_B.override` is read in after the `ALU_B.palette` file when MAX is started up with this technology, and therefore overrides the layer color and fill patterns.

A pop-up will open, asking if you'd like to override the default palette file. You can type in a new palette name in the field to create a new palette file, or override the current palette file.

Step 6 ⇨ Exit MAX by either typing the hotkey **Ctrl-d** or selecting **Exit** from the **File** menu.

USING A BLACK BACKGROUND

If you prefer working with a black background, MAX has a pre-assigned palette already set up.

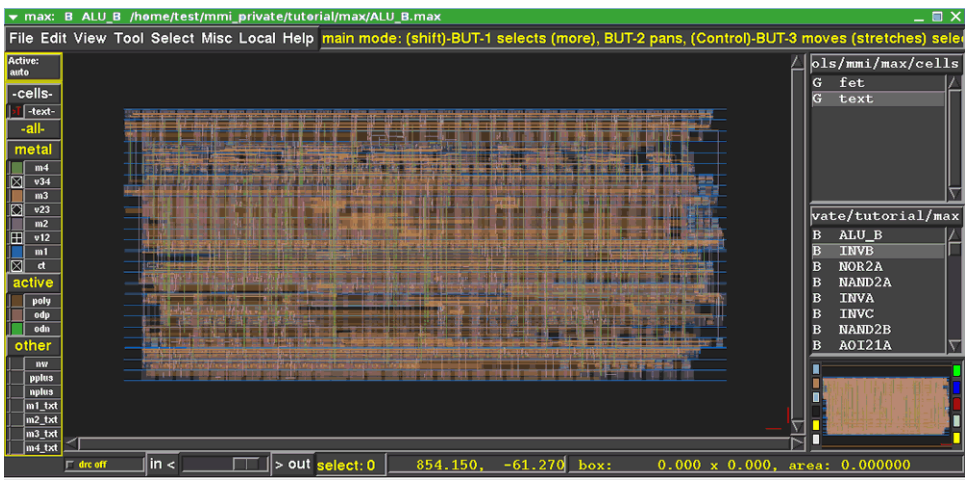
Step 7 ⇨ Let's try this with our technology. On the command line, type:

```
max -tech ALU_B-bb ALU_B.gds
```

Step 8 ⇨ View the internals (hotkey: i).

You should now see something similar to Figure 116.

Figure 116:ALU_B Technology with Black Background



Step 9 ⇨ Exit MAX (hotkey: **Ctrl-d**).

Importing Layers and Palette Styles from Another Program

In addition to GDS and ASCII file formats, MAX can also handle industry-standard formats, including **.DRF** and **OpenAccess**.

Importing Technology Information From A PDK

IMPORTING A "DISPLAY.DRF" FILE

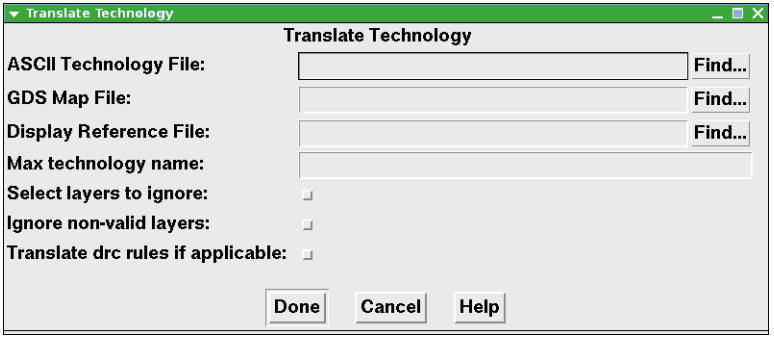
The foundry generally provides you with a Physical Design Kit (PDK) which should contain an ASCII tech file and sometimes a separate GDS layer map file. If you have layer colors and fill patterns specified for Cadenc's *Virtuoso* layout editor, MAX can also read in that information from the **.drf** (display reference file) file.

Step 1 ⇨ From the command line (shell window) type:

```
cds_convert
```

This opens the **Technology Translation Menu**, as shown in Figure 117.

Figure 117:Technology Translation Menu for "cds_convert" to Import "display.drf" files



Step 2 ⇨ Click on the **Find...** button to select the desired **ASCII Technology File**. This file usually has an extension of `.tf`, `.asc`, or `.tech`. It includes the layer names and usually the GDSII layer number mapping information, as well as some basic DRC rules.

If the ASCII technology file does not contain the mapping information from GDSII layer number to layer name, then it will be in a separate file (usually with a `.map` extension). Specify it as the **GDS Map File**.

- The **Display Reference File** (`.drf`) is optional and is where layer colors and stipple patterns are defined. If not specified, `cds_convert` uses MAX default colors and stipple patterns.

  The MAX default colors use **layer transparency** which can make it easier to see multiple layers on top of each other. Each layer can have a transparency value from 0 to 100 (completely see-through to completely opaque). We suggest that you first do the conversion without the `.drf` file to see how MAX transparent layers look on your designs. You can always change the colors as shown in the above section. If you are used to using stipple patterns for transparency, you may find true transparency easier on the eyes.

- Type the name of the technology in the **Max technology name** text area. If no technology name is specified, **cds_convert** will use the rootname of the ASCII technology file.

- To use this technology file, on the command line, type:

  ```
  maxview -tech <tech_name>
  ```

The technology file gets put in `~/mmi_private/max/tech/<technology name>`. To make the technology accessible to all users, put the technology files in `$MMI_TOOLS/../mmi_local/max/tech`. MAX automatically looks in `$MMI_TOOLS/../mmi_local/max/tech` and then in `...~/mmi_private/max/tech` for technology files.

Step 3 - Since this tutorial does not contain a PDK, click on **Cancel**.

Importing OpenAccess Database Files

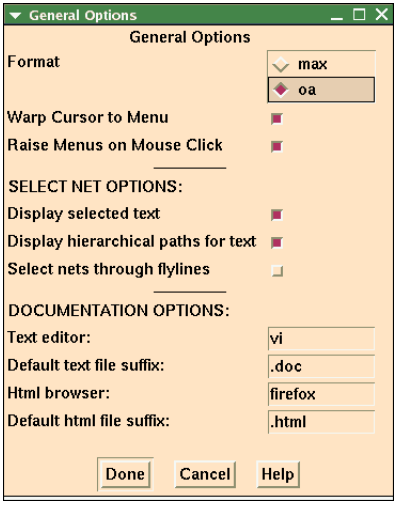
Micro Magic, Inc. tools support the OpenAccess database. MAX can read the technology information from the OA database and create a MAX technology file automatically. For information on this format, please refer to ***www.si2.org***.

IMPORTING AN OPENACCESS FILE

To read in OA databases, the format of the layout files needs to be set **oa**.

Step 1 ⇨ From the **File** menu, select **User Preferences**, and then select **General Setup...**. The following pop-up menu will appear.

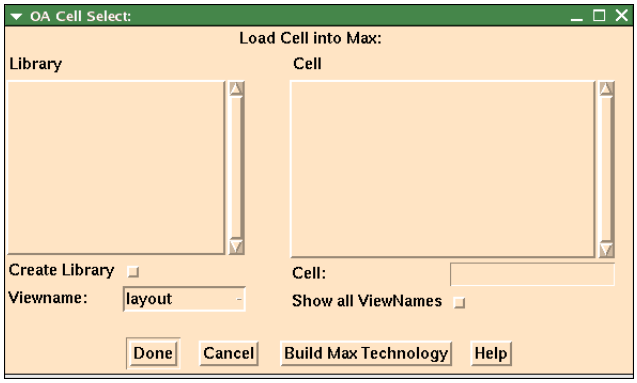
Figure 118:General Options Pop-up Menu: Selecting OpenAccess Format



Step 2 ⇨ Click on the "**oa**" radio button to select OpenAccess format, then click **Done**. You are now in OpenAccess mode.

Step 3 ⇨ From the **File** menu select **Open**, and you will see the following pop-up. From this menu, you may access your OpenAccess library and cells.

Figure 119:OA Cell Selection Menu for OpenAccess Format Files



- If you click on **Build Max Technology**, MAX will get the layer names and other technology information from the OA database file. Note that you can change these assignments afterwards as necessary.
- To have MAX **automatically** start up on OA mode, add the following line to your `max.rc` file:

```
set FILE_MODE oa
```

- Once you have created the technology file from the OA database, start up MAX with:

```
max -tech <tech_name>
```

and MAX will launch using the correct technology and in OA mode.

Changing Hotkeys

With MAX, the user can change the definitions of hotkeys. Not all menu commands have hotkeys, so if you find that you are using a command often that does not have a hotkey, you might want to create one.

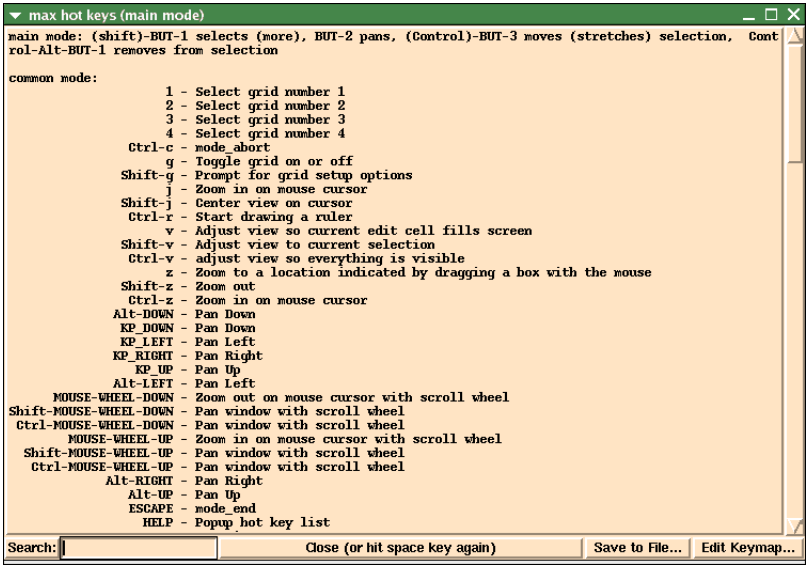
Listing Hotkeys By Command

- If MAX is not currently up, start it by typing "`max`" in a terminal window.

Step 1

- To bring up the current list of hotkeys, type **SPACE** or select **Current Hot Keys...** from the **Help** menu. You should see the form shown in Figure 120.

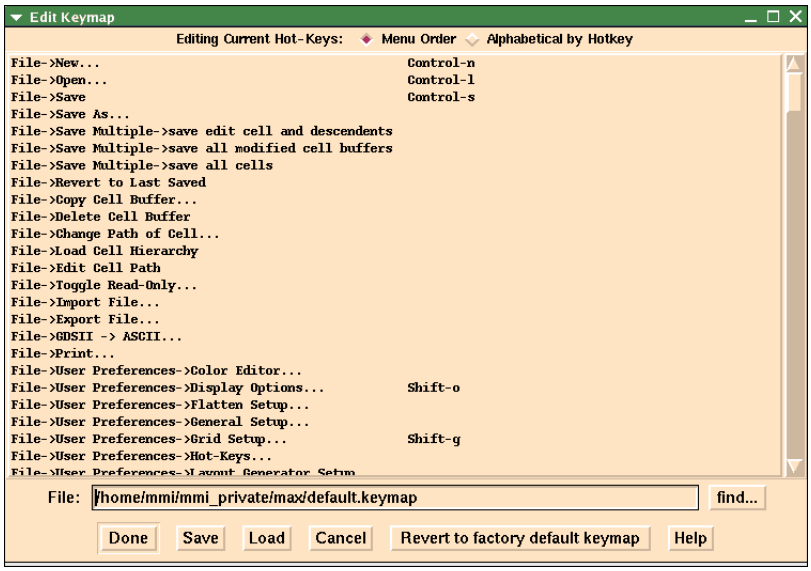
Figure 120:Current Hot Keys Pop-up Menu



This pop-up form is useful if you want to find the hotkey assigned to a command. Most of the hotkeys are also found in the menus, to the right of the menu command. If you want to change or add a hotkey, you need to edit this list.

Step 2 ⇨ Click on **Edit Keymap...** in lower right of pop-up. You should now see a pop-up like Figure 121.

Figure 121:Edit Keymap Pop-up Menu



By default, the hotkeys are ordered by their menu appearance. If you want to edit the hotkey of a specific menu command, this makes it easy to find.

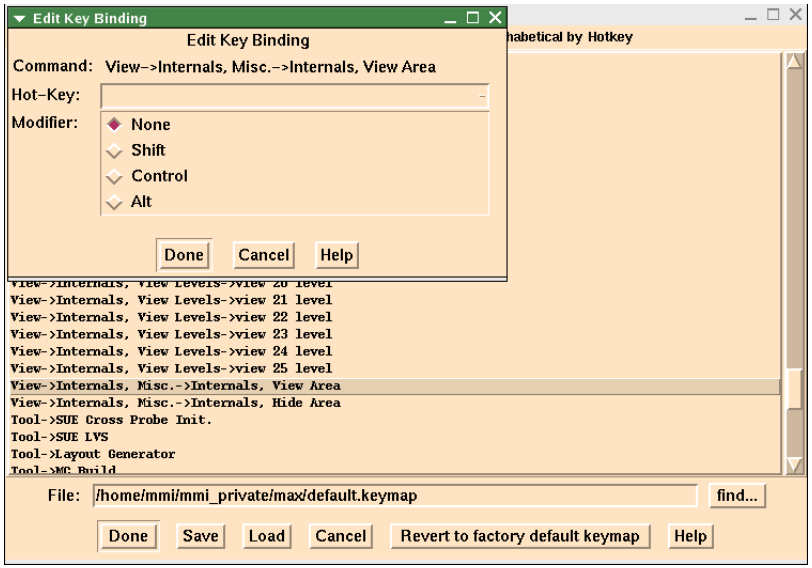
LISTING HOTKEYS ALPHABETICALLY

If you want to find out which hotkeys have already been assigned, you want them listed in alphabetical order.

Step 3 ⇨ Click on the **Alphabetical by Hotkey** toggle button at the top of the pop-up form. Notice that the **Shift-k** hotkey is not currently used.

Step 4 ⇨ Scroll down the list to the bottom and locate **View-> Internals, Misc.-> Internals, View Area**. Click here with the **LEFT mouse button** (**Button-1**) to open the menu shown in Figure 122.

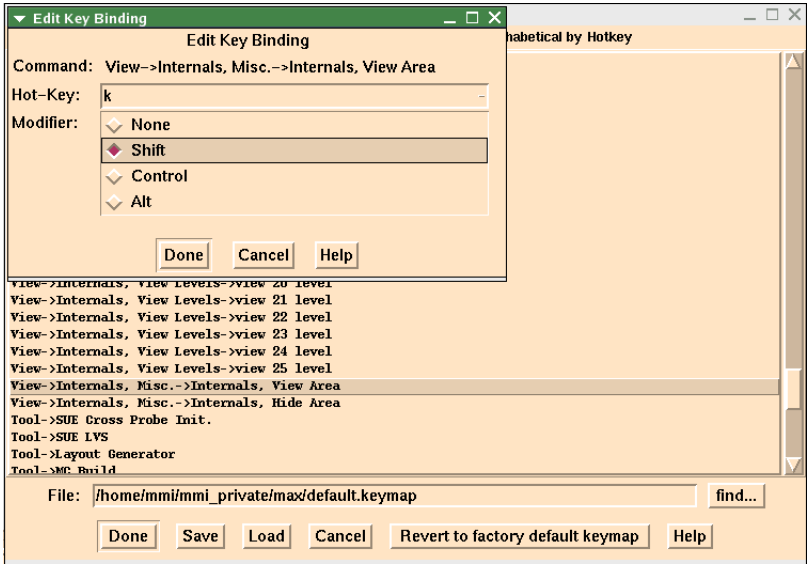
Figure 122:Edit Key Binding Pop-up Menu



RE-MAPPING A HOTKEY BINDING

Step 5 ⇨ Type the letter "**k**" in the **Hot-Key** field. Click on the **Shift** toggle button as shown in Figure 123. This means that the hotkey for this command is now **Shift-k**. Click on **Done**.

Figure 123:Changing Hotkey Binding



You should now see the **Shift-k** hotkey highlighted in the **Edit Keymap** pop-up.

- If you wanted to save this information, you would click on the **Save** button so that this hotkey binding will be set the next time you start MAX.
- You can always return to the original hotkey definitions by clicking on **Revert to factory default keymap.** If you want to test it out or just use it for this session of MAX, click on the **Done** button.

Step 6

- Click on **Done**. Go to the **View** menu and select **Internals, Misc**. Notice that **Internals, View Area** command now has the hotkey **Shift-k** listed.

You can probably see by now that after loading your GDSII files, the ability to customize the display and hotkeys makes using MAX pleasant and easy. We at Micro Magic, Inc. hope that MAX makes loading, viewing, editing, and creating your layout as simple, fast, and painless as possible.

If you have purchased MAX-3D 3-Dimensional Layout Editor, continue on with the special tutorial section for "3D Designs".