User Manual

MAX & MAX-LS
Layout System

Micro Magic, Inc.

Version 5.1.12

Micro Magic, Inc.

Table of Contents

Table of Contents

Table of Contents

Table of Contents

Preface
About This Document

This chapter provides an overview of the *Micro Magic, Inc. MAX and MAX-LS Layout Editor User Manual*:

Objectives

This manual is a detailed guide to the Micro Magic, Inc. MAX, and MAX-LS Layout Editor. Once you are familiar with the tool, this manual will act as a quick reference and how-to guide. The best way to get started with MAX or MAX-LS is to run the MAX Tutorial.

Audience - MAX

This document is designed for layout designers and architects. It assumes that you have a strong foundation in IC design and development, and layout design. This document also assumes that you are familiar with the following programming languages and operating systems:Tcl/Tk, and UNIX or Linux.

Organization

This manual is divided into several chapters. Each chapter deals with a different aspect of the Micro Magic MAX/MAX-LS Layout Editor.

This manual contains the following chapters:

- Preface, “About This Manual” (this chapter), provides a brief description of the content and organization of this manual.
- Chapter 1, “MAX Manual Overview”, introduces the basics of the MAX and MAX-LS Layout Editors.
- Chapter 2, “The MAX Layout Editor”, explains each section of the MAX window as well as covering the basics of how to create layout with MAX.
- Chapter 3, “MAX Commands”, goes through each menu in MAX and describes each command.
- Chapter 4, “The MAX Wire Tool”, covers how to draw wires in MAX.
- Chapter 5, “The Layout Generator and MAX-LS”, goes through how to use the layout generator, which is part of the MAX-LS Layout System, to generate schematic-driven layout.
- Chapter 6, “MAX Technology Targeting”, covers indetail how to create MAX technology files.
- Appendix A, “MAX Hotkeys”, references the hotkeys available in each mode in MAX.

- Appendix B, “MAX Text Commands”, points you to the documentation on the text commands available in MAX.
- Appendix C, “Tcl/Tk and the MAX API”, covers the basics of the MAX API, including an example of a gcell program.

Chapter Organization

Each chapter contains descriptions of the commands and methodology necessary to create layout using MAX, with step-by-step, illustrative examples describing the usage of commands and procedures for generating schematics.

Documentation Conventions

General Conventions

This guide uses the following text conventions:

- Statements, commands, command output, filenames, directory names, and configurations are shown in a boldface, fixed-width font. The following example shows a full path name:

```
~/mmi_private/max/.maxrc
```

- In examples, text that you type literally is shown in bold. In the following example, you type the word *max*:

  To launch max, type **max**

- Menus, menu options, and pop-up menus are generally shown in a boldface, sans serif font. For example:

  Select `Save As` from the `File` Menu.

Conventions for Software Commands and Statements

When describing the Micro Magic, Inc. software, this guide uses the following type and presentation conventions:

- Statement or command names that you type literally are shown nonitalicized. In the following example, the statement name is *set MC(default_generator)*:

```
set MC(default_generator) sram
```

- Variables for which you substitute appropriate values, are shown in itallics. Optional commands within a string of code are often enclosed within square brackets ([ ]). Variables are enclosed within angle brackets (<>). When you type the **`setMACRO`** statement, you substitute a value for *name*, *row1* and *row2*. *CELL* is an optional command.

```
set MACRO(<name>) {[CELL] <row1> <row2> ...}
```

Conventions for Mousing

Throughout this document, we assume you are using a 3-button mouse. When we refer to using the mouse, the following settings apply:

- Mouse-Button-1, or Button-1, refers to the left-most button (index finger) when you are using a right-handed mouse.
- Mouse-Button-2, or Button-2, refers to the middle, or center button (middle finger).
- Mouse-Button-3, or Button-3, refers to the right-most button (ring finger) when using a right-handed mouse.

  Reverse these settings if you are using a left-handed mouse, using your mouse control program, so that you click Button-1 with your index finger, Button-2 with your middle finger, and Button-3 with your ring finger.
- "Click" means to depress the designated mouse button once.
- "Double-click" means to depress the designated mouse button twice, quickly.
- "Drag" means to depress and hold down the designated mouse button while moving the mouse.

Documentation Feedback

We are always interested in hearing from our users. Please let us know what you like and do not like about the Micro Magic, Inc. documentation, and let us know of any suggestions you have for improving the documentation. If you find any mistakes or out of date information, please send email to **support@micromagic.com**.

Support

For product problems or technical support issues, contact Micro Magic, Inc. at **support@micromagic.com**.

Getting set up

MAX runs independently from any of the Micro Magic, Inc. tools suite.

First, ensure that the Micro Magic, Inc. software has been installed, and that you have a valid MAX license.

Chapter 1 Overview

Introduction

This manual is for both MAX and MAX-LS. MAX is a full-custom IC layout tool for physical layout of leaf cells, large blocks and complete SoC products. MAX also has a complete programming interface via Tcl/Tk and a well-documented API. Whether you need full-custom layout, cell-assembly, chip-assembly or the ability to write your own generators, MAX is the tool to choose for your physical design needs.

MAX is equally at home in all aspects of physical design, from creating cells for a library, to interacting with place-and-route at the block-level, to assembling an entire chip. While MAX is powerful enough to handle the largest of chips, it's easy to learn and easy to use.

MAX-LS is a layout system that incorporates our best tools for IC physical layout design of leaf cells, large blocks and complete SoC products. MAX-LS features true schematic driven layout design, including interactive cell generation based on LVS and DRC-correct generators, and cross-probing between schematic and layout.

MAX-LS schematic driven layout offers the ability to generate layout that is DRC and LVS correct with devices automatically sized. Based on your schematic, MAX-LS can generate every transistor and show flylines as to how they should be connected. This gives the layout designer complete control, yet assures rapid physical design development.

MAX Features

- Interactive viewing and editing of hierarchical layout
- Continuous DRC feedback during layout
- Hierarchical and incremental DRC
- Interactive connectivity tracing
- Interactive wiring tool
- Generators for layout structures such as nfets or pfets.
- Interfaces to other tools, including schematic capture (for example SUE), and batch DRC and LVS (for example Dracula or Calibre).
- Smart palette for easy control and feedback on layers.
- Reads/writes GDSII.
- Full customization and extension via Tcl/Tk scripting language and API.
- Technology independence via technology description files.
- Optimized for large databases.

MAX-LS Features

- All the features of the MAX layout editor listed above.
- SUE schematic viewing and editing (Refer to the SUE User Manual for details on how to use SUE).
- Layout generator for automatically generating layout from a schematic.
- Cross-probing between layout and schematic.

MAX Basics

MAX uses the idea of *paint* as well as objects. Paint defines the mask geometries: where each mask is opaque and where it is clear. MAX is different from many layout editors in that mask geometries are not maintained as a list of fixed rectangles, wires, etc. as entered by the user, but rather as paint regions which have no memory of the constituent rectangles from which they were originally constructed. Additionally, MAX layers do not always correspond exactly to Mask layers. The `See Mask` command allows you to preview actual masks.

In addition MAX can also do object-based layout using *Gcells* (generator cells), polygons, circles and wirepaths. Gcells can be flattened down to paint for complete editing control.

MAX organizes layout into cells. Each cell, saved in an ASCII file of the same name, contains paint, Gcells, polygons, labels and instances of other cells.

Labels associate text with points or rectangles in the design on a given layer. Labels and text come in flavors and are used to define cell input and output ports, and comments.

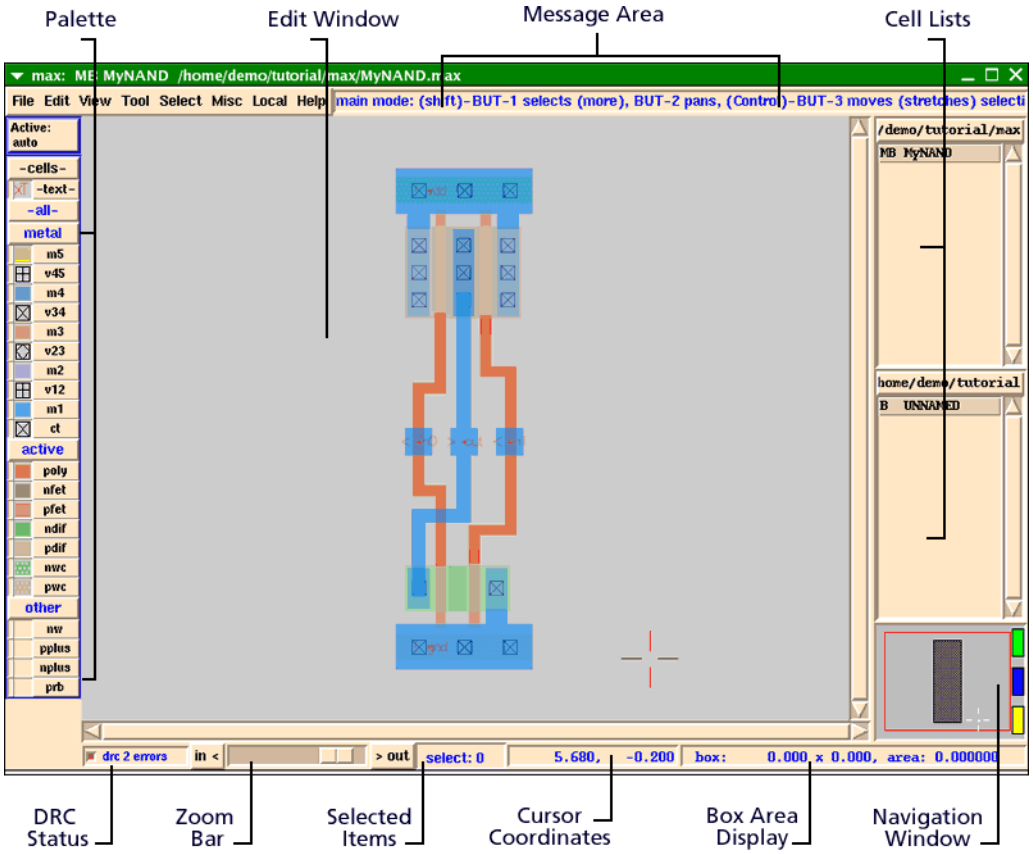
Instances of cells are placed by clicking on the cell names in the lists along the right side of the MAX window (see Figure 1) while holding down the Shift key.

- Clicking on a cell entry without the Shift key changes the view to that cell. If the cell you want is not in the cell lists, you will first have to load it into memory (See Open on page 34). An entire directory of cells can be loaded into MAX by selecting `Autoload directory` at the top of a cell list.
- Instances can be shown in two ways: abstractly (only name, ports and bounding box shown), or with internals displayed (refer to `Internals, View Area` (see page 84)).
- When a cell is loaded, only the abstracts are shown for the instances.

The function of the mouse buttons and hotkeys in MAX vary with the *mode* MAX is in. For a summary of the current function of mouse buttons and hot keys, hit the **Space** bar at any time. Also check the message area to the right of the menu bar for information on the current mode. (See page 151 for a summary of the hotkeys.)

The hotkeys for panning and zooming around, and for controlling the grid, work in all modes, (even when in the middle of dragging something around with a mouse button depressed).

Figure 1: Figure 1: MAX Layout Window



- MAX's *main* mode, mouse **Button-1** (left) is used to select things, **Button-2** (middle) is used to pan, and **Button-3** is used to move selected around. If you are using a scroll-wheel mouse, the **scroll-wheel** is used for zooming in/out as well as scrolling throught the cell list boxes and window slider bars.

In addition to the traditional cursor, MAX has a *box* (with the default color of brown) that is used to specify areas.

- To position the box, in main mode, type **b** and then drag out a box with **Button-1**.
- To paint a rectangle (specify the layer), point to the layer to paint (either in the layout or in the *palette* on the left side of the MAX Window) and type the hotkey **p**.

The `Wire Tool` is a much easier way to create interconnections as opposed to specifying each rectangle. In addition, Gcells shown in the top cell list are an easy way to generate fets and vias. The `Layout Generator` which is part of MAX-LS gives you the ability to quickly generate layout from a schematic. Also as part of MAX-LS, you can cross-probe between layout and schematic.

As you create and modify the layout, you will probably notice white dots appearing from time to time. This is the continuous background DRC's way of complaining. To find out the specific design rule violations behind the white dots, position the box over the region you are interested in and type **Shift-y**, or choose `Explain DRC under Box` (see page 104)

A popular feature in MAX is the ability to quickly trace connectivity. To select an entire net, just position the cursor over a geometry and type the hotkey **s** (See "Select Net" on page 92.). The entire net is highlighted and the labels found on the net are listed in the Message Area.

Where to Get Help

Demonstration

If you are new to MAX, the best place to start is with a demonstration from the friendly folks of Micro Magic, Inc. If you have not already seen a demo, pick up the phone and schedule one now!

Tutorial

The *Micro Magic, Inc. MAX Tutorial* is a step by step, hands on, introduction to MAX. It can be started at any time from the `Help` menu in MAX. It is in HTML form, and will come up in Netscape (or another browser of your choice). The browser used is specified by the **`DEFAULT_BROWSER`** environment variable.

Manual

You are currently reading the *Micro Magic, Inc. MAX User Manual*. The manual is the authoritative reference for MAX. It is available at any time from the `Help` menu in MAX. Like the tutorial, it is in HTML form, and will come up in Netscape (or another browser of your choice). There are also postscript and PDF versions available for printing in **`$MMI_TOOLS/doc/max/max_manual.`**

Menus

For information on commands, browse through the MAX menus. Note that, as you move over menu items with the cursor, on-line descriptions appear in the Message Area directly to the right of the menu bar (see Figure 1).

Hotkeys

Hotkeys are provided for most menu items, and are shown at the right of the menu entries. These hotkeys are available from MAX's main mode. When in submodes, such as when drawing wires or editing labels, other hotkeys are in effect.

A list of currently active hotkeys is always available by clicking on `Hot Keys` in the `Help` menu, (alternately just hit the **Space**-bar). The current functions of the mouse buttons are also listed.

Text Commands

An extensive set of text commands is available for use in scripts. These commands are mainly of interest to developers implementing new menu items, startup scripts, etc. However, text commands can be invoked directly, by typing into the window from which MAX was started.

- For documentation on text commands, click on `Text Commands` (see page 107) in the `Help` menu or type the hotkey **?**. Click on the one line description of a command for more detailed information.

- You can search for commands whose documentation contains selected keywords, by typing them into the Search field at the bottom of the text commands window.

  For example, to search for commands related to saving files, you can enter **`file save`** into the Search field.

Text commands are built on top of **`Tcl/Tk.`** All **`Tcl`** and **`Tk`** commands are available providing a complete scripting language and toolkit for GUI extensions. This version uses **`Tcl/Tk`** 8.4.9 with Micro Magic enhancements. Complete information on **`Tcl/Tk`** is available from the official **`Tcl/Tk`** website at **`http://scriptics.com`**. We also recommend the book *Tcl and the Tk Toolkit*, authored by John Ousterhout, and published by Addison-Weseley.

Micro Magic Documentation Guide

MAX is closely tied to a number of other programs in the Micro Magic Inc. design tools suite. For example, MAX can be used together with SUE Design Manager to simultaneously view a design at schematic and physical levels, with cross-probing between the two. Several utility programs, such as *Ext2sim*, post-process MAX output for use with other programs. For information on these other programs, check out the *Micro Magic, Inc. Documentation Guide*. The guide summarizes all the tools in the Micro Magic design suite and provides pointers to complete documentation.

The *Micro Magic, Inc. Documentation Guide* can be accessed from inside MAX by clicking on `MMI Documentation Guide` in the `Help` menu. Alternately, the program **`mmidoc`** brings up the guide in your favorite browser. Simply type **`mmidoc`** in a shell window and the Micro Magic, Inc. documentation will be brought up in your browser.

The Friendly Folks of Micro Magic, Inc.

If the above sources are insufficient, do not hesitate to contact the Micro Magic group. You can send email to us at: **`support@micromagic.com`**

Chapter 2 MAX Layout Editor

Running MAX

This chapter covers information on how to run the MAX layout editor.

To start MAX, simply use the **max** command as defined below. Each release of MAX comes with a set of generic technology files, so you can get started entering layout using one of these technologies. The **gds_input** (page 125) command gives the user the ability to easily read in GDSII files without an existing technology file.

MAX Command Line Description

Synopsis

```
max [-switch value] [[cell_name1, cell_name2,...] | [gds_file]]
```

Description

This launches the MAX Layout Editor. The **-tech** option is used to specify which technology will be used. Numerous cells can be specified to load up at runtime, or they can be opened after MAX is started. If a cell name is specified, MAX starts with the technology specified in that MAX file. Below is a list of the switches for the **max** command.

Table 1: List Of Switch Options For MAX Command

Option	Description
-tech tech_name	Specify the technology which will be used. The default technology is used if –**tech** is not specified. It is defined by the environment variable **MAX_DEFAULT_TECH**. If a MAX cell is specified on the command line, the technology of that cell overrides **MAX_DEFAULT_TECH**. If the technology specified on the command line does not match the technology of the MAX file, a warning is printed and the cell is not loaded.
-h | -help	Prints out syntax of the **max** command
-v | -version	Print the version of MAX and exit.
-geometry XxY+T+Z	Start MAX with the given X and Y size at the given T and Z coordinates.
-colormap new	Start MAX with a private colormap
-iconify 1	Start up MAX iconified
-batch 1	Run MAX in "**batch**" mode, which means no pop-ups or confirmations required.
-new <cell>	Bring up MAX with a new cell.
-command <tcl_cmd>	After the **.maxrc** files have been read in, execute the Tcl command at startup.
-set <var>=<value>	Set a global variable at startup. Variables can also be set in a **.maxrc** file.
cell_name1,...	Specify the cells to open while loading MAX. All cells must have been laid out with the same technology.
gds_file	If a GDSII file is specified, MAX will first translate the GDSII file to MAX format. If the extension of the file is anything other than **.max**, it is assumed that it is a GDSII file. Specifying a GDSII file on the command line requires that there is already a technology file created for this cell.

Environment Variables

The following are the MAX environment variables.

MMI_TOOLS

The environment variable **MMI_TOOLS** must be set to the directory where the MMI tools were installed. All Micro Magic, Inc. software is installed under a single directory.

Example:

```
setenv MMI_TOOLS /tools/mmi
```

MMI_LOCAL

The environment variable **MMI_LOCAL** can be used to specify the location of the **mmi_local** directory. If this variable is not set, MAX looks for the **mmi_local** directory in the default location **$MMI_TOOLS/../mmi_local**.

The **mmi_local** directory is where site or company specific information is stored. This is where the technology files should be located. Any settings specified in **mmi_local** for hotkeys, colors, etc. override the default settings in MAX.

```
setenv MMI_LOCAL /tools/mmi_local
```

MMI_PRIVATE

The environment variable **MMI_PRIVATE** can be used to specify the location of the **mmi_private** directory. If this variable is not set, MAX looks for the **mmi_private** directory in the default location directly under you home directory at **~/mmi_private**. The **mmi_private** directory is where user specific information is stored. You may have your own hotkey and color settings.

- Any settings specified in **mmi_private** override the default settings for MAX and the settings defined in the **mmi_local** directory.

```
setenv MMI_PRIVATE ~/mmi_private
```

MAX_DEFAULT_TECH

If **MAX_DEFAULT_TECH** is set, MAX defaults to this technology. The **-tech** option overrides the environment variable. the default technology for MAX is set to mmi25 which is a generic technology file provided with MAX.

```
setenv MAX_DEFAULT_TECH mmi25
```

MMI_BROWSER

This determines which web browser is used when bringing up documentation or tutorials.

```
setenv DEFAULT_BROWSER netscape
```

MMI_LICENSE_FILE

This specifies the location of the MMI license file. If not set, MAX looks for the license file in **$MMI_TOOLS/../mmi_local/mmi_license.lic** or in **$MMI_LOCAL/mmi_license.lic**.

```
setenv MMI_LICENSE_FILE $MMI_LOCAL/mmi_license.lic
```

MMI_EDITOR

This environment variable is used mainly in SUE. If the tool needs to bring something up in a text editor, it will use the editor specified by **MMI_EDITOR**.

MAX_PROBE_DISPLAY

MAX_PROBE_DISPLAY is used with MAX-LS. It specifies which display MAX will attempt to bring up SUE in for crossprobing.

The .maxrc File

The **.maxrc** file contains commands which are added to `Local` or `Tool` menus, define variables, and execute text commands. In addition, Tcl programs can be defined or sourced in the **.maxrc** file. Below is an example of a **.maxrc** file which sets the variable **ZOOM_BUTTONS** (See "Zoom Buttons" on page 19) then sources a Tcl script, adds a command to the `Local` menu and turns off the interactive DRC on startup.

The file can be named **.maxrc** or **max.rc.**

```
set ZOOM_BUTTONS "green yellow purple orange blue"
source $MMI_LOCAL/max/tree.tcl
menu_local_cmd "Print hierarchy tree" print_tree
pal_special_off drc
```

There are a number of locations where MAX looks for a **.maxrc** file. Below are the locations in the order in which they are sourced. The last **.maxrc** file found (usually the one in the current directory) will override settings from the previous files.

```
$MMI_TOOLS/max/.maxrc
$MMI_TOOLS/../mmi_local/max/.maxrc
~/mmi_private/max/.maxrc
~/.maxrc (home directory)
./.maxrc (current directory)
```

The MAX subdirectory can contain a version number, allowing you to run different versions of the software. For example:

```
$MMI_TOOLS/../mmi_local/max5.0/.maxrc
$MMI_TOOLS/../mmi_local/max/.maxrc
~/mmi_private/max/.maxrc
```

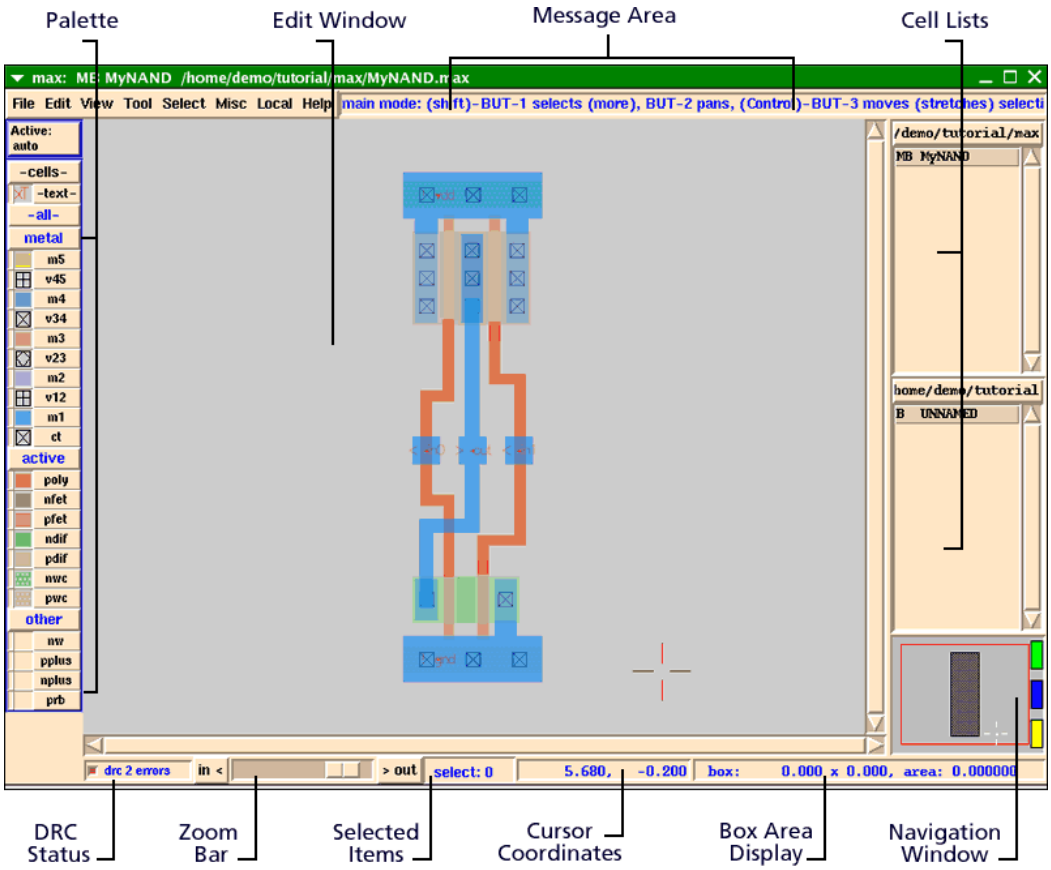
If you are running MAX version 5.0 or later, MAX will get the **.maxrc** files from **mmi_local/max5.0** and **mmi_private/max.** If you are running a version of MAX earlier than 5.0 (for example, MAX 3.2), MAX will get the **.maxrc** files from **mmi_local/max** and **mmi_private/max**. MAX looks for a **max** directory with the latest version not exceeding the version of MAX which is being run. For versions before MAX 3.0, MAX only looks at the **max** directories, no version numbers

Remember, you can name your file **max.rc** as well as **.maxrc**.

Tour of MAX Window Elements

When MAX starts up, it creates a new main window. The main window has a large grey (default color) area for displaying layout in the center, surrounded by various other elements. (See Figure 2). The purpose and use of these other elements is described below. Most of the elements are optional and can be removed (see “Display Options” on page 42) to increase the area available for displaying layout.

Figure 2: The MAX Window



Title Bar

The window title bar (displayed at the top of the window by most window managers) identifies the window as a MAX window and gives the name of the cell loaded into the window. If an edit-in-place is currently in progress, the cell being edited is also identified.

Menu Bar and Message Area

At the top of the window, normally just below the title bar, are the *Menu Bar* (on the left) and *Message Area* (on the right). The menus are scanned, and menu items are invoked, by clicking the menu with **Button-1** and then clicking on the menu item. You can also select menu options by holding down **Button-1**, dragging down to highlight the desired item, and then releasing.

You can *tear off* a menu, and place it in a permanent window of its own. To do so, just click on the dashed "tear line" at the top of the menu.

Message Area

Note that when the cursor is over a menu item, a short description of it appears in the Message Area to the right of the menus. When the cursor is not over a menu item, the Message Area normally displays information on the current mode and function of the mouse buttons. Occasionally the Message Area is used to display other useful information, such as the highlighted DRC error when stepping through DRC errors.

Palette

The MAX palette, located along the left border of the main MAX window, displays the available layers, visibility, selection status, and cursed layers (layers under the cursor). Furthermore, layer visibility and/or selectability is changed on a layer by layer basis, by groups of layers or by all layers. Also, the selectability of subcells and Gcells is changed by clicking on the box labeled `-cells-`. Refer to Figure 3 for details.

Active Layer

The top button in the palette specifies the *Active Layer*. The default setting is `auto`, used for the Wiring Tool on page 27. The Active Layer controls what layer polygons and circles are drawn in. It also controls the layer wires start in.

Layers Under Cursor

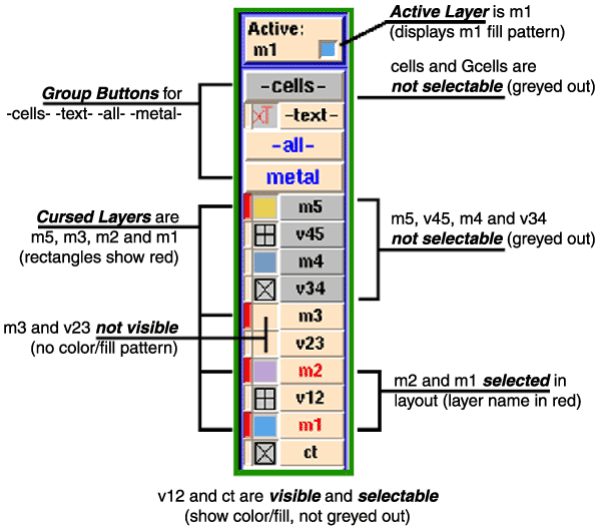
From left to right, for each layer, the palette contains a small rectangle that turns red when the cursor is over that layer in the Layout Window — known as a *cursed layer*. The layer does not need to be visible to be cursed. However, layers in unexpanded subcells (cells for which internals are not visible) are not cursed.

Layer Visibility

The next rectangle displays the *visibility* of the layer. If this layer is visible, the rectangle will contain the fill pattern and color of the layer. Toggle layer visibility on or off by clicking on the rectangle with **Button-1**.

- To toggle on or off **visibility** of groups of layers, click with **Button-1** on the *group name* (for example: `-active-`). (Refer to Figure 3, below.)

Figure 3: Regions of the MAX Palette



Layer Selectability

The right-most rectangle shows the layer name and indicates **layer selectability** and the **selected layer.** All layers start out being *selectable*. Click **Button-1** on this rectangle to "grey out" the rectangle, and the layer is no longer selectable. When you drag out a selection in the main MAX window, only those layers both visible AND selectable will be selected.

If the layer is selected anywhere in the MAX layout (it does not have to be in the visible window), the text will turn red. Otherwise it is black. Note that it is possible to be selected (displaying red text) but with the selectability off. For example, when you select a wire and trace its connectivity, you may select geometries on layers that are not selectable.

Painting and Erasing Layers

In addition to selecting layers, in the Main Mode you can also paint and erase layers.

- Clicking the paint hotkey **p** over either the visibility or the selectability area will ***paint*** that layer into the box in the Layout Window.
- Clicking the erase hotkey **o** will ***erase*** that layer from inside the box in the edit cell.

Groups of Layers

You can control the **visibility** and **selectability** on a layer-by-layer basis, as described above, or by *groups* or *all layers*.

The **group buttons** (`metal, active, other`) control all of the layers below them until the next group button.

The **group buttons** (**`-cells-, -all-, -text-`**) control the visibility and selectability of cells, text, or all layers.

- Clicking **Button-1** on a group button toggles the *visibility* of all the layers in the group. If any of the layer visibilities are off, then all the layers will be turned on. Otherwise they will all be turned off.
- Clicking **Button-3** on the group button toggles the *selectability* of each of the layers in the group in the same way as with the visibility.
- The **`-all-`** button has the same functionality as the group button but controls **all** layers — a kind of super-group.
- The **`-text-`** button has the same functionality as the layer buttons but controls the selectabililty and visibility of text.
- The **`-cells-`** button controls the selectabililty of cells instances

Groups of layers can be hidden and restored in the palette by clicking with **Button-2** (middle mouse button) on the group button. This is useful if the palette is cluttered with infrequently used layers. Note that all layers are displayed on startup.

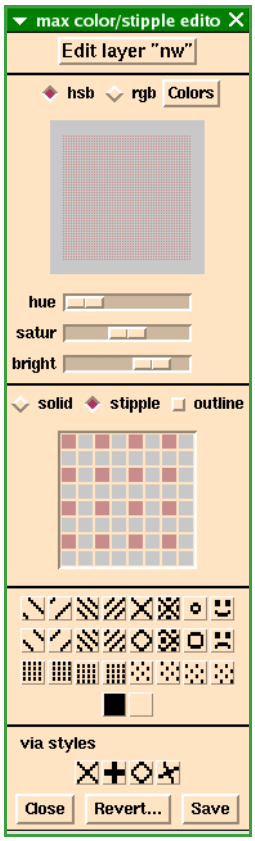
Changing Color and Fill Patterns of Layers

To edit the color, fill pattern, and outline style of any layer, click with **Button-2 or Button-3** over that layer's palette entry (either the layer name or color/fill square). This brings up a `color/stipple editor,` as shown in Figure 4.

If the editor is already open, just change the layer to the desired layer by selecting the layer in the `Edit layer` popup, or by clicking with **Button-2 or Button-3** on the desired layer.

Note that if the editor is obscured by other windows or iconified, this will deiconify or raise it.

Figure 4: MAX Color/Stipple Editor



Edit Layer

By holding down **Button-1** over the `Edit Layer` button, a list of available layers appears. You can select the layer you wish to edit this way or by clicking **Button-3** on the layer in the palette. To change the color and stipple patterns of *special layers* (background, grid, etc.), you must select the layer through this pop-up.

Editing Special Layers

Contained within the `color/stipple editor` layer selection is a list of special layers that you can modify. These include:

- the *background color*,
- the *grid color*,
- the *label* or *text color*,
- the *highlight color and stipple*,
- the *online DRC feedback color and stipple*, and
- the *bounding box color*.

Simply click with **Button-1** on the desired special layer to edit it. Note that, other than background, you will not see the current color of the special layer in the top color window. Instead you must look at the stipple window or the MAX Layout Window to see the effect.

For example, turn on the grid in the MAX Layout Window before trying to change the grid color.

- **annotation** — The color of objects created with the API commands **`lay_line`** and **`lay_rect`**. Examples are the ruler and the edit lines/vertices when editing a polygon.
- **background** — The background color for the MAX Edit Window.
- **cell bbox/text** — The color of the outline of an instance (bounding box), instance type, and instance name for which no internals are viewed (unexpanded instances). If the instance is selected, the highlight color is used.
- **box** — The color of the box (See "The Box" on page 25)
- **drc** — The color and stipple pattern for the interactive DRC errors.
- **feedback** — The feedback layer is used for DRC errors from an external tool. It is also used for displaying the mask layers. (See "See Mask" on page 80)
- **flyline** — The color of flylines used by the layout generator (See "Introduction To The MAX Layout Generator" on page 117).
- **grid** — The color of the grid.
- **label** — The color of labels or text when not highlighted.
- **selection** — The color and stipple pattern for items which have been selected.

Layer Color

From top to bottom, the `color/stipple editor` lets you edit the color of the current layer. Simply change the hue, saturation, and brightness (HSB) sliders or RGB (red, green, blue) sliders.

- The hue is essentially a color wheel. Choose the desired color with the hue slider.
- Next change the saturation and brightness if needed. There is also a color select list which gives you a fixed set of colors to start from. To select a predefined color, click on `Colors` and select the desired color.
- If you would rather specify colors with RGB, then click on the `RGB` toggle and move the sliders to the desired locations.
- There is also a color select list which gives you a fixed set of colors to start from. To select a predefined color, click on `Colors` and select the desired color.

Because any changes you make in the editor affect the MAX Layout Window in real time and cause MAX to repaint the screen, the editor may appear sluggish if you are viewing a large layout. If this occurs, simply zoom in on a small part of the layout and the performance will improve.

Layer Fill and Outline

Next in the `color/stipple editor`, you can select whether the fill for a layer is *solid* or *stippled*.

- **Solid layers** appear solid in the layout but are also transparent to other layers. Solid layers are the easiest to see and it is suggested that all common layers be made solid, such as diffusion, poly, and metal layers (but not vias). Solid layers do not have stipple patterns nor outlines.
- **Stippled layers** are suggested for vias and less common layers like n-well and pplus. Stipples can be *outlined*.
  - You can edit the stipple pattern of a stipple by clicking with **Button-1** on the squares of the stipple to toggle them on or off.
  - You can also click with **Button-1** on any of the provided stipple patterns to set the current stipple.
  - Selecting the `outline` button will draw a single pixel outline around any region with that stipple pattern.
- **Via Styles**
  - If you are changing the color of vias, you can also change the **via style**. Often times vias don't use stipple patterns, but have a simple "x" or "+" in it. MAX provides you with four different via styles.

Palette File

The bottom of the `color/stipple editor` provides a `file find...` bar and the buttons `Close`, `Revert...`, `Load` and `Save`.

`find...` allows you to specify a file name to load or save which contains the layer color and stipple patterns.

The default location for this file is:

`~/mmi_private/max/tech/<tech name>/<tech name>override.`

If you save your color and stipple information in this file, it will be automatically loaded when you start MAX with the same technology.

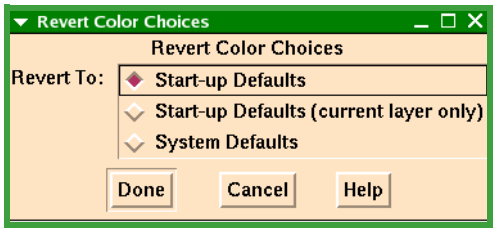
`Close` closes the `color/stipple editor` without saving edits.

`Revert...` brings up the pop-up shown in Figure 5, giving several options to discard all or only the current layer colors/stipples from the last saved version (usually the same version that you started the MAX session with)., and to

`Load` loads the palette file specified in the `find...` bar. This is useful if you want to load your color and stipple patterns while running MAX in a different account.

`Save` saves the current colors/stipples so that they are the default for future MAX sessions.

Figure 5: Revert Options



Saving the Palette and the `.override` file

The palette layers, groups, and colors/stipples are saved in a file called **`<tech>.palette`** where **`tech`** is the name of the technology (for example, **`mmi25`**). This file is created when the technology files are made using **`make_tech`** (see Chapter 6, "MAX Technology Targeting").

- When you click `Save`, MAX will attempt to save a local copy of the palette color and stipple patterns into the file specified in the `find...` bar. By default the file is stored in::

  `~/mmi_private/max/tech/<tech>/<tech>.override`

- If you do not have this directory, MAX will create the appropriate directory in UNIX. If you want these palette options to affect all other users, you can UNIX move it to:

  `${MMI_TOOLS}/../mmi_local/max/tech/<tech>/<tech>.override`

- The **`<tech>`**.override file only changes the color and stipple patterns of layers. It does not change the grouping or layer order in the palette. This information is specified in the **`<tech>.palette`** file and is only loaded when MAX is invoked.

- The order of the layers in the **`<tech>.palette`** file determines the order of layers in the palette in MAX. You also edit the palette file to modify the placement and names of **groups**.

  - The **`.override`** file is an ASCII file that you can edit directly. Colors are specified as RGB triplets, and stipples as 1's and 0's.

  - If you want to rearrange layer orders or add/subtract/modify groups, you must presently do it in the palette file. You can then re-launch MAX to view the results.

  - The **`SAMPLE_STIPPLES`** variable allows you to change the default stipples showing in the color palette. Refer to "Text Commands" on page 107 for more information.

If you do edit the **`.palette`** file or the **`.override`** file directly, be careful to preserve the syntax or else MAX won't be able to read it.

Cell Lists

Along the right side of the window are lists containing names of cells currently loaded into memory. Each list pertains to cells in a particular UNIX directory.

- The top cell list is generally the `Gcells` (page 25) provided with MAX. The number of lists, and the directories they display, can be controlled by clicking on the directory names at the top of the lists.
- To customize the list boxes that display loaded MAX cells, use the API command **`list_box_configure.`** (Refer to Appendix C, "TCL/TK and The MAX API" for information about API commands.)

Clicking on a cell name (except for Gcells) with **Shift-Button-1** causes that cell to be displayed in the Layout Window.

- **Button-1** puts an *instance* of that cell in the Layout Window. You can position the instance by moving the mouse using **Button-1** or **Button-3** to place the instance.
- Clicking with **Button-1** on a Gcell brings up the Gcell `Edit Properties` (page 67) form.
  - An "`M`" to the left of a cell indicates that it has been ***modified*** since last saved to disk.
  - An "`R`" indicates that the disk file is ***read only***.
  - An "**`S`**" indicates that the cell has been ***saved*** to disk.
  - A "**`B`**" indicates that the cell resides only in the ***memory buffer*** and has never been saved to disk.
  - A "`G`" indicates that the cell is a ***gcell***.

The Navigator Window

If you have zoomed in on a portion of a cell, the Navigator Window in the lower right corner shows you a rectangle representing the entire cell.

- A red box represents the zoomed in area.
- The white box in the Navigator Window shows the current location of `The Box` (page 25).
  - Clicking once with **Button-2** or **Button-3** in the Navigator Window moves you to that region in the edit window.
  - Drag out an area in the Navigator Window with **Button-1** to zoom to a specified region of the cell in the edit window.

Zoom Buttons

On the right side of the Navigator Window are three *zoom buttons*. The top zoom button (green by default) lets you go backward and forward in the zoom stack.

- Clicking **Button-1** on this top button lets you step backwards one view at a time through your previous views.
- Clicking **Button-3** on this top zoom button lets you step forwards one view at a time.

  This only works if you have already gone backward to previous views.
- The other zoom buttons allow you to *save a specific view* (for example, if you have zoomed into a specific area of the layout) and return to that view later.

- To set the zoom buttons:
  - First zoom in on a desired area of the layout.
  - Then click on a zoom button (other than the top button) with **Button-2** or **Button-3**. Notice that you now see a box — of the same color as that button in the Navigator Window — indicating the location of this view.
  - To get back to a view, click on the desired zoom button with **Button-1**.
  - To customize the Navigator Window, you can set **`ZOOM_BUTTONS`** in the **`.maxrc`** file as shown below.

  In this example, we will add four additional zoom buttons, and change the top zoom button to yellow, with the command:

  ```
  set ZOOM_BUTTONS "yellow green purple orange blue"
  ```

  This command can be typed into the Command Window to change the zoom buttons on the fly or can be added to a **`.maxrc`** file. (See "Creating and Editing Layout with MAX" on page 22).

The Bottom Bar

The bar at the bottom of the window holds four elements:

- DRC Status Area
- Zoom Bar
- Selected Area
- Cursor Coordinates
- Box Area Display

DRC Status Area

The *DRC Status Area* is at the left end of the bottom bar. The messages it displays and their meaning are described below. When active, the background DRC runs continuously and DRC errors are displayed as white dotted areas.

- For an explanation of a DRC error, drag a box around the error with **Button-1** and select `Explain DRC under Box` (page 104) from the `Misc` menu.

- If only one DRC error occurs under the box, that single error is displayed in the Message Area.

  Otherwise, the DRC errors are listed in the Command Window (the shell window from which you started MAX).

- The `DRC Find Next Error` (page 103) command, also in the `Misc` menu, can be used to step through all current DRC errors. The text explanation of the DRC error is displayed in the Message Area.

drc off

The DRC processing is turned off. ***When a large GDSII file is first read in, MAX runs DRC on the entire layout in the background if DRC is turned on***. "`drc off`" means that the interactive DRC is not running.

- The visibility of DRC errors can be toggled by clicking on the `drc` button to the left of the DRC Status Area. This also toggles the DRC on/off

  In other words, making DRC errors visible automatically turns the DRC on, and making errors invisible turns it off.

drc busy

The background DRC is on and running, and has unfinished business.

drc clean

The DRC is up-to-date and there are no MAX DRC errors.

drc n errors

The DRC is up-to-date and there are *n* DRC errors.

drc >10 errors

The DRC is up-to-date and there are more than 10 DRC errors.

Zoom Bar

The *Zoom Bar* is located in the middle of the bottom bar. It works like a scroll bar, except that it zooms in or out, rather than panning over.

Selected Area

The *Selected Area* is located to the right of the Zoom Bar. This area indicates how many things are selected. By default, if more than 100 things are selected, it displays ">100".

- You can change this number with the variable **`SELECT_MAX_DISPLAY`**. By default, everything is counted when displaying the number.

- To change this, you can use the variable **SELECT_DISPLAY**. For example, if you want only cells or text counted in the Selected Area, type the following command in the Command Window or add it to a **.maxrc** file.

```
set SELECT_DISPLAY "cells text"
```

If you click on this area with **Button-1**, the Selection Probe window appears, showing details of what is selected.

Cursor Coordinates

To the right of the Selected Area is an area displaying the *Cursor Coordinates*. The coordinates are for the cell which was opened.

- If you do an Edit Cell or Object in Place (page 80), the coordinates displayed will be for the original (top level) cell.
- Clicking on the Cursor Coordinates area with **Button-1** opens a pop-up form allowing you to specify exact coordinates of where to move the cursor.

Box Area Display

The *Box Area Display* at the right end of the bottom bar gives the current box dimensions as width (dx) *x* height (dy) in microns. Also displayed is the area of the box.

- You can change what is displayed in the Box Area Display by clicking in Box Area with **Button-1**.
- You change what is viewed by editing "Display on status bar, box:". This pop-up form also appears when you select Box Dimensions from the Misc menu.
- The Box Area can be used to "measure" objects by selecting them. (When you select something the box is automatically placed around it.)
- The Box Area can also be used to measure the space between things, using the hotkey **m**. The hotkey **m** is for the Measure (page 96) command in the Misc menu.

You really need to use the hotkey, since you need the cursor positioned at the space you are measuring when issuing this command.

Scroll Bars

Scroll bars for the layout area allow you to pan vertically or horizontally through the layout. There are also scroll bars for the cell lists.

Creating and Editing Layout with MAX

Command Window

The *shell window* MAX is started from is also of interest during the operation of MAX. Informational messages are posted to this window.

- For example, the results of the `Explain DRC under Box` (page 104) command are displayed in the command window.

- In addition, text commands, or even entire Tcl scripts can be typed into the Command Window (See "Text Commands" on page 107). Text commands are of use primarily to developers, but are occasionally useful to end-users who wish to access (obscure) features not available from the menus.

Reading In A GDSII File

In order to read in an existing GDSII file, you must first create a basic MAX technology file which defines the layers in the GDSII input. If there is no existing technology file, use the `gds_input` (page 131) command to create a basic technology file by looking at what layers exist in the GDSII file.

This technology file will contain only the layer definitions with no DRC rules, no layer connectivity, and no definitions for Gcells. It is useful for quickly viewing a GDSII file or as a starting point for a new technology file.

The `Import File` (page 34) command (under the `File` menu) assumes that you have a technology file.

Creating A MAX Technology File

MAX comes with generic technology files (**`mmi25`** and **`mmi18`**) for a 0.25 µm and 0.18µm process. These can be used as a starting point for creating new technologies. A technology for MAX is defined in the technology source file. The `make_tech` (page 129) program is then used to convert the technology source file into the technology files used by MAX.

If you have a *Dracula* DRC deck for the technology, the `drac_convert` (page 134) program can be used to create a MAX technology source file.

Please refer to Chapter 6, "MAX Technology Targeting", for detailed information on `gds_input`, MAX technology source files, `make_tech` and `drac_convert`.

Basic MAX Usage

This section of the manual gives a quick overview of the basic things you can do in MAX.

Refer to Chapter 3, "MAX Commands" for a complete description of all MAX commands. A great source of information on the basics of using MAX is the Micro Magic, Inc. MAX Tutorial.

- Typing **Ctrl-c** aborts out of the current mode, undoing any changes and returning to the Main Mode. As an example, in Wire Mode, **Ctrl-c** will abort the wire you are working on and return you to the Main Mode.

- Typing **ESC** ends the current mode and returns to the Main Mode. For example, if you are working in Wire Mode, **ESC** ends the wire you are working on and returns you to the Main Mode.

Moving Around the Layout

Zoom, Pan, Scroll Bars, Zoom Window

Once you have loaded a layout in MAX, there are many ways of moving around in the layout.

- Use the command `Zoom to Area` (page 82) (hotkey: **z**) to specify the region to zoom in on. You type the hotkey **z** and then drag out the region to zoom to with **Button-1**.
- Use the command `Zoom to Fit Selected` (page 82) (hotkey: **Shift-v**) to zoom in to fit what is selected.
- The `Zoom Out` (page 82) command (hotkey: **shift-z**) allows you to zoom out by a fixed amount.
- The `Zoom In on Cursor` command (hotkey: **j**) allows you to zoom in by a fixed amount, centered on the cursor.
- Use the **scroll-wheel** on the mouse to zoom in and out on the layout. The zoom area will be centered around the point of the cursor.
- Use the scroll bars on the bottom and right of the Layout Window to pan around the layout.
- In addition, you can use the `The Navigator Window` (page 19) to move around the layout.

Viewing Internals of Cells

When a cell is first loaded into MAX, only the top level of hierarchy is visible. The internals of instances are not visible.

- To view all internals of all cells, use the `Internals, View Area` (page 84) command (hotkey: **i**) to view the internals.
- To view internals of only selected cells, first select the cells (`Select cell`, hotkey: **f**) (`Select additional cell`, hotkey: **Shift-f**), and then use the `Internals, View Cell` command (hotkey: **Shift-i**).
- You can **hide internals of all** cells (hotkey: **h**) or **hide internals of selected** cells (hotkey: **Shift-h**).

Viewing Layers

The palette on the left of the layout area allows you to turn on/off the visibility of individual layers or groups of layers.

- Clicking once with mouse **Button-1** (left mouse button) on an *individual layer* (the square with the layer color and fill) toggles the visibility of that layer on or off.
- Clicking once with mouse **Button-1** on one of the *group names* (for example, `active`) toggles that group of layers on or off.

Refer to the Palette (page 12) section for more information.

Drawing/Painting Layout

The Box

MAX makes extensive use of a rectangular box, displayed in brown (by default) on the screen. *The Box* is a sort of second cursor that designates an *area* rather than a point. Many MAX commands work on the area defined by the Box.

> For example, the normal way to add a rectangle on a layer is to place the Box where the rectangle is to go, and then click with **Button-3** (right mouse button) over a sample of the layer you wish to draw.

You can also draw polygons and circles (See "Add Polygon" on page 59) with MAX.

To paint a rectangle on a mask layer in MAX,

- Move the Box to the rectangle you want to paint (See "Make/move Box" on page 94), and then click over a sample of the layer you wish to paint with mouse **Button-3** (in Main mode).
- If there is no sample handy in the Layout Window, you can click over a palette entry instead with mouse **Button-3**.
- Notice that multiple layers can be painted at once, and that only layers that are currently visible (turned on in the palette) get painted.
- You can also use **Button-3** to erase all mask layers under the Box, by clicking on empty space; or click **Ctrl-Button-3** over a layer to erase just that layer.
- To change the layer of a rectangle:
  - First select the rectangle,
  - Erase that layer by clicking over empty space and then
  - Select a new layer with **Button-3**.
- You can also use the Edit Properties (page 67) command (hotkey: **p**) to change the layer of a rectangle.

Gcells

Generator cells or *Gcells* are parameterized cells that regenerate themselves when they get different inputs. Devices (such as fets) and vias are good candidates for Gcells since a given layout may have multiple types that differ only by size or some other attribute.

Gcells are user definable using the Tcl scripting language. Presently they must be created and modified using a text editor. To discern Gcells from conventional MAX cells, Gcells are saved in **.maxg** files as opposed to **.max** files.

Refer to Gcells Tcl Programs (page 155) for information on creating Gcells.

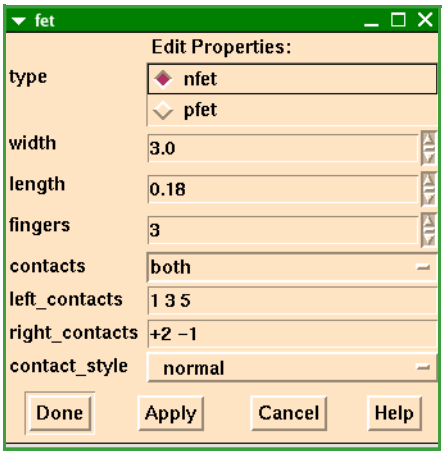
- To insert a Gcell into the layout, click on it with **Button-1** (left mouse button) in the cell list on the right side of the MAX window. This will bring up the Gcell `Edit Properties` form (See Figure 6).

Currently MAX comes with fet and via generators. The `Edit Properties` form for a fet allows you to change the width, length, number of fingers and location of the contacts.

- Once you have specified the properties, click on `Done`.
- You can rotate (hotkey: **r**) or flip (hotkey: **x** or **y**) the fet before you place it.
- Move the fet to the desired location and click with **Button-1**. The Wiring Tool automatically drops via Gcells.

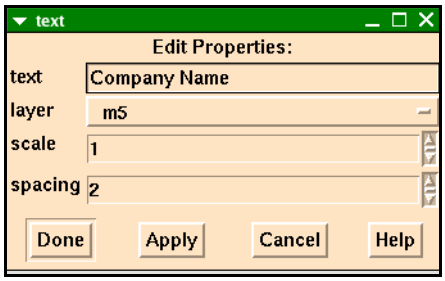
You can edit the properties of a Gcell with the `Push into Cell` (page 79) command in the `Edit` menu (hotkey: **e**). This will bring up the `Edit Properties` form (See Figure 6) for the selected Gcell.

Figure 6: FET Gcell



There is also a Gcell for text as show in Figure 7. This can be used to add text, such as the company name or chip name, to the layout.

Figure 7: Text Gcell



Stretching Gcells

You can change the size of a Gcell by either editing the properties of a Gcell (hotkey: **e**) or stretching the Gcell.

- To stretch a Gcell, first select the Gcell and go into the Gcell edit mode (hotkey: **Shift-e**).
  - Using the Gcell stretch capability, you can only change the width of a fet Gcell.
  - A via Gcell can be stretched in either direction.
- Once in the Gcell edit mode, hold down the middle mouse button (**Button-2**) near the edge of the Gcell and drag to the desired size. Additional vias will be added automatically.

Wiring Tool

The Wiring Tool makes it easy to lay out wires which include contacts/vias and multiple layers. Refer to "Introduction To The MAX Wire Tool" on page 109 for detailed information on how to use the Wiring Tool.

Editing Layout

Selecting Things

Many commands in MAX operate on the selection.

- For example, to move, duplicate or delete something, you must first select it.
- The current selection is highlighted in white (by default).
- In addition, the names of layers currently contained in the selection are highlighted in the palette in red.
- Editing functions are based on what is selected.

Refer to Figure 3 on page 13.

Selecting by Pointing

Mouse **Button-1** (in Main mode) is used for selection.

- To select something, simply click on it with **Button-1** (left mouse button).
- If there are several items under the cursor, you can cycle through them by clicking repeatedly with **Button-1** at the same location.

  Layers and labels/text can be selected in this way. For layers, the largest rectangle containing the cursor is selected.

  The `Cursor Probe` (page 93) command (hotkey: **Ctrl-Button-1**) brings up a pop-up form listing all layers, Gcell or instances under the mouse. Instances can be selected using the `Select Cell` (page 91) command (hotkey: **f**).

- Clicking **Button-1** on a space with no paint or objects will *deselect* everything.

Selecting Nets

- To *select an entire net*, move the mouse over a geometry and use the `Select Net` (page 92) command (hotkey: **s**).
- In the `Cursor Probe` (page 93) form, **Button-2** (middle mouse button) selects the net for the geometry.

Selecting Areas

- To *select everything in an area*, drag a box over it with mouse **Button-1**.
- Holding down the **Shift** key while dragging a box adds objects to the selection.

Selecting (and Excluding) Layers

The layers that are selectable can be controlled by clicking on the layer name with **Button-1** in the palette. Only layers that are currently visible and selectable can be selected.

Selecting (and Excluding) Labels

Labels that are attached to a layer are selected along with the layer. Free floating labels (with no layer below) are selectable only if visible, controlled through `Display Options` (page 42).

Adding to the Selection

- To add more objects to the current selection, use **Shift-Button-1** in place of **Button-1**.
- The `Selection Probe` (page 92) command (`hotkey:` **Shift-f**) can be used to add a cell to the selection.
  - If the item is already selected, **Shift-Button-1** and the **Alt-f** hotkeys *remove* the geometry, Gcell or instance from the selection.

Moving Things Around (And Other Manipulations)

Once you select something, typically with mouse **Button-1**, you can move it around with **Button-2**.

- To constrain the motion to one dimension only (horizontal or vertical) use **Shift-Button-2**.
- The selection can also be stepped over, one design-grid at a time, using the `Move` (page 71) command (`hotkeys:` **Shift-arrow-keys**).

Other Edit Commands

Other operations on the selection include:

- *Rotations* (See “Rotate” on page 71),

- *Flips* (See “Flip Upside-Down” on page 71),
- *Stretching* (See “Stretch” on page 75),
- *Duplication* (See “Duplicate” on page 70) and
- *Deletion* (See “Delete” on page 70).

These operations can all be found in the `Edit Menu` (page 57).

Hierarchy

Each cell in MAX is saved as a separate ASCII file on disk. A cell contains geometries and/or instances. In order to place an instance of a cell, it must be first loaded into MAX. If you have a standard cell library where all layout cells are in one directory, use the **`max_auto_load`** text command to load all cells in a directory. This command can be put into your **`.maxrc`** file.

Placing instances

To place an instance in the current edit cell, you need to have first loaded the cell into MAX.

- Once it is in one of the cell lists you can place the instance by holding down the **Shift** key while clicking on the cell with mouse **Button-1**.
- You then use **Button-1** in the Layout Window to position the instance.
- You can use the `Duplicate` (page 70) command (`hotkey:` **d**) to place multiple instances of the same cell.

Editing in place

Once you have placed instances in a cell, you can edit the lower level cells using the `Edit Cell or Object in Place` (page 80) command (`hotkey`: **Shift-e)** to edit the cell while viewing the rest of the layout.

- The command `Push into Cell` (page 79) (`hotkey:` **e**) pushes you down into the cell for editing.
- The command `Pop out of Cell` (page 80) (`hotkey`: **Ctrl-e**) pops you back up.

To use the `Edit Cell or Object in Place` or `Push into Cell` commands, you must first select the instance.

- Use the `Select Cell` (page 91) command (`hotkey:` **f**) to select an instance of the cell.

Chapter 3 MAX Commands

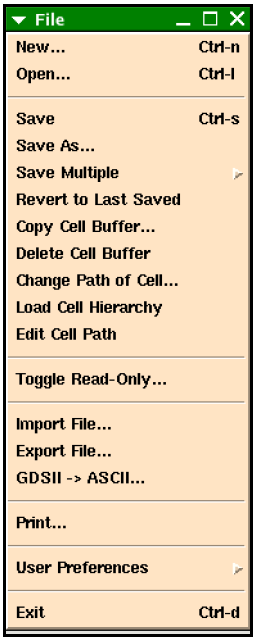
MAX Menus

This section describes each menu item in the MAX Layout Editor. A brief description of each command is displayed in the MAX `Message Area` (page 12) when the cursor is over the menu item. (Before reading this chapter, it is a good idea to have run the **MAX Tutorial** and/or read **Chapter 2**.) Each menu item is described starting with the `File` menu on the left.

File Menu

This menu is used create and open cells, save cells, import and export GDSII, and so on. Each menu option is described below.

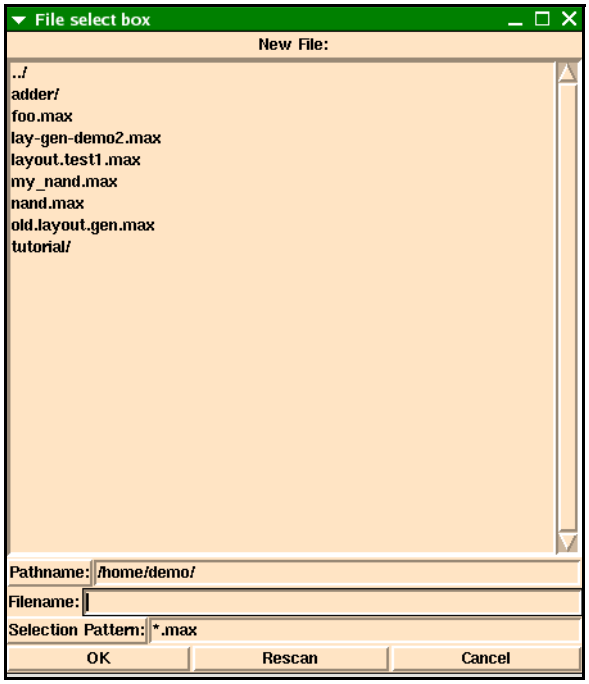
Figure 8: The File Menu



`New` *Hotkey:* ***Ctrl-n***

This is used to create a new cell. A form will pop up (see Figure 9) where you can specify the path and name of the new cell. Until the cell is saved for the first time, it is only in memory and has not been created in the UNIX file system. The first time a file is saved, a file **`<cell name>.max`** will be created in the specified directory. All MAX files must have the **`.max`** extention.

Figure 9: File Menu: New... Create a New Cell Pop-up



Open *Hotkey:* ***Ctrl-l***

Open an existing cell. A directory navigator box similar to Figure 9 comes up which allows you to search for the desired cell.

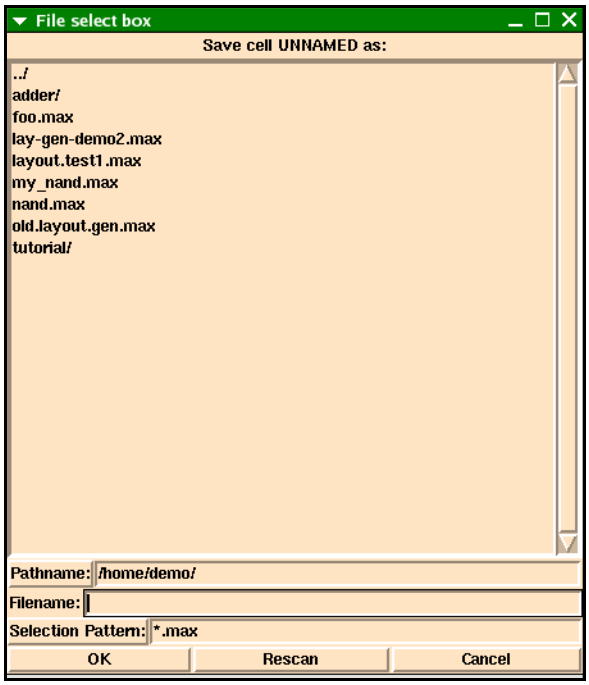
Save *Hotkey:* ***Ctrl-s***

Save the current cell. The cell is saved in a file called **`<cell_name>.max`** in the directory from which it was opened or created (New). If the cell is UNNAMED, a pop-up box (see Figure 10) appears where you specify the path and cell name.

Save As *Hotkey:* ***None***

Save the current cell to a different name. The cell is saved to the new name and MAX opens the new cell. The original cell will not be updated.

Figure 10: File Menu: Save As... Pop-up



Save Multiple

Hotkey: ***None***

This command brings up a sub-menu allowing you to save multiple cells. The options are:

Save edit cell and descendents — Save the current cell (edit cell) and all cells in the hierarchy below the current cell (descendents). This only saves cells that have been modified.

Save all modified cell buffers — Save all cells currently open in MAX which have been modified.

Save all cells — Save all cells in memory (loaded into MAX) except UNNAMED. This will save cells whether they have been modified or not.

Revert to Last Saved

Hotkey: ***None***

Restore current cell to the version which was last saved to disk. The user is prompted if this should really be done.

Copy Cell Buffer

Hotkey: ***None***

Make a copy of the current cell. This copies the existing version of the cell in memory to a new cell. The new cell must be saved in order for there to be a **`<cell_name>.max`** file on disk for the cell. This is used if you want to save the current state of the cell you are editing into a different cell name.

Delete Cell Buffer

Hotkey: ***None***

Delete the edit cell buffer (the cell you are currently editing). This only affects the version of the cell in memory. The version on disk, the **`<cell_name>.max`** file, remains unchanged.

`Change Path of Cell`

Hotkey: ***None***

Change directory where current cell will be saved. The default is the location from which it was loaded.

`Load Cell Hierarchy`

Hotkey: ***None***

Load current cell's descendents into memory. This can also be accomplished by `Showing internals`, which first has to load all the subcells in order to view them.

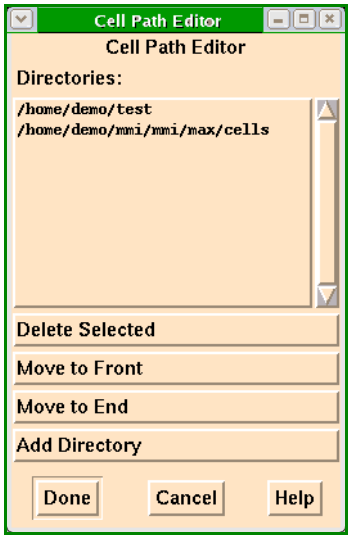
`Edit Cell Path`

Hotkey: ***None***

This menu allows you to more easily manage various directories. You can add, delete, and reprioritize directories. These are the directories where MAX will look for cells or gcells that are instantiated. It looks in the first directory first and if not found, looks in the next directory.

Figure 11 shows two directories, the directory from which MAX was started and the gcell directory. If you add new directories, these directories will not be displayed in the list boxes until a cell is loaded from the directory. Generally this command is added to your **`.maxrc`** file using the **cell_path_add** text command. That way if you have one cell that calls cells from other directories, they can be found.

Figure 11: File: Edit Cell Path



When you click `Done`, all cells in those directories that you have deleted will be removed from MAX memory.

`Toggle Read Only`

Hotkey: ***None***

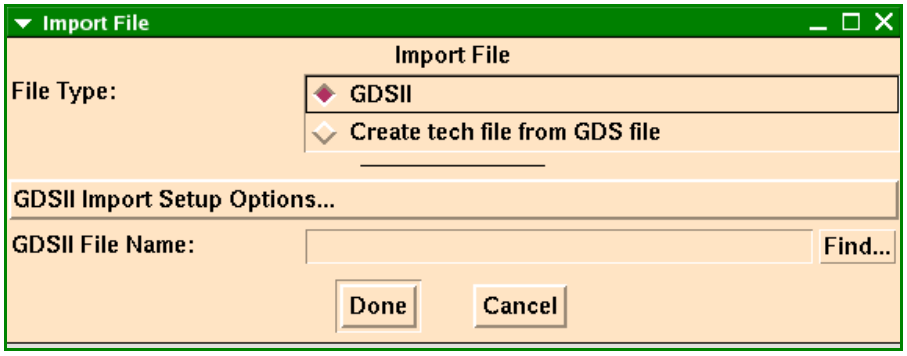
Toggle the read-only status of the current edit cell. If the cell is not read-only, this makes the cell read-only, and vice-versa.

`Import File`

Hotkey: ***None***

This brings up the `Import File` pop-up shown in Figure 12, allowing you to import a file into MAX. Currently only GDSII files are supported. The `GDSII Import Setup Options` menu is described below.

Figure 12: File Menu: Import File... Pop-up

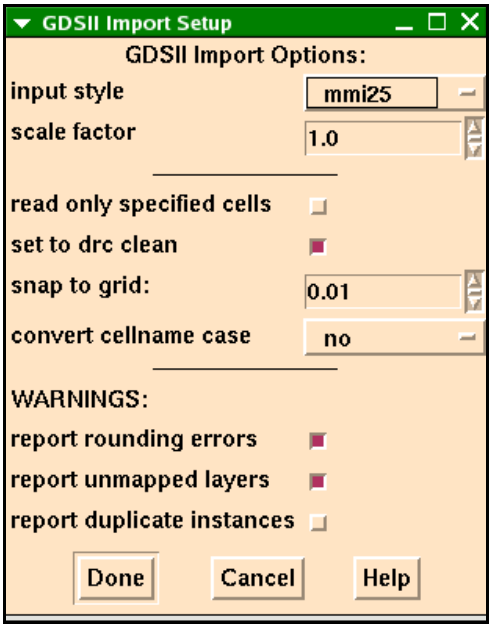


If you click on `Find`, a `File select box` appears. If there is an existing cell in memory with the same name as a cell in the GDSII file, it will be overwritten by the `Import File` command. This will not affect the version of the cell saved on disk.

The default suffixes for GDSII files are **`.gds`**, **`.strm`**, or **`.gdsii`**. You may need to change the suffix in the file selector box to find the GDSII files if your suffix is something else.

Clicking on `GDSII Import Setup Options` brings up the pop-up shown in Figure 13. Each of the options are described below:

Figure 13: File Menu: Import File... GDSII Import Options

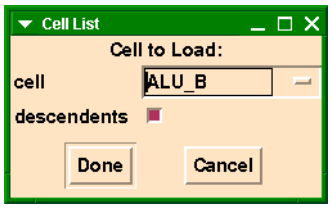


input style — If the technology file contains multiple styles, you can select between styles. In general there will be only one input style.

scale factor — While reading in the GDSII, the layout can be scaled by the specified factor. The default scale factor, 1.0, does not scale the layout.

read only specified cells — This allows you to read only a single cell or a cell and its descendents, not the entire GDSII file. Once you click on `Done`, MAX searches the GDSII file for all cell names and brings up the pop-up as shown in Figure 14. You can then select the cell to load from the generated list as well as specify whether to load its descendents.

Figure 14: File Menu: Import File... Import Options Cell List Pop-up



set to drc clean — Do not check for DRC errors once the GDSII file has been read in. You generally want this to be selected because running MAX DRC on an entire chip or block can take quite a while. When you make edits in MAX, the DRC rules will be checked in the area near the edits only.

snap to grid — Snap all coordinates in the layout to the grid specified. The default is the design grid.

convert cellname case — Convert cell names to all uppercase, all lowercase or leave them as is (default).

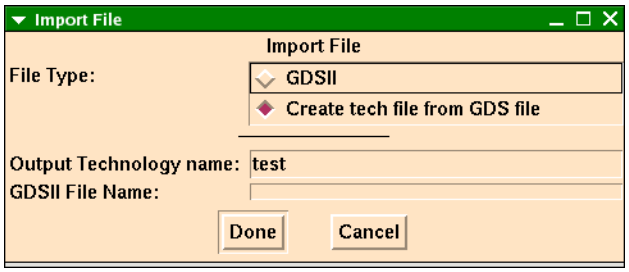
report rounding errors — Report rounding errors when reading in a GDSII file. Rounding errors can occur when converting from GDSII units to internal MAX units. They can also happen when snapping to a user grid, or when scaling (both GDSII input options).

report unmapped layers — Report any data found in the GDSII file on layers for which there is no definition (mapping) in the MAX technology file.

report duplicate instances — Report if duplicate instances are found at the same location. This should generally be turned off if the GDSII has lots of duplicates of cells at the exact same location (such as via cells).

All of the above options assume that there is an existing MAX technology file which matches the GDSII file. If no technology file exits, click on `Create tech file from GDS file` and the form will update to look like Figure 15.

Figure 15: File Menu: Import File... GDSII File and Create Tech File



The `Output Technology name` is the name of the technology which will be created using the `gds_input` (page 131) program. The technology file will be very basic with mappings for layers. It will not include DRC rules, but is a good starting place for creating a technology file. This option also allows you to look at a GDSII file without requiring an existing technology file.

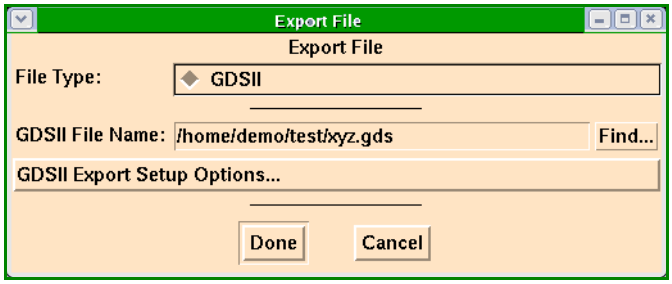
`Export File` *Hotkey:* ***None***

Export the current MAX cell and its descendents to a file, currently only GDSII.

- If you click on `Find`, a `File select box` pop-up (see Figure 16) will appear, allowing you to specify the name of the GDSII file. The default is **`<cell_name>.gds`**.

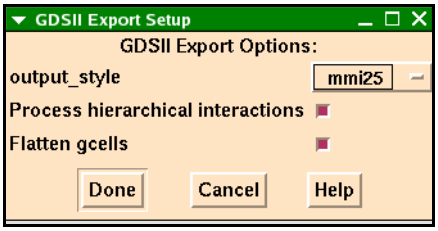
  If there is an existing **`.gds`** file with the same name in the directory, MAX will first have the user confirm overwriting the existing file.

Figure 16: File Menu: Export File... Pop-up



- Clicking on `GDSII Export Setup Options`, brings up the pop-up as shown in Figure 17.

Figure 17: File Menu: Export File... GDSII Options



output_style — Specify the output style for exporting GDSII. In general there is only one output style.

Process hierarchical interactions — Turning this off speeds up the output of GDSII data.

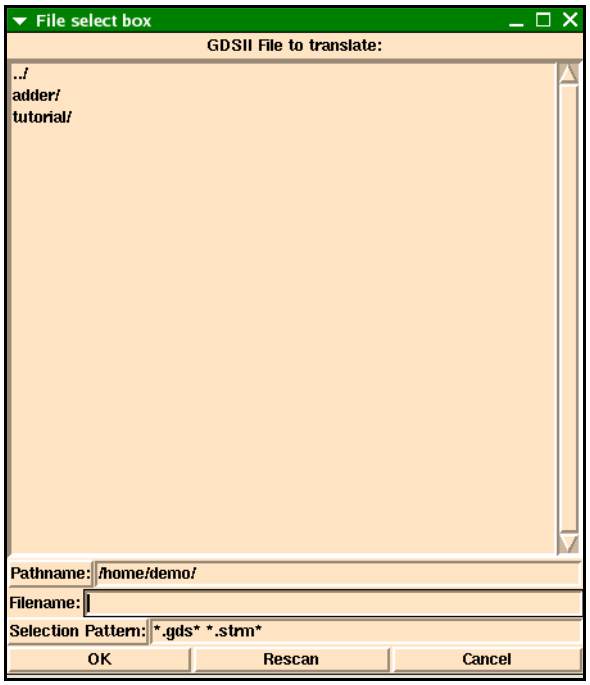
- If you used **`gds_input`** to create a technology file, then you can safely turn off the `Process hierarchical interactions` option and get MUCH faster (by approximately 100x) GDSII output.
- If you are using the **`mmi25`** technology provided with MAX, GDSII output does much more than just write out the GDSII file. It does layer generation (nplus, pplus, and nwell) with notch and gap filling, so hierarchical interactions must be processed in this case.

Flatten gcells — Also, if you are using the **`mmi25`** technology, use the `Flatten gcells` option to tape out the design to get proper layer generation of the nplus and pplus layers.

`GDSII->ASCII` *Hotkey:* ***None***

Converts a GDSII file into an ASCII format. This is a useful option for debugging a GDSII file. A pop-up appears (see Figure 18) where you can select a GDSII file to convert. This converted file is placed in the same directory as the original GDSII file. The extension of the converted file is **`.gds_ascii`**.

Figure 18: GDSII -> ASCII Menu



`Print` *Hotkey:* **None**

Using the `Print` command in the `File` menu, MAX prints the current cell to a postscript printer.

MAX prints all layers that are currently visible in the layout with the same colors, outlines, and stipples as set in MAX.

- Adjust colors and fill patterns before printing using the `Color Editor` (page 41) under `User Preferences` in the `File` menu.
- Additionally, turn off any layers in MAX that you do not want to plot.

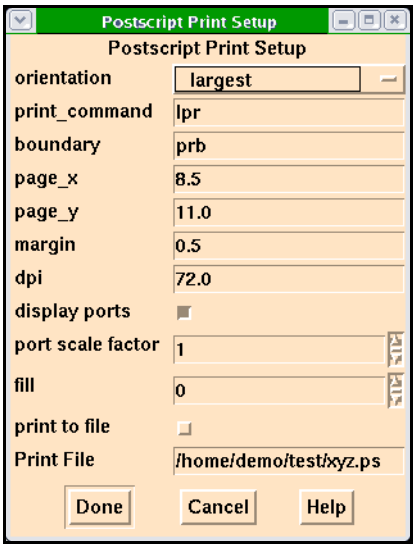
  For example, to plot only metal layers, turn off all other layers in the MAX palette before running the `Print` command.

MAX prints the entire cell, including all hierarchy shown, despite what is visible on the screen.

- To see what will be printed, do a `Zoom to Fit Edit Cell` (page 82) (hotkey: **v**), and then move the mouse outside of the cell and `Select Cell` (page 91) (hotkey: **f**).
- Once the top level cell is selected, select `Internals, View All` (page 82) (hotkey: **i**).

Before printing, MAX brings up a `Postscript Print Setup Form` (see Figure 19) where you select the printer (print command to use), the size of the plot, its orientation, the font scaling, a title, and whether to print directly or to a file. Even if you print to a file, you must choose the eventual printer name and image size so MAX can create the appropriate file type.

Figure 19: File Menu: Print... Setup Options



If the `Image Orientation` is set to `largest`, MAX automatically selects the orientation which will give you the largest printout. You can also select the standard `landscape` and `portrait` orientations.

For plotters, you can specify the length and width by entering values in the `page_x` and `page_y` fields.

- For example, if your plotter has a carriage width of 36 inches and you want a 70 inch long plot, you would specify "`page_x` **`36`**" and "**`page_y 70`**".
- in the "`print_command`" field you can type in direct commands to plot, like this:

  ```
  cat %s | lpr -s -P%p
  ```

  where **`%s`** is replaced with the plot file name and **`%p`** is replaced with the printer/plotter name when printing.

  After the plot is complete, the plot file is removed..

- Why not use the simple command:

  ```
  lpr -P%p %s
  ```

  In short, because plot files can get very large, the **`-s`** option is required to prevent the plot file from being copied to the print queue and possibly overflowing or jamming it.

  Unfortunately, you must be careful to not delete the plot file until the plot is completely sent to the plotter. By using the **`cat`** command, this is partially insured.

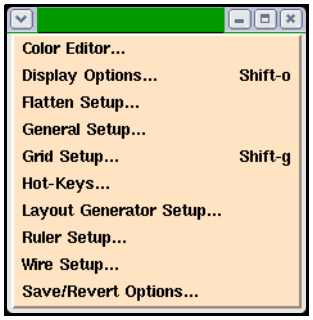
Even with this command, you may still have to plot large plots to a file and then send them to the printer.

User Preferences

Hotkey: ***None***

This brings up a sub-menu (see Figure 20) allowing you to specify user options. Each of these options are described below.

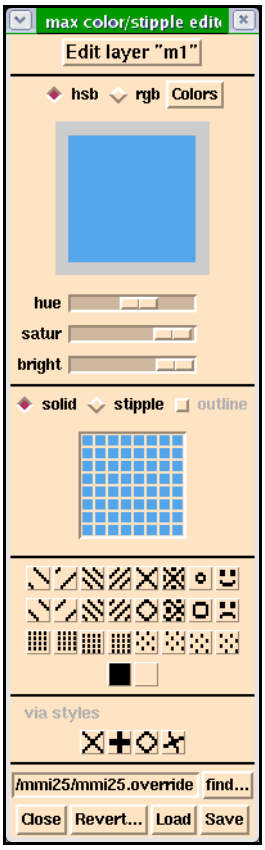
Figure 20: File Menu: User Preferences Options



`Color Editor` *Hotkey:* **None**

This brings up the MAX color editor as shown in Figure 21. You can specify the color and fill/stipple pattern for each layer.

Figure 21: File Menu: Color Editor... Pop-up

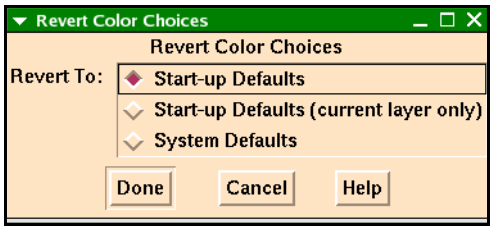


- Hold down **Button-1** over `Edit layer` at the top of the pop-up to select a layer to edit. The layers to edit include the ***background***, ***highlight***, ***feedback*** (external DRC tools) and ***grid*** layers.
  - You specify the color using the HSB or RGB sliders.
  - You can select a specific color name by holding down **Button-1** over `Colors`.
- In the lower section, you can specify whether to use a `solid` or `stipple` fill for the layer.
  - The `solid` fill is a transparent color and is typically used for poly and metal layers.
  - If you select `stipple`, you can choose from one of the predefined stipple patterns, or create your own. You may also select a solid `outline` for a stippled layer.
- For each via layer you can specify via styles, what kind of "x" to put over it. This allows you to set different via styles for different layers.

- You can specify a file to load or save the color information to. The default is always **~/mmi_private/max/tech/<tech_name>/<tech_name>.override.** If it is saved in this file, then the color information will automatically be loaded the next time you start MAX as the same user.

- To revert to the original colors and fill types, click on the `Revert` button and the pop-up in Figure 22 will appear.

Figure 22: File Menu: User Preferences Color Editor... Revert Colors Form



- If you choose `Start-up Defaults`, colors will be reset to what they were when you started this MAX session, or from the last time you saved the colors in this session. You can choose to revert all colors to these default settings or only the color that is currently in the color/stipple editor.

- If you choose `System Defaults`, MAX removes all your personal color preference settings by deleting any color file in **~/mmi_private/max/tech** for this technology. All colors will be reset to the defaults specified in the system **tech** file.

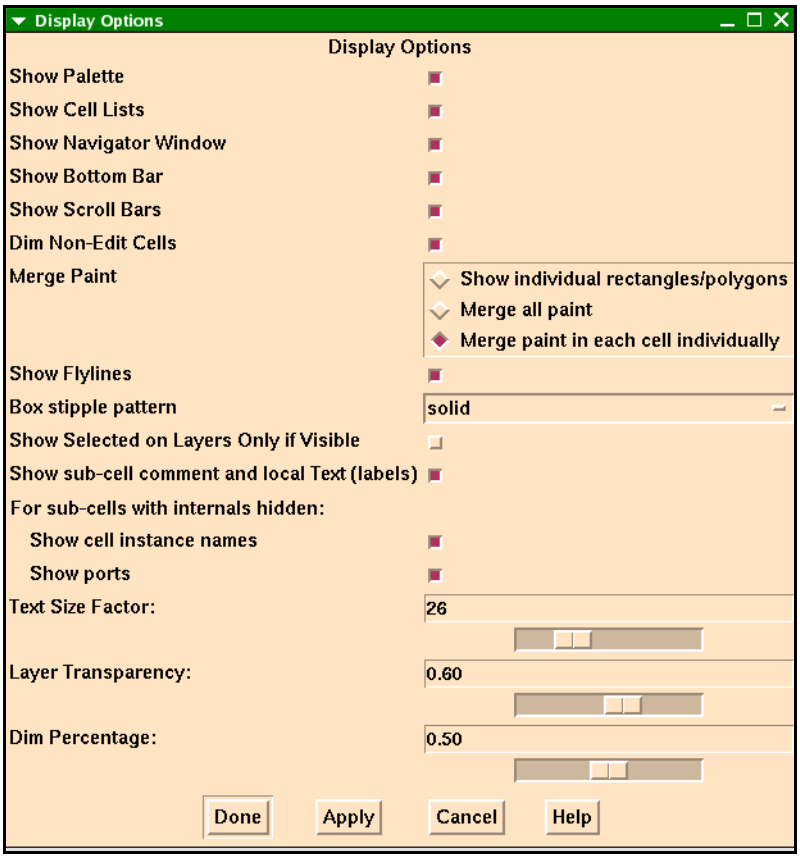
- If you select `Save` in the `Color Editor`, you current settings for all layers will be saved in your **mmi_private** directory in **~/mmi_private/max/tech/<tech_name>**, where **<tech_name>** is the name of the current MAX technology. MAX loads these color settings the next time you bring up MAX with this technology.

`Display Options`

Hotkey: ***Shift-o***

This brings up a sub-menu allowing you to specify user options, as shown in Figure 23. Each of these options is described below.

Figure 23: File Menu: User Preferences Display Options... Pop-up



Show Palette — This is a toggle button to turn on or off viewing of the palette on the left side of the MAX window.

Show Cell Lists — This toggle button turns on or off the viewing of the cell lists on the right side of the MAX window. If the cell lists are turned off, the Navigator Window is also turned off.

Show Navigator Window — This toggle button turns on or off the viewing of the Navigator Window on the right side of the MAX window. The cell lists will expand down into the region reserved for the Navigator Window.

Show Bottom Bar — This toggle button turns on or off the bottom bar containing the DRC Status Area, the Zoom Bar, Cursor Coordinates and the Box Display Area.

Show Scroll Bars — This toggle button turns on or off the scroll bars on the bottom and right of the edit area. The scroll bars appear only when you are zoomed in on the layout.

Dim Non-Edit Cells — Normally mask information that is not in the edit cell is dimmed. To display all mask information at the same brightness, turn this button on.

Merge — Paint is rectangles or polygons in a specific layer. This controls what happens when you merge paint. Paint merging, which provides correct alpha blending and outlining for layers works for both rectangles and polygons. This controls how overlapping polygons and rectangles are viewed in MAX.

There are three `Merge Paint` options, controlled in the `Display Options` menu.

Show individual rectangles/polygons: If paint merging is set to this, then MAX shows the individual rectangles and polygons. If there are two rectangle of the same layer in the same cell and same hierarchy, the the rectangles are always merged and look as if they are a single polygon. If a paint rectangle and a polygon overlap, then this option will show the overlap area in a darker shade than the layout layer color.

Merge all paint: This causes all paint and polygons to be merged across all cells. All rectangles and polygons will be shown with the same shade of color. Remember, though, that the current edit cell is always displayed in a darker shade unless the **Dim Non-Edit Cells** options is turned off.

Merge paint in each cell individually: This causes paint to be merged for each subcell individually.

Show Flylines — Flylines show connections yet to be wired, as straight lines between the circuit elements to be connected. The display of flylines is controlled by this toggle button. Flylines are only visible when you are zoomed in closely enough.

Box stipple pattern — This controls the outline pattern for the box layer. In the Color Editor, you can only change the color of the box. The options for the outline are solid, dash, and dotdash. Use the toggle list to select the desired option. You can also specify your own stipple pattern, for example for a dash pattern, use 0377. Stipple pattern is a 16-bit binary number where a 1 means the pattern is on and 0 means it is off.

Show Selected on Layers Only if Visible — If this is toggled on, then when you do a select net, only layers which are visible are highlighted. Select net still traces the connectivity through layers which are not visible.

Show sub-cell comment and local Text (labels) — This toggle button turns on or off the viewing of comment labels and local labels in a hierarchical design. If labels are defined as *input*, *output*, *inout*, or *global* they will always be displayed if `labels` is turned on and the internals are visible. By default, any labels which are comment labels or local labels and are in lower level cells, are not visible.

- To make these labels visible the `Show` **sub-cell comment and local** `Text (labels)` toggle must be turned on. Note that these Text types are always displayed in the edit-cell.

For sub-cells with internals hidden: - When internals are hidden or not shown, there are the two display options below.

Show cell instance names — Normally subcell names and instance identifiers are displayed for subcell instances whose internals are hidden. If this button is turned off, these names will not be displayed. The subcell names will still be displayed if the subcell is selected.

Show ports — Show/hide Text ports (Text of types "**`input`**", "**`output`**" or "**`inout`**") even when sub-cell internals are hidden.

Text Size Factor — This controls the zoom level where Text labels disappear. Larger numbers make Text labels more visible when zoomed out.

Layer Transparency — Controls the transparency of layers in the edit cell. The values are a percentage of "solid", where 100% would be completely opaque, and 0% would be completely transparent (invisible). Transparency is generally most useful with values of 0.40 to 0.80, or 40% - 80% opaque.

Dim Percentage — Controls the transparency of layers that are not in the edit cell. The transparency is a percentage value of opaqueness of the specified layer, as explained above.

Flatten Setup

Hotkey: ***None***

This menu controls setup for the `Flatten Cells` (page 98) command, accessible through the `Misc` menu. The setup menu specifies what happens to labels in a cell when you flatten it.

For each type of label or text (**input, output, inout, global, local, comment**) you can either:

- Delete that type of label/text, or
- Change it into any other type of label.

  For example:

  - If you specify **`global turns into: global`** then global labels in the flattened cell will be copied into the parent cell without changes.
  - If you specify **`global turns into: delete`**, then global labels will be deleted.

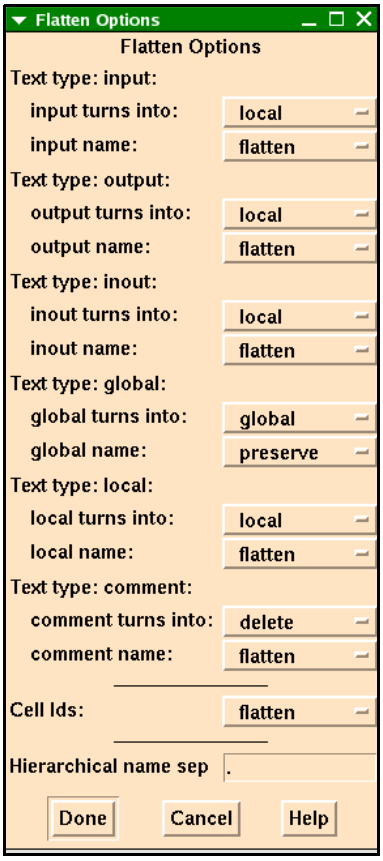
You can also specify whether to **flatten** or **preserve** the label names. If they are preserved, they may conflict with other existing labels with the same name.

- To avoid this, you can `flatten` the label names, which will cause the labels to be renamed to something like: **`cellname.labelname`** where **`cellname`** is the instance name of the cell that was flattened, and **`labelname`** was the label name in the cell that was flattened.

  Cell Ids — If set to "`flatten`", Cell Id names are flattened by prepending the name of the parent cell to the cell's name. If set to "`preserve`", the Cell Ids are retained, but may be made unique by appending "_n", where `n` is a number.

  Hierarchical name sep — this field allows you to enter a label to differentiate cell names within a hierarchy, such as "`.parent`" or "`.secondary`". These names will automatically be appended to the cell name.

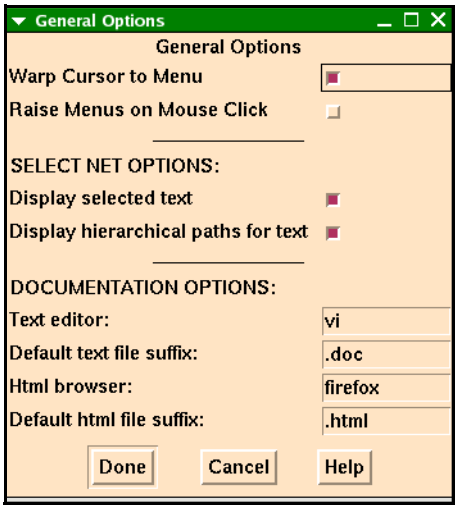
Figure 24: File Menu: User Preferences Flatten Setup... Pop-up



`General Setup` *Hotkey:* ***None***

This brings up a sub-menu, shown in Figure 25,which allows you to specify user options. Each of these options are described below.

Figure 25: File Menu: User Preferences General Setup... Pop-up



Warp Cursor to Menu — If set, the mouse cursor will move automatically (warp) to the default "OK" or "Done" button within most pop-up menus. Be aware that this option may not work if you are using a Linux emulator.

Raise Menus on Mouse Click — If set, then clicking the mouse anywhere in the MAX window will automatically raise and make visible the **current** property menu or dialog box. For times when a property menu or dialog box is obscured by the main MAX window; that is, when the menu drops behind the main MAX window in the window stacking order and is no longer visible, this is a helpful option.

SELECT NET OPTIONS — These next two options apply to selected nets and how net names are displayed.

Display selected text — When you select a net, this displays all unique text on the net, which can be a very long list. But it can be a very useful option if trying to detect shorts. If a net has both a VDD and a GND label on it, then you know there is a problem.

Display hierarchical paths for text — When you select a net, this will include hierarchical text names in the list of selected text. Otherwise, it will only list the names in each cell, which could be confusing in some cases. You would want to turn this option off though if you're looking at global signals such as CLK.

DOCUMENTATION OPTIONS — These options control which text and html editors to use for displaying documentation.

Text editor — This is used to specify the default text file editor for `Display Cell Doc` (page 84) in the `View` menu. The default is **emacs**.

Default text file suffix — This is used to specify the default text file suffix for `Display Cell Doc` (page 84) in `View` menu. This defaults to **.doc**.

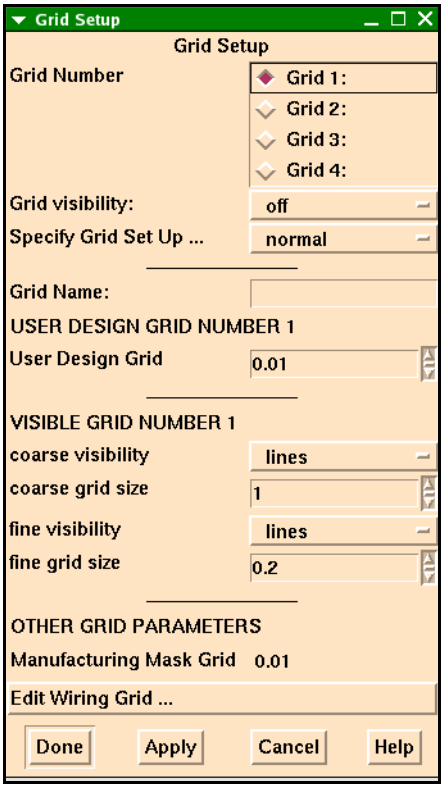
Html browser — This is used to specify the default HTML file browser for `Display Cell Doc` (page 84) in `View` menu. The default is **firefox**.

Default html file suffix — This is used to specify the default HTML file suffix for `Display Cell Doc` (page 84) in `View` menu. This defaults to **.html**.

`Grid Setup` *Hotkey:* ***Shift-g***

This command opens a pop-up form as shown in Figure 26. There are both a coarse grid and a fine grid which can be defined and viewed separately. In addition, the grid can be displayed as either lines (default), dots or dashes.

Figure 26: File Menu: User Preferences Grid Setup... Pop-up



Grid Number — MAX maintains information for several different grids. This option controls which grid is the current grid. The information on the current grid is displayed in this menu. You can set up the grids differently and then switch between them rapidly using this menu, or using hotkeys. (The default hotkeys are: **1**, **2**, **3**, **4**.)

Grid visibility — Determines if visible grid is currently on or off. Can be changed by the `Toggle Grid` (page 81) command in the `View` menu.

Specify Grid Set Up — If set to `normal`, the basic grid options are displayed in this menu. If set to `detailed`, additional available grid options are displayed.

Grid Name: — Allows you to enter a discriptive name for this particular grid setup.

User Design Grid — Points and rectangles entered by the mouse will snap to this resolution.

coarse visibility — Show a coarse grid when grid is on and zoomed in far enough. The grid can be displayed as either lines, dots or dashes.

coarse grid size — The size of coarse grid, if visible.

fine visibility — Show a fine grid when grid is on and zoomed in far enough. The grid can be displayed as either lines or dots.

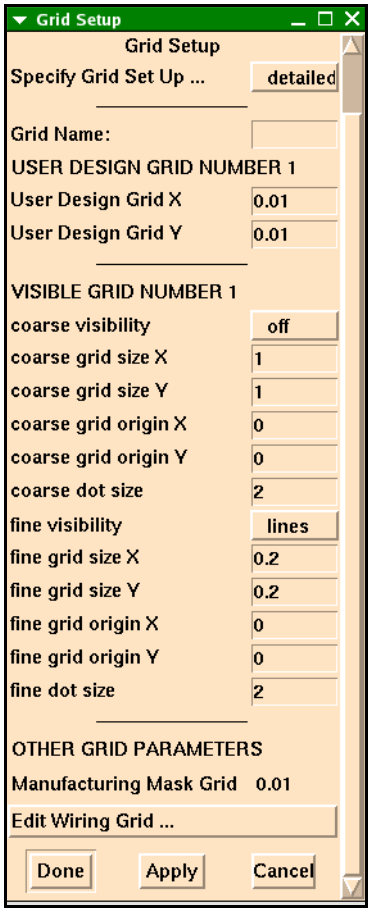
fine grid size— The size of fine grid, if visible.

Manufacturing Mask Grid — The minimum feature size determined by the manufacturing process; this value is set in the tech file See "`Technology Source Files`" on page 134.. and cannot be changed here. It is listed for reference.

Edit Wiring Preferences — This brings up the Wiring Parameters popup menu, as detailed in **Wiring Tool** chapter. This is where you can change the width, spacing and grid for use with the Wiring Tool.

If `Specify Grid Setup` is set to `detailed`, the following additional options are displayed as shown in Figure 27.

Figure 27: File Menu: User Preferences Grid Setup... Detailed Grid Setup



User Design Grid X — Points and rectangles entered by the mouse will snap to this resolution in the X direction.

User Design Grid Y — Points and rectangles entered by the mouse will snap to this resolution in the Y direction.

coarse visibility — The coarse grid can be turned **off**, or be **lines**, **dots**, or **dashes**. The coarse grid is displayed when you are zoomed out far enough.

coarse grid size X — The size of coarse grid in X direction, if visible.

coarse grid size Y — The size of coarse grid in Y direction, if visible.

coarse grid origin X — The origin of coarse grid in X direction.

coarse grid origin Y — The origin of coarse grid in Y direction.

coarse dot size — The size of grid dots if grid type is dots.

fine visibility — The display type of the fine grid. This can be set to either **lines** or **dots**, **dashes** or turned **off**.

fine grid size X — The size of coarse grid in X direction, if visible.

fine grid size Y — The size of coarse grid in Y direction, if visible.

fine grid origin X — The origin of fine grid in X direction.

fine grid origin Y — The origin of fine grid in Y direction.

fine dot size — The size of grid dots if grid type is dots.

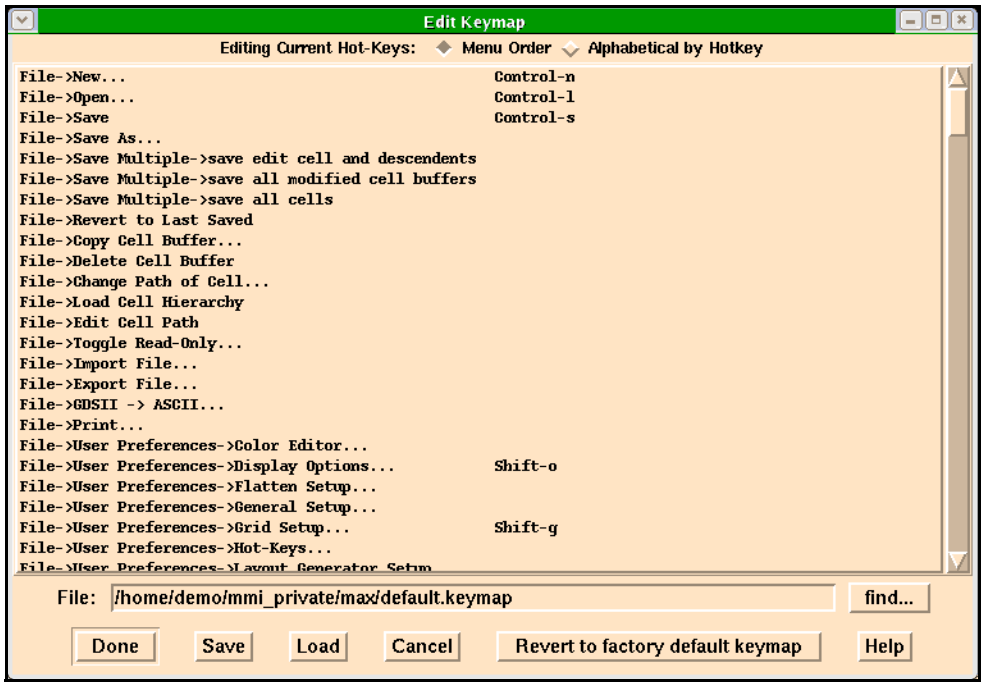
Hot-Keys

Hotkey: ***None***

This brings up a pop-up form, shown in Figure 28, where you can view, specify or modify the hotkey bindings used in MAX.

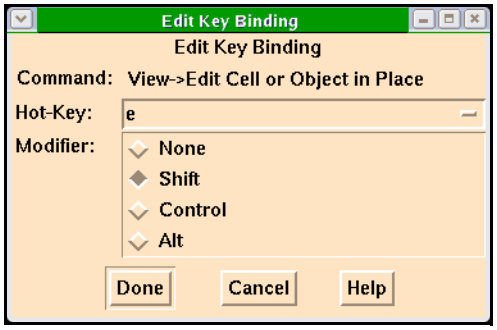
- The `Edit Keymap` pop-up lists the menu entries and commands with reprogrammable hotkeys.
  - The left column contains a description of the menu or command.
  - The right column shows the current hotkey binding, and is empty if no hotkey is assigned.
  - Some hotkeys can not be changed (for example, **Ctrl-C**), and so those menu items do not appear in the list.
  - The hotkeys can either be displayed in `Menu Order` if you're looking for a specific command from a menu, or they can be displayed `Alphabetical by HotKey`. This is helpful if you're trying to find out what hotkeys have already been defined.

Figure 28: File Menu: User Preferences Hot Keys... Edit Keymap Pop-up



- To edit the hotkey for a command, click on the item with **Button-1** and a pop-up form appears as shown in Figure 29.

Figure 29: File Menu: User Preferences Hot Keys... Edit Key Bindings



Hot-Key — The hotkey designation can be be any single character (a-z, 0-9, or symbols like $ or %), a function key designator (F1 - F10 only, not F11 or F12) or a valid "keysym" name for special keys on your system, such as "Left" for the left-arrow key.

CASE IS IMPORTANT!
"Left" is not the same as "left".

- To remove a hotkey binding, leave the entry blank.

- Valid keysym names vary from system to system and can be difficult to discover. Use the pop-up menu on the right edge of the `Hot-Key:` entry area to display a list of possibly valid keysyms.

Sometimes the symbolic key you select (such as "<") will automatically be converted to a keysym name (like "less") to make sure it will be a valid binding for the X-window system.

Many keys have multiple keysym names; MAX does not detect conflicts if you use synonyms. For example, if you bind the hotkey "%" to one function and "percent" (a valid keysym name synonymous with "%") to another function, no error will be printed. Obviously, the % hotkey will perform only one of the two functions.

- **Modifier** — The modifers can be **Shift**, **Control** or **Alt**. If you want a hotkey to be a capital "b", then specify the hotkey as **b** and the modifier as **Shift**.

The "**Alt**" key modifier is pre-empted by some window managers and not passed to application programs, so it may not be usable in MAX. MAX automatically maps all "**Alt**" key combinations to "**Meta**" key combinations as well, so if you create a hotkey using the "**Alt**" modifier, you may be able to use the hotkey combination by pressing the "**Meta**" key (if you have one) instead of the "**Alt**" key.

- If you want to Save or Load a keymap file, you first enter the file name in the `File:` box. You can use **find** button to help search for a file. The default location for a user defined keymap is **`~/mmi_private/max/default.keymap`**. If this file exists, then it is automatically loaded when you start MAX. After specifying the file name, click on `Save` or `Load` to save or load the file.

- At any time you can `Revert to factory default keymap`. This reverts back to the hotkey settings that came with MAX.

The keymap files are found in the following places and loaded in this order.

```
$MMI_TOOLS/max/
$MMI_LOCAL/max/
~/mmi_private/max/
```

The final keymap loaded overrides the others.

- System administrators can add keymaps to this list by creating and saving a keymap in MAX, then copying their file **`~/mmi_private/max/default.keymap`** to **`$MMI_LOCAL/max/new.keymap`**, where "**`new`**" is a descriptive name of the keymap.

- If you select `Done`, the new hotkeys will be used in the current MAX session. The new hotkeys will appear in the help for `Current Hot Keys`, and will also appear in the menus.

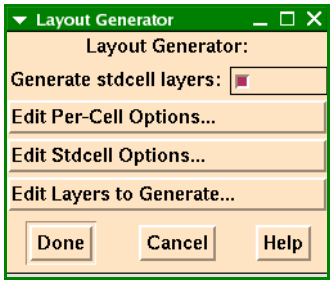
Layout Generator Setup

Hotkey: ***None***

This brings up a sub-menu allowing you to specify user options, as shown in Figure 30, below. These are options which are used by the Layout Generator which is part of MAX-LS.

Each option in this menu is described fully in Chapter 5, "Introduction To The MAX Layout Generator".

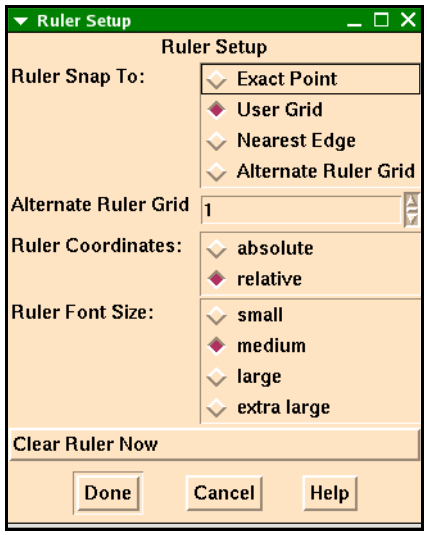
Figure 30: File Menu: User Preferences Layout Generator Setup... Pop-up



`Ruler Setup` *Hotkey:* ***None***

This brings up the `Ruler Setup` pop-up form, shown in Figure 31, which controls how the ruler snaps and whether to user absolute or relative coordinates.

Figure 31: File Menu: User Preferences Ruler Setup... Pop-up



Ruler Snap To — This specifies what the ruler will snap to. The options are **Exact Point** (the exact point you click on), **User Grid** (the nearest user grid point), **Nearest Edge** (The nearest edge of any polygon), or **Alternate Ruler Grid**.

Alternate Ruler Grid — The alternate grid, in microns, as opposed to the User Grid. This is only used if `Ruler Snap To` is set to `Alternate Ruler Grid`.

Ruler Coordinates — If set to `relative`, the ruler is numbered starting at 0. If set to `absolute`, the ruler is numbered with absolute coordinates.

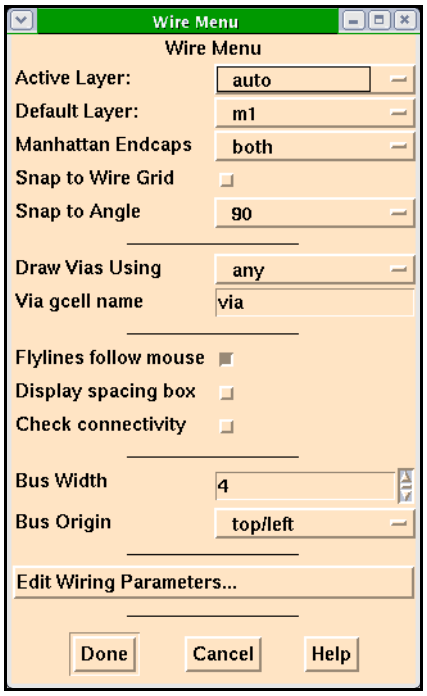
Ruler Font Size — The size of text displayed by the ruler. This option does not affect rulers that have already been drawn.

Clear Ruler Now — Clears all rulers. You can also use the **Alt-r** hotkey.

`Wire Setup` *Hotkey:* ***Shift-w***

The `Wire Setup` menu (see Figure 32) contains several options for controlling wiring parameters. These options are described below.

Figure 32: File Menu: User Preferences Wire Setup... Pop-up



Active Layer — This is the layer the wire tool uses to draw the wire. If the tool is set to **`auto`**, the wire tool attempts to determine the correct layer by looking under the mouse when the wire is started. If there are multiple layers under the mouse, it uses the top layer. The **Active Layer** can also be set using the **`Active`**: indicator at the upper left of the MAX screen.

Default Layer — Sets the first layer the wire tool looks for under the cursor if **Active Layer** is set to `auto`, and either the Default Layer is found under the cursor, or there are no layers under the cursor.

Manhattan Endcaps — This is used only for drawing Manhattan wires. It can be set to **both**, **none**, **begin**, or **end**. If set to "`both`", then the wire will be drawn with beginning and ending endcaps with a size half the width of the wire. So, the wire will extend half a wire width past the beginning and ending points.

Snap to Wire Grid: — Wire end-points and vertices will snap to the wiring grid specified in the `Wiring Parameters` menu. This is NOT the same as the `User Grid` in the `Grid` menu. You can specify a different wiring grid for each layer being wired in the `Wiring Parameters` menu. When placing a via, the via will be moved to the nearest valid intersection of the grids of the two layers being connected by the via.

Snap to Angle — Wires are constrained to the specified angle. If the angle is set to 0, all angle wiring is allowed. The `Snap to Angle` can be 90. 45, 30, 15, or 0 degrees.

Draw Vias Using — If set to `any`, the code draws vias using the first method that works.

- If there is a via `gcell` installed, that will be used;
- If there is a via sub-`cell` found, that will be used;
- As a last resort, vias will be `painted` using rectangles.

  You generally want to use the `gcell` option.

Via gcell name — Specify the name of gcell to use for vias, and default properties in the form of: **`-propname value`**. This is used only if `Draw Vias Using` is set to `any` or `gcell`. You can use this to specify default properties for the via. For example, to make vias symmetric by default, set it to: **`via -symmetric 1`**. You can also change this if you have created your own via gcell for use by the wiring tool. To force the wiring tool to use 2 vias when changing layers, use the gcell name: **`via -min_x_cuts 2`**.

Flylines follow mouse — If this is set, any flylines attached to the net being wired will follow the end of the wire. When the wire is connected, the flylines disappear.

PERFORMANCE WARNING: If this option is set, and there are any flylines anywhere in the edit cell, then the wire tool will trace the wire connectivity, using the **sel_net** function, each time a new wire is started.

If the wire is extensive (example: vdd or gnd), and the cells through which the wire is connected are currently expanded (ie, their contents are visible), then the wire tool may be slow.

Display spacing box — A visible spacing box will be displayed showing the current spacing of the wire being drawn. The spacing info is taken from the `Wiring Parameters` menu. This can be helpful if the wiring spacing rules are larger than the DRC spacing rules.

Check connectivity — If this is set, MAX will check wire connectivity when the wire is finished, and report any shorts. Connectivity is not traced through unexpanded cells; only conflicts among labels in the edit cell are reported.

PERFORMANCE WARNING: If this option is set, the wiring tool will trace the wire connectivity using the **`sel_net`** function, which can be slow if the wire is extensive (for example, vdd or gnd).

Bus Width — If you are wiring busses, then this is the width of the busses.

Bus Origin — If the origin is set to top/left, the bus is built downward or to the right of the cursor. If the origin is set to bottom/right, the bus is built upward or to the left of the cursor.

Edit Wiring Parameters — Edit the parameters for the wiring tool. See `Wiring Menu` (page 113) for detailed information on the wire parameters. This is where you specify the width, spacing, and grid for each layer for use in the wiring tool.

`Save/Revert Options` *Hotkey:* ***None***

This saves the current settings for the colors, grid, ruler, and so on, to the file:

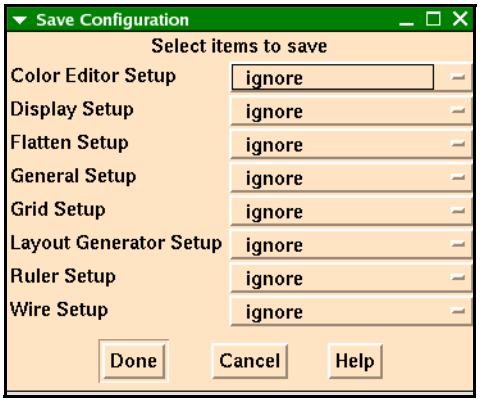
`~/mmi_private/max/tech/<tech_name>/<tech_name>.pref.`

The palette information will be written to a separate file:

`~/mmi_private/max/tech/<tech_name>/<tech_name>.override.`

The palette information can also be saved from the Color Editor. The next time MAX is started as the same user, the preferences files will be sourced.

Figure 33: File Menu: User Preferences Save/Revert Options... Pop-up



For each item, shown in Figure 33, you can choose how to handle the configuration in future MAX sessions. The options are:

ignore — This function does not save the configuration item at this time, and any previously saved information is unchanged.

save — The current state of the configuration item is saved from the current MAX session for future MAX sessions.

revert_to_default — Any configuration information that has been saved the this or a previous session for this item is deleted, and the NEXT MAX session will use start-up defaults. However, the CURRENT MAX session is unaffected (except for the `palette/color editor`, which reverts instantly).

`Exit` *Hotkey:* ***Ctrl-d***

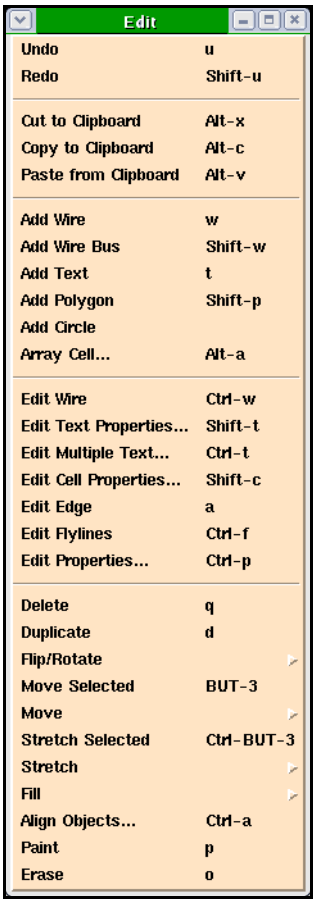
Exit out of MAX. If cells have been modified and not saved, a warning pop-up will appear. If you need to save cells, select `Cancel` and save the cells before quitting.

- Typing the **Ctrl-d** hotkey over the command window (the window from which MAX was started) exits MAX without any prompts.

Edit Menu

The `Edit` menu, shown in Figure 34, contains commands to modify geometries, Gcells, and cells.

Figure 34: Edit Menu



`Undo` *Hotkey:* ***u***

Undo last edit. There are 99 levels of undo. MAX keeps track of operations in every cell edited, so it can undo back to a previously edited cell. This command will undo back to the last time the cell was saved.

`Redo` *Hotkey:* ***Shift-u***

Redo last undo. There are 99 levels of redo.

`Cut to Clipboard` *Hotkey:* ***Alt-x***

Cut the selected geometry(s) or portions of geometries, Gcell(s) and/or instance(s) to the MAX clipboard. They can then be pasted into the current cell or into another cell. This also works for cutting out a section of a geometry, not just entire geometries.

Copy to Clipboard

Hotkey: ***Alt-c***

Copy the selected geometry(s) or portions of geometries, Gcell(s) and/or instance(s) into the MAX clipboard. The contents of the MAX clipboard can be pasted into the current cell or another cell.

Paste from Clipboard

Hotkey: ***Alt-v***

Paste the geometry(s) or portions of geometries, Gcell(s) and/or instance(s), which were previously cut or copied into the MAX clipboard, into the current cell. The geometry(s) are then visible and can be positioned with the mouse.

Add Wire

Hotkey: ***w***

Use the wire tool to connect geometry(s). The wire tool uses the connectivity defined in the technology file. Refer to `Introduction To The MAX Wire Tool` (page 109) for detailed information on how to use the wire tool.

To add a wire:

- Select `Add Wire` from the `Edit` menu or type the hotkey: **w**.
- Click with **Button-1** on the location to start a wire. If there is a wiring layer under the cursor and the Active layer is set to Auto, the wire will start in the layer under the cursor. Otherwise it will start in the default wiring layer.
- Move the mouse to draw the wire. Click with **Button-1** to turn corners.
- To drop a via and go up to the next layer, type the hotkey: **d.** To drop a via and go down a layer, type the hotkey: **Shift-d**.
- To end a wire, click with **Button-3**.

Add Wire Bus

Hotkey: ***Shift-w***

Use the wire tool to draw a bus. The bus will be drawn with the width specified by `Bus Width` in the `Wire properties` menu.

- After you have typed **Shift-w**, you can change the width of the bus, by holding down **Button-2** and selecting `Wiring Menu` (hotkey: **Shift-w**). This will change the bus width for this bus and all subsequent busses until the width is changed again.

  The wire tool uses the connectivity defined in the technology file. Refer to `Introduction To The MAX Wire Tool` (page 109) for detailed information on how to use the wire tool.

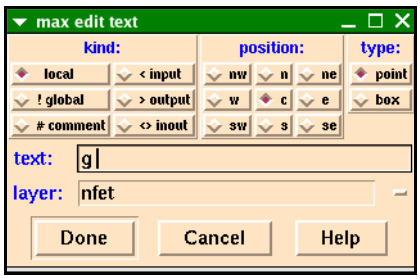
Add Text

Hotkey: ***t***

Add a piece of text or a label to the selected geometry.

- The easiest method is to first select a geometry on the desired layer and then select `Add Text`. A form comes up, as shown in Figure 35, where you enter the text for the label.
- If the layout is only going out to GDSII, the default options are adequate. Text is automatically scaled and rotated to maximize visibility.

Figure 35: Edit Menu: Add Text Form



local, global, comment — These options are used when you extract a SPICE or LEF file. This information has no effect on the GDSII output file.

input, output, inout — These options are used for the MCC Critical Path tool. These options are also used when you extract a SPICE or LEF file. This information has no effect on the GDSII output file.

point, box — The **box** option is used mainly when outputting LEF, to define the port region for a routing tool. The **point** option simply defines the origin of the label or text in a GDSII file. If a box is defined for a label, the origin of the text in the GDSII file will be the lower left corner of the box.

direction — This controls the origin of the label relative to the text.

layer — The **layer** option determines what GDSII layer the label will be generated in. It defaults to the layer under the box or point when the `Add Label` command was executed. To change the layer, select a the new layer name.

`Add Polygon` *Hotkey:* ***Shift-p***

To create a non-manhattan polygon in MAX, use the `Add Polygon` command. The polygon will be drawn in the active layer. To create a rectangle, it is much easier to draw the rectangle with **Button-1** and then fill it with a layer using the Paint command (hotkey: **p**).

- If you wish to change the active layer, you can do so by changing the active layer at the top of the palette before you start the polygon.
- If the active layer is `auto`, the polygon will be drawn in poly.
  - Use **Button-1** to draw the outline of the polygon by single-clicking at each vertex location.

  MAX will fill in the polygon when it becomes a valid polygon. Polygons can only have 45 or 90 degree corners (since most processes only allow 45 or 90 degree angles), cannot have acute angles, and cannot be self-intersecting.
- When you are happy with the polygon, simply click with **Button-3** to finish it.

  The MAX message window (shell window) will display the message "`Can't finish polygon. Move cursor and try again.`" if you try to end the polygon without connecting on a strict 45- or 90-degree angle.
  - Otherwise, you can hit **u** for undo (or click the middle mouse button) to remove vertices and try again.

- To edit an existing polygon, select it and use `Edit Properties` from the `Edit` menu. The outline and vertices of the object will be displayed.
  - Move the mouse close to the vertex that you wish to change and click the left mouse button. You can now add new vertices (**Button-1**) or remove old vertices (**Button-2**) just like you were adding a new polygon.
  - Inside the edit mode, you can also change the polygon layer by clicking the middle or right button over the desired layer in the palette. While in edit mode, the hotkey **m** toggles between all angle and 45-90 degree mode.

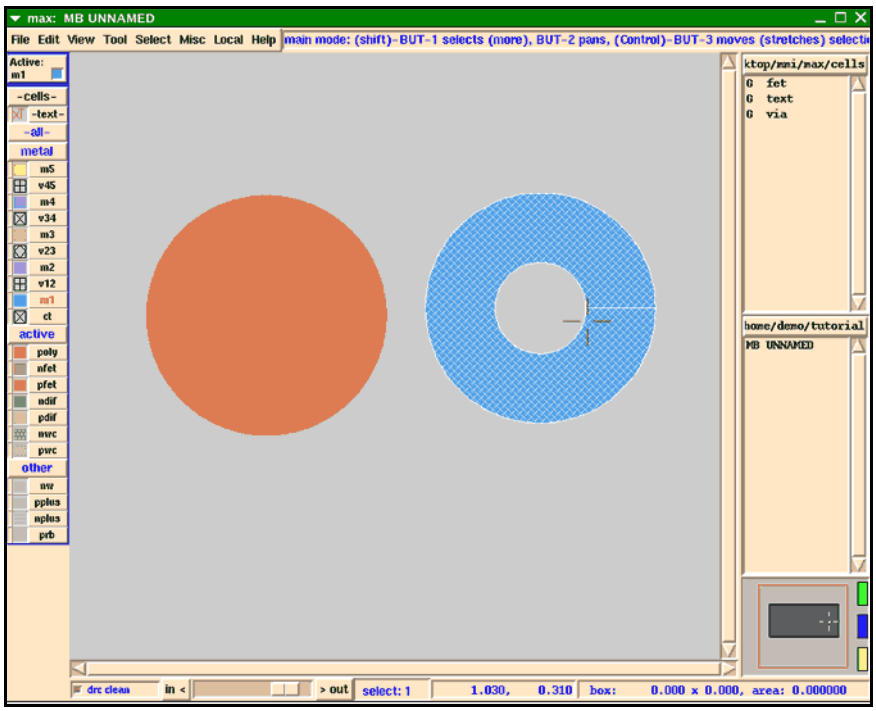
Interactive DRC does not work for all-angle polygons. Interactive connectivity tracing in MAX does work for polygons, but netlist extraction does not work in this release.

`Add Circle` *Hotkey:* **None**

To create a circle or a donut in MAX, use the `Add Circle` command. The circle/donut will be drawn in the active layer.

- Select the layer you want to draw the circle in by clicking on the **Active Layer** label at the top of the palette, and then selecting the desired layer name. A small square with the layer name and color will be displayed. Refer to the information on `Active Layer` on page 12.
  - Click **Button-1** where you want the origin of the circle/donut to be and drag out the circle. Clicking **Button-1** toggles between dragging the inside and outside radius of the circle and so you can make a donut. (Refer to Figure 36.) Click on **Button-3** to finish the circle.
  - If you want to edit the circle, simply delete it and create a new one.

Figure 36: Edit Menu: Circle and Donut in MAX

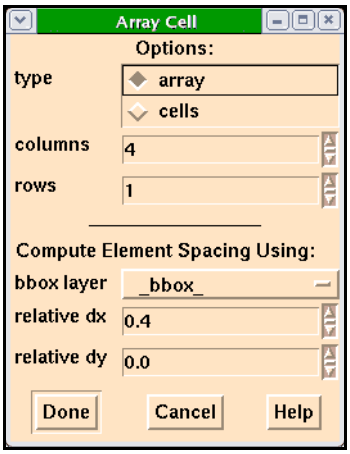


Interactive DRC does not work for circles. Interactive connectivity tracing does work, but netlist extraction does not work in this release.

`Array Cell` *Hotkey:* ***None***

- First select a the Gcell, group (See "`Group Objects`" on page 97.) or instance you want to array. When you select `Array Cell`, the pop-up in Figure 37 will appear, where you specify how many `columns` and `rows` in the array.
  - By default, the cells are abutted using the bounding box of all layers in the cell.
  - Alternately, you can select a specific layer to use for abutment. You may select the displayed bounding box of a cell based on a particular layer, since certain layers my overlap the cell's actual abutment box.
  - If you want additional spacing between cells, use the `relative dx` and `relative dy` options.

Figure 37: Edit Menu: Array Cell... Pop-up



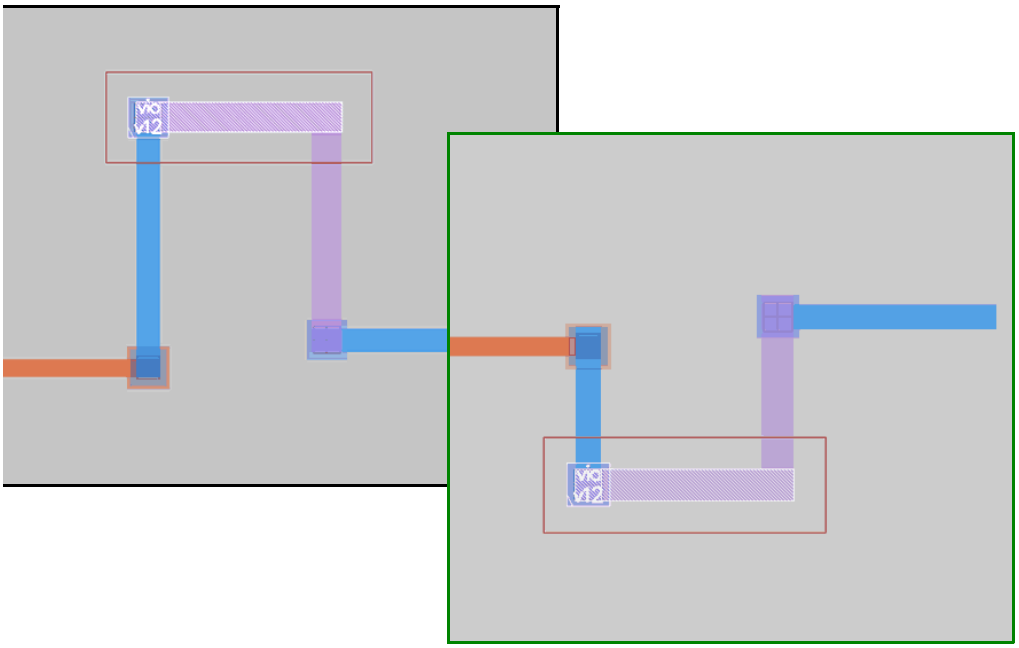
`Edit Wire` *Hotkey:* ***Ctrl-w***

With this command, you can edit existing wires.

- After you have selected `Edit Wire`, hold down **Button-1** on a wire segment and drag the segment to the new location.
- When you release **Button-1**, the editing ends. The segment you select will also move the vias if present.

  In Figure 38, the top segment was selected and moved down past the other wires. Notice that the wire stays connected and the via moves with the segment.

Figure 38: Edit Wire: a) Selecting Wire Segment to Edit; b) Moving Wire Segment Down

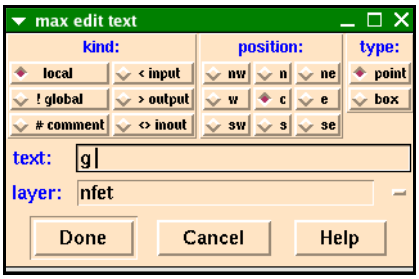


`Edit Text Properties`

Hotkey: ***Shift-t***

Edit the selected text. Select the text you want to edit and then click on `Edit Text Properties`. The form in Figure 39 will come up allowing you to enter any changes in the properties.

Figure 39: Edit Menu: Edit Text Properties... Pop-up



The different property types associated with text are:

Kind — This selection specifies the connectivity of the net to which the label is attached. This information is typically used during various types of extraction.

A "**local**" text label provides a name for the attached net, and indicates that the attached net is local to the current cell.

A "**global**" lavel indicated that the attached net is a global connecton, such as Vdd or Gnd.

An "**input**" "**output**" or "**inout**" label indicated that the attached net is a port. Ports are important to various extractors; for example, to mark nets that must be connected by routing to the parent cell.

The "**comment**" label is just a comment, and does not affect the connectivity of any attached net.

Position — These options specify the visual positioning of the text relative to the point location. If set to "**c**", the text is *centered* over the point. The other options cause the text to appear on the specified sides (north, west, southeast, etc.) of the point position.

Type — The "**point**" option attaches text to a single point, and is the most common type.

The "**box**" text option sets text in a rectangle, and is used in some applications to indicate bounding boxes.

Text — This is the actual text you type in for the text label.

Layer — If this field is *blank* (no entry) it means the text is not attached to any layer. If a layer name is specified, then the text label wll be physically positioned over geometry in that layer, affecting the connectivity of that layer according to the options chosen in the "**Kind**" label.

- Mouse **Button-1** repositions either the point or the box.

- If the label is a **point**, click once with **Button-1** at the new location.
- If the label is a **box**, click and drag out the new location and size of the box.
- Change the text by entering new text in the `Text` field.

Text does not specify connectivity for a net unless it is physically positioned over mask geometry on the layer specified by "Layer". Typically, no warning is produced for textual labels whose specified "Layer" does not actually match any of the layers that are underneath the text.

Edit Multiple Text

Hotkey: ***Ctrl-t***

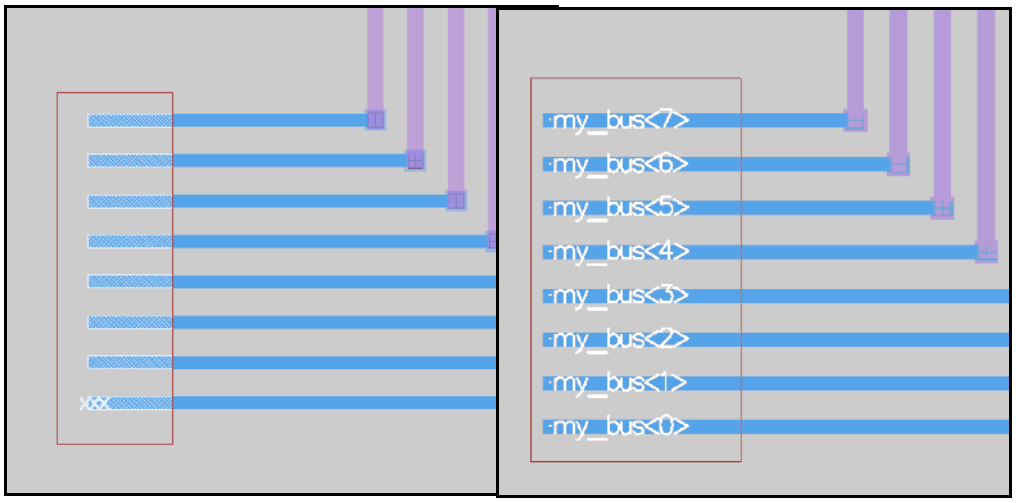
This command is used to automate the naming of busses. It can also be used to change properties of the selected text without changing the actual text.

To add text to busses using this menu:

- First, put a label/text on one bit of the bus. It does not matter what the label/text is. The label should be placed in the location on the rectangle/wire where you want the text to be added on all of the rectangles/wires.
- Next select each bit of the bus by drawing a box around all of the rectangles/wires that make up the bus making sure that the text you added is also selected.

  Figure 40 shows an eight bit bus with one piece of text and all the bits of the bus selected.

Figure 40: Edit Multiple Text: a) Selecting Area on Bus; b) Adding Multiple Text to Selected Area



- You could also put a label/text on each bit of the bus. The easiest way to do this is to `duplicate` the label/text. Then draw a box around the labels for that bus (be sure that no other labels are selected). Then select `Edit Multiple Text`.

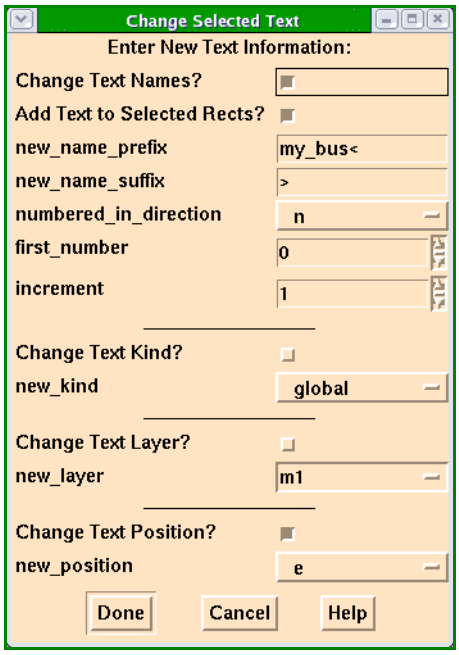
- To change the names of the text or add text to the bus, select the `Change Text Names` toggle button.

  In the example in Figure 41, the labels will be named **my_bus<0>**, **my_bus<1>**, and so on. Bit 0 is on the bottom, and they increment going up (**n** for **n**orth).

  Figure 41 also shows `Change Text Position` turned on and `new_position` set to **e**, which means the the names will display to the east or right of the text origin.

- If any of the rectangles/wires do *not* have text on them, then you will also see the `Add Text to Selected Rects` option. Select this to add text to these wires. If you now click on `Done`, the new text will be added as shown in Figure 40, above.

Figure 41: Edit Menu: Edit Multiple Text Menu... Change Selected Text



The `Change Selected Text properties menu` applies changes to all of the currently selected text labels. You can change the text **Kind**, **Layer**, **Position**, and rename the text to have or be a **<prefix> <number>** or **<suffix>**.

Change Text Names? — If set, the selected text will be renamed based on the prefix, suffix, direction, first number and increment..

Add Text to Selected Rects? — If set, text will be added to the selected rectangles. This is used when you want to add text to the rectangles of a bus.

new_name_prefix — The prefix attached to new text names.

new_name_suffix — The suffix attached to new text names.

numbered_in_direction — Text names are numbered, in order, in this direction. The variable `numbered_in_direction` controls where the bus naming starts and, therefore, in which direction the naming increments. A direction of `n` means that the naming starts at the bottom label and increments "north" (up).

first_number — The number given to the first text name.

increment — Numbers are incremented by this amount for each new text name. For example, if you want to number the odd bits of a bus, you can specify **first_number** as **1** and **increment** as **2**.

Change Text Kind? — If set, the "**Kind**" of text will be changed for all selected text. If `Change Text Kind?` is toggled off, the type of label is taken from each selected label.

new_kind — If desired, this is the new text "**Kind**". To change the `Kind` of all selected labels/text, select the `Change Label Kind?` toggle. `new_kind` can be *input, output, inout, global, local,* or *comment*.

Change Text Layer? — If set, the "**Layer**" type will be changed for all selected text.

new_layer — Optional new "**Layer**" specification for all selected text.

Change Text Position? — If set, the text "**Position**" will be changed for all selected text. This controls where the text is drawn relative to the origin of the text.

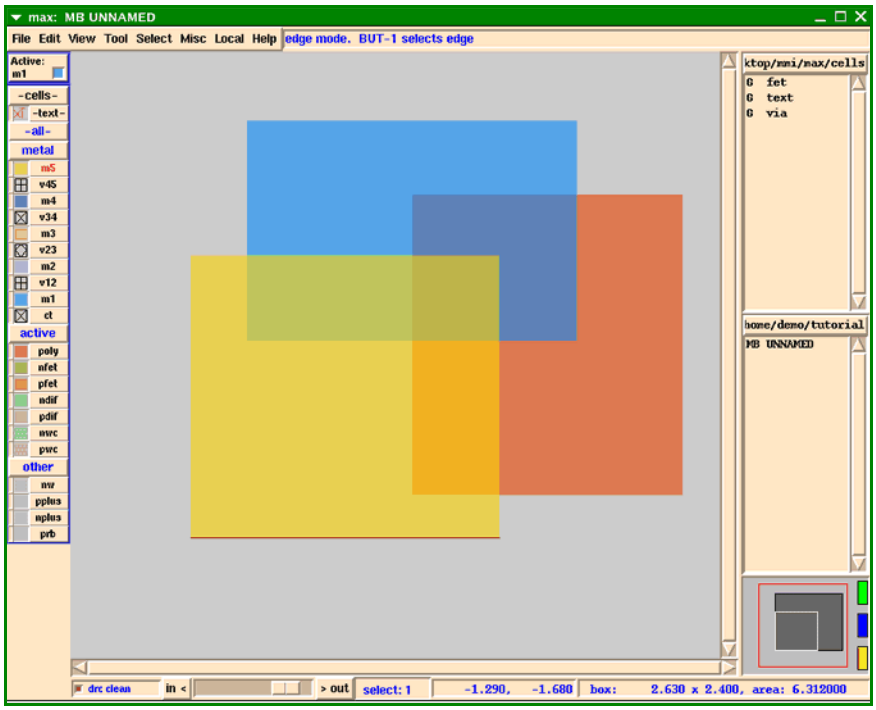
new_position — Optional new "**Position**" of all selected text. The **Position** will be *n, s, e, w, ne, ne, se, sw,* or *c*.

`Edit Edge` *Hotkey:* ***a***

Edit an edge of a geometry. This allows you to "stretch" the edge of a single rectangles or polygons.

- First, select `Edit Edge` or type hotkey: **a**. As you move over an edge of a rectangle or polygon, a highlight bar will appear, shown in red in Figure 42 below.
- Once you have the desired edge highlighted, hold down **Button-1** and drag the edge to the desired location. In Figure 42, we are moving the bottom edge of the yellow rectangle downward.

Figure 42: Edit Menu: Edit Edge Function



`Edit Flylines`

Hotkey: ***Ctrl-f***

This allows you to add flylines to the layout cell. If you have used the Layout Editor (See "`Introduction To The MAX Layout Generator`" on page 117.), flylines are automatically added to the layout. When you complete a connection, the flyline disappears. If you then break that connection, currently the flyline will not reappear. You would use the `Edit Flylines` command to manually re-insert the flyline.

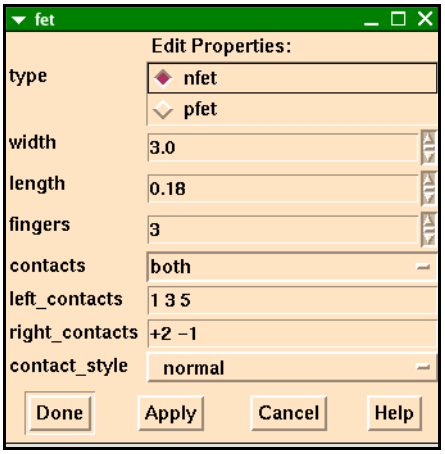
- Select `Edit Flylines` and then click with **Button-1** on the two geometries you want the flyline to connect.

`Edit Properties`

Hotkey: ***Ctrl-p***

View and edit the properties of an object. For Gcells, this would bring up the `Gcells properties` form as shown in Figure 43. This is the same form which was used when the Gcell was placed in the layout. Each of the options are described below.

Figure 43: Edit Menu: Edit Properties... FET Gcell Properties



type — Specifies the type of fet that will be produced. The types of fets this gcell can create are specified by the `device` (page 140) statements in the technology file.

width — Specifies the width of the fet. This fet generator has a minimum fet width of one contact width.

length — Specifies the length of the fet.

fingers — Specifies the number of fet gates. If `fingers` is greater than 1, then a stacked fet will be created.

contacts — Contacts can be specified `right` (on the right side of the fet only), `left` (on the left side of the fet only), `both` (on both the right and left sides of the fet), or `all` (on all source/drain regions of the fet, different from `both` only if the fet has multiple fingers).

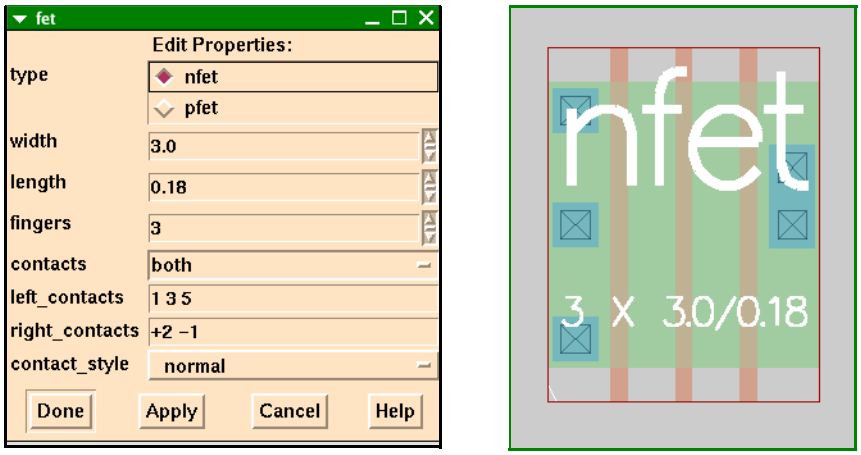
left_contacts — With this option you can enter specific locations for left contacts. Figure 44 shows the `left_contacts` in positions 1, 3, and 5 starting from the bottom.

right_contacts — With this option you can enter specific locations for right contacts. Figure 44 shows using the "+" and "-" operators to control where not to put contacts. "+2" removes the bottom two contacts. "-1" removes the top one contact.

- Click on the `Apply` button to see how these changes affect your gcell. When you are satisfied, click on `Done` to implement them.

"Dog-bone" style fets can be created with the fet gcell by modifying the necessary properties.

Figure 44: Edit Menu: Edit Properties... FET Gcell Properties



contact_style — If the technology file supports a metal-enclose-contact rule that is assymetric — one that allows the enclosure to be smaller in one direction than the other — this option specifies how the metal over the contact will be drawn.

If `contact_style` is set to `normal`, the metal enclosure of contact will be the same on all sides.

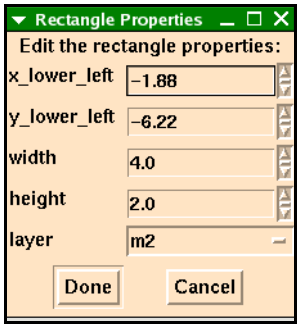
If `contact_style` is set to `minimum_x_overlap` or `minimum_y_overlap`, the contact will be drawn to minimize the metal in the direction corresponding to the x or y direction of the fet (in standard orientation).

The `Apply` button allows you to see the effects of your changes to the cell before actually committing them.

Click `Done` to commit your changes to the cell.

If you have a rectangle selected, `Edit Properties` brings up a form (see Figure 45) where the lower-left corner, width, and height of the rectangle can be changed. This is also one way you can change the layer of a rectangle.

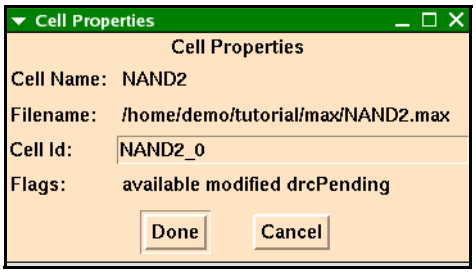
Figure 45: Edit Menu: Edit Properties... Rectangle Properties



If the selected object is an *instance*, the `Cell Properties` form, as shown in Figure 46, will appear. Here, you can change the instance name (**`Cell Id`**) for an instance. This form also tells you which directory it is getting the parent cell from and any flags on the cell (*available, readonly, modified*).

If the selected object is a polygon, it will put you into polygon edit mode where you can modify verticies and change the layer of the polygon.

Figure 46: Edit Menu: Edit Properties... Cell Properties Form



`Delete` *Hotkey:* ***q or <Del>***

Delete the selected geometry(s) or portions of geometries, Gcell(s) and/or instance(s). This differs from `Cut` in that the geometries and/or cells are *not* put into a buffer, so they cannot be pasted.

- If you want to delete a portion of a rectangle, draw a box (with **Button-1**) which includes the area to cut.
- Then use either the **Delete** command, or go to a area in the Layout Window which does not have paint and click the erase hotkey **o**.
  - If there are multiple layers of paint under the box you drew, you can specify to only delete a specific layer by selecting that layer in the palette or in the layout window and typing the hotkey **o**.
  - If you want to delete an entire rectangle on a layer, select that rectangle by clicking **Button-1** over it. If there are multiplayers under the mouse, you may need to toggle through them by clicking with **Button-1** at the same location.

  Note: This only works for rectangles, not for polygons or circles.

`Duplicate` *Hotkey:* ***d***

Duplicate the selected geometry(s), Gcells and/or instances. Once the geometry(s) have been duplicated, you use **Button-3** to move and place the geometry(s).

`Flip/Rotate` *Hotkey:* ***None***

This activates a sub-menu as shown in Figure 47 that allows you to flip or rotate the selected geometry(s) in several ways.

Figure 47: Edit Menu: Flip/Rotate Sub-menu

Flip Upside-Down

Hotkey: ***y***

Flip the selected geometry(s) vertically or upside down. This flips around the center of the bounding box of the selected items.

Flip Sideways

Hotkey: ***x***

Flip the selected geometry(s) horizontally or sideways. This flips around the center of the bounding box of the selected items.

Rotate

Hotkey: ***r***

Rotate selected geometry(s), Gcell, or instance clockwise by 90 degrees.

Rotate Counter-Clockwise

Hotkey: ***Shift-r***

Rotate selected geometry(s), Gcell, or instance counter-clockwise by 90 degrees.

Move Selected

Hotkey: ***Button-3***

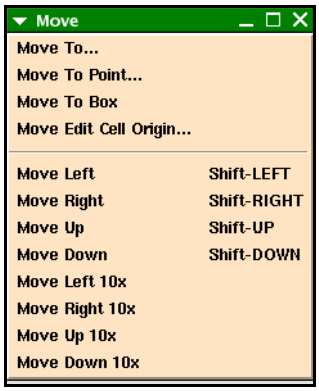
You first select what you want to move. It's generally easiest to move things by using **Button-3** as opposed to the menu.

Move

Hotkey: ***Shift <arrow keys>***

The `Move` menu contains several options for moving your geometries:

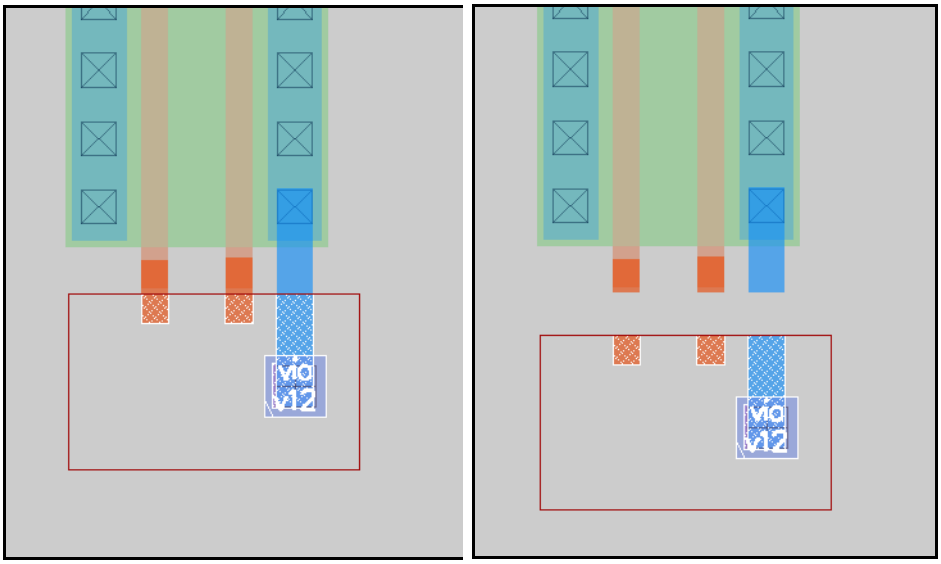
Figure 48: Edit Menu: Move Commands Sub-menu



The commands for moving affect selected rectangle(s) or portions selected by the box, text, polygons, Gcell(s) and/or instance(s).

- If an entire rectangle is selected, the entire rectangle will be moved. It's generally easiest to select rectangles by clicking on them with the left mouse button (**Button-1**). **Shift-Button-1** will add to the selection.
- If only a portion of a rectangle(s) is selected (surrounded by the box) the rectangle(s) will be cut or separated into two pieces when the selection is moved (see Figure 49).
- For everything other than rectangle (which include wires you've drawn), the `Move` commands move the entire Gcell, text, polygon, etc.

Figure 49: Edit Menu: Move Down Command Example



- The **Shift-arrow** keys allow moves of one grid point at a time. Using just the right mouse button (**Button-3**) allows free movement. **Shift-Button-3** locks the movement to either the X or Y direction.

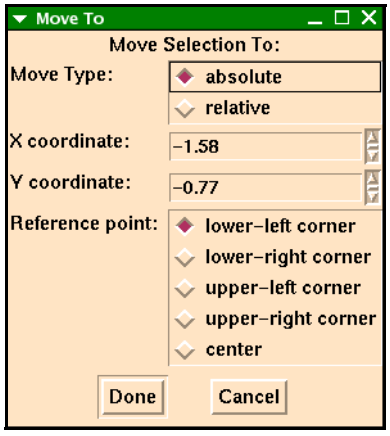
- If you select the `Move` sub-menu from the `Edit` menu (see Figure 50), you will notice that you can move object by one user grid at a time (**Shift-Arrow-Keys**) or move by 10x the user grid.

  - You can specify hotkeys for the 10x options in the `Hot-Keys` (page 50) options pop-up.

`Move To` ***Hotkey: None***

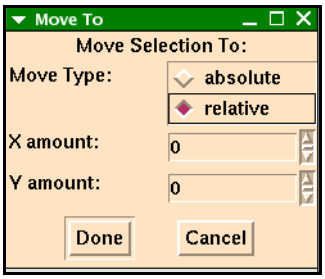
If you select `Move To`, the pop-up in Figure 50 appears for an `absolute` move.You specify coordinates to move to and the reference point for the move. If `center` is selected for the reference point, the center of the selection is rounded to the nearest manufacturing grid.

Figure 50: Edit Menu: Move Sub-menu Move To Form



- If you select a `relative` move for `Move To`, the form will update as shown in Figure 51. You then specify how far to move in the X and Y direction.

Figure 51: Edit Menu: Move Sub-menu Move To - Relative Form



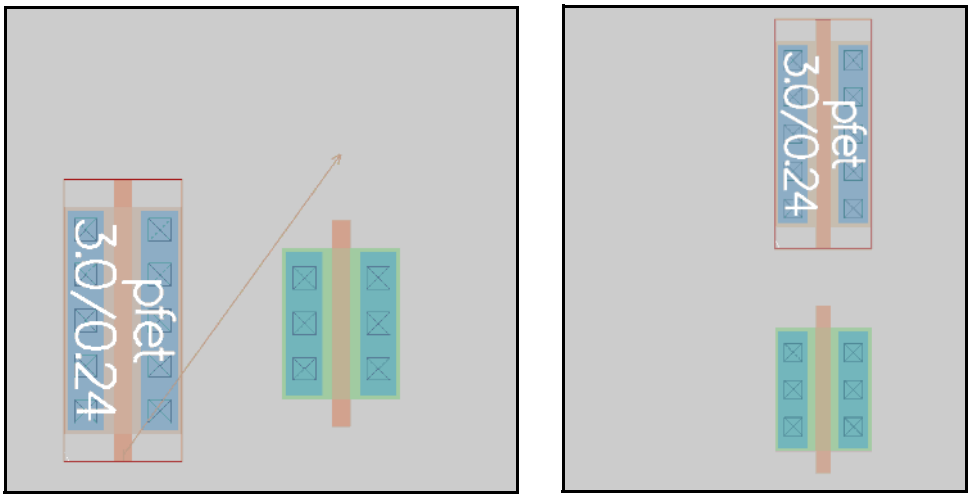
`Move to Point` ***Hotkey: none***

If you select `Move To Point...` from the `Move` sub-menu, you choose a point on the selected object to move from and a destination point to move to.

In the example in Figure 52, the pfet lower center is being moved above the nfet.

- Click on pfet at lower edge center to select it for moving.
- Click on `Edit`, then `Move > Move to Point...`
  - As you move the cursor, you will see an arrow dragging between the selection point on the pfet and the cursor.
- Click on the destination point (in this example, above the nfet) to finalize the pfet move.

Figure 52: Move: Move To Point... a) Arrow Showing pfet "Move To" Destination; b) pfet Moved



The `Move To Box` option moves the lower left corner of the selected objects to the lower left corner of the box.

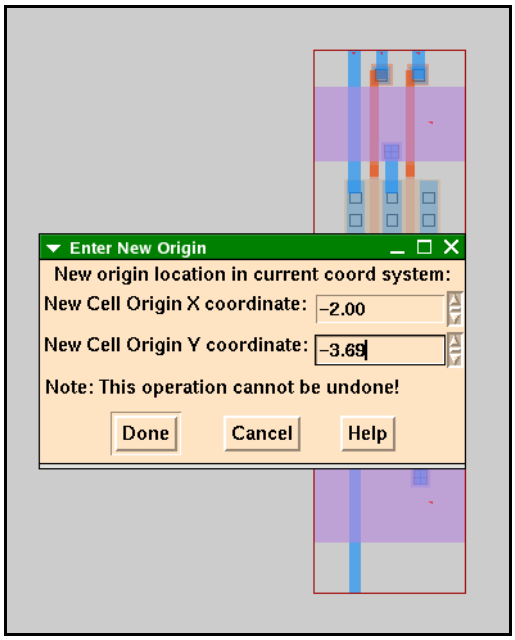
Move Edit Cell Origin

Hotkey: none

The `Move Edit Cell Origin...` option shown in Figure 53 moves the origin to the specified location.

- The easiest way to specify the location for the origin is to use the Box to get the coordinates, as shown in Figure 53.
  - You first set `Display on status bar, box to origin+size` by clicking with **Button-1** on the `Box Area Display` (page 22) in the lower right corner of the MAX window.
  - In this example, if you want the new origin of the cell to be the lower left corner of the cell, select the top level cell by moving the mouse outside of the cell and typing the hotkey **f** (`Select Cell` from the `Misc` menu). The `Box Area Display` now shows the coordinates for the lower left corner of the cell.
  - Enter these coordinates as the new cell origin.

Figure 53: Edit Menu: Move Sub-menu Move Edit Cell Origin



Stretch Selected

Hotkey: ***Ctrl-Button-3***

Stretch whatever is selected. This works for gcells as well as wires and rectangles. Any wires and rectangles must be at the current cell level of hierarchy.

- If you have just a gcell selected, select an edge on the gecell with stretch to change the gcell size.
- If anything else is selected, the the gcell will move and any selected wires attached to it will stretch. Figure 55 shows an example of stretching two wires down.

`Stretch Selected Gcell`

Hotkey: ***None***

For gcells, this command does the same thing as `Stretch Selected`.

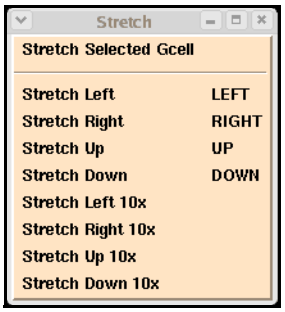
- First, select the gcell you want to stretch. Then once you select the `Stretch Selected` command, the cursor turns into a hand.
- Move the mouse over an edge on a gcell you which to stretch. As soon as the mouse is over an edge, the cursor turns into a stretch arrow.
- Using the left or right mouse button, press and hold the button and drag the mouse to the desired gcell size.

`Stretch`

Hotkey: ***<Arrow keys>***

Stretch the edges of the geometry(s) selected. If an entire geometry is selected, this command simply moves the geometry.

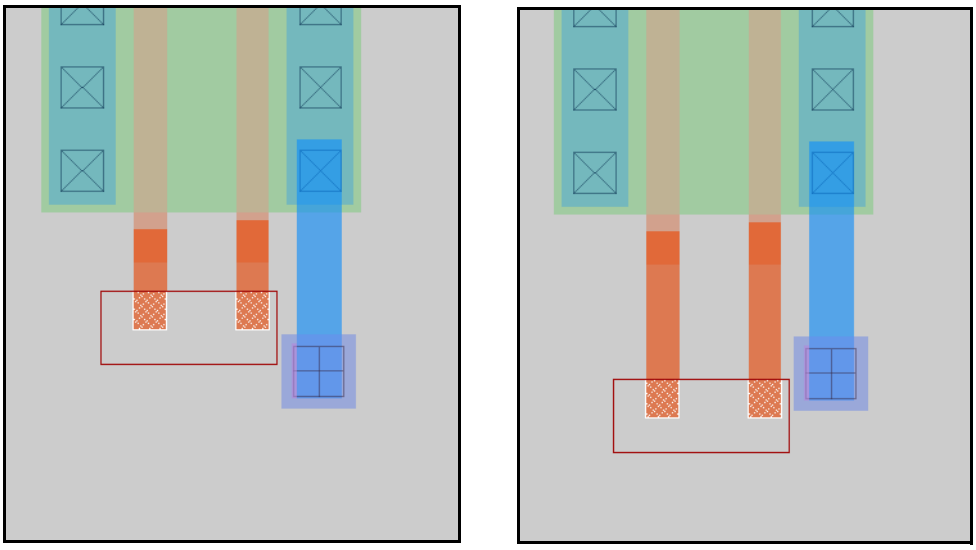
Figure 54: Edit Menu: Stretch Sub-menu



The arrow keys allow stretches of one grid point at a time. Holding down **Ctrl-Button-3** allows free movement for stretching while locking in either the X or Y direction.

See Figure 55 for an example of the `stretch down` command. In this case, the horizontal edges enclosed within the box are stretched down.

Figure 55: Edit Menu: Move Sub-menu Stretch Down Command Example



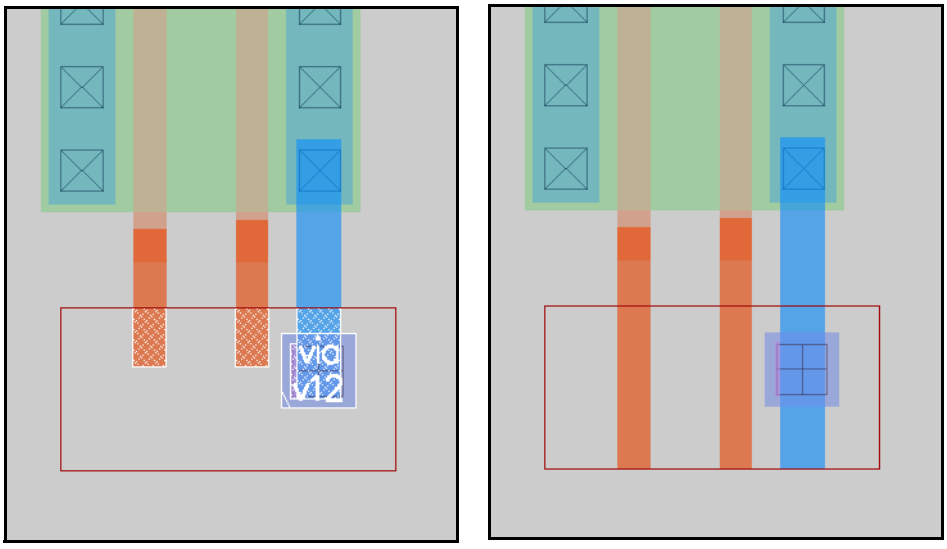
- If you select the `Stretch` sub-menu (see Figure 54) from the `Edit` menu, you will notice that you can also stretch by 10x the grid.
- You can specify hotkeys for these options in the `Hot-Keys` (page 50) options pop-up.

`Fill` *Hotkey:* ***Ctrl<arrow keys>***

Extend the geometry(s) to the edge of the box in the direction specified

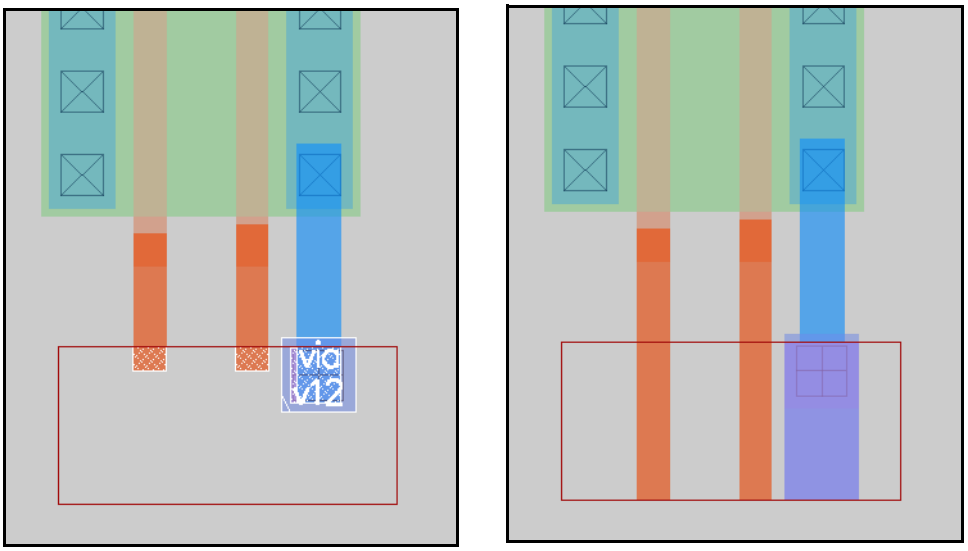
In the example shown in Figure 56, the `fill down` option is selected. The geometries selected by the box will be extended, or filled, down to the bottom of the box. Which layer is extended and the width of that layer is determined by the top edge of the box.

Figure 56: Edit Menu: Fill Sub-menu Extend Down Command - Example 1



In the example shown in Figure 57, the `fill down` option is selected again, but this time notice the top edge of the box intersects the wider via region. This time the width of the Metal1 layer extended to the bottom of the box is the width of the via region.

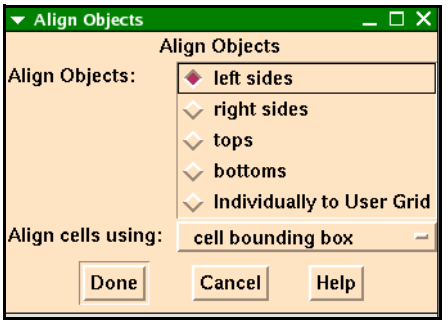
Figure 57: Edit Menu: Fill Sub-menu Extend Down Command - Example 2



`Align Objects` *Hotkey:* ***Ctrl-a***

The `Align Objects` pop-up form, shown in Figure 58, allows several choices for alignment. Each choice is described below.

Figure 58: Edit Menu: Align Objects... Pop-up



Align Objects — You can align only gcells, cells, rectangles and text.

- If you select `left, right, tops` or `bottoms`, the specified side of each object is aligned with the specified side of the current box. Usually the box is over the last object selected, so other objects are aligned with the last object selected.
- Alternatively, you can place a box with the **b** command, to which the objects will be aligned. If you select `Individually to User Grid`, all selected objects are nudged until their lower left corners are aligned on the current `User Grid`, as specified in the `Grid` menu.

Align cells using — `cell bounding box`, aligns cells using the edges of their bounding boxes.

- If you select `cell origin`, the cells are aligned using the origin point inside the cell.

"Align center" is not allowed because this could create off-grid geometries.

Paint *Hotkey:* **p**

This is the command to create rectangles in MAX. This method may initially seem different, but ends up being very efficient.

The steps to create a rectangle in MAX are the following:

1. Draw a box of the desired size.

   - Click and hold down the left mouse button (**Button-1**) and drag out a box of the desired size. Notice that drawing a box this way also selects what ever is under the box.
   - Type the **b** hotkey or select `Make/move Box` from the `Misc` menu. Then hold down the left mouse button (**Button-1**) and draw a box.
   - You can specify the exact location and size of the box using the `Box Dimensions` command under the `Misc` menu (hotkey: **Shift-b**). If you have already drawn a box, it will default to the origin and size of the box.

2. Fill the box with the desired layer using the paint command (hotkey: **p**).

   - Move the mouse of the desired layer in either the palette or the layout window and type **p**. If there are multiple layers under the mouse when you type **p**, all those layers will be drawn in the box.
   - If you select `Paint` from the `Misc` menu, then you click with **Button-1** on the desired layer after calling the command.

The paint command has two hotkeys defined: **p** and **Alt-But-3**. Some windowing systems already use the **Alt** key, so you have to use the **p** hotkey or change the meta key in the windowing system. You can always change hotkeys for commands.

If you type the **p** hotkey where there is no layout, then all layers under the box are **erased**. This only affects the current cell and does not affect gcells.

Erase

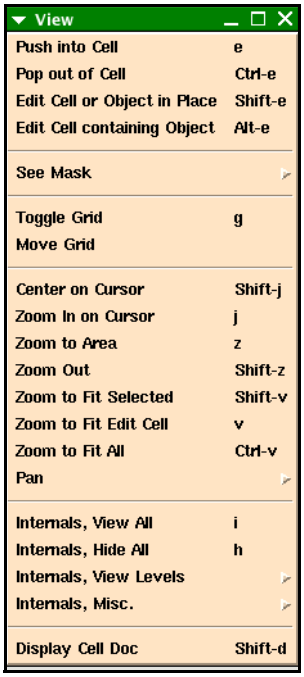
Hotkey: **o**

The **Erase** command is similiar to the **Paint** command except that it erases the layer under the cursor.

- You first draw a box in the area where you want to erase layout.
- Then, move the mouse over the layer in the palette, or the layer/layers in the layout window, you want to erase under the box. Type the **Erase** hotkey **o**.
  - If you type **o** anywhere there is no layout, ***all*** layers under the box are erased.

View Menu

Figure 59: View Menu



Push into Cell

Hotkey: **e**

Push into the selected instance (See `Select` menu for info on how to select cells). If an instance is visible, it can be selected and pushed into for editing no matter how far down in the hierarchy it exists. You can also iteratively push down through the hierarchy. MAX remembers which cells you have pushed into so that when you use the `Pop` command it reverses the order of the `Push` commands.

- If a Gcell is selected, the `Push` command brings up the `Edit Properties` (page 67) form for the Gcell.
- If a polygon is selected (see `Add Polygon` on page 59), you are put into the Edit Polygon mode where you move, add, or delete vertices.

Pop out of Cell

Hotkey: ***Ctrl-e***

Pop back to the previous edit cell or the root (top level) cell.

Edit Cell or Object in Place

Hotkey: ***Shift-e***

Edit the selected cell or object in place. For cells, this command works the same as the `Push into Cell` (page 79) command, except that the non-edit cells are visible.

The cell currently in edit mode will appear brighter. The paint in the non-edit cells is dimmed.. Dimming of the non-edit cell paint can be turned off with the `Dim Non-Edit Cells` (see page 43) toggle button in the `Display Options`.

The `Pop out of Cell` (page 80) command can be used to pop back up the hierarchy.

- If a Gcell has been selected, `Edit Cell or Object in Place` brings up the `Stretch Selected Gcell` command allowing you to stretch the gcell. See `Stretching Gcells` (page 27) for more information on this procedure.
- If a polygon was selected, this command allows the user to edit the polygon. Refer to `Add Polygon` (page 59) for more information on editing polygons.

Edit Cell containing Object

Hotkey: ***Alt-e***

Edit in place the cell containing visible paint under cursor. This command is similar to the `Edit Cell or Object in Place` (page 80) command except that an instance does not have to be selected. This command can be useful if you want to edit some layout and you're not sure what cell it is in.

- Place the mouse over the layout you want to edit and type the **Alt-e** hotkey. The paint in the cell in edit mode appears brighter. The paint in the non-edit cells is dimmed.
- Dimming the non-edit cell paint can be turned off with the `Dim Non-Edit Cells` (see page 43) toggle button. The `Pop out of Cell` (page 80) command can be used to pop back up the hierarchy.

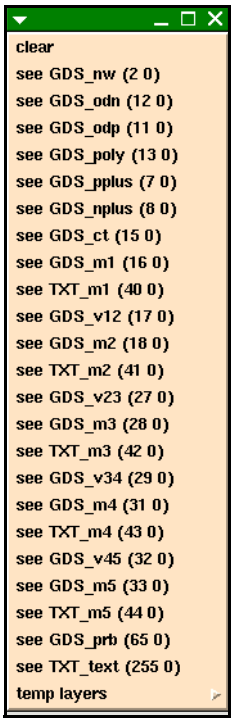
See Mask

Hotkey: ***None***

`See Mask` lists all of the layers which will be output with the `Export File` (page 37) command when it outputs GDSII. The layers below the line are temporary layers defined in the technology file. Viewing them can be useful when debugging a technology file.

- Clicking on one of the mask layers in the `Mask` menu (see Figure 60) draws that layer in MAX on the feedback layer (See "`Editing Special Layers`" on page 15).
- The `clear` option clears all mask layers.
- The `Mask` menu changes based on the technology you are using.

Figure 60: View Menu: See Mask Sub-Menu



- Selecting `clear` will clear any mask layers which have been displayed. This also clears the ruler and DRC violations from external tools.

`Toggle Grid` *Hotkey:* ***g***

This command toggles on or off viewing of the grid.

`Move Grid` *Hotkey:* ***None***

Once in this mode, hold down **Button-1** to move the grid. This is useful if you want to move the grid to overlap a specific corner on a rectangle.

`Center on Cursor` *Hotkey:* ***None***

Change the center of the layout (pan) to the location of the cursor. If you select this command from the menu, you will the be prompted to click with **Button-1** at the point in the layout you want centered in the edit window.

- You can specify a hotkey for this command in the `Hot-Keys` (page 50) options pop-up.
  - If you have a hotkey defined, first move the cursor to the desired location for the center and then type the hotkey.

`Zoom In on Cursor` *Hotkey:* ***j***

Zoom in by 2x centered on the current location of the cursor. This command works best if you use the hotkey.

- Position the cursor over the point you want to be the center of the screen and then use the hotkey **j**.

`Zoom to Area` *Hotkey:* ***z***

Zoom in to a specified region of layout. Select `Zoom to Area`. The cursor now becomes a hand.

- Hold down **Button-1** and drag out a box around the region you want to zoom in on.

`Zoom Out` *Hotkey:* ***Shift-z***

Zoom out by 2x on the layout. The center of the layout remains the same.

`Zoom to Fit Selected` *Hotkey:* ***Shift-v***

Zoom to fit the selected geometry(s) in the layout window.

`Zoom to Fit Edit Cell` *Hotkey:* ***v***

Zoom to fit the current cell in the layout window.

- If you have done an `Edit Cell or Object in Place` (page 80), the `Zoom to Fit Cell` command will zoom to fit the edit cell.
- If you have not edited a cell in place, then this command does the same as the `ZOOM to Fit All` command.

`Zoom to Fit All` *Hotkey:* ***Ctrl-v***

Adjust view to see entire root cell.

- If you have edited a cell in place, this command will zoom out to view the top level cell.

`(Zoom Wheel)` *Hotkey:* ***Mouse-Wheel-Up/Down***

Zoom in and out of the MAX window by using the mouse scroll wheel.

- Scrolling forward will zoom in; scrolling backward (toward you) will zoom out.

`Pan` *Hotkey:* ***<Keypad Arrows>***

Using the arrow keys on the keypad (usually on the far right side of the keyboard) you can pan up, down, left, and right. You generally won't have keyboard arrows on a laptop, just the regular arrows.

- You can also pan up and down the MAX window by holding down the **Ctrl** key while scrolling with the mouse wheel. **Ctrl-Scroll-Up** moves the MAX window upwards; **Ctrl-Scroll-Down** moves the window downwards.
- Another way to pan is to hold down the middle mouse button (scroll wheel on some mice) and pan around the layout by moving mouse.

`Internals, View All` *Hotkey:* ***i***

Show internals for all instances in the current cell all the way down to the bottom level of hierarchy. .

`Internals, Hide All` *Hotkey:* ***h***

This is the opposite of `Internals, View All`. The internals of all instances in the current cell will be "turned off" or hidden.

`Internals, View Levels`

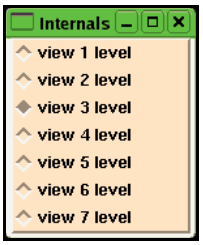
Hotkey: ***None***

The easiest way to use this command is to tear off the `View Levels` menu. Your menu should look something like Figure 61.

- If you have just started MAX, opened a cell, and viewed all internals, the toggle button will be on the bottom level of hierarchy. In this design, as you can see from Figure 61, there were only 3 levels of hierarchy.
- If you do an `Internals, Hide All`, then the `View Levels` menu toggle will be on **view 1 level**. If you want to view levels other that all levels or top level, click on the appropriate **view level.**

This command can be useful if you're investigating a layout you are not familiar with. In general, this is not needed because MAX can quickly view all layer and all levels of hierarchy.

Figure 61: View Levels menu

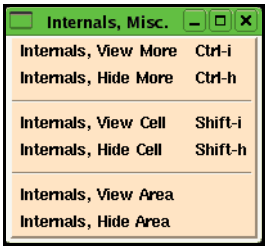


`Internals, Misc.`

Hotkey: ***None***

This is a sub-menu for additional Internals commands. The sub-menu will look like Figure 62.

Figure 62: View Internals, Misc. sub-menu



`Internals, View More`

Hotkey: ***Ctrl-i***

This views one more level of hierarchy. So, for example, if you are viewing 2 levels of hierarchy, this command would have you now viewing 3 levels down of hierarchy.

`Internals, Hide More`

Hotkey: ***Ctrl-h***

This command is the opposite of `Internals, View More`. If, for example, if you are viewing 3 levels of hierarchy, this command would have you now viewing 2 levels down of hierarchy.

`Internals, View Cell`

Hotkey: ***Shift-i***

Show internals of selected instance(s) one hierarchy level down.

- The easiest way to use this command is to use the `Select Cell` command (hotkey: **f**) to select an instance of a cell.
  - Then use the `Internals, View Cell` command. One level of hierarchy down from the selected instance is now visible.
  - You can now select one of the visible instances and show the internals of that cell. This command can be used if more than one instance is selected.

MAX remembers which instances were "turned on", so if you use the `Internals, View Cell` command to view the internals, when you do another `Internals, View Cell` of the same instance MAX remembers the state of the sub-instances (whether they were visible or not).

`Internals, Hide Cell`

Hotkey: ***Shift-h***

Hide internals of selected instance(s).

- The easiest way to use this command is to use the `Select Cell` command (hotkey: **f**) to select an instance of a cell.
  - Then use the `Internals, Hide Cell` command. Now the level of hierarchy of the selected cell/s is hidden.

MAX remembers which instances were "turned on", so if you use the `Internals, Hide Cell` command to hide the internals, when you do another `Internals, View Cell` of the same instance MAX remembers the state of the sub-instances (whether they were visible or not).

`Internals, View Area`

Hotkey: ***None***

Show all internals for the instance(s) under the box all the way down to the bottom level of hierarchy. Any instance that is partially or completely enclosed by the box will have internals shown.

- You first need to draw a box around an area that you want the internals to be visible and then select `Internals, View Area`. This option will show (make visible) all levels of hierarchy.

`Internals, Hide Area`

Hotkey: ***None***

Hide internals for the instance(s) under the box. Any instance that is partially or completely enclosed by the box will have internals hidden.

- You first need to draw a box around an area that you want internals to be hidden and then select `Internals, Hide Area`.

`Display Cell Doc`

Hotkey: ***Shift-d***

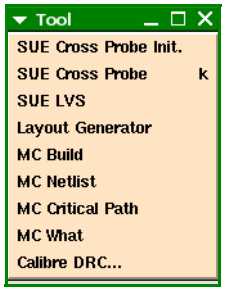
Display the **`<cell_name>.doc`** or **`<cell_name>.html`** in an editor or browser. If there is no **`.doc`** or **`.html`** file in the same directory as the **`.max`** file, you will be prompted to create a new **`.doc`** file.

The text editor is the default editor specified in the `File Menu: User Preferences General Setup... Pop-up` (page 47) form. The file suffixes for text and HTML files are also specified in the `General Setup` form.

Tool Menu

The `Tool` menu, shown in Figure 63, contains links to additional development tools from MMI, such as the *MegaCell Compiler* and the *Layout Generator*. The MegaCell Compiler (MCC) is described in more detail in the *Micro Magic, Inc. MCC MegaCell Compiler User Manual*. If you have not purchased the MegaCell Compiler, then the four MC commands will not be available in the `Tool` menu.

Figure 63: Tool Menu



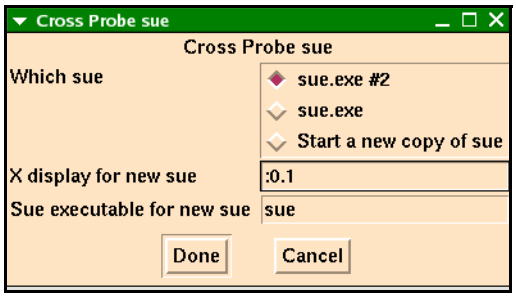
`SUE Cross Probe Init.`

Hotkey: ***None***

When you select this command, the pop-up shown in Figure 64 will appear. It lists all the SUE sessions currently running on your computer.

- If you already have the corresponding cell loaded in SUE, select that copy of SUE. Otherwise, start a new copy of SUE.

Figure 64: Tool Menu: SUE Cross Probe Init Popup



The `SUE executable for new sue` specifies which command to execute to run SUE. You may have a shell wrapper script that gets executed at your site. If you have multiple monitors on your computer, `X display for new sue` controls the initial SUE startup monitor.

SUE needs to find a matching schematic in your current directory, **`<cell_name>.sue`**. If your SUE schematic is in a different directory, it is easiest to first start up SUE with the schematic loaded.

Refer to Search Paths in MAX (page 126) for more details on how to crossprobe if cells have different names or are in different directories.

- SUE Cross Probe Init will first load the cell. If there is a matching schematic and sim netlist of the current cell in SUE, this command runs Gemini LVS to compare the layout to the schematic. Gemini LVS actually compares the **.sim** netlist files between layout and schematic, therefore a sim netlist must first be generated from the schematic before SUE Cross Probe Init is run.
- The nets that match between MAX and SUE will be highlighted in MAX using the highlight layer (see "Editing Special Layers" on page 15), and highlighted in SUE in red.

If SUE is already running, it must be able to find the cell in the loaded libraries.

- To view the results of Gemini LVS, look at the command window (the window from which MAX was started).

Crossprobing works only on flat designs, not hierarchical ones. The layout and schematic must both be flat, with no hierarchy.

SUE Cross Probe

Hotkey: ***k***

- First select a net or part of a net in MAX and then select SUE Cross Probe.
  - If there is a match for the selected net in SUE, that net will be highlighted in red in SUE and highlighted in MAX in the highlight color.
  - You can also cross probe from SUE to MAX. See the *SUE User Manual* for details.

SUE LVS

Hotkey: ***None***

Run GEMINI LVS for the current cell. This works without having to bring up the schematic in SUE.

- This command looks for the **.sim** netlist from the SUE schematic named **<cell_name>.sue.**
- It extracts a **.sim** netlist from MAX and calls the netlist **<cell_name>_lay.sim**.
- Finally, it runs Gemini and compares the two netlists. The results will appear in the command window (the window from which MAX was started).

Gemini LVS only works for flat designs, ones with no hierarchy.

Layout Generator

Hotkey: **None**

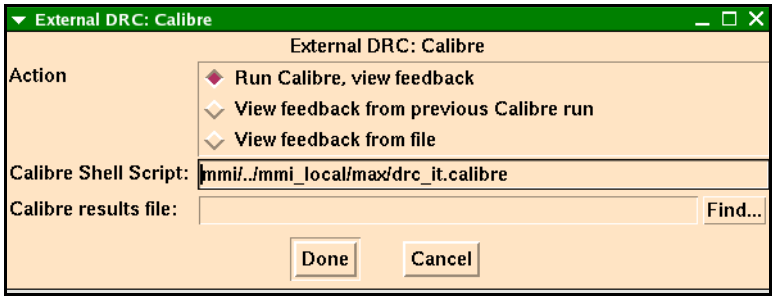
The layout generator converts a schematic into a layout containing fets with flylines to indicate connectivity. Refer to `Introduction To The MAX Layout Generator` (page 117) for detailed information.

Calibre DRC

Hotkey: **None**

This will run Mentor Graphics' *Calibre* DRC program on the current cell. It brings up the pop-up shown in Figure 65.

Figure 65: Tool Menu: Calibre DRC... Run External DRC Pop-up



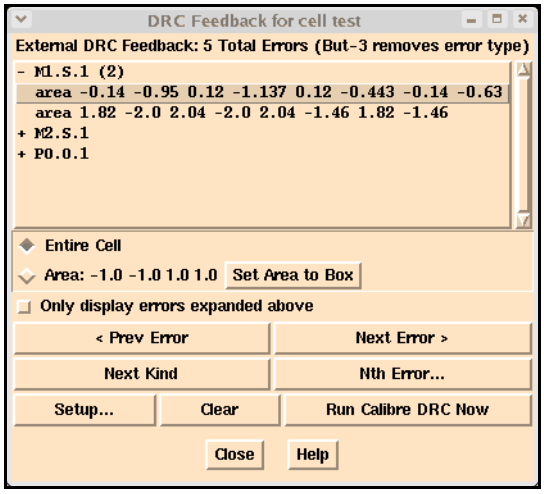
- If you click on `Run Calibre, view feedback`, MAX writes out a GDSII file for the current cell, then uses the **`drc_it.calibre`** script specified by `Calibre Shell Script` to run Mentor Graphics' *Calibre*.

  An example **`drc_it.calibre`** shell script is provided in: **`$MMI_TOOLS/mmi_local.sample/max`**.

  - If your system administrator followed the install instructions and copied over the **`mmi_local.sample`** directory, this script will be in **`$MMI_TOOLS/../mmi_local/max`**. This script must be edited to point to your *Calibre* DRC decks for your technology.
  - While *Calibre* is running, messages will print in the MAX Command Window (the shell window from which you started MAX).
  - Once *Calibre* has finished running, MAX displays the *Calibre* DRC errors in your layout using the `annotation` layer.

- If you select `View feedback from previous Calibre run`, MAX loads the DRC results from a previous *Calibre* run and brings up the `DRC Feedback` window. Refer to the `DRC Results` (page 100) command for more details on the MAX *Calibre* interface.
- If you select `View feedback from file`, MAX loads the DRC results from a previously saved *Calibre* run and brings up the `DRC Feedback` window as shown in Figure 66.

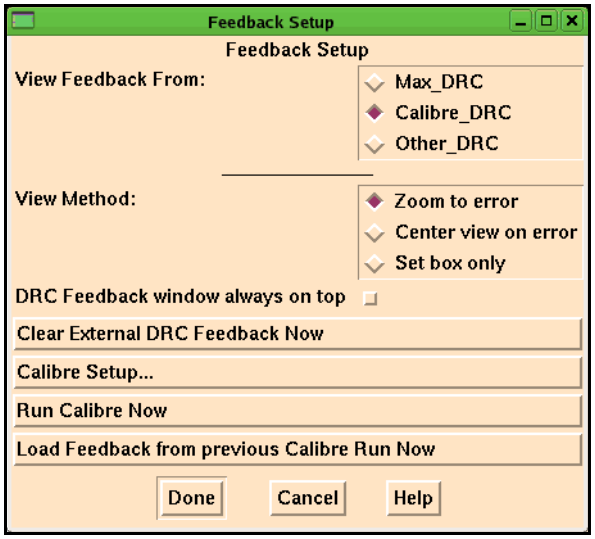
Figure 66: DRC Calibre feedback... popup



The `Next Error` and `Prev Error` buttons allow you to step through each of the DRC errors. You can skip to the `Next Kind` of error or to a specific error.

- If you click on `Setup`, the form in Figure 67 comes up. You can control if the DRC Results window shows DRC results from MAX or from a Calibre run.

Figure 67: DRC Feedback Setup... popup



The `View Method` specifies how you view the results when you click on `Next Error` or `Prev Error`.

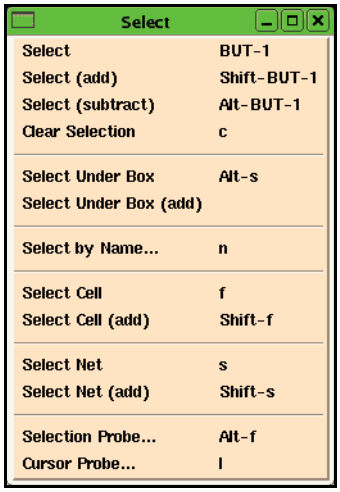
`Zoom to error` — zooms in close on the DRC error.

`Center view on error` — centers the DRC error on the window maintaining the current zoom.

`Set box only` — highlights the DRC error only.

Select Menu

Figure 68: Select Menu



`Select` *Hotkey:* ***Button-1***

Select objects by clicking with **Button-1**.

- You can select any gcell, instance, polygon, text or rectangle at the current level of hierarchy by clicking over the object with **Button-1**.
  - If there is more than one object under the cursor, you can step through each object by clicking **Button-1** at the same location.

You can control what is selectable using the palette. You can choose selectability by layer, cells (instances) or text.

- To select multiple objects, you can draw a box around them by holding down and dragging **Button-1**. Any instance, gcell or polygon that is at least partially enclosed by the box will be selected. Only the portion of a rectangle enclosed in the box will be selected. Clicking with **Button-1** on a rectangle selects the entire rectangle.
  - If you are selecting instances or gcells, it is generally easier to use the `Select Cell` (hotkey: **f**) command. The `Select Cell` command allows you to select cells throughout the hierarchy. The `Select` command only selects cells at the current level of hierarchy.

`Select (add)` *Hotkey:* ***Shift-Button-1***

Add to the current selection.

- By clicking **Shift-Button-1** over an object, it is added to the selection. When using `Select (add)` you can not step through each of the objects under the cursor. If you need this kind of control, set the selectability in the palette.

- Drawing a box around objects using **Shift-Button-1** adds objects under the box to the current selection.

`Select (subtract)`

Hotkey: ***Alt-Button-1***

Subtract objects from the current selection.

- You can subtract objects from the selection by either clicking on an object with **Alt-Button-1** or drawing a box while holding down **Alt-Button-1**.

`Clear Selection`

Hotkey: ***c***

Clear (deselect) all instances, gcells, rectangles or nets that have been selected. Rulers and other annotations will not be cleared.

- Clicking with **Button-1** in an area where there are no objects also deselects them.

`Select under Box`

Hotkey: ***Alt-s***

Selects paint (portions of rectangles) enclosed by the box, as well as any cells, gcells, groups or polygons which are either partially or entirely enclosed by the box.

- Use the `Make/move Box` (hotkey: **b**) to draw a box.

`Select under Box (add)`

Hotkey: ***None***

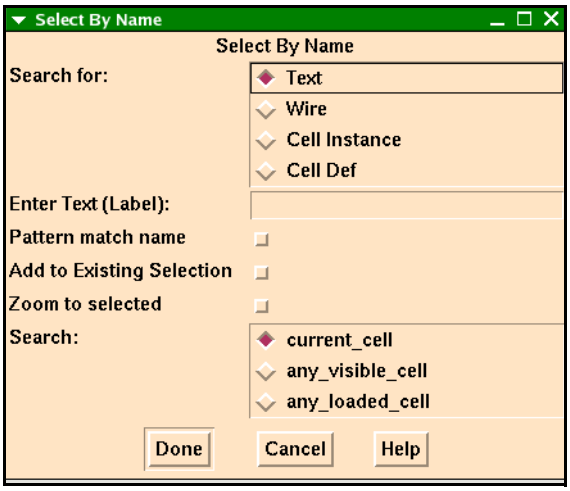
Add to current selection by adding paint enclosed by box as well as any cells, gcells, groups or polygons which are either partially or entirely enclosed by the box.

`Select by Name`

Hotkey: ***n***

Select any instances, cell defs, wires, or text by name. A pop-up form as shown in Figure 69 will appear where you enter the name of the element to select from the list. All instances of the specified selections will be highlighted.

Figure 69: Select Menu: Select by Name... Pop-up



Search for — Specify whether to search for **Text**, **Wire** (text and rectangles selected with a net), **Cell Instance** (all instances with specific name), or **Cell Def** (all instances of specified cell).

Enter Text (Label) — Use this field to enter a particular piece of text for which to search. This is and instance name, cell name, or text.

Pattern match name — If selected, MAX pattern matches using the following characters:

```
?      match any character;
*      match zero or more characters;
\c     match character c
[abc]  match any one of the characters in brackets;
[a-z]  match range of characters;
```

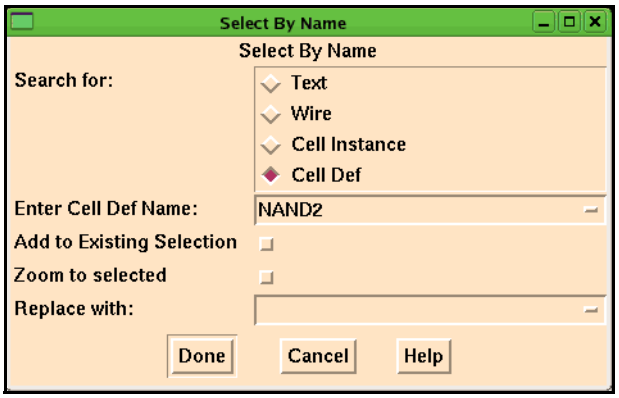
Add to Existing Selection — If selected, the text, instances, cells or nets are added to the selection. Otherwise, all other selections are removed and then the specified object(s) is selected.

Zoom to selected — If selected, MAX adjusts or zooms the view to center on the selection.

Search — Click to choose which cell you would like to have searched. **current_cell** specifies to look only at the current level of hierarchy. **any_visible_cell** specifies all cells for which internals are visible. **any_loaded_cell** specifies to search down through the hierarchy even if internals are not visible.

If Search for is specified as Cell Def, then the form changes to Figure 69. `Enter Cell Def Name` allows you to select lists all cells at the current level of hierarchy. You can also `Replace` all instances of this cell with a different cell.

Figure 70: Select Menu: Select By Name



`Select Cell` *Hotkey:* ***f***

Select the instance of a cell or a Gcell that is under the cursor.

- If there are multiple levels of hierarchy under the cursor, selecting the `Select Cell` command multiple times shuffles through the different levels of hierarchy. The `Select Cell` command starts at the lowest level of hierarchy under the cursor and moves up the hierarchy.
- For this command, you should use the **f** hotkey.

`Select Cell (add)`

Hotkey: ***Shift-f***

Add instance or gcell under cursor to the current selection.

`Select Net`

Hotkey: **s**

Select an entire net. MAX traces the connectivity from the specified rectangle (the rectangle under the cursor) through the hierarchy.

- If `Select Net` is invoked from the menu, click with **Button-1** on a geometry. The connectivity is traced and highlighted.
- To use the hotkey (**s**), move the mouse over a geometry and then type **s**. The entire net is highlighted through the hierarchy.

All the unique labels associated with the net throughout the expanded hierarchy are listed in the message area. If there are multiple geometries under the mouse, you can toggle through them by typing **s** again without moving the mouse.

Only geometries which are visible are highlighted. To select a net through all levels of hierarchy, `internals` (see page 82) must be viewed.

`Select Net (add)`

Hotkey: ***Shift-s***

This allows you to select and add more nets to the one initially selected or to any selection.

- Choose `Select Net (add)`, place the mouse over the net you wish to add to the current selection and click **Shift-s**.

`Selection Probe`

Hotkey: ***Alt-f***

Brings up the selection probe which displays all objects that have been selected as shown in Figure 71. What gets displayed in the probe is controlled by the two toggle buttons.

`Button-1 selects in:` controls what gets selected when you click or draw a rectangle with **BUT-1**.

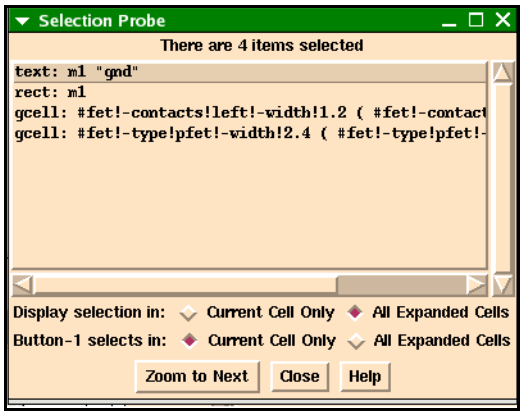
- The default is `Current Cell Only` which only selects objects that are at the current level of hierarchy.
- `All Expanded Cells` allows objects at any visible layer of hierarchy to be selected.

Setting `Button-1 selects in` to `All Expanded Cells` should only be used when investigating the hierarchy. Using this setting at other times can be very confusing when editing layout.

`Display selection in:` controls what is displayed in the `Selection Probe`. This only affects the results if `Button-1 selects in:` is set to `All Expanded Cells`.

- `Current cell only` displays only rectangles, text and polygons at the current level of hierarchy.
- `All Expanded Cells` displays rectangles, text, and polygons throughout the hierarchy.

Figure 71: Selection Probe Menu

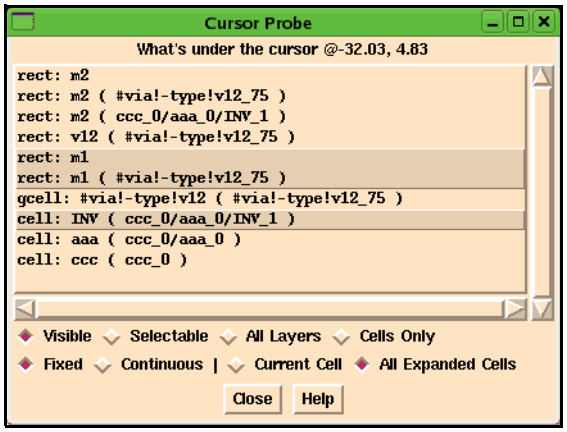


`Cursor Probe` *Hotkey: I*

The `Cursor Probe` displays a list of all the visible objects under the cursor in the current edit cell, as shown in Figure 72. Objects that are currently selected are highlighted in the `Cursor Probe` list. In Figure 72, cell **INV**, and rectangles **m1** and **m1** in a **via** are selected.

- Mouse **Button-1** over an item in the `Cursor Probe` window selects or deselects that item in the MAX window.
- Mouse **Button-2** over a cell, rectangle, or polygon in the `Cursor Probe` list pushes in place into the cell containing the item.

Figure 72: Select Menu: Cursor Probe Pop-up



The top options contol what is displayed in the Cursor Probe.

Visible — displays everything that is visible.

Selectable — displays only layers or cells which specified as selectable in the palatte.

All layers — shows everything including gcells, cell instances, text and rectangles.

Cells Only — shows only cell instances and gcells.

Fixed — means that what is displayed only changes when the Cursor Probe (hotkey: **I**) is selected.

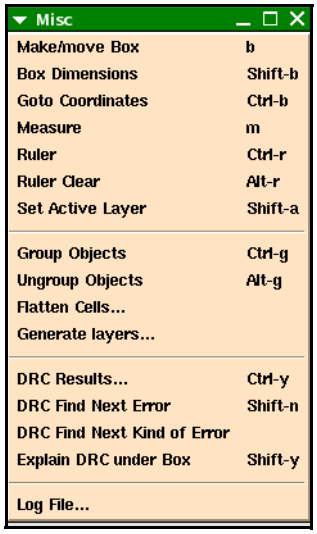
Continuous — means that the Cursor Probe information is continuously updated as you move the cursor.

Current Cell — only shows objects at the current level of hierarchy.

All Expanded Cells — shows objects in all visible (internals viewed) cells.

Misc Menu

Figure 73: Misc Menu



Make/move Box *Hotkey:* **b**

This is used to draw a box. (See "The Box" on page 25.)

- Type the hotkey **b** or select Make/move Box from the menu, then hold down and drag with **Button-1** to draw the box.
  - Simply holding down and dragging **Button-1** creates a box, but *also selects* what is underneath the box.

If there is already a box drawn, you can use this command to graphically move or resize the box:

- If you move the cursor over a corner or edge of the box, then click and drag with **Button-1**, the box is resized.
- If the cursor is anywhere other than over a corner or edge, **Button-3** moves the box.

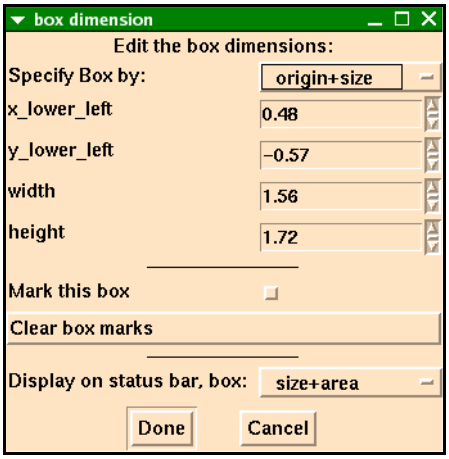
Box Dimensions

Hotkey: ***Shift-b***

Show and edit the box dimensions and location. A pop-up form appears (see Figure 74) which displays the x and y dimensions for the box.

- You can either specify the **size** by giving the width and height, or specify the **corners** by giving the X and Y coordinates of the box. This is useful if you want to have a box with exact dimensions at an exact location.

Figure 74: Misc. Menu: Box Dimensions



Specify Box by:

- **origin+size** — Specify the lower left corner for the box and the width and height.
- **corners** — Specify the lower left and upper right corners.
- **center** (x_center, y_center, width, height) — Specify the center for the box and the width and height.

x_lower_left Button Enter the coordinate value for the lower left corner of the box in the x orientation.

y_lower_left — Enter the coordinate value for the lower left cornef of the box in the y orientation.

width — Enter the value in units for the width of the box.

height — Enter the value in units for the height of the box.

Mark this box — Marks your created box with a dimmed outline. This allows you to have multiple boxes drawn.

Clear box marks — Removes any of the previously marked boxes.

Display on status bar, box:

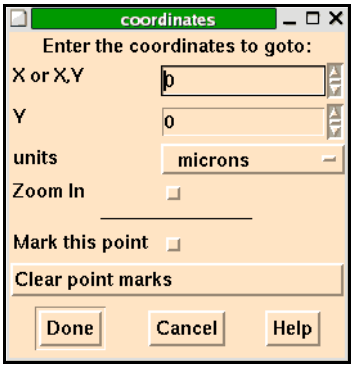
- **size** — Display the size only of the box in the Box Area Display (lower right corner of the MAX window).
- **size+area** — Display the size and area of the box in the Box Area Display (lower right corner of the MAX window).
- **corners** — Display the lower left and upper right corners in the Box Area Display.
- **origin+size** — Display the lower left corner and the width and height in the Box Area Display.
- **disable** —Turn off the Box Area Display.

Goto Coordinates

Hotkey: ***Ctrl-b***

Go to the coordinates specified in the pop-up box. You specify the X and Y coordinates as shown in Figure 75.

Figure 75: Go To Coordinates popup....



Units — are either microns, nanons, or half_nanons.

Zoom In — zooms in on the coordinate. Otherwise, the layout is centered around the coordinate at the current zoom level.

Mark this point — leaves a mark at the coordinate until **Clear point marks** — is selected.

Measure

Hotkey: ***m***

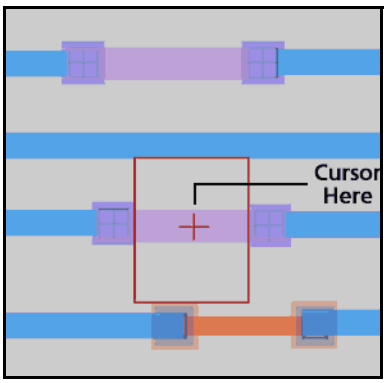
Place the box in the largest area unoccupied by visible layers. The dimensions of the box are displayed in the `Box Area Display` (page 22).

- In the example shown in Figure 76, if the cursor is placed over the M2 wire as shown, the box is drawn out until it finds a layer other than M2, or any visible layer if there is empty space in between.

  In the x direction, MAX searches for any non-M2 layer.

  In the y direction, since there is blank space before finding another layer, it searches for any layer.

Figure 76: Misc. Menu: Measure Command Example



`Ruler` *Hotkey:* ***Ctrl-r***

Draw a ruler to show the distance between any two points.

- Select the `Ruler` command.
  - Click once with **Button-1** to start the ruler. **B**
  - **utton-3** ends the ruler.
  - **Button-2** brings up the `Ruler Setup` form. This is the same form found under `File, User Preferences`.
- To clear the ruler, use the `Ruler Clear` command.
  - Holding down the **Shift** key allows the ruler to be drawn at any angle. Without the **Shift** key the ruler is locked to the vertical or horizontal.

`Ruler Clear` *Hotkey:* ***None***

Clear all rulers.

`Set Active Layer` *Hotkey:* ***Shift-a***

This allows you to set a new active layer by simply pointing to a layer in the layout window and using a hotkey, instead of clicking the `Active Layer` button in the palette.

- To select a new active layer, position the mouse pointer over the desired layer and press **Shift-a**.
  - The new active layer will show in the palette.
  - This is the layer on which polygons will be drawn and wires started by default.

`Group Objects` *Hotkey:* ***Ctrl-g***

Put the selected objects into a group.

- You can group rectangles, portions of rectangles, polygons, circles, Gcells, instances and other groups.

- A group can be selected and moved just like a cell instance. If you `Push into Cell` or `Edit Cell or Object in Place`, the group can be edited like a cell instance.

`Ungroup Objects` *Hotkey:* ***None***

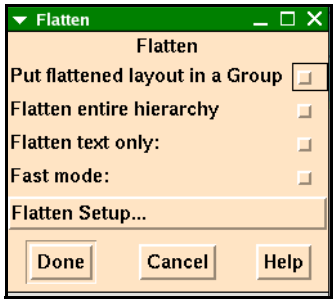
Ungroup selected groups.

`Flatten Cells` *Hotkey:* ***None***

Flatten the selected cell(s). A pop-up form appears, as shown in Figure 77. By default, this command removes the level of hierarchy of the selected cells. This is sometimes called an explode operation.

- The cell(s) selected must be at the edit cell level of hierarchy. The level of hierarchy associated with the selected cell is removed.

Figure 77: Misc. Menu: Flatten Cells... Pop-up



Put flattened layout in a Group — If set, the flattened layout will be placed into a new group sub-cell, which can be moved or edited more easily.

Flatten entire hierarchy — If set, the flatten operation will be run on subcells until entire cell hierarchy is flattened. Otherwise, only the selected cells themselves will be flattened; only one level deep.

Flatten text only — If set, only text (labels) will be flattened, not cells. In other words, text in the sub-cell(s) will be copied into the current cell, according to the options set in the `File Menu: User Preferences Display Options... Pop-up` (page 43) menu, but the cells themselves will not be affected.

Fast mode — This a faster method for flattening large cells. It does not leave the result selected, and will not preserve flylines.

Flatten Setup... — Opens a pop-up menu allowing you to specify the desired flattening options, as shown in Figure 78. The menu specifies what happens to labels in a cell when you flatten it.

- Options for flattening `Text types` include: `local, global, input, output, inout, comment`, and `delete`.
- Options for `name` are either `preserve` or `flatten`.

For each type of label (input, output, inout, global, local, comment) you can either delete that type of label, or change it into any other type of label.

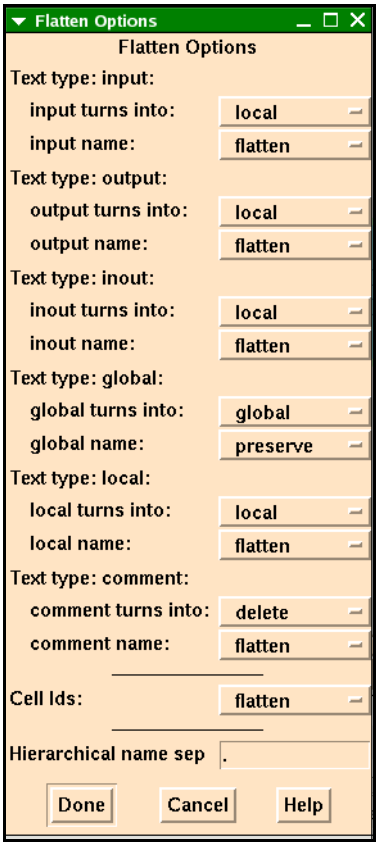
- For example, if you specify "`global turns into: global`" then global labels in the flattened cell will be copied into the parent cell wthout changes.

- If you specify "`global turns into: delete`", then all global labels will be deleted during flattening.

You can also specify whether the label names are preserved or flattened. If they are preserved, they may conflict with other existing labels with the same name.

- To avoid this, flattening the label names will cause the labels to be renamed to something like: "`cellname.labelname`" where "`cellname`" is the *instance* name of the cell that was flattened, and "`labelname`" was the *label* name in the cell that was flattened.

Figure 78: Misc. Menu: Flatten Cells... Flatten Setup Options



Cell Ids — If set to "**flatten**" Cell Ids are flattened by prepending the name of the parent cell to this name. If set to "**preserve**", the Cell Ids are retained, but may be made unique by appending "_n" where n is a number.

Hierarchical name sep — Allows you to enter a unique character to act as a hierarchical name separator.

Generate Layers

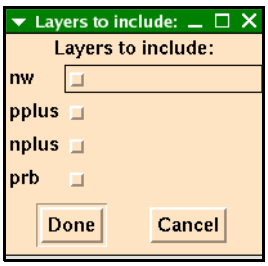
Hotkey: **None**

Auto-generate the specified layers by using DRC rules from the current technology file to surround existing geometry in the current cell.

New layers are placed into an overlay cell that can then be flattened, if desired.

There must be layer generation statements in the MAX technology file in order to be able to auto-generate layers. The form shown in Figure 79 will open, with toggle buttons for the various selections.

Figure 79: Misc. Menu: Generate Layers... Pop-up

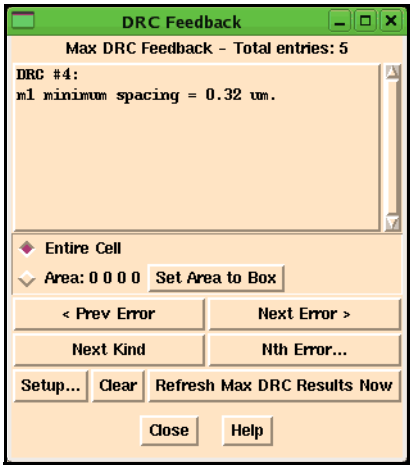


`DRC Results` *Hotkey:* ***Ctrl-y***

Display the DRC error feedback window. This is used for both displaying the MAX DRC errors and the DRC errors from an external tool. Currently, Mentor Graphics' *Calibre* is the only external DRC tool supported with this interface.

- When you select `DRC Results`, the form in Figure 80 appears.

Figure 80: Misc. Menu: DRC Results... Pop-up



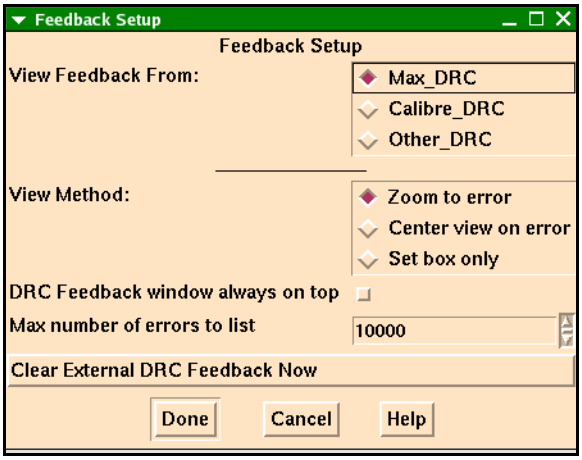
- If you click on `Next Error`, the first MAX DRC error will be displayed in the feedback window. MAX will also zoom in on the error. The text of the DRC error is also displayed in the MAX `Message Area` (page 12).

  `Next Kind` — will skip to the next type of error.

  `Nth error` — brings up a form where you can specify the error to review.

- Clicking on `Setup` will bring up the form shown in Figure 81.

Figure 81: Misc. Menu: DRC Results... MAX DRC Setup

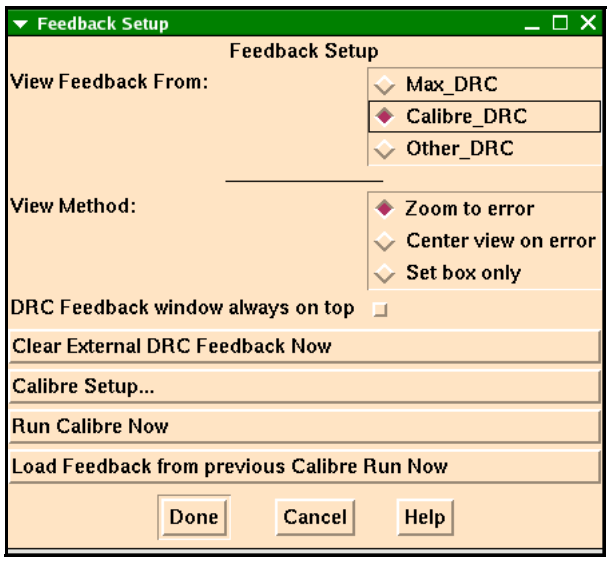


- Click the button to select which type of DRC feedback to view: `MAX` , `Calibre`, or `Other`.
  - You can choose to `Zoom to error` (default) which zooms in on the DRC error each time you step to a new error.

    `Center view on error` keeps the current zoom level, but centers the view on the DRC error each time you step to a new error.

    `Set box only` does not touch the current zoom, it only highlights the DRC error each time you step to a new error.
  - When the `DRC Feedback window always on top` toggle button is turned on, the DRC feedback window will stay resident on top of the MAX window. This can be helpful if you're editing layout to correct DRC errors, but want to leave the `DRC Feedback` window up. This way it will not get "lost" behind the MAX window.
  - The `Max number of errors to list` allows you to set a top end for the list of errors.
- If you click on the `Calibre_DRC` toggle, the setup form will update to look like Figure 82.
  - `Clear External DRC Feedback Now` will clear/erase the errors from *Calibre* which were displayed when `Load Feedback from previous Calibre Run Now` was selected.

Figure 82: Misc. Menu: DRC Results... External DRC Setup



- Clicking on `Calibre Setup`, causes the popup form in Figure 83 will appear. This is where you specify the name and location of the script which runs Calibre DRC.

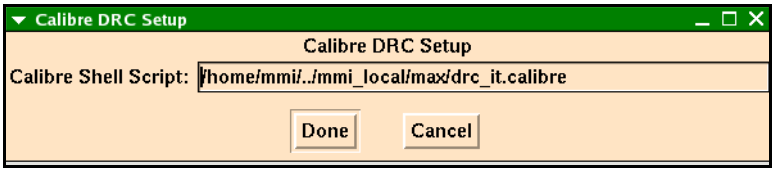
- Selecting `Run Calibre DRC Now...` tells MAX to write out a GDSII file for the current cell, then use the **`drc_it.calibre`** script specified by `Calibre Shell Script` to run Mentor Graphics' *Calibre*.

  An example **`drc_it.calibre`** shell script is provided in the **`$MMI_TOOLS/mmi_local.sample/max`**.

  - If your system administrator followed the install instructions and copied over the **`mmi_local.sample`** directory, this script will be in **`$MMI_TOOLS/../mmi_local/max`**. This script must be edited to point to your *Calibre* DRC decks for your technology.

While *Calibre* is running, messages will print in the MAX Command Window (the shell window from which you started MAX).

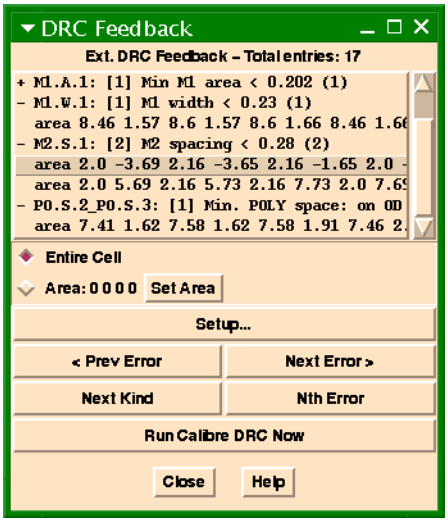
Figure 83: Misc. Menu: DRC Results... Calibre Setup



Once *Calibre* has finished running, the feedback (DRC errors) will automatically be loaded into MAX. MAX displays the *Calibre* DRC errors on your layout using the `annotation` layer.

- Click on `Done` in the `Feedback Setup` form. The *Calibre* DRC errors will be displayed in the `DRC Feedback` form, as shown in Figure 84.

Figure 84: Misc. Menu: DRC Results... View Calibre Results



All of the errors are grouped by type of error.

- If you click on the "+" next to an error type, the list is expanded to show all of the DRC violations for that type of error.
- Clicking on one of the errors will zoom you in on the error. The text of the DRC error (from *Calibre*) is displayed in the MAX Message Area.

You can display DRC errors for the `Entire Cell` (default) or specify an area.

- To specify and area in which to view DRC errors (both for MAX and *Calibre* errors), click on `Set Area` and then draw out a region with **Button-1**.

  This only affects the viewing of DRC errors. *Calibre* DRC is always run on the entire cell.

`DRC Find Next Error`

Hotkey: ***Shift-n***

Step through each of the DRC errors in MAX. As each DRC error is highlighted, MAX also zooms in on the error. The DRC error explanation is printed in the `Command Window` (page 22) and in the MAX Message Area. This has the same functionality as clicking on `Next Error` in the DRC Feedback window.

`DRC Find Next Kind of Error`

Hotkey: ***None***

Step to next type of error. For example, if you are looking at `Metal1 min. spacing errors`, this command would skip to the next type of error (for example, `Metal2 min. spacing`). This command has the same functionality as clicking on `Next Kind` in the DRC Feedback window.

When viewing errors from the MAX built-in DRC, they are not sorted by error kind, so this command scans until it finds a new kind of error. This does not mean that all the previous kind of errors have been viewed.

`Explain DRC under Box`

Hotkey: ***Shift-y***

Gives a text explanation of the kind of DRC error currently selected in the box.

- Draw a box over one or multiple DRC error(s) in MAX (showing as white dots).
- Select `Explain DRC under Box` and the DRC information will be printed in the MAX Command Window (the window from which you started MAX).
  - If multiple DRC errors are under the box, all of the DRC violations will be listed.
  - If there is only one DRC error under box, the DRC error information is also displayed in the **Message Area**.

This only explains MAX DRC errors, not errors from an external DRC tool such as *Calibre*.

`Log File`

Hotkey: ***None***

Put log of MAX into a log file. This allows you to keep track of what commands have been run in the current MAX session.

Local Menu

This menu contains **Tcl**-based MAX functions. You can add your own **Tcl** functions to this menu. An example of this would be running a batch DRC tool on the current cell. The **Tcl** code for the examples in the `Local` menu is found in:

```
$MMI_TOOLS/../mmi_local/max
```

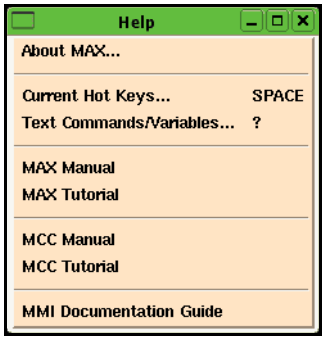
This menu remains empty until you have added functions.

Figure 85: Local Menu (Empty - No Functions)

Help Menu

Any time the mouse is over a menu option, a basic description of the command is displayed in the `Message Area` (page 12). The `Message Area` also displays the mouse functions in the current mode. The `Help` menu (see Figure 86) provides you with additional help.

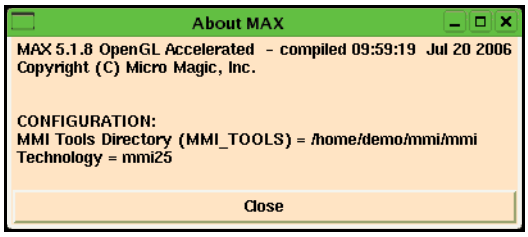
Figure 86: Help Menu



`About MAX` *Hotkey:* ***None***

Brings up a form (see Figure 87) with current information about MAX. `About MAX` lists the version of MAX used, the location of the MAX software, and the current technology.

Figure 87: Help Menu: About MAX... Help Form



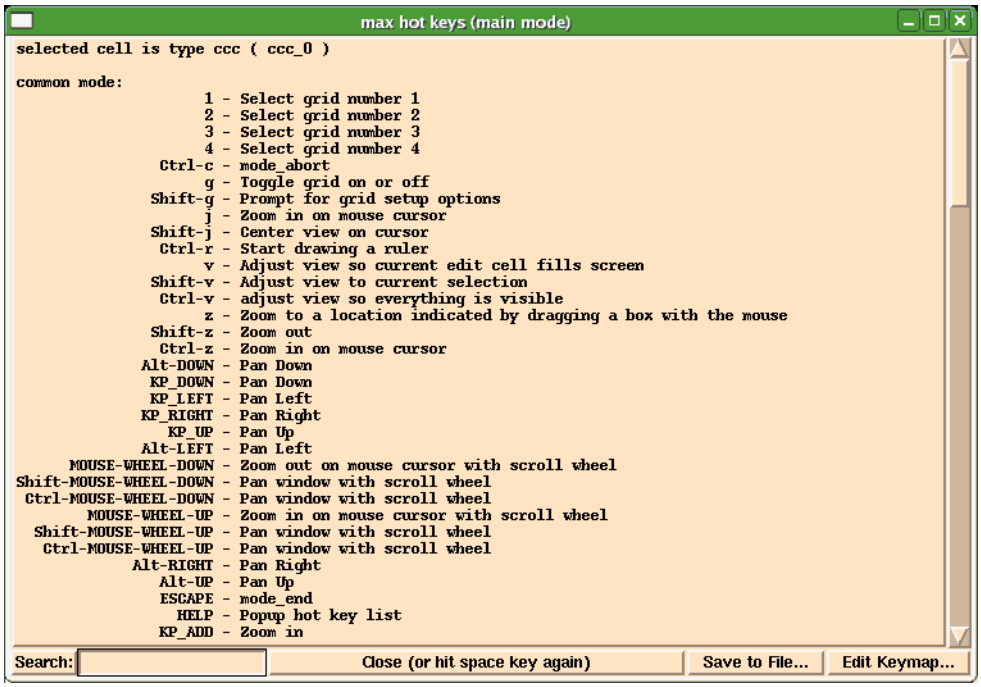
Current Hot Keys

Hotkey: **Space**

Hitting the **Space** bar in any mode brings up the list of hotkeys for that mode along with a description of each hotkey (see Figure 88).

- The `Save to File` button writes out this hotkey information into a text file.
- The `Edit Keymap` button brings up the form for editing the MAX hotkeys (see page 50).

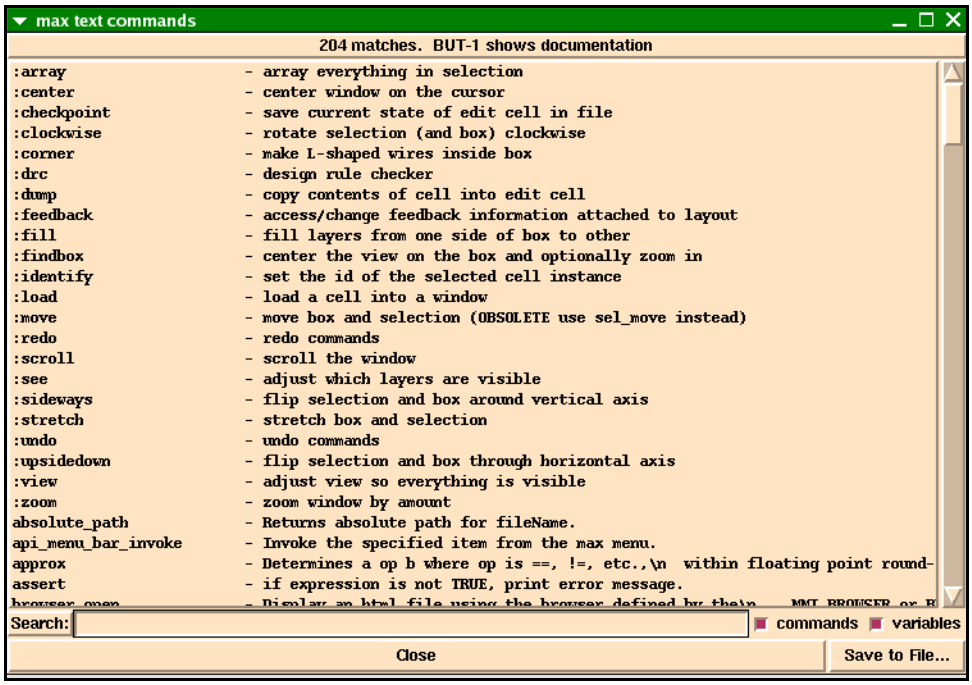
Figure 88: Help Menu: MAX Current Hot Keys List



Text Commands *Hotkey:* **?**

Brings up a listing of all text commands, as shown in Figure 89.

Figure 89: Help Menu: MAX Text Commands/variables...



- Use the `Search` field to search for specific functionality. This command searches through the text command names and descriptions for the text string typed in.
- Click on the text command with mouse **Button-1** to see a description of the syntax. The syntax of the text commands is also listed in Appendix B, “Text Commands”.
  - To execute a text command, you can type it into the `Command Window` (page 22).

The text commands can also be part of a **Tcl** script. Example API scripts can be found in:

```
$MMI_TOOLS/mmi_local.sample/max/api_examples.
```

MAX Manual *Hotkey:* ***None***

Brings up the MAX User Manual (this document) in your web browser.

MAX Tutorial *Hotkey:* ***None***

Brings up the MAX Tutorial in your web browser. The tutorial is a step by step, hands-on introduction to MAX.

MCC Manual *Hotkey:* ***None***

Bring up the MCC User Manual in your web browser.

MCC Tutorial

Hotkey: ***None***

Brings up the MCC Megacell Compiler Tutorial in your web browser. The tutorial is a step by step, hands-on introduction to MCC.

MMI Documentation Guide

Hotkey: ***None***

Brings up an HTML document that points to the complete set of documentation for Micro Magic, Inc. This can also be brought up from any shell window by typing **`mmidoc`**. It has pointers to the MAX Tutorial and this manual, among others.

Chapter 4 The MAX Wire Tool

Introduction To The MAX Wire Tool

The MAX Wire Tool facilitates drawing new wire connections. Wires can be drawn in any metal or poly layer, and include vias to switch between layers. The layers used by the Wire Tool are specified in the MAX Technology File. (See "MAX Technology Targeting" on page 129.)

The Wire Tool does not modify existing wires; however, you can delete part of an existing wire and rapidly draw new wires using the Wire Tool. You can also Stretch (page 75) a segment of the wire. The Edit Wire (hotkey: **Ctrl-w**) command allows you to stretch segments of wires and have vias automatically stretched as well.

Wires are drawn one at a time by entering Wire Mode, optionally selecting the layer and/or wire width, and then using the mouse to point to the wire vertices. You can also draw buses using the Add Wire Bus command.

Starting a Wire

Before you can draw any wire, you must be in Wire Mode.

- To create a wire, use the **w** hotkey or select Add Wire (page 58) from the Edit Menu.

  This places MAX in Wire Mode, which is indicated by the mouse cursor changing to a pointing finger, and the status message (at the top of the screen) changing to wire mode.

After you enter Wire Mode, but before you start drawing the wire, you can change the layer and/or the width of the wire you will draw. If you do not select a layer, the wiring tool will use the Active Layer (page 12). If the active layer is set to auto, then Auto Layer Selection will be used, meaning it will pick one of the layers under the cursor when you start drawing the wire.

Changing the layer used to draw the wire

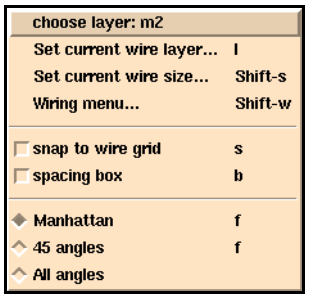
The active layer is specified in the palette. If there is no wireable layer under the cursor and the active layer is set to auto, the Default Layer (see "Wire Setup" on page 54) from the Wiring Menu (page 113) is used.

- Alternatively, you can select a layer in one of the following ways:
  - Place the mouse over the desired layer in the palette and press **Button-3**.
  - Place the mouse over any paint in the design window, hold down **Button-2** and select choose layer: <layer> from the pop-up menu. (see Figure 90).
  - Select Set current wire layer (hotkey: **l**) from the pop-up menu. A pop-up will appear where you can choose the layer in which to start.

- Change the `Active Layer` in the palette.
- Change the `Default Layer` in the `Wire` menu (described below).

The new layer will be reflected in the palette under the Active layer.

Figure 90: Wire Pop-up Menu Before Wire is Started



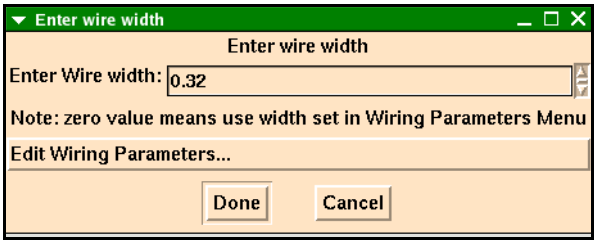
Changing Wire Width

- To change the wire width, use the **Shift-s** hotkey, or select `Set Current Wire Size` from the pop-up menu that appears when you press mouse **Button-2** in the MAX design window.

  The `Enter wire width` form will appear (see Figure 91). If the wire width is set to 0, it means use the default wire width for the layer.

  - The default wire width for each layer is specified in the `Wiring Parameters Menu` (page 114) and is initialized to draw minimum width wires. You can actually change the width of the current wire segment even after you have started drawing it, but you cannot change the width of wire segments you have already completed.

Figure 91: Edit Wire Width Form



You can also set the wire width to match the width of the existing wiring layer, if any, under the cursor.

- To do this, press the **Shift** key when you start the wire (using mouse **Button-1**).

Drawing a Wire

- ***To actually start drawing the wire***, point the mouse to the location where you want the wire to start and press **Button-1** (or **Shift-Button-1** to match the width of the layer under the cursor).

  You can then move the mouse to create the wire, which will follow the mouse movements.

- ***To create a vertex*** (bend in the wire) press **Button-1** again. Notice that the vertex slides along the previous wire segment so that you can move the mouse in any direction and the wire will follow.

  - To prevent the vertex from sliding, press the **a** key, or select `anchor vertex` from the pop-up menu that appears when you press mouse **Button-2** (see Figure 92).

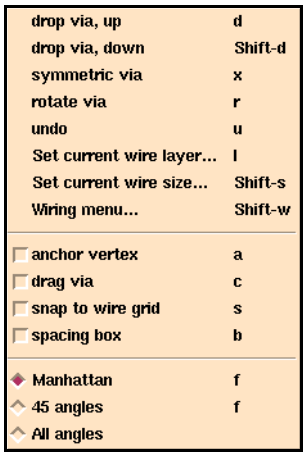
- ***To end the wire***, press **Button-3** over an existing metal or poly of the same type as the wire. You can also end the wire at any time with **Button-3** not over current wire layer, which allows you to exit wiring mode without completing a connection.

  When you end the wire by completing a connection, the wire tool automatically snaps the final wire segment to make the wire align properly with the existing rectangle.

  - To prevent this, end the wire by using **Ctrl-Button-3** at the wire end-point.

Once a wire has been started, the `Wire Menu` updates its selections, as shown in Figure 92.

Figure 92: Wire Pop-up Menu After Wire Has Been Started



- ***To change layers***, `drop a via` using either the **d** or **Shift-d** hot keys, or use the pop-up menu on mouse **Button-2**.

- ***To undo a wire segment or via***, use the **u** hot key, or select **undo** from the pop-up menu on mouse **Button-2**.

Example: If you are wiring in Met1 horizontally and `drop a via, down` (hotkey: **Shift-d**), the via down to poly still moves in the horizontal direction (see Figure 93). In this case, you are wiring the two left gates.

Having the via follow the mouse is helpful, for instance, if you are trying to line up the vertical poly wire with a gate.

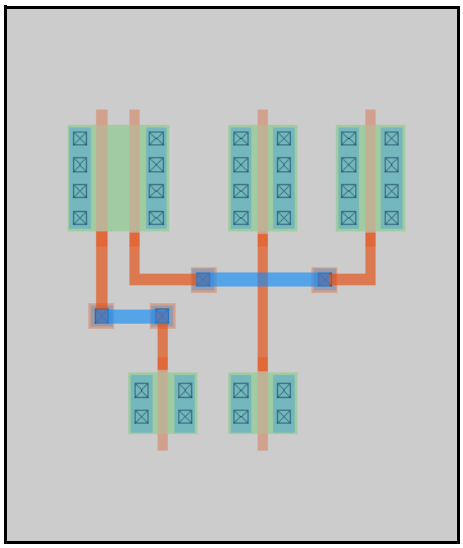
- ***To line up the wire with a gate***, move the mouse over the gate, click with **Button-3** on the gate and the wire and via automatically align.

If you instead want to continue with poly in the horizontal direction, you must anchor the via.

- ***To anchor vertex***, move the via to the desired position and select `anchor vertex` (hotkey: **a**).

In the example shown in Figure 93, this is how the 2nd and 4th pfet gates were wired.

Figure 93: Wire Tool in Use - Dropping Vias



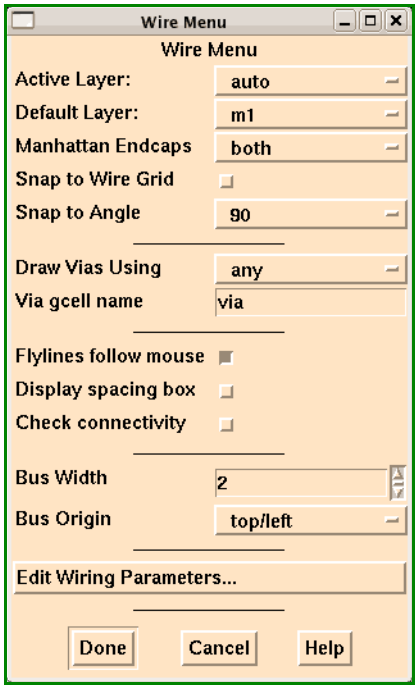
By default, the Wire Tool locks the angles to 90 degrees (Manhattan-style geometry). You can toggle between 45 degree and Manhattan with the **f** hotkey or by selecting the desired angle from the pop-up menu. You may also draw all-angle wires.

Interactive DRC does not work for 45 degree or all angle wires or circles. Interactive connectivity tracing in MAX does work, but the netlist extraction does not work for 45 degree or all angle wires or circles in this release.

Wiring Menu

The `Wiring Menu` (see Figure 94) can be opened using the **Shift-w** hotkey once in wire mode or select `Wire Setup` under `User Preferences` in the `File` menu.. It can also be accessed from the pop-up menu in Wire Mode. Each of the `Wiring Menu` options are described below.

Figure 94: Wire Menu



Active Layer — If you are in Wire Mode and select a layer from the palette using mouse **Button-3** or in the layout window with the **l** (lower case L) hotkey, the `Active Layer` is updated in this menu to reflect your layer selection.

Default Layer — The default layer is the layer that will be used if the `Active Layer` is set to **`auto`** and if no wireable layer is under the cursor when you start the wire.

Manhattan Endcaps — For drawing manhattan wires only. `Endcaps`, if selected, are drawn at one-half the width of the wire. If set to `both`, then MAX draws endcaps on the beginning and end of the wire of a size equal to half the width of the wire.

Snap To Wire Grid — This option constrains all wire vertices to snap to the wiring grid for the current layer. The wiring grid for each layer is specified in the `Wiring Parameters Menu` (page 114), and defaults to a one micron grid.

Snap To Angle — This is the default angle constraint for wire segments, and is normally 90 degrees.

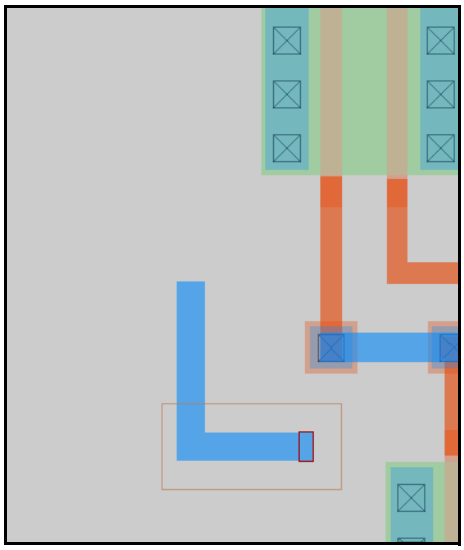
Draw Vias Using — If set to `any`, the code draws vias using the first method that works: if there is a via `gcell` installed, that will be used; if there is a via `subcell` found, that will be used; as a last resort, vias will be **painted** using rectangles. You generally want to use the `gcell` option.

Via gcell name — Specify the name of gcell to use for vias, and default properties in the form: **`-propname value`**. Only used if `Draw Vias Using` is set to `any` or `gcell`. You can use this to specify default properties for the via. For example, to make vias symmetric by default, set it to: **`via -symmetric 1`**. You can also change this if you have created your own via gcell for use by the wiring tool.

Flylines follow mouse — If on, any flylines attached to the net being wired will follow the end of the wire. When the wire is connected, the flylines disappear.

Display Spacing Box — If selected, this displays a simple aid to wiring consisting of a visible box around the wire as it is drawn (see Figure 95). The width of this visual box indicates the correct wire separation (as defined in the `Wiring Parameters Menu` (page 114)) for the current wiring layer. This is useful for spacing wires at more than the minimum width.

Figure 95: Wire Spacing Box



Check connectivity — If set, MAX will check wire connectivity when the wire is finished, and report any shorts. Connectivity is not traced through unexpanded cells. Only conflicts among labels/text in the edit cell are reported. Shorts are reported in the MAX command window (the window from which MAX was started).

Edit Wiring Parameters — Edit the parameters for the wiring tool. Refer to `Wiring Parameters Menu` (page 114) for detailed information on the wire parameters.

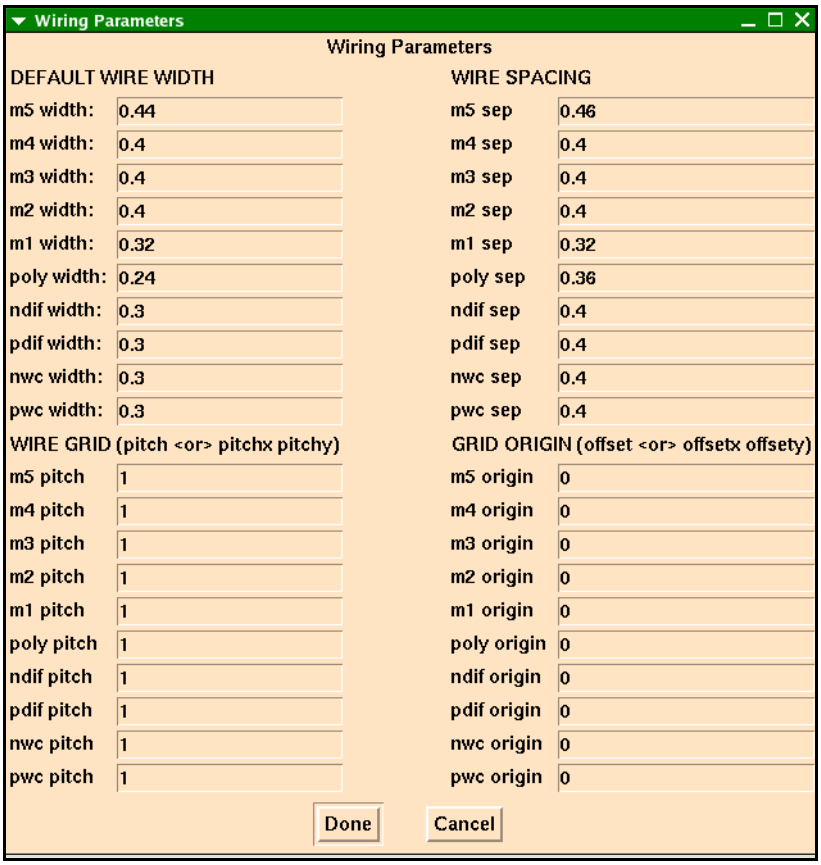
Wiring Parameters Menu

The `Wiring Parameters Menu` (see Figure 96) can be entered from the `Wire Menu` (obtained by using the **Shift-w** hotkey while in wire mode). This menu lets you specify the default wire widths, separation, and grid for the wires on each layer. The wire width cannot be less than the minimum allowed width for the current technology, or it is silently ignored.

The `GRID ORIGIN` is normally (0,0).

- You can specify either a single number, for example 0.5, which would place the origin at (0.5,0.5), or two numbers separated by a space, for example ".5 0", which would place the grid origin at (0.5,0).
- `WIRE GRID` specifies the grid to snap to if `snap to grid` is specified in the `Wire Menu`. `WIRE SPACING` is the value used for the spacing box.

Figure 96: Wiring Parameters Menu



Wire Tool Hotkeys

Below is a summary of the hotkeys for the wiring tool. The first set of hotkeys apply *after* the `Add Wire` command (hotkey: **w**) and *before* the wire has been started.

Table 2: Hotkeys Immediately After Wire Command but Before Wiring

Hotkey	Function
Button-1	Start a minimum width wire on the active layer.
Shift- Button-1	Start a wire is same width as geometry on the active layer.
Button-3	End Wire Mode.
Button-2	Wire tool pop-up menu
b	Set a spacing box to guide wiring. This is useful if your routing width is more than the minimum spacing.
f	Toggle between 45/90 degree angle wiring mode
s	Toggle snap to grid
Shift- s	Set current wire size
Shift- w	View the `Wire Menu`
ESC	End Wire Mode

The next set of hotkeys are active after the wire has been started.

Table 3: Hotkeys During Wiring Mode

Hotkey	Function
Button-1	Add a wire segment. Whenever you select Button-1, a new segment (corner) is added.
Button-3	End wire, aligned. If when you press mouse Button-2, the wire overlaps another wire of the same layer in the same direction, the wire will be automatically aligned.
Ctrl- Button-3	End wire, unaligned. The wire will end at location specified without doing any aligning.
Button-2	Wire Tool pop-up menu
a	Anchor/unanchor wiring vertex or via
b	Set a spacing box to guide wiring. This is useful if your routing width is more than the minimum spacing.
c	Change drag via method (toggles)
d	Drop a via, up one layer
Shift- d	Drop a via, down one layer
f	Toggle between 45/90 degree angle wiring mode
r	Rotate via, if any
s	Toggle snap to grid
Shift- s	Set current wire size
u	Undo last via or wire segment
Shift- w	View the `Wire Menu`
x	Symmetric via
u	Undo last via or wire segment
ESC	End Wire Mode. End wire at location of cursor.
Ctrl-c	Cancel wire. Delecte wire you are currently drawing.
(MOTION)	As you move the mouse, the end of the wire segment follows the cursor. The vertex slides to follow the mouse.

Chapter 5 The Layout Generator and MAX-LS

Introduction To The MAX Layout Generator

The MAX Layout Generator converts a schematic to a layout containing fets, with flylines to indicate connectivity. The fets are laid out in two rows with p-fets on the top and the n-fets on the bottom, and are optionally connected to power and ground rails. Additional optional features include contact sharing, folding of fets larger than a specified minimum, and pre-wiring of connections that can be wired outside of the center wiring channel.

The schematic to be converted may be hierarchical, but all hierarchical components must ultimately consist of n-fets and p-fets.

- The location of the fets in the generated layout from left to right is taken from the location of the fets in the schematic, looking from left to right.
- Where p or n fets appear vertically aligned in a column in the schematic, the order is from top to bottom.
- The process is fully hierarchical. That is, if an icon for a sub-schematic is encountered in the left-to-right traversal of a parent schematic, all fets in the sub-schematic are inserted at that point, followed by the remaining fets to the right or below the sub-schematic icon.

The Layout Generator can be used to assist in developing a standard cell library. For cells that will be regenerated for different technologies, the designer would move the fets on the schematic so that the fets are placed ideally in the layout.

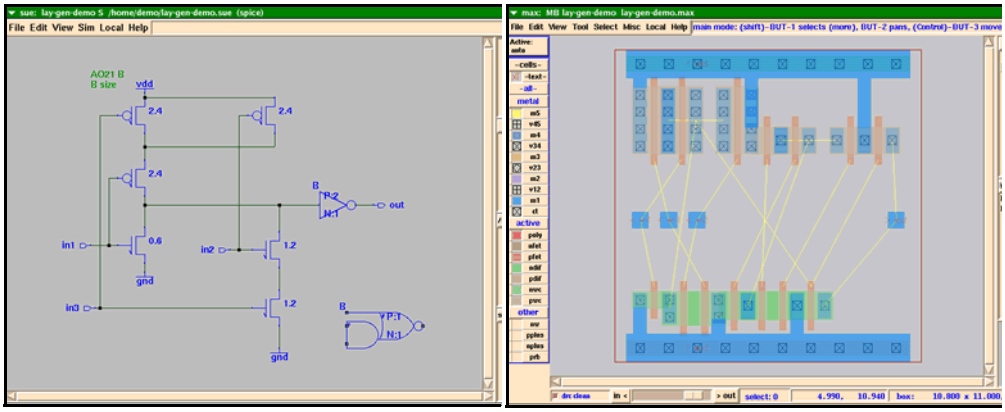
Note: About SUE

When SUE is launched using the standard default settings, your schematic will look something like Figure 97.

Note: About MAX

The default settings for the layout generator would have placed routing pads in the center for each of the ports, as shown in Figure 97. They were left out of these examples for easier viewing, by selecting `Option: No Router Pads` from the `Edit Per-Cell Options form` described in the section `Edit Per-Cell Options` on page 120 below.

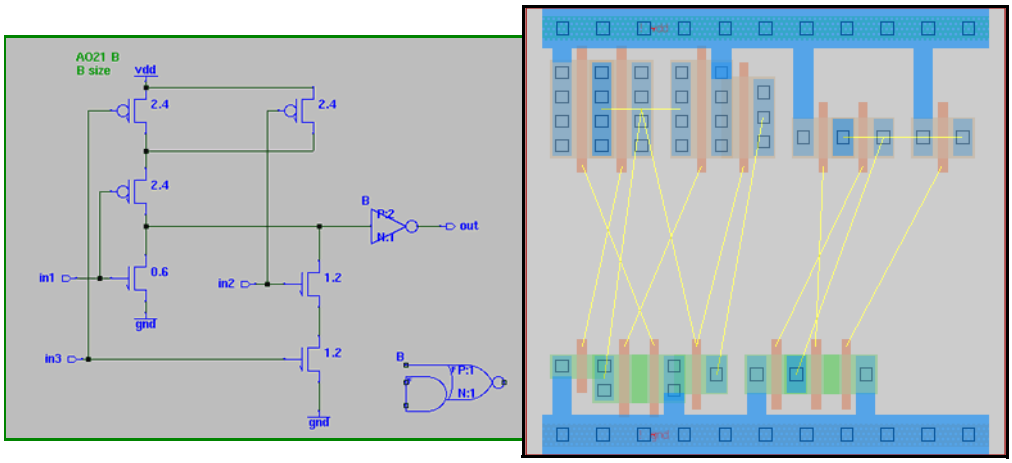
Figure 97: A) SUE Schematic; B) MAX Layout Generator Default Showing Router Pads



Layout Generator Examples

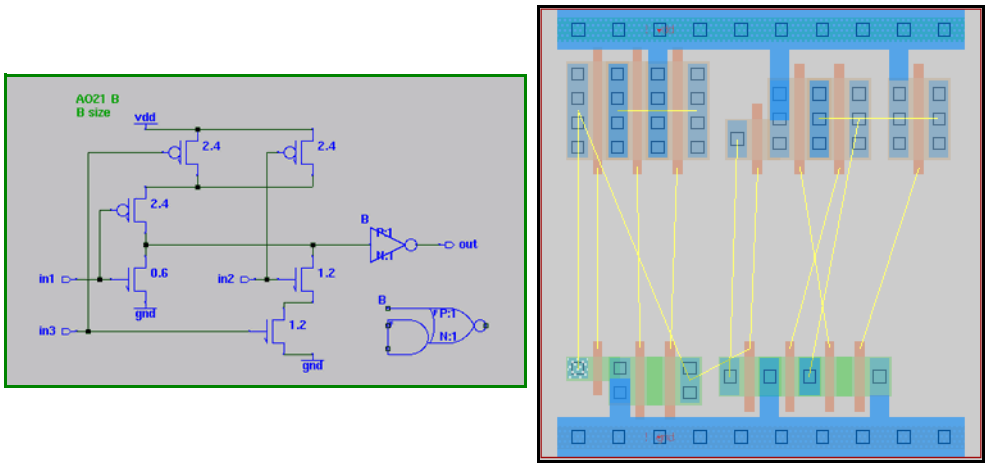
In the first example in Figure 98, notice that the order of the fets are such that a number of the net connections have to cross each other.

Figure 98: Layout Generator - First Placement



By changing the order of the fets in the schematic, as shown in Figure 99, the resulting layout is now much easier to route. In the schematic in Figure 99, the fets in the upper left and lower right have been moved toward the center.

Figure 99: Layout Generator - Better Placement



The default settings for the layout generator would have placed routing pads in the center for each of the ports. They were left out for these examples for easier viewing.

To generate the layout, you must first generate a **`.sim`** netlist file for the schematic.

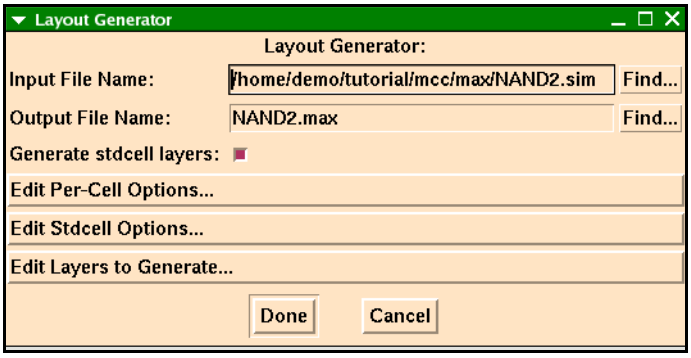
- To do this, start SUE and edit the top-most schematic representing the layout to be generated.
  - From the `Sim` menu, select `Change Simulation Mode`, and select `sim`, then `Done`.
  - From the `Sim` menu, select `sim netlist`. This generates the **`.sim`** file. (Note: The **`.sim`** format has been extended by Micro Magic, Inc. to contain the layout information.)
- Start MAX and select `Layout Generator` from the `Tool` menu. This brings up the `File Select Box`. MAX is asking you for the name of the **`.sim`** file that you generated in SUE.

  For example, if the SUE schematic was **`NAND2B.sue`**, then you need to enter **`NAND2B.sim`**.

  Next, MAX will show you the `Layout Generator` menu, as shown in Figure 100. This includes the `Input File Name` you entered above (and which you can change here), and the `Output File Name`, which is the cell name of the MAX output file, the `Generate Stdcell Layers` option, and buttons for a number of sub-menus. Changes that you make in the menus are maintained throughout your current MAX session.

If `Generate Stdcell Layers` is turned off, then the fets will be placed in two rows, flylines drawn, and routing pads placed. No other standard cell layers are generated. If router pads are turned off, then only the fets will be placed with flylines shown.

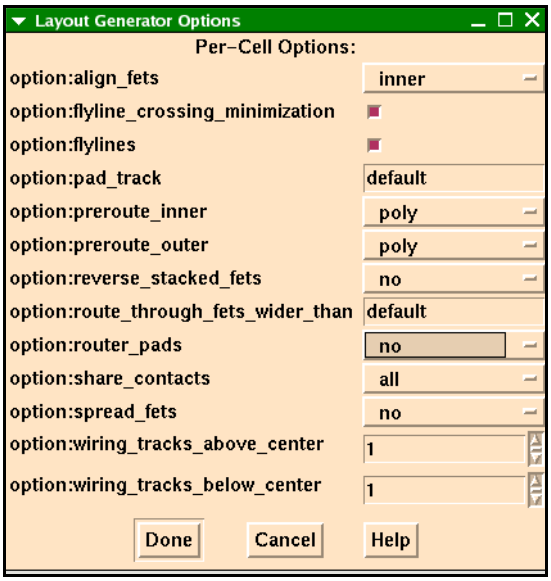
Figure 100: Layout Generator Form



Edit Per-Cell Options

This menu (See Figure 101) contains options you might want to change to get a better layout of this particular cell; for example, whether to align the fets toward the inner channel or the outer power rails, and so on. Each option is described below.

Figure 101: Layout Generator: Edit Per-Cell Options Form



align_fets — If set to `outer`, fets will be placed as close to power rails as possible; if set to `inner`, fets will be placed near the central wiring channel.

flyline_crossing_minimization — Rearrange flylines to minimize crossings. This is a time-consuming step, and probably not necessary if you are going to hand-route the cell.

flylines — Create flylines to indicate connectivity from the schematic.

pad_track — If router pads are being created, this specifies the y coordinate, in **dpc_router_pitch** units, where pads will be placed. If this is set to `default`, pads are placed near the Nplus/Pplus boundary. See also: **router_pads**.

preroute_inner — With this option enabled, the layout generator attempts to pre-route between the fets and the inner wiring channel where possible. Metal preroutes will reduce the contact sizes and place metal wires directly over fets.

preroute_outer — With this option enabled, the layout generator attempts to pre-route above and below the fets where possible. Metal preroutes will reduce the contact sizes and place metal wires directly over fets.

reverse_stacked_fets — This option reverses the layout order of specified types of fets (nfets or pfets) in the layout for stacked fets (fets that share diffusion.) This is used to ease congestion.

route_through_fets_wider_than — If `option: preroute_inner` or `preroute_outer` is enabled, this option specifies the minimum size fet that may NOT be routed over. If the option is set to `default`, the minimun fet width will be set to preserve one contact.

router_pads — If this option is set, the layout generator places on-grid router landing pads for nets that are ports. If this option is set to `contact`, the layout generator also provides contacts for nets that are on poly. Note: The pad size is specified by `router_pad_size` in the `Stdcell Options` menu.

share_contacts — This option controls whether fets share contacts with neighboring fets.

spread_fets — If this option is `no`, fets will all be jammed to the left. If it is `all`, the shorter row of fets will be spread out. If it is set to `larger`, only larger fets will be pushed to the right. If it is set to `best`, MAX splits connected fets at vdd/gnd contacts, or if fets are larger.

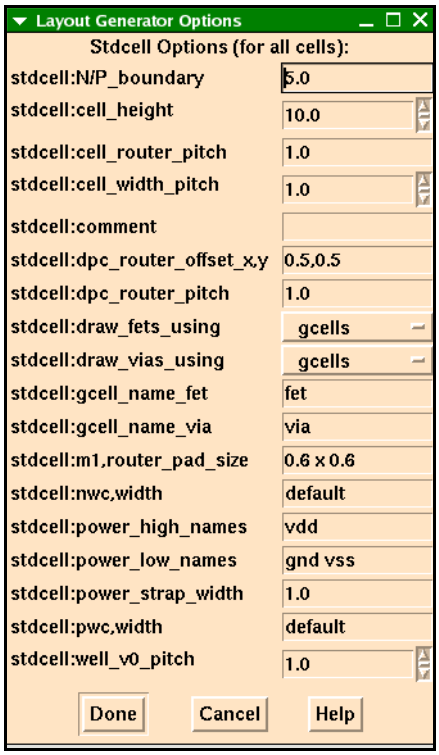
wiring_tracks_above_center — When this option is set, *pfets* will be folded to preserve wiring space between the pfets and the cell N/P boundary, which allows a set number of wiring tracks. The wiring track size is taken from `cell_router_pitch` in the `Stdcell Options` menu.

wiring_tracks_below_center — When this option is set, *nfets* will be folded to preserve wiring space between the nfets and the cell N/P boundary which allows the set number of wiring tracks. The wiring track size is taken from `cell_router_pitch` in the `Stdcell Options` menu.

Edit Stdcell Options

This form (See Figure 102) contains options that you should probably set once for all the cells that you intend to generate in a particular stdcell family; for example, the height of the cells.

Figure 102: Layout Generator: Edit Stdcell Options Form



N/P_Boundary — This sets the height of the Nplus/Pplus boundary from the bottom of the cell, in microns.

cell_height — This sets the height of cell in microns.

cell_router_pitch — The pitch used for inter-cell wiring, which may be finer than the `dpc_router_pitch`, is entered here. See also: `wiring_tracks_above_center` and `wiring_tracks_below_center` in the `Per-Cell Options` menu.

cell_width_pitch — The cell width will be rounded up to this pitch, normally equal to `dpc_router_pitch`.

dpc_router_offset_x,y — The offset of final router tracks, in microns, is entered here. Values to be entered in the form: **`<xoffset>, <yoffset>`**. Usually both `xoffset` and `yoffset` are equal to one-half the `dpc_router_pitch`.

dpc_router_pitch — The router pitch used for final (post-DPC) routing is entered here. Router landing pads are created on this pitch.

draw_fets_using — Fets are created using the method specified here.

draw_vias_using — Vias are created using the method specified here.

If you specify any `preroute_` options that require modifications to the fets, the fets will be drawn using paint regardless of the setting of this option.

If you specify `subcells` you should flatten the subcells created for fets before writing out the standard cell, to avoid naming conflicts among the subcells created for the fets.

gcell_name_fet — The name of the fet gcell, when `draw_fets_using` is ***gcells***, is entered here.

gcell_name_via — The name of via gcell, when `draw_vias_using` is ***gcells***, is entered here.

m1,router_pad_size — The size of Metal1 landing pads required for the final router, in the form: **`<width> x <height>`** in microns, is entered here. The default size is determined from the minimum Metal1 area rule.

nwc,width — This is for the size of the nwell contact under the power strap, if any.

power_high_names — Enter the name(s) of the positive power supply here. If multiple names are used, they will all be considered equivalent. The positive power rail is labeled with the first name.

power_low_names — Enter the name(s) of the negative power supply. If multiple names are used, they will all be considered equivalent. The negative power rail is labeled with the first name.

power_strap_width — This sets the size of the power strap.

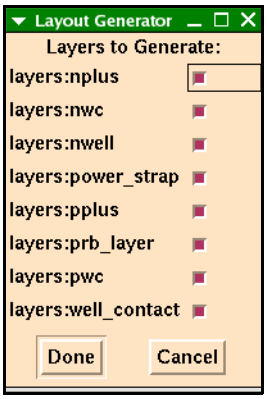
pwc,width — This sets the size of the pwell contact under the power strap, if any.

well_v0_pitch — If enabled, well contacts under the power straps will be created on this pitch, normally equal to `dpc_router_pitch`.

Edit Layers to Generate

This lets you individually control various Layout Generator layers. These layers are only generated if the `Generate Stdcell Layers` option in the main `Layout Generator` menu (See Figure 103) is enabled.

Figure 103: Layout Generator: Edit Layers to Generate Form



- Once you have set the options, select `Done` to generate the layout. The layout cell name will default to **`<cell_name>.max`**, where **`cell_name`** is the name of the cell in SUE.

You now see the layout in the MAX window with flylines (turned on by default) showing connectivity. Refer to Figure 98 and Figure 99.

- Use the MAX `Wire Tool` (see "`Introduction To The MAX Wire Tool`" on page 109) to make the connections. The flylines disappear as each connection is made.
  - If the connection is broken, the flylines currently do not reappear. Use the `Edit Flylines` (page 67) command to manually add the flyline back into the layout.

Cross-probing

Once the layout has been generated for a cell, you then wire up the connections using the flylines as guides. The Wire Tool makes the process of wiring up the cells quick and painless.

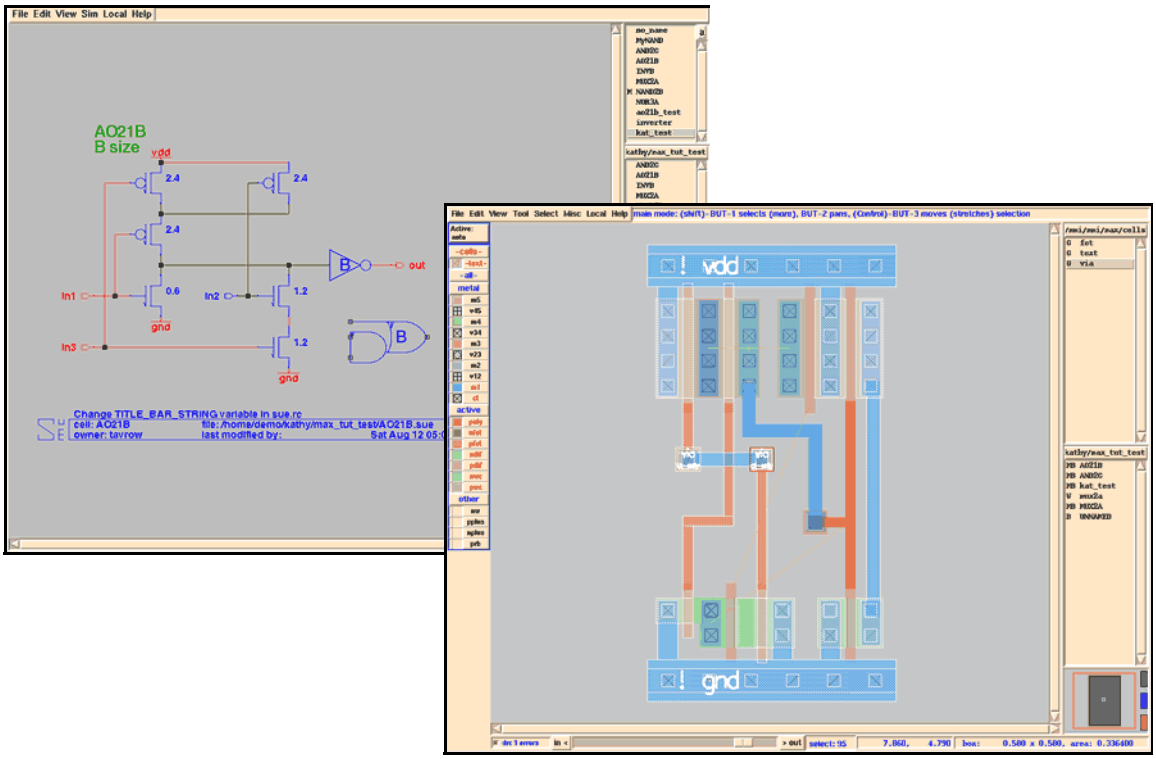
Once some of the wires have been completed, you can cross-probe between MAX and SUE to check the connections.

- To do this, select `SUE Cross Probe Init.` (page 85) from the `Tool` menu.

Before you ran the Layout Generator, you created a **`.sim`** file of the schematic. `SUE Cross Probe Init` first extracts a **`.sim`** netlist from the layout and then runs *Gemini* LVS between the SUE schematic and the MAX layout. Cross-probing can also be invoked from SUE by selecting `max cross probe init` from the `Sim` menu.

LVS crossprobing only works on FLAT designs NOT hierarchical ones. If the SUE schematic is hierarchical, you can only crossprobe at the top level of hierarchy.

Figure 104: Cross-Probe Init: a) SUE Schematic; b) MAX Layout



Once *Gemini* LVS has run, the nets which match are highlighted in MAX and SUE.

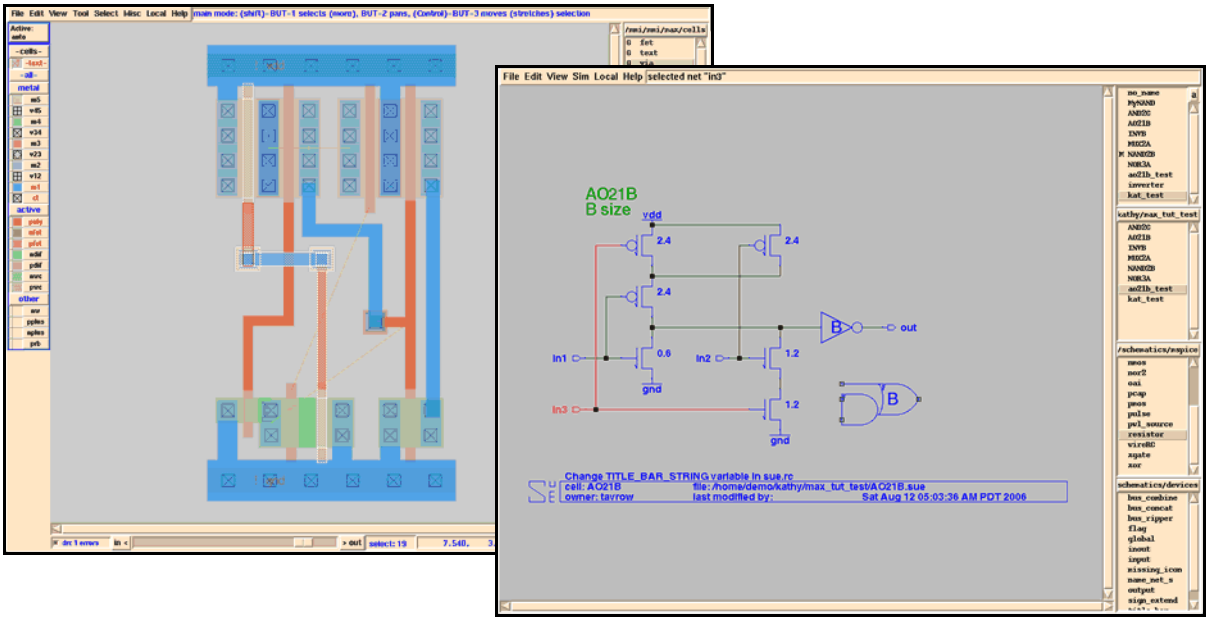
- Matching nets are highlighted in MAX using the highlight layer (see "`Editing Special Layers`" on page 15).
- The nets which match in SUE are highlighted in red.

You can select a net in MAX either by using the hotkey, **s**, or simply selecting a portion of the net and then using the `cross-probe` (hotkey: **k**) command to find the corresponding net in SUE.

- To do this, select `SUE Cross Probe` (page 86) from the `Tool` menu, or use the hotkey **k**. The corresponding net will be highlighted in SUE (See Figure 105).

If there is no corresponding net in SUE, nothing will be highlighted and an error message will be printed in the MAX Command Window.

Figure 105: Cross-Probe Net: a) Net selected in MAX Layout; b) Highlighted net in SUE Schematic



- You can cross-probe from SUE to MAX by selecting a net in SUE, clicking on the net with **Button-1**, and then selecting `max cross probe` from the `Sim` menu in SUE (or using the hotkey, **k**). The corresponding net in MAX will be highlighted.

If there is no corresponding net in MAX, nothing will be highlighted and an error message will be printed in the SUE command window.

Cross-probing without the Layout Generator

You can cross-probe any layout that has recognizable fets in it (and no other types of devices). No text entry at all is required. The cell does not need to be DRC clean to cross-probe.

Search Paths in MAX

When you do a `SUE Cross Probe Init.` (page 85) from the `Tool` menu in MAX, MAX will first run LVS on the current layout cell versus the schematic with the same name.

- For instance, if you are editing **`foo.max`**, MAX will try to run LVS against the schematic **`foo.sue`** using the **`foo.sim`** netlist.

  If **`foo.sue`** and **`foo.sim`** are in the same directory as **`foo.max`**, then great. If they are not in the same directory, you have to specify where MAX should look for them.

  - The list of paths to search is stored in the MAX variable **`LVS_SEARCH_DIRS`**.

  Therefore, if you have some schematics in the current directory and some in the parallel SUE directory, you would add the following line to your **`.maxrc`** file:

  ```
  set LVS_SEARCH_DIRS ". ../sue"
  ```

Sometimes you want to run LVS against a schematic that has a different name than the layout. This occurs when you have two or more different layouts for the same schematic.

- In the case of an SRAM, you might have two different SRAM cells — **`cell_L`** and **`cell_R`** — but only one schematic — **`cell.sue`**. In this case you want to map the layout cell name to the schematic cell name. Use the LVS associative array for this:

```
set lvs(<layout_name>) <schematic_name>
```

In the SRAM case, you would add the following lines into your **`.maxrc`**:

```
set lvs(cell_L) cell
set lvs(cell_R) cell
```

Search Paths in SUE

When cross-probing from SUE to MAX, SUE needs to be able to find the extracted netlists (**`.sim`** files) from the MAX layouts. If these layouts are not in the same directory, SUE needs to know where they are.

This search path is stored in the SUE variable **`BACK_ANNOTATE_PATH`**.

- For example, if you stored some MAX layouts in the same directory as the schematics, and some in the parallel `max` directory, you would add the following to your **`.suerc`** file:

```
set BACK_ANNOTATE_PATH ". ../max"
```

- In this version, if you have SUE bring up MAX and load the cell, that cell must be in the same directory as where you started SUE. Otherwise, you have to load the MAX cell yourself.

Chapter 6 MAX Technology Targeting

Introduction To MAX Technology Files

MAX technology files are created from a *source* file which includes layer definitions, GDS layer number assignments, and, optionally, DRC rules and colors. This source file must be converted to MAX'S internal technology files using the `make_tech` (page 129) program before running MAX.

The MAX source file consists of two parts:

1. A table of layer definitions.
2. A list of commands that derive layers, add additional DRC rules, add information needed for Gcells, the wire tool and the layout generator.

Only the Layer Definition table is required to create a technology. The additional statements after the Layer Definition table are only used to improve MAX's interactive DRC coverage of your layout, to derive any new layers and to add the commands necessary for Gcells, the Wire Tool and the Layout Generator.

The Layer Definition table can be created by one of three ways:

1. By hand using a text editor. You can also use the **`mmi18.source`** and **`mmi25.source`** files as starting points. These are the example technologies provided with MAX. They can be found in **`$MMI_TOOLS/max/tech`**.
2. From a *Dracula* DRC deck using the Micro Magic `drac_convert` (page 134) program.
3. From a GDS file using `gds_input` (page 131) program from Micro Magic.

make_tech

Once you have created the source file, you must run it through `make_tech` which is a stand-alone program included with the MMI software. `make_tech` has the following syntax:

```
make_tech -file <source_file> [-tech <tech_name>] [-r] [-install]
```

If you don't specify the technology name, `make_tech` will default to the technology "test".

Also, if you want to overwrite an existing technology of the same name, you have to use the "**`-r`**" option to replace the existing technology files.

Use the **`-r`** option with care! Changing the layer names in a technology can lead to lost data by making existing **`.max`** files partially or wholly unreadable.

By default, make_tech will install the technology into your private area (typically **~/mmi_private/max/tech/<tech_name>**). This will not change or make this technology available for any one else.

- If you want to install it for everyone's use, use the **-install** option which will place it in:

  ```
  $MMI_LOCAL/max/tech/<tech_name>
  ```

  or if you haven't defined **MMI_LOCAL**:

  ```
  $MMI_TOOLS/../mmi_local/max/tech/<tech_name>
  ```

make_tech will also copy the technology source file into the technology directory so you will always have it. The technology source file will be renamed to **<tech_name>.source**.

- The make_tech program will also allow users to make technology variations which are of the form:

  ```
  <tech_name> = <base_name>-<variation>
  ```

When MAX saves layout into **.max** files, it also saves the base technology name used to create them. MAX will only let you read **.max** files if the base technology name inside the **.max** file corresponds to the current MAX base technology. Otherwise, you might load a layout with incorrect layer names and other problems.

Sometimes you may want to have two different technologies that are virtually the same but have minor differences. In this case, you do want to be able to load **.max** files into different technologies. MAX solves this problem with technology variations. Technology variations have the same base technology name followed by a dash (-) and then a technology variation. Only the base technology name is saved into the **.max** files and checked on read in.

- One example of a technology variation might be a process with the choice of different numbers of layers of metal. In one case the technology might have 3 layers of metal and in the other case, 4.

  Therefore, you might have the two technology variations:

  ```
  my_tech-tlm and
  my_tech-flm
  ```

  Note that you should not include a dash (-) in a technology name unless you want to create a technology variation.

- Once you have run make_tech, you can use this technology by starting MAX with the following line:

  ```
  max -tech <tech_name>
  ```

- Alternatively, you can set the UNIX environment variable **MAX_DEFAULT_TECH** to be the desired technology and then every time you start MAX, it will default to that technology:

  ```
  setenv MAX_DEFAULT_TECH <tech_name>
  max
  ```

Reading in GDSII Files

If there is an existing technology file which matches the GDSII file, you can use the `Import File` (page 34) command. Otherwise, you can use the `gds_input` command which finds all the layers in the GDSII file and creates a technology file for the cell.

gds_input

The `gds_input` program is meant for importing arbitrary GDS files directly into MAX without first requiring a technology file.

- `gds_input` will first parse the GDS file (using MAX) and determine what GDS layers are in the file.
- It will then create a simple technology source file as described below, assigning arbitrary names and colors to each layer.
- It then runs `make_tech` on the source file to create a new MAX technology.
- Finally, it will run MAX with the new technology and import the GDS file.

  The syntax for `gds_input` is:

  ```
  gds_input [-expand_datatypes] [-file <source_file>] [-max
  <executable>][-tech <name>] [-rebuild] [-dont_load_file] <gds_file>
  ```

  where **<gds_file>** is the full pathname to the gds file to be loaded.

- If you specify "**-expand_datatypes**" then every datatype on everylayer is assigned a unique MAX layer. Otherwise, all datatypes of a given layer are merged together (default).
- If you specify "**-rebuild**" then uses your modified "source file". The source file defaults to "**tech.source**" and technology defaults to "tech".
- If you specify "**-dont_load_file**" then the tech file is built from the given gds file but the gds file is not loaded.
- If you don't specify the **source_file**, it will default to the file **test.source**. If you don't specify the technology, it will default to **test**.
- So, if you just want to view the GDS file called **aaa.strm**, you can simply run:

  ```
  gds_input aaa.strm
  ```

  That's it!

Synopsis

```
gds_input [-expand_datatypes] [-file <source_file>] [-max
<executable>] [-tech <name>] [-rebuild] [-dont_load_file] <gds_file>
```

Description

The above command will run the `gds_input` script.

- It will first load the `gds_file` into MAX and find all GDS layers which are used in the file, with `<gds_file>` being the full pathname to the GDS file to be loaded.
- It then creates a technology template file and runs `make_tech`.

- Once the technology files have been created, `gds_input` brings up the `gds_file` in MAX using the new technology file.

Table 4: Options

Option	Description
-file <source_file>	The source file is the file which contains the GDSII layer-number-to-layer-name mapping. The name of source_file defaults to <tech>.source. Refer to `Technology Source Files` (page 134) for the syntax of the source file.
-tech <tech_name>	Specify the name of the technology to be created. tech_name defaults to "test". The new technology files will be put in: **~/mmi_private/max/tech/<tech_name>**
-h | -help	Prints out the syntax of the **gds_input** command
gds_file	The name of the GDSII file from which the new technology will be created. You should select a cell or block which contains all or at least most of the layers, or the entire chip.
-expand_datatypes	If you specify "-expand_datatypes" then every datatype on every layer is assigned a unique MAX layer. Otherwise, all datatypes of a given layer are merged together (default).
-dont_load_file	If you specify "-dont_load_file" then the tech file is built from the given GDS, file but the GDS file is not loaded.
-rebuild	Take the <tech_name>.source file as input and run **make_tech**, and then load the GDSII file. This allows you to edit the technology source file created by a previous run of **gds_input**, change layer names, colors, etc. and rebuild the MAX technology.

Layers

Once you have created a technology source file with **`gds_input`**, you will likely want to add layer names and group layers, etc.

- For example, if you ran:

```
gds_input aaa.strm
```

`gds_input` would create the technology source file called **`tech.source`** which might look like this:

```
#layer  gds:dt txt:dt   type    width   space   color
#====== ====== ====== ======== ======= ======= =======
L8      8       -       -       -       -
L12     12      -       -       -       -
L13     13      -       -       -       -
L15     15      -       -       -       -
L16     16      -       -       -       -
L17     17      -       -       -       -
L18     18      -       -       -       -
L62     62      -       -       -       -
```

You might then edit **`tech.source`** to add meaningful layer names and types, for example:

```
#layer  gds:dt txt:dt   type    width   space   color
#====== ====== ====== ======== ======= ======= =======
diff    8       -       -       -       -
poly    12      -       -       -       -
```

```
ct      13      -       via     -       -
m1      15      -       metal   -       -
v12     16      -       via     -       -
m2      17      -       meal    -       -
L18     18      -       -       -       -
L62     62      -       -       -       -
```

- You would then need to regenerate the technology file using the command:

```
make_tech -r -file tech.source
```

- Now load the GDS file with the new tech file by entering:

```
max -tech tech aaa.strm
```

Environment

The following are the environment variables which are used by `gds_input`.

`MMI_TOOLS` The environment variable **`MMI_TOOLS`** must be set to the directory where the Micro Magic, Inc. programs were installed. All of the Micro Magic, Inc. programs are installed under a single directory.

```
setenv MMI_TOOLS /usr/tools/mmi
```

More on “gds_input”

Typically, once you have run `gds_input`, you want to assign meaningful layer names to the layers, specify the correct `layer type` (page 136), order the layers (see `layer type` on page 136), and possibly tell MAX to ignore some layers or merge other layers.

- To do this, simply edit the technology source file that `gds_input` creates and then rerun `gds_input` with the “`-rebuild`” option.

  With “`-rebuild`”, `gds_input` will skip parsing the GDS file and use the specified source file, run `make_tech`, and then load the GDS file into MAX with the new technology.

  Otherwise, you could simply run `make_tech` yourself and then start MAX with the appropriate technology and `import` the file.

`gds_input` can’t determine where instance names are hidden in the GDS file (if they are there at all).

- If your GDS file has instance names, and you want MAX to read them, you have to add the `iname` type to the technology source file as described in the `layer type` (page 136) section.
- Note that `gds_input` does not use the `-install` option to `make_tech` so you will need to run `make_tech` to install this technology for other people to use. This also prevents you from conflicting with other people trying to run `gds_input` simultaneously.

However, you will overwrite any local technology that you already have with the same name (for example, it calls `make_tech` with the `-r` option).

Converting a Dracula DRC File

The `drac_convert` script will take a *Dracula* DRC file and convert the layer information, some of the basic layer generation and the basic DRC rules into a **`layers_file`** used by `make_tech`. This is an alternative to creating the technology source file by hand or running `gds_input`.

drac_convert

The `drac_convert` program will attempt to parse a *Dracula* DRC deck and create a technology source file from it for use by `make_tech`.

- The syntax of `drac_convert` is:

  ```
  drac_convert <dracula_command_file> > <source_file>
  ```

  For example, if you had a *Dracula* DRC deck for the xxx25 technology, you might run:

  ```
  drac_convert xxx25.drac > xxx25.source
  ```

- You can then run `make_tech` on **`xxx25.source`**. You might want to first edit **`xxx25.source`** to change layer names and add additional rules.

`drac_convert`, presently, will find most width, spacing, and overlap rules to add to your technology source file. It is not guaranteed to find all rules but is only meant to shorten technology creation times.

Technology Source Files

The technology source file has the following basic characteristics:

- Comment lines are lines which contain a pound sign (#) as the first non-white-space character.
- Blank lines are ignored.
- Lines may be continued by ending the line with a backslash (\) character.

The technology source file for the mmi25 technology is located in:

```
$MMI_TOOS/max/tech/mmi25/mmi25.source
```

- An example of a very basic technology source file is located at the end of this chapter. It has the minimum information needed for reading and writing GDSII, using MAX Gcells, and the Wiring Tool.

  This example does not contain all of the information needed for the layout generator or all of the DRC rules.

- MAX currently limits the number of layers to 1000 in any given techology file.

The Layer Definition table is a list of white-space separated fields, one per each layer to be defined. White spaces are either spaces or tabs. Generally, layers are ordered from lowest mask layer down to highest mask layer. Each column defines a separate attribute of the layers which in brief are, from left to right:

- Layer name
- GDS layer number and datatype
- GDS text layer number and datatype (optional)
- Type of layer (optional)
- Minimum width of that layer in microns (optional)
- Minimum space from that layer to itself in microns (optional)
- Desired color of that layer (optional)

Where a field is optional and you do not wish to specify a value for it, use the dash character (-) to specify that you have not set the field. Do not simply add spaces. If you do, the parser will then use the next field instead. You can only omit the dash if no other fields to the right of it are defined on that line.

Each column of the Layer Definition table is now described in detail.

layer name

The layer name is the unique means by which MAX will refer to this layer. MAX saves all layouts with specific reference to this name and NOT to the GDS layer number or any other parameter.

- If you change this name, for example, from **`MET1`** to **`M1`**, and run `make_tech`, all existing MAX files that use this technology file will not read correctly into MAX.
- If you do need to change this name, you will need to export all of your existing MAX layouts to GDS, then change the name in the technology source file and run `make_tech`.
- Finally, import the GDS layouts back into MAX and save them back out to MAX format.

If you create the Layer Definition table from either the `drac_convert` program or the `gds_input` program, you will probably want to edit the layer names. Make sure you do this before saving out any MAX files.

gds layer number and datatype

This field specifies the GDS layer numbers and datatype for both importing and exporting GDS into and out of MAX. At present you cannot easily specify a different GDS mapping between input and output. You can do so on a limited basis by deriving new layers with different GDS layer numbers.

- The format of this column is:

```
<gds_num> or
<gds_num>,<gds_num>[,...] or
<gds_num>:<data_types> or
<gds_num>[:<data_types>];<gds_num>[:<data_types>][;...]
```

If no data type is given, then it defaults to all data types on input (or '*') and data type 0 on output.

- Data types can be any comma-separated list of numbers or ranges of numbers (for instance, **<begin>-<end>**).
- Furthermore, if more than one GDS layer number or data type is given, then the first GDS layer, data type pair are used for output. Here are some legal examples:

  Imports geometries from layer 12, all data types, and exports to layers 12, datatype 0.

  ```
  12
  ```

  Imports geometries from GDS layers 12 and 13, all data types, but only exports them out to layer 12, datatype 0:

  ```
  12,13
  ```

  Imports geometries from GDS layer 12, datatypes 1, 3, and 5. Exports to layer 12, datatype 1:

  ```
  12:1,3,5
  ```

  Imports geometries from GDS layer 12, datatypes 1-3, and 5, layer 13 all datatypes, and layer 14 datatype 1. Exports to layer 12, datatype 1:

  ```
  12:1-3,5;13;14:1
  ```

- Remember not to put any spaces in any of the GDS layer designations since the space is the column separator.

text layer number and datatype

By default, text (also called *labels* in MAX) will be imported/exported from/to the same GDS layer number as the geometry that it is attached to. By specifying this field, you can instruct MAX to import/export text from/to different GDS layers. The format of this field is identical to the format of the GDS layer numbers as described above.

- Note that in MAX, datatypes and texttypes are considered synonymous. Therefore, we say that text resides on layer 34, datatype 12; identical to saying layer 34, texttype 12:

  ```
  34:12
  ```

layer type

This optional parameter helps MAX to better build a technology file. The valid types are:

```
act, poly, metal, via, ignore, gdsonly, bbox, iname, other (default)
```

- It is recommended that all conducting layers be given a type of either **act**, **poly**, **metal**, or **via**.

- Specify all diffusions as **act** (for active layer) and all poly layers as **poly**. Metal layers should be **metal** and contacts/vias should be **via**.

MAX uses these types to determine connectivity and build fets. (If MAX cannot recognize a fet it will think that the source and drain are connected).

If a different layer type is specified other than the ones given above (i.e. not **act**, **poly**, **metal**, etc.) then all of the layers with this new type will be grouped together in the palette.

- For example, if you have 6 different implant layers, you might give them each the type "**implant**" and they will be grouped together in the palette. Any layers without a group will be placed in the "**other**" group at the end of the palette.

The order of the **via** and **metal** layers in the Layer Definition table determines the order in which they get connected.

- The first **via** layer becomes the contact and connects to all of the **act** and **poly** layers. It is also connected to the next **metal** layer.

- The next **metal** layer is then connected to the following **via** layer and so on.

  For example, for the following technology, you would need this line order for the **via** and **metal** types in the technology source file to get connected properly:

Table 5: Layer Type

# layer	gds:dt	txt:dt	type	width	space	color
ct	15	-	via	0.30	0.30	-
m1	16	40	metal	0.32	0.32	8,139,255
v12	17	-	via	0.36	0.35	-
m2	18	41	metal	0.40	0.40	182,134,222
v23	27	-	via	0.36	0.35	-
m3	28	42	metal	0.40	0.40	255,160,65
...	...	...	...	...	...	...

Any other order and MAX would not get the connectivity correct. For example, if you specified the layers in the order **ct,v12,v23,m1,m2,m3,** MAX would not understand connectivity. You need to alternate layers starting from `via` (which will become the contact layer) just as in the above example.

The order of all layers *other* than the **via** or **metal** layers is unimportant.

- If there are layers in your GDS input file that you want to *ignore*, specify them with the type **ignore**. Otherwise you will get warnings that they are not defined when you import them (if you have enabled GDS warnings in the `import GDS` dialog).

The **gdsonly** type is a special type for layers that you don't want to see directly in MAX, but that you want to derive other layers from.

- For example, in MAX, you can carve up the diffusion layer during GDS, then importing into ndif and pdif using the derive command described below.

The **bbox** type is a special type to specify the ***bounding box*** of cell instances.

- Only the GDS layer and data type are significant in the **bbox** layer line, and there should only be one **bbox** layer line in the Layer Definition table.
- MAX uses this layer data to display the cell boundaries, until a cell is viewed at which time MAX recomputes the bounding box from the cell layout itself.
- MAX will also output the bounding box to this GDS layer for other tools.

The **iname** type is a special type to specify where ***instance names*** are stored in the GDS file.

- Not all tools write out instance names to GDS files.
- Only the GDS layer and data type are significant in the **iname** layer line, and there should only be one **iname** layer line in the Layer Definition table.

There are two forms of **iname** types, depending on your GDS input file or how you wish to write a GDS output file:

Table 6: Two Forms of "iname" types

# layer	gds:dt	txt:dt	type	width	space	color
...	...	...	...	...	...	...
iname	63	-	iname	-	-	-
...	...	...	...	...	...	...

OR

# layer	gds:dt	txt:dt	type	width	space	color
...	...	...	...	...	...	...
iname	propattr	102	iname	-	-	-
...	...	...	...	...	...	...

- In the first case, the instance name is stored in **GDS layer 63, datatype 0**.
- In the second case, the instance name is stored in **property attribute 102**.
- Note that in the second form, the **102** is actually in the text column.

The rest of the layers, such as implants, should be specified as **other**, which is also the default; therefore, you can use the dash (-) for them.

Every line in the Layer Definition table, except for those of type **gdsonly**, **ignore**, and **bbox**, will show up in the MAX palette.

minimum width

This optional value in microns will create a DRC rule for the MAX interactive DRC checker for the minimum width of the layer.

minimum space

This optional value in microns will create a DRC rule for the MAX interactive DRC checker for the minimum space of the layer to itself.

color

This optional parameter will set the default color and style for this layer in the layout. If no color or no style is specified, a color and style will be assigned to the layer based on its type. Remember, these colors and styles can always be changed inside of MAX using the color editor and saved for subsequent use.

- Note that the stipple pattern (also called the fill pattern) and outline style must be specified through the color/stipple editor (see `Palette` on page 12) in MAX, and can be saved for subsequent MAX sessions.

Colors can be specified in one of two ways:

- By specifying the RGB triplet of the form **`red,green,blue`** (comma separated, no spaces) with each value in the range of 0-255);
- Or by specifying a color name using the names provided in the file **`rgb.txt`** that is provided with the X window system.

Styles can be specified using the following keywords:

```
solid, stipple, x, +, diamond, outline, outline_only
```

- To specify both a style and a color, separate them with a ":" (colon), style first. For example, to specify *solid* and *red*, you would enter:

```
solid:red or solid:255,0,0
```

The stipple patterns are selected initially to be reasonable for the type of layer — this is another reason why you should specify the layer types.

- Note that MAX has its own version of **`rgb.txt`** that it refers to which might be different than yours.
- If no color is given, MAX will choose one. Remember, these colors can always be changed inside of MAX using the color editor and saved for subsequent use.

Additional Statements

Following the Layer Definition table, you can specify additional statements that will define devices, create derived layers, add further connection information, and add more DRC rules.

The statements are all single lines and must start with one of the following keywords:

- device
- connect
- derive
- drc

device

For connectivity tracing and extraction, it is important that MAX understand what constitutes a device. Presently, only fets are supported.

- Fets are defined using the device statement. If you do not specify any devices but you specify an `act` layer type and a `poly` layer type, MAX will automatically create a fet device from the intersection of those layers.

- If you want to define your own fets, use the following syntax:

```
device <device_name> from <poly_layer> <diffusion_layer>
```

  For example, if you have defined a separate ndif layer and pdif layer, you can define an nfet and a pfet with the following lines:

```
device nfet from poly ndif
device pfet from poly pdif
```

Note that the nfet and pfet become layers and will show up in the MAX layer palette. However, MAX will not let you directly modify the colors or fill patterns of these layers since they are derived from the other two layers that make them up.

connect

For connectivity tracing and extraction, MAX must know what layers are conducting and how they connect to other layers.

- `make_tech` can derive most connectivity information from the order of the layer types **`metal`** and **`via`**, and from its knowledge of fet devices.

- However, if you want to add additional connectivity, you can do so with the connect statement.

  The connect statement has the syntax:

```
connect <layer_1> <layer_2>
```

- If you want to allow butting contacts between the n-well contact (nwc) and the p-diffusion (pdif), for example, you would add the following line:

```
connect nwc pdif
```

derive

- In MAX, you can derive a new layer from one or more other layers using any of the following layer operations:

  ```
  and <layer>
  or <layer>
  and-not <layer>
  grow <value>
  growx <value>
  growy <value>
  shrink <value>
  shrinkx <value>
  shrinky <value>
  ```

  The syntax of the derive statement is as follows, which is parsed from left to right:

  ```
  derive <new_layer> from <starting_layer> <operation> <layer_or_value>
  [<operation> <layer_or_value> ...]
  ```

- No parentheses or grouping is allowed. Instead you need to define temporary layers.

  For example, you need to use two derive statements to generate the following:

  ```
  derive tmp1 from nplus grow 0.1
  derive new from nw and-not tmp1
  ```

  In this case the layer "**new**" is derived from the "**nw**" layer except for where there is nplus grown by 0.1 microns.

- You can view any layers that you derive in MAX using the `See Mask` (page 80) command and clicking on the desired layer.

  That layer will then be overlaid on the current window in a temporary feedback layer.

- Use the `clear` button at the top of the `Mask` menu to remove this feedback layer.

- Only one layer at a time can be displayed in this way; for example, when you click on a second layer, it first clears the initial layer.

  Note that MAX will only draw this feedback layer to cover the current viewable window, so make sure that you have zoomed out sufficiently before running this command. This feature prevents MAX from potentially wasting a lot of time generating a layer for an entire chip when all you care about is a small section.

You may notice that some of the layers in the `Mask` menu have the words **`GDS_`**, **`TXT_`** and **`DRC_`** appended to their names. This is to differentiate the layers from the internal MAX layer from which they are derived. Sometimes to create a DRC rule, as described in the next section, MAX must create a special temporary layer. These are the layers that begin with **`DRC_`**.

- If you wanted to select out only those contacts that landed on poly instead of diffusion, you could **`derive`** a new layer, *`poly_ct`*, as follows:

```
derive poly_ct from poly and ct
```

- If you wanted to generate the od layer from the all the different types of diffusion, you would do the following:

```
derive od from ndif or pdif or nwc or pwc
```

- You can also '**or**' layers together by simply separating them with commas, as in the following example:

```
derive od from ndif,pdif,nwc,pwc
```

  which is equivalent to the first example.

- The **and-not** operation allows you to select parts of layers that are not coincident with other layers. For example, n-type diffusions are made up of diffusions that don't have n-well and don't have pplus over them:

```
derive ndif from od and-not nw and-not pplus
```

The **grow** operation grows a layer by a specified amount in all directions.

- If two geometries on the same layer are separated by a distance of *2x* and you grow them by a distance *x*, they will merge together.
- If you then shrink them by a distance *x*, they will stay merged together.

  This grow/shrink operation is useful for notch and gap filling.

The **shrink** operation is also handy for eliminating undersized geometries. Note that the shrink operation can sometimes yield unexpected results, in particular, if you have touching geometries in adjacent sub-cells. If those geometries are shrunk, even slightly, they will no longer touch.

- For example, if you want to generate the n-well (nw) layer to be 0.6 microns around pdif and 0.16 microns around n-well contacts (nwc) and you want to notch and gap fill it, you can use the following line:

```
derive pdif_grow pdif grow 0.6
derive nw from nwc grow 0.16 or pdif_grow grow 0.3 \
shrink 0.3
```

- A more clever way to do this is as follows:

```
derive nw from pdif grow 0.44 or nwc grow 0.16 \
grow 0.3 shrink 0.3
```

In this release of MAX, layer operations only affect manhattan geometries.

drc

The **drc** line allows you to add rules beyond the simple width and spacing rules in the Layer Definition table. DRC rules can check layers defined in the Layer Definition table and also any derived layers or any combination thereof.

- The types of rules that are allowed are:

```
width <value>
space <value>
space_to <layer> <value>
enc <layer> <value>
ext <layer> <value>
area <value>
```

- The syntax of the **drc line** is:

```
drc <layer> <operation> [<layer>] [<value>] [-msg <text>] [; <comment>]
```

MAX understands the following **drc** operations: **width, space, space_to, enc, ext, area, illegal, width>**.

The operations **width** and **space** are the minimum widths and spaces of the given layer and are typically defined in the Layer Definition table.

- **space_to** allows you to specify the minimum space between two different layers.
- **enc** specifies the minimum enclosure or overlap of the first layer to the second layer.
- **ext** specifies the minimum extension past another layer and is typically used for specifying the poly overlap of gate, and diffusion overlap of gate in fets.
- **area** specifies the minimum area of an isolated geometry of the layer.

  All values are in microns, except for the area, which is in microns squared.

  - To specify the minimum spacing between ndif and pdif at 1.2 µm, use the following line:

```
drc ndif space_to pdif 1.2
```

- **illegal** specifies that any geometries on the given layer are a DRC error. **illegal** is typically used on a derived layer.

  - For example:

```
derive tmp1 from bipolar and nwell
drc tmp1 illegal -msg nwell not allowed over bipolar
```

- **width>** is used for specifying wide metal rules.

  - For example, the following line:

```
drc m2 width> 0.72 space 0.2
```

    tells MAX that if an m2 geometry is wider than 0.72 µm, then the minimim space to any other m2 must be at least 0.2 µm.

- If MAX sees a violation of this rule in the layout, it will return a message of the form:

```
ndif to pdif minimum spacing = 1.2 um.
```

  - If you want to add any additional information to this message, do so with the comment line, for example:

```
drc ndif space_to pdif 1.2 ; rule 31.2
```

    which will lead to the error message:

```
ndif to pdif minimum spacing = 1.2 um. rule 31.2
```

    Note that you cannot use the pound sign (#) in the comment.

    MAX will attempt to make a reasonable error message out of the drc line. However, sometimes a completely different error message is desired. To create a custom error message, use the **-msg** option.

  - For example:

```
derive tmp_v12 from v12 grow 0.2 shrink 0.2 and m1 and m2
drc tmp_v12 space 0.16 -msg V12 on different nets space >= 0.16
```

    Without the **-msg** in the above drc, on a violoation of the above drc rule, MAX would alert the user that "**tmp_v12 minimum spacing = 0.16 um.**", which is not very helpful.

  - If you want to specify the minimum enclosure of contact by Metal1, you would use the following rule:

```
drc m1 enc ct 0.09
```

  - If you want to specify the minimum extension of poly past an nfet, use the following rule:

```
drc poly ext nfet 0.36
```

- Only the first occurrence of any rule will be used.

  `make_tech` will give a warning that it is ignoring any subsequent duplicate rule definitions. Thus, if you define the minimum width of a layer in the Layer Definition table and then redefine it with the DRC line, the first definition will be used.

Note that some rules have restrictions. For now, the **ext** rule must only apply to layers with types of **poly** or **act**. Also the **area** rule requires that the **width** of the layer is first defined.

Advanced Topics

- One way to simplify viewing and creating layout when importing from GDS is to separate diffusion into four different types, depending on its function, as shown here:

```
derive ndif from od and-not nw and-not pplus
derive pdif from od and nw and pplus
derive nwc from od and nw and-not pplus
derive pwc from od and-not nw and pplus
```

- When exporting, you simply "**od**" all the layers together with the line:

```
derive od from ndif,pdif,nwc,pwc
```

- If you do this, you have to add some special directives to the Layer Definition table as shown below:

Table 7: Special Directives to Layer Definition Table

# layer	gds:dt	txt:dt	type	width	space	color
...	...	...	...	...	...	...
od	4	-	gdsonly	0.30	0.40	-
ndif	derived	-	act	-	-	66,213,66
pdif	derived	-	act	-	-	202,160,115
nwc	derived	ndif	act	-	-	-
pwc	derived	pdif	act	-	-	-
...	...	...	...	...	...	...

- The **od** layer is only used during import and export and not internally, and therefore is declared as "**gdsonly**".

- The '**ndif**', '**pdif**', '**nwc**', and '**pwc**' layers are derived on input — they are not explicitly part of the GDS file in this case, and therefore you must use the special keyword *derived* for the GDS layer number, as shown above.

- Finally, you can add an additional special keyword to the **nwc** and **pwc**, layers as shown above, to cause them to track the colors of the **ndif** and **pdif** layers.

Other Commands

Some information needed for the wire tool, Gcells and the layout generator is defined with the Tcl set statement. Any set statements in the technology source file gets transferred to the MAX technology files. See the **mmi25.source** file in **$MMI_TOOLS/max/tech/mmi25** for descriptions of the commands needed.

The following are the currently used variables for storing technology data:

DRC_DATA

For certain rules, the value checked by the MAX online DRC checker may be different than the value used to build a device using gcells.

- For example, you might want to check for a minimum poly space in a fet but want the gcell to create fets with a larger space.

  Furthermore, there might be rules that you don't want to be checked at all but that you want the gcells or wiring tool to know about.

- Finally, in specifying the layer operations to create a layer, MAX might not be able to determine the intent or the rule. In either of these cases, you can specify these different values using the **DRC_DATA** variable, for example:

```
set DRC_DATA(space_to,poly,diff) 0.10
```

Note: You may need to speak with an MMI Applications Engineer for further assistance if you need to use this variable.

GRID

Set the grid parameters to control resolution.

- For example, to specify the smallest mask resolution allowed in this techology use:

```
set GRID(mask) 0.005
```

MAX internally represents numbers to 0.001, or nanometer, resolution.

- To specify the default grid resolution use:

```
set GRID(resolution) 0.01
```

LAYER_NAME

The gcells in MAX use canonical names to refer to layers. If your layer names are different, you must provide themapping from your layer names to MAX layer names using the **LAYER_NAME** array variable:

```
set LAYER_NAME(pdiff) <your_p_diffusion_name>
set LAYER_NAME(ndiff) <your_n_diffusion_name>
set LAYER_NAME(nplus) <your_n_plus_name>
set LAYER_NAME(pplus) <your_p_plus_name>
set LAYER_NAME(nwell) <your_n_well_name>
set LAYER_NAME(poly)  <your_poly_name>
set LAYER_NAME(contact) <your_contact_name>
```

- For example, if your poly layer is called **PG**, use the following to remap your layer name:

```
set LAYER_NAME(poly) PG
```

GCELL_FETS

If the technology source file contains enough data to fully specify a fet, MAX's gcell fet generator can be used to place and modify fets in a layout.

- A fet requires at least one layer to be specified with type "`act`" or "`active`" and one layer with type "`poly`".

  If you specify two or more layers as active, for example ndif and pdif, then you need to specify the fet types with lines like the following:

```
device nfet from poly ndif
device pfet from poly pdif
```

This will create an nfet using ndif and a pfet using pdif.

The fet gcell also requires the following technology data to be specified:

```
ndif width
poly width and space [for fingered devices only]
poly extension past ndif (nfet)
ndif extension past gate poly (nfet) [for uncontacted devices only]
contact width and space
contact space to gate poly (nfet)
ndif enclosure of contact
metal1 enclosure of contact
```

- For example, the following minimum technology source file would be sufficient:

```
ndif       2       -       act     0.2
poly       7       -       poly    0.1     0.2
ct         14      -       via     0.15    0.2
m1         15      -       metal

drc poly ext nfet 0.2
# only needed for uncontacted devices
drc ndif ext nfet 0.15

drc ct space_to nfet 0.1

# Must specify enclosure for both ndif and poly around contact.
derive ct_ndif from ct and ndif
drc ndif enc ct_ndif 0.1

drc m1 enc ct 0.05
```

- The default fet size will have width of 1 um and length of the poly minimum width. To change the default fet width to, for example, 0.2 um, add the following line:

```
set GCELL_FET_DEFAULT_WIDTH 0.2
```

Often processes support additional fets beyond the normal nfets and pfets. There might be low-threshold fets or high-voltage fets. With a single line in the technology source file, most of these additional fets can be specified and then the gcell fet generator will be able to make them.

- To specify a fet variation, use the following form:

```
set GCELL_FETS(<name>) {<base_type> [-lmin <value_in_um>] \
  [-gate_space <value_in_um>] [-contact_space_to_gate <value_in_um>] \
  [-add {<layer> <reference_layer> <x_overlap_in_um> <y_overlap_in_um>]
```

where **<name>** is the name of the fet to be created, such as **pmos_lvt** or **nmos_25**.

To make this fet:

- First draw a fet with the given **<base_type>** like nmos or pmos, potentially augmented with different values for *lmin*, *contact space to gate* or *gate spacing*.

- Then add layers with the `name` <**`layer`**> that is drawn with reference to <**`reference_layer`**> but with a given X overlap <**`x_overlap_in_um`**> and Y overlap <**`y_overlap_in_um`**>.

  For example, the low-threshold pfet could be specified with the following line:

  ```
  set GCELL_FETS(pmos_lvt) {pfet -add {vtl_p poly 0.15 0.2}}
  ```

  This would make the device "**`pmos_lvt`**" from a pfet with the additional layer "**`vtl_p`**" added around the poly layer but wider than the poly by 0.15µm in the X dimension and 0.2 µm in the Y dimension.

  Additionally, a high-voltage nfet could be specified with the following line:

  ```
  set GCELL_FETS(nmos_25) {nfet -lmin 0.3 -contact_space_to_gate 0.12 \
    -add {od25 ndif 0.4} -gate_space 0.3}
  ```

  This would make the device "**`nmos_25`**" from an nfet, and having the following characteristics:

  - The default (and smallest) channel length would be 0.3 µm.
  - The contact space to gate would be 0.12 µm and, for multiple-fingered fets,
  - the gate spacing would be 0.3 µm.
  - Additionally, the layer *`od25`* would be placed around the ndif layer, overlapping it by 0.4 µm on each side.

LAYINFO

If you are using the MAX layout generator, you should setup the default values for certain parameters of the cells that you are generating.

Here are some examples:

```
set LAYINFO(stdcell:cell_height) 7.4
set LAYINFO(stdcell:dpc_router_pitch) 0.74  ;# Router pitch for m1, m2, m3, etc.
set LAYINFO(stdcell:power_strap_width) 1.0
```

MAX interactive DRC Limitations

MAX cannot check what are referred to as *non-local* rules. These are rules that require it to look outside of the immediate vicinity of where any modifications to the database occur.

Example of these rules are:

- antenna rules
- wide metal rules
- connectivity based rules
- well/substrate tie spacing

Presently, MAX also cannot check the new via overlap rules that many fabs are currently using in submicron technologies. These rules stipulate that the via overlap (also called the surround) can be very small in two opposite directions if it is larger in the two other opposite directions.

- If you intend on using these rules, you will need to set the DRC rule to be the smaller number and rely on an external DRC checker to test them.

Also, at present the technology source files do not handle parasitic extraction data.

Basic Technology Source File Example

```
# This is a simple technology file which includes the information
# necessary for GDSII input and output, MAX Gcells and the wiring
# tool. Additional information is needed for the layout generator
# and for all DRC rules.
#
```

# layer	gds:dt	txt:dt	type	width	space	color
nw	2	-	-	1.2	0.6	-
ndif	11	-	act	0.3	0.4	-
pdif	12	-	act	0.3	0.4	-
poly	13	-	poly	0.24	0.36	-
pplus	7	-	-	0.44	0.44	-
nplus	8	-	-	0.44	0.44	-
ct	15	-	via	0.30	0.30	-
m1	16	40	metal	0.32	0.32	-
v12	17	-	via	0.36	0.35	-
m2	18	41	metal	0.40	0.40	-
v23	27	-	via	0.36	0.35	-
m3	28	42	metal	0.40	0.40	-
v34	29	-	via	0.36	0.35	-
m4	31	43	metal	0.40	0.40	-
prb	65	-	-	-	-	
text	-	255	text			
bbox	62	-	bbox			
iname	propattr	102	iname			

```
# define fet devices
device nfet from poly ndif
device pfet from poly pdif

drc nfet space 0.5
drc pfet space 0.5

# until max4 must be on same plane
# for fet Gcell generator
drc poly ext nfet 0.36
drc poly ext pfet 0.36
drc ndif ext nfet 0.44
drc pdif ext pfet 0.44

# DRC rules needed for via Gcell
drc m1 enc ct 0.09
```

```
drc m1 enc v12 0.09

drc m2 enc v12 0.09
drc m2 enc v23 0.1

drc m3 enc v23 0.09
drc m3 enc v34 0.09

drc m4 enc v34 0.09

# add other tcl variable definitions with the set statement

# for wire tool
set ROUTE(order) "m4 m3 m2 m1 poly ndif pdif"
set ROUTE(default_layer) m1

##################################################################
# USER DESIGN GRID
# This is the initial value shown in the Grid menu.
##################################################################
set GRID(resolution) 0.01

##################################################################
# ADDITIONAL DRC RULES
# Additional rules that are not specified in the technology
information table.
# Needed for the fet and via Gcell generators
##################################################################
set DRC_DATA(space_to,ct,nfet) 0.22
set DRC_DATA(space_to,ct,pfet) 0.22
set DRC_DATA(enclose,ndif,ct) 0.14
set DRC_DATA(enclose,pdif,ct) 0.14
set DRC_DATA(enclose,poly,ct) 0.14
```

Appendix A
MAX Hotkeys

To view or print out a current list of hotkeys for each mode, use the “`spacebar`” option in the `Help` menu

Appendix B
MAX Text Commands

Text Commands

Refer to the online `Text Commands` (page 107) documentation found in the `Help` menu in MAX for the most current documentation.

The text commands documentation can also be found in the **`text_commands.html`** file found in **`$MMI_TOOLS/doc/max/max_manual`**. This file contains documentation for the MAX text commands.

Appendix C
TCL/TK and The MAX API

Writing Tcl/Tk Scripts

Tcl/Tk is used for the programming interface for MAX. All Tcl and Tk commands are available providing a complete scripting language and toolkit for GUI extensions. Complete information on Tcl/Tk is available from the official Tcl/Tk website at **http://scriptics.com**. We also recommend the book "*Tcl and the Tk Toolkit*", authored by John Ousterhout, and published by Addison-Weseley.

There is an extensive list of text commands which are built on top of Tcl/Tk. (See "Text Commands" on page 153.) Tcl/Tk commands can be typed directly into the MAX `Command Window` (page 22), can be put into a **`.maxrc`** file or into a separate file which can be sourced. Examples of Tcl/Tk scripts can be found in **`$MMI_TOOLS/mmi_local.sample/max/api_examples`**.

Useful Tcl Commands (MMI Extensions)

```
setl {a b c} "4 12 foo"
```

is the same as

```
set a 4
set b 12
set c foo
```

Creating associated lists:

```
set my_list [list "a 25" "type $type" "xyz 5"]
set my_var [get_assoc "xyz" $my_list]
```

The above commands put **`5`** in **`my_var`**

Gcells Tcl Programs

Generator cells or Gcells are parameterized cells that regenerate themselves when they get different inputs. Devices (for example, fets) and vias are good candidates for Gcells since a given layout may have multiple types that differ only by size or some other attribute.

Gcells are user definable using the Tcl scripting language. Presently they must be created and modified using a text editor. To discern Gcells from conventional MAX cells, Gcells are saved in **`.maxg`** files as opposed to **`.max`** files.

A Gcell layout requires the following two structures to be defined:

```
proc gcell_make_<name> {properties} { ... }
define_gcell <name> <property_list>
```

where **`<name>.maxg`** is the name of the file that they are in. Note that other Tcl procedures can also be defined and used in the Gcell file.

The **gcell_make_<name>** procedure is called whenever the Gcell is created or modified and is passed the properties for that given instantiation. The procedure should draw the Gcell oriented around the origin — translations and rotations/mirrors are handled automatically by MAX. Gcells can ONLY use the following drawing procedures to create layout:

```
gcell_paint <layer> <x1> <y1> <x2> <y2>
gcell_label <text> <type> <layer> <x1> <y1> [<x2> <y2>]
gcell_polygon <layer> <x1> <y1> <x2> <y2> ...
```

You CANNOT use the conventional drawing commands like **db_paint** or **db_polygon.**

The `define_gcell` command defines the default user-definable properties of the Gcell and optional choices and checks on those parameters. The simplest form of the **property_list** is the form:

```
{{<prop1> <default_value>} {<prop2> <default_value>} ...}
```

which is also known as an associative list. You can also restrict the choices for a property with the line:

```
{<prop> <default_value> choice|radio <list_of_choices>}
```

For example, a fet Gcell might have the property:

```
{type nfet choice {nfet pfet}}
```

The difference between **choice** and **radio** is the look on the `property` menu that the user will see. The use of **radio** will cause all choices to be listed with radio buttons before them.

You can also do simple checks on property values by using the **-number, -min, -max** and **-incr** switches, for example:

```
{<prop <width> <1.2> -number -incr 0.1 -min 0.6 -max 20.0}
```

This will insure that the user can only enter widths between 0.6 and 20.0 in increments of 0.1.

Typically the first few lines of any Gcell procedure (such as **gcell_make_<name>**) are the following:

```
 # get values out of properties list
set type [get_assoc type $properties]
set width [get_assoc width $properties] ...
```

which will get the actual values out of the property list and assign them to more manageable variables.

Example Gcell

```
# Example generator cell (Gcell) to draw a simple fet.

# This procedure draws a simple fet given a list of properties

proc gcell_make_simplefet {properties} -desc {
  Gcell procedure to make a fet
} {
```

```
  # get values out of properties list
  set type [get_assoc type $properties]
  set width [get_assoc width $properties]
  set length [get_assoc length $properties]

  # fet gate is vertical.  Origin is lower left corner of gate

  # what layers make up this fet?
  setl {poly diff} [techinfo device $type]
  if {$poly == ""} {
    puts "Aborting, Illegal fet type $type."
    return
  }

  # get some key params
  set poly_ext_gate [max [techinfo enclose $poly $type opt] 0.1]
  set diff_ext_gate [max [techinfo enclose $diff $type opt] 0.1]

  set x -$diff_ext_gate

  # draw the poly
  gcell_paint $poly 0 -$poly_ext_gate $length [expr $width + \
    $poly_ext_gate]

  # draw the diffusion
  gcell_paint $diff -$diff_ext_gate 0 [expr $length + \
    $diff_ext_gate] $width

  # the return string will show up when the Gcell is selected
  return "$width/$length"
}

# Fix up the props used to make a fet Gcell.
# Allow fet width only to stretch.

proc gcell_make_props_simplefet {prop_list} {

  # get layers and key parameters
  set type [get_assoc type $prop_list]
  setl {poly diff} [techinfo device $type]
  if {$poly == ""} {
    # illegal type
    return $prop_list
  }

  set minwidth 0.4

  if {[set bbox [get_assoc _BBOX_  $prop_list]] != ""} {
    # compute width from bbox
    set prop_list [rm_assoc _BBOX_ $prop_list]
    set prop_list [rm_assoc width $prop_list]

    setl {x1 y1 x2 y2} $bbox
    set poly_ext_gate [max [techinfo enclose $poly $type opt] 0.1]
    set newwidth [expr $y2 - $y1 - 2.0 * $poly_ext_gate]
    lappend prop_list [list width [uusnap [max $newwidth \
   $minwidth]]]

  } elseif {[set newwidth [get_assoc width $prop_list]] != ""} {
    # Check width validity.
    set prop_list [rm_assoc width $prop_list]
```

```
    lappend prop_list [list width [uusnap [max $newwidth \
    $minwidth]]]
  }

  return $prop_list
}

# now define it and give default arguments

set fets [techinfo devices]

set properties ""
lappend properties "type [lindex $fets 0] choice \{$fets\}"
#
lappend properties "width 1 -number -incr 0.1"

set min_length [techinfo width [lindex [techinfo device \
    $default_fet] 0]]
lappend properties "length $min_length -number \
    -incr [res] -snap 0.1"

define_gcell simplefet $properties
```

Adding Commands to the Local Menu

You can add commands to the `Local` menu in MAX. To add a command into the `Local` menu, you need to add the following lines to a **.maxrc** file.

```
source ~/mmi_private/max/my_commands.tcl
menu_local_cmd "My command 1" command_one
```

You first load in the Tcl file containing your Tcl procedures. You then add the command to the `Local` menu.

The syntax of the `menu_local_command` is:

```
menu_local_command <name in menu> <name of tcl command>
```

This will add the command at the bottom of the `Local` menu. At present you cannot add hotkey bindings to any `Local` menu commands.

.

Index

Symbols

A

B

C

D

E

F

G

H

I

L

Index

M

N

O

P

R

Index

Z