

Cadence Tutorial: Layout Entry

Instructional 'named' Account

1. Get one by logging in to instructional server (in 199 Cory, 273 Soda or over the net using 'ssh' to *cory.eecs.berkeley.edu*) as 'newacct' (passwd: 'newacct') and fill in your information step by step.
2. After request, you will receive an email with your account and password.

Software Installation

To connect to linux/unix machines from your home windows machine, you need to:

3. Download xmanager (Xmanager 3.0 build 0128) from www.netsarang.com. The software is free to try for 30 days but it will continue to work even after this trial has expired.
4. Install the software and open **xstart**
5. Enter quasar.eecs.berkeley.edu in the 'host' field
6. Enter your instructional account and password and select **SSH** as 'protocol'
7. For 'execution command' click the arrow on the right and select **3 xterm (Solaris)**
8. Click ok and a terminal window should open for you.

Alternatively, you can use Exceeds (or other X windows software) + SSH secure shell, but the method given here is simpler relatively.

Setup

1. **(Optional) Make a Cadence directory:**
mkdir cadence
Cadence generates a lot of files and directories, so it is recommended that you make a separate directory (i.e. ~/cadence)
2. **Set the executable and license paths:**
/share/b/bin/cadence-setup.csh
Then it will automatically pop out a line:
'What directory [/home/aa/grad/****/ee141]?', then you type in your cadence directory, e.g.
~/cadence

Starting

1. Go to your cadence directory:
cd cadence
2. After going to your cadence directory, in a UNIX command window, type
/share/b/bin/virtuoso6 &
*Alternately, type **virtuoso6 &** in your cadence directory to run virtuoso (ver. 6)*
The Cadence "log file" window should pop up on your screen, and you can start using Cadence
3. Commands that start Cadence tools on the Instructional UNIX systems include: /
share/b/bin/icfb2

/share/b/bin/layout2

/share/b/bin/spectre2

/share/b/bin/cdsdoc2

(The *2 indicates that it is a script that EECS Instruction developed.)

4. Library Manager

After loading Cadence the command interpreter window (CIW) should be visible and the library manager window should also be open. If it is not, start the manager from the Tools menu in the command window:

CIW -> Tools -> Library Manager

Creating a Technology File/Library

1. Cadence requires that you have both:

- A design library that will store the imported file
- A technology file/library, which defines the layers.

The design library and the technology library can be the same library, or they can be different libraries.

2. If you don't have a technology library, you need to create a new one:

CIW -> Tools -> Technology File Manager -> New

Technology Library Name: your_library_name

Load Existing Technology Library: cdsDefTechLib

(You can choose a different existing technology file if you want, but **cdsDefTechLib** is recommended if you are going to create a completely new technology.) At this point, I believe that you have to quit and restart Cadence in order to be able to use your new technology library.

3. If you don't have a technology library but you have a technology file (e.g. **ee245tech.tf**), you need to create a new technology library:

CIW -> Tools -> Technology File Manager -> New

Technology Library Name: your_library_name

Load ASCII Technology File: ee245tech.tf

4. If you already have a technology library, but you want to create another new destination library:

Library Manager -> File -> New

Library Name: your_library_name

Attach to an Existing techfile

Attach to Technology Library: your_existing_library_name

Add/Edit Layer Appearance

If your imported file has layers that were not previously included in your technology file, or if you created a new technology file, you might want to define or edit the color and stipple pattern of the layers. Two steps are needed as follows:

1. ADD layers to your technology file

CIW -> Tools -> Technology File Manager-> Edit Layers

Technology Library: your_library_name

Click "**Add**" to add the layers you want to have in your layout and then name each layer and choose appearance in the pop out "**Add Layer Purpose Pair**" window or you can revise the layer properties later by selecting a layer in the "**Layer Purpose Pair Editor**" window and click "**Edit**"

To change the color and stipple pattern, choose a different predefined display packet under the **Display Resources heading** (e.g. POLY1, POLY2)

To edit another layer, unselect the current layer in the "**Layer Purpose Pair Editor**" window, and select the next layer. A selected layer is surrounded by a thin box. It can be quite difficult to see if a layer is selected.

After this step, you can define all the layers needed in your layout without having to keep track of which display.drf file you are using, but it is more tedious to change the layer appearance, because you can only do that by selecting different display resources which are predefined. Therefore, the following step is needed for you to revise the layers to your desired appearances.

2. Edit layer appearances

CIW -> Tools -> Display Resources Manager -> Edit

Application: Virtuoso

Tech Lib Name: your_library_name

Layers: LSW

(Select a layer and edit the color, pattern, fill etc. as you desire)

File -> Save

If you call the file display.drf and save it in your Cadence start directory (e.g. ~/cadence/display.drf), it will automatically load whenever you start. Otherwise, you will have to load it yourself (File -> Load) later on.

Since your DRF is unique, other users won't see the same layer colors unless they copy your DRF. If you are sharing layouts, this can be a problem.

Layout

1. In your target library choose:

File > New > Cell View

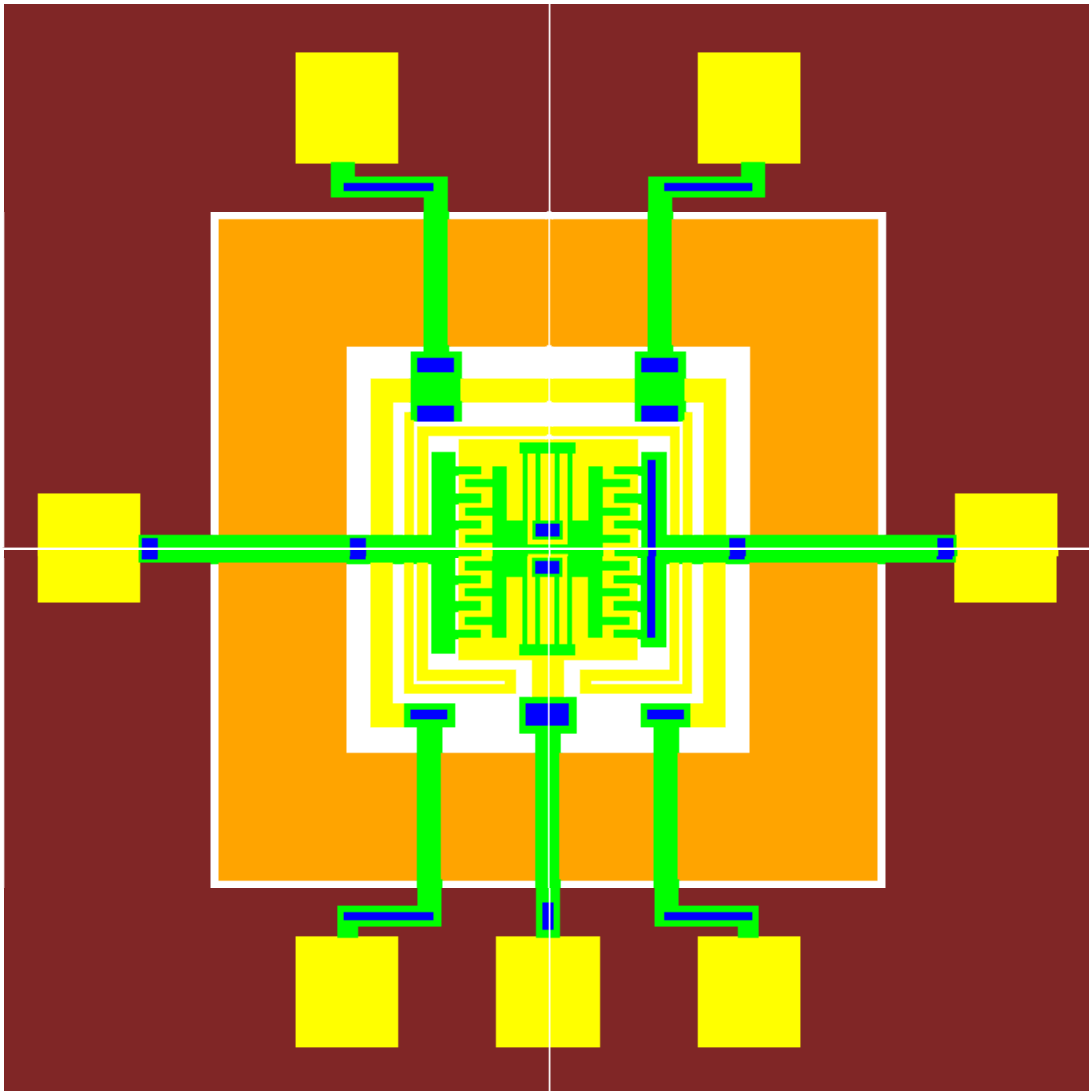
Select **Virtuoso** as the Tool. The View Name should automatically change to *layout* if you click in a different field. Click Ok and click Ok once again in the ensuing pop-up "Add AMS...". A Virtuoso Layout window will now open. Before entering any layout, set the grid with:

Options > Display or type e

Change the X and Y Snap spacing to facilitate your drawing and click on Ok. Depends on dimensions of different feature, the snap spacing is adjusted very often. Now you can start to draw your mask.

2. To draw the mask, say a **SNC**, first choose the corresponding layer in the LSW window by clicking on the layer. Then move your cursor into the layout window where you want to draw the **SNC** layer and type **r** and move your mouse. A brown box will appear indicating the boundary of the **SNC** mask. Just click on the left mouse button to draw the **SNC** rectangle. To change the dimension of the rectangle, move your cursor to the side where you want to extend or shorten such

that the side is highlighted and then type **s**. The side will move with your cursor. Likewise draw the **SNT** and so on. The layout will look like:



When you draw, in order to make sure the dimensions are correct, type **k** and LMB click on the layout the starting point that you are measuring and move the mouse till you hit the ending point. If you want to zoom in, type **z** and drag a rectangle area that you would like to zoom in. Type **f** if you want to fit your entire view to the window. Often times you need to array your devices across the wafer, then to implement that you can just simply copy the feature that you already finished in layout. To do that, select the blocks that you want to duplicate by dragging a rectangle containing all of them and then type **c**, move you mouse to put the copy.

Export

1. If you are exporting a GDS file
CIW -> File -> Export -> stream
2. If you are exporting a CIF file

CIW -> File -> Export -> CIF

In the "Stream Out" or "CIF Out" window

Library Name: your_library_name

Top Cell Name: your_export_cell_name

View Name: layout

Output: Stream DB -or- CIF DB

Output File: your_output_file_name.gds (or .cif)

The following settings convert the layout to a micron scale (i.e. 1.0 in the layout equals 1micron in the exported file).

Scale UU/DBU: 0.001

Units: micron