ELE462: Design of Large Scale Integrated Systems

Tutorial 1: Cadence Design Environment Setup

Introduction

This tutorial covers schematic entry, simulation, layout, and RC extraction using Cadence (`icfb`) and a generic 90nm Process Development Kit (gpdk090).

Cadence is installed on the OIT servers nobel, lxiv and sixtyfour, but we assume nobel in this tutorial (we used to use hats, but it has since gone off-line and is no longer in use). You can run Cadence from any Linux or Windows based local machine.

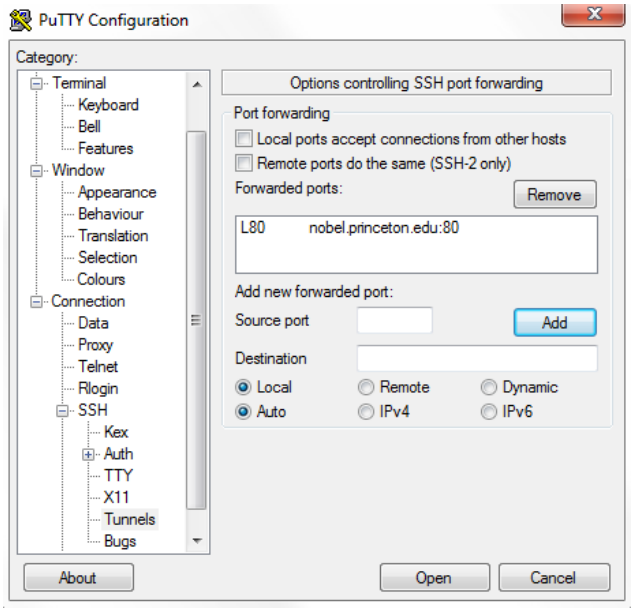
NOTE: In these tutorials, the commands you need to enter on a terminal or operations you do on menu buttons are shown after a ">" and in red. You should type the command without the ">" on the terminal. Copy pasting it directly from this file may not work.

If you are using a linux machine, you can connect to nobel as follows.

```
> ssh –X netID@nobel.princeton.edu
```

You'll be prompted for your password. Enter it and you are in.

If you are using a windows machine, you can use SecureCRT, SSHsecure Shell or putty to connect to hats. You can google and download free versions of any of these. Find out how to set up the "x11 forwarding" options and connect to nobel.princeton.edu using your netID and password. These options are available in the "tunneling" tab in putty. You need to select a local port "80" (say) and a remote port "80" (say) and enter remote server ID as nobel.princeton.edu for x11 forwarding.



If you are using SecureCRT to connect, make sure in the session properties, `Connection > Authentication > Primary` is set to Keyboard Interactive. Use the *ssh2* protocol. In `Port Forwarding > X11`, check Forward X11 Packets. Also make sure your computer display is set to 32-bit color.

If you are using putty, go to SSH > X11 and check Enable X11 forwarding.

The instructions from now are shown for a linux terminal. They hold equivalently for a windows machine as well.

Environment Setup

You need to set up the cadence environment path to use it.

If you are using csh, open the file ".cshrc" from your home directory (note the "dot" before the file name. This means that the file is "hidden" and not visible when you list ('ls') the files; it is visible if you use the 'ls –l' option). You can open .cshrc by typing say,

```
> emacs .cshrc
```

Now copy, the following to this file.

```
source /usr/licensed/cadence-20050411/cshrc
setenv CDS_Netlisting_Mode Analog
if ( $OS == "SunOS5" ) then
setenv CDSHOME /usr/licensed/cadence-20050411/IC5141.sun4v
else
setenv CDSHOME /usr/licensed/cadence-20050411/IC5141
endif
setenv ASSURAHOME /usr/licensed/cadence-20050411/ASSURA41
setenv PATH "$ASSURAHOME/tools/bin:$ASSURAHOME/tools/assura/bin:${PATH}"
```

If you prefer to use the bash shell, you can open up the ".bashrc" or the ".profile" file in your home directory (if this file is not there, create one). Then copy the following to it.

```
##########################################################################
# Cadence environment set up - 5.10.41
##########################################################################

source /usr/licensed/cadence-20050411/profile

# Analog netlist extraction variables

export CDS_Netlisting_Mode=Analog

if [ $OS = "SunOS5" ]; then
export CDSHOME=/usr/licensed/cadence-20050411/IC5141.sun4v

elif [ $OS = "Linux2" ]; then
export CDSHOME=/usr/licensed/cadence-20050411/IC5141
fi
export ASSURAHOME=/usr/licensed/cadence-20050411/ASSURA41
export PATH=$ASSURAHOME/tools/bin:$ASSURAHOME/tools/assura/bin:$PATH
```

After you are done copying the above to the .cshrc/ .bashrc, type

```
> source ~/.cshrc
```

OR

```
> source ~/.bashrc
```

File locations

Here are locations of various files that you will use throughout the tutorial.

Cadence GPDK: ~nverma/public_html/cadenceSetup_5.10.41/gpdk090_v4.4
Cadence GPDK models: ~nverma/public_html/cadenceSetup_5.10.41/gpdk090_v4.4/models
Cadence GPDK documentation: ~nverma/public_html/cadenceSetup_5.10.41/gpdk090_v4.4/docs
Spectre library file: ~nverma/public_html/cadenceSetup_5.10.41/gpdk090_v4.4/gpdk090/gpdk090.scs

Creating a Design Project in Cadence

A unique directory should be created for your `ele462` activities. Use the `mkdir` command to create the desired directory structure, for example:

```
> mkdir ~/ele462
> cd ~/ele462
```

Now, you will create the cds.lib file which contains the names and paths of Cadence libraries. You will also create an initialization file which contains commands to be executed by Cadence at startup. These files must be in the same directory where you start Cadence (e.g. ~/ele462). First copy the template files as follows:

```
> cd ~/ele462
> cp
/n/homeserver2/eehome2a/PUBLIC_HTML/nverma/cadenceSetup_5.10.41/cds.lib .
> cp
/n/homeserver2/eehome2a/PUBLIC_HTML/nverma/cadenceSetup_5.10.41/cdsinit .
> cp
/n/homeserver2/eehome2a/PUBLIC_HTML/nverma/cadenceSetup_5.10.41/display.drf .
```

In the working directory, copy the gpdk090 model and stream files.

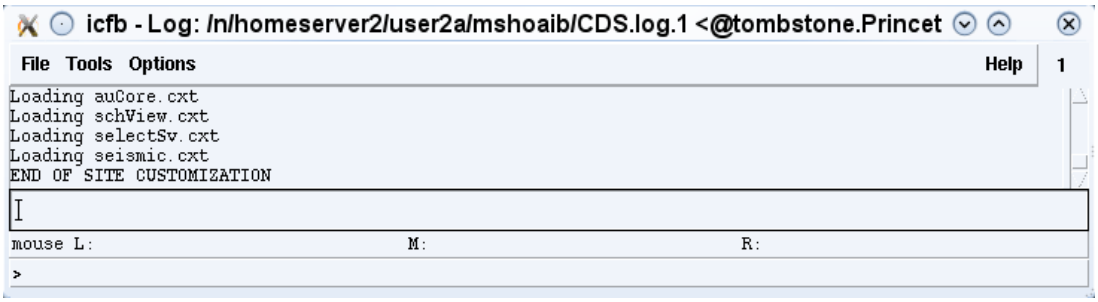
```
> cp –rf /n/homeserver2/eehome2a/PUBLIC_HTML/nverma/cadenceSetup_5.10.41/gpdk090_v4.4/models .
> cp –rf /n/homeserver2/eehome2a/PUBLIC_HTML/nverma/cadenceSetup_5.10.41/gpdk090_v4.4/stream .
```

Starting up Cadence

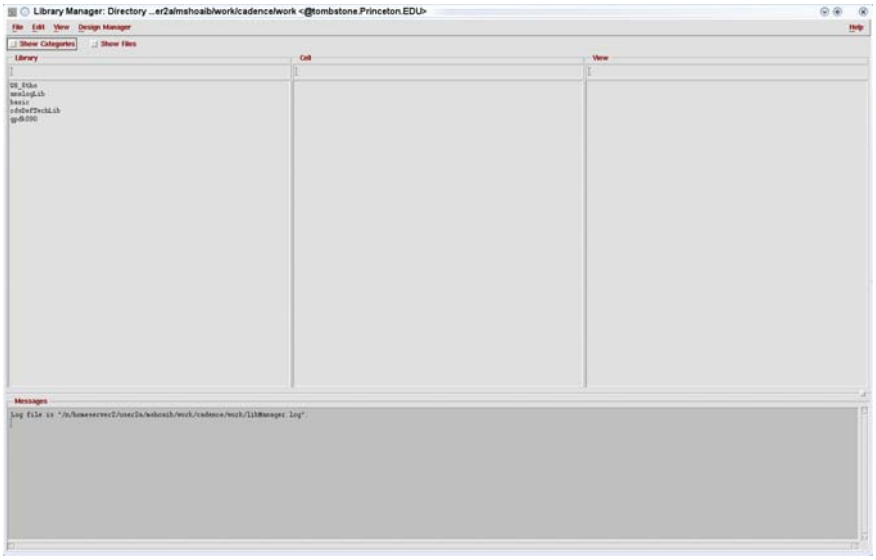
To start cadence from the working directory, type

```
> icfb &
```

This will start the Cadence Integrated Circuit Front to Back environment. The main `CDS.log` window should appear:

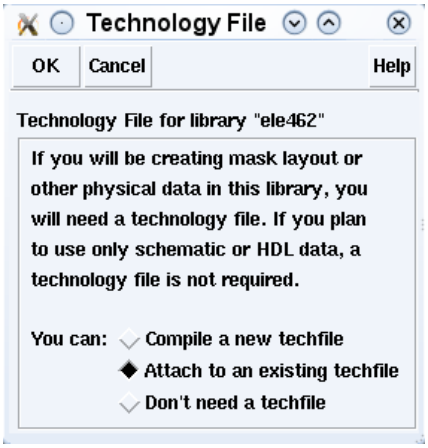


Start library manager (`Tools > Library Manager`). The library manager displays all the libraries referenced in cds.lib.



Now you are going to create a new library. In the library browser, select `File > New > Library`, then type a library name (e.g. ele462) in the name field. Click `OK`.

Select `Attach to an existing techfile` option. Click `OK`.



Select `gpdk090` from the `Technology Library` menu and then click `OK`. This attaches the 90nm PDK to your new library, which means that designs in the library will make use of the technology file, layer definitions, and design rules of the PDK. Your `CDS.log` window should now read “Design library ‘ele462’ successfully attached to technology library ‘gpdk090’.

You will see `ele462` in the list of libraries in the `Library Manager` window. You are now ready to create schematics in the new library.