

# **EE4610 Digital IC I**

## **Instruction Manual**

*Last revised November 2025*

This document is a guide for the setting up and accessing Cadence for the course EE4610 Digital IC I Design. The technology used in this course is updated to TSMC 180 nm due to license issue in ADE (Previously it was 90 nm).

## **1 | Cadence Setup**

We access the Cadence design tool from a remote server, namely *ee4610.ewi.tudelft.nl*. Windows users will connect over SSH using MobaXterm. The procedure for setting up is listed below:

1. Open MobaXterm and type *ssh -X <netid>@ee4610.ewi.tudelft.nl*  
(For eg. if your netid is abc then type abc@ee4610.ewi.cdtudelft.nl)
2. Enter your password in the terminal
3. Change the password to your own by command *passwd*. You need to enter your current password firstly, and type new password two times.
4. Create a directory '*tsmcBCD*' using the command *mkdir tsmcBCD*
5. Change home directory using *cd tsmcBCD*
6. Type */opt/ei/DK/tsmc/oa180/mini018BCDG2/216A/et4382/start* in the terminal
7. Download CalibreSetup folder from Brightspace. There are 2 documents in it, ".cdsinit" and "sourceme". Copy them to *tsmcBCD* folder and cover the initial ones. (Note that the ".cdsinit" is a hidden file, you need to set it visible after download.) If you cannot copy files, open the initial version with command *gedit sourceme* and *gedit .cdsinit*, and copy the new texts covering the initial ones.
8. Type *source sourceme* in the terminal
9. Type *virtuoso &* to start up cadence

These steps needs to be done only once. From next time onwards, to start up cadence follow the below steps.

1. Open MobaXterm and type *ssh -X <netid>@ee4610.ewi.tudelft.nl*  
(For eg. if your netid is abc then type abc@ee4610.ewi.tudelft.nl)
2. Enter your password in the terminal
3. Change home directory using *cd tsmcBCD*
4. Type *source sourceme* in the terminal
5. Type *virtuoso &* to start up cadence

### **NOTE:**

While creating your library, make sure to link your library to *tsmc18*.

While opening schematics and analog environment (ADE L or ADE XL) you may face warnings regarding license, please click *OK* or *Always*.

For design of oscillators and TDC, use transistor models nmos2v, nmos5v, pmos2v and pmos5v.

Suggestion from us is that in case Cadence is slow, you can implement the initial design or the entire design in LTspice (Technology used is similar to the one used in Analog CMOS I and II so you can reuse that setup files).

**For Macbook:**

If you're using a Macbook, please download a software "XQuartz". Here is the link: <https://www.xquartz.org/>  
It is a command-based software with the same property as MobaXterm. Since it doesn't have a graphical interactive page as MobaXterm, here are some useful command:

*ls*: search files and folders under the current path;

*ls -a*: Search everything including hidden files under the current path;

*cd tsmcBCD*: Go to folder tsmcBCD;

*gedit sourceme*: Open file sourceme with gedit editor.

*gedit .cdsinit*: Open file sourceme with gedit editor. //Open these 2 files and replace them with the given texts in "CalibreSetup" folder.

".cdsinit" is a hidden folder. In MacOS, using "command + shift + .", you can see the hidden file.

(There is a software for Mac named Filezilla, which can transfer files from your Mac to server. However, I found it's not free. So, copy and paste texts in a file is an alternative way when you need to copy a file.)

Other procedures are similar as Windows.

Tip: if your Macbook could not type ".", please find the Setup of Xquartz → Input → select "Follow system keyboard layout"