Starting Mentor Graphics' *xDxDesigner* Tool Suite for the First Time

(ENGN1630 - Fall 2016)

The Mentor Graphics tool suite is available on all machines in room 194 and 196. You log on to those machine using your regular Brown account.

Setup: The first time you use Mentor Graphics' *xDxDesigner* software suite or the *PADS* printed circuit layout tool, you must run a batch file that sets up a directory system to hold your work and puts a first example of a project file and schematic into your account. It also sets up environmental variables and registry entries for system support and fixes problems with the "freeze" software that we use to protect the software binaries. Among other things, the setup operation gives you access to some schematic symbol libraries and printed circuit land patterns.

To set up the directory system, use the mouse to select the start-menu sequence: "Start/All Programs/Electrical/PADSVX.1.2/Copy_INI." This may take a minute or more to complete but when it does, you will find three new directories off your root directory on the U:\ drive. One called "PADS_WDIR" holds system files and you should not change these. The other thing that U:\PADS_WDIR holds is the Excel spreadsheet you use to format your bill of materials.

The second directory called, "Engineering Classes," has a full default project space. In it there is a default schematic, "Example.1" that has only a first page with a border but no circuit. It has all the symbol libraries ready to use and may be the basis for the schematic required in ENGN1630. Finally, the symbol libraries are stored in U:\PCB Libraries," where you are free to edit or add to the sets.

Setup for Returning Customers: If you have used *xDxDesigner* before and are just moving to the newest (PADSVX.1.2) version, you need to update some of the files that set initial conditions and directory pointers. In the directory U:\PADS_WDIR delete the four files: DxDesigner.wsp, DxDesigner.xml, DxDesignerNL.wsp, and borders.ini. Then rerun the start-menu sequence: "Start/All Programs/Electrical/PADSVX.1.2/Copy_INI." This will replace links to the executable files, refresh the symbol libraries, and update your directories.

Startup: The simplest way to start the *xDxDesigner* schematic entry tool is to click on the green icon on your desktop. Other ways to invoke the tool are the start menu actions Start/All Programs /Electrical/PADS VX.1.1/ Design Entry/xDxDesigner VX.1.1 or Start/All Programs /PADS Favorites/xDxDesigner. Usually the splash screen as the program opens will have a link to the example project. If that is not there, use the menu sequence in *xDxDesigner*: Files/Open/Project and, if necessary, browse to U:\ Engineering Classes. (Most likely that directory will be the default when the tool opens.) Select the project file, "EngineeringClasses.prj". When that opens, go to the leftmost "Navigator" pane. Expand the "Board" item in the tree and double click "Example1". The schematic that opens will only have a blank sheet with a block frame for documentation. Symbols

come from symbol libraries you access through the "DxDatabook" tabbed pane at the bottom of the screen. See the ENGN1630 manual for more details.

Known Bug: Mentor designed this software to be used for sharing schematics between designers and implemented a scheme that prevents simultaneous changes to the schematic by different users. The method they used creates a lock file when you start editing and deletes that file after you close the software. Our file server is slow at responding to the deletion command and if you log off too quickly the deletion will not take place. The next time you try to open the schematic, you get an error message about your being unable to access the database that holds your work. To fix that problem:

- Browse to U:\<your project directory>\database\cdbsvr

- Delete the file: sAddress.adr.

- Now reopen your schematic.

- Next time wait to log out for about 10 seconds after the command window for *xDxDesigner* closes.