

Starting and Using Mentor Graphics' *PADS Designer* Tool Suite

(ENGN1630 - Fall 2019)

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

First-time Setup: The first time you use Mentor Graphics' *PADS Designer* 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 land patterns for printed circuit board design.

To set up the directory system, use the mouse to open the Win 10 start menu and select "PADSVX.2.5 (32-bit)/_Copy_INI." This macro may take a minute or more to complete but when it does, you will find three new directories off your root directory on the H:\ drive. One called "H:\PADS_WDIR" holds system files and you should not change these. That directory also has ProcessBOMwithDigikeyRFQ_v4.xlsm, an Excel spreadsheet you use to format your bill of materials.

You should also find four new icons on your desktop. Use the green one labeled "PADS Designer VX 2.5" to start the schematic generation tool. The icon calls a macro that copies a couple of files before starting the DxDesigner module. When the program closes, wait long enough before logging out that the command window for the macro closes.

The first time you open the program, a pop-up window will open with a choice between two licenses. Using the mouse, choose the PADS Standard license, not the AMS license.

The second directory on H:\, called, "Engineering Classes," has a full default project space. In it there is a default schematic, "SimpleSchematic.1" under the board "TestPADSSetup." It has symbol libraries ready to use as the basis for the schematic required in ENGN 1630. To check that you have the ENGN 1630 libraries, start PADS Designer using the green icon. Once open, use the menu bar to select View → PADS Databook and expand the partition ENGN163. Finally, the symbol libraries are stored in "H:\PCB Libraries," where you are free to edit or add to the sets. For example, if you want to add some notation to the symbol, you can select the symbol in the Databook, right mouse-click and choose "Edit library symbol."

Setup for Returning Customers: If you have used *PADS Designer* before and are just moving to the newest (PADSVX.2.5) version, you need to update some of the files that set initial conditions and directory pointers and you may need to change directory mapping. In the summer of 2019, our IT group finished a process of moving all user files for Engineering from the U:\ drive to an identical drive called H:\ to coincide with usage

in other departments. If you have your PADS files on a U:\, you should move them to your H:\ drive immediately.

The next step is to delete the five files: DxDesigner.wsp, DxDesigner (an XML file), DxDesignerNL.wsp, DxDesignerPADS.wsp, and borders (a Configuration settings file) from the directory H:\PADS_WDIR. Then rerun the start-menu sequence: “Start /PADSVX.2.5 (32-bit)/_Copy_INI.” This move will replace links to the executable files, refresh the symbol libraries, and update your directories. It will add new icons for starting the Mentor tools to your desktop but will not delete the old ones. You should delete those by hand if they are still there.

Startup: The simplest way to start the *PADS Designer* schematic entry tool is to double-click on the green icon on your desktop. Another way to invoke the tool is the start menu action PADS VX.2.5 (32-bit)/PADS Designer VX.2.5. The program should open with a start page pane that has a link to the H:\ Engineering Classes project. Select that link. When that opens, look at the leftmost “Navigator” pane. The “Board” item in the tree should be expanded to show the “Schematic” item “SimpleSchematic.1”. The schematic should be open, showing a simple *RLC* circuit. These and other symbols come from symbol libraries you access through the “PADS Databook.” See the ENGN 1630 lab 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 H:\<your project directory>\database\cdbsvr
- Delete the file: sAddress.adr.
- Now reopen your schematic.
- Next time wait to log out at least 5 seconds after the command window for *PADS Designer* closes. (You will see the close-out script executing in a command window; when the window closes, it is safe to log out.)

Bill of Materials: When your schematic is complete and you are ready to generate a bill of materials, select Tools → Part Lister to get a dialog box that will let you select the scope for this BOM: a block (= your schematic), a board (possibly comprising several schematics), or the project (possibly of several boards). After you click “Run,” there should be a new .txt file in your Engineering Classes folder with the name corresponding to the selected scope. Next, navigate to H:\PADS_WDIR and open the Excel file “ProcessBOMwithDigikeyRFQ_v4.xlsm.” Select View → Macros → View Macros →

ProcessBOM, which will open a file dialog that asks you to locate your .txt file for input, and then select a location and name for the output spreadsheet. This output contains the bill of materials (all of the parts referenced in your schematics). If there are parts that are not fully specified, for example, there is no supplier part number (VENDOR parameter in PADS Designer), then you need to go back to the schematic, go on-line to find a vendor part number, and enter that on the schematic. For ENGN1630, the preferred vendor is Digikey.com as they have a good professional web site and good prices.