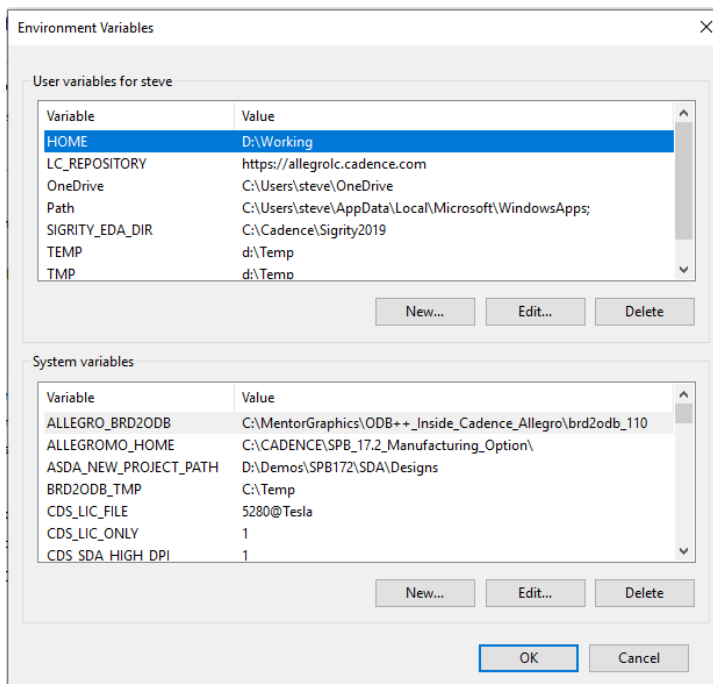# Introduction

There are several settings that can be defined to help manage the PCB Editor software. User settings like HOME, PCBENV and ENV which enable the user to pre-define paths for libraries, default settings and function keys. There are also Administration level settings like CDS_SITE and CDSROOT that allow an Administrator to preset options for all users within a company. This Technical note describes how to define and manage these environment settings.

PCB Editor Settings – Home Variable.

HOME is a system variable that defines the location for your PCB Editor Environment settings. It is recommended that every user has their own HOME folder. This can be defined during the installation of the Cadence PCB Editor software.

To change the HOME variable go to Windows button – Type Path followed by a return then choose Environment Variables.

HOME can be a user variable or a system variable. If you set HOME as user Variable, every user needs their own variable. If only one user works on the system you can define it as a system variable.

If you need to modify this setting, select it using a left click on Edit to edit the variable.
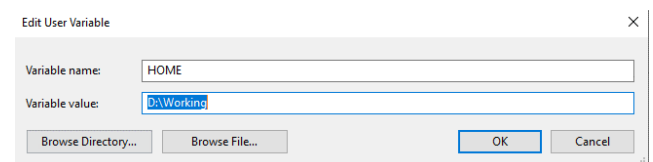
Modify the Variable value to point to the relevant directory. Ensure that the user has full privileges for the directory specified.

PCBENV folder.

Pcbenv is a folder in the HOME folder and contains the PCB Editor environment. This directory will be auto-generated if it does not exist. Below is a list of some of the files normally found in this directory: -

Env   This is the Environment file. It is read by PCB Editor when the software is started. It contains individual user settings such as aliases, function key definitions, library paths to access files on the system, and system variables used by PCB Editor to find the software.

allegro.ini	This file keeps track of the path where you're working file is located. It keeps track of the size and location of the main tool window. DO NOT EDIT THIS FILE! If you are having problems with PCB Editor, this file can be deleted as a form of troubleshooting. It will be created automatically the next time you start PCB Editor.
allegro.mru	This file stores a list of the most recently used board files. DO NOT EDIT THIS FILE!
allegro.ilinit	This file contains the location of any skill files that are auto-loaded when the software is started. Please refer to How to add skill routines for further information on skill.
allegro.geo	This file remembers where the forms last came up and places the same type of form in the same location. DO NOT EDIT THIS FILE!
myfavorites.txt	This file contains which class/subclass(es) are to be displayed in the My Favorites folder of the Color Dialog form.
my_favorites	This file contains any user preferences that have been set in User Preferences as favorite settings.
pad_designer.geo	This stores the Pad Designer form location and size.
pad_designer.mru	This stores a list of most recently used padstacks and paths.
license_cache_allegro_17.4-2019.txt	This file stores a cache of the available licenses for PCB Editor and is auto-generated. If you get access to a new license file you can either manually delete this file or use the Reset License Cache button on the PCB Editor license picker dialog.

cdssetup folder

Cdssetup is a folder in the HOME folder that contains the OrCAD Capture (CIS or DE CIS) user setup structure. This directory will be auto-generated if it does not exist. Initially there will be an OrCAD_Capture folder and an OrCAD _PSpice folder (license dependant) and then there will be the release version folders like 17.4.0 or 22.1.0 as well as a tclscripts folder which will contain any user defined / installed apps. In the OrCAD_Capture\Release_Version folder are some default files that can include some of the following files:-

Capture.ini	When Capture starts up, it uses a pre-defined set of default values for the application settings. These default values are defined in the Capture configuration (Capture.ini) file. If you are running Capture for the first time on a computer, it uses a pre-defined set of configurations to create the INI file. After this, every time you make any configuration changes, this file is updated when you close Capture.
Spinfo.ini	This lists the recent files list that is available on the OrCAD Capture Start Page.
TreeInfo.lbt	This is an auto-generated file that stores the library indexing. DO NOT MODIFY this file.
Backini.unk	This is an auto-generated file that stores the default library settings for Capture including the PCB Footprint locations and any libraries that have been added.

BackupCaptureCIS.ini   This is an auto-generated file that stores the default library settings for Capture CIS including the PCB Footprint locations, Part Library Directory locations and the CIS DBC file that has been defined under Options – CIS Configurations.

Path Definitions in the ENV file.

The env file contains all path definitions which are different from the default system settings. When you change these setting in PCB Editor by using Setup > User Preferences > Paths > Library or Config the changes are written to the env file. You can also change the env file by opening it with a text editor like WordPad. Below is an overview of some of the possible path settings. The two most important ones for library definition are padpath and psmpath. These store the default paths for PCB Footprint and padstack definitions.

For a description of all user preference information open the user preferences from Setup > User Preferences then select Info with the LMB.

Category Paths – Library

devpath	Search path for library devices (.txt). Not used for OrCAD Capture or DE HDL. Only required for third party netlist's.
interfacepath	Search path for Interface files (.idf)
miscpath	Search path for miscellaneous file types. Supported types are dxf conversion (.cnv).
modulepath	Search path for design reuse modules (.mdd).
padpath	Search path for library padstacks (.pad).
parampath	Search path for parameter files (.prm). These allow reuse of physical design data option settings like text, visibility and grid settings.
psmpath	Search path for library symbols (.psm .osm .bsm .ssm .fsm).
step_facet_path	Search path for STEP facet files (.xml).
step_mapping_path	Search path for STEP mapping files (.map) for device.
steppath	Search path for STEP models (.stp .step).
techpath	Search path for technology files (.tech).
topology_template_path	Search path for topology template files (.top).

Category Paths – Config

accpath	Search path for ACC project.
aptpath	Search path for aperture flash files -- this is obsolete with .fsm support (.bsm).
artpath	Search path for artwork parameter files (.txt) and artwork aperture files (.txt).

clippath	Search path for sub-drawing files (.clp).
dclpath	Search path for decoupling capacitor list files (.dcf).
dfaauditpath	Search path for DFA Audit (.arl .rle).
dfacnspath	Search path for dfa constraints spreadsheet files (dfa).
Idxfilterpath	Search path for IDX object filter configuration file(.config).
Idxpath	Search path for IDX files(.idx).
ipc2581attrpath	Search path for IPC2581 property configuration file(.atr).
ipc2581spec_path	Search path for IPC2581 spec configuration file(.xml).
ldfpath	Search path for Library definition file (.ldf).
lstpath	Search path to locate list files (.lst).
materialpath	Search path to locate materials.dat (Allegro) or mcmmat.dat (APD) (.dat).
ncdpath	Search path for NC Drill parameter files (.txt).
pcell_lib_path	Search path for pcell component implementation (.il .ile).
prfeditpath	Search paths for user preferences files.
scriptpath	Search path for scripts.
textpath	Search path for extracta command files (.txt).
tilepath	Search path for reusable die pin tiles (.til) (APD).
viewpath	Search path for visibility schema files (.color).
wizard_template_path	Search path for Allegro templates (.brd .dra).
xtalk_table_path	Search path for Cross talk tables (.xtb).

File Management and Structure.

In User Preferences there is a category called file_management. This category is used to define a file structure that will assist in the project directory standardisation if required.

|	
Autosave	Enables autosaving. It must be set/unset before starting Allegro.
autosave_dbcheck	Enables quick database check before an autosave. In default mode, this is turned off because this increases the time for a save.

autosave_name	Sets base name used for the autosave file. The default name is AUTOSAVE. Do NOT provide a file extension. Allegro will use the appropriate extension for the type of database under edit.
autosave_time	Controls autosave intervals. The default is 30 minutes. The minimum is 10 minutes and the maximum is 300 minutes.
ads_logrevs	Enables file versioning for Allegro log files. Value <n> = number of versions you want maintained.
ads_sdart	The subdirectory to which artwork files should be written.
ads_sdlog	The subdirectory to which log files should be written.
ads_sdmcad	The subdirectory to which IDF and IDX files are written. Default is the same directory as the design.
ads_sdplot	The subdirectory to which plot files should be written.
ads_sdreport	The subdirectory to which report files should be written.
ads_autosaverevs	Enables file versioning for AUTOSAVE database files. Value <n> = number of versions you want maintained. Default is no versioning.
ads_boardrevs	Enables file versioning for allegro layouts (.brd) and symbol (.*sm) files. Value <n> = number of versions you want maintained. Default is 1 version.
ads_logrevs	Enables file versioning for Allegro log files. Value <n> = number of versions you want maintained.
ads_textrevs	Enables file versioning of allegro files which are not .brd .*sm or .log Value <n> = number of versions you want maintained.
allegro_nolocking	By default, Allegro programs create a lockfile when a design is opened. This allows other Allegro programs to sense that the design is in use. This option disables advisory file locking (pre-16.5 behavior). Users setting this option will not create file locks but still be notified of locks set by other users.
directory_cache	Ignores fully qualified directories in Allegro PATH variables that do not exist. These directories when located remotely can have slow access. The Allegro command, bad_directories, lists these directories but requires you to use Allegro with features that use the PATH variables.
dump_library_directory	Specifies the export directory that Export Libraries (dlib UI command) uses as its directory. Default is the current directory. Location may be a relative or absolute path. Command attempts to create the rightmost directory component if it is not present.
ecadmcad_status_update_interval	Controls update interval of ECAD/MCAD status. The default is 1 minute. The minimum is 1 minute and the maximum is 30 minutes.

ext_artwork	File extension used for artwork (film) files. Default is .art. Recommendation is to set this at the CDS_SITE level. Use caution before changing the extension to ensure all your post-processing tools can handle the new extension.
ext_drill	File extension used for ncdrill files. Default is .drl. Recommendation is to set this at the CDS_SITE level. Use caution before changing the extension to ensure all your post-processing tools can handle the new extension.
ext_script	Change extension used for Allegro scripts. Default extension is .scr. Recommendation is to set this at the CDS_SITE level.
import_file_alarm_enable	Enables import file alarm. It must be set/unset before starting Allegro.
import_file_alarm_interval	Controls import file alarm interval. The default is 10 minutes. The minimum is 1 minute and the maximum is 720 minutes.
journal_nobuffer	By default, the journal file has buffered output to improve performance when writing the file over a network. This option changes the output to unbuffered. The only advantage of unbuffered output is if a program crashes more lines are written to the journal file.
journal_prefix	Prefix appended to the program's journal file name. Two special prefix keys are available; "user" substitutes user's login and "host" which substitutes system name. Prefix must be a legal file name (no / or :). Example journal_prefix = user for login "me" would create an allegro journal file of me_allegro.jrl
temp	or TMP, <directory path with a large quantity of free disk space> This variable causes allegro processes to use the specified directory for temporary storage of data files. Most often, artwork requires more temporary space than the /tmp directory contains.

System Setting CDS_SITE

CDS_SITE is a variable set at the system level. It allows the administrator to define standard settings and paths for all users. CDS_SITE is normally located on a server. Note: With CDS_SITE you can only administrate PCB Editor and DE HDL. This variable is not used for DE CIS or OrCAD Capture.

To define or change the CDS_SITE variable go to Windows button – Type Path followed by a return then choose Environment Variables.
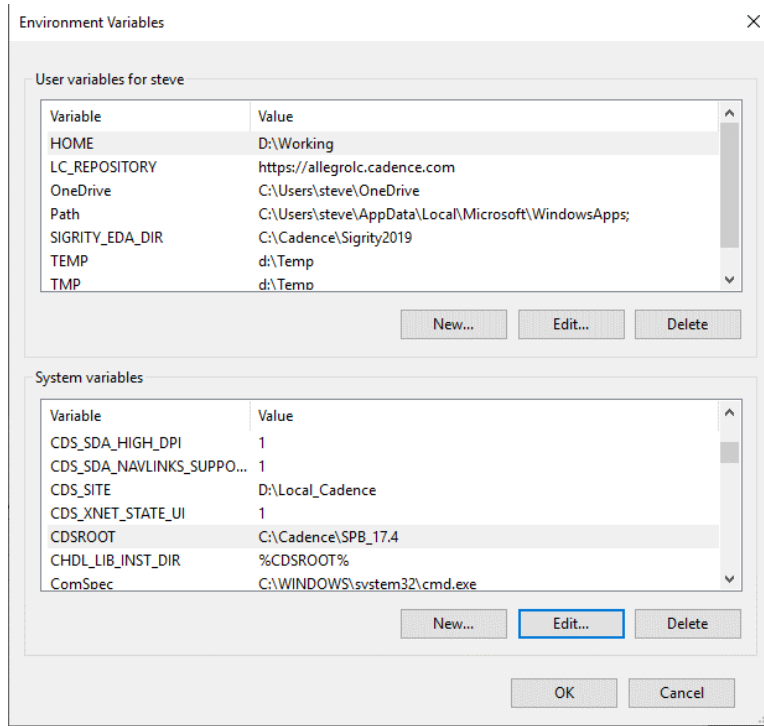
Once the CDS_SITE variable is defined copy any folder structure required from <your_install_dir>\share\ to the CDS_SITE folder e.g. If you require a user defined menu copy c:\Cadence\SPB_22.1\share\pcb\text\nclegend to c:\my_cds_site\pcb\text\nclegend. Then modify the nclegend contents and ncdpath so that PCB Editor looks for the CDS_SITE menu first.

With the CDS_SITE definition you can administrate nearly all of the PCB Editor settings. Here are some examples: Paths to symbols and padstacks, Funckeys, Sub directories for artwork and reports, Paths to Scripts, Paths to Views, Skill Routines.

PCB Editor searches for settings in the following order except for : -
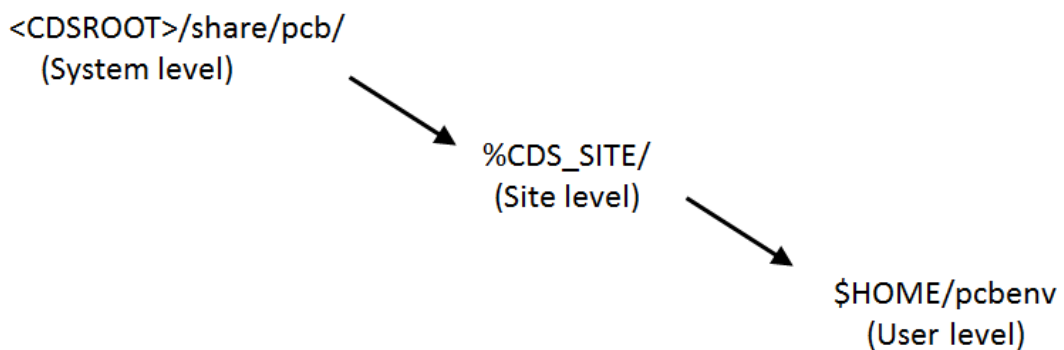
If you want to define a SITE based env file that defines default settings for all users like padpath, psmpath, steppath etc then this can be administered by:-

Define a site.env file that has the SITE settings you require. Store this file in $CDS_SITE\pcb\site.env.

CDSROOT - Working with different Software Versions

CDSROOT is the system variable which defines the install directory you use to start the tools. During installation this variable automatically is set. If you only work with one software version, you do not need to use this function. If you work with two or more software versions, it's important to know which value is set. You can modify the CDSROOT manually, with Cadence Switch Release command.

Cadence Switch Release sets the CDSROOT and all necessary Path settings. You start it by selecting Start > Cadence > Cadence Switch Release.

FYI – Cadence release 17.x no longer requires the use of CDSROOT (or any Computer based environment variables like PATH). When you install 17.x and you no longer require access to 16.6 or older versions the installation will remove any Cadence environment variables. IF you need to use both versions please ensure you leave the environment variables set. You can leave the CDSROOT variable set to C:\Cadence\SPB_16.6 and run both 16.6 and 17.x from the Start menu without the need for the Switch Release

Note: Cadence Switch Release does not change the HOME variable setting.