Title: How to roll out PCB Editor and Capture CIS/DE CIS using SITE and HOME

Product: OrCAD / Allegro PCB Editor
Capture CIS / DE CIS

Summary: There are different ways to administrate Cadence PCB Tools. This Note shows how the different mechanisms work together.

Version 16.6

Author/Date: Beate Wilke / 20.03.2014

This Application Note replaces the 3 Application Notes:

- FlowCAD_AN_PCB_site - How to roll out PCB Editor using Site
- FlowCAD_AN_PCB_environment - PCB Editor Environment
- FlowCAD_AN_PCB_SiteAndHome - How to roll out PCB Editor using SITE and HOME

You download example Site definition from www.flowcad.de/AN

Or watch the webinar video at Youtube.

Table of Contents

1 Introduction

PCB Editor and DE HDL can use up to 3 different sources for path and environment settings.
These are:

- Global env definitions - default settings after installation, do not change
- SITE definition - company/team settings
- HOME definition - personal settings

Up to 16.5 software version, DE CIS had only one user capture.ini in installation structure
Since 16.6 DE CIS can use 2 different sources for path and environment settings.
These are:

- SITE definition - company/team settings
- HOME definition - personal settings

Global env and HOME definitions are used always. SITE is not a must be, it can be used. It is designed to manage environment for larger design teams or whole companies.

In this document the main focus are definitions for PCB Editor and DE CIS.

1.1 Global env definitions

The global env file for PCB Editor is stored in Cadence installation directory under share\pcb\text. If you don't know the exact location of the installation, look at system or user variable CDSROOT. In chapter 2 we will explain how to get the variables and how to change or add a variable.

Important: Never change the global env File and all files which are stored in %CDSROOT/share/pcb. All changes should be stored in Site or Home definition to make them independent from Hotfix changes or reinstallation.

1.2 SITE definition

A Site manages all path and environment settings which are unique for all users.

After software installation PCB Editor and DE HDL use an empty global site structure.
It's located in %CDSROOT/share/local/pcb for PCB Editor and %CDSROOT/share/local/cdssetup for DE HDL. Because the software installation often is on local system this structure isn't useful to manage global settings. Users should use the Site definition.
If you have 3 or more users working with PCB Editor, DE HDL or DE CIS it's better to define your own Site. This Site should be located in a central folder. A good idea is to create a shared folder on a server which contains Site, library and other global files which are used by all users. It would be perfect if you could map this shared folder under the same network drive character for all users.
CAD admin or IT has to make changes only in SITE definition and all users automatically get the changes.

1.3 HOME definition

HOME stores all user specific path and environment information. If you work with less than 3 persons you don't need SITE definition. You can store all information in a HOME definition.

1.4 Cadence Variables Summary

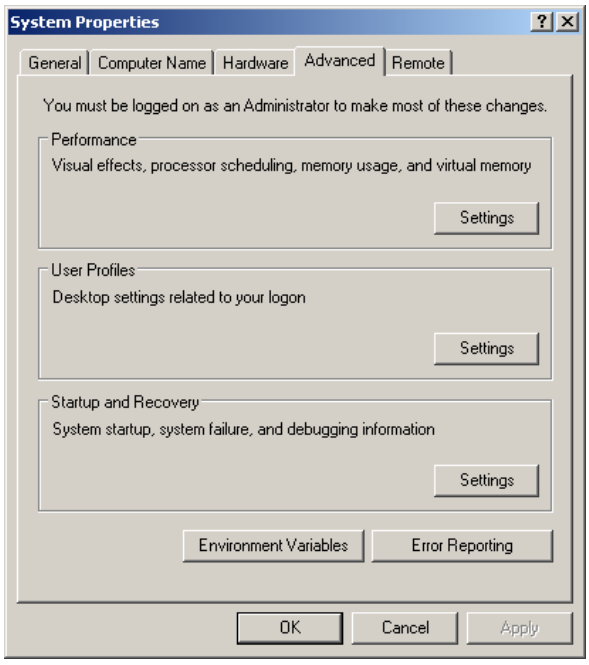
PCB Editor, DE HDL and DE CIS use these main environment variables:

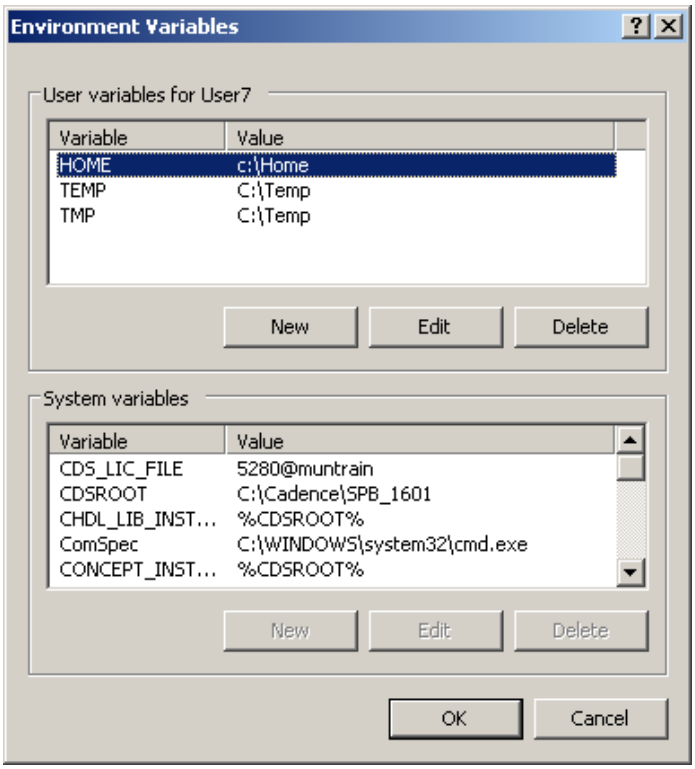
- CDSROOT - Path to installation directory
- CDS_LIC_FILE - Port and Name of license manager
- HOME - Path to user environment
- CDS_SITE - Path to site environment for PCB Editor and DE HDL

2 Defining a Variable

2.1 Example: HOME Variable

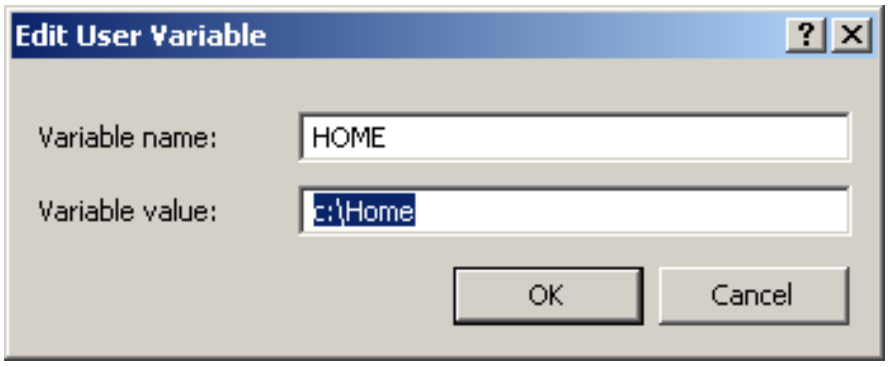
HOME is a user variable that defines the location for your Cadence environment settings. Since 16.6 capture.ini is also stored in HOME folder.
To change the HOME variable, go to **Start -> Settings -> Control Panel -> System -> Advanced -> Environment Variables.**





Because HOME is a user variable, every user needs his own variable. If only one user works on the system you can also define it as system variable.

If you want to change the variable select it and click on Edit.



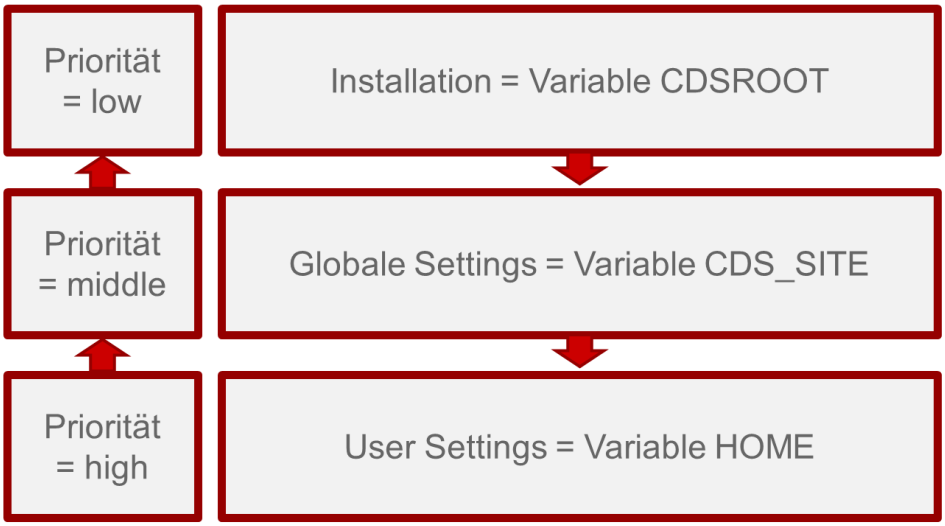
You can change the value to all existing folders where the user has full privileges.

Use the same steps to define CDS_SITE.

If you work with 2 different versions for example 15.7 and 16.5 it would be good to have two different HOME folders. One for 15.7 and one for 16.5 with different path definitions for library because the binary code of 15.7 and 16.5 board files and symbols is different.

The same is with SITE definitions. We recommend saving current SITE and HOME before switching to a new release, for always having a defined backup.

3 Read Order



PCB Editor searches for settings in the following order:

$cdsroot/share/pcb/text
(System level) → $CDS_SITE/pcb
(Site level) → $HOME/pcbenv
(User level)

Allegro DE HDL searches for settings in the following order:

$cdsroot/share/cdssetup
(System level) → $CDS_SITE/cdssetup
(Site level) → $HOME/cdssetup
(User level)

DE CIS searches for settings in the following order:

$CDS_SITE/cdssetup/OrCAD_Capture/16.6.0
(Site level) → $HOME/cdssetup/OrCAD_Capture/16.6.0
(User level)

4 PCB Editor - Global Content

All definitions in this chapter can be done in local user env file or global site.env. It depends, if you want to set them for all users or if a single user works in a different way. Good examples are Funckeys. They map a keyboard key to a function. You will have global settings, unique for all users and every user will create additional own Funckeys. He will use the summary of all Funckey definitions. If the same key is mapped to different functions in user env and site.env the user definition will win. Remember Read Order in chapter 3.

You can use Setup > User Preferences in PCB Editor to set the definitions you want and test them. They are stored in your %HOME/pcbenv/env file. Later you can move the definitions you want as unique to site.env in Site definition.

The env file contains all path and environment definitions which are different from the default system settings. You can also change the env file by open it with a text editor, it's written in ASCII. Attached to this Application Note is a site definition including some examples.
If you want to, you can also get a Site with an example component library for PCB Editor and DE CIS. Please contact FlowCAD Support for this.

This is an overview of the most used path and environment settings.
We used the Setup > User Preferences structure for listing them.

Two important path settings are padpath and psmpath. If you work with your own component library you need to set them first.

4.1 File_management > Output_dir

Ads_sdart	The subdirectory to which artwork files should be written
Ads_sdplot	The subdirectory to which plot files should be written
Ads_sdreport	The subdirectory to which report files should be written

These sub directories are automatically created in the project folder where the BRD file is stored.

4.2 File_management > Versioning

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.

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.

4.3 Manufacture > NC_legend

nclegend_file	Override the default filename convention for NC Drill Legend. By default, NC Drill Legend uses a name of default-<units> where units are the current board units. The override name should contain only the filename not a PATH component.

4.4 Paths > Config

Clippath	Search path for sub-drawing files (.clp).
Dfaauditpath	Search path for DFA Audit (.arl .rle).
Dfacnspath	Search path for dfa constraints spreadsheet files (dfa).
Ipc2581attrpath	Search path for IPC2581 property configuration file(.atr).
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 files (.txt).
Scriptpath	Search path for scripts.
Textpath	Search path for extracta command files (.txt).
Viewpath	Search path for visibility schema files (.color).
wizard_template_path	Search path for Allegro templates (.brd .dra).

4.5 Paths > Editor

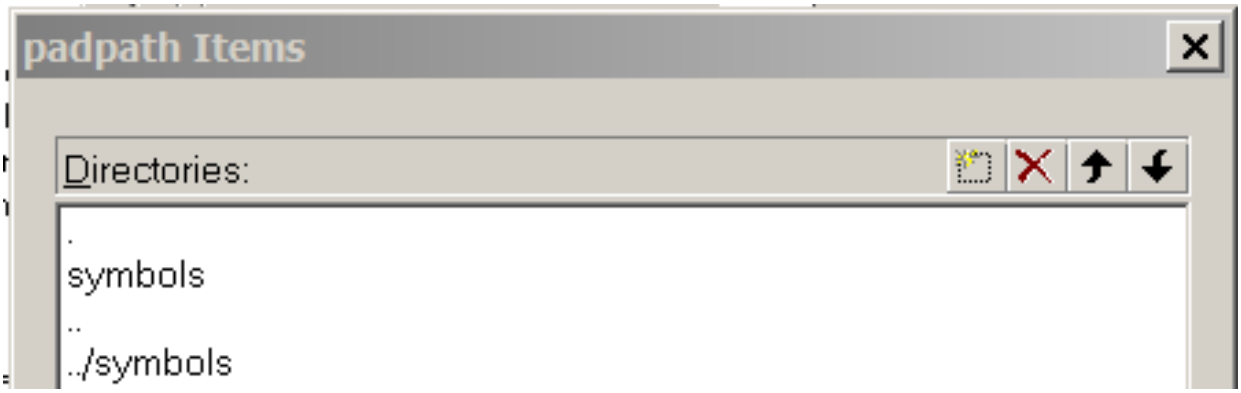
Menupath	Search paths for menu files.

4.6 Paths > Library

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).
Steppath	Search path for STEP files (.stp .step)
Techpath	Search path for technology files (.tech).

4.7 Path Syntax

When you first open padpath after installation and enable Expand box you will always find these path settings at the beginning.



.	Dot is for your local folder (where .brd file was opened). We recommend adding the dot at the beginning of every path definition. PCB Editor “sees” all local files first, which make it easier to test new configuration files or all other new files.
symbols	Looks for the sub folder symbols in local folder and use it for search.
..	Looks one level above local folder
../symbols	Looks for the sub folder symbols one level above (parallel folder).

Hierarchy is from top to bottom here. All folders are read in the order they appear.

4.8 Funckey

In the PCB Editor training you will learn how to define temporary Funckeys. To get a permanent definition you have to define the funckeys in the env file. Here some examples for Funckey definitions:

These funckeys switch layers on and off. It's similar to key functions of OrCAD Layout. They use additional skill routines to run.

```
funckey 0 'settoggle lay all_layer all_layer_nil; $lay'
funckey 1 'settoggle lay lay_top lay_top_nil; $lay'
funckey 2 'settoggle lay lay_bot lay_bot_nil; $lay'
funckey 3 'settoggle lay lay_2 lay_2_nil; $lay'
funckey 4 'settoggle lay lay_3 lay_3_nil; $lay'
funckey 5 'settoggle lay lay_4 lay_4_nil; $lay'
funckey 6 'settoggle lay lay_5 lay_5_nil; $lay'
funckey 7 'settoggle lay lay_6 lay_6_nil; $lay'
funckey 8 'settoggle lay lay_7 lay_7_nil; $lay'
funckey 9 'settoggle lay lay_8 lay_8_nil; $lay'
funckey ~0 'settoggle lay lay_9 lay_9_nil; $lay'
funckey ~1 'settoggle lay lay_10 lay_10_nil; $lay'
funckey ~2 'settoggle lay lay_11 lay_11_nil; $lay'
funckey ~3 'settoggle lay lay_12 lay_12_nil; $lay'
funckey ~4 'settoggle lay lay_13 lay_13_nil; $lay'
funckey ~5 'settoggle lay lay_14 lay_14_nil; $lay'
funckey ~6 'settoggle lay lay_15 lay_15_nil; $lay'
funckey ~7 'settoggle lay solder_top solder_top_nil; $lay'
funckey ~8 'settoggle lay solder_bot solder_bot_nil; $lay'
funckey ~9 'settoggle lay paste_top paste_top_nil; $lay'
funckey ~A0 'settoggle lay paste_bot paste_bot_nil; $lay'
funckey ~A1 'settoggle lay silk_top silk_top_nil; $lay'
funckey ~A2 'settoggle lay silk_bot silk_bot_nil; $lay'
funckey ~A3 'settoggle lay assembly_top assembly_top_nil; $lay'
funckey ~A4 'settoggle lay assembly_bot assembly_bot_nil; $lay'
funckey ~A5 'settoggle lay drill drill_nil; $lay'
funckey ~A6 'settoggle lay test_top test_top_nil; $lay'
funckey ~A7 'settoggle lay test_bot test_bot_nil; $lay'
funckey ~A8 'settoggle lay dimension dimension_nil; $lay'
funckey ~A9 'settoggle lay drawing drawing_nil; $lay'
```

Funckeys for the standard functions of right mouse button

```
funckey Esc cancel
funckey u undo
funckey n next
funckey o oops
```

Funckeys for often used placement and routing functions

```
funckey r "iangle 90"
funckey m "pop mirror"
funckey f "pop finish"
funckey w 'settoggle width 0.1 0.2 0.3 0.5 1 2; echo "Using width“ $width; options
acon_line_width $width'
funckey v 'settoggle grid_val 0.001 0.1 0.2 0.5 1 2; echo "Using grid" $grid_val; replay
$CDS_SITE\pcb\script\set_grid.scr'
```

Funckeys for managing the view

```
funckey a 'settoggle rats "rats all" "unrats all"; $rats; echo "Toggle Airlines"'
funckey q '"show element";echo "Query Element"'
funckey g 'grid toggle; redraw; echo "Toggle Grid"'
funckey d 'color -globvis on'
funckey c 'color -globvis off'
funckey z "zoom center"
funckey b "settoggle togfilm 'Film: TOP' 'Film: GND1' 'Film: I1' 'Film: POW1' 'Film: GND2'
'Film: I2' 'Film: POW2' 'Film: BOTTOM'; FORM vf_vis colorview_list $togfilm"

funckey h hilight
funckey j dehilight
funckey s 'shadow toggle'
funckey . drc update
funckey , "pick_to_grid -cursor;pick_to_grid -cursor"
```

Funckeys which are executed by mouse

```
button Swheel_up plus_global_transp
button Swheel_down minus_global_transp
button SCwheel_up plus_shape_transp
button SCwheel_down minus_shape_transp
button Cwheel_up plus_shadow
button Cwheel_down minus_shadow
```

Alias to move one grid unit with arrow keys, supported only in Placement Mode

```
alias SUp    "move; ipick_to_gridunit 0 +1"
alias SDown  "move; ipick_to_gridunit 0 -1"
alias SLeft  "move; ipick_to_gridunit -1"
alias SRight "move; ipick_to_gridunit +1"
```

Every user can have its own set of Funckeys. Normally the System Administrator defines a set of Funckeys for all users to get a unique handling of the tool.

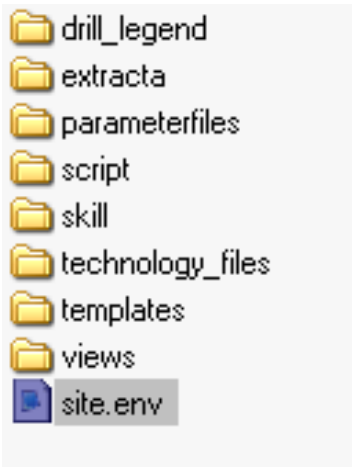
We recommend not to use i, x and y for funckeys. These characters are used to put in coordinates and can confuse users.

5 PCB Editor – Structures

In this chapter we describe the structure of the attached site example and additional files.

5.1 Site structure

The structure of attached example is …/site_166/pcb/:



You can modify the example, add or remove folders or create your own structure.

We will use the views folder as an example, for showing how all this works.

In folder views you will find several views. A view contains the information which layers are switched off or on in PCB canvas. You can use them to quickly change visibilities.

Normally HOME definitions overwrite SITE definitions. To use a combination of path definitions from HOME and SITE you need to add something to path im SITE.

Example:

site.env located in $CDS_SITE/pcb folder contains this row:

- set viewpath = . $CDS_SITE/pcb/views

HOME env contains this row:

- set viewpath = $viewpath D:\Data\Home\166\pcbenv\views

$viewpath in Home env is to command to hold viewpath information from SITE.

With this syntax PCB Editor reads two folders, $CDS_SITE/pcb/views and D:\Data\Home\166\pcbenv\views to get the predefined views from Site and Home.

Without $viewpath, Home viewpath overwrites SITE viewpath and PCB Editor will only look into Folder D:\Data\Home\166\pcbenv\views.

5.2 PCB Editor - Skill and Allegro.ilinit

Skill is the PCB Editor internal programming language. You can define custom specific menus and functions using Skill. The attached site includes 3 free skill routines and the matching allegro.ilinit file.
At startup, Allegro PCB Editor looks for the allegro.ilinit file. This file contains the path of SKILL files and loads the SKILL commands for use. The allegro.ilinit file is located in $HOME/pcbenv folder and / or $CDS_SITE/pcb.

Example for an allegro.ilinit file:

```
load("./skill_data/cds2f.il")
load("./skill_data/layout.il")
load("./skill_data/display.il")
```

Skill can also be used in combination with funckeys. Here are some examples of funckeys which start a skill routine to switch layers on or off.

```
funckey 0 'settoggle lay all_layer all_layer_nil; $lay'
funckey 1 'settoggle lay lay_top lay_top_nil; $lay'
```

5.3 PCB Editor - PCBENV

PCBENV is a folder in your HOME definition and contains the user env file and additional user files. These are the possible files:

Env	The User env file
allegro.ini	This file keeps track of the path where your 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 recreated automatically the next time you start PCB Editor.
allegro.mru	This file records a list of the most recently used board files. DO NOT EDIT THIS FILE!
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! This file can be deleted as a form of troubleshooting. It will be recreated automatically the next time you start PCB Editor.
my_favorites	Favorites from Setup > User Preferences
myfavorites.txt	This file contains which class/subclass(es) to be displayed in the MyFavorites folder of the Color Dialog form.
pad_designer.geo	This records the Pad Designer form location and size.
pad_designer.mru	This records a list of most recently used padstacks and paths.

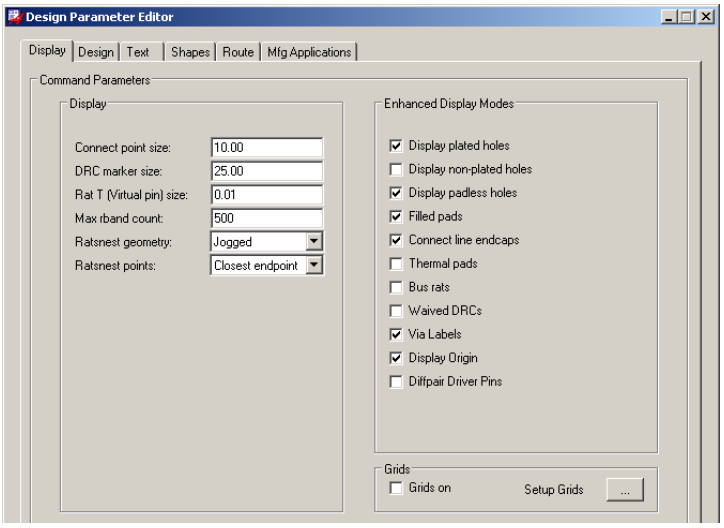
6 PCB Editor - Rollout

During rollout of PCB Editor it's important to define structures and information which helps all users to start quickly an efficient with PCB Editor. Many of the following configuration files are explained in PCB Editor Training. Because PCB Editor is a complex system we always recommend booking Cadence trainings.

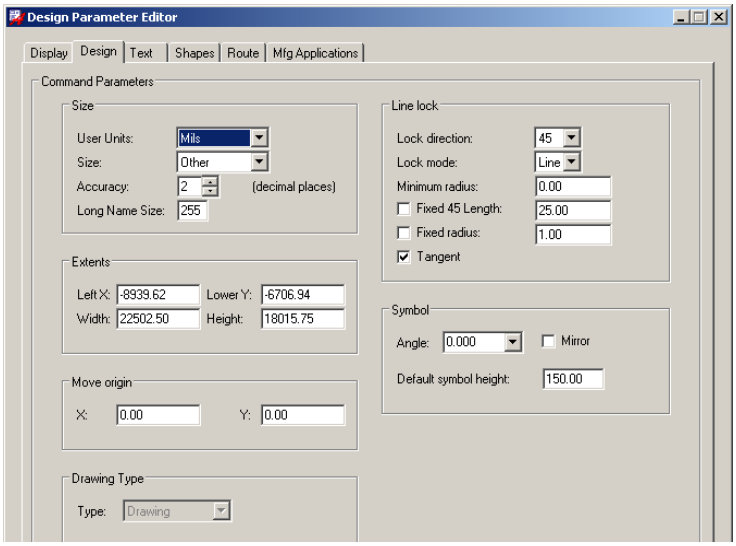
6.1 Board Template

A Board Template is a board file which has no netlist information. It includes all settings which are uniform for all boards or a type of boards. These settings could be:

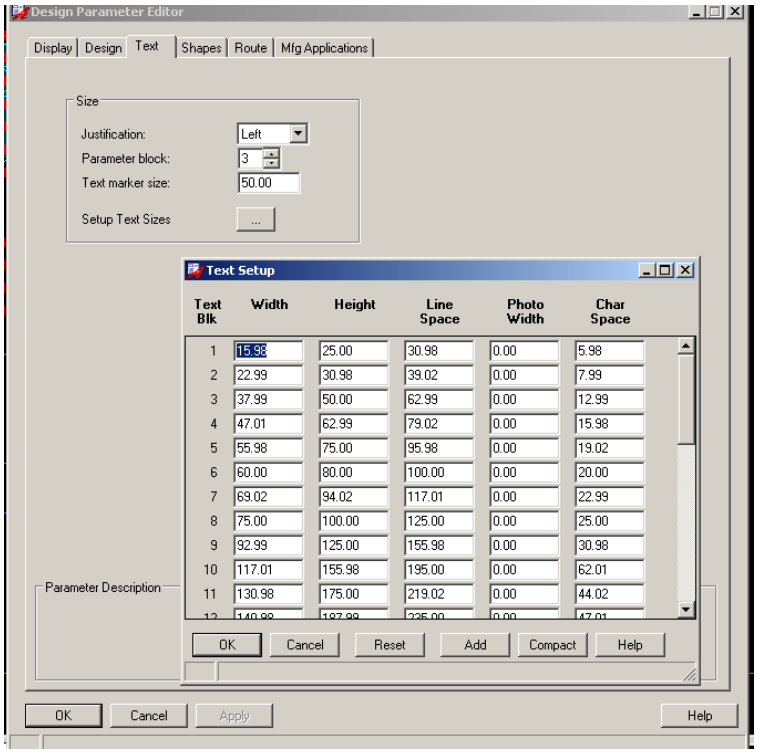
Setup > Design Parameter > Display



Setup > Design Parameter > Design



Setup > Design Parameter > Text



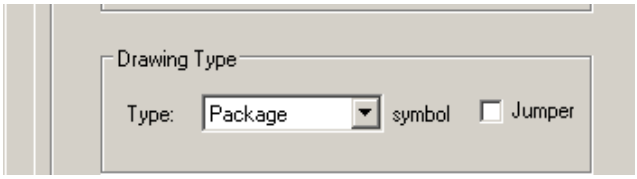
A template can also include dimensions, preplaced mechanical and format symbols you need for documentation. If you have components which are always on the same position like connectors you can place them as Package Symbol with dummy reference designator.

6.2 Symbol Template

A Symbol template is a symbol file (*.dra) which has no placed pins or any graphic information. It includes all settings which are uniform for all symbols. These settings could be Setup > Design Parameter > Display, Design and Text like in board file template.

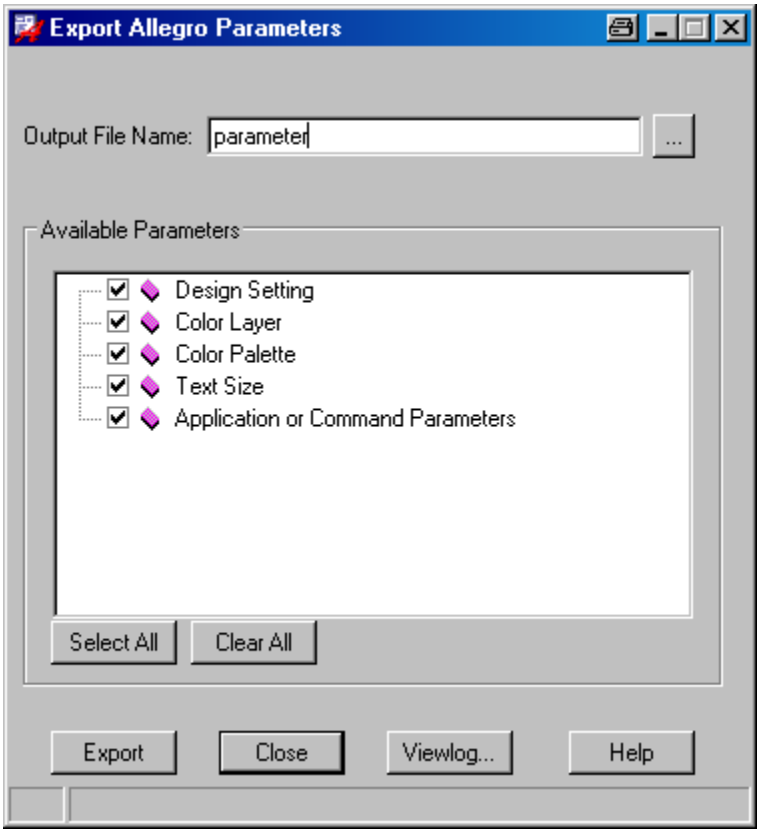
A symbol template can also include preplaced labels.

You can switch between the different symbol types by using Setup > Design Parameter > Design > Drawing Type. This allows you to change a mechanical symbol to a package symbol or a format symbol to mechanical symbols and so on.



6.3 Parameter Files

Parameter files contain all color settings and settings from Setup > Parameter menu.



6.4 Technology Files

Technology Files contain spacing, physical and electrical rule sets and layer stackup.

6.5 Scripts

Scripts contain repeating actions. You can record them with absolute coordinates or relative coordinates (macro mode)

6.6 Drill Legends

A Ncdrill file defines context and view of drill legends.

6.7 Material

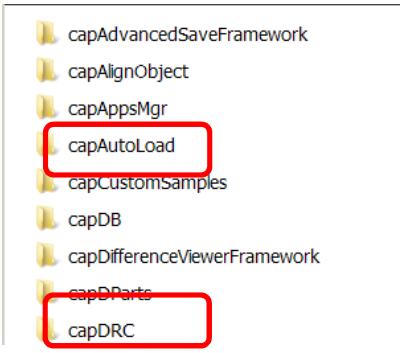
In Cross Section you can set all available materials from material file on conductor and dielectric layers to define the real stackup und use it to calculate impedances. Add your materials to material.dat file and store it in Site or Home.

6.8 DFA

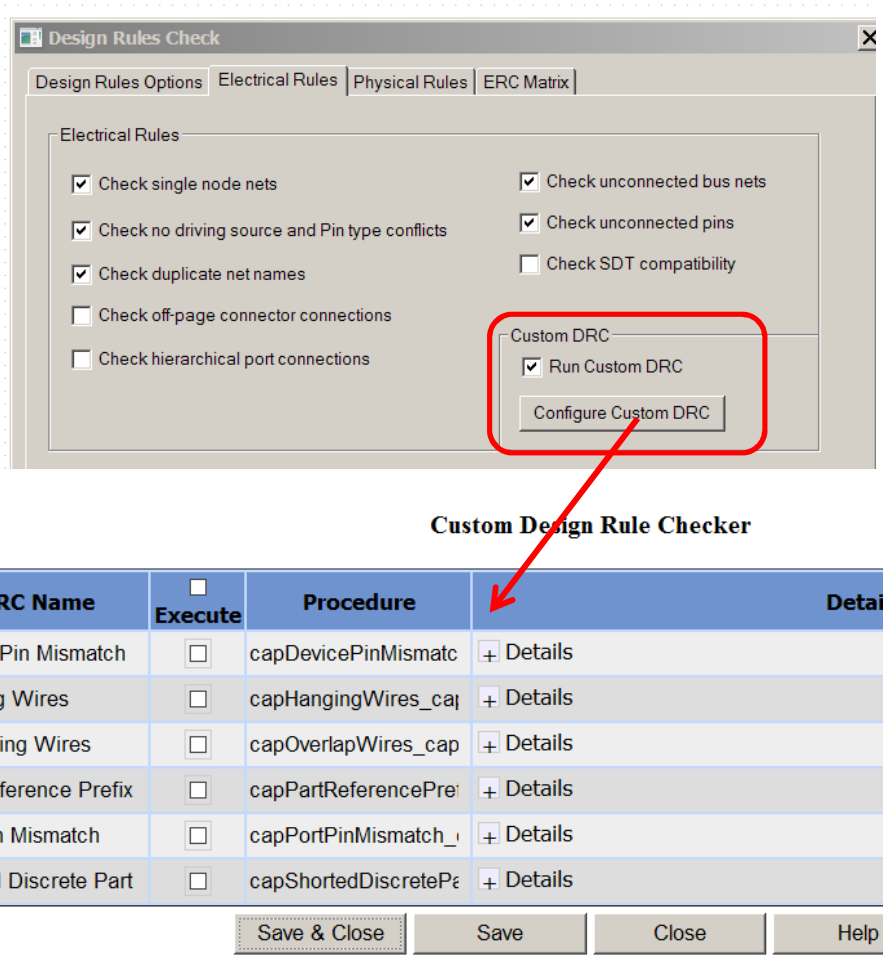
DFA stands for “**D**esign **F**or **A**ssembly”. We use a spreadsheet to manage the different minimum spaces between components to design boards with correct placement for assembly.

7 DE CIS - TCL

DE CIS uses TCL to define interfaces or special functions. Cadence provides a lot of examples and documentation. It's located in $CDSROOT\tools\capture\tclscripts. All files in $CDSROOT\tools\capture\tclscripts\capAutoLoad are loaded during DE CIS Start.



There are other TCL examples which are loaded by Capture functions. E.g. capDRC contains examples for user defined DRC which can be activated by DRC menu.



When you create your own TCL scripts you should store them in Site or Home environment. The paths are $HOME\cdssetup\OrCAD_Capture\16.6.0\tclscripts and $CDS_SITE\cdssetup\OrCAD_Capture\16.6.0\tclscripts. Use the same subfolder names like in $CDSROOT\tools\capture\tclscripts.

7.1 Capture.ini

DE CIS has no global environment available in installation folder like PCB Editor.
When you start DE CIS first, it creates the configuration file capture.ini.
Capture.ini contains all setting for DE CIS from DE CIS menus:

- Option > Preferences
- Option > Template
- Option > Autobackup
- Tools > DRC > ERC Matrix
- Bom Templates

Capture also reports window settings, recent file list and many other things in capture.ini.
For 16.6 it's located in %HOME\cdssetup\OrCAD_Capture\16.6.0.
In all earlier releases it's located in $CDSROOT/tools/capture.

Like PCB Editor Capture CIS needs global settings like library paths and user settings like colors. In default installation all settings are in one user capture.ini. This makes it difficult for CAD administrators to change e.g. library path.
Better split this information in a global capture.ini file located at $CDS_SITE\cdssetup\OrCAD_Capture\16.6.0 an a local user capture.ini file located at $HOME\cdssetup\OrCAD_Capture\16.6.0

First create one capture.ini file in HOME environment with all path settings and test how it works. You need to add these paths for CIS manually.

[Part Management]
Configuration File=Path to DBC file

[Footprint Viewer Type]
Type=Allegro

[Allegro Footprints]
Dir0=Path to footprint folder 1
Dir1=Path to footprint folder 2

[Part Library Directories]
Dir0=Path to olb folder 1
Dir1=Path to olb folder 2

[CIS Browse Directories]
Dir0=Path to document folder 1
Dir1=Path to document folder 2

Check all Capture and CIS functions including templates, preferences settings, ERC Matrix and all other global things you want to define for all users. Open capture.ini and delete all unimportant entries.

Move this new created INI file to your Site environment under $CDS_SITE\cdssetup\OrCAD_Capture\16.6.0.

7.2 Managing Master and local user INI

There are two ways to manage master.ini and local capture.ini. You can use Capture INI Manager or CIS Admin Tool.

7.3 Capture INI Manager

First you need CaptureINIManager21.rar file. It's attached to this AN. Using OrCAD Marketplace in Capture Startpage you can get a free download of latest version.

Extract the files and move capIniMgr folder to
$CDS_SITE\cdssetup\OrCAD_Capture\16.6.0\tclscripts.

Create a new text file called capIniMgr.tcl in folder
$CDS_SITE\cdssetup\OrCAD_Capture\16.6.0\tclscripts\capAutoLoad.
Open this file with text editor and add these two rows.

```
package require capIniMgr
capIniLoad {D:\Home\site_166\cdssetup\OrCAD_Capture\16.6.0\master.ini}
```

In second row you need to use a matching path. It must point to your site environment. Save the file and start Capture. This TCL script copies all values from master.ini to local capture.ini during start of Capture and replaces existing entries. This method makes sure that all global settings are always correct in local user INI.
So changes in master.ini are automatically distributed to all users.

Attached you can find Capture INI Manager Application File inimngr.pdf to get more information how to use it.

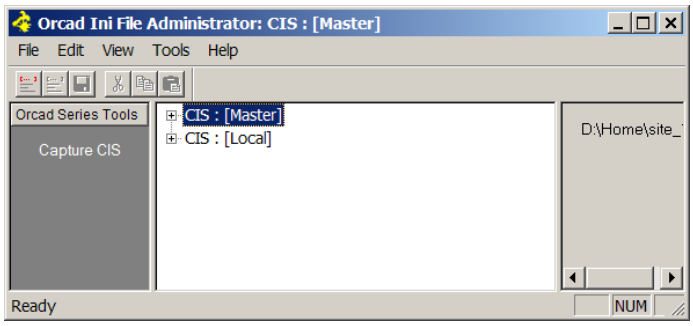
7.4 CIS Admin Tool

CIS Admin Tool helps to create master capture.ini and user capture.ini with a GUI.

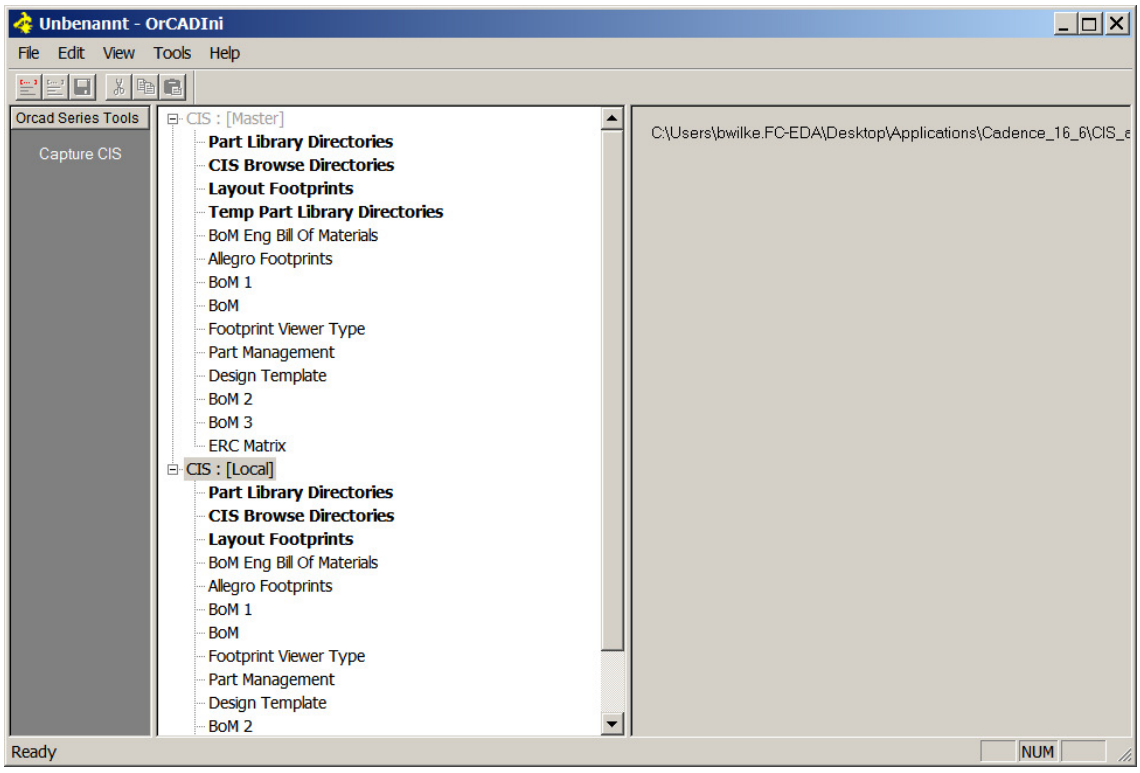
You can download the tool at:

http://www.cadence.com/products/orcad/pages/downloads.aspx#cis16

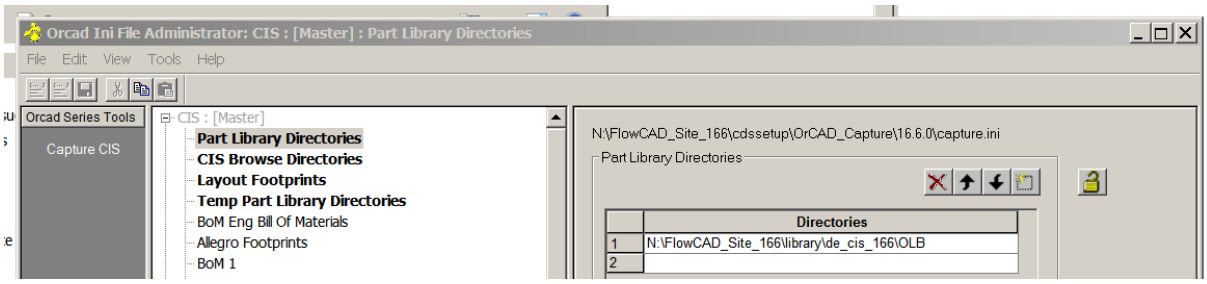
Unzip the file and start OrCADIni.exe.



Use File > Select Master ini to open capture.ini from Site and File > Select Local ini to open capture.ini from Home.



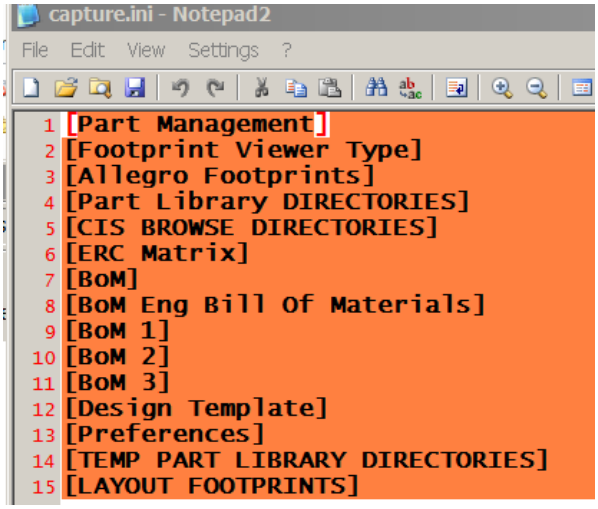
Select Master INI Sections and check entry on right side. Use lock icon to lock the entries you want to protect. Ignore Section Layout Footprint and Temp Part Library Directories.



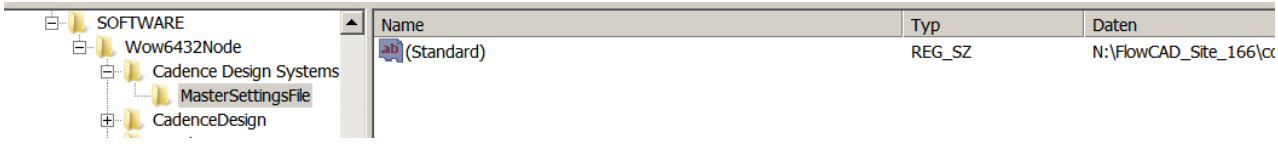
File > Save.
Open capture.ini from Site with text editor. You will find a list of Locked Entries. Close without save.

```
[LOCKED ENTRIES]
Allegro Footprints$$Dir6=1
Allegro Footprints$$Dir7=1
Allegro Footprints$$Dir8=1
BoM 1$$Max Rows To Output=1
Allegro Footprints$$Dir10=1
Allegro Footprints$$Dir9=1
Allegro Footprints$$Dir11=1
Allegro Footprints$$Dir12=1
Footprint Viewer Type$$Type=1
PART LIBRARY DIRECTORIES=1
Allegro Footprints$$Dir13=1
BoM 1$$Property Count=1
BoM 1$$Exclude Part References=1
BoM 1$$Horizontal Bom Output=1
BoM 1$$Include Header=1
BoM 1$$Property0=1
BoM 1$$Output Relational Data=1
BoM 1$$Property1=1
BoM 1$$Reference List Separator=1
CIS BROWSE DIRECTORIES=1
BoM 1$$Property2=1
```

Open capture.ini from Home with text editor. You see all command lines but no entries. All entries are stored in Site INI.



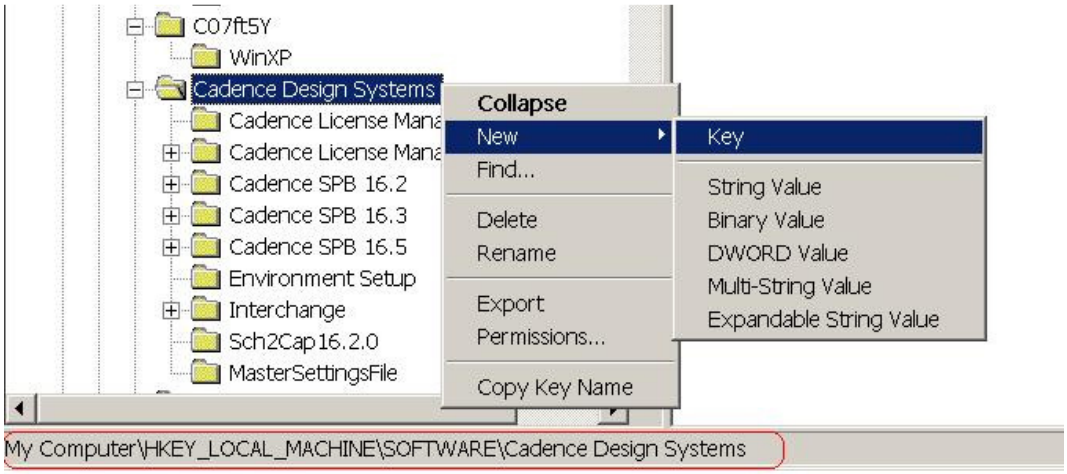
Open regedit and search for capture.ini.



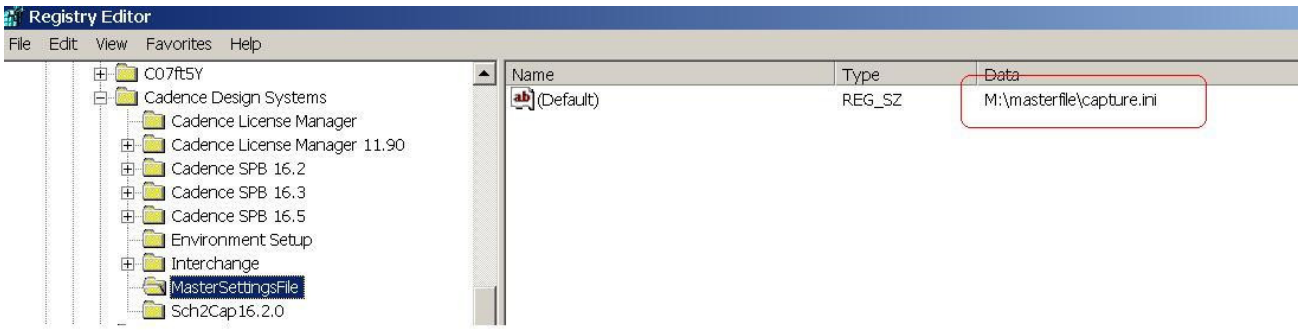
You will find several entries in user setting. Every client system needs a registry entry to find master INI.

On 32 Bit machines

- Go to HKEY_LOCAL_MACHINE\SOFTWARE\Cadence Design Systems.
- Right click on Cadence Design Systems > New > key



- Name this key "MasterSettingsFile"
- Assign the value of this key as <path to master capture.ini>\capture.ini.



On 64 bit machines

- Go to HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\Cadence Design Systems
- Create the key "MasterSettingsFile" and assign the same value as above under Cadence Design Systems.

7.5 3D Footprint Viewer

With current version (16.6 s019) 3D Footprint Viewer can't read footprint information from master INI. Users who want to use this viewer have to copy [Allegro Footprints] entries from master ini to user ini. In this case the best way is to use Capture INI Manager.

8 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 is automatically set. If you only work with one software version you don't need to take care of it. But if you work with 2 or more software versions it's important to know which value is just set. You can modify CDSROOT manually, with Cadence Switch Release or use Batch Files.

8.1 Cadence Switch Release

Cadence Switch Release sets CDSROOT and all necessary Path settings.
You start this by selecting Start -> All Programs -> Cadence SPB Switch Release.
Always use Switch Release of the highest software version.

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

8.2 Batch Files

Batch Files are short script file which sets variables and start programs. The Batch File creates a local environment to start the tool. For example you work with 15.7 and have a test installation for 16.6 it would be the best to set all variables to 15.7 and use Batch Files to start 16.6. In this environment you have no risk to open and save a 15.7 board file with 16.6 because you use totally different settings.

Example:

```
echo off
set CDSROOT=D:\SPB_16.5
set HOME=E:\HOME\165

Path=
C:\WINDOWS\system32;
C:\WINDOWS;
C:\WINDOWS\System32\Wbem;
%CDSROOT%\tools\PSpice\Library;
%CDSROOT%\tools\Capture;
%CDSROOT%\tools\bin;
%CDSROOT%\tools\pcb\bin;
%CDSROOT%\tools\specctra\bin;
%CDSROOT%\tools\libutil\bin;
%CDSROOT%\tools\fet\bin;
%CDSROOT%\tools\dfII\bin;
%CDSROOT%\tools\PSpice

call allegro
```

Note:
The path setting must be one row. It's divided here into different rows for a better overview. Copy your Path value from system setting and create your own path for batch files.
With a Batch File you can set different HOME variables for different tools or versions. This helps to manage different component libraries.