Why is a site environment interesting and how will it help me?

It is all about making a structured and formalized configuration. Having a company configuration on a network drive will improve the team work and ease installation of new clients or setting up a new client with Cadence Allegro or OrCAD.

With a site environment there is 1 shared place for storing everything like footprints, padstacks, schematic symbols, output preferences, shortcut definitions etc. A change to a site environment will be reflected at the client the next time the tool is launched.

With a user setup also being configured the individual user will be able to setup their own preferences as long as those are not setup as read only from the company configuration.

The folder structure and configuration is described in more detail further down in this guide.

Content

Decide a network location for all company configurations

1. In the ***network location create*** a ***folder*** named ***"company_setup"***
   a. Example: \\RDDRIVE\Cadence\company_setup
2. ***Copy the structure*** from the company_setup folder in site_setup.zip into \\RDDRIVE\Cadence\company_setup

Decide a location for all user configurations

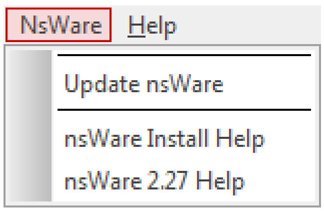
1. On the client create a folder for any individual users setting
   a. Example: C:\Cadence\user_setup
2. Copy the structure from the user_setup folder in the site_setup.zip file into C:\Cadence\user_setup

Setup clients to point to company and user configuration

1. Go to ***Control Panel and select System***
2. Select ***Advanced system settings*** in the right side
3. Select ***Environment Variables***
4. Create 2 new user environment variables
   a. CDS_SITE=\\RDDRIVE\Cadence\company_setup (don't specify pcb folder, it is expected in given path)
   b. ALLEGRO_PCBENV=C:\Cadence\user_setup

Check correct nsWare installation

Start OrCAD or Allegro PCB Editor and check if a nsWare menu exist, it will have a default of at least 2 menu entries.



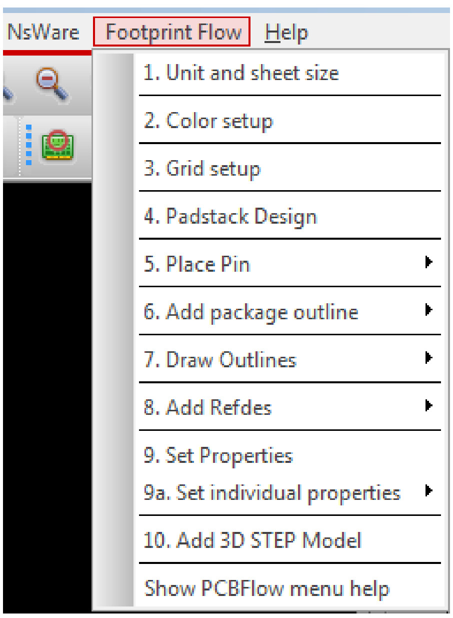
Notice that the position of the nsWare menu is not necessarily next to the Help menu.

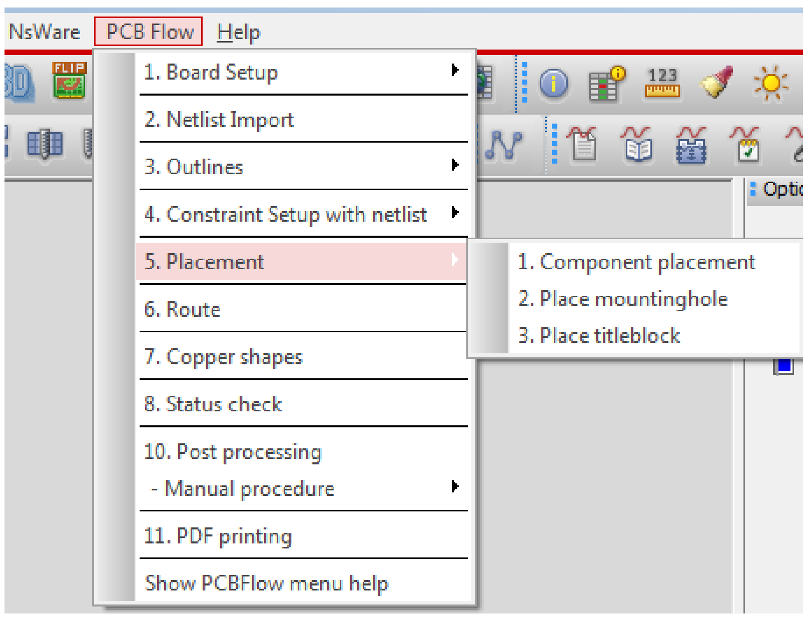
Both help topics are updated whenever necessary and hence refer to a website.

Latest NsWare menu should show ***nsWare 2.63 or newer Help***

PCB flow and Footprint flow

2 menus are added to help guide the casual user through the steps involved in creating PCB designs and footprints respectively.





The menus can be disabled setting the environment variable "ns_no_pcbflowmenu" in either of the files "env" (allegro_pcbenv directory) or "site.env" (cds_site directory). This can be done by adding "set ns_no_pcbflowmenu" to one of the files.

Shared configuration for Allegro Design Entry CIS (ADE CIS) / OrCAD Capture CIS

The configuration allows controlling preferences that are stored in Capture.ini with respect to setting up titleblock information, libraries, CIS configuration etc.

After copying data as described above there will be a folder \\RDDRIVE\Cadence\Company_setup\OrCAD_Capture that contain 2 important files

First_Capture.ini can be added information that are added to the local Capture.ini the first time ADE CIS/Capture CIS is started at the client. First_Capture.ini settings are typically used for preferences that are company recommended but still leaves the user the flexibility to change locally like background color on the schematic editor canvas.

Master_Capture.ini is used for any preferences that should be used all the time. Everytime ADE CIS/Capture CIS is started the settings from Master_Capture.inin are added to the local Capture.ini file. Master_Capture.ini are typically used for preferences like

- CIS configuration file path (.dbc)
- Datasheet path
- Schematic part/symbol library path
- Footprint paths
- Titleblock preferences
- Design Rule Check (DRC) preferences
- BOM setup
- And much more

Both First_Capture.ini and Master_Capture.ini are seeded with a few suggested settings.

Notice: Any paths need to be replaced with the paths selected above for %cds_site%

Shared configuration for Allegro/OrCAD PCB Editor

The configuration allows controlling settings both at a company and an individual user level. The settings can be shortcuts keys, directories (paths) to padstacks, symbols, views etc. and output preferences.

Settings are read by PCB Editor in a specific order and later settings overrule anything set earlier unless the settings is readonly.

1. Cadence installation settings (default paths, shortcuts etc. installed together with the Allegro/OrCAD software)
    a. It is recommended not to change settings within the installation directory since those settings can be deleted or changed at re-install, uninstall or change of PC.
2. \\rddrive\Cadence\Company_setup\pcb\site.env (%cds_site%\pcb\site.env)
    a. This file is used to control any company settings. So anything in this file will be used whenever PCB Editor is started as long as the environment variable cds_site is defined correctly
    b. Site.env can be edited using a text editor, notepad++ or another reasonably good text editor is recommended
3. C:\Cadence\user_setup\env (%allegro_pcbenv%\env) where env is a file without any file extension.
    a. Env file can also be edited in a text editor. Do not edit anything below the section marked as computer generated.

Company_setup description

The pcb folder within the directory selected for the company configuration (%cds_site%) contain a number of directories that can be used for storing different configurations, footprint etc. A description of many of these is found further down. The folder also contain 2 important files

art_param.txt: This file specifies the default artwork parameters like Gerber 274X, number of decimal and integer places etc.

site.env: This file specifies company wide settings, paths and shortcuts. Many of the paths specified refers to the directory structure within the pcb folder

For example “set PADPATH = . symbols padstacks $CDS_SITE\pcb\padstacks” which means that the folder ***padstacks*** within the directory %cds_site%\pcb is referred as a padpath for PCB Editor to search for padstacks

Also shortcuts are listed in site.env and can be overruled by user shortcuts as described in the section about user_setup

A number of user preferences found in Setup → User Preferences are set in site.env.

Site.env must be edited in a text editor like Notepad++. It is possible to setup preferences etc. using Setup → User preferences by copy and paste from the env file described in user_setup further down this document.

Notice: Please check that all paths, user preferences and shortcut keys (funckey and alias) are as wanted within your company.

Pcb folder directory structure

To migrate your existing setup to this type of setup you will need to move your existing files into the folder structure below. The most important directories are explained below

Dfa is for files with design for assembly constraints (.dfa)

Extracta is for user defined reports (.txt)

Menus contain example customized menus for PCB Editor (.men)

Modules is for design reuse modules and placement replicates

Nsware contain setup files for some nsware utilities

Padstacks is where existing and new padstacks should be placed (.pad)

Parameter is for parameter files with colors, grids etc. (.prm)

Scripts is where existing and new scripts should be placed (.scr)

Skill is where existing and new skill files should be placed (.il) – do not touch or place anything within the nsWare subdirectory

Step is where existing and new step files should be placed (.step/.stp)

Symbols contain a number of subdirectories where existing symbol files should be placed (.dra, .psm, .ssm, .bsm, .osm, .fsm)

Tech is where technology files should be placed (.tcfx, .tcf)

Template is where board, symbol and padstack templates should be placed (.pad, .brd, . dra)

Views is where existing and new view files should be placed (.color)

User_setup description

The file named “env” within %allegro_pcbenv% is used to store any settings made at the user level. Whenever anything is changed from the user preference editor (Setup → User Preferences) the settings are stored in the file “env”. These settings are saved in the “Computer generated” section and should not be edited manually.

Shortcut definitions and any other manual preferences can be added above the “computer generated” section.

Shortcut example

funckey i zoom in

funckey (all lowercase) is a keyword followed by the shortcut key and finally the command assigned to that keyboard key. The above command defines lowercase “i” as a shortcut for zoom in within PCB Editor. Likewise other shortcuts can be defined.

Files within the user_setup 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 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 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_17.2-2016.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.

Getting help and more information

Each of the menus contain a Help menu item at the bottom that will go to a webpage with help.

Notice that not all help is completely up-to-date but it is still valid. Many of the utilities within the menus contain their own help.

Shortcuts

Most shortcuts defined are described in Nordcad → Show Nordcad menu help

Also feel free to contact us directly for help and installation etc.

Contact details at www.nordcad.no and www.nordcad.dk