nsWare framework release 2.83

To run nsWare products it is necessary to install the nsWare framework.

Download and unzip

- http://nordcad.dk/download/nsware/nsWare.zip

Unzipping will show a directory structure as shown

allegro.ilinit.txt

nsWare (folder)

nsware.il

Installation

1. Start OrCAD or Allegro PCB Editor and write **"set"** in the command prompt to get a list of environment variables.

Installation can be done to 3 directories

- Company SITE environment (%cds_site%\pcb\skill)
- User configuration environment (%localenv%)
- Cadence installation directory (%cdsroot%\share\local\pcb\skill)
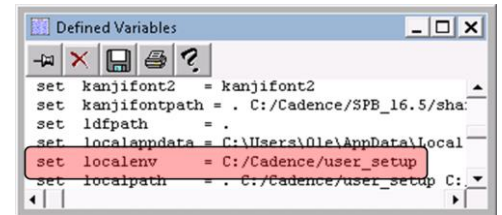
2. Locate the **desired installation path** by localizing the appropriate variable name (embraced in %-signs above - **cds_site, localenv or cdsroot**)

3. Move the **nsWare folder and allegro.ilinit.txt** to the selected directory, for the example above the content will be

    C:\Cadence\user_setup\allegro.ilinit   (see note below if allegro.ilinit already exist in this folder)

    C:\Cadence\user_setup\nsWare\nsware.il

4. **If allegro.ilinit** already exist in the chosen directory then copy the lines from allegro.ilinit.txt into the existing allegro.ilinit file.


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 2.83 or newer Help**
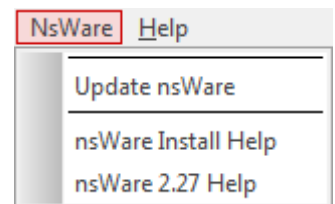
Install nsWare products from **OrCAD MarketPlace**

Any nsWare products will need to be placed inside the nsWare folder. After restarting OrCAD/Allegro PCB Editor the appropriate menu choices will be available inside the nsWare menu.

The example shows having just the PDF generator and PCB Stacking Stripe symbol products

With nsWare 2.43 or newer running Update nsWare will install all free nsWare apps
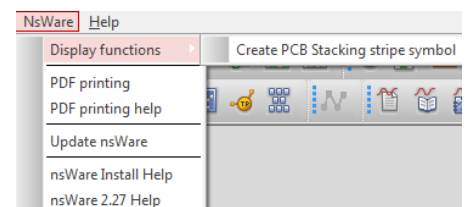
Support for proxy server

If a proxy server is used this will block the use of **Update nsWare**. To work around this issue the proxyserver settings can be specified using an environment variable named "nswareupdate_proxy".

This variable can either be specified at the system level, in the "env" file or "site.env" file.

The value of the variable should be of the format proxy-server:portnumber

Example: set nswareupdate_proxy website-xyz:80

PCB flow and Footprint flow

Version 2.35: 2 menus are added to help guide the casual user through the steps involved in creating PCB designs and footprints respectively.
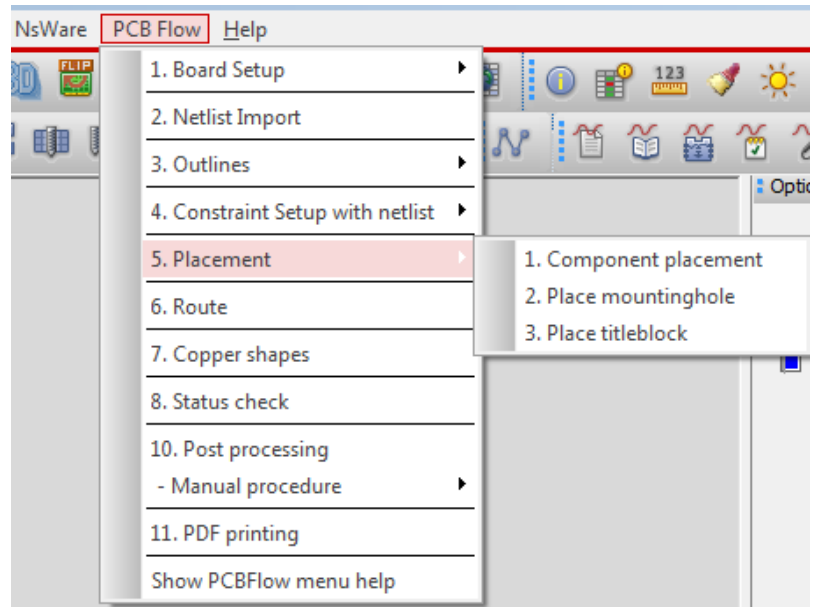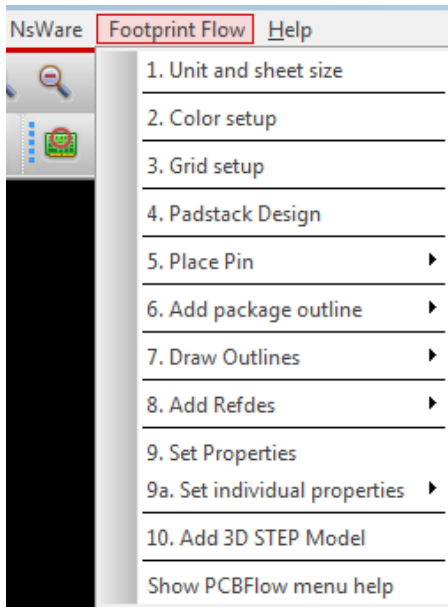
The menus can be disabled setting the environment variable "ns_no_pcbflowmenu" in either of the files "env" (localenv directory) or "site.env" (cds_site directory). This can be done by adding "set ns_no_pcbflowmenu" to one of the files.

FreeWare apps

1. ***NsWare → Change grid***
   a. fast change of grid settings
2. ***NsWare → Change route width***
   a. fast change of cline width
3. NsWare → Output functions → Pick and place data TOP
   a. Pick and place data for top layer mounted components
4. NsWare → Output functions → Pick and place data BOTTOM
   a. Pick and place data for bottom layer mounted components
5. NsWare → Output functions → Layout 2 PCB Editor footprint names
   a. List of footprint names after translation of board files. Can be back-annotated to OrCAD Capture. Also works for other translated board files.
6. NsWare → Output functions → CAMCAD data
7. NsWare → Output functions → CircuitCAM data
8. NsWare → Output functions → Pick and place data to Capture
   a. Exports pick and place data in a format that can be back-annotated to OrCAD Capture
9. NsWare → Shape functions → Force update all shapes, Delete all voids and Disable all shapes
   a. Ability to create all auto-voids and or user defined voids
10. NsWare → Symbol functions
    a. Calculate the total footprint area for placed components. Can be used for board size evaluation.
11. NsWare → Configuration Utility
    a. Can be used to store and recall different settings for paths, funckeys, env variables etc.
12. NsWare → Display functions → Create views from artwork films
    a. Can create view files from the artwork films in the current board file
13. NsWare → CAD Translators → Eagle
    a. Translate Eagle board and library files in XML format to OrCAD/Allegro PCB Editor
    b. Removed with 2.62 due to integration with 17.2-2016 Quartly Intermediate Release (QIR) 1
14. NsWare → PDF Output
    a. Will genereate PDF files for each film in the board. Offers great flexibility with respect to ordering, fonts, adding film names to plots etc.
15. NsWare → Output functions → Run Post Process
    a. Automated post processing – Gerber, NC Drill, IPC356A etc.
16. NsWare → Planar Designer
    a. Design of planar transformerss

Read more about NsWare : http://www.nordcad.dk/filer/nsware/nswaremenu.htm

Revision history

Revision 2.45: Update nsWare now supports the use of a proxy server. Set an environment variable or use set in env or site.env file : Example "set nswareupdate_proxy website-xyz:80"

Revision 2.43: Update nsWare will now install all freeware apps

Revision 2.35: Added "PCB Flow" and "Footprint flow" menu. Can be disabled setting variable "ns_no_pcbflowmenu"

Revision 2.34: Fixed issue inconsistency with NsWare menu in SigXplorer

Revision 2.31: Added Eagle translator to the NsWare→CAD Translators submenu.

Revision 2.27: Added *Update nsWare* menu item. This function will search for and update the installed apps to the latest release.

Revision 2.43: Added freeware apps as part of update nsWare

Revision 2.48: Added force update where it will update all apps to the latest version without looking at timestamps.

Revision 2.50: Added Eagle2Allegro translator as a free app. It will translate to both OrCAd and Allegro format.

Revision 2.62 : Tested on Release 17.2-2016