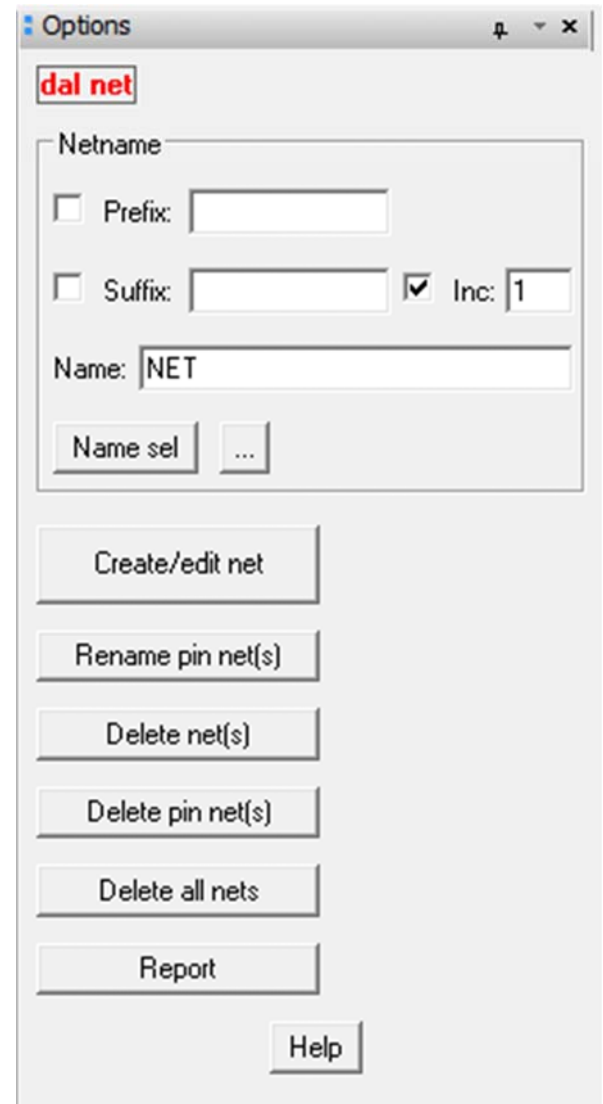
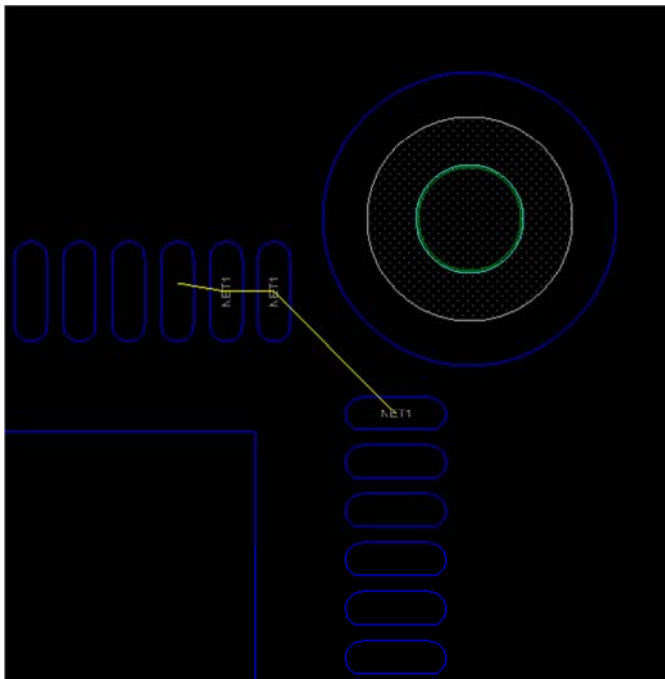


## Net

Allows for on the fly creation, deletion and editing of net connections in the PCB editor. **USE CAUTION WHEN CHANGING SCHEMATIC BASED DESIGNS.**

Features:

- Quickly and easily modify the netlist on the fly.
- Auto increment and prefix/suffix naming.
- Disabled unless the Cadence env variable LOGIC\_EDIT\_ENABLED is set for safety. See page 2.



Video demonstration available at:

[www.dalTools.com](http://www.dalTools.com)

# Configuration

---

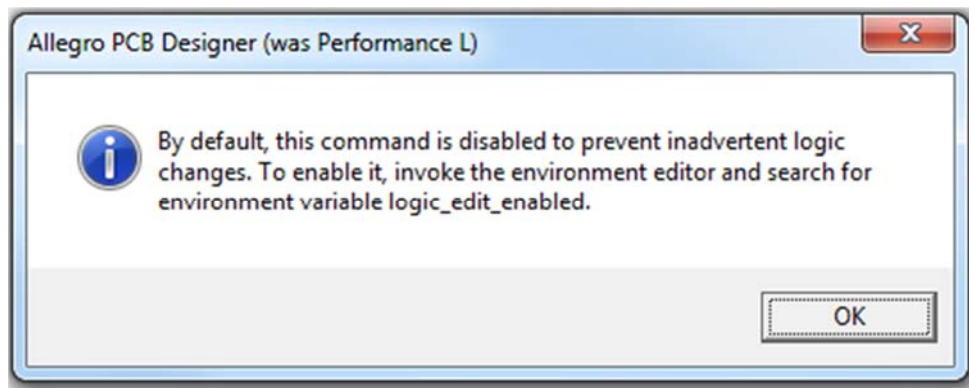
Normally the Cadence supplied environment variable “**LOGIC\_EDIT\_ENABLE**” is unset to prevent netlist changes. In order to change the netlist data in the board this variable must be set.

**To prevent users from being able to modify the net list do the following:**

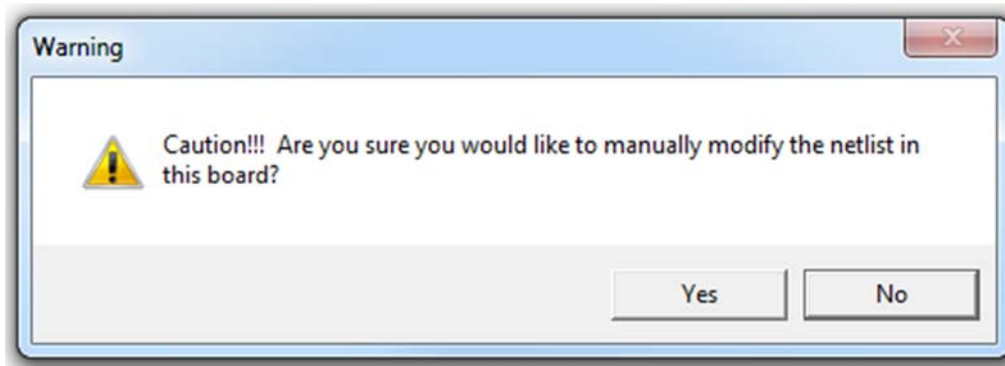
1. In the site env file add this line:  
**UNSET LOGIC\_EDIT\_ENABLED**
2. Add this line under the above line so the users cannot override the setting :  
**readonly LOGIC\_EDIT\_ENABLED**

If a user then tries to change the value this is reported:

E- LOGIC\_EDIT\_ENABLED is set read only, cannot be unset



In any event daTools also warns the user that changing the database netlist can be dangerous in schematic driven designs:



# Help

