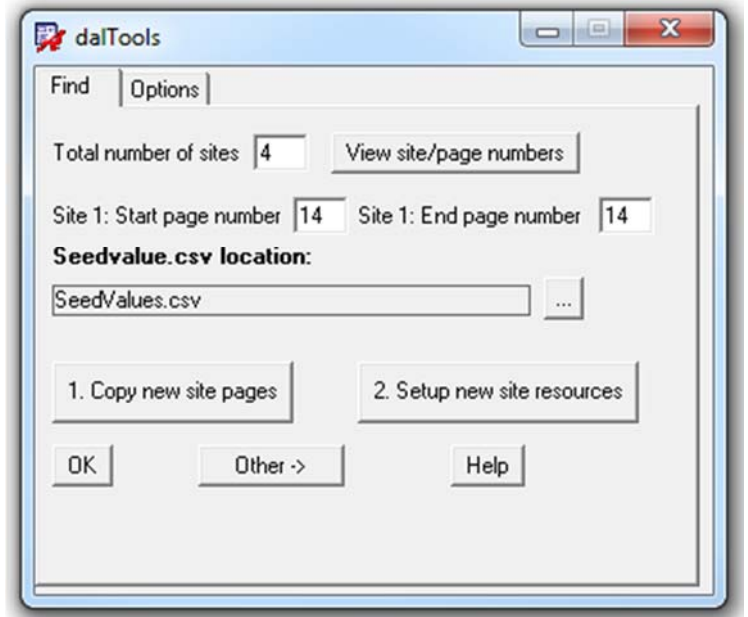


HdlMultiSiteCopy

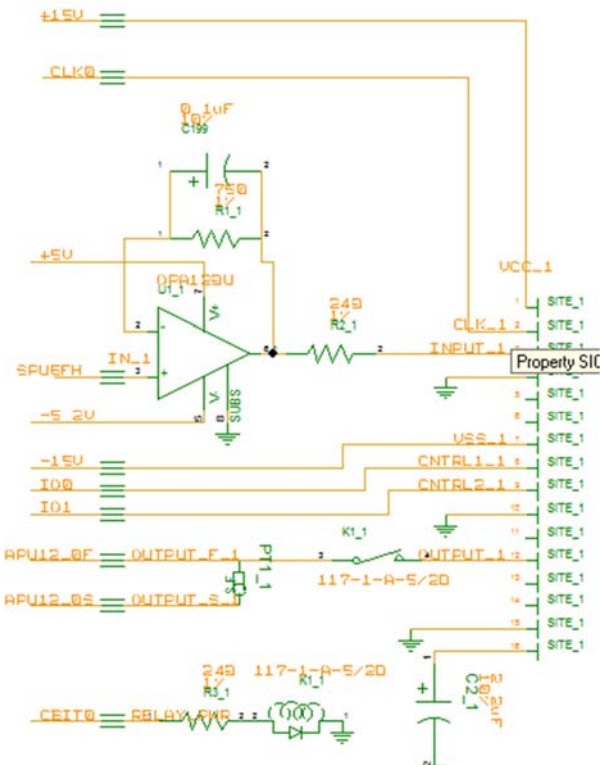
A GUI based tool to facilitate the copying of single site flat schematics into multi-site schematics.

Features:

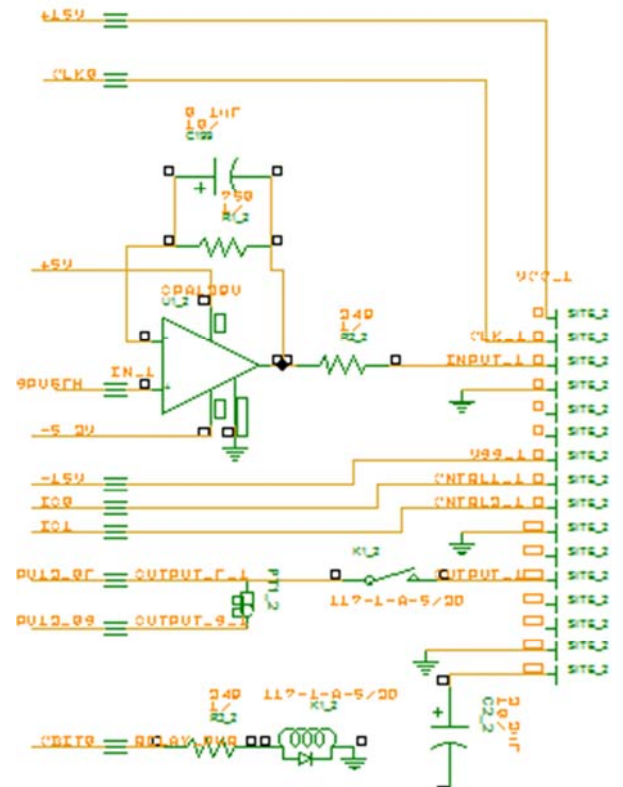
- ❑ Automatically copies the number of sites by start and end page numbers.
- ❑ Quickly and easily change the copied site refdes, netname and note values.
- ❑ Seedvalue and existing netname selection GUI..



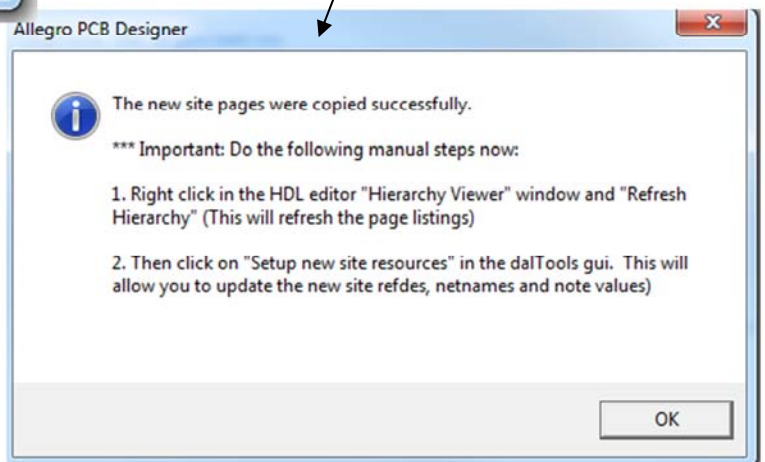
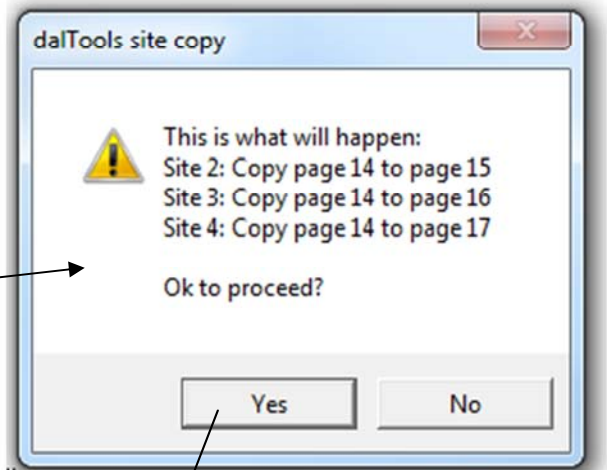
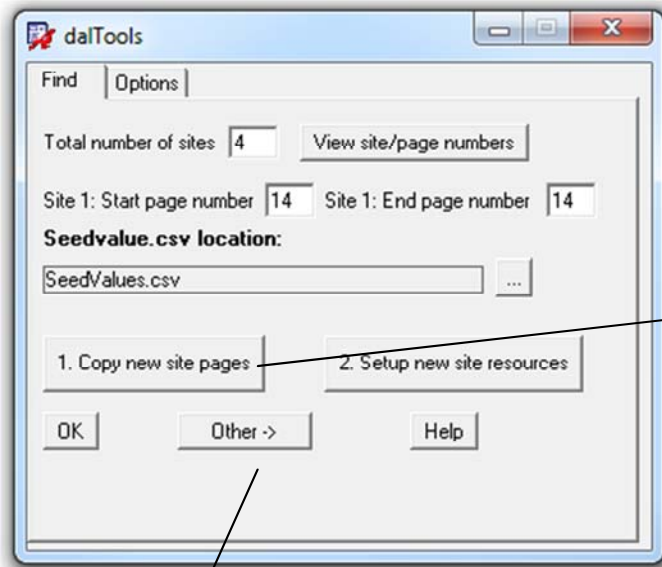
SITE_1



SITE_2



Automatic page copy

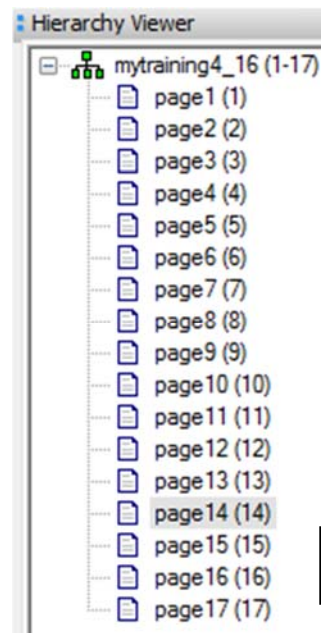
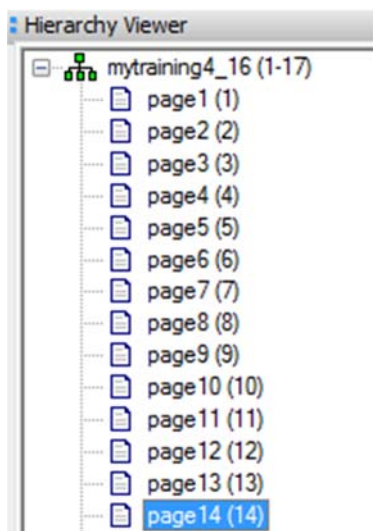


- View/Edit page.map file
- View/Edit module_order.dat file

- View netlist data
- View XML .xcon file
- View DCF .dcf file
- File Manager... (sch_1)

- View seedValue.csv file

- View page.map data from .csa files



3 new pages added

Change Site Resource Values

dalTools - HdI Multi Site Copy

Site: 2 Display: Refdes Nets Notes *Yellow nets are matched in the seedvalues.csv file. *Yellow refdes locations are duplicates.

Select	Page	Copied from page	Site page seq num	Property	Site 1 Value	Site 2 value	Site 2 value [New]
1	15	14	1	LOCATION	R3_1 (4)	R3_1 (4)	R3_2
2	15	14	1	LOCATION	U1_1 (4)	U1_1 (4)	U1_2
3	15	14	1	LOCATION	R1_1 (4)	R1_1 (4)	R1_2
4	15	14	1	LOCATION	C199(4)	C199(4)	
5	15	14	1	LOCATION	K1_1 (8)	K1_1 (8)	K1_2
6	15	14	1	LOCATION	PT1_1 (4)	PT1_1 (4)	PT1_2
7	15	14	1	LOCATION	C2_1 (4)	C2_1 (4)	C2_2
8	15	14	1	LOCATION	SITE_1 (64)	SITE_1 (64)	SITE_2
9	15	14	1	LOCATION	SITE_1 (64)	SITE_1 (64)	SITE_2
10	15	14	1	LOCATION	SITE_1 (64)	SITE_1 (64)	SITE_2
11	15	14	1	LOCATION	SITE_1 (64)	SITE_1 (64)	SITE_2
12	15	14	1	LOCATION	R2_1 (4)	R2_1 (4)	R2_2
13	15	14	1	LOCATION	K1_1 (8)	K1_1 (8)	K1_2
14	15	14	1	LOCATION	SITE_1 (64)	SITE_1 (64)	SITE_2
15	15	14	1	LOCATION	SITE_1 (64)	SITE_1 (64)	SITE_2
16	15	14	1	LOCATION	SITE_1 (64)	SITE_1 (64)	SITE_2
17	15	14	1	LOCATION	SITE_1 (64)	SITE_1 (64)	SITE_2
18	15	14	1	LOCATION	SITE_1 (64)	SITE_1 (64)	SITE_2
19	15	14	1	LOCATION	SITE_1 (64)	SITE_1 (64)	SITE_2

Items: 25

Update schematic ? Close Note: Right click on a row number for more options. Select: All on All off Change values -> Change values Prefix Suffix Change above ?

Allegro PCB Designer

Updates the schematic with the values in the "New value" column.

OK

1. Change C100 -> C200
 2. Change Clk1 -> Clk2
 3. Change Site_1 -> Site_2
 4. Change Site -> Site_1
- Clear all visible new values

Seedvalue net browser ...
All net browser ...

Allegro PCB Designer

"Change above" replaces the value from the "Site 1 Value" into the "Site X value [New]" column by matching path or xy location. When the new value column is correct, click on the "Update schematic" button to change the schematic values.

OK

Seedvalue net browser ...

+12V
+15V
+24V
+5V
-12V
-15V
-24V
5.2V
APU12_0_5_FL
APU12_0_5_SL
APU12_0F
APU12_0S
APU12_10F
APU12_10S
APU12_11F
APU12_11S
APU12_12_17_FL
APU12_12_17_SL
APU12_12F
APU12_12S
APU12_13F
APU12_13S

Total elements: 746

Help

