OrCAD New Enhancements

Addi

13/Aug/2013

Topic

- ## GraserWARE FrontendPack
  - Replace BUS Alias
  - Import/Export Properties
  - Design NC-Part Setting
  - Design Compare
  - Graser Allegro Netlist

Replace BUS Alias

- Rename BUS Alias
  - By Selection
  - By Page
  - By Design

Replace BUS Alias

- Rename BUS Alias
  - By Selection
  - By Page
  - By Design

Import/Export Part Properties

- Is the property editor in Schematic always annoying??
- Export Design Properties to Excel

Import/Export Part Properties

- Import Part Properties from Excel to Design

NC-Part Setting

- ## Design component sets are not present
  - Manually add strings in Value
  - Add a new field in the part property and then add a string to display on the page
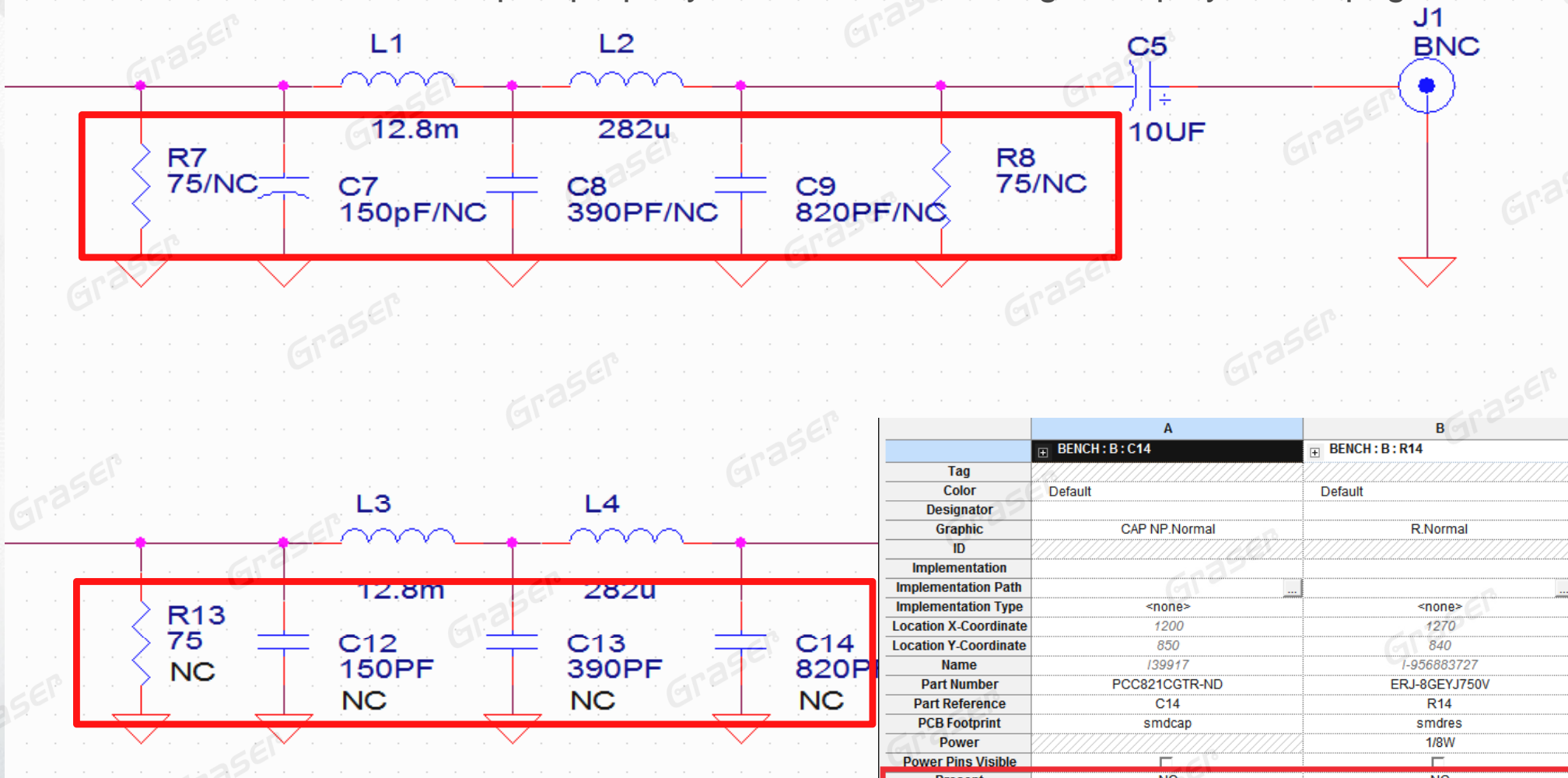
	A	B
	⊞ BENCH : B : C14	⊞ BENCH : B : R14
Tag		
Color	Default	Default
Designator		
Graphic	CAP NP.Normal	R.Normal
ID		
Implementation		
Implementation Path	...	...
Implementation Type	<none>	<none>
Location X-Coordinate	1200	1270
Location Y-Coordinate	850	840
Name	I39917	I-956883727
Part Number	PCC821CGTR-ND	ERJ-8GEYJ750V
Part Reference	C14	R14
PCB Footprint	smdcap	smdres
Power		1/8W
Power Pins Visible	☐	☐
Present	NC	NC
Primitive	DEFAULT	YES
PSpice	C	R
Reference	C14	R14
Source Library	D:\ORCADWIN\V9\CAPTURE\LIBRARY\	C:\WINDOWS\TEMP\XLATTEMP.OLB
Source Package	CAP NP	R

NC-Part Setting

- GraserWARE NC-Part Setting
  - Simply set up part status
  - Auto Parts, increase Value string or BOM_IGNORE property and change part color
  - New BOM Export

Capture Design Compare

- Design version1 and version 2

  How do you know which part or Net-list is different??

Project Design1

Project Design 2

Difference

?
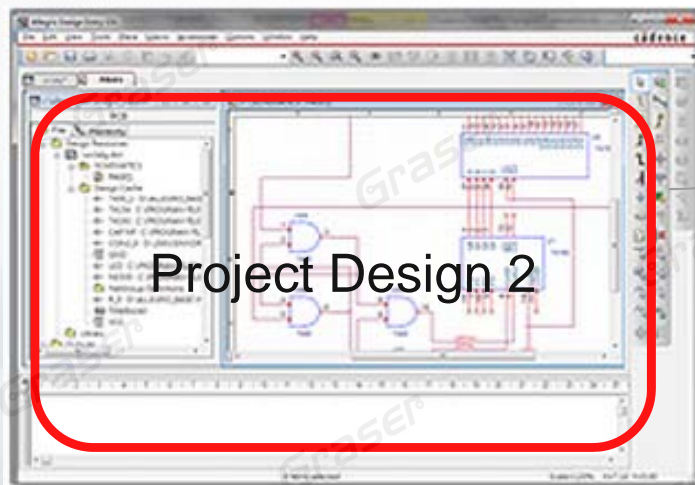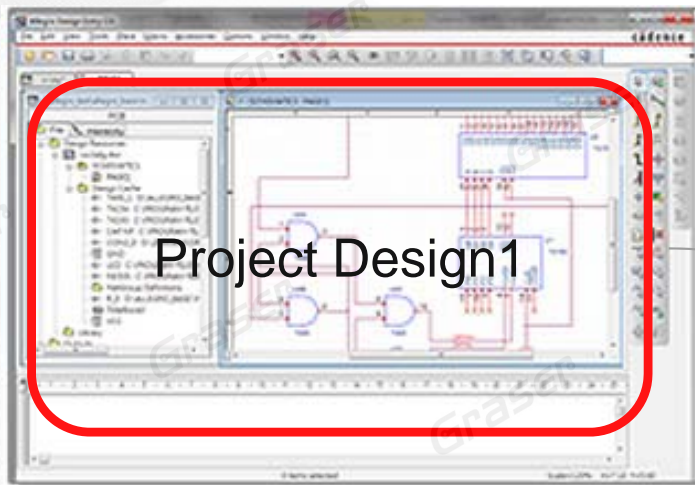
Capture Design Compare

- Design Compare
  - By Part
  - By Net
  - Both
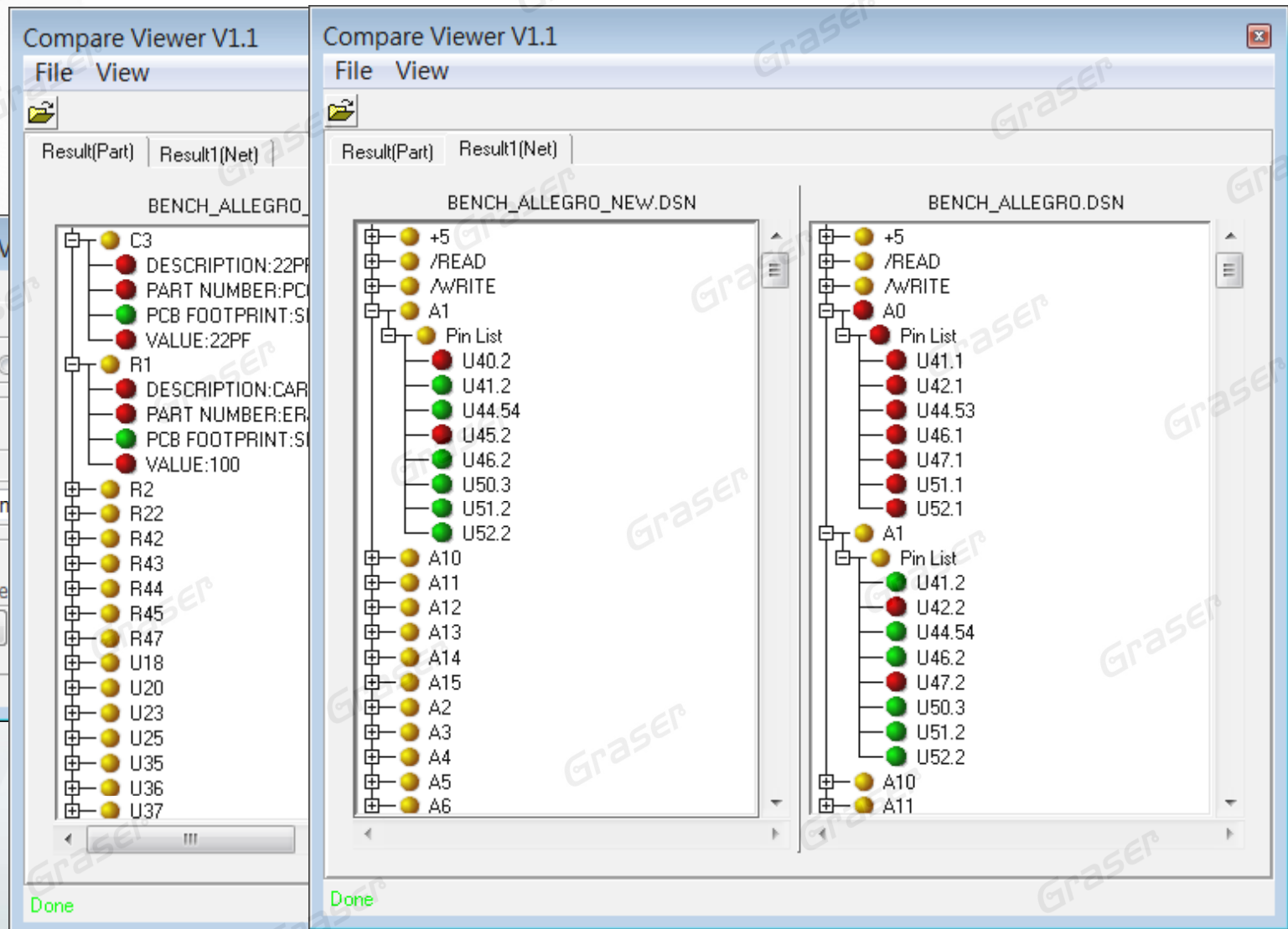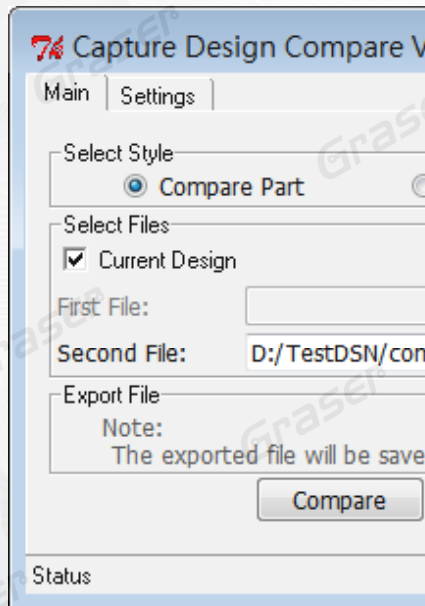
Graser Allegro Netlist

- Using the OrCAD Create Netlist, the schematic of the parts or net properties can be transferred to Allegro Board File.