

NetEditorLite User Manual

Foreword

NetEditorLite is a program that can extract circuit data based on image of chip. It supports automatic or human-computer interactive ways to extract circuit data which is faithful to the chip image. It also can export Verilog or Edif200 file that can be imported into Synopsys, Cadence and other EDA software for redesign.

IST (Beijing) Co., Ltd.

Technical support: gaofeng_chao@isti.com.cn

Company Homepage: www.isti.com.cn

Directory

Foreword	1
Chapter 1 Requirements.....	3
1.1 Operating System Requirements.....	3
1.2 Hardware Configuration Requirements	3
Chapter 2 Installation steps	4
2.1 Installation file describes	4
2.2 Installation and Open a project	4
Chapter 3 Interface introduction	7
3.1 Menu Bar	7
3.1.1 File Menu	9
3.1.2 Edit Menu.....	9
3.1.3 View Menu.....	9
3.1.4 Tools Menu	10
3.1.5 Options Menu.....	10
3.1.6 Window Menu.....	10
3.1.7 Help Menu	11
3.2 Tool Bar.....	11
3.3 Status Bar	12
3.4 Multilayer image navigation bar	12
3.5 Work window	13
3.6 Cell List.....	14
3.8 Radar window	15
Chapter 4 Extraction process	17
4.1 Establish the top cell	17
4.2 Divided Function modules	18
4.3 Extraction within each function module	20
4.4 The net of specify wire	21
4.5 Electrical Rule Check (ERC)	22
4.6 SVS	22
Chapter 5 Check Design Rules	24
Chapter 6 Export schematic data	25
Chapter 7 Export layout data	27
7.1 Export CIF file	27
7.2 Import data into Layout Editor	29
7.3 import data into Cadence	29
Chapter 8 About Setting Options	30
8.1 Color Settings.....	30
Frequently Asked Questions	33
Appendix: Shortcuts	35

Chapter 1 Requirements

1.1 Operating System Requirements

NetEditorLite can be running under the following operating systems:

- (1) Windows 2000
- (2) Windows 2003
- (3) Windows XP

The recommended operating systems are as follows:

Server: Windows Server 2003

Clients: Windows XP Professional

1.2 Hardware Configuration Requirements

- (1) CPU: Pentium IV 1.6G or above
- (2) 256M RAM or more
- (3) 40G hard disk or higher
- (4) 1024 × 768 screen resolution or larger

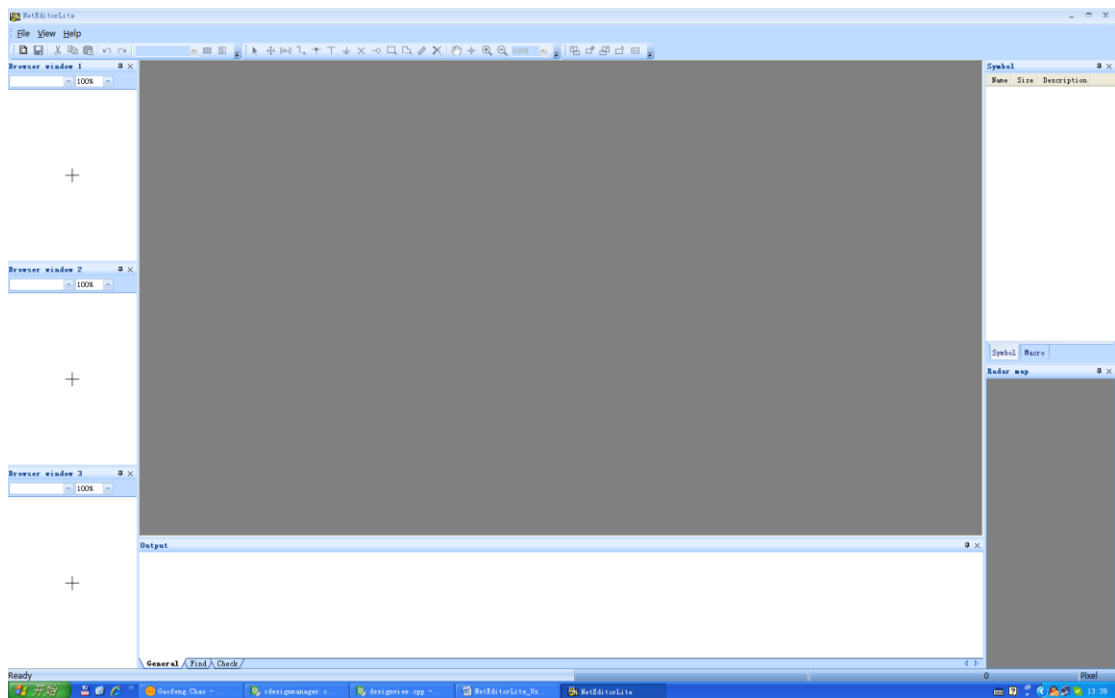
Chapter 2 Installation steps

2.1 Installation file describes

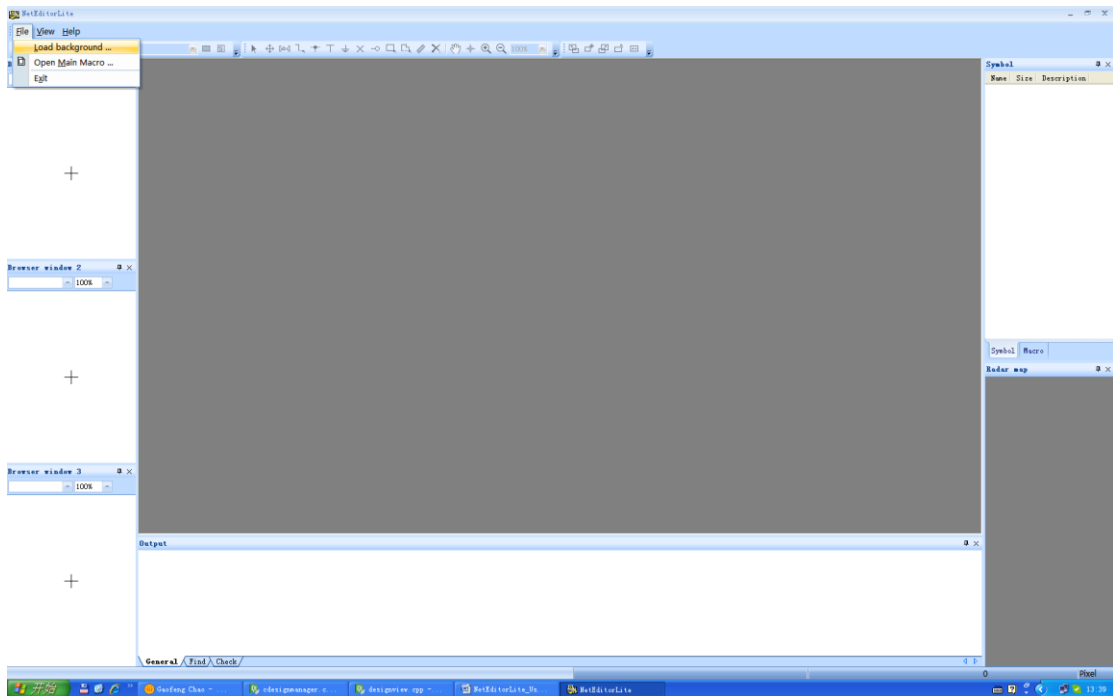
NetEditorLiteSetup.msi: installation package, installing the NetEditorLiteSetup program for extract netlist data.

2.2 Installation and Open a project

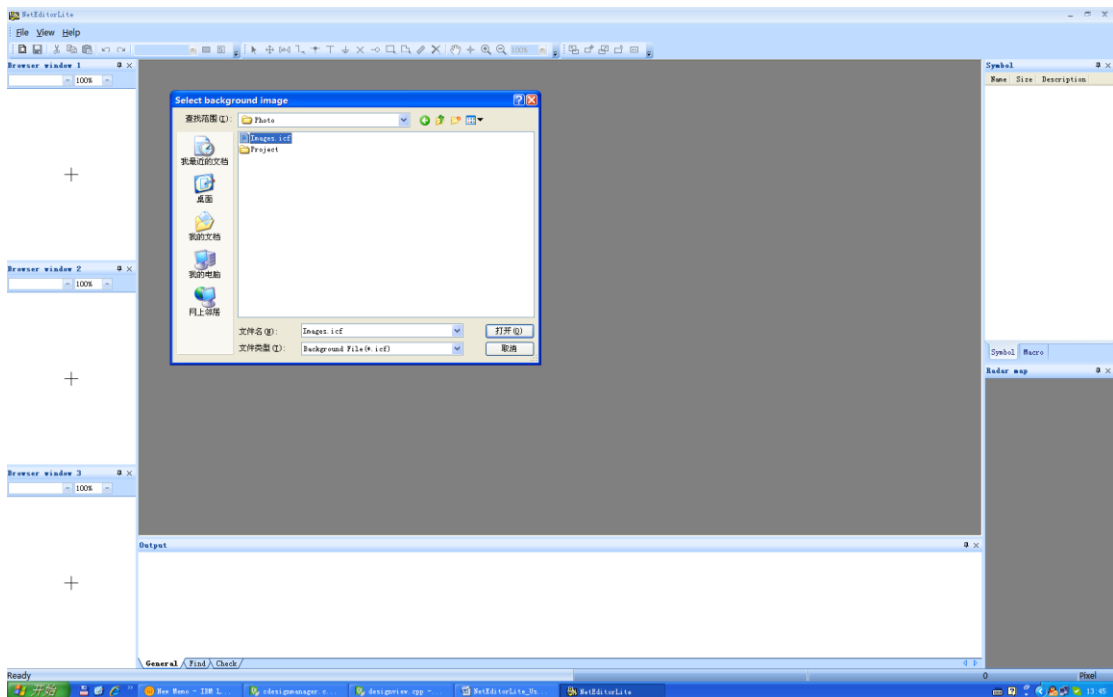
1. Executive NetEditorLiteSetup.msi, according to the installation wizard prompts, step by step installation.
2. After installation has been complete, click desktop shortcut NetEidtorLite, as shown below:



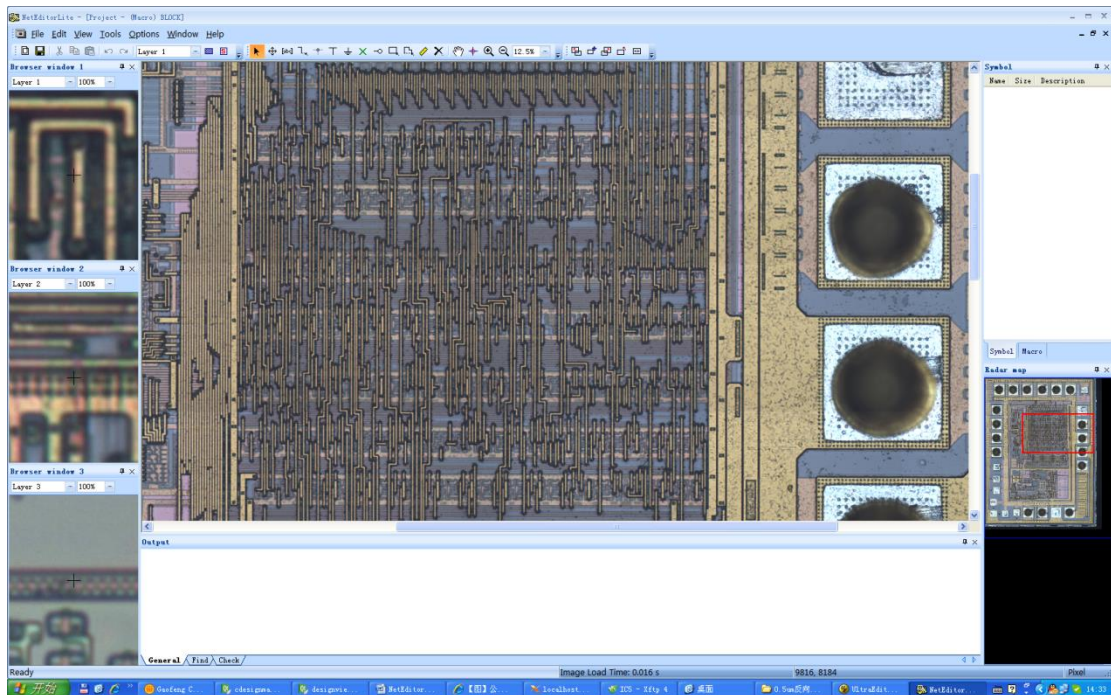
Click menu "File", select "Load Background Image", as shown below:



Select image file(". icf"), then click "Open", as shown below:




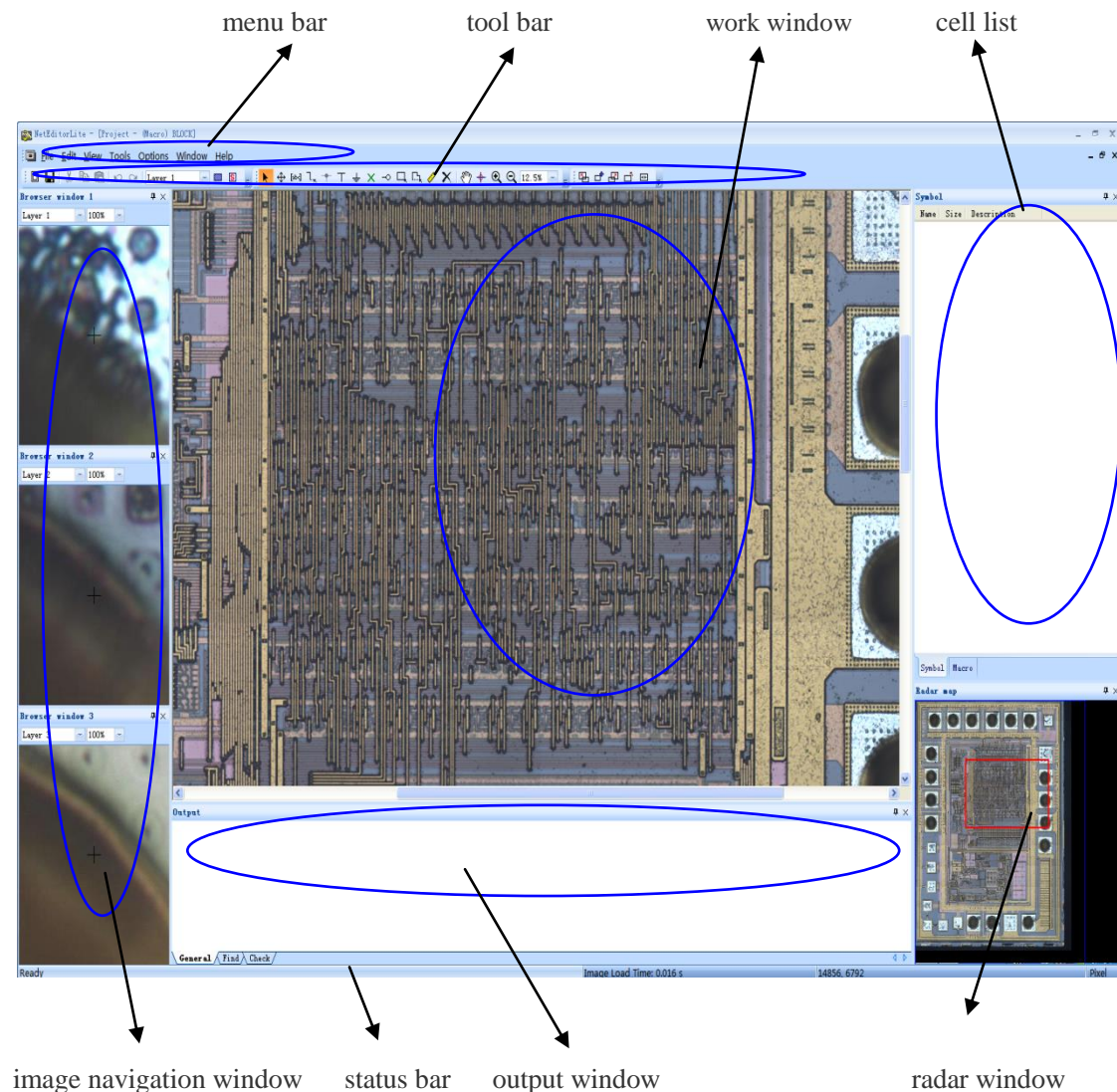
Click menu "File" and select "Open Main Macro...", the image show in the work window, as shown below:



Open a project end.

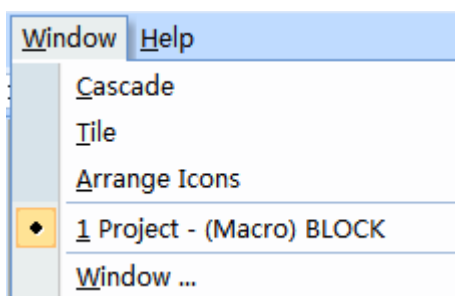
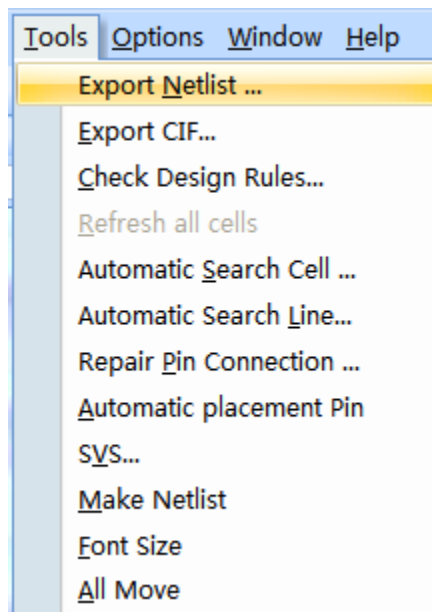
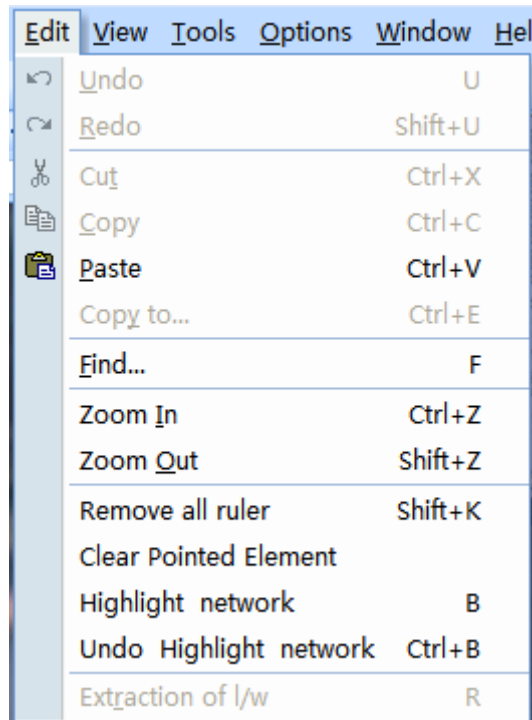
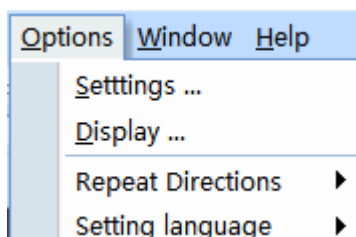
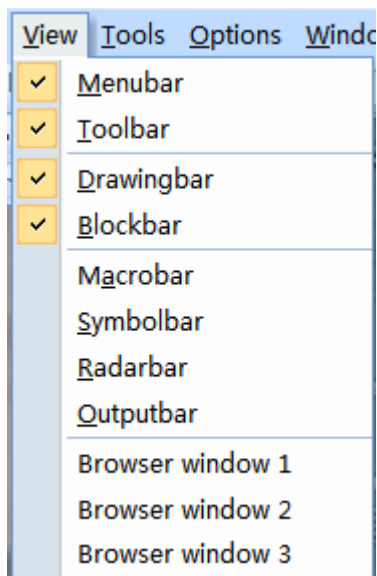
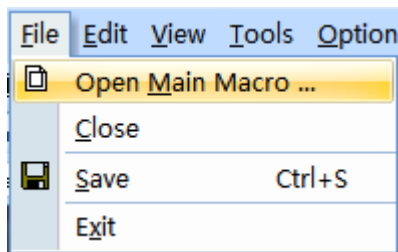
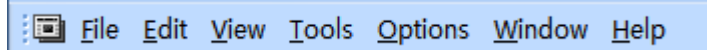
Chapter 3 Interface introduction

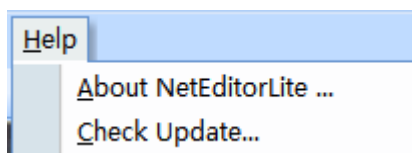
Click icon  (Create a top macro cell), as shown below. The interface includes the following sections: menu bar, tool bar, status bar, multilayer image navigation window, work window, cell list, output window and radar window for positioning.



3.1 Menu Bar

Menu bar as shown below, it corresponds to some of the basic operation of software system. Each menu described below:





3.1.1 File Menu

Menu	Function	Shortcuts
Open Main macrocell	New/Open top cell	
Close	Close the active document	
Save	Save the active document	Ctrl + S
Exit	Exit the application	

3.1.2 Edit Menu

Menu	Function	Shortcuts
Undo	Undo the last action	U
Redo	Re-execute the last action	Shift + U
Cut	Cut the selection content	Ctrl + X
Copy	Copy the selected content	Ctrl + C
Paste	Insert Clipboard contents	Ctrl + V
Copy to		Ctrl + E
Find	Find the specified text	Ctrl + F
Zoom In	Enlarge the current view	Ctrl + Z
Zoom Out	Shrink the current view	Shift + Z
Remove all ruler	Clear all ruler	Shift + K
Clear Pointed Element	Delete by element type	
Highlight network	Highlight network	B
Undo Highlight network	Highlight Cancel	Ctrl + B
Extraction of l / w	Extraction length & width	R

3.1.3 View Menu

Menu	Function	Shortcuts
Menubar	Show or hide the menu bar	
Toolbar	Show or hide toolbars	
Drawingbar	Show or hide the drawing column	
Blockbar	Show or hide the BlockBar	
Macrobar	Show or hide the Macrobar	
Symbolbar	Show or hide the Symbolbar	
Radarbar	Show or hide the Radarbar	

Outputbar	Show or hide the Outputbar	
Browse window 1	Show or hide the browser window 1	
Browse Window 2	Show or hide the browser window 2	
Browse Window 3	Show or hide the browser window 3	

3.1.4 Tools Menu

Menu	Function	Shortcuts
Export Netlist	Generated shematic data file	
Export CIF	Generated l layout data file	
Check Design Rules	Check design rules	
Refresh all cells	Refresh cell	
Automatic Search Cell	Automatic search cell instances	
Automatic Search Line	Automatic search wire	
Repair Pin Connection	Fix broken connection between	
Automatic Placement Pin	Place PIN to an empty line head	
SVS	schematic data comparison	
Make NetList	Recalculate the logic relation	
Font Size	Set font size	
All Move		

3.1.5 Options Menu

Menu	Function	Shortcuts
Settings	Grid and automatic connection settings	
Display	Settings display of each Figure	
Repeat Direction	Setting naming repeat method	
Setting language		

3.1.6 Window Menu

Menu	Function	Shortcuts
Cascade	Arrange the windows to overlap	
Tile	Arrange windows as non-overlapping	
Arrange Icons	The icons at the bottom of the window	
Project-(Macro) BLOCK	Currently open window	
Window	Manage the current window	

3.1.7 Help Menu

	Menu	Shortcuts
About NetEditorLite	Display program information, version number and copyright	
Check Update	Find the highest version	

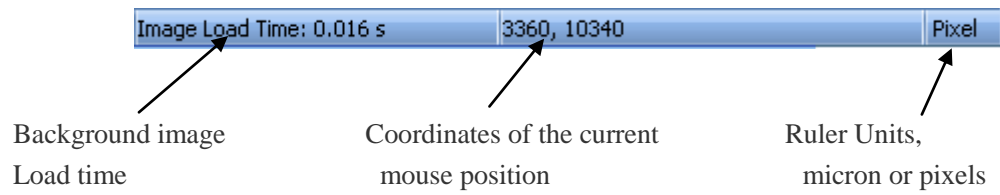
3.2 Tool Bar

Some button icon of commonly command on the toolbar, each function of button icon and their corresponding Shortcut in the following table.

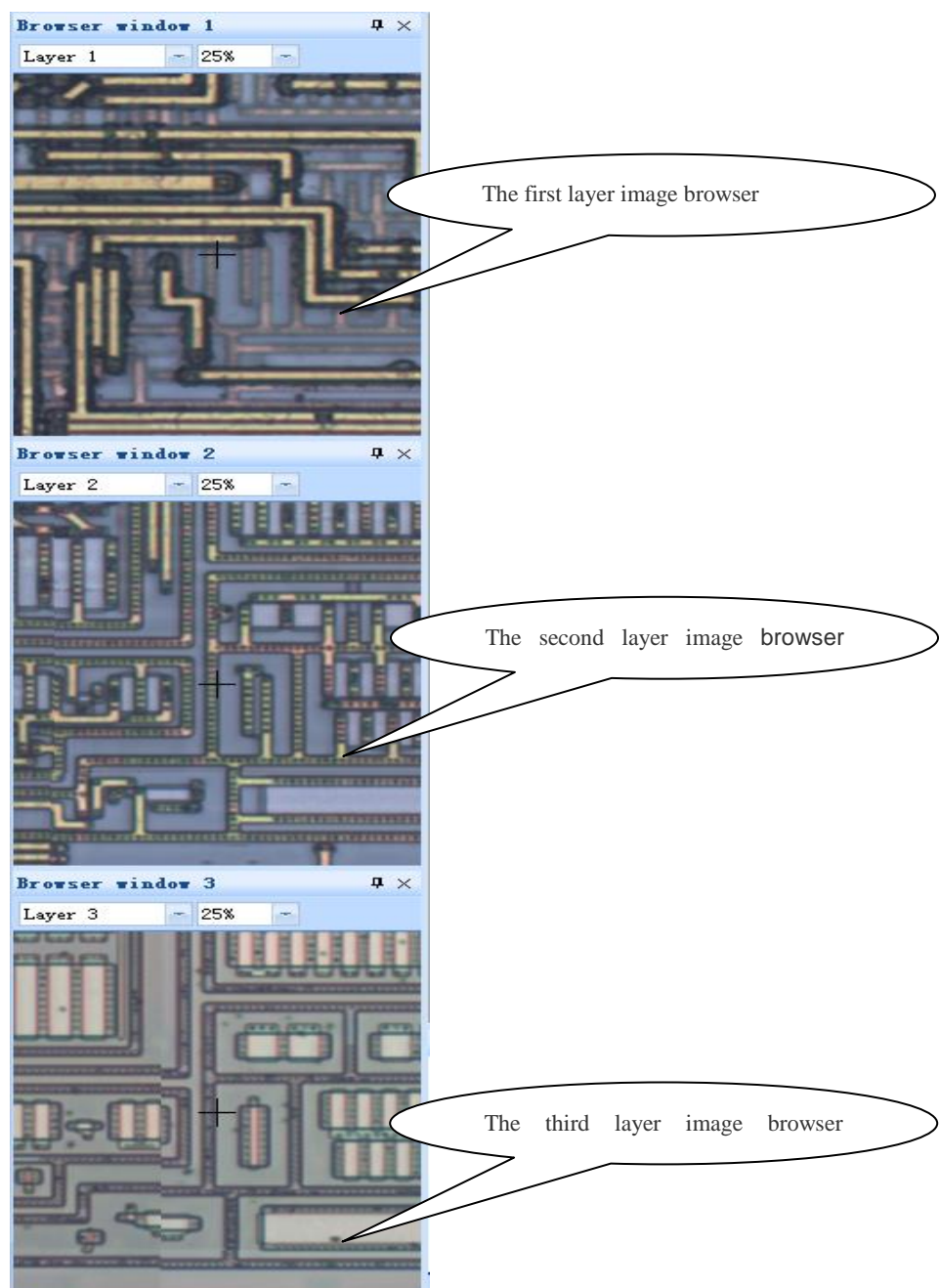


Icon	Function	Shortcuts
	Show / Hide Image	~
	Show all / selected objects	F8、 E
	Add label	F1
	Add wire	F2
	Add junction	
	Add vdd instance	F4
	Add gnd instance	F5
	Add noConn instance	F6
	Add pin	F7
	Rectangle / Create Cell	
	Polygon/Create Cell	
	Add Ruler	K
	Delete	DEL
	Move screen	
	Coordinate positioning	G
	Enlarge	Ctrl+Z
	Shrink	Shift+Z
	Drag (remain the connection relationship)	
	Move (not remain connected relationship)	
	Copy	
	Rotation	
	Two selected points automatic connection	F3

3.3 Status Bar



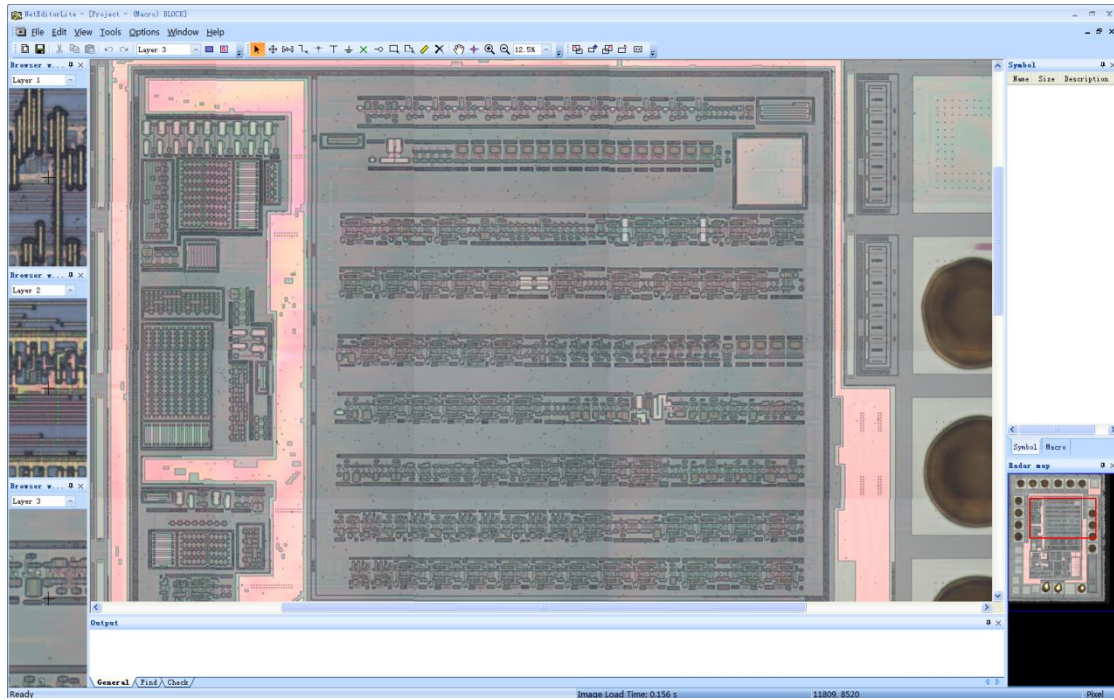
3.4 Multilayer image navigation bar



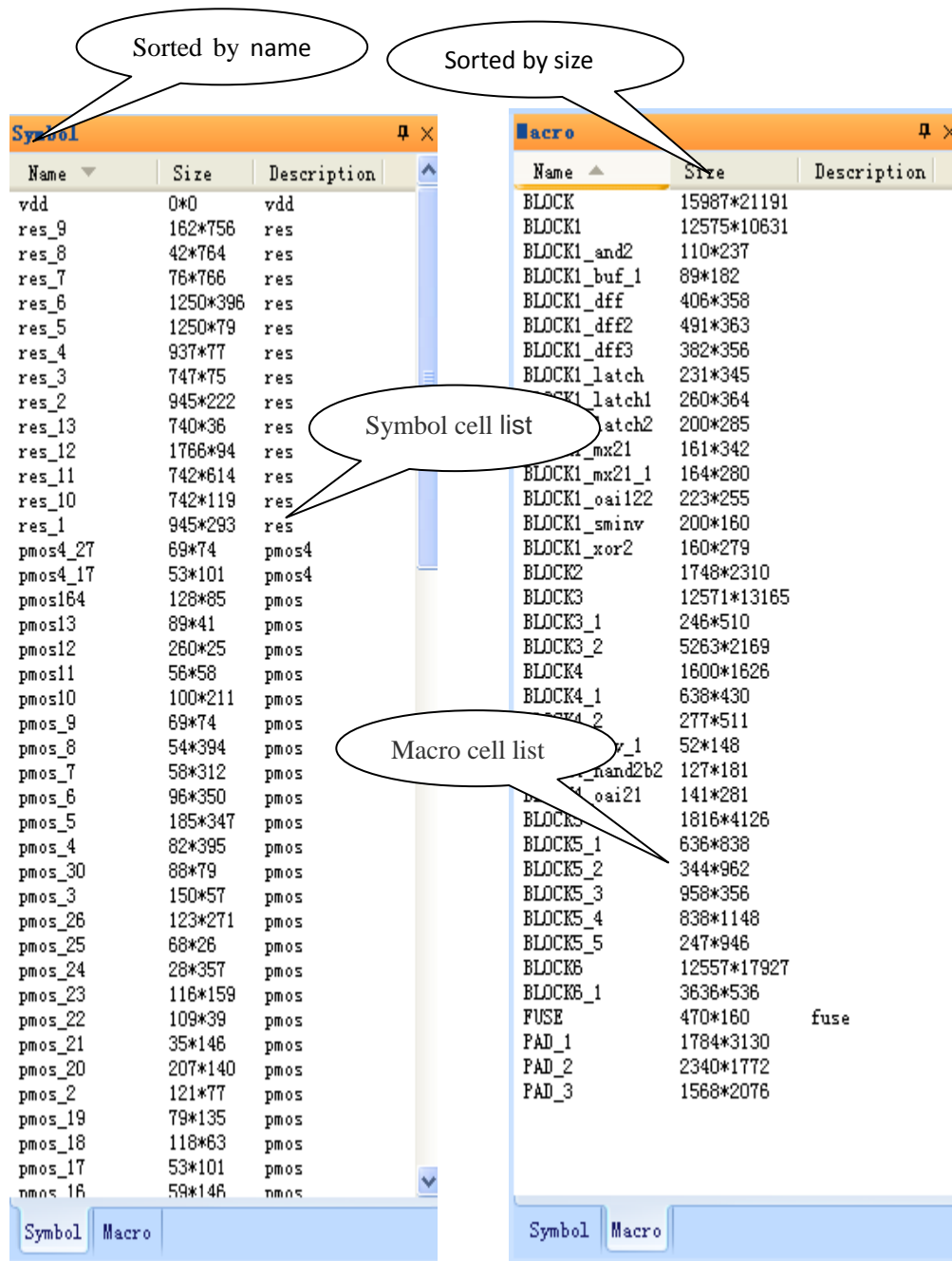
3.5 Work window

Work window shows the current position of the image, Switch to a different layer image by press the number keys or ' + ' , ' - ' .

Press ~ to display or hide the background image.



3.6 Cell List

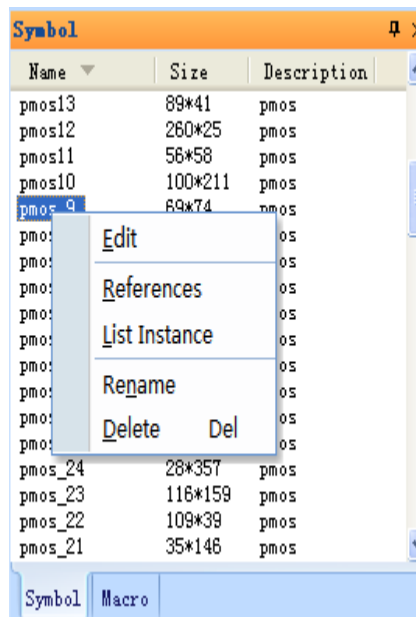


Related operations:

Click on "Name", "size", "Description" tab to make the appropriate sort.

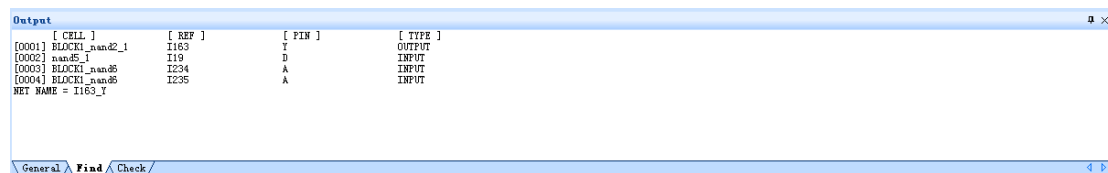
Double-click the cell name, open the corresponding cell.

Right-click to display the menu below:



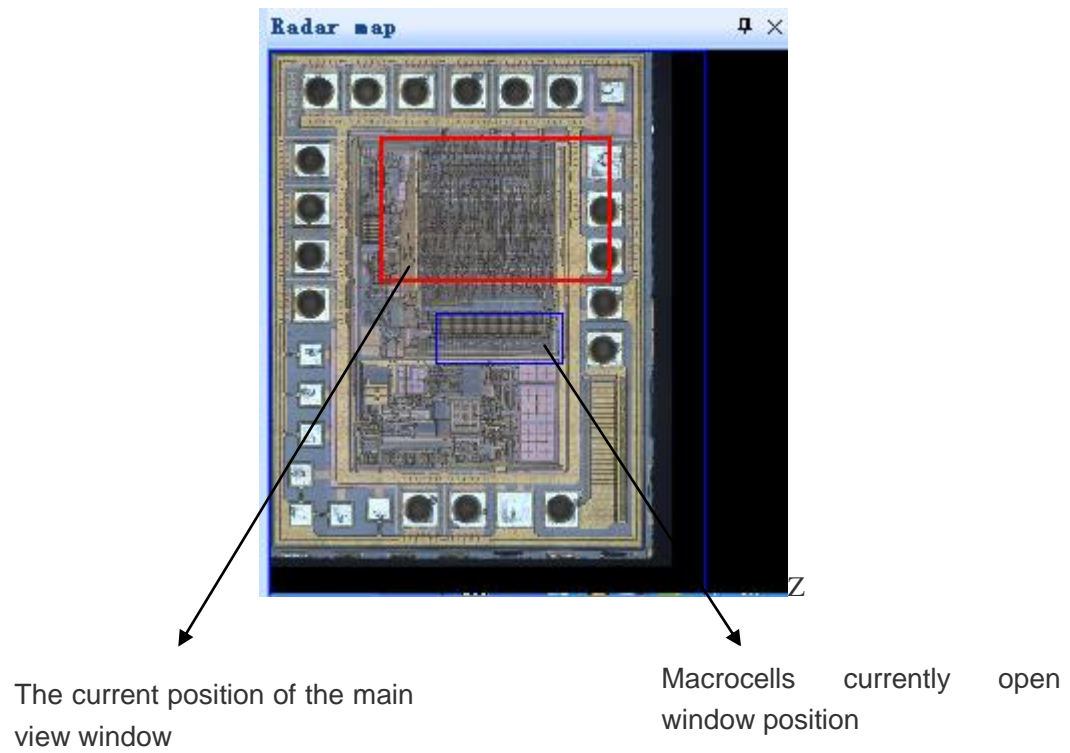
3.7 Output window

Output window record some result of the corresponding operation, such as pins on net, find, ERC etc.



3.8 Radar window

By radar navigation bar, user can clearly know the current cell in the entire chip image position. Also user can easily location position of the entire chip image by clicking on radar window.




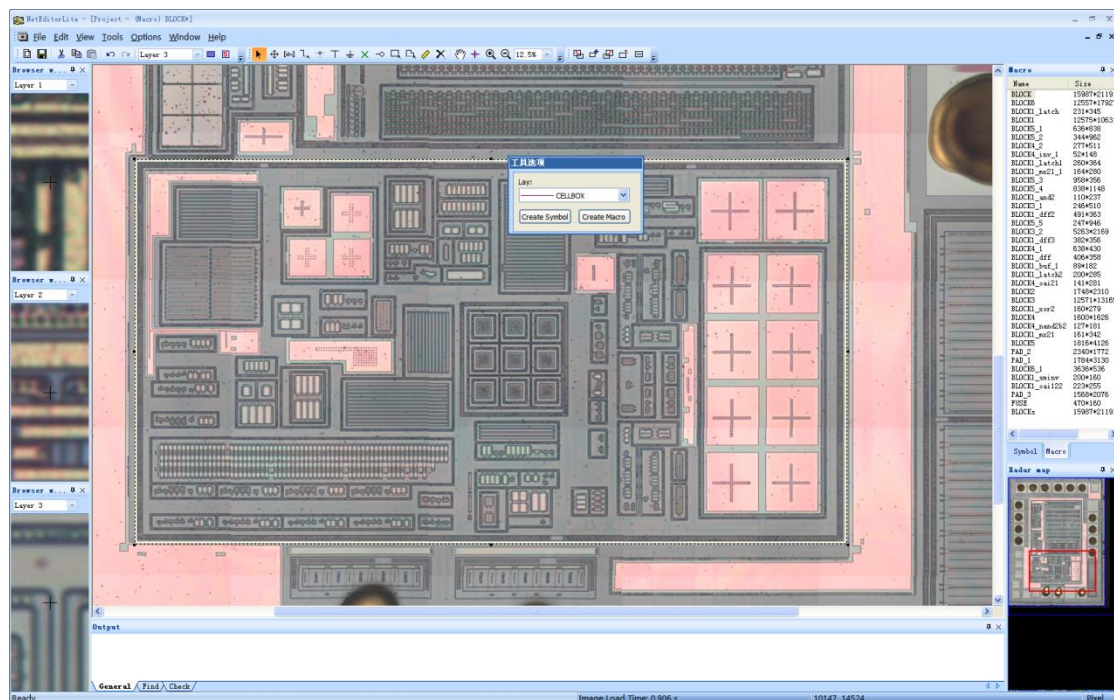
Chapter 4 Extraction process

Electric circuit extracted by the following processes:

1. Establish the top cell
2. Divided function module
3. Extraction within each function module
4. Electrical Rule Check (ERC)
5. Export

4.1 Establish the top cell

Main macro cell corresponding to the entire chip work area, it is the top cell. Establish the main macro cell purpose is to future operations are limited to inside this master macro cell, including the establishment macro cells (function module) and symbol cells (component) . Main macro cell established according to the following methods: reduce the image to a suitable proportion, click on the toolbar  button, drawing an area, as shown in white border:



In the pop-up "Tools Options" dialog, click the "New macro" and then in the pop-up "Cell

properties" dialog, click the "Save" button, as shown below. In the "Cell properties" dialog, you can modify its Name property, or can also add RefPrefix parameter that are used as a prefix of instance name within cell .By this method, you can easily find out it belongs cell through the prefix of instance name. This example its Name property to BLOCK, RefPrefix parameter value is BL_, so in BLOCK cell all referenced names of instance are the beginning of BL_.

Parameter	Show?	Sync?	Default value
Name	Default to show	Yes	BLOCK
Reference	Default to show	Yes	M?
RefPreview	Default to hidden	Yes	BL_

Direction: Top

Description:

External libraries

Library:

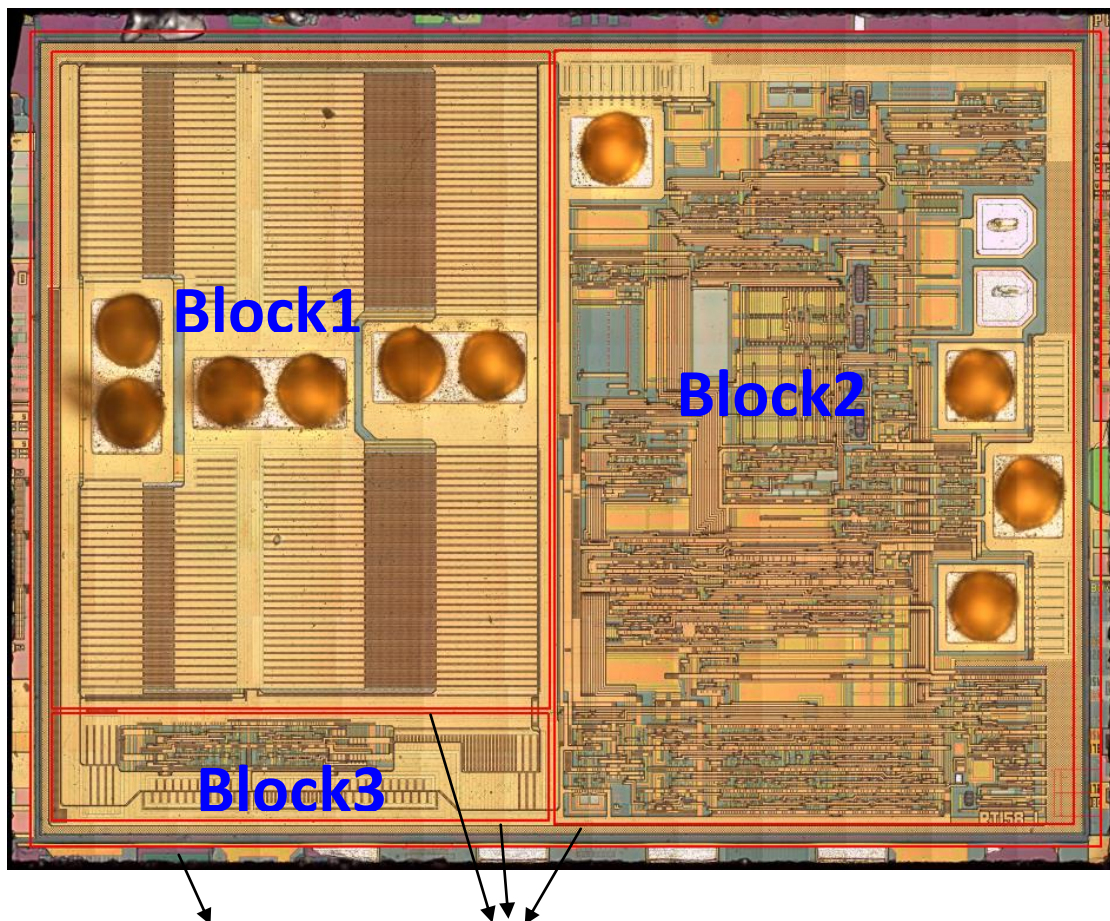
Cell:

External cell in external library contains the symbolic figure geometric information and pins. It must be specified when exporting edf200 file.

4.2 Divided Function modules

According to working principle and electrical functions of various parts of the chip by image, the cell can be further divided into several macro cells (function module) and modules should not overlap each other. Division method has the same steps as establish the main macro cell

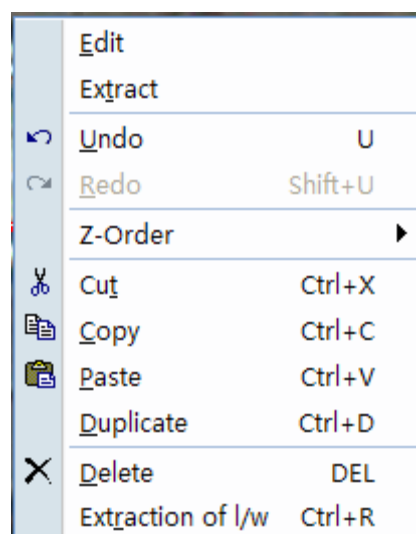
A division of functional modules in top cell as show below:

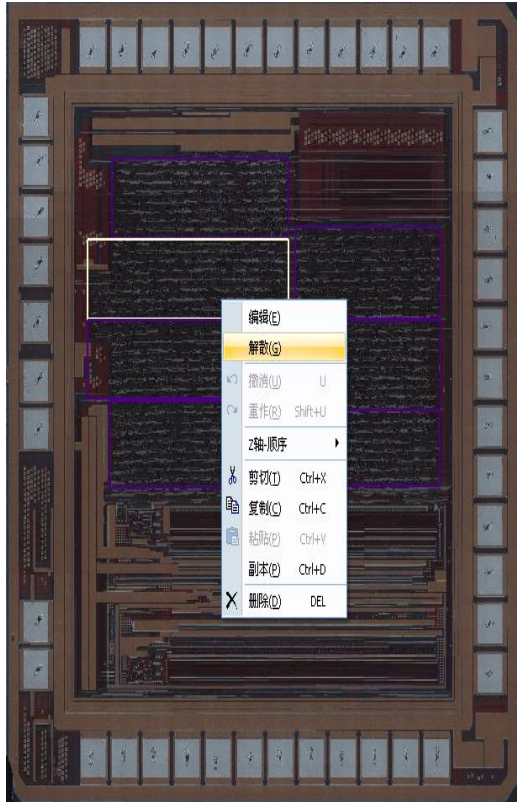


Main macro cell border

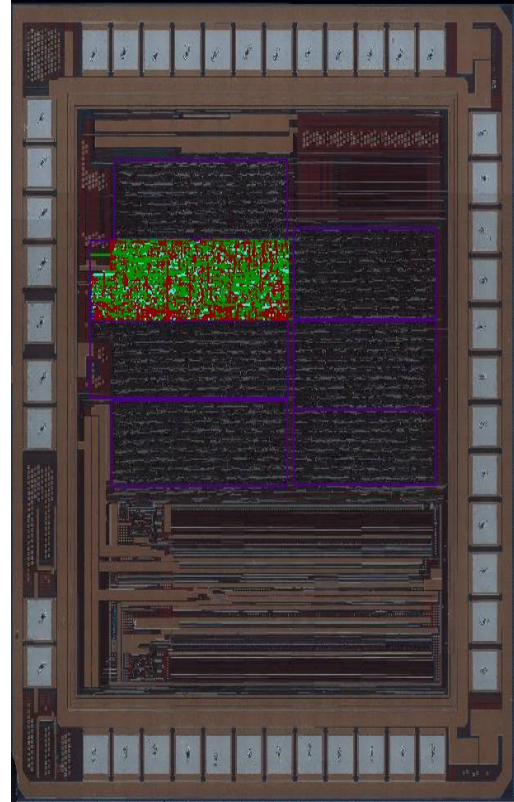
Macro cell (function module) border

After the end of extraction of each functional module (Block1 or Block2 or Block3) in the top cell(BLOCK), you can put netlist data of instance that represent a functional module into the master Macro by pop-up menu by right clicked on the boundary of instance as illustrated "Extract" .






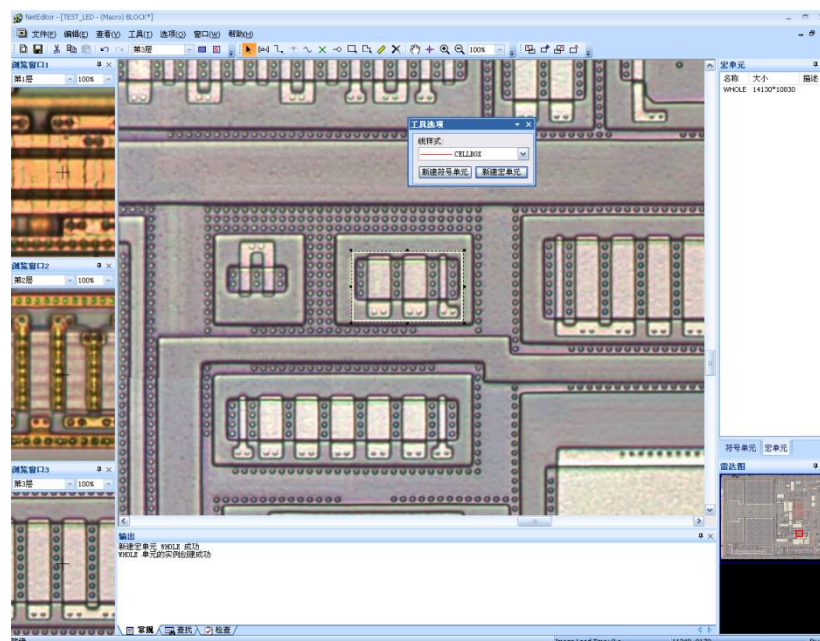
Before



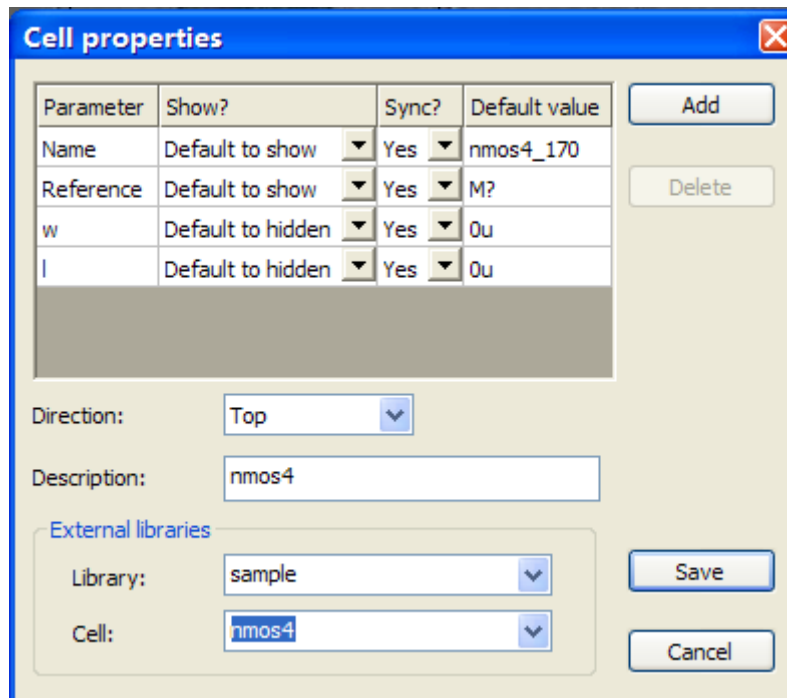
After

4.3 Extraction within each function module

a) First, symbol cell (component) is defined by the various functional of the module. Symbol cell defined as follows: click on the toolbar  button, draw boundary of cell, as shown:



In the pop-up "Tool Options" dialog, click the "Create Symbol", and modify its Property, and click "Save", as shown.



b) Second, placing instances of cell by Automatic search cell command and manual confirmation or Manual placement instance (detailed reference to Chapter V).

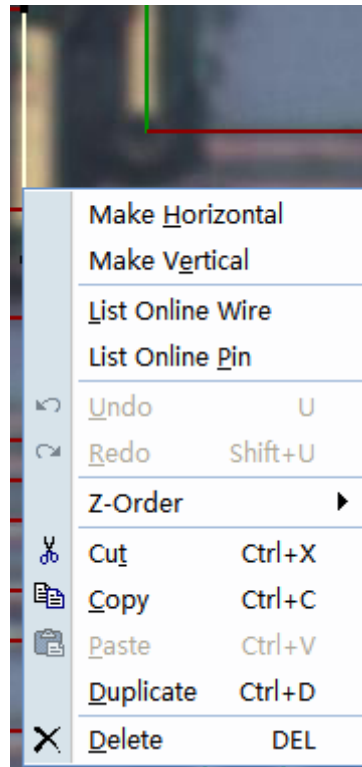
c) Third, drawing line by Automatic search cell command manual confirmation or Manual draw line (detailed reference to Chapter VI).

d) Fourth, Electrical Rule Check.

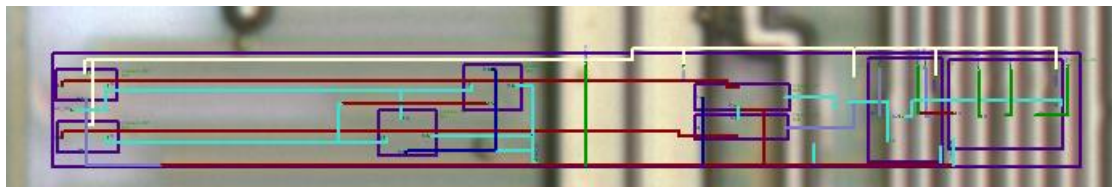
e) Finally, Generation of external pin that comprise a complete electric circuit.

4.4 The net of specify wire

Extraction of each functional module is over, you can view pins or wires on a net by right-clicked that pop-up menu and selected "list online wire "or "list online pin" on the specify wire. The result will be list the output window. Press TAB key one by one positioning view (press SHIFT + TAB back to the previous article).



You also can see the whole composition of net by double-click the specify wire, all wires and pins are listed, as shown below:



4.5 Electrical Rule Check (ERC)

Electrical rule checking should run at the end of extraction in each functional module. After By electrical rule checking, you can eliminate simple errors included in the circuit, as physical、logical、reference name and so on (detailed reference to Chapter V).

4.6 SVS

In order to improve correct rate of the extraction data, we generally performed twice or more times extraction, then proceed comparison (SVS) between each other. Two or more times extraction should be based on the same reference cell data, then wires of extracting alone (Get the same reference cell data by save as).

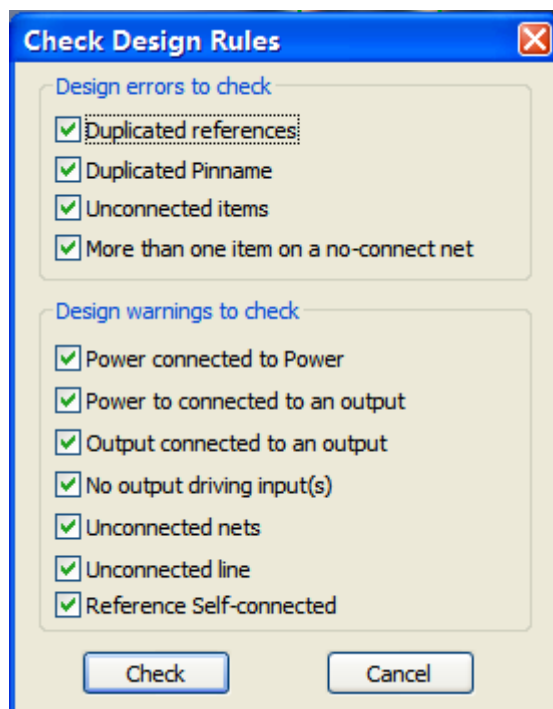
4.7 Export

Finally, you can export the electric circuit that extracted to file that can import to Synopsys, Cadence for redesign. Verilog, Edif200 format of file can be exported (detailed reference to Chapter VI)

Chapter 5 Check Design Rules

Electrical rule checking should be checked after extraction of each functional module finished and before export schematic data. By electrical rule checking, you can eliminate the circuit errors included reference name, physical and logic and so on.

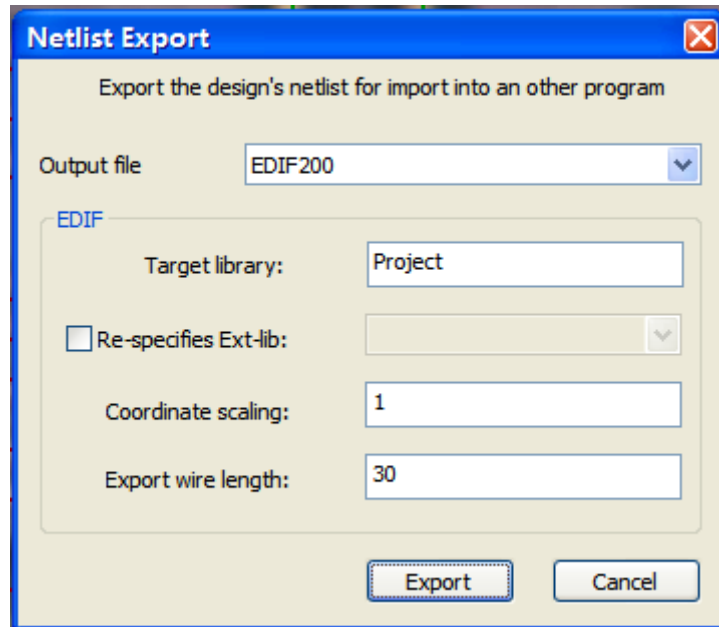
Click menu "Tools" -> "Check Design Rules" will pop up the following dialog:



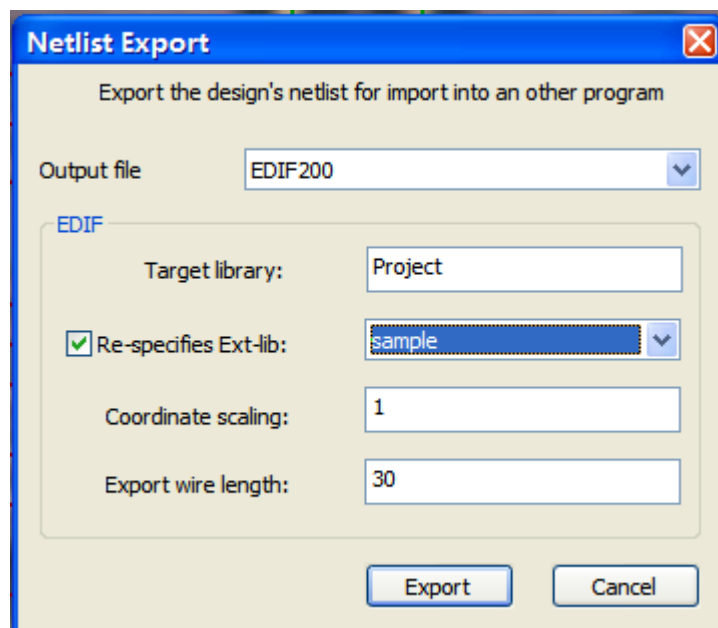
The selected items on dialog as shown above are checked after clicked the button "Check." The result will be displayed in the output window. Like with the previous operation, press Tab or SHIFT + Tab key to locate wrong location.

Chapter 6 Export schematic data

Click the menu “Tools” -> "Export Netlist" will pop up the following dialog:



Currently the output file format includes Verilog and Edif200. Click the "Export" will be generated in the specified format file.



Re-specifies Ext-lib: Changed reference symbol of external library that is different form

created

Coordinate scaling: setting when cells are too dense or too sparse between each other.

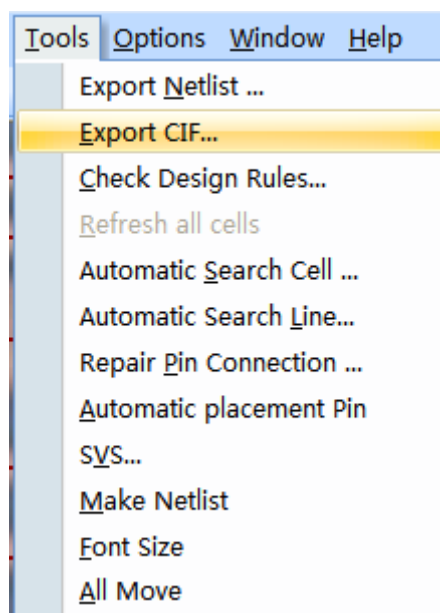
Export wire length: wire length in edif200.

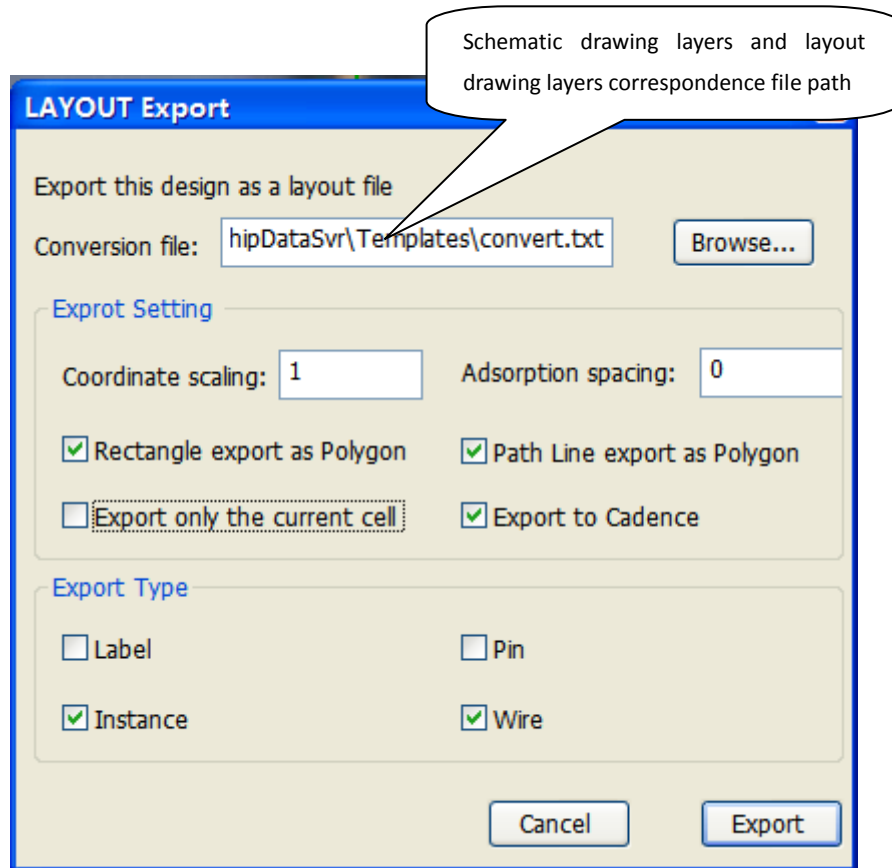
Chapter 7 Export layout data

According to schematic data that extracted, you can export brief layout data, and then import into LayoutEditor or Cadence to see Corresponding layout.

7.1 Export CIF file

In the menu bar "Tools", click "Export CIF ...", as shown below:





Correspondence between Schematic drawing layers and layout drawing layers file content is as follows:

LayerMap	CELLBOX Boundary 0.45		
LayerMap	M1	MET1	0.45
LayerMap	M2	MET2	0.5
LayerMap	M3	MET3	0.5
LayerMap		M4	MET4 0.5
LayerMap		M5	MET5 0.5
ViaMap	M1	M2	VIA1
ViaMap	M2	M3	VIA2
ViaMap	M3	M4	VIA3
ViaMap	M4	M5	VIA4
ViaParam VIA1	0.45	0.15	
ViaParam VIA2	0.45	0.15	
ViaParam VIA3	0.45	0.15	
ViaParam VIA4	0.45	0.15	

LayerMap: drawing layer convert keyword,
Layer name of schematic、layer name of layout 、width of layout.

ViaMap: via convert keyword,
Adjacent layer name of schematic、via layer name of layout.

ViaParam: via defined keyword

Layer name of via、 width of via layer、 Distance covered

Note: case sensitive, a line must has a keyword

7.2 Import data into Layout Editor

Detailed reference to LayoutEditor user manual

7.3 import data into Cadence

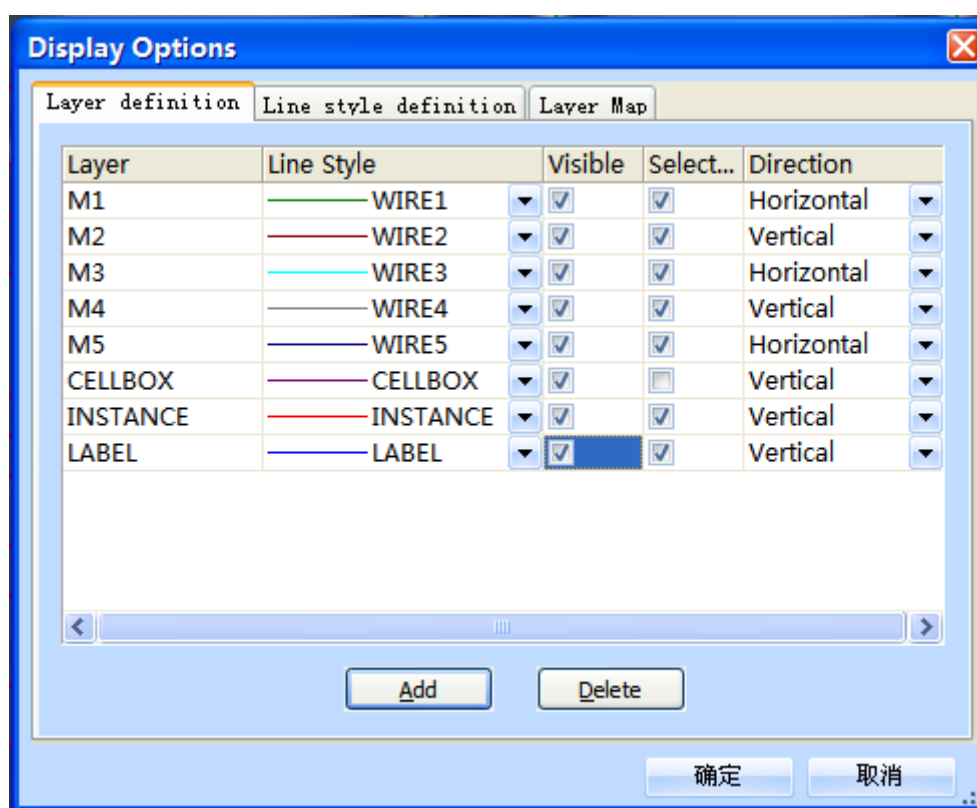
Detailed reference to LayoutEditor user manual

Chapter 8 About Setting Options

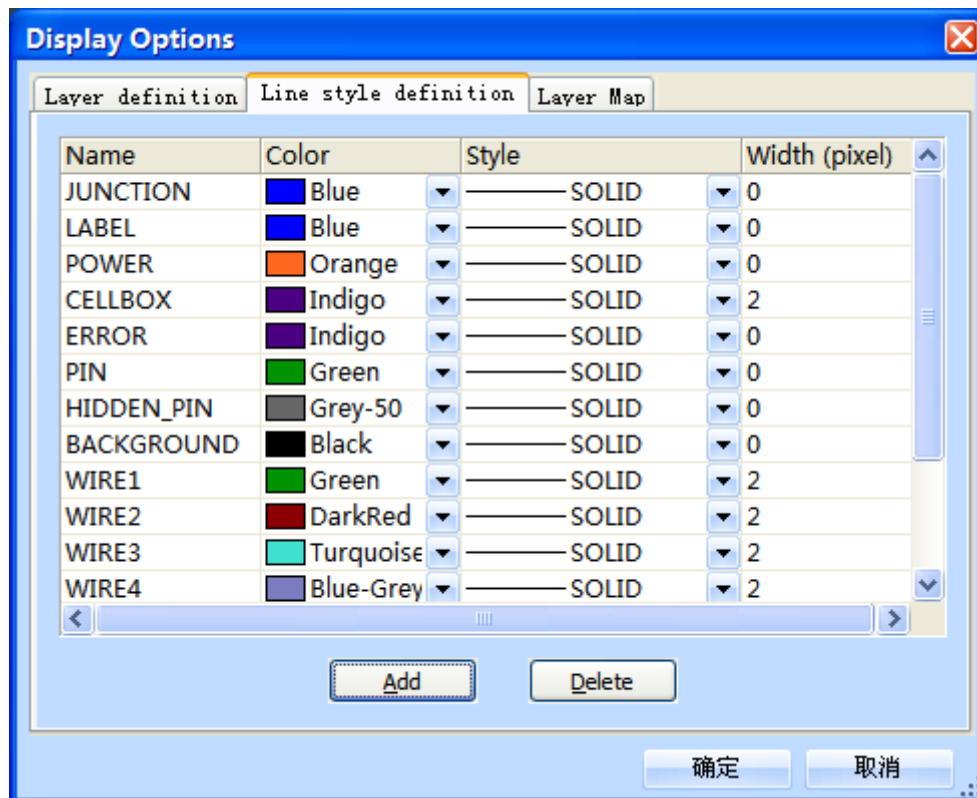
Some setup options of the software is very important, such as color settings, grid settings, the following will be described one by one.

8.1 Color Settings

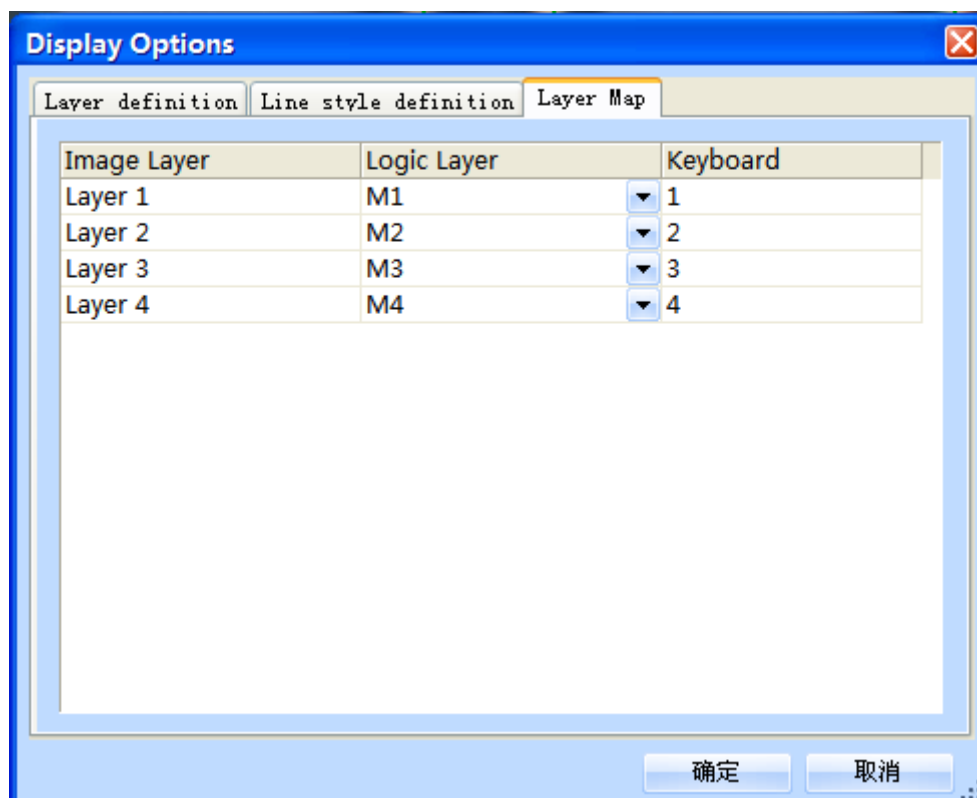
Clicked menu "Options" ->"Display", pop up a dialog about Display Options, as shown below. You can click "Add" to add a new layer lines, and set it to the corresponding style and direction attributes.



Click on the tab "Line style definition", dialog will pop up as shown below. You can set the line name, color, style, line width and other attributes. By clicking "Add" you can increase the new line style.



Click on the tab "Layer Map" ,dialog will pop up as shown below. You can set the Image layer, Logic layer (Draw layer), and the Numeric shortcut keys.



8.2 Option settings

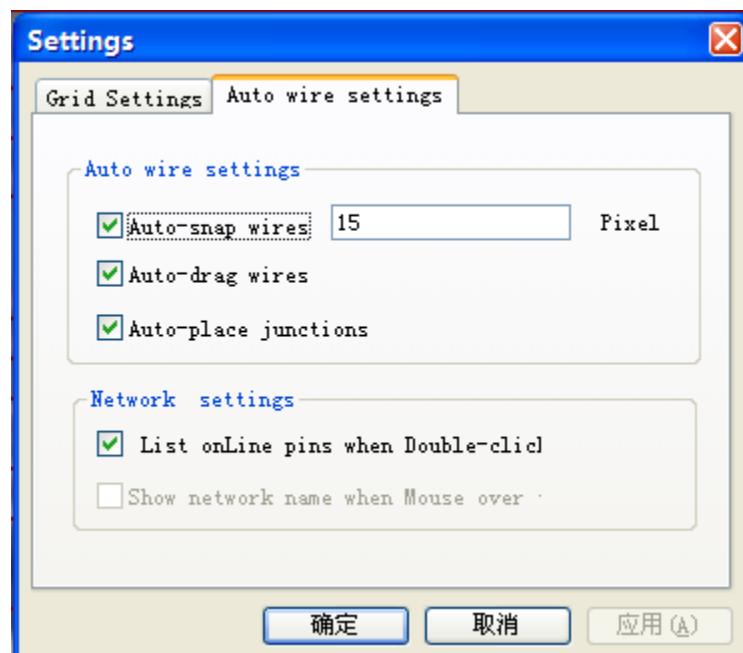
Clicked menu "Options"-> "Settings", the following dialog will be displayed. In the "Grid Settings" tab, you can set precision and units of ruler.

Users can precision of ruler by setting the grid spacing.

When you need to use measuring the size of components, generally choose "unit is microns."



"Auto wire Settings" tab as shown below. The options have the following meanings:

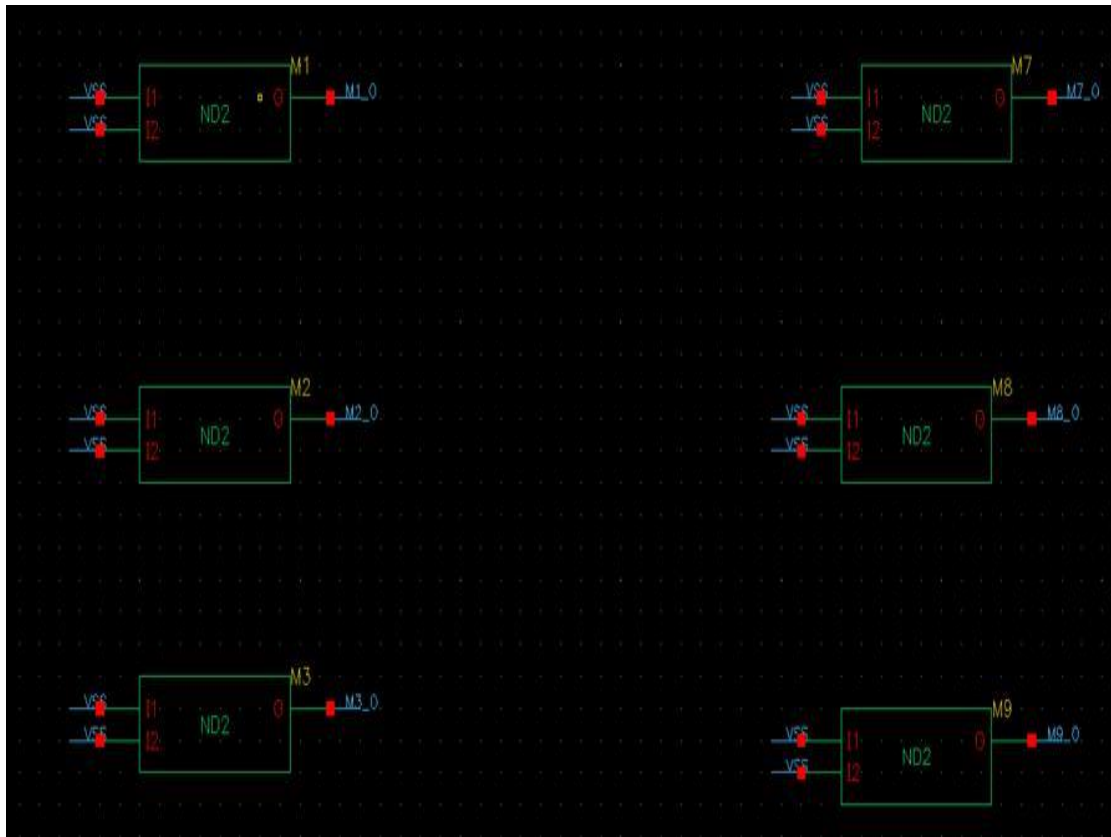


Check the "Auto-snap wires", the current mouse position from the nearest element is less than a certain threshold (default is 15 pixels, the user can modify), the line will automatically snap to the nearest element when drawing wire.

Check the " List onLine pins when Double-click ", list all elements included pins connected line when double-click a line.

Frequently Asked Questions

Q1: After import edf200 file as below, although there is wire name, but no full connection, ask how to operate? Whether connect manually for the full connection?

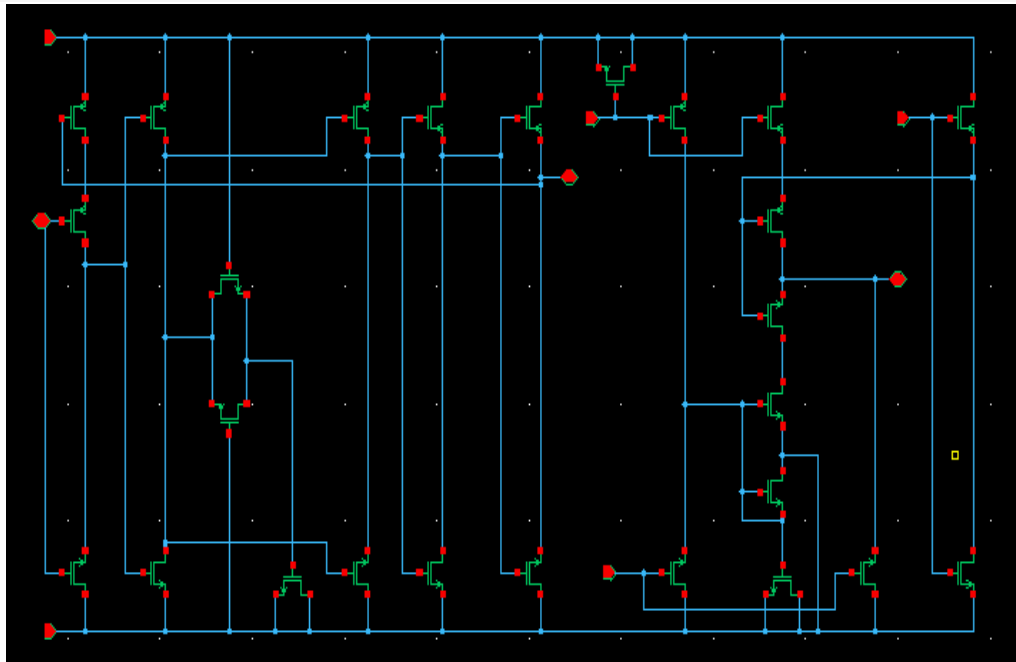


Re:

Edf200 file exported from NetEditorLite, full wire was shortened into small wire, But retaining its logical connection relationship, as: M1_0, M2_0, M3_0 are representing a logical connection relationship by wire name.

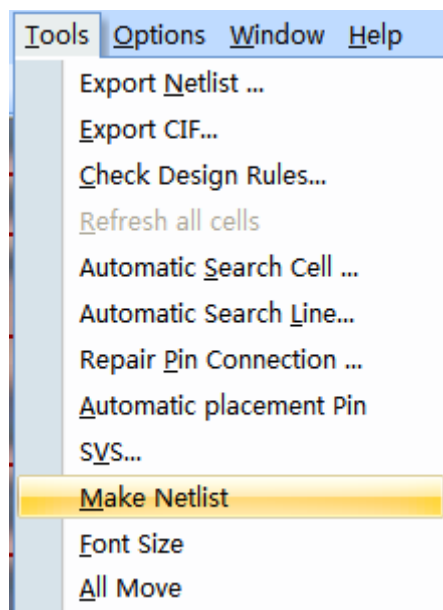
Because edf200 file without losing its logical connection relationship, it does not affect subsequent simulation and verification work.

Pictured below shown after manually structured circuit:



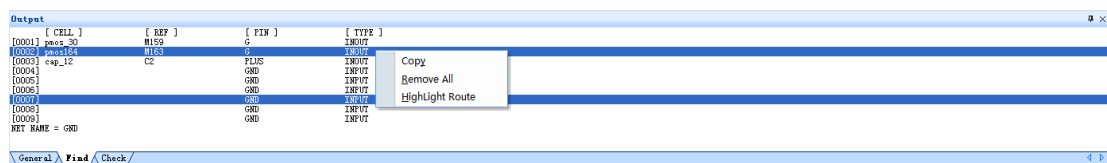
Q2: Logical relationship disorders found, how to solve.

Re: Run "Make Netlist" command




















Q3: How to quickly find the shorted connection points in the network

Re: First list the network pin, and then in the output window, for as the following operation:



Appendix: Shortcuts

Shortcut	Icon	Function
Double-click the cell in Macro list or Symbol list		Open the cell
Numeric keys as 1、2、3 ...or +、-		Switch different layers of image
~		Show / Hide Image
F8、 E		Show all / selected elements
Arrow keys or W, A, S, D		Move the view up and down
F1		Add label or wire name
F2		Add wire
F3		
F4		Add vdd Instance
F5		Add gnd Instance
F6		Add noConn Instance
F7		Add Pin
F3		Connect the two selected points
DEL		Delete
K		Add ruler
Shift + K		Clear ruler
U		Undo the last operation
Shift + U		Re-execute the last action
Ctrl + X		Cut the selection to clipboard
Ctrl + C		Copy the selection to clipboard
Ctrl + V		Insert clipboard contents
F		Find the specified text
G		Loaction x,y
Ctrl + Z		Enlarge the current view
Shift + Z		Reduce the current view
Ctrl + S		Save the active document
B		Highlight the network
Ctrl + B		Cancel highlighted network
R		Extracting ruler value as cell's parameter
Ctrl + R		Extracting ruler value as instance's parameter (First ruler as a parameter l value, Second ruler as the parameter w value)
Q		Display template background