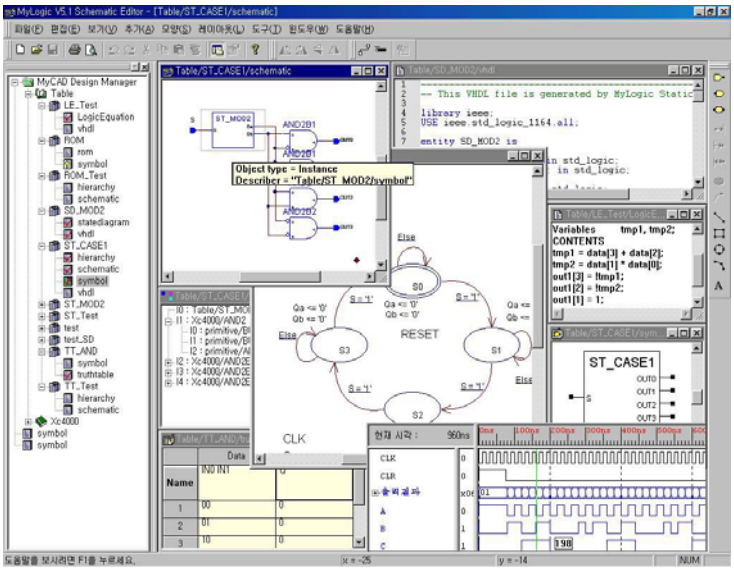
Powerful Design Solution on Windows

MyLogic Station
Schematic Editor Manual

Version 5.1



Copyright Information

SELOCO, Inc.
3F Namyang Bldg, 60-5 Bangi-dong
Songpa-gu, Seoul, 138-828 Korea
Phone: +82-2-3433-0000 Fax: +82-2-3433-0099
E-mail: support@seloco.com, sales@seloco.com
Home page: http://www.seloco.com

MyCAD, Inc.
528 E. Weddell Drive, Suite #3
Sunnyvale, California 94089, USA
Phone: 408-745-6785 Fax: 408-745-6783
Email: support@mycad.com, sales@mycad.com
Home page: http://www.mycad.com

Contexts

Introduction

The page describes the layout for pages to follow:

Pull-down menu:
* signifies a command in the Symbol Editor only.
** signifies a command which is not available at MyAnalog Station.

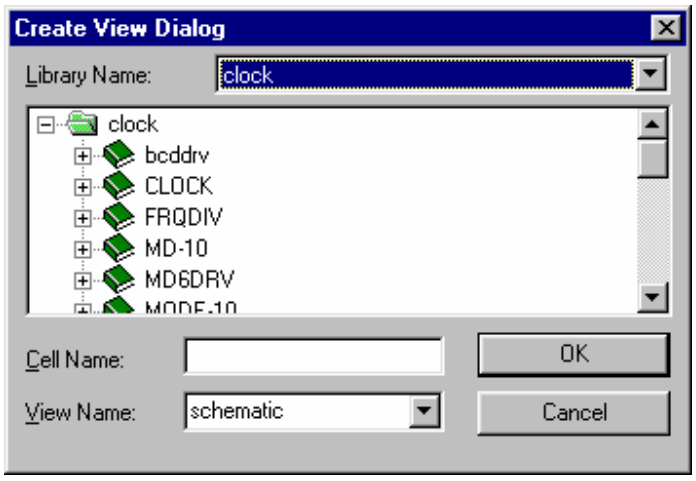
File→New…

Command Description

It creates a cell.

Description of a command

Usage & Dialog Box



Dialog box explanation

SchEd: Schematic Editor

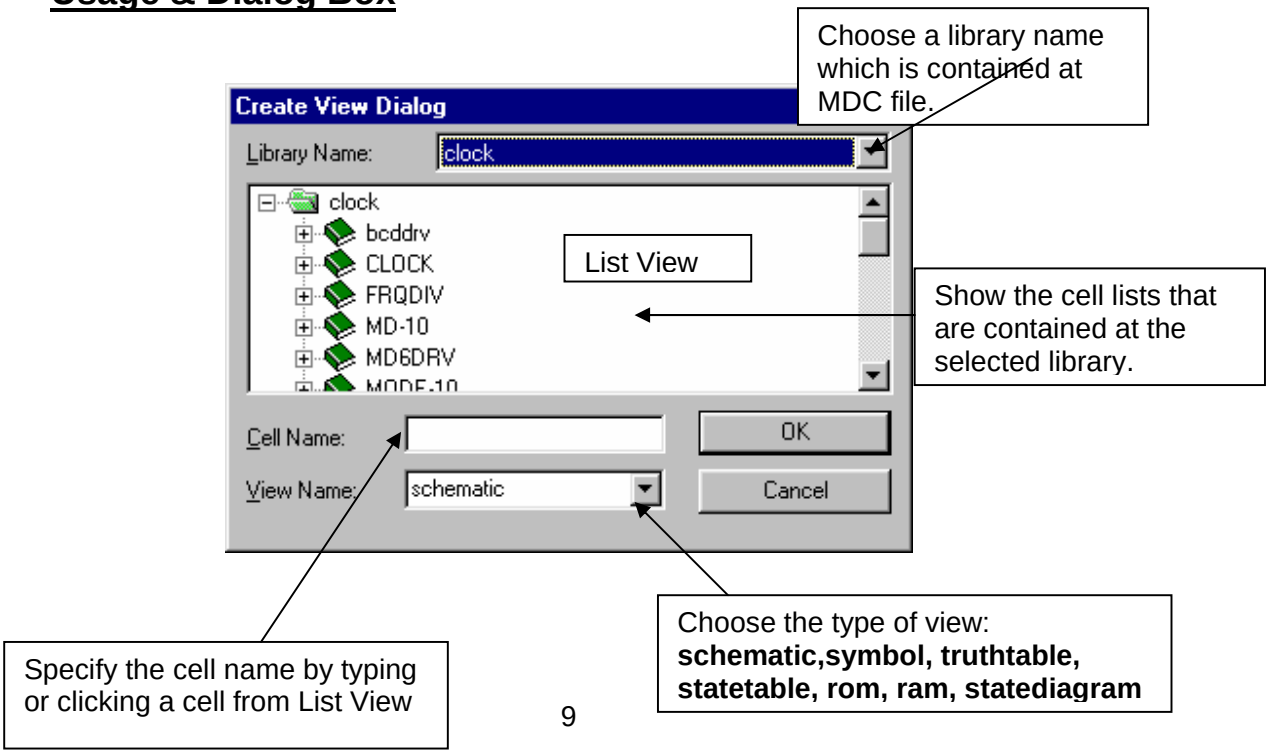
File → New…

Command Description

It creates a cell. It is activating when an MDC file was opened.

Note: MDC (MyCAD Design Context) file contains the information about library names.

Usage & Dialog Box



Library Name: If you made several working libraries that were maintained by the mdc file, you can choose one of library lists on this selection box.

List View: It shows the cell lists contained to the mdc file.

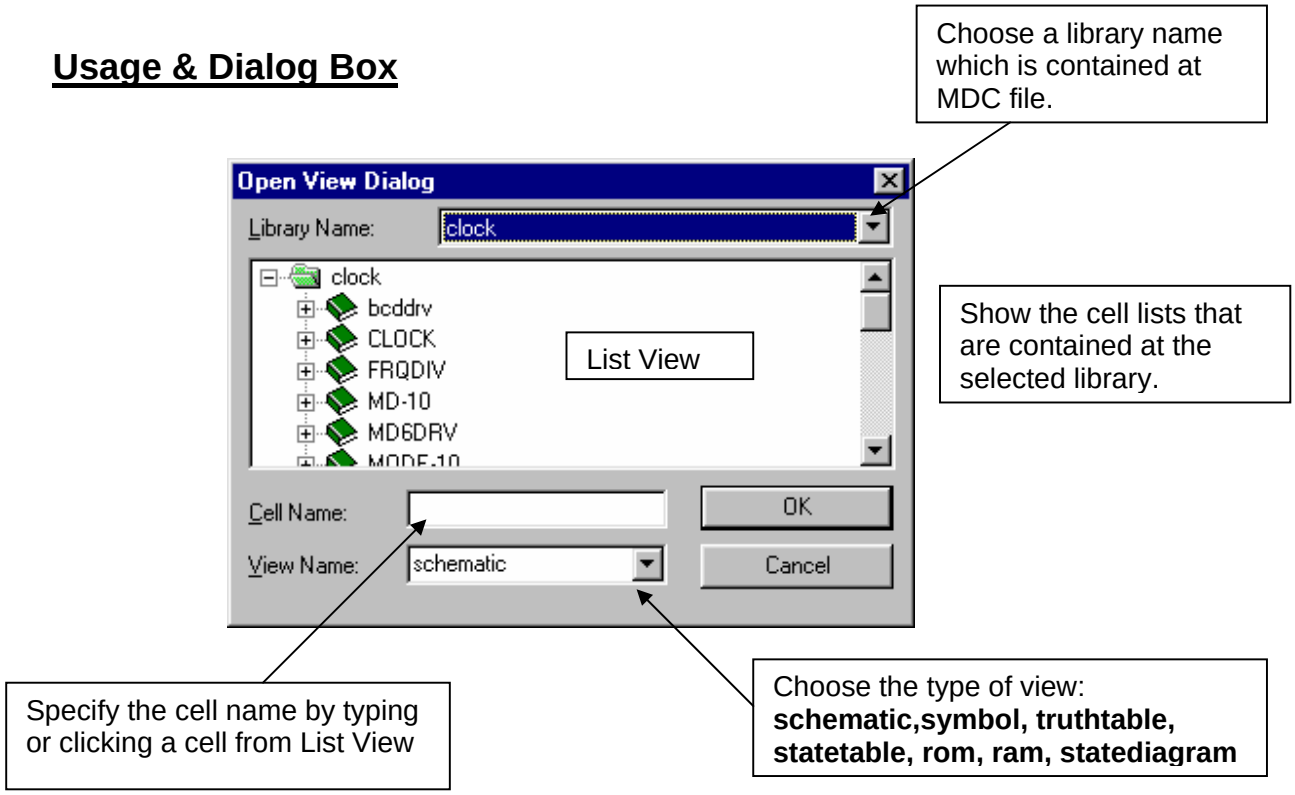
Note: The schematic view and the symbol view are only available at the analog design.

File → Open…

Command Description

It opens a cell and its view.

Usage & Dialog Box



File → Save

Command Description

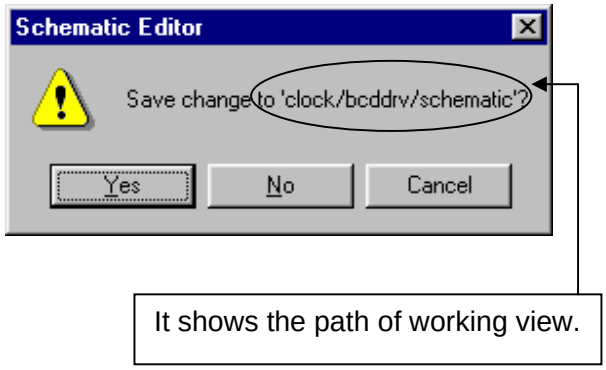
It saves the current working view.

File → Close

Command Description

It closes the current working view.

Note: SchEd checks whether the view is changed or not, before closing it. If not, you can see the following dialog box.

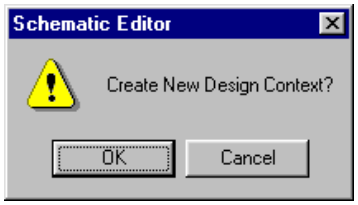


File → New Design Context…

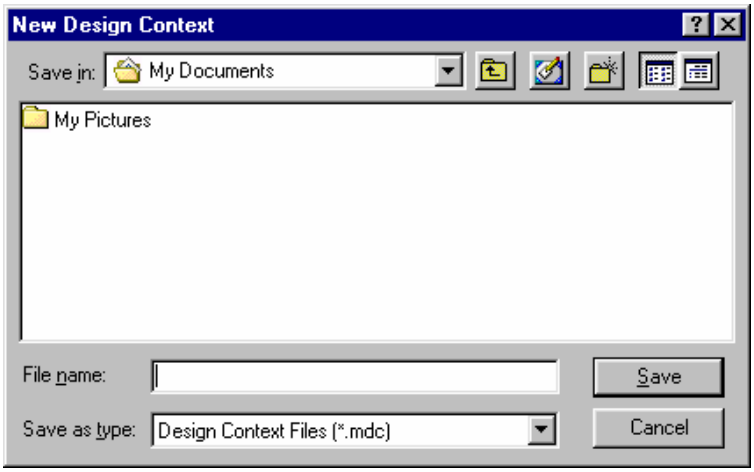
Command Description

It creates an mdc file.

Note: If an mdc file is working, then the following dialog box is appeared. Click on ***"OK"*** *button with the left mouse button.*

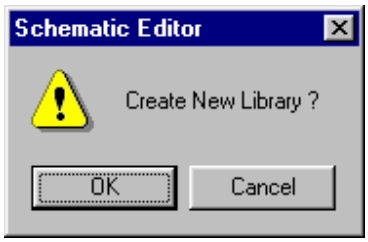


Usage & Dialog Box



File name: Move to the directory and type new mdc filename. The extension mdc does not need to be typed because it is automatically attached.

If you click on the ***"OK"*** button with the left mouse button on **New Design Context** dialog box, then the following dialog box will be shown.



Click on ***"OK"*** button with the left mouse button, then SchEd creates new library named after mdc file's name.

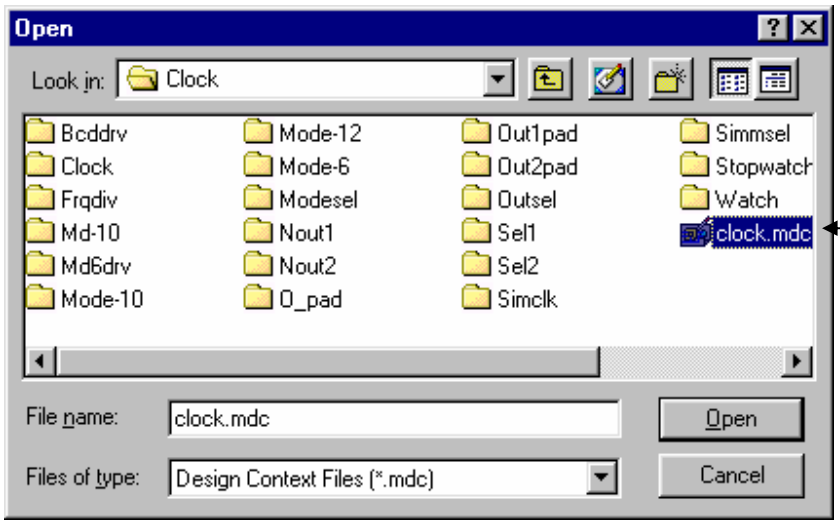
File → Open Design Context…

Command Description

It opens an mdc file.

Note: It is working an mdc file. Then, SchEd automatically closes it before opens another mdc file.

Usage & Dialog Box



Click an mdc file with the left mouse button

File → Save Design Context

Command Description

It saves the current working mdc file.

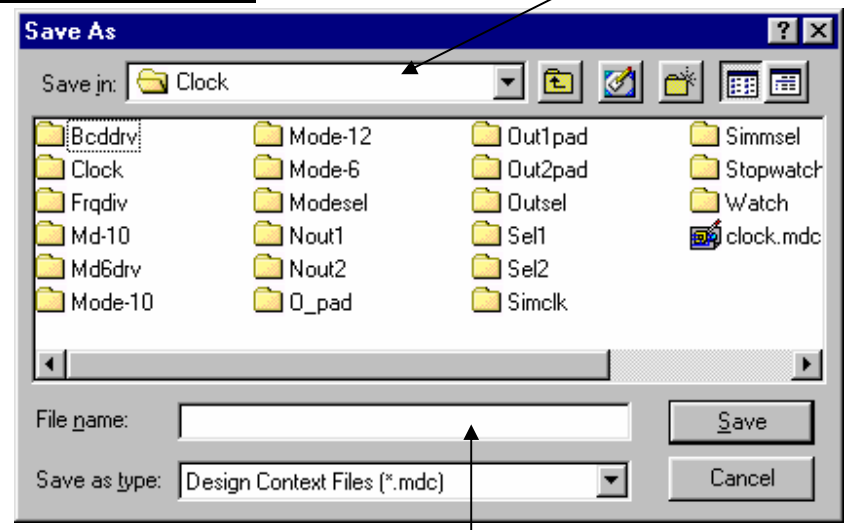
Note: When an mdc file is closed, then it is automatically saved.

File → Save As Design Context…

Command Description

It saves as the current working mdc file into another.

Usage & Dialog Box



File → Close

Command Description

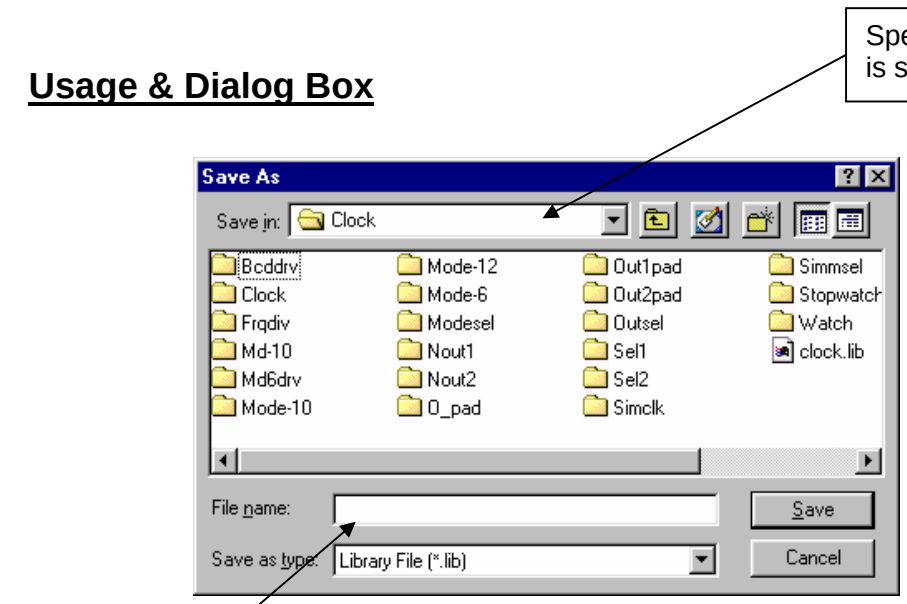
It closes the current working mdc file.

File → Create Library…

Command Description

It creates a new library.

Usage & Dialog Box



File → Add/Remove Library…

Command Description

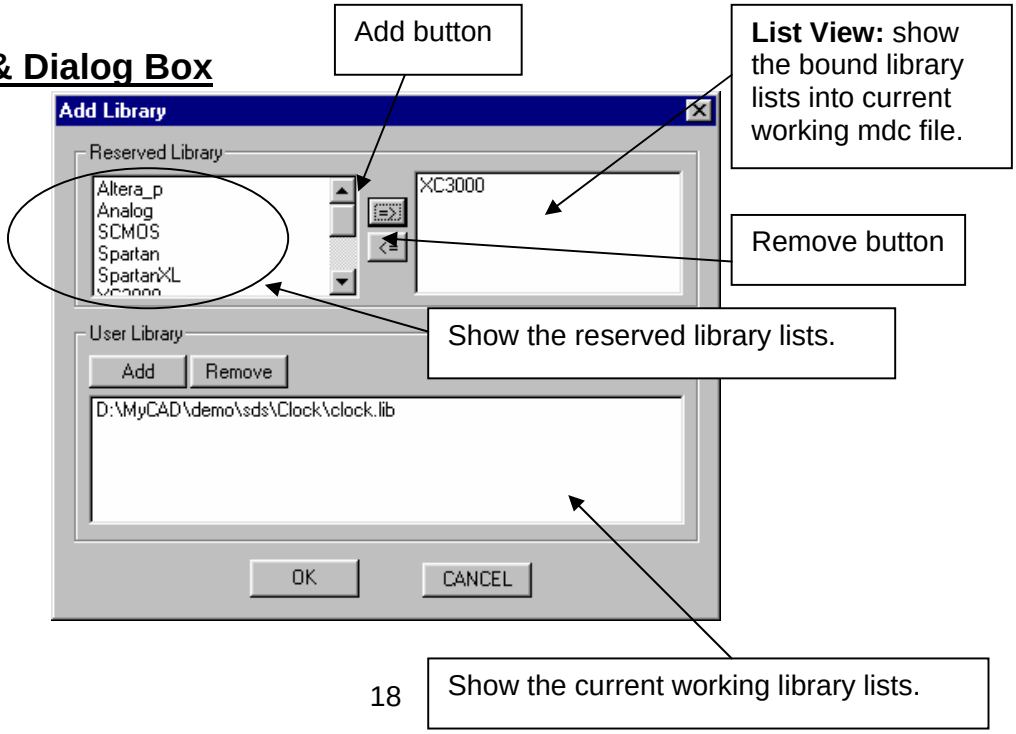
It adds the library into the current working mdc file and also removes the library from the current working mdc file.

Note: MyLogic Station provides two kinds of library parts. One part can be used for both Xilix FPGA design and Altera FPGA design, the other can be only used for drawing a schematic. And then, user library sets are also available to add/remove library.

There are Spartan XL, Spartan, and XC4000E series for Xilinx's FPGA design.

And, Analog library subsets are only available at MyAnalog Station.

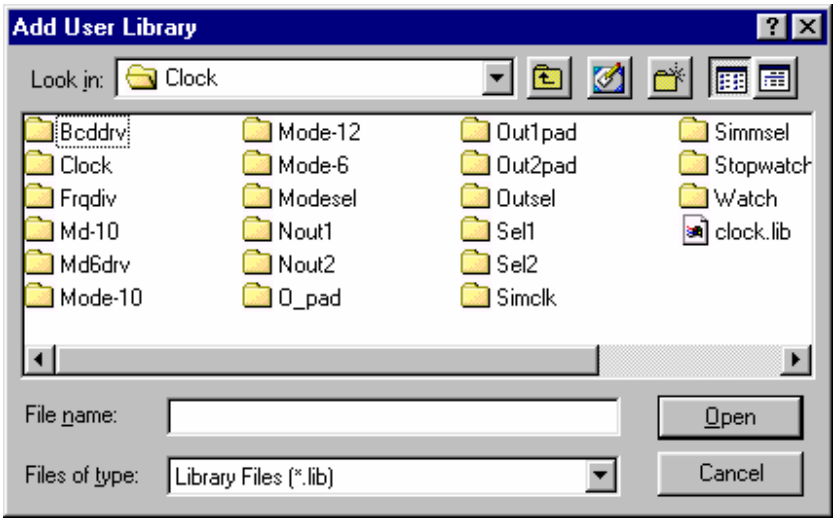
Usage & Dialog Box



To add a library into current working mdc file, choose one of library at **Reserved Library** view on **Add Library** dialog box. And click on ***"Add"*** button with the left mouse button at **Reserved Library** view.

Note: The reserved library names are added during the installation into windows registry.

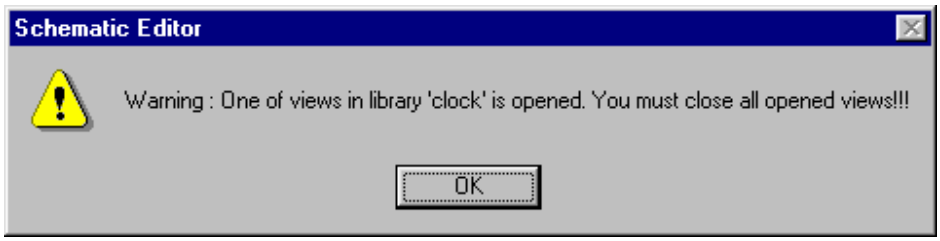
If you want to add the library which is not included at the reserved library list, click on ***"Add"*** button at **User Library** view. Then, the following dialog will be shown.



Locate the directory of the library and resister it at **Add User Library** dialog box.

To remove the library from the current working mdc file, select the library at **List View** with the left mouse button and click on ***"Remove"*** button at **Reserved Library** view.

Note: You cannot remove the current using library, then you can see the following dialog box.

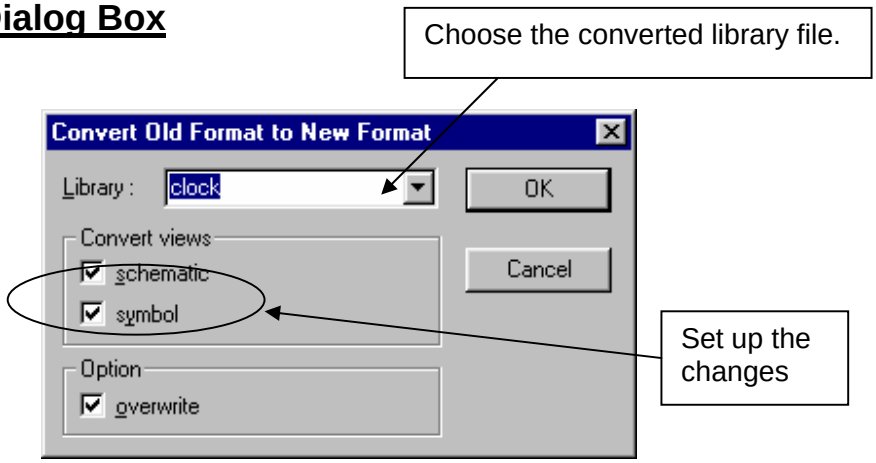


File → Convert Project…

Command Description

It converts the MyLogic V2.1 data format into V5.1 data format because there is lot of difference between them.

Usage & Dialog Box

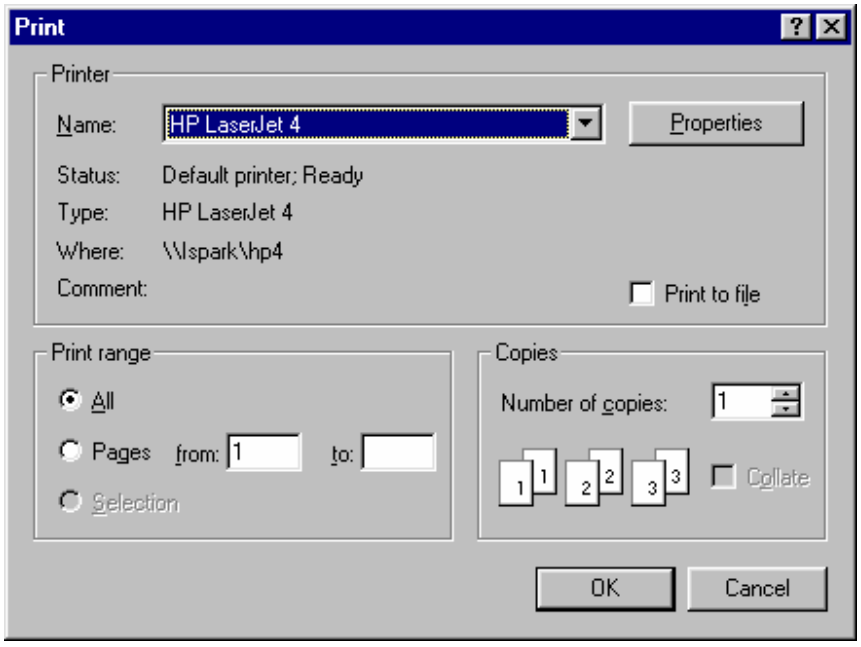


File → Print…

Command Description

It prints out the current working view.

Usage & Dialog Box

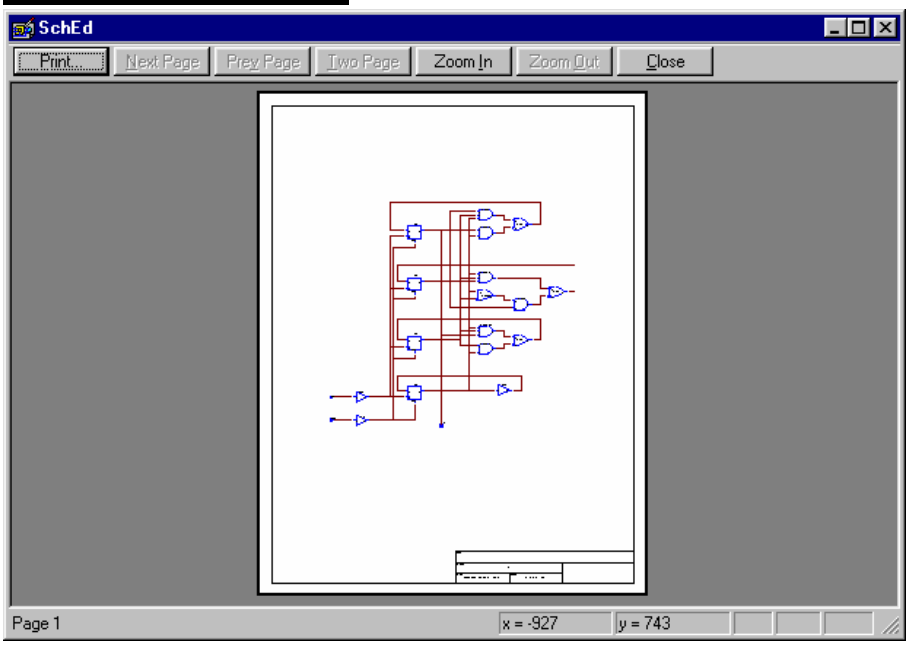


File → Print Preview

Command Description

It will preview the printing.

Usage & Dialog Box



Related Commands

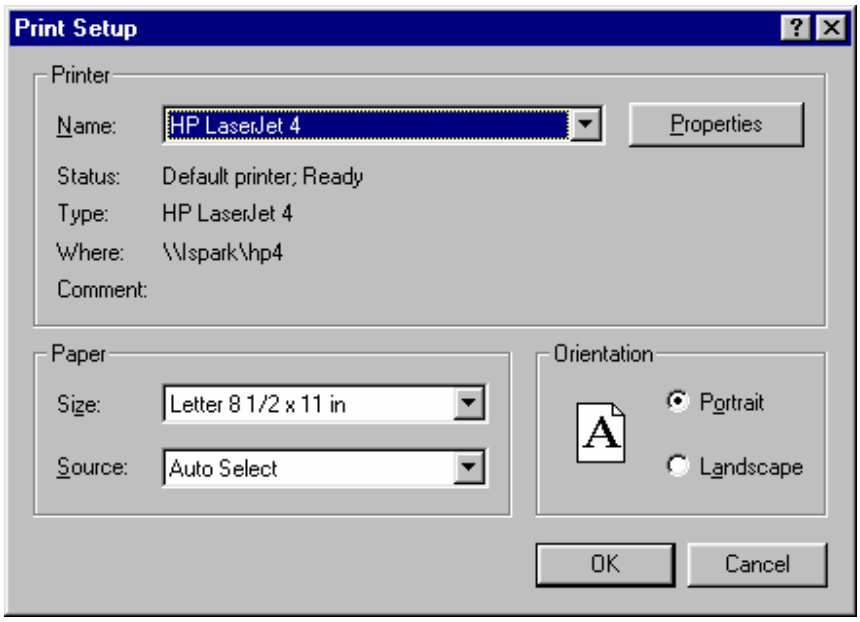
Whether to print frame or multiple pages can be configuration at ***Tools → Options… →Print (tab)*** from pull-down menu of SchEd.

File → Print Setup…

Command Description

It will setup the printer and paper.

Usage & Dialog Box

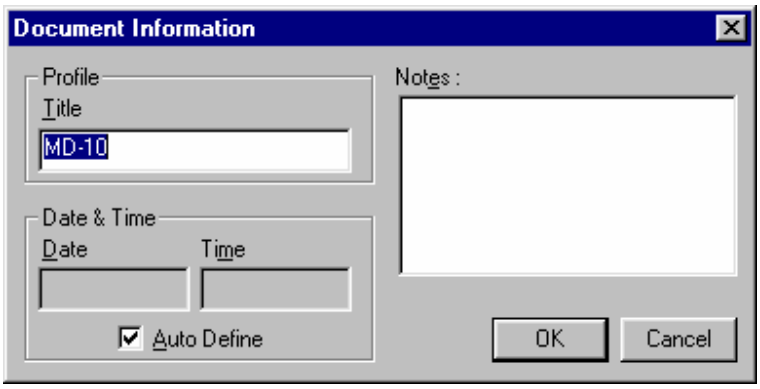


File → Document Info…

Command Description

It will bring up the Document Information dialog box.

Usage & Dialog Box



Title: It shows the name of current working view.

Auto Define: It setup whether date and time are printed automatically or not.

Notes: You can describe the relevant information.

File → Exit

Command Description

It closes the SchEd.

Edit → Undo

Command Description

It undoes the editing. It is only activate when the object on the Schematic view or the Symbol view was edited.

Edit → Redo

Command Description

It redoes the editing which was undone.

Edit → Cut

Command Description

It wipes out the selected object and put it into the clipboard.

Note: No other application except SchEd can paste the object in the clipboard.

Edit → Copy

Command Description

It copies the selected object to the clipboard.

Note: No other application except SchEd can paste the object in the clipboard.

Edit → Paste

Command Description

It pastes the object which has been copied or cut to the clipboard.

Note: The object to be pasted will appear in the middle of the active window.

Edit → Delete

Command Description

It deletes the selected object.

Edit → Select All

Command Description

It selects all the objects.

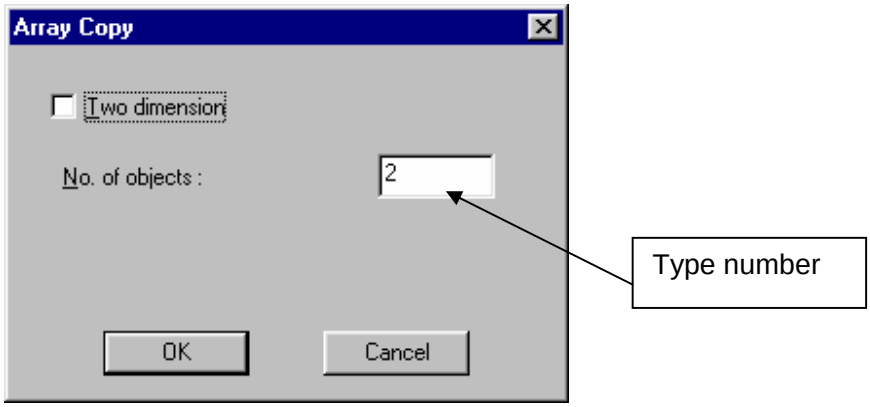
Edit → Array Copy…

Command Description

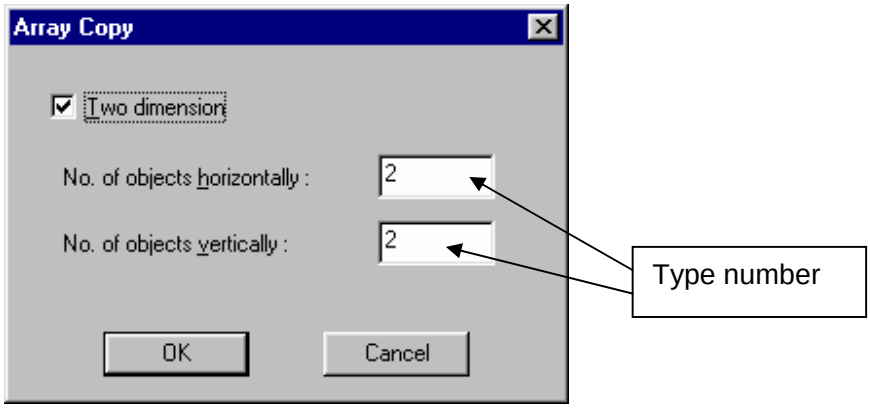
It will copy the selected object array based on the specified number on **No. of objects** field.

Usage & Dialog Box

Start by selecting objects from current working view, and selecting ***Edit → Array Copy…*** from pull-down menu of **SchEd**. Then, the following dialog box will be shown. Type the number at **No. of object** field.



Two dimension: It checks on when you want to copy two-dimensional array. And you check on this box. Then, the following dialog box will be shown.



No. of objects horizontally: You can specify the number of horizontal array.

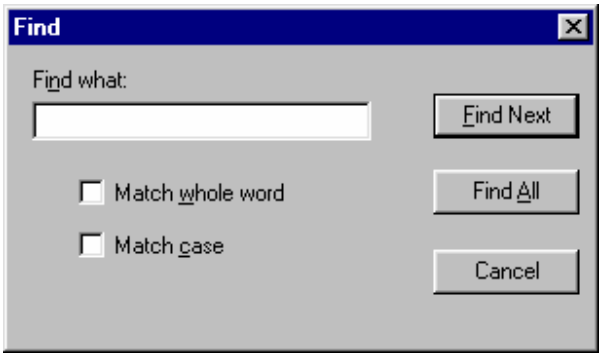
No. of objects vertically: You can specify the number of vertical array.

Edit → Find…

Command Description

This command enables for you to find a symbol, text, and port name at SchEd. It will locate the symbol, text, and port name on the working window.

Usage & Dialog Box



Edit → Find Next

Command Description

It will find a symbol, text, and port name at the different place after finding it by ***Edit → Find*** command.

View → Redraw

Command Description

It will redraw the screen and refresh the image of the current working window.

View → Whole Page

Command Description

It will display the whole fit in the current working window.

View → ZoomIn

Command Description

It will zoom in by 2.

View → ZoomOut

Command Description

It will zoom out by 2.

View → Design Manager

Command Description

It toggles the display of Design Manger on SchEd.

View → Status

Command Description

It toggles the display of Status bar on SchEd.

View → Properties…

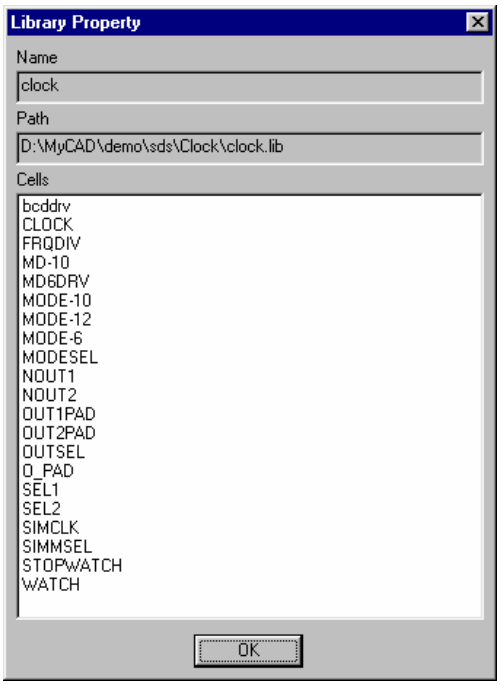
Command Description

It will show the properties of selected object.

Usage & Dialog Box

Library Property

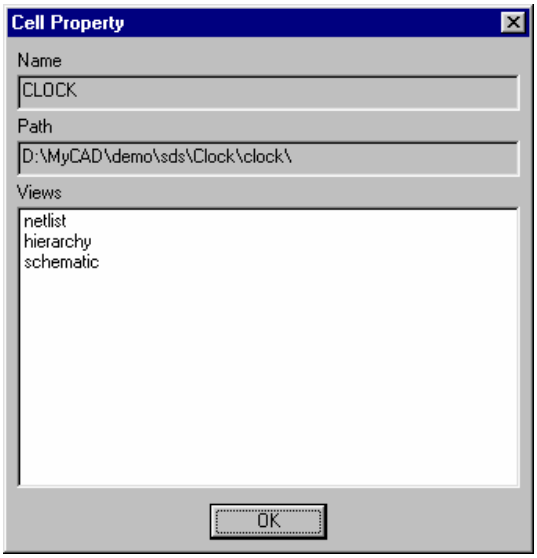
Select a library on Design Manager of SchEd with the left mouse button, and choose ***View → Properties…*** from pull-down menu of SchEd. Then, the following dialog box will be shown.



You can see the cell lists that are contained at the selected library and its path.

Cell Property

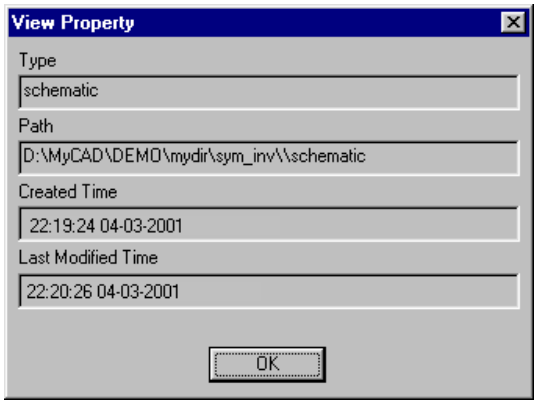
Select a cell on Design Manager of SchEd with the left mouse button, and choose ***View → Properties…*** from pull-down menu of SchEd. Then, the following will be shown.



It just displays the view lists contained to the cell.

View Property

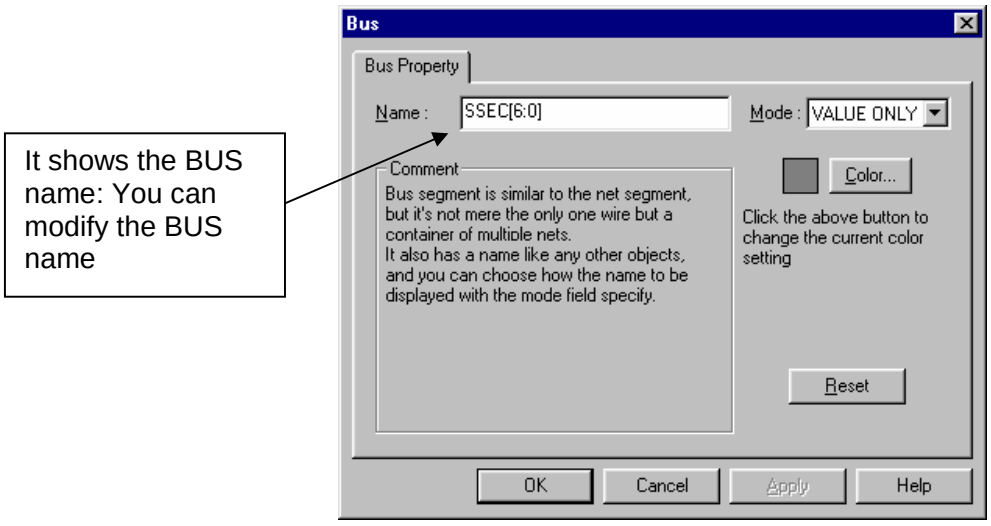
Select a view on Design Manager of SchEd with the left mouse button, and choose ***View → Properties…*** from pull-down menu of SchEd. Then, the following will be shown.



It displays the created time and the last modified time of the view.

BUS Property

Select a BUS on the Schematic View, and choose ***View → Properties…*** from pull-down menu of SchEd. Then, the following dialog box will be shown.



Mode: It controls the display mode.

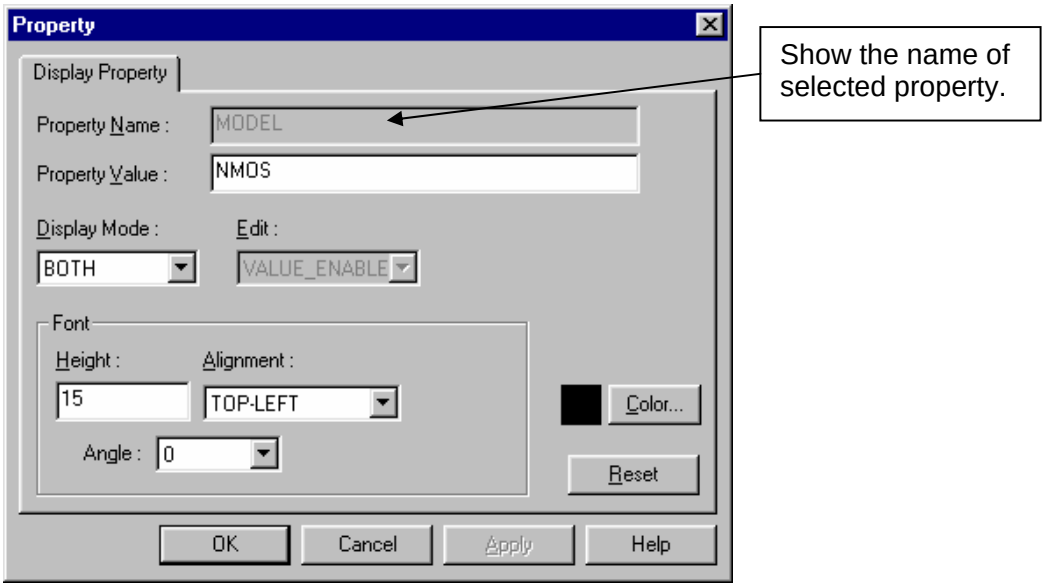
***OFF**: It does not display both name and value of BUS on the Schematic View.*

***NAME ONLY**: It only displays the name of BUS on the Schematic View.*

***BOTH**: It displays both name and value of BUS on the Schematic View.*

Display Property

If you double click the property of object on whether the **Schematic View** or **Symbol View**, then, the **Property** dialog box will be shown.



Property Name: It shows the name of selected property. And, it is only activate when the object of symbol is selected at the **Symbol View**.

Property Value: It shows the value of selected property. And it is modifiable.

Display Mode: It controls the display mode of selected property.

***OFF**: It does not display both name and value of property.*

***NAME ONLY**: It only displays the name of property.*

***BOTH**: It displays both name and value of property.*

Edit: It controls the editable of value of property.

***DISABLE**: It makes all unselectable.*

***VALUE_ENABLE**: It makes the value of property editable.*

***ALL_ENABLE**: It makes all editable.*

Height: It setup the height of text of property.

Alignment: It sets up the alignment of property.

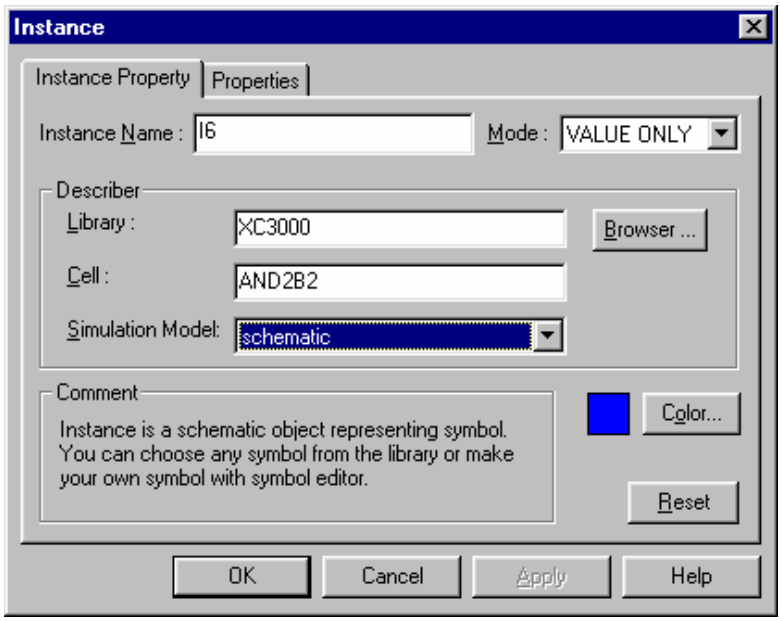
Angle: It rotates the property by degree.

Color: It changes the display color of property into another.

*Note: In general, the **Edit** mode is not activating except when you edit the symbol instance. However, the symbol instance becomes no modifiable at the Schematic View.*

Instance Property

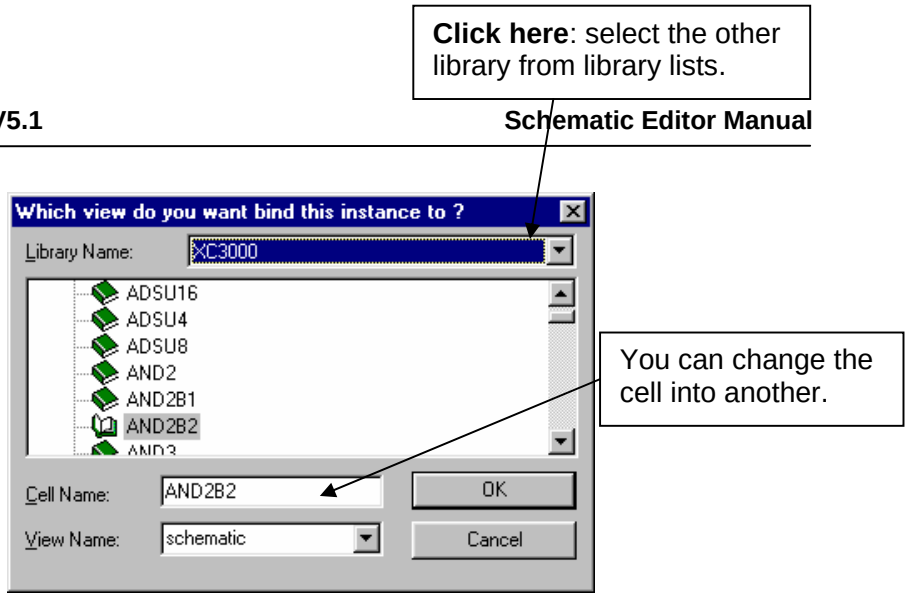
Select the instance at the **Schematic View**, and choose ***View → Properties…*** from pull-down of SchEd. Then, the **Instance** dialog box will be shown.



Instance Property tab:

Instance Name: It shows the instance name.

Library: It shows the information of library of instance. You can change the instance into another library by clicking "***Browser***" button. Then, the following dialog box will be shown.



Cell: You can change the current instance into another at the current library by clicking "***Browser***" button. Then, the above dialog box will be shown.

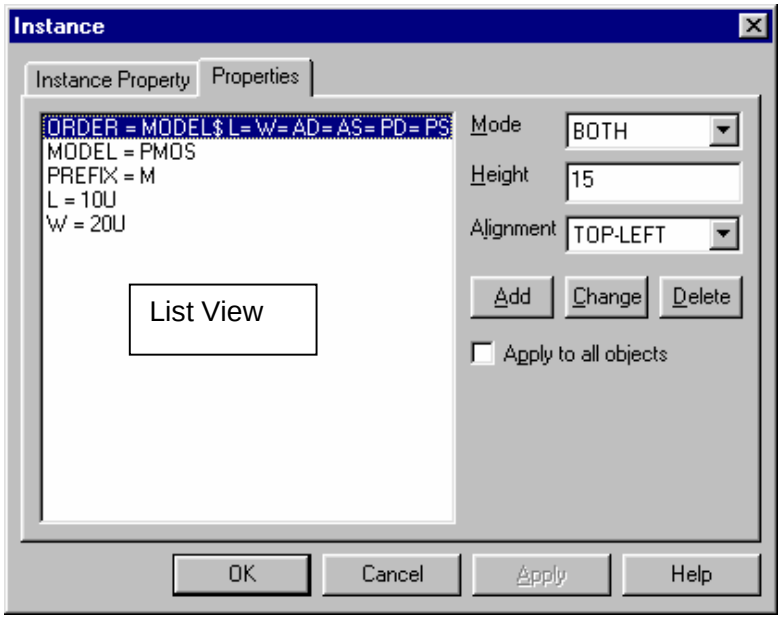
Simulation Model: It displays the simulation model of selected instance. You can change the simulation model into another.

Properties tab:

This tab is useful at a circuit design. In general, you have better avoid using this tab on the logic design.

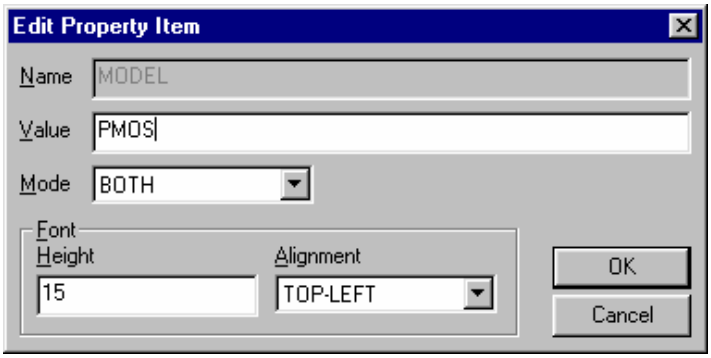
The following diagram is showing the properties of **NMOS** instance.

The format of property is : "NAME=VALUE"



*** Change Property**

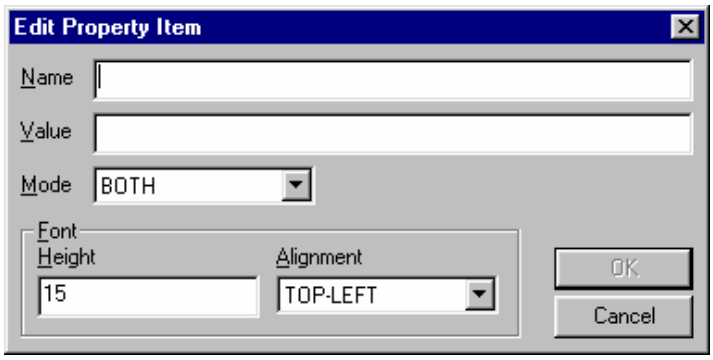
Start it by selecting one from List View. And click *"Change"* button on the **Instance** dialog box with the left mouse button. Then, the following dialog box will be shown.



If the selected name is given by default, then you could not change it. In that case, you only change the value.

*** Add Property**

Click "***Add***" button on the **Instance** dialog box with the left mouse button. Then, the following dialog box will be shown.



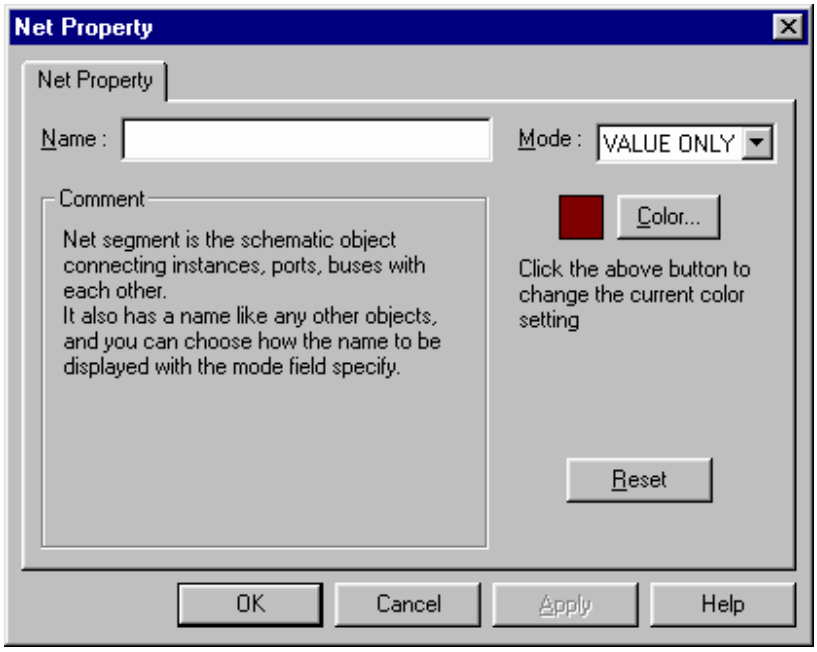
Specify the name at the "***Name***" text input box. And Specify the value at the "***Value***" text input box.

*** Delete Property**

Select the property that you want to delete on the **List View** with the left mouse button. And, click "***Delete***" button on the **Instance** dialog box. Then, selected property will be deleted.

Net Property

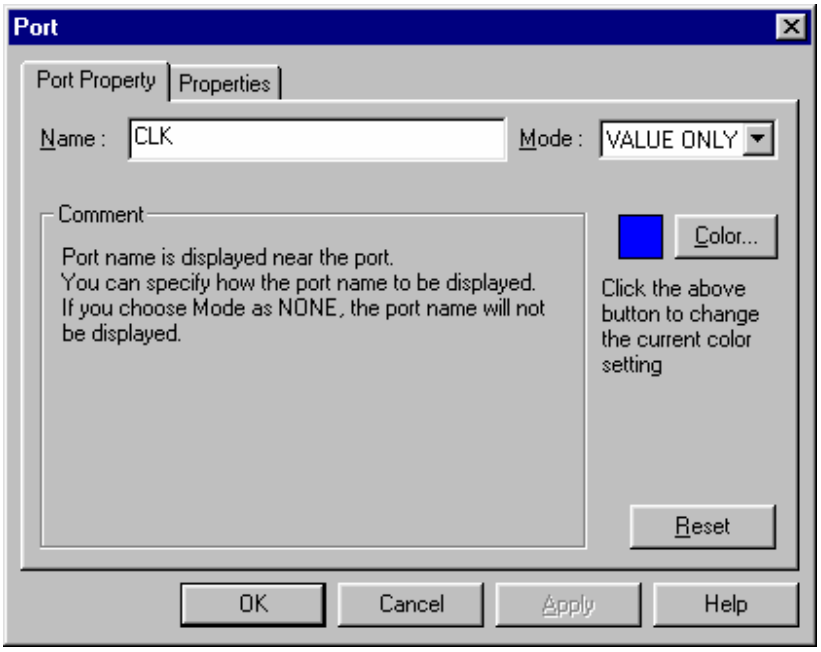
Select a net on the **Schematic View** with the left mouse button. And, choose ***View→ Properties…*** from pull-down menu of SchEd. Then, the **Net Property** dialog box will be shown.



Name: You can specify the local name of net. If you do not assign the local name of net, SchEd will automatically assign local name into each net.

Port Property

Select the port on the **Schematic View** with the left mouse button. And, choose ***View→Properties…*** from pull-down menu of SchEd. Then, the **Port** dialog box will be shown.



Name: You can specify the port name. If you do not assign the port name, SchEd will automatically assign port name.

Add → Port → Input

Command Description

It adds the input port.

Usage & Dialog Box

The input port will locate the middle of the **Schematic View**. Move it into properly location by dragging.

Add → Port →Output

Command Description

It adds the output port.

Usage & Dialog Box

The output port will locate the middle of the **Schematic View**. Move it into properly location by dragging.

Add → Port → InOut

Command Description

It adds the bi-directional port.

Usage & Dialog box

The bi-directional port will locate the middle of the **Schematic View**. Move it into properly location by dragging.

Add → Figure → Line *

Command Description

It adds the line object.

Usage & Dialog Box

The line object will locate the middle of **Symbol View**. Handles are shown at both end of the line. The line is positioned and moving the handle changes the length.

Add → Figure → Rectangle*

Command Description

It adds the rectangle object.

Usage & Dialog Box

The rectangle object will locate in the middle of **Symbol View**. The four handles are displayed. By poisoning each handle to the desired position, the rectangle is formed with positioned corners.

Add → Figure → Circle *

Command Description

It adds the circle object

Usage & Dialog Box

The circle object will locate in the middle of Symbol View. The four handles are displayed. The size of circle is determined by moving four handles.

Add → Figure → Arc *

Command Description

It adds the arc object.

Usage & Dialog Box

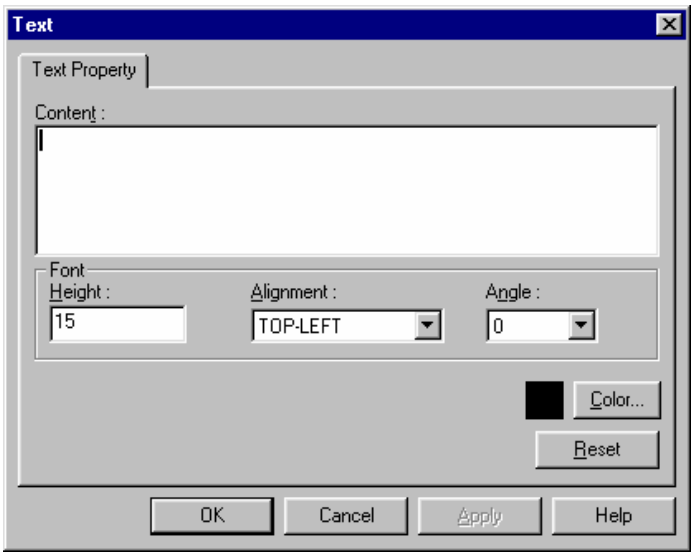
The three handles are displayed. Positioning the two end points to the desired points, and moving the middle handle to the desired are shape. The position and form of the arc is determined by the positions of three handles.

Add → Figure → Text *

Command Description

It adds the text object.

Usage & Dialog Box



Type the text at the **Content** input box. Click "***OK***" button with left mouse button. Then, the text object will locate in the middle of **Symbol View**. Move it into properly location.

Add → Net Mode

Command Description

It toggles the net mode.

Usage & Dialog Box

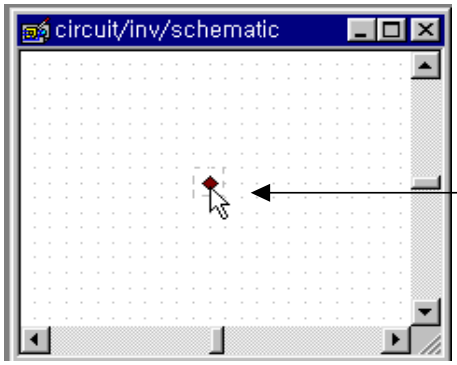
In the net mode, drawing wire is only possible. To get out of net mode, double click the left mouse button or choose ***Add → Net*** Mode from pull-down menu of SchEd.

Wiring (It does not include in the pull-down menu of SchEd)

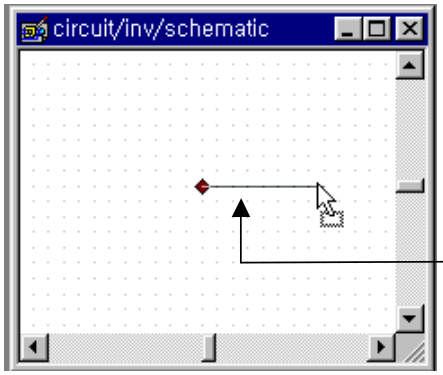
Wiring is used only in the Schematic View.

The basic of drawing a wire

Click with the mouse left button on working area of the Schematic View. The red-dot is shown as in the following diagram. (Only if you use the default color of SchEd.)
Drag the muse cursor to the end point, and then drop. You can draw wire to any direction from the red-dot point.



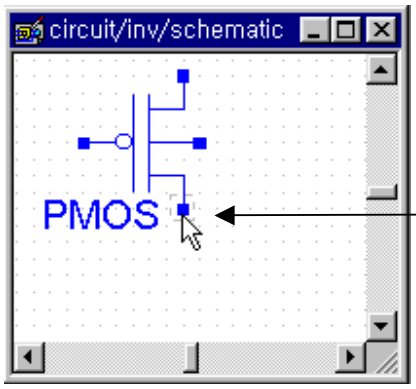
Click on the working area of the Schematic View with the left mouse button:
the bounding box is appeared



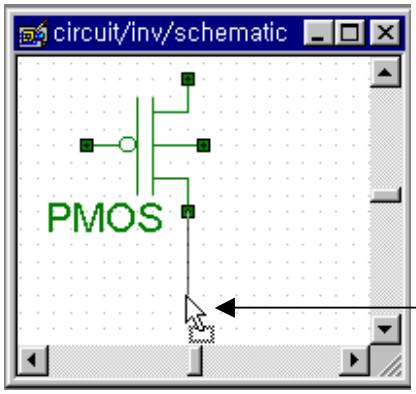
Draw wire to the desired position by dragging the mouse:
The mouse cursor shape is changed during dragging.

Another way of wiring (from the symbol object's end point)

Click the left mouse button on the net point of a symbol object, then drag the mouse to the desired end point of the net and drop.



Place the mouse cursor on the net point:
The bounding box of the net point is appeared

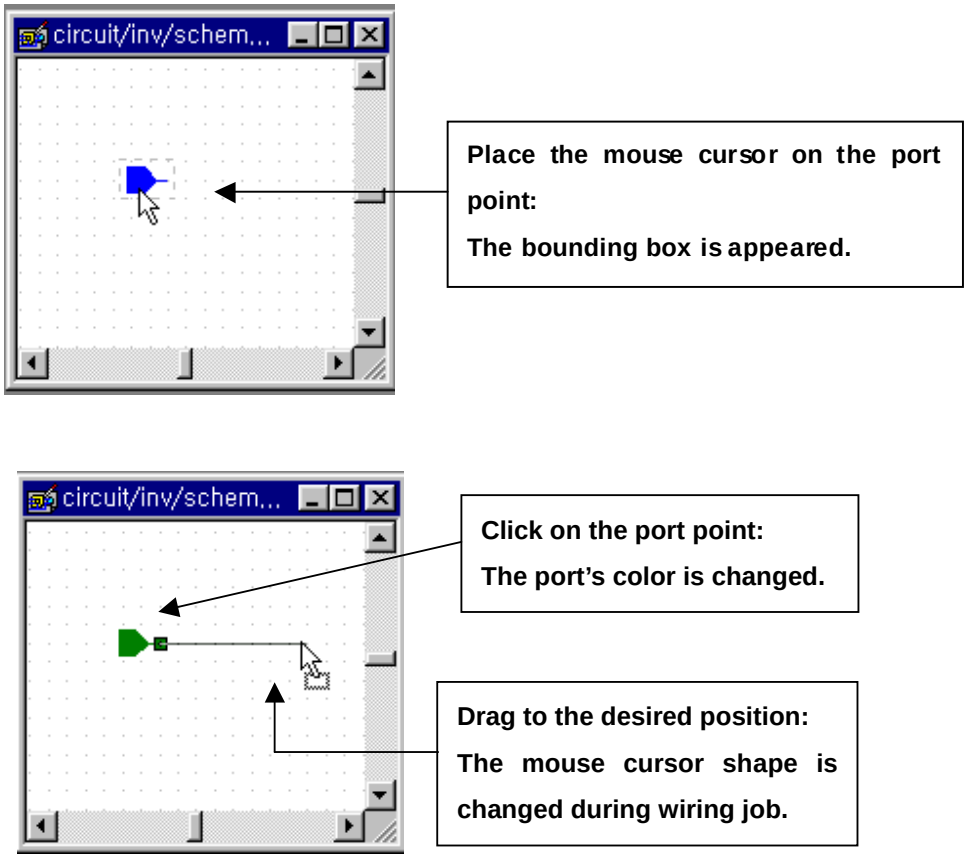


Click on the net point of symbol, symbol's color is changed to green

Drag to the desired position:
the mouse cursor shape is changed during wiring job.

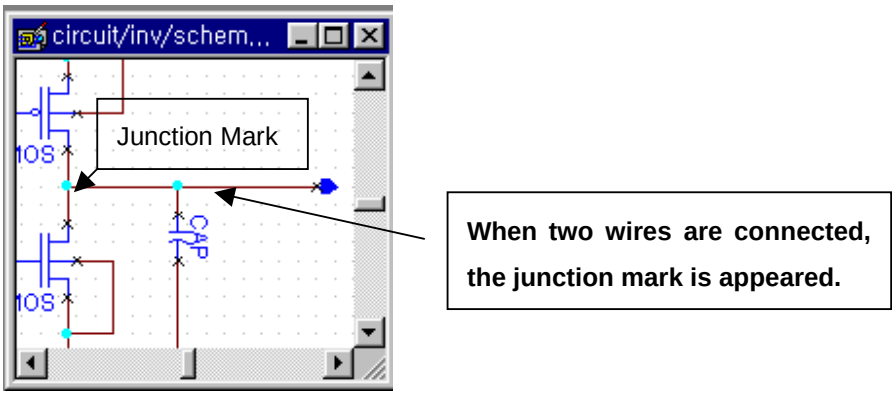
Another way of wiring (from the port point)

Click the left mouse button on the port point. Then drag the cursor to the desired end point of the net and drop.



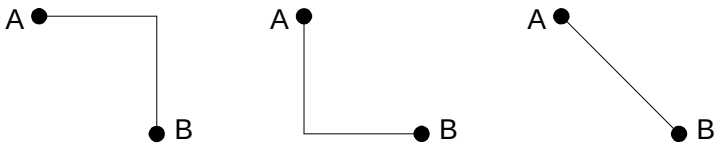
Junction

If two wires are connected, then the junction mark is appeared. You distinguish it whether they are connected or not.



Wire Mode

There are three kinds of wire mode, horizontal first, vertical first, and diagonal. Click Shift key or Ctrl key on the keyboard, then three wire mode displays.



Horizontal first (Shift key)

Vertical first (Shift key)

Diagonal (Ctrl key)

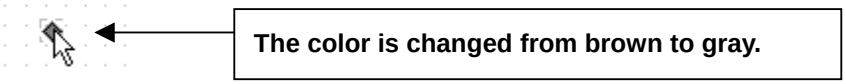
Add → BUS

Command Description

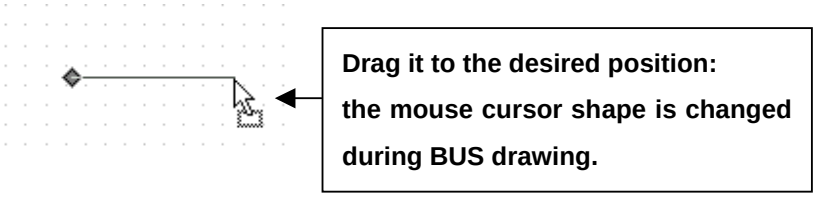
It toggles the BUS mode.

Usage & Dialog Box

Choose ***Add → Bus*** from pull-down menu of SchEd. The color of port changes from brown to gray.



Drag it to the desired position and then drop the end point.



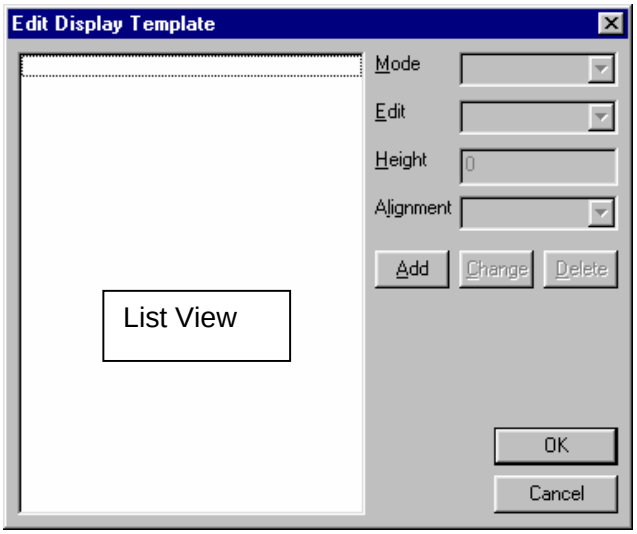
*Note: In the BUS mode, BUS drawing is only possible. To get out of the BUS mode, double click the left mouse button or choose **Add → BUS** from pull-down menu of SchEd.*

Add → Property Template…*

Command Description

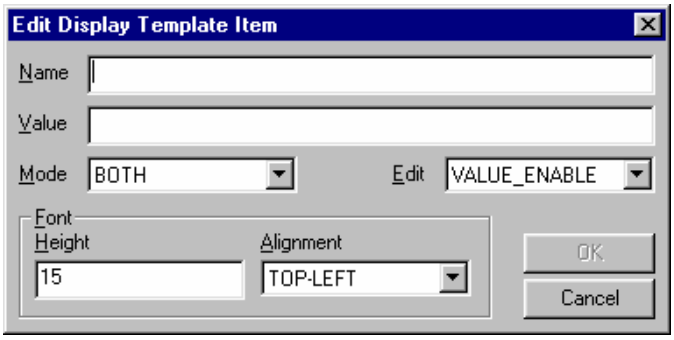
It adds the property into symbol object.

Usage & Dialog Box



Add property

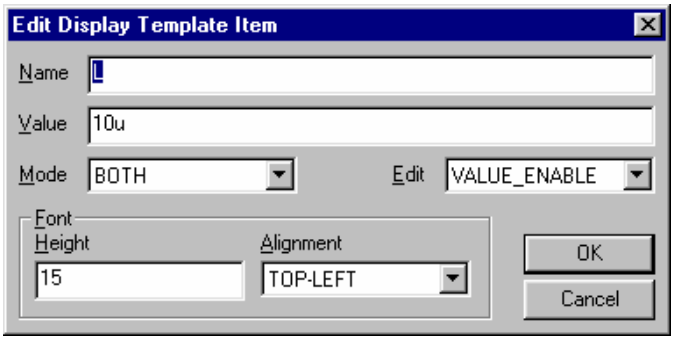
Click "*Add*" button on the **Edit Display Template** box with the left mouse button. Then, the **Edit Display Template Item** dialog box will be shown.



Specify the name of property at the "**Name"** text input box by typing. And, specify the value of name of property at the "**Value"** text input box by typing.

Change property

Select one of property on the **List View** of **Edit Display Template** box, and click Change button with the left mouse button. Then, the **Edit Display Template Item** dialog box will be shown.



Change the name or value of property, and click "***OK"*** button with the left mouse button.

Shape → Rotate Left

Command Description

It rotates the object to the left by 90 degree.

Shape → Rotate Right

Command Description

It rotates the object to the right by 90 degree.

Shape → Flip Vertical

Command Description

It flips vertically the object.

Shape → Flip Horizontal

Command Description

It flips horizontally the object.

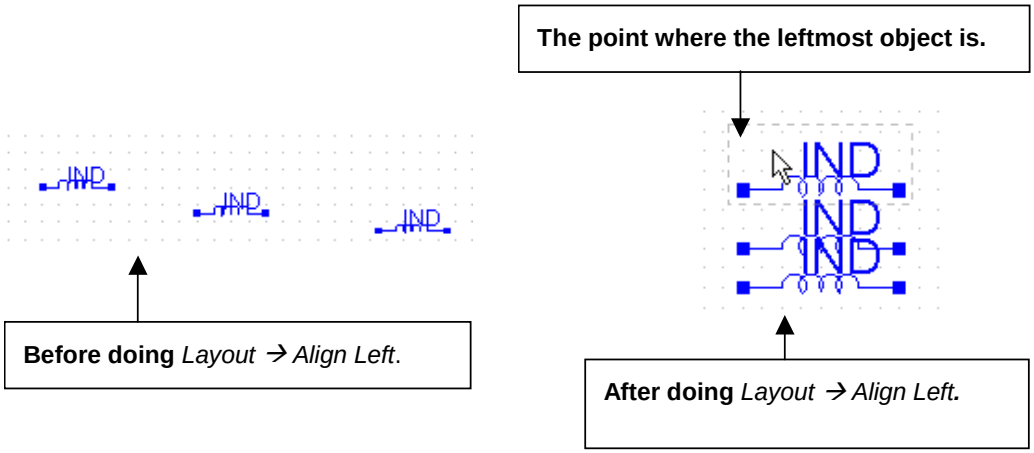
Layout → Align Left

Command Description

It aligns the selected objects to the leftmost object's position.

Usage & Dialog Box

Select objects. And, choose ***Layout → Align Left*** from pull-down menu of SchEd. Then, the selected objects are aligned to the leftmost object's position.



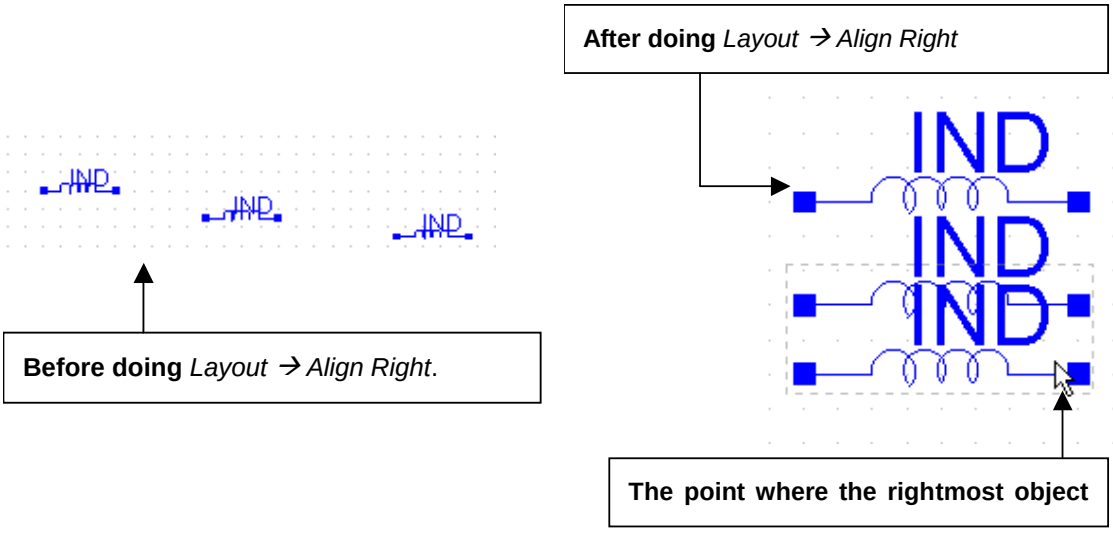
Layout → Align Right

Command Description

It aligns the selected objects to the rightmost object's position.

Usage & Dialog Box

Select objects. And, choose ***Layout → Align Right*** from pull-down menu of SchEd. Then, the selected objects are aligned to the rightmost object's position.



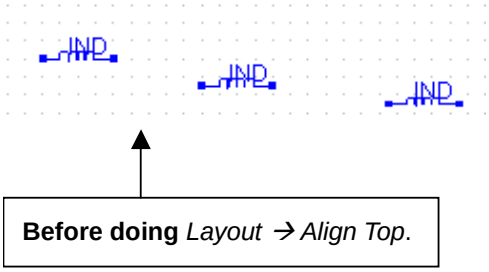
Layout → Align Top

Command Description

It aligns the selected objects to the uppermost object's position.

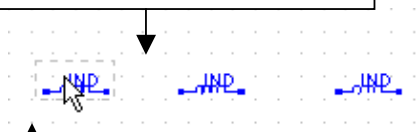
Usage & Dialog Box

Select objects. And, choose ***Layout → Align Top*** from pull-down menu of SchEd. Then, the selected objects are aligned to the uppermost object's position.



Before doing *Layout → Align Top*.

After doing *Layout → Align Top*.



The point where the uppermost object is.

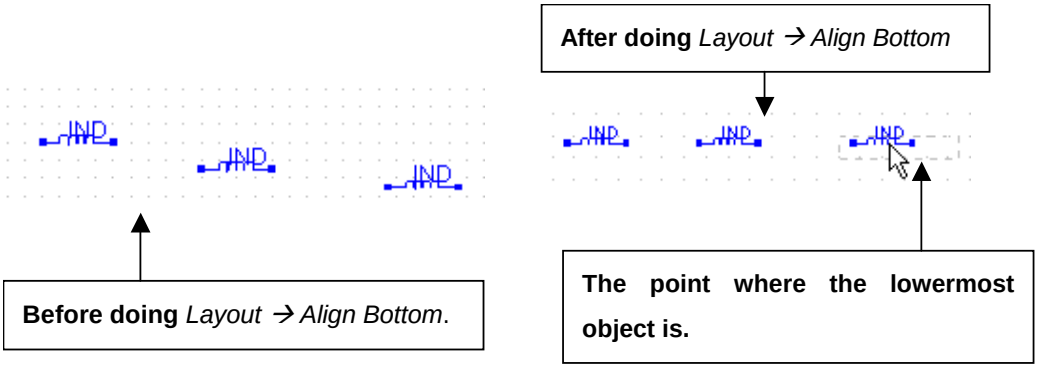
Layout → Align Bottom

Command Description

It aligns the selected objects to the lowermost object's position.

Usage & Dialog Box

Select objects. And, choose ***Layout → Align Bottom*** from pull-down menu of SchEd. Then, the selected objects are aligned to the lowermost object's position.



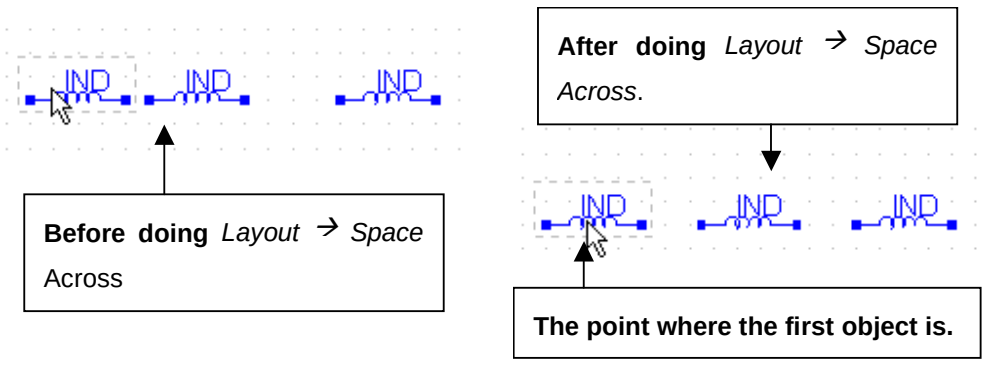
Layout → Space Across

Command Description

It makes same space between right and left of selected objects.

Usage & Dialog Box

Select objects more than three. And choose ***Layout → Space Across*** from pull-down menu of SchEd. Selected objects are arranged so that spacing between them becomes the same. The space between objects becomes even in between the leftmost object and the rightmost object.



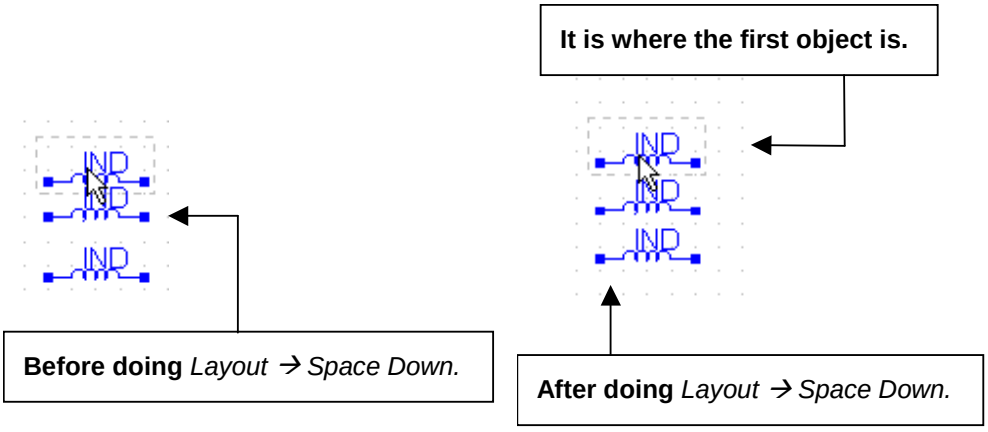
Layout → Space Down

Command Description

It makes same space between top and bottom of selected objects.

Usage & Dialog Box

Select objects more than three. And choose ***Layout → Space Down*** from pull-down menu of SchEd. Selected objects are arranged so that spacing between them becomes the same.

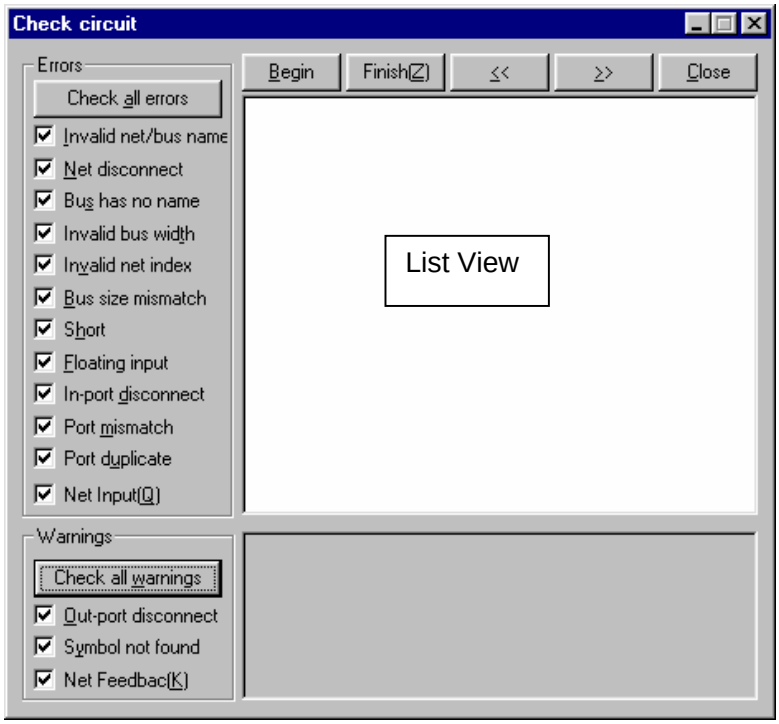


Tools → Check Circuit…

Command Description

It brings up the Check Circuit dialog box. It is only activating when the Schematic View is opened.

Usage & Dialog Box



Start by clicking ***Checks all errors*** button with the left mouse button on the **Errors View** and if you want to see the warning message, click ***Warnings*** button wit the left mouse button on the **Warnings View**. And, click ***Start*** button with the left mouse button on the **Check Circuits** dialog box. Then, the error messages will be shown at the **List View** if there is error on your schematic design. If you click the error message with the left mouse button on the **List View**, **SchEd** will display the location of selected error message into the middle on the **Schematic View**.

Error Conditions

Invalid net/bus name: it checks the difference between net name and BUS name

Net disconnect: it checks whether there is floating net or not.

Bus has no name: it checks whether there is the name of BUS or not.

Invalid bus width: it checks the difference of bit-number between connected two BUS signals.

Invalid net index: it checks the index of BUS.

Bus size mismatch: it checks the difference between the bit-number of BUS and the number of signals that were connected to the BUS.

Short: it checks the short at the schematic.

Floating input: it checks the name of port whether it has its name or not.

In-port disconnect: it checks the name of symbol port whether it has its name or not.

Port mismatching: it checks the name of port between symbol and schematic.

Port duplicate: it checks the name of port whether they are using same name or not.

Net input: it checks whether the name of net is assigned properly or not.

Warning Conditions

It is useful if you want to check the latent error before implementing your schematic through Hardware System. Sometimes, the schematic will be failed to implement even though there is no error on the schematic.

Out-port disconnect: it checks whether the out port has its mapping information for implementing Hardware System or not.

Symbol not found: it checks whether the instance has its symbol or not.

Check Feed Back: it checks whether the net has its feedback information for back-annotation or not.

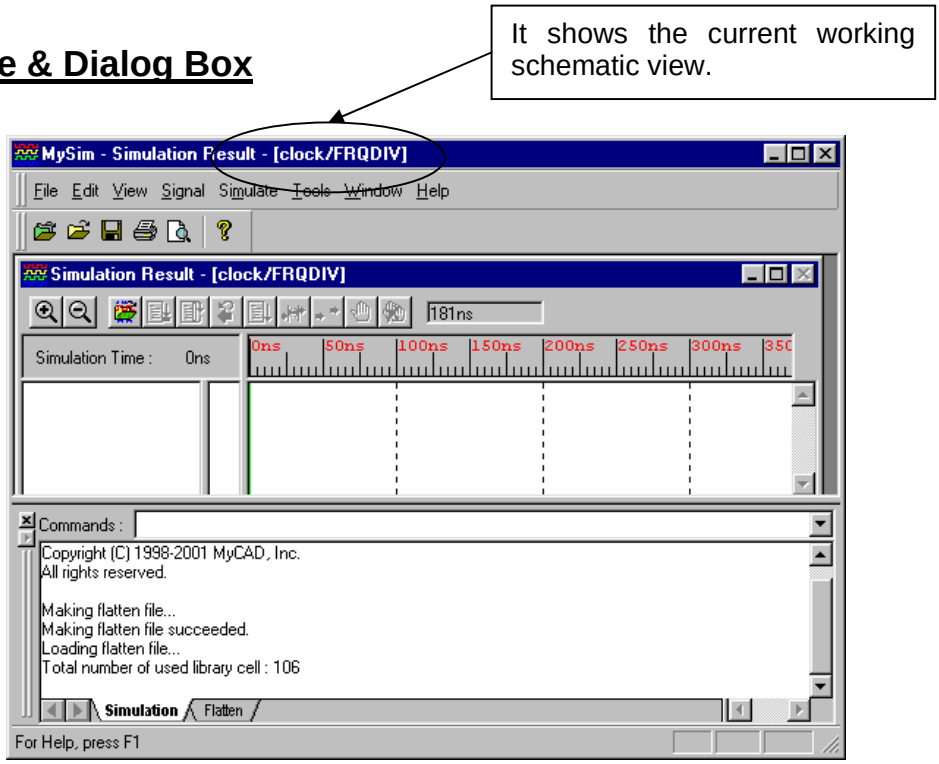
Tools → Simulator

Command Description

It invokes the MySim, logic simulator.

Note: If you want to simulate the circuit level schematic, you should change the path of simulator into the location of SPICE simulation program. You can change by clicking Tools→Options…→Path (tab) *from pull-down menu of SchEd.*

Usage & Dialog Box

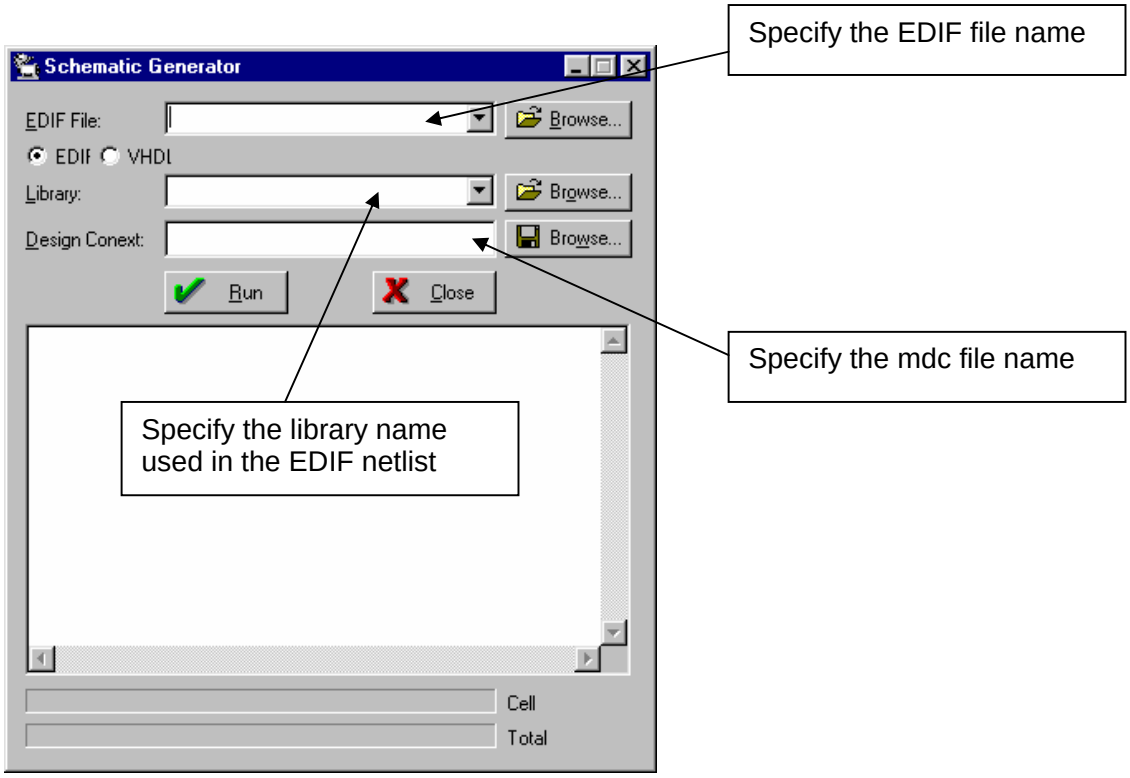


Tools → Import EDIF Netlist...

Command Description

It invokes the schematic generator.

Usage & Dialog Box



Note: The schematic generator of MyLogic Station does not generate the schematic data from VHDL netlist for SchEd. And then, it is not available at circuit level design.

Tools → Export → VHDL

Command Description

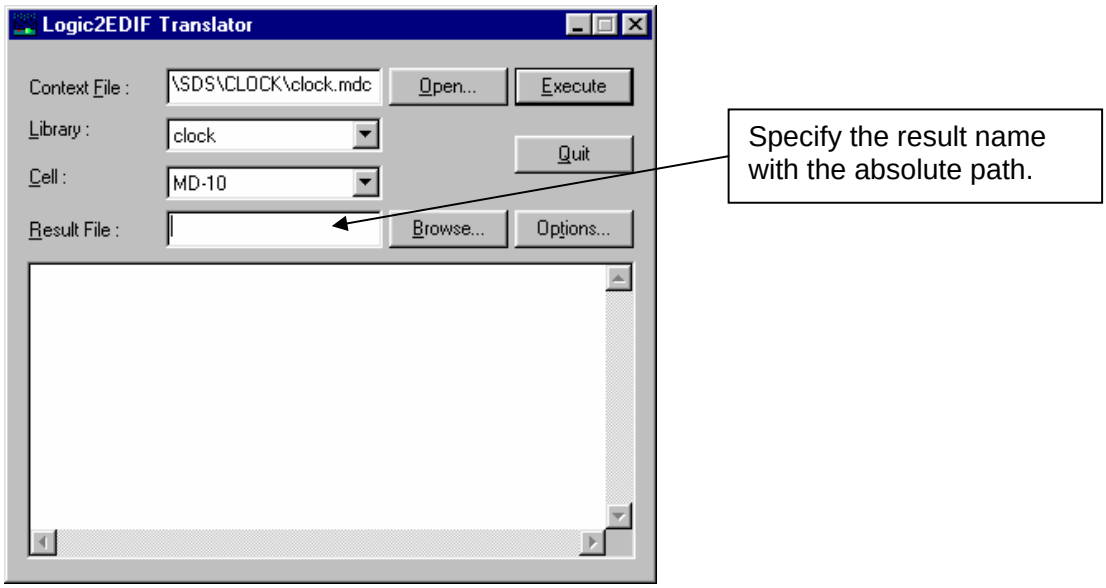
It will extract VHDL netlist from schematic data.

Tools → Export → EDIF 200

Command Description

It invokes the Logic2EDIF Translator.

Usage & Dialog Box

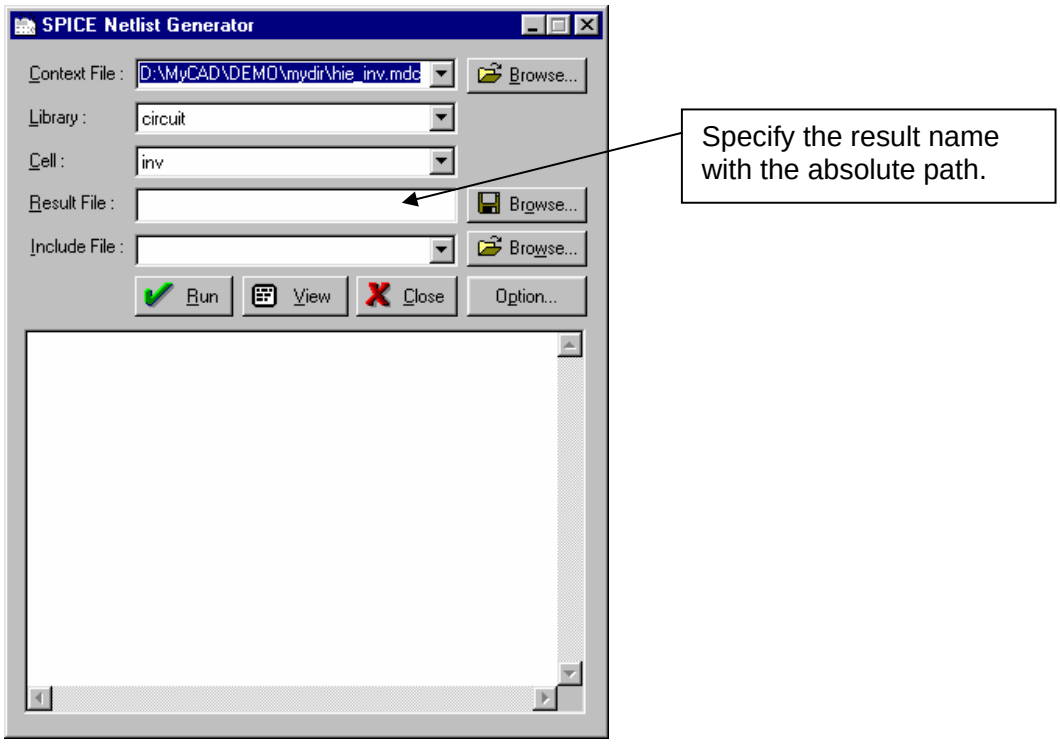


Tools → Export → SPICE**

Command Description

It invokes the SPICE Netlist Generator.

Usage & Dialog Box

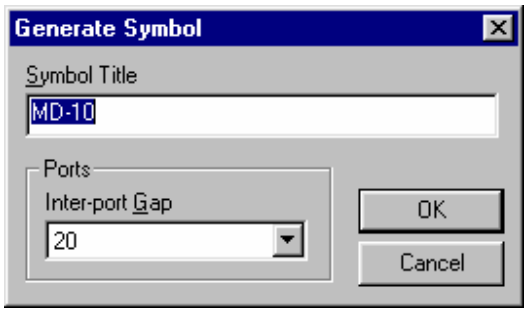


Tools → Make Symbol…

Command Description

It makes a symbol with the schematic data.

Usage & Dialog Box



Symbol Title: it is automatically named by the name of current working Schematic View.

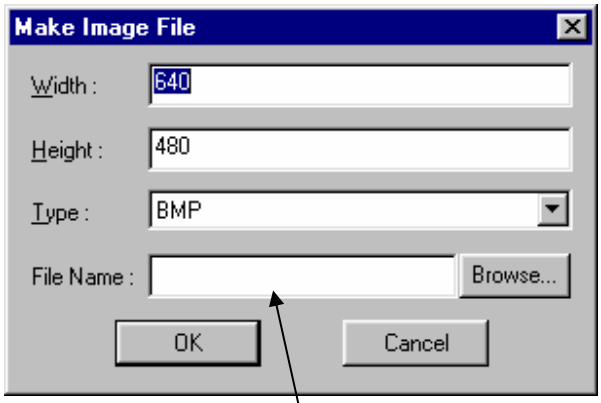
Inter-port Gap: it means the distance between symbol's port. The distance units are the space unit number of SchEd.

Tools → Make Image File…

Command Description

It copies the whole data of current working Schematic View into clipboard. And then, the copied data can be saved as an image file such as BMF, GIF, or TIFF.

Usage & Dialog Box



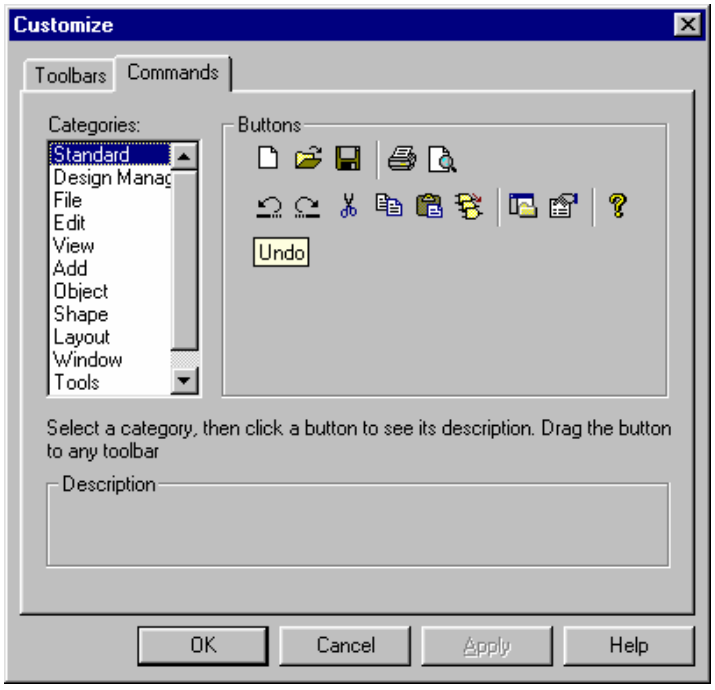
Specify the result file name with the absolute path.

Tools → Customize…

Command Description

It will bring up the customize dialog box.

Usage & Dialog Box



Toolbars can be customized. To customize toolbar, just drag the icon from the Buttons View at the right of Commands tab and drop it the toolbar you are customizing. To remove a command from the toolbar, just drag it from the toolbar and drop it outside of the toolbar.

Tools → Options…

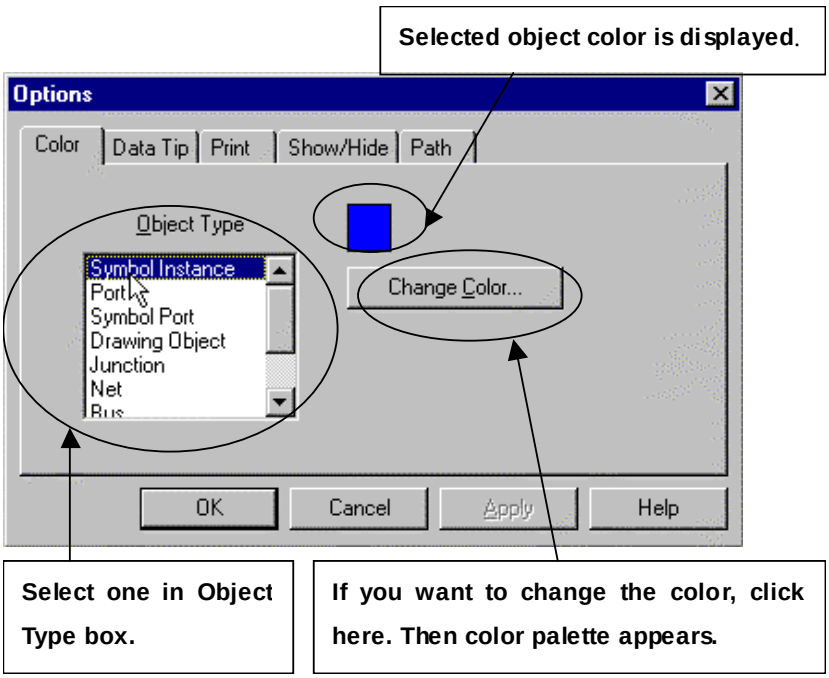
Command Description

It brings up the options dialog box.

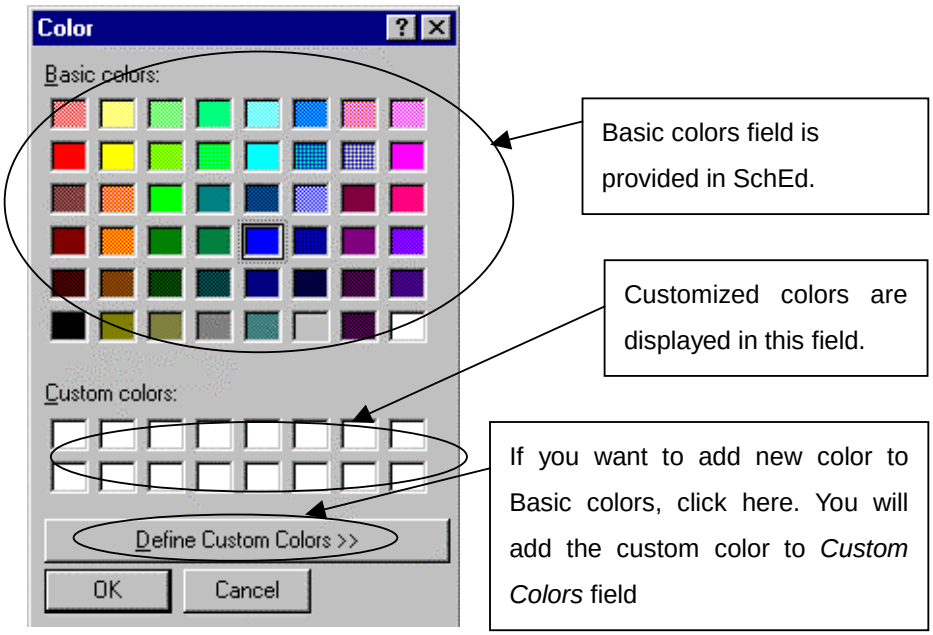
Usage & Description

Options dialog box is consisted of five tabs namely **Color**, **Data Tip**, **Print**, **Show/Hide**, and **Path**.

Color Tap



The following **Color** tab determines the default color of each object. Select one of "**Object Type"** view whose color to be changed, and then selected object color is displayed in **Color** tab. If you want to change the color to another, click ***Change Color...***button on **Color** tab. The following dialog box is appeared.

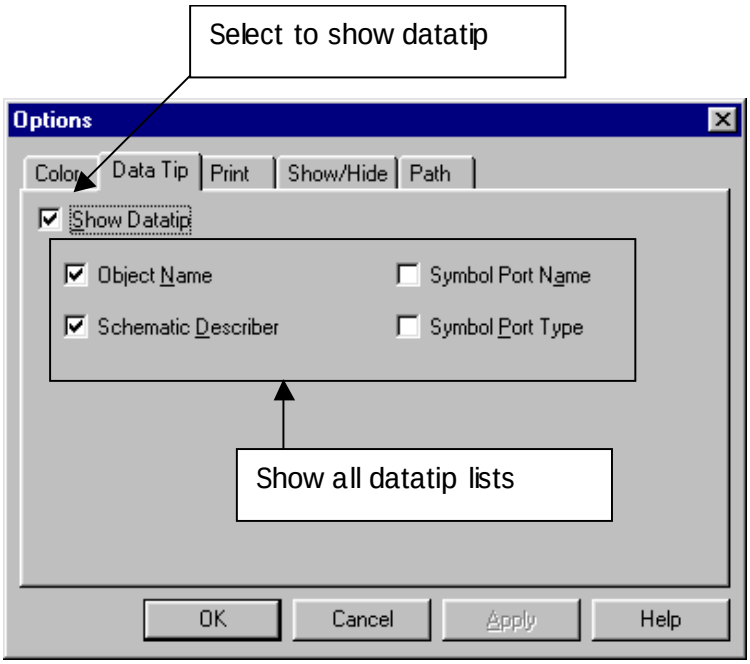


Select the color which you want to change. The changed color does not automatically apply to the object that was drawn before changing the color. It only applies to the drawing object after changing. The default color of SchEd is following in the table.

Object Type	Color	Object Type	Color
Symbol Instance	Blue	Port	Blue
Symbol Port	Blue	Drawing Object	Blue
Junction	Sky Blue	Net	Brown
BUS	Gray	Selected Region	Green
Background	White		

Data Tap

The Data Tip page follows.

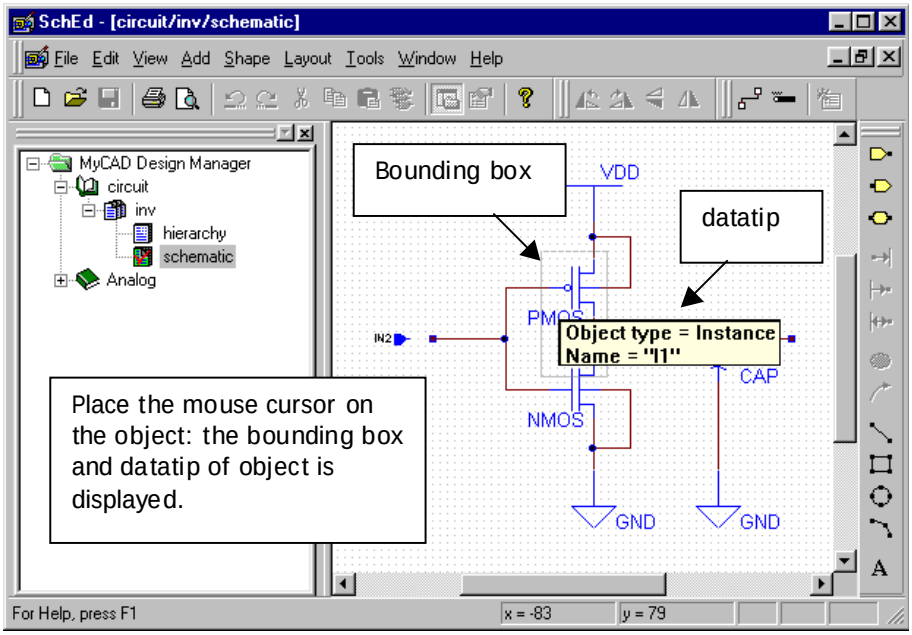


Check on "**Show Data"** check button if you want to show object's datatip on the schematic view of SchEd.
Checking on "**Object Name"**, "**Schematic Describer"**, "**Symbol Name"** and "**Symbol Port Direction"** determines whether related object's datatips on the schematic view of SchEd are showed or not.

Object Name

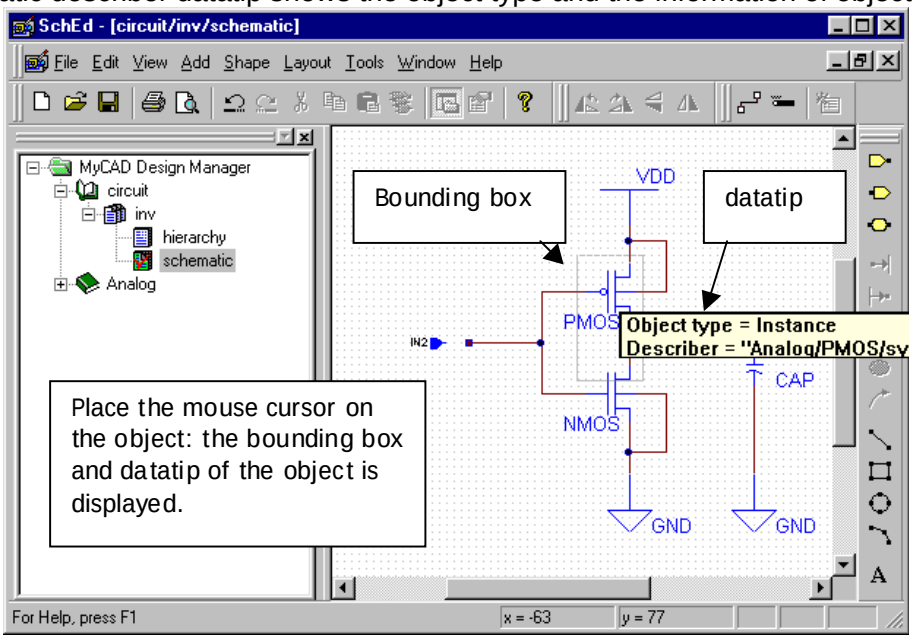
If you check on "**Object Name"** check box, object name datatip will be displayed in Schematic View.

Object name datatip are the object type and the object name.



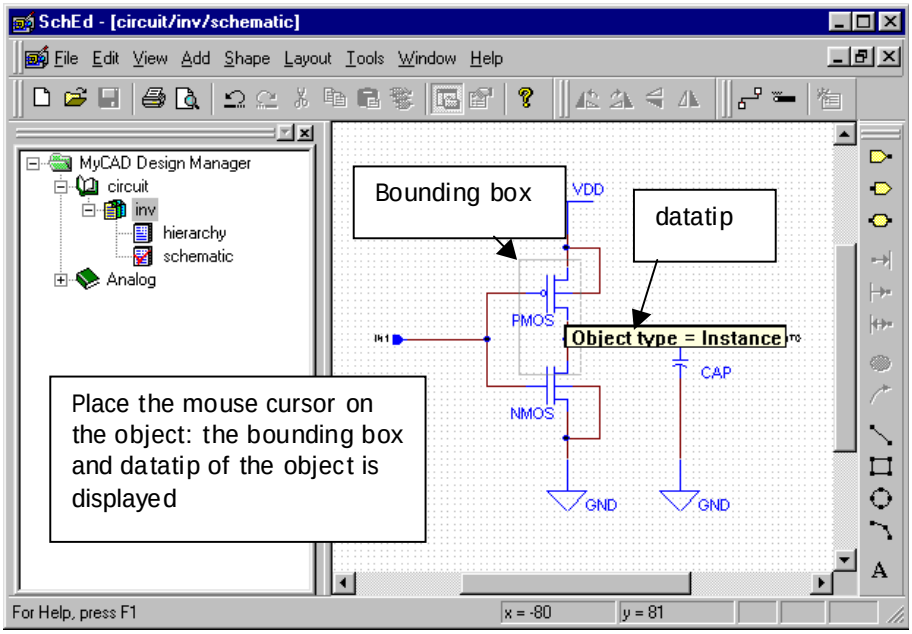
Schematic Describer

If you check on "**Schematic Describer"** check box, schematic describer datatip will be displayed in Schematic View.
Schematic describer datatip shows the object type and the information of object.



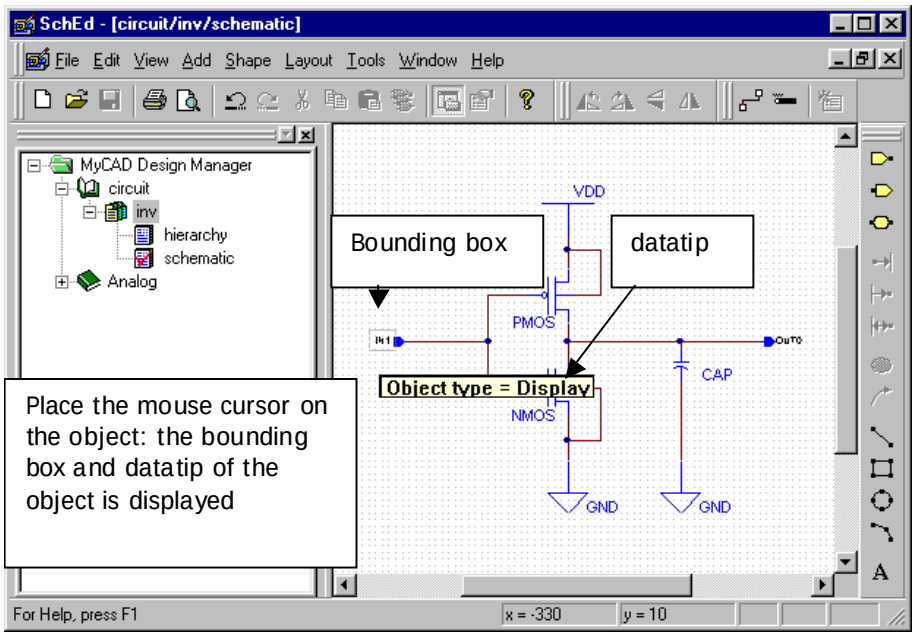
Symbol Name

If you check on "**Symbol Name"** check box, symbol name datatip will be displayed at the Schematic view.

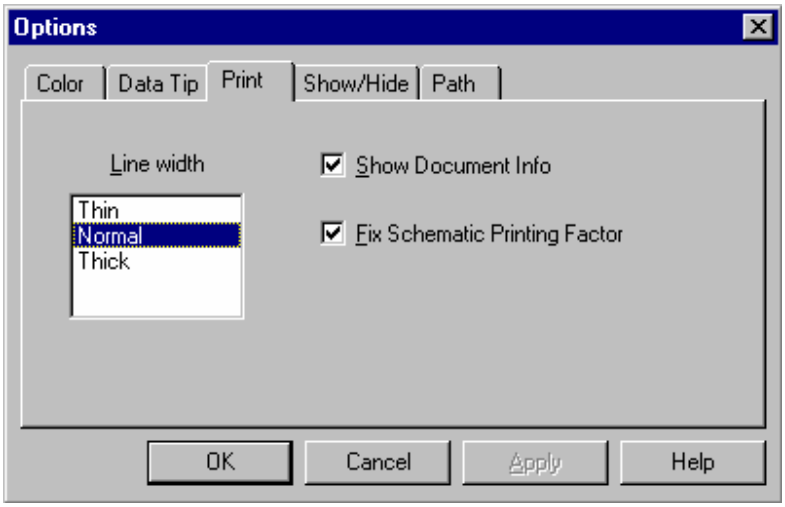


Symbol Port Direction

If you check on "**Symbol Port Direction"** check box, Symbol port direction datatip will be displayed in Schematic view.



Print Tab



Line width view on **Print** tab determines the thickness of the lines to be printed. Select "**Thin"**, depicts the thinnest line width to be printed. "**Normal"** depicts the standard thickness and "**Thick"** the thicker lines.

Show Document Info

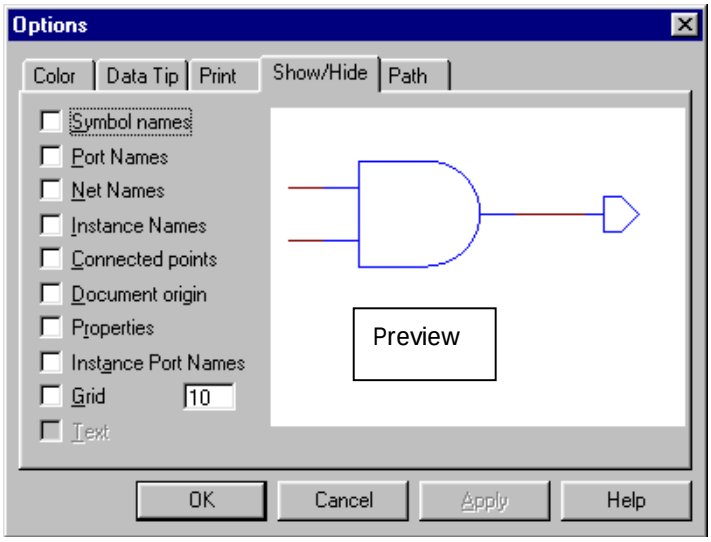
It determines whether to print the Document Info. or not. If you check off this field, then the Document Info will not be printed. (Detail things are refer to *Document Info…*)

Turn on Fix Schematic Printing Factor

It determines whether Fix the Schematic Printing Factor is applied or not when print. If you check on this field, then the printing scale is fixed regardless of schematic design size.

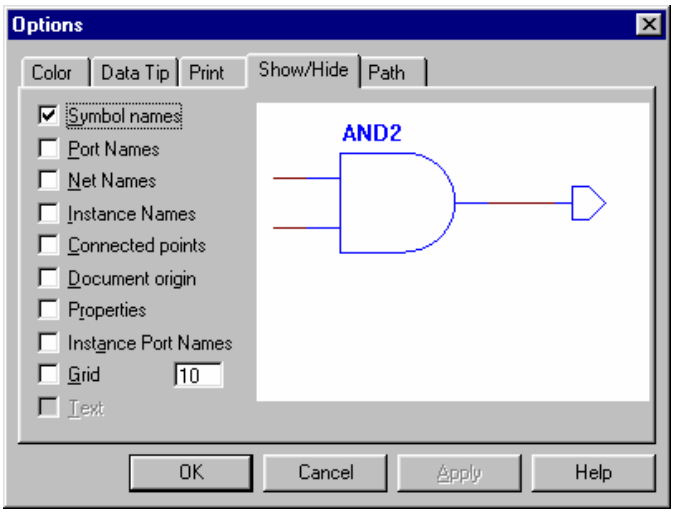
Show/Hide tab

Only checked selection is displayed at the **Preview** on **Show/Hide** tab.



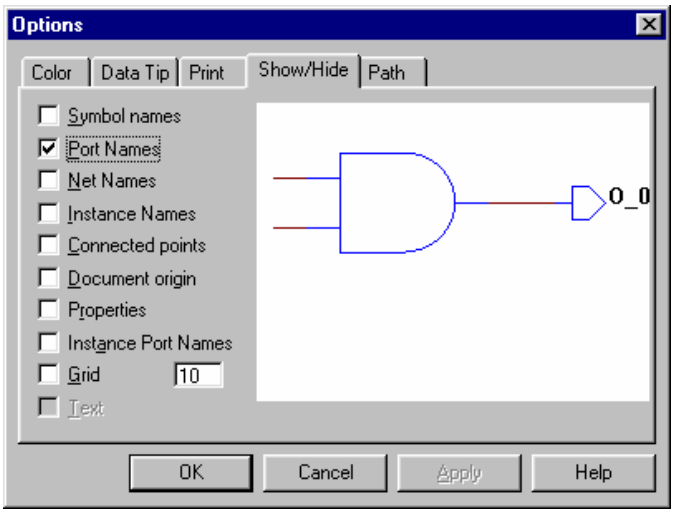
Symbol names

It determines whether to show symbol name or not at the Schematic View. And you can also see that on **Preview** on **Show/Hide** tab like to the following.



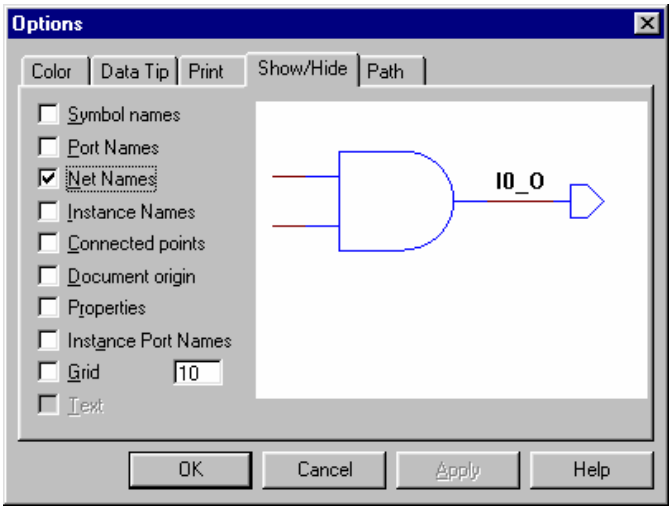
Port Names

It determines whether to show the name of port or not at the current working view. And you can also see that on **Preview** on **Show/Hide** tab like to the following.



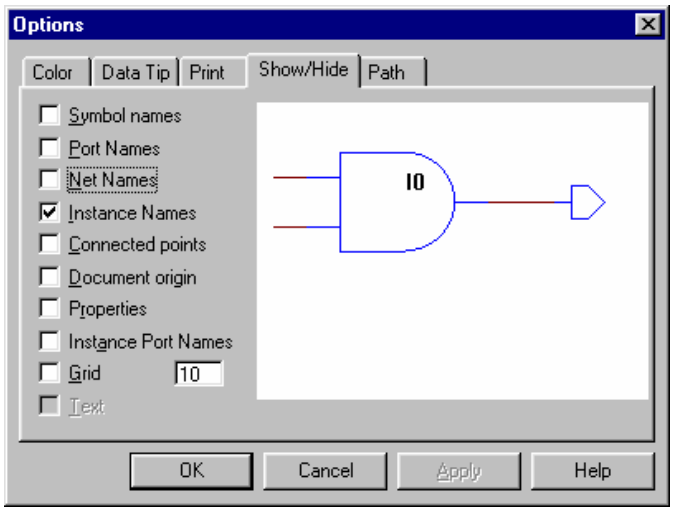
Net Names

It determines whether to show the net names between symbols or not at the current working view. And you can also see that on **Preview** on **Show/Hide** tab like to the following.



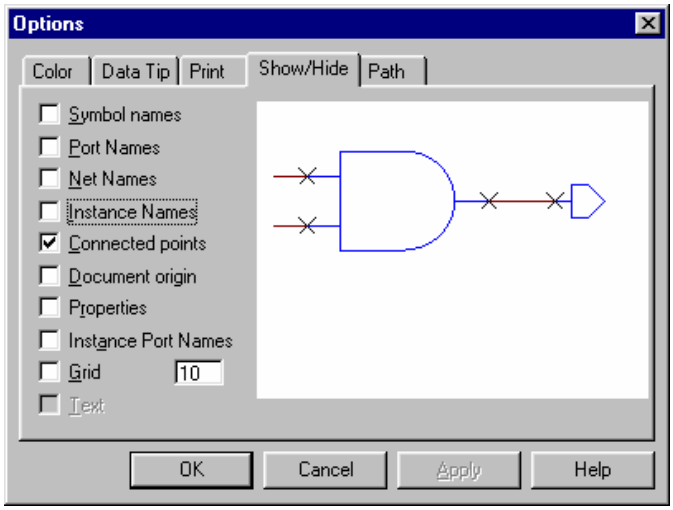
Instance Names

It determines whether to show instance name or not at the current working view. And you can also see that on **Preview** on **Show/Hide** tab like to the following.



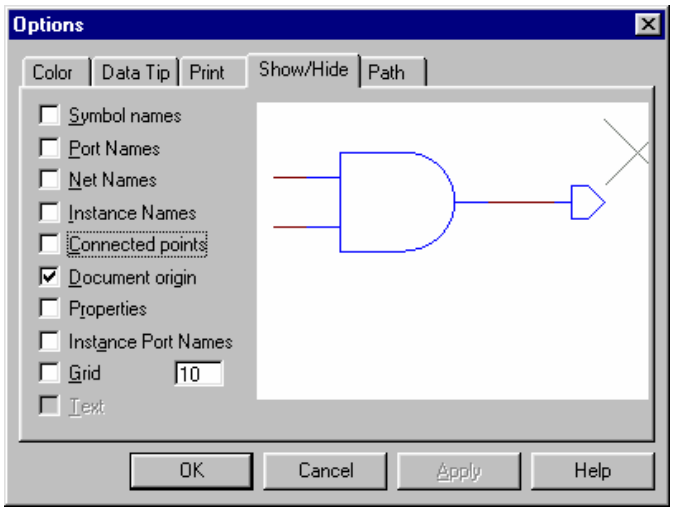
Connected points

It determines whether to show the connection between symbol and wire or not at the current working view. And you can also see that on **Preview** on **Show/Hide** tab like to the following.



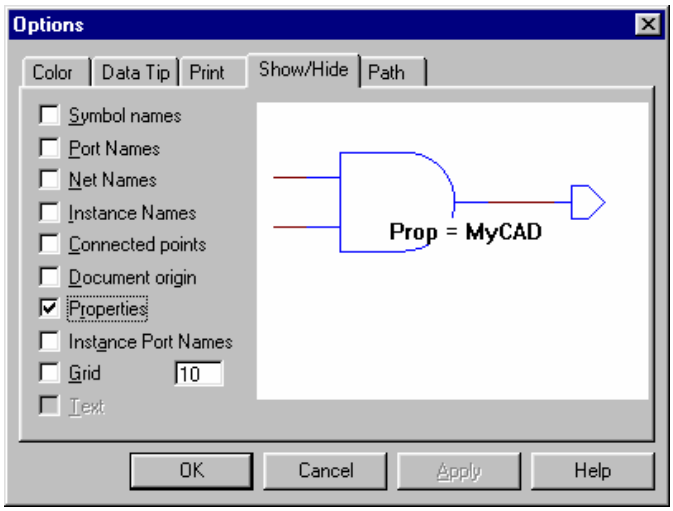
Document Origin

It determines whether to show the document origin or not at the current working view. And you can also see that on **Preview** on **Show/Hide** tab like to the following.



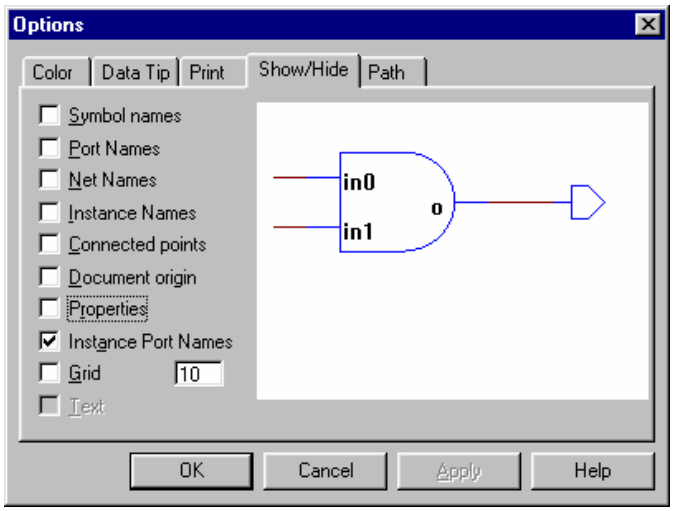
Properties

It determines whether to show properties of each symbol or not at the current working view. And you can also see that on **Preview** on **Show/Hide** tab like to the following.



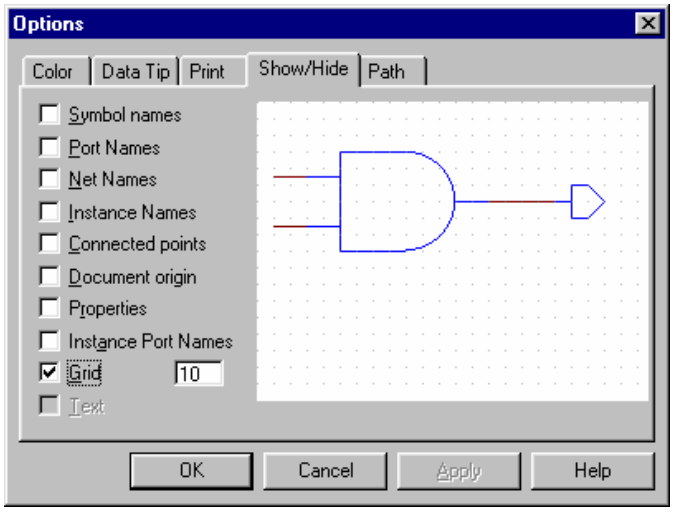
Instance Port Names

It determines whether to show instance port names or not at the current working view. And you can also see that on **Preview** on **Show/Hide** tab like to the following.

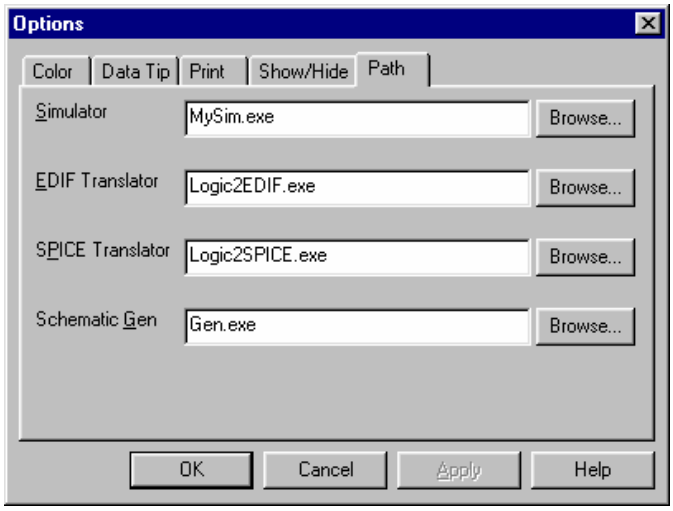


Grid

It determines whether to show the grid or not at the current working view. Type specified number at the blank box next to Grid check box. The number of the blank box next to Grid check box is the unit space of SchEd. The default value is 10. If you want to specify the space of grid, type value at the blank box next to Grid check box.



Path tab



It links other tools of MyAnalog Station to SchEd.
"Simulator" text input box is automatically set up for Logic Simulator of MyLogic Station, so if you want to link SPICE simulator, MySPICE, click on ***Browe…*** button and then locate the directory of MySPICE.
"EDIF Translator" text input box is automatically set up for MyLogic Station. It is not available on MyAnalog Station.
"Schematic Gen" text input box is automatically set up for schematic generator of MyLogic Station. It is also not available on MyAnalog Station.
"SPICE Translator" text input box is automatically set up for netlist extractor of MyAnalog Station.

*Note: The default paths of programs are set up for MyLogic Station at **Path** tab.*

Window → New Window

Command Description

It will bring up another window for the current activate window.

Window → Close All

Command Description

It will close all current activate windows except for Design Manager.

Window → Cascade

Command Description

It will cascade the windows.

Window → Title Horizontally

Command Description

It will arrange the windows in horizontal pattern.

Window → Title Vertically

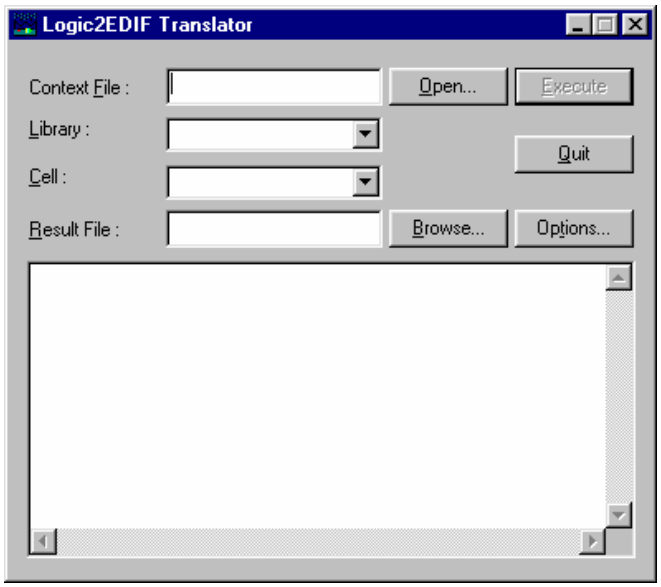
Command Description

It will arrange the windows in vertical pattern.

Translator

Logic2EDIF

Logic2EDIF is a translator from MyLogic Database to EDIF (Electrical Design Interchange Format) 200 netlist.



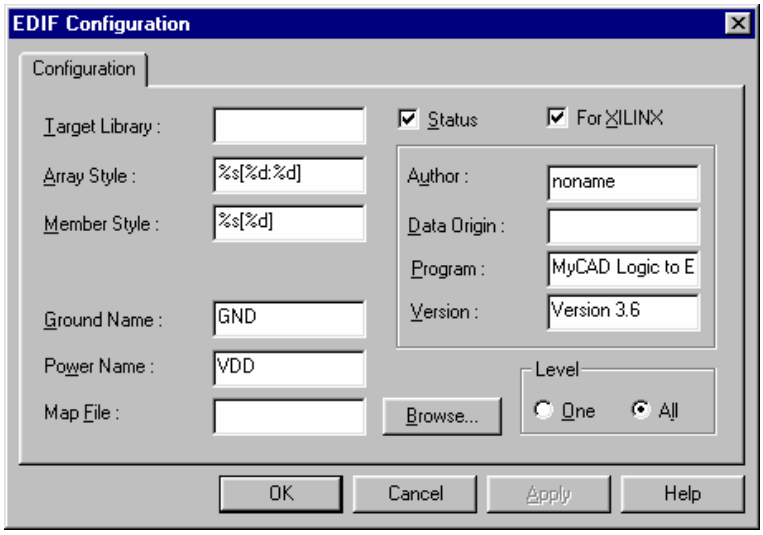
Context File: Specify the design context file by typing with the absolute path or by clicking ***Open...*** button.

Library: Choose the library. You should select one from library lists that are contained at the design context file.

Cell: Choose the cell. You should select a cell from cell lists of the chosen library. **Logic2EDIF** translates the data by cell.

Result File: Specify the result file by typing with the absolute path or clicking ***Browse...*** button.

Options…: If you want to specify the option of **Logic2EDIF**, click it. Then, the **EDIF Configuration** dialog box will be shown like to the following.



Target Library: Specify the library name for using at the other program.

Array Style: It is used for BUS translation. If you want to change the BUS name's rule from MyLogic to the other program, you should use this option.

%s: string type

%d: integer type

Member Style: It is used for BUS translation. If you want to change the BUS number's rule from MyLogic to the other program, you should use this option.

%s: string type

%d: integer type

Ground Name: If you want to change the ground name used at MyLogic Station during EDIF translation into another, specify the ground name at this field.

Power Name: If you want to change the power name used at MyLogic Station during EDIF translation into another, specify the power name at this field.

Map File: You can specify the map file, which is contained mapping information between components that used at the different library.

Note: the port number should be same between components that used at the different library.
The format of map file is the following these:
Edifout*cell_map: "scr_cell=dest_cell" "s_port=d_port"
***src_cell**: specify the name of source cell*
***dest_cell**: specify the name of destination cell*
***s_port**: specify the source port of source cell*
***d_port**: specify the destination port of destination cell*
***Example**: edifout*cell_map: "IN=INV" "IN1=A" "OUT=B"*

Status: It determines whether adds user's information or not when you translate from MyLogic Database into EDIF 200 netlist.

***Author**: Used to include the designer name.*
***Data Origin**: Used to include the vendor name used at the schematic job.*
***Program**: Used to include the name of program used to make the EDIF netlist.*

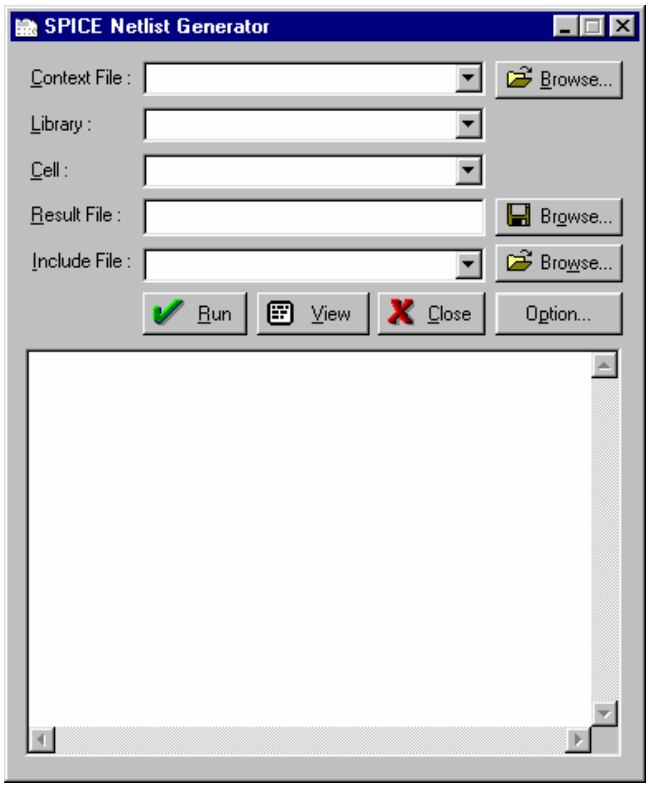
***Version**: Used to include the version number of Logic2EDIF program.*

For XILINX: If this box is checked on, then Logic2EDIF would add the extra information for Xilinx when generate EDIF 200 netlist.

Level: It determines whether only one-level translation or all-level translation. In general, the database of MyLogic Station has the hierarchy so that you should specify the level for Logic2EDIF to translate whether only current level or all.

SPICE Netlist Generator

SPICE Netlist Generator is a netlist extractor from MyLogic Database to SPICE format netlist such as Standard SPICE format or HSPICE format. The extracted netlist from SPICE Netlist Generator can be used for both SPICE simulation and LVS (Layout Versus Schematic) with MyChip Station.



Context File: Specify the design context file by typing with the absolute path or by clicking ***Browse...*** button.

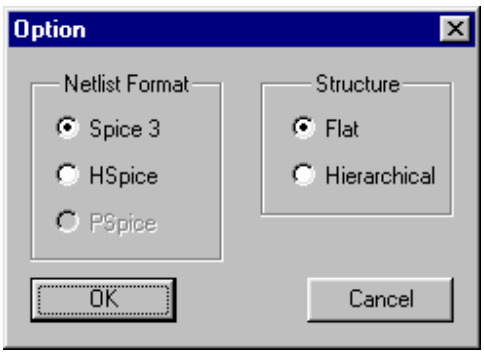
Library: Choose the library. You should select one from library lists that are contained at the design context file.

Cell: Choose the cell. You should select a cell from cell lists of the chosen library. **SPICE Netlist Generator** extracts the data by cell.

Result File: Specify the result file by typing with the absolute path or clicking ***Browse...*** button.

Include File: Specify the model parameter file for SPICE simulation by typing with the absolute path or clicking ***Browse...*** button. It is the optional.

Option…: If you want to specify the option of **SPICE Netlist Generator**, click it. Then, the **Option** dialog box will be shown like to the following.



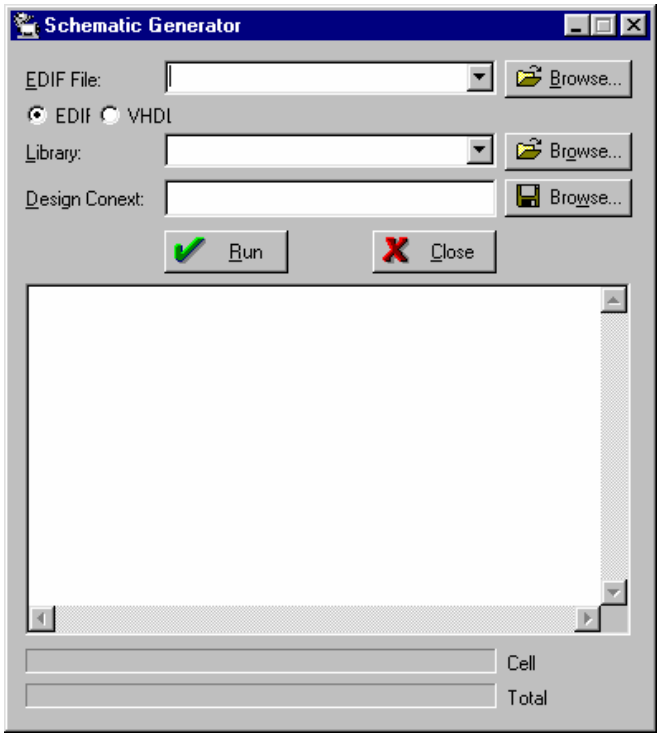
Netlist Format*: It determines the netlist format generated from SPICE Netlist Generator whether standard SPICE (Spice3) or HSPICE (HSpice).*

Structure*: It determines the structure format whether flatten (Flat) or Hierarchy (Hierarchical). If you select "**Spice3"** at **Netlist Format** view, then, you could not select "**Hierarchycal"** at **Structure** view because standard spice does not support hierarchy netlist format.*

Schematic Generator

Schematic Generator is used for generating schematic data with EDIF netlist or VHDL netlist.

Note: Schematic Generator does not support to generate from VHDL netlist to MyLogic database.



EDIF File: Specify the EDIF netlist by typing with the absolute path or clicking ***Browse...*** button.

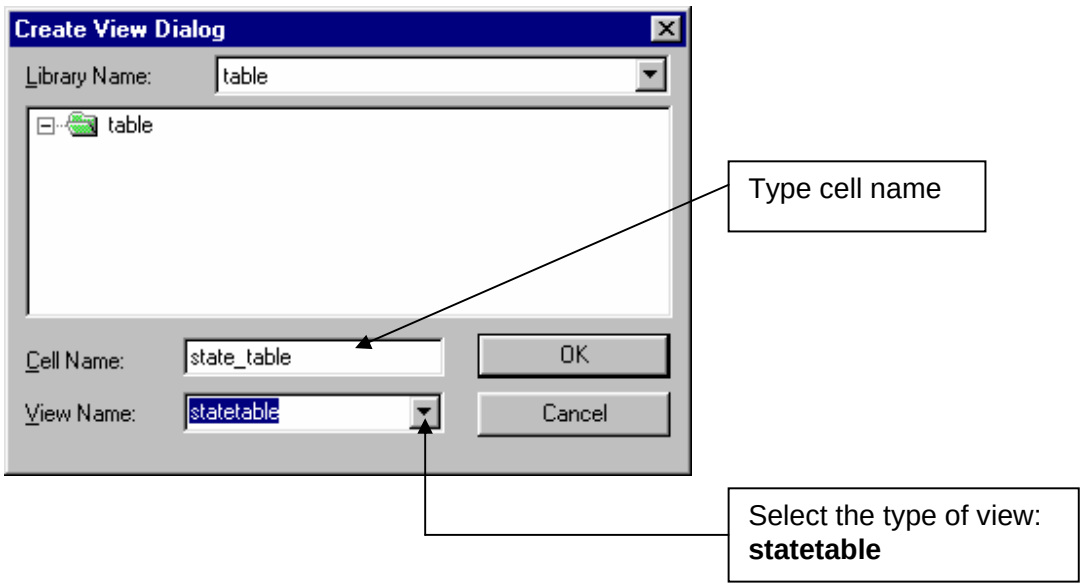
Library: Specify the library name used at EDIF netlist by typing with the absolute path or clicking ***Browse...*** button.

Design Context: Specify the design context file for SchEd by typing with the absolute path or clicking ***Browse...*** button.

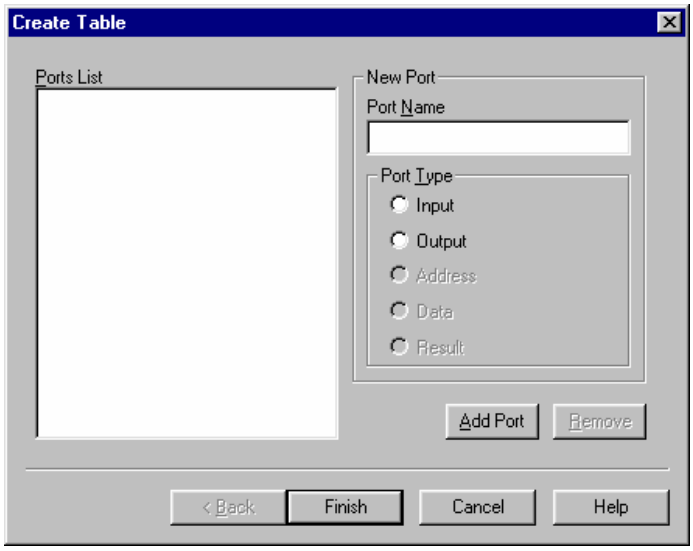
Appendix I: State Table

In this section, the procedure of the State Table's usage is described.

Choose ***File→New*** from pull-down menu of SchEd. Then, the **Create View Dialog** dialog box will be shown.

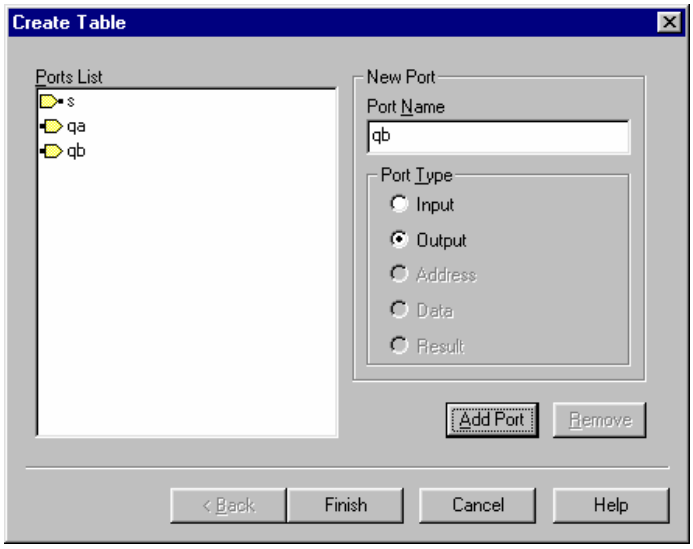


Type cell name at the "**Cell Name** "text input box. And choose the type of view as statetable. Then, the **Create Table** dialog box will be shown.



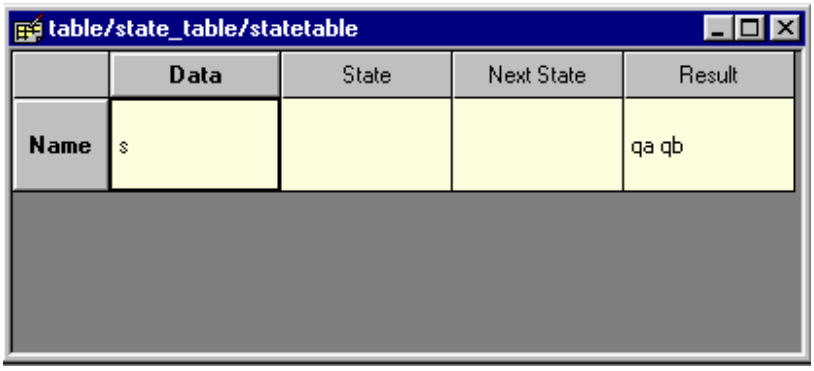
Type port name at "**Port Name"** text input box on **New Port v**iew. And choose the port type either input or output by checking on at **Port Type** selection view.

Click ***Add port*** button with the left mouse button on the **Create Table** dialog box. Then, the port will be appeared at the **Port List** view on the **Create Table** dialog box. (The following is one of example. Where "**s"** is input port and "**qa"** and "**qb"** is output port.)

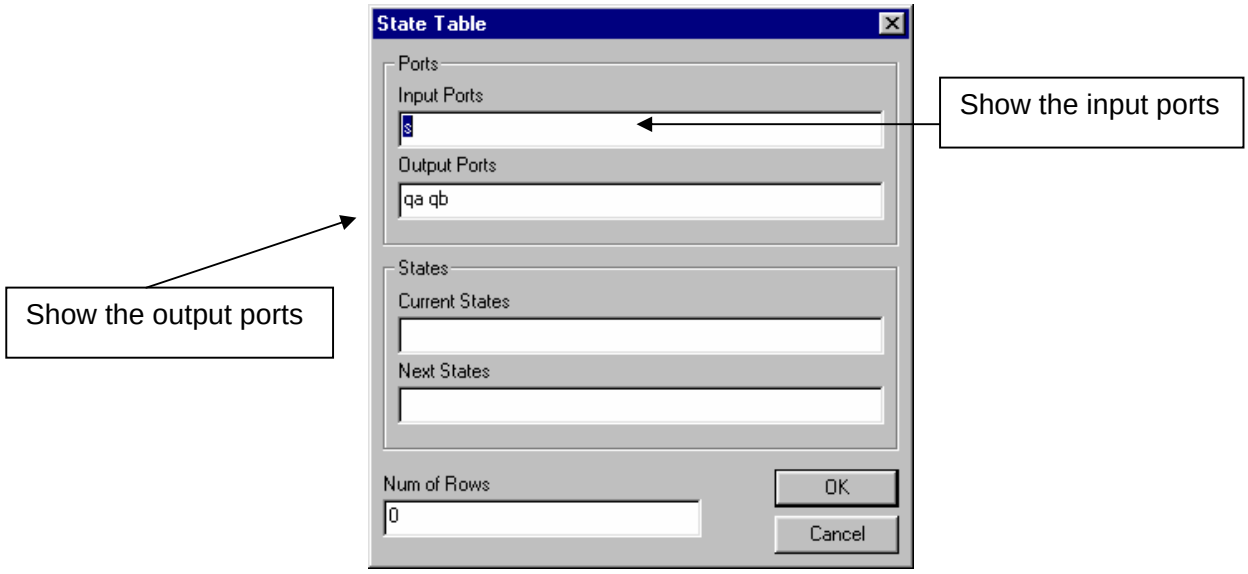


Click ***Finish*** button with the left mouse button after finishing to create ports.

Then, the Statetable view will be shown on SchEd.



And, choose *View → Properties…* from pull-down menu of SchEd. Then, the **State Table** dialog box will be shown. You can specify the property of state table using by the **State Table** dialog dox.



Current States: specify the variable names of current state. During that, you should consider the total number of variable names before specifying the current state. For example, type **B A** at the **"Current States"** text input box.

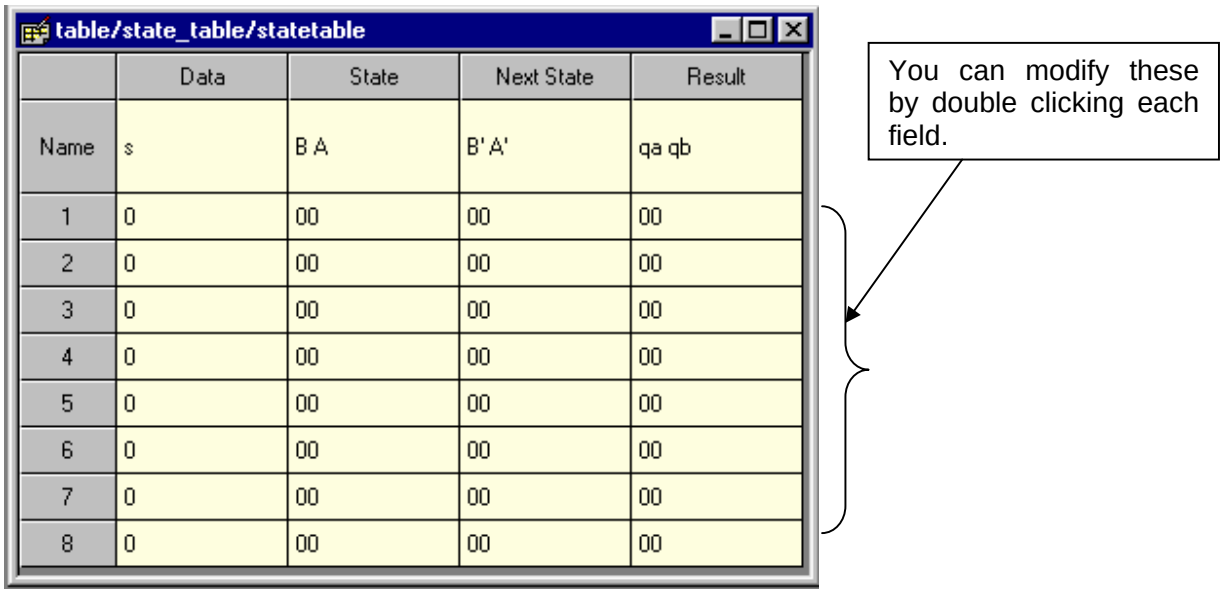
> *Note: Between variable names should be the space. If not, SchEd will recognize them into a single variable.*

Next States: specify the names of next state. The total number of next states should be same of that defined at the Current States text input box. For example, type **B' A'** at the **"Next States"** text input box.

Note: Between variable names should be the space. If not, SchEd will recognize them into a single variable.

Num of Rows: type the total number of the state. And the number of port and of state determines it. For example, the number of state and of port is each **2(two)** and **3(three),** then the value of **"Num of Rows"** text input box should be **8(eight)** because the total number of row is calculated by the formula, **(the number of state)**(the number of port).**

If you follow the above, then the **Statetable** View will be shown the following:

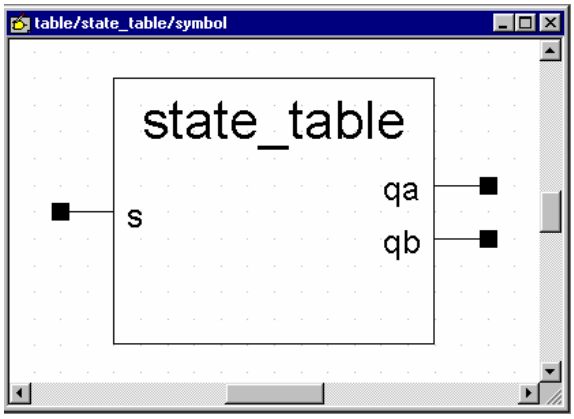


Then, you modify the above table by the following.

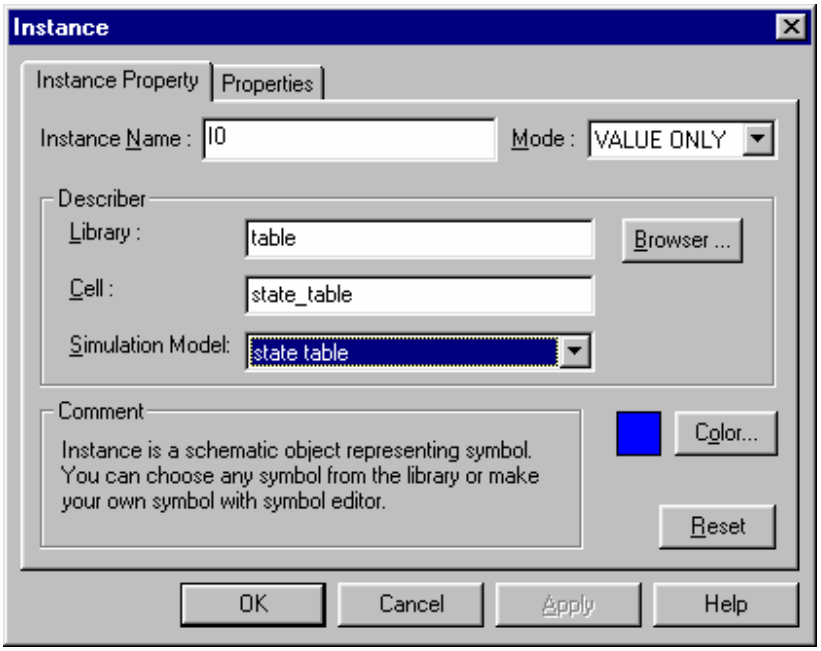
Note: To modify a field at the state table, double click it. Then, type values.

Current State	Input	Next State	Result(Output)
B A	s	B′ A′	qa qb
0 0	0	0 0	0 0
0 0	1	1 0	0 1
1 0	0	1 0	0 1
1 0	1	0 1	1 0
0 1	0	0 1	1 0
0 1	1	1 1	1 1
1 1	0	1 1	1 1
1 1	1	0 0	0 0

After finishing to type values for each field at the state table, you should make it into a symbol to use at the other schematic by choosing ***Tools → Make Symbol...*** from pull-down menu of SchEd.



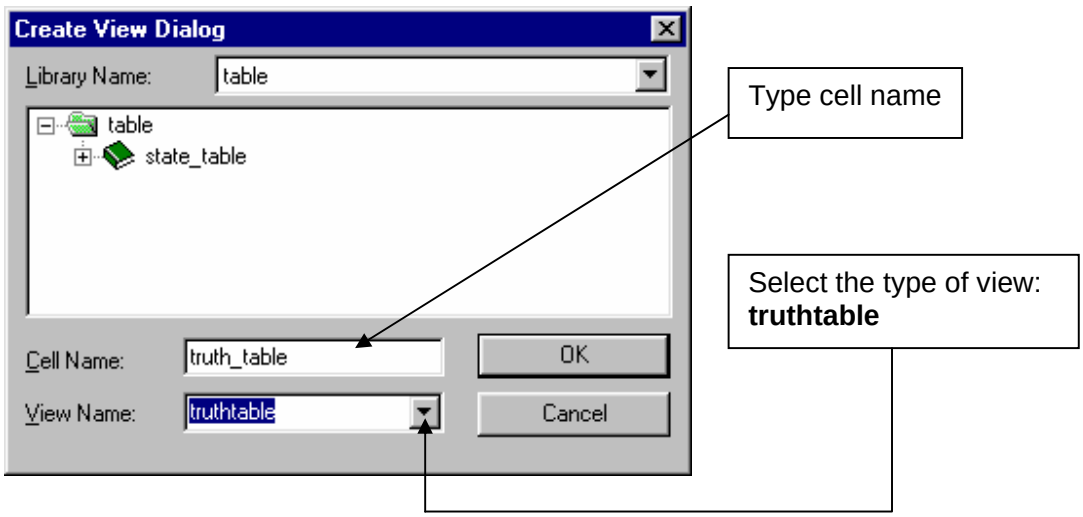
At the other schematic, you should set up the **Simulation Model** to **"state table"** at the property of the instance for simulation on MySim.



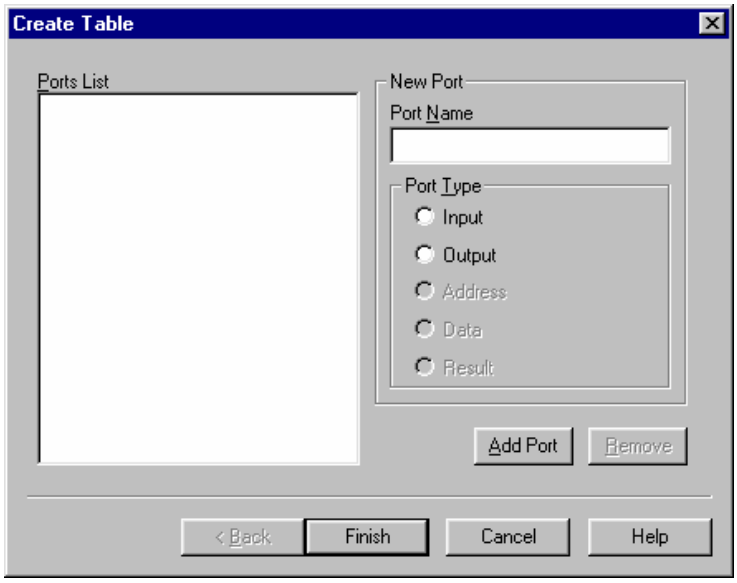
Appendix II: Truth Table

In this section, the procedure of the Truth Table's usage is described.

Choose ***File→New*** from pull-down menu of SchEd. Then, the **Create View Dialog** dialog box will be shown.

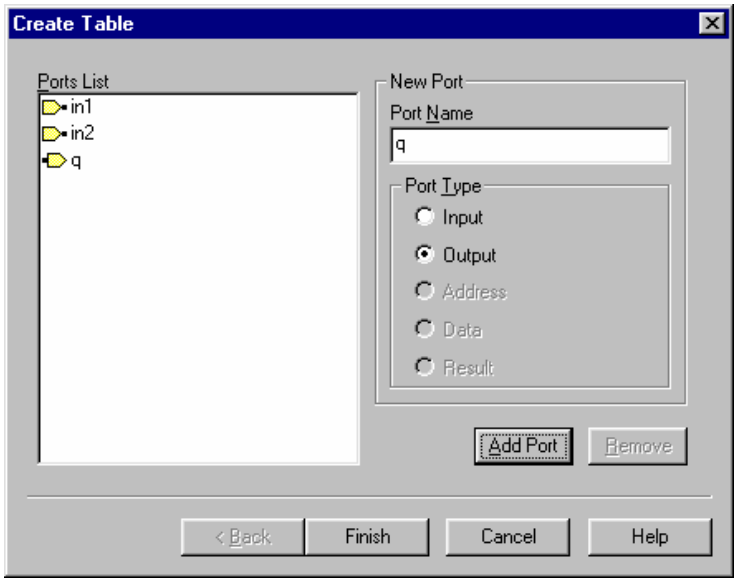


Type cell name at the **"Cell Name"** text input box. And, choose the type of view as truthtable. Then, the **Create Table** dialog box will be shown.



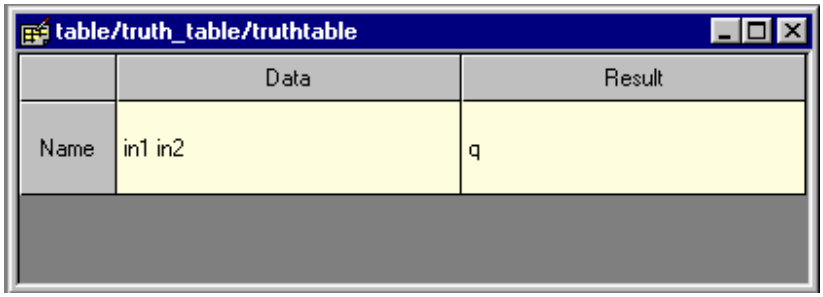
Type port name at **"Port Name"** text input box on **New Port** view. And choose the port type either input or output by checking on at **Port Type** selection view.

Click ***Add Port*** button with the left mouse button on the **Create Table** dialog box. Then, the port will be appeared at the **Port List** view. (The following is one of example. Where **"in1"** and **"in2"** is input port and "**q"** is output port.)

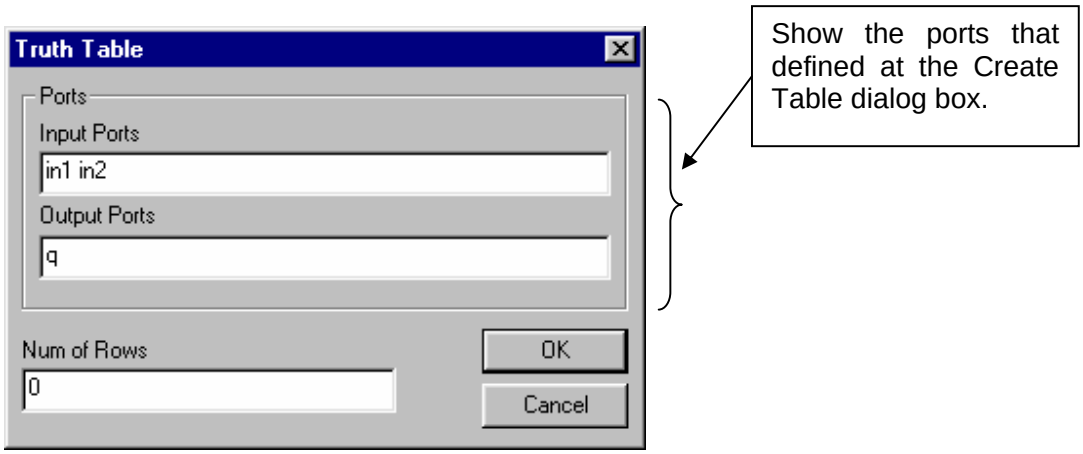


Click ***Finish*** button with the left mouse button after finishing to create ports.

Then, the **Truthtabl**e view will be shown on SchEd.

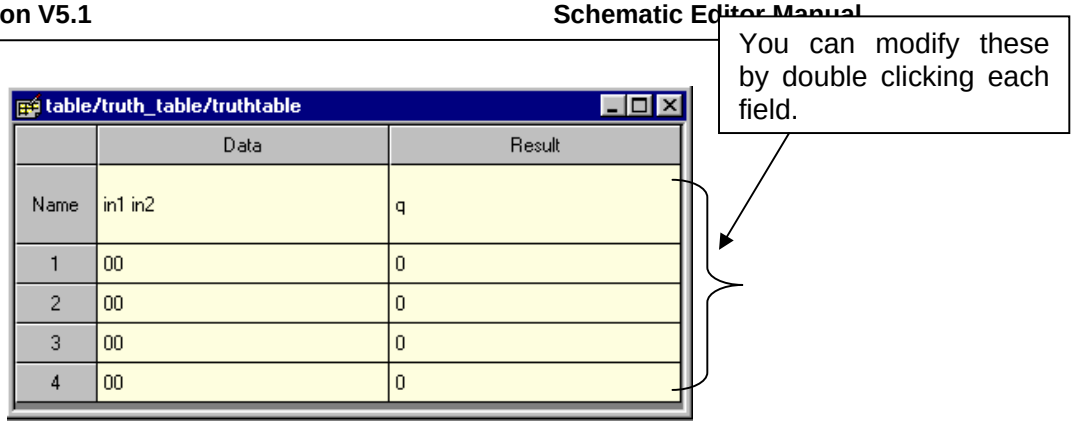


And, choose **View →Properties…** from pull-down menu of SchEd. Then, the **Truth Table** dialog box will be shown. You can specify the property of truth table using by the **Truth Table** dialog box.



Specify number on **"Num of Rows"** text input box. The total number of input ports determines the value of row. In this example, the total number of input ports is **"2(two)"** so that the number of row becomes **"4(four)"** by the formula, 2***(the total number of input ports).

If you type "4(four)" at the **"Num of Rows"** text input box, then, the following will be shown.

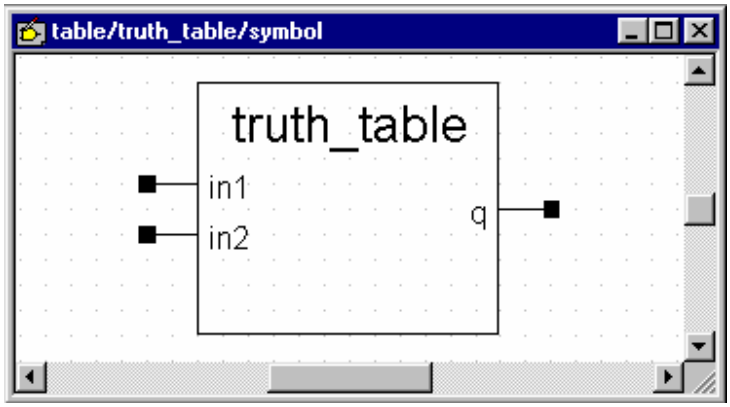


Then, you modify the above table by the following.

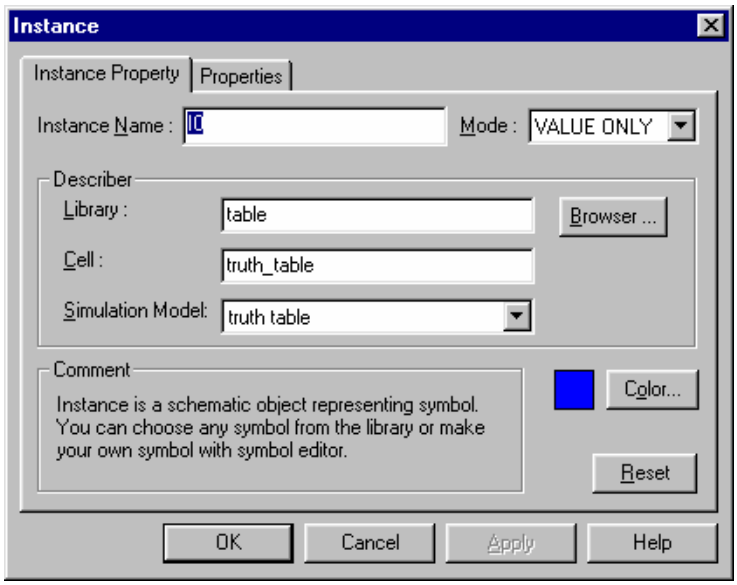
	Data	**Result**
Name	in1 in2	q
1	00	0
2	01	0
3	10	0
4	11	1

Note: If you want to modify the name field, you should space between names. If not, SchEd will recognize it as a name.

After finishing to type values for each field at the truth table, you should make it into a symbol to use at the other schematic by choosing ***Tools → Make Symbol...*** from pull-down menu of SchEd.



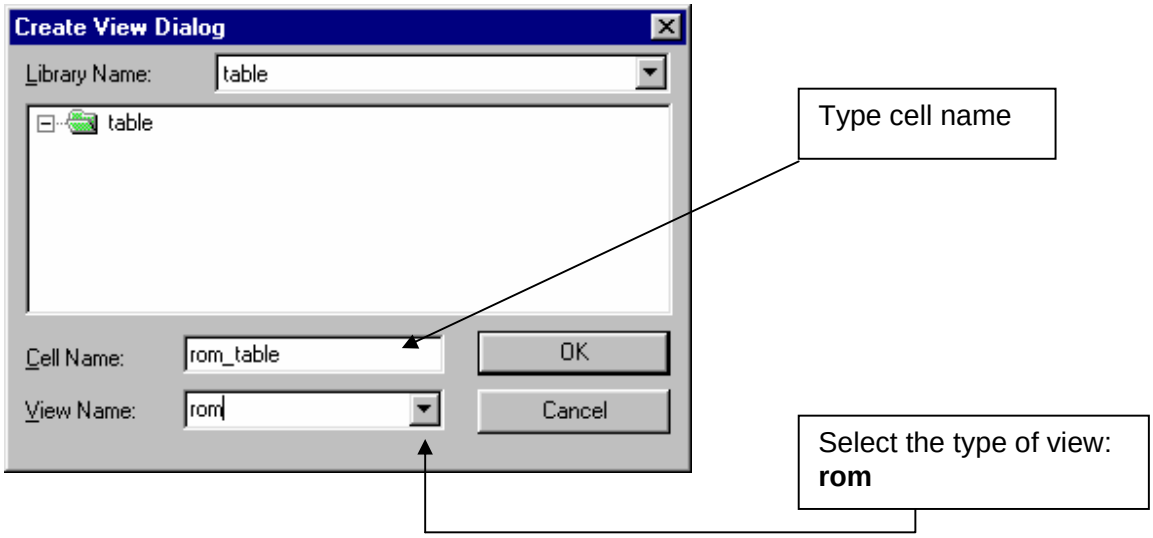
At the other schematic, you should set up the **Simulation Model** to **"truth table"** at the property of the instance for simulation on MySim.



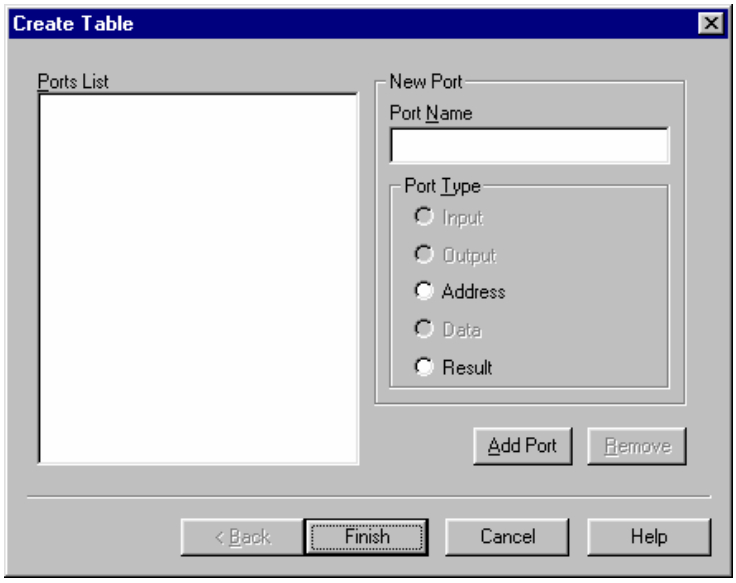
Appendix III: Rom Table

In this section, the procedure of the Rom Table's usage is described.

Choose ***File→New*** from pull-down menu of SchEd. Then, the **Create View Dialog** dialog box will be shown.

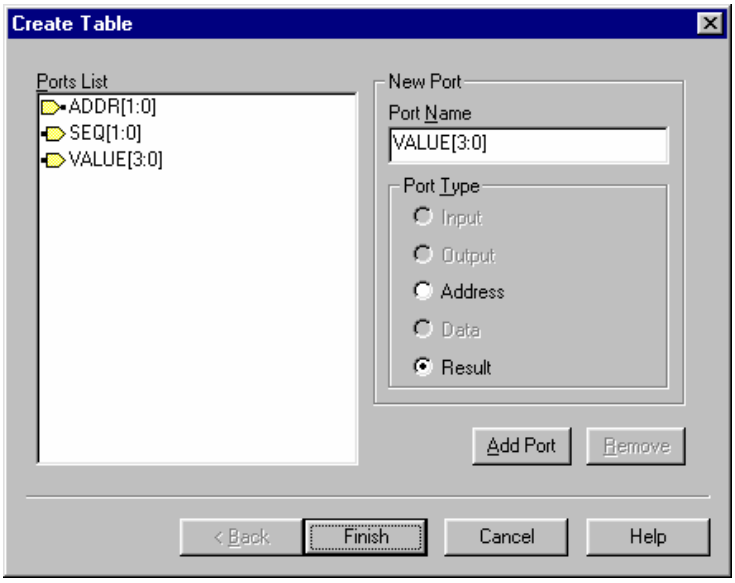


Type cell name at the **"Cell Name"** text input box. And, choose the type of view as rom. Then, the **Create Table** dialog box will be shown.



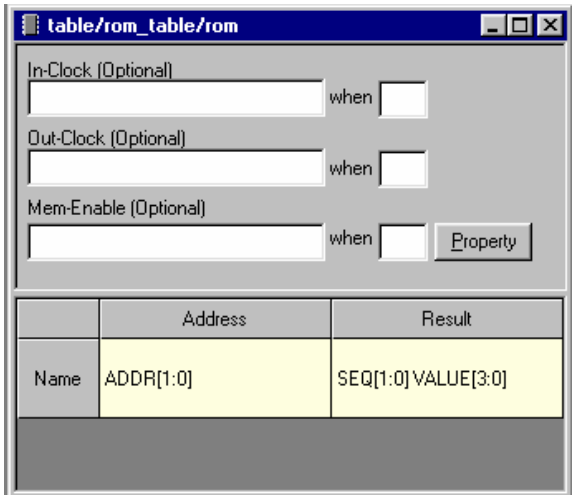
Type port name at **"Port Name"** text input box on **New Port** view. And choose the port type either input or output by checking on at **Port Type** selection view.

Click ***Add Port*** button with the left mouse button on the **Create Table** dialog box. Then, the port will be appeared at the **Port List** view. (The following is one of example. Where **"ADDR[1:0]"** is address port and **"SEQ[1:0]"** and **"VALUE[3:0]"** is result port.)

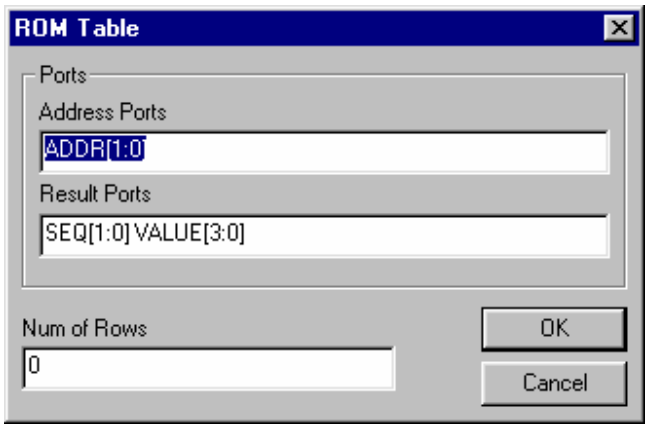


Click ***Finish*** button with the left mouse button after finishing to create ports.

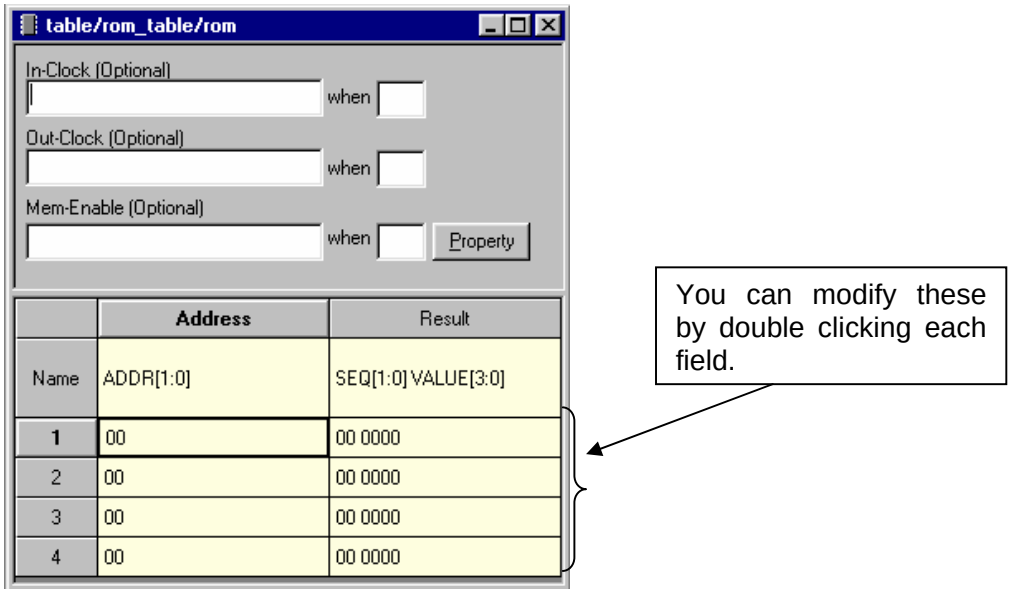
Then, the **ROM** view will be shown on SchEd.



And, choose **View →Properties…** from pull-down menu of SchEd. Then, the **ROM Table** dialog box will be shown. You can specify the property of truth table using by the **ROM Table** dialog box.



Type "**4**" at "**Num of Rows"** text input box. It is due to the bit-number of address port is 2. [2**(bit-number of address port) = 4]. Then, the ROM view is changed into the following diagram.



In-Clock (Optional): It controls the address port of ROM whether it works or not. You can specify the port name of in-clock by typing the name if in-clock port at ***"In-Clock (Optional)"*** text input box and the condition of in-clock port by typing the value at "***when"*** text input box next to "***In-Clock (Optional)"*** text input box. If you do not set up the port name of in-clock, then, the address port of ROM will always work.

Note: The condition of in-clock port should be "0" or "1". Where "0" means the falling edge of in-clock signal, "1" means the rising edge of in-clock signal. If you set "when=1", then the register of address port start to work when the

value of in-clock port is falling edge. In the other case, the previous value of register of address will be kept.

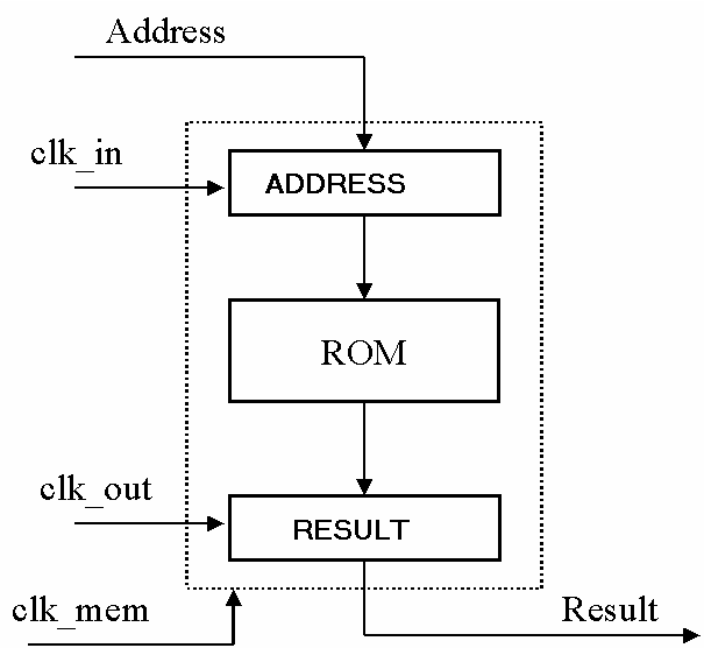
Out-Clock (Optional): It controls the result port of ROM whether it works or not. You specify the port name of out-clock at "***Out-Clock (Optional)"*** text input box and the value of "***when"*** next to "***Out-Clock (Optional***)" text input box. If you do not set up the port name of out-clock, then, the result port of ROM will always work.

Note: The condition of out-clock port should be "0" or "1". Where "0" means the falling edge of out-clock signal, "1" means the rising edge of out-clock signal. If you set "when=1", then the register of address port start to work when the value of out-clock port is falling edge. In the other case, the previous value of register of result will be kept.

Mem-Enable(Optional): It controls to work ROM or not. If you set up the memory clock with condition, ROM will work when it meets the condition. And the other case, ROM will be high impedance. If you do not set up the memory clock, then ROM will always work.

Note: The condition of memory clock port should be "0" or "1". Where "0" means the falling edge of memory clock signal, "1" means the rising edge of memory clock signal.

The following diagram might help you to know the option setting of ROM.

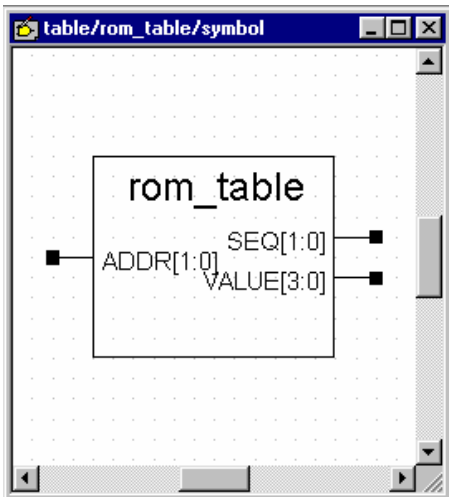


Then, you modify the above table by the following.

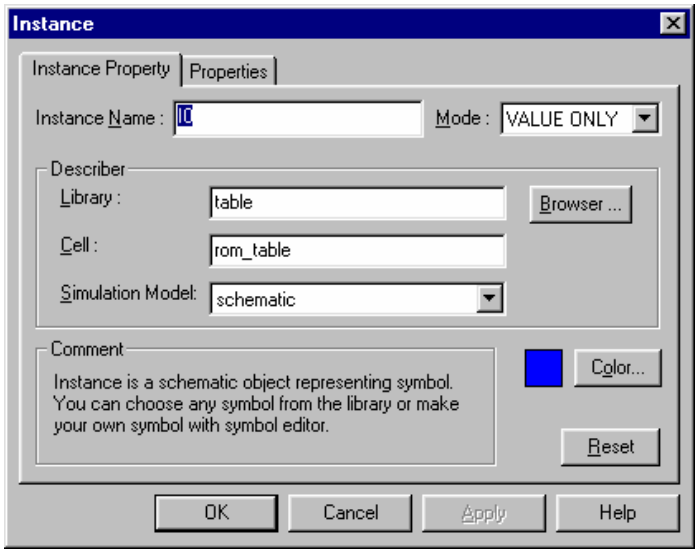
	Data	Result
Name	ADDR[1:0]	SEQ[1:0] VALUE[3:0]
1	00	00 0000
2	01	01 0011
3	10	10 1100
4	11	11 1111

Note: If you want to modify the name field, you should space between names. If not, SchEd will recognize it as a name.

After finishing to type values for each field at the ROM table, you should make it into a symbol to use at the other schematic by choosing ***Tools → Make Symbol...*** from pull-down menu of SchEd.



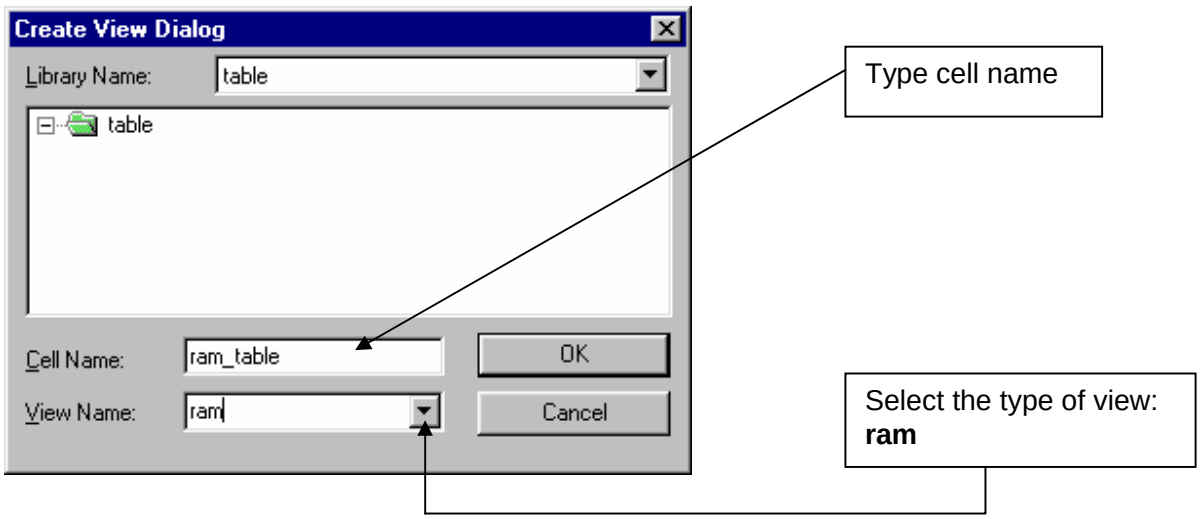
At the other schematic, you should set up the **Simulation Model** to **"truth table"** at the property of the instance for simulation on MySim.


Instance
Instance Property
Properties
Instance Name :
I0
Mode :
VALUE ONLY
Describer
Library :
table
Browser ...
Cell :
rom_table
Simulation Model:
schematic
Comment
Instance is a schematic object representing symbol.
You can choose any symbol from the library or make
your own symbol with symbol editor.
Color...
Reset
OK
Cancel
Apply
Help

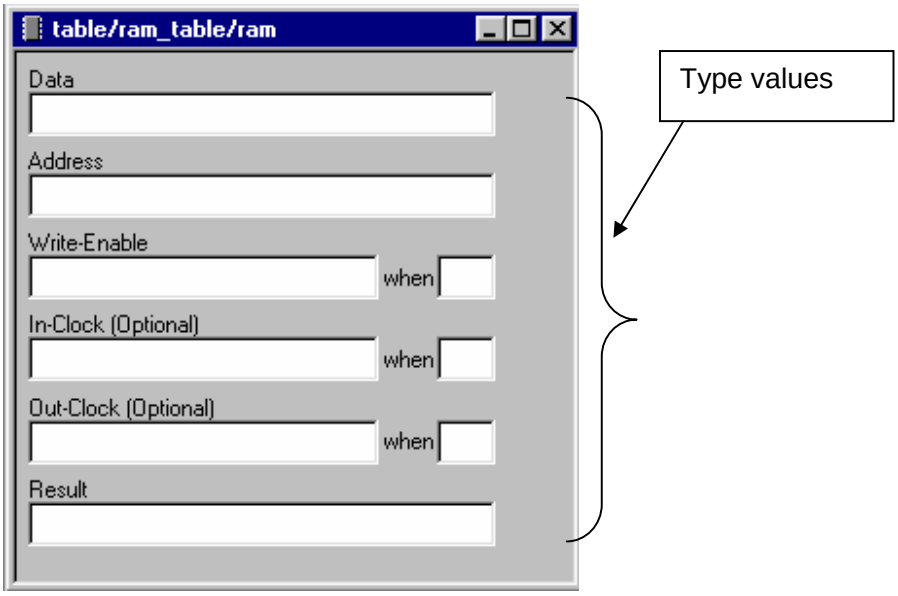
Appendix IV: Ram Table

In this section, the procedure of the Ram Table's usage is described.

Choose ***File→New*** from pull-down menu of SchEd. Then, the **Create View Dialog** dialog box will be shown.



Type cell name at the **"Cell Name"** text input box. And, choose the type of view as ram. Then, the following dialog box will be shown.

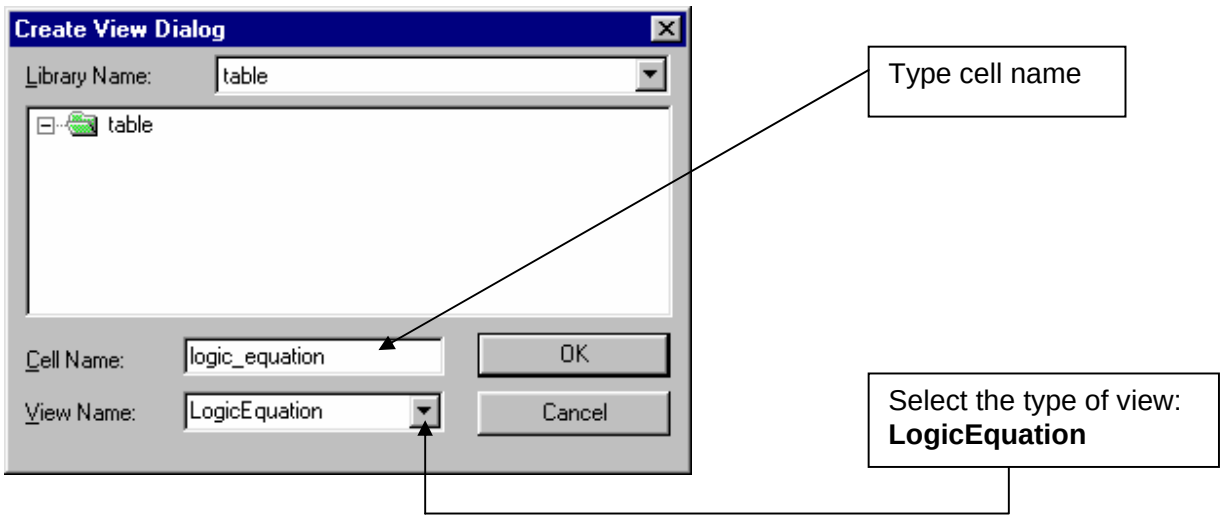


RAM table has the write enable port. It controls the status of ROM whether write mode or read mode. And, RAM does not need values of address port. The usage of in-clock port and out-clock port is similar with RAM table.

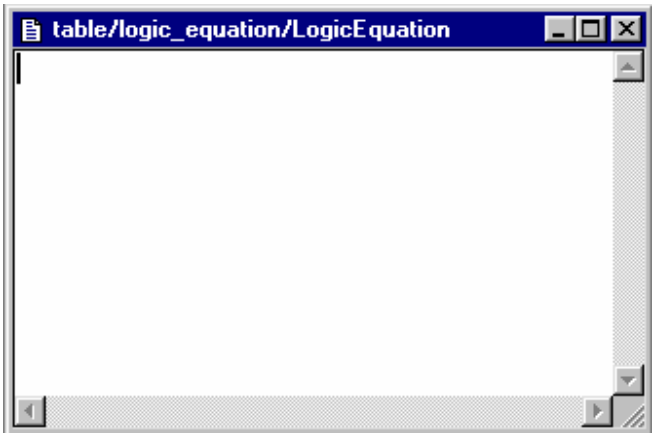
Appendix V: Logic Equation

In this section, the procedure of the Logic Equation's usage is described.

Choose ***File→New*** from pull-down menu of SchEd. Then, the **Create View Dialog** dialog box will be shown.



Type cell name at the **"Cell Name"** text input box. And, choose the type of view as LogicEquation. Then, the following dialog box will be shown.



The input method of logic equation is the text at **LogicEquation** view. The following table shows the list of operator that is available at logic equation design.

Operator	Meaning
+	OR
*	AND
!	NOT

SchEd will recognize the line which is starting with **"//"** into comment.

The following shows the syntax of logic equation.

LogicEquation :=header cell_name interface contents

Header	:=|LogicEuation-V100|
Cell_name	:=CELLNAME string
Interface	:=input outputs variables
Inputs	:=INPUT (signal_name)*SEMI_COLON
Outputs	:=OUTPUT(signal_name)*SEMI_COLON
Variables	:=VARIABLE (local_signal_name)*SEMI-COLON
Contents	:=CONTENTS end_of_line (assignment_expr)*END
Assignment_expr	:=signal ASSIGN_OP expression SEMI-COLON
Expression	:=and_expr
	| or_expr expression
Or_expr	:=primary_expr OR_OP primary_expr
And_expr	:=unary_expr
	|and_expr AND_OP unary_expr
Unar_expr	:=NOT_OP primary_expr
	|primary_expr
Primary_expr	:=signal
	|1|0

	|(expression)
Signal	:=[signal_name | local_signal_name]
Comment	:=//string end_of_line
OR_OP	:= +
AND_OP	:= *
NOT_OP	:=!
SEMI_COLON	:= ;

Note:
-. The width of operand signal between left-hand side and right-hand side should be same.
-. You can only assign the value of result signal.
-. You do not need to type the header of Logic Equation because SchEd automatically save it when it is closed.

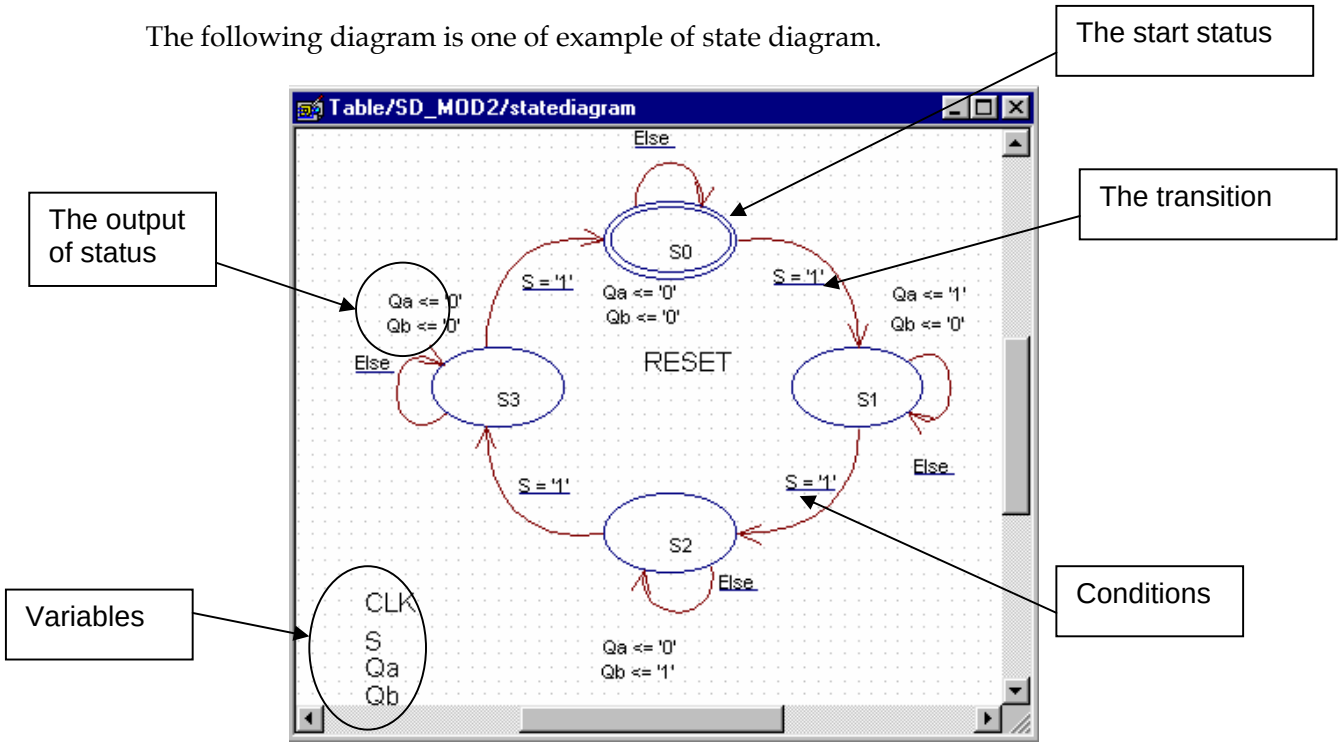
The following is an example of Logic Equation:

```
// This is comment line
CELLNAME    LESample;
INPUTS              data[3:0];
OUTPUTS         out1[3:0];
Variables           tmp1, tmp2;
CONTENTS
tmp1 = data[3] + data[2];
tmp2 = data[1] * data[0];
out1[3] = !tmp1;
out1[2] = !tmp2;
out1[1] = 1;
out1[0] = 0;
END
```

Appendix VI: State Diagram

Some of full-down menu of SchEd will be changed when you design the schematic with state diagram at State Diagram view.

The following diagram is one of example of state diagram.



Note: To simulate the state diagram, you should extract to VHDL netlist by choosing **Tools → Extract → VHDL** *from pull-down menu of SchEd.*

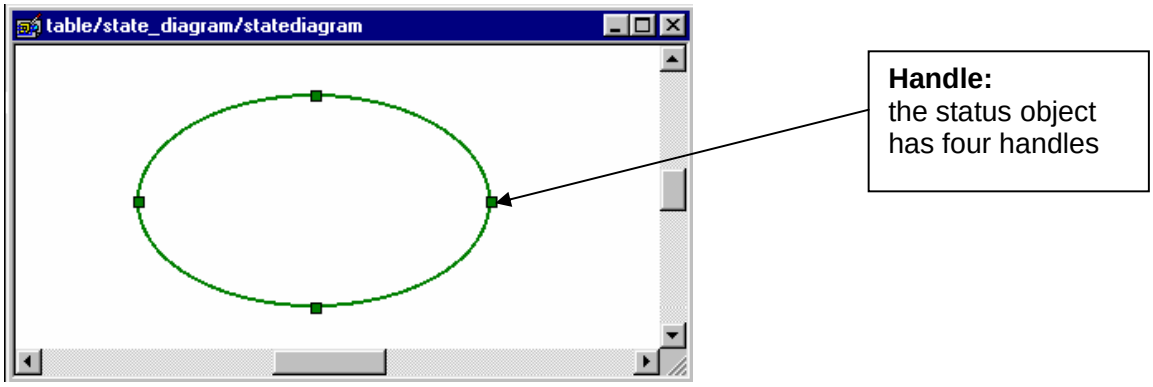
The following is description concerning to the new menu that are only appeared at State Diagram view.

Add→State

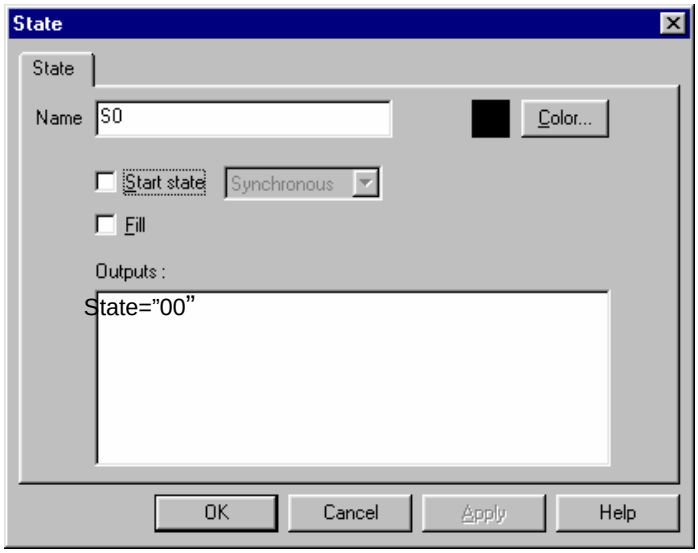
Command Description

It adds the state object into **State Diagram** view.

Usage & Dialog Box



Choose a handle and drag the mouse to a desired location and release it. Then, you can make a status object's shape. Double click the status object. Then, State dialog box will be shown.



Name: It shows the name of selected status object. You can change the name by typing.

Color: It shows the color of selected status object. The default color of it is black. You can change the color by clicking ***Color..*** button. Then, **Color** dialog box will be shown. Choosing a color at **Color** dialog box.

Start state: If you check on it, then selected status will be the start of state diagram and change the boundary into double-line.

Synchronous: You can set the status into synchronous.

Asynchronous: You can set the status into asynchronous.

Fill: If you check on it. Then, selected status will be filled. When the filled status is current working, the output will be the filled status.

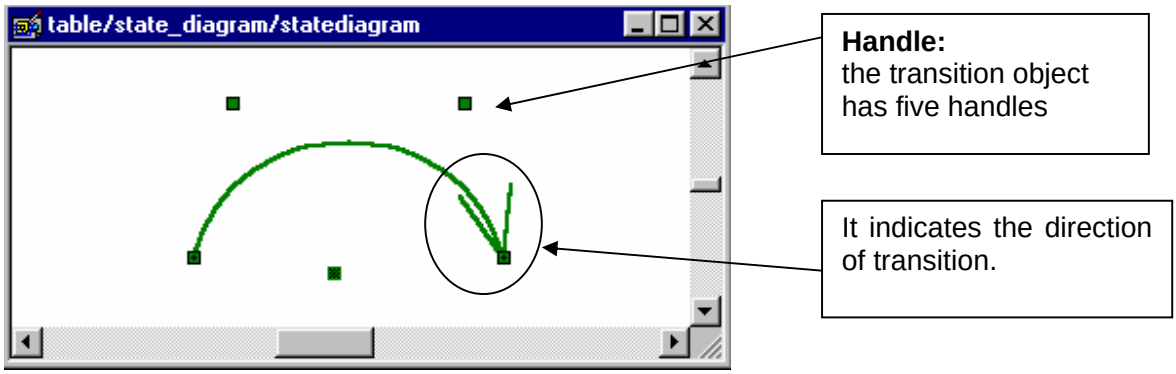
Outputs: Type the output value of state.

Add → Transition

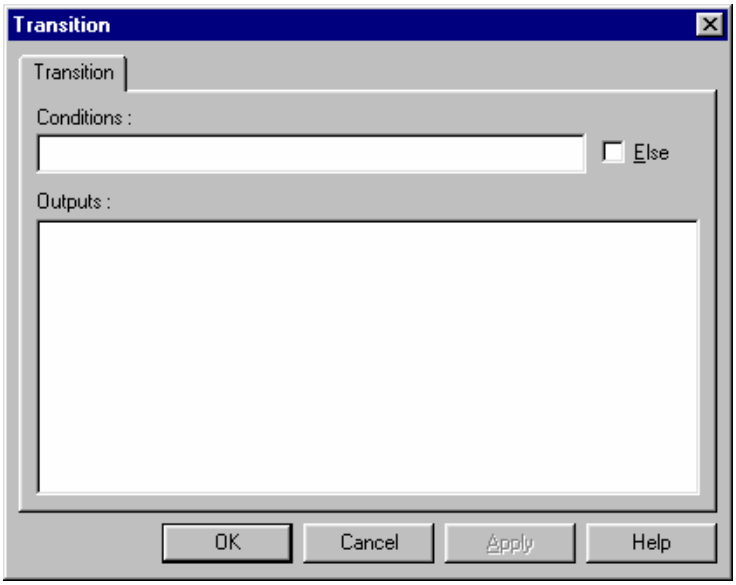
Command Description

It adds the transition object into **State Diagram** view.

Usage & Dialog Box



Choose a handle and drag the mouse to a desired location and release it. Double click the transition object, **Transition** dialog box will be shown.



You can the condition of transition at **Transition** dialog box.

Conditions: Set the condition of transition.

Else: If you check on this box, you could set the state for else condition.

Outputs: Type the output value of state.

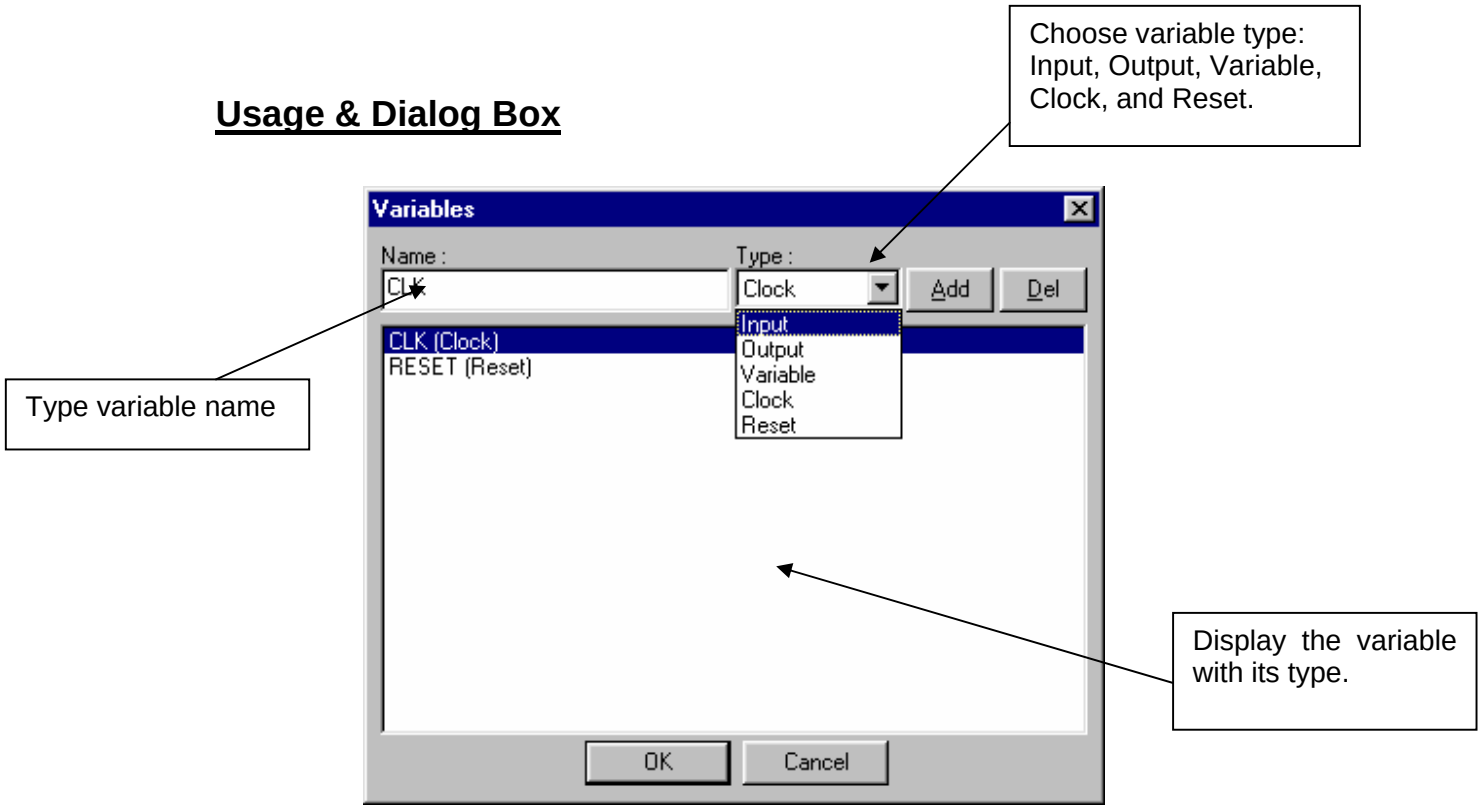
Note: SchEd will display the transition condition with the underline on State Diagram view.

Add → Variables…

Command Description

It adds the variables.

Usage & Dialog Box



Tools → Verify

Command Description

It checks the state diagram whether it contains error or not.