



# **Module Setting**

© Norlinvest Ltd, BVI. Visionics is a trade name of Norlinvest Ltd. All Rights Reserved.

No part of the Module Setting document can be reproduced in any form or by any means without the prior written permission of Visionics. Module Setting document is subjected to change without notice. Visionics will make changes in a manner that will not affect dependent systems.

Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Visionics.

Visionics, EDWinXP, Docone, EDCOMX, SimWinXP and Mixed Mode Simulator and their respective logos are trademarks or registered trademarks of Visionics. Unauthorized duplication of this work may also be prohibited by local statute.

**Disclaimer:** Information in this publication is subject to change without notice and does not represent a commitment on the part of Visionics. The information contained herein is the proprietary and confidential information of Visionics or its licensors, and is supplied subject to, and may be used only by Visionics's customer in accordance with, a written agreement between Visionics and its customer. Except as may be explicitly set forth in such agreement, Visionics does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Visionics does not warrant that use of such information will not infringe any third party rights, nor does Visionics assume any liability for damages or costs of any kind that may result from use of such information.

---

## Contents

<b>CONTENTS .....</b>	<b>2</b>
<b>MODULE SETTINGS .....</b>	<b>4</b>
GENERAL SETTINGS .....	4
Steps for undo/redo.....	4
Angle Snap.....	5
Grid Setting.....	5
Symbol Line Width .....	5
Unit Setting.....	5
SCHEMATIC EDITOR SETTINGS.....	7
Thickness of Wires/Buses.....	7
Page Size (Default) .....	7
Setting colors in Schematic Editor.....	7
Packaging details of components.....	7
Labeling scheme .....	7
Setting component values.....	8
Predefined Net Name.....	9
Predefined Wire Label.....	9
MIXED MODE SIMULATOR SETTINGS .....	10
Set the component values.....	10
Setting the Ground .....	10
Simulation Function.....	10
EDSPICE SIMULATOR SETTINGS .....	11
Setting the values for different components.....	11
LAYOUT EDITOR SETTINGS .....	12
Via Size .....	12
Airgap Size .....	12
Board Size (Default) .....	12
Change the color of the various layout items such as background, board outline, etc.....	13
Change the color/ name of the layers. ....	13
Labeling scheme .....	13
User Defined Lay Prefix .....	14

---

Routing Layers & Direction.....	14
Via Rules .....	14
Trace Rules .....	14
Clearances .....	14
Autorouter.....	15
Locked Edit.....	15
Layout Component Placement.....	15
ELECTROMAGNETIC ANALYZER SETTINGS .....	16
Settings required to conduct the analysis. ....	16
THERMAL ANALYZER SETTINGS.....	16
Settings required to conduct the analysis. ....	16
FABRICATION MANAGER SETTINGS.....	17
Set net for the copper pour area. ....	17
Change the color/ name of print layers. ....	177
Generate Gerber Data.....	17
Generate NC Drill Data .....	18
Generate PCB Assembly outputs .....	18
Bare Board Testing outputs.....	18
SYMBOL EDITOR SETTINGS.....	19
Symbol Line Width .....	19
Simulation Function.....	19
LIBRARY BROWSER/ LIBRARY EXPLORER SETTINGS.....	20
Path setting for the library files.....	20

---

---

## Module Settings

The topics covers briefly the necessary settings required to be done in individual modules.

- **General Settings**
- **Schematic Editor Settings**
- **Mixed Mode Simulator Settings**
- **EDSpice Simulator Settings**
- **Layout Editor Settings**
- **Thermal Analyzer Settings**
- **Electromagnetic Analyzer Settings**
- **Fabrication Manager Settings**
- **Symbol Editor Settings**
- **Library Browser/ Library Explorer Settings**

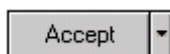
### General Settings

Most of the general settings common to almost all the editors of EDWinXP are set from the **Options Setup** window.

- i. Open **EDWinXP project explorer**
- ii. Right click **System → Options**.

#### Steps for undo/redo

Select **Options Setup → General → Recovery & Units** .The Undo/ Redo level may be set to vary from 1 to 10 for all the editors except Library Editor, where it can vary from 1 to 30. As the number of level increases, the system may slow down and hence the user is provided with an option to set it as per his need. The user can also switch On or Off the Undo/ Redo feature from all the editors. Now to effect this change as the default setting, select the “**Save & Exit**” from the



dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the **Apply** button and exit.

---

## Angle Snap

The Angle snap determines the steps for angular rotation of a component, component name, notes etc while relocating an item. The angle snap set from the options window will be the default angle snap for all the editors. The angle snap may be changed afterwards from each of the editors individually as per the requirement.

- i. Select **Sizes → General → Angle Snap** from the **Options Setup** window.
- ii. Now to effect this change as the default setting, select the “**Save & Exit**” from the



dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the **Apply** button and exit.

## Grid Setting

Grid helps to align the components well. The grid value set from the options window will be the default grid value for all the editors. The grid value may be changed afterwards from each of the editors individually as per the requirement.

Now to effect this change as the default setting, select the “**Save & Exit**” from the



dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the **Apply** button and exit.


## Symbol Line Width

The symbol line width of the symbols in the schematic diagram may be changed from the **Project Properties** window. The project properties window is invoked from the project explorer. Right click on the task **Project** and select **Project Properties**. The **Project Properties** window opens up. Select the **Advanced** tab. The symbol line width may be set as required in terms of dimension or percentage. Note that this setting is only for the current schematic diagram. In order to change the symbol's line width permanently, it must be edited from the Symbol editor.

## Unit Setting

The Unit setting in the options window will apply to all the editors. The unit may be changed from each of the editors as per the users need but there is a possibility to override the unit setting of individual editors.

---

To override the unit settings, from the options window, Select **General** → **Recovery & Units**. Check the option **Override Editors with System Units**. Now to effect this change as the default setting, Select the “**Save & Exit**” from  dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the **Apply** button and exit.

VISIONICS

---

## Schematic Editor Settings

### Thickness of Wires/Buses

The thickness of wires/buses in the schematic diagram may be changed from the Project properties window. The project properties window is invoked from the project explorer. Right click on the task Project and select Project Properties. The project properties window opens up. Select the **Advanced** tab. The wire/bus width may be set as required in terms of dimension or percentage. Note that this setting is only for the current schematic diagram.

- i. In order to change the default wire width, it must be set from the options window. The Options setup window is invoked from the project explorer. Right click on the task System and select Options.
- ii. From the options setup window, select **Sizes → Schematic → Bus → Wire Width**. Set the value as required. Now to effect this change as the default setting, select the

“**Save & Exit**” from the  dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the **Apply** button and exit.

### Page Size (Default)

The default page size is set from the options window.

- i. The Options setup window is invoked from the project explorer.
- ii. Right click on the task **System** and select **Options**. Select **Format → Schematic Page**.
- iii. Select the required size.
- iv. Now to effect this change as the default setting, select the “**Save & Exit**” from the

 dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the **Apply** button and exit.

### Setting colors in Schematic Editor.

The color of almost all the items present in the schematic diagram may be changed from the options window and can be set as the default color.

---

- i. Right click on the task **System** and select **Options**.
- ii. Select **Color & Layer → Schematic**.
- iii. Set the color by picking the required color from the color palette and filling it against the required items listed.
- iv. Now to effect this change as the default setting, select the “**Save & Exit**” from the



dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the **Apply** button and exit.

### Packaging details of components

The packaging details may be manipulated as per the user need.

- i. Open **EDWinXP project explorer** and select **System → Options → Packaging Pref.**
- ii. The component name, pinout details and the component description may be switched On/ Off. The font and size of these texts may be selected.
- iii. Now to effect this change as the default setting, select the “**Save & Exit**” from the



dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the **Apply** button and exit.

### Labeling scheme

Labels are attached to symbols when placed on to the schematic. The font and size of the labels may be set as required.

Open **EDWinXP project explorer** and select **System → Options → Comp. Label Scheme → Schematic** and edit the settings as required. Now to effect this change as the default setting,

select the “**Save & Exit**” from the  dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the **Apply** button and exit.

### Setting component values

The component values may be set either from the schematic editor module or from the mixed mode simulator itself. The values of certain components such as resistors, capacitors, etc may be set from the schematic editor.

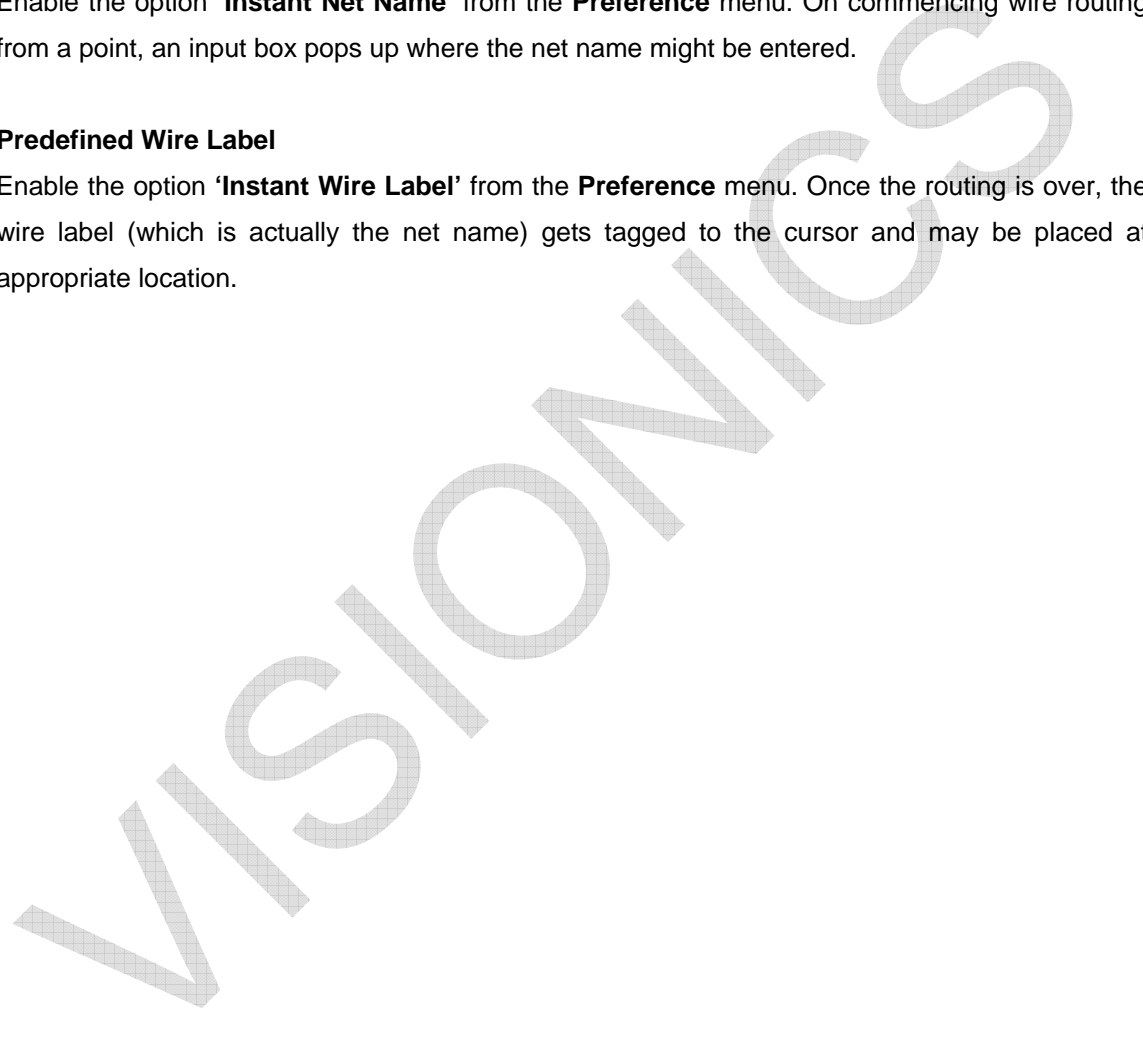
Select the option tool **Add/Change Component values** (function tool) **Add /Change Component Text** and click on the component. An input box opens up where the component values separated using commas may be entered. The input voltages for generators etc. are set in a similar fashion from either the schematic editor or from mixed mode simulator.

#### **Predefined Net Name**

Enable the option '**Instant Net Name**' from the **Preference** menu. On commencing wire routing from a point, an input box pops up where the net name might be entered.

#### **Predefined Wire Label**

Enable the option '**Instant Wire Label**' from the **Preference** menu. Once the routing is over, the wire label (which is actually the net name) gets tagged to the cursor and may be placed at appropriate location.



---

## Mixed Mode Simulator Settings

### Set the component values

The component values may be set either from the schematic editor module or from the mixed mode simulator itself. The component values are set using the option tool, Set Parameter, of the first function tool of mixed mode simulator. Select the tool and click on the component. The parameters of the selected component may be edited in the window that opens up. The parameters may be borrowed from the available SPICE parameters or the parameters available from mixed mode parameter library.

The values of certain components such as resistors, capacitors, etc may be set from the schematic editor itself. Select **Add/Change Component values** (option tool) → **Add/Change Component Text** and click on the component. An input box opens up where the component values separated using commas may be entered.

The input voltages for generators etc. are set in a similar fashion from either the schematic editor or from mixed mode simulator.

### Setting the Ground

The GND may be set in two ways. From the schematic editor, load the GND symbol and place it appropriately in the circuit and complete the connections. Or else from the mixed mode simulator module, place the GND reference marker at appropriate point in the circuit using the function tool Set Ground.

### Simulation Function

Select **Assign Simulation Model** (option tool ) → **Set Parameters | Models** (function tool). Click on the symbol to open a window. Click the button **Assign Simulation Function** to assign the simulation function as required.

To change the simulation function of the symbol permanently, open the symbol in the symbol editor for editing. Select the symbol for editing in the Symbol Editor. Select the option tool **Symbol Properties** from the function tool **Properties**. In the window that opens up, select the simulation function as required.

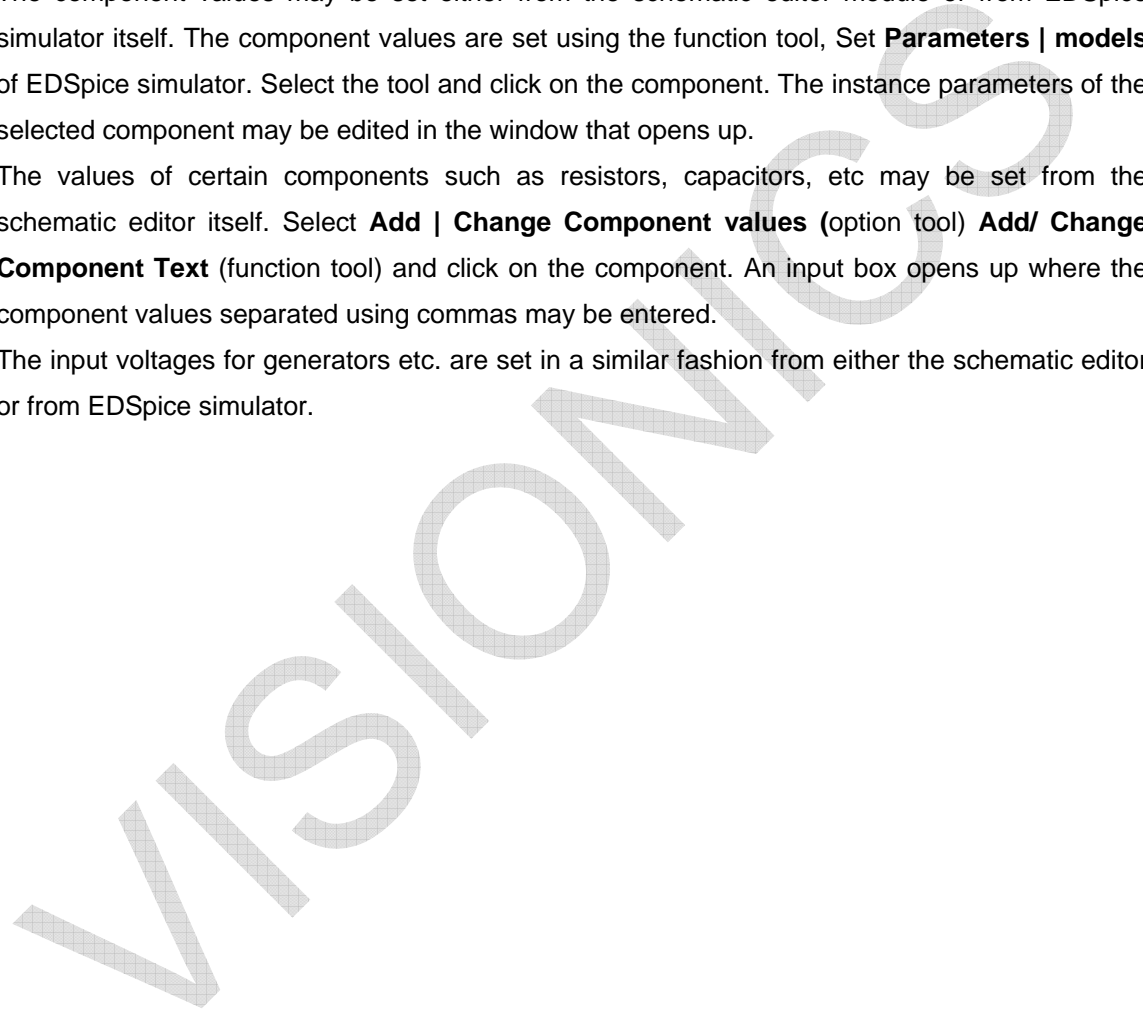
## EDSpice Simulator Settings

### Setting the values for different components

The component values may be set either from the schematic editor module or from EDSpice simulator itself. The component values are set using the function tool, Set **Parameters | models** of EDSpice simulator. Select the tool and click on the component. The instance parameters of the selected component may be edited in the window that opens up.

The values of certain components such as resistors, capacitors, etc may be set from the schematic editor itself. Select **Add | Change Component values** (option tool) **Add/ Change Component Text** (function tool) and click on the component. An input box opens up where the component values separated using commas may be entered.

The input voltages for generators etc. are set in a similar fashion from either the schematic editor or from EDSpice simulator.



---

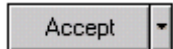
## Layout Editor Settings

### Via Size

Seven types of Vias are available in EDWinXP. Each of the via has different size and the size may be changed as per the user need. To change the size of via, Select **Tools → Via Padstack** from the menu of the layout editor. A window “**Edit Via Padstack**” opens up where all the properties of the via such as size, shape, air gap, etc. may be edited. The setting remains until EDWinXP is closed.

### Airgap Size

From the options setup window, Select **Sizes → Layout → Airgap Size**. The airgap size set from the options window will be the default Airgap Size for all the editors. The airgap size may be changed afterwards from Layout Editor or Fabrication Manager individually as per the requirement. Now to effect this change as the default setting, select the “**Save & Exit**” from the



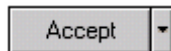
dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the **Apply** button and exit.

### Board Size (Default)

The default board size is set from the options window.

- i. The **Options** setup window is invoked from the **project explorer**.
- ii. Right click on the task **System** and select Options. Select **Format → Layout board**.
- iii. Select the required size.
- iv. Now to effect this change as the default setting, select the “**Save & Exit**” from the



dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the **Apply** button and exit.

---

### Change the color of the various layout items such as background, board outline, etc.

The color of almost all the items present on the board may be changed from the options window and can be set as the default color.

- i. Right click on the task **System** and select **Options**.
- ii. Select **Color & Layer → Layout**.
- iii. Set the color by picking the required color from the color palette and filling it against the required items listed.
- iv. Now to effect this change as the default setting, select the “**Save & Exit**” from the



dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the **Apply** button and exit.

### Change the color/ name of the layers.

The color and name of all the layers in the layout may be changed from the options window and can be set as the default color and name for the layer.

- i. Right click on the task **System** and select **Options**.
- ii. Select **Color & Layer → Layers**.
- iii. Set the color by picking the required color from the color palette and filling it against the layers listed.
- iv. Appropriate color may be assigned to the category by keeping the **Shift** key pressed and click on the colors in the palette. This will randomly change the color of the category to the color selected. To know the right color the category is assigned, press CTRL and click on the category. This will highlight the color used in the color palette. The layer name may be edited directly in the edit box.
- v. Now to effect this change as the default setting, select the “**Save & Exit**” from the



dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the **Apply** button and exit.

### Labeling scheme

The font and size of the prefix attached to the symbol when placed on the board may be set as required.

---

- i. Open EDWinXP project explorer and select **System → Options → Comp. Label Scheme → Layout** and edit the settings as required.
- ii. Now to effect this change as the default setting, select the “Save & Exit” from the



dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the APPLY button and exit.

### User Defined Lay Prefix

The prefix for the component to be loaded on the board may be set as per the user need.

On enabling the option **Preference → User Defined Lay.Prefix**, an input box pops up where the prefix may be entered. The components loaded here after will have this pre-defined prefix.

**Note:** All the settings given below are set in Default Design Rules Setup window, which is invoked, from EDWin Project Explorer. These settings may be set as default, for the current Project, or for the current Circuit. Most of the options set may or may not be used while preparing the design. To use these settings certain options may be set within the Layout Editor or corresponding modules.

### Routing Layers & Direction

The layers and directing of the traces to be used may be set in **Default Design Rules Setup** window.

### Via Rules

The type of Vias to be used and the number of vias may be specified in the **Default Design Rules Setup** window.

### Trace Rules

The minimal and maximal trace parameters like width, airgap etc. may be set different using **Default Design Rules Setup** window.

### Clearances

The clearance value i.e. the Pad to Pad distance, Trace to Pad distance, Trace to Trace distance and the Signal Trace Check Width may be specified here.

**Autorouter**

Autorouter settings parameters may be specified in this **Default Design Rules setup** window.

**Locked Edit**

This option is particularly used at the final stage of the design to prohibit further edition to the design, ie. Certain operations may be locked using this option.

**Layout Component Placement**

Certain Design rules related to components may be set in the **Design Rules Setup** window. A special option i.e. Component snap may be specified in this window. When specified, this value is used by the layout editor but this value set can be surpassed by **Ctrl** key.

How to use an alternate Snap value (for Layout Components only) using design rule setup dialog: Open **Design Rule** dialog from EDWin Project Explorer and enter the snap value say .1000" in Layout component Placement. In Layout Editor, set the snap value to .0500". Now try relocating the components, you may find that the components move with Snap value .1000". To use the Snap value selected in Layout editor, press CTRL key and try to relocate. Now the Layout Components move with Snap value .0500".

---

---

## Electromagnetic Analyzer Settings

### Settings required to conduct the analysis.

The settings required to conduct the analysis can be set

- i. **EDWinXP Project Explorer → PCB Layout**
- ii. **Select Board Analyzer (task list) → Analysis (menu) → Settings.**
- iii. Set the parameters related to the board, field values and the display options.
- iv. In addition to that, the electrical parameters, net parameters and trace parameters may be edited before the analysis, using the function tools provided.
- v. Select **Analysis → Run Electromagnetic Analysis** from the menu to run the analysis.

## Thermal Analyzer Settings

### Settings required to conduct the analysis.

- i. The settings required to conduct the analysis is set from the menu **Analysis → Settings.**
- ii. Set the parameters related to the board, cooling parameters, temperatures and the display options.
- iii. In addition to that, each of the component parameters may be edited before the analysis, using the **Component Parameters (function tool)**.
- iv. Select this tool and click on the required component to open a window where the parameters may be edited.
- v. Select **Analysis → Run Thermal Analysis** from the menu to run the analysis.

---


## Fabrication Manager Settings

### Set net for the copper pour area.

- i. Once the editing mode is set as **Tools(menu) → Copper**, the tool to select the net for the copper gets enabled.
- ii. Select View | Toolbars | Sizes from the menu to obtain the tool, Select Net.
- iii. Select the required net, from the drop down list containing all the net names
- iv. Now enable the tool for defining the copper pour area from the toolbar.

### Change the color/ name of print layers.

The color and name of the print layers may be changed from the options window and can be set as the default color and name for the layer. Right click on the task System and select Options. Select Color & Layers | Layers. Set the color by picking the required color from the color palette and filling it against the layers listed. Appropriate color may be assigned to the category by keeping the SHIFT key pressed and click on the colors in the palette. This will randomly change the color of the category to the color selected. To know the right color the category is assigned, press CTRL and click on the category. This will highlight the color used in the color palette. The layer name may be edited directly in the edit box. Now to effect this change as the default setting,

select the "Save & Exit" from the  dropdown button before exiting from this window.

**Note:** If this change is just for the current project, then click the APPLY button and exit.

### Generate Gerber Data

- i. Open **Fabrication Manager**.
  - ii. Select **Fabrication-> Setup-> Gerber Plotter Data-> Gerber Artworks**.
  - iii. Select the layers for which the Gerber data is to be generated.
  - iv. Click the **Execute** button.
  - v. You may notice that the gerber files are generated for each layer with an extension .GBR file. Click the **Execute** button and close the window.
-

- vi. Select **File-> Gerber Viewer Setup** to open a window.
- vii. Select the gerber file from '**Gerber ASCII File**' and click the **Preprocess** button to preprocess the file.
- viii. To view, Select **Tools-> Gerber View**.

#### **Generate NC Drill Data**

- i. Open **Fabrication Manager**.
- ii. Select **Fabrication-> Setup-> NC-Drill Data-> Output Contents**.
- iii. Click the **Execute** button to open a window.
- iv. Check the options to generate the drill database.
- v. Click **Execute** button and close the window.
- vi. Select **Tools-> Template (Notes/Dim)** for viewing.

#### **Generate PCB Assembly outputs**

- i. Open **Fabrication Manager**.
- ii. Select **Fabrication-> Setup-> PCB Assembly Outputs**.
- iii. Select either **Generic** or **IPC-D-355** format data output.

#### **Bare Board Testing outputs**

- i. Open **Fabrication Manager**.
- ii. Select **Fabrication-> Setup-> Bare Board Test Outputs**.
- iii. Select either **Generic** or **IPC-D-356A** format data output.

---

## Symbol Editor Settings

### Symbol Line Width

The symbol line width of the symbols in the schematic diagram may be changed from the Project properties window.

- i. Invoke **Project Properties** from Project Explorer.  
Or  
Right click on the task Project and select **Project Properties**.
- ii. The project properties window opens up. Select the **Advanced** tab. The symbol line width may be set as required in terms of dimension or percentage. Note that this setting is only for the current schematic diagram. In order to change the symbol's line width permanently, it must be edited from the Symbol editor.
- iii. Select the symbol for editing in the Symbol Editor.
- iv. Select the **Item Properties** (option tool) from the **Properties** (function tool).
- v. Select **Edit → Select All** from the menu.
- vi. Right click on the workspace to obtain a pop up menu.
- vii. Select **Properties → Symbol Items** and change the line width in the properties window.
- viii. Save the symbol and the part to the user created library.

### Simulation Function


In order to change the symbol's simulation function permanently, select the symbol for editing in the Symbol Editor. Select the option tool Symbol Properties from the function tool Properties. In the window that opens up, select the simulation function as required. Save the symbol and the part in the user created library. It can also be set from the mixed mode simulator but it applies only to the current project.

- i. Select **Set Parameters | Models** (function tool) → **Assign Simulation Model** (option tool).
- ii. Click on the symbol to open a window.
- iii. Click **Assign Simulation Function** button to assign the simulation function as required.

## Library Browser/ Library Explorer Settings

### Path setting for the library files.

The default path for the library files may be set from the options window.

- i. Right click **System → Options**.
- ii. Select **Files → Folders**.
- iii. Set the path.
- iv. Now to effect this change as the default setting, select the “**Save & Exit**” from the  dropdown button before exiting from this window.
- v. The library path may be set from the menu File/ Change library Path of Library Explorer/ Browser but this holds only for the current project.

**Note:** If this change is just for the current project, then click the **Apply** button and exit.