

# **EDWinXP Getting Started**



© Norlinvest Ltd, BVI. Visionics is a trade name of Norlinvest Ltd. All Rights Reserved. No part of the EDWinXP Getting Started document can be reproduced in any form or by any means without the prior written permission of Visionics. EDWinXP Getting Started 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

CONTENTS	2
SCHEMATIC EDITOR	4
Loading the Components	4
Loading Components using Component Browser	4
Packaging the Components	5
Routing the wire connections	5
Viewing the output	5
Entering Page Notes and Design Notes	6
Saving the Project	7
Printing the Schematic Diagram	7
SIMULATION	
MIXED MODE SIMULATION	8
Steps for Mixed Mode Simulation	8
Assign Component Parameter Values	9
ANALYSIS	9
Bias Point Calculation	10
Transient analysis	10
Parameter Analysis	10
Fourier Analysis	13
DC Sweep Analysis	
AC Sweep Analysis	16
Monte Carlo Analysis	16
Sensitivity Analysis	19
EDSPICE SIMULATOR	
Steps for EDSpice Simulation	
ANALYSIS	
Transient Analysis	
Small Signal AC Analysis	
DC Transfer Function Analysis	
Distortion Analysis	25
Operating Point Analysis	
Noise Analysis	
DC / AC Sensitivity Analysis	
Transfer Function Analysis	
Pole- Zero Analysis	

PCB LAYOUT	5
Define board outline	5
Relocating the components	5
Routing the Components 3	6
Auto routing using Arizona Auto router 3	7
Testing the Board	9
BOARD ANALYZERS 4	2
Thermal Analyzer	.2
Steps for Thermal Analysis 4	2
Electromagnetic Analyzer 4	4
Steps for Electromagnetic Analysis 4	4
Signal Integrity Simulation4	.7
Steps for Signal Integrity Analysis 4	7
Field Analyzer	0
Field Analyzer - Operation5	0
FABRICATION MANAGER	5
Introduction5	5
Gerber Output	6
Preview the GERBER Data	7
Gerber Mechanical Plots 5	9
Introduction to NC Drill 6	0
NC Drill Output Parameters 6	0
Preview NC- Drill data: 6	2
Introduction to PCB Assembly outputs6	2
Introduction to Bare Board Testing outputs6	3
Generate Bare Board Test outputs6	4

# Schematic Editor

Invoke Schematic Editor from Project Explorer in the following ways.

Right click Page [Main Page] and Select Edit Page from the list.

Or

Select Edit Page from the task list or from the task toolbar.

- 1. Turn ON Grid by enabling the grid from the dropdown, in Standard Toolbar.
- 2. The value for grid may be selected from the drop down list as .1000".
- 3. Set Snap value to .0500" for better placement of the components.

#### Loading the Components

Select **Tools** → **Components** toolbar and right click on the workspace to popup a set of Component editing tools. By default, **relocate** tool is enabled.

#### Loading Components using Component Browser

- 1. The component browser is launched from View → Schematic → Browser.
- The required components can be obtained either by using the Browse option in the window or by Search for the element in the quick menu
- 3. Using the **Place** button in the bin we can place the components in the workspace.

Note: Hotkeys can also be assigned to element names by which quick launch of components is possible.

- 4. Voltage source, Ground and signal generators can be obtained from the Component Browser. These are external sources (which will not be housed on the PCB) and hence connections to them can be shown with the help of connectors. Select connectors from the component browser, then select List Connector and select the required header from the list appearing.
- 5. Load the connector and using **Repeat Component** function tool, create the required number of instances. Similarly load **SPL0** (Hotkey **G**).
- 6. To relocate the component, right click and select tool **Relocate**. Position the part where required and click the mouse to place component

## Packaging the Components

If the PCB of the circuit has to be designed, then the components need to be **packaged**. Packaging is nothing but information that the system requires identifying the Package associated to Part placed on the Schematic. Only packaged components are automatically **front annotated** to layout.

#### Auto Packaging

Select **Tools** → **Component** → **Pack/Unpack Component** → **Auto packaging** from the option tool. An Automatic Packaging window appears Click **Execute** button.

Automatic Packaging 🛛 🗙					
-Name pr	efixes fou	ind:			
Prefix in	Part: P	ack:	Change to:		Start N 🔺
T		<b>N</b>	Т		1
C		ব	C		1
S		ব	S		1
U		ব	U		1 💌
					Þ
			Execute		Cancel

#### Routing the wire connections

- 1. Connections between components are established using wires and buses.
- 2. Select **Tools → Connections** to enable a set of tools required for routing.

智 Tip: Adjust zoom precision to view the terminals being wired. Turn ON grid and snap to help in positioning the wires.

- 3. Before creating the connections, enable **Preferences/Instant net name** and **Preferences/Instant wire label.**
- 4. Right click on the work space and select **Connections**.
- 5. Enable **Pin to Pin** option tool whenever an entry or pin is clicked for making a connection. This ensures that the connection is made to the pin.
- 6. An input box appears prompting the user to enter the name of the net, enter the name most suited for the net.

#### Viewing the output

- Select Tools → Instruments Set wave form Contents (function tool) → Voltage Waveform (option tool).
- 2. Click on the nets where the voltage waveforms to be displayed.

## **Entering Page Notes and Design Notes**

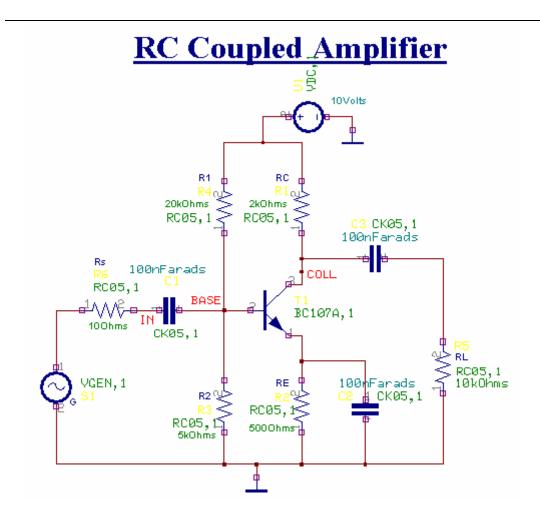
- 1. **Page notes** appear only on the current page (e.g. Page No.) of the project whereas **Design notes** appear on all the pages (e.g. Project name) of the selected circuit.
- Select Tools → Page Notes toolbar to enable a set of tools for creating page notes. Right click and enable

Edit & Add Text	Block 🛛		
Font name:	Vector Font		
Font size:	0.0600''		
Font pitch:	0.0120''		
Text: B Z U O S RC Coupled Amplifier			
	Accept Cancel		

Create PN Graphic Item tool → Create Text to open a window Edit & Add Text block. Enter the required notes and accept it.

The text gets tagged to the cursor. Now place the text at desired position. To edit this text, Press **Ctrl** key and click on the text to select it. Perform the various operations using bullets.

Schematic Circuit designed is given below.



#### Saving the Project

In order to prevent loss of work, save the project periodically using the following steps.

- 1. On the File menu of Project Explorer, Click Save Project or Press Ctrl + S.
- 2. The first time Save Project is selected, a dialog pops up prompting to enter the name for the project.
- 3. Type in the name of project as **RC Coupled Amplifier.EPB**.

#### Printing the Schematic Diagram

To print the Schematic diagram,

- 1. Select **File → Print page** from the main menu of Schematic Editor.
- 2. A window pops up with the preview of circuit diagram.
- 3. Click on 'Fit graphic to one page with margin' icon. Click on Print Page icon to print the page.

# Simulation

# **Mixed Mode Simulation**

#### **Steps for Mixed Mode Simulation**

#### Preprocess the circuit

The simulator analyzes the schematic first and checks for the simulation function. It generates the data along with the statistics of the circuit. Data includes the number of digital and analog nets, list of symbols, which may be simulated along with their simulation functions, number of digital inputs and outputs, number of A/D input/ outputs. This is presented as a dialog box. Observe that the analog primitives are assigned negative simulation function.

Click the **OK** button to close this dialog box. The simulator is now ready for making further analysis on the circuit.

Mixed-Mode Simulation Preprocess Result				
Analog Nets : SPL0	8 (3) 💌	Digital Nets :	0	
Dig. Inputs : Dig. Outputs : Input A/Ds : Output D/As : Components : Primitives :	0 0 0 12 5	-2 Capacitor -4 Voltage Source -7 Voltage Generator -1 Resistor -6 NPN Transistor		
Close Preprocess completed				

Note: Any change in the circuit element values or topology gets implemented only if the preprocess is invoked. Preprocess menu is invoked from Simulation menu. Preprocessing the circuit resets the values set.

## **Assign Component Parameter Values**

- Select Component Properties (function tool) → Change Simulation
   Parameters (option tool).Click on the component
- 2. Component Parameter Setup window opens.
- 3. Change the parameter values.
- 4. Click Accept.

# Analysis

For setting up simulation time and Start analysis types,

- 1. Select Simulation → Analysis.
- 2. Set the parameters by selecting General settings from the tree view on the left side of the window.

Setup Simulation Parameters		×	
Analysis Type	ype Parameter Setup		
		Simulation Limits	
General Settings	Ambient Temperature	25	
	Max.Iteration Error	10 m	
🛨 🕂 🎇 Transient Analysis	Iteration Limit	100	
DC Sweep Analysis	Floating No. Precision	4	
AC Sweep Analysis		Min. Displayable Units	
Monte Carlo Analysis	Voltage	mV	
	Current	μΑ	
🦾 🛲 Sensitivity Analysis	Display Iteration Errors		
	Display TP Values	V	
		Accept Cancel	

3. Set the values as

Simulation Limits:	
Ambient Temperature	25
Max. Iteration Error	100m
Iteration Limit	100
Floating No. Precision	4

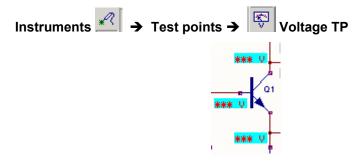
Min. Displayable Unit:	
Voltage	mV
Current	μA

Click Accept button to accept the changes and to automatically switch to Analysis option

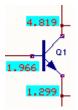
# **Bias Point Calculation**

Bias Point Calculation is used to determine the DC behaviour of the circuit.

 Inorder to view the DC values, fix the test points at the base, collector and emitter of the transistor as shown below. To place the test points, select **Tools →**



- 2. Select Simulation → Analysis.
- 3. A window **Setup Simulation Parameters** opens with the option **Analysis** being highlighted on the left side of the window by default.
- 4. Check Bias Point Calculation check box.



Click **Start** button, and observe that the node voltages are displayed at the locations where test points were placed. Similarly current values can also be viewed by placing the test points at the required nets.

#### **Transient analysis**

Transient analysis is used to view the input and output with respect to time. Inorder to view the performance of the circuit in time frame run transient analysis.

- 1. Select Simulation → Analysis.
- 2. Select **Transient analysis** from tree view.

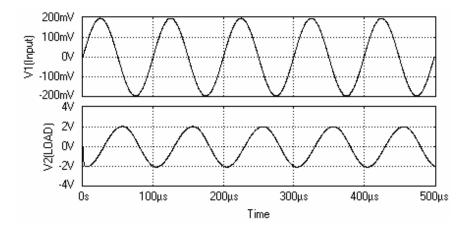
3. Enter following values:

Analog simu	ulation step	time:	2μ
-------------	--------------	-------	----

Simulation time limit: 1m

Setup Simulation Parameters		
Analysis Type	Parameter Setup	
	Analog Sim. Step Time	2μ
General Settings	Sim. Time Limit	1 m
	Initialize LC	Solve
	Display Waveform	
DC Sweep Analysis	Transfer Function Analysis	
AC Sweep Analysis		
Sensitivity Analysis		
		Accept Cancel

- 4. Check **Display waveform** check box.
- 5. Click on Accept to switch into the Analysis option
- 6. Click Start to begin the analysis.



## **Parameter Analysis**

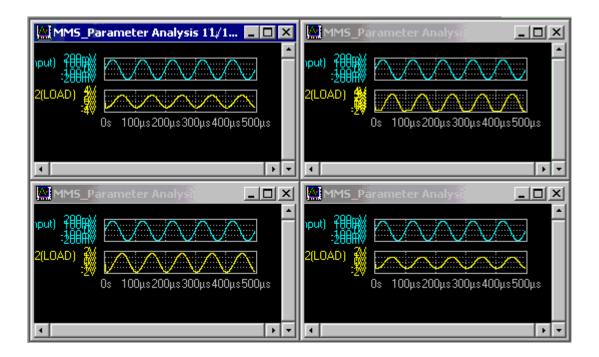
Parameter analysis helps to study the effect of variation of component parameters on the circuit. This analysis calculates the change in the output of a given circuit when a selected parameter of a particular circuit element is varied over a range of values.

- 1. Select Simulation → Analysis.
- 2. Select **Parameter Analysis** from the tree view.
- 3. Select Sweep Variable as Component Parameter type.
- 4. Move the cursor over the resistor and click on it. Selected component name and parameter name appears in the dialog box.
- 5. Set the values for the following parameter as

Start Value	1K Ohm
End Value	3K Ohm
Steps	750 Ohm

Setup Simulation Parameters				
Analysis Type	Parameter Setup			
	Sweep Variable	Comp. Parameter		
🔛 🔛 General Settings	Comp. Name	R1/1		
	Parameter Desc	Resistance [Ohm]		
Transient Analysis	Parameter Name	R		
🚧 Parameter Analysis	Analysis Type	Sweep		
ዓው Fourier Analysis	Sweep Mode	Linear		
DC Sweep Analysis	Start Value	1 kOhm		
	End Value	3 kOhm		
	Steps	750 Ohm		
Monte Carlo Analysis	Display Waveform	<b>N</b>		
Sensitivity Analysis				
		Accept Cancel		

- 6. Click Accept to switch into Analysis option.
- 7. Check Parameter analysis check box.
- 8. Now click **Start** button for analysis to take place. The result may be viewed in the Waveform viewer.



# **Fourier Analysis**

Fourier analysis helps to calculate the total harmonic distortion of analog waveforms generated during transient analysis.

- 1. Select Simulation → Analysis.
- 2. Select Fourier analysis from the tree view.
- 3. Set the parameters in the box as

Fundamental Frequency	10K
No of harmonics (nfreq)	10
Deg. of Polynomial (polydegree)	2

Setup Simulat	ion Parameters			×
Analysis Type		Parameter Setup		
	rsis	Fundamental Frequency	100 k	
🛄 🛄	Settings	No. of Harmonics (nfreq)	10	
-	- nt Analysis	Deg. of Polynomial (polydegree)	2	
- 🎋 Pa	arameter Analysis			
ዓም Fo	ourier Analysis			
	eep Analysis			
— AC Swe	ep Analysis			
	Carlo Analysis			
Sensitiv	rity Analysis			
_				
		]		
			Accept	Cancel

- 4. After setting the parameter click **Accept** button to accept the parameters.
- 5. Check Fourier Analysis and click **Start** button. Fourier analysis results are displayed in a text format.

File Viewer (D:\EDWINXP\JOB\MM5_Fourier.out) - [D:\EDWINXP\.	JOB\ 🗆 🗙
=================== Mixmode Simulation output =	
Project Name : 1_11_06RC Coupled Amplifier Analysis Type : Fourier Analysis Date : 2/11/2006 Time : 10:38:54:690	
Fourier Analysis Temperature : 25 Deg. C	
Fourier Components of Transient Response Net : LOAD	•
1 open document(s).	NUM INS //

## **DC Sweep Analysis**

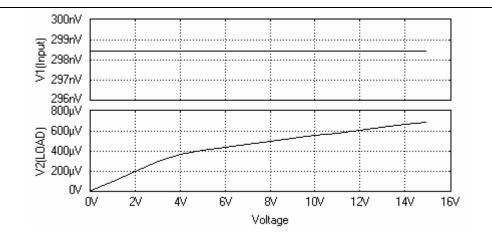
DC Sweep analysis helps to observe the circuit operating parameters by varying the value of a component. One or more sources (current and voltage) may be stepped over a range. Results of the analysis at each source value may then be viewed in the waveform generator.

- 1. Select Simulation → Analysis.
- 2. Select **DC Sweep Analysis** from the tree view.
- 3. Move the cursor over DC supply and click on it. Selected component name and parameter name appears in the dialog box.
- 4. Set the sweep limits as

Start value	0V
End value	15V
Steps	1V

Setup Simulation Parameters		×
Analysis Type	Parameter Setup	
		Component[1]
General Settings	Sweep Variable	Comp. Parameter
	Comp. Name	U1/1
Transient Analysis	Parameter Desc	Voltage [V]
🕀 DC Sweep Analysis	Parameter Name	E
AC Sweep Analysis	Start Value	0V
	End Value	15V
Monte Carlo Analysis	Steps	1 V
🛄 💹 Sensitivity Analysis	Component[2]	
	Sweep Mode	Linear
	Display Waveform	
	,	Accept Cancel

- 5. Check Display Waveform check box to display the waveform.
- 6. Select Accept to switch into Analysis Option.
- 7. Check DC Sweep Analysis check box.
- 8. Click Start



#### AC Sweep Analysis

**AC sweep analysis** is used to calculate the small signal frequency response of the circuit assuming its current biasing.

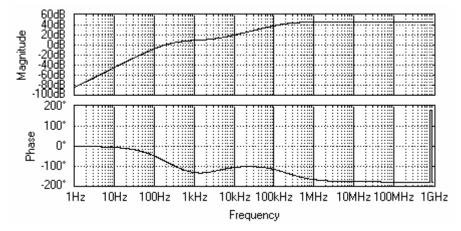
- Select Instruments → Set Reference Points (function tool) → AC IN+ and click on the input net.
- 2. Select Instruments → Set Reference Points (function tool) → AC INoption tool and click on the input net.
- 3. Note: This tool specifies the input given to the input side to be a voltage source or a current source for running the AC Sweep Analysis.
- 4. Select Instruments → Set Reference Points → AC OUT+ → and click on the load net.
- 5. Select Instruments → Set Reference Points → AC OUT- I and click on the load net.
- 6. Note: This tool specifies the output variable taken from the output side to be either the open voltage or the short current.
- 7. Select Simulation → Analysis.
- 8. Select **AC Sweep Analysis** from the tree view.

Setup Simulation Parameters		
Analysis Type	Parameter Setup	
		Sweep Variables
🔛 🔛 General Settings	Start Frequency	1
	End Frequency	16
🛛 🅀 🖓 Transient Analysis	Points/Decade	100
DC Sweep Analysis	Phase Range	[-180,+180]
AC Sweep Analysis	IN Source	Voltage
	OUT Source	Open Voltage
Monte Carlo Analysis		Output
📃 🛄 Sensitivity Analysis	Real Part	
	Imaginary Part	
	Magnitude	<b>N</b>
	Phase	<b>N</b>
	Group Delay	
	Display Waveform	
		Accept Cancel

9. Set the values for the following parameters as

Start frequency	100 Hz
End frequency	1GHz
Points / decade	100
Phase range	[-180, +180]
IN source	voltage
OUT source	Open voltage

- 10. Click **Accept** button to switch into Analysis option.
- 11. Check AC Sweep Analysis check box.
- 12. Click Start for AC Simulation Pass to take place.



#### Monte Carlo Analysis

The Monte Carlo analysis is used for simulations with a given error on different components. This test is very useful for visualizing how the circuit will run with imperfect components as are used in reality.

- 1. Select Simulation → Analysis.
- 2. Select **AC Sweep Analysis** from the tree view.
- 3. Select Instruments → Set Reference Points → Monte Carlo voltage and click on any of the nets.
- 4. Select Monte Carlo Analysis from the tree view.
- 5. Select the component at which the analysis has to be done, by clicking on it.
- 6. Enter the parameters and click **Accept** to accept the parameters.
- 7. Click Start.

Setup Simulation Parameters		×
Analysis Type	Parameter Setup	
		Component[1]
🔛 🔛 General Settings	Comp. Name	R3/1
	Parameter Desc	Resistance [Ohm]
🛨 🕀 🏠 Transient Analysis	Parameter Name	R
DC Sweep Analysis	Tolerance	10
	No. of Samples	10
	Component[2]	
		Function
🦾 🛄 Sensitivity Analysis	Deviation	Uniform
	Output	YMaximum
	Time	10m
		Accept Cancel

The result is displayed in a text file.

📱 File Viewer (D:\EDWINXP\JOB\MM5_Montecarlo.out) - [D:\EDWINXP\ 💶 🗖 🗙
Elle Edit Search Window Help
Mixmode Simulation
MonteCarlo Analysis for Maximum and minimum deviation done
Output for the simulation follows
Nominal output
Component [1]: R3/1 Parameter : 5 k Voltage : 5.40715
1 open document(s). CAPS NUM INS

#### **Sensitivity Analysis**

Sensitivity Analysis is used to calculate the component that is most sensitive to an output reference point.

- 1. Select Simulation → Analysis.
- 2. Select **Sensitivity Analysis** from the tree view.
- 3. Inorder to determine the sensitivity of a component with reference to a net select

Instruments - Set Reference Points - Sensitivity voltage

- 4. Click on any of the nets and place the reference points.
- 5. Select **Sensitivity Analysis** from the tree view.
- 6. Specify a value of tolerance for which the analysis is to be carried out. This is the common tolerance value for all parameters of all components.

Setup Simulation Parameters		2	×
Analysis Type	Parameter Setup		
	%Change in Parameter Value	5	I
🖳 🏪 General Settings			I.
🕂 🌺 Transient Analysis			
🔁 DC Sweep Analysis			I.
Monte Carlo Analysis			
🛲 Sensitivity Analysis			
	,	Accept Cancel	1

- 7. Click **Accept** button to switched into Analysis option.
- 8. Check Sensitivity Analysis check box.
- 9. Click **Start** button, result is displayed in a text file.

File Viewer (D:\EDWINXP\JOB\MMS_Sensitivity.out) - [D:\EDWINXP\J	l ×
Eile Edit Search Window Help	P ×
F=====================================	-
Project Name : 1_11_06RC Coupled Amplifier Analysis Type : Sensitivity Analysis Date : 2/11/2006 Time : 11:54:35:940	
Percentage increase of parameter value : 5%	
DC Sensitivity of output Voltage at net "COLL"	
1 open document(s).	

# **EDSpice Simulator**

# Steps for EDSpice Simulation

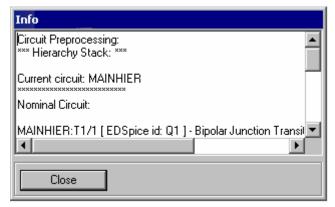
#### Preprocess

Preprocessing confirms whether the circuit is ready for simulation. Preprocessing must be performed at all times, when elements have been added or deleted from the circuit or connectivity between them is changed.

Inorder to preprocess the circuit select **Simulation**  $\rightarrow$  **Preprocess** Click **Close**.

# Analysis

Select Simulation → Analysis



Analysis Setup :		×
Simulator Variables(Options)	Select Analyses	
Initial Node Voltage Guesses Transient Initial Conditions Analysis Transient Analysis Small-Signal AC Analysis D C Transfer Function Analysis Distortion Analysis Operating Point Analysis Noise Analysis	<ul> <li>Transient Analysis</li> <li>Small-Signal AC Analysis</li> <li>DC Transfer Function Analysis</li> <li>Distortion Analysis</li> <li>Operating Point Analysis</li> <li>Noise Analysis</li> </ul>	Setup Setup Setup Setup Setup Setup
DC/AC Sensitivity Analysis Transfer Function Analysis Pole-Zero Analysis	<ul> <li>DC/AC Sensitivity Analysis</li> <li>Transfer Function Analysis</li> <li>Pole-Zero Analysis</li> </ul>	Setup Setup Setup
	Run	Cancel

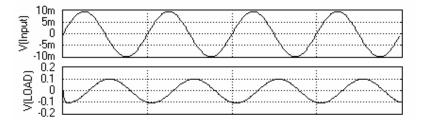
## **Transient Analysis**

- 1. Transient analysis is used to view input and output with respect to time.
- 2. Select Simulation → Analysis.
- 3. Select **Transient Analysis** from the tree view.
- 4. Enter the values as

Step1Final time. 4mStart time0

Analysis Setup:		×	
Simulator Variables(Options)	Set Parameters :		
Initial Node Voltage Guesses	tran 1μ.4m		
Transient Initial Conditions	J		
Analysis	Step	1μ	
🖻 Transient Analysis	Final Time	.4m	
	Start Time	0	
Print(Standard Output)	Max. Step	0	
Plot(Standard Output)	Temperature		
Fourier(Setup)	As Setup		
🗄 - Small-Signal AC Analysis	Fourier		
⊕ DC Transfer Function Analysis	Results	Waveform	
🗄 - Distortion Analysis			
Operating Point Analysis			
Noise Analysis			
- DC/AC Sensitivity Analysis			
- Transfer Function Analysis	J		
Pole-Zero Analysis	Run	Accept Cancel	

- 5. Click **Accept** button after entering the values to automatically switch to Analysis.
- 6. Click **Run** button to start simulation.



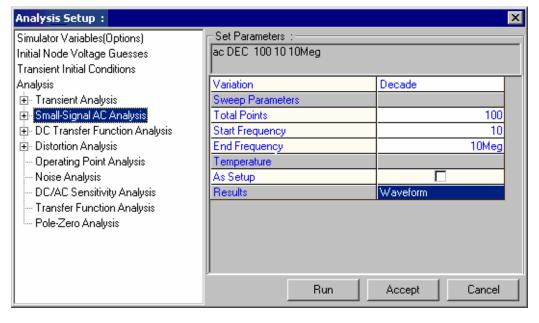
# **Small Signal AC Analysis**

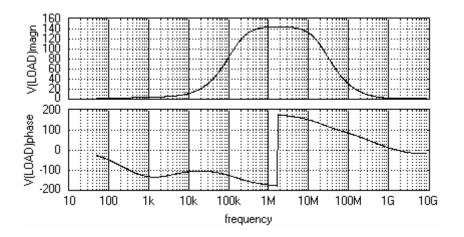
This analysis is used to obtain the small signal AC behaviour of the circuit.

- 1. Select Simulation → Analysis.
- 2. Select Small Signal AC Analysis from the tree view.
- 3. Enter the values as

Total points	100
Start frequency	10 Hz
End frequency	100GHz

- 4. Select **Waveform** for displaying the output.
- 5. Click Accept button after entering the values to automatically switch to Analysis.
- 6. Click Run button to start simulation





# **DC Transfer Function Analysis**

DC Transfer Function Analysis gives the behaviour of the circuit with respect to the varied voltage/current.

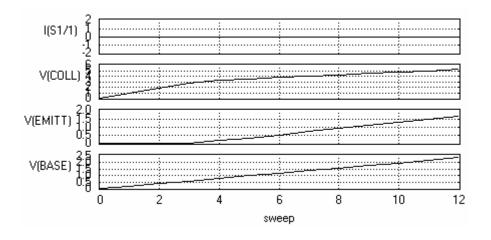
- 1. Select Simulation → Analysis.
- 2. Select DC Transfer Function Analysis from the tree view.
- 3. Set the values as

Start Voltage 0V **Stop Voltage** 12V 1

Step

Analysis Setup:		×		
Simulator Variables(Options)	Set Parameters :			
Initial Node Voltage Guesses	dc V11 0 12 1			
Transient Initial Conditions	]			
Analysis	Sweep Parameters			
吏 Transient Analysis	First Source	01/1 [V11]		
🗄 Small-Signal AC Analysis	Start Voltage	0		
DC Transfer Function Analysis	Stop Voltage	12		
连 Distortion Analysis	Step	1		
Operating Point Analysis	Second Source			
- Noise Analysis	Start Voltage	0		
- DC/AC Sensitivity Analysis	Stop Voltage	12		
- Transfer Function Analysis	Step	1		
Environal Pole-Zero Analysis	Temperature			
	As Setup			
	Results	Waveform		
	J			
	Run	Accept Cancel		

- Select Waveform for displaying the output. 4.
- 5. Click Accept button after entering the values to automatically switch to Analysis.
- 6. Click Run button to start simulation



## **Distortion Analysis**

The distortion analysis computes steady state harmonic and intermodulation products for small input signal magnitude. If signals of a single frequency are specified as the input to the circuit, the complex values of the second and third harmonics are determined at every point in the circuit. If two frequencies are specified at the input of the circuit the analysis finds out the complex values of the circuit variables at the sum and difference of the input frequencies, and at the difference of the smaller frequency from the second harmonic of the larger frequency.

- Set the frequency, Select Component parameters (function tool) →
   Change Simulation Parameters option tool.
- 2. Select Simulation → Analysis.
- 3. Select **Distortion analysis** from the tree view.
- 4. Inorder to run distortion analysis, specify the analysis parameters by selecting Distortion Analysis from the tree view on the left side of the **Analysis Setup**.
- 5. Set the values as

Total points	100
Start frequency	10Hz
End frequency	10Meg

- 6. Click Accept button after entering the values to automatically switch to Analysis.
- 7. Click **Run** button to start simulation. The output obtained is a text file.

Analysis Setup:		×	
Simulator Variables(Options)	Set Parameters :		
Initial Node Voltage Guesses	disto DEC 1001010Meg 25		
Transient Initial Conditions			
Analysis	Variation	Decade	
🗄 - Transient Analysis	Sweep Parameters		
🗄 - Small-Signal AC Analysis	Total Points	100	
DC Transfer Function Analysis	Start Frequency	10	
Distortion Analysis	End Frequency	10Meg	
Waveform Viewer	Spectral Analysis		
Print(Standard Output)	F20VERF1	25	
Plot(Standard Output)	Temperature		
- Operating Point Analysis	As Setup		
- Noise Analysis	Results	Waveform	
- DC/AC Sensitivity Analysis			
- Transfer Function Analysis			
E-Pole-Zero Analysis	J.		
	Run	Accept Cancel	

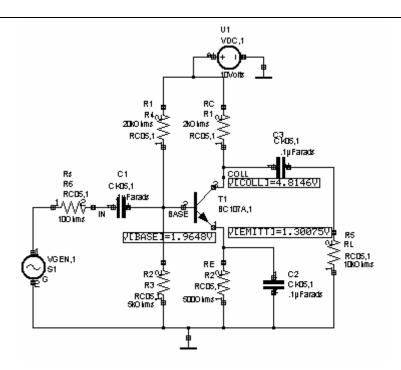
File Viewer (D:\EDWINXP \EDS_WRK\EDS_DISTO_ANALYSIS_11_20
■ File Edit Search Window Help
Title: 1_11_06RC Coupled Amplifier: - Distortion Analysis Date: Mon Nov 20 12:48:50 2006 Temperature: 27.000 Plotname: disto3:DISTORTION - IM: 2f1-f2 Flags: complex No. Variables: 3 No. Points: 1
Variables: 0 frequency frequency 1 v(2) voltage 2 v(6) voltage Values: 0 1.00000000000000000000000000000000000
2 open document(s). CAPS NUM INS

# **Operating Point Analysis**

Operating Point Analysis is used to determine the DC behaviour of a circuit. This is done automatically when an AC analysis and Transient Analysis is done.

- 1. Select Simulation → Analysis.
- 2. Select **Operating point Analysis** from the tree view.
- **3.** The default selection is '**As marked**' means that results will be displayed at all markers placed on the circuit schematic. After analysis, the values will be presented by updating the markers.
- 4. If we want to know the values at all nodes and branches of the circuit, select 'All Points' the results can be viewed in the RAWSPICE.RAW file, For that Select Options → View EDSpice Files → Raw file, from that Open rawspice.raw file.

Analysis Setup:	×
Simulator Variables(Options) Initial Node Voltage Guesses Transient Initial Conditions Analysis Transient Analysis Small-Signal AC Analysis DC Transfer Function Analysis Distortion Analysis Vaveform Viewer Print(Standard Output) Plot(Standard Output) Plot(Standard Output) Operating Point Analysis Noise Analysis DC/AC Sensitivity Analysis Transfer Function Analysis Pole-Zero Analysis	Set Parameters :
	Run Accept Cancel



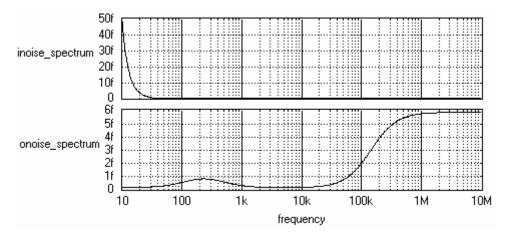
#### **Noise Analysis**

Noise Analysis is used to analyse the noise existing at any point in a circuit, due to the combined effect of all noise sources in the circuit.

- 1. Select Simulation → Analysis.
- 2. Select **Noise Analysis** from the tree view.

Analysis Setup:			×
Simulator Variables(Options) Initial Node Voltage Guesses Transient Initial Conditions Analysis	Set Parameters : noise V(6) V10 DEC 100 1		
<ul> <li>Transient Analysis</li> <li>Small-Signal AC Analysis</li> <li>DC Transfer Function Analysis</li> <li>Distortion Analysis</li> <li>Operating Point Analysis</li> </ul>	Selec Output Variable V(LO/ Input Source S1/1		Ref. Net
<ul> <li>Noise Analysis</li> <li>DC/AC Sensitivity Analysis</li> <li>Transfer Function Analysis</li> <li>Pole-Zero Analysis</li> </ul>	Variation Sweep Parameters Total Points Start Frequency End Frequency Points/Summary	Decar	de
	Run	Acc	ept Cancel

- 3. Set the simulation parameters such as start frequency, end frequency, total points etc.
- 4. After analysis the **Noise Spectral Density Curves** will be presented in the Waveform viewer as shown below.



The total integrated Noise will be presented in the rawspice.raw file. It can be viewed from Options → View EDSpice Files → Raw file, this .raw file is pasted below.

🗉 File Viewer (D:\EDWINXP \EDS_WRK\RAWSPICE.RAW) - [D:\EDWINXP 🗖 🗖 🗙
■ File Edit Search Window Help
Title: 1_11_06RC Coupled Amplifier: - Noise Analysis
Date: Mon Nov 20 13:10:43 2006
Temperature: 27.000
Plotname: noise2:Integrated Noise - V^2 or A^2
Flags: real
No. Variables: 2
No. Points: 1
Variables:
O onoise_total voltage
1 inoise_total voltage
Values:
0 5.754142702638551e-008
1.957670426641511e-013
Title: 1_11_06RC Coupled Amplifier: - Noise Analysis
1 open document(s).

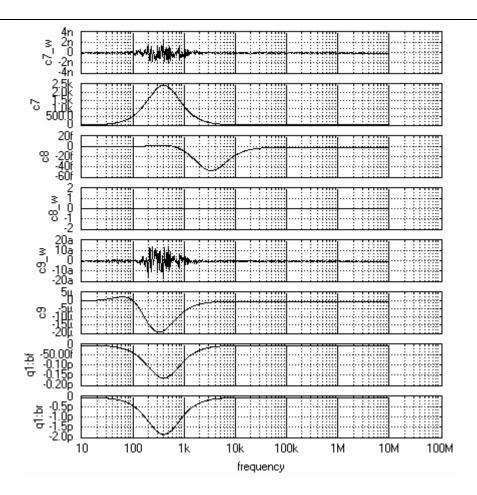
# DC / AC Sensitivity Analysis

This analysis helps to calculate sensitivity of an output variable (voltage and current) with respect to all circuit variables, including model parameters.

- 1. Select Simulation → Analysis.
- 2. Select DC / AC Sensitivity Analysis from the tree view.

Analysis Setup:				×
Simulator Variables(Options) Initial Node Voltage Guesses Transient Initial Conditions	- Set Parameters : sens I(V11) AC DEC	: 100 10 10Meg	]	
Analysis Transient Analysis Small-Signal AC Analysis DC Transfer Function Analysis Distortion Analysis Operating Point Analysis	Output Variable Output Variable Temperature As Setup	Select Net/Sc [(U1/1 [V11])	urce Rel	Net
Noise Analysis DC/AC Sensitivity Analysis Transfer Function Analysis Pole-Zero Analysis	AC Sensitivity Analy Variation Sweep Parameters Total Points Start Frequency End Frequency	Bun	Decade Accept	100 10 10Meg

- 3. Click Accept button after entering the values to automatically switch to Analysis.
- 4. Click Run button to start simulation



- 5. After analysis the results of the **DC Sensitivity Analysis** will be presented in the rawspice.raw file, open this file from **Options View EDSpice Files Rawfile**.
- 6. The results of **AC Sensitivity Analysis** will be presented in the Wave form viewer.

🗮 File Viewer (D:\EDWINXP \EDS_WRK\rawspice.raw) - [D:\EDWINXP \E 💻 🗖 🗙
Eile Edit Search Window Help
Title: 1_11_06RC Coupled Amplifier: - DC/AC Sensitivity Analysis
Date: Mon Nov 20 13:25:32 2006
Temperature: 27.000
Plotname: sens1
Flags: complex
No. Variables: 50
No. Points: 602
Variables:
0 frequency frequency grid=3
1 c7_w voltage
2 c7 voltage
3 c8 voltage
4 c8_w voltage
5 c9_w voltage
6 c9 voltage
1 open document(s).

#### **Transfer Function Analysis**

The transfer function analysis calculates the small signal ratio of the output node to the input source, and also the input and output impedence of the circuit.

- 1. Select Simulation → Analysis.
- 2. Select **Transfer function Analysis** from the tree view.
- 3. Set parameters and click **Accept** to accept these values.

Analysis Setup:			×
Simulator Variables(Options) Initial Node Voltage Guesses	Set Parameters :- tf V(6,2) V10		
Transient Initial Conditions		_	
Analysis		Select Net/Source	Ref. Net
庄 - Transient Analysis	Output Variable	V(LOAD)	Input
🗄 Small-Signal AC Analysis	Output Variable		
⊡ DC Transfer Function Analysis	Input Source	S1/1 [V10]	
庄 - Distortion Analysis			
- Operating Point Analysis			
- Noise Analysis			
- DC/AC Sensitivity Analysis			
Transfer Function Analysis			
Pole-Zero Analysis			
	ļ		
		Run Acc	cept Cancel

- 4. Click **Run** button to execute the analysis.
- After the analysis, the results will be displayed in the .raw file. Open this file from
   Options → View EDSpice Files → Rawfile.

File Viewer (D:\EDWINXP \EDS_WRK\RAWSPICE.RAW) - [D:\EDWINXP • •					
Eile Edit Search Window Help					
Title: 1 11 06RC Coupled Amplifier: - Transfer Function Analysis					
Date: Mon Nov 20 15:01:20 2006					
Temperature: 27.000					
Plotname: tf1					
Flags: real					
No. Variables: 3					
No. Points: 1					
Variables:					
0 transfer_functionvoltage					
1 output_impedance_at_v(6,2) voltage					
2 v10#input_impedance voltage					
Values:					
0 -1.000000000000e+000					
1.000000000000e+004					
1.000000000000e+020					
1 open document(s).					

#### Pole- Zero Analysis

Pole- Zero Analysis is most commonly used for determining the stability of control circuits.

This computes the poles and / or zeros in the small signal ac transfer function.

- 1. Select Simulation → Analysis.
- 2. Select the **Pole Zero Analysis** from the tree view.

Analysis Setup:		×	
Simulator Variables(Options) Initial Node Voltage Guesses	Set Parameters : pz 3 0 6 0 CUR ZER		
Transient Initial Conditions	Select Nodes		
Analysis Ē- Transient Analysis	Node 1	IN [3]	
⊡ - Small-Signal AC Analysis ⊕ - DC Transfer Function Analysis	Node 2 Node 3	SPL0 [0] LOAD [6]	
Distortion Analysis	Node 4	SPL0 [0]	
<ul> <li>Operating Point Analysis</li> <li>Noise Analysis</li> </ul>	Select Poles/Zeroes Transfer Function	CUR ZER	
DC/AC Sensitivity Analysis		CONCEN	
<ul> <li>Transfer Function Analysis</li> <li>Pole-Zero Analysis</li> </ul>			
	Run	Accept Cancel	

- 3. Click Accept button to accept these values.
- 4. Click **Run** button to execute the analysis.
- 5. After analysis, the results will be displayed in the .raw file .For opening this file select **Options →View EDSpice Files →Rawfile**.

📕 File Viewer (D:\EDWINXP \EDS_WRK\RAWSPICE.RAW) - [D:\EDWINXP 🗖 🗖 赵	<				
Eile Edit Search Window Help	<				
Title: 1_11_06RC Coupled Amplifier: - Pole-Zero Analysis	1				
Date: Mon Nov 20 15:16:14 2006					
Temperature: 27.000					
Plotname: pz1	I				
Flags: complex	I				
No. Variables: 5	I				
No. Points: 1	I				
Variables:	I				
O zero(1) voltage	I				
1 zero(2) voltage	I				
2 zero(3) voltage	1				
3 zero(4) voltage					
4 zero(5) voltage					
Values:					
0 -2.267406215754985e+010,0.000000000000e+000					
1 open document(s).	//				

# PCB LAYOUT

**PCB Layout** is invoked from Project Explorer in the following ways.

Right click PCB Layout and select Edit PCB Layout from the list.

Or

Select Edit PCB Layout from the task list or from the task toolbar.

## Define board outline

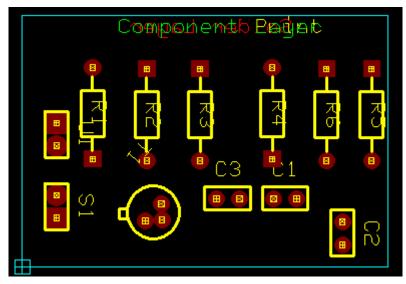
- 1. Select Tools → Board Outline
- Select Define Outline (function tool) → Create Board (option tool). This tool enables free hand drawing, using which a board of desired shape can be drawn on the workspace.
- To select a pre-defined board format, select the option tool Textual Mode. The Properties Board Outline (Create) dialog pops up.
- 4. Enter the desired values and click Accept button.

Properties Board Outline(Create)				
Board Outline	Property	Value		
	X position	0.000mm		
	Y position	0.000mm		
	Line width	0.127mm		
	Gap	0.3048mm		
	Shape	Rectangular 🗾		
	Width	Rectangular		
	Height	Circular		
		Polygon		
	l			
	Apply	Accept Cancel		
I				
		///		

#### **Relocating the components**

The components placed on the schematic, which contain both symbol and package, are front annotated to the layout after packaging. These components are positioned at board datum (0,0) automatically and may be relocated either manually or automatically **Note:** Only parts that contain both symbol and package will be automatically back annotated to the schematic.

- 1. Before we start loading Parts on to the page, turn ON **Grid** by enabling grid from the dropdown, in Standard Toolbar. The value for grid may be selected from the drop down list as .1000".
- 2. Similarly, set Snap value to .0500" for better placement of the components.
- 3. Select **Tools** →**Components Relocate Component** (function tool) to relocate the components.
- 4. Enable Ratsnest (F7 key) option tool of Relocate Component function tool to view ratsnest while relocating the components to ensure that components having a large number of interconnections are positioned close to each other. Pressing shift key while relocating/ stretching an item allows the item to move/ stretch smoothly.



## **Routing the Components**

Connections between components may be established by using **Traces** and **Copper Pour** areas.

- Select Tools → Connections → Route (Function tool) to enable a set of Routing tools on right clicking the workspace.
- 2. Adjust **zoom precision** to view the pins properly.
- 3. Turn on **Preference/ Guidelines (Next unconnected node**) before routing because this option guides you to take the easiest path to route.
- 4. Select **True Size** and **Pad frames** from **View/ Layout**, enabling you to select proper trace size. This also prevents you from creating errors such as traces crossing over pad, traces very close together etc.

- First route power and ground signals (Net SPL0 and +5V). In Project Explorer, select Project / Project Design Rules and set the routing width to 0.030" for Pwr/Gnd lines and 0.013" for Signal lines.
- 6. To start routing traces, first select the power points
- Select layer for routing from Layers in the main menu. 28 layers are available for routing. By default COMP LAYER is selected. Click on pin to route on SOLD LAYER.

Tips: While routing, enable the tool Snap Trace by 45 degree to change routing directions in steps of 45 deg only.

- 8. Move cursor with 45° angle through a short distance and click at the nearest point.
- 9. Terminate routing of the trace by pressing END or F4 key on your keyboard. Or click on the tool **End Connections**.

# **Auto Routers**

The three Auto routers available are:

- 1. Autorouter-Arizona
- 2. Autorouter-Specctra
- 3. Autorouter-Maxroute

## Arizona Autorouter

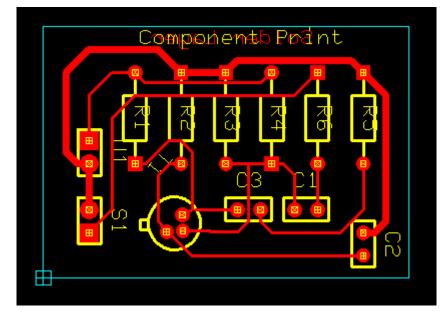
Arizona auto router is an integrated module of the EDWinXP. It uses its own temporary project and simplified graphics. The Arizona auto router allows routing the traces of a PCB Layout automatically.

## Auto routing using Arizona Auto router

- 1. Select Auto → Auto Router → Arizona.
- 2. Select File → Load board to route from project.
- 3. Select Parameters Setup (function tool) → Routing Parameter Settings (option tool).
- 4. Check Solder Layer in the Routing Parameters Settings window

Layers Cleara	e <b>ter Settings</b> nce Setup costs Width	ı&Via   Fano	out   Op	otimize c	:ost
	Layer	Direction	Pow.	Sig.	
	COMP.LAYER	Horizontal			
	ALT.GLUE MASK				
	D				
	E				1
	F				
	G				
	H				
	J K				
		J	<u> </u>		
		Accept		<u>C</u> ance	el

- 5. Select Auto routing Routines (function tool) → Start Auto Router.
- 6. Select Auto routing Routines → Miter.



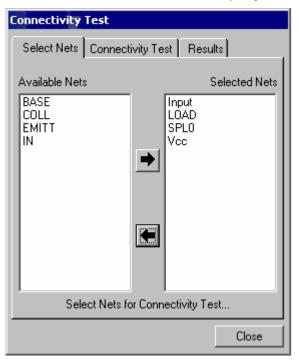
7. Close Arizona Auto router window, Click Update project and Exit to save the project.

# **Testing the Board**

While designing a PCB, it is quite obvious that a number of errors may occur. These errors may be in the form of overlapping pads, unconnected Nodes, traces crossing another trace, etc. Such errors must be taken care of before printing the PCB. To find out such errors, certain checks on the board are done. **Connectivity and DRC check** are the two checks.

# **Connectivity Test**

Connectivity test may be used to check whether there is any electrical discontinuity (unconnected nodes or deleted trace segments) in a single net. By setting certain parameters, the test can be performed either on individual nets or all the selected nets.



1. Select Tools → Connections → Connection Property → Test Connectivity

- 2. Click on the board or on a net. The **Connectivity Test** dialog pops up.
- 3. Select the required nets using the move keys and click the connectivity test tab.
- Select Connectivity test tab, set anyone of the three options:
   Stop at first fail: Test stops at the first occurrence of error.
   Test all selected Nets: Checks whether all Nodes of the selected Nets are

connected.

Test single Net: Checks connectivity of the Nodes of the selected Net.

- 5. Click the **Test** button to display the results.
- Perform this test until "Tested Nets Fully Connected" message is displayed in this window.

#### **Design Rule Check (DRC)**

This utility is used to create an error free board to enhance the efficiency of your board. It automatically smoothes, miters, and checks for both aesthetic and manufacturing problems that might have been created in the process of manual or automatic routing. This test helps us to check the clearance between pad to pad, pad to trace and trace to trace. Select **Autocheck** from **Auto** Menu. The **Clearance Check** dialog pops up.

Select the layers and enter the clearance value in the window.

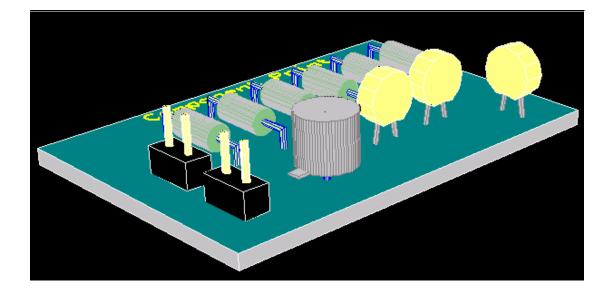
To select all the layers used in the project click the **Set To Used** button. Click **Execute**.

Cleara	nce Check Setup	×						
Check Clearances Check other DR Violations								
#	Layer names Check 🔺							
1	COMP.MASK							
2	COMP.LAYER							
3	GLUE MASK							
4	ALT.GLUE MASK							
5	C							
6	D							
7	E							
8	F							
9	G							
10	H	────────────						
11								
		Set to Used						
Che	cks & Clearances							
	Pad to Pad:	0.2032mm						
	Pad to Trace:	0.2032mm						
	Trace to Trace:	0.2032mm						
	nnel width for Single ce Check:	1.270mm						
	✓ Use clearances as in design rules for layers							
		Execute						
	Accept	Cancel						

#### **3D Board Viewer**

3D Board Viewer gives idea on how the components are located, whether there is risk of friction between components due to their height and shape, the side on which components are placed, etc.

Select Tools -> 3D Board Viewer to view the 3D board



**3D View Control** box appears, in this we can select options to view the board in any direction.

# **Board Analyzers**

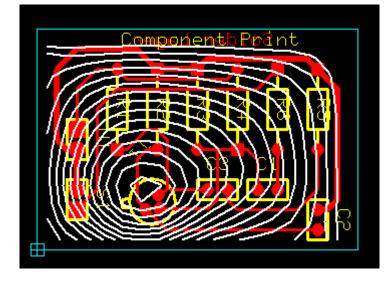
EDWinXP provides two board analyzers. Thermal Analyzer Electromagnetic Analyzer

# **Thermal Analyzer**

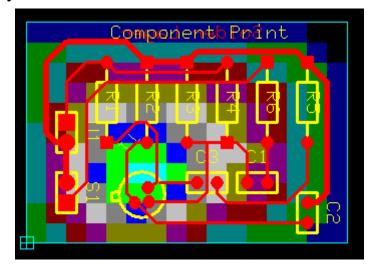
The Thermal Analyzer is intended to be used for analyzing and identifying potential thermal problems on a Printed Circuit Board. It evaluates the Temperature Distribution on a finished PCB, at steady state conditions.. The result of the analysis may be displayed using isotherms or color mapping scheme.

# **Steps for Thermal Analysis**

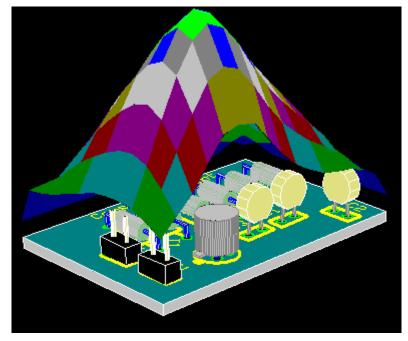
- 1. Right click on the **PCB Layout** in the Project Explorer. Select **Board Analyzer** in the list.
- 2. Select the Thermal Analyzer tab.
- 3. From **Analysis**, choose **Settings** to open the settings window and click the **Display Options** tab. Click on **Set to default** button .
- 4. From the Analysis menu select Settings click and select the tool Board Parameters Click on Set to default button
- **5.** After supplying necessary parameters, right click and select the tool **Execute** to execute the analysis. The result is displayed in the form of isotherms.



 Color mapping of the board may be viewed by switching on the option View->Analysis Results->Colored board.



3D Board may be viewed by switching on the option View->Analysis Results >3D Board Viewer. This option allows to view the temperature distribution on the circuit in the form of a 3D graph. The higher the temperature, the higher will be the location of the point on the graph.



- 8. The analysis result is obtained. From the result it may be easy to identify the most heated up component so a heat sink of about may be used.
- 9. To simulate the heat sink effect, Select the tool Component Parameters and click on the component. A Thermal Parameters window pops up where you may set the cooling parameter for the component. Check Component Mounted Heat Sink check box, enter the Thermal Resistance of Heatsink (e.g. 1.2°C/Watt) value and click OK. Repeat the same step for all other components and execute

the analysis again. You can notice the change in the isotherms after conducting the thermal analysis.

10. Labels may be placed to display the temperature at various points on the board.For this right click and select the tool Set/ Delete Label and click on the board at required locations. Label gets tagged to the cursor. Now drag the mouse to place the label at proper place.

Note: To delete the isotherm label click again on the label.

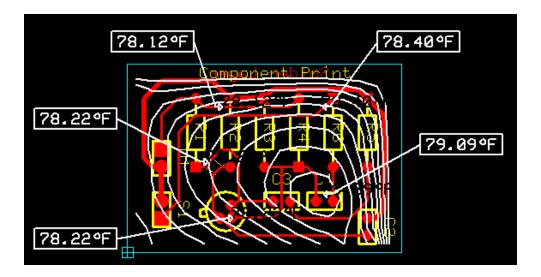
# **Electromagnetic Analyzer**

Electromagnetic Analyzer is used to predict the intensity of electromagnetic field generated by the working circuit on the PCB. An electromagnetic field develops when voltage passes through the traces on board. Once the routing of traces is complete, electromagnetic analysis may be performed on the board. The Electromagnetic Analyzer measures the distribution of the Electric Field Intensity on a finished PCB. The isolines show the distribution of the field intensity on the board. Under the electromagnetic analyzer you can perform Signal Integrity Analysis and Field Analysis.

# **Steps for Electromagnetic Analysis**

- 1. Select Electromagnetic Analyzer tab, Right click and select the tool Electrical Parameters .
- 2. The board parameters are set to the default values by clicking the **Set to default** button.

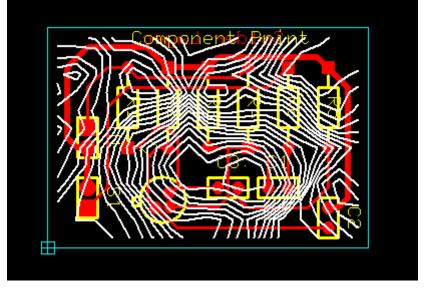
Net Electrical Parameters			
Select Net	Currently Set Net F	arameters	
	Net Name	Voltage (V)	Frequency
	BASE	5	1M
	COLL	5	1M
	EMITT	5	1M
	' IN	5	1M
	Input	5	1M
	LOAD	5	1M
	SPLO	5	1M
-	Vcc	5	1M
View Trac	ces		Set to Default
		(	Accept



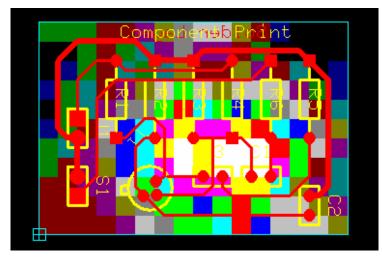
3. If you select the Set to default button, the default parameter gets loaded into the text boxes. To move Net from Select Net to Currently Set Net parameters use

key and to move backward use key.

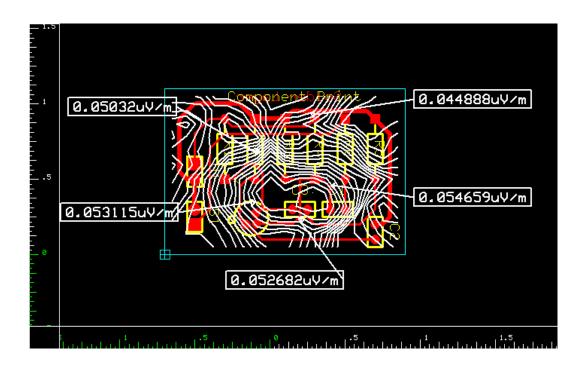
- 4. View the electrical parameters of the trace from point to point using the View Traces button after selecting the particular Net Name
- 5. After adding these nets you may execute the analysis directly. For this right click and select the tool Execute Analysis. The result is displayed in the form of isolines.



 Color mapping on the board and may be viewed by switching on the option Colored board under View → Analysis Results → Colored Board. The analysis result is obtained. The Maximum and minimum field intensity locations can be observed.



7. Labels may be placed to display the field intensity at various points on the board. For this select the tool Set/ Delete Isolines and click on the board at required locations. Label gets tagged to the cursor. Now drag the mouse to place the label at proper place.



# **Signal Integrity Simulation**

The Signal Integrity Analyzer examines the probable distortion of high-speed signals as they pass through traces on the PCB. The results of the analysis are displayed in the Waveform Viewer. The purpose is to predict how a signal deviates from its ideal (or intended) behavior in a real-world setting.

# **Steps for Signal Integrity Analysis**

Under normal working conditions, signals propagate from some pins of the components in the circuit (technically called **Driver Nodes**), through the interconnecting traces and into pins on the same or other components (called **Receiver Nodes**).

This analysis detects the degree of distortion of the signal from its ideal behavior while it passes through the trace and graphically presents the comparison result with the help of the **Waveform Viewer**.

1. Signal Integrity Analyzer is invoked from within the Electromagnetic Analyzer.

Right click on the task PCB LAYOUT in the Project Explorer to unfold a list of functions. Clicking on Electromagnetic Analyzer in this list opens up this module.

2. Right click on the workspace and select the tool Signal Integrity and click on any of the trace, which is to be analyzed.

- 3. Now select a net. A window pops up where you may set all the parameters for simulation. To check the integrity of an electromagnetic signal while it passes through this trace, go through the following steps.
- 4. All nets are displayed in the frame Available Nets. Select a net from the list and click . The segment nodes of the selected net get displayed in the frame. Right click on any of the node and set it as driver node and another node as receiver node.

Net: LOAD - Signal Integrity Simulation (Propag	jating node:R5/2) 🛛 🗙
Node Parameters Simulation 🤐	
Selected Nets	Net : LOAD, Node : R5/2
IN   Driver node   Reciever node   Simple node     Set all nodes to receivers     Remove Net     Vec     Available Nets	No Parameters Available for the Selected Item! Show Trace Segments
	Board Parameters Close

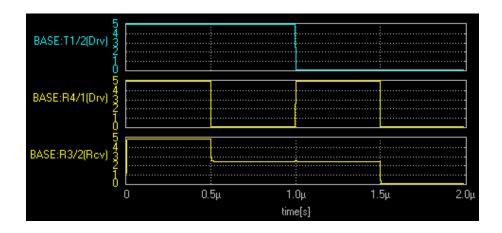
5. The electrical properties of the trace will influence the signal as it passes through it and this is shown in the respective columns. The Trace Segment parameters of the driver node may be viewed by clicking on Show Trace Segments.

Net: BASE - Signal Integrity Simulation (Propagating node:T1/2)						
Node Parameters   Simulation   🥌						
Selected Nets	Net : BASE, Node : T	1/2				
BASE	Parameter	Value	U			
	Technology	TTL Course Dutys	_			
B3/2 B4/1 C1/2	Function Resistance (R out)	Square Pulse 150	Ohm			
1/2	Inductance (L out)	3n				
	Capacitance (Clout)	7p	F			
	Voltage min	0	<u>v</u>			
	Voltage max	5	<u>v</u>			
	TL->H	U	<u>s</u>			
Available Nets	T High T H->L	1μ 0	s s			
	TLow	1μ				
COLL ,5,1M EMITT ,5,1M IN ,5,1M Input ,5,1M						
LOAD ,5,1M SPL0 .5.1M	r	Show Trace S	<u>egments</u>			
	Board Parame	iters (	Close			

6. Set the following parameters.

```
Max voltage to 5V
T L->H = 0
T H = 1us
T H-> L = 0
T L = 1us
```

- Set TTL technology to the receiver nodes. Check the check box corresponding to Test Points. The **Test points** allow displaying the output of the selected node in the waveform viewer. If no test point is selected then output waveform alone is displayed on the waveform viewer.
- To execute simulation, Start time and End time of the simulation as well as the sampling time interval must be specified. This corresponds to the parameters Start time, Time limit and Time step. Enter 1n for Time Step and 2µs for Time limit.
- Click on Calculate button. The Calculate button allows to calculate the maximum time limit and will update the text box Time limit. Here the time step is set to 1ns and Time limit to 2us.
- 10. Check the box **Generate Waveform.** Click the **Simulate** button to process the simulation.
- 11. The waveform obtained after simulation is shown below.



## **Field Analyzer**

The Field Analyzer is a tool for studying the electromagnetic fields that are created when power and/or signal traces on the board are energized. The results of the analysis may be view as a color graph, isolines, 3D-wire mesh graph, etc. It must be particularly mentioned that, just as with the other two tools, the Field Analyzer does not make any decisions or suggestions about the proper functioning of the design. The Field Analyzer predicts the variations in the selected field, due to electromagnetic properties of physical connections (traces), within a spatial area and time frame specified by the user.

#### **Field Analyzer - Operation**

Invoke Field Analyser from Signal Integrity Simulation window. Check the option 'Display Fields' in the Signal Integrity Simulation window. Click the Simulate button to open the Field Analyzer.

Tool Box	×
- Graph Modes	
Graph extent[m]:	
1.40 🗨	
Select Field:	
Hz E A Fi	
Graph:	
in the second se	
- Graph Params.	1
Isolines density: 10 🗧 🗧	
Color graph	
Precision raster: 40x40	
Display grid: 🛛 式	
Graph center	
×[m]: 0.00	
Y[m]: 0.00	
- Controls	1
Time Step: 100pSx 1 = 1 ▼	
No. of steps: 2000	
- Plot settings	1
Field values	
Autoscale maximal 🔽	

The Tool Box on the right side of Field Analyzer includes

#### **Graph Modes**

The Graph mode includes

#### Graph Extent:

Size of the square covered in the graph can be adjust using the zoom out and zoom in button.

## Select field

It includes Electric field, Magnetic field, Vector potential, Scalar potential. Output can be viewed by selecting each field.

# Graph.

In this option of Field Analyzer, five types of graph can be viewed. Colored

graph , Isoline , field direction graph , Polar coordinates graph , 3D graph .

Graph Params.

Isolines density

More detailed presentation of the result can be obtained by increasing the Isolines density in the Graph Params.

Color graph:

Precision raster:

The precision to calculate the field values can be adjusted using the Precision raster.

Display grid:

Applying higher values to the Display grid will provide smoother variations to the colored graph.

# Controls

Run Simulation button used to begin the Analysis. .

The simulation may be interrupted (and continued later) at any desired point by

using the **Pause** button.

In order to continue the interrupted analysis, simply click the **Continue Run** button.

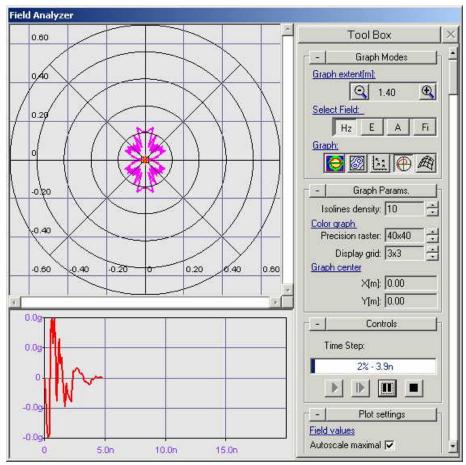
**Stop** button stops the simulation.

## Plot settings

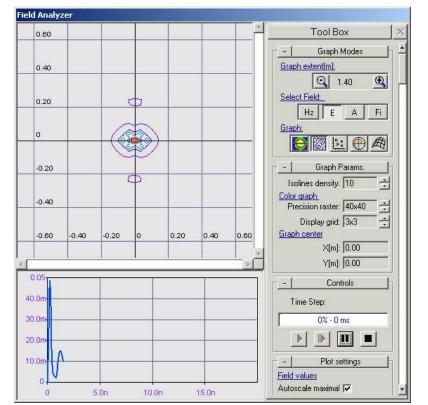
You can view the Field Analyzer Diagram by setting the parameters as your need. If Auto Scaling has been selected, the maximum values displayed as well as the scales of the graphs are automatically updated. On the other hand, if Auto Scaling has been turned off, these values are clipped at the values specified. (In other words, values higher than those specified are ignored.).

Different magnetic field, electric field, vector potential and scalar potential graphs can be viewed by selecting **Select Field** and **Graph** in **Graph Modes**.

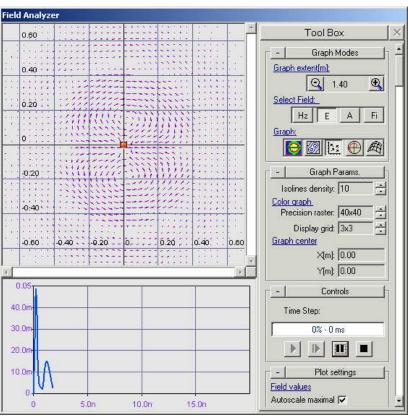
# Electric Field Polar Co-ordinate Graph



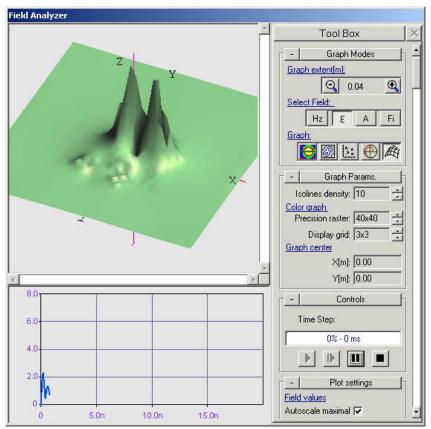
# **Magnetic Isoline Graph**



# **Magnetic Field Direction Graph**



Magnetic Field 3D Graph



54

# **Fabrication Manager**

# Introduction

Fabrication is the last stage of the electronic design process. The PCB information is converted into ASCII output files – GERBER (.gbr), NC Drill (.ncd), PCB Assembly Outputs Generic (.pck), IPC-D-355(.355), Bare Board Testing Generic (.bbt) and IPC-D-356A(.356) which are input to machines to finally create the hardware.

- 1. Invoke **PCB Layout** in the Project Explorer.
- Select Fabrication Manager from the tasklist or Select from the task toolbar. The Fabrication Manager window opens with a default board size and gets aligned with the Project Explorer to fit the screen.



3. Invoke Fabrication Data Manager choose Setup from Fabrication.

🔚 Fabrication Data Manager				X
Category	Ge	rber Photoplotter Data		
Gerber Photoplotter Data		Output File Locations and Names		
Gerber Artworks		Location	D:\EDWINXP\JOB\	
Gerber Mechanical Plots		File Name Prefix	RC Coupled Amplifier_MAINHIER	
⊡- NC-Drill Data		Gerber Output Format		
DCD Accessible Output		Output File Format Standard	RS-274-X	
PCB Assembly Output		Output Units	Inches	
bale board resurg		Precision Format	2.3	
		Omitted Zeroes	Leading	
		Scale Factor	1.0000	
		Photoplotter Options		
		Circular Interpolation supported		
		Filled Polygons Supported		
		Square Flash Fill for Rectangles		
		Embed Offset and Stepping Commands		
		Mixed Polarity Plot (Copper Pour Areas)	V V	
		Aperture Table		
		Aperture Table type	Embedded	
		Macros/Special Apertures supported		
		Offset and Stepping Definitions		
		Units	Inches	
		Number of Horizontal Steps	1	Ţ
1	-		1	
		[	Accept Cancel	

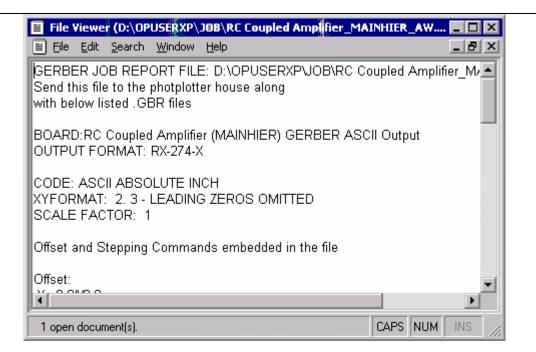
4. Choose Gerber Artworks in the left pane of Fabrication Data Manager .

# **Gerber Output**

GERBER is the standard format used as input to photoplotters that generate the design data on films. These films are used as masters for manufacturing the PCB, the physical realization of the schematic data. GERBER format is a vector format for defining various elements of the layout. This format represents all traces as draw and the pads that are part of the component footprint as flashes. The Photoplotter uses this command language

- 1. Select the layers for which we want the Gerber Artwork files.
- 2. Click Execute.
- 3. 'Gerber output window' opens, Click on **Execute**.
- 4. This completes the generation of GERBER data.

🔚 Fabrication Data Manager					_ 🗆 X
Category	Gerber Artworks	;			
Gerber Photoplotter Data     Gerber Artworks     Gerber Artworks     Gerber Mechanical Plots     NC-Drill Data     Output Contents     PCB Assembly Output     Bare Board Testing	Include in Artwork	Layer Name L M N O P Q Q R S	Mirror Image	Copper Pour Areas	Boa
		S T U V W X Y Z SOLD.LAYER			
		SOLD.MASK SOLD.PRINT	Set to Use		



# Preview the GERBER Data

# Preprocess the GERBER file

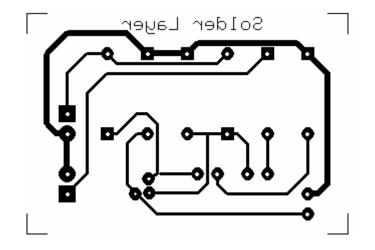
File → GERBER Viewer Setup option. Click on Grid cell in the column GERBER ASCII
 File, an ellipsis (a button with 3 dots) appears, click on it and select the required GERBER
 ASCII files (\*.GBR)..

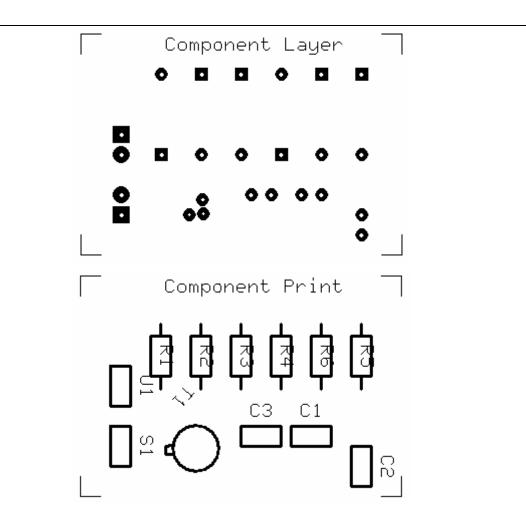
Gerber Preprocessor and Viewer Setup								
Gerber			sor 🗎 A	rtwork Viewer Setup				
#	Gerber ASCII File	Superi Positiv Image	e	Gerber ASCII File (superimposed)	Artwork Displ	av File	Selection	
1	RC Coupl			<u>`````````````````````````````````````</u>	RC Coupled	·		1
2	RC Coupl	ed [			RC Coupled			1
3	RC Coupl	d De			RC Coupled			1
4	RC Coupl	d De			RC Coupled			
5	RC Coupl	d De			RC Coupled			1
6		I I						
7		I I						
8		1						
9		]						
				·	·			
Gerb	er Input File Paramete	rs :				<u>D</u> e:	stination Patł	ר ו
Outp	ut Units	Inch	nes					
Omitt	ed Zeroes	Lea	ding					
Scale	e Factor:X	1.0					Clear	
Scale	e Factor:Y	1.0					Clear All	
Preci	sion Factor:X	2.3					cical <u>A</u> ll	
Preci	Precision Factor:Y 2.3							
Apert	Aperture Table Autogenerated					E	reProcess	
Close								
Selected	Selected: 4 D:\EDWinXP\JOB\RC Coupled Amplifier_MAINHIER_91.GBR							

Multiple Artwork files may also be preprocessed Click '**Preprocess**' button. The preprocessing status may be displayed on the status bar and the completion is marked by a **Successfully processed selected files.'** message. As a result, artwork file (\*.**ART**) is created with the name of the source Gerber

Viewing Artwork

- 1. Select Artwork Viewer Setup from Gerber Preprocessor and Viewer Setup window.
- 2. Choose Browse Artworks
- 3. Select the .ART files and click on open
- 4. Select the files one by one and click on Apply.
- 5. The Artwork of a PCB is the actual display of the various copper routes, padstack, pinholes, etc.. The design data may be viewed layer by layer as it is got on the photoplotted film or pen plotted artwork. This menu option must be active to view the various layers of the design individually.
- 6. Tools Artwork / Power & Gnd planes .
- 7. Select Define Artworks (function tool) Set Used to Accept.
- 8. Select Centreholes from View Artwork Centreholes as 1/3 Size.
- In order to view layers, Select Tools Artwork & Pwr/Gnd Planes use the Layer( Comp.side & Sold.side) drop down menu and select the required layer. The artwork for that particular layer may be viewed as it appears. Select Solder Layer from tool bar.





# **Gerber Mechanical Plots**

This window allows setting the parameters for gerber mechanical plots on both Component and Solder side. The window opens with the Component Side, by default. Here the Generate Mechanical Plot for Component Side checkbox is unchecked. Enabling this checkbox displays the list of parameters that may be set.

1. Select **Fabrication → Setup.** This dialog box is shown below.

Category	Ge	rber Mechanical Plots	
⊡- Gerber Photoplotter Data Gerber Artworks		mponent Side Solder Side	
Gerber Mechanical Plots		Generate Mechanical Plot for Component Side	
Output Contents	E	Layer for which (Mechanical Plot - Component Side) is to be generated	
- PCB Assembly Output		Current Layer	COMF
Bare Board Testing	E	Board entities to be plotted	
		Board Outline	
		Board Description Notes	
		Dimensions	
		Component Outlines	
	E	Padstack entities to be plotted	
		Pad Frames	
	11	Hole Outlines	
		Execute	
		Accept Ca	ncel

- 2. Click Execute, then a Gerber Output window appears.
- 3. Click **Execute.** 'Gerber ASCII output completed' message appears in the bottom of the window. The output obtained will be a text file.

#### Introduction to NC Drill

In generating the NC-Drill, the system extracts information about the whole positions, sorts them according to drill size information. The different hole sizes are grouped together and generally sorted to drill in a specific order. This grouping may be done based on sizes, type of drilled hole, like through plated or non-plated, via type like buried or unburied etc. This is generated as a drill tape data. The data may be used directly on any numerically controlled drilling machine. This NC-Drill data is of the EXCELLON format, which is the accepted industry standard.

# **NC Drill Output Parameters**

 Select Fabrication → Setup → Fabrication data manager → NC-Drill Data. Depending on the equipment used by the PCB manufacturer some parameters must be set prior to generation of NC drill files.

🔚 Fabrication Data Manager					
Category	NC	Drill Data (Excellon Format)			
Gerber Photoplotter Data     Gerber Artworks     Gerber Mechanical Plots     NC-Drill Data     Output Contents     PCB Assembly Output     Bare Board Testing		Excellon Output Format Output Units Omitted Zeroes Tools Tool Size Units Tool Size Precision Tool Magazine Size NCDrill bit movement Optimization Optimization method		Inches Trailing Inches 3 digits 8 Tools None	
			Accep	ot	Cancel

2. Set the following parameters

Output units - Inches or Millimeters.

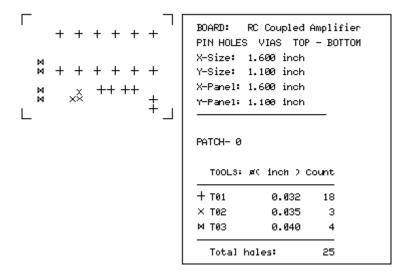
**Omitted Zeroes-** Leading or Trailing.

- 3. Click on Execute, NC Drill Output window appears.
- 4. In this window check Prepare drill database for Template only and Check for multiple holes in the same XY position.
- 5. Click on Execute. In this window after execution displays a message 'NC Drill Output Completed'.
- 6. Click **Close** and **Accept** Fabrication Data Manager.

NC-Drill Output						
NC Drill Data	abase Contents					
Patch:	Tool:	Size:	Holes:			
Patch 0	Start					
	T01	0.032"	18			
	T02	0.035"	3			
	T03	0.040"	4			
Patch 0	End					
	Total:		25			
	<ul> <li>Prepare Drill Database for Template only</li> <li>Check for multiple holes in the same XY position</li> </ul>					
Dutput To: Template						
	NC-Drill Ou	utput Completed				
		Execute	Close			

## **Preview NC- Drill data:**

Select Tools Template (notes)



## Introduction to PCB Assembly outputs

When the manufacturing volumes are high and high in component density, designs like SMD, manual assembly becomes difficult. In such cases, automatic component insertion

technique is required in a specified format to drive automated machinery to position the components at exact location. As this data is machine specific, there is a provision for both Generic as well as IPC-D-355 standard format of the output. Generate PCB Assembly outputs.

1. Select Fabrication → Setup → PCB Assembly Output.

🔚 Fabrication Data Manager				
Category	PCB Assembly Outputs			
Gerber Photoplotter Data	Ge	meric IPC-D-355	,	
Gerber Artworks Gerber Mechanical Plots		General		
⊡-NC-Drill Data		Coordinates Unit	Inches and Degrees	
Output Contents		Sequence of Component	PMD Components first	
PCB Assembly Output		Sort	Part Name	
Bare Board Testing		Job Description		
		Job Name	RC Coupled Amplifier	
		Title	RC Coupled Amplifier	
		Drawing	?	
		Revision	?	
		PCB Parameters		
		Thickness of PCB	0.05906"	
		Height of PMD components over PCB	0.03937"	
		Number of PCB Fiducials	2	
		PCB Fiducials Size	0.0600''	
		PCB Fiducials Horizontal Shift	0.0900''	
		PCB Fiducials Veritcal Shift	0.0900''	
		Output PCB Patern Tolerance		
		Output PMD Components Tolerance		
			Execute	
			Accept Cancel	

- 2. Click on **Execute**. Output is a extension .**PCK** extension file.
- 3. Save it
- 4. Select IPC-D-355 tab on top of the Fabrication Data Manager window.
- 5. Click on **Execute**. Output is a **.355** extension file, save it in the desired location.

# Introduction to Bare Board Testing outputs

Bare Board Test (BBT) is like quality check usually done by the fabricator to check the open & short circuits in and between the tracks. A supply of 50 volts is passed through the tracks and this job is done by automated machines. A probe sort of thing is inserted through the pads so as to probe. A netlist is generated from the gerber (it is not generated from the design) so that to cross check the actual physical PCB manufactured with respect to actual gerber in order to find out the fabrication errors. This is a special on board test outputs generated in Generic and IPC-D-356A.

#### Generate Bare Board Test outputs

Select Fabrication → Setup. To setup parameters for Bare Board outputs choose Bare Board Testing node from Fabrication data manager choose either format and set the required parameters. Click Execute to generate the Bare Board Testing output in either Generic or IPC-D-356A format.

🔚 Fabrication Data Manager				
Category	Bare Board Test Output			
Gerber Photoplotter Data		neric IPC-D-356A		
Gerber Mechanical Plots		General		
⊡- NC-Drill Data		Co-ordinates Reference	Board Xmin, Ymin	
Output Contents		Board Viewed	From Top	
PCB Assembly Output		Coordinates Unit	Inches	
Bare Board Testing		Contents (Connection Points:)		
		Component Pads X,Y Coordinates		
		Component Pads X,Y Sizes		
		Via Hole Pads X,Y Coordinates		
		Via Hole Pads X,Y Sizes		
		Contents (Conductors):		
		Trace Segments		
		Copper Pour Areas		
		Plain Copper Items		
			Execute	
			Accept Cancel	

The output for Bare Board Testing is obtained as a text file.