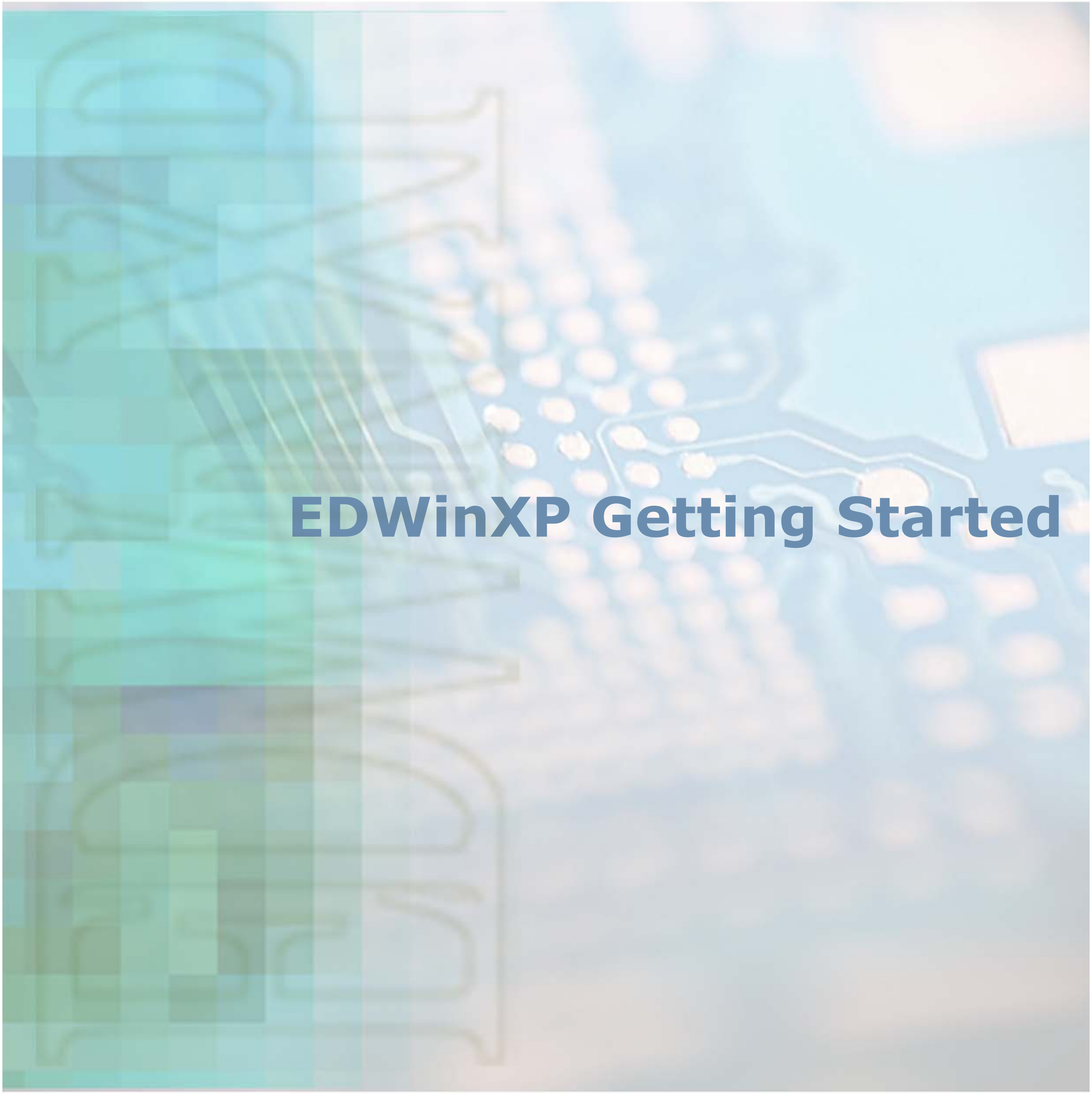
ELECTRONIC DESIGN for WINDOWS EDWinXP



EDWinXP Getting Started

VISIONICS

CONTENTS

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**.
2. 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.

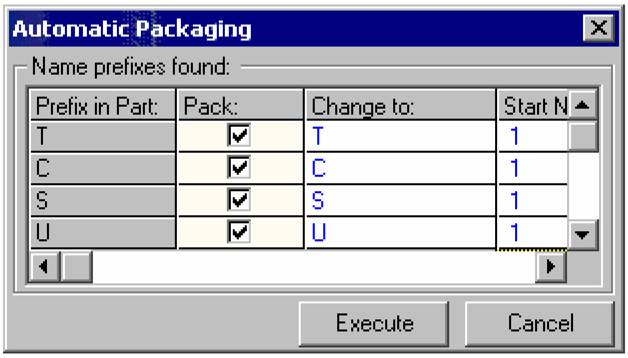
4. Voltage source, Ground and signal generators can be obtained from the Component Browser. These are external sources (which will not be housed on the PCB) and hence connections to them can be shown with the help of connectors. Select **connectors** from the component browser, then select **List Connector** 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**.**



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.

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

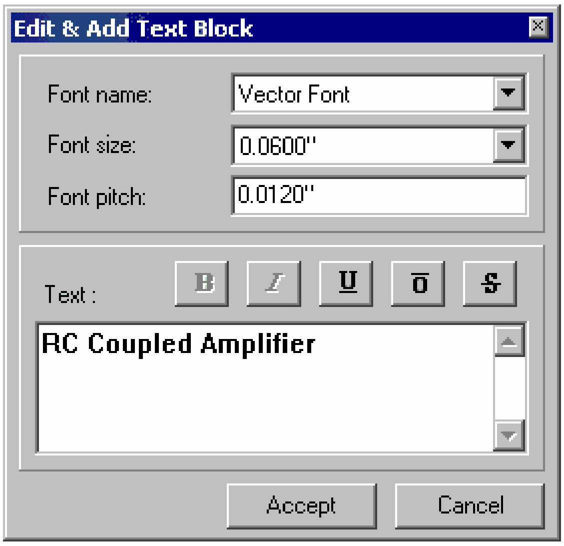
3. Before creating the connections, enable **Preferences/Instant net name** and **Preferences/Instant wire label.**
4. Right click on the work space and select **Connections**.
5. Enable **Pin to Pin** option tool whenever an entry or pin is clicked for making a connection. This ensures that the connection 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

1. 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

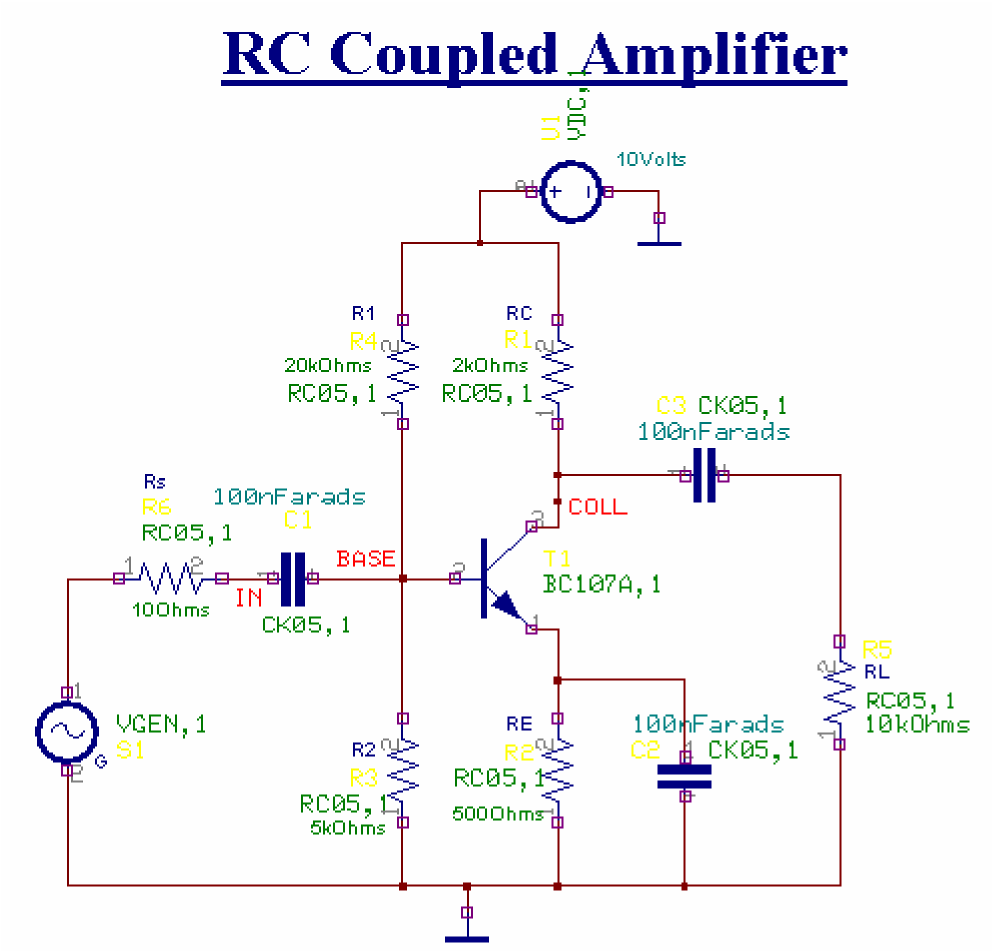
1. **Page notes** appear only on the current page (e.g. Page No.) of the project whereas **Design notes** appear on all the pages (e.g. Project name) of the selected circuit.
2. Select **Tools ➔ Page Notes** toolbar to enable a set of tools for creating page notes. Right click and enable



3. 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

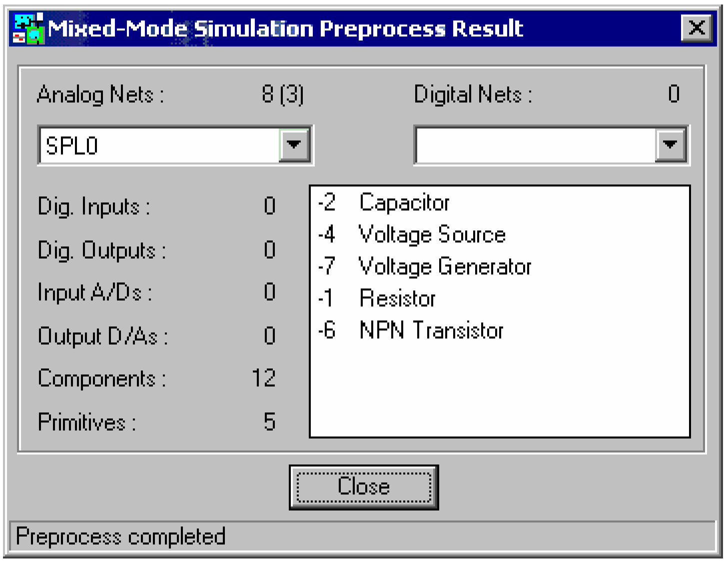
Mixed Mode Simulation

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.



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.

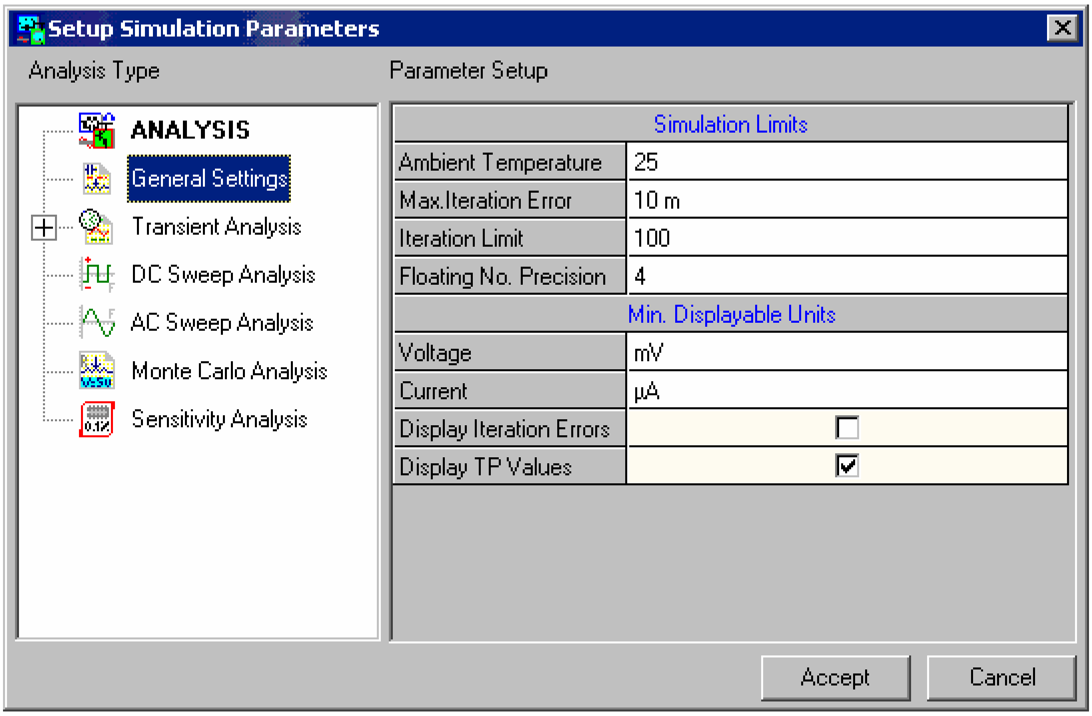
Assign Component Parameter Values

1. 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.



3. Set the values as

Simulation Limits:

Ambient Temperature	**25**
Max. Iteration Error	**100m**
Iteration Limit	**100**
Floating No. Precision	**4**

Min. Displayable Unit:

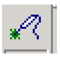
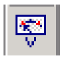
Voltage	**mV**
Current	**μA**

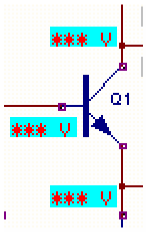
4. Click **Accept** button to accept the changes and to automatically switch to Analysis option

.

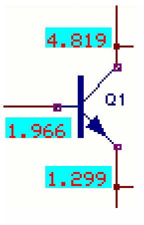
Bias Point Calculation

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

1. 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 ➔ Instruments**  **➔ Test points ➔**  **Voltage TP**



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

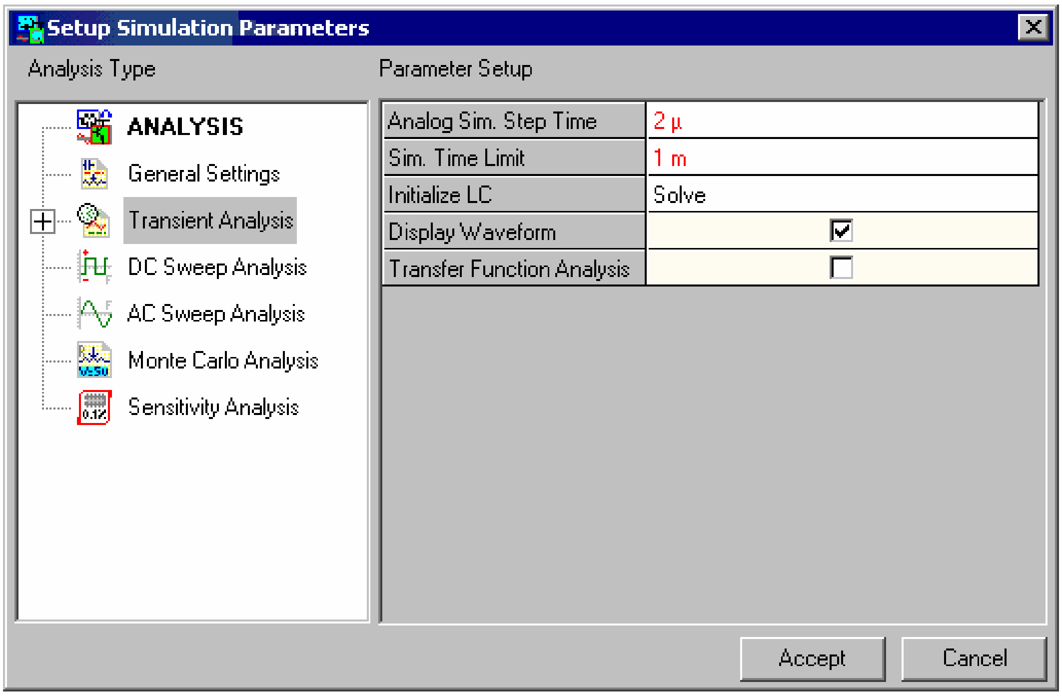
Transient analysis is used to view the input and output with respect to time. Inorder to view the performance of the circuit in time frame run transient analysis.

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

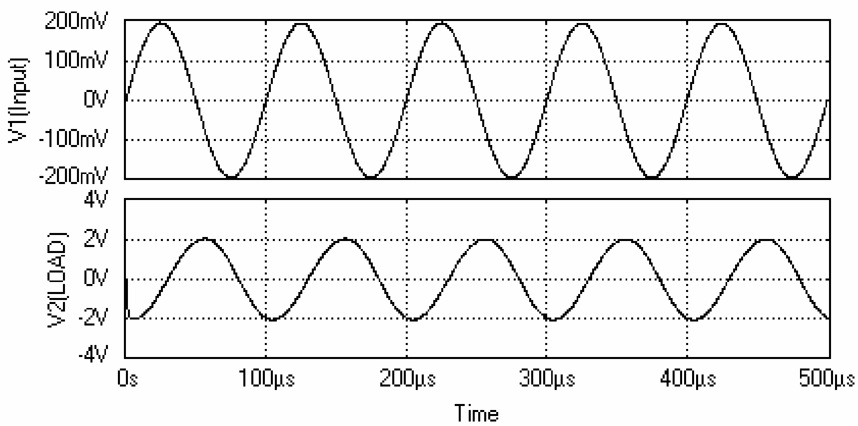
3. Enter following values:

   Analog simulation step time: 2µ

   Simulation time limit: 1m



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



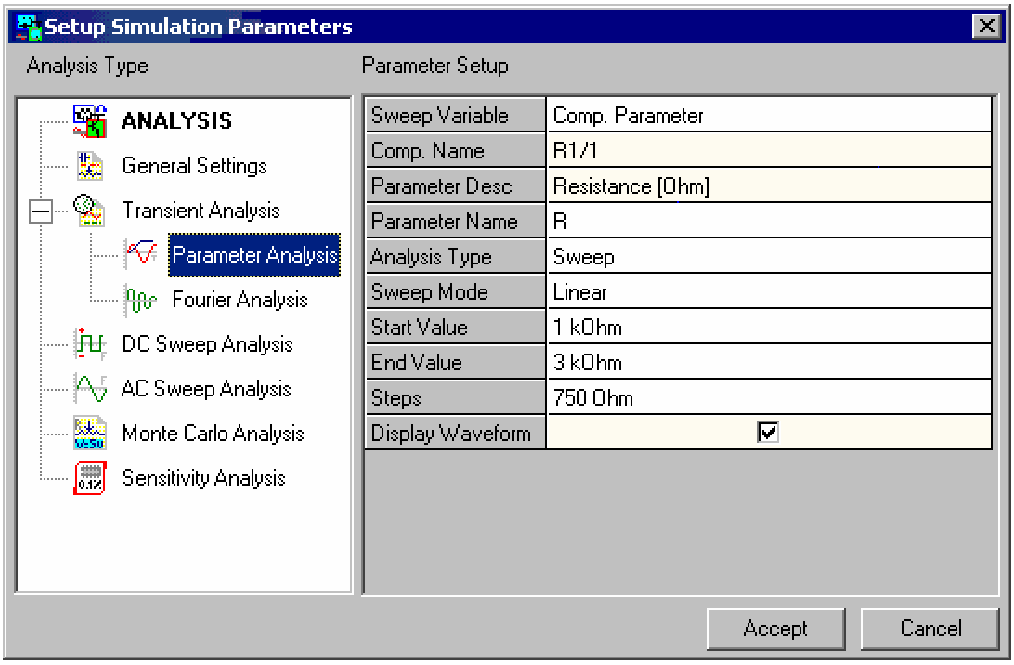
Parameter Analysis

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.

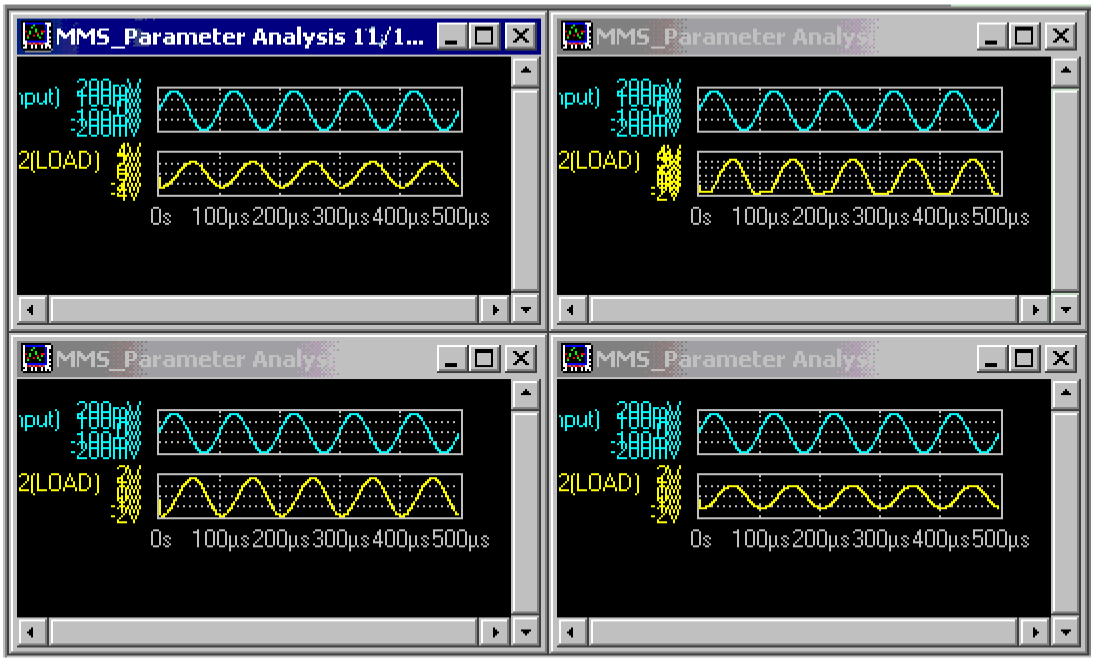
1. Select **Simulation ➔ Analysis.**
2. Select **Parameter Analysis** from the tree view.

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**



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.

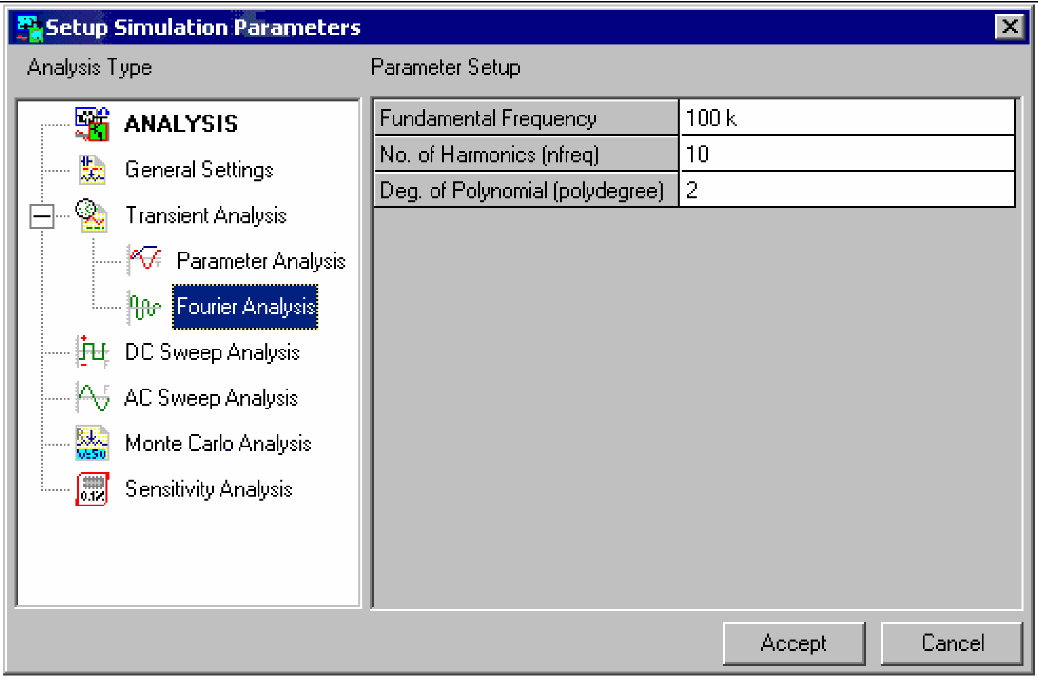


Fourier Analysis

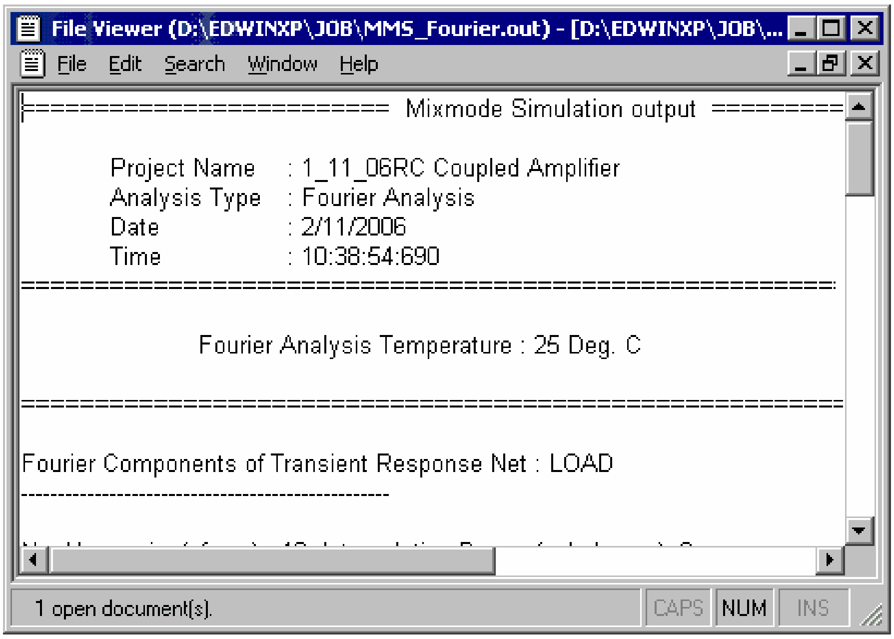
Fourier analysis helps to calculate the total harmonic distortion of analog waveforms generated during transient analysis.

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**



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.

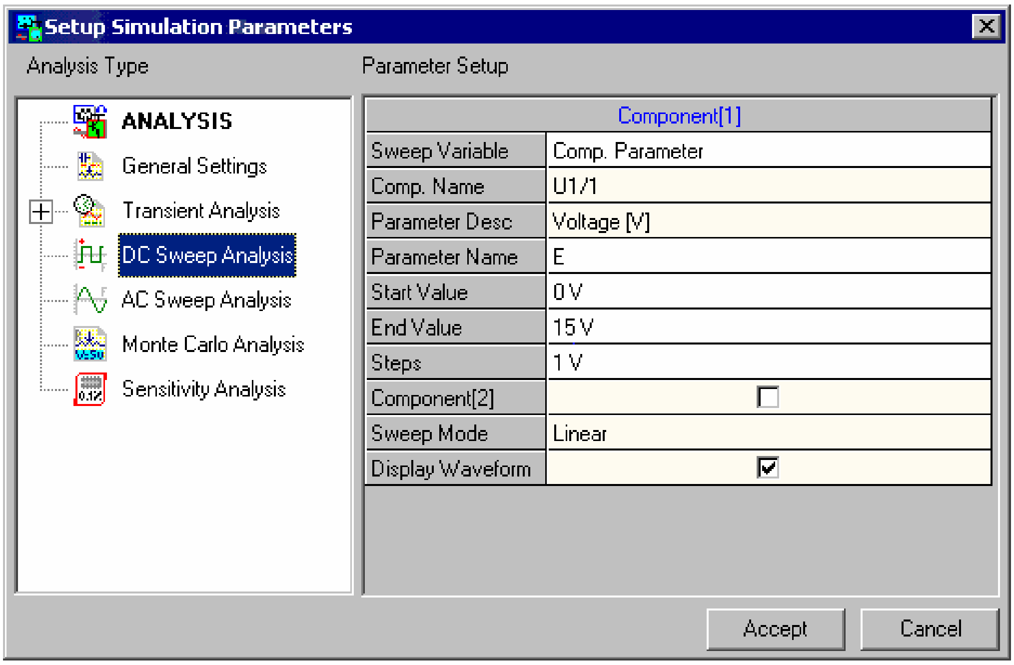


DC Sweep Analysis

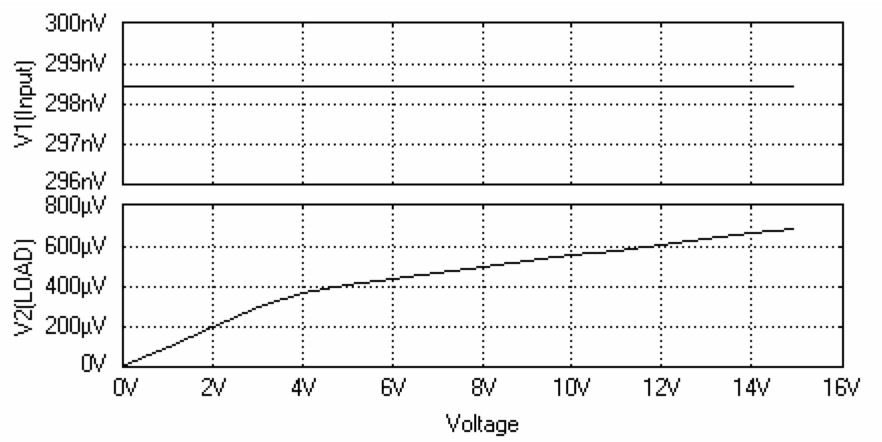
DC Sweep analysis helps to observe the circuit operating parameters by varying the value of a component. One or more sources (current and voltage) may be stepped over a range. Results of the analysis at each source value may then be viewed in the waveform generator.

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**



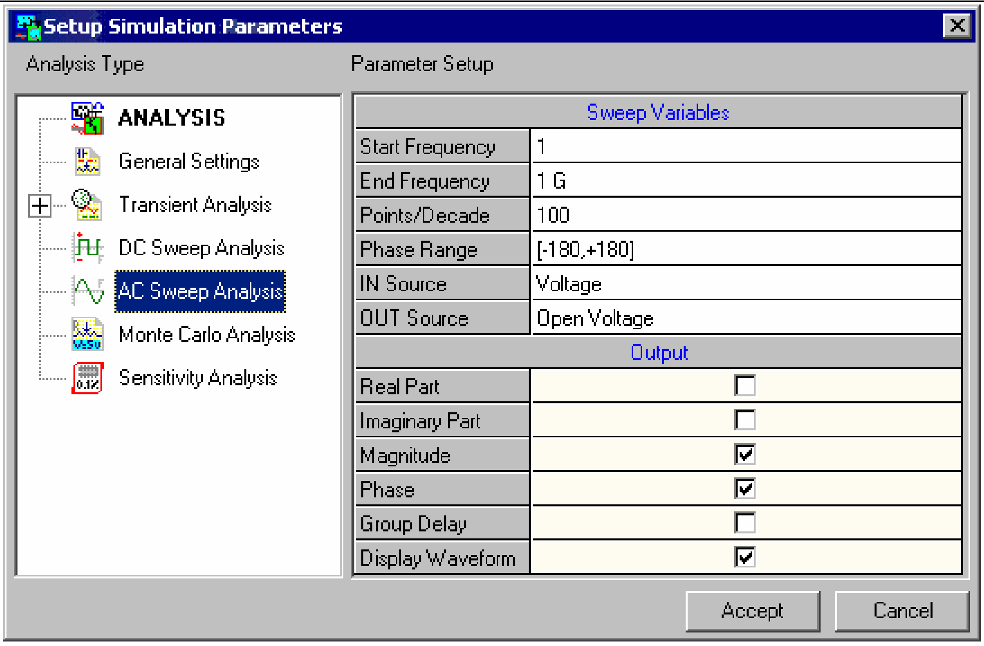
5. Check **Display Waveform** check box to display the waveform.
6. Select **Accept** to switch into Analysis Option.
7. Check **DC Sweep Analysis** check box.
8. Click **Start**



AC Sweep Analysis

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

1. Select **Instruments ➔** **Set Reference Points** (function tool) ➔ **AC IN+** and click on the input net.
2. Select **Instruments ➔** **Set Reference Points** (function tool) ➔ **AC IN-** option 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-** 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.

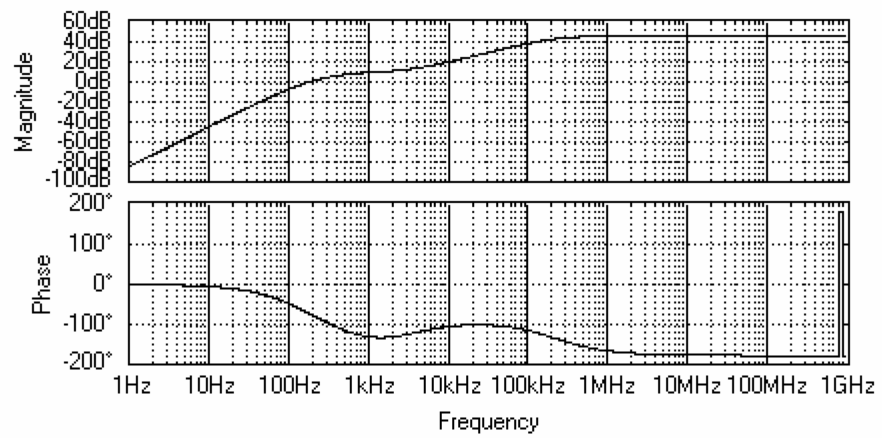


9. Set the values for the following parameters as

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.

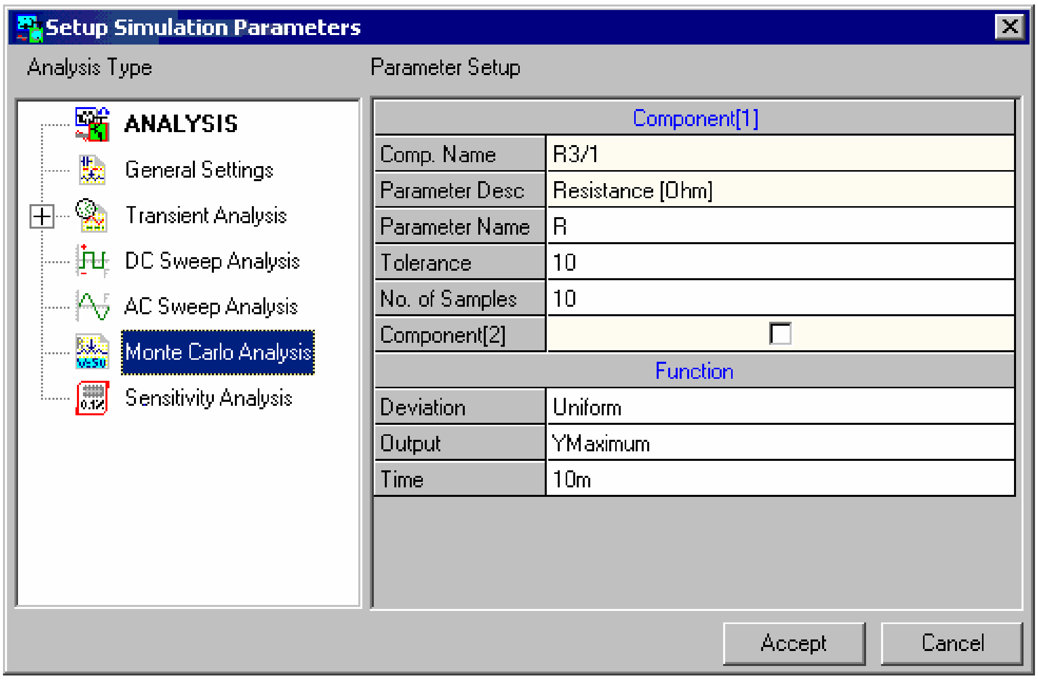
11. Check **AC Sweep Analysis** check box.
12. Click **Start** for AC Simulation Pass to take place.



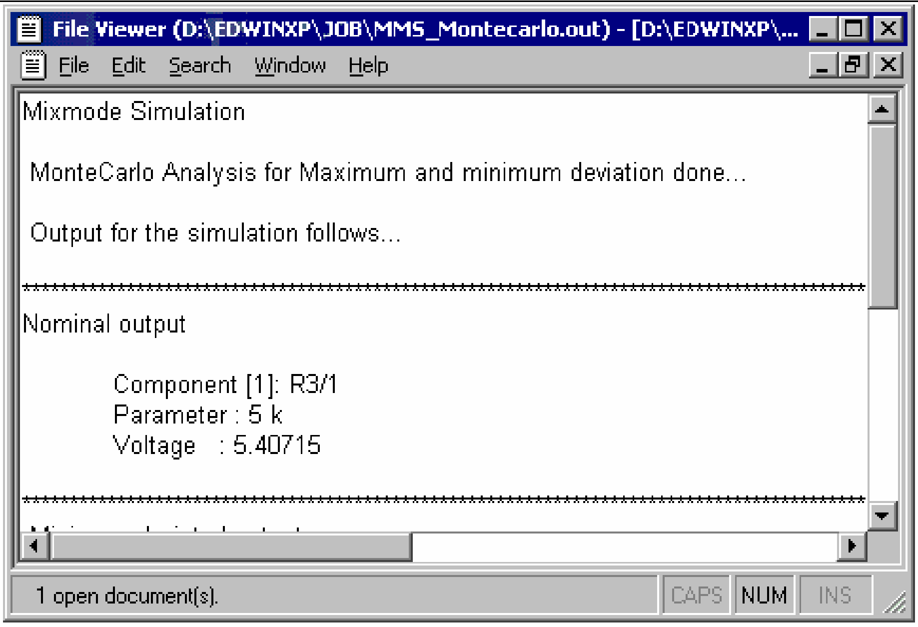
Monte Carlo Analysis

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.**



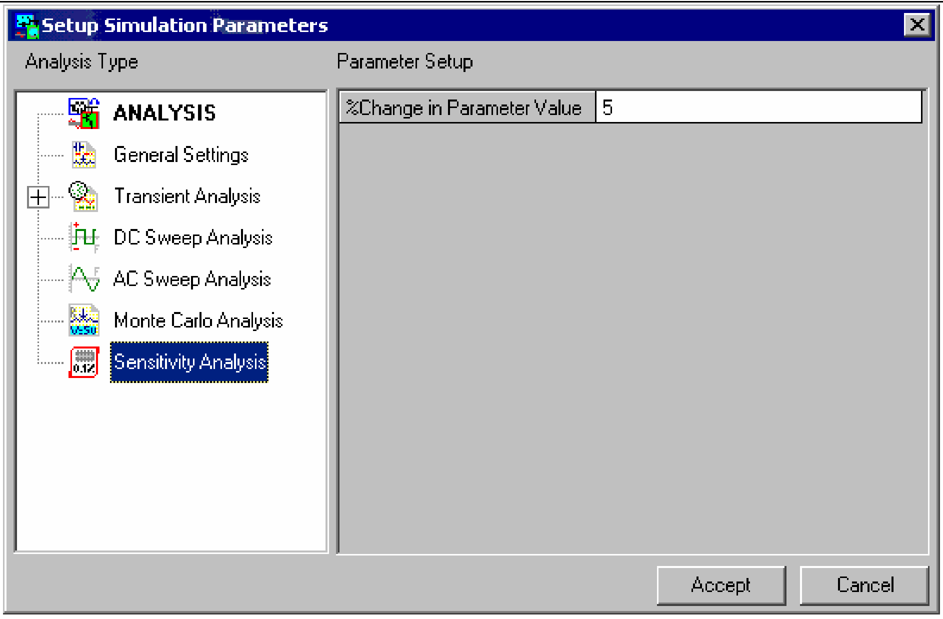
The result is displayed in a text file.



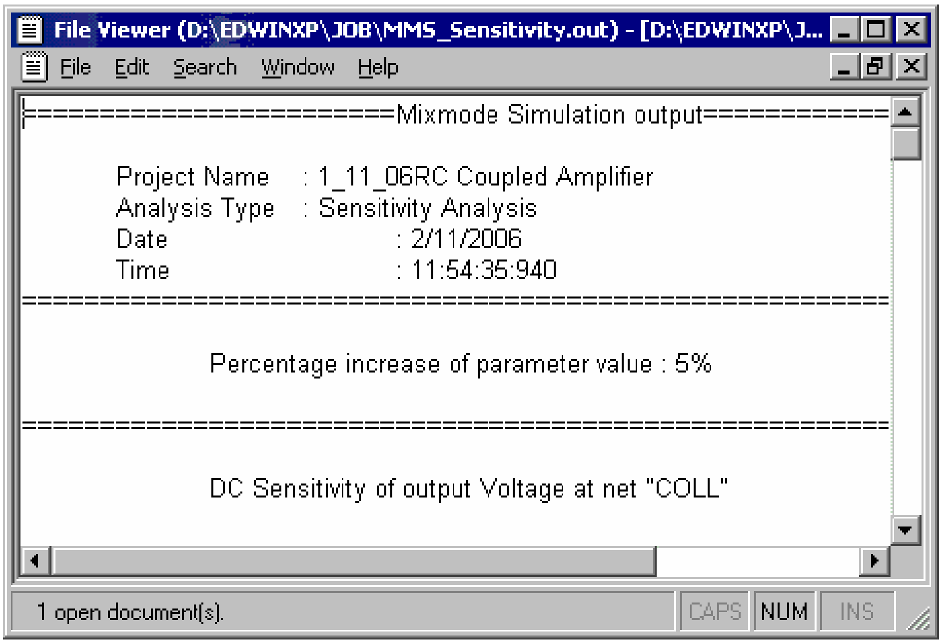
Sensitivity Analysis

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.



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.



EDSpice Simulator

Steps for EDSpice Simulation

Preprocess

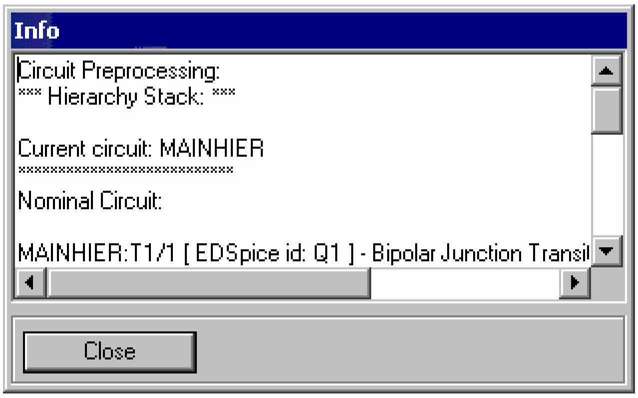
Preprocessing confirms whether the circuit is ready for simulation. Preprocessing must be performed at all times, when elements have been added or deleted from the circuit or connectivity between them is changed.

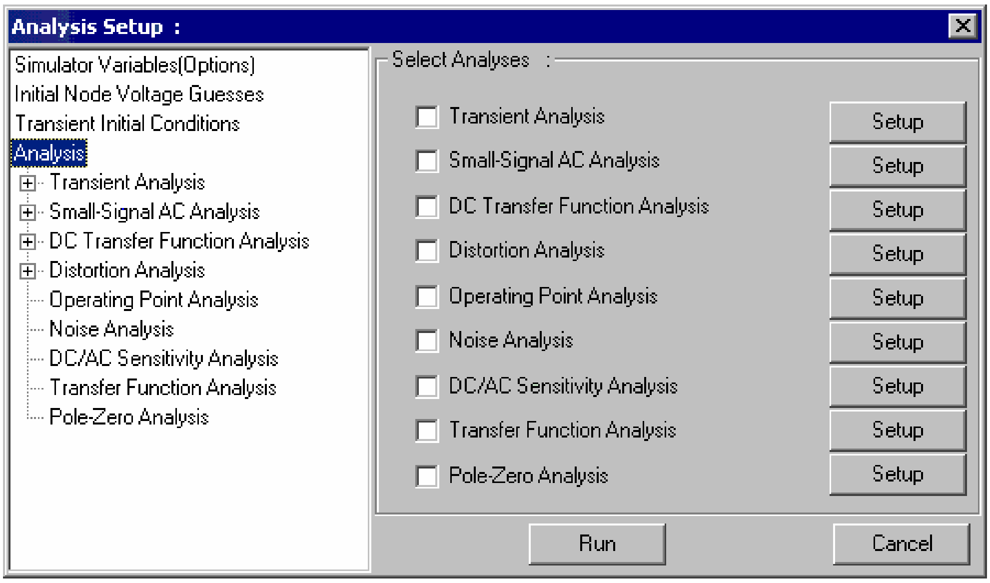
Inorder to preprocess the circuit select **Simulation ➔ Preprocess**

Click **Close**.

Analysis

Select Simulation ➔ Analysis

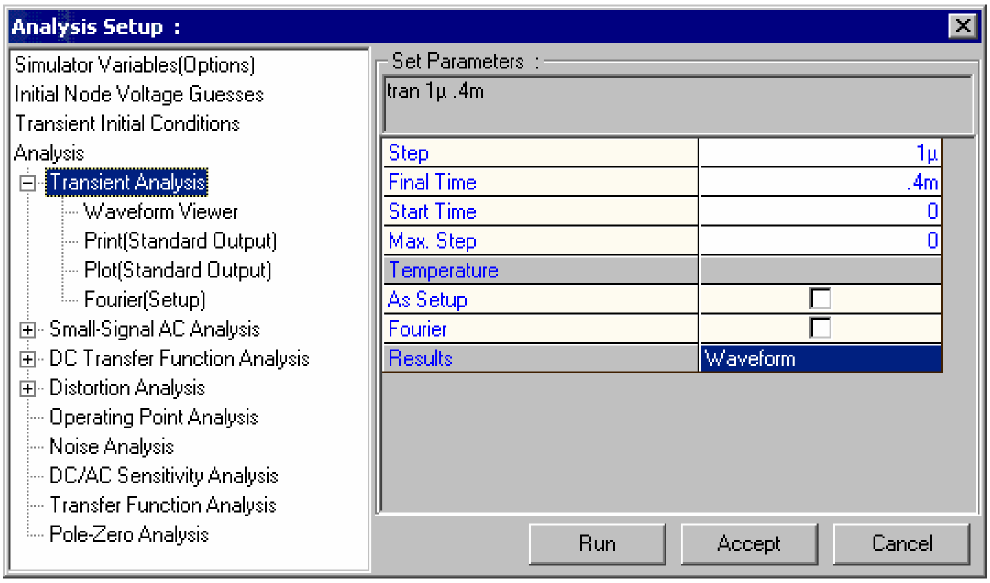




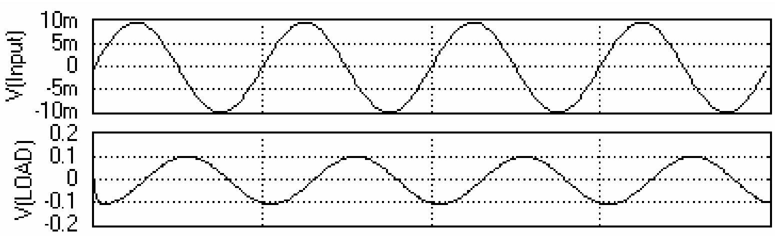
Transient Analysis

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

Step	**1**
Final time	**. 4m**
Start time	**0**



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



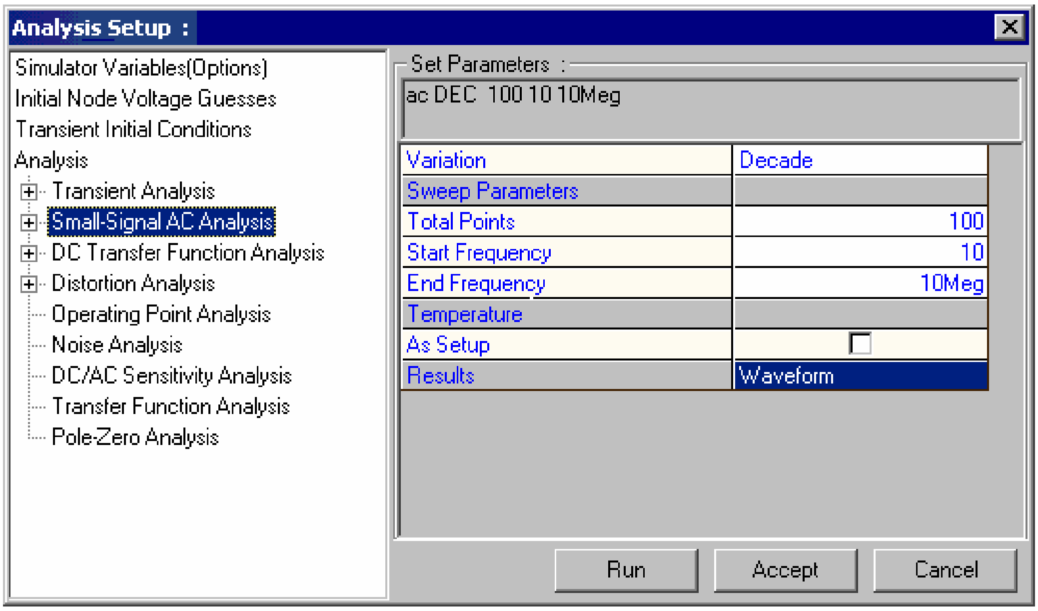
Small Signal AC Analysis

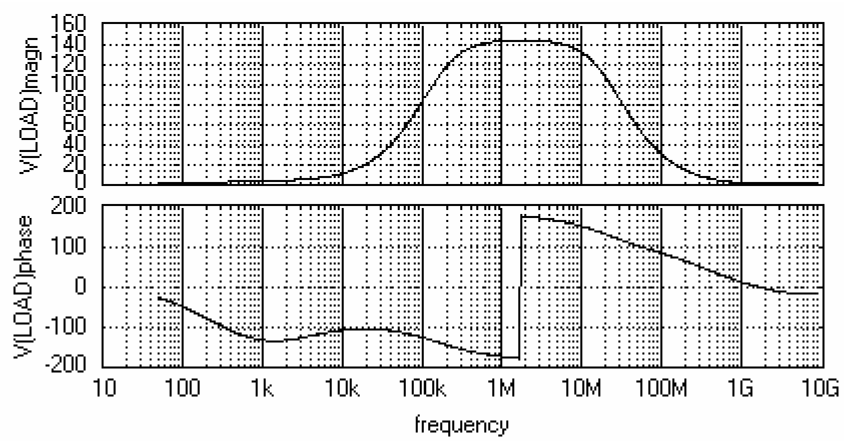
This analysis is used to obtain the small signal AC behaviour of the circuit.

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

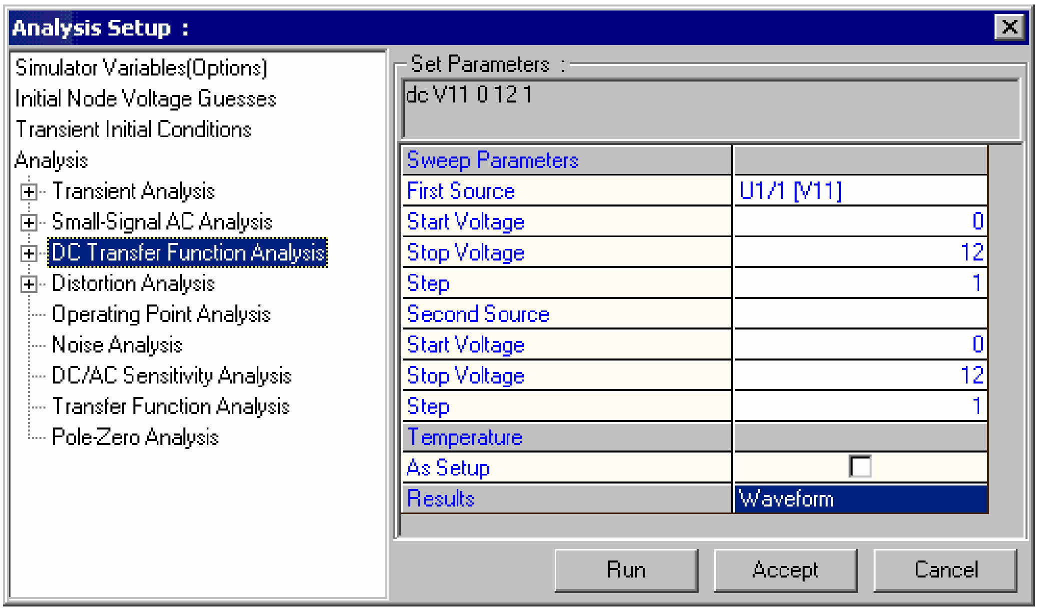
DC Transfer Function Analysis gives the behaviour of the circuit with respect to the varied voltage/current.

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

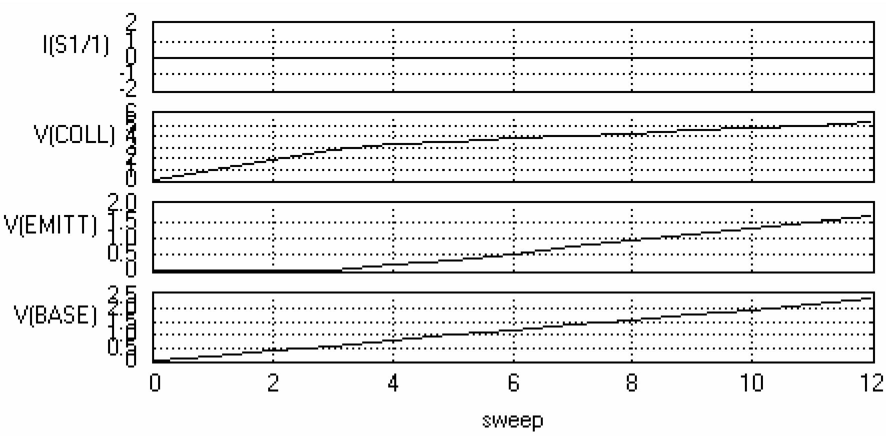
   Start Voltage **0V**

   Stop Voltage **12V**

   Step **1**



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



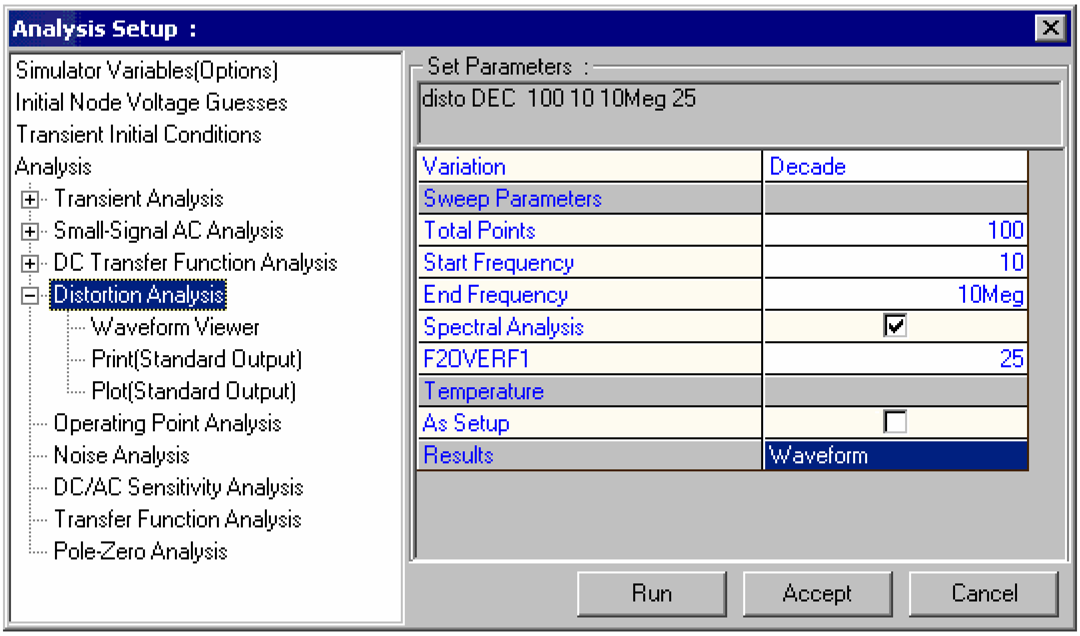
Distortion Analysis

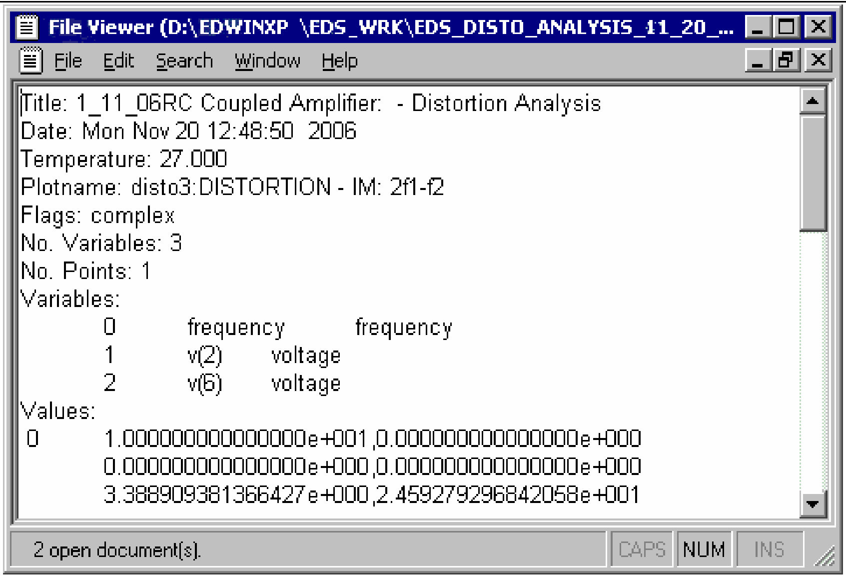
The distortion analysis computes steady state harmonic and intermodulation products for small input signal magnitude. If signals of a single frequency are specified as the input to the circuit, the complex values of the second and third harmonics are determined at every point in the circuit. If two frequencies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

1. 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.

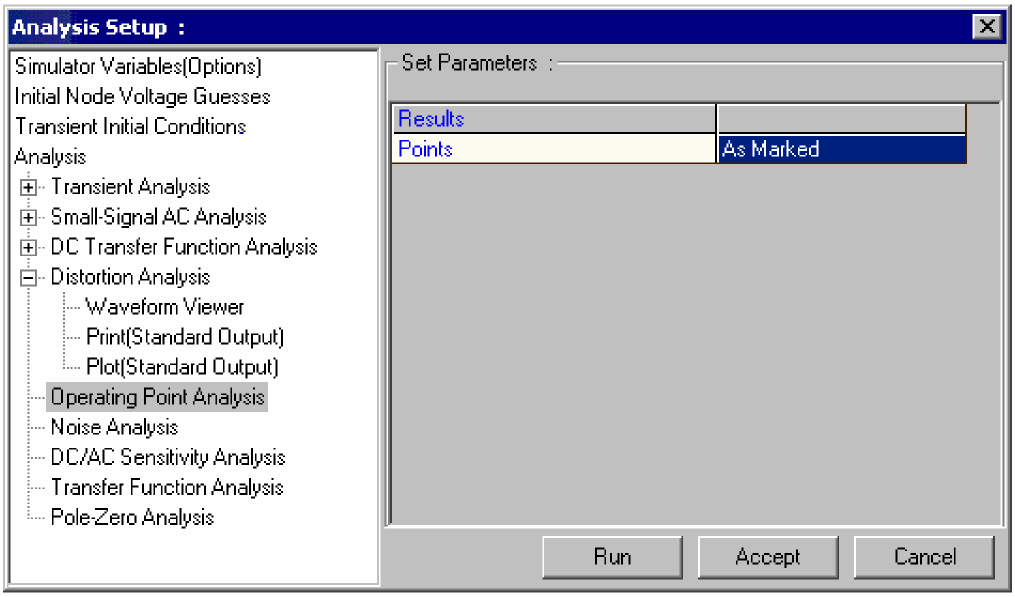


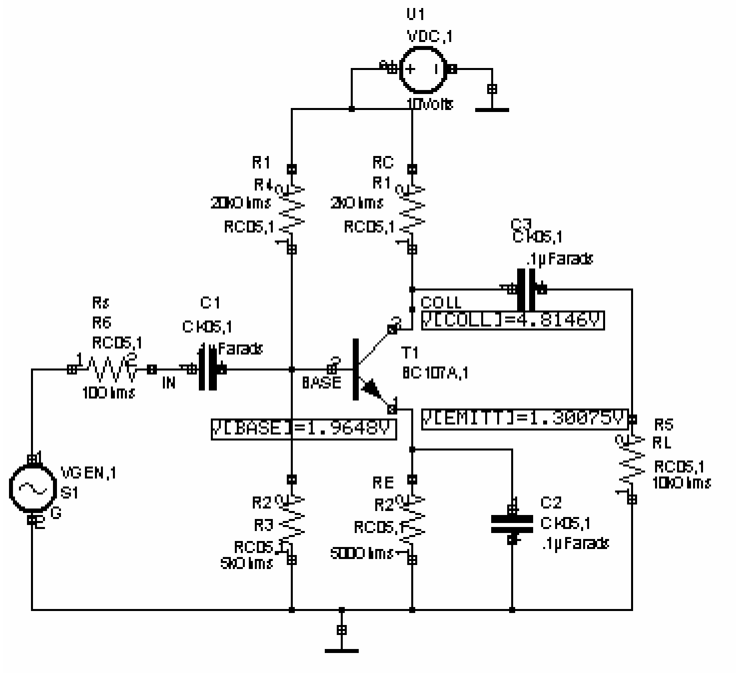

File Viewer (D:\EDWINXP \EDS_WRK\EDS_DISTO_ANALYSIS_11_20_...
File Edit Search Window Help
Title: 1_11_06RC Coupled Amplifier: - Distortion Analysis
Date: Mon Nov 20 12:48:50 2006
Temperature: 27.000
Plotname: disto3:DISTORTION - IM: 2f1-f2
Flags: complex
No. Variables: 3
No. Points: 1
Variables:
0 frequency frequency
1 v(2) voltage
2 v(6) voltage
Values:
0 1.000000000000000e+001,0.000000000000000e+000
0.000000000000000e+000,0.000000000000000e+000
3.388909381366427e+000,2.459279296842058e+001
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.

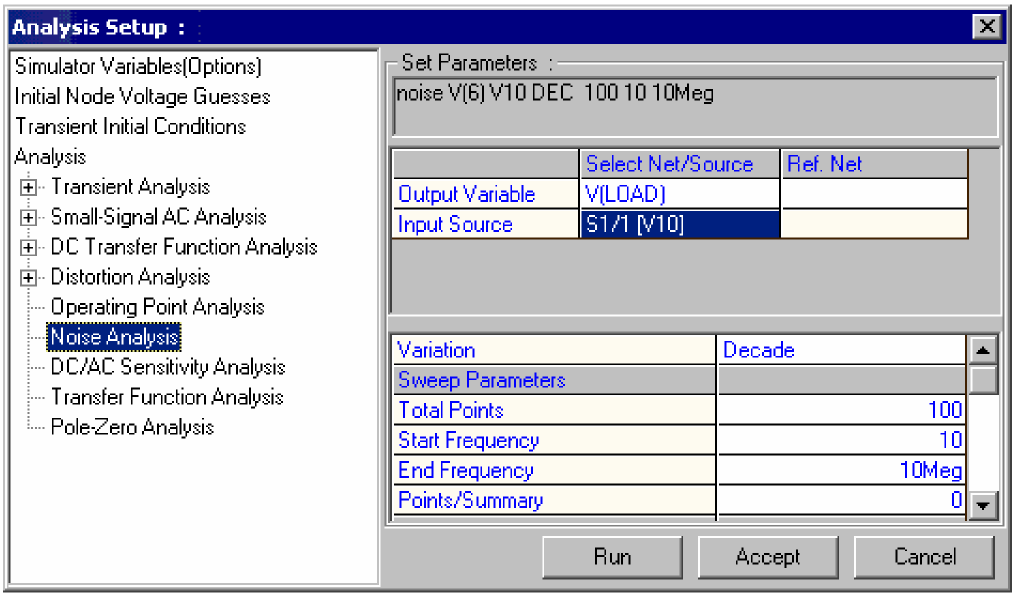



U1
VDC,1
10Volts
R1
R4
20kOhms
RCD5,1
RC
R1
2kOhms
RCD5,1
C3
CK05,1
.1μFarads
Rs
R6
RCD5,1
10Ohms
C1
CK05,1
.1μFarads
IN
BASE
COLL
V[COLL]=4.8146V
T1
BC107A,1
V[BASE]=1.9648V
V[EMITT]=1.30075V
R5
RL
RCD5,1
10kOhms
VGEN,1
S1
G
R2
R3
RCD5,1
5kOhms
RE
R2
RCD5,1
500Ohms
C2
CK05,1
.1μFarads

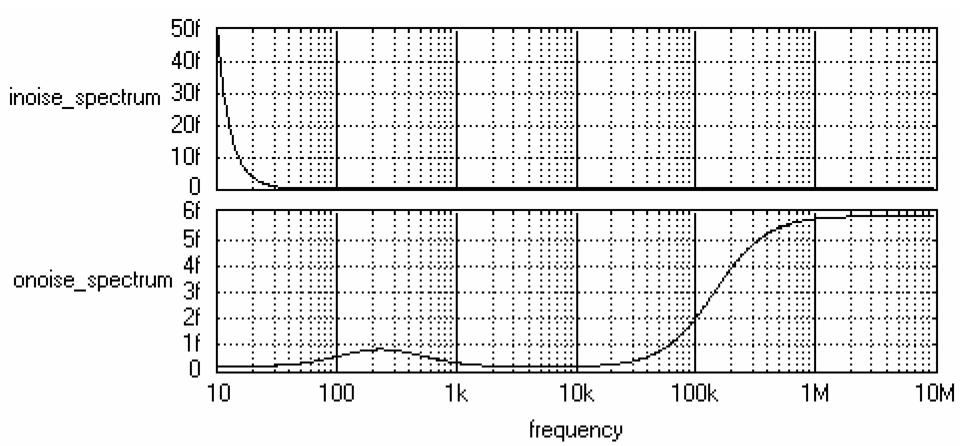
Noise Analysis

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.

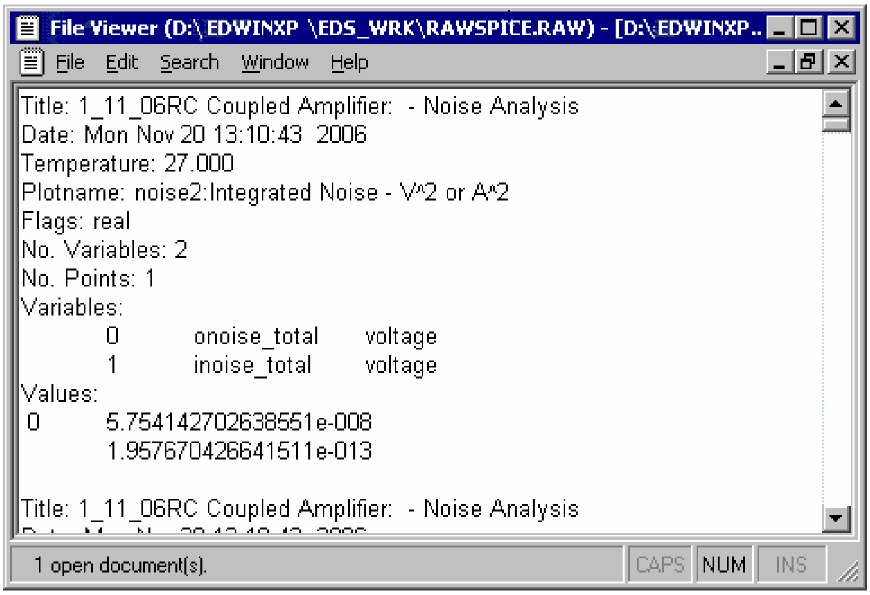
1. Select **Simulation ➔ Analysis.**
2. Select **Noise Analysis** from the tree view.



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.



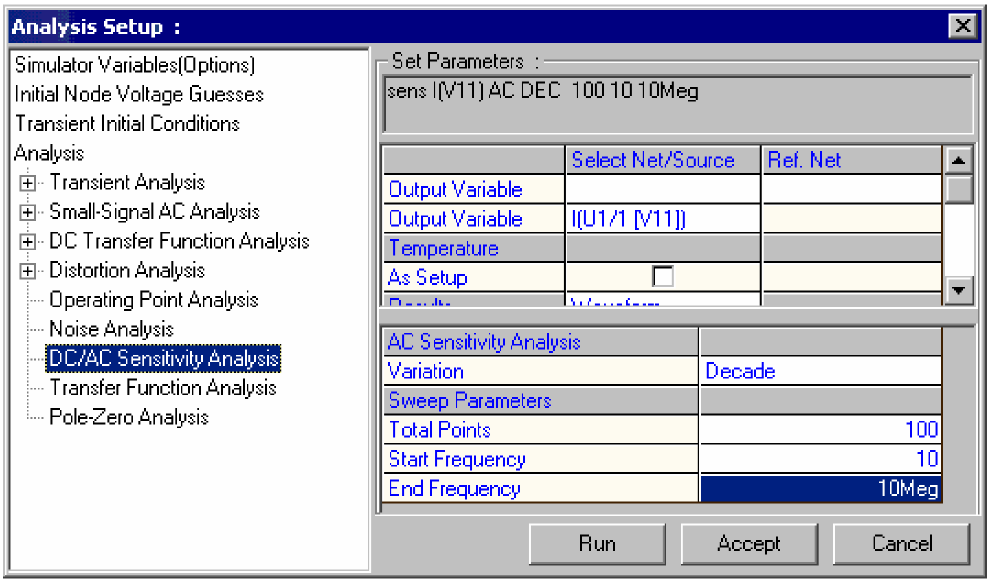
5. The total integrated Noise will be presented in the rawspice.raw file. It can be viewed from **Options ➔ View EDSpice Files ➔ Raw file,** this .raw file is pasted below.


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:
0 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).
CAPS NUM INS

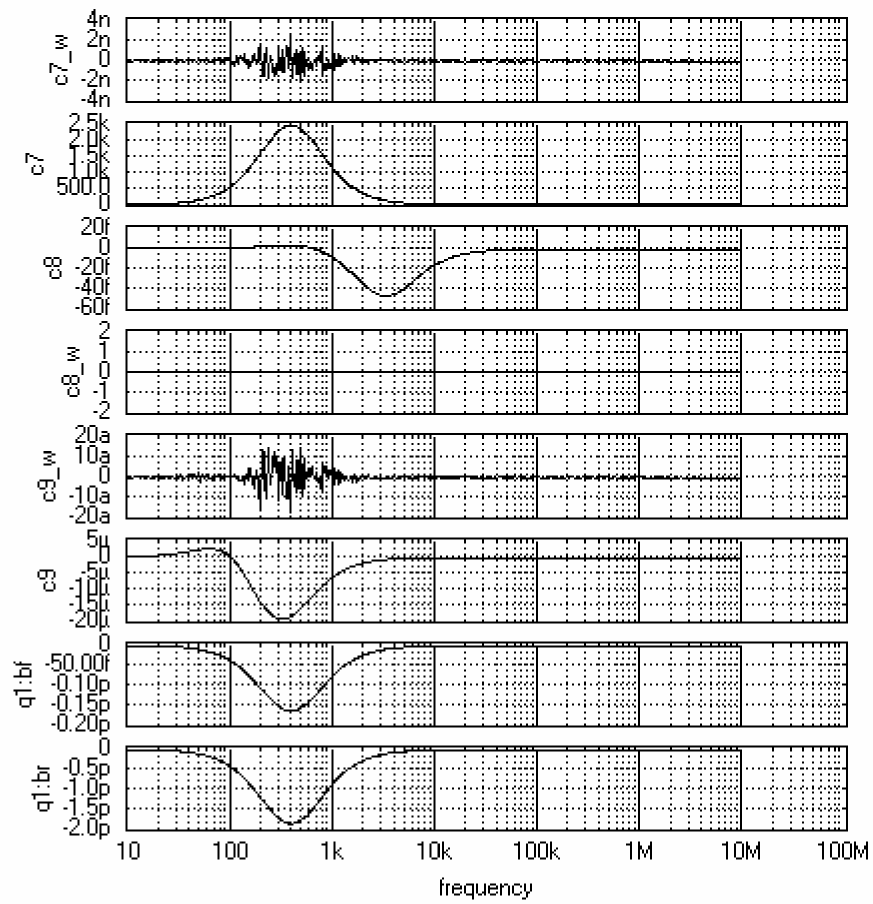
DC / AC Sensitivity Analysis

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

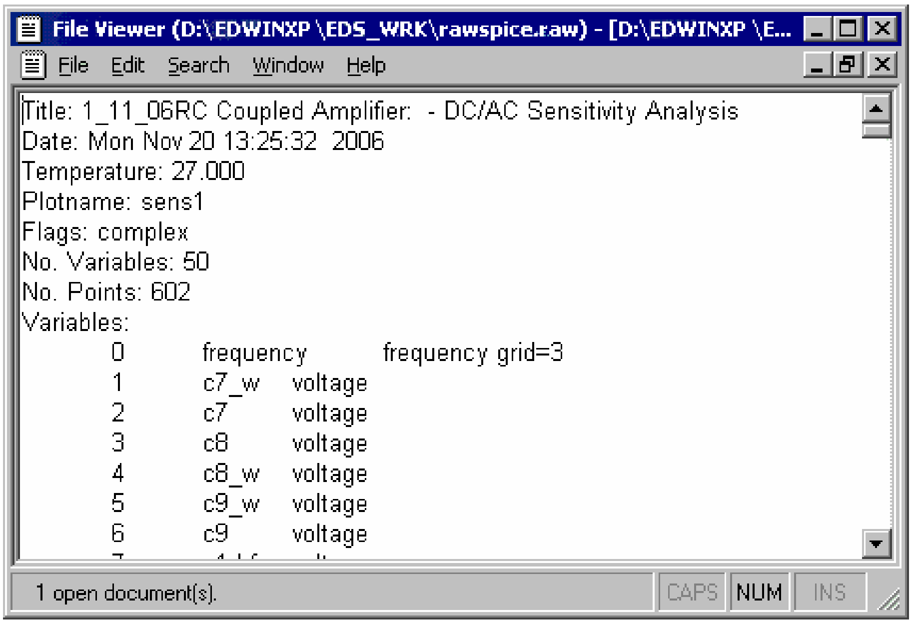
1. Select **Simulation ➔ Analysis.**
2. Select **DC / AC Sensitivity Analysis** from the tree view.



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



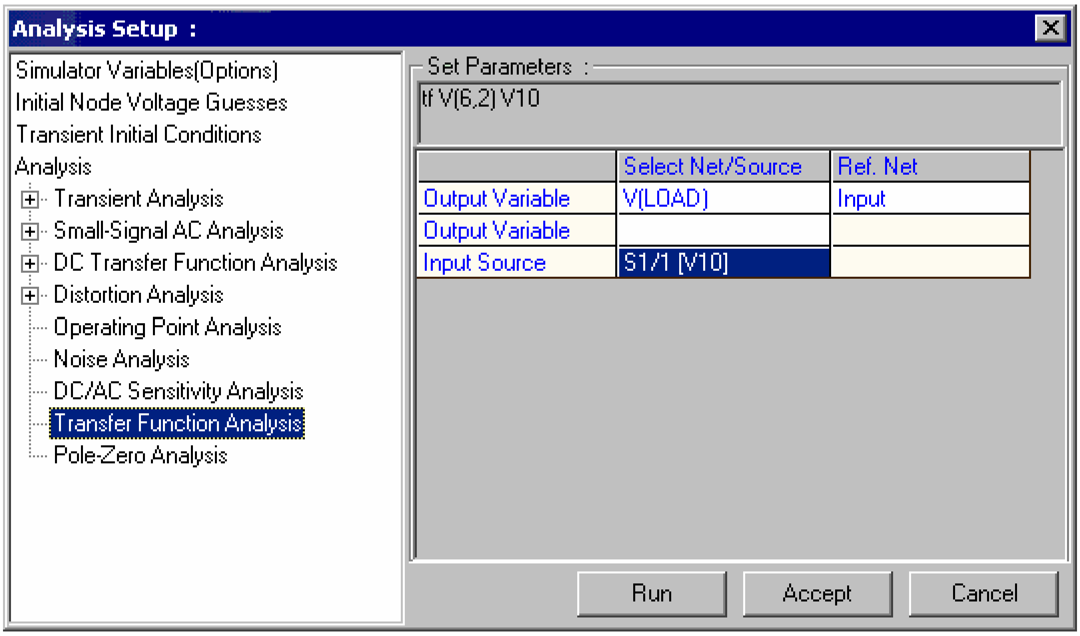
5. After analysis the results of the **DC Sensitivity Analysis** will be presented in the rawspice.raw file, open this file from **Options View EDSpice Files Rawfile.**
6. The results of **AC Sensitivity Analysis** will be presented in the Wave form viewer.



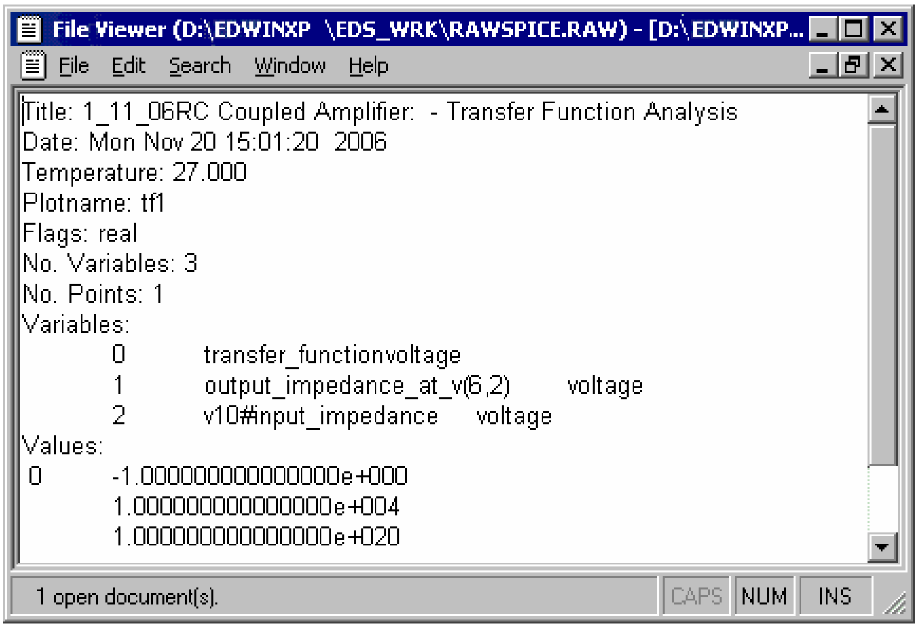
Transfer Function Analysis

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.



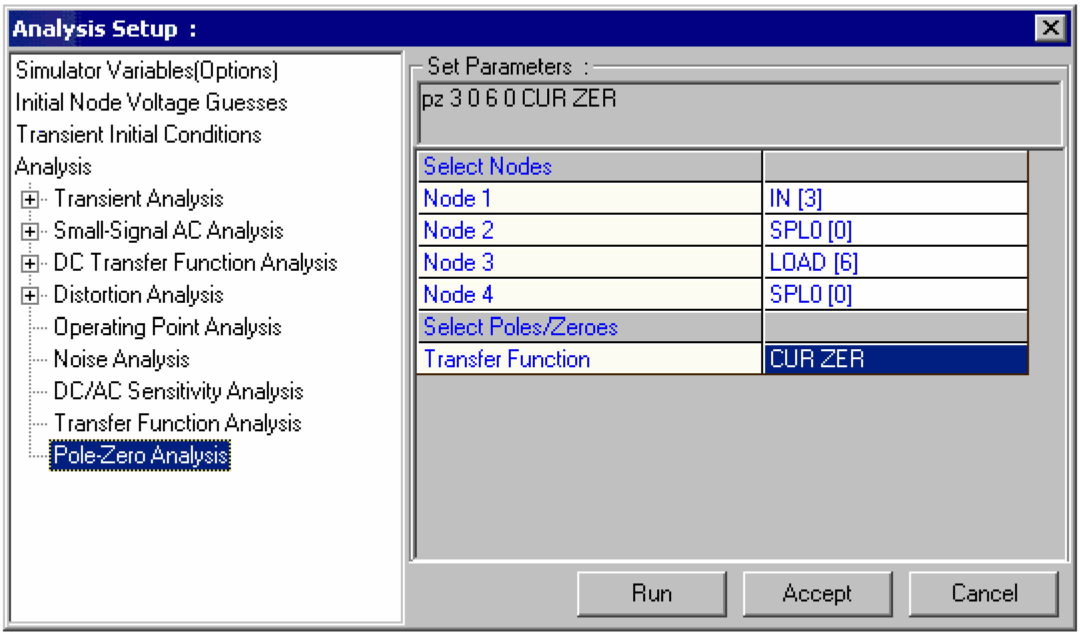
4. Click **Run** button to execute the analysis.
5. After the analysis, the results will be displayed in the .raw file. Open this file from **Options ➔ View EDSpice Files ➔ Rawfile.**



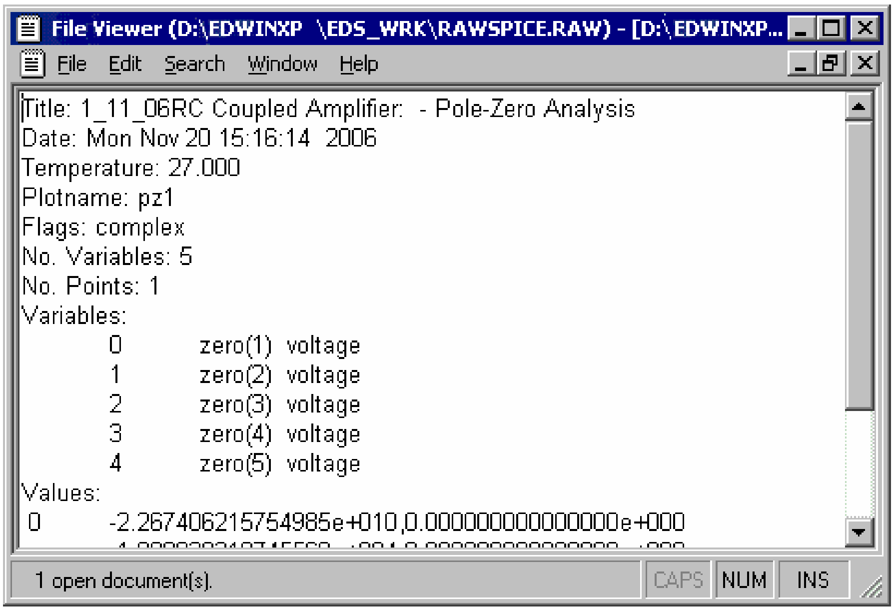
Pole- Zero Analysis

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.



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**.



PCB LAYOUT

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

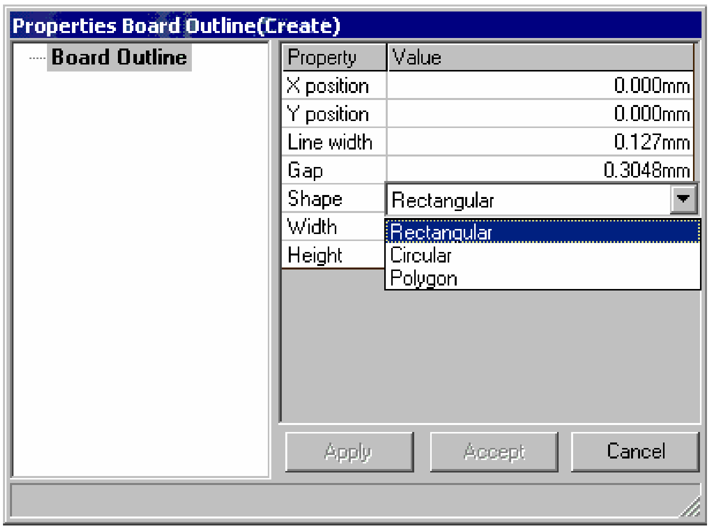
Right click **PCB Layout** and select **Edit PCB Layout** from the list.

Or

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

Define board outline

1. Select **Tools ➔ Board Outline**
2. 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.
3. 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.

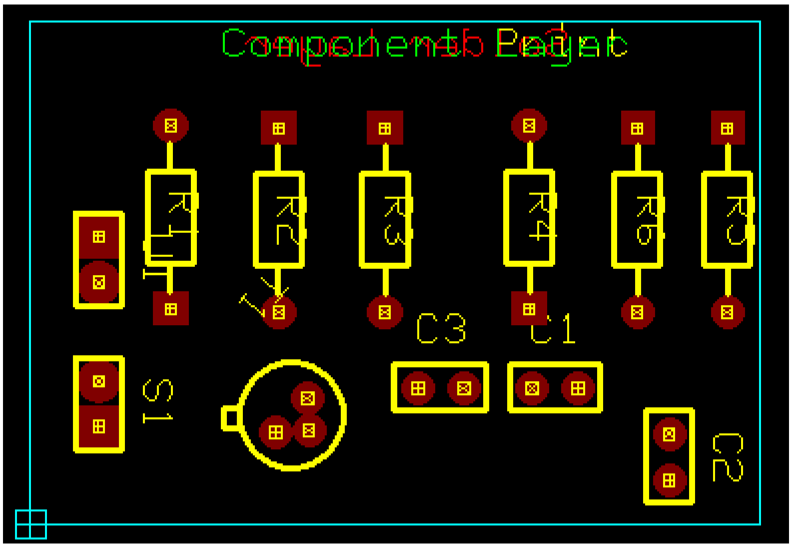


Relocating the components

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.

1. 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.

5. 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
7. 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:

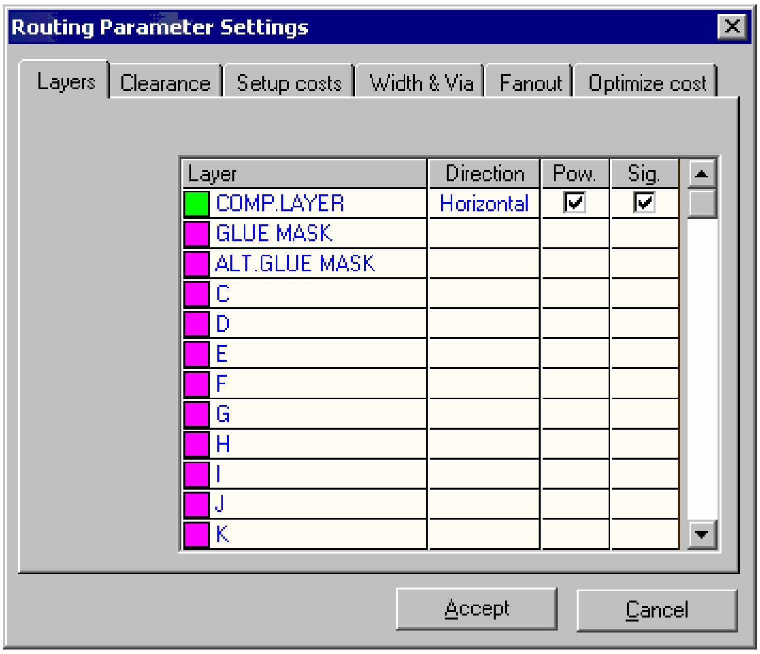
1. Autorouter-Arizona
2. Autorouter-Specctra
3. Autorouter-Maxroute

Arizona Autorouter

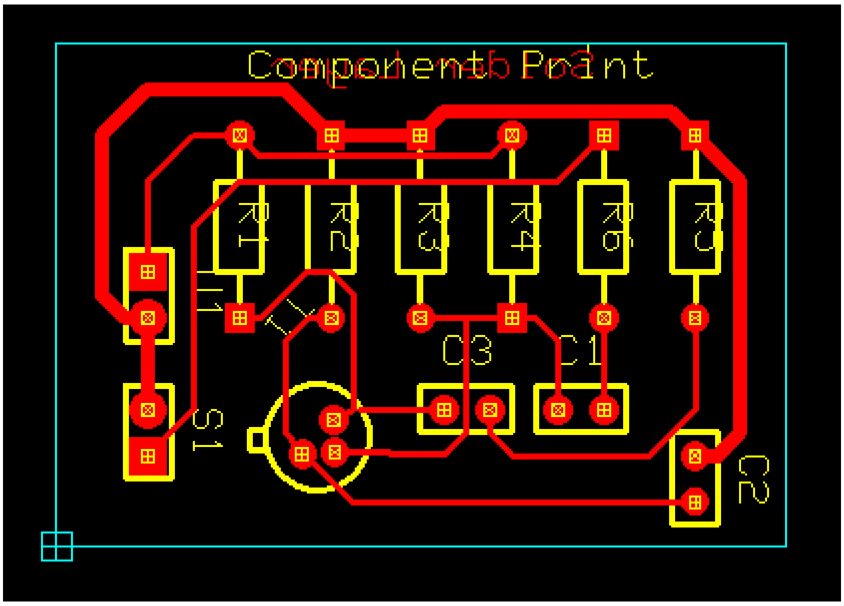
Arizona auto router is an integrated module of the EDWinXP. It uses its own temporary project and simplified graphics. The Arizona auto router allows routing the traces of a PCB Layout automatically.

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



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.

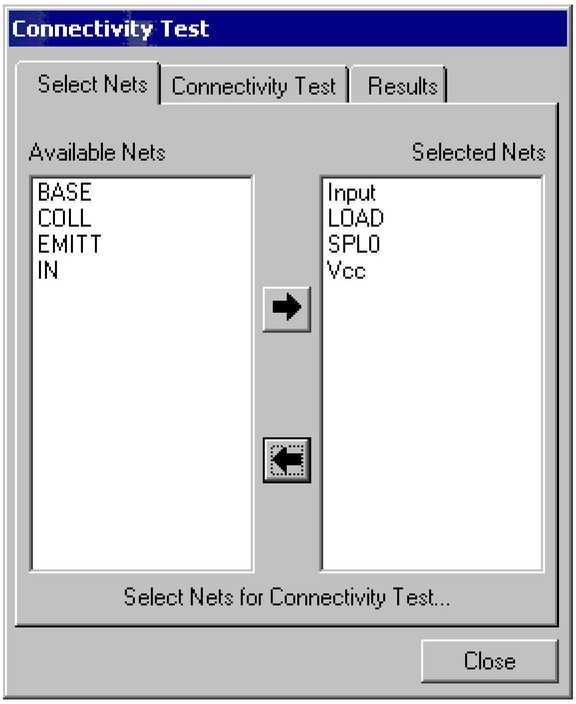
Testing the Board

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.
4. 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.
6. Perform this test until **"Tested Nets – Fully Connected"** message is displayed in this window.

Design Rule Check (DRC)

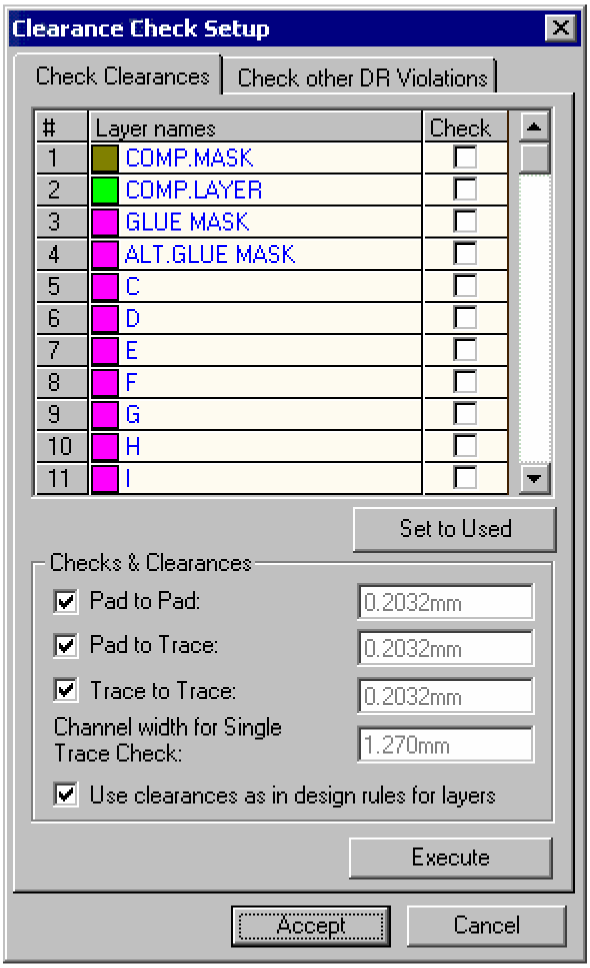
This utility is used to create an error free board to enhance the efficiency of your board. It automatically smoothes, miters, and checks for both aesthetic and manufacturing problems that might have been created in the process of manual or automatic routing. This test helps us to check the clearance 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.

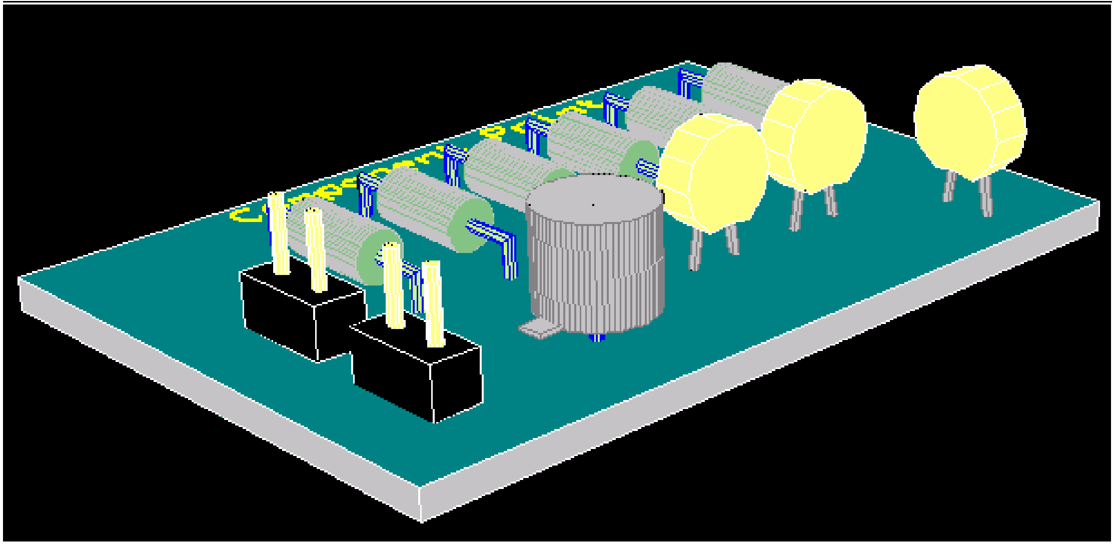
Click **Execute**.



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

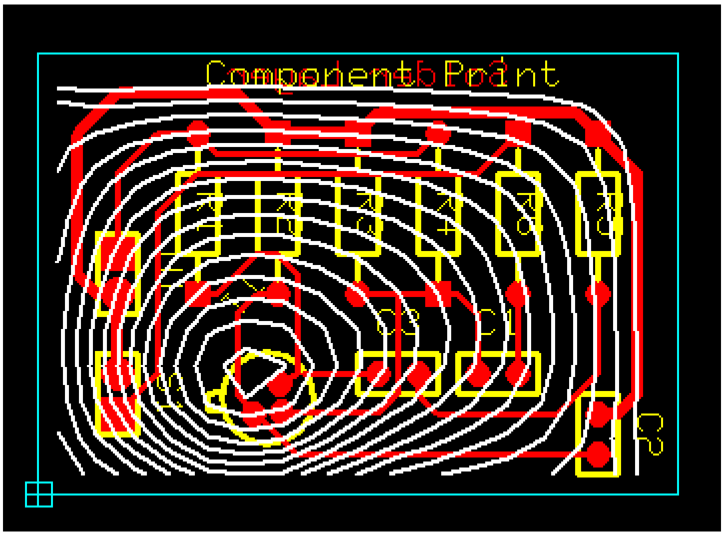
Electromagnetic Analyzer

Thermal Analyzer

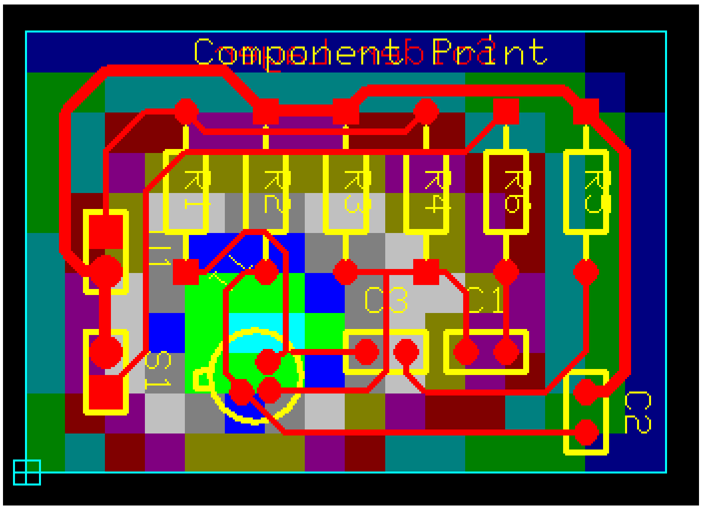
The Thermal Analyzer is intended to be used for analyzing and identifying potential thermal problems on a Printed Circuit Board. It evaluates the Temperature Distribution on a finished PCB, at steady state conditions.. The result of the analysis may be displayed using isotherms or color mapping scheme.

Steps for Thermal Analysis

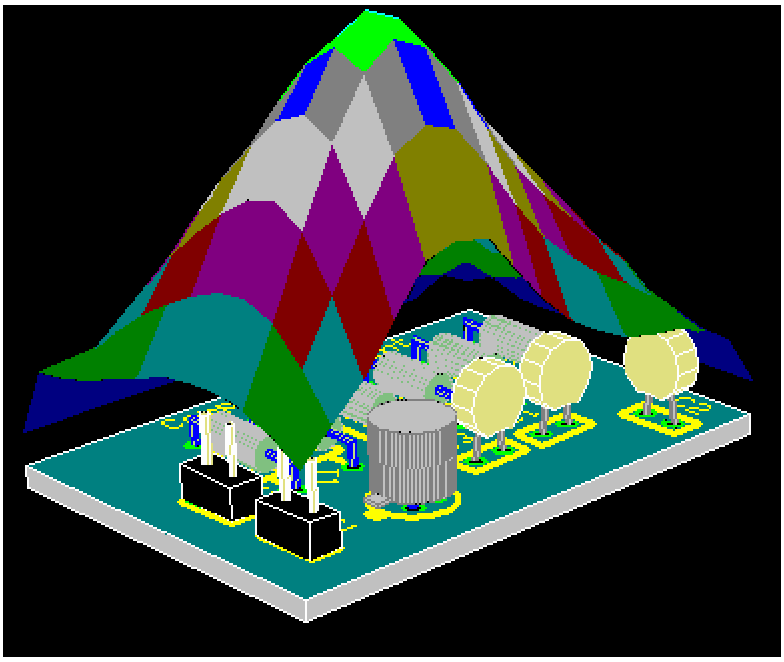
1. Right click on the **PCB Layout** in the Project Explorer. Select **Board Analyzer** in the list.
2. Select the **Thermal Analyzer** tab.
3. From **Analysis,** choose **Settings** to open the settings window and click the **Display Options** tab. Click on **Set to default** button .
4. From the **Analysis** 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.



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



7. 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.

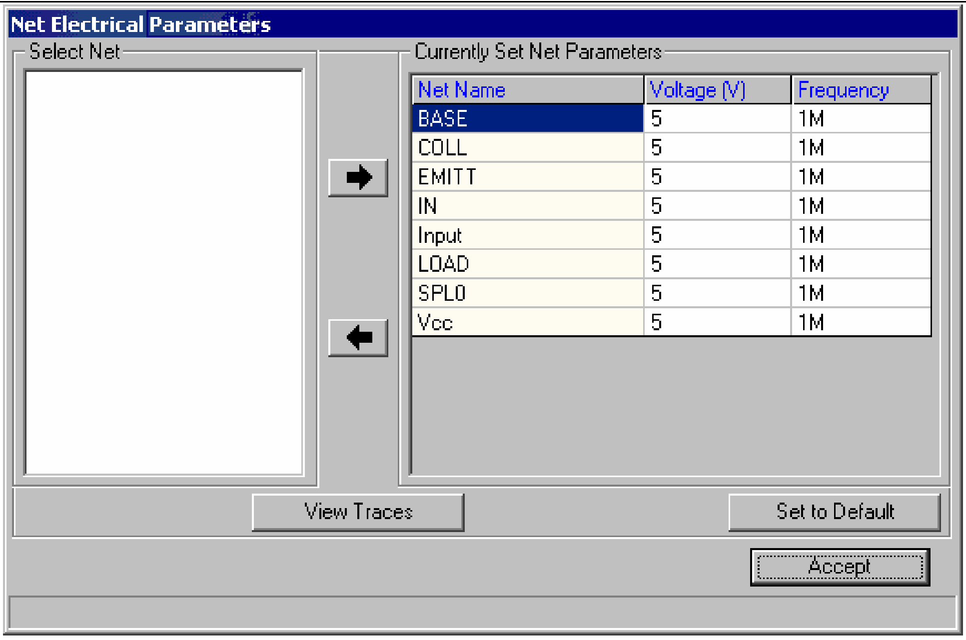
Note: To delete the isotherm label click again on the label.

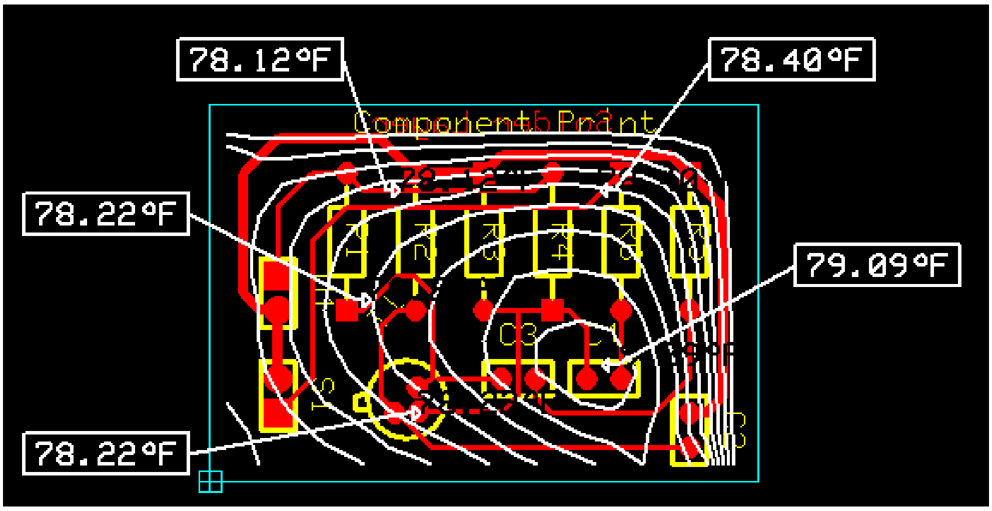
Electromagnetic Analyzer

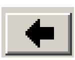
Electromagnetic Analyzer is used to predict the intensity of electromagnetic field generated by the working circuit on the PCB. An electromagnetic field develops when voltage passes through the traces on board. Once the routing of traces is complete, electromagnetic analysis may be performed on the board. 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.

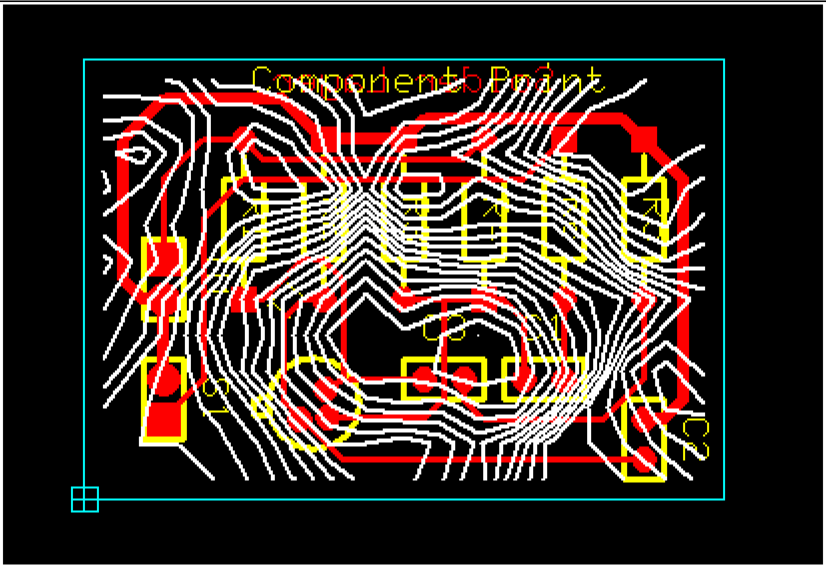
Steps for Electromagnetic Analysis

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.

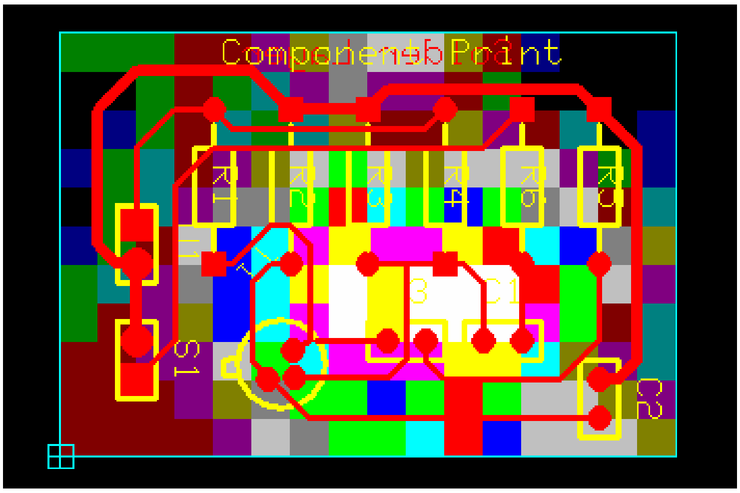




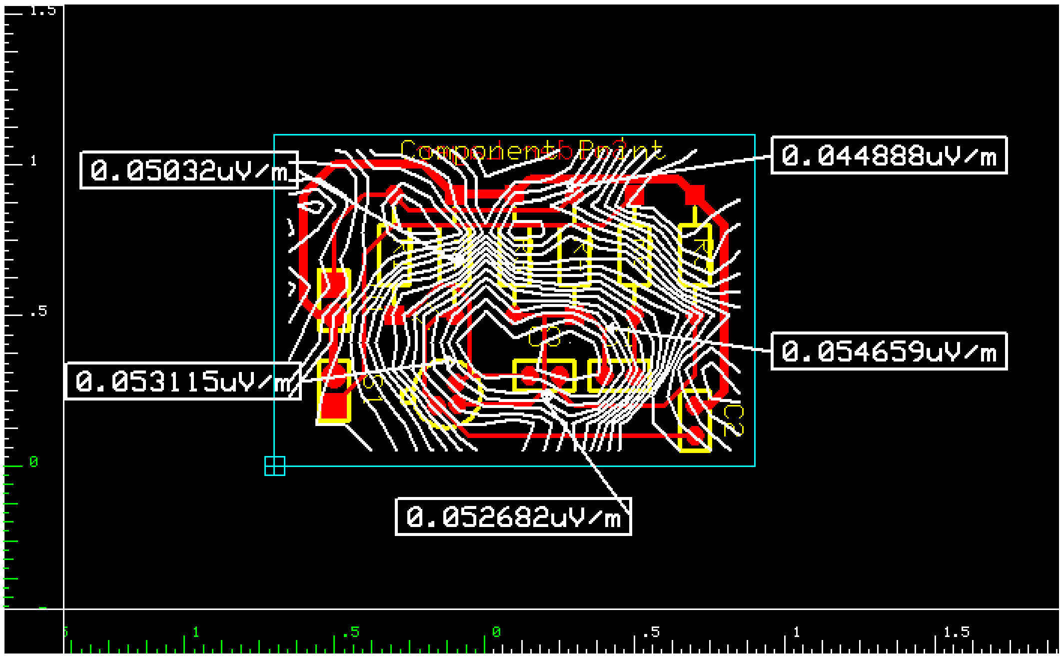
3. If you select the **Set to default** button, the default parameter gets loaded into the text boxes. To move Net from Select Net to Currently Set Net parameters use  key and to move backward use  key.
4. View the electrical parameters of the trace from point to point using the **View Traces** button 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.



6. 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.

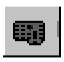
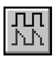


Signal Integrity Simulation

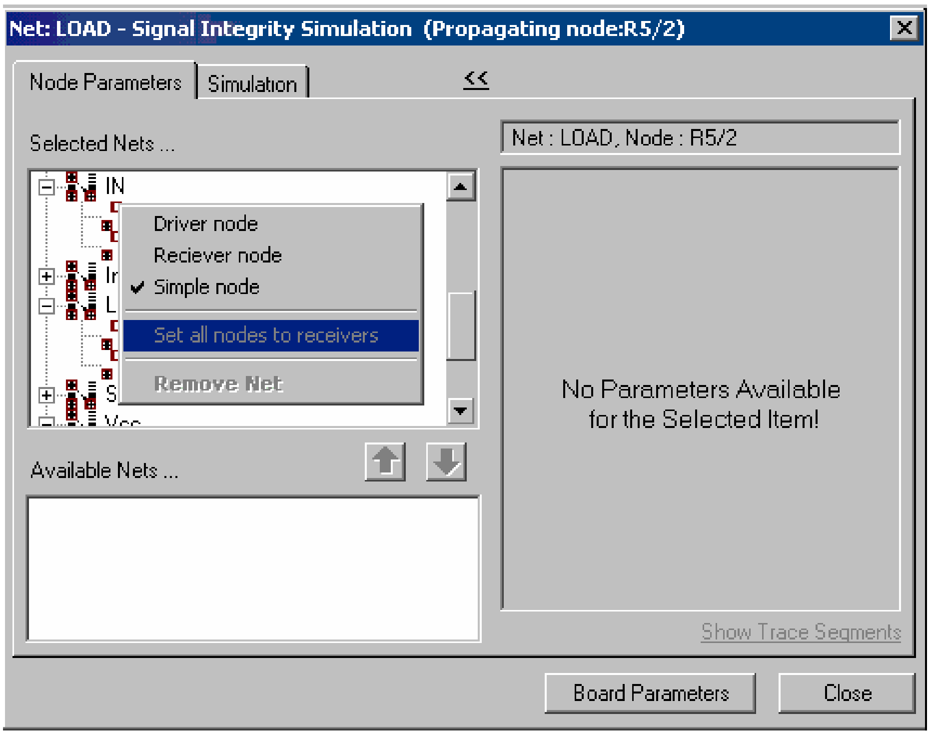
The Signal Integrity Analyzer examines the probable distortion of high-speed signals as they pass through traces on the PCB. The results of the analysis are displayed in the Waveform Viewer. The purpose is to predict how a signal deviates from its ideal (or intended) behavior in a real-world setting.

Steps for Signal Integrity Analysis

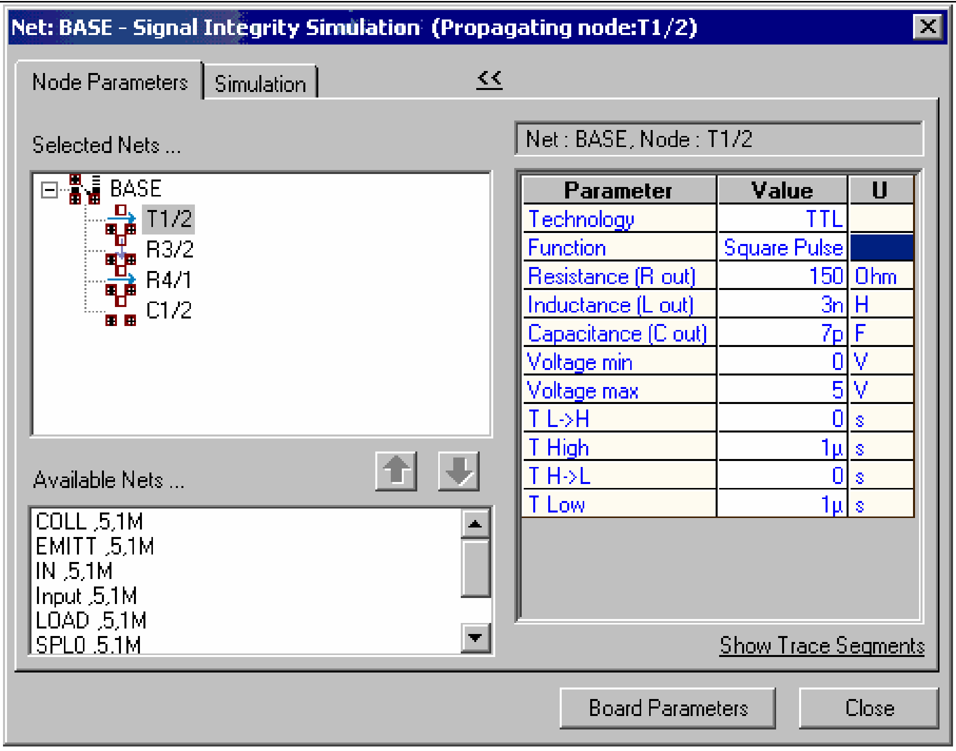
Under normal working conditions, signals propagate from some pins of the components in the circuit (technically called **Driver Nodes**), through the interconnecting traces and into pins on the same or other components (called **Receiver Nodes**).
This analysis detects the degree of distortion of the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



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** .



6. Set the following parameters.

   Max voltage to 5V

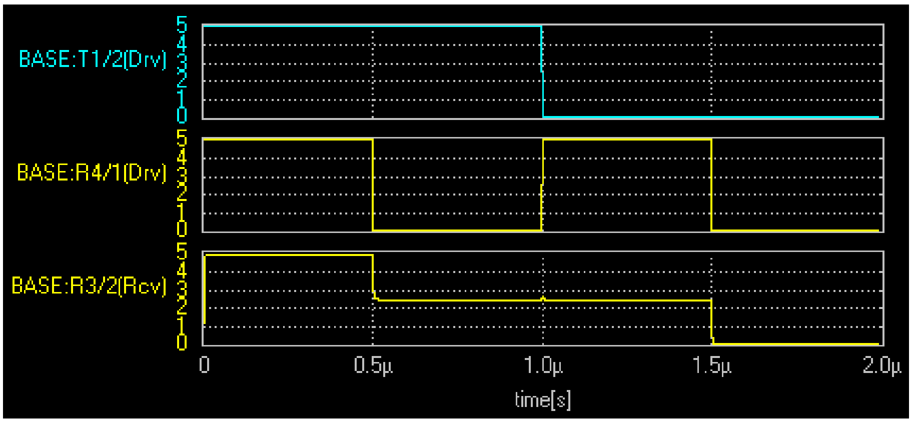
   T L->H = 0

   T H = 1us

   T H-> L = 0

   T L = 1us

7. 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.
8. 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.
9. 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.

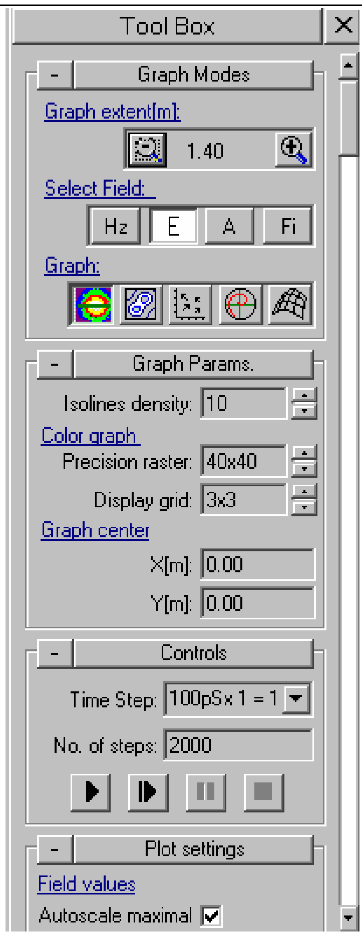


Field Analyzer

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.**



The Tool Box on the right side of **Field Analyzer** includes

Graph Modes

The Graph mode includes

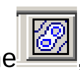
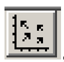
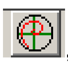
Graph Extent:

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.

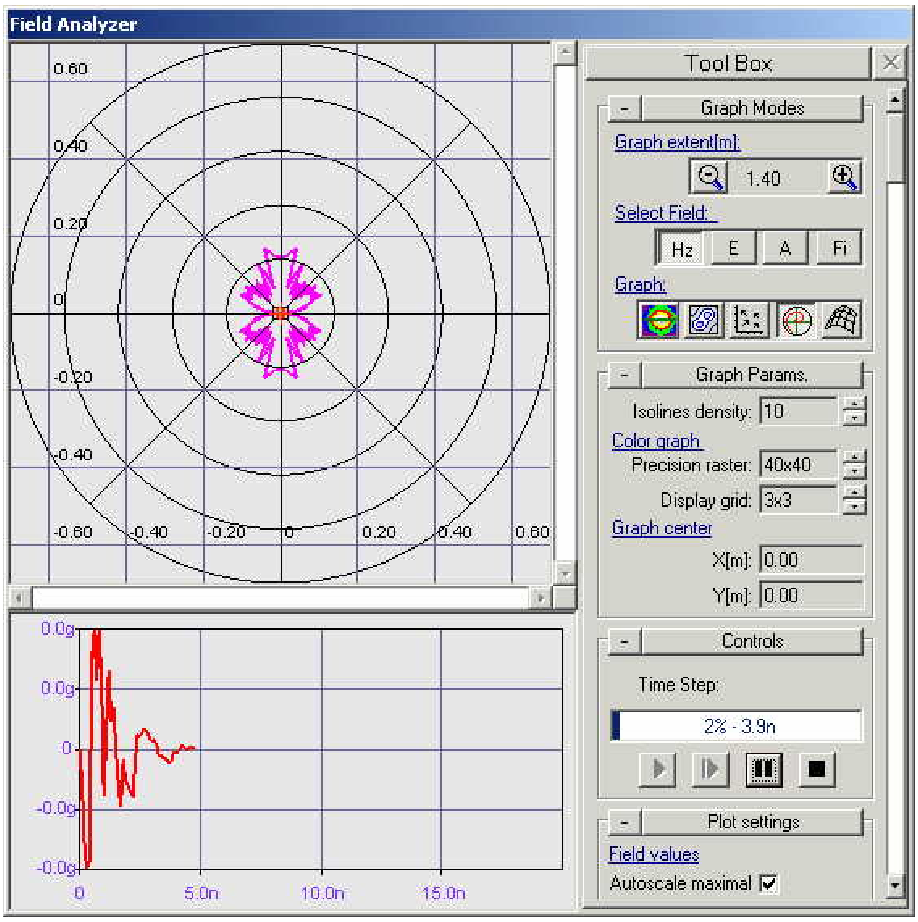
Plot settings

You can view the Field Analyzer Diagram by setting the parameters as your need.

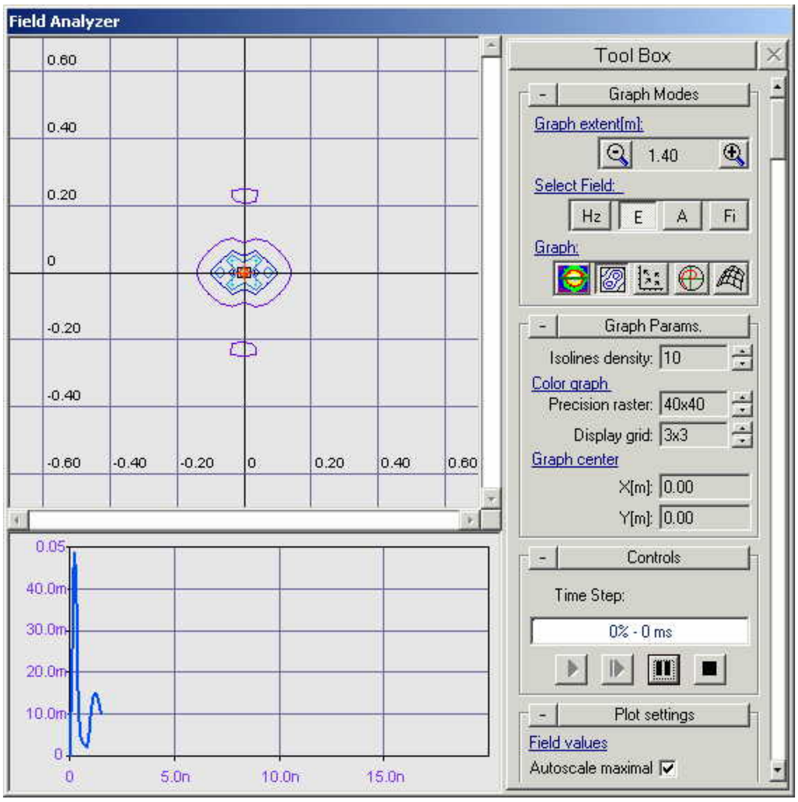
If Auto Scaling has been selected, the maximum values displayed as well as the scales of the graphs are automatically updated. On the other hand, if Auto Scaling has been turned off, these values are clipped at the values specified. (In other words, values higher than those specified are ignored.).

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

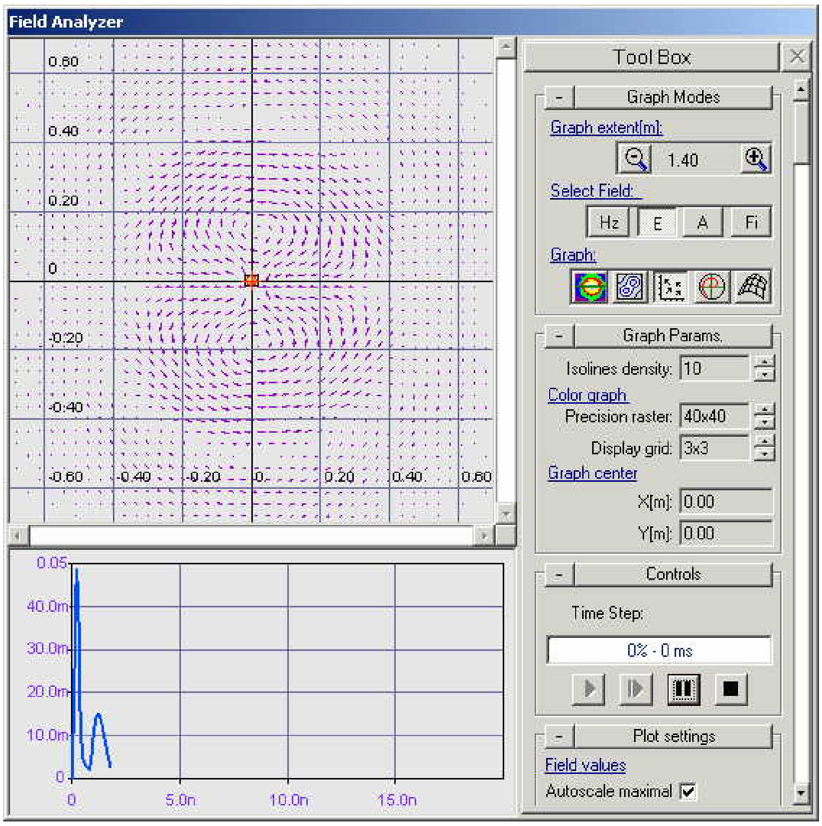
Electric Field Polar Co-ordinate Graph



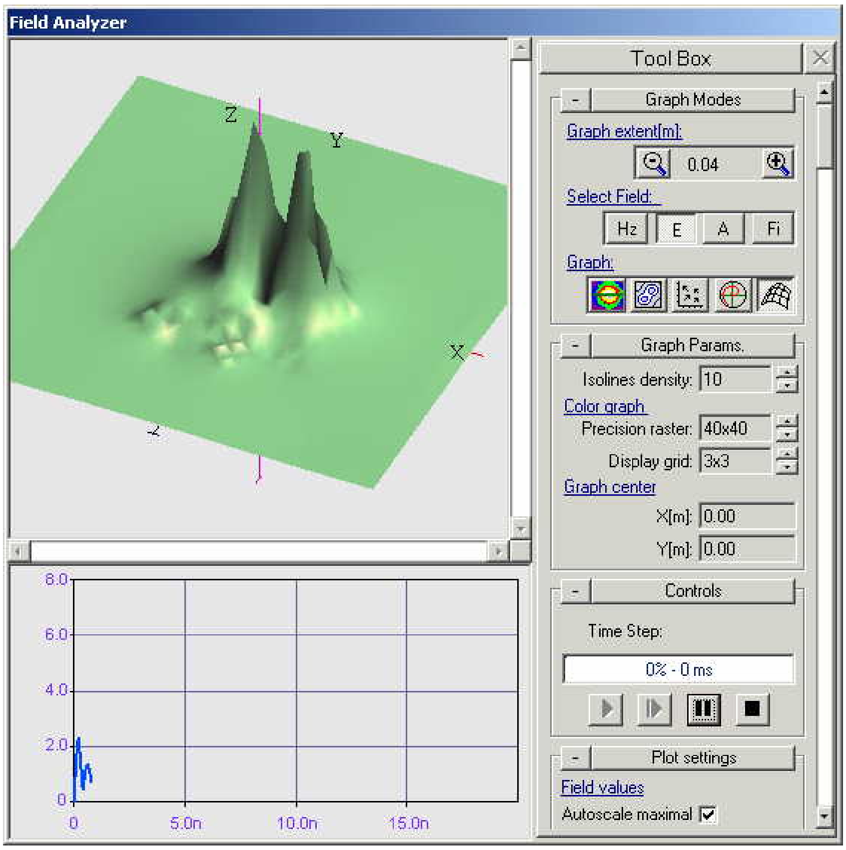
Magnetic Isoline Graph



Magnetic Field Direction Graph



Magnetic Field 3D Graph



Fabrication Manager

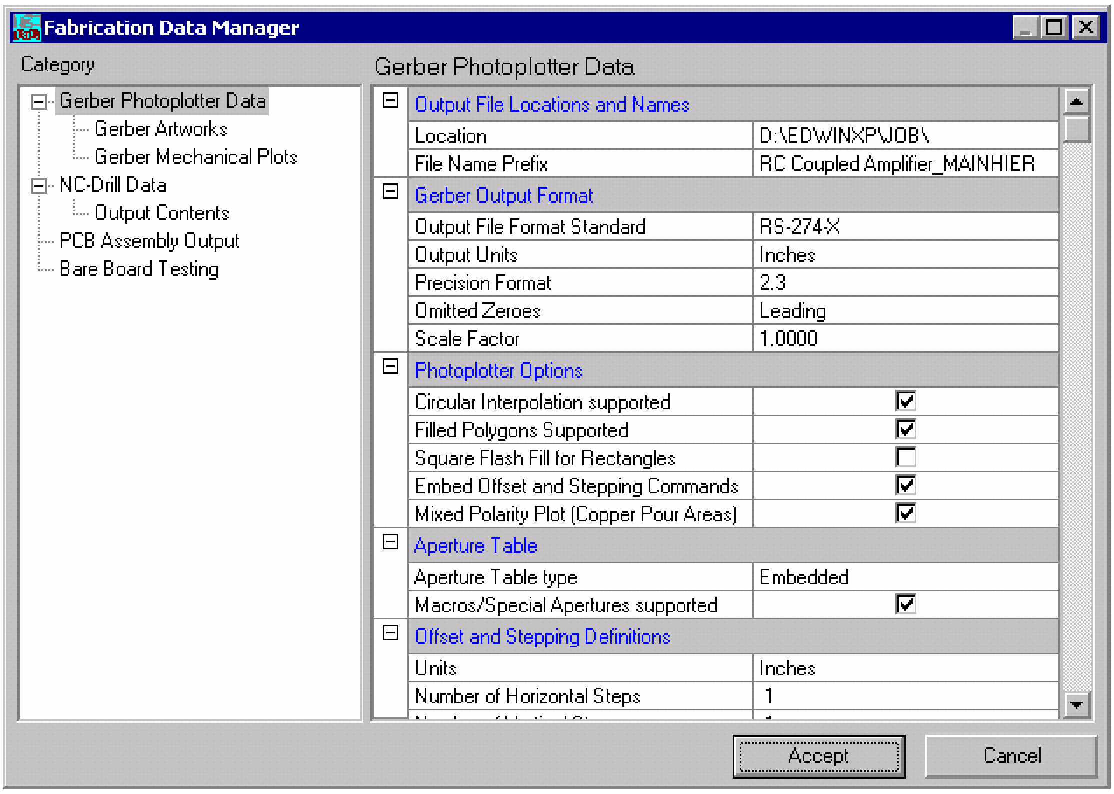
Introduction

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.
2. 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.

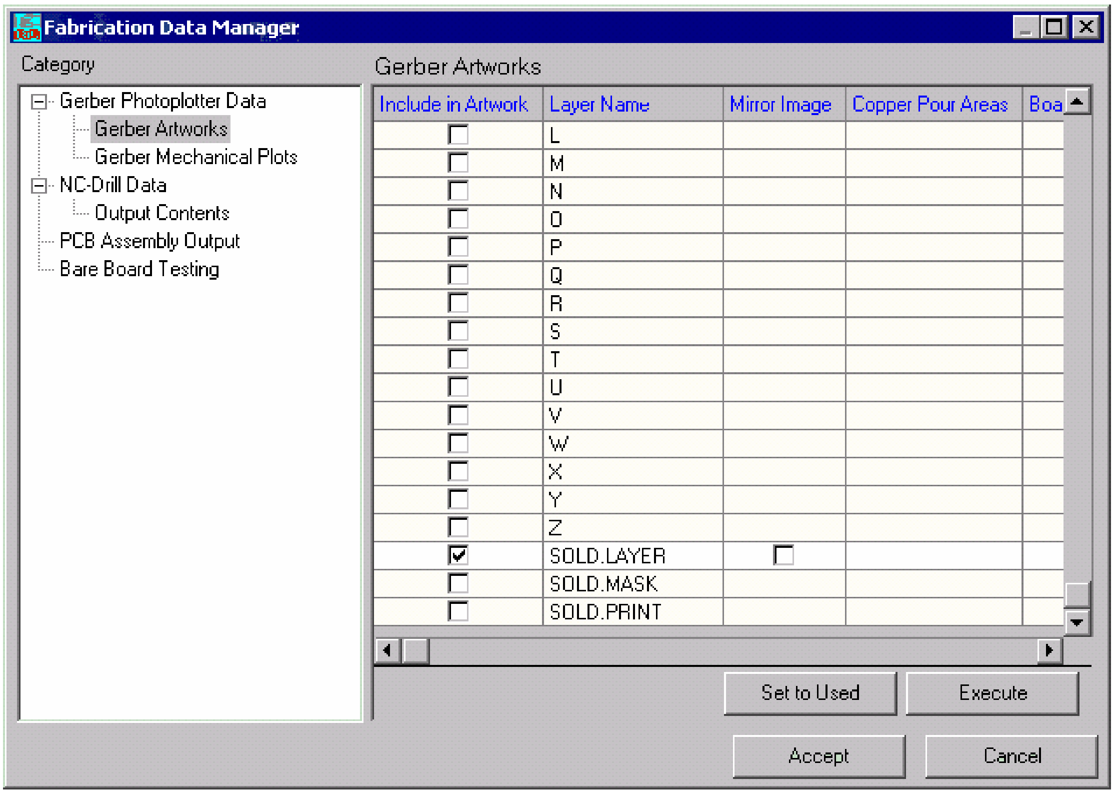
4. Choose Gerber Artworks in the left pane of Fabrication Data Manager .

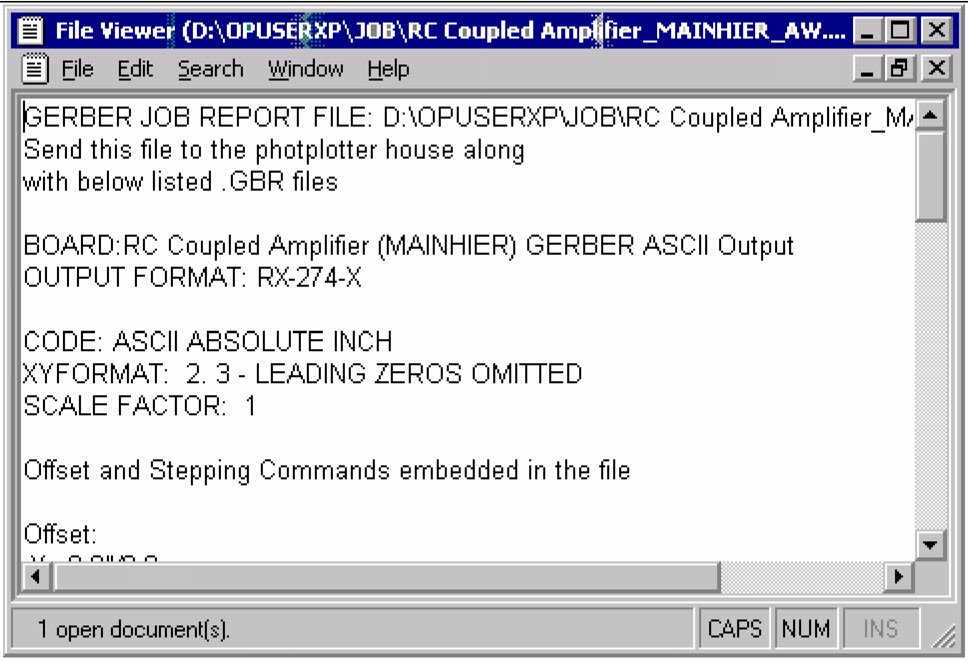


Gerber Output

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.

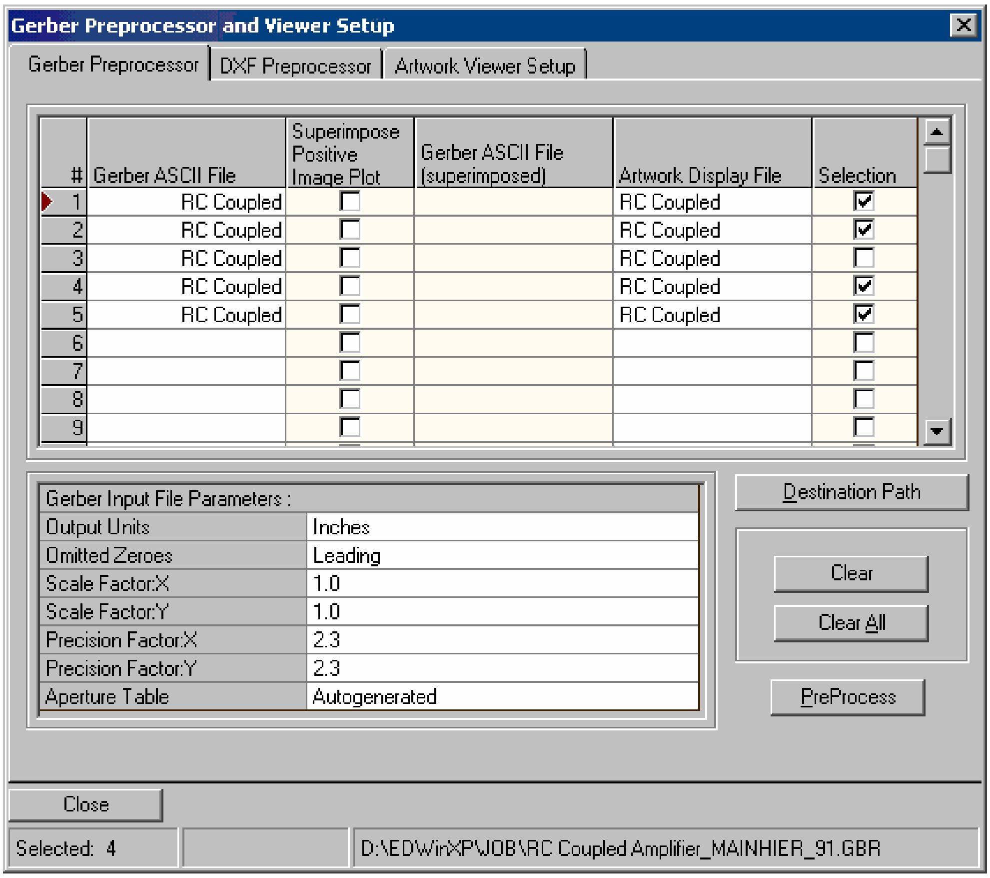




Preview the GERBER Data

<u>Preprocess the GERBER file</u>

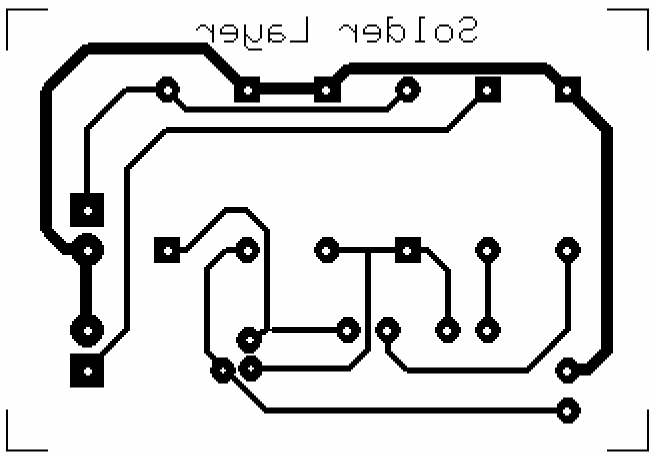
File ➔ GERBER Viewer Setup option. Click on Grid cell in the column **GERBER ASCII File,** an ellipsis (a button with 3 dots) appears, click on it and select the required GERBER ASCII files (***.GBR)**..

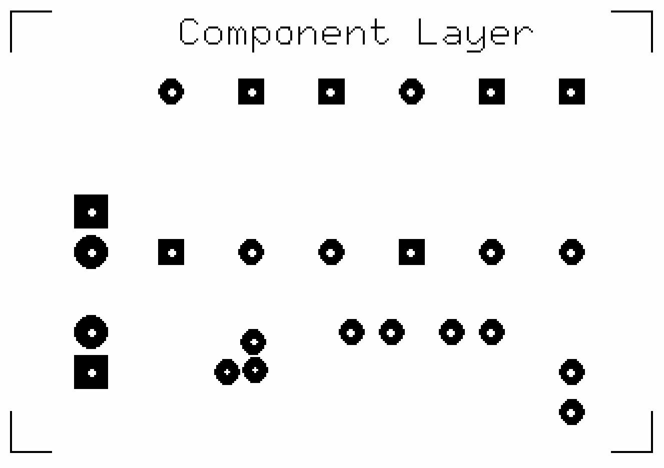


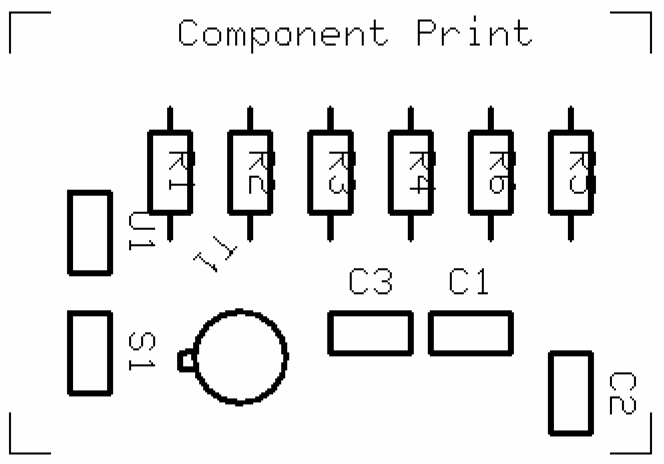
Multiple Artwork files may also be preprocessed Click **‘Preprocess’** button. The preprocessing status may be displayed on the status bar and the completion is marked by a **Successfully processed selected files.’** message. As a result, artwork file (***.ART**) is created with the name of the source 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.
9. 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.**



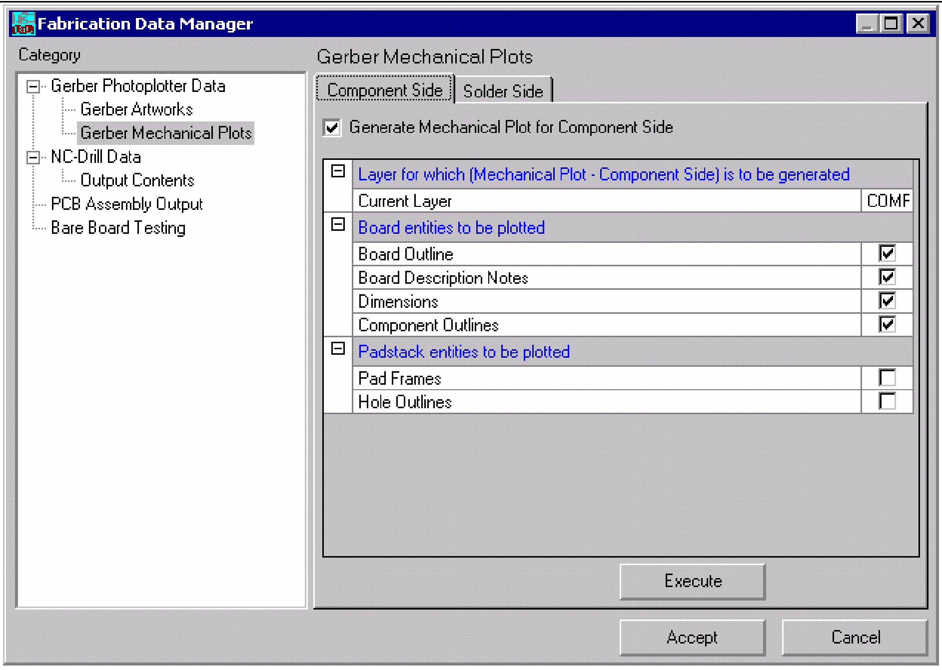




Gerber Mechanical Plots

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.



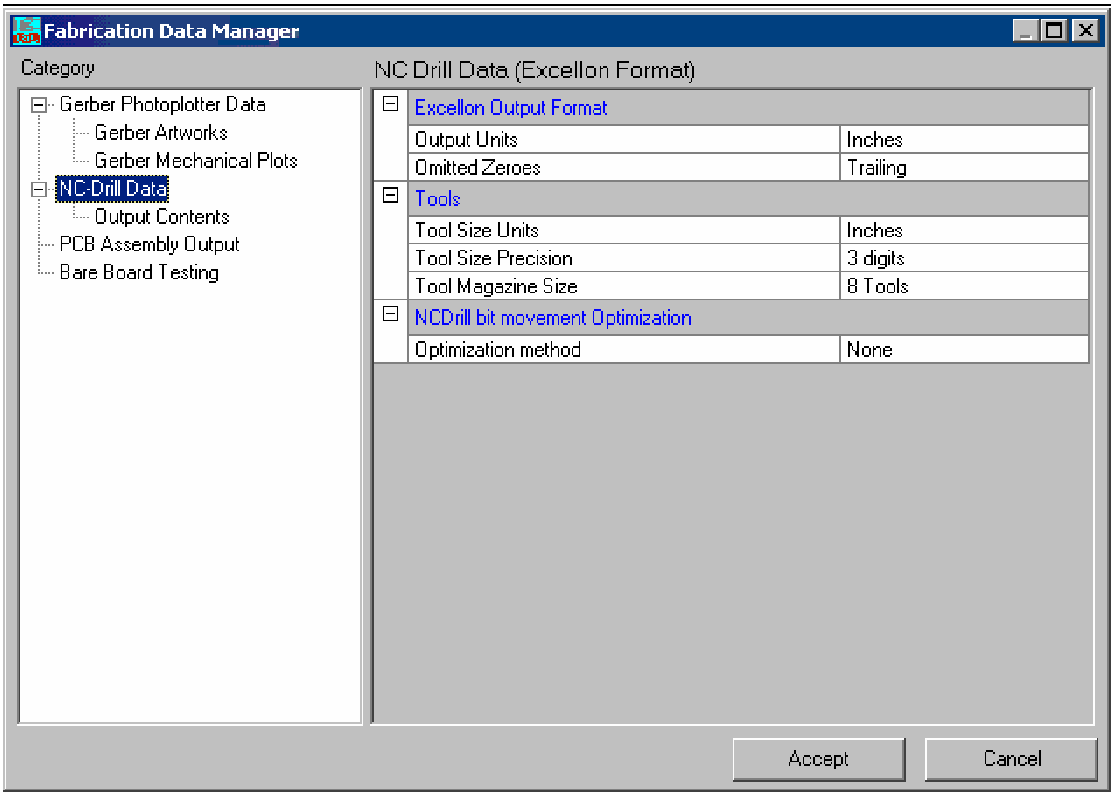
2. Click **Execute,** then a Gerber Output window appears.
3. Click **Execute.** **'**Gerber ASCII output completed' message appears in the bottom of the window. The output obtained will be a text file.

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

1. 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.

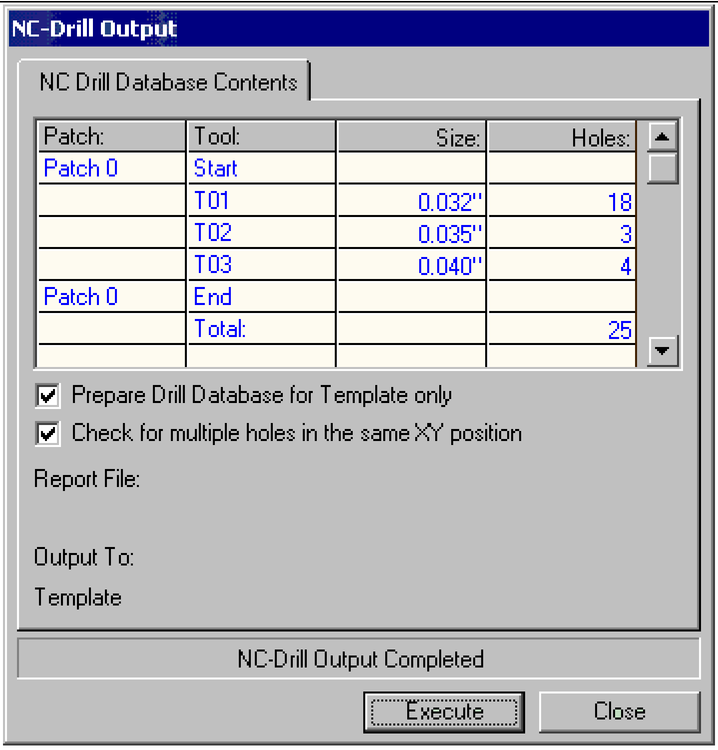


2. Set the following parameters

   Output units - Inches or Millimeters.

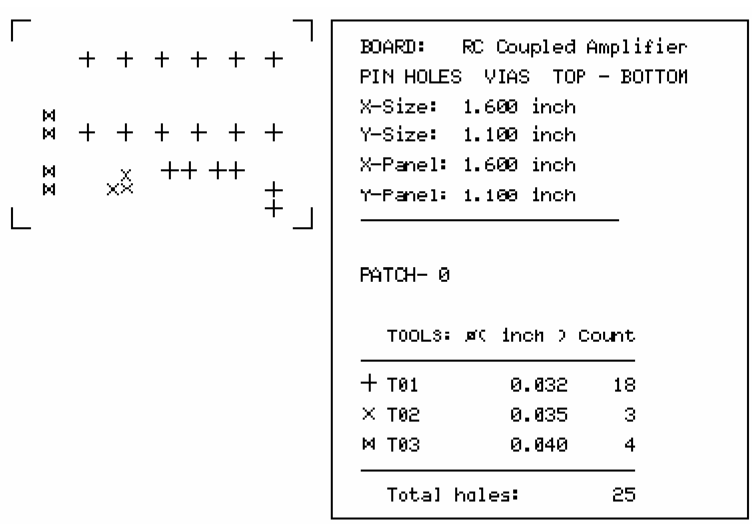
   Omitted Zeroes- Leading or Trailing.

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.



Preview NC- Drill data:

Select Tools ▯ Template (notes)

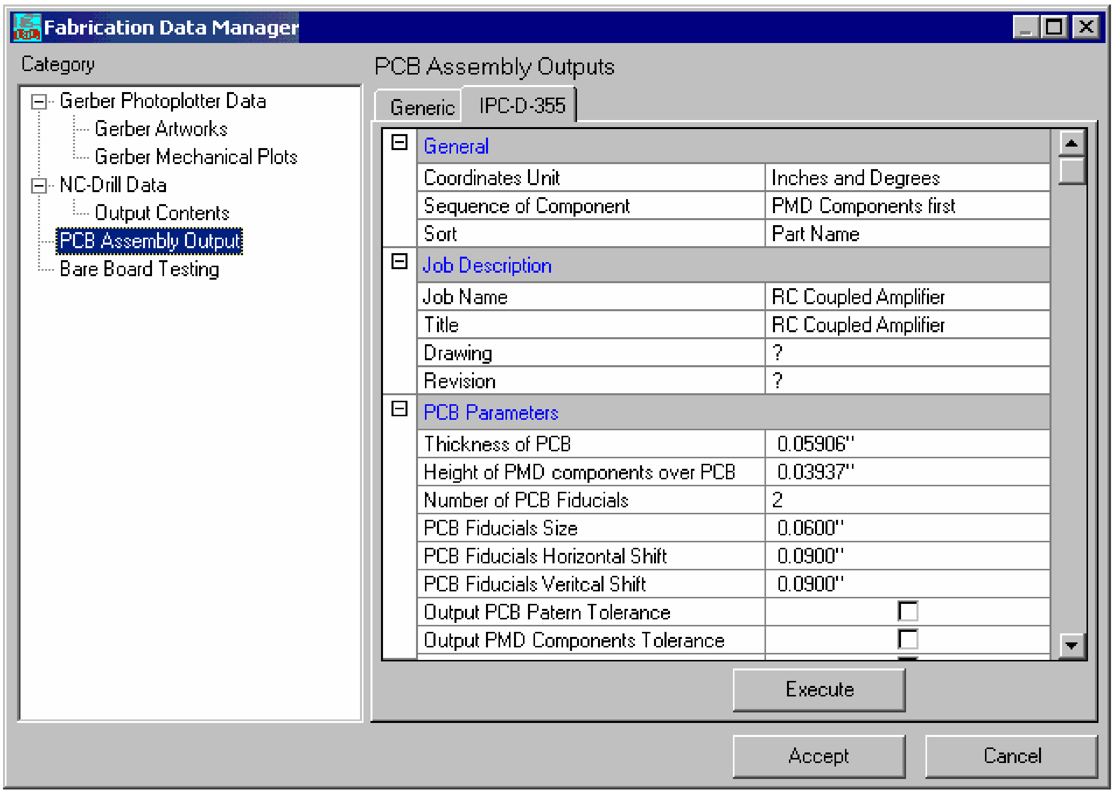


Introduction to PCB Assembly outputs

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.**



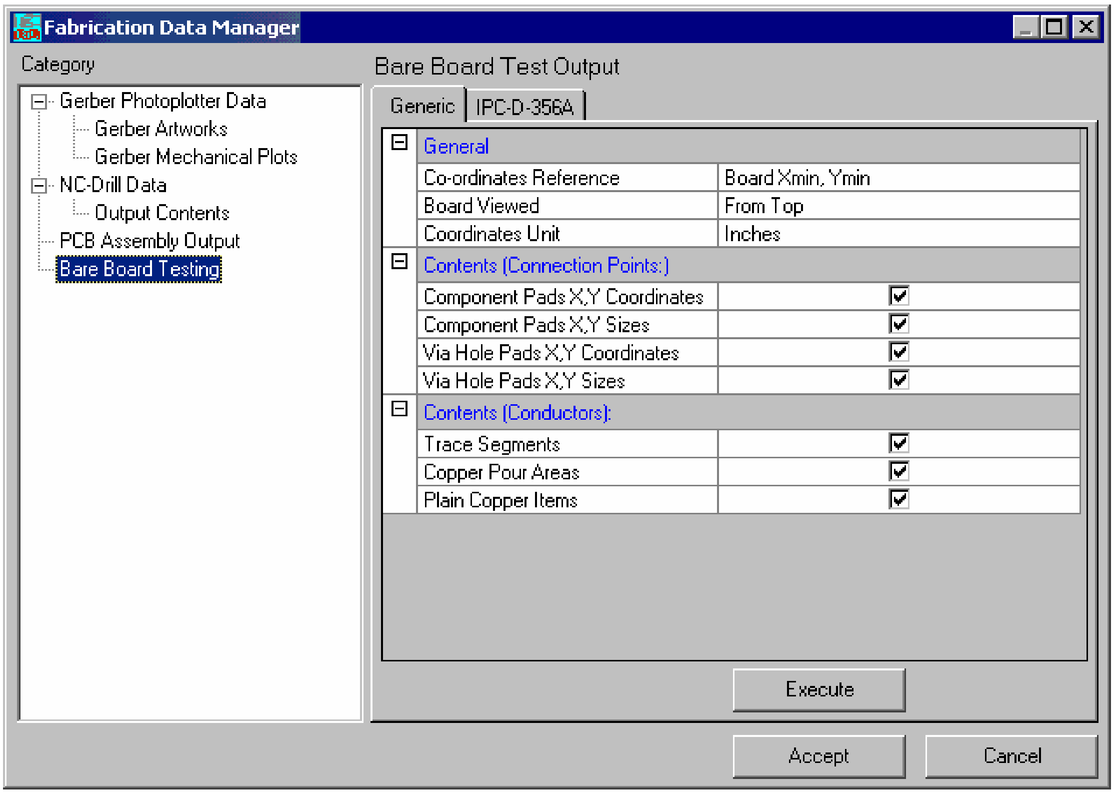
2. Click on **Execute**. Output is a extension **.PCK** extension file.
3. Save it
4. Select **IPC-D-355** tab on top of the **Fabrication Data Manager** window.
5. Click on **Execute**. Output is a **.355** extension file, save it in the desired location.

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.



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