EDWINXP

ELECTRONIC DESIGN FOR WINDOWS

TUTORIAL ON SCHEMATIC CAPTURE , SIMULATION

&

PCB LAYOUT DESIGNING

INDEX

TOPIC	PAGE NO.
Role of EDA in Learning Electronics	1
Introduction to EDWINXP	2
Getting Started (GUI)	3-4
Major Functions in Schematic Editor	4-7
Simulation Parameter Setting	7-12
Working with Different Analysis	12-21
Working with Microcontroller Co-Simulation	22-28
Working with EDSPICE Simulator	28-36
Truth Table to Circuit Converter Wizard	37-38
Active Filter Designing Wizard	39-40
Tutorial on PCB Designing	**41**
Introduction to PCB Layout	42-46
DRC Check	46-47
3D Viewer	48
Thermal Analysis	49-52
Electro-Magnetic Analysis	52-53

Role of EDA (ELECTRONIC DESIGN AUTOMATION) **in Learning Electronics**

Learning is a gradual process, which can be expedited by experimentation. Electronic experiments therefore constitute a fundamental part of an electronic engineering student's curriculum. Physical experimentation no doubt has its own merits but it has the serious drawback of limiting the student's practical knowledge to the prescribed experiments. It takes away the opportunity of doing something that is so characteristic of their age, exploration. To satisfy the students' inherent scientific curiosity what is required is a cost-effective system, which will allow them to do 'just their own thing'. That is what makes the learning process complete, an environment conducive to experimentation without the fear of destroying components or devices. Not having to work with physical devices, the students can experiment till they achieve desired results.

SCOPE OF EDWINXP IN CONTRAST TO OTHER EDA TOOLS

EDWinXP includes the complete set of tools you need to save time and money while designing Printed Circuit Boards: Schematic Editor, Layout Editor, Auto Placers, Auto Routers, Edspice simulator, Mixed Mode Simulator, Microcontrollers co-simulation, VHDL co-simulation, Fabrication Manager, Thermal analyser, Electromagnetic analyser, Signal Integrity analyser, Library Manager and more

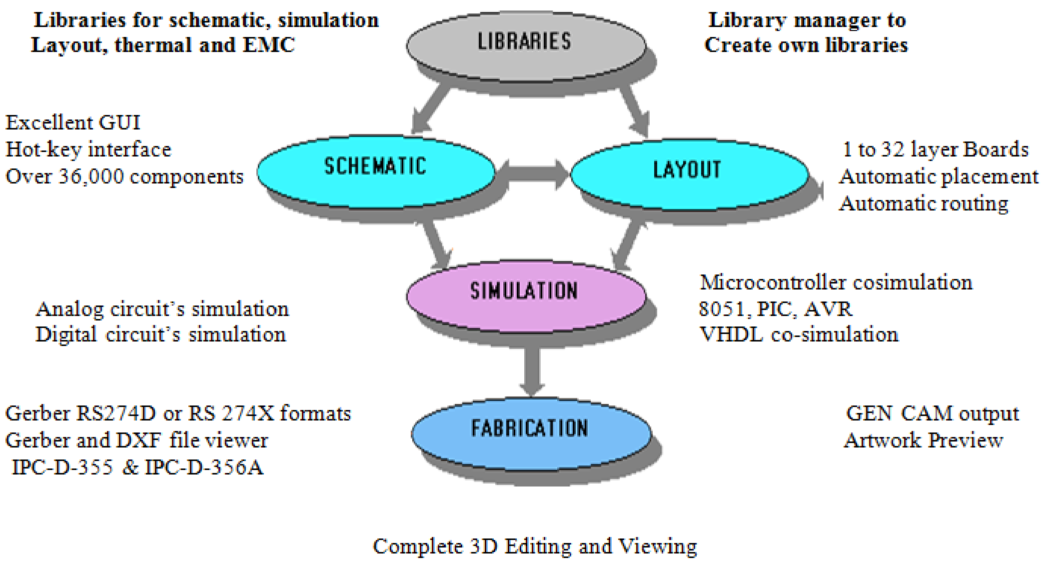


Fig: Design flow of EDWINXP

EDWINXP is a single window based fully integrated EDA tool, which offers seamless integration, forward and back annotation between various modules. With EDWINXP you are not only getting a simulation or a layout tool but you are getting a complete board level design flow which starts from schematic capture, simulation, layout, board level analysis and tap out to fabrication. EDWINXP has got over 80,000 licenses worldwide.

So as for as educational institutes are seen, their basic purpose concerns with giving the students complete technology exposure from concept to product development and in-between methodologies and challenges, rather than just exploring only one segment.

Being an industrial standard EDWinXP is used by many reputed engineering colleges and universities in India.

Introduction to EDWINXP

In the past, students traditionally verified their laboratory electronic circuits by building them on breadboards and measuring the various nodes with the appropriate laboratory equipment. By using a computer simulation program, such as EDWINXP, students can obtain results before they come to lab. Hence the laboratory experiments become reinforcement to the subject matter at hand. The use of a computer simulation program allows the student to easily subject the circuit to various stimuli (such as input signals and power supply variations) and to see the results in either a tabular format or plotted out graphically using various analysis.

An Outline of EDWINXP

EDWINXP simulates the behavior of electronic circuits on a digital computer and tries to emulate both the signal generators and measurement equipment such as multimeters, oscilloscopes, curve tracers, and frequency spectrum analyzers.

Types of Analysis Performed by EDWINXP

EDWINXP is a general-purpose circuit simulator capable of performing four main types of analysis: Bias Point, DC Sweep, AC Sweep, and Time Domain (transient).

Bias Point

The Bias Point analysis is the starting point for all analysis. In this mode, the simulator calculates the DC operating point of the circuit. Options include calculating the detailed bias points for all nonlinear controlled sources and semiconductors, performing sensitivity analysis, and calculating the small signal DC gain.

DC Sweep

The DC Sweep analysis varies a voltage source over a range of voltages in an assigned number of increments in a linear or logarithmic fashion.

AC Sweep/Noise

The AC Sweep/Noise analysis varies the operating frequency in a linear or logarithmic manner. It linearizes the circuit around the DC operating point and then calculates the network variables as functions of frequency. The start and stop frequencies as well as the number of points can be assigned.

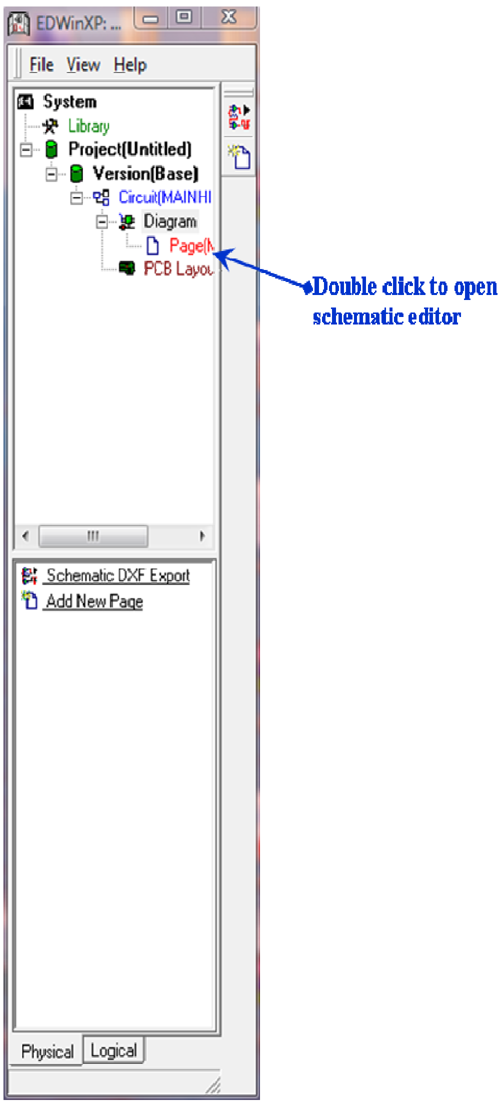
Time Domain (transient)

The Time Domain (transient) analysis is probably the most popular analysis. In this mode, you can plot the various outputs as a function of time. The starting and ending times for the various plots can be input. The accuracy (smoothness) of the output plots can also be controlled by regulating the maximum (time) step size.

Getting started with EDWINXP:

Go to start >program files> EDWinXP 1.71> **EDWinXP Main ----- (double click on it)**

It will open a small window. Select **New Project** from the files Tab for creating new project Page. Then double click to open schematic editor as shown below.



It will open a window as shown in the picture below.

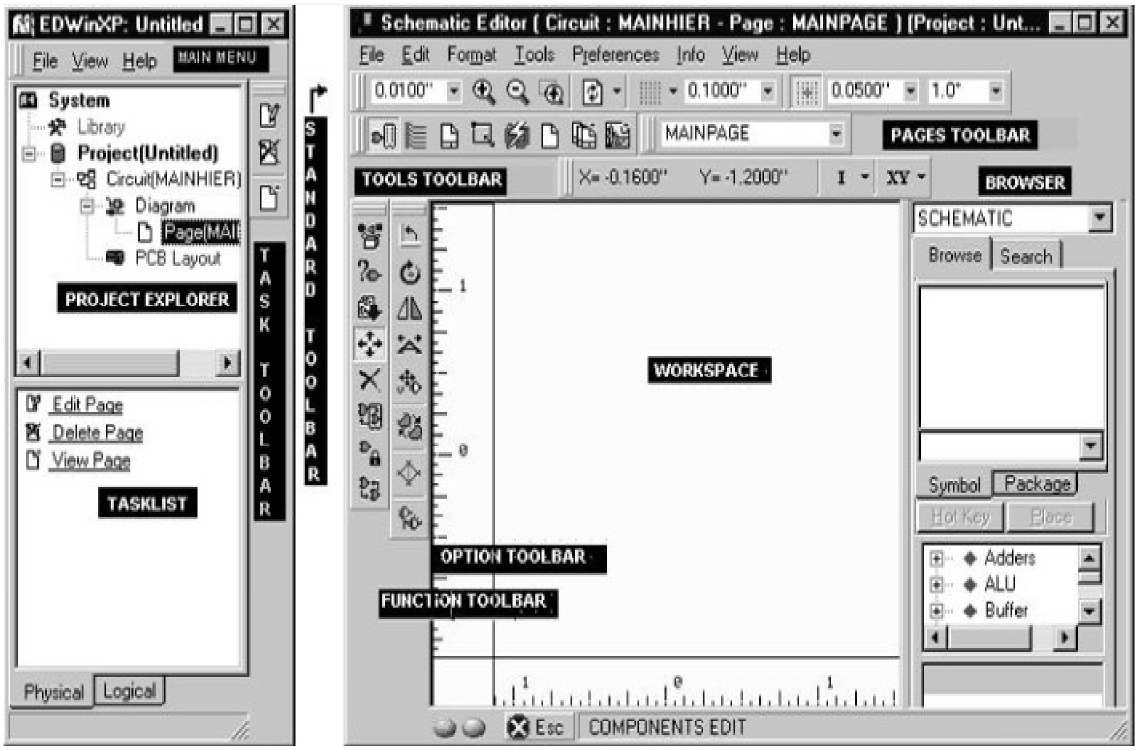
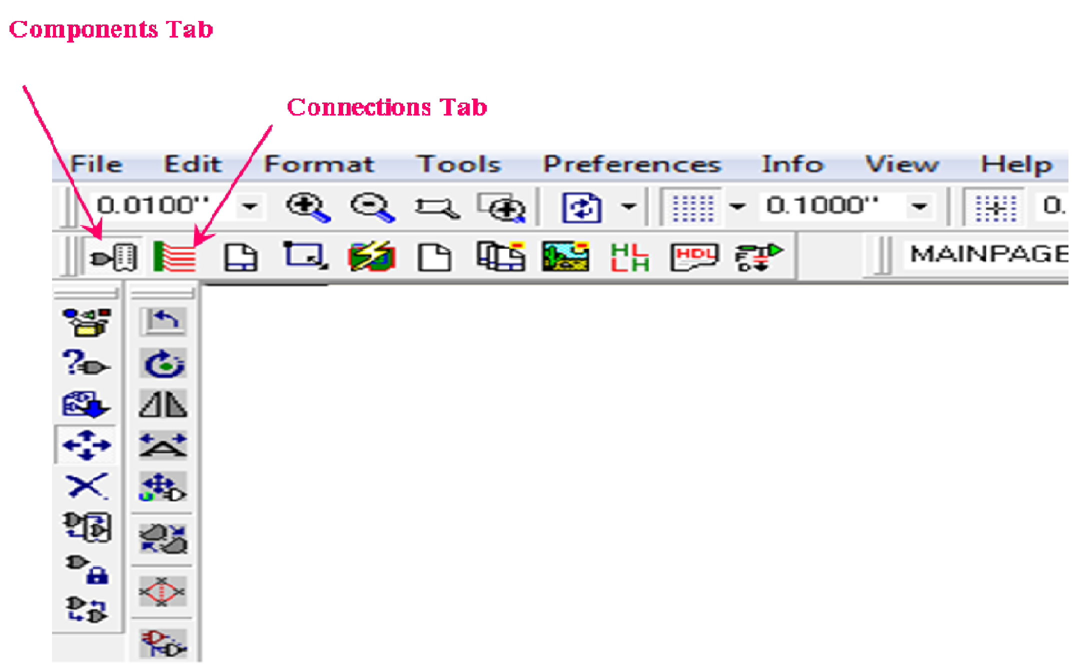


Fig. Complete GUI of EDWINXP

Major functions to be performed in the schematic editor are:

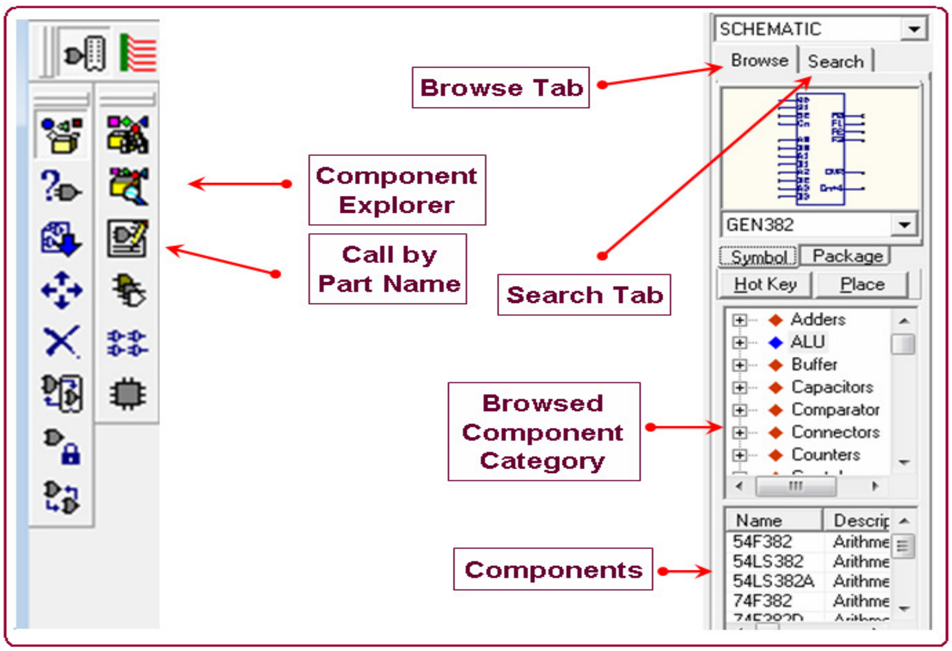
1. Bring components to the workspace from library.
2. Make connections between them



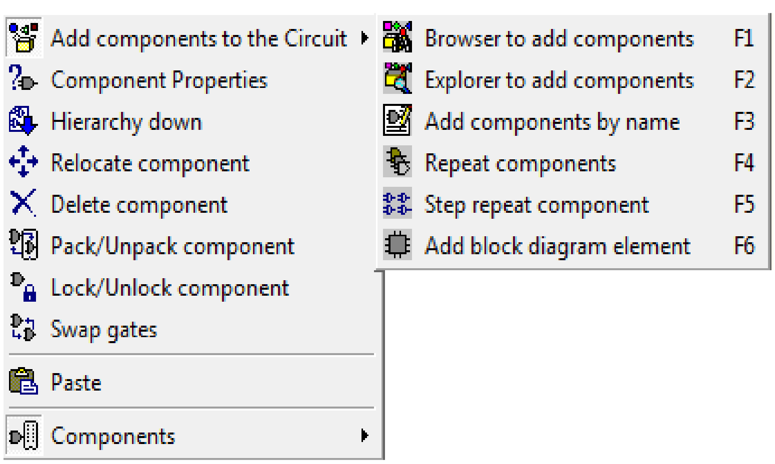
Select **components Tab** to Add components to workspace.

EDWinXP provides 5 ways to call the component into the workspace.

a) Component Explorer
b) Search mode
c) Component Browser
d) Call component by Part Name
e) Call component using Hot Keys.



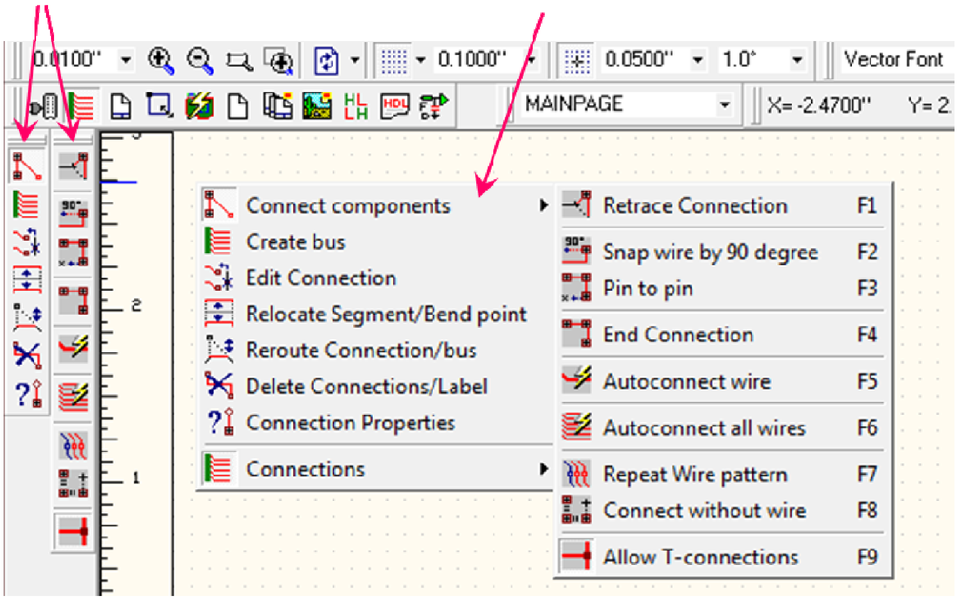
After selecting **components Tab** option, Right click in the workspace window, a pop up window will open, select the appropriate options as shows below.



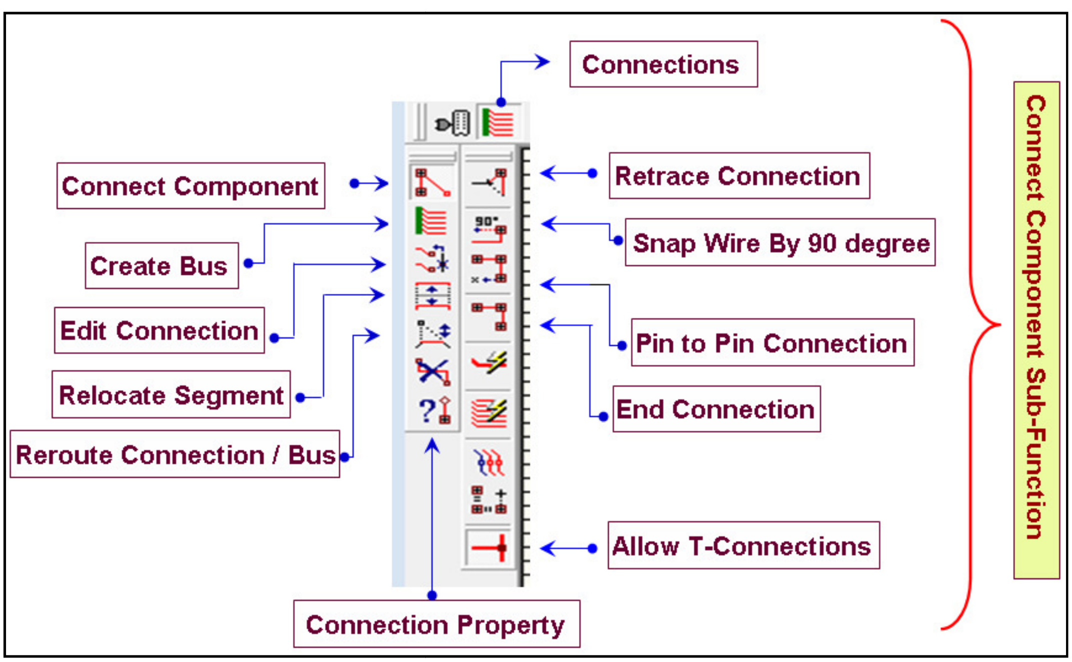
Once you have placed components on the work space select the connections Tab and by selecting the appropriate corresponding Tabs, you can make connections efficiently. As shown

Select the appropriate options for connection from here

Otherwise same options can be found by right click on the workspace



Connections toolbars are shown in details below:



Generally flow of any project or experiments will follow steps as given below:

1. Call the components, selecting the correct package.
2. Inter-Connect the components.
3. Assign the values (component simulation parameters) for the components used in the circuit.
4. Simulate the circuit to check the design.

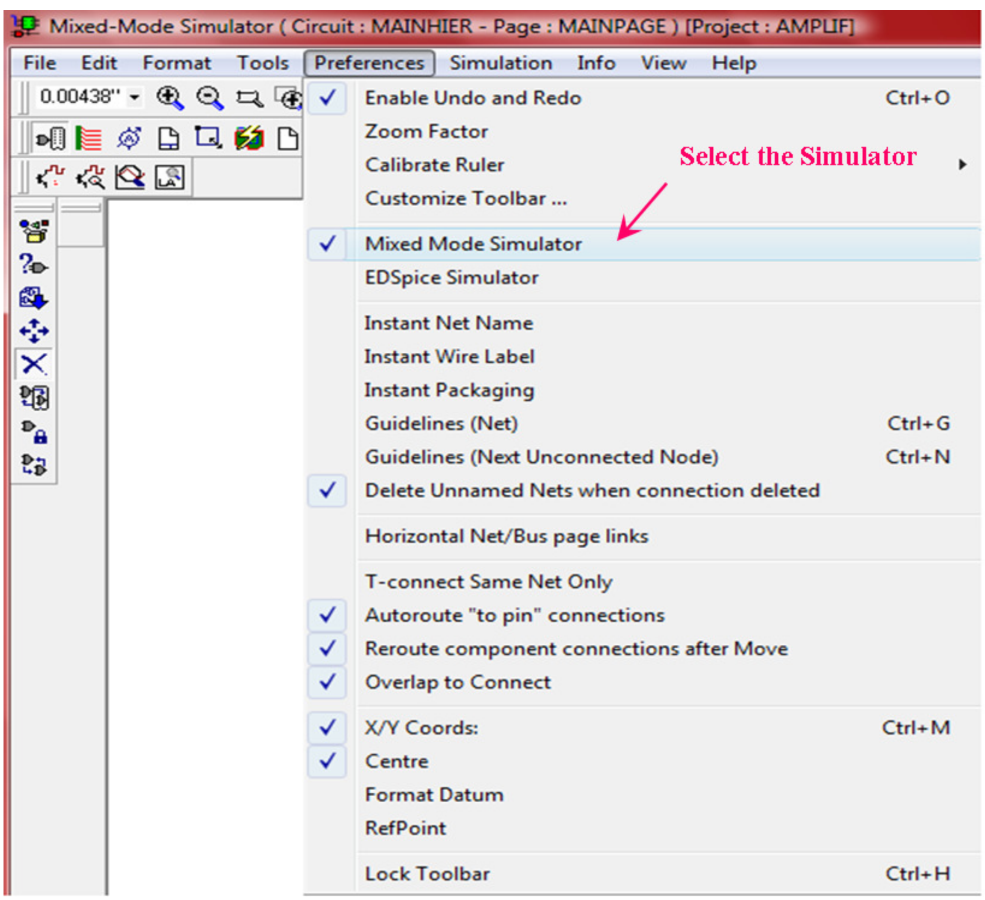
First two steps we have already done in the above exercises. For 3rd and 4th select the simulation first either mix mode simulator or Edspice simulator.

EDWINXP has two simulators associated with it names as:

Mixed Mode Simulation.

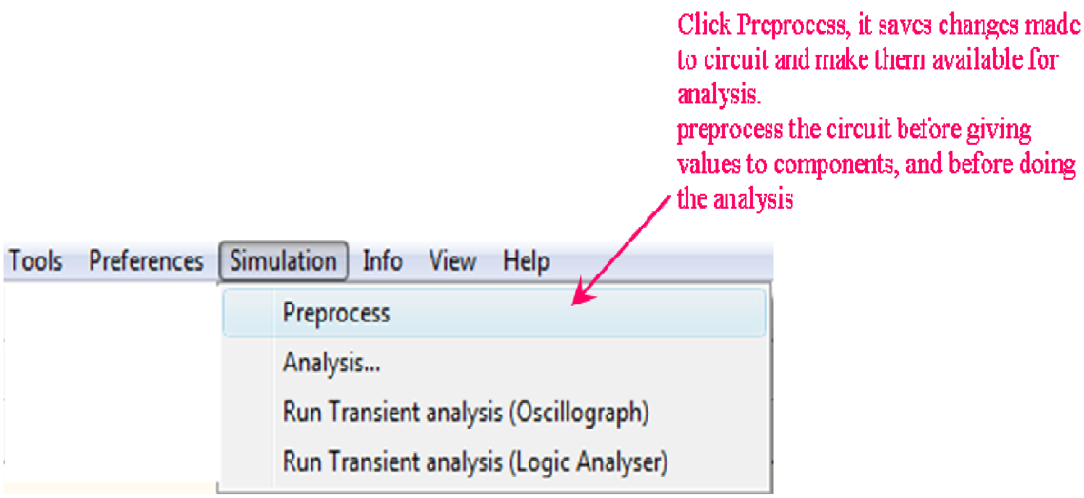
Edspice Simulation.

Each Simulation Pack offers many number of the analysis for the user to meet his requirement for example Transient Analysis, Small Signal Analysis, and Sweep (AC/DC) Analysis….etc.



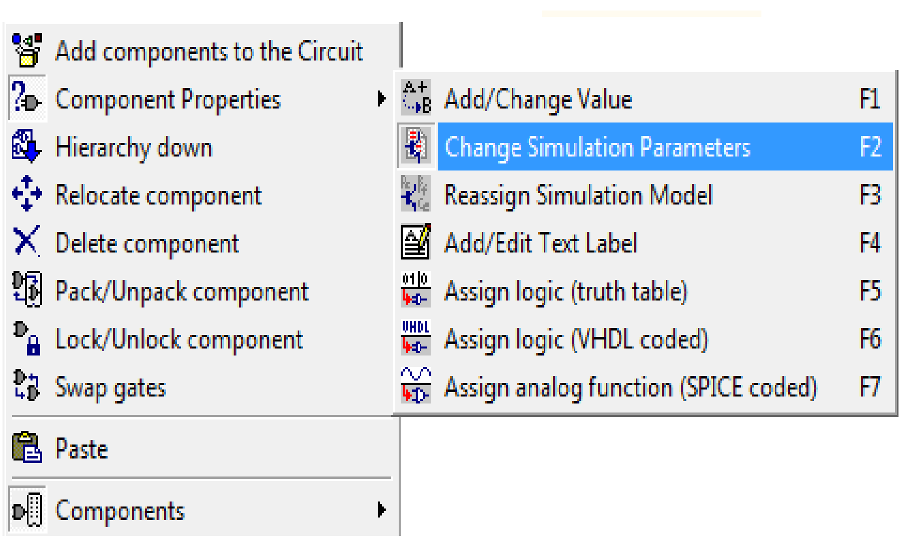
Assigning the values (simulation parameters to components), follow the following steps.

Before changing the component simulation parameter the user should do the pre-process analysis so that the simulator recognizes the components used in the circuit. To do the pre-process the required simulator is selected using the Preference tab as shown in the figure below:



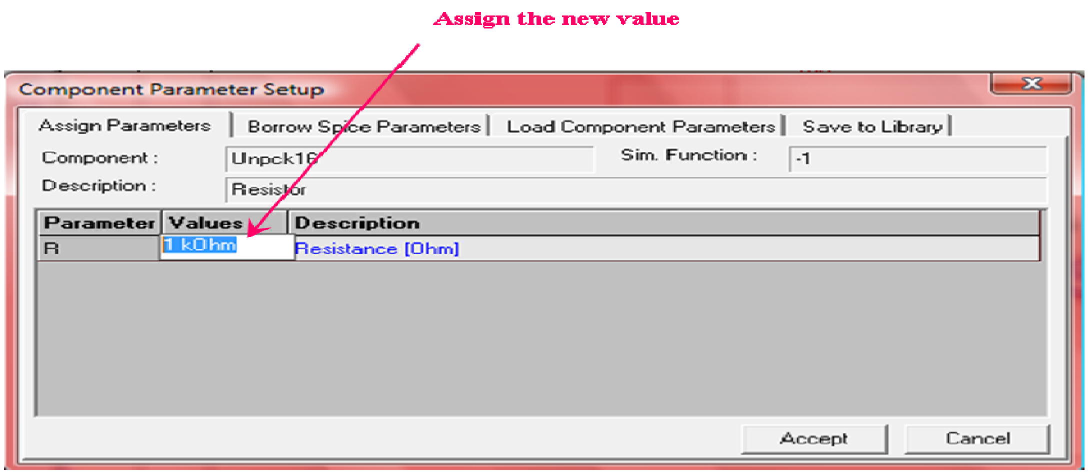
Right click on the workspace and as shown below select change simulation parameters.

Component properties>change simulation parameters

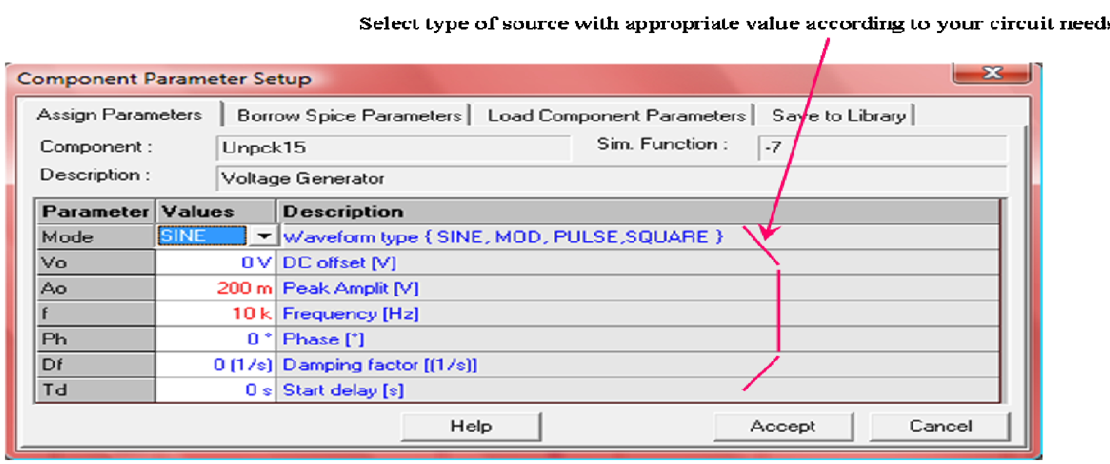


Now double click on the component for which you want to assign the new value, it will open a pop up window as shown below.

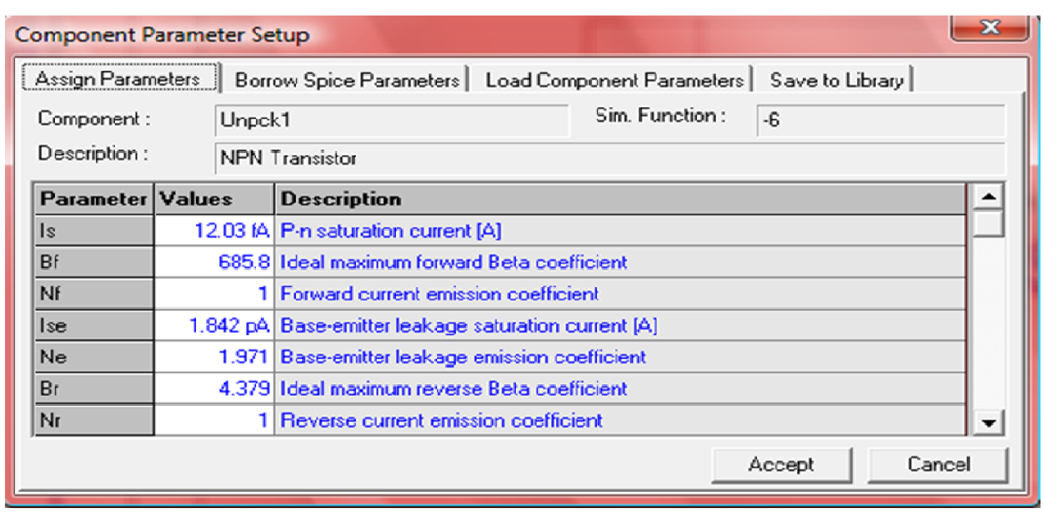
Component parameter setup for resistor component



Component parameter setup for Voltage Generator (function generator component

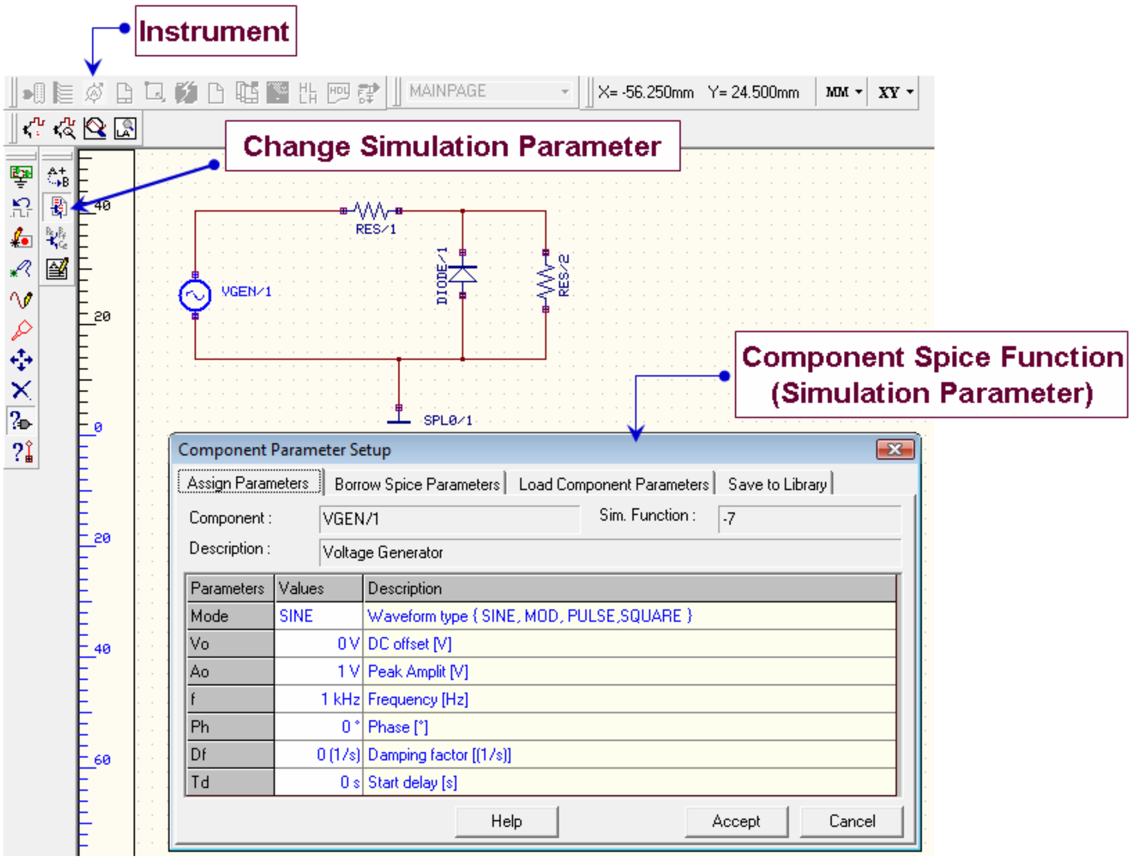


Component parameter setup for transistor component

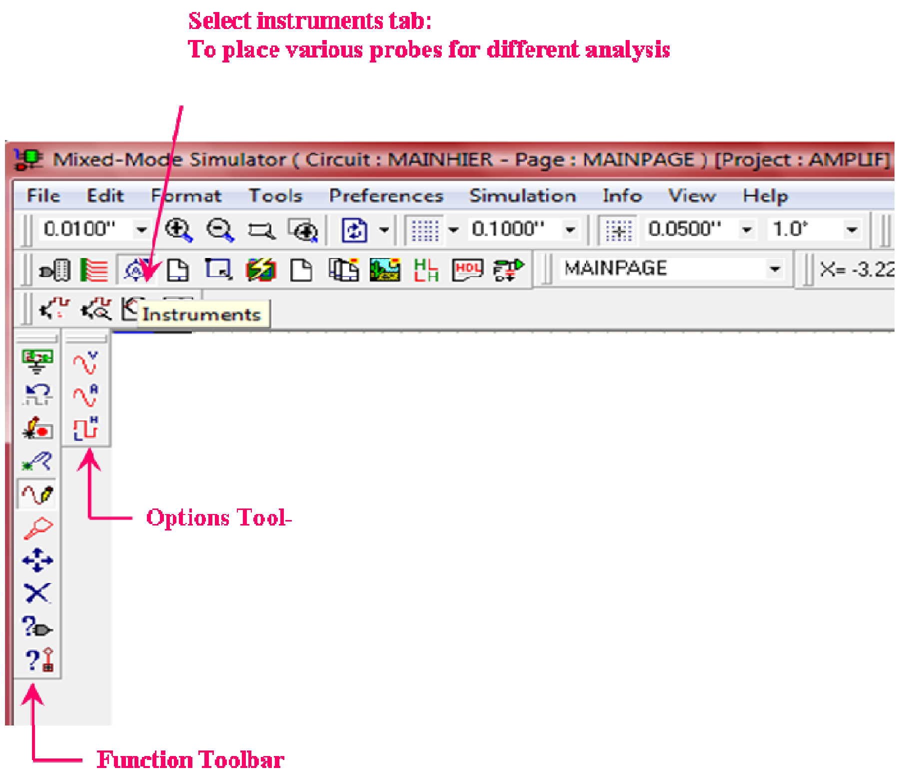


You can also use the toolbars for doing the above functions as shown in the diagram below:

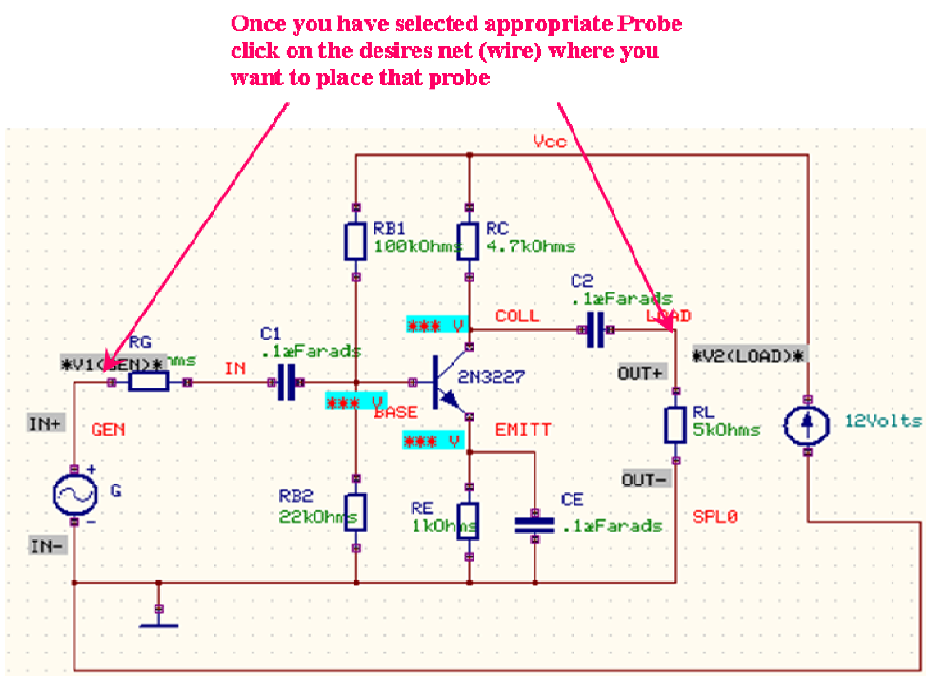
Once the required simulation pack(mix mode or Edspice) is chosen the user will given another tool called **Instruments.** Using the instrument the user can able to change the simulation parameter of any component and also do the required analysis. To change the simulation parameter of the component chose the Instrument Tab and select the **change simulation parameter function** which displays the Spice function window of the selected component. (See fig given below)



Now to place various probes on the circuit to see various waveforms follow the following steps:



Select: instruments tab >appropriate function toolbar >then appropriate options in the options toolbar, to select any appropriate probe.



After placing all the probes on the circuit next step is to do the analysis, first we will cover the Analysis supported by MIX MODE SIMULATOR

Analysis Supported in Mixed Mode Simulator

- Bias Point Calculation.
- Transient Analysis.
- Parameter Analysis.
- Fourier Analysis.
- DC Sweep Analysis.
- AC Sweep Analysis.
- Monte Carlo Analysis.
- Sensitivity Analysis.

BIAS POINT CALCULATION:

(1) Calculation of all node voltages and components pin currents, assuming a steady state of the circuit (means assuming time t = 0 sec).

(2) All Capacitive and Inductive elements are inactive during the Bias Point Calculation.

(3) Logical Simulation of digital part of the circuit is not performed.

Select ***Analysis*** menu from the main menu to open ***setup simulation parameter*** with option Analysis being highlighted on the left side of the window by default. Set the general parameter from the ***general settings***.

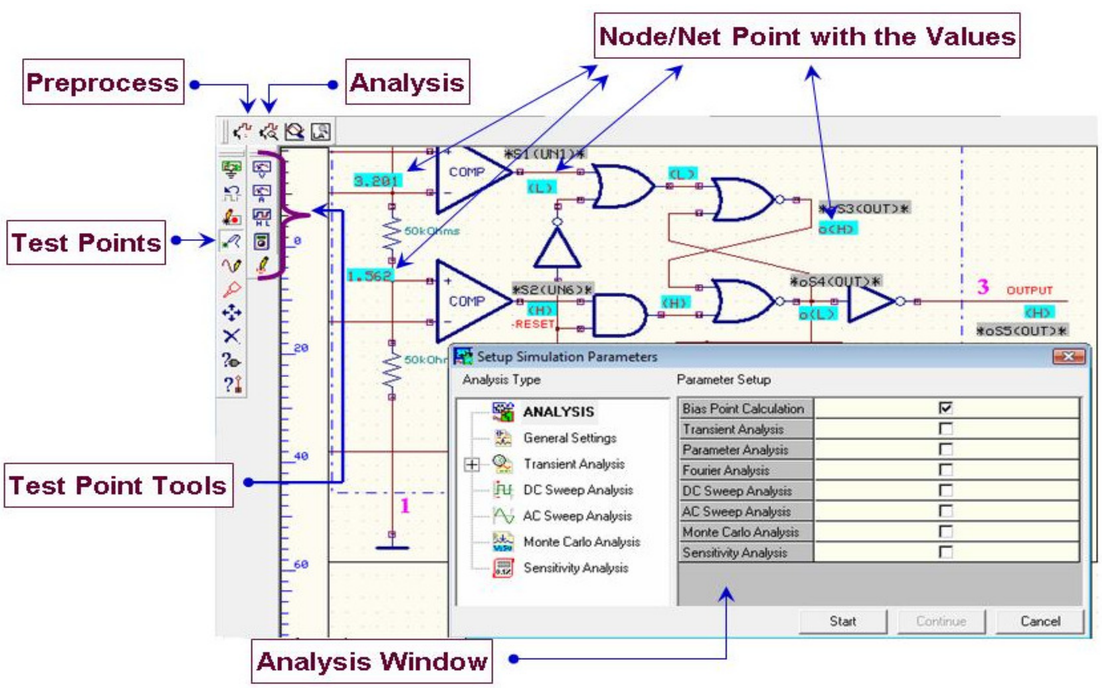


Figure: Analysis Window

Bias Point calculation pass which calculated all the voltages and currents in the circuit (Assuming time = 0)

Once the analysis is done, by selecting the test points, user can check the currents, voltages and logical state at any node or net by just clicking on that.

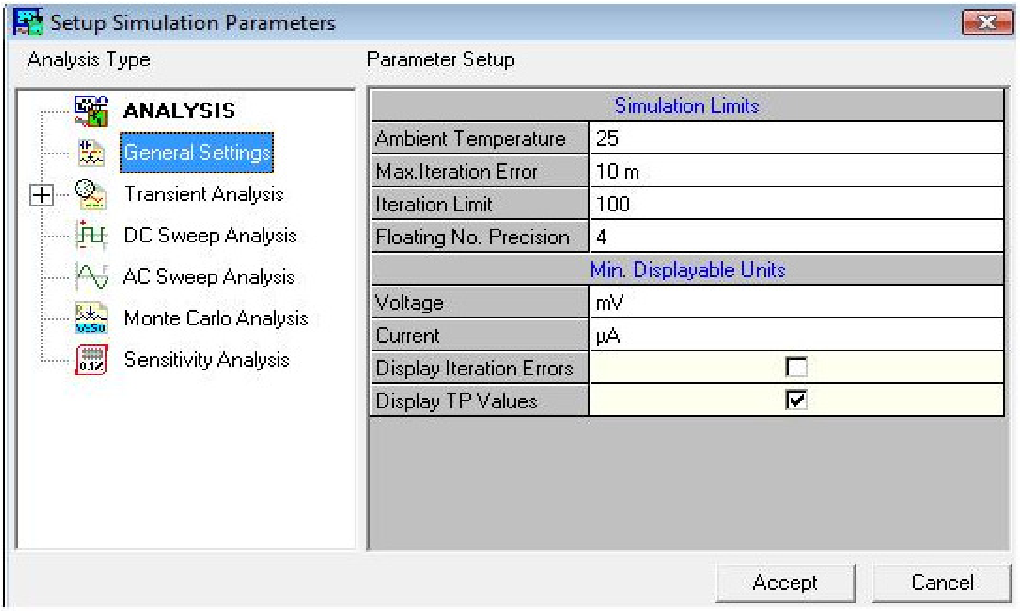


Figure: General Settings window

TRANSIENT ANALYSIS:

Transient Analysis is used to study the circuit in the time domain. Here all the parameters set or to be calculated are done with respect to time.

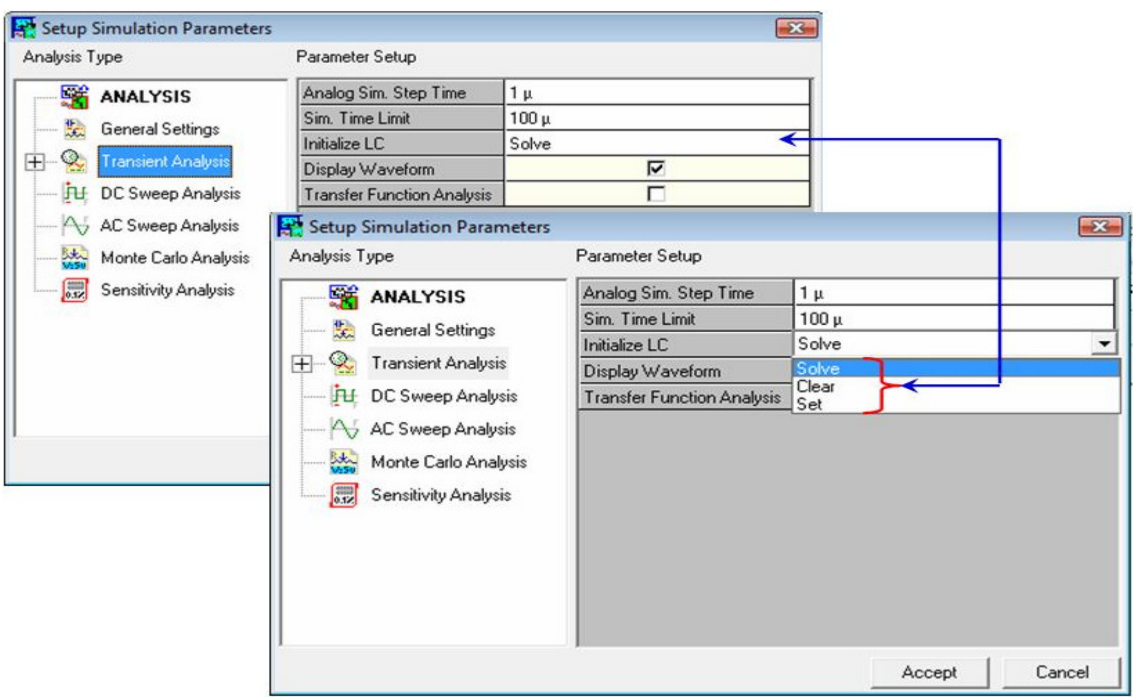


Figure: Transient Analysis Window – Initialize LC Option

Initialize LC: This option allows user to specify the initial values for Capacitor and Inductors.

It has three options:

(1) ***Solve:*** The initial conditions are solved using the supply voltage and current in the circuit.
(2) ***Clear:*** Selecting this option clears all the initial values, means all the initial values are set to zero during simulation.
(3) ***Set:*** The initial value mentions in the component simulation parameters are taken into consideration during simulation.

PARAMETER ANALYSIS:

Parameter analysis allows the user ti study the effect of variation of component parameters on the circuit.

Parameter analysis is performed with the parameter value of one component, **sweeped** through a range of values. The parameter value can also be varied in terms of **steps.**

Parameter analysis can also be carried out by considering ambient temperature as a Sweep/Step parameter.

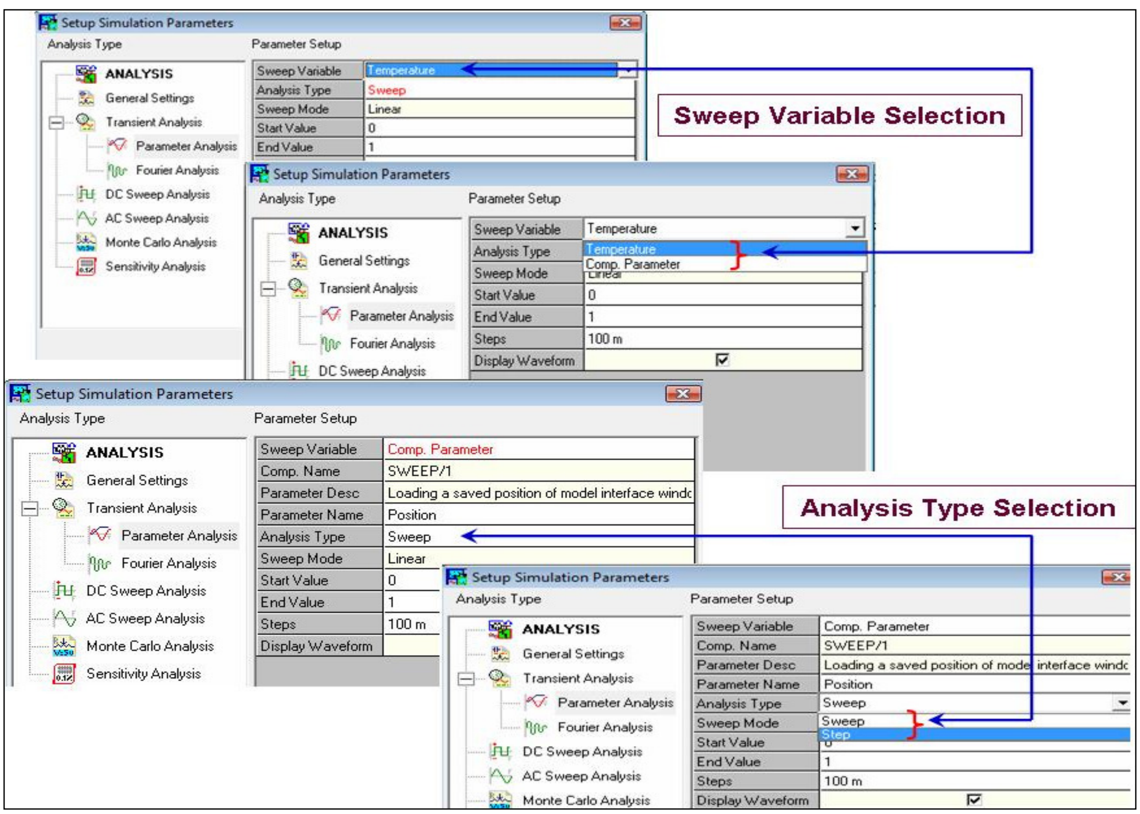


Figure: Parameter analysis window – Sweep variable and Analysis Type description.

To set the component parameter as a Sweep variable, point the cursor to the desired component in the schematic editor and click on the component, then the analysis window will get updated with the component details. Choosing the required range for which the variable has to be varied the analysis will be carried out.

FOURIER ANALYSIS:

Fourier analysis is used to calculate the total harmonic distortion of analog waveforms generated on conducting **transient analysis.**

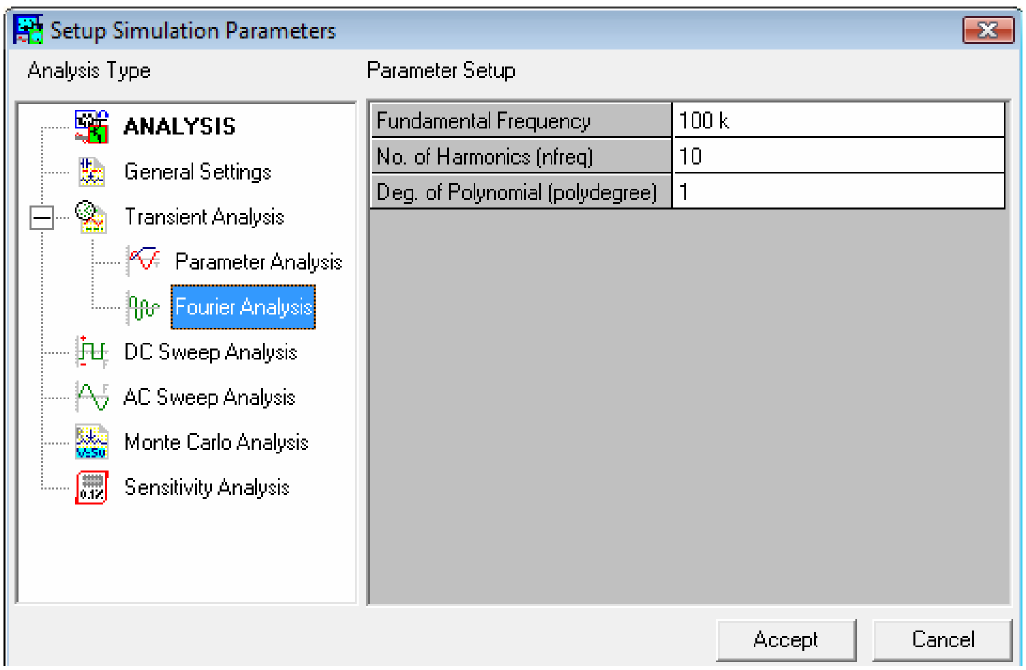


Figure: Fourier analysis window

The input parameters for the Fourier analysis are

(a) Fundamental Frequency
(b) Degree of Polynomial (Polydegree)
(c) Number of Harmonics: Number of the Sinusoidal components of frequencies that are multiples of fundamental frequency.

The point to be noted here is that as the Number of Harmonics increases the accuracy of the analysis result will increases. And also as the Degree of polynomial increases the accuracy of the analysis result increases.

The result of the analysis will be displayed in the text format.

DC SWEEP ANALYSIS:

Before proceeding to the detailed analysis of any circuit, it is very important to set up the correct operating point (Condition) of the circuit. This is achieved using the **sweep analysis.** The DC Sweep Analysis allows studying the effect due to the variation of the one circuit element by keeping another circuit element constant through out the range of values.

The DC Sweep Analysis is a non-linear analysis, which determines the DC Operating point of the circuit. During the analysis/Simulation of the circuit, the capacitors are open circuited and all inductors are short circuited.

When user choose one source as the Sweep variable then it is called as Single DC Sweep analysis and when more than one source is used then the analysis is called as Nested DC Sweep analysis.

Note that either the component parameter or ambient temperature can be chosen as the Sweep variable. To set the component parameter as a Sweep variable, point the cursor to the desired component in the schematic editor and click on the component, then the analysis window will get updated with the component details. Choosing the required range for which the variable has to be varied the analysis will be carried out.

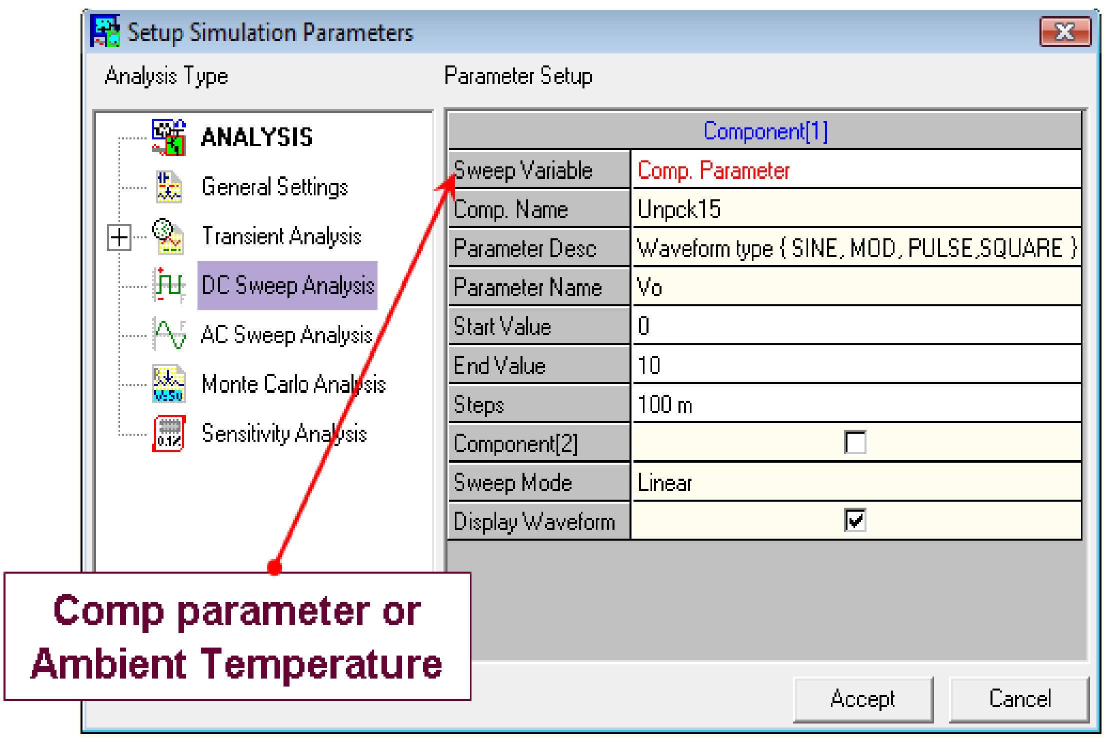


Figure : DC Sweep Analysis Window

AC SWEEP ANALYSIS:

It is linear small signal analysis, over a user defined range of frequency. Usually this analysis is used to calculate the frequency response, gain of the circuit. That is the small signal AC behaviour of the circuit. The various modes of the output are Magnitude, Phase, real, imaginary and 20log of gain.

In order to carry out AC Sweep analysis, first the user should define the AC +IN and –IN and also AC +OUT and –OUT. To define the above said parameters please refer the figure.

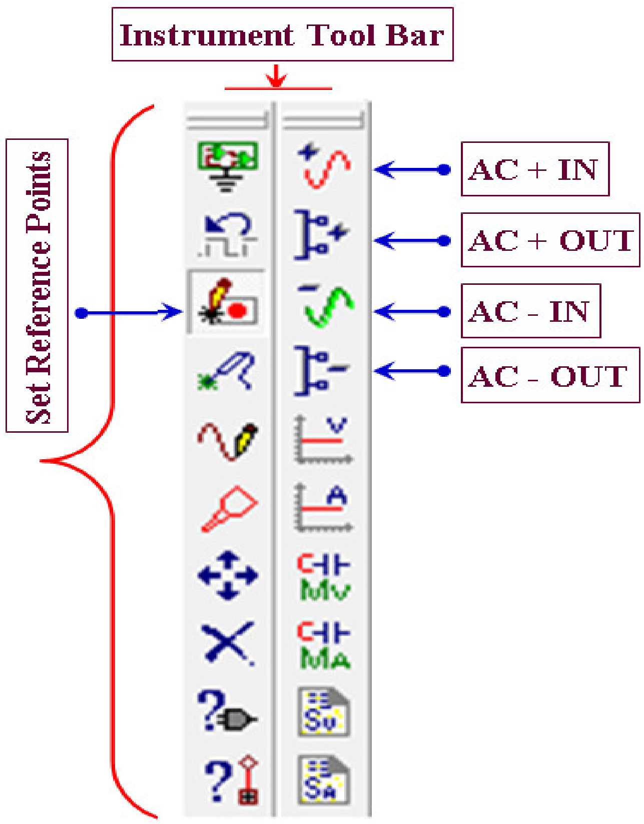


Figure : Setting – AC IN and AC OUT parameters

Once the parameters are defined, select the analysis tool and click on the AC SWEEP Analysis.

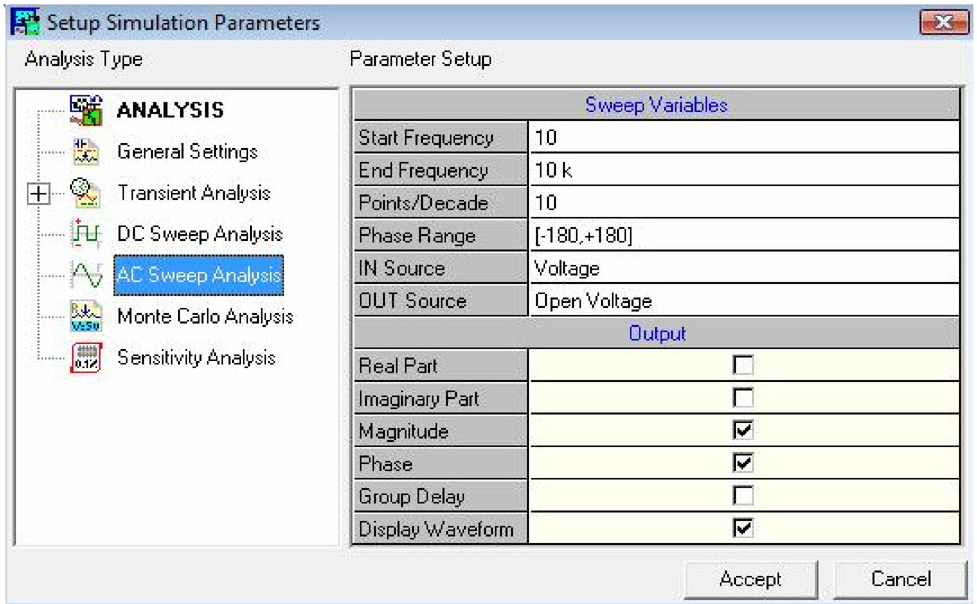


Figure : AC Sweep Analysis Window

Enter the range of the frequency for which the analysis has to be done. Mark the required type of the output and run the analysis. The result of the analysis is displayed in waveform viewer window. The below waveform is the simple example of the AC Sweep analysis result of the amplifier.

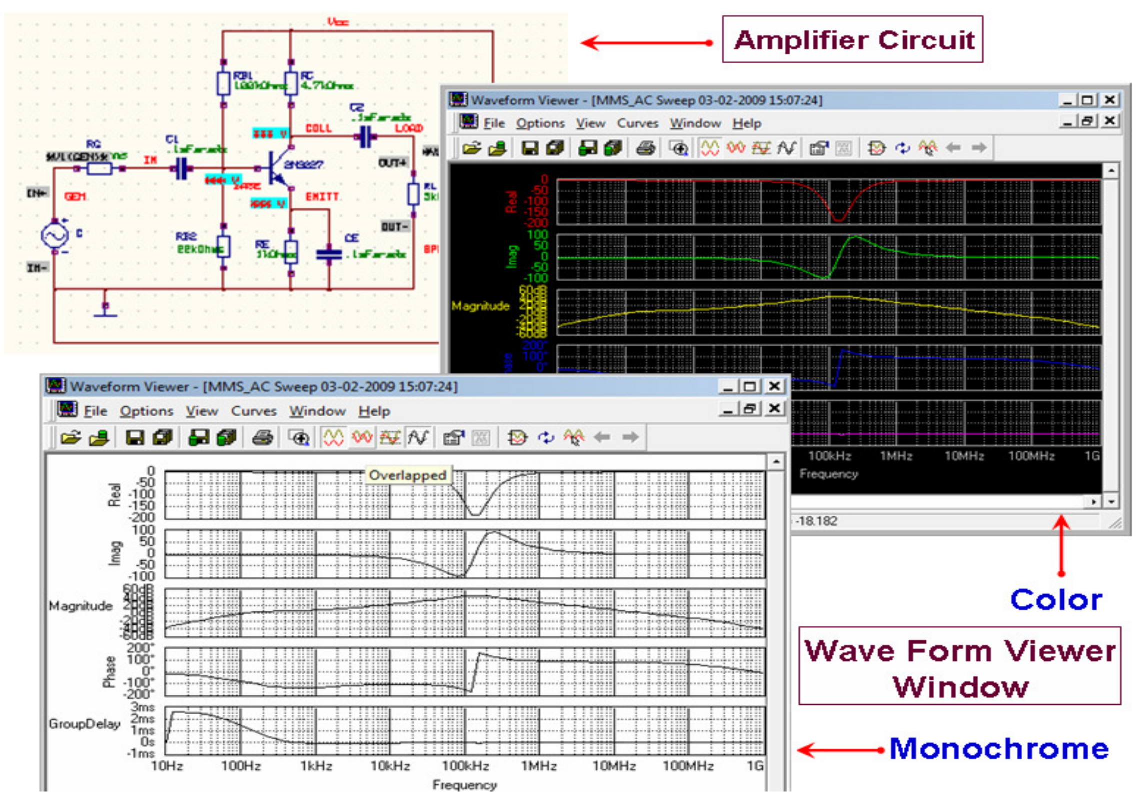


Figure: AC Sweep Analysis Result Window. (Example: Single Stage RC Coupled Amplifier)

MONTE CARLO ANALYSIS:

Simulation of the circuit with a given error on the different component helps the user to visualize how the circuit will behave with the imperfect component. This can be achieved using the Monte Carlo Analysis.

Here in this option the user can define two parameters. One is the Percentage of the error of a given circuit.

Another is the type of the error. Here the user is given two options.

(i) Gaussian – Normal
(ii) Uniform.

As in the other analysis, here also the Monte Carlo Parameters has to be defined. To do so please refer the below diagram:

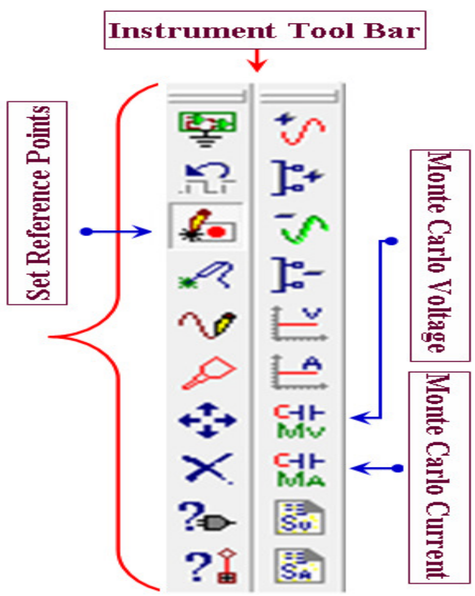


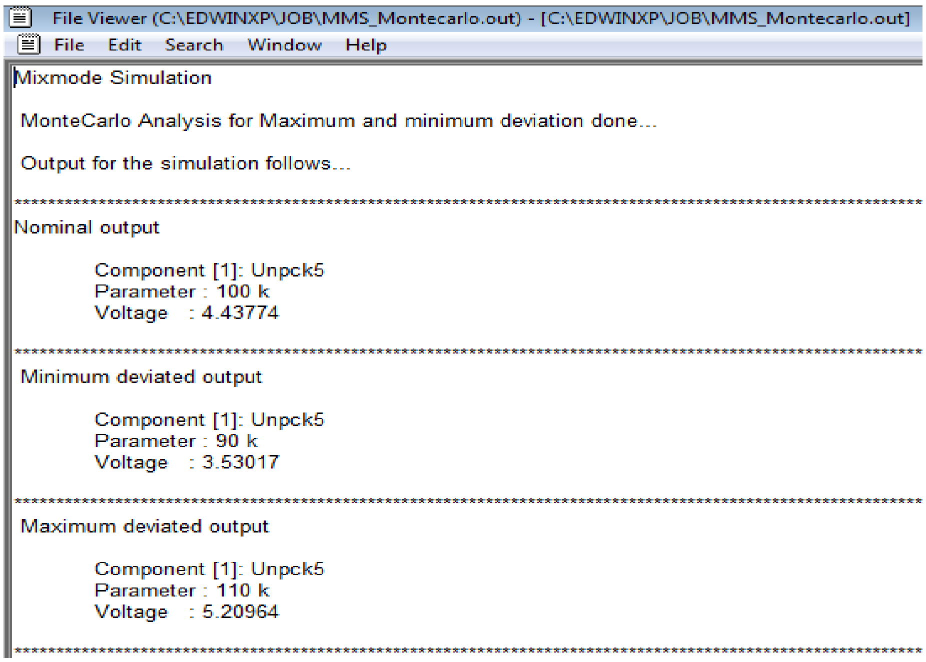
Figure : Setting : Monte Carlo Voltage and Current Parameters.

The important points to be noted here is that when the user chooses the Gaussian mode for the simulation, it uses the bell curve approach as it randomly pick values in the range of error specified but with the majority of the values near the resistor value. But when the user chooses the normal mode for the simulation, it selects values within the specified oblivious of what the actual value of the resistor is.

The result will be displayed in the text form. Please refer the below figure for the sample Monte Carlo Analysis report for the Amplifier project.

Values taken during analysis:

Error type: Normal type   Tolerance: 10%   Number of samples: 10   Time: 10ms



```
File Viewer (C:\EDWINXP\JOB\MMS_Montecarlo.out) - [C:\EDWINXP\JOB\MMS_Montecarlo.out]
File   Edit   Search   Window   Help
Mixmode Simulation

 MonteCarlo Analysis for Maximum and minimum deviation done...

 Output for the simulation follows...

*****************************************************************************************************
Nominal output

        Component [1]: Unpck5
        Parameter : 100 k
        Voltage   : 4.43774

*****************************************************************************************************
 Minimum deviated output

        Component [1]: Unpck5
        Parameter : 90 k
        Voltage   : 3.53017

*****************************************************************************************************
 Maximum deviated output

        Component [1]: Unpck5
        Parameter : 110 k
        Voltage   : 5.20964

*****************************************************************************************************
```

Figure: Output of the Monte Carlo Analysis for the project RC Coupled Amplifier.

SENSITIVITY ANALYSIS:

Sensitivity Analysis calculates the sensitivity of the output variable with respect to all circuit elements including the model parameters. The result will display the highest sensitive element in the circuit.

Defining the sensitive elements (Reference Points) in the designed circuit, choose the sensitive analysis option in the general analysis tool.

Assign the percentage change in parameter value for which the reference point is defined. The analysis report will be displayed in the text format.

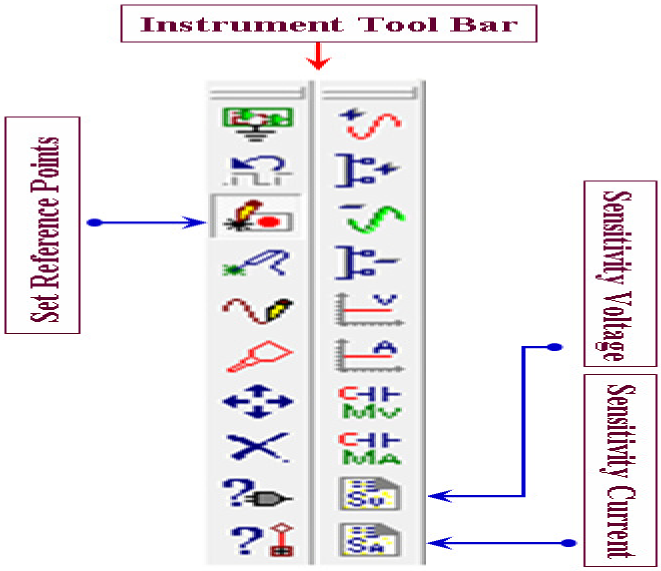
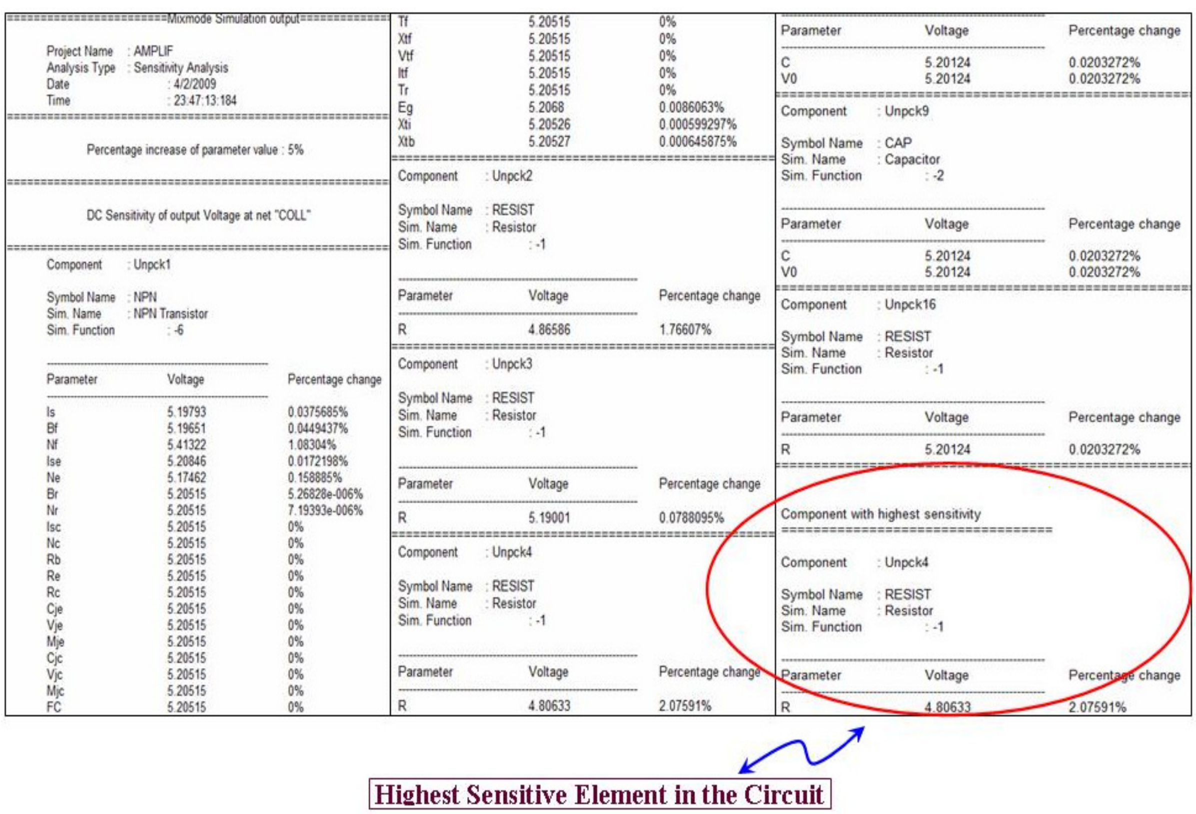


Figure: Setting voltage and current parameter

Please refer the below figure for the sample Monte Carlo Analysis report for the Amplifier project.

Values taken during analysis:

%change in Parameter Value: 10%



========================Mixmode Simulation output=============

Project Name : AMPLIF
Analysis Type : Sensitivity Analysis
Date : 4/2/2009
Time : 23:47:13:184

==

Percentage increase of parameter value : 5%

==

DC Sensitivity of output Voltage at net "COLL"

==

Component : Unpck1

Symbol Name : NPN
Sim. Name : NPN Transistor
Sim. Function : -6

Parameter	Voltage	Percentage change
Is	5.19793	0.0375685%
Bf	5.19651	0.0449437%
Nf	5.41322	1.08304%
Ise	5.20846	0.0172198%
Ne	5.17462	0.158885%
Br	5.20515	5.26828e-006%
Nr	5.20515	7.19393e-006%
Isc	5.20515	0%
Nc	5.20515	0%
Rb	5.20515	0%
Re	5.20515	0%
Rc	5.20515	0%
Cje	5.20515	0%
Vje	5.20515	0%
Mje	5.20515	0%
Cjc	5.20515	0%
Vjc	5.20515	0%
Mjc	5.20515	0%
FC	5.20515	0%
Tf	5.20515	0%
Xtf	5.20515	0%
Vtf	5.20515	0%
Itf	5.20515	0%
Tr	5.20515	0%
Eg	5.2068	0.0086063%
Xti	5.20526	0.000599297%
Xtb	5.20527	0.000645875%

==

Component : Unpck2

Symbol Name : RESIST
Sim. Name : Resistor
Sim. Function : -1

Parameter	Voltage	Percentage change
R	4.86586	1.76607%

==

Component : Unpck3

Symbol Name : RESIST
Sim. Name : Resistor
Sim. Function : -1

Parameter	Voltage	Percentage change
R	5.19001	0.0788095%

==

Component : Unpck4

Symbol Name : RESIST
Sim. Name : Resistor
Sim. Function : -1

Parameter	Voltage	Percentage change
R	4.80633	2.07591%

Parameter	Voltage	Percentage change
C	5.20124	0.0203272%
V0	5.20124	0.0203272%

==

Component : Unpck9

Symbol Name : CAP
Sim. Name : Capacitor
Sim. Function : -2

Parameter	Voltage	Percentage change
C	5.20124	0.0203272%
V0	5.20124	0.0203272%

==

Component : Unpck16

Symbol Name : RESIST
Sim. Name : Resistor
Sim. Function : -1

Parameter	Voltage	Percentage change
R	5.20124	0.0203272%

==

Component with highest sensitivity

==================================

Component : Unpck4

Symbol Name : RESIST
Sim. Name : Resistor
Sim. Function : -1

Parameter	Voltage	Percentage change
R	4.80633	2.07591%

Figure: Output of the Sensitivity Analysis for the project RC Coupled Amplifier.

Micro Controller Co-Simulation

The modern technologies built now days, most of the new products are controlled using the microcontrollers. The students must know the behaviour of the microcontrollers. The EDWinXP gives a room for the student to study the behaviour of the micro controller and also to take use of the same in their projects.

EDWinXP Supports:

AVR

PIC16C58A/SS

PIC16CR54/SS

PIC16CR57A/SS

8051 and in the Ver 1.70, the Motorola Kit is also introduced.

The point to be noted here is that any component to be simulated should have the behavioural structure of the same. The EDWinXP has built-in behavioural structure for the above mentioned microcontrollers.

To select the required microcontroller use the browser option in the EDWinXP. Refer the below shown figure.

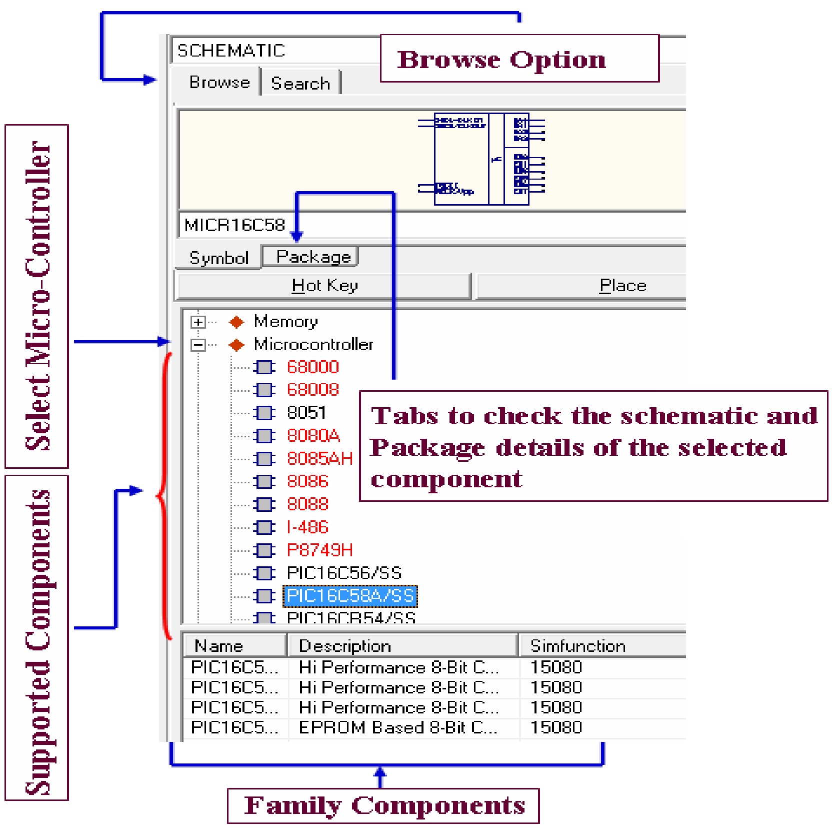


Figure: Micro-Controller

Once the required Micro-Controller is selected, to program the same select the *Mixed Mode Simulator* and choose the *Instrument* tool option. Select *Change Simulation Parameter* option to display the window as shown below.

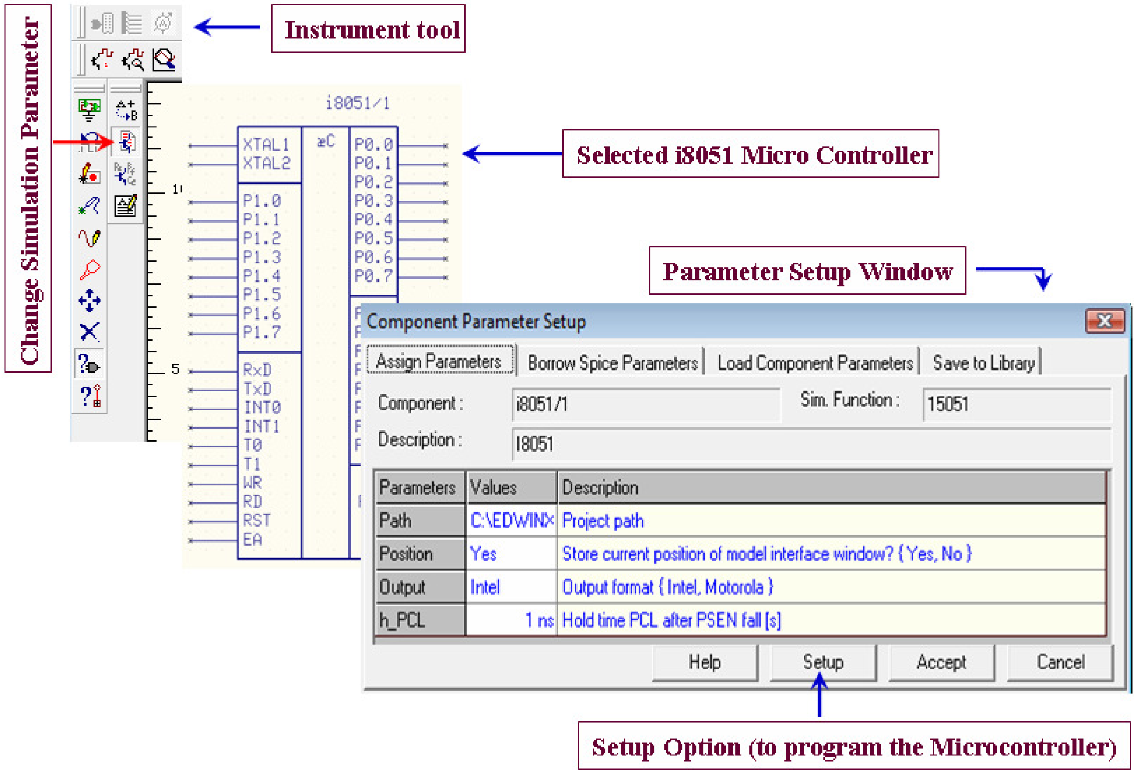


Figure: Invoking the Micro-Controller

To Program the Microcontroller, click on Setup push button and the windows will popup as shown below:

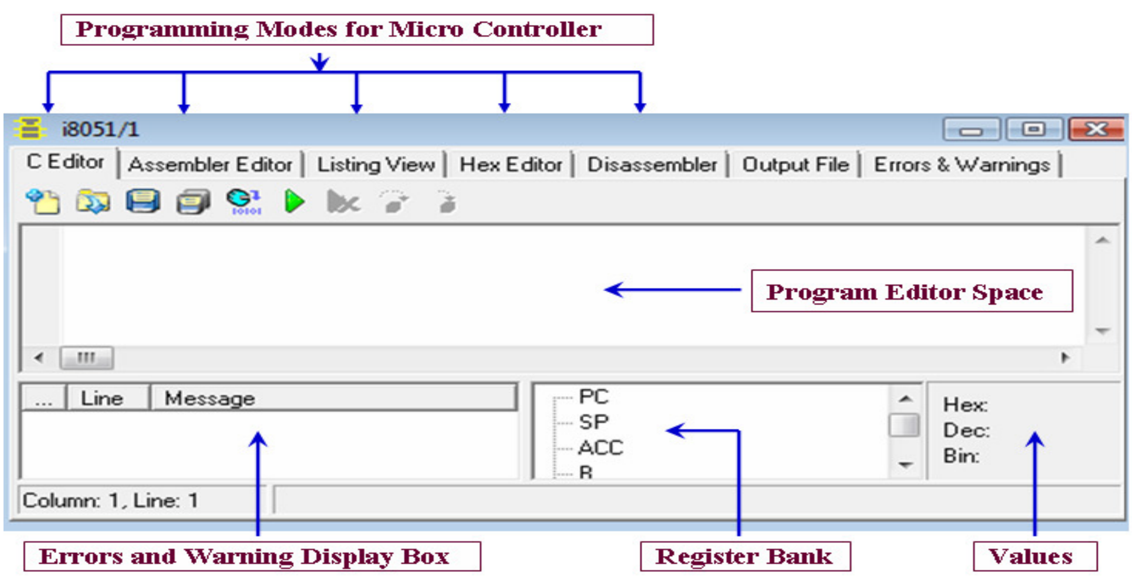


Figure: Setup Window for i8051 Micro-Controller

C-Editor:

In the C-Editor option the user is subjected to program the microcontroller using the C language. Once the program is done the user can built the microcontroller by a single click on the built option provided in the C-Editor Tab. The error and warning message box displays the built status.

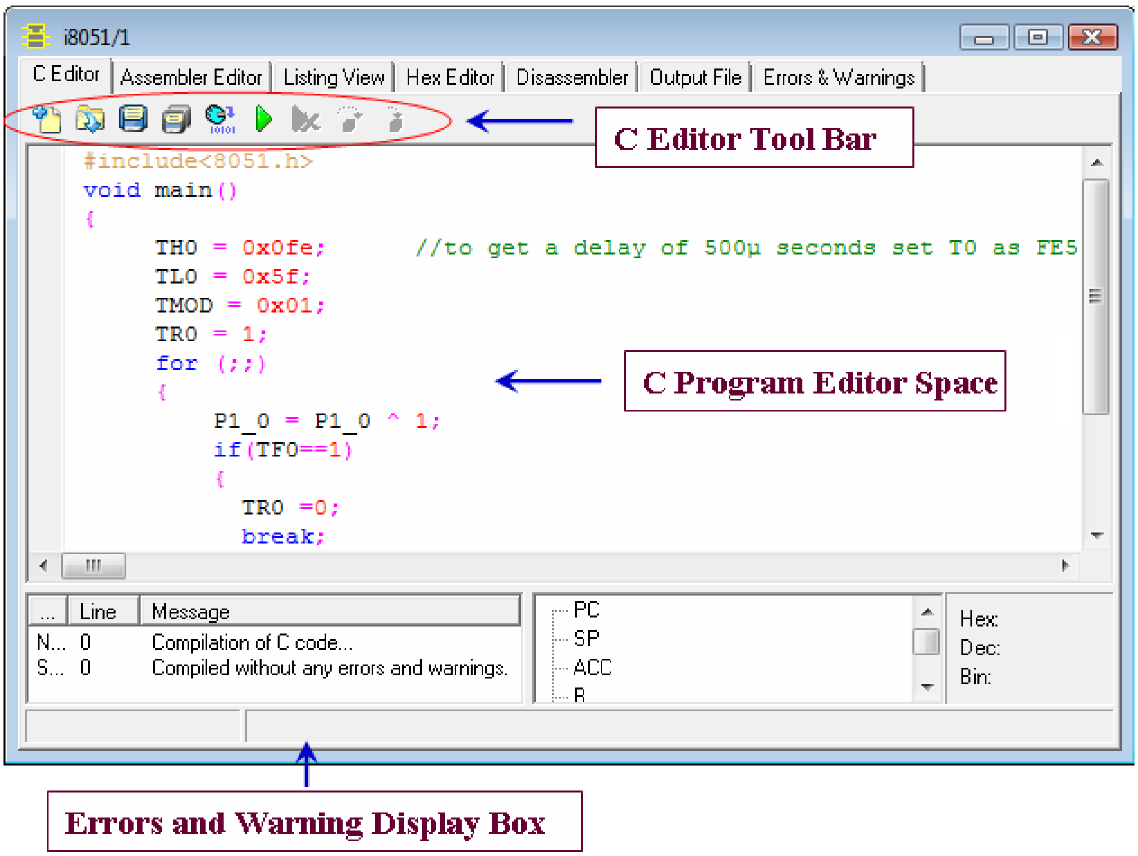


Figure: Setup Window for i8051 Micro-Controller - C-Editor ---Example

Tool Bar in C-Editor:

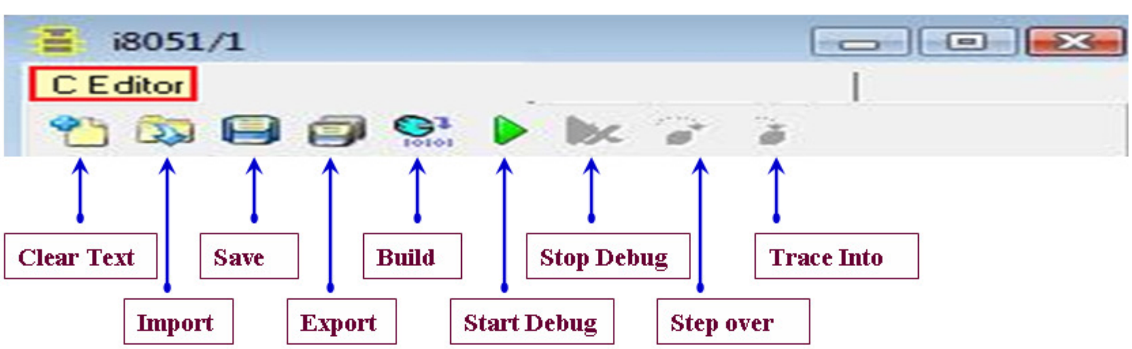


Figure: Tool Bar in C-Editor

Export: The Export option displays the *On Space Dialog Box* to prompt the user for output file name and saves the source code to the disk.

Compile and Build: The Compile option enables the user to *Generate* the *On compile* event to notify model to compile the source code and the Build option *Generate* an *On Build* event to notify model to compile the source code.

Start Debug: This option enables the user to *Generate* the *On Debug* event to notify model to debug the source code.

Stop Debug: This option allows the user to stop debugging which is associated with the generation of an *On Stop Debug* event to notify model to stop debugging the source code.

Step Over: The option allows the user to generate an *On Step Over* event to notify model to step over the being currently debugged part of code.

Trace Into: The option allows the user to generate an *On Trace Into* event to notify model to Trace Into the being currently debugged part of code.

Assembler Editor:

If the user wants to program the Micro-Controller using the Assembly Program concept, the user can take use of the Assembler Tab.

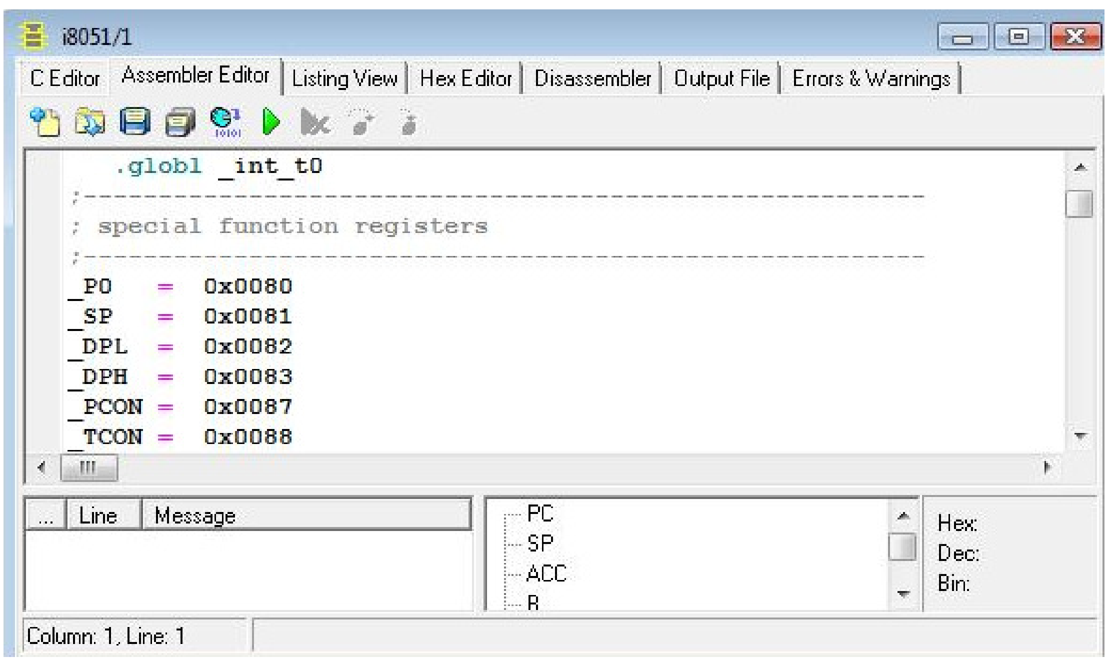


Figure: Window for i8051 Micro-Controller - Assembler-Editor --- Example

Note that the tool bar in the Assembler Editor is same as the tool bar in the C-Editor

Listing View:

Listing view represents the listing format of assembler code.

Most of the Complex projects designed using the Micro-Controller requires some of the information as mentioned below:

- Real addresses of all assembler instructions used in the code.
- Physical addresses of all the data in the code scope.
- Real addresses of global variables which use the internal memory of Micro-controller.
- The initial address of stack pointer
- Information about used banks of registers.

The above mentioned information will be available to the programmer after the compilation of the program through the *Listing View.*

Hex Editor:

This editor is used to display the binary data in the Hex and ASCII formats. Here the user can edit the binary data. This editor provides the functions like Export, Import and Save option.

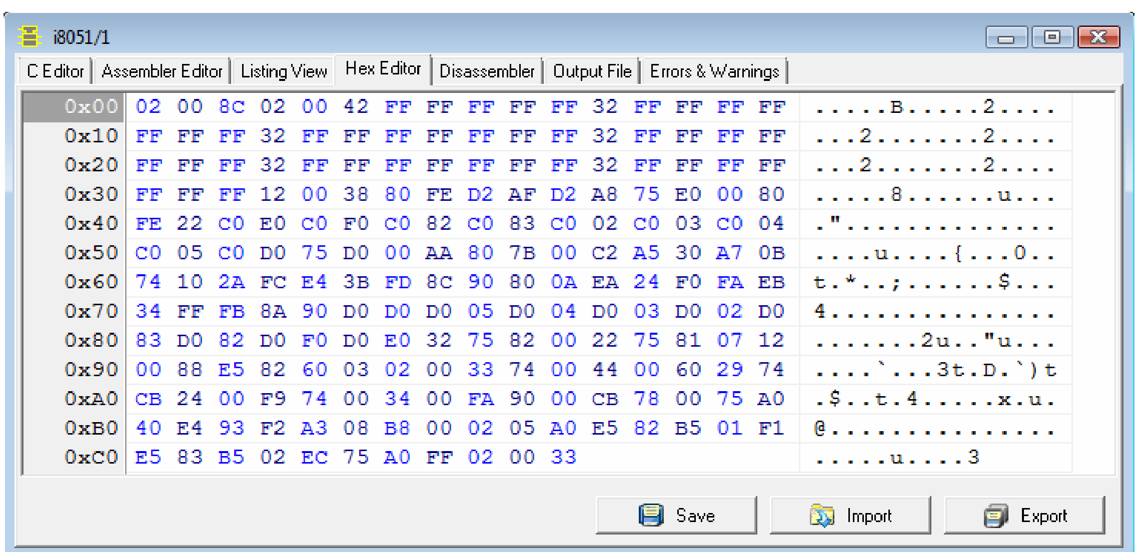


Figure: Hex Editor Window

Dissembler:

This shows the actual instructions loaded in the micro controller. Here the user is able to see the continuous address area. Here the user can come to know the actual library functions loaded.

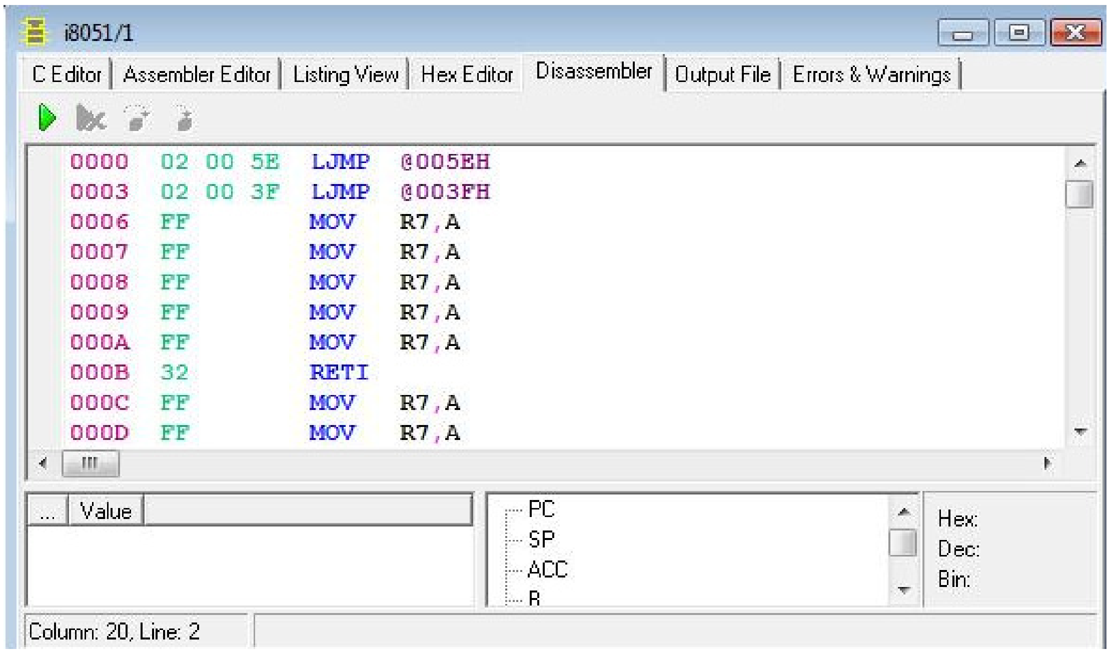


Figure: Dissembler Window

Output File:

The Output File generates the binary code for the source code written in the C-Editor after compilation.

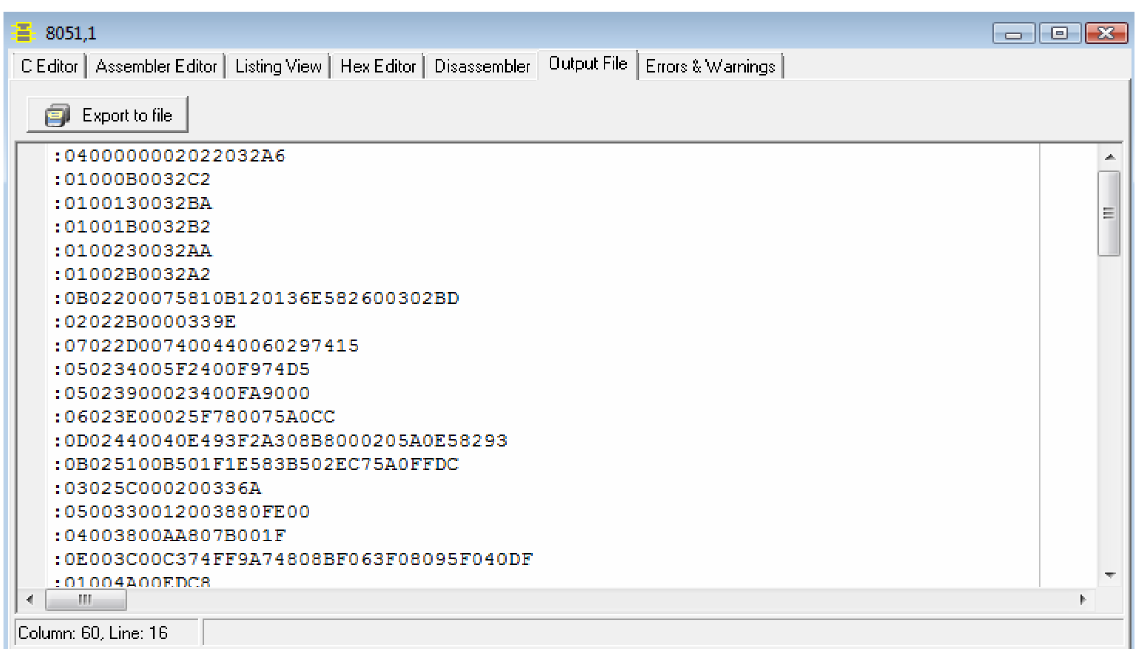


Figure: Output File Window

Errors & Warnings:

Errors & Warnings tab allows the user to check the errors and warning generated during the simulation (Mixed Mode Simulation in time domain). Please note that this window/tab does not generate any event. This window also displays the simulation time.

SPICE Simulation (EDSPICE SIMULATOR)

SPICE: SIMULATION PROGRAM FOR INTEGRATED CIRCUIT EMPHASIS.

The circuit can be represented in the form of text called "circuit file". This file can be created using any text editor and can be submitted to the SPICE for analysis of the circuit.

Steps for using SPICE:

1. Start with the schematic diagram and notate it for SPICE consisting of three steps:

a) Give each component or circuit element a name
b) Assign one node with the node number zero. This is the ground node and all other circuit node voltage will be expressed with respect to this node.
c) Each additional node in the circuit is given a node number, which must be a positive integer. The order in which the nodes are numbered is arbitrary, and node numbers need not be sequential. Each node must be connected to al least two elements.

2. Decide the type of the analysis to be carried out to check the circuit. SPICE supports the following analysis to be carried out.

a) DC AC Transient
b) DC Transfer function
c) DC Small signal sensitivity
d) Distortion analysis
e) Noise analysis
f) Fourier analysis

Based on the analysis to be carried out, one or more control line in the program has to be added, like DC Voltage and Current, range of Frequencies…etc.

3. Create an input file for SPICE using a text editor; the first line must be a title line and the last line of the input file must be .END

Simulation of the circuit files (SPICE File):

First select the EDSpice Simulation from the preference menu and then using the ***simulation*** menu select the ***circuit file editor*** option. This pops up the window (please refer the figure given) where the user can open the circuit file which is already exists or to create the new file. Once the circuit file is written, then select ***simulate*** for the analysis to be carried out. Note that the output is displayed in the waveform viewer.

EDSpice Interactive Interpreter

This window is invoked automatically by running any analysis in EDSpice Simulator.

It is very important to note that, it is not necessary to draw the circuit diagram and then simulate it or to get the spice netlist for that. It is possible to write the Spice Netlist in the text editor with the file extension as *.CIR. And using the EDSpice Simulator we can simulate or can analyze the circuit and can generate the required type of the output.

The input file of the Spice should always contain:

- A title line
- Element line for each and every element in the schematic
- .END line

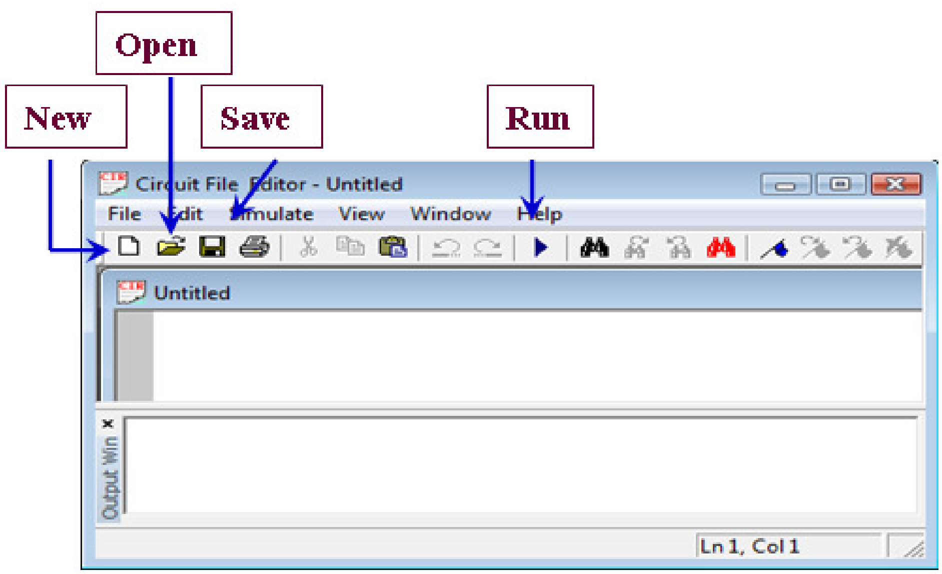


Figure: Circuit File Editor Window

Spice Circuit file for Simple Astable Multivibrator circuit is given below for the reference:

```
Astable Multivibrator: (Netlist view)
*
*
.IC V(3)=100m V(5)=6 V(7)=-6 V(6)=800m V(4)=6
*
C8 5 6 4.7U
C7 3 7 4.7U
R6 7 4 4.7k
R5 6 4 4.7k
R4 5 4 1k
R3 3 4 1k
Q2 5 7 0 Q2
Q1 3 6 0 Q1
V9 4 0 6
.MODEL Q2 NPN CJE=1p TF=10n CJC=1p
.MODEL Q1 NPN CJE=1p TF=10n CJC=1p
*CODE MODEL DEFINITIONS
*
*
.END
```

Spice control lines for the output:

.PRINT: this is used to display the result of each selected analysis in the textual format as a set of values

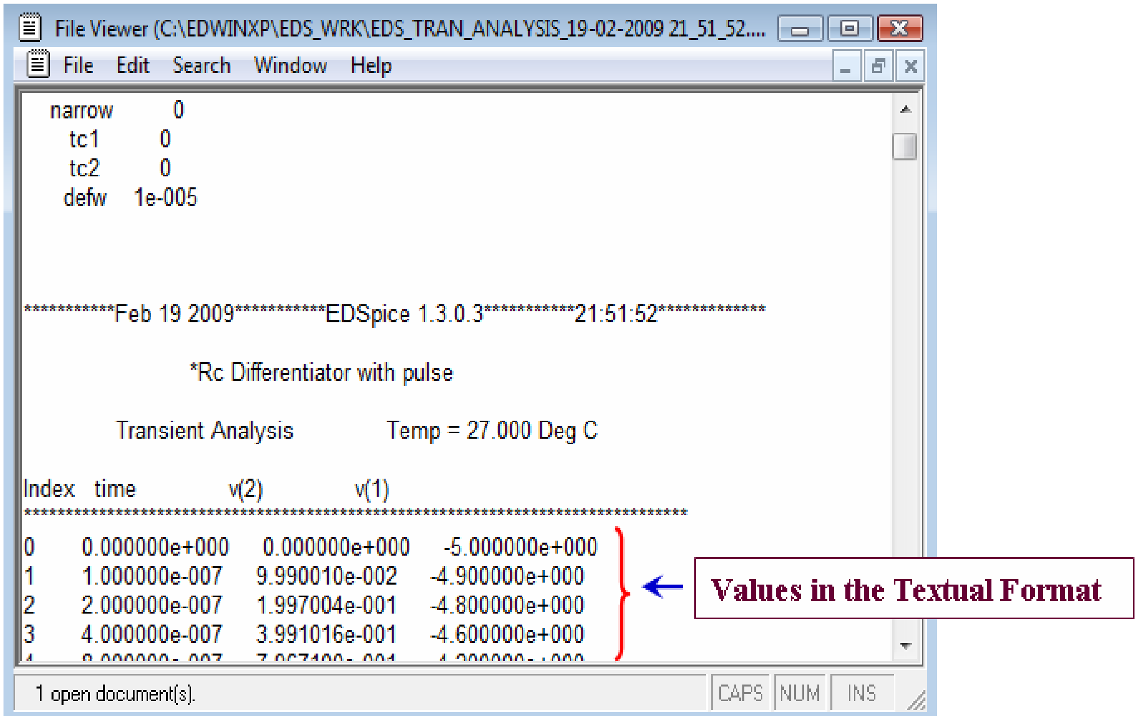


Figure: Result window for .PRINT Spice Control Line

.PLOT: this is used to plot the points obtained after each selected analysis.

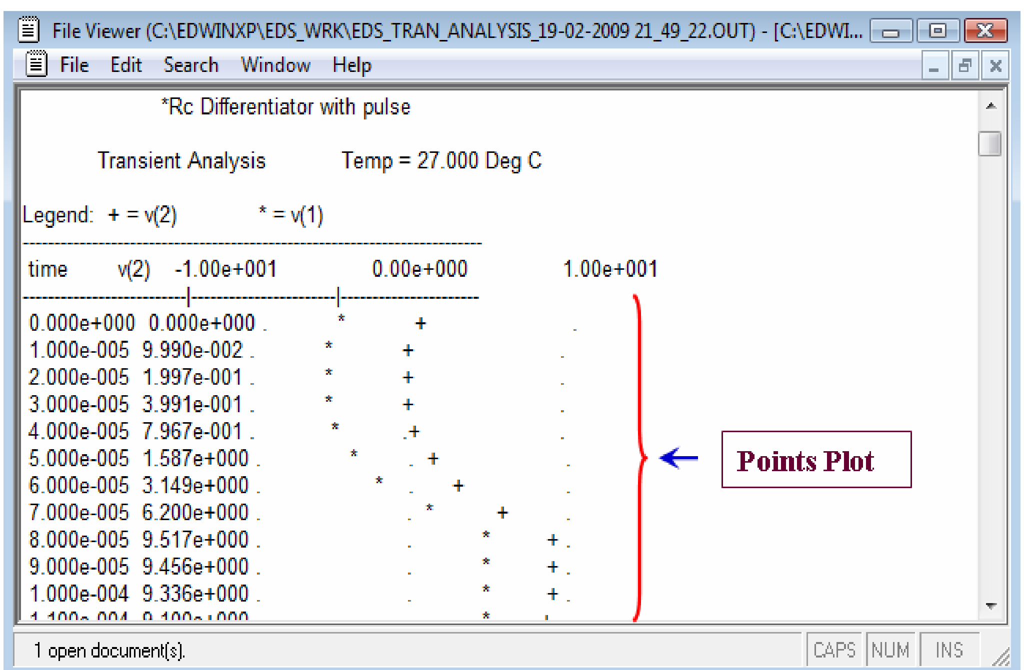


Figure: Result window for the .PLOT Spice Control Line

Analysis Supported in EDSpice Simulator

TRANSIENT ANALYSIS:

Regardless the Spice Circuit file simulation; the user can design the hard circuit in the Schematic Editor and analyze the circuit in all aspects. First the circuit is build using the Schematic editor and then by selecting the Spice Simulation from the preference, enable the instrument tool. Assign the required values for the elements/components used in the circuit using the Change Simulation Parameter. Choose the transient Analysis in the Analysis window and give the values such as Start time, Stop time, Step….etc - Please refer the below figure. Once the parameters are set select the Waveform Viewer option from the tree. Do the proper settings as explained the figure shown below:

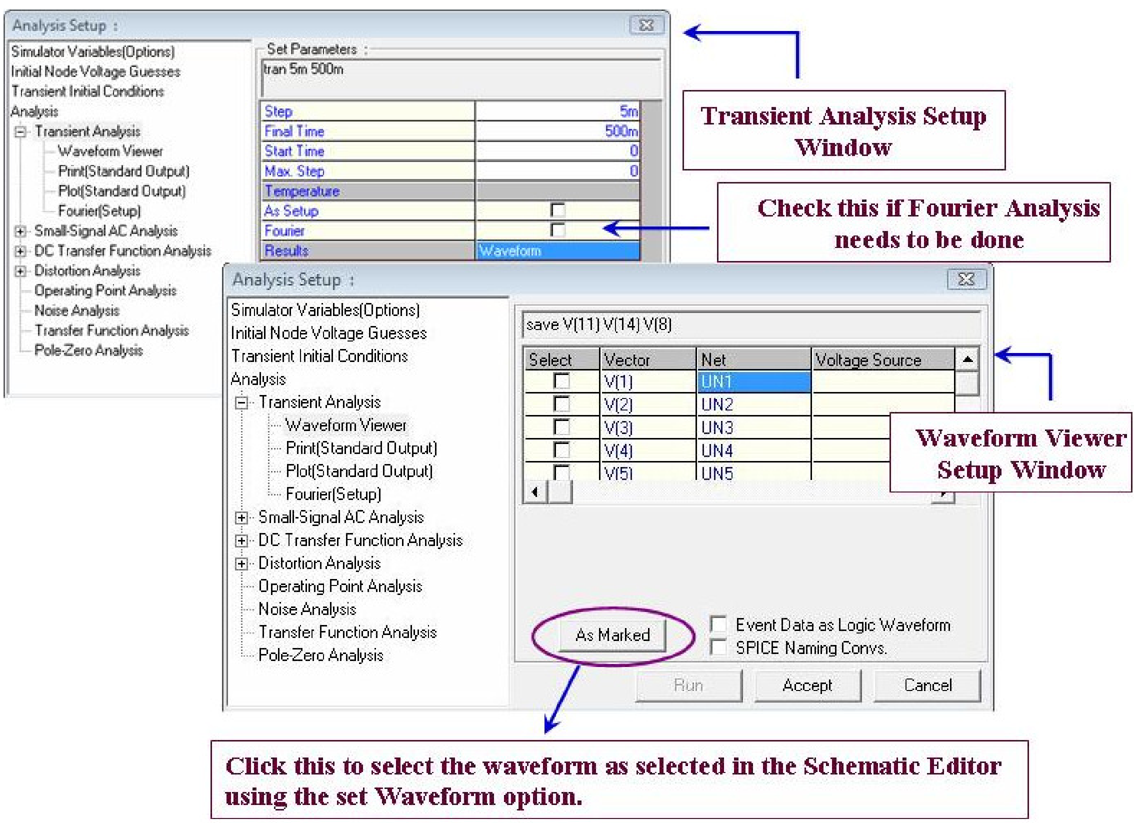


Figure: Transient Analysis Setup Window with the Waveform Viewer Options

The output type selected in the transient setup option, (supported Standard Outputs, No output, Waveform), assign the parameters required in the respective options from the tree of the transient analysis. Standard output covers both the PLOT and PRINT option.

PLOT option plots the points calculated during the analysis of the circuit.

PRINT option represents the output in the textual format. All the calculated values with respect to the time is defined and displayed in the output.

No Output option compiles the circuit; means for the circuit designed, the circuit file will be generated but displays NO OUTPUT.

Please refer the window Setup for the PLOT and PRINT Standard Output option.

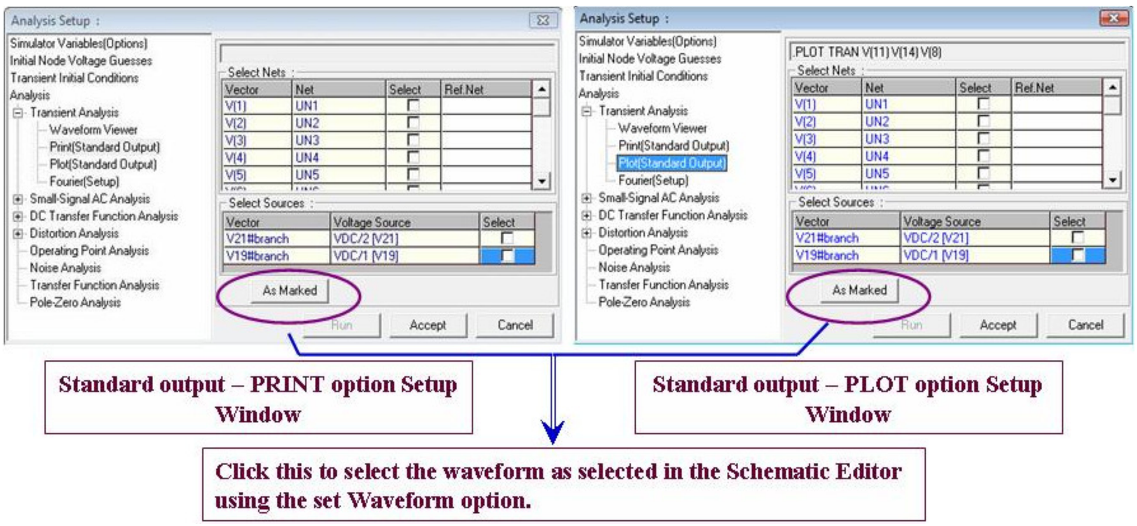


Figure: PLOT and PRINT Standard Output Option.

The Fourier analysis can be done by just checking the Fourier Option in the Transient Analysis Setup Window. The Setup Window for the Fourier analysis is shown below:

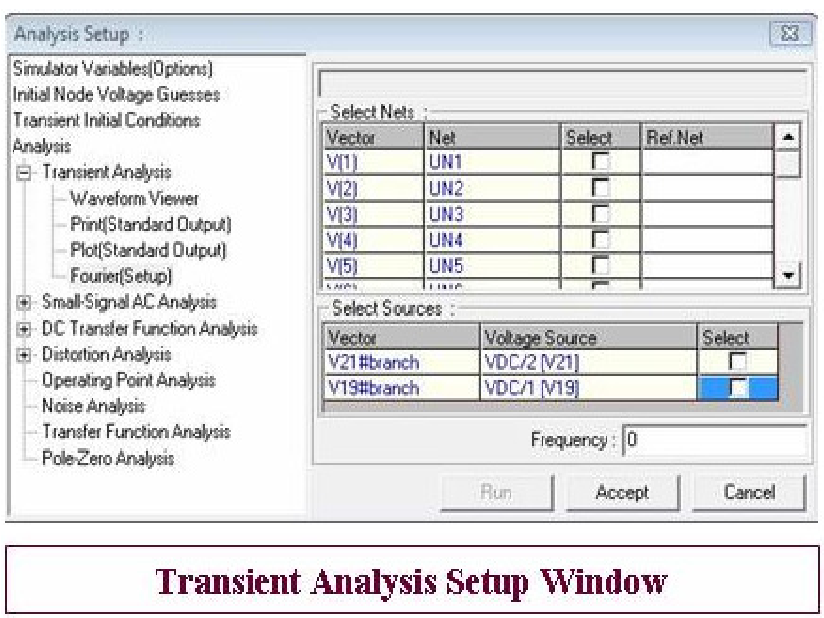


Figure: Fourier Setup Window.

SMALL SIGNAL AC- ANALYSIS:

As every knows that the digital circuit simulation and the analysis is simple or easy, since it has only two levels that is Zero's or One's. Whereas the analog circuits analysis and the simulation is quiet complicated. Most of the analysis done includes many parameters to be set and the parameters to be calculated.

The circuits such as the amplifier, oscillators or the circuits including the AC parameters requires sweep analysis to get the gain and phase response to the frequency variations. The SWEEP analysis or SMALL SIGNAL AC – analysis performs the analysis of the circuit with respect to the variation of the frequency. Please refer the figure given below for the setting the parameters required for the analysis.

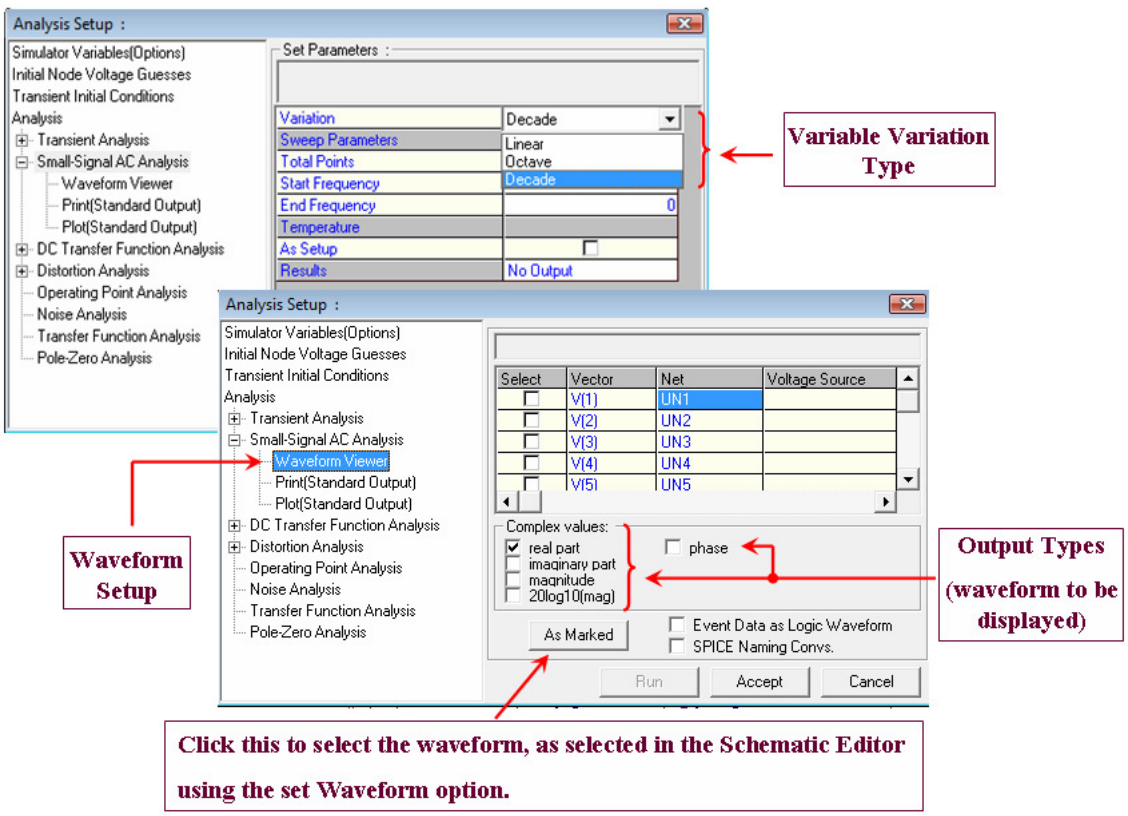


Figure: Small Signal AC Analysis

First the circuit is build using the Schematic editor and then by selecting the Spice Simulation from the preference, enable the instrument tool. Assign the required values for the elements/components used in the circuit using the Change Simulation Parameter. Choose the Small Signal AC Analysis in the Analysis window and give the values such as total points, start frequency, end frequency….etc. Once the parameters are set select the Waveform Viewer option from the tree. Do the proper settings as explained the figure shown above.

Note that the user is allowed to select the complex values to be displayed in the waveform. And also the user can select the variation type that is Decade, Octave or Linear.

The output type selected in the small signal AC analysis setup option, (supported Standard Outputs, No output, Waveform), assign the parameters required in the respective options from the tree of the small signal AC analysis. Standard output covers both the PLOT and PRINT option.

PLOT option plots the points calculated during the analysis of the circuit.

PRINT option represents the output in the textual format. All the calculated values with respect to the time is defined and displayed in the output.

No Output option compiles the circuit; means for the circuit designed, the circuit file will be generated but displays NO OUTPUT.

DC TRANSFER FUNCTION - ANALYSIS:

For any user to simulate the complex circuit, it is very necessary to know the operations of the component behaviour for the changes of simulation parameters. This can be achieved using the DC Transfer Function analysis.

The students can take use of this type of analysis to study the input and output characteristics of semiconductors and also the forward and reverse bias characteristics of the diodes.

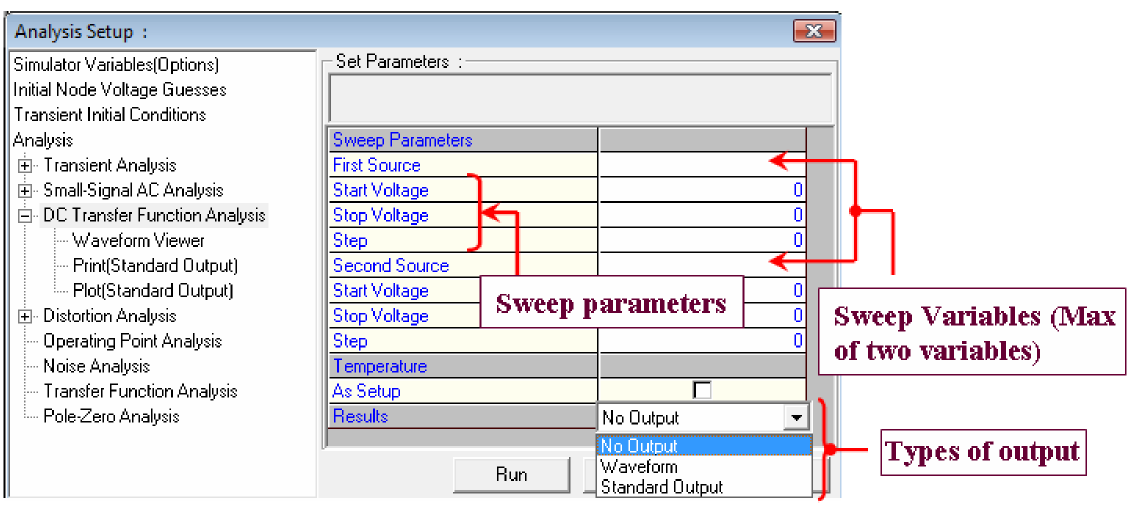


Figure: DC Transfer Function Setup Window

The above figure shows the setup window for the DC Transfer Function analysis. Here the user can select maximum of two sweep variables for the analysis.

Please refer the below figure for the waveform showing the output of the simple experiment.

In the figure shown below, the rounded part shows the Control Line for the DC Transfer Function. Note that it also includes the temperature for the analysis is also mentioned. The temperature for which the analysis has to done can be varied using the setup for the respective analysis.

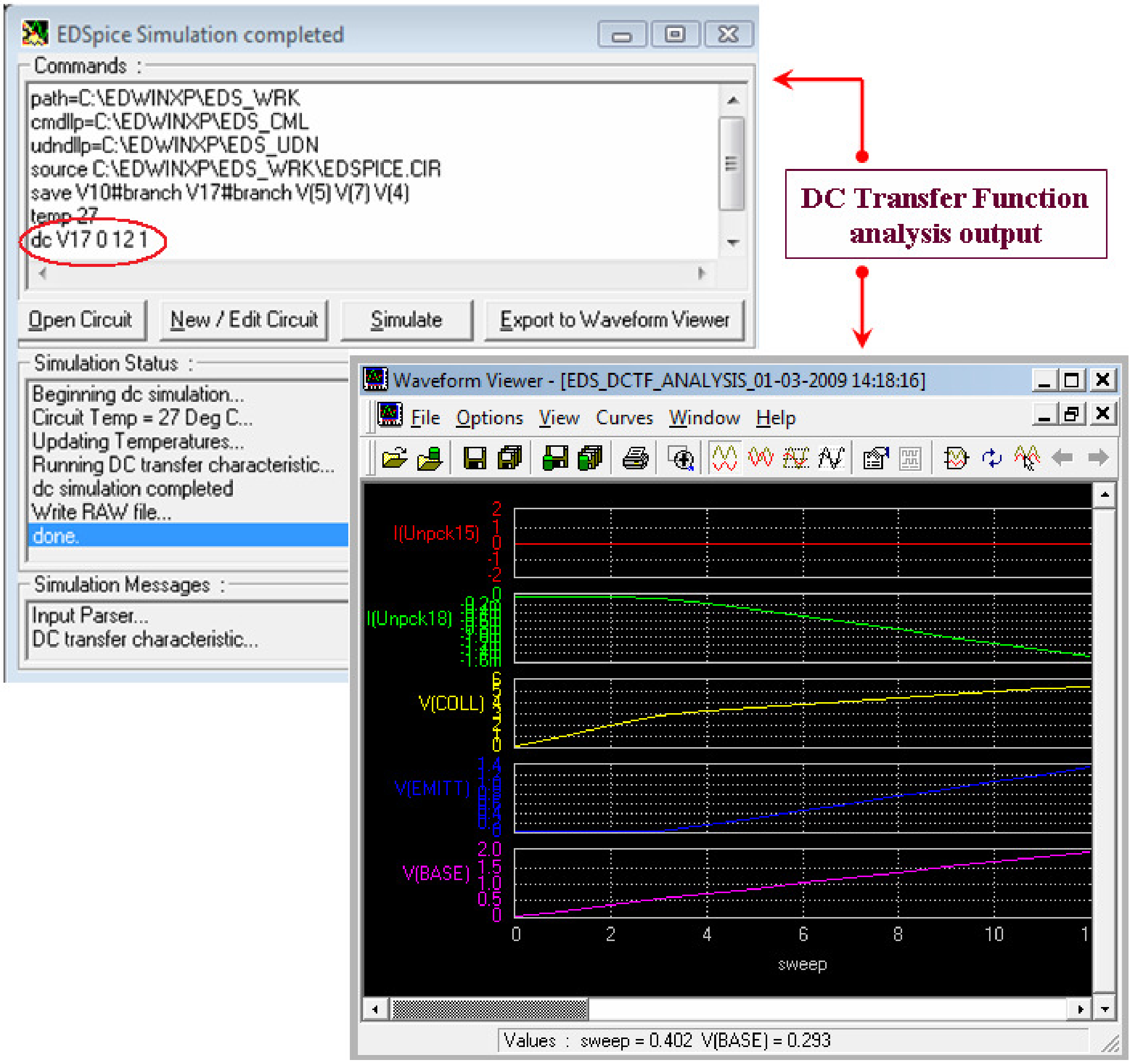
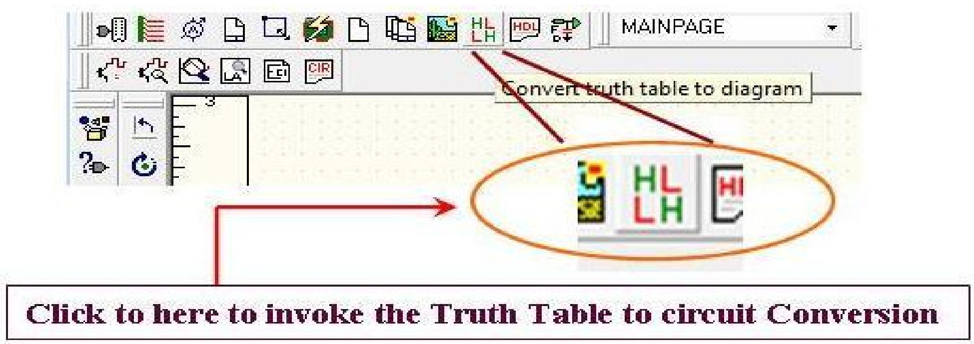


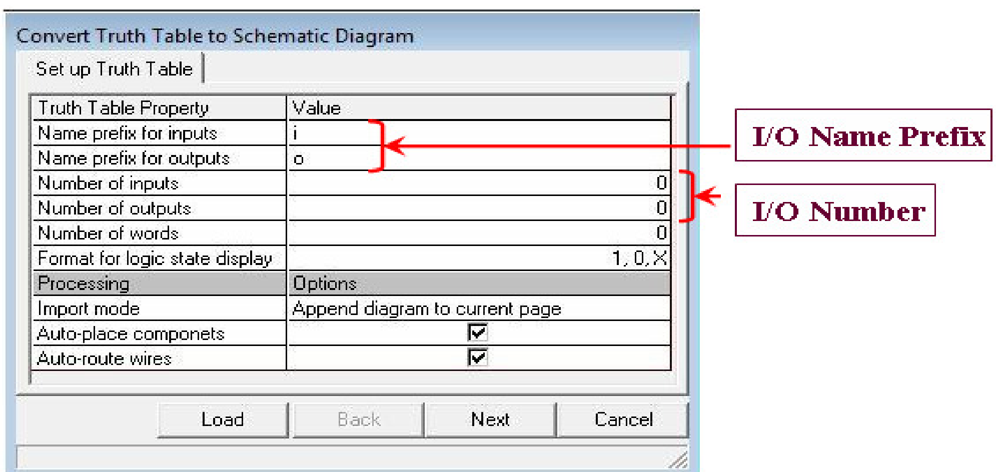
Figure: Simulation output and Output waveform for the DC transfer function of the Amplifier circuit

TRUTH TABLE TO CIRCUIT CONVERSION

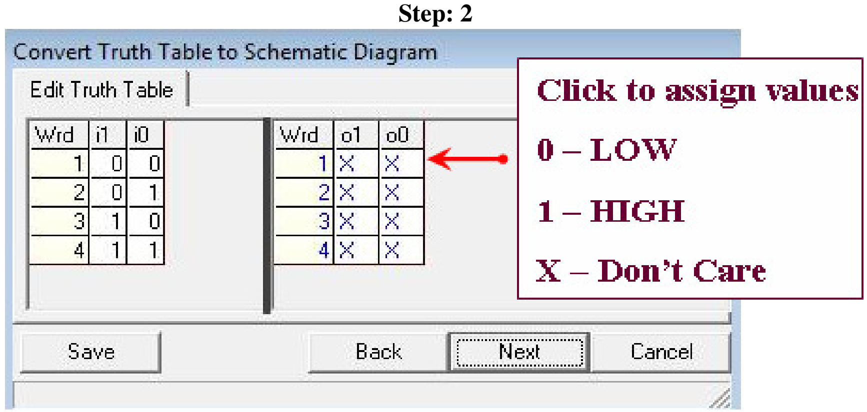
EDWinXP supports the user to design the circuit using the Truth Table to Circuit conversion. This option is invoked by clicking the Truth Table to Circuit Conversion. To design the circuit using the Truth Table to Circuit conversion option follow the steps as given below:



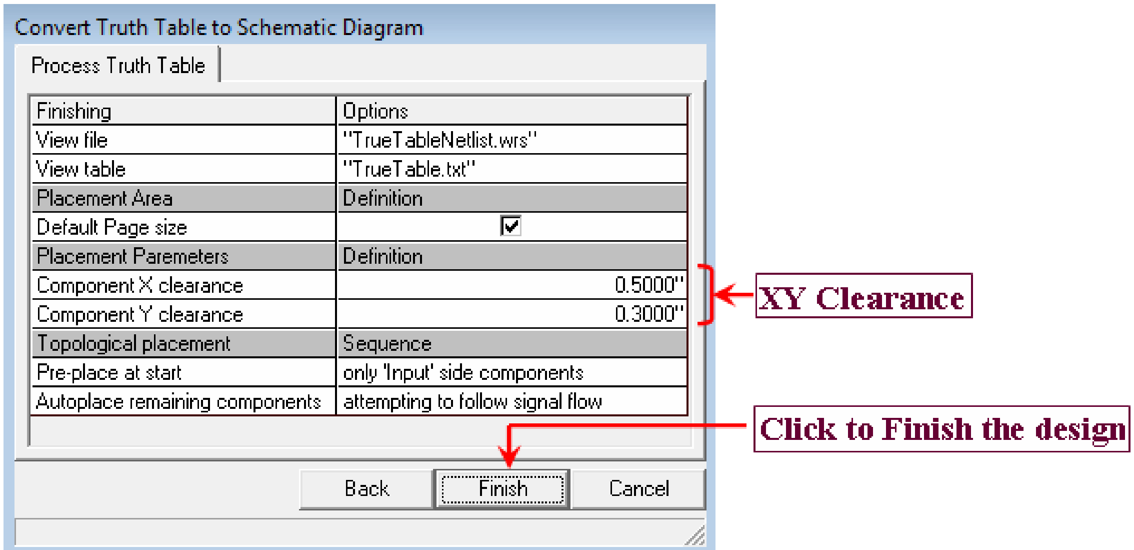
Step: 1



Step: 2



Step: 3



Step: 4

Remark:

- The result of the steps shown above gives the circuit and all the components generated will be the universal gates only.

- The user can assign don't care also in the truth table.

- TT to Circuit conversion makes the user to define the circuit easily. This will allow to define the sub circuit for the project including analog and digital design.

ACTIVE FILTER DESIGNING

The designing of the Active Filter includes the complex calculations depending on the degree of the filter. To make the designing of the Active Filter easy, EDWinXP provides the user friendly Active Filter Design by one click option.

EDWinXP supports the following Active Filters:

Chebyshev Filter

Butterworth

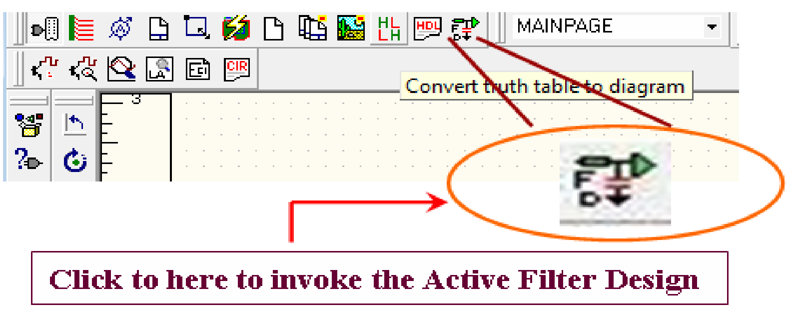
Bessel

Elliptic (Cauer)

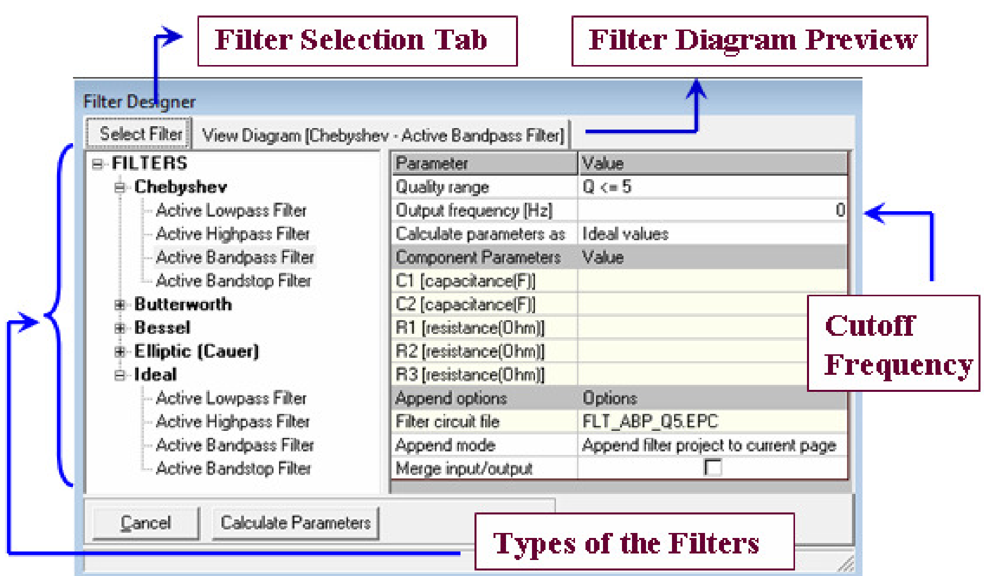
Ideal

All the above mentioned filters support Lowpass, Highpass, Bandpass and Band StopFilters.

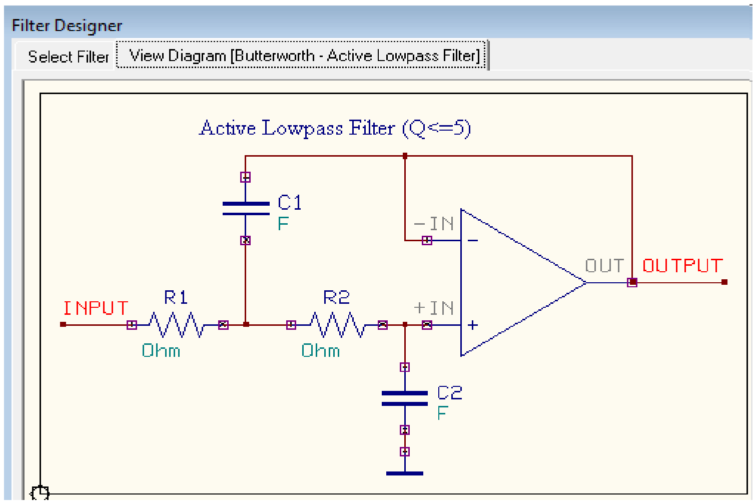
To design the Active filter using the option provided using the EDWinXP; follow the steps as given below:



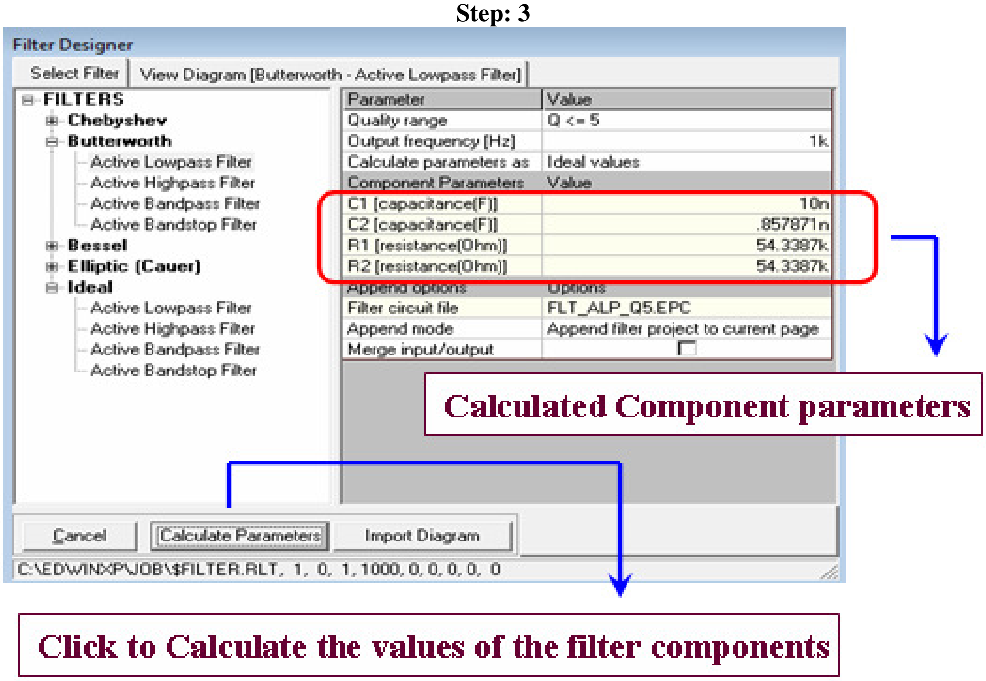
Step: 1



Step: 2


Filter Designer
Select Filter
View Diagram [Butterworth - Active Lowpass Filter]
Active Lowpass Filter (Q<=5)
C1
F
-IN
OUT
OUTPUT
INPUT
R1
R2
+IN
Ohm
Ohm
C2
F

Step: 3


Filter Designer
Select Filter
View Diagram [Butterworth - Active Lowpass Filter]
FILTERS
Chebyshev
Butterworth
Active Lowpass Filter
Active Highpass Filter
Active Bandpass Filter
Active Bandstop Filter
Bessel
Elliptic (Cauer)
Ideal
Active Lowpass Filter
Active Highpass Filter
Active Bandpass Filter
Active Bandstop Filter
Parameter | Value
Quality range | Q <= 5
Output frequency [Hz] | 1k
Calculate parameters as | Ideal values
Component Parameters | Value
C1 [capacitance(F)] | 10n
C2 [capacitance(F)] | .857871n
R1 [resistance(Ohm)] | 54.3387k
R2 [resistance(Ohm)] | 54.3387k
Append options | Options
Filter circuit file | FLT_ALP_Q5.EPC
Append mode | Append filter project to current page
Merge input/output
Calculated Component parameters
Cancel
Calculate Parameters
Import Diagram
C:\EDWINXP\JOB\$FILTER.RLT, 1, 0, 1, 1000, 0, 0, 0, 0, 0
Click to Calculate the values of the filter components

Step: 4

EDWINXP

ELECTRONIC DESIGN FOR WINDOWS

SIMULATION EXAMPLES

PROMOTED AND DISTRIBUTED BY:

Ambition Technologies
206, 2nd Floor, Deepak Plaza, DC Chowk, Sector-9
Rohini, New delhi-110085
Phone: 011-32041225, 45733636
Email: support@ambitiontech.com
Website: www.ambitiontech.com

LIST OF EXPERIMENTS

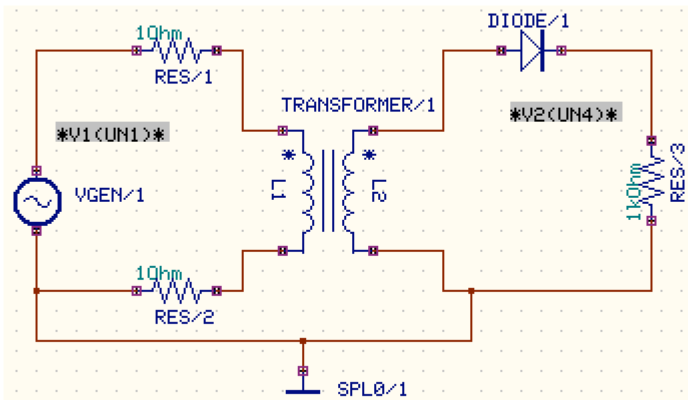
1. Half wave rectifier circuit
2. Full wave rectifier circuit
3. Integrator circuit using op-amp
4. Differentiator circuit using op-amp
5. Inverting amplifier circuit using op-amp
6. Non- Inverting amplifier circuit using op-amp
7. summing amplifier circuit using op-amp
8. Differential amplifier circuit using op-amp
9. Negative clamper circuit
10. Negative clamper with the given ref voltage circuit
11. Positive clamper circuit
12. Negative clipper circuit
13. Double-sided clipper circuit
14. Series voltage regulator circuit
15. Shunt voltage regulator circuit
16. Astable multivibrator using 555 timer circuit

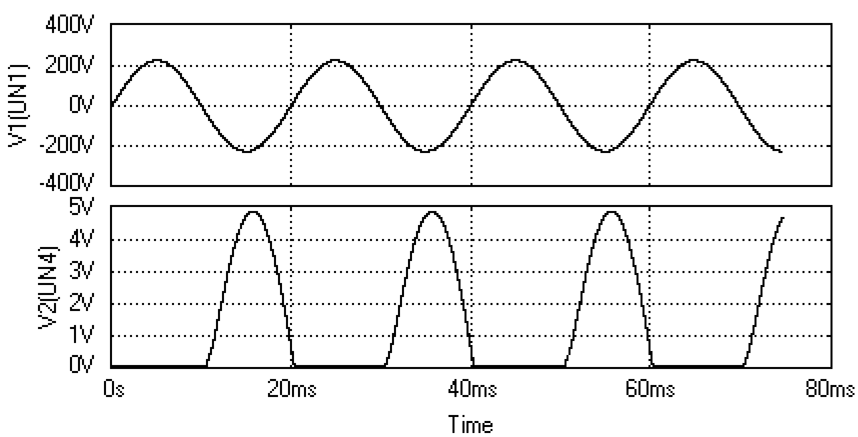
Half Wave Rectifer Circuit

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
RES	RES	Resistor	3
DIODE	DIODE	Diode – 1N4001	1
VGEN	VGEN	Voltage Generator	1
TRANSFORMER	TRANSFORMER	Transformer	1
GND	SPLO	Ground	1

Circuit and the Output Waveform:





EDWinXP->
Preference ->Mixed Mode Simulator: The circuit is preprocessed first and by selecting the transient analysis options, the output of the circuit is displayed in the waveform viewer. Note that to view the waveform output first set the waveform markers wherever required. This is can done by selecting the Set Waveform Content from the Instrument Option. Please see the fig1.bmp.

In the circuit the transformer is used to achieve the isolation between the input and output circuits and also used as the step down transformer to achieve the 5V output from the 230V input with changing the frequency.

Circuit: A Rectifier is an electrical device that converts alternating current to direct current, a process known as rectification. Rectifiers have many uses including as components of power supplies and as detector of radio signals. Rectifiers may be made of solid state diodes, vacuum tube diodes, mercury arc valves, and other components.

A device, which performs the opposite function, is known as an inverter.

When only one diode is used to rectify AC, the difference between the term diode and the term rectifier is merely one of usage, i.e., the term rectifier describes a diode that is being used to convert AC to DC. Almost all rectifiers comprise a number of diodes in a specific arrangement for more efficiently converting AC to DC than is possible with only one diode. Before the development of silicon semiconductor rectifiers, vacuum tube diodes and copper (I) oxide or selenium rectifier stacks were used.

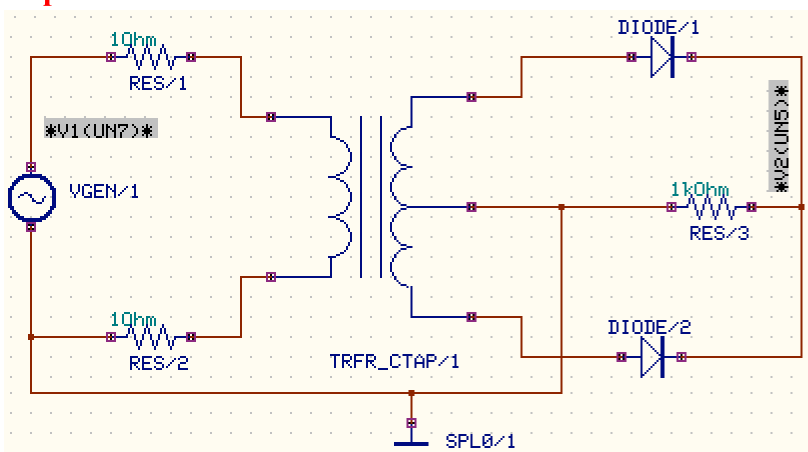
Early radio receivers, called crystal radios, used a “cat’s whisker” of the fine wire pressing on a crystal of galena (lead sulfide) to serve as a point-contact rectifier or “Crystal detector”. In gas heating systems flame rectification can be used to detect a flame. Two metal electrodes in the outer layers of the flame provide a current path and rectification of an applied alternating voltage, but only while the flame is present.

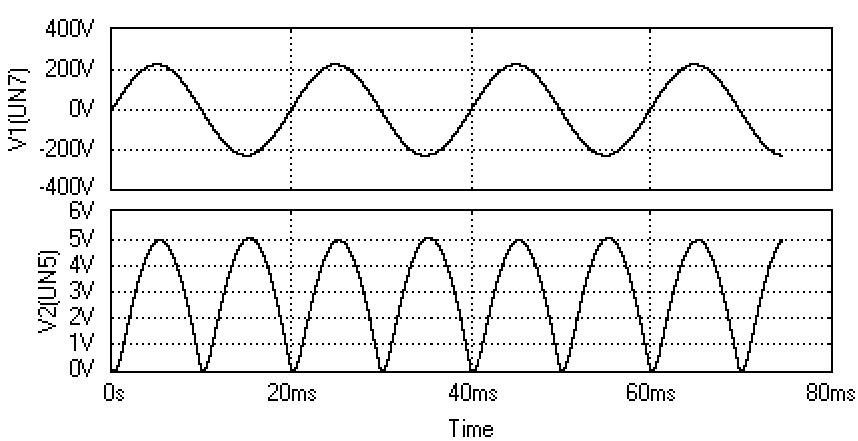
Full Wave Rectifier Circuit

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
RES	RES	Resistor	3
DIODE	DIODE	Diode – 1N4001	2
VGEN	VGEN	Voltage Generator	1
TRFR/CATP	TRFR/CATP	Center Tapped Transformer	1
GND	SPLO	Ground	1

Circuit and the Output Waveform:





EDWinXP->

Preference ->Mixed Mode Simulator: The circuit is preprocessed first and by selecting the transient analysis options, the output of the circuit is displayed in the waveform viewer. Note that to view the waveform output first set the waveform markers wherever required. This is can done by selecting the Set Waveform Content from the Instrument Option. Please see the fig1.bmp.

In the circuit the transformer is used to achieve the isolation between the input and output circuits and also used as the step down transformer to achieve the 5V output from the 230V input with changing the frequency.

Circuit: A Rectifier is an electrical device that converts alternating current to direct current, a process known as rectification. Rectifiers have many uses including as components of power supplies and as detector of radio signals. Rectifiers may be made of solid state diodes, vacuum tube diodes, mercury arc valves, and other components.

A device, which performs the opposite function, is known as an inverter.

When only one diode is used to rectify AC, the difference between the term diode and the term rectifier is merely one of usage, i.e., the term rectifier describes a diode that is being used to convert AC to DC. Almost all rectifiers comprise a number of diodes in a specific arrangement for more efficiently converting AC to DC than is possible with only one diode. Before the development of silicon semiconductor rectifiers, vacuum tube diodes and copper (I) oxide or selenium rectifier stacks were used.

Early radio receivers, called crystal radios, used a “cat’s whisker” of the fine wire pressing on a crystal of galena (lead sulfide) to serve as a point-contact rectifier or “Crystal detector”. In gas heating systems flame rectification can be used to detect a flame. Two metal electrodes in the outer layers of the flame provide a current path and rectification of an applied alternating voltage, but only while the flame is present.

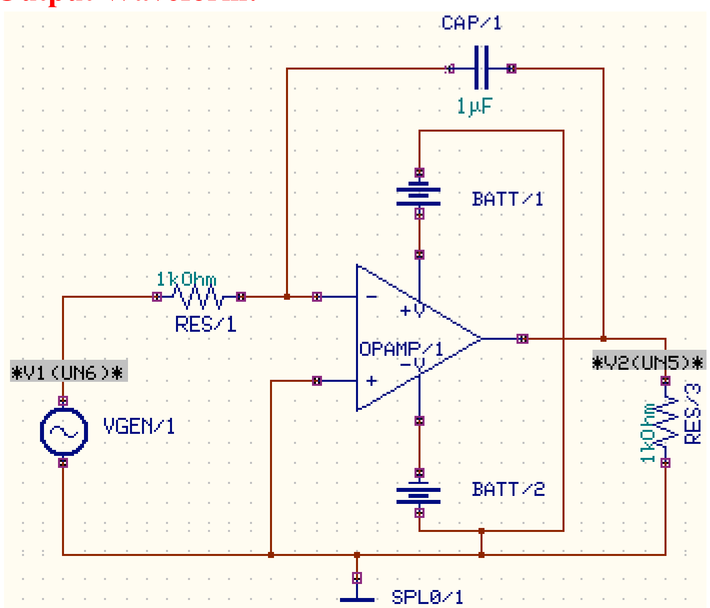
The above figure shows the full wave rectifier circuit. The operation if the circuit is very simple. The transformer used here is a center-tapped transformer to provide the isolation between the input circuit and output circuit and also the step down the input voltage from 230V to 5V with out changing the frequency of the signal source. During the positive half cycle of the input signal the Diode D1 conducts and during the negative half cycle of the input signal the diode D2 conducts, thus achieving the pulsating DC output. Note that, to reduce the ripples in the output just connect the filter circuit, (parallel connection of the capacitor in the output circuit) to the output circuit.

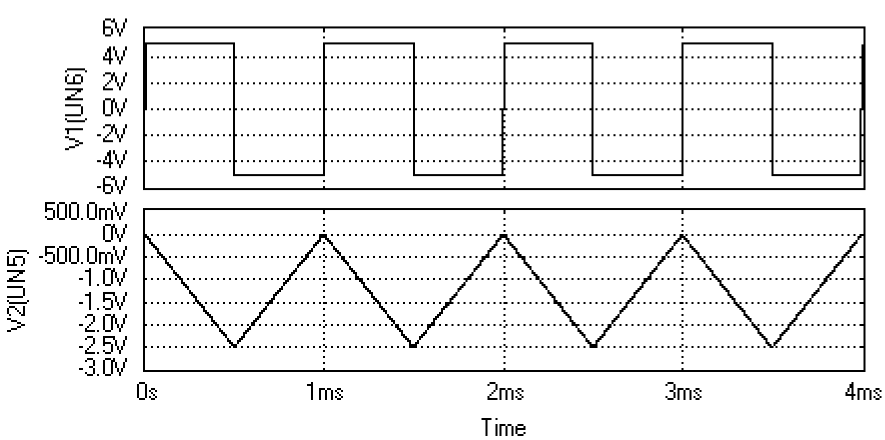
Integrator Circuit Using Op-amp

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
Opamp	Opamp	Operational Amplifier	1
RES	RES	Resistors	2
VGEN	VGEN	Voltage Generator	1
BATT	BATT	Battery (DC Supply)	2
CAP	CAP	Capacitor	1
GND	SPLO	Ground	1

Circuit and the Output Waveform:





EDWinXP->
Preference ->Mixed Mode Simulator: The circuit is preprocessed first and by selecting the transient analysis from the analysis options the output of the circuit is displayed in the waveform viewer. Note that to view the waveform output first set the waveform markers wherever required. This is can be done by selecting the Set Waveform Content from the Instrument Option. Please see the fig1.bmp.

$$V_{out} = \int_{0}^{t} -\frac{V_{in}}{RC}\,dt + V_{initials}$$

Operational Amplifier:

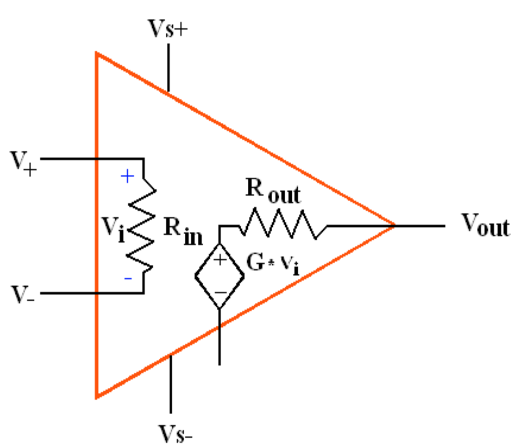
An operational Amplifier, often called an op-amp, is a DC-coupled high-gain electronic voltage amplifier with differential inputs ad usually a single output. Typically the output of the op-amp is controlled either by negative feedback, which largely determines the magnitude of its output voltage gain, or by positive feedback, which facilitates regenerative gain and oscillation. High input impedance at the input terminals and low output impedance are important typical characteristics.

The Op-amp is one type of differential amplifier. Other types of differential amplifier include the,

- Fully differential amplifier (similar to the op-amp, but with 2 outputs).
- The instrumentation amplifier (usually built from 3 op-amps).
- The isolation amplifier (similar to the instrumentation amplifier, but which works fine with common-mode voltages that would destroy an ordinary op-amp).
- Negative feedback amplifier (usually built from 1 or more op-amps and a resistive feedback network).

Ideal Op-amp:

The below figure shows an example of an ideal operational amplifier. The main part in an amplifier is the dependent voltage source that increases in relation to the voltage drop across Rin, thus amplifying the voltage difference between V+ and V-. Many uses have been found for Op-amp and an ideal Op-amp seeks to characterize the physical phenomena that make Op-amps useful.



Vs+ and Vs- are not connected to the circuit within the Op-amp because they power the dependent voltage source's circuit. These are notable, however, because they determine the maximum voltage the dependent voltage source can output.

For any input voltage the ideal Op-amp has,

- Infinite open-loop gain.
- Infinite bandwidth.
- Infinite input impedance (resulting in zero input currents)
- Zero offset voltage.
- Infinite slew rate.
- Zero output impedance and
- Zero noise.

History:

1941: A DC Coupled, high gain, inverting feedback amplifier, is first found in US patent 2,401,779 " summing amplifier" filed by Karl D.Swartzel Jr, of Bell Labs in 1941. This design used three vacuum tubes to achieve a gain of 90dB and operated on voltage rails of (+ or -) 350 volts.

1947: in 1947, the Opamp was first formally defined and named in a paper by Prof.John R.Ragazzini of Columbia University. This Op-amp designed by Loebe Julie, was superior in a variety of ways. It had two major innovations. Its inout stage used a ling-tailed triode pair with loads matched to reduce drift in the output and far more importantly, it was the first p-amp design to have two inputs (inverting and non-inverting).

1961: 1961 were producing solid state, discrete Op-amps. The P45 had a gain of 94dB and a ran on (+ or -) 15V.

1962: First Op-amp in potted modules.

1963: First monolithic IC Op-amp.

1986: Release of the µA741-would be seen as a nearly ubiquitous chip.

1966: First Varactor bridge Op-amps.

1970: First High-speed, low input current FET design.

1972: Single sided supply Op-amps being produced.

Applications:

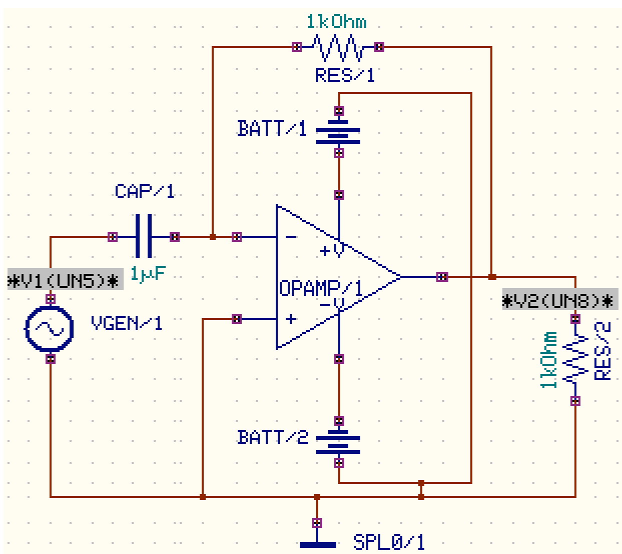
¤ Audio-and vide-frequency pre-amplifiers and buffers
¤ Voltage comparators
¤ Differential amplifiers
¤ Differentiators and integrators
¤ Filters
¤ Precision rectifiers
¤ Precision peak detectors
¤ Voltage and current regulators
¤ Analog calculators
¤ Analog-Digital converters
¤ Digital-Analog converters
¤ Voltages clamps
¤ Oscillations and waveform generators.

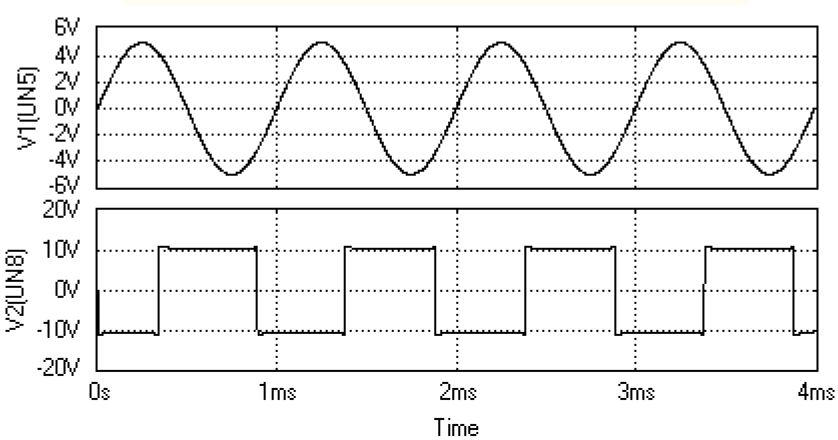
Differentiator Circuit Using Op-amp

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
Opamp	Opamp	Operational Amplifier	1
RES	RES	Resistors	2
VGEN	VGEN	Voltage Generator	1
BATT	BATT	Battery (DC Supply)	2
CAP	CAP	Capacitor	1
GND	SPLO	Ground	1

Circuit and the Output Waveform:





EDWinXP->
Preference ->Mixed Mode Simulator: The circuit is preprocessed first and by selecting the transient analysis from the analysis options the output of the circuit is displayed in the waveform viewer. Note that to view the waveform output first set the waveform markers wherever required. This is can be done by selecting the Set Waveform Content from the Instrument Option. Please see the fig1.bmp.

$$V_{out} = -RC\left(\frac{dV_{in}}{dt}\right)$$

Operational Amplifier:

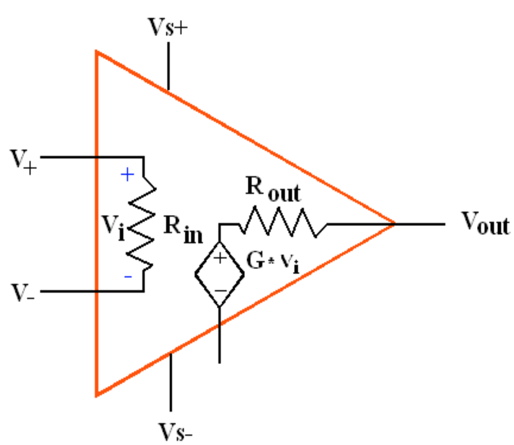
An operational Amplifier, often called an op-amp, is a DC-coupled high-gain electronic voltage amplifier with differential inputs ad usually a single output. Typically the output of the op-amp is controlled either by negative feedback, which largely determines the magnitude of its output voltage gain, or by positive feedback, which facilitates regenerative gain and oscillation. High input impedance at the input terminals and low output impedance are important typical characteristics.

The Op-amp is one type of differential amplifier. Other types of differential amplifier include the,

- Fully differential amplifier (similar to the op-amp, but with 2 outputs).
- The instrumentation amplifier (usually built from 3 op-amps).
- The isolation amplifier (similar to the instrumentation amplifier, but which works fine with common-mode voltages that would destroy an ordinary op-amp).
- Negative feedback amplifier (usually built from 1 or more op-amps and a resistive feedback network).

Ideal Op-amp:

The below figure shows an example of an ideal operational amplifier. The main part in an amplifier is the dependent voltage source that increases in relation to the voltage drop across Rin, thus amplifying the voltage difference between V+ and V-. Many uses have been found for Op-amp and an ideal Op-amp seeks to characterize the physical phenomena that make Op-amps useful.



Vs+ and Vs- are not connected to the circuit within the Op-amp because they power the dependent voltage source's circuit. These are notable, however, because they determine the maximum voltage the dependent voltage source can output.

For any input voltage the ideal Op-amp has,

- Infinite open-loop gain.
- Infinite bandwidth.
- Infinite input impedance (resulting in zero input currents)
- Zero offset voltage.
- Infinite slew rate.
- Zero output impedance and
- Zero noise.

History:

1941: A DC Coupled, high gain, inverting feedback amplifier, is first found in US patent 2,401,779 " summing amplifier" filed by Karl D.Swartzel Jr, of Bell Labs in 1941. This design used three vacuum tubes to achieve a gain of 90dB and operated on voltage rails of (+ or -) 350 volts.

1947: in 1947, the Opamp was first formally defined and named in a paper by Prof.John R.Ragazzini of Columbia University. This Op-amp designed by Loebe Julie, was superior in a variety of ways. It had two major innovations. Its inout stage used a ling-tailed triode pair with loads matched to reduce drift in the output and far more importantly, it was the first p-amp design to have two inputs (inverting and non-inverting).

1961: 1961 were producing solid state, discrete Op-amps. The P45 had a gain of 94dB and a ran on (+ or -) 15V.

1962: First Op-amp in potted modules.

1963: First monolithic IC Op-amp.

1986: Release of the μA741-would be seen as a nearly ubiquitous chip.

1966: First Varactor bridge Op-amps.

1970: First High-speed, low input current FET design.

1972: Single sided supply Op-amps being produced.

Applications:

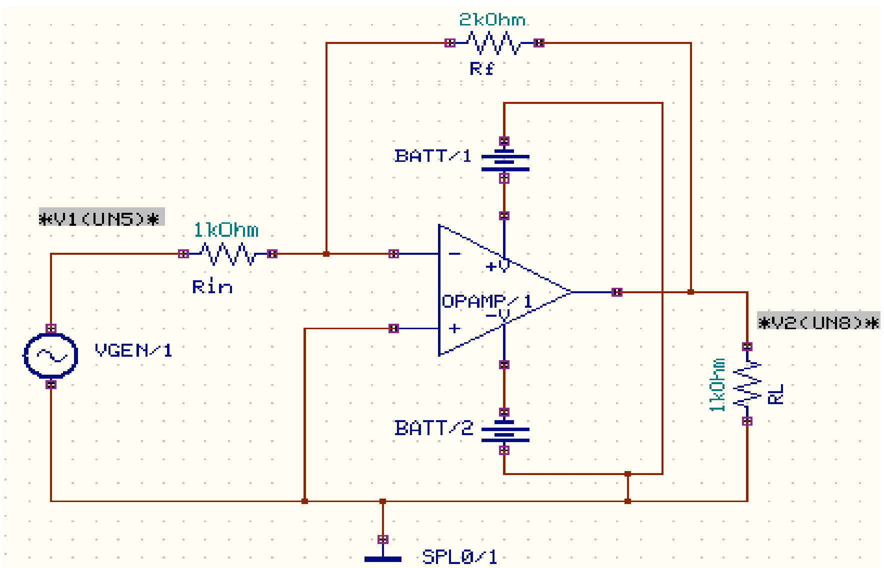
¤ Audio-and vide-frequency pre-amplifiers and buffers
¤ Voltage comparators
¤ Differential amplifiers
¤ Differentiators and integrators
¤ Filters
¤ Precision rectifiers
¤ Precision peak detectors
¤ Voltage and current regulators
¤ Analog calculators
¤ Analog-Digital converters
¤ Digital-Analog converters
¤ Voltages clamps
¤ Oscillations and waveform generators.

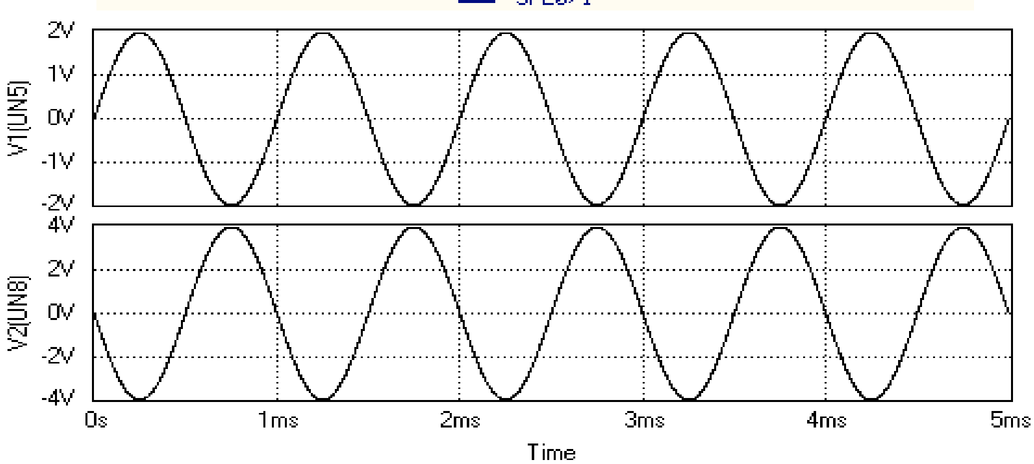
Inverting Amplifier Circuit Using Op-amp

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
Opamp	Opamp	Operational Amplifier	1
RES	RES	Resistors	5
VGEN	VGEN	Voltage Generator	2
BATT	BATT	Battery (DC Supply)	2
GND	SPLO	Ground	1

Circuit and the Output Waveform:





EDWinXP->
Preference ->Mixed Mode Simulator: The circuit is preprocessed first and by selecting the transient analysis from the analysis options the output of the circuit is displayed in the waveform viewer. Note that to view the waveform output first set the waveform markers wherever required. This is can be done by selecting the Set Waveform Content from the Instrument Option. Please see the fig1.bmp.

$$V_{out} = -V_{in}\left(\frac{R_f}{R_{in}}\right)$$

Operational Amplifier:

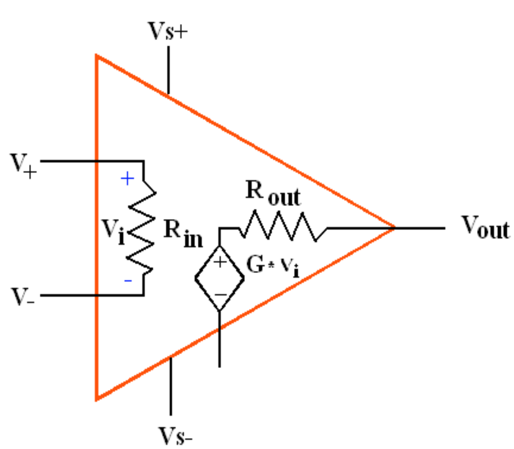
An operational Amplifier, often called an op-amp, is a DC-coupled high-gain electronic voltage amplifier with differential inputs ad usually a single output. Typically the output of the op-amp is controlled either by negative feedback, which largely determines the magnitude of its output voltage gain, or by positive feedback, which facilitates regenerative gain and oscillation. High input impedance at the input terminals and low output impedance are important typical characteristics.

The Op-amp is one type of differential amplifier. Other types of differential amplifier include the,

- Fully differential amplifier (similar to the op-amp, but with 2 outputs).
- The instrumentation amplifier (usually built from 3 op-amps).
- The isolation amplifier (similar to the instrumentation amplifier, but which works fine with common-mode voltages that would destroy an ordinary op-amp).
- Negative feedback amplifier (usually built from 1 or more op-amps and a resistive feedback network).

Ideal Op-amp:

The below figure shows an example of an ideal operational amplifier. The main part in an amplifier is the dependent voltage source that increases in relation to the voltage drop across Rin, thus amplifying the voltage difference between V+ and V-. Many uses have been found for Op-amp and an ideal Op-amp seeks to characterize the physical phenomena that make Op-amps useful.



Vs+ and Vs- are not connected to the circuit within the Op-amp because they power the dependent voltage source's circuit. These are notable, however, because they determine the maximum voltage the dependent voltage source can output.

For any input voltage the ideal Op-amp has,

- Infinite open-loop gain.
- Infinite bandwidth.
- Infinite input impedance (resulting in zero input currents)
- Zero offset voltage.
- Infinite slew rate.
- Zero output impedance and
- Zero noise.

History:

1941: A DC Coupled, high gain, inverting feedback amplifier, is first found in US patent 2,401,779 " summing amplifier" filed by Karl D.Swartzel Jr, of Bell Labs in 1941. This design used three vacuum tubes to achieve a gain of 90dB and operated on voltage rails of (+ or -) 350 volts.

1947: in 1947, the Opamp was first formally defined and named in a paper by Prof.John R.Ragazzini of Columbia University. This Op-amp designed by Loebe Julie, was superior in a variety of ways. It had two major innovations. Its inout stage used a ling-tailed triode pair with loads matched to reduce drift in the output and far more importantly, it was the first p-amp design to have two inputs (inverting and non-inverting).

1961: 1961 were producing solid state, discrete Op-amps. The P45 had a gain of 94dB and a ran on (+ or -) 15V.

1962: First Op-amp in potted modules.

1963: First monolithic IC Op-amp.

1986: Release of the μA741-would be seen as a nearly ubiquitous chip.

1966: First Varactor bridge Op-amps.

1970: First High-speed, low input current FET design.

1972: Single sided supply Op-amps being produced.

Applications:

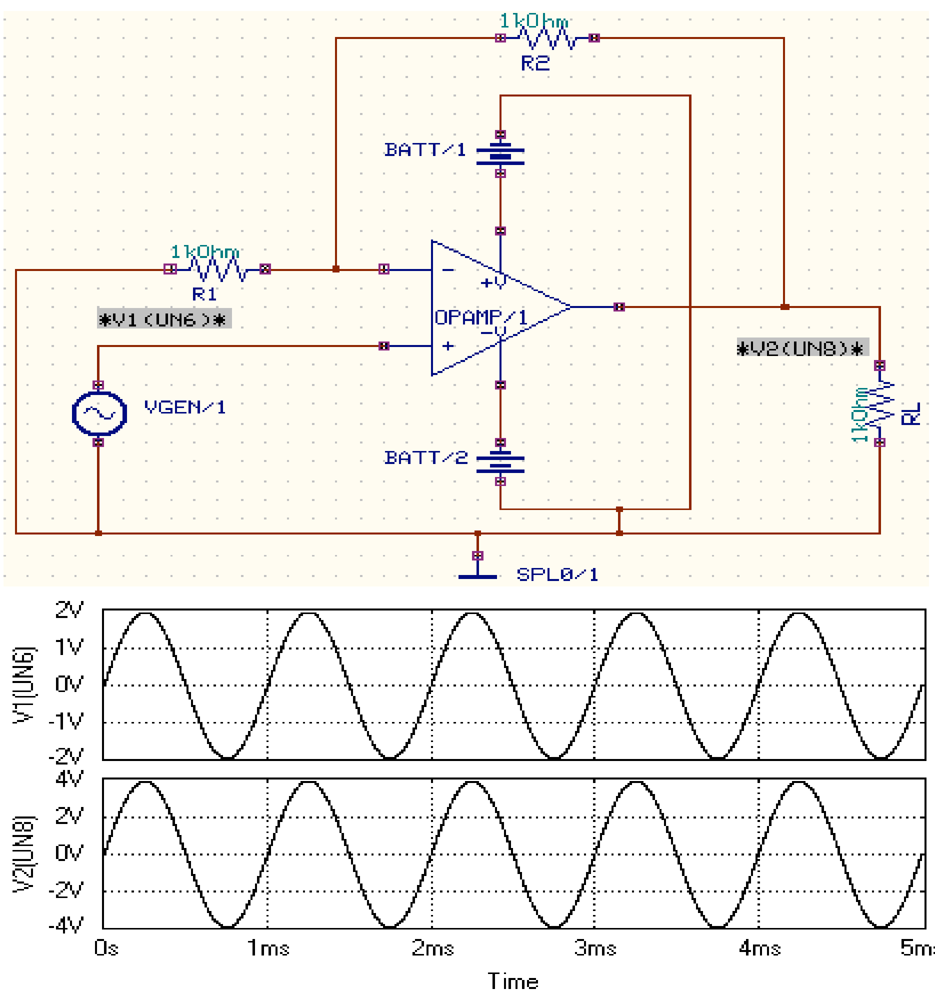
- Audio-and vide-frequency pre-amplifiers and buffers
- Voltage comparators
- Differential amplifiers
- Differentiators and integrators
- Filters
- Precision rectifiers
- Precision peak detectors
- Voltage and current regulators
- Analog calculators
- Analog-Digital converters
- Digital-Analog converters
- Voltages clamps
- Oscillations and waveform generators.

Non-Inverting Amplifier Circuit Using Op-amp

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
Opamp	Opamp	Operational Amplifier	1
RES	RES	Resistors	3
VGEN	VGEN	Voltage Generator	1
BATT	BATT	Battery (DC Supply)	2
GND	SPLO	Ground	1

Circuit and the Output Waveform:



EDWinXP->
Preference ->Mixed Mode Simulator: The circuit is preprocessed first and by selecting the transient analysis from the analysis options the output of the circuit is displayed in the waveform viewer. Note that to view the waveform output first set the waveform markers wherever required. This is can be done by selecting the Set Waveform Content from the Instrument Option. Please see the fig1.bmp.

$$V_{out} = V_{in}\left(1 + \frac{R_2}{R_1}\right)$$

Operational Amplifier:

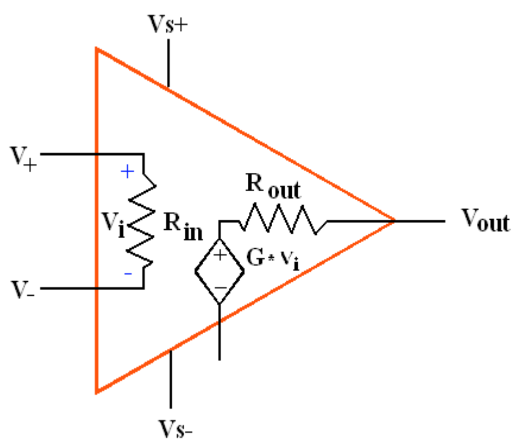
An operational Amplifier, often called an op-amp, is a DC-coupled high-gain electronic voltage amplifier with differential inputs ad usually a single output. Typically the output of the op-amp is controlled either by negative feedback, which largely determines the magnitude of its output voltage gain, or by positive feedback, which facilitates regenerative gain and oscillation. High input impedance at the input terminals and low output impedance are important typical characteristics.

The Op-amp is one type of differential amplifier. Other types of differential amplifier include the,

- Fully differential amplifier (similar to the op-amp, but with 2 outputs).
- The instrumentation amplifier (usually built from 3 op-amps).
- The isolation amplifier (similar to the instrumentation amplifier, but which works fine with common-mode voltages that would destroy an ordinary op-amp).
- Negative feedback amplifier (usually built from 1 or more op-amps and a resistive feedback network).

Ideal Op-amp:

The below figure shows an example of an ideal operational amplifier. The main part in an amplifier is the dependent voltage source that increases in relation to the voltage drop across Rin, thus amplifying the voltage difference between V+ and V-. Many uses have been found for Op-amp and an ideal Op-amp seeks to characterize the physical phenomena that make Op-amps useful.



Vs+ and Vs- are not connected to the circuit within the Op-amp because they power the dependent voltage source's circuit. These are notable, however, because they determine the maximum voltage the dependent voltage source can output.

For any input voltage the ideal Op-amp has,

- Infinite open-loop gain.
- Infinite bandwidth.
- Infinite input impedance (resulting in zero input currents)
- Zero offset voltage.
- Infinite slew rate.
- Zero output impedance and
- Zero noise.

History:

1941: A DC Coupled, high gain, inverting feedback amplifier, is first found in US patent 2,401,779 " summing amplifier" filed by Karl D.Swartzel Jr, of Bell Labs in 1941. This design used three vacuum tubes to achieve a gain of 90dB and operated on voltage rails of (+ or -) 350 volts.

1947: in 1947, the Opamp was first formally defined and named in a paper by Prof.John R.Ragazzini of Columbia University. This Op-amp designed by Loebe Julie, was superior in a variety of ways. It had two major innovations. Its inout stage used a ling-tailed triode pair with loads matched to reduce drift in the output and far more importantly, it was the first p-amp design to have two inputs (inverting and non-inverting).
1961: 1961 were producing solid state, discrete Op-amps. The P45 had a gain of 94dB and a ran on (+ or -) 15V.
1962: First Op-amp in potted modules.
1963: First monolithic IC Op-amp.
1986: Release of the µA741-would be seen as a nearly ubiquitous chip.
1966: First Varactor bridge Op-amps.
1970: First High-speed, low input current FET design.
1972: Single sided supply Op-amps being produced.

Applications:

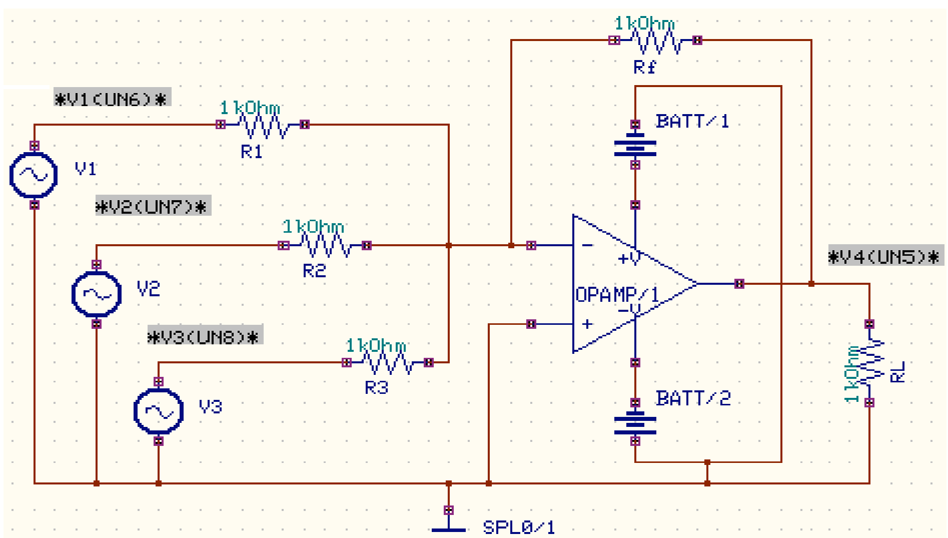
¤ Audio-and vide-frequency pre-amplifiers and buffers
¤ Voltage comparators
¤ Differential amplifiers
¤ Differentiators and integrators
¤ Filters
¤ Precision rectifiers
¤ Precision peak detectors
¤ Voltage and current regulators
¤ Analog calculators
¤ Analog-Digital converters
¤ Digital-Analog converters
¤ Voltages clamps
¤ Oscillations and waveform generators.

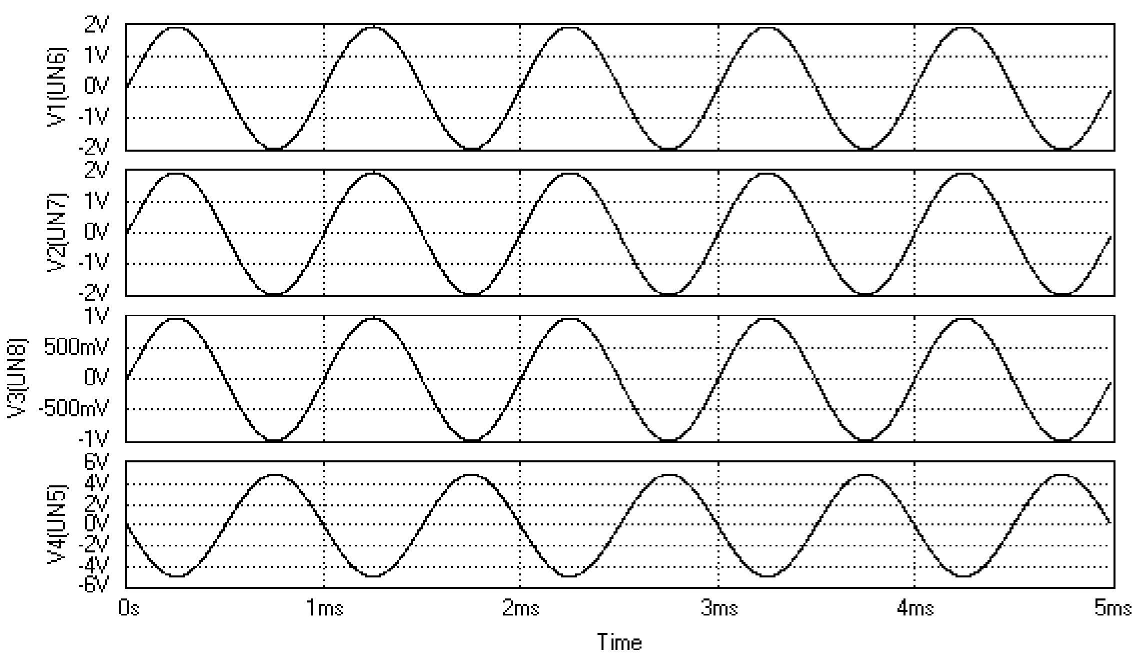
Summing Amplifier Circuit Using Op-amp

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
Opamp	Opamp	Operational Amplifier	1
RES	RES	Resistors	5
VGEN	VGEN	Voltage Generator	2
BATT	BATT	Battery (DC Supply)	2
GND	SPLO	Ground	1

Circuit and the Output Waveform:





EDWinXP->
Preference ->Mixed Mode Simulator: The circuit is preprocessed first and by selecting the transient analysis from the analysis options the output of the circuit is displayed in the waveform viewer. Note that to view the waveform output first set the waveform markers wherever required. This is can be done by selecting the Set Waveform Content from the Instrument Option. Please see the fig1.bmp.

$$V_{out} = -R_f\left(\frac{V_1}{R_1} + \frac{V_2}{R_2} + \frac{V_3}{R_3}\right)$$

When $R_1 = R_2 = R_3$ and Rf independent then,

$$V_{out} = -\frac{R_f}{R_1}\left(V_1 + V_2 + V_3 \right)$$

When $R_1 = R_2 = R_3 = Rf$, then

$$V_{out} = -\left(V_1 + V_2 + V_3 \right)$$

Operational Amplifier:

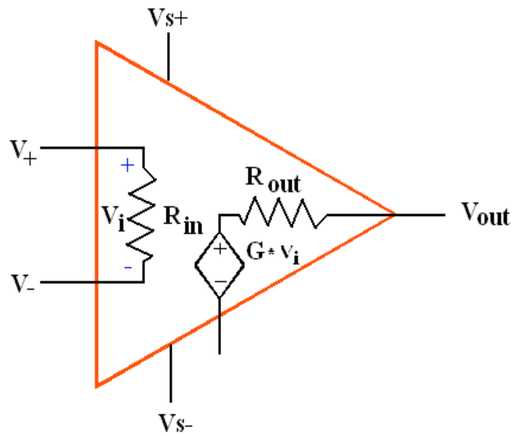
An operational Amplifier, often called an op-amp, is a DC-coupled high-gain electronic voltage amplifier with differential inputs ad usually a single output. Typically the output of the op-amp is controlled either by negative feedback, which largely determines the magnitude of its output voltage gain, or by positive feedback, which facilitates regenerative gain and oscillation. High input impedance at the input terminals and low output impedance are important typical characteristics.

The Op-amp is one type of differential amplifier. Other types of differential amplifier include the,

- Fully differential amplifier (similar to the op-amp, but with 2 outputs).
- The instrumentation amplifier (usually built from 3 op-amps).
- The isolation amplifier (similar to the instrumentation amplifier, but which works fine with common-mode voltages that would destroy an ordinary op-amp).
- Negative feedback amplifier (usually built from 1 or more op-amps and a resistive feedback network).

Ideal Op-amp:

The below figure shows an example of an ideal operational amplifier. The main part in an amplifier is the dependent voltage source that increases in relation to the voltage drop across Rin, thus amplifying the voltage difference between V+ and V-. Many uses have been found for Op-amp and an ideal Op-amp seeks to characterize the physical phenomena that make Op-amps useful.



Vs+ and Vs- are not connected to the circuit within the Op-amp because they power the dependent voltage source's circuit. These are notable, however, because they determine the maximum voltage the dependent voltage source can output.

For any input voltage the ideal Op-amp has,

- Infinite open-loop gain.
- Infinite bandwidth.
- Infinite input impedance (resulting in zero input currents)
- Zero offset voltage.
- Infinite slew rate.
- Zero output impedance and
- Zero noise.

History:

1941: A DC Coupled, high gain, inverting feedback amplifier, is first found in US patent 2,401,779 " summing amplifier" filed by Karl D.Swartzel Jr, of Bell Labs in 1941. This design used three vacuum tubes to achieve a gain of 90dB and operated on voltage rails of (+ or -) 350 volts.

1947: in 1947, the Opamp was first formally defined and named in a paper by Prof.John R.Ragazzini of Columbia University. This Op-amp designed by Loebe Julie, was superior in a variety of ways. It had two major innovations. Its inout stage used a ling-tailed triode pair with loads matched to reduce drift in the output and far more importantly, it was the first p-amp design to have two inputs (inverting and non-inverting).
1961: 1961 were producing solid state, discrete Op-amps. The P45 had a gain of 94dB and a ran on (+ or -) 15V.
1962: First Op-amp in potted modules.
1963: First monolithic IC Op-amp.

1986: Release of the µA741-would be seen as a nearly ubiquitous chip.
1966: First Varactor bridge Op-amps.
1970: First High-speed, low input current FET design.
1972: Single sided supply Op-amps being produced.

Applications:

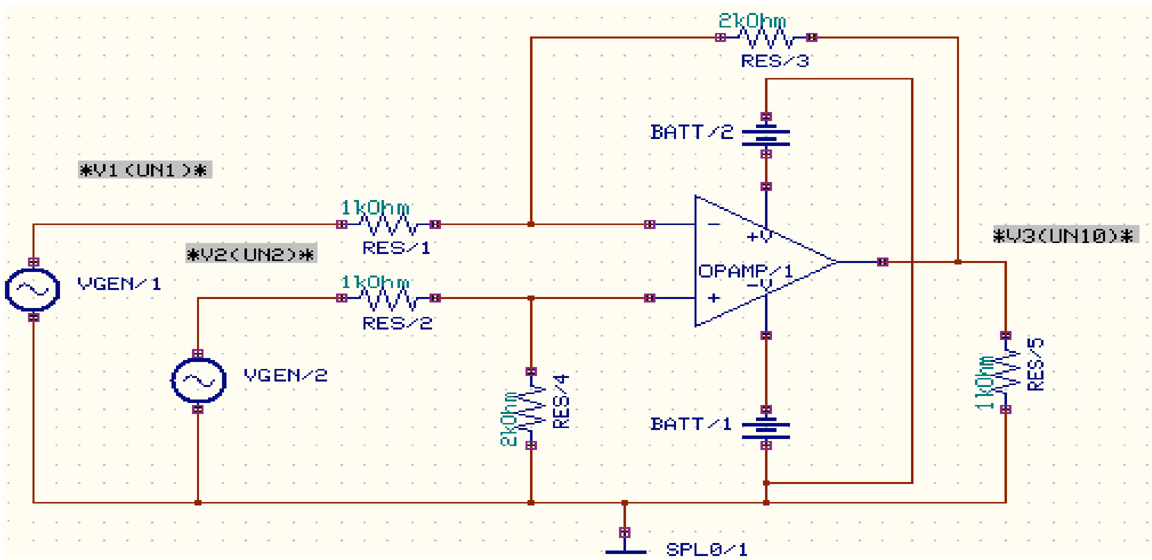
- ¤ Audio-and vide-frequency pre-amplifiers and buffers
- ¤ Voltage comparators
- ¤ Differential amplifiers
- ¤ Differentiators and integrators
- ¤ Filters
- ¤ Precision rectifiers
- ¤ Precision peak detectors
- ¤ Voltage and current regulators
- ¤ Analog calculators
- ¤ Analog-Digital converters
- ¤ Digital-Analog converters
- ¤ Voltages clamps
- ¤ Oscillations and waveform generators.

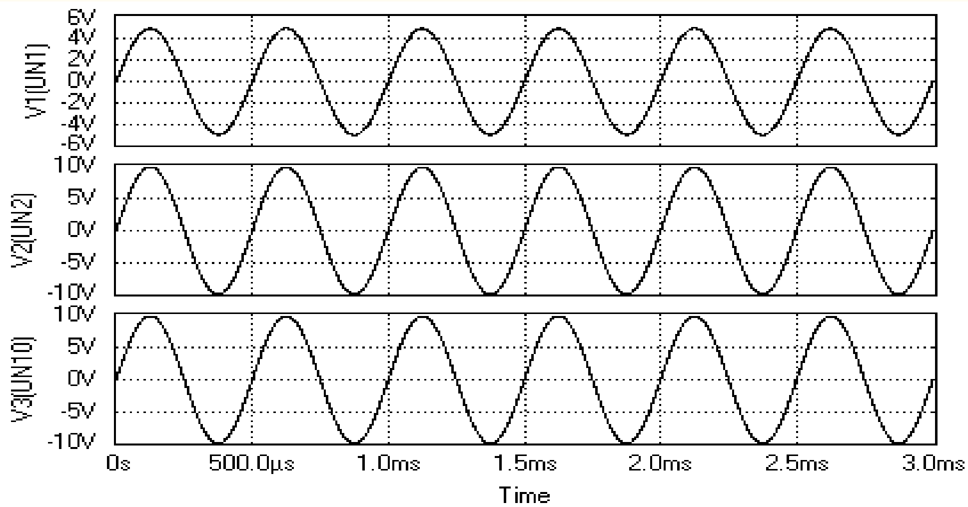
Differential Amplifier Circuit Using Op-amp

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
Opamp	Opamp	Operational Amplifier	1
RES	RES	Resistors	5
VGEN	VGEN	Voltage Generator	2
BATT	BATT	Battery (DC Supply)	2
GND	SPLO	Ground	1

Circuit and the Output Waveform:





EDWinXP->
Preference ->Mixed Mode Simulator: The circuit is preprocessed first and by selecting the transient analysis from the analysis options the output of the circuit is displayed in the waveform viewer. Note that to view the waveform output first set the waveform markers wherever required. This is can be done by selecting the Set Waveform Content from the Instrument Option. Please see the fig1.bmp.

$$V_{out} = V_2\left(\frac{(R_f + R_1)\,R_g}{(R_g + R_2)\,R_1}\right) - V_1\left(\frac{R_f}{R_1}\right)$$

Whenever $R_1 = R_2$ and $R_f = R_g$,

$V_{out} = A\,(V_2\text{-}V_1)$ and $A = R_f/R_g$.

Operational Amplifier:

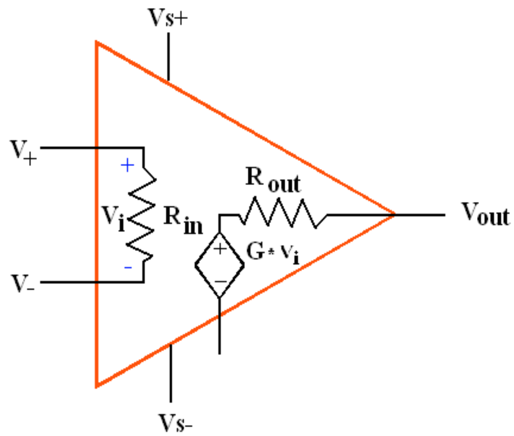
An operational Amplifier, often called an op-amp, is a DC-coupled high-gain electronic voltage amplifier with differential inputs ad usually a single output. Typically the output of the op-amp is controlled either by negative feedback, which largely determines the magnitude of its output voltage gain, or by positive feedback, which facilitates regenerative gain and oscillation. High input impedance at the input terminals and low output impedance are important typical characteristics.

The Op-amp is one type of differential amplifier. Other types of differential amplifier include the,

- Fully differential amplifier (similar to the op-amp, but with 2 outputs).
- The instrumentation amplifier (usually built from 3 op-amps).
- The isolation amplifier (similar to the instrumentation amplifier, but which works fine with common-mode voltages that would destroy an ordinary op-amp).
- Negative feedback amplifier (usually built from 1 or more op-amps and a resistive feedback network).

Ideal Op-amp:

The below figure shows an example of an ideal operational amplifier. The main part in an amplifier is the dependent voltage source that increases in relation to the voltage drop across Rin, thus amplifying the voltage difference between V+ and V-. Many uses have been found for Op-amp and an ideal Op-amp seeks to characterize the physical phenomena that make Op-amps useful.



Vs+ and Vs- are not connected to the circuit within the Op-amp because they power the dependent voltage source's circuit. These are notable, however, because they determine the maximum voltage the dependent voltage source can output.

For any input voltage the ideal Op-amp has,

- Infinite open-loop gain.
- Infinite bandwidth.
- Infinite input impedance (resulting in zero input currents)
- Zero offset voltage.
- Infinite slew rate.
- Zero output impedance and
- Zero noise.

History:

1941: A DC Coupled, high gain, inverting feedback amplifier, is first found in US patent 2,401,779 " summing amplifier" filed by Karl D.Swartzel Jr, of Bell Labs in 1941. This design used three vacuum tubes to achieve a gain of 90dB and operated on voltage rails of (+ or -) 350 volts.

1947: in 1947, the Opamp was first formally defined and named in a paper by Prof.John R.Ragazzini of Columbia University. This Op-amp designed by Loebe Julie, was superior in a variety of ways. It had two major innovations. Its inout stage used a ling-tailed triode pair with loads matched to reduce drift in the output and far more importantly, it was the first p-amp design to have two inputs (inverting and non-inverting).

1961: 1961 were producing solid state, discrete Op-amps. The P45 had a gain of 94dB and a ran on (+ or -) 15V.
1962: First Op-amp in potted modules.

1963: First monolithic IC Op-amp.
1986: Release of the μA741-would be seen as a nearly ubiquitous chip.
1966: First Varactor bridge Op-amps.
1970: First High-speed, low input current FET design.
1972: Single sided supply Op-amps being produced.

Applications:

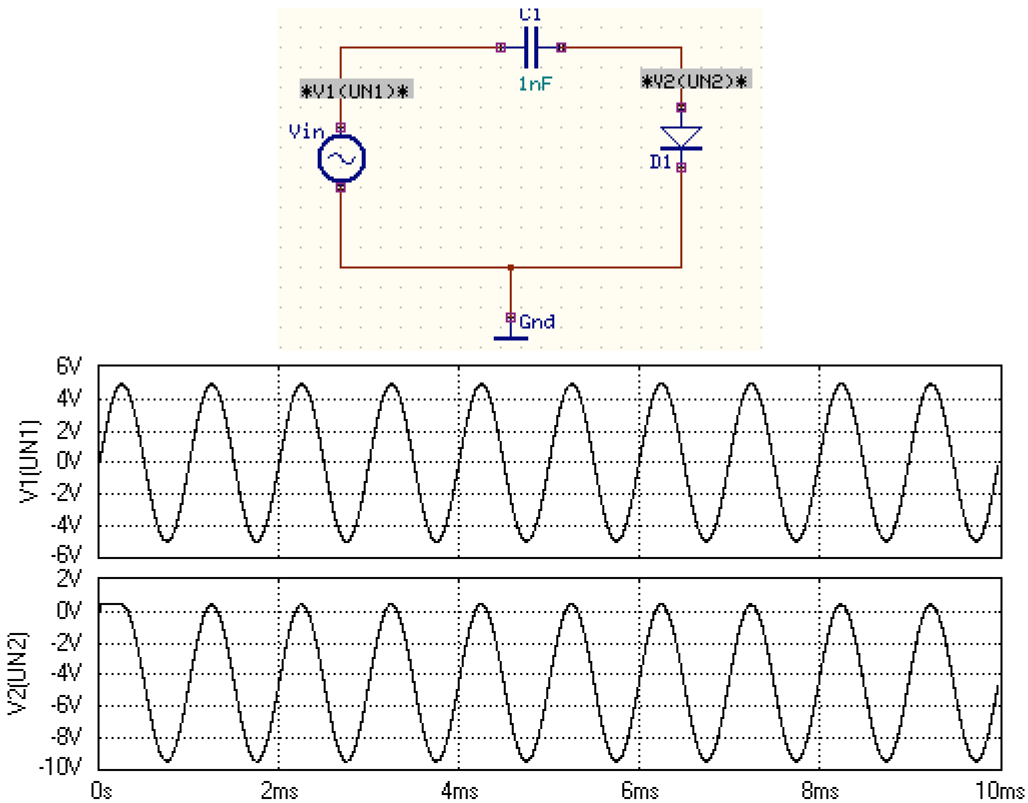
- ¤ Audio-and vide-frequency pre-amplifiers and buffers
- ¤ Voltage comparators
- ¤ Differential amplifiers
- ¤ Differentiators and integrators
- ¤ Filters
- ¤ Precision rectifiers
- ¤ Precision peak detectors
- ¤ Voltage and current regulators
- ¤ Analog calculators
- ¤ Analog-Digital converters
- ¤ Digital-Analog converters
- ¤ Voltages clamps
- ¤ Oscillations and waveform generators.

Negative Clamper Circuit

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
CAP	CAP	Capacitor	1
DIODE	DIODE	Diode – 1N4001	1
VGEN	VGEN	Voltage Generator	1
GND	SPLO	Ground	1

Circuit and the Output Waveform:



EDWinXP->
Preference ->Mixed Mode Simulator: The circuit is preprocessed first and by selecting the transient analysis from the analysis options the output of the circuit is displayed in the waveform viewer. Note that to view the waveform output first set the waveform markers wherever required. This is can done by selecting the Set Waveform Content from the Instrument Option. Please see the fig1.bmp.

Certain applications in electronics require that the upper or lower extremity of a wave be fixed at a specific value. In such applications, CLAMPING-CLAMPER circuit is used. A clamping circuit clamps or restrains either upper or lower extremity of a waveform to a fixed de potential. This circuit is also known as DIRECT-CURRENT RESTORER or a BASE-LINE STABILIZER. Such circuits are used in test equipment, radar systems, electronic counter measure systems and solar systems. Depending upon the equipment, you could find the negative or positive clampers with or without bias.

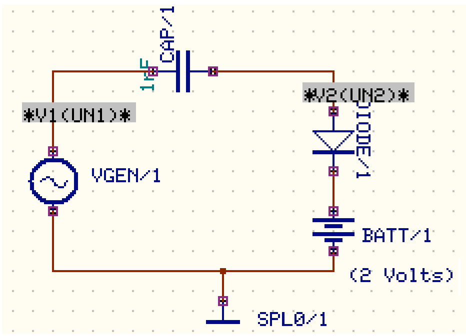
The circuit given clamps the input signal to the negative part of the signal. Note that clamping is nothing but adding the DC signal to the input signal.

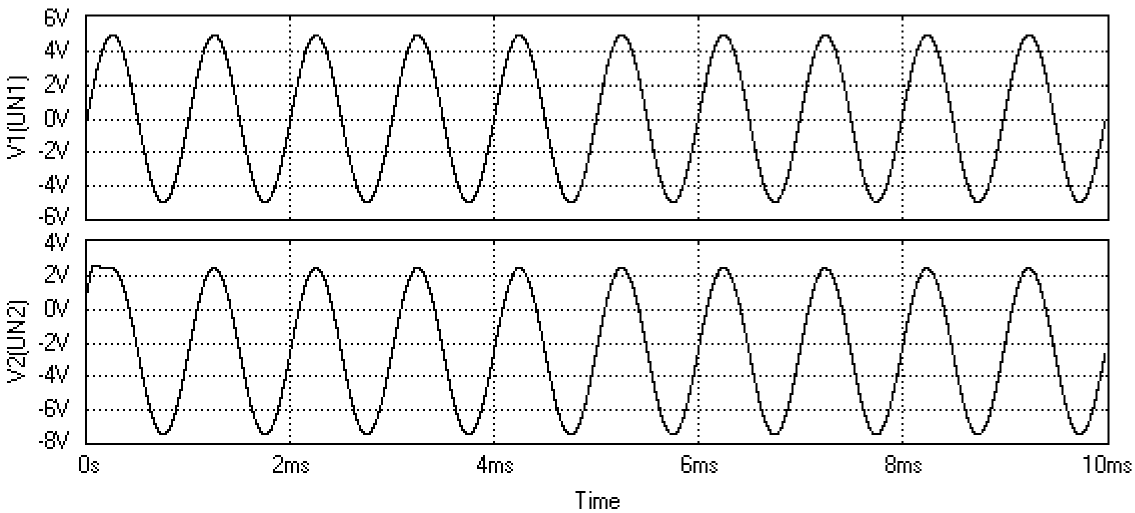
Negative Clamper with the given Ref Voltage Circuit

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
CAP	CAP	Capacitor	1
DIODE	DIODE	Diode – 1N4001	1
VGEN	VGEN	Voltage Generator	1
BATT	BATT	Battery (DC Voltage)	1
GND	SPLO	Ground	1

Circuit and the Output Waveform:





EDWinXP->

Preference ->Mixed Mode Simulator: The circuit is preprocessed first and by selecting the transient analysis from the analysis options the output of the circuit is displayed in the waveform viewer. Note that to view the waveform output first set the waveform markers wherever required. This is can done by selecting the Set Waveform Content from the Instrument Option. Please see the fig1.bmp.

Certain applications in electronics require that the upper or lower extremity of a wave be fixed at a specific value. In such applications, CLAMPING-CLAMPER circuit is used. A clamping circuit clamps or restrains either upper or lower extremity of a waveform to a fixed de potential. This circuit is also known as DIRECT-CURRENT RESTORER or a BASE-LINE STABILIZER. Such circuits are used in test equipment, radar systems, electronic counter measure systems and solar systems. Depending upon the equipment, you could find the negative or positive clampers with or without bias.

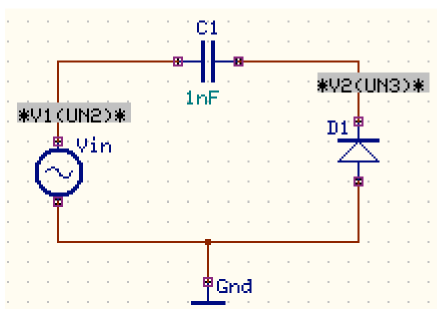
The circuit given clamps the input signal to the negative part of the signal. Here the clamping is controlled using the battery connected (BATT/1)

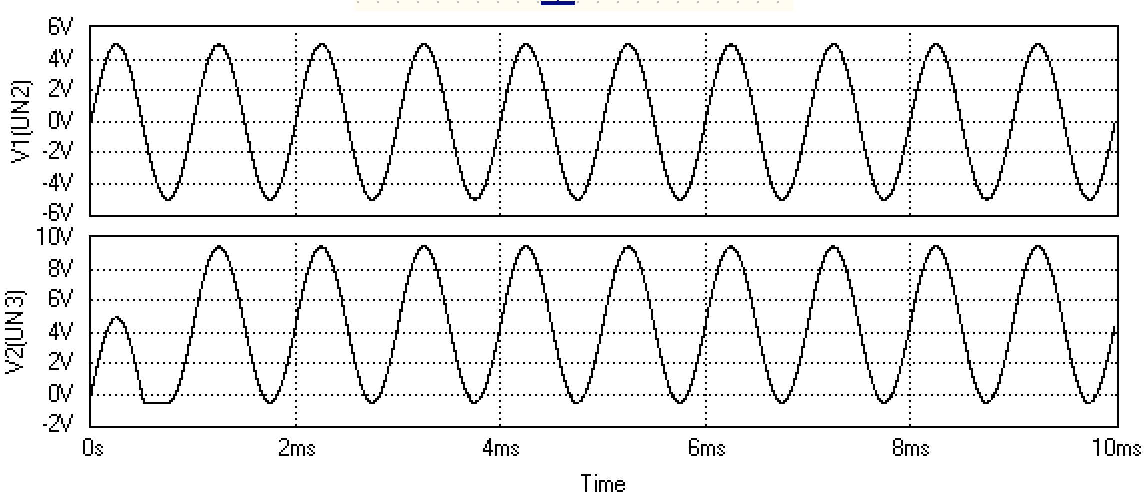
Positive Clamper Circuit

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
CAP	CAP	Capacitor	1
DIODE	DIODE	Diode – 1N4001	1
VGEN	VGEN	Voltage Generator	1
GND	SPLO	Ground	1

Circuit and the Output Waveform:





EDWinXP->

Preference ->Mixed Mode Simulator: The circuit is preprocessed first and by selecting the transient analysis from the analysis options the output of the circuit is displayed in the waveform viewer. Note that to view the waveform output first set the waveform markers wherever required. This is can done by selecting the Set Waveform Content from the Instrument Option. Please see the fig1.bmp.

Certain applications in electronics require that the upper or lower extremity of a wave be fixed at a specific value. In such applications, CLAMPING-CLAMPER circuit is used. A clamping circuit clamps or restrains either upper or lower extremity of a waveform to a fixed de potential. This circuit is also known as DIRECT-CURRENT RESTORER or a BASE-LINE STABILIZER. Such circuits are used in test equipment, radar systems, electronic counter measure systems and solar systems. Depending upon the equipment, you could find the negative or positive clampers with or without bias.

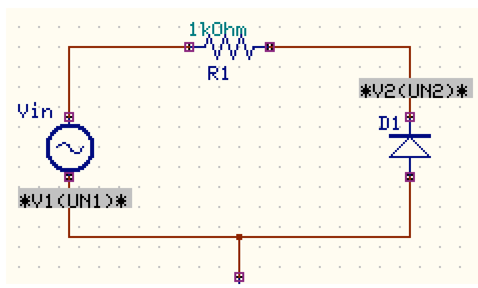
The circuit given clamps the input signal to the Positive part of the signal. Note that clamping is nothing but adding the DC signal to the input signal.

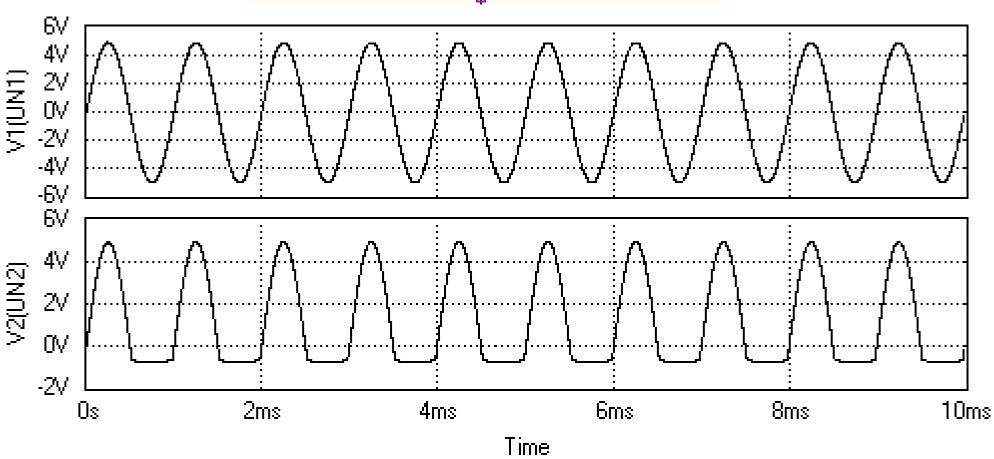
Negative Clipper Circuit

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
RES	RES	Resistor	1
DIODE	DIODE	Diode – 1N4001	1
VGEN	VGEN	Voltage Generator	1
GND	SPLO	Ground	1

Circuit and the Output Waveform:





EDWinXP->

Preference ->Mixed Mode Simulator: The circuit is preprocessed first and by selecting the transient analysis from the analysis options the output of the circuit is displayed in the waveform viewer. Note that to view the waveform output first set the waveform markers wherever required. This is can done by selecting the Set Waveform Content from the Instrument Option. Please see the fig1.bmp.

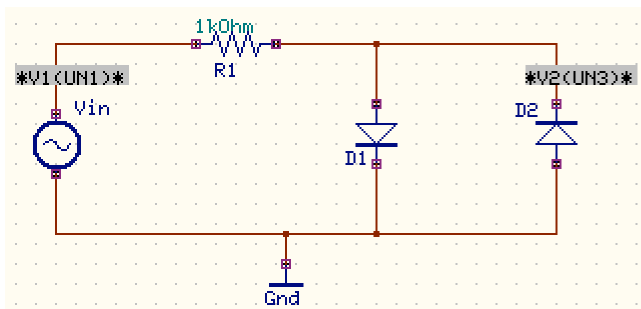
The circuit just clips the negative part of the input signal since the diode will be in the forward condition only during the positive cycle of the input signal. During the negative cycle of the input signal the diode will be in OFF State since it will be in the reverse biased condition.

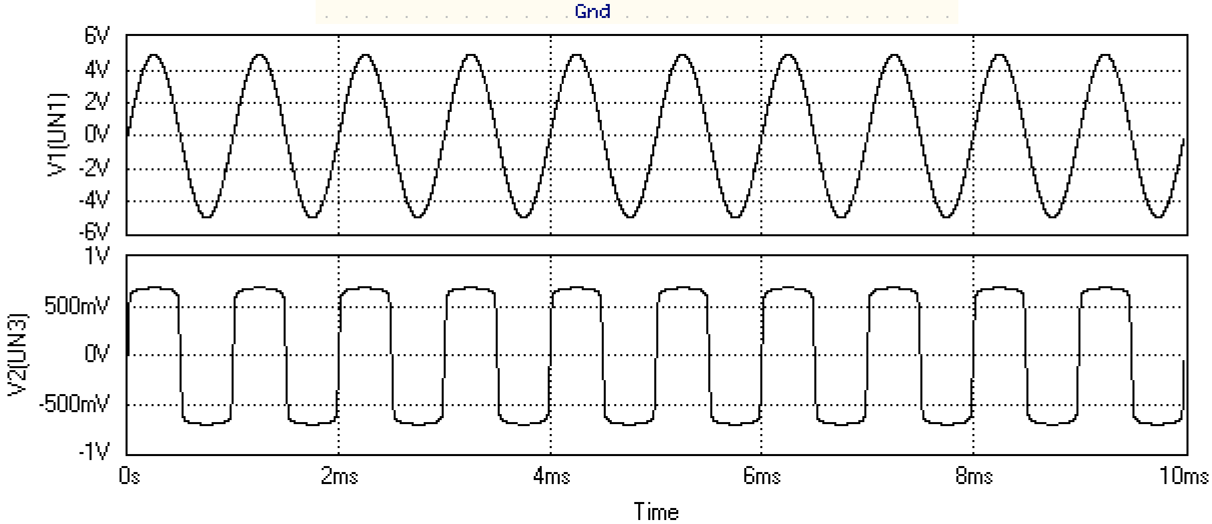
Double Sided Clipper Circuit

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
RES	RES	Resistor	1
DIODE	DIODE	Diode – 1N4001	2
VGEN	VGEN	Voltage Generator	1
GND	SPLO	Ground	1
BATT	BATT	Battery (DC Voltage)	2

Circuit and the Output Waveform:





EDWinXP->

Preference ->Mixed Mode Simulator: The circuit is preprocessed first and by selecting the transient analysis from the analysis options the output of the circuit is displayed in the waveform viewer. Note that to view the waveform output first set the waveform markers wherever required. This is can done by selecting the Set Waveform Content from the Instrument Option. Please see the fig1.bmp.

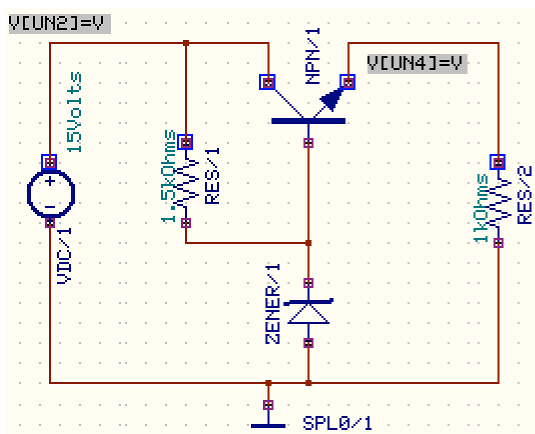
The circuit clips some part of both the cycles from the input signal since the one of the diode will be in the forward condition during both cycle of the input signal. During the Positive cycle of the input signal the Diode/1 will be in ON State and during the Negative cycle of the input signal the Diode/2 will be in ON State.

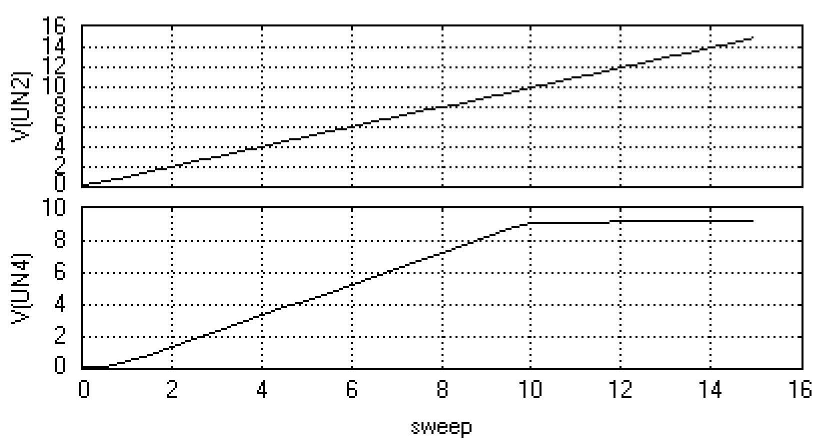
Series Voltage Regulator Circuit

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
RES	RES	Resistor	2
ZENER	ZENER	Zener Diode – 1N3016A	1
VGEN	VGEN	Voltage Generator	1
GND	SPLO	Ground	1
NPN	NPN	NPN Transistor BC 107-A	1

Circuit and the Output Waveform:





EDWinXP->
Preference ->EDSpice Simulator-> First preprocess the circuit. Make the VDC as the sweep element and vary the input voltage from 0v to 15 volts with step increase of .5v. This sweep analysis is carried out using the DC Transfer Analysis. By selecting the waveform as the output type, the result is displayed by means of waveform.

The schematic for a typical series voltage regulator is shown. Because the total load current passes through this transistor, it is sometimes called a "pass transistor." Other components, which make up the circuit, are the current limiting resistor and the Zener diode.

Recall that a Zener diode is a diode that block current until a specified voltage is applied. Remember also that the applied voltage is called the breakdown, or Zener voltage. Zener diodes are available with different Zener voltages. When the Zener voltage is reached, the Zener diode conducts from its anode to its cathode (with the direction of the arrow).

In this voltage regulator, transistor has a constant voltage applied to its base. This voltage is often called the reference voltage. As changes in the circuit output voltage occur, they are sensed at the emitter of transistor producing a corresponding change in the forward bias of the transistor. In other words, transistor compensates by increasing or decreasing its resistance in order to change the circuit voltage division.

The Zener used in this regulator is a 10-volt Zener. In this instance the Zener or breakdown voltage is 10 volts. The Zener establishes the value of the base voltage for transistor. The output voltage will equal the Zener voltage minus a 0.7-volt drop across the forward biased base-emitter junction of transistor.

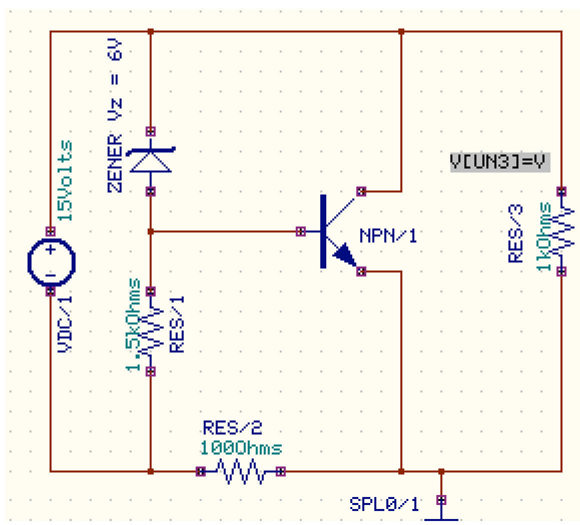
Notice the input and output voltages of 15 and 9.4 volts, respectively. The 9.4 output voltage is a momentary deviation, or variation, from the required regulated output voltage of 9.3 and is the result of a rise in the input voltage to 15 volts. Since Zener holds the base voltage of transistor at 10 volts, the forward bias of transistor changes to 0.6 volt. Because this bias voltage is less than the normal 0.7 volt, the resistance of transistor increases, thereby increasing the voltage drops across the transistor to 5.8 volts. This voltage drop restores the output voltage to 9.3 volts. The entire cycle takes only a fraction of a second and, therefore, the change is not visible on an oscilloscope or readily measurable with other standard test equipment.

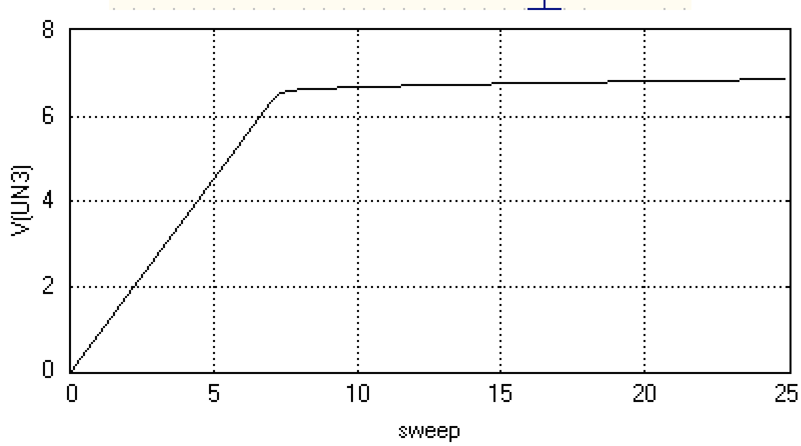
Shunt Voltage Regulator Circuit

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
RES	RES	Resistor	3
ZENER	ZENER	Zener Diode – 1N3016A	1
VGEN	VGEN	Voltage Generator	1
GND	SPLO	Ground	1
NPN	NPN	NPN Transistor BC 107-A	1

Circuit and the Output Waveform:





EDWinXP->
Preference ->EDSpice Simulator-> First preprocess the circuit. Make the VDC as the sweep element and vary the input voltage from 0v to 15 volts with step increase of .5v. This sweep

analysis is carried out using the DC Transfer Analysis. By selecting the waveform as the output type, the result is displayed by means of waveform.

The schematic shown is that of a shunt voltage regulator. Notice that transistor is in parallel with the load. Components of this circuit are identical with those of the series voltage regulator except for the addition of fixed resistor RES/2. As you study the schematic, you will see that this resistor is connected in series with the output load resistance. The current limiting resistor and Zener diode provide a constant reference voltage for the base-collector junction of transistor. Notice that the voltage drop determines the bias of transistor across RES/2 and RES/1. As you should know, the amount of forward bias across a transistor affects its total resistance. In this case, the voltage drop across R_S is the key to the total circuit operation.

The voltage drop across the Zener diode remains constant at 5.6 volts. This means that with a 15volt input voltage, the voltage drop across RES/1 is 9.4 volts. With a base-emitter voltage of 0.7 volt, the output voltage is equal to the sum of the voltages across Zener and the voltage at the base-emitter junction of transistor.

The increases the forward bias on transistor to 0.8 volt. Recall that the voltage drop across Zener remains constant at 5.6 volts. Since the output voltage is composed of the Zener voltage and the base-emitter voltage, the output voltage momentarily increases to 6.4 volts. At this time, the increase in the forward bias of transistor lowers the resistance of the transistor allowing more current to flow through it. Since this current must also pass through RES/2 there is also an increase in the voltage drop across this resistor. The voltage drop across RES/2 is now 8.8 volts and therefore the output voltage is reduced to 6.3 volts.

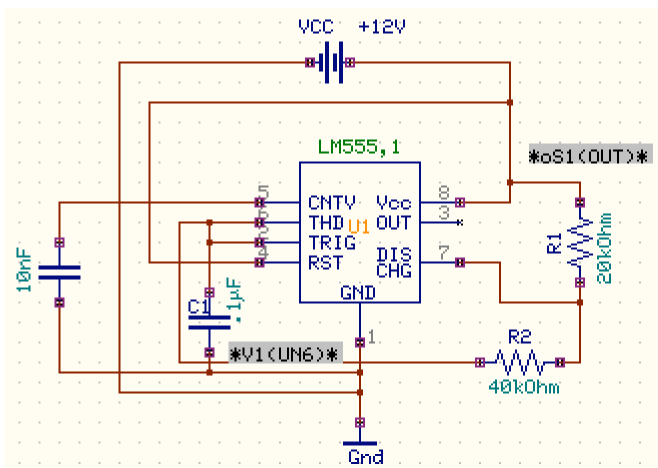
The load current has increased causing a momentary drop in voltage output to 6.2 volts. Recall that the circuit was designed to ensure a constant output voltage of 6.3 volts. Since the output voltage is less than that required, changes occur in the regulator to restore the output to 6.3 volts. Because of the 0.1 volt drop in the output voltage, the forward bias of transistor is now 0.6 volt. This decrease in the forward bias increases the resistance of the transistor, there by reducing the current flow through transistor by the same amount that the load current increased. The current flows through RES/2 returns to its normal value and restores the output voltage to 6.3 volts.

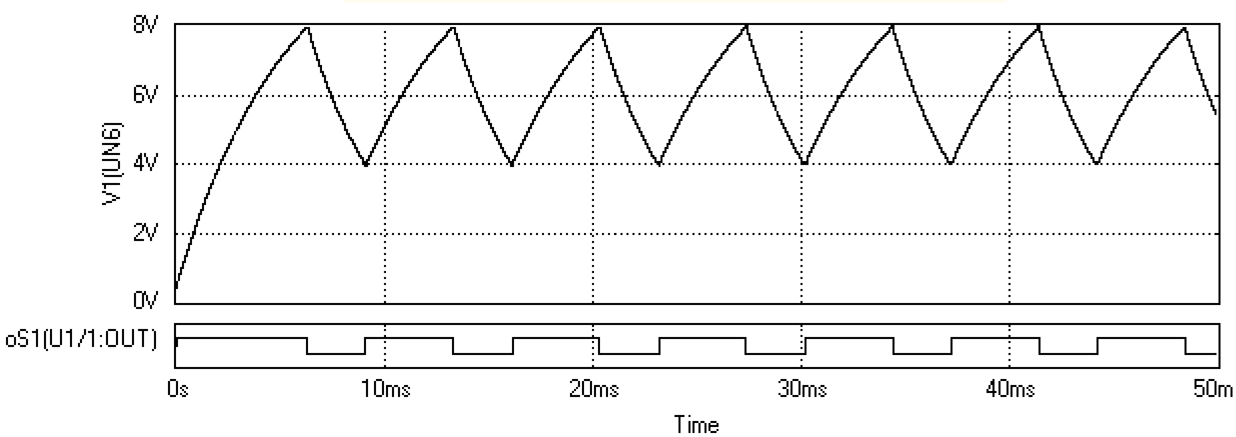
Astable Multivibrator using 555 timer- Circuit

Components Required:

NAME	EDWinXP COMPONENT	DESCRIPTION	NUMBER OF COMPONENTS REQUIRED
RES	RES	Resistor	2
555 timer	LM555	Timer	1
CAP	CAP	Capacitors	2
BATT	BATT	Battery	1
GND	SPLO	Ground	1

Circuit and the Output Waveform:





EDWinXP->

Preference ->Mixed Mode Simulator: The circuit is preprocessed first and by selecting the transient analysis from the analysis options the output of the circuit is displayed in the waveform viewer. Note that to view the waveform output first set the waveform markers wherever required. This is

can done by selecting the Set Waveform Content from the Instrument Option. Please see the fig1.bmp.
Multivibrator is an electronic circuit used to implement a variety of simple two-state systems such as oscillators, timers and flipflops. It is characterized by two amplifying devices (transistors, electron tubes or other devices) cross-coupled by resistors and capacitors. The most common form is the astable or oscillating type, which generates a square wave-the high level of harmonics in its output is what fives the Multivibrator its common name, the Multivibrator originated as a vacuum tube (Valve) circuit described by Williams Eccles and F.W.Jordhan in 1919.

There are 3 types of Multivibrator circuits:

ASTABLE: in which the circuit is not stable in either state – it continuously oscillates from one state to the other.

MONOSTABLE: in which one of the states is stable, but the other is not-the circuit will flip into the unstable state for a determined period, but will eventually return to the stable state. Such a circuit is useful for creating a timing period of fixed duration in response to some external event. This circuit is also known as **one shot**. A common application is in eliminating switch bounce.

BISTABLE: in which the circuit will remain in wither state indefinitely. The circuit can be flipped from one state to the other by an external event or trigger. Such a circuit is important as the fundamental building block of register or memory devices. This circuit is also known as flip-flop.

NOTE:
In its simplest form the multivibrator circuit consists of two cross-coupled transistors. Using resistor-capacitor networks within the circuit t define the time periods of the unstable states, the various types may be implemented. Multivibrator find applications in a variety of systems where square waves or timed intervals are required. Simple circuits tend to be inaccurate since many factors affect their timing. So they are rarely used where high precision is required.

Before the advent of low-cost integrated circuits, chains of multivibrators found use as frequency dividers. A free-running multivibrator with a frequency of one-half to one-tenth of the reference frequency would accurately lock to the reference frequency. This technique was used in early electronics organs, to keep notes of different octaves accurately in tune. Other applications included early television systems, where the various line and frame frequencies were kept synchronized by pulses included in the video signal.

Multivibrator Frequency:

The period of each half of the multivibrator is given by $t = \ln(2)\, RC$. The total period of ascillation is given by:

$$T = t1 + t2 = \ln(2)\, R_2 C_1 + \ln(2)\, R_3 C_2$$

$$f = \frac{1}{T} = \frac{1}{\ln(2)\cdot(R_2C_1 + R_3C_2)} = \frac{1}{0.693\cdot(R_2C_1 + R_3C_2)}$$

For the special case: where t1 = t2, (50% duty cycle)
R2 = R3
C1 = C2

$$f = \frac{1}{T} = \frac{1}{\ln(2)\cdot(2RC)} = \frac{0.721}{RC}$$

EDWINXP

ELECTRONIC DESIGN FOR WINDOWS

TUTORIAL ON PCB DESIGN

PROMOTED AND DISTRIBUTED BY:

Ambition Technologies
206, 2nd Floor, Deepak Plaza, DC Chowk, Sector-9
Rohini, New delhi-110085
Phone: 011-32041225, 45733636
Email: support@ambitiontech.com
Website: www.ambitiontech.com

PCB LAYOUT

To complete any Electronics project, one should follow the predefined steps as follows.

- ♠ Transfer the logic or design to the schematic editor.
- ♠ Simulate to check the design.
- ♠ Re-simulate with the variations in the simulation parameters to check the operating point and to check the sensitivity of the circuit.
- ♠ Design the PCB Layout.
- ♠ Do Board analysis and conduct signal integrity to verify the crosstalk and to find the field intensity at every point.
- ♠ Conduct Thermal analysis to find the hot spot and provide the cooling parameters at the required
- ♠ Generate the manufacturing document for the fabrication. Using the CAM tool check the generated Gerber files.

** Please note that before starting the routing the PCB design, after placement of the components it is very important to check the signal crosstalk to verify the placement of the components.

** EDWinXP Supports 32 layers of PCB design whereas 28 out of that will be for copper layers, 2 layers for silkscreen and 2 for Mask layers.

** The PCB layout in the EDWinXP has Auto Routers and Auto Placers which makes easy to the users to design the PCB layout.

** EDWinXP supports 3 types of Auto Routers, Viz:

- ✓ Standard
- ✓ Arizona
- ✓ Spectra
- ✓ Max Route(Interface)

The first and foremost thing to be done before starting of the design is to set the outline of the layout. This can be done by two ways. One is to define the outline through the *Text-window* and another is to draw using the *Create Board Tool.*

The board shape can be defined as per the users' requirement. The textual mode offers all the shapes like Polygon with the customizable side length, Circle with the radius input and rectangle with the width and height as the input parameters.

To call the components used in the design to the PCB layout, the user has to pack the component using the *PACK/UNPACK COMPONENT* option in the Schematic editor. The Packing and Unpacking of the component can be done manually choosing only the required components or the user can take use of the *Auto Packing* which packs all the components used. (The Components to be packed should have the Foot Print or the **Package Details**).

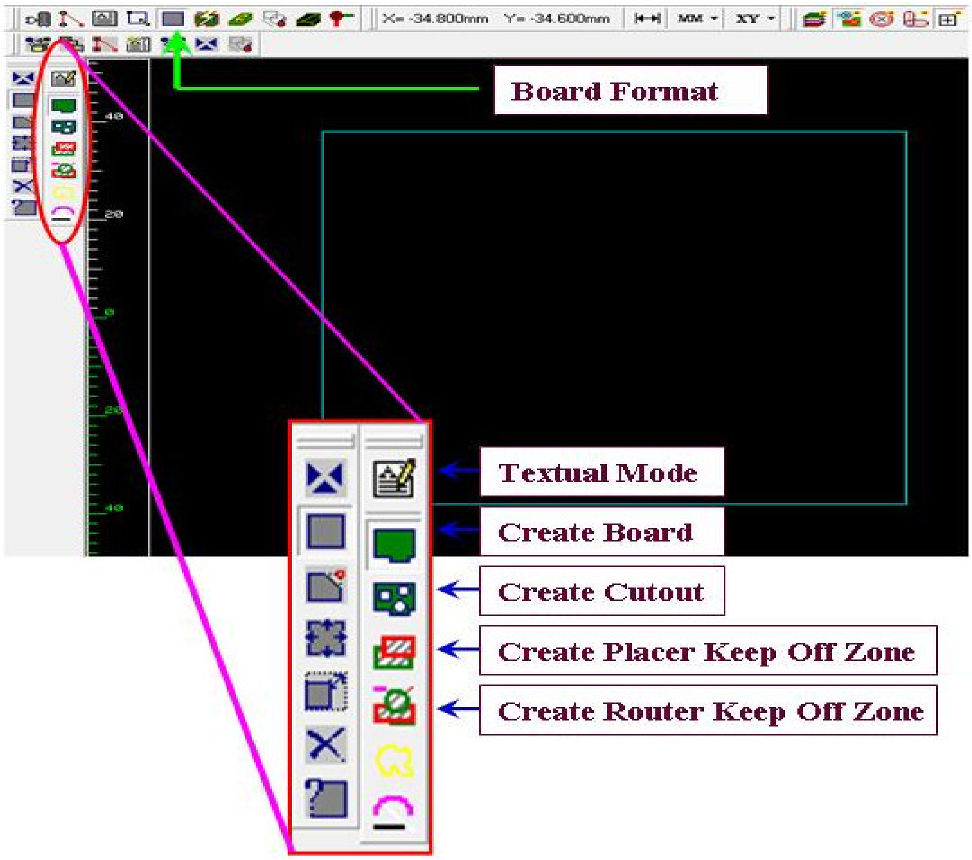


Figure: PCB Layout Board format Tool bar details

Please refer the above figure for the details of the PCB Layout format toolbar. Here the user will have to define the PCB size, have to create the Placer Keep Off Zone, Router Keep Off Zone and also the shape of the PCB Board.

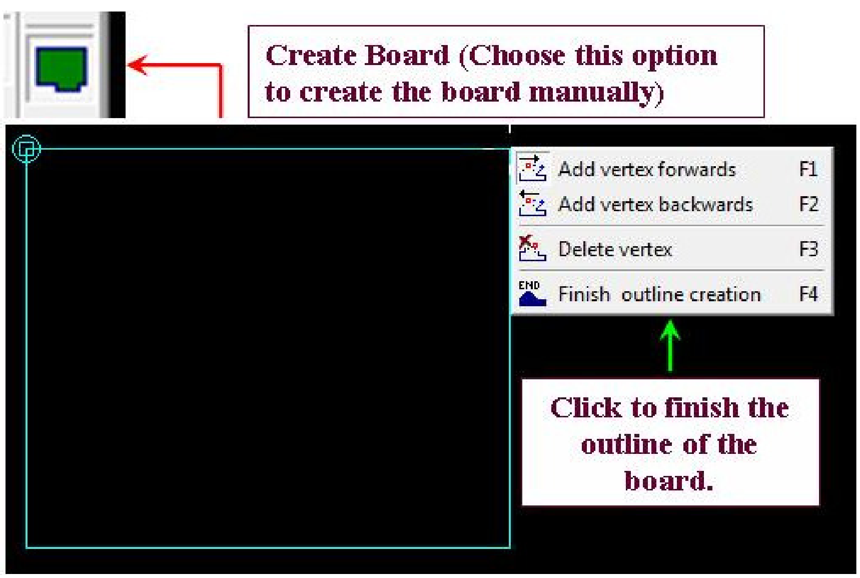


Figure: Manual drawing of the PCB Board

Placer Keep Off Zone: The user defined area in which the placement of the component is not allowed.

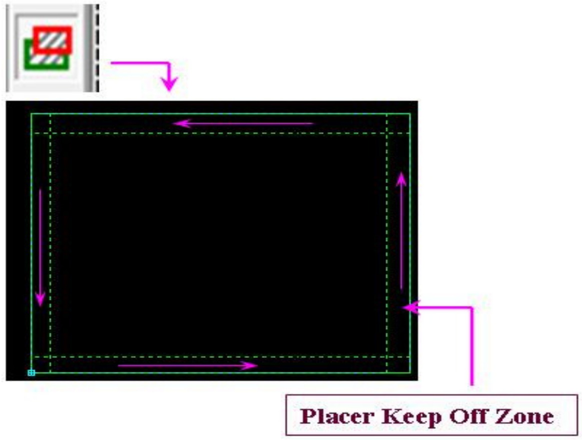


Figure: Placer Keep Off Zone

Router Keep Off Zone: The user defined area in which the routing (traces) are not allowed to draw.

Cut Out: The user can provide the Cut Outs in the PCB for placing the PCB through Components/ for some mechanical requirements. In this area the placement and the routing is not allowed.

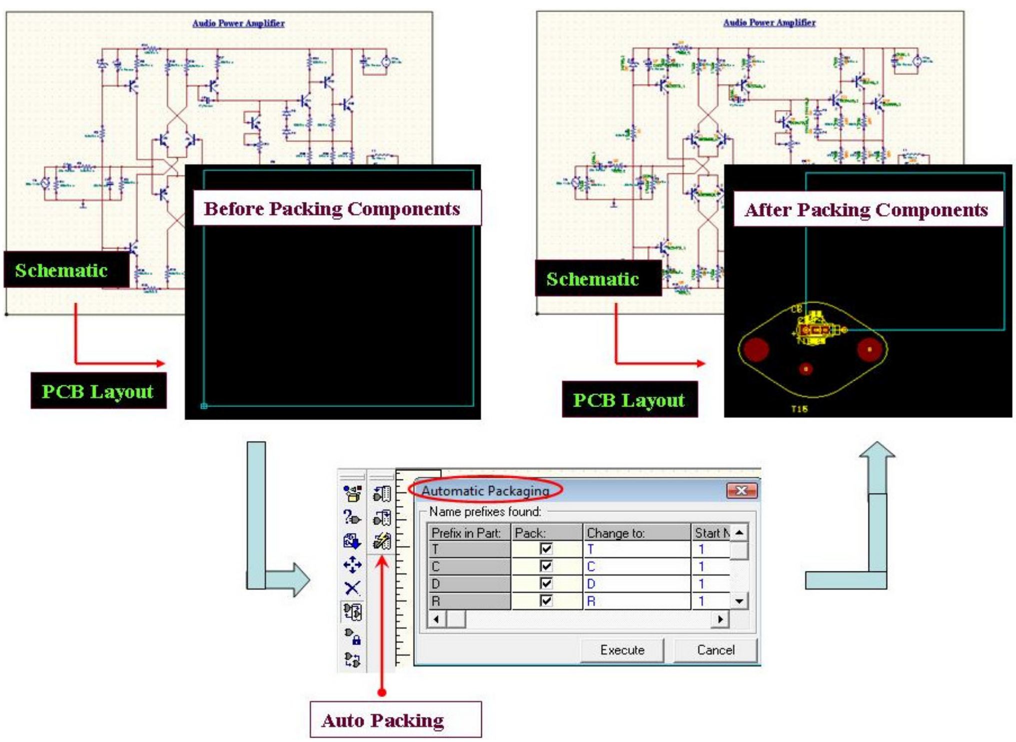


Figure: Concept of packing of the components

Once calling of the components to PCB Layout is done, the Auto placer is used to place the components automatically. Please refer the figure given below:

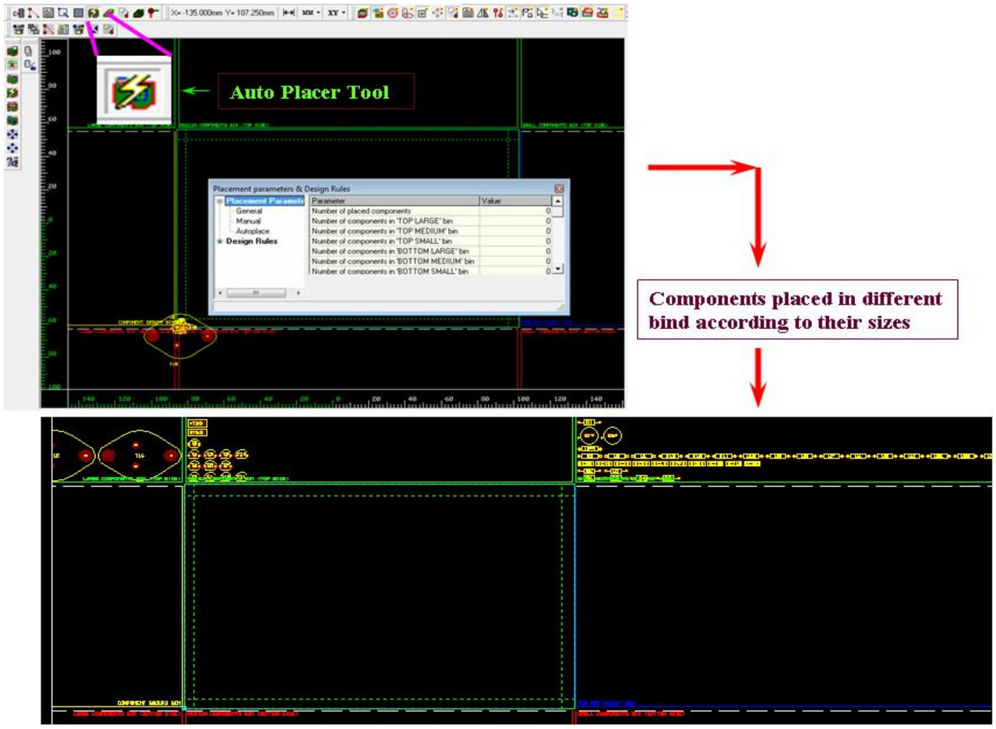


Figure: Auto placer Options

The main advantage of the Auto Placer in EDWinXP is that the components are placed in different bins according to the size and also according the layers.

Figure: Auto Placement of the Components

Once the components are placed, the user can now start the routing manually. The EDWinXP supports the Auto Routing as explained earlier. Pleaser refer the below figure for the steps to use the Auto Router

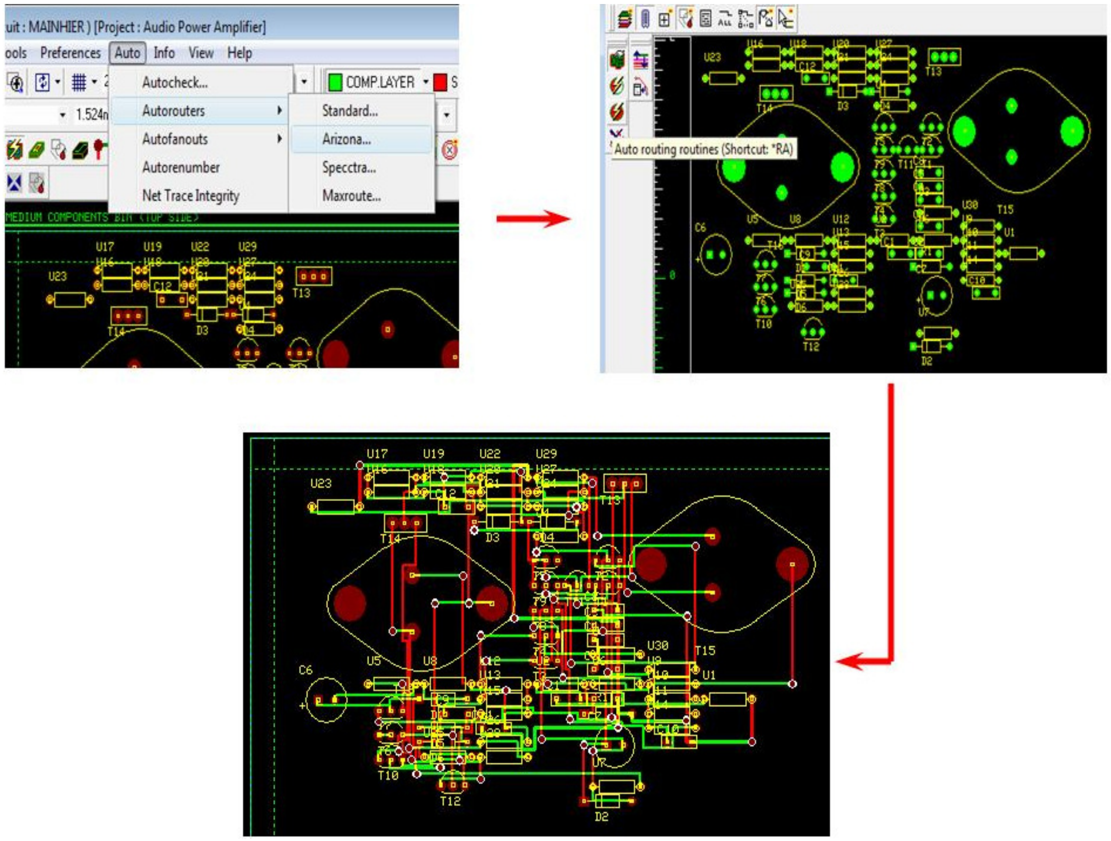


Figure: Steps - Auto Router

DRC CHECK

The next job or the step to follow after the design is to GO for ***DRC CHECK***. DRC Check means Design Rule Check. Here the user can verify his/her design with respect to the Trace to Trace distance, Trace to PAD distance, and PAD to PAD distance. This is very important to reduce the cross talk between the components.

The figure given below defines the procedure to carry on DRC Check of the PCB Layout designed.

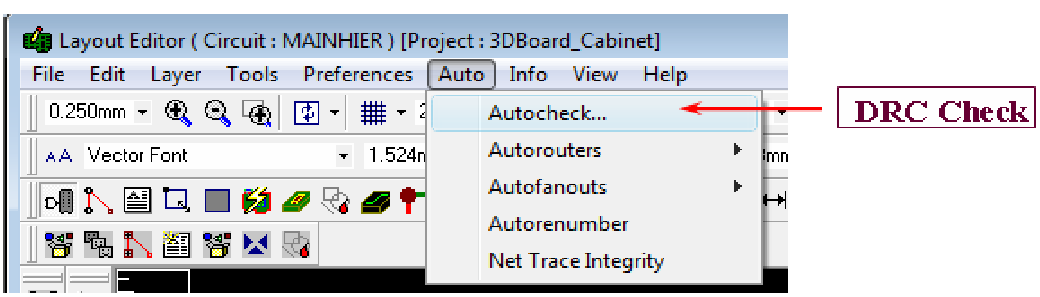


Figure: DRC Check

Select Auto option from the Mail tool bar (refer figure), and choose Auto Check Option from the tree drop down. This will invoke the window as shown in the figure.

Here the users has to define the design Rule, such as PAD to PAD distance, Trace to Trace distance and Trace to PAD distance. The above mentioned same rules can be verified for layers individually.

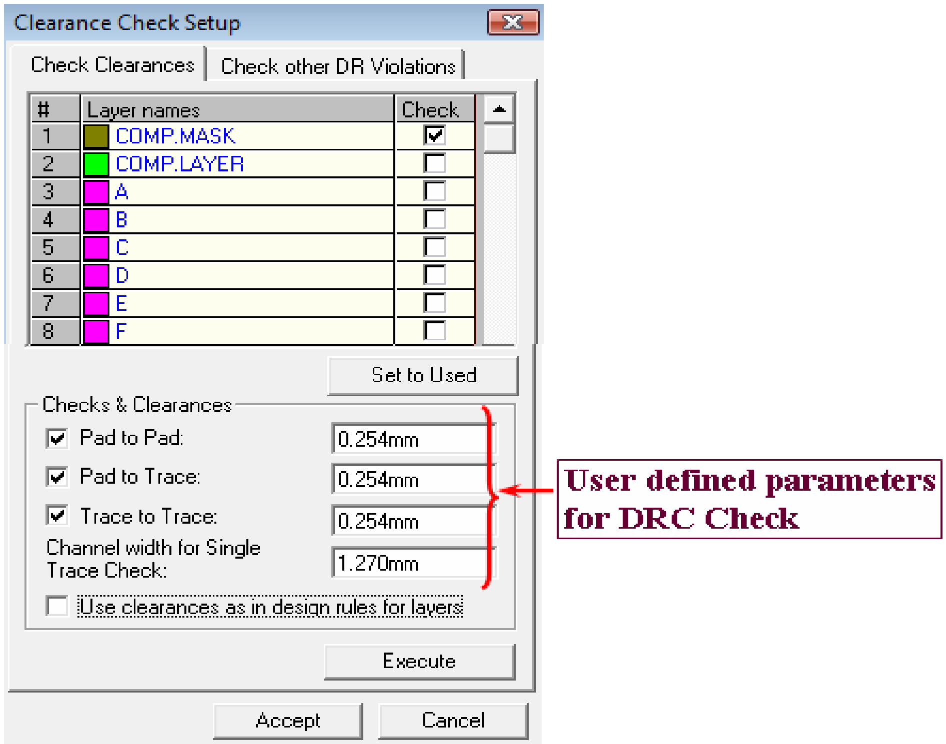


Figure: Clearance Check Setup for DRC Check.

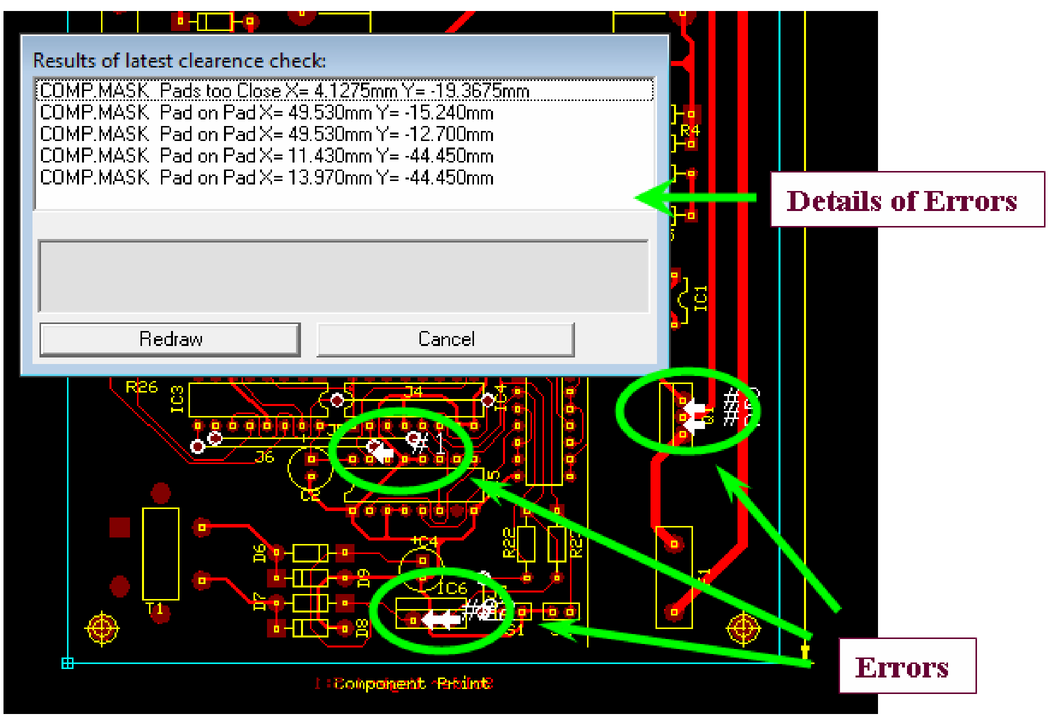


Figure: DRC Check Error Window

The above window will be displayed if any error violation of the design rule found during the DRC Check. The textual window will give all the details of the error and also it gives the details of the x-y Position of the point where the error occurred.

The graphical mode of the result will show the arrow mark where the error had occurred. (Refer the figure shown above).

3D Viewer

EDWinXP supports the 3D View of the design by one click option. The view is customizable and also EDWinXP supports 3D Trace Viewer giving details of the traces of layer by layers in 3D format.

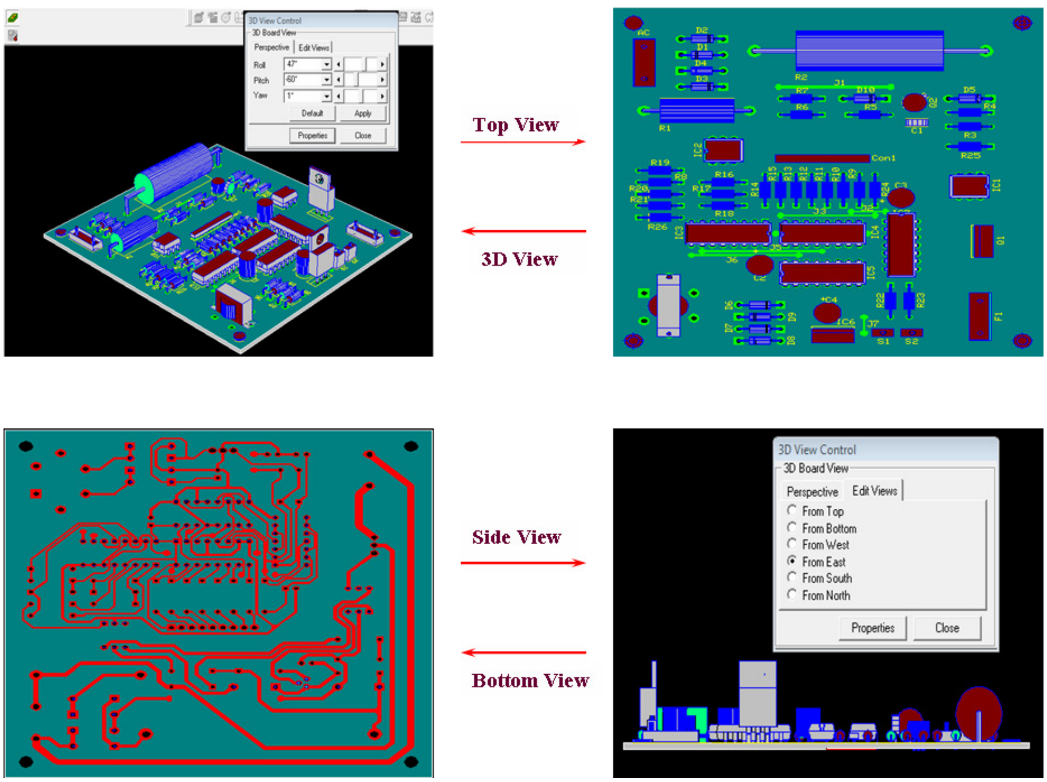


Figure: 3D Viewer

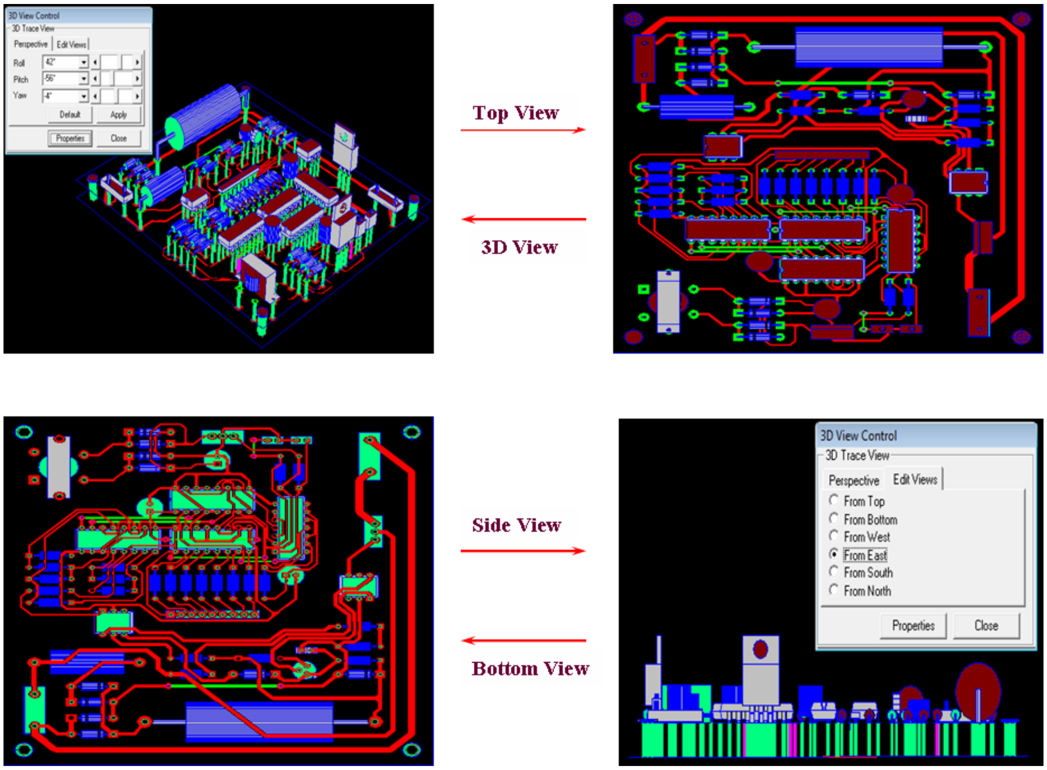


Figure: 3D Trace Viewer

Board Analyzer

The board has to analyze for the cross talk and the Field Intensity and also the temperature for the efficiency of the system. EDWinXP supports the Board Analysis. Here using this tool the user can carry on Thermal and Electro Magnetic Analysis.

Thermal Analysis:

Thermal Analyzer is used to identify the potential thermal problems on the PCB board. Here the Isothermal lines will be displayed and also the user can view the colored graph. The result of the analysis evaluates the temperature distribution on the PCB board. This will help the user to identify the potential thermal problem – gives an ideal for the user to overcome the problem.

Using the Set/Delete Label option in the option tool bar, the user can display the temperature at any point of the PCB board.

The analysis setting allows the user to define his/her predefined values for the analysis to carried out, since for every project, their will be a variation in the set up parameter. Please refer the figure given for the details on the Setting options in the Thermal Analysis.

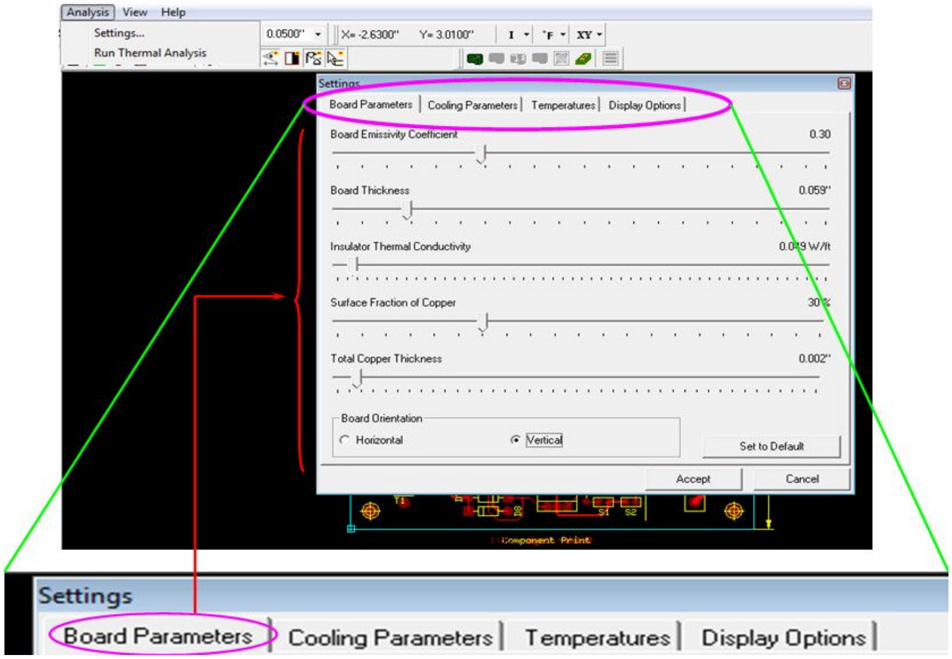


Figure: Setting Option for Thermal Analysis – Board parameters

Board Parameters:

- Board Emissivity Coefficient
- Board Thickness
- Insulator Conductor Conductivity
- Surface Fraction of Copper
- Total Copper Thickness

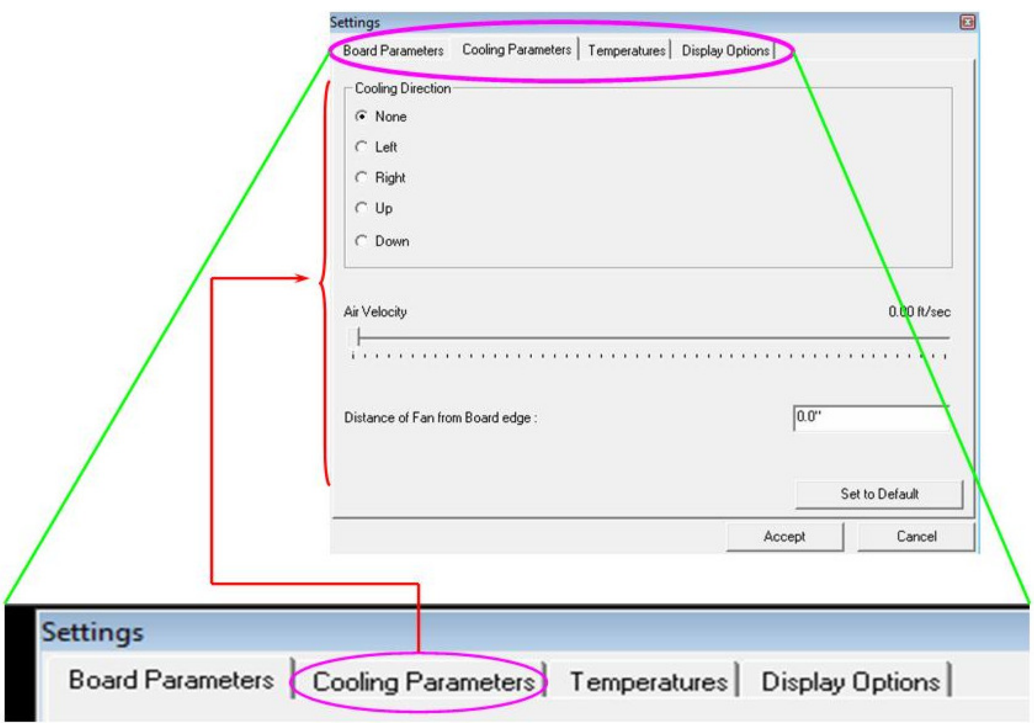


Figure: Setting Option for Thermal Analysis – Cooling Parameter

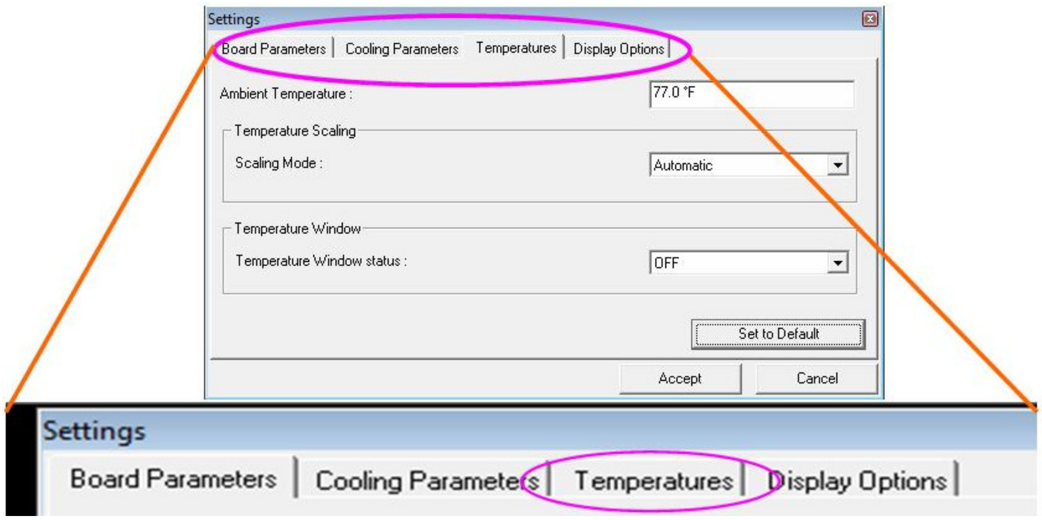


Figure: Setting Option for Thermal Analysis – Temperatures

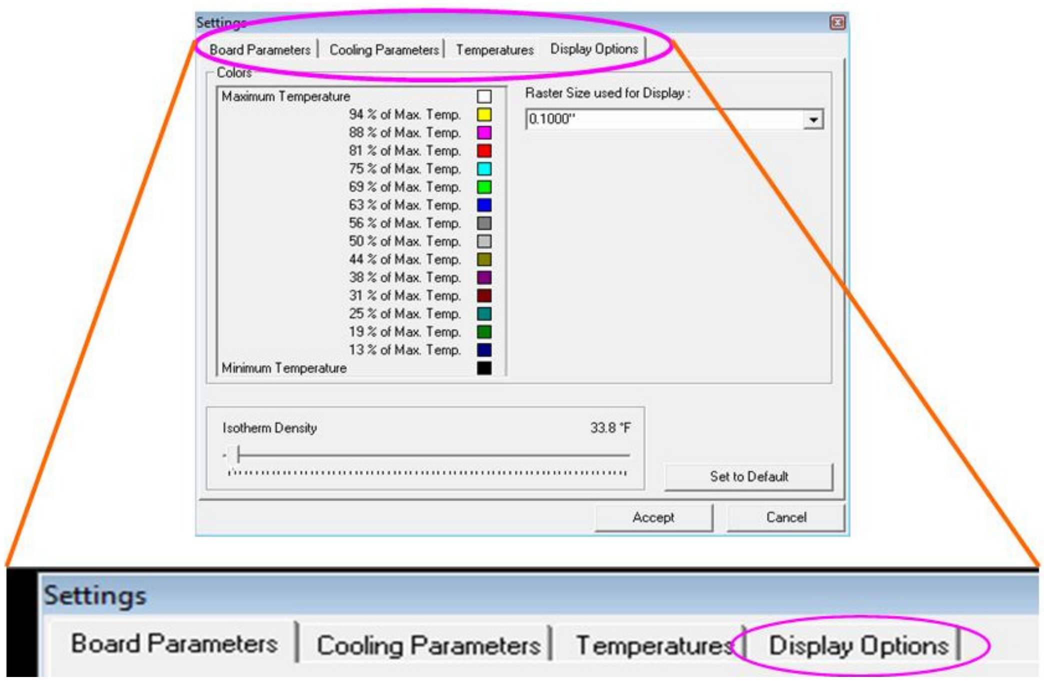


Figure: Setting Option for Thermal Analysis – Display Options

Note that in the value of the temperatures option provided in the Setting option will be the ambient temperature of the circuit. This temperature will be taken as the operating temperature for the analysis.

Once all the parameters are set then proceed with the analysis. To do the thermal analysis, select the *Thermal Analysis* tab from the *Board Analyzer* option. Select *Analysis* tool from the main tool bar and choose ***Run Thermal Analysis***.

The output of the analysis results in the Isothermal Lines. The user may switch over to the colored graph also. And also if the user wants the Alpha Numerical output (temperature value), the user can take use of the Set/Delete label option. Please refer the figure shown below for all the three options.

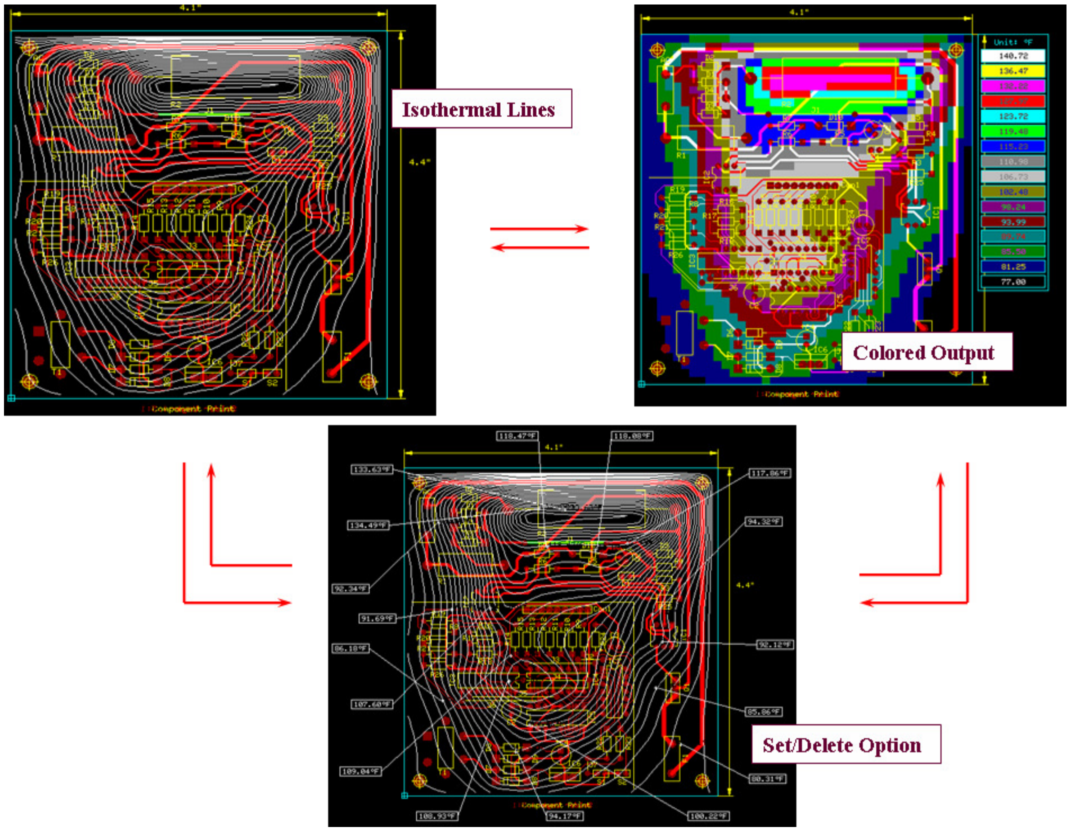


Figure: Thermal Analysis Output

Electro Magnetic Analysis:

To determine the intensity of the electromagnetic field on the PCB board can be done using the Electro Magnetic Analysis. I would like to add a point that Electro Magnetic will generate as voltage passes through the trace.

Note that the placement of the components plays a big role with the cross talk inside the board. It is true that when voltage flows through the traces on the board, but the right placement of the component and also the spacing between the components and also the spacing between the component and trace, reduces the cross talk and also the reduces the electromagnetic field in the board.

In the Electro Magnetic Analysis tab also the user can define parameters for the analysis can be carried out.

Please refer the figure given below for all the parameter setting for the Electro Magnetic analysis.

Once all the parameters are set then proceed with the analysis. To do the *Electromagnetic* analysis, select the *Electromagnetic Analysis* tab from the *Board*

Analyzer option. Select *Analysis* tool from the main tool bar and choose ***Run Electromagnetic Analysis***.

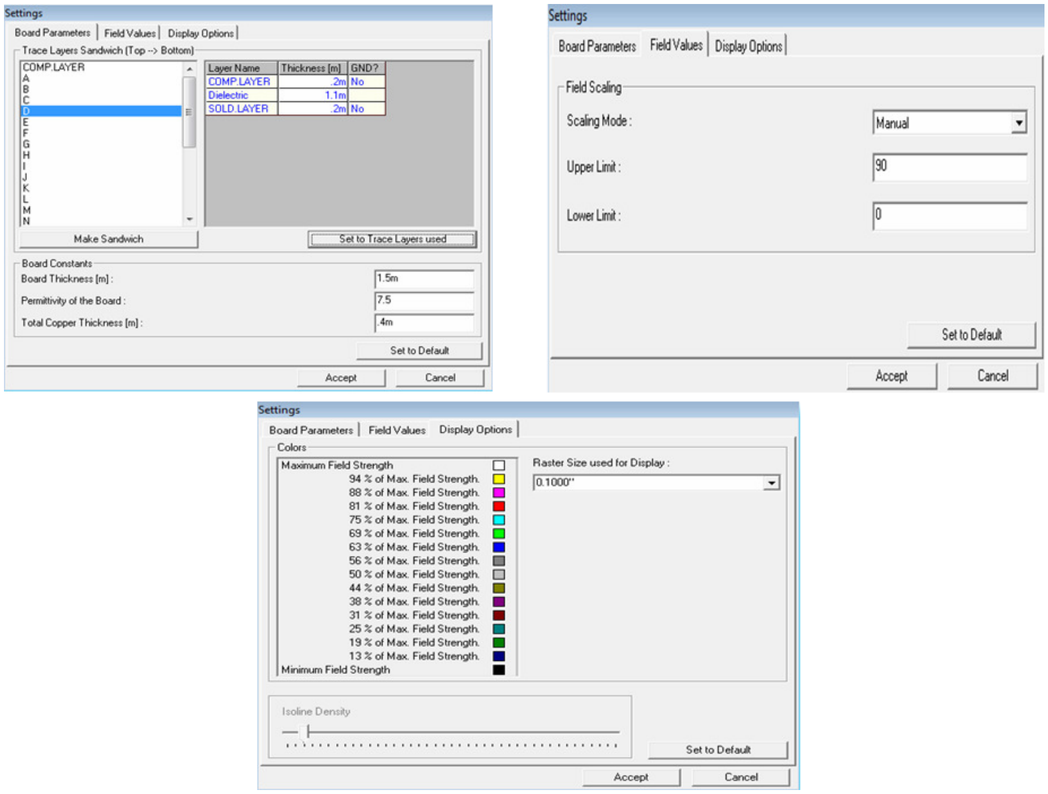


Figure: Setting options for the Electromagnetic Analysis

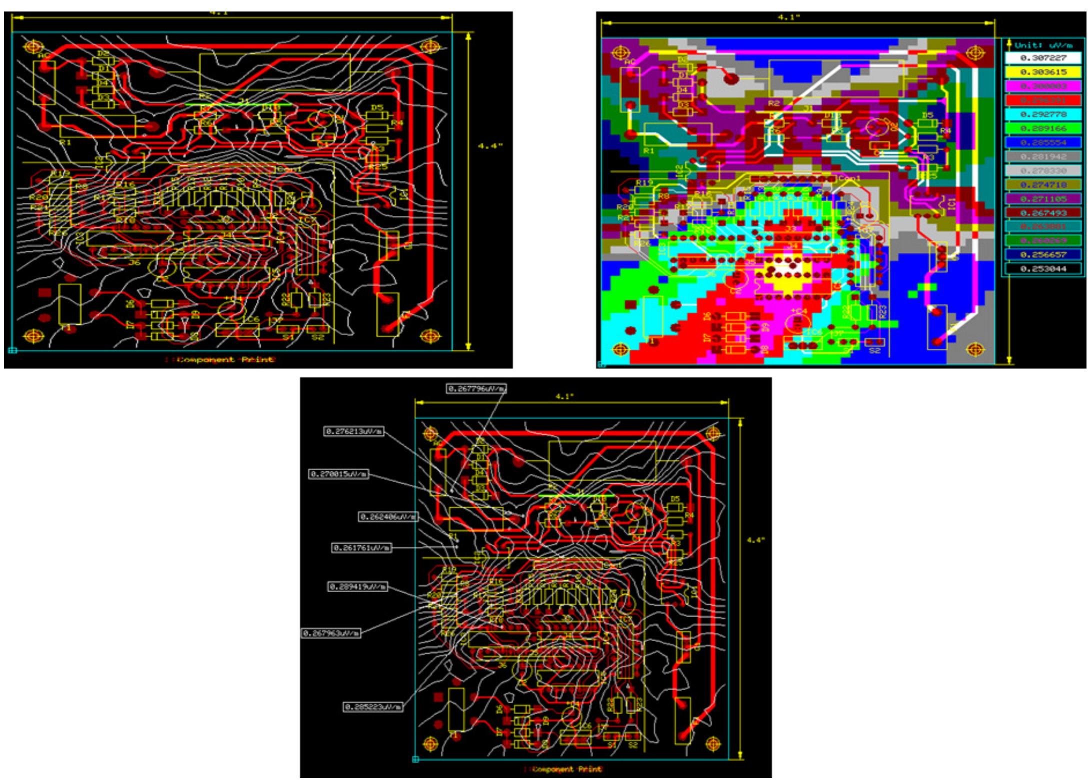


Figure: Electromagnetic Analysis Output