Manual for Wavenology EM Graphic Circuit Editor

Wave Computation Technologies, Inc.

Jan., 2013

Introduction

- WCT Graphic Circuit Editor is used to build a Spice circuit model in WCT EM full wave simulator.

- WCT also provides a Text Circuit Editor for Spice circuit. By comparing these two editors, the Graphic Circuit Editor is more convenient.
    - The Graphic Circuit Editor provides a function to check the circuit connection.
    - User doesn't need to assign the node index for the circuit.
    - The Graphic Circuit Editor provides V|I probes to let user to observe the values by his interest.

- Current WCT Graphic Circuit Editor supports following Spice circuit elements
    - Independent Source (V & I): DC, Pulse, Piece-wise linear, Sin
    - Resistor
    - Capacitor
    - Inductor
    - Diode
    - BJT transistor

    - **Other circuit elements and sub-circuit will be implemented in the future.**

- Current WCT Graphic Circuit Editor supports following probes
    - Voltage probe
    - Differential Voltage probe
    - Current probe

Basic Requirements for a Lumped Circuit in WCT

- There is at least ONE circuit element in the circuit.

- There are N port pairs for a circuit. Following is (A) a simplest circuit with a resistor and 1 port pair (two ports). (B) a 2 port pairs (4 ports) circuit.

1

Port pair 1

0

R1

(a)

1

Port pair 1

0

R2

R1

Port pair 2

2

0
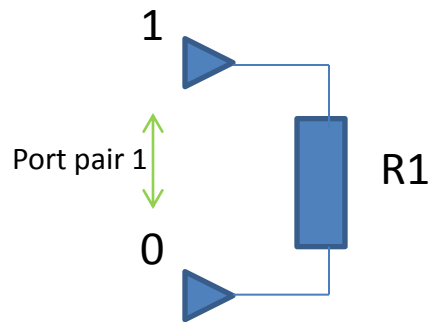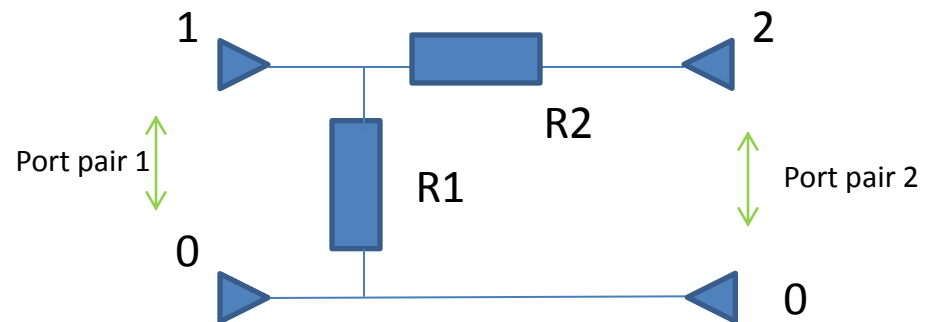
(b)

- Port is not a real circuit element. It is a 3D connection point that connect the circuit with WCT 3D EM solver. Therefore, each port should have a 3D position.

- Every two ports compose a port-pair to let the EM solver can feed voltage or current into the circuit.

- *All items and circuit name must use the characters in the range '0'-'9', 'a'-'Z' and '_'. Not other characters are allowed. Low character has the same meaning as upper character.*

Enter Graphic Circuit Editor
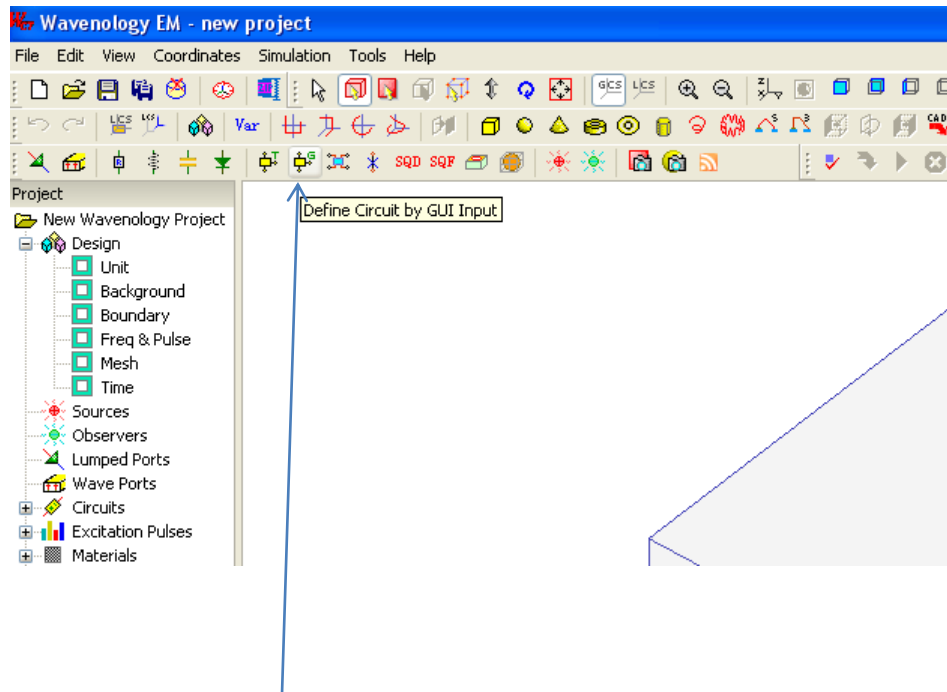
To create a new Spice circuit by Graphic Circuit Editor, click this "*Define Circuit by GUI Input*" button.
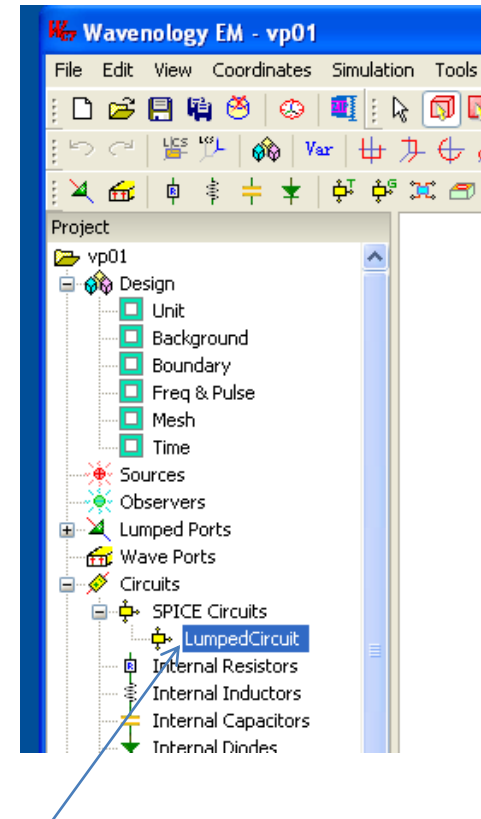
To edit a existing Spice circuit that created by Graphic Circuit Editor, double click the circuit.

Graphic Circuit Editor Layout

Toolbar Items

Quit &
Save
without
checking

undo

Zoom in

Ready
to drop
voltage
probe

Ready
to drop
current
probe

Refresh
Canvas

Quit
without
save

Quit &
save with
checking

undo

Zoom out

Ready to
drop
differenti
al voltage
probe

Circuit
Property:
Including
circuit
name,
port-pair
definition

Supported Items in the Canvas
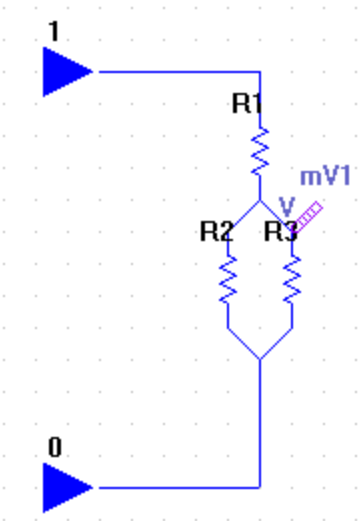
A circuit example
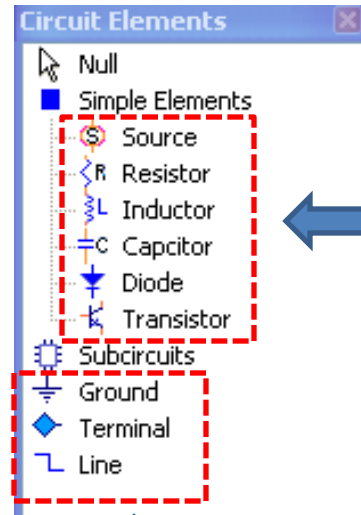
Circuit Elements

Null
Simple Elements
- Source
- Resistor
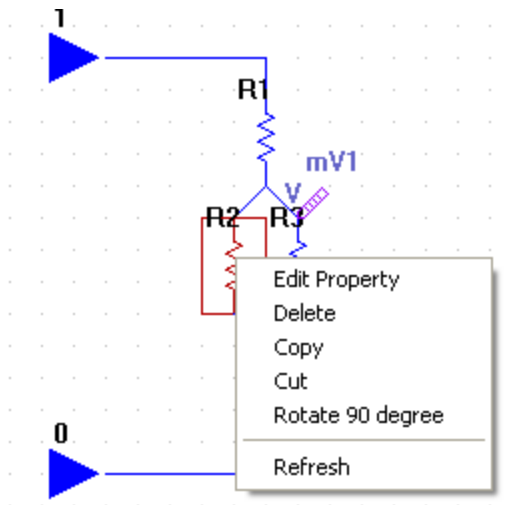- Inductor
- Capcitor
- Diode
- Transistor
Subcircuits
Ground
Terminal
Line

Independent Source (V & I): DC, Pulse, Piece-wise linear, Sin
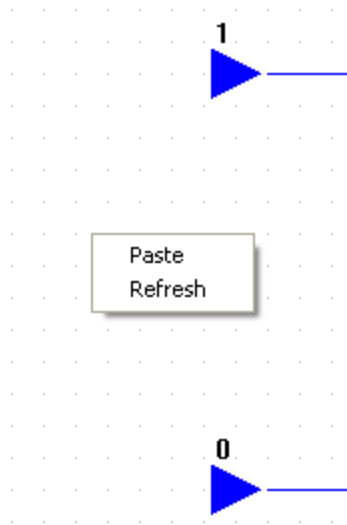Resistor
Capacitor
Inductor
Diode
BJT transistor

Ground, circuit port, line

Supported Operations for a Selected Item in Canvas

1. Property editing: modify name, parameters

2. Delete

3. Copy: copy selected items to clipboard

4. Cut : cut selected items to clipboard

5. Rotation 90 degree: the original connections will be removed.

Operations on Canvas
(not selected items)

Refresh canvas and Paste items from clipboard (if clipboard has items)
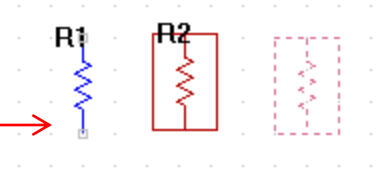
Refresh canvas only (if clipboard has not items)

Refresh canvas and Cancel dropping a new item on canvas

Item Status on Canvas

selected
state

R1   R2

If a terminal of an item is not connected (OPEN), there is a rectangle displayed

R1

R1

Regular
state

Dragging
state

Terminals of two items will be connected , there is a red box displayed

There is not rectangle on the connected terminal in the Regular state

Supported Items' information Independent Source

Independent Source (V & I)

DC: *icon with*   -
Pulse: *icon with*   п
Piece-wise linear: icon with м
Sin: *icon with*   ~

V1 V2 V3 V4

I5 I6 I7 I8
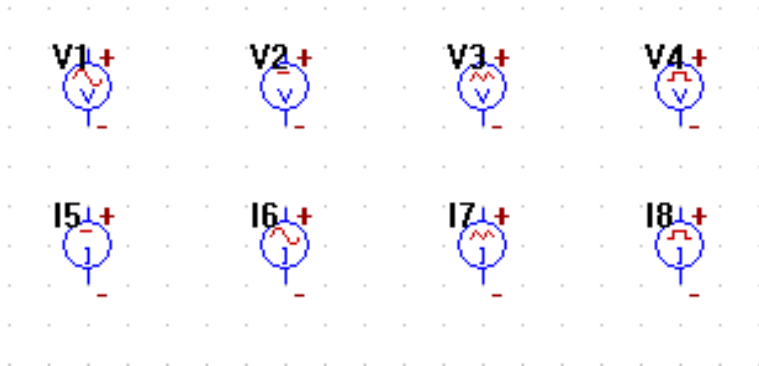
Voltage source Unit: V

Current source Unit: A

Voltage source start with V
Current source start with I

Circuit Source Editor

Category
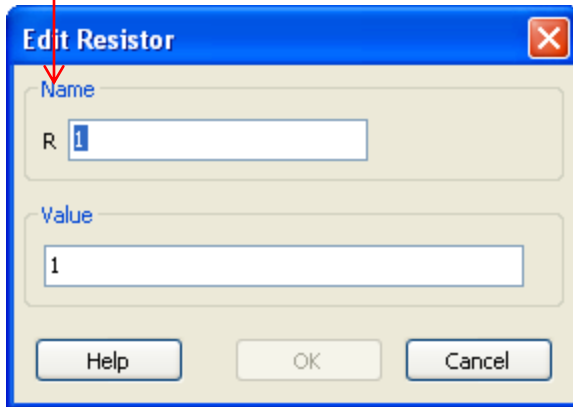Voltage Source
Voltage Source
Current Source

Name
V 1

Waveform

○ DC   Magnitude 1

◉ Sinusoidal

○ Piece-Wise Linear

○ Pulse

○ Exponential

Waveform

Modify the pulse

Help   OK   Cancel

Cont.

Sinusoid waveform editor
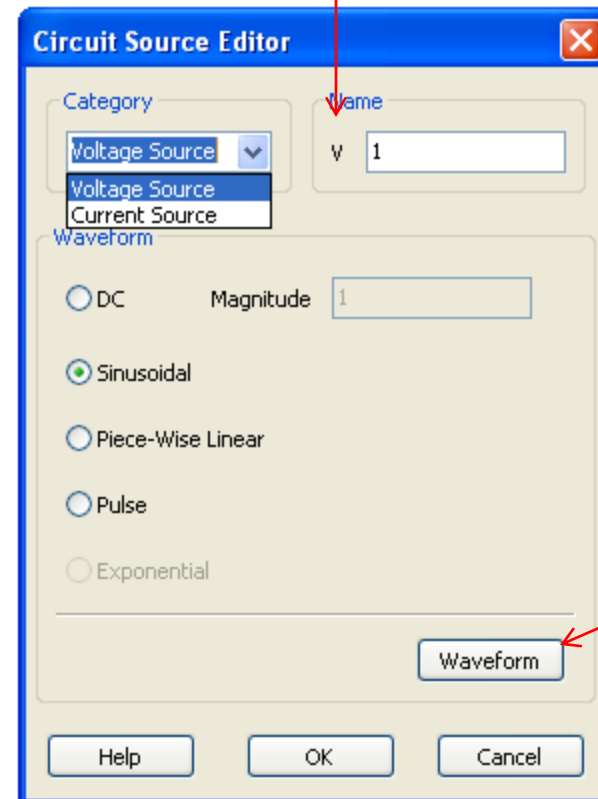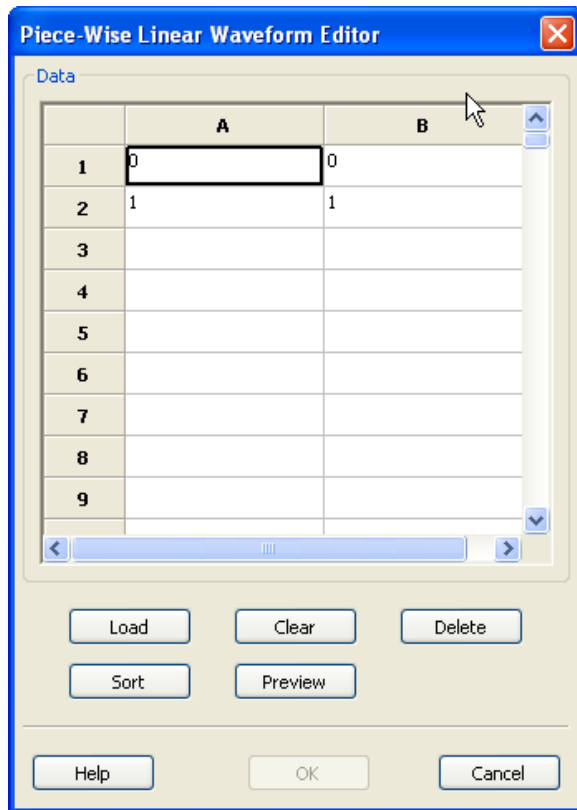
Piece-Wise Linear
waveform editor
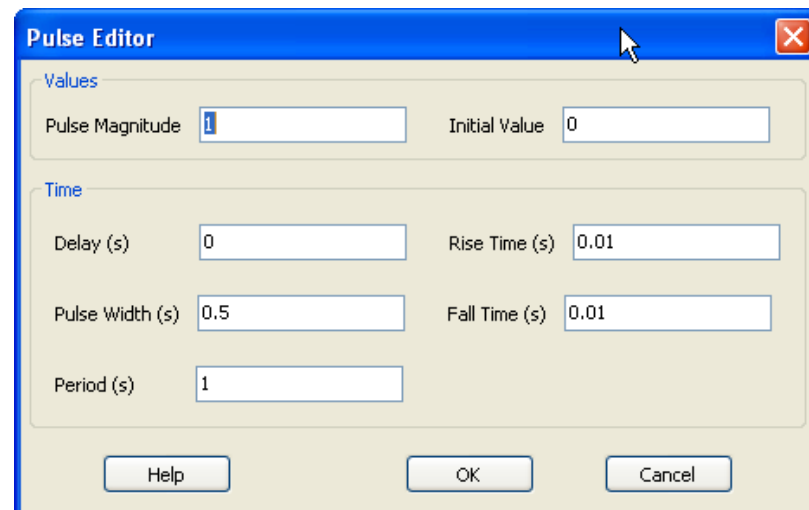
Pulse editor

Resistor & Capacitor

Resistor start with R
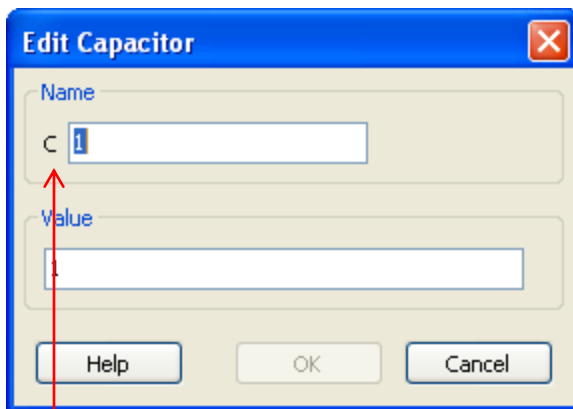
Unit: Ω
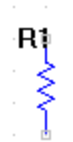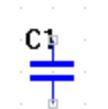
Unit: F

Resistor start with C
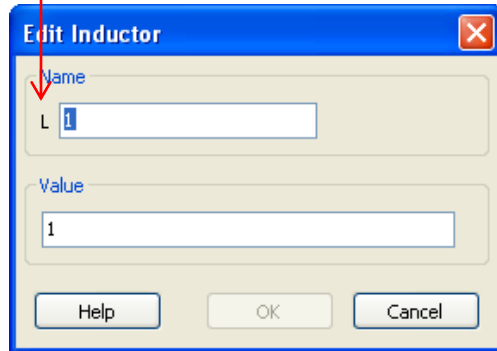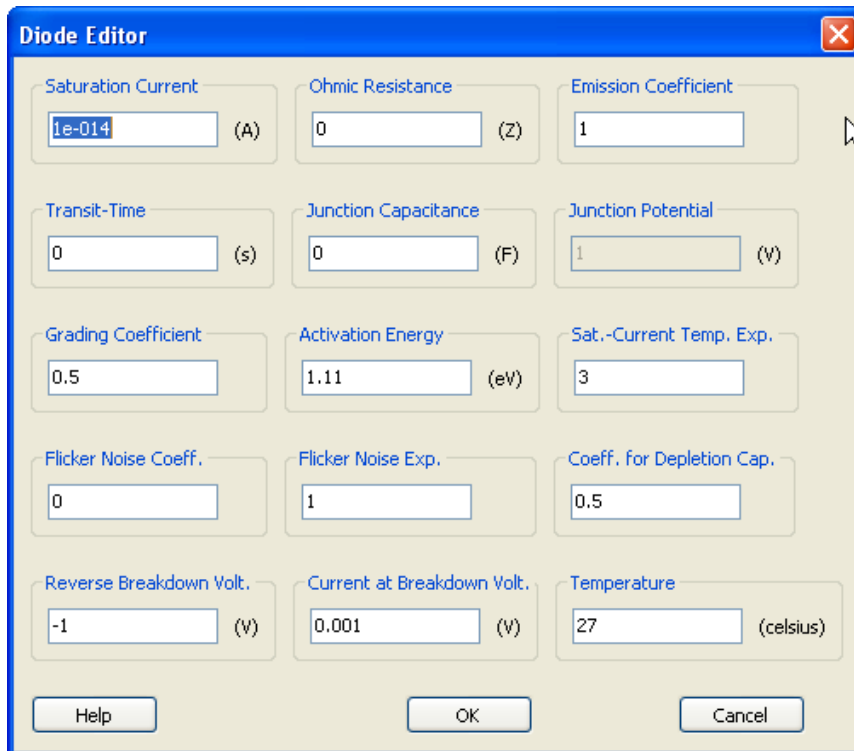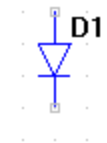
Inductor & Diode

Inductor start with L

Unit: H

Diode name is automatically assigned.

BJT Transistor
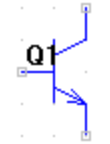
BJT Transistor start with Q

Spice BJT transistor has 41 parameters. Currently, we only support the most frequently used 11 parameters are editable.

For the meaning of each parameters, please refer to Spice manual.
http://newton.ex.ac.uk/teaching/CDHW/Electronics2/userguide/sec3.html

Ground, Line & Circuit Port

Not Editable

Not Editable. The line terminal can be dragged separately.

Edit Circuit Port

Name

`1`

Physical Position (x,y,z)

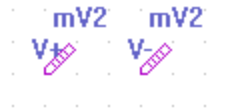`0.0, 0.0, 0.0`

Help   OK   Cancel

Probes

All probes are not editable, cannot be rotated.
Probe must be placed on the line or item terminal.

mV1

Voltage probe measures the voltage from ground.

mV2   mV2

Differential voltage probe measures the voltage between two nodes.

mI1

Current probe measures the current goes through a item or a line.
Therefore, it is an invalid current probe position on a connection with more than 2 items.

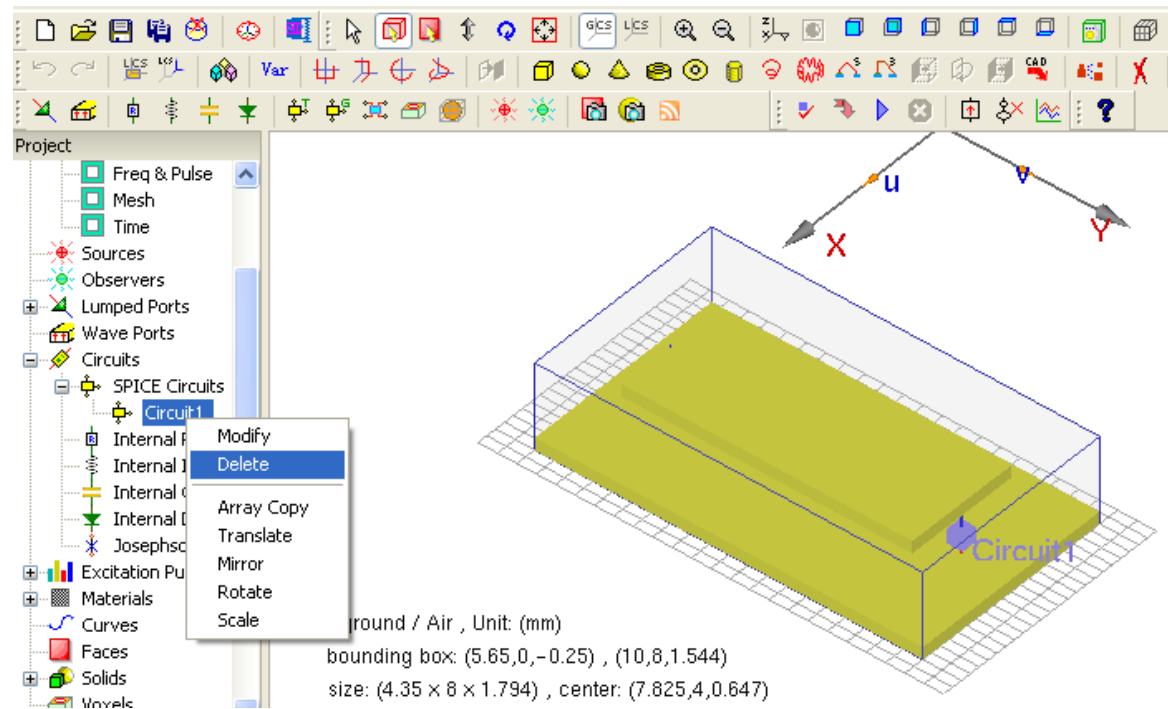# Circuit Properties

Use this toolbar button

Tutorial Case 1
A Circuit with Three Resistors

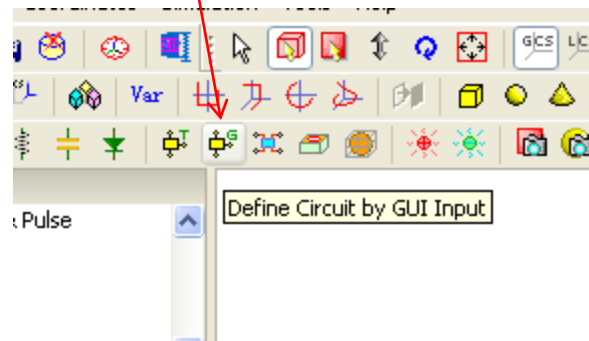This case is modified from the WCT EM tutorial case: ……\Circuits\Resistor\ SpiceRes.wnt
1. Open case **'SpiceRes.wnt'** and *SaveAs* '….\…..\vp01.wnt'
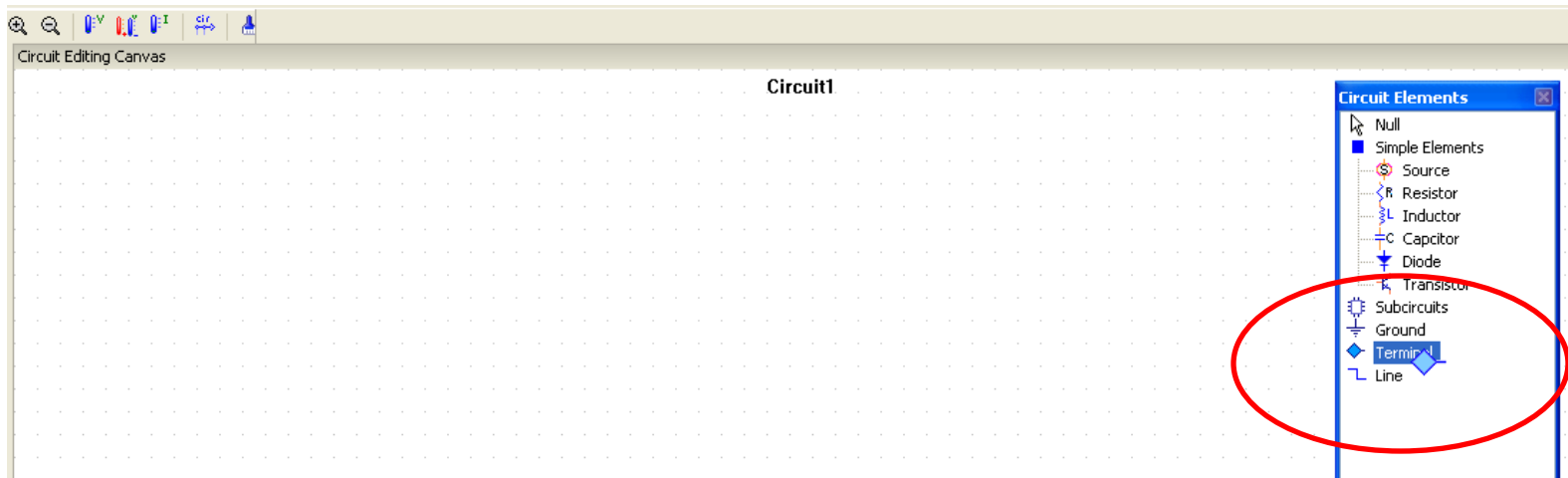2. Delete original text format Spice circuit '**Circuit1**'

3. Press this button to enter graphic circuit editor

Define Circuit by GUI Input

4. Press the icon "Terminal" in the "Circuit Elements" list. The mouse cursor will become a *Port*.

Circuit Editing Canvas

Circuit1

Circuit Elements

- Null
- Simple Elements
  - Source
  - Resistor
  - Inductor
  - Capcitor
  - Diode
  - Transistor
- Subcircuits
- Ground
- Terminal
- Line
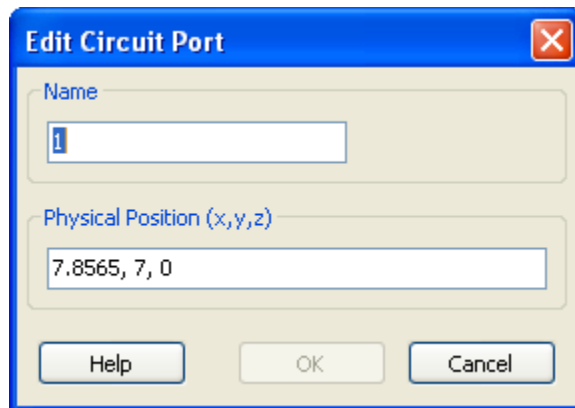
5. Move mouse to circuit editor canvas, place two ports at any places.
Then press "*ESC*" key or use mouse (right button) menu "*Cancel*", or move
mouse back to "Circuit Elements" list and press "NULL" item, to quit "dropping
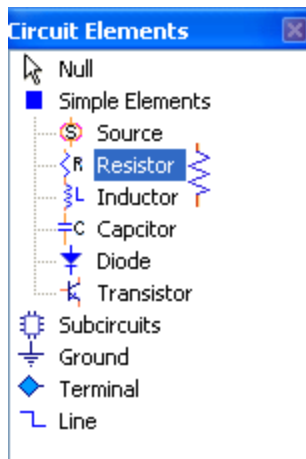item" mode.

6. Modify the name of Port 1 & 2 to 1 & 0, 3D positions are "7.8565, 7, 0" and
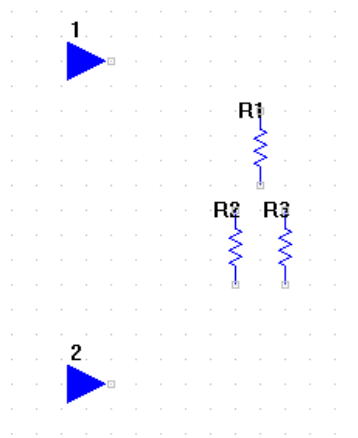"7.8565, 7, 1.044" , respectively.

21
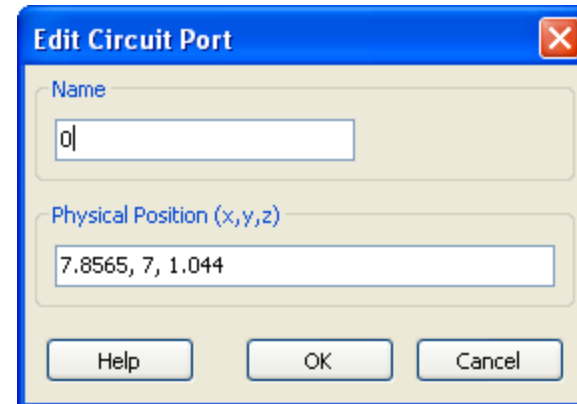
Cont.

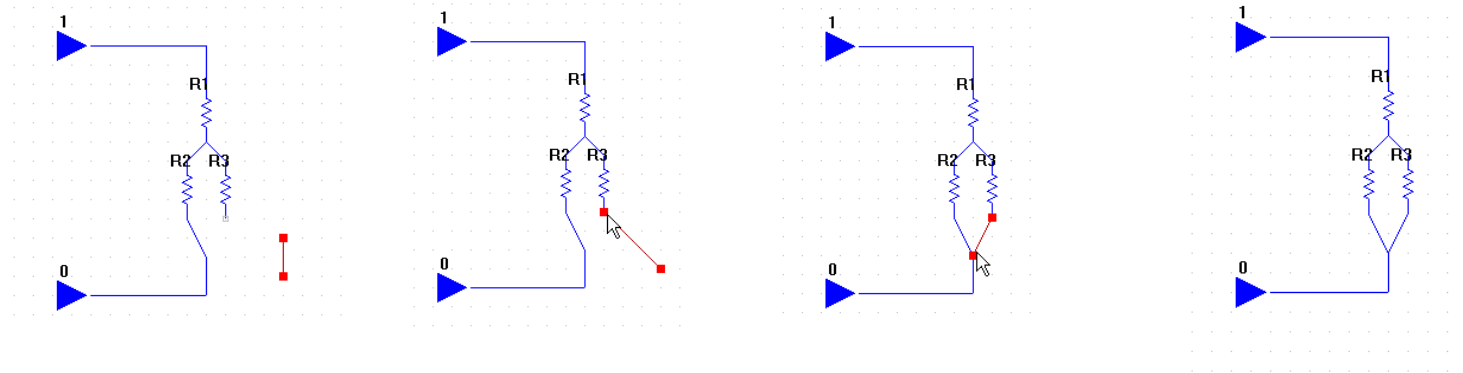7. Press the icon "Resistor" in the "Circuit Elements" list. The mouse cursor will become a *Resistor*.

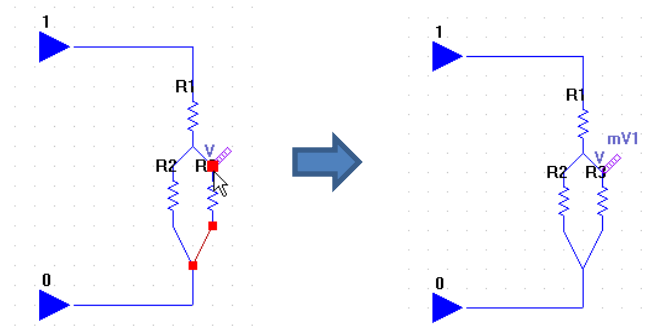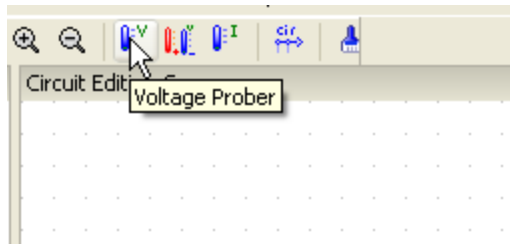8. Drop three resistors in the canvas. Then modify R1=25 Ω, R2=50 Ω and R3=50 Ω respectively.

9. Drop enough lines in the canvas, and dragging lines' terminals to connect resistors and ports, as following steps.

10. Press "Voltage Probe" button in the toolbar, drop a voltage probe at the position as following.
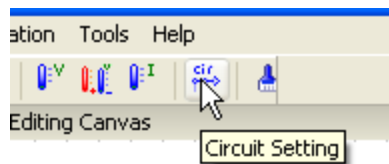

Before dropping

After dropping

11. Check the circuit property.

Default port-pair setting is wrong. Because this circuit has not port *2*.

Correct port-pair

12. Press "Check, Save and Exit" button to check the circuit and back to WCT 3D main Canvas.
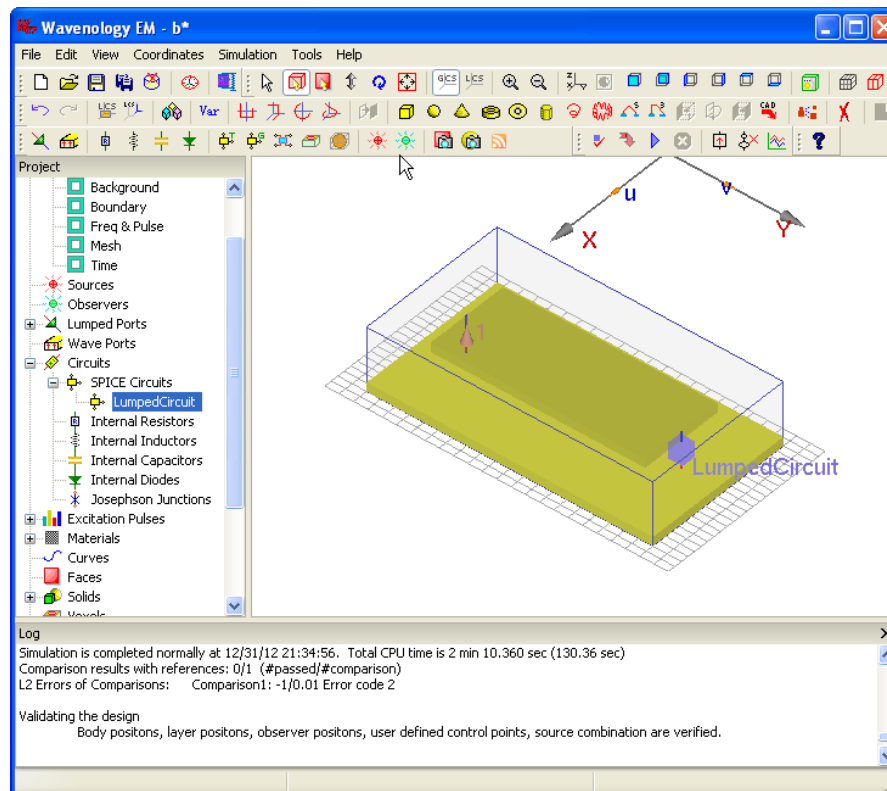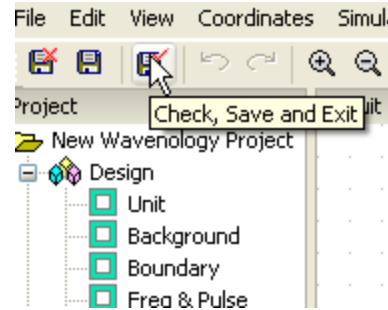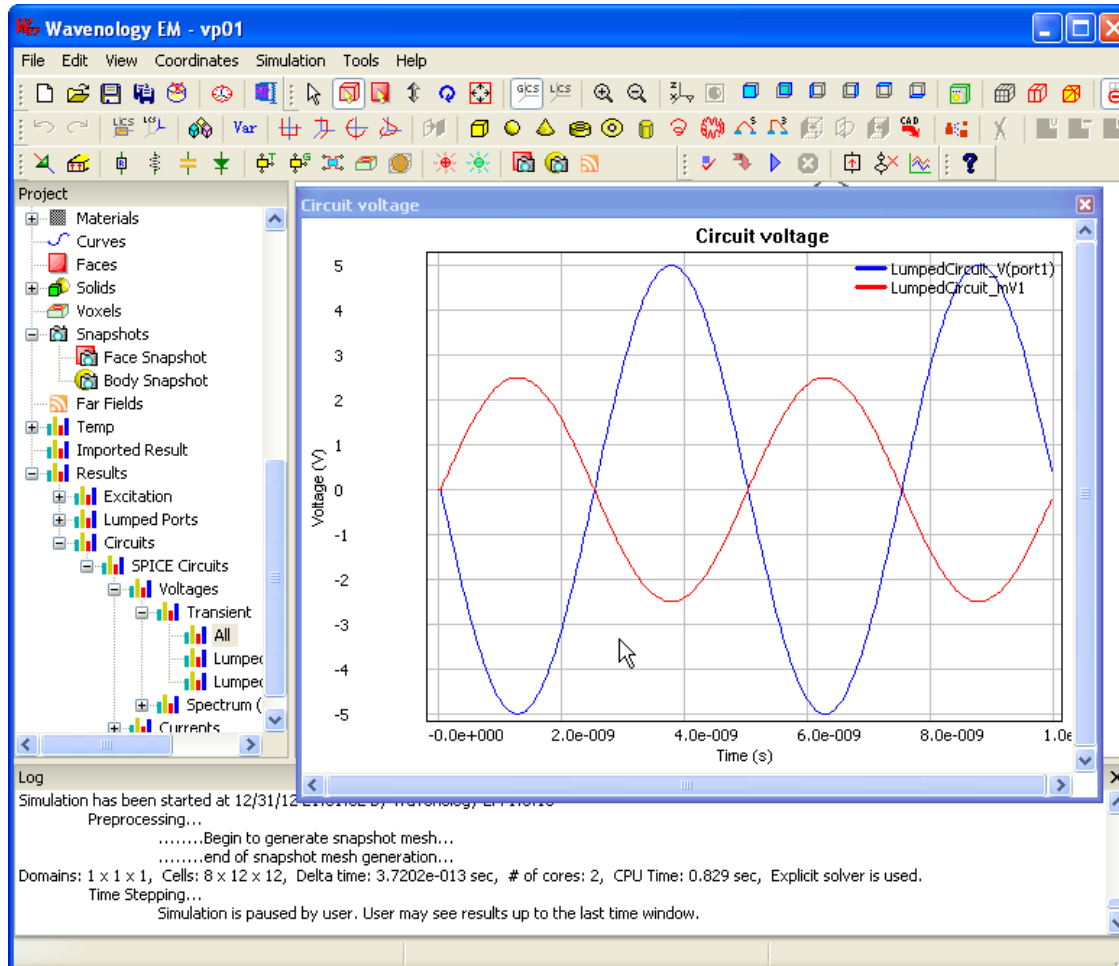
13. Simulate the case and check the result.

Demo Cases List

1. Single resistor: graphic_circuit_demo\r01\r01.wnt
2. three resistors: graphic_circuit_demo\r01\r03.wnt

3. Single capacitor: graphic_circuit_demo\c01\c01.wnt

4. Single inductor: graphic_circuit_demo\l01\l01.wnt

5. Single diode: graphic_circuit_demo\diode\d01.wnt

6. V probe with three resistors: graphic_circuit_demo\vProbe\vp01.wnt
7. Differential V probe with three resistors: graphic_circuit_demo\vProbe\vp02.wnt

8. I probe with three resistors: graphic_circuit_demo\IProbe\ip01.wnt
9. I probe with resistor, source and inductor: graphic_circuit_demo\IProbe\ip02.wnt