R. Moser - Desk Copy

P

S

Beginner's User Manual for PSpice™

Berrigan • Beals • Bird

TABLE OF CONTENTS

ILLUSTRATIONS

FIGURES

SECTION 1

Create a New Project

SECTION 2

Circuits: Resistive Circuit

INTRODUCTION

For many years the Electronics Department of Penn College has relied on the PSpice™ computer program to simulate and analyze electronic circuits. "SPICE" is an acronym for Simulation Program with Integrated Circuit Emphasis. While the PSpice™ program is an excellent tool for evaluating circuits, it can be difficult for a beginner to learn because of the program's atypical format. Frustration can abound for a beginner who uses the program without an understanding of its basic structure.

As a result, three Penn College Advanced Technical Communication students chose to write this manual to alleviate student and faculty frustration when electronic students use the PSpice™ program for the first time. The *Beginner's User Manual for PSpice™* explains the basic structure of the program and uses a step-by-step format to help students succeed when they simulate and analyze the PSpice™ circuits required by the Electronics program. When students use this manual successfully, many hours of instructional time can be saved for the electronics faculty.

The *Beginner's User Manual for PSpice™* demonstrates how to open the Orcad® program, create a PSpice folder, open new projects, add libraries, build resistive circuits, and analyze the results of the circuits. The manual clarifies the libraries needed to select components so that the analysis functions will run properly; and explains how to correctly save projects, close projects, close the PSpice folder, and close the program.

A note to the user:
Once you understand the basic structure of the program, you can apply that knowledge to other, more complicated procedures and circuits. This manual, however, is not an answer key. While the manual helps you learn the basics of the PSpice™ program, it does not contain every answer to every possible dilemma that the program may present. As an electronics student, you can continue to explore and gain more program knowledge on your own.

The manual is designed to help you develop a confidence in circuit simulation programs, realize the value of SPICE programs as a circuit evaluation tool, and learn the PSpice™ program as easily as possible. With the *Beginner's User Manual for PSpice™* as a guide, you can quickly master the skills necessary to complete your first required assignments and have a positive experience when you simulate your first circuits.

CHECK LIST

Before **Creating a New Project**, check the following:

- ✓ **IBM compatible compute is used**
- ✓ **Windows 98 operating system or higher is installed on computer**
- ✓ **Orcad Family Release 9.2 Lite Edition** is **installed on computer**
- ✓ **3 ½ Floppy Disk is inserted into the (A:\) drive (for saving PSpice folder and circuit schematic files)**

MOUSE CONVENTIONS *(consistent with PSpice™printed literature)*

The following mouse conventions are used throughout the manual:

- **CLICKL** (*click left* once) to select an item.
- **DCLICKL** (*double click* left) to end a mode or edit a selection.
- **CLICKR** (*click right* once) to abort a mode.
- **DCLICKR** (*double click right*) *to repeat an action.*
- **CLICKLH** (click left, *hold down, and move mouse*) to drag a selected item. Release left button when placed.
- **DRAG** (*no clicks, move mouse*) to move an item.

BOLD TEXT

A Glossary (at the end of this manual) contains **bold text** terms with definitions according to their use in this manual.

Create a Folder and Save to Floppy Disk

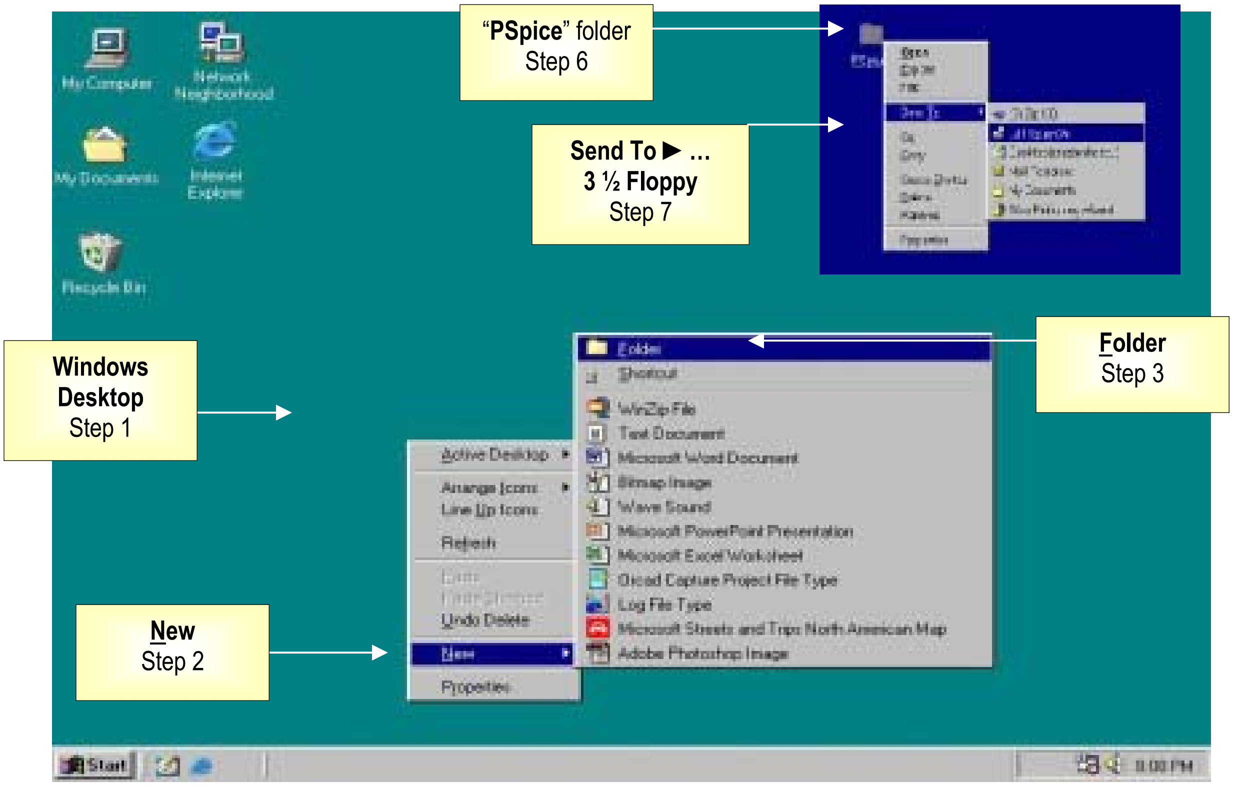


Figure 1-1 Create a Folder and Save to Floppy Disk

Instructions	Icon *(Position on Figure)*	Description
1. **CLICKR** (click right mouse button) on **Windows Desktop** . . .		Open pull-down menu
2. **CLICKL** (click left mouse button) on **New** option . . .		Open **New** pull-down menu
3. **CLICKL** on **Folder** option.		Open **New Folder**
4. Type "**PSpice**" while **New Folder** is highlighted . . .		Name **New Folder**
5. Use **Enter** ← ("key" on keyboard).		End name **New Folder**
6. **CLICKR** on "**PSpice**" **Folder** . . .		Open pull-down menu
7. **CLICKL** on **Send To** ► **3 ½ Floppy**.		Send **Folder** to (**A**:) drive

Open PSpice™ Program

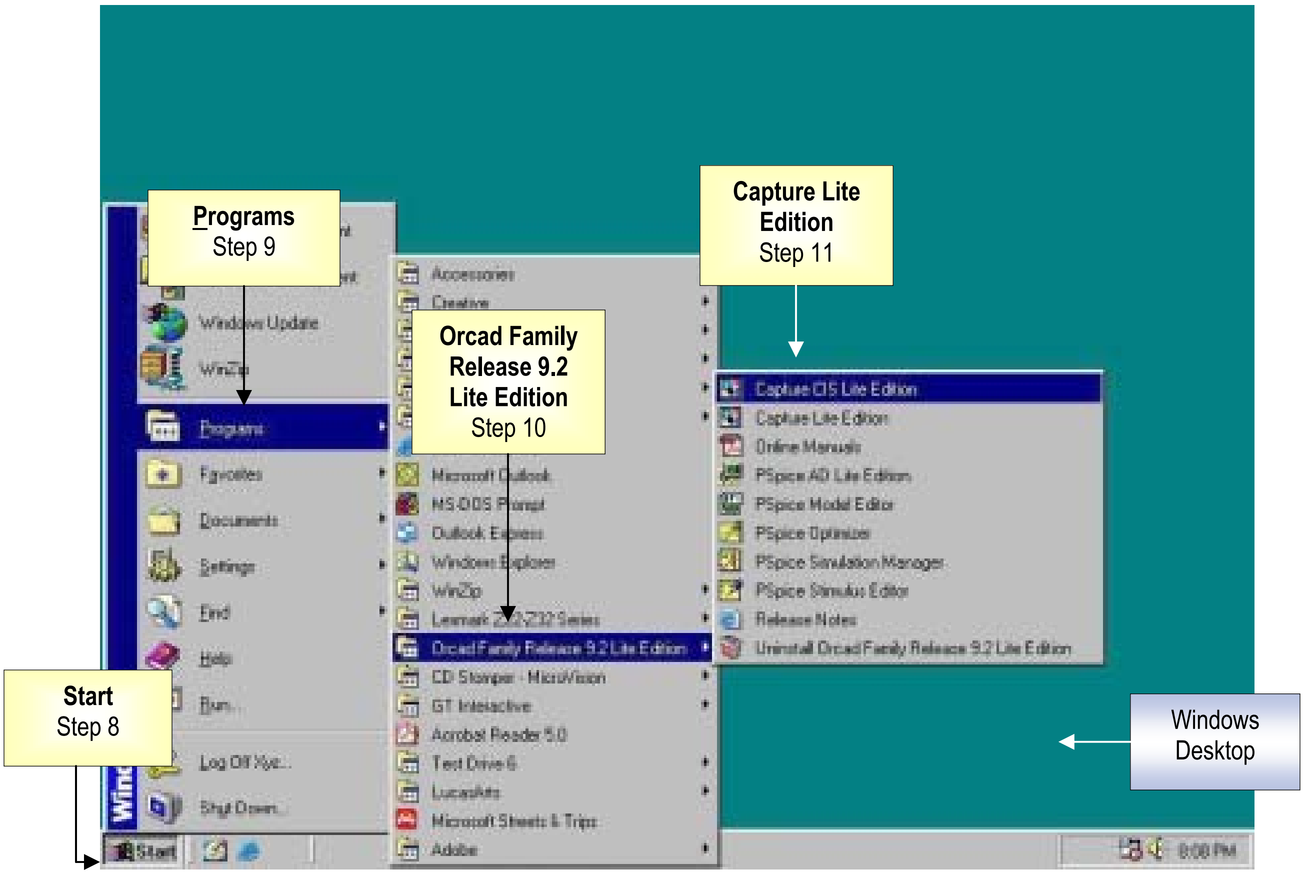


Figure 1-2 Open PSpice™ Program

Instructions	Icon (Position on Figure)	Description
8. **CLICKL** on **Start** icon on **"Windows Desktop"** . . .	 *(lower left)*	Open **Windows** options
9. **CLICKL** on **Programs** option . . .		Open **Programs** menu
10. **CLICKL** on **Orcad** (**Orcad Family Release 9.2 Lite Edition)** option . . .		Open **Orcad** program menu
11. **CLICKL** on **Capture Lite Edition** option.		Open **Capture – [Session Log]** window

Open a New Project

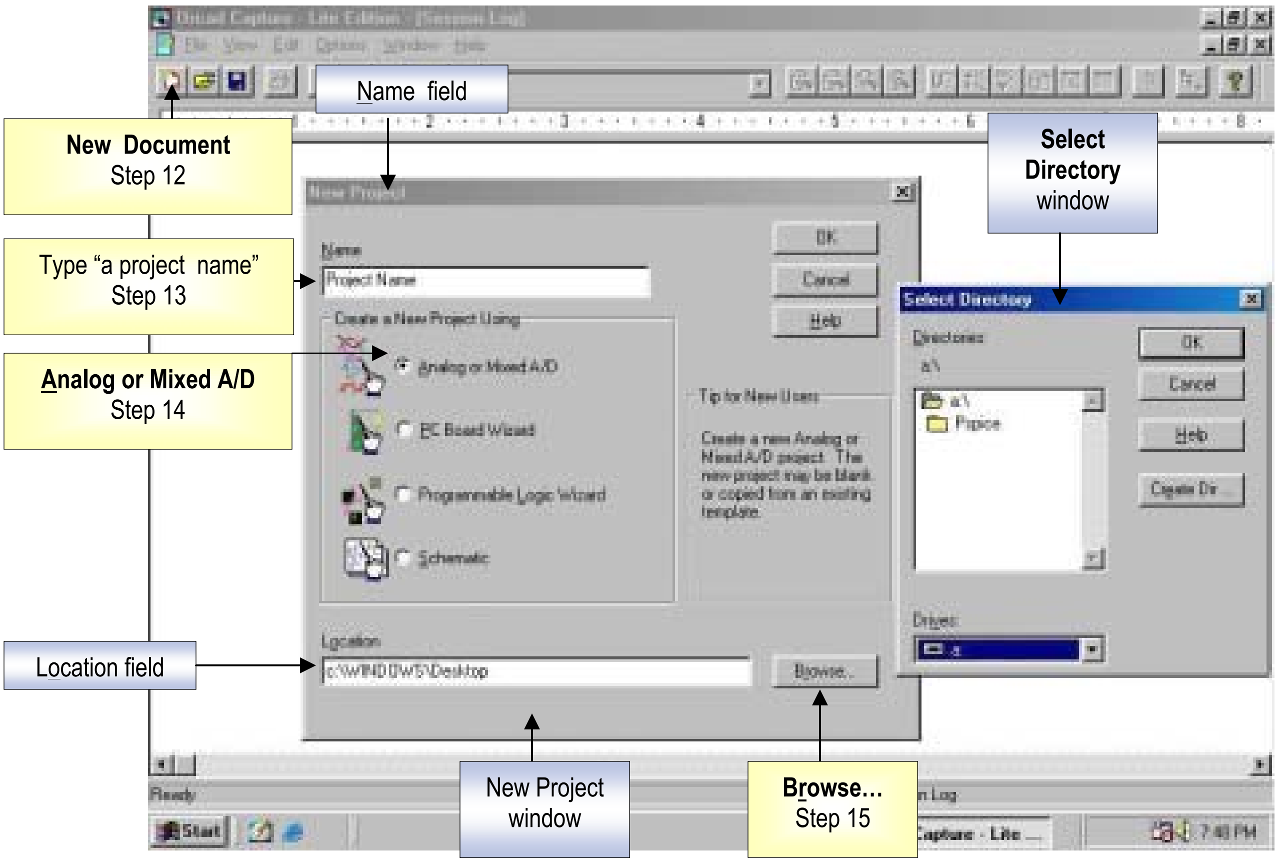


Figure 1-3 Open a New Project

Instructions	Icon (Position on Figure)	Description
12. **CLICKL** on **New Document** icon.	 *(upper left)*	Open **New Project** window
13. Type a “***project name***” in the **Name:** field window.		Name **Project**, i.e. Circuit 1, Circuit 2, etc.
14. **CLICKL** on **Analog or Mixed A/D** radial option.	 *(center left)*	Select **Analog** or mixed **Analog/Digital** project
15. **CLICKL** on **Browse…** option.	 *(lower middle)*	Open **Select Directory** window

Select a Directory

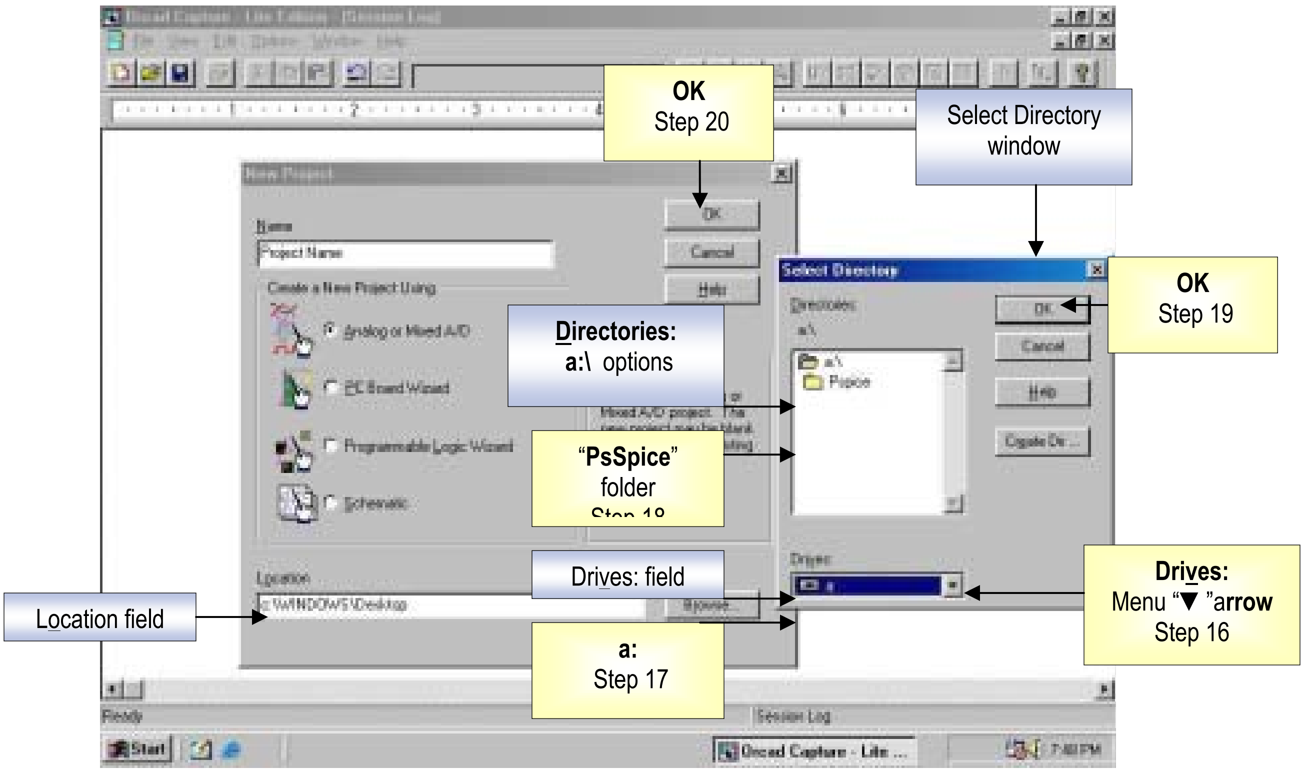


Figure 1-4 Select a Directory

Instructions	**Icon** *(Position on Figure)*	**Description**
16. **CLICKL** on **Drives:** "▼" arrow . . .	*(lower right)*	Open **Drives:** pull-down menu
17. **CLICKL** on "**a:**" **Drives:** option . . .	a:	Select (**a:**) **Drive**
18. **DCLICKL** on "**PSpice**" folder option in "**a:**" field . . .	Create a blank project	Highlight and select "**PSpice**" folder
19. **CLICKL** on **OK** option in **Select Directory** window.	OK *(center right)*	Close **Select Directory** window
20. **CLICKL** on **OK** option in **New Project** window.	OK *(upper middle)*	Close **New Project** window

Create Blank PSpice Project

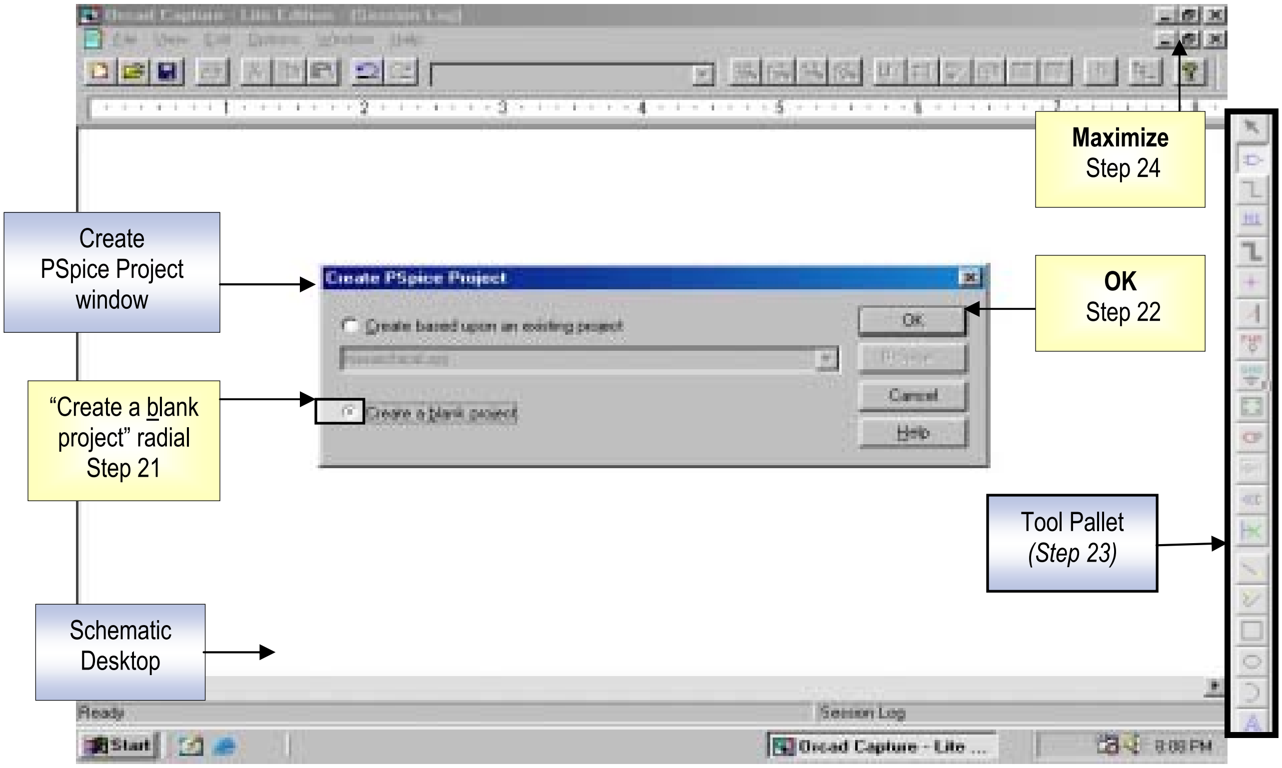


Figure 1-5 Create Blank PSpice Project

<u>**Instructions**</u>	<u>**Icon**</u> *(Position on Figure)*	<u>**Description**</u>
21. **CLICKL** on **Create a blank project** radial option . . .		Select **Create a blank project**
22. **CLICKL** on **OK** option in **Create PSpice Project** window.	OK *(center right)*	Close **Create a Project** window
23. **CLICKL** on "**Schematic Desktop**".		Open **Tool Palette**
24. **CLICKL** on "**Maximize**" icon.	 *(upper right)*	**Maximize** schematic desktop

Add Libraries

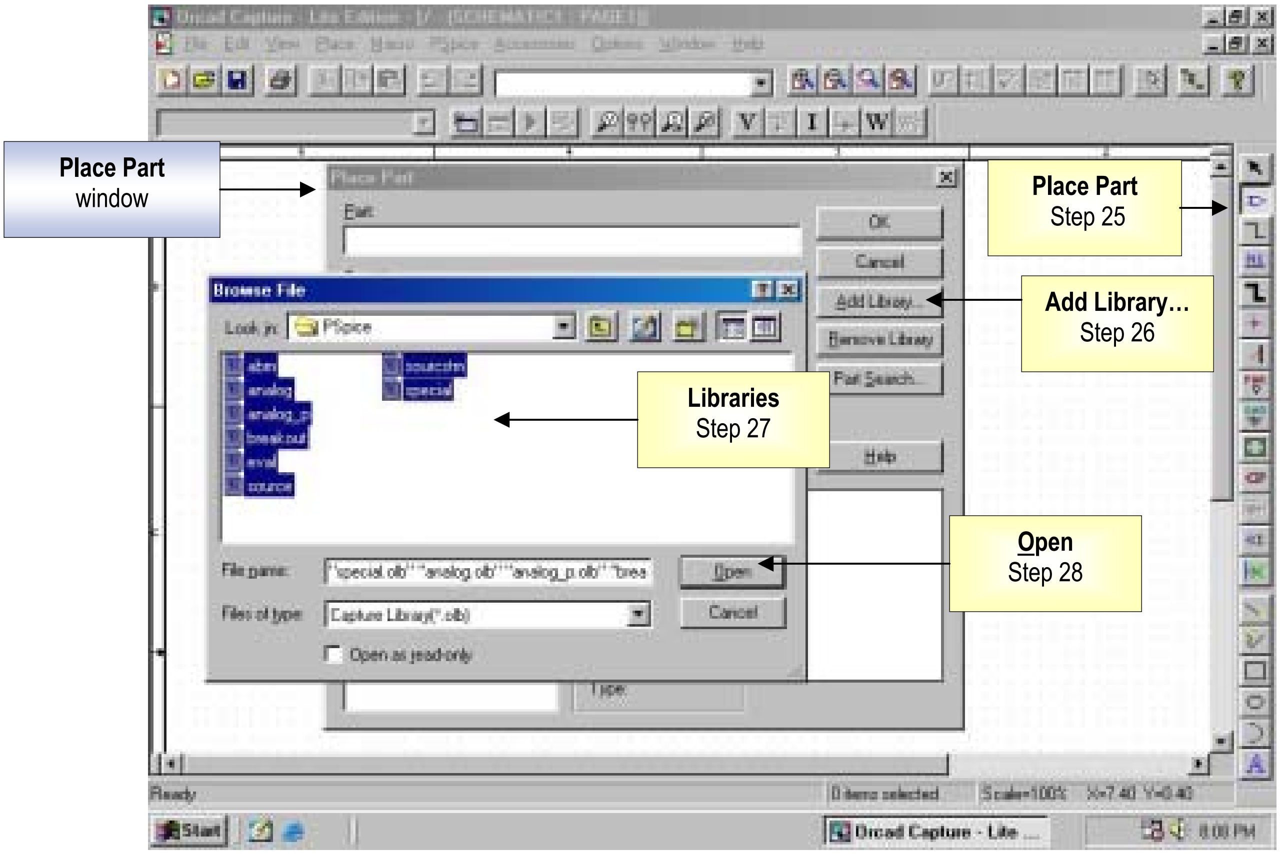
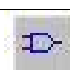


Figure 1-6 Add Libraries

Instructions	Icon *(Position on Figure)*	Description
25. **CLICKL** on **Place Part** icon . . .	 *(upper right)*	Open **Place Part** window
26. **CLICKL** on **Add Library…** option.	 *(center right)*	Open **Browse File** window
27. While holding down **Shift** key on keyboard, **CLICKL** (highlight) all **Library** options . . .		Select all **Libraries**
28. **CLICKL** on **Open** option.	 *(lower middle)*	Open highlighted **Libraries** and close **Browse File**

Place Part

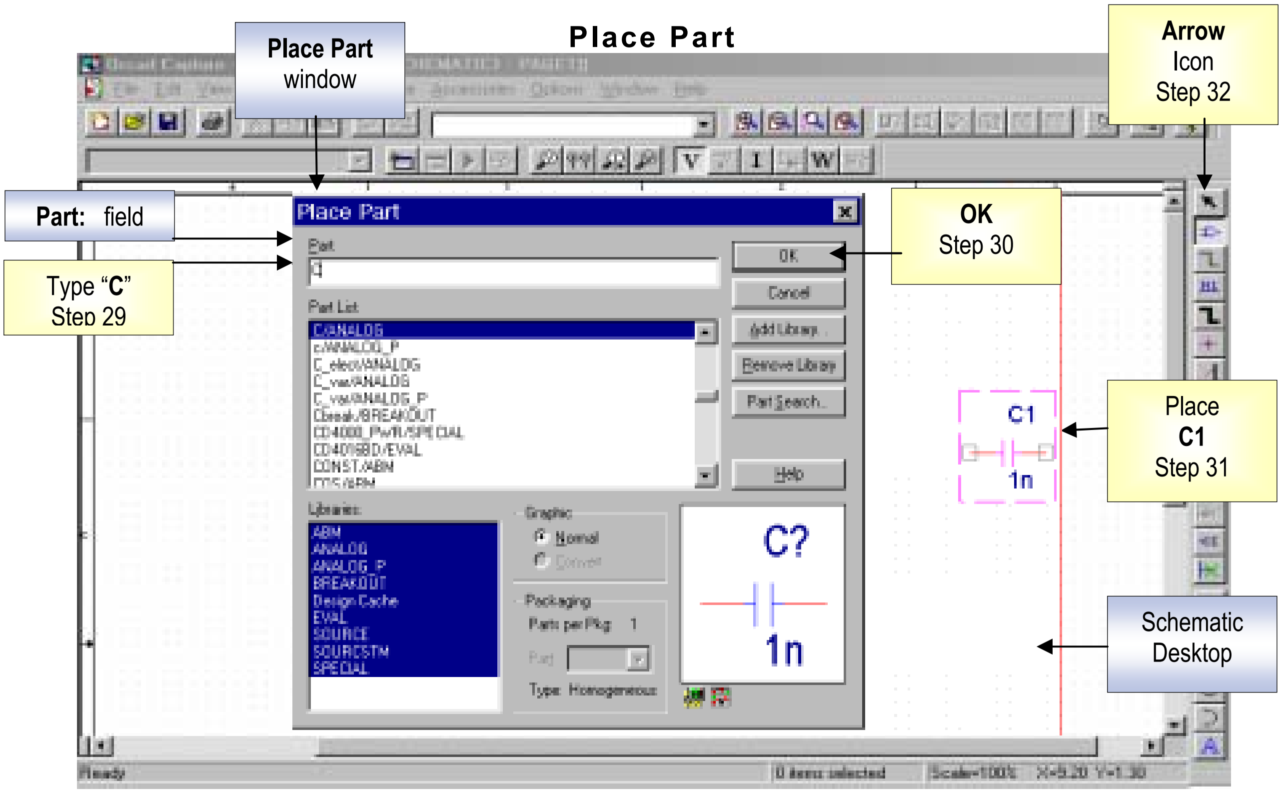
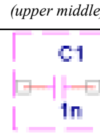
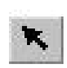


Figure 1-7 Place Part

Instructions	**Icon** *(Position on Figure)*	**Description**
29. From **Place Part** window, type "**C**" or "**c**" in **Part:** entry field . . .		Locate **Capacitor**
30. **CLICKL** on **OK** option.	 *(upper middle)*	Select **Capacitor**
31. **CLICKL** on "**Schematic Desktop**" to place capacitor (**C1**) . . .	 *(center right)*	Place **Capacitor** on schematic desktop
32. **CLICKL** on "**Arrow**" icon . . .	 *(upper right)*	End place **Capacitor** mode
33. **CLICKL** on "**Schematic Desktop**".		End **Place Part** mode

Move, Enlarge, Shrink, and Remove a Part

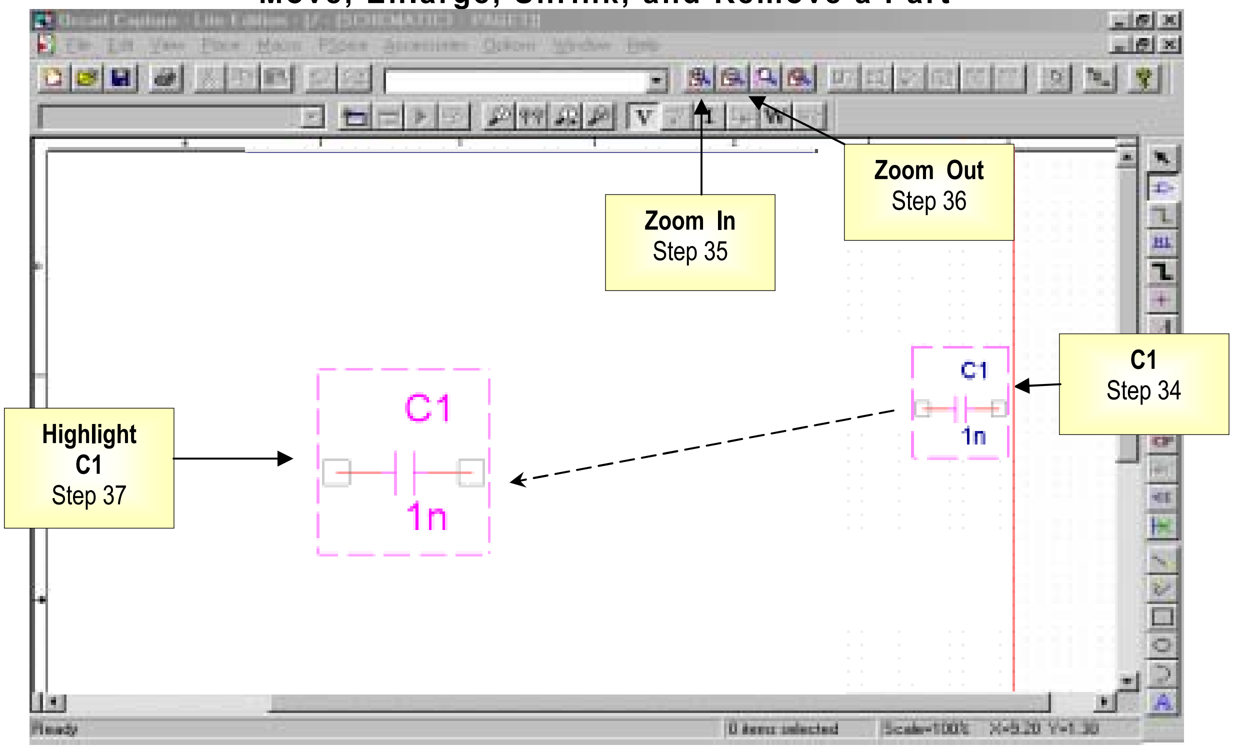
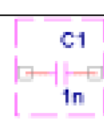


Figure 1-8 Move, Enlarge, Shrink, and Remove a Part

Instructions	Icon (Position on Figure)	Description
34. **CLICKLH** *(click and hold on mouse)* on center of capacitor **C1** to "**drag**" to another location . . .	 *(center right to left)*	Highlight entire **Capacitor** (**C1**) and move to another location
35. **CLICKL** on **Zoom in** icon.	 *(upper center)*	Enlarge **Capacitor** size on schematic desktop
36. **CLICKL** on **Zoom out** icon.	 *(upper center)*	Shrink **Capacitor** size on schematic desktop
37. To remove capacitor, **CLICKL** on center of capacitor "**C1**" to highlight entire part . . .	 *(lower left)*	Highlight **Capacitor** (**C1**) part
38. Press **Delete** key on keyboard.		Remove **Capacitor** from schematic desktop

Undo Command, Save Schematic, Close Project, and Close Program

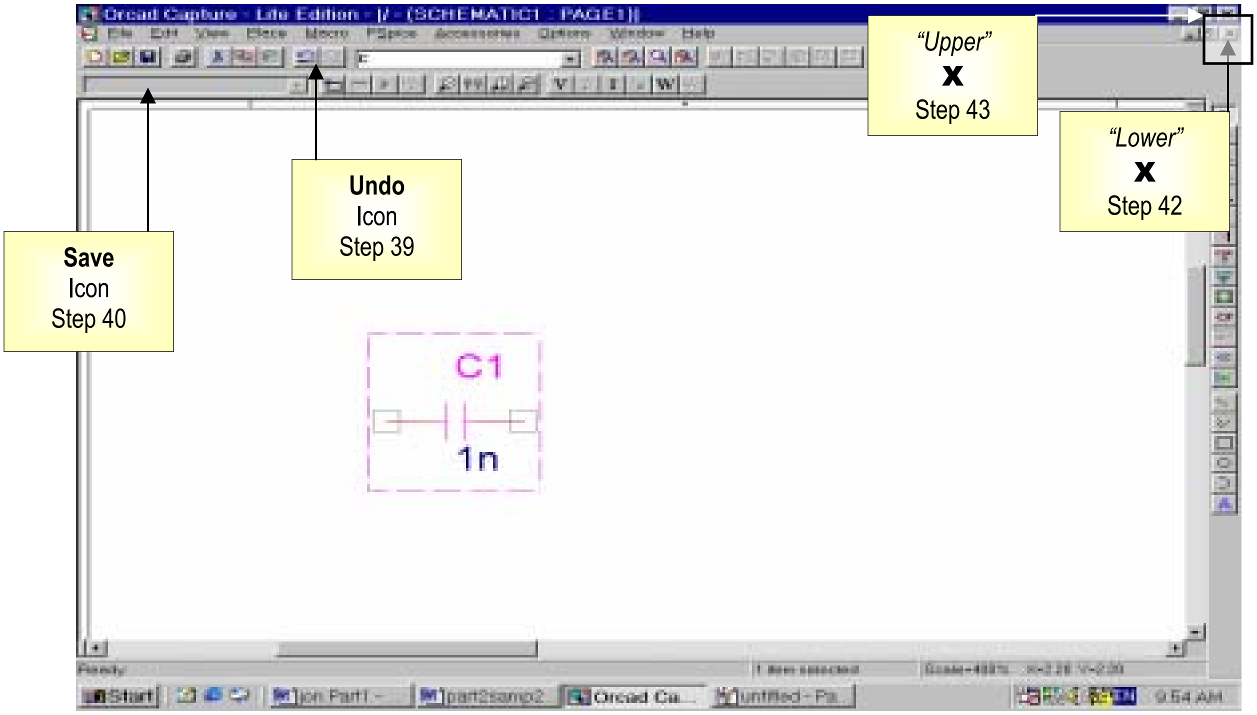
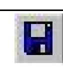
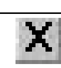
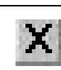


Figure 1-9 Undo Command, Save Schematic, Close Project, and Close Program

<u>Instructions</u>	<u>Icon</u> (Position on Figure)	<u>Description</u>
39. **CLICKL** on **Undo** icon.	 *(upper left)*	**Undo** last command
40. **CLICKL** on **Save** icon.	 *(upper left)*	**Save** schematic to **A:** drive
41. **CLICKL** on **Undo** icon.	 *(upper left)*	**Undo** last command
42. **CLICKL** on "*lower* **X**" **Close** icon.	 *(upper right)*	**Close** "***new project***" - "Schematic" page1 file
43. **CLICKL** on "*upper* **X**" **Close** icon.	 *(upper right)*	**Close** "**PSpice**" folder and Orcad® program

CHECK LIST

Before simulating a **Resistive Circuit**, check the following:

- ✓ **Section 1** of this manual "**Creating a New Project**" is complete
- ✓ **3 ½ Floppy disk with saved PSpice folder is inserted into the correct drive**

MOUSE CONVENTIONS *(consistent with PSpice™printed literature)*

The following mouse conventions are used throughout the manual:

- **CLICKL** (*click left* once) to select an item.
- **DCLICKL** (*double click* left) to end a mode or edit a selection.
- **CLICKR** (*click right* once) to abort a mode.
- **DCLICKR** (*double click right*) *to repeat an action.*
- **CLICKLH** (click left, *hold down, and move mouse*) to drag a selected item. Release left button when placed.
- **DRAG** (*no clicks, move mouse*) to move an item.

BOLD TEXT

A Glossary (at the end of this manual) contains **bold text** terms with definitions according to their use in this manual.

Re-open Orcad Program

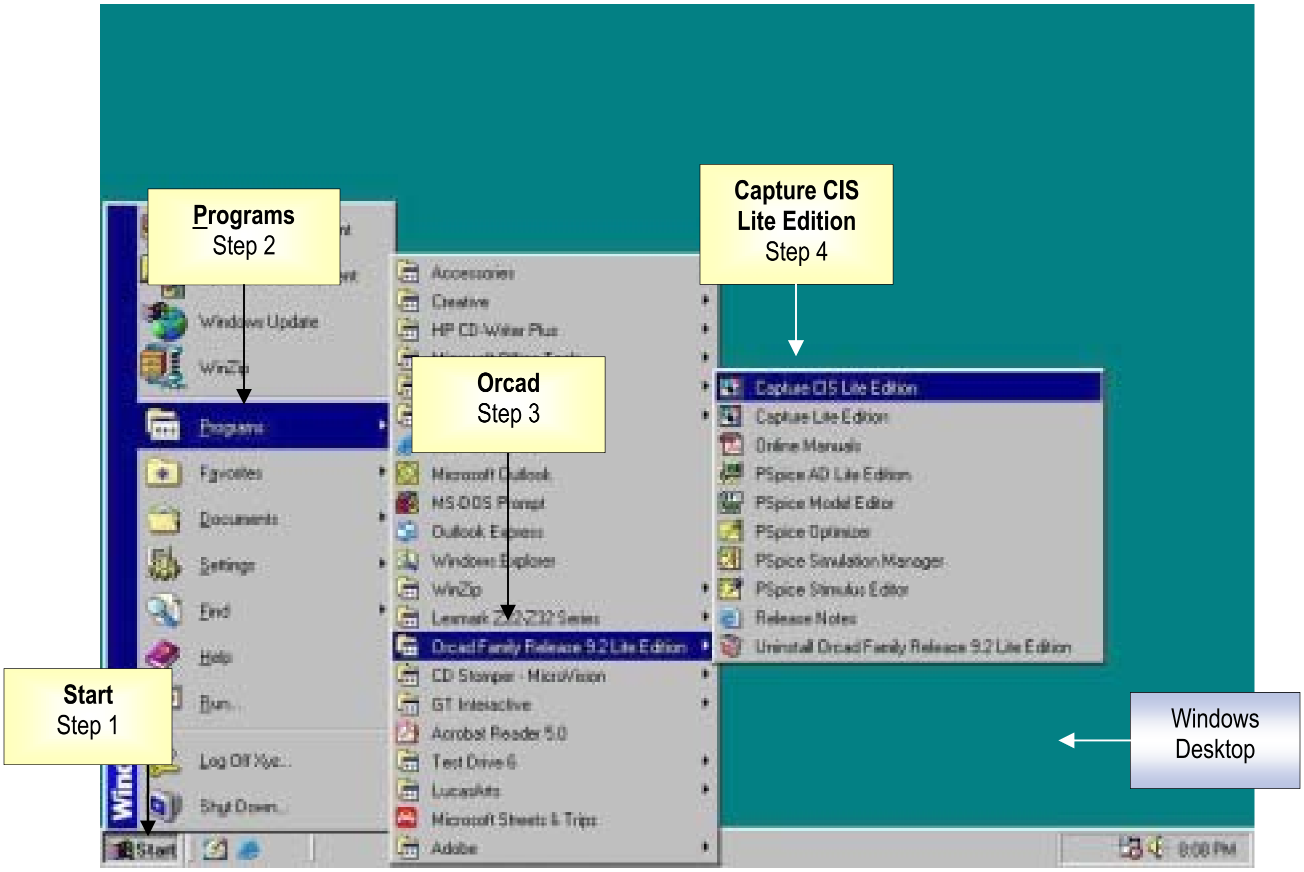


Figure 2-1 Re-open Orcad Program

Instructions	**Icon** *(Position on Figure)*	**Description**
1. **CLICKL** on **Start** icon on **"Windows Desktop"** . . .	 *(lower left)*	Open **Windows** options
2. **CLICKL** on **Programs** option . . .		Open **Programs** menu
3. **CLICKL** on **Orcad (Orcad Family Release Lite Edition)** option . . .		Open **Orcad** program menu
4. **CLICKL** on **Capture CIS Lite Edition** option . . .		Open **Capture – [Session Log]** window

Re-Open PSpice™ Folder and Project

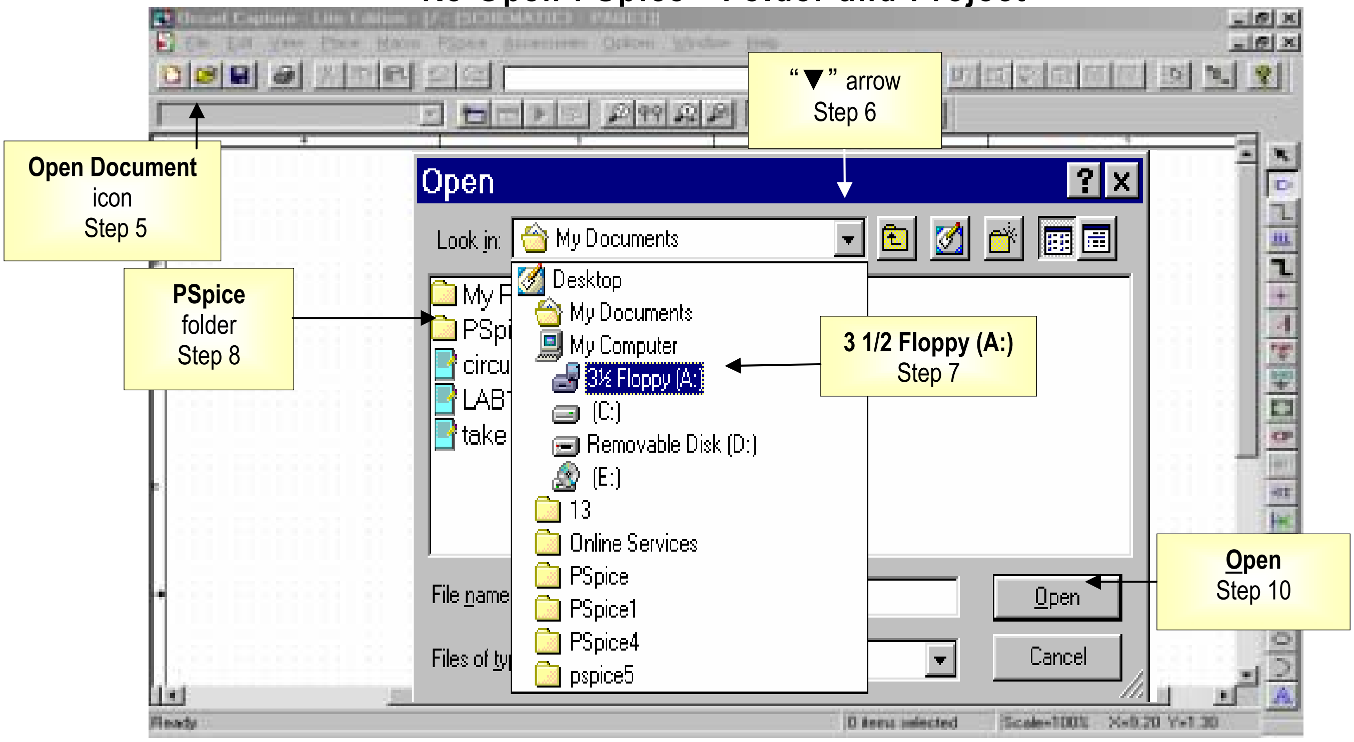


Figure 2-2 Re-Open PSpice™ Folder and Project

Instructions	**Icon** *(Position on Figure)*	**Description**
5. **CLICKL** on **Open Document** icon . . .	*(upper left)*	Open **Open** window
6. **CLICKL** on “▼” arrow in the **Look in:** filed . . .	*(upper middle)*	Open **My Documents** pull-down menu options
7. **CLICKL** on **3 ½ Floppy (A:)*** . . . *Note: (*computer drive letter may vary*)	3½ Floppy (A: *(center)*	Open **3 ½ Floppy (A:)** pull down **folder** options
8. **CLICKL** on **PSpice** folder . . .	PSpice *(center left)*	Select **PSpice** folder
9. **CLICKL** on “***Project Name***” file . . .		Select “***Project Name***” file
10. **CLICKL** on **Open** option.	Open *(lower right)*	Open **Project** file

Open PSpice™ Project Schematic Page 1

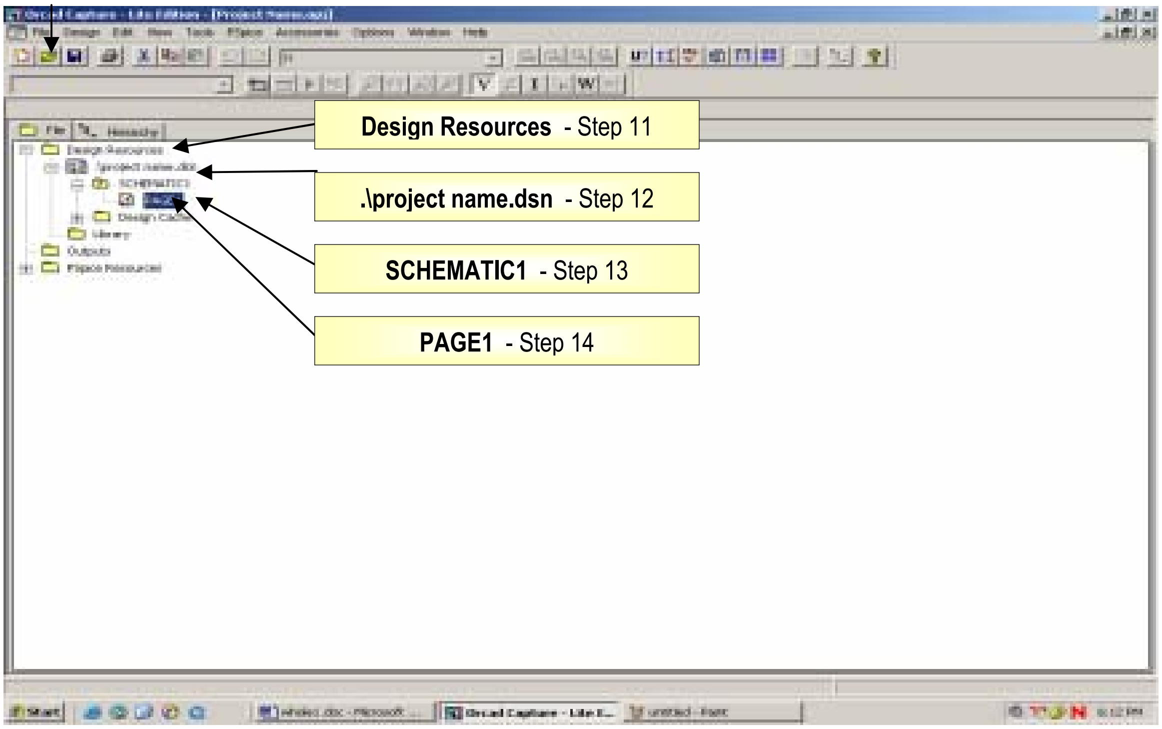


Figure 2-3 Open PSpice™ Project Schematic Page 1

Instructions	Icon *(Position on Figure)*	Description
11. **CLICKL** on **Design Resources.**	Design Resources *(upper left)*	Open **Design Resources** folder
12. **DCLICKL** on ".\project name.dsn project file . . .	.\project name.dsn *(upper left)*	Select **.\project name.dsn** project file
13. **DCLICKL** on **SCHEMATIC1** folder . . .	SCHEMATIC1 *(upper left)*	Open **SCHEMATIC1** folder
14. **DCLICKL** on **PAGE1**.	PAGE1 *(upper left)*	Select **PAGE1** schematic

Place Ground

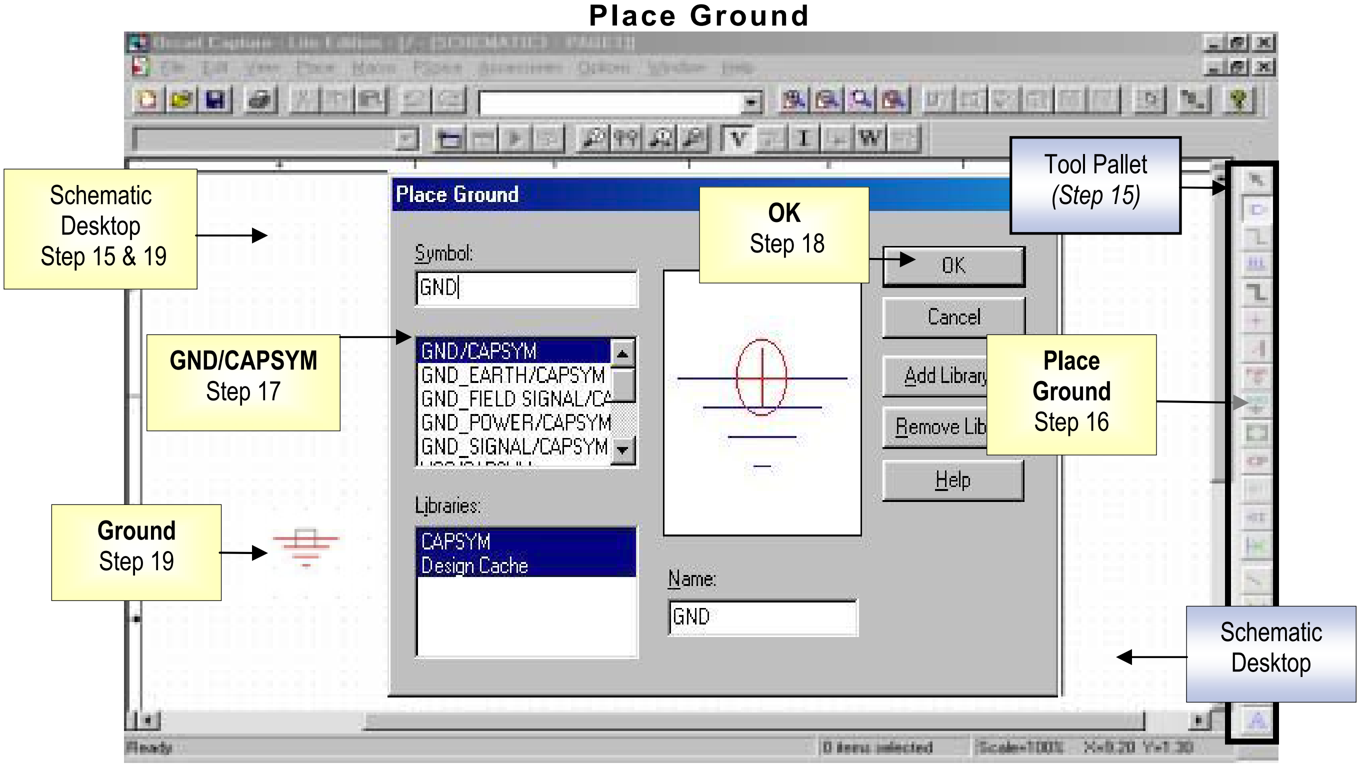
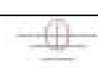
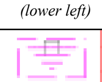


Figure 2-4 Place Ground

Instructions	Icon (Position on Figure)	Description
15. **CLICKL** (click left mouse) any where on Schematic Desktop.		Open **Tool Palette**
16. **CLICKL Place Ground** icon to begin to simulate resistive circuit.	 *(center right)*	Open **Place Ground** window
17. **CLICKL GND/CAPSYM** in window.		Select specific **Ground**
18. **CLICKL OK** to select ground.	 *(upper middle)*	
19. **CLICKL** on "**Schematic Desktop**" . .	 *(lower left)*	Place **Ground**
20. **CLICKR** on **Ground** to highlight . .		Open pull-down menu
21. **CLICKL** on **End Mode**.		End **Place Ground** mode

Assign Value to Ground

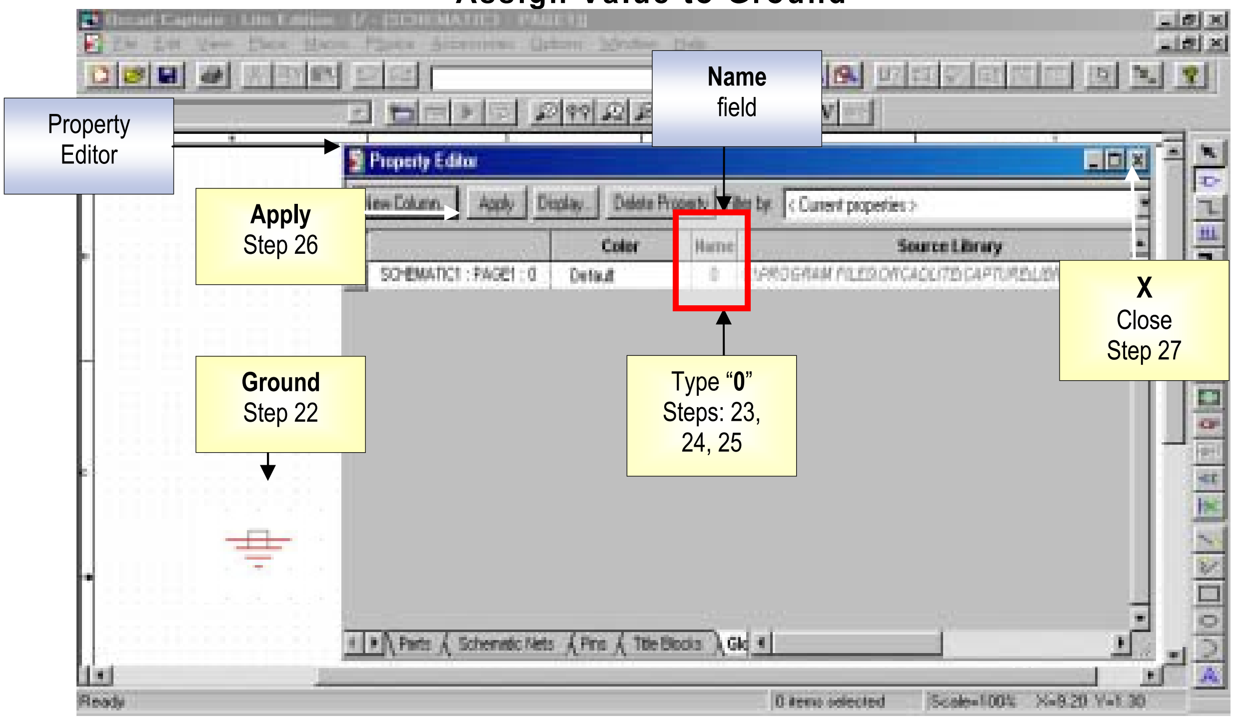
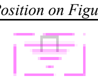


Figure 2-5 Assign Value to Ground

Instructions	Icon *(Position on Figure)*	Description
22. **DCLICKL** on **Ground** symbol on schematic desktop . . .	 *(lower left)*	Highlight **Ground** and open **Property Editor** window
23. * Place cursor <u>after</u> "D" in "**GND**" in **Name:** field of **Property Editor** window.	 *(upper middle)*	*Caution: Program will **<u>close</u>** if "**GND**" is highlighted to delete.
24. Backspace to remove "**GND**" . . .		
25. Type "**0**" in **Name** field of **Property Editor** . . .		Set **Ground** value at "**0**"
26. **CLICKL** on **Apply** . . .	 *(upper left)*	Assign **Ground** value "**0**"
27. **CLICKL** on **Close** icon in **Property Editor** window.	 *(upper right)*	Close **Property Editor** window

Place Resistor

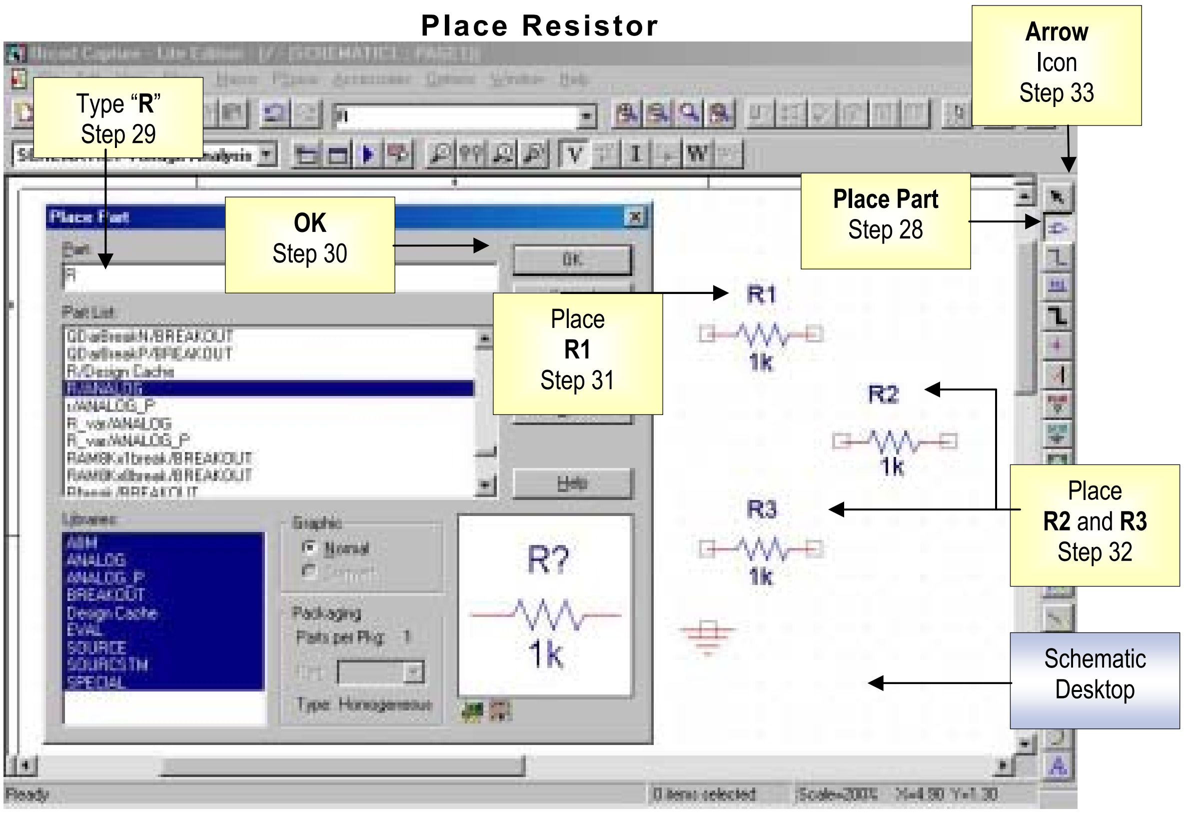
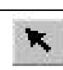


Figure 2-6 Place Resistor

Instructions	**Icon** *(Position on Figure)*	**Description**
28. **CLICKL** **Place Part** icon to begin to place first resistor . . .	 *(upper right)*	Open **Place Part** window
29. Type "**R**" (or "**r**") in **Part:** entry field . . .		Locate a **Resistor**
30. **CLICKL** on **OK** . . .	 *(upper middle)*	Select **Resistor**
31. **CLICKL** on "**Schematic Desktop**" to place first resistor (**R1**).	 *(upper middle)*	Place **Resister**
32. **CLICKL** on "**Schematic Desktop**" to place more resistors (**R2**, **R3**).		Place more **Resistors**
33. **CLICKL** on "**Arrow**" icon.		End **Place Part** mode

Rotate Resistor

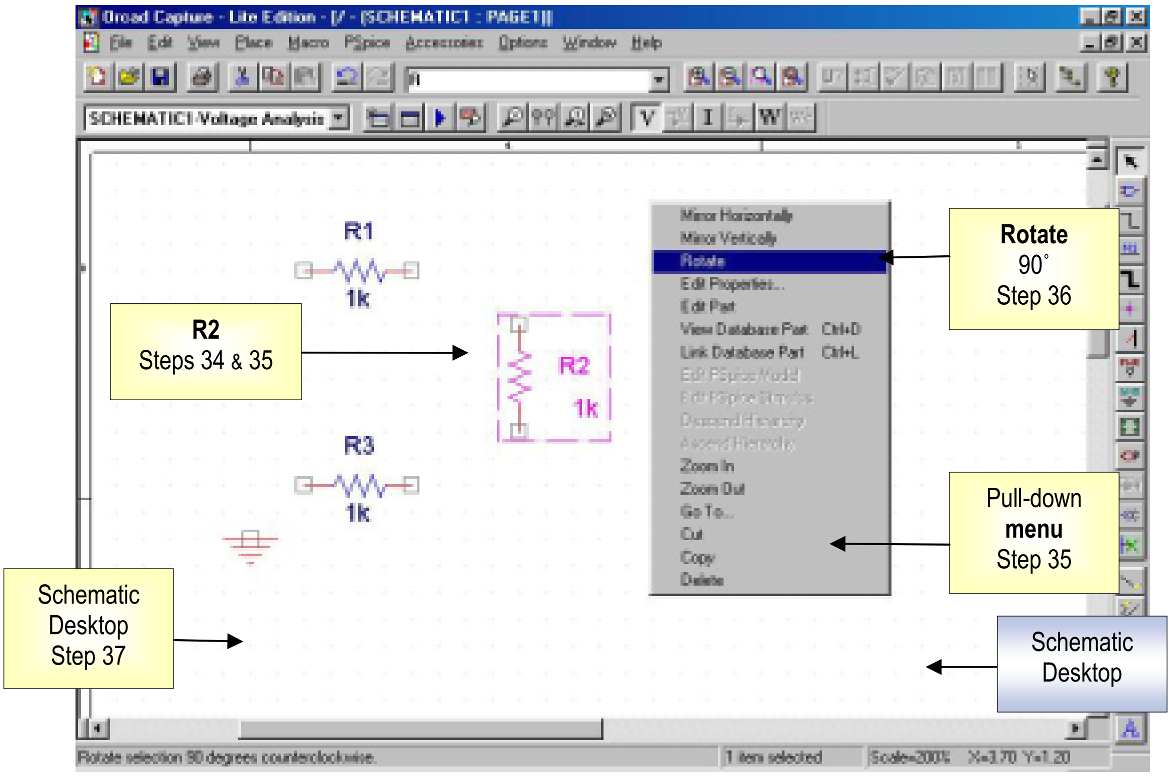
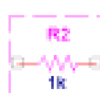
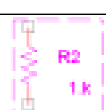


Figure 2-7 Rotate Resistor

Instructions	**Icon** *(Position on Figure)*	**Description**
34. **CLICKL** on center of **Resistor (R2)** symbol to highlight . . .	 *center*	Highlight entire **R2**
35. **CLICKR** on center of **Resistor (R2)** symbol . . .		Open pull-down **menu**
36. **CLICKL** on **Rotate** in pull-down menu . . .		**Rotate Resistor** 90˚ on schematic desktop
37. **CLICKL** on "**Schematic Desktop**" to end **Rotate** mode.		End **Rotate** mode

Change Resistor Value(s)

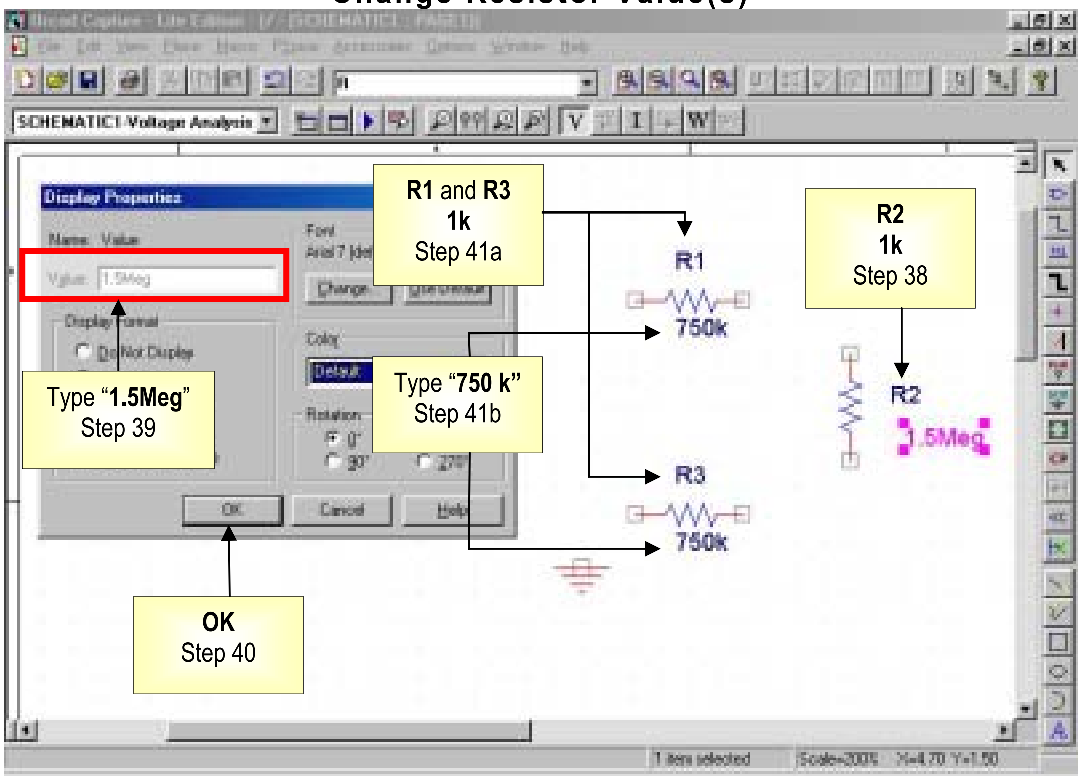
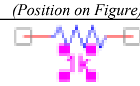


Figure 2-8 Change Resistor Value(s)

Instructions	**Icon** (*Position on Figure*)	**Description**
38. **DCLICKL** on "**1k**" of **R2** in the to change ohm value of **Resistor**.	 *(center right)*	Open **Display Properties** window
39. *Type "**1.5Meg**" (MEG or meg) for 1.5 M* in **V**<u>**a**</u>**lue:** entry field. (**PSpice does not recognize "M"*)	 *(upper left)*	* Caution: **DO NOT** highlight **1k** to change **Resistor** (**R2**) value from **1k Ω** to **1.5M Ω**
40. **CLICKL** on **OK** to close screen.	 *(lower left)*	Close **Display Properties** window
41. Repeat steps 34 – 36 to change: a. **R1** from "**1k**" to: type "**750k**" (or 750K). b. **R3** from"**1k"** to: type "**750k**".		Change **Resistors** value(s) from **1k Ω** to **750k Ω**

Place Power Supply

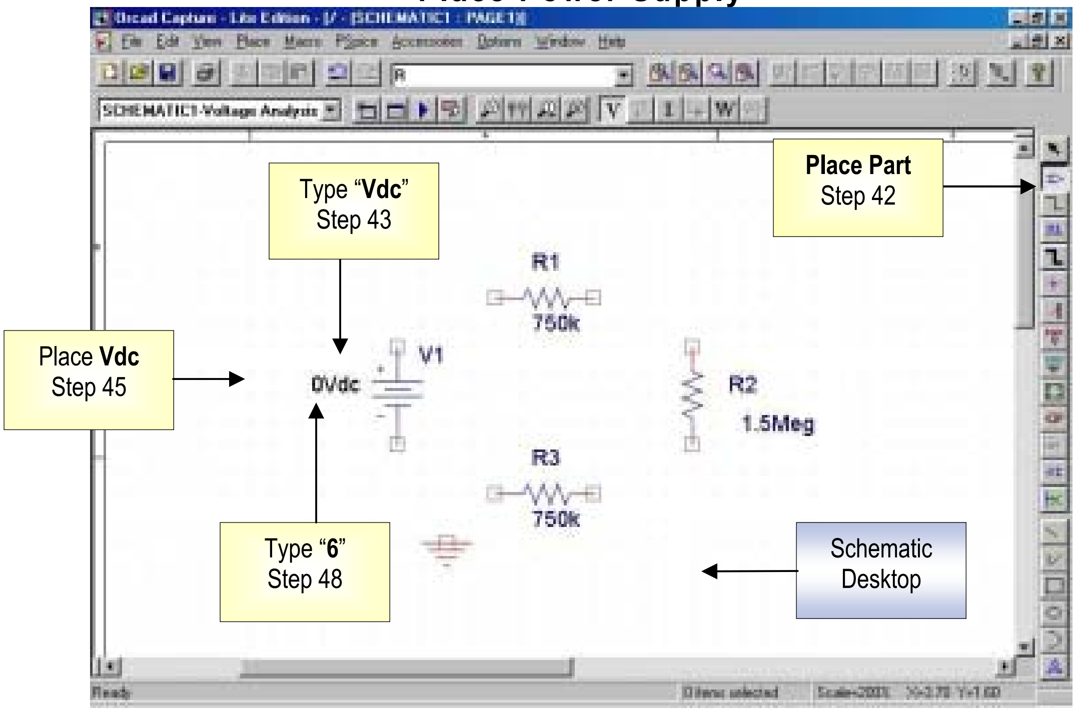
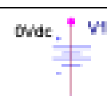


Figure 2-9 Place Power Supply

Instructions	**Icon** *(Position on Figure)*	**Description**
42. **CLICKL** on **Place Part** icon to place power supply in circuit.	 *(upper right)*	Open **Place Part** window
43. Type "**Vdc**" in **Part:** entry field . . .		Locate **Power Supply**
44. **CLICKL** **OK** to select a dc voltage power supply (source voltage) . . .	OK	Selects **Vdc** power supply
45. **CLICKL** on "**Schematic Desktop**".		Place **Vdc** in circuit
46. Press "**ESC**" key on upper left of keyboard.		End **Place Part** mode
47. **DCLICKL** on "**0Vdc**" of Power Supply . . .		Opens **Display Properties** window
48. Type "**6**"**Vdc** in **Value:** field . . .	6Vdc	Change from **0** to **6 Vdc**
49. **CLICKL** on **OK** option.	OK	Select **6Vdc** as source V

Wire Circuit

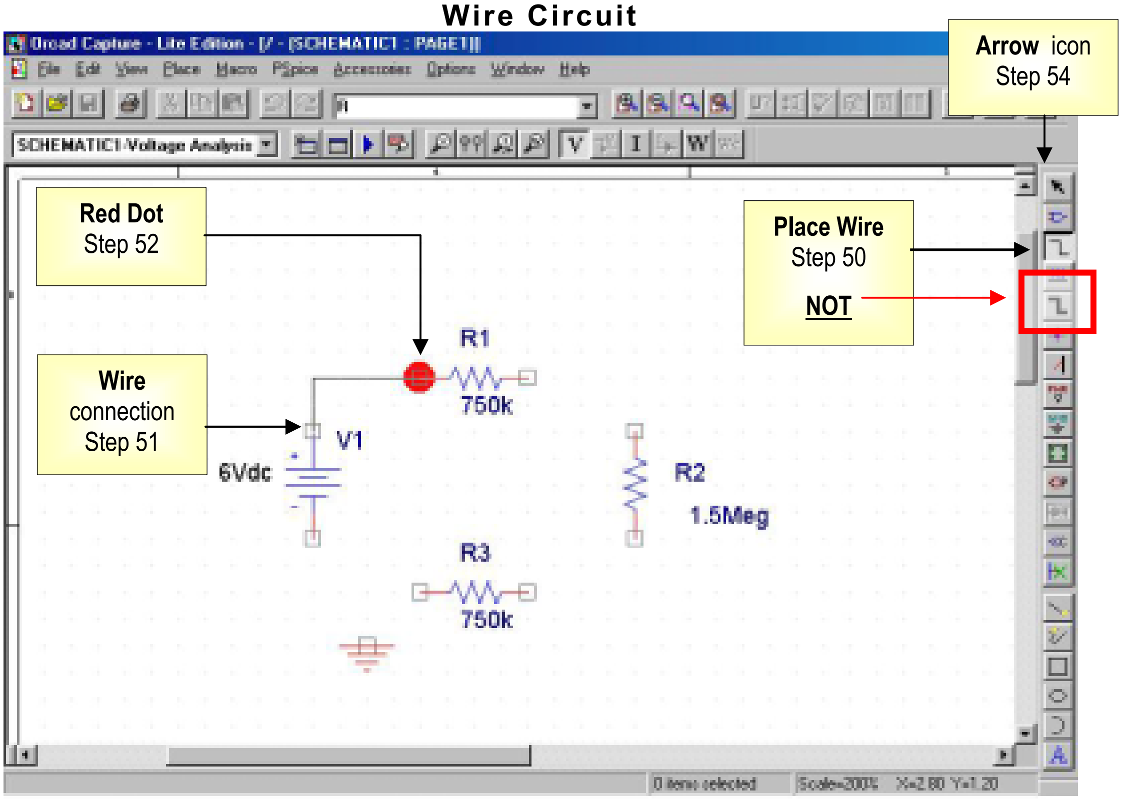
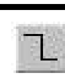
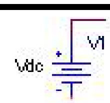
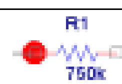
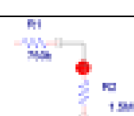


Figure 2-10 Wire Circuit

Instructions	**Icon** *(Position on Figure)*	**Description**
50. ***CLICKL** on "**Place Wire**" icon To begin to wire circuit . . .	 *(upper right)*	* Caution: See Figure 2-9 Open **Place Wire** mode
51. Place cursor over the "□" top end of **Vdc**, then **CLICKL** . . .		Attach first part of **Wire**
52. Move cursor to end of next part (**R1)** until the "•" appears, then **CLICKL**.		Attach last part of **Wire**
53. Continue to **Place Wire(s)** between parts until continuous path is formed.		Note: **DO NOT** wire **Ground** yet
54. **CLICKL** on "**Arrow**" icon.	 *(upper right)*	End **Place Wire** mode

Place Junction (Node)

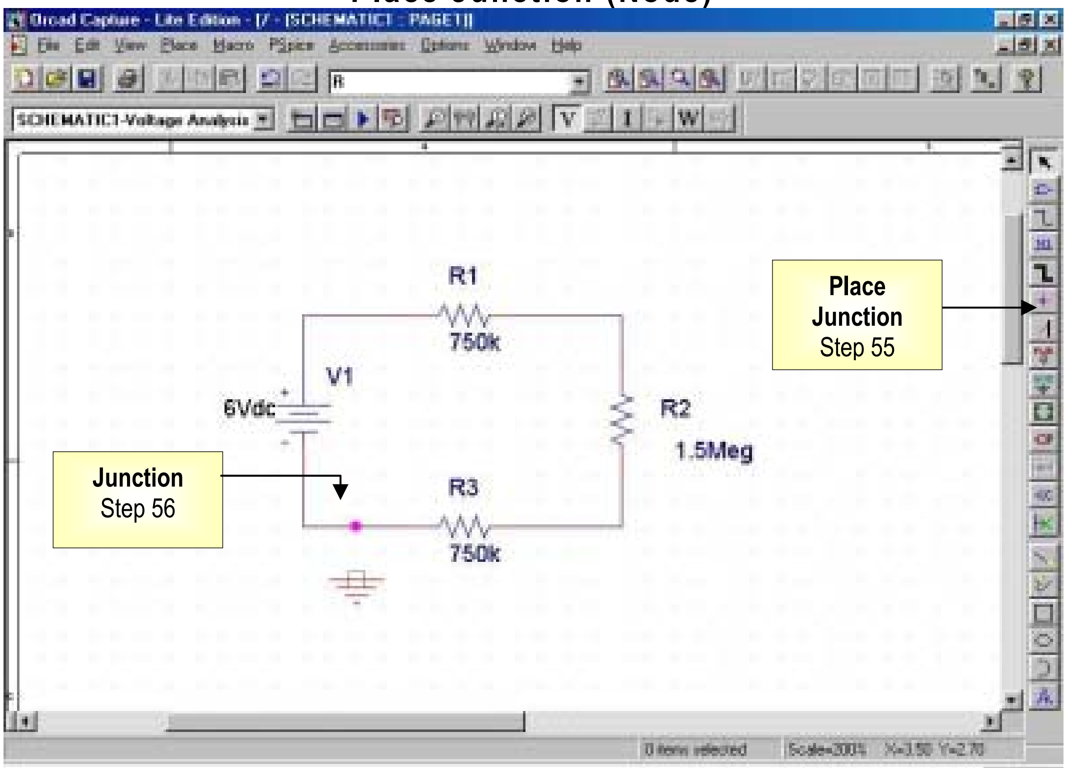
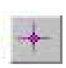
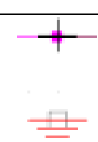
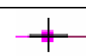


Figure 2-11 Place Junction (Node)

Instructions	**Icon** *(Position on Figure)*	**Description**
55. **CLICKL** on "**Place Junction**" icon to begin to connect **Junction** to **GND** . . .	 *(center right)*	Open **Place Junction** mode
56. **CLICKL** on wire in circuit where junction is desired . . .		Place **Junction** in **Wire**
57. **CLICKR** on **Junction** . . .		Opens pull-down menu
58. **CLICKR** on **End Mode**.		Ends **Place Junction** mode

Wire Junction to Ground

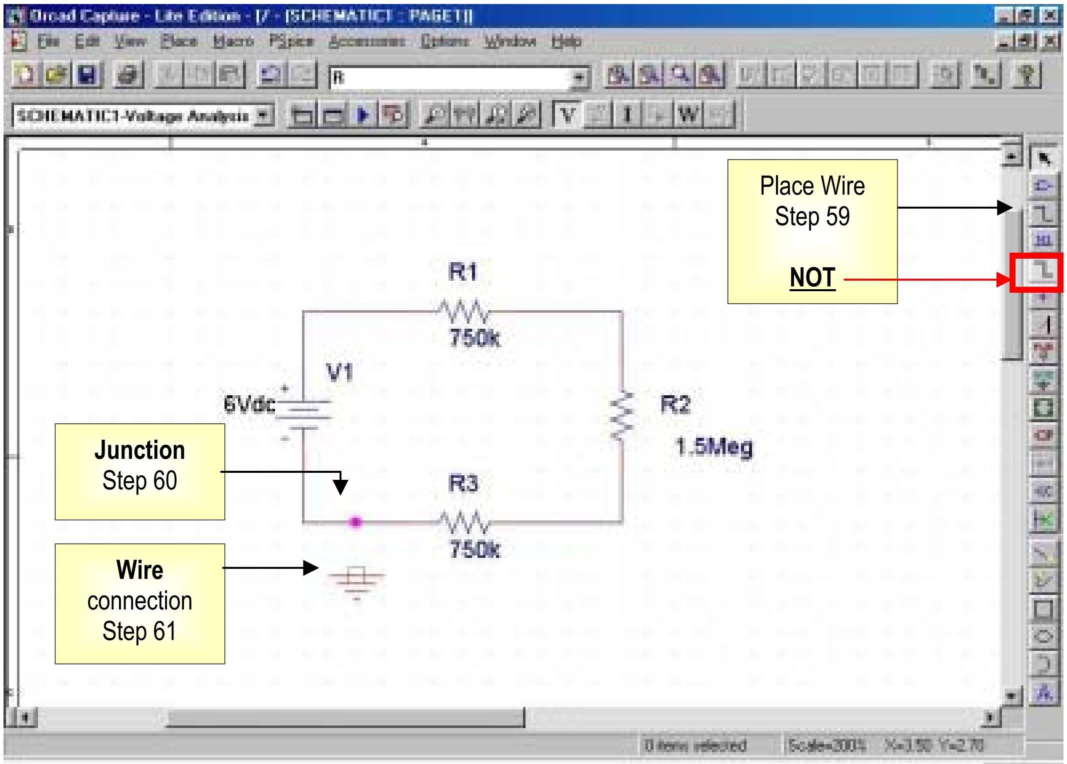
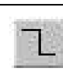
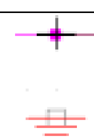
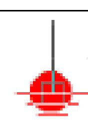


Figure 2-12 Wire Junction to Ground

Instructions	Icon *(Position on Figure)*	Description
59. **CLICKL** on "**Place Wire**" icon to begin to wire **Junction** to **GND**.	 *(upper right)*	Open **Place Wire** mode
60. Place cursor "+" over **Junction** at node "•"; then **CLICKL** . . .		Attach first part of **Wire** to **Junction**
61. Move cursor "+" over the top of **GND** symbol until a "•" appears, then, **CLICKL** . . .		Attach last part of **Wire** to **Ground**
62. Press "**ESC**" key on keyboard.		End **Place Wire** mode

Create a New Simulation File

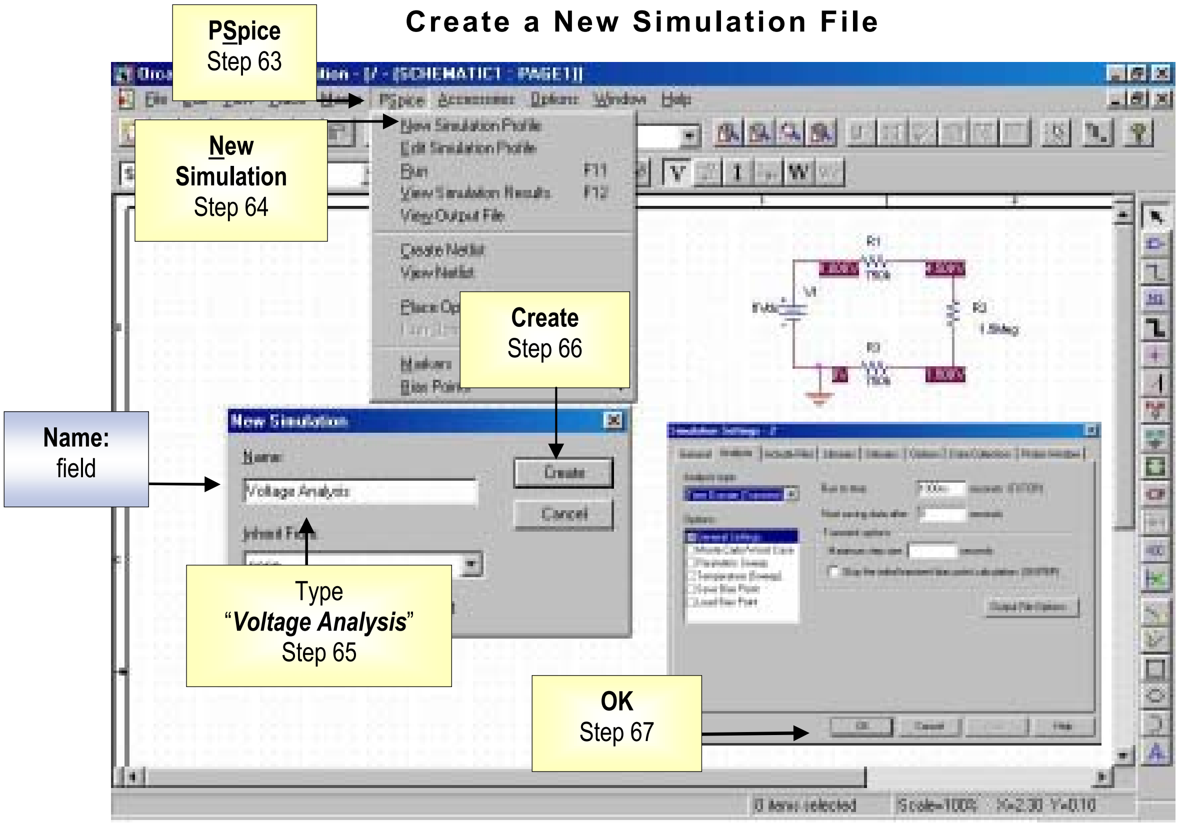


Figure 2-13 Create a New Simulation File

Instructions	Icon *(Position on Figure)*	Description
63. **CLICKL** on **PSpice** option in menu bar . . .	PSpice *(upper left)*	Open **New Simulation** window
64. **CLICKL** on **New Simulation Profile** . . .	New Simulation Profile *(upper left)*	Open **New Simulation Profile** file
65. Type "***Voltage Analysis***" in **Name:** field . . .		Name **New Simulation Profile** file
66. **CLICKL** on **Create** .	Create *(center)*	Close **New Simulation** window
67. **CLICKL** on **OK** option.	OK *(lower right)*	Return to **Schematic Desktop**

Voltage Analysis

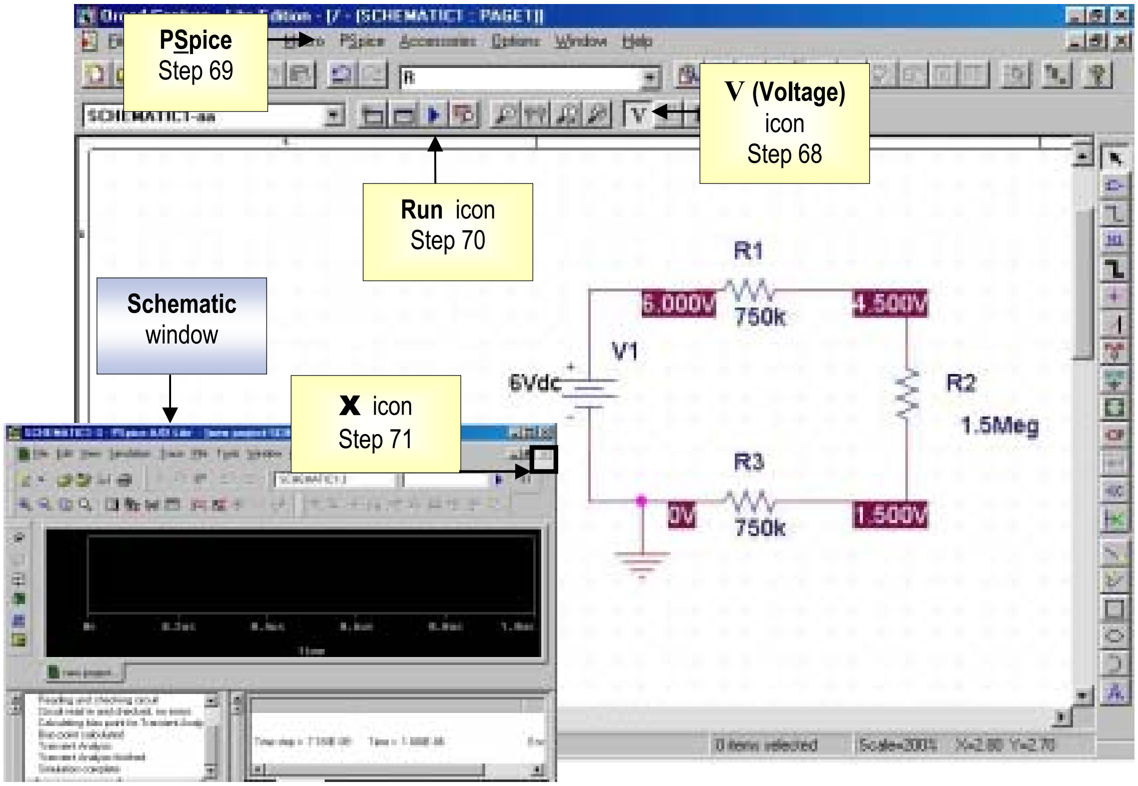


Figure 2-14 Voltage Analysis

Instructions	**Icon** *(Position on Figure)*	**Description**
68. **CLICKL** on “**V**” (**Voltage)** icon *(may already be selected)*. . .	V *(upper middle)*	Select **Voltage** analysis for every node in circuit
69. **CLICKL** on **PSpice** option in menu bar . . .	PSpice *(upper left)*	Open pull-down **PSpice** menu options
70. **CLICKL** on **Run** icon . . .	 *(upper left)*	Open “**New Simulation**” file
71. **CLICKL** on **X** of **Schematic** window to close and view **voltage**.	X *(upper left)*	Close **Schematic** window

Current Analysis in Amperes and Power Analysis in Watts

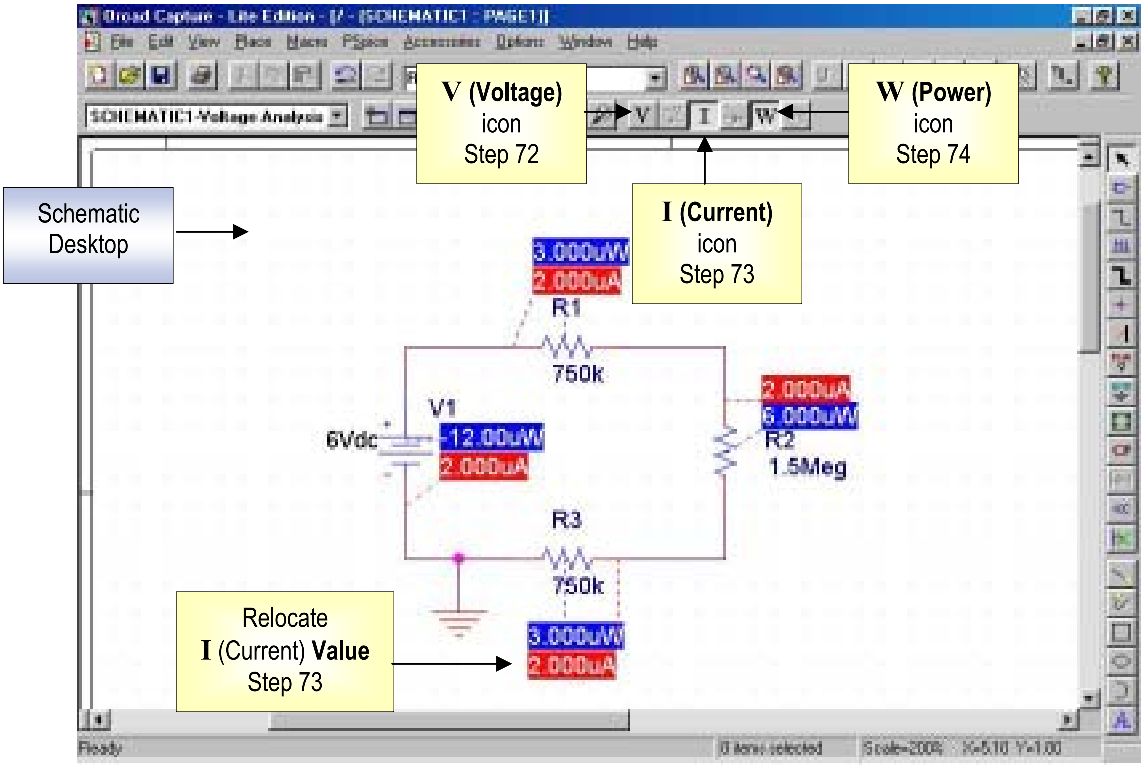


Figure 2-15 Current Analysis in Amperes and Power Analysis in Watts

Instructions	**Icon** *(Position on Figure)*	**Description**
72. **CLICKL** on "**V**" (**Voltage**) icon.	V *(upper middle)*	Deselect **Voltage** values from every node in circuit
67. **CLICKL** on "**I**" **Current** icon . . .	I *(upper middle)*	Select **Current** analysis for every node in circuit
68. **CLICKL** on "**W**" **Power** icon . . .	W *(upper middle)*	Select **Power** analysis for every node in circuit
69. **CLICKH** (*click and hold on mouse)* on center of any analysis value to highlight and **drag** to new location. .	2.000uA *(lower middlet)*	Highlight and **drag** to new location
70. **CLICKL** on "**I**" and "**W**" icon	I W *(upper middlet)*	Deselect **Current** and **Voltage** analysis

Save File, Print Schematic, and Close Program

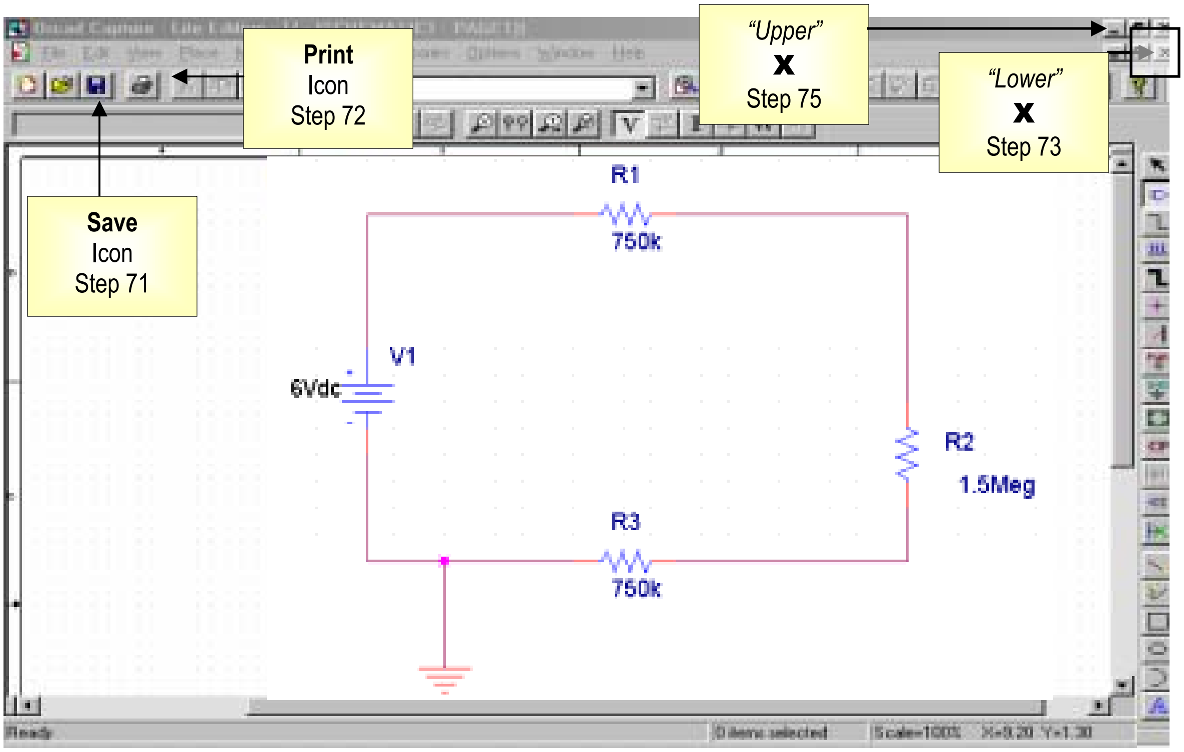


Figure 2-16 Save File, Print Schematic, and Close Program

Instructions	Icon (Position on Figure)	Description
71. **CLICKL** on **Save** icon.	*(upper left)*	**Save** schematic to **A:** drive
72. **CLICKL** on **Print** icon . . .	*(upper left)*	Print **Resistive Circuit** simulated schematic
73. **CLICKL** on “*lower* **X**” **Close** icon.	*(upper right)*	**Close** “***new project***” - “Schematic” page1 file
74. **CLICKL** on **Yes** option.	Yes	Save changes to schematic page.
75. **CLICKL** on “*upper* **X**” **Close** icon . . .	*(upper right)*	**Close** “**PSpice**” folder and Orcad® program
76. **CLICKL** on **Yes All** option.	Yes All	**Save All** to floppy disk

GLOSSARY

Note: *The terms are defined according to their use in the manual.*

Ω (symbol) Ohms, a unit of resistance

1k 1,000; a resistor value in Ohms (Ω)

1Meg 1,000,000; a resistor value in Ohms (Ω)

750k 750,000; a resistor value in Ohms (Ω)

Analog or Mixed A/D Option for Analog or Analog and Digital

Arrow (icon) End current mode

Browse Search through file directories

Capture Lite Edition Open PSpice [Session log]

CLICKL Click Left on mouse

CLICKLH Click Left and Hold on mouse to drag

CLICKR Click Right

Create PSpice Project Open a new document and blank project

DC Direct Current

DC Direct Current

DCLICKL Double Click Left on mouse

Drag Moves the highlighted object on desktop

GND/CAPSYM Specific ground type

Edit Properties Pull-down menu with editing options

End Mode End the mode of selected function

I Current analysis option in Amperes

Junction A node or specific place in the wire of between components

New Open new folder on “Windows” desktop

New Project Menu options for new project

Orcad Family Release 9.2 Lite Edition Program that contains PSpice™

GLOSSARY

Continued…

Part : Entry field for part search

Place Ground Mode to place selected ground symbol on schematic desktop

Place Junction Mode to place junction node in wire

Place Part Mode to locate and place selected part on schematic desktop

Place Wire Mode to place wire between parts on schematic desktop

Power Supply Voltage source, i.e. Vdc, Vac

Programs ► Open “Program” menu options

Property Editor Window with component property options

R Resistor

R2 Resistor label for 2nd resistor

Rotate Mode to rotate a selected part 90°

Select Directory Open “Directory” options

Send To 3 ½ Floppy ► Pull-down menu option to move folder from “Windows” desktop to (A:\) drive

V Voltage analysis option in volts

Vac Voltage source with Alternating Current

Vdc Voltage source with Direct Current

W Power analysis option in Watts