

---

**Information in this manual is subject to change without notice. No liability is assumed for damages resulting from the use of this information or from the software described. Said software is furnished with the understanding that only one copy may be made for backup purposes. No part of this manual may be reproduced or transmitted in any form or by any means without the written permission of Mental Automation.**

**Microsoft ®, Windows™, and MS ® are registered trademarks of Microsoft Corporation.**

***SuperCAD™/Mobile***  
***For Windows Mobile 2003 and 5.0***

**© 1992-2006 Mental Automation, Inc.**

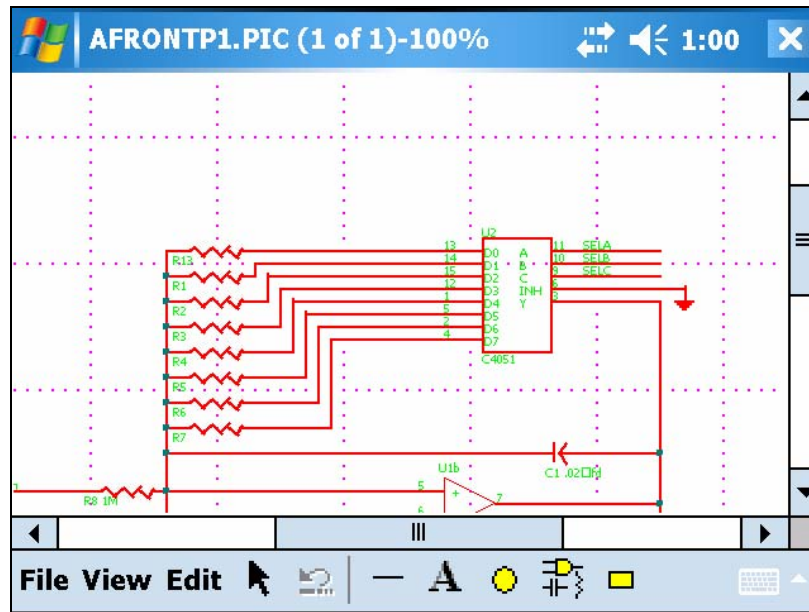
---

# Contents

<b>1 Introduction</b> .....	<b>3</b>
1.1 SCHEMATIC EDITOR FEATURES .....	3
1.2 INSTALLATION.....	4
1.3 RECOMMENDED SYSTEM.....	5
1.4 GETTING STARTED .....	6
1.4.1 TAP OPERATION .....	6
1.4.2 CREATING A NEW SCHEMATIC.....	6
1.4.2.1 PLACING PARTS.....	6
1.4.2.2 WIRING THE SCHEMATIC.....	7
1.4.2.3 ADDING NET NAMES.....	7
1.4.2.4 SAVING THE SCHEMATIC.....	8
1.4.2.5 USING OTHER FEATURES.....	8
<b>2 Edit Operations</b> .....	<b>8</b>
2.1 ROTATE.....	9
2.2 MIRROR.....	9
2.3 VERTICAL FLIP.....	9
2.4 REFERENCE NUMBER.....	9
2.5 RE-GRID OPERATION .....	10
2.6 UNDO .....	10
2.7 MOVE.....	10
2.8 STRETCH .....	10
2.9 INSERTING OBJECTS .....	10
2.9.1 ONE TAP OBJECTS.....	11
2.9.2 TWO TAP OBJECTS .....	12
<b>3 File Operations</b> .....	<b>12</b>
3.1 SAVE AND SAVE AS COMMANDS.....	12
3.2 OPEN COMMAND.....	12
3.3 PRINT OPERATION .....	12
3.4 PAGE ITEMS.....	13
3.4.1 NEW PAGE.....	13
3.4.2 DELETE CURRENT PAGE.....	13
3.4.3 NEXT PAGE.....	13
3.4.4 PREVIOUS PAGE.....	13
3.4.5 EXIT SUPERCAD.....	13
3.5 VIEW OPERATIONS .....	13
3.6 SETUP .....	14
3.6.1 SHEET SIZE.....	14
3.6.2 GENERAL SETUP.....	14
3.6.3 SELECTIVE EDITING.....	14

# 1 Introduction

SuperCAD/Mobile is a fully functional schematic editor that runs under Windows Mobile 2003 or 5.0. A typical screen is shown below.



## 1.1 Schematic editor features

SuperCAD/Mobile is compatible with the desktop PC-based SuperCAD which runs on Windows 95/98/XP or 2000

- File and library part compatible with SuperCAD V6.0
- Schematics with sheet sizes A-E
- Multiple sheets per schematic—up to 5 sheets
- Easy object placement (wires, parts, and other objects)
- Automatic connector dot placement
- Automatic bus generation

SuperCAD does not include every feature of the full PC-based program; here are some of the missing features:

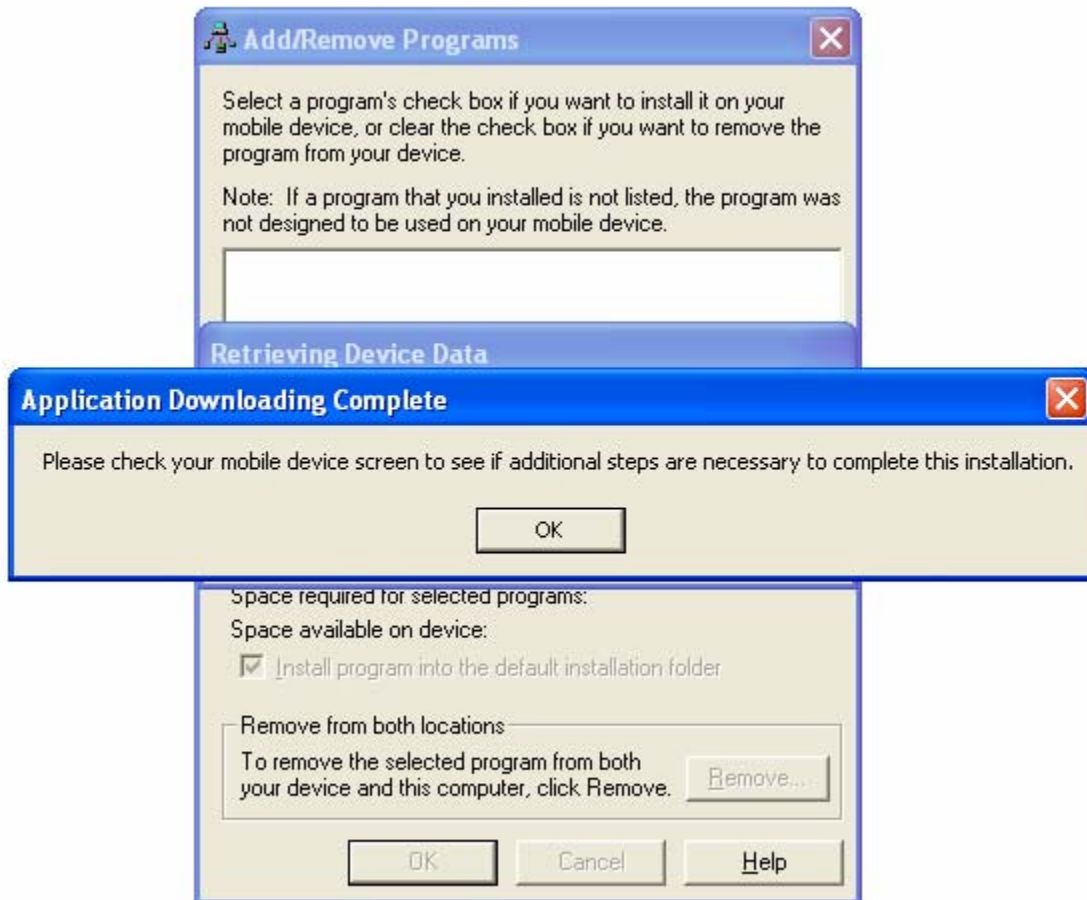
- Printing (use the evaluation or PC version)
- Netlister (use the PC version)
- Library Parts Editor (use the evaluation or PC version)
- Simulation (PC version)

## 1.2 Installation

Installation of SuperCAD requires two files:

- Register.bat
- SuperCAD.CAB

To install, place these files in the same directory, and start **ActiveSync** (which comes with the Pocket PC). Then click on Register.bat. This will start the Add/Remove Programs utility in ActiveSync and install the software. The screen is shown below.



In the pocket PC you will get a screen similar to the following (for Windows Mobile 5.0—you can safely ignore the message about the version of Windows Mobile, since SuperCAD runs on both Windows Mobile 2003 and 5.0)



The SuperCAD files are installed as follows:

<b>Files</b>	<b>Location</b>
SuperCAD executable, Title.PIC	Program Files\SuperCAD
Library parts (.CON files)	Program Files\SuperCAD\LIB
Audio prompts	Program Files\SuperCAD\Sounds
SuperCAD Help file	Windows
SuperCAD example files	My Documents

Using ActiveSync you can transfer your custom library files to the **LIB** directory, and you can transfer .PIC file to/from the **My Documents** directory. SuperCAD cannot open files in the SuperCAD directory itself.

Note: you should run SuperCAD in **Landscape** mode. To set this mode, tap **Settings** in the **Start** menu and then tap on the **System** tab. Then tap on the **Display** icon.

### ***1.3 Recommended System***

SuperCAD runs best on the latest Pocket PC's having a VGA resolution of 640 x 480; older Pocket PC's typically have 320 x 240 resolution and should also work, but you will do more scrolling to view/edit a drawing.

## ***1.4 Getting Started***

### ***1.4.1 Tap Operation***

All drawing in SuperCAD/Mobile relies on using the PDA stylus to either place objects or to select/manipulate them. To place objects (such as a library part or line), first select the object type either from the toolbar or from the Edit/Insert Object menu item. Objects are either placed with a single tap (text and library parts), or with two taps (lines, ellipses, rectangles and others).

To edit single objects, first enter edit mode by tapping the **Edit** button (or tapping the Edit/Select Objects menu item). Then select single objects by tapping on them; if an object is selected, then object handles will be displayed. For the most part, the object handles are small solid rectangles located at the object stretch points. In the case of library parts only a dashed rectangle about the part shows when a part is selected.

Note that diagonal lines can be selected by clicking at the midpoint of the line.

To edit multiple objects first tap you also tap on the **Edit** button, but you draw an edit box about the objects you want to select: 1) tap at the first corner (upper left) of the edit box and 2) tap at the second corner (lower right) of the box; a dashed rectangle then shows the outline of the edit box. Make sure you do not tap on top of an object or you will select that object.


To unselect an object or objects, tap the toolbar **Edit** button.

### ***1.4.2 Creating a new Schematic***

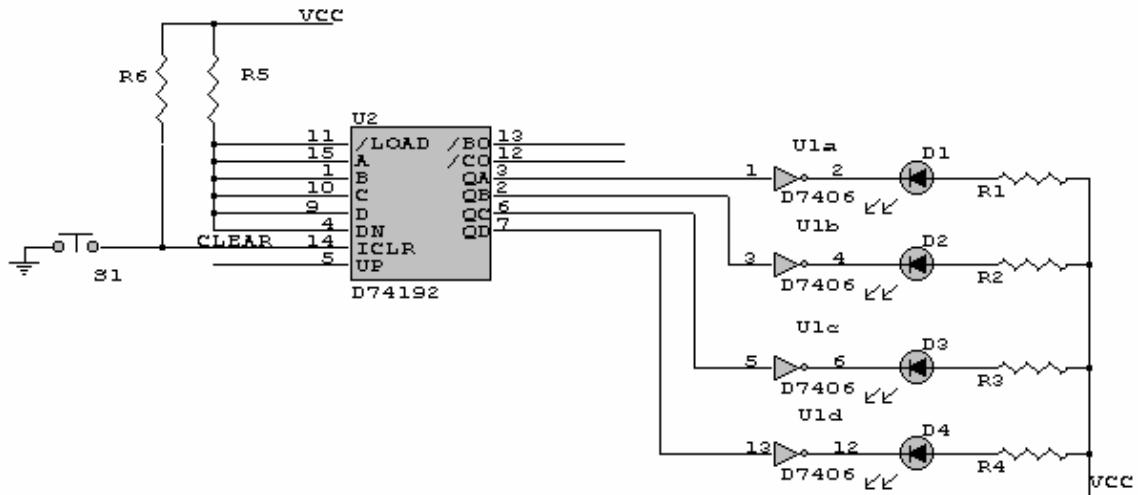
To start a new schematic, tap on **New** in the File menu. This clears the screen so that we can start a new drawing. If there is any work already on the screen, you will be prompted to save it. The default name for a new schematic is “untitled”. Change the name by tapping on **Save As...** in the **File** menu. Enter a name of up to 80 characters; you do not have to add an extension. Then tap on the Ok button. The new name will now show up in the SuperCAD window title bar and also in the title block of the schematic.

#### ***1.4.2.1 Placing Parts***

The example schematic requires 6 different kinds of parts: a D74192 decade counter, a D7406 inverter, a resistor, a LED (diode), a push button, and a ground symbol.

To load the D74192, tap on the library button . Select the GENXXX library in the library dialog box and select the D74192 from the part section. You will have to slide the scrollbar button to find the part. When you find it, tap once on it and tap on the **Load**

**Part** button. Then place the cursor at the center of the screen and tap; the part will be placed there. Note that the reference designator is U1. Next load the other parts (GENXX\D7406, LED, PB2, HRES, and GND) and place them as shown below (in the completed schematic):



Note that the reference designators (U1, R1, R2, etc.) of the parts are automatically set as you place the parts. The designator values are set by the order in which you place the part. For example the first resistor placed is R1, the second one placed is R2, and so on. You can set your own designator value for a part by first selecting the part in edit mode and then right-mouse tapping over it to get the edit part dialog which has a field to set this value.

### 1.4.2.2 Wiring the Schematic

To wire the parts together, tap on the LINE object button. Then go to the endpoint of each terminal that needs to be wired and draw horizontal and vertical line segments to the connecting terminal(s). To draw each line segment just push the left mouse button at the first point and then drag the new line point to the second location (and then release the button). Do not use diagonal lines. Connector dots are added automatically where needed.

### 1.4.2.3 Adding Net Names

In SuperCAD net names are optional, since the netlister will give each unique net a default net name. However, you may want to name at least some of the nets so that the function of a circuit is more clearly understood. For example, you may want to call a clock line CLOCK, and a set of memory address lines A0-A11. In the example schematic, we have explicitly named two nets CLEAR and VCC. To name these nets tap on the text button (or push the "t" key), place the cursor on one of the lines making up the net and tap; enter the net name (CLEAR or VCC) and then push the Esc key. You should be in snap-to-grid mode.

Note that VCC is a standard name, which refers to the digital IC +5 power supply. Although you cannot see these pins on the IC's in the schematic they are in the database. When the netlister extracts the netlist it connects any terminals connected to a signal "VCC" to the IC +5 power. The same is true for signals attached to the ground symbol (GND part); they are all attached to the IC ground terminals by the netlister.

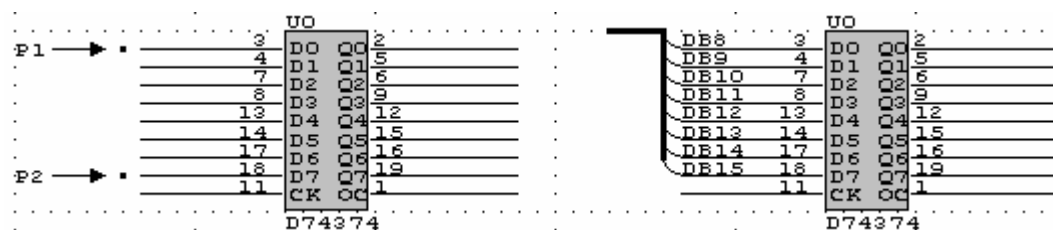
Important Note: Use only net names with 8 or less characters and do not use any space characters for the name (For example, do not use CLK IN, instead use CLK\_IN).

#### 1.4.2.4 Saving the Schematic

After working on the schematic you should periodically save it by tapping on the Save item in the File menu.

#### 1.4.2.5 Using Other Features

One feature that is a real labor saver when you are drawing schematics with microprocessors and memory parts is the automatic bus feature. Try the following experiment. Clear the schematic. Load the D74374 (octal register) part and place it in the center of the screen. Now tap on the bus object button. Then tap at the point labeled p1 (one standard grid unit to the left of the terminal--refer to diagram below). A dialog box will appear prompting you for a bus name and starting index. Enter "DB" for the bus root name and "8" for the starting index; tap on the Ok button. Now move the cursor to the point labeled p2 and tap. A bus, complete with "bus rippers" and signal name text will then appear as shown below on the right. You can draw a simple bus line (just a thick line) without the bus rippers and signal names by just tapping on Ok at the first-point prompt, and then tapping at a second point (this is how the horizontal bus segment below was made).



## 2 Edit Operations

You can edit in either single-object mode or multiple object mode. To edit in single object mode, tap the toolbar **Edit** button (with arrow tip on it). Then select an object by tapping on it.

To edit multiple objects, you also tap on the **Edit** button, but you draw an edit box about the objects you want to select:

- 1) Tap at the first corner (upper left) of the edit box (a small circle will appear to mark the location)
- 2) Tap at the second corner (lower right) of the box; a dashed rectangle then shows the outline of the edit box. Make sure you do not tap on top of an object or you will select that object.

De-select selected objects by tapping on the **Edit** button in the tool bar.

### ***2.1 Rotate***

To rotate selected objects through a fixed angle increment, tap over the rotate button in the toolbar. In single-object mode, objects are rotated by the fixed angle about their origin point. In group mode, the selected objects will be rotated about the center point of the edit box. The value for the angle of rotation is by default +90 degrees (counter-clockwise).

### ***2.2 Mirror***

The mirror operation works nearly the same as the rotate operation (see last section), except that objects are reflected about a vertical line. In the case of library parts, the vertical line goes through the origin. After selecting the object(s) tap on the mirror button to do the mirroring.

### ***2.3 Vertical Flip***

You can flip a single object or group of objects by first selecting the object(s) and then tapping on the vertical flip button. A single object is flipped about a horizontal line going through its origin. In group mode object(s) are flipped about a horizontal line going through the center of the edit box.

### ***2.4 Reference Number***

This operation allows you to assign reference numbers to library parts. The operation can be performed on one part, or automatically on every part of a selected category (for example, IC parts) on a given drawing page. To initiate the operation, tap over the **Set Reference Numbers** function in the **Edit** menu (**Modify Objects** section) pull down menu. A dialog box will then appear allowing you to select whether the operation is for one part or for the whole drawing page. If the whole page option is selected, enter either ALL or the reference letter type corresponding to the part type. For example, for ICs, you enter "u"; for resistors and capacitors, you enter "r" and "c" respectively. After selecting the category, enter a one to three digit starting reference number. All parts of the selected

category will be numbered in the time sequence in which they were entered, using the specified number as a starting point. Parts with multiple components per package are given the same reference number until the number of components/package is exceeded; then a new number is assigned. Note that these parts are automatically sequenced---that is, in making the drawing, if you don't use the F6 key to select different instances of a part, this can be done in the reference numbering operation.

If the operation is for one part, the part to be numbered needs to be selected. Do this by simply tapping over the part. After you do this, a dialog box will appear, allowing you to enter the desired reference number. Enter a number having from one to four digits (up to 9999) and tap over the Ok button. The part will then be numbered, and the display will be updated accordingly

### ***2.5 Re-Grid Operation***

This operation snaps all objects to the current grid.

### ***2.6 Undo***

The undo operation is used to undo the last edit operation (move, copy, delete, etc.) or to remove the last object added to a drawing.

### ***2.7 Move***

To move a single object that has been selected, simply tap at the new position. Likewise, to move a group of objects that have been selected, tap at the new location. You can move multiple times by just tapping again.

### ***2.8 Stretch***

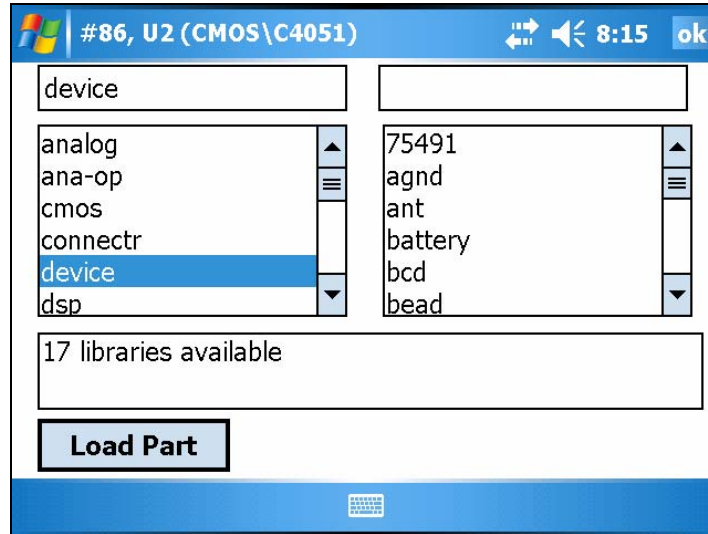
If a selected object has handles (e.g. the line object) , then the object can be stretched or resized. To do this, first tap on one of the object handles and then tap at a new position for the handle

### ***2.9 Inserting Objects***

Objects are entered with one or two taps. One tap objects include library parts and text. Two-tap objects include lines, rectangles, ellipses, and others.

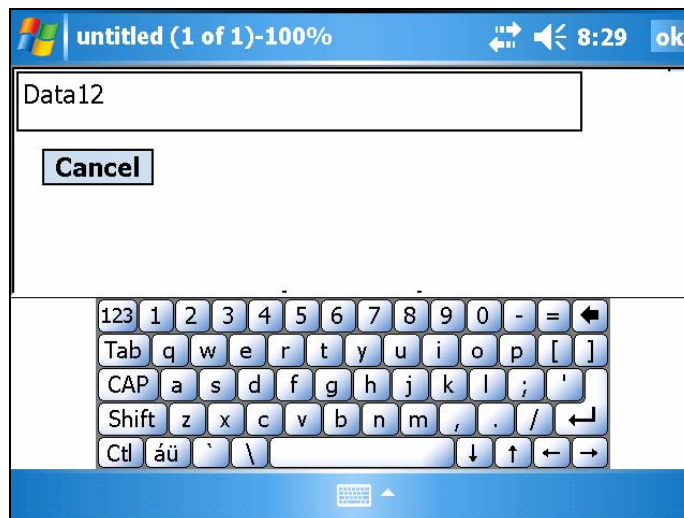
### 2.9.1 One Tap Objects

To enter a library part, tap on the library button in the tool bar or tap on **Insert Object/Library Part** in the **Edit** menu. This results in the following dialog which allows you to select the specific part:



Once you have selected a part and have clicked Load Part, you can tap at a screen location to place the part. To place multiple parts just repeat the tap.

To place text, tap on the text button on the tool bar or **Insert Object/Text** in the **Edit** menu. Then tap at a screen location for the text. This results in a display of the text dialog:



Enter the text and tap the OK button.

### ***2.9.2 Two Tap Objects***

To enter a two-tap object, such as a line, first select the object type (from the tool bar or from the **Insert Object** section of the **Edit** menu). Then tap at the first point and then the second point; if you make a mistake just tap the **undo** button on the toolbar and do the operation again.

Note that a small circle will appear when you start drawing an object that uses two taps (such as the line); the circle marks the first tap position.

## **3 File Operations**

### ***3.1 Save and Save As Commands***

The **Save As** operation allows the current drawing to be saved in a specified file. To start the operation, select **Save As...** from the **File** menu; a dialog box will then appear with the current (default name: “UNTITLED” if you just started) filename shown in an edit box. If you want the default name to be the save filename, click on the Ok button; otherwise, enter the desired filename and then click on the button. The current drawing will then be saved in the specified file.

If you want to save the current drawing using the current name, just click on the Save menu item in the File menu.

### ***3.2 Open Command***

To perform both directory and load operations, first access the File menu and then click on the **Open** menu item. When the **Open** item is activated, the .PIC extension filenames and sub-directory names are read from the current directory and displayed in a dialog box. To load a file or change a directory, first select an entry by clicking over it with the mouse; the selected entry will be highlighted. You can also use the up and down cursor keys to move through the menu items. Next, click over the Open button. If the entry is a filename, the associated file is then loaded and displayed. If the entry is a sub-directory, the current working directory is changed to this directory, and the contents of the resulting directory are again displayed in the popout menu.

### ***3.3 Print Operation***

To print a schematic, use Active Sync to transfer it to the host PC and then use SuperCAD (purchased or evaluation version) to print.

### ***3.4 Page Items***

SuperCAD/Mobile allows up to 5 pages in a single schematic file.

#### ***3.4.1 New Page***

The New Page menu item causes a new page to be added to your schematic. This allows you to continue your circuit on a second page rather than create a second schematic.

#### ***3.4.2 Delete Current Page***

The Delete Current Page menu item allows you to delete the current page of the schematic.

#### ***3.4.3 Next Page***

The Next Page menu item causes you to go to the next page of the schematic.

#### ***3.4.4 Previous Page***

The Previous Page menu item causes you to go to the previous page of the schematic.

#### ***3.4.5 Exit SuperCAD***

Clicking over Exit in the File menu ends the current SuperCAD session. Various settings, including color setups, utility menu setups and others are saved in the SUPERCAD.INI file.

### ***3.5 View Operations***

#### ***Zoom In***

The zoom in operation allows you to increase (magnify) the drawing scale by one step; for example, from 100% to 125%. If you click on the Zoom In item in the View menu, the drawing is increased in scale about a point in the center of the current drawing window.

#### ***Zoom Out***

The zoom out operation allows you to decrease the drawing scale by one step; for example, from 125% to 100%. When the drawing scale is decreased you see more of the

drawing. If you click on the Zoom Out item in the View menu, the drawing is decreased in scale about a point in the center of the current drawing window.

### Center Operation

This operation centers the viewing window over the drawing and re draws the screen. After this operation, the scroll bars will be centered in each of the scroll bar regions. You can do this operation by tapping over Center in the View menu.

## ***3.6 Setup***

### ***3.6.1 Sheet Size***

To set one of the standard sheet sizes (A-E), tap one of the selections in Setup/Sheet Size section of the Edit menu. The default sheet size is A (8.5" x 11").

### ***3.6.2 General Setup***

Tap Setup/General in the Edit menu to get the general setup dialog. This dialog allows enabling or disabling of the following:

- Grid display
- Title block
- Hidden pins
- Snap to grid
- Rubber band operation
- Drawing limited to orthogonal directions
- Automatic references

If the Hidden pins checkbox is checked then pin numbers are displayed on certain parts such as resistors and capacitors.

If **Automatic references** is checked then, when a library part is added to a schematic, its reference designator (and instance letter in the case of multiple instance parts) is automatically set. If you don't use this feature, you can use the **Set Reference Numbers** operation in the Edit menu (**Modify Object** section).

### ***3.6.3 Selective Editing***

Tap **Setup/Selective Editing** to get the selective editing dialog. This dialog allows you to select just one type of drawing element (for example just text), or combinations of

element types to edit. To select just one type of element, tap the **No Objects** button and then tap one of the object types check boxes. Later, when you want to select all types of objects to edit, get the selective editing dialog again and tap the **All Objects** button.