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Save and SaveAs Commands

The SAVEAS operation allows the current drawing to be saved in a specified file. To start the operation, select SAVEAS 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.

Load Command

To perform both directory and load operations, first access the FILE pull down menu and then click on the LOAD menu item. When the LOAD item is activated, the .PIC extension filenames and sub directory names are read from the current directory and displayed in a dialog box. Up to 7 file names and 7 sub directory/drive names can be displayed at one time, and you can view more names by utilizing the scroll bars. The entries enclosed in square brackets ([]) are sub directories. If you are in a sub directory, there is also an entry (the first entry) denoted as "[..]". 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. If you click on the entry "[..]", you will change the current directory to the parent of the directory. You can move around the entire directory of a given drive in this manner, by simply selecting directories (including "[..]") and clicking with the mouse key.

Format Operation

This operation allows you to format a schematic for use as a library part. Prior to using the FORMAT operation, the schematic must be drawn according to rules for making a library part. In making the schematic, however, you do not have to worry about the exact sequence of drawing elements (lines and text), pin numbers, or library keywords---this is handled with the FORMAT operation. To use the FORMAT command, first load the schematic that you wish to format. Click over FORMAT in the file menu, and respond with a "y" to the initial message box. The formatter then will execute the following formatting steps:

- 1) identify all lines used as terminals and copy these to a temporary buffer
- 2) identify pin text on terminal lines and copy these to the buffer
- 3) identify library keywords such as reference designators and the number of parts indicator ("p="); these are copied to the buffer
- 4) clear the original schematic and copy all the identified elements back from the temporary buffer in the correct sequence

These steps are repeated for each plane of a part if it has more than one symbol plane (for example, parts like the D7400 which have a De Morgan equivalent symbol).

As the formatter runs, watch for message box displays. These report the number of terminals, the number of voltage pins, and the number of parts/package. If these reports do not agree with your part design, recheck the schematic and run the FORMAT operation again.

Note that you do not have to format parts which have been built by the BUILD utility, unless you have reedited the part. Also you do not need to format simple parts such as symbols (e.g. ground symbol) that do not contain pins and keywords.

Print Operation

To print the current drawing page, select the print function from the FILE menu. This will activate the print dialog box. You can specify either single sheet or paginated print out using the radio buttons. If you have selected the paginated printer option, there will be one printed page for each page delineated by the page break display; in general, you have to tape the separate pages of a drawing together for a continuous drawing. The total number of pages printed for each drawing size is summarized as follows:

- A size....1 page
- B size....2 pages
- C size....2 pages
- D size....2 pages
- E size....4 pages

If you have selected single sheet operation, the drawing (A-E size) will be scaled to fit the printer sheet size.

To initiate the actual printing, click over the OK button.

You can change printer setups such as paper orientation and print out quality by clicking over the SETUP button.

Macro Command

The macro operation allows a previously stored drawing to be loaded into the current drawing without overwriting it. To initiate a macro operation click over the MACRO menu item in the FILE menu; a file dialog box will appear, having exactly the same format as in the LOAD operation. Select the macro file as described in the preceding section for load files. Once you have clicked over the OPEN button, the file will load, but not be displayed. At this point a prompt message will appear, asking you to position the cursor at a desired load (reference) point. Double click at the point, and the loaded file will be drawn, referenced to the point, and added to the current drawing. Any size information (i.e., A-E size) of the loaded macro drawing will be discarded.

If you make an error in the macro operation---for example, if you get the wrong file---you can delete the added picture elements by pushing the F7 (undo) key.

Exit SuperCAD

Clicking over EXIT in the FILE menu (or pushing the F2 key) ends the current SuperCAD session.

Drawing

Lines and Arrows

Autowiring of Lines

Busses

Rectangles

Circles and Ellipses

Arcs

Connector Dots

Text

Library Parts

Editing Library Parts

Lines and Arrows

To draw a line or arrow, first select the line or arrow function from the tool bar. Move the cursor to the first endpoint and drag to the other endpoint. The line or arrow is drawn as you drag. Lines and arrows that are constrained to be only vertical or horizontal will be drawn if the limit option is set to "on". The limit option is be chosen in the setup menu.

Lines and arrows can optionally be selected by entering "L" or "W" respectively on the keyboard.

Autowiring of Lines

Note: This function is not enabled unless you have SuperCAD+ or have purchased the MA8040 Utilities package.

The autowiring function allows you to wire together two points on a schematic sheet by simply clicking at the two points. The wiring function software automatically draws in all the intervening signal lines. If there are any library objects between the two points, the signal lines are drawn around them. Also connector dots are automatically added where needed, for example if one of the two points lies on the midpoint of a line. All lines drawn by the router are orthogonal.

To use the wiring function, first make sure that the wiring function is enabled in the setup menu. Also, you have to be in snap-to-grid mode (use the setup menu or the F10 key for this). Next, select the line function from the tool bar and drag at the first point. Move the cursor to the second point, and release the left mouse button. The router then connects the points.

Note some of the restrictions of the autowire function: 1) It will not wire points that are inside of a part----meaning inside of the part perimeter outline. This outline can be viewed when you move or place a part, or if you have the outline display enabled (via the SETUP menu or through use of the F3 key). If a point is ON the perimeter (for example, a terminal end point), then it can be wired. 2) The wiring function may add in more lines than desired; this is because the software tries to follow contours around objects present in the drawing. 3) The autowiring function is restricted to routing lines on the grid.

Busses

Busses are drawn in the same manner as lines and arrows, except that they are always drawn vertically or horizontally; the drawing is always constrained to 90 degree direction increments. They busses are drawn as thick lines, three pixels wide. Busses that connect to parts, such as registers, must be drawn within 2 grid units of the part terminals to which they are to be connected. To draw a bus, first select the bus function and then click at the first point of the bus. A dialog box will then appear asking for the signal name and for a starting index number; enter a one or two digit number as required. Now click at the second point of the bus. The bus will then be drawn with the signal names automatically sequenced, and the connecting bus signal lines attached.

Rectangles

To draw a rectangle, click on the rectangle button in the tool bar. Move the cursor to the desired location of the first corner of the rectangle; then drag to the desired opposite corner location of the rectangle. The specified rectangle will be drawn. as you drag with the mouse. The rectangle is drawn using the current outline styles, thickness, and fill patterns set in the STYLE menu.

Rectangles can be selected by entering "R" on the keyboard.

Circles and Ellipses

To draw circles or ellipses, click on the ellipse button in the tool bar. Move the cursor to a point which will be the center and then drag to the desired circumference of the ellipse or circle. The ellipse is drawn using the current outline styles, thickness, and fill patterns set in the STYLE menu.

You can optionally enter "E" to select circle or ellipse objects.

Note that the aspect ratio (ratio of vertical to horizontal diameters) of the ellipse is set in the STYLE menu.

Connectors

Connector symbols (solid dots) are drawn by first clicking on the connector button in the tool bar. After this, each time you click in the drawing area of the screen, a connector symbol is drawn at the cursor location.

The "C" key optionally selects the connector object.

Connectors are typically used for connecting line or library part terminal endpoints to midpoints of lines.

Text

Entering text is accomplished by selecting the text function from the tool bar, moving the cursor to the desired location, and clicking. At this point, the text entry is accepted from the keyboard. You can enter text in either overwrite mode or insert mode; toggle between the modes by pushing the INS key (a message on the dialog line will tell you what mode you are in. If a carriage return (using the IBM ENTER key) is given, text entry occurs one character below the current line, at the horizontal position of the initial click. Exit from text mode is accomplished by using the ESC (escape) key. A text element on one line can have up to 80 characters.

If you want to reedit a previously entered text element, click over the element. The text cursor will appear at the location of the nearest character in the (line) element.

The font size and type are selected using the FONT selection function in the SETUP menu section.

Optionally, the text function can be selected by entering "T" on the keyboard.

If you are entering text which needs to be read as a signal name by the netlister program NLST, make sure that the origin lies near the associated line.

Arcs

SuperCAD allows generation of quarter-ellipse arcs. To enter an arc that connects two points, first select the arc function by clicking over the arc button in the tool bar. Then place the cursor at the first point and drag the cursor to the second point.

Note that you can make a half-ellipse arc by joining together two quarter ellipse arcs.

Library Parts

To draw a library part, click on the library button on the tool bar. A file dialog box will then appear with the path set to the library sub directory. Select the part by clicking over its entry in the file list box; you may have to change sub directories to find the part you want. Then move the cursor to the desired location for the part and click or drag. The part will then be drawn on the screen. Note that when you drag the part you just see the part outline.

Editing Library Parts

You can change any characteristic of a part by entering EDIT mode and double-clicking over the given part, which causes a dialog box to appear.

Characteristics that you can change include:

- o reference designator
- o part instance
- o value
- o pinout (DIP or surface mount)
- o symbol type (NORMAL or DE MORGAN)

These characteristics are changed by entering the appropriate value in the associated edit box (reference designator, instance, & value) or by using the radio buttons (pinout and symbol selection).

Edit Operations

Entering Edit Mode

Single-object and Group Edits

Move

Copy

Erase

Rotate

Mirror

Origin

Draw-to-fit

Undo

Change

Center

Clipboard Operation

Undo Operation

The undo operation is used to remove the last object or set of objects added to a drawing. The objects may have been added by one of the object entry commands (line, text, etc.), by the copy command, or with a macro (file) operation.

Center Operation

This operation centers the viewing window over the drawing and redraws the screen. After this operation, the scroll bars will be centered in each of the scroll bar regions. You can do this operation either via the F4 key or by clicking over CENTER in the EDIT menu.

Change Operation

To change any style (including font style) of an object, first select the object or group of objects. Then select the new style or font from the menu(s).

Clipboard Operation

To copy objects to the clipboard, first select them in GROUP mode, and then click over CLIP in the EDIT menu.

Entering Edit Mode

The edit functions are accessed by first entering edit mode. To do this, simply click the right mouse button, or click over the edit button (having the dashed square) in the toolbar menu. Once in edit mode you generally start the edit operation by selecting one or more objects; objects are selected by clicking over them or by drawing an edit box around them.

Single-object and Group Edits

The following commands can be used in either single or group edit mode:

move, erase, copy, mirror, rotate, and change.

The edit mode is selected automatically, depending simply on where you place the cursor. If you place the cursor over an object (on the boundary in the case of a rectangle) and click, you select the object in single object mode. The following objects are highlighted with "handles" when they are selected:

- lines
- arcs
- rectangles
- ellipses & circles
- arrows

The "handles" provide selection points for re-sizing or stretching the associated objects.

Other objects including connector dots, text and library parts are highlighted with a dotted rectangle.

If you click in any area of the screen where there is no object, you enter group mode. A dot symbol, indicating that you just entered group mode is placed on the screen at the point where you clicked. To select a set of objects, you now draw an edit box around the origin points of the given objects. The first corner of the edit box is at the location of the dot symbol. The edit box itself is drawn by clicking or dragging for the second corner of the box.

Note that all objects in a drawing have a unique origin point; this point is always the first point entered by clicking or dragging with the leftmost mouse key. Once an object has been drawn on the screen, its origin point can be located by clicking over the ORIGIN button. All area-oriented (group) edit commands are referenced to the origin point of each object.

Move Operation

To move a single object that has been selected, simply move the mouse cursor over it and drag it to the new position. If the object has handles (e.g. the line object) and you drag over these, then the associated endpoint or corner is moved, and the object is stretch or resized.

To move a group of parts that have been selected, simply click at the new point (or drag). Note that if there are more than 10 objects, only the first 10 are shown while dragging; this speeds up the dragging operation.

Copy Operation

The copy operation works just like the move operation, except that you hold down the SHIFT key while dragging or placing the selected objects.

The copy operation can be used to copy objects from one plane to another. To do this, select objects (either one object or a group of objects), and then push the "P" key. At the prompt, enter the destination plane. Click or drag the objects at the desired position on the destination plane.

Erase Operation

To erase one or more selected drawing objects, click over the ERASE button or push the DEL key on the keyboard.

Rotate Operation

To rotate selected objects click over the rotate button in the tool bar. In single-object mode, objects are rotated +90 degrees (counterclockwise) 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 also +90 degrees.

Note that text characters for the bit-mapped fonts (Helv, Courier, Tms Rmn) cannot actually be rotated, although they are printed in a different location, corresponding to the rotated origin point.

Mirror Operation

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.

Origin Operation

The origin operation is used to identify the origin points of the objects in a drawing. To use the origin function click over the origin button in the tool bar. The SuperCAD software will then draw an origin symbol (a small circle about the origin point) for each object shown on the screen.

Draw to fit

To use the zoom operation, click on the ZOOM button in the tool bar; then click anywhere in the drawing window. The entire schematic will be scaled and displayed within the drawing window (draw-to-fit mode). Move the cursor to a location in the drawing you wish to zoom in on; click, and the display will be drawn in normal size, centered at the cursor position of the click. If the cursor position was near a drawing edge in the fitmode display, the centering in the zoomed display will be limited by the display borders.

The drawtofit/zoom combination provides an easy way to move around in a large drawing, particularly if your drawing size is B or greater.

Display Operations

The operations initiated in this pull down menu allow you to display the output of both digital and analog simulators. These simulators can be run from within the SuperCAD environment without returning to DOS or Windows, using input from the current schematic sheet. Typically a simulation is started by loading a circuit schematic, and editing it to provide control information such as start times, clock rates, input values, etc. Signals which are to be monitored are marked using the MARK function. Next, both the netlister and the simulator are run from the RUN menu, producing a display file. The display file can be accessed after the simulation run, from the DISPLAY menu in one of the pop out windows. For logic simulations (using SuperSIM), the results are displayed in the logic analyzer window, and for Spice (analog) simulations, results are displayed in the oscilloscope window.

Important note: Digital and analog simulation software packages are sold separately from SuperCAD; you cannot view circuit wave forms without these packages. Contact Mental Automation for further information.

Oscilloscope Display

Logic Analyzer Display

Marking Signals for Display

Oscilloscope Display

This display window is accessed by clicking over the SCOPE menu item in the DISPLAY menu, and allows you to view one or two channels of time wave forms. The wave forms can be obtained from disk files produced by a Spice analog simulator. See Figure 2-15. The window can be moved by dragging over the window title bar. Control over the timebase of the scope is accomplished by clicking over the "T" button. If the radio button is on (indicated by a set or dark button), then each click increases the time per division, otherwise it decreases the time. Likewise, the offset and gain of each scope channel are controlled by clicking over the associated buttons and using the radio button to control increase or decrease. The offset button is marked with the letter "O", while the gain button (immediately to the left of the "O" button) is marked with the current gain. The gain setting can be adjusted in 9 steps ranging from .1 units/division to 50 units/division, where "units" can be volts, db, or phase degrees depending on the type of SPICE analysis.

To display a signal in the window, first click over one of the two trace labels in the window; these are initially specified as "trace 1" and "trace 2" in the display. Then, click over a node number in the schematic; the node has to be marked with an asterisk ("*"), and you have to have a display output file that was generated from the given schematic by the spice simulator. Currently, the ISSPICE simulator from Intusoft is supported, but other spice packages may work as well.

To view different portions of a wave form, use the scroll bar or the scroll arrows.

Note that you can read out the time of a specific point in a wave form by using the readout cursor. This cursor consists of a vertical line which can be moved by using the mouse and dragging the center point either left or right within the display window. The time at the horizontal position of the cursor is displayed at the bottom of the screen on the prompt line. Also the amplitude of each channel is displayed.

If you are doing an AC analysis of a circuit, both phase and magnitude traces are generated and stored in the SPICE display file. To select which trace is loaded when you click over the marked node number, the "m" key is used. The default selection is magnitude; if you push the "m" key once, phase is selected, and pushing it again returns to a selection of magnitude.

You can obtain a screen dump of the oscilloscope window by clicking over the "p" button. The dump will be done using the printer settings last selected in the PRINT menu, or in the last Windows application that used the current printer.

Logic Analyzer Display

This display window is accessed by clicking over the ANALYZE box in the DISPLAY menu, and allows you to view up to four channels of time wave forms. The window can be moved by dragging over the window title bar. Control over the timebase of the scope is accomplished by clicking over the "T" button. If the radio button is on (indicated by a set or dark button), then each click increases the time per division, otherwise it decreases the time. To view different portions of the display, use the scroll bar or scroll arrows.

To display a given trace, click over one of the four display labels, and then click over a marked signal name; if a given node on the circuit doesn't have a name, you have to add one, and then re-run the netlister and simulator.

As with the Oscilloscope display, you can read out the time at a point on a wave form by using the readout cursor (the vertical line). This cursor is moved (and a value consequentially displayed on the dialog line) by using the mouse to drag the line.

You can obtain a screen dump of the logic analyzer window by clicking over the "p" button. The dump will be done using the printer settings last selected in the PRINT menu, or in the last Windows application that used the current printer.

Marking Signals for Display

To mark a signal for display, click over MARK in the DISPLAY menu and then click over the desired signal in the schematic sheet. A marked signal has a "*" character appended to it.

Keyboard Interface

Object Entry Keys (Summary)

T.....Enter text.
B.....Enter bus.
W.....Enter arrow.
R.....Enter rectangle.
L.....Enter line.
E.....Enter ellipse.
C.....Enter connector dot.
K.....Enter Arc.

Function Keys

F1.....Enter Help Mode
F2.....Exit Program
F3.....Toggle automatic wiring mode
F4.....Center schematic
F5.....Access library menu
F6.....Change part instance
F7.....Remove last object
F8.....Clear schematic
F9.....Re-draw schematic
F10.....Toggle snap-to-grid mode


Other Keys


A.....Toggle pinout mode (SURFACE or DIP)
S.....Toggle symbol mode (NORMAL or DE MORGAN)
P.....Change drawing plane (0,1, ALL)
ESC.....Re-draw/clear edit mode
HOME.....Move to library origin point


Buttons


The tool bar region contains two groups of buttons. The left group is used for selecting library parts or drawing objects (e.g., lines, rectangles). The right group selects edit operations (zoom, erase, rotate, etc.).


The individual buttons are defined as follows:

 text object

 draw-to-fit


 bus object


 enter edit mode


 arrow object

 erase

 rectangle object

 rotate


 line object

 mirror

 ellipse object

 origin

 connector object

 arc object

 library part

Utilities Menu

The UTILITIES menu is a user customizable menu. Functions displayed in the menu are user defined, and allow you to run any DOS or Windows executable file from within SuperCAD. Typically the menu is used for functions such as the netlister and parts (bill of material) lister. If the given function is already on the menu, you simply click over the associated menu item and the function is started. If the function is not there, click over DEFINE in the menu; a dialog box will appear. To add the new function enter the desired menu text in the menu text box and a command string in the command box. Click on the ADD button to enter.

If the function takes a file name as a parameter, specify this as "%s" in the command box; for example:

```
nlst %s n
```

in the case of the netlist utility.

You can delete any current item in the UTILITIES menu by selecting it from the list box and clicking over the DELETE button. likewise, you can modify an existing item by first selecting it, editing the fields in the COMMAND and MENU TEXT boxes, and then clicking over UPDATE.

When you exit SuperCAD the UTILITIES settings are saved in the WIN.INI file.

Setup Operations

Drawing Size

Library Part Defaults

Drawing Scale

Drawing Plane

Grid Size

Editing

Font Selection

Color Selection

General Setup

Drawing Size Selection

To select a drawing size, click over the DRAWING SIZE menu item in the SETUP menu; a cascade menu will appear listing drawing sizes A through E. The letters select standard drawing sizes defined as follows:

A size 11" x 8.5" (640 x 480 pixels)
B size 17" x 11" (960 x 640 pixels)
C size 22" x 17" (1280 x 960 pixels)
D size 34" x 22" (1920 x 1280 pixels)
E size 44" x 34" (2560 x 1920 pixels)

Click over the associated menu item to select a new size. Immediately, on choosing the drawing size, the screen will be redrawn to reflect the new size.

Library Part Setups

The LIBRARY PART item of the SETUP menu selects a cascade menu from which you can choose the following library part features:

Package type---DIP or surface mount

Symbol---Normal or DeMorgan

Instance

Hidden pins (with "z" indicators)---Hidden or visible

Part outline---ON or OFF

Note that the first three selections in the menu only apply to the current active part which is the last part loaded.

General Setups

The GENERAL section of the SETUP menu allows you to specify the following options:

<u>option</u>	<u>choices</u>
limit	on, off
border	on, off
grid	visible/clear grid snap on/off
route	on, off
rubberband	on, off
page breaks	on, off
rulers	on, off

To do the setup operation, move the cursor to the option under the option entity you wish to change and click; the new option will be selected and marked on the screen with the check mark.

Drawing Scale

This section of the SETUP menu allows the drawing scale to be set to 50%, 100% (normal), 150%, or 300%. You can do any schematic entry operation (including editing and object placement) at any of these three scales.

Grid Size

This section of the SETUP menu allows the grid size to be set to four possible sizes denoted as 1, 1/2, 1/3, and 1/6. The default grid size (1) is one tenth of a ruler inch, which is 9 pixels at 100% scale. This size allows you to snap easily to library part terminals. The other grid sizes (1/2, 1/3, and 1/6 of the basic size) are useful for making parts using the SuperCAD editor. To change the current grid size click on the GRID menu item, and then click over the desired grid size option in the cascade menu.

Editing

If you click over the EDITING menu item of the SETUP menu you get a cascade menu which allows selection of an edit mode. There are essentially two edit modes:

- o edit all objects
- o edit one class of objects

The default mode is to edit all objects. If you select one of the other modes to edit just one class of objects---for example, lines---then you can only select, move, copy, or erase, etc. objects of the given class. The other objects become "invisible" with respect to the selection operation. This facilitates schematic editing when you have several type of objects (for example lines and text in library parts) at the same position on the drawing.

Drawing Plane

You can select the drawing plane (0,1,ALL) by clicking first over the PLANE menu item and then clicking over the appropriate item in the cascade menu. Drawing plane 0 is main (default) drawing plane for schematics, while drawing plane 1 is used to hold the De Morgan equivalent symbol for parts like the 7400 NAND gate.

Font Selection

The font selection operation allows you to change font size and font style. Fonts can range from 8-point sizes up to 38-point sizes, except for the STANDARD font (default) which can be just 8-point.

To set a specific font and size, click over any font type in the typeface list box; then select the size in the Size list box. You can also select italic mode for any font by clicking in the italic ON or OFF check box. Click on the "OK" button to confirm the selection. The font selection remains in affect until a new selection is made.

Note that the following fonts are bitmapped: Helvetic, Courier, Times Roman. These fonts cannot be rotated.

Color Selection

In color video modes you can choose any one of 16 colors for the following:

- drawing window background
- drawing window objects (all text, lines, etc.)
- rulers
- grid
- edit mode
- oscilloscope and analyzer windows

You select the color item and its associated color by first selecting the radio button of the color item, and then clicking over one of the 16 color squares. You then click on the OK button. Only one item can be changed at a time. Note that not all colors can be used together. For example, if you choose RED for the drawing window and RED for the edit color, you can't see text that is being edited.

Auxiliary Operations

Enter Reference Number

Enter Value

Find Text or Reference Designators

Replace text

Enter Auto-incremented Text

Check Schematic Page

Enter 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, click over the "ref no." function in the AUX 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 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 clicking 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 three digits (but less than 511) and click over the OK button. The part will then be numbered, and the display will be updated accordingly.

Value

The value operation lets you append a value to any library part, with the exception of ICs. For example, you can add component values to resistors and capacitors; you can also specify part numbers for transistors and diodes. The appended value can be extracted by the parts utility, and it is displayed on the schematic next to the reference number. To enter the value, first click over VALUE in the AUX menu. Then click over any part you want to add a value to, and enter the value in the dialog box. Note that the value can be any text string with 8 or less characters.

Find Operation

The find operation is used to locate text strings in a drawing page; it can also be used to find library parts with a specific reference number. To start the operation, first select it from the menu; a dialog box will then appear. Select either search for text or search for reference numbers, by clicking on one of the two radio buttons. To search for text, enter a string of 80 characters or less in the text box. The schematic will then be searched for a matching string in a text element. If a match is found, the window will be redrawn with the found text centered and highlighted. A dialog box with two buttons, labeled CANCEL and CONTINUE, will appear. Click over CONTINUE if you want to continue searching, or over CANCEL, if you wish to stop. This operation will not find strings inside library parts.

If you selected the option to look for a reference designator, enter a reference designator (for example, "u7" or "r15") in the text box. The schematic will then be searched for a part having the given designator. If it is found, the screen is re-drawn, and the reference designator string of the part is highlighted. You can then search for further parts (having the same designator) or quit.

Replace Operation

The replace operation is used to replace all text strings in a given drawing page, which match a given search string, with a specified replacement string. To initiate the operation, select it from the menu; a dialog box will then appear. Enter search string (of 80 characters or less) in the "Search for text:" edit box. Next, enter a replacement string in the "Change to:" edit box. Also specify whether you want to be prompted before a change is made, using the radio buttons. After, this click on the OK button. The SuperCAD software will then search all text elements in the drawing and make the necessary replacements as directed.

Note that this operation cannot be applied to library parts, since the text in these parts is not directly referenced.

Enter Auto-incremented Text

The autoincrement text operation allows any text consisting of a root name string and an index number string to be quickly entered. To start the operation, select AUTO from the AUX menu. A dialog box will be displayed, which allows you to enter a root name string (e.g., XYZ) and a starting index (e.g., 12). Next click on the OK button. After this, click on the screen drawing area wherever you want an autoincremented string drawn. The string will then be drawn and the index number portion will be incremented for the next click. Note that if you leave the root text edit box empty, and enter just the starting index, you will get a sequence of numbers at each click on the drawing.

Check Schematic Page

The check operation allows you to check a schematic sheet prior to doing a netlist operation. The check operation does the following:

- 1) It checks all input terminals of all library parts on the sheet for connection to an orthogonal line, another library part (including ground and connector symbols), or a text element (i.e., a label). If it can't find such a connection, the message "Can't find connection on <input pin>" is displayed.
- 2) It checks for unattached text, displaying it in a message if encountered. This check can be used to see that text intended for a label is properly placed.
- 3) It checks for unspecified reference designators; if a designator such as "u0" or "r0" is found, a warning message is produced.

To initiate the check operation click over the CHECK menu item. A message box will then ask you if you want to display the messages on screen or store them in a file. If you expect a lot of errors, the file method is the most appropriate. The file name for the error reporting is REPORT.DOC.

Style Selection

Outline Style Selection

Setting Fill Patterns

Setting Outline Border Widths

Setting Ellipse Aspect Ratios

Outline Style Selection

The outline style selection operation selects one of four drawing styles for lines, arrows, ellipses, arcs and rectangles: solid (the default), dash, dot, or dot-dash. To select the outline drawing style, simply select the STYLE menu and then click over the desired drawing style. All subsequent lines, arrows, ellipses, and rectangles will be drawn in the selected style until a new style is selected. Note that when the width of a line or other object is greater than one, the outline style is always solid.

Setting Fill Patterns

Fill patterns for rectangles and ellipses are selected by first clicking over the FILL menu item. Then in the fill dialog box simply click over one of the 8 filled rectangles in the upper part of menu, and the given pattern is displayed in the rectangle at the bottom next to the "OK" button. If you now click on "OK" this pattern is selected until you select a new one. Note that the unfilled rectangle in the upper left corner of the dialog display specifies an empty or transparent fill pattern.

Setting Outline Border Widths

Line, arc, rectangle border, ellipse border widths can be entered by clicking over the WIDTH menu item. In the width dialog box, enter a number from 1 to 9; this is the number of units (pixels) used for line or rectangle thickness when these objects are next entered. The default for width (used when the program starts) is 1. Note that when the width is greater than one, the outline style is always solid.

Setting Ellipse Aspect Ratios

Clicking over the ASPECT menu item allows you to change the current aspect ratio (defined as the ratio of x to y axes). The default is 1, and you can set any ratio between .1 and 99. Note that the appearance of ellipses is altered both by the video adapter (CGA, Hercules, EGA, VGA, SVGA) and by the printer/plotter type. To get a perfect circle on a printer set the aspect ratio to 1.2 for A size drawings and .833 for B size drawings; these ratios come about because a printer's resolution is 60 dots vertically, and 72 dots horizontally. For an HPGL plotter the aspect ratio should be set to 1 (in any case).