OrCAD®

Electronic Design Automation Tools

Schematic Design Tools 386+

User's Guide

McBoole is a public domain process developed by Michel Dagenais of McGill University.

Seventh Edition 23 Sept 94

OrCAD®

9300 S.W. Nimbus Avenue
Beaverton, Oregon 97008-7137
U.S.A.

Sales & Administration	(503) 671-9500
FAX	(503) 671-9501
Technical Support	(503) 671-9400
24-Hour Bulletin Board System	(503) 671-9401

C O N T E N T S

Welcome to OrCAD Schematic Design Tools

Welcome to practical electronic engineering. You now own OrCAD's **Schematic Design Tools 386+**, a design automation tool set with the power of an engineering workstation. You can complete complex design tasks using **Schematic Design Tools 386+** in a fraction of the time it takes by hand.

Schematic Design Tools 386+ was developed specifically to run on DOS-based personal computers using an 80386 or later processor. In addition, it supports most popular graphics boards, printers, and plotters.

Finding the information you need

These four manuals accompany **Schematic Design Tools 386+**:

- *ESP Design Environment User's Guide*
- *Stony Brook M2EDIT Text Editor User's Guide*
- *Schematic Design Tools 386+ User's Guide*
- *Schematic Design Tools 386+ Reference Guide*

Installation

Before you begin to explore **Schematic Design Tools 386+**, take a few minutes to install the tool set and register for technical support. Just follow the steps in the installation instructions that accompany **Schematic Design Tools 386+**.

Project-oriented design environment

Schematic Design Tools 386+ is one part of a fully integrated electronic design automation (EDA) system. The design environment is structured so you can focus on what's important: the design. Designs are organized by project, with all the design files—schematics, netlists, part lists, simulation results, and board layouts—stored together.

The *ESP Design Environment User's Guide* introduces the graphical environment under which **Schematic Design Tools** and the other OrCAD tool sets operate. In this environment, OrCAD tools and tool sets, such as **Schematic Design Tools**, are accessed via buttons. There are four OrCAD tool sets. They are:

- Schematic Design Tools
- Digital Simulation Tools
- Programmable Logic Design Tools
- PC Board Layout Tools

Buttons to access all four OrCAD tool sets display on the main screen, even if you only have one tool set installed on your computer.

Learning Schematic Design Tools

This user's guide introduces **Schematic Design Tools.** The best way to get to know **Schematic Design Tools** is to start with *Chapter 2: Introducing Draft,* then proceed chapter by chapter through this book. You will be guided through several practice sessions that show you the basics about using **Schematic Design Tools.**

Beyond the basics

Once you master the basics, refer to the *Schematic Design Tools 386+ Reference Guide* for information that will help you plan and create your design. The reference guide explains how to tailor the configuration of the software to match your personal requirements, provides detailed information about commands and concepts, and tells how to transfer a design between OrCAD tool sets. It is designed to be a continuing source of instruction and reference as you use **Schematic Design Tools 386+.**

The design environment

Schematic Design Tools 386+ is one part of an integrated electronic design automation environment. Using this design environment you can:

- Run the tools within a tool set. The tools that make up **Schematic Design Tools** are listed in the next section.
- Move between tool sets without switching directories or copying files.
- Configure tools. Each tool can be configured and the configuration stored. This eliminates the need to enter command-line switches every time a tool is used.
- Organize designs by project. All files associated with a design—schematics, netlists, reports, PLD source code, simulation results, and layouts—are stored in one location. This location is actually a directory on your computer's hard disk. Each design has its own directory containing all of the files described above.

Tools

The tools in a tool set are organized by function:

- Editors
- Processors
- Librarians
- Reporters
- Transfers

Figure 1-1 shows how these tools are organized on the **Schematic Design Tools 386+** screen.

These functions are described briefly on the pages that follow. The explanations assume that you are already familiar with common electronic design terms and concepts. If you are just learning about schematic design, some terms we use to describe the tools may not be familiar to you. Don't worry: basic, essential concepts and skills are thoroughly covered in chapters 2 through 8 of this guide. Advanced concepts are explained in the *Schematic Design Tools 386+ Reference Guide.*

You can run all OrCAD tools on a single worksheet or on a multiple-sheet design. Multiple-sheet designs can be either flat designs or hierarchical designs. To learn about these different types of files, see the *Design structures* section later in this chapter.

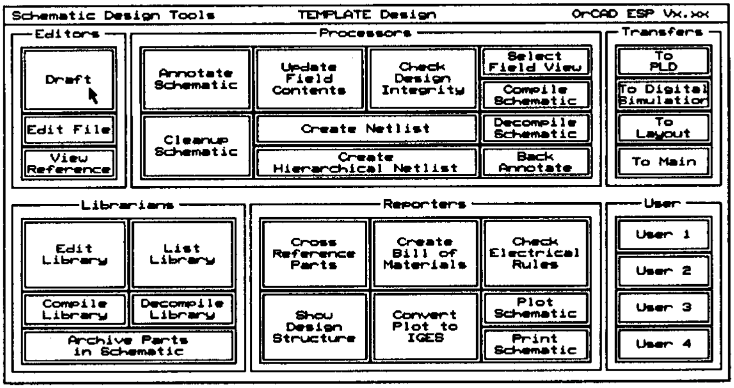


Figure 1-1. Schematic Design Tools 386+ screen.

Editors

Editors create or modify design files. The three editor tools in **Schematic Design Tools 386+** are:

- **Draft.** The heart of **Schematic Design Tools** is the schematic editor, **Draft. Draft** is used to create schematics, which are part of the design database.
- **Edit File.** This text editor is used to create and edit text files.
- **View Reference.** This tool runs a text editor in a reference-material directory provided by OrCAD. You can view files about drivers, libraries, netlist formats, and other topics of interest.

Processors

Processors are tools that subject a design file to a specific process. The seven processors in **Schematic Design Tools 386+** are:

- **Annotate Schematic.** This tool automatically updates part reference designators (such as U?, R?). It also updates the pin numbers associated with the reference designators in multiple-element parts. **Annotate Schematic** can handle very large, complex, multiple-sheet designs. It can update incrementally (leaving previously assigned reference designators alone) or unconditionally.

- **Create Netlist.** This tool creates a text file listing the logical interconnections between signals and pins. The netlist information is used for board layout. **Create Netlist** creates a netlist in one of over 30 different formats. See *Appendix B: Netlist formats* in the *Schematic Design Tools 386+ Reference Guide* for a list of available formats.

  You can also create your own netlist formats. See *Appendix D: Creating a custom netlist format* in the *Schematic Design Tools 386+ Reference Guide* for instructions.

  Create Netlist also creates the connectivity database. The connectivity database is used when transferring to OrCAD's **Programmable Logic Design Tools** and **Digital Simulation Tools.**

- **Create Hierarchical Netlist.** This tool operates similarly to the **Create Netlist** tool; it differs in that it maintains the hierarchical structure of a design and in the type of netlist created.

  Both **Create Netlist** and **Create Hierarchical Netlist** can be used with any design structure, but most board layout packages use the netlist formats of **Create Netlist.**

- **Update Field Contents.** This tool updates part value and part fields. Every part has ten fields that are used to hold text or data associated with the part. One data field holds reference designator values, such as "U1A" and "Q1." Another holds the part's name, such as "74LS04" or values relevant to the part, such as ohm (Ω) values for resistors. Another is recommended for module values, such as "CK05" or "8DIP300." The other seven data fields can store any information you might find useful: part tolerance, vendor name, part number, and so on.

  Update Field Contents can change information in all but the reference designator field. It changes fields based on the contents of an update file. You create the update file using **Edit File.**

- **Back Annotate.** This tool updates part reference designators by using a list of old and new reference designators called a Was/Is file. You create the Was/Is file using **Edit File.**

- **Cleanup Schematic.** This tool checks to see if any wires, buses, junctions, labels, module ports, or other objects have been placed on top of one another.

- **Select Field View.** This tool makes the contents of a part field either visible or invisible on the schematic.

- **Check Design Integrity** prepares a design for the netlist process by running **Cleanup Schematic, Cross Reference Parts,** and **Check Electrical Rules.** These three tools check for drawing errors, duplicate and mismatched reference designators, and violations of electrical rules.

- **Decompile Schematic.** This tool creates an ASCII text representation of a design. You typically use **Decompile Schematic** to create text files from worksheets, use a text editor to modify the text files, and then use **Compile Schematic** to create new worksheets.

- **Compile Schematic.** This tool creates schematic (.SCH) files from ASCII text files. The ASCII text file format is described in *OrCAD Technical Note # 47: Decompile Schematic's ASCII Export (AEX) Specification.*

Librarians

Schematic Design Tools 386+ includes libraries containing more than 20,000 parts. These parts represent TTL, IEEE, CMOS, memory, ECL, discrete, analog, microprocessor, and peripheral devices.

Schematic Design Tools 386+ includes five librarians, which are tools for managing and creating library parts. Three of these tools work directly on libraries:

- **List Library**. This tool creates a file which lists all the parts in a library.
- **Archive Parts in Schematic**. This tool scans a single worksheet or an entire design and collects all the parts used. It then creates a library file containing those parts.
- **Edit Library**. This tool is a graphical editor for creating or modifying library parts. You can save an edited part in a new or existing library.

You can also create or modify library parts with a text editor, such as **Edit File**. These two tools convert libraries from source form (text) to compiled form and vice versa:

- **Compile Library**. This tool converts a library source file into a compiled library object file. The compiled library object file can be used by the other **Schematic Design Tools.**
- **Decompile Library**. The inverse of the **Compile Library** tool, this tool converts a compiled library object file into a library source file. You can edit the library source file using **Edit File.**

Reporters

Reporters produce reports but do not modify design data. The seven reporter tools in **Schematic Design Tools** are:

- **Cross Reference Parts.** This tool scans specified schematic files, gathers information for all the parts used in the schematic files, and creates a report that lists each part's location in the design.
- **Create Bill of Materials.** This tool creates a summary list, sorted by reference designator, of all the parts used. You can also merge additional information into the summary list by using an include file.
- **Check Electrical Rules.** This tool checks for conformity to basic electrical rules. It checks for shorts, inputs with no driving source, unconnected pins, bus contention, and other common electrical hookup problems.
- **Show Design Structure.** For a hierarchical design, this tool displays the design's root worksheet filename plus all the associated sheet symbol names and the filename of each sheet. For a flat design, it displays the design's root worksheet filename plus the schematic filenames of the linked sheets.
- **Convert Plot to IGES.** This tool translates a schematic plot file (created by **Plot Schematic**) into IGES (Initial Graphics Exchange Specification) text format. This common data format allows schematic plot files to be stored on a mainframe computer or used with other applications that accept IGES input (such as VersaCAD®).
- **Plot Schematic.** This tool plots a single schematic or all the schematics in a design. It produces high-resolution plots of your designs, is capable of scaling, and can handle much larger designs than **Print Schematic** can.

  Devices that accept *vector* commands are considered to be plotters. A vector is one or more points and an associated function. For example, a line has two points and a "shortest-distance" function; a circle has a center point and a "radius" function. Postscript is a vector language.

- **Print Schematic**. This tool prints a single schematic or all the schematics in a design. It produces draft-quality printouts of your designs at the default scale.

  Devices that accept *raster* commands are considered to be printers. A raster is an array of dots which may define any shape.

Transfers

Three of the transfer tools perform the steps needed to prepare a design for use by another OrCAD tool set. The **To Main** transfer tool simply displays the main screen. The four transfer tools in **Schematic Design Tools** are:

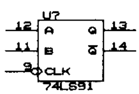
- **To PLD**. This tool updates part value and part fields, annotates the reference designators, extracts the **PLD** information, and displays the **Programmable Logic Tools** screen.
- **To Digital Simulation**. This tool annotates the reference designators, builds a connectivity database, creates trace and stimulus files, and displays the **Digital Simulation Tools** screen.
- **To Layout**. This tool updates part value and part fields, annotates the reference designators, builds a netlist, and displays the **PC Board Layout Tools** screen.
- **To Main**. This tool displays the main screen.

Graphic objects

Schematics are made up of a variety of graphic objects. You can include any of the following graphic objects in your schematic designs:

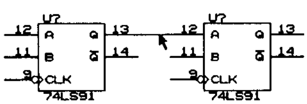
- Parts
- Wires
- Buses
- Junctions
- Power objects
- Module ports
- Sheet symbols
- Labels
- Text
- Title block
- Stimulus objects
- Vector objects
- Trace objects
- Layout objects

Parts



Parts are graphic objects you place on the schematic worksheet to represent the electronic parts in your design.

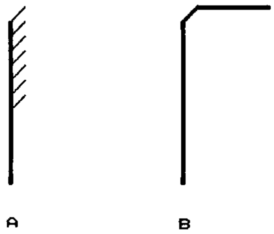
Wires



Wires are graphic objects you place on the worksheet to represent connections between objects, such as pins of parts and power objects.

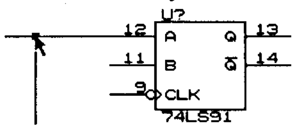
Buses

Buses are graphic objects that represent arrays of signals as single units on your worksheet.



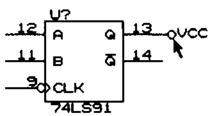
Junctions

Junctions are graphic objects that represent physical connections between wires, buses, and nodes. Junctions look like small squares.



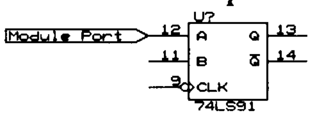
Power objects

Power objects are graphic objects that represent connections to a power source.



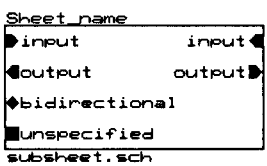
Module ports

Module ports are graphic objects that indicate where signals are conducted between worksheets.

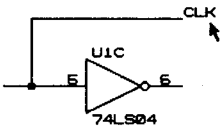


Sheet symbols

Sheet symbols are block-shaped symbols representing other worksheets. Each sheet symbol represents a subsheet. Sheet symbols are only used in hierarchical designs.

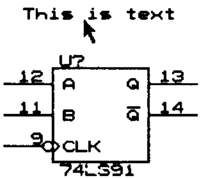


Labels



Labels assign names to wires or to buses. Signals which share a name are considered to be electrically connected even though the physical connection is not shown on the worksheet.

Text



You can also place *text* in your worksheet. Text is used to leave notes or descriptive text (that isn't required by the circuit) on a schematic diagram. Such text helps you and others understand the functions being performed or documents some aspect of circuit operation.

Title block

The *title block* is used to label your worksheets so that you can tell them apart. It contains information such as company name and address, schematic title, number, size, and revision.

Stimulus objects

OrCAD's **Digital Simulation Tools** uses *stimulus objects* to determine where a stimulus is to be applied to a circuit.

Vector objects

OrCAD's **Digital Simulation Tools** uses *vector objects* to determine where sets of stimuli are to be applied to a circuit.

Trace objects

OrCAD's **Digital Simulation Tools** uses *trace objects* to determine which signals to trace.

Layout objects

You can place hidden text on your worksheet and flag its location with a *layout object*.

Working with schematic diagrams

As its name suggests, **Draft** is designed to be analogous to pencil-and-paper design tools and to support the design process from the conception of a design to the final sets of detailed schematic diagrams.

In **Draft**, *worksheets* are analogous to the sheets of drafting paper on which you draw schematics. A worksheet appears on the computer screen as a rectangular area in which you can place parts and draw wires.

When you save your work in **Draft**, the worksheet is saved as a schematic file in the current design directory. If you do not provide a name, **Draft** uses the name of the design directory and the extension .SCH.

A design directory contains all the files for a particular design. In the default OrCAD directory structure, all design directories are subdirectories of the \ORCAD directory.

Design structures

Some designs are small enough to be represented entirely on a single schematic worksheet. **Schematic Design Tools 386+** standard page sizes correspond to the five standard sheet sizes for plotters and printers (A through E for U.S. inch, and A4 through A0 for ISO metric). You can also create custom page sizes up to 65 inches square.

However, some designs are too large for even the biggest sheet and, even if a very complex design could fit on one sheet, there are good reasons for dividing it:

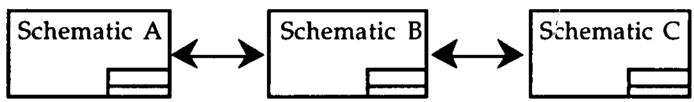
- ❖ To partition a design so that several people can work on it at once.
- ❖ To develop the design using a top-down approach. That is, you may want to begin with a block diagram in which each block represents a major function and then construct more detailed diagrams for each of the blocks.
- ❖ To organize your design by functional parts.
- ❖ To maximize the performance of your tools.

Schematic Design Tools 386+ offers two ways of handling multiple-sheet designs: flat designs and hierarchical designs.

Flat designs

Best suited for small designs with no more than ten sheets, flat designs laterally connect the output signals from one schematic to the input signals of another.

All schematics in a flat design are on a single level, as shown below.



Since you must manage all of the interconnections between the sheets of a flat design by the names assigned to the module ports, it is best to keep a flat design relatively small.

Module ports

Module ports that have identical names on both schematics are considered to be electrically connected.

Figure 1-2 shows an example of connections between schematics in a simple two-sheet flat design.

The module ports in figure 1-2 that connect between the schematics are named CLEAR, LOAD, and RCO. The module ports named Hi[0..3] and Lo[0..3] are not connected.

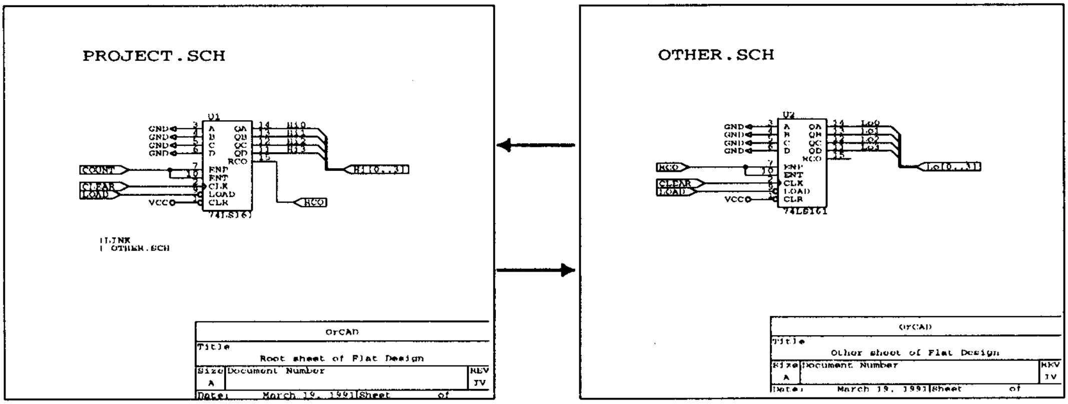
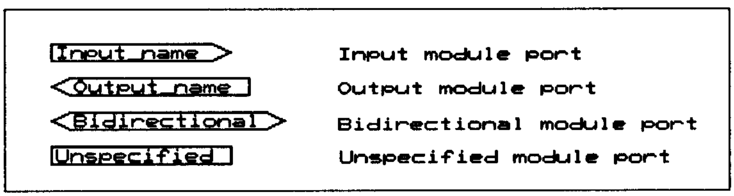


Figure 1-2. Schematics linked by module ports.

Figure 1-2 shows only input and output module ports. **Draft** has two other types of module ports: bidirectional and unspecified. The four types of module ports are represented as follows:



You can use module ports to connect single wires, as shown in figure 1-2, and to connect buses.

You place module ports on a schematic using **Draft's PLACE Module Port** command.

| LINK command

Module ports indicate the names of connected signals but do not specify which worksheets are included in the design. In flat designs, one worksheet must have a list of the worksheets in the design.

The list heading consists of the "pipe" character (the vertical bar on your keyboard) followed by the keyword "LINK." The list elements are filenames of the worksheets that are linked to the root worksheet. Each filename must be preceded by the pipe character.

Notice the | LINK (read as "pipe-link") command on the PROJECT.SCH worksheet in figure 1-2. The worksheet with the list is considered to be the root sheet.

You place the worksheet list on the root worksheet using **Draft's PLACE Text** command. The example at right shows text as it appears on a root worksheet that has module ports that link to worksheets called SCHEM1.SCH, SCHEM2.SCH, and SCHEM3.SCH. This text can appear anywhere on the worksheet.

```
|LINK
|SCHEM1.SCH
|SCHEM2.SCH
|SCHEM3.SCH
```

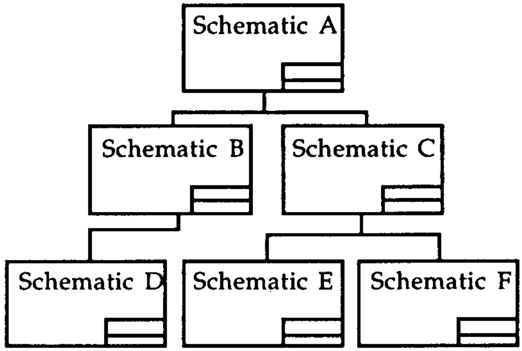
It is important that the "|" characters are vertically aligned. If they are not, **Schematic Design Tools 386+** processors and reporters will not be able to find the linked worksheets. In the example at right, worksheet SCHEM2.SCH will be ignored because it is not aligned correctly. A good test of the | LINK command is the **Show Design Structure** tool.

```
|LINK
|SCHEM1.SCH
 |SCHEM2.SCH
|SCHEM3.SCH
```

△ *NOTE: For details about module ports, see the* ***PLACE Module Port*** *command in chapter 2 of the* Schematic Design Tools 386+ Reference Guide. *For details about placing text on a worksheet, see the* ***PLACE Text*** *command in chapter 2 of the* Schematic Design Tools 386+ Reference Guide.

Hierarchical designs

As an alternative to the flat design, you can create schematics that contain symbols representing other schematics. These symbols are called *sheet symbols*. The layered arrangement created by placing schematics inside other schematics is called a *hierarchy*. Any hierarchy—whether it is a schematic design or a corporate organizational chart—has "higher" and "lower" levels.



Any schematic can contain sheet symbols that refer to other schematics, and this nesting structure can be made many levels deep. The schematic at the top of a hierarchy, which directly or indirectly refers to all other schematics in the design, is called the *root worksheet*.

You place sheet symbols on a schematic using **Draft**'s **PLACE Sheet** command.

How signals enter and leave sheet symbols

Just as module ports indicate which signals connect between schematics, *sheet nets* indicate which signals connect between a sheet symbol and its associated schematic. In figure 1-3, sheet nets are the small black objects shown on the borders of the sheet symbols. The sheet nets on a sheet symbol correspond to module ports on the schematic named in the sheet symbol.

You place sheet nets using **Draft**'s **Add-NET** command, which becomes available when you select the **PLACE Sheet** command. To associate a particular sheet net with a particular module port, assign the same name to both.

The bracketed notation (*A*[*m*..*n*]) shown on the module ports and sheet nets designates the number of signals carried by a bus. For example, X[0..3] indicates four signals: X0, X1, X2, and X3.

Buses must have a label or a module port with similar bracket notation to indicate the number of signals they carry, as shown in figure 1-3. In addition, wires connected to buses must have labels identifying the signals they carry.

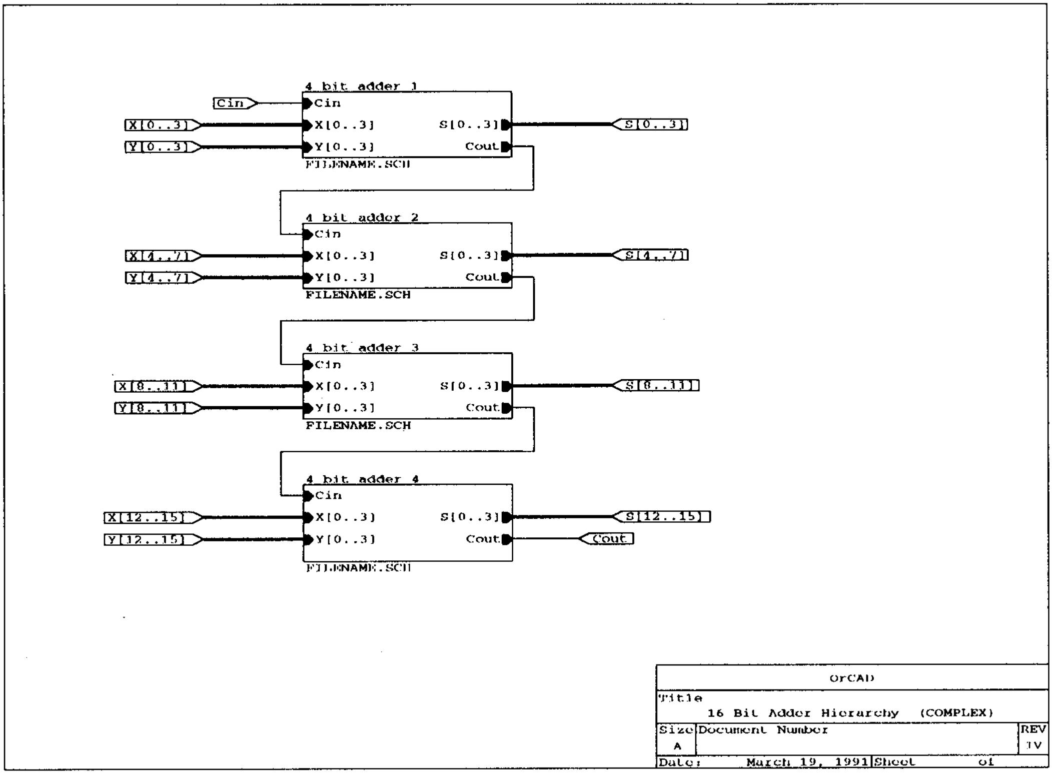


Figure 1-3. Hierarchical design.

Simple and complex hierarchies

A one-to-one correspondence between sheet symbols and the worksheets they refer to defines a structure called a *simple hierarchy.*

Several sheet symbols can refer to a single worksheet, resulting in a structure called a *complex hierarchy.* For example, figure 1-3 shows a complex hierarchy in which there are four references to a single worksheet called FILENAME.SCH.

How sheet symbols refer to schematic logic

To give a sheet symbol access to the logic of a particular worksheet, you assign the sheet symbol a worksheet's filename. The filename displays at the bottom of the sheet symbol, as shown in figure 1-3.

You assign a sheet symbol its corresponding worksheet's filename using the **Filename** command, which becomes available when you select the **PLACE Sheet** command.

In addition to filename markers, each sheet symbol must have a unique name. In figure 1-3, the sheet symbol names are shown just above each sheet symbol.

You name the sheet symbols using the **Name** command, which becomes available when you select the **PLACE Sheet** command or when you edit the sheet symbol.

Moving between levels in a hierarchy

Draft makes it easy to move up and down in a hierarchy, from sheet symbol to associated worksheet and back again.

To go from a sheet symbol to its associated worksheet, put the pointer on the sheet symbol and select **QUIT Enter Sheet**. To go from this schematic back to the schematic with the sheet symbol, select the **QUIT Leave Sheet** command.

More about hierarchical design structures

The worksheet represented by a sheet symbol can itself contain sheet symbols. This means you can create hierarchies that are many levels deep, each level containing greater detail.

This is particularly useful for very complex designs. It encourages a logical, function-oriented approach to partitioning designs and makes them easier to manage.

Another advantage offered by hierarchical structure is the ability to use sheet symbols to refer repeatedly to "stock" worksheets containing common circuit functions. This is often used in gate array and field-programmable gate array (FPGA) designs.

△ *NOTE: A deep hierarchy is much more efficient than a wide hierarchy. A wide hierarchy, while not a flat design, has many of the same limitations in organization, presentation, and structure. A deep hierarchy more clearly represents the functional nature of the design.*

For more information, study the hierarchy examples in *Chapter 8: Structuring your design.*

Learning Schematic Design Tools

The remainder of the *Schematic Design Tools 386+ User's Guide* shows how to design schematics by guiding you through the process of creating the schematic diagrams for a digital clock. To do this, you use the schematic editor called **Draft** to create the schematic of the clock circuitry. Within the schematic are three smaller circuits:

- A clock oscillator circuit
- A power regulator circuit
- A logic and display circuit

Each chapter builds on the skills and concepts from the previous chapter. As you complete each chapter, you create a series of design files.

The summaries that follow describe the design concepts and skills you learn in each chapter.

Chapter 2: Introducing Draft

This chapter introduces **Draft**, the **Schematic Design Tools** schematic editor. You learn how to run the design environment, run **Design Management Tools**, set up work conditions for **Draft**, run **Draft**, capture and save an initial macro, and save your work.

Chapter 3: Capturing the clock oscillator schematic

In this chapter, you create (or *capture*) a small schematic and learn the basic procedures required for schematic capture. You learn how to get and place library parts, how to place wires, how to place junctions, and how to place labels and text.

Chapter 4: Capturing the power regulator schematic

In this chapter, you capture a schematic that is slightly more complex than the previous schematic. You learn how to move a group of parts, delete a part, undo a delete operation, rotate a part, place a power symbol, set a tag, jump to a tag or a reference, and print a hard copy of the schematic.

Chapter 5: Creating a custom part

In this chapter, you use **Edit Library** to define a custom part (a seven-segment display). You learn how to draw the part body, draw special shapes, use shading, add pins to the part body, add pin names, and save the new part in a library.

Chapter 6: Capturing the logic and display circuit schematic

In this chapter, you capture the final portion of the digital clock schematic. You learn how to draw a repeatable portion of the schematic, make and place multiple copies of it, write and use a macro, and use repeat parameters to place wires and labels.

Chapter 7: Using other Schematic Design Tools

This chapter introduces you to some of the other tools included in **Schematic Design Tools**. You learn to use **Annotate Schematic, Check Electrical Rules, Create Netlist, Back Annotate, Create Bill of Materials,** and **Plot Schematic.**

Chapter 8: Structuring your design

This chapter describes and reviews a complex hierarchy and shows how to convert a complex hierarchy to a simple hierarchy. Flat designs and how to link sheets are also reviewed.

Chapter 9: Tips and techniques

This chapter provides a collection of tips and techniques that you can use to enhance your ability to use **Schematic Design Tools** productively. This chapter does not follow the tutorial style of the other chapters.

Introducing Draft

In this chapter, you establish **Draft**'s work conditions. You learn how to:

- Run **ESP**
- Run **Design Management Tools**
- Set up work conditions for **Draft**, the schematic editor
- Run **Draft**
- Capture and save an initial macro
- Save your work

Before you begin

Before you begin the exercises presented in this part of the user's guide, take a minute to review the conventions used in this user's guide, and to learn some operating system basics.

Keys

Schematic Design Tools 386+ is designed to operate on a wide variety of computer systems. Since many computers label their keyboard keys differently, OrCAD has adopted standards to name two of the most widely used keys.

<Enter>

Whenever you see <Enter>, it means to press the <Enter> key on your keyboard. On your keyboard, the <Enter> key may be labeled Enter, New Line, Next, Return, or Send.

Throughout the user's guide you are instructed to enter text. For example, the instructions may read "Enter the filename." This means to type the name of the file and press <Enter>. If you are instructed to "Type the following characters," you should type the specified characters *without* pressing the <Enter> key.

<Ctrl>

Whenever you see <Ctrl>, it means to hold down the <Ctrl> key and press another key. For example, if the instructions say "press <Ctrl><A>," you should hold down the <Ctrl> key and press the <A> key.

Other keys

Alphanumeric, function, and special keys are shown in angle brackets. Examples include <F1>, <Enter>, and <Esc>.

Mouse basics

Clicking the left mouse button is the same as pressing the <Enter> key. In this user's guide, when you are instructed to "press <Enter>," you can use either the keyboard or the mouse.

Clicking the right mouse button is the same as pressing the <Esc> key. In this user's guide, when you are instructed to "press <Esc>," you can use either the keyboard or the mouse.

Keyboard input

Characters that you enter are shown in bold monospace font, such as "enter **`tutor.sch`**." This text can also be enclosed in a box:

```
tutor.sch
```

```
load file? tutor.sch
```

In the last example, you enter only the characters shown in bold. The nonbold characters show what the computer displays.

Operating system prompt

In this user's guide, the operating system prompt is shown as:

```
C:>
```

Callouts

In the later chapters of this guide, callouts such as ①, ②, and ③ appear on schematic diagrams. These callouts refer to the corresponding step numbers in the instructions.

Commands

In this guide, commands are shown in **bold** type. Main menu commands are shown in uppercase letters. Other commands are shown as they appear on the menu. When you are asked to select a command, usually both the main menu command and other command are specified.

Filenames

Filenames can be from one to eight characters long. A filename may also have a period and an extension consisting of up to three characters. You can use either uppercase or lowercase letters when entering a filename or extension, but the operating system converts all the letters to uppercase.

Filenames and extensions usually contain only letters and numbers. However, you can use additional characters supported by the operating system. For compatibility with OrCAD's environment, use only letters (A–Z and a–z), numbers (0–9), underscores (_), number signs (#), and "at" signs (@).

Most OrCAD software works with any characters your operating system supports. Some applications used in conjunction with OrCAD software—including SPICE programs, some PCB layout programs, and some text editors—support a more limited character set. You should keep any such limitations in mind as you design and avoid using characters that are allowed by one piece of software but not another.

Designs

In the OrCAD design environment, all files pertaining to a design are kept in one directory on your disk. Putting different designs in different directories lets you organize your files, much as you would organize a file cabinet.

By following the steps in this tutorial, you will work on a design named "TUTOR." All of the files for this design are contained in the directory named "TUTOR." Files in the directory have the filename "TUTOR" and an extension that indicates the type of file. For example, the TUTOR schematic worksheet that you create in chapters 2 through 6 is named TUTOR.SCH.

Running the design environment

To run an OrCAD tool, you must first display the main screen.

1. Be sure that your computer is turned on.
2. At the operating system prompt, enter the command shown in bold:

```
C:> orcad
```

In a moment, the main screen displays (figure 2-1).

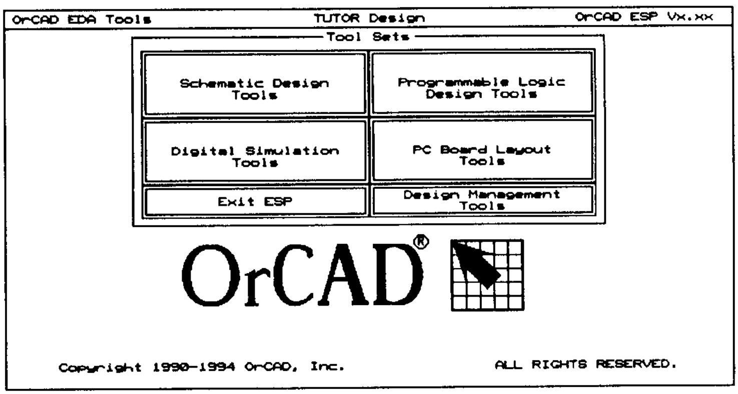


Figure 2-1. Main screen.

Changing to the TUTOR design

Before you work with any of the tools accessed from the main screen, you need to change to the TUTOR design. Remember, a design is a directory in which all the files related to a project are stored.

Run Design Management Tools

Follow these steps to run **Design Management Tools** and change to the TUTOR design:

1. Place the pointer on the **Design Management Tools** button and click the left mouse button. The menu at right displays. The **Execute** command is highlighted.

```
Design Management Tools
Execute
Local Configuration
Assign Hot Key
Configure ESP
Help
```

2. Click the left mouse button to select the **Execute** command. The screen shown in figure 2-2 displays.

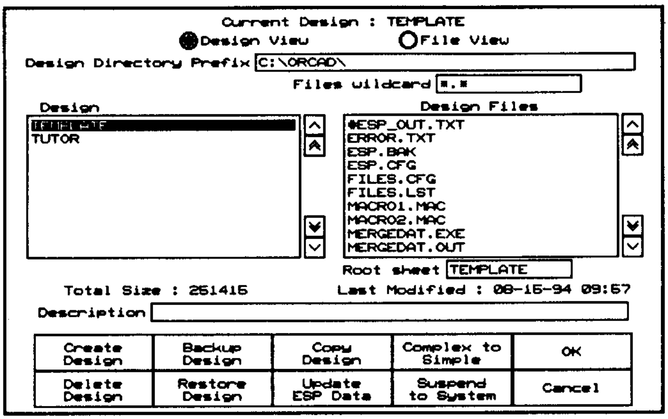


Figure 2-2. ***Design Management Tools*** *screen, Design View.*

3. Place the pointer on the design named TUTOR and click the left mouse button. This selects the TUTOR design.

4. Select **OK** to return to the main screen. Notice that the heading in the upper center of the screen has changed to **TUTOR Design.**

△ *NOTE: See the* ESP Design Environment User's Guide *for instructions on how to use* ***Design Management Tools.***

Change the start-up design

The design environment is configured to start in the TEMPLATE design each time you run the design environment. Since you will be working in the TUTOR design throughout this guide, you need to change the start-up design to TUTOR. Follow these steps:

1. Select **Design Management Tools.** The menu at right displays.

```
Design Management Tools
Execute
Local Configuration
Assign Hot Key
Configure ESP
Help
```

2. Select **Configure ESP**. The screen in figure 2-3 displays.

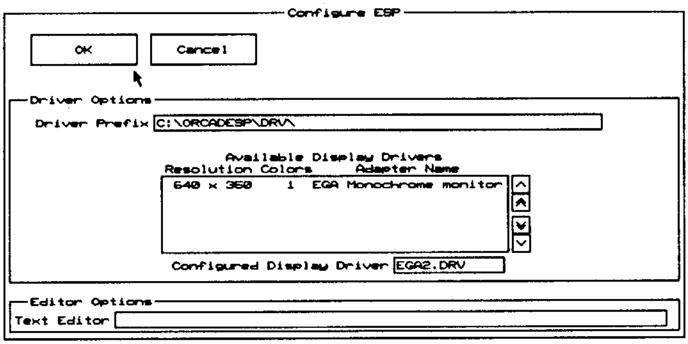


Figure 2-3. Top portion of the ***Configure ESP*** *screen.*

3. Move the pointer to the bottom of the screen. The display pans to show more of the **Configure ESP** screen. Continue panning until you reach the **Design Options** area.

```
Design Options
Startup Design  TEMPLATE
```

4. Place the pointer in the **Startup Design** entry box and click the left mouse button. The pointer becomes a cursor in the entry box. Delete `TEMPLATE`, and enter **`TUTOR`** as the start-up design.

```
Design Options
Startup Design  TUTOR
```

5. Move the pointer to the top of the screen and select **OK.** An easy way to move to the **OK** button is to press the <Home> key.

   The changes you made to the **Configure ESP** screen are saved and the main screen displays.

△ *NOTE: See the* ESP Design Environment User's Guide *for detailed instructions on how to configure the design environment.*

Creating a custom library

During this tutorial, you add a new part to a library file. It is recommended that you make no changes to libraries provided by OrCAD, but create a custom library whenever you need to make edits. This practice protects your custom library parts. Follow these steps:

1. Select **Design Management Tools** and **Execute.**
2. Make sure that * . * shows in the **Files wildcard** entry box.
3. Select **File View** at the top of the screen. The screen shown in figure 2-4 displays.

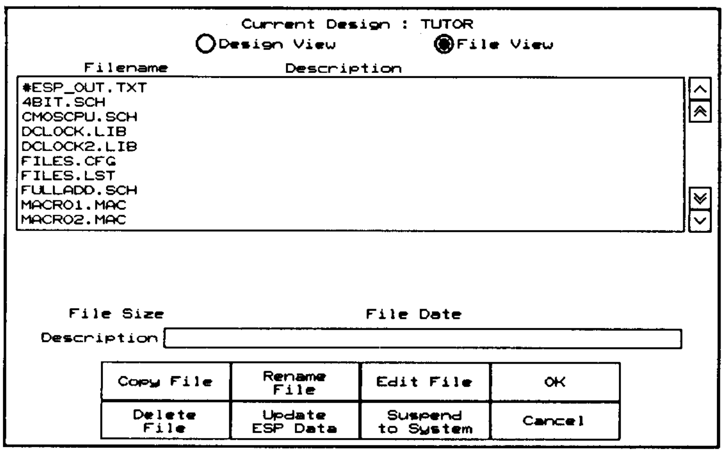


Figure 2-4. File View screen.

4. Select **Copy File**. The screen shown in figure 2-5 displays.

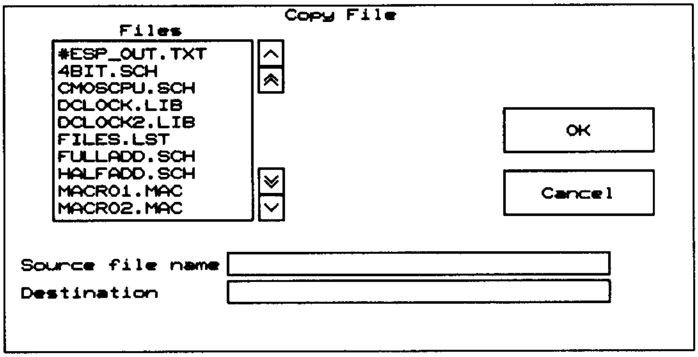


Figure 2-5. Copy File screen.

5. Select the PRESDT.LIB file from the **Files** list box. You will need to scroll down to see it.
6. Move the pointer to the **Destination** entry box and press <Enter>.
7. Enter the new name for the file, `TUTOR.LIB`, and then select **OK**.
8. Select **Cancel** to dismiss the **Copy File** screen and **OK** to return to the main screen.

Running Schematic Design Tools

Follow these steps to display the **Schematic Design Tools** screen.

1. Select **Schematic Design Tools.** The menu at right displays.
2. Select the **Execute** command. The **Schematic Design Tools** screen displays (figure 2-6).

```
Schematic Design Tools
Execute
Local Configuration
Assign Hot Key
Configure ESP
Help
```

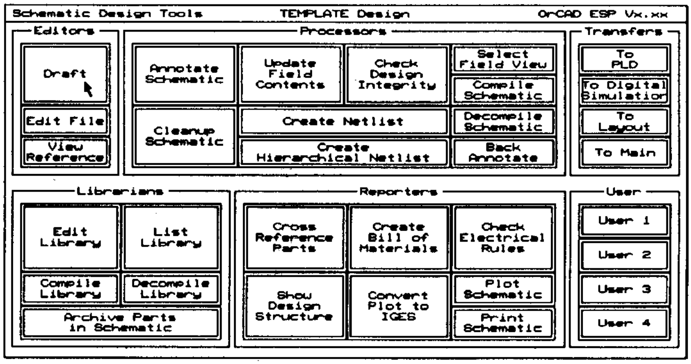


Figure 2-6. Schematic Design Tools 386+ screen.

Defining title block information

Before you run the schematic editor **Draft**, take a few minutes to configure the information to appear in the title block of the worksheet you will create. To do this, you must display the **Configure Schematic Design Tools** screen.

View the configuration for Schematic Design Tools

1. Select **Draft**. The menu at right displays.
2. Select the **Configure Schematic Tools** command.

```
Draft
Execute
Local Configuration
Assign Hot Key
Show Version
Configure Schematic Tools
Help
```

Figure 2-7 shows the top portion of the **Configure Schematic Design Tools** screen. The parameters you see may differ from those in the figure, because some of the configuration information depends on your system hardware. For more information about the **Configure Schematic Design Tools** screen, see *Chapter 1: Configure Schematic Tools* in the *Schematic Design Tools 386+ Reference Guide.*

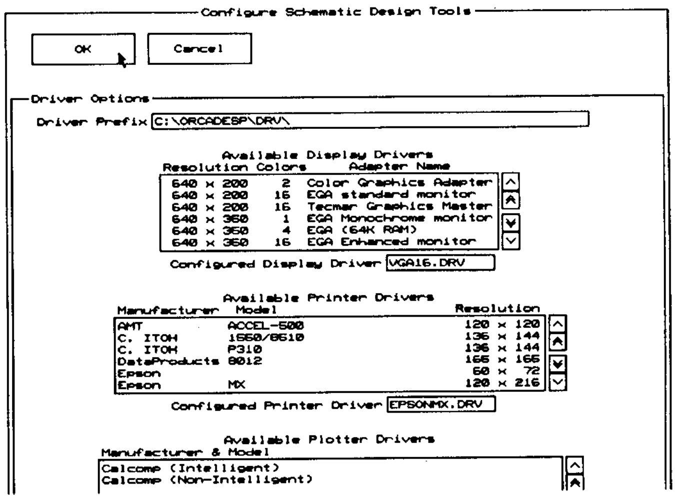


*Figure 2-7. Top portion of the **Configure Schematic Design** Tools screen.*

3. Scroll to the **Worksheet Options** area (figure 2-8).

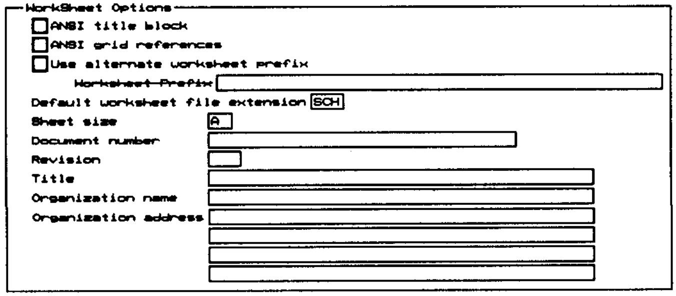


Figure 2-8. ***Worksheet Options*** *area of the* ***Configure Schematic Design Tools*** *screen.*

Notice the **Document number, Revision, Title, Organization name,** and **Organization address** entry boxes. Any information entered in these fields becomes a part of your worksheet's title block. For this tutorial, you enter information in the **Title, Organization name,** and **Organization address** entry boxes.

4. Position the pointer within the **Title** entry box and press <Enter>. Enter the title, **`Digital clock schematic`**.

5. Press <Tab> to move to the next entry box—in this case, **Organization name**—and press <Enter>. Enter the name of your organization.

△ *NOTES: Extended ASCII characters can be used in any title block entry box except* ***Sheet Size****. To enter extended ASCII characters, enable your keyboard's NumLock function, and press <Alt> while typing the appropriate number on the numeric keypad. Using the number keys on the keyboard will not produce any characters.*

The extended ASCII character set is from the MS DOS English code page table #437.

▲ *CAUTION: These extended ASCII characters are not permitted: 128 (null), 141 (carriage return), and 167 (single quote). If you use one of these extended ASCII characters, you can expect unpredictable results when you try to open your worksheet.*

6. Press <Tab> to move to the first entry box of **Organization address,** and press <Enter>. Enter the street address of your organization.
7. Press <Tab> to move to the second entry box of **Organization address,** and press <Enter>. Enter the city and state of your organization.
8. Move the pointer to the top of the screen and select **OK.** This updates the configuration and displays the **Schematic Design Tools** screen.

Running Draft

Now that you have changed the start-up design to TUTOR and set up your title block information, you are ready to begin learning about the schematic editor **Draft**.

1. Select **Draft**. The **Draft** menu displays.
2. Select the **Execute** command to run **Draft**.

   The top and left edges of the new worksheet display, as shown in figure 2-9. Because the screen is smaller than the worksheet, the right and bottom edges of the worksheet are not visible. You can think of the screen as a window into the larger worksheet area.

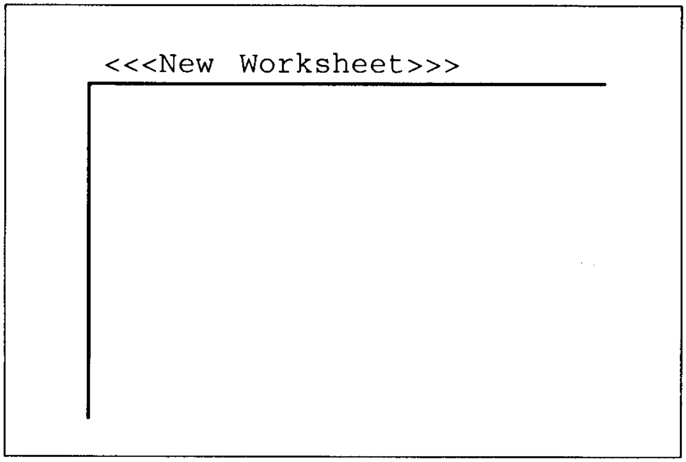


*Figure 2-9. New worksheet in **Draft**.*

Learning OrCAD basics

Pop-up menus guide you step by step through OrCAD software. **Draft** organizes commands using menus and command lines. You can select a command by either clicking the mouse or pressing a key.

△ *NOTE: For complete command descriptions, see the* Schematic Design Tools 386+ Reference Guide.

Main menu

Press <Enter> or click the left mouse button to see the main menu (shown at right). Press <Esc> or click the right mouse button to remove the main menu from the screen.

To return to the main menu—no matter where you are in **Draft**—press <Esc> or click the right mouse button as many times as necessary until no menu or command line displays in the upper left corner of the screen, and then press <Enter> or click the left mouse button.

```
Again
Block
Conditions
Delete
Edit
Find
Get
Hardcopy
Inquire
Jump
Library
Macro
Place
Quit
Repeat
Set
Tag
Zoom
```

Commands

There are several ways to select a command. You can use the methods shown in table 2-1 in any combination. The method you use is a matter of personal preference.

	Using the keyboard	*Using the mouse*
To highlight a menu command	Press the up and down arrow keys to slide the highlighting over the command.	Move the mouse to slide the highlighting over the command.
To select a highlighted menu command	Press <Enter>.	Click the left mouse button.
To select any command	Press the first capital letter in the command name.	

Table 2-1. Selecting commands.

Draft responds to a command either by performing the command's function or by displaying another menu or a command line.

Menus

All menus look and work just like the main menu. **Draft** displays the menu name on the top line of the screen. Press <Esc> or the right mouse button to return to the menu or command line that called the current menu. Follow these steps to familiarize yourself with these processes:

1. Press <Enter> to display the main menu.
2. Select the **BLOCK** command. The menu at right displays.
3. Press <Esc> to dismiss the **BLOCK** menu.

```
Block
Move
Drag
Fixup
Get
Save
Import
Export
ASCII Import
Text Export
Copy
```

Command lines

Command lines are a series of commands listed across the top of the screen. When a command line displays, you can move the pointer around the working area or select a command (by pressing the first capitalized letter in its name). Press <Esc> or the right mouse button to return to the menu or command line that called the command line. Follow these steps to familiarize yourself with these processes:

1. Press <Enter> to display the main menu.
2. Select the **EDIT** command. The **EDIT** command line appears at the top of the screen:

```
Edit  Find  Jump  Zoom
```

3. Press <Esc> to dismiss the **EDIT** command line.

Returning to the main menu

To return to the main menu—no matter where you are in Draft—press <Esc> as many times as necessary until no menu or command line displays in the upper left corner of the screen, and press <Enter>.

How commands are shown in this guide

As described earlier in this chapter, commands are shown in **bold** type. Main menu commands are shown in uppercase letters. For example, the statement "Select the **PLACE Wire** command" means "Select the **PLACE** command from the main menu, and select the **Wire** command from the **PLACE** menu."

When you are asked to select a command, usually both the main menu command and other command are specified. Where the context is clear, though, the main menu command is not specified. For example, if the **PLACE** menu already displays, and you are asked to select the **Wire** command, the instruction is simply "Select the **Wire** command."

Setting up Draft's work conditions

Now that you understand how **Draft's** commands, menus, and command lines operate, you will use the **SET** command to change the default work conditions that govern the way **Draft** displays and maintains schematics.

Display work conditions settings

Follow these steps to display the **SET** menu:

1. Press <Enter> to see the main menu.
2. Select **SET** from the main menu. The **SET** menu displays, as shown below.

```
Set
Auto Pan            YES
Backup File         YES
Drag Buses           NO
Error Bell          YES
Left Button          NO
Macro Prompts       YES
Orthogonal          YES
Show Pins           YES
Title Block         YES
Worksheet Size        A
X,Y Display          NO
Grid Parameters
Repeat Parameters
Visible Lettering
Cursor Style
```

Using the commands in the **SET** menu, you can control features such as automatic backup of schematic files, the angles at which you can place wires, and whether or not pin numbers display on part symbols. For more information about **Draft's** work conditions, see the **SET** command description in the *Schematic Design Tools 386+ Reference Guide.*

The next few sections describe a few of **Draft's** work conditions and the commands controlling them.

Pan across the schematic

Auto Pan is the first command in the **SET** menu. When you start work on a new worksheet, **Auto Pan** is set to **Yes**.

When **Auto Pan** is set to **Yes**, the worksheet follows the movement of the pointer. If part of a worksheet is off the screen and you move the pointer beyond the edge of the display, the hidden part of the worksheet pans into view.

If you set **Auto Pan** to **No**, the screen does not pan. In this case, you must use the **JUMP** and **ZOOM** commands to see different parts of the worksheet.

Follow these steps to pan across the schematic:

1. Press <Esc> to dismiss the **SET** menu. **Auto Pan** remains set to **Yes**.

2. Move the pointer to the lower right corner until the title block displays. The screen pans to keep up with the pointer. Notice that the title block information you entered earlier in this chapter displays.

3. Move the pointer toward the upper left corner until the upper left corner of the worksheet displays.

Redisplay the SET menu

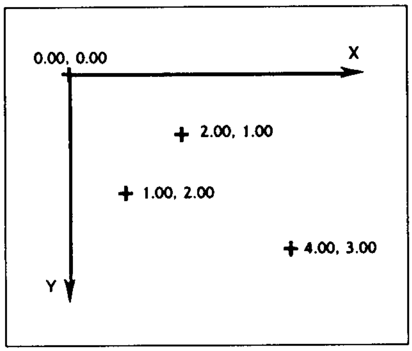
1. Press <Enter> to recall the main menu. When the menu displays, you'll see the highlight bar is on the **AGAIN** command.

2. Press <Enter> to select **AGAIN**. This selects the main menu command you chose last—in this case, **SET**.

Display X,Y coordinates

Draft uses a coordinate system to locate points on the worksheet, as shown in the illustration below.

An X coordinate specifies horizontal location and a Y coordinate specifies vertical location. Thus any point on the worksheet can be indicated by an X and Y coordinate pair in the form (X,Y). The (0.00, 0.00) point is always at the upper left of the worksheet.



If **X,Y Display** is set to **Yes**, the X and Y coordinates of the pointer's position display in the upper right corner of the screen. The default setting is **No**.

Follow these steps to display X, Y coordinates:

1. Select **X,Y Display**. "Show Coordinates?" and a short menu display.
2. Select **Yes**. The prompt and the menu disappear.
3. Move the pointer in any direction and watch the X,Y coordinates in the upper right corner of the screen.

The units shown in the X,Y display represent inches on the printed schematic. The upper left corner is (.00, .00) and the lower right corner is (9.50, 7.00). On a sheet 8.5 inches by 11 inches, the actual drawing area is 7 inches by 9.5 inches. This allows for borders around the drawings.

Select worksheet size

The **Worksheet Size** command selects one of five sizes for your schematic. Follow these steps to change the worksheet size:

1. Press <Enter> to display the main menu, then select **AGAIN**. The **SET** menu displays.
2. Select **Worksheet Size**. A menu lists the five options available for the size of a worksheet, as shown at right.

```
Set Worksheet Size
(Area inside borders)
A    9.70 x  7.20
B   15.20 x  9.70
C   20.20 x 15.20
D   32.20 x 20.20
E   42.20 x 32.20
```

3. Select **C** size.
4. Move the pointer to the edges and corners of the worksheet to explore the size of the editable region of a C-size sheet. The dimensions shown in the **Worksheet Size** menu are the worksheet's borders. On a C-size sheet 22 inches by 17 inches, the actual drawing area is 20 inches by 15 inches.

△ *NOTE: If* ***Schematic Design Tools*** *is configured to use metric dimensions, the* ***Set Worksheet size*** *menu displays the International Standards Organization (ISO) paper sizes: A4 through A0. In addition, the X, Y display is given in millimeters. For information about configuring* ***Schematic Design Tools*** *to use metric dimensions, see* Chapter 1: Configure Schematic Tools *in the* Schematic Design Tools 386+ Reference Guide.

Changing your view of the worksheet

Draft can display worksheets at five different scales. You change the view using the **ZOOM** command. The worksheet can be zoomed in or out to magnify or reduce its visible image.

When **Draft** is zoomed out, you can see a large portion of the worksheet. Zooming in enlarges a small portion of the worksheet and displays more details. You can zoom in to draw intricate portions of your worksheet with exacting detail and then zoom out to look at the whole schematic.

ZOOM in and out

Follow these steps to zoom out and see more of the worksheet on the screen at one time:

1. Move the pointer to lower right corner until the title block displays.
2. Select **ZOOM** from the main menu. The menu at right displays.
3. Select **Out**. A view of the worksheet at one-half the original scale displays.

```
Zoom (present
scale=1)
Center   ( 1)
In       ( 1)
Out      ( 2)
Select
```

4. Experiment with the scale using **In**, **Out**, and **Select**. If you use **Select** you can choose the scale at which to view the worksheet, as shown below. Notice that this menu shows the keyboard shortcuts on the left (1, 2, 5, 0, and T) and the zoom scale associated with each keyboard short cut on the right (1, 2, 5, 10, and 20).

   If you choose **1**, you view the worksheet at full size. This shows the most detail ("zooms in" the closest). If you choose **2**, you view the worksheet at one-half the original size. If you choose **T** (for 20), you view the worksheet at one-twentieth the original size—you see the maximum working area and the least detail.

```
Zoom - Select Scale
(present scale=1)
1        (1)
2        (2)
5        (5)
0        (10)
T        (20)
```

5. When you finish experimenting with zooming, return to full-size view (scale level 1).

Set grid parameters

While working on a large worksheet, it is useful to have visual cues that tell you approximately where you are on the sheet.

The **Grid Parameters** command on the **SET** menu sets up some of these visual cues. The **Set Grid Parameters** menu is shown at right.

```
Set Grid Parameters
Grid References     NO
Stay On Grid        YES
Visible Grid Dots   NO
```

Display grid references

Grid References turns grid reference guides along the top and left edges of the display on and off. The guides divide the worksheet into blocks. Horizontally, the guides divide the worksheet from 8 to 1. Vertically, they divide the worksheet from D to A. For example, the title block (lower right corner) is located at A-1, as illustrated in figure 2-10.

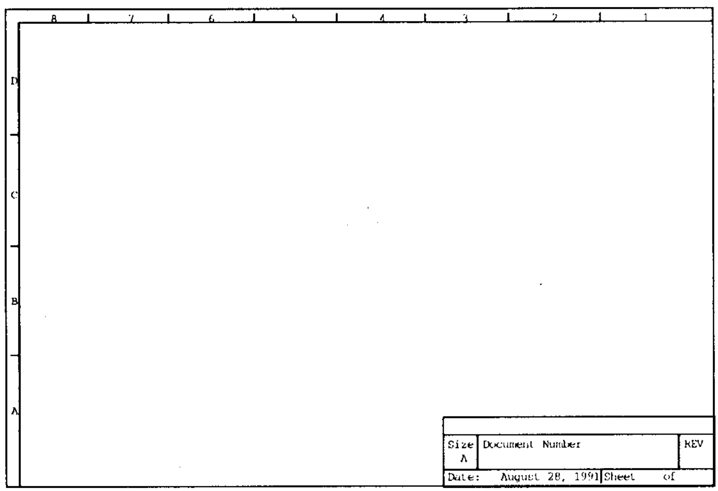


Figure 2-10. Grid references.

Use the **JUMP Reference** command on the main menu to move to specific locations using these map-like coordinates.

Follow these steps to display grid references:

1. Select **SET** from the main menu.
2. Select **Grid Parameters.**
3. Select **Grid References.**
4. Select **Yes.** The grid reference guides display at the top and left edges of the screen.

△ *NOTE: Schematic Design Tools 386+ can be set up to use ANSI Y14.1 drawing standards. Refer to the* Schematic Design Tools 386+ Reference Guide *for details.*

Stay on grid

Stay on Grid determines whether or not pointer movement is restricted to grid intersections. **Stay On Grid** is set to **YES.** Do not make any changes here.

△ *NOTE: Keep Stay on Grid set to Yes unless you have a compelling reason to be off-grid. Anything placed off-grid—such as text and labels—may be hard to select and edit later.*

Make the grid visible

Visible Grid Dots turns the dots representing intersections on and off. The space between the dots represents 0.1 inch on the printed worksheet. Follow these steps to make the grid dots visible:

1. Select **SET** and **Grid Parameters** again.
2. Select **Visible Grid Dots,** then select **Yes.** Grid dots display on the worksheet. You can make the grid dots brighter or dimmer by adjusting the intensity on your monitor or by changing the color selection in the **Color and Pen Plotter Table** area of the **Configure Schematic Design Tools** screen.

Updating the worksheet

When you work on a schematic for a long time, it is important to periodically save your work on disk as a precaution against power failures and other unexpected events.

Update the file

Follow these steps to save the worksheet without changing its filename:

1. Select **QUIT** from the main menu. **Draft** displays the filename and the **Quit** menu, as shown at right.
2. Select **Update File. Draft** saves the file.
3. Press <Esc> to dismiss the **Quit** menu.

```
Quit TUTOR.SCH
Enter Sheet
Leave Sheet
Update File
Write to File
Initialize
Suspend to System
Abandon Edits
Run User Commands
```

Creating a macro

Macros can record virtually anything you do in a program—so you can automate many repetitive tasks and speed up your work.

Capture a macro

Earlier in this chapter, you used the **SET** command to change work conditions. Follow the steps below to capture commands for setting work conditions in a macro.

△ *NOTE: This macro only works when you are at the main menu level of* ***Schematic Design Tools****.*

1. Select **MACRO** from the main menu. The **MACRO** menu at right displays.
2. Select **Capture**. The prompt "Capture macro?" displays.

   Macros can be run by a single key or a combination of keys.

```
Macro
Capture
Delete
Initialize
List
Read
Write
```

Single keys that can run macros are the function keys (<F1> through <F10>) and special keys in the numeric keypad (<Ins>, <PgUp>, and <PgDn>).

Key combinations that can run macros include:

- <Ctrl> with function keys
- <Ctrl> with alpha keys (except C, H, and M)
- <Alt> with function keys
- <Alt> with alpha keys
- <Shift> with function keys

If you choose a prohibited key combination, **Draft** displays "ERROR: Key cannot be defined as macro" and displays the "Capture macro?" prompt again.

3. Press <Ctrl><A> to assign a keystroke to this macro. "^A" displays at the "Capture macro?" prompt.
4. Press <Enter>. The message "<macro>" displays to remind you that you are defining a macro. Any commands you select while "<macro>" displays are added to the list of commands stored in the macro.
5. Type the keys shown in the left column below. **Draft** performs the commands as it captures them.

Key sequence	*Commands*	*Effect*
<Enter> <Esc>	*none*	Clear screen
SXY	**SET** **X,Y Display** **Yes**	Display coordinates
SGGY	**SET** **Grid Parameters** **Grid References** **Yes**	Display grid references
SGVY	**SET** **Grid Parameters** **Visible Grid Dots** **Yes**	Display grid dots
ZS1	**ZOOM** **Select** **1**	Display schematic at full size

6. Press <Ctrl><End> to end the macro definition. **Draft** displays "<<<MACRO END>>>" to confirm that the macro definition is complete.

   The macro is now stored in the computer's memory. You can run it when you are at the main menu level of **Draft** by pressing the key combination you specified, <Ctrl><A>.

△ *NOTE: Some keyboards have two keys labeled <End>. If you press <Ctrl><End> and "<<<MACRO END>>>" does not appear, try using the other <End> key.*

Save the macro

1. Select **MACRO Write**. The prompt "Write all macros to?" displays.
2. Enter `tutor.mac`. **Draft** writes the macro to the TUTOR.MAC file in the TUTOR design directory.
3. To tell **Draft** to read macros from the TUTOR.MAC file, select **MACRO Read**. The prompt "Read all macros from?" displays.
4. Enter `tutor.mac`.
5. If you wish to test the macro you just saved, change some of the work conditions and press <Ctrl><A> to restore them.

Exiting Draft

You are nearly finished with this chapter. Follow these steps to exit **Draft**:

1. Select **QUIT** from the main menu. **Draft** displays the filename and the **QUIT** menu shown at right.
2. Select **Update File**. **Draft** saves the file.
3. Leave **Draft** by selecting **Abandon Edits**. **Draft** exits to the **Schematic Design Tools** screen.

```
Quit TUTOR.SCH
Enter Sheet
Leave Sheet
Update File
Write to File
Initialize
Suspend to System
Abandon Edits
Run User Commands
```

Setting up automatically

In addition to using **SET** to control **Draft's** work conditions, you can automate the process of defining **Draft** work conditions by configuring **Schematic Design Tools** so that the macro you just created runs every time you run **Draft.** A macro that runs when the tool starts is called an *initial macro.*

1. Select **Draft** from the **Schematic Design Tools** screen.
2. Select **Configure Schematic Tools** from the menu that displays. The **Configure Schematic Design Tools** screen displays.
3. Scroll to the **Macro Options** area of the **Configure Schematic Design Tools** screen.
4. Position the pointer within the **Draft Macro File** entry box and press <Enter>. This entry box defines the name of the macro file you created earlier in this chapter.
5. Enter the macro path and filename:

   ```
   Draft Macro File  \orcad\tutor\tutor.mac
   ```

   Notice that the **Draft Initial Macro** entry box becomes accessible (not dimmed) when you enter text in the **Draft Macro File** entry box.
6. Position the pointer within the **Draft Initial Macro** entry box and press <Enter>. This entry box defines a macro that automatically runs when you run **Draft.**

   <Ctrl><A> is the keystroke that runs the macro. In the **Draft Initial Macro** entry box, however, you use a caret symbol (^) to represent the <Ctrl> key.
7. In the **Draft Initial Macro** entry box, simultaneously press <Shift> and <6> to enter the caret symbol (^), and then enter **A**. The entry box should look like this:

   ```
   Draft Initial Macro  ^A
   ```
8. Move the pointer to the top of the screen and select **OK.** This updates the configuration and displays the **Schematic Design Tools** screen.

△ *NOTE: Once you configure ^A in the **Draft Initial Macro** entry box on the **Configure Schematic Design Tools** screen, the macro runs automatically each time you run **Draft**. You can also run it when you are at the main menu level of **Draft** by pressing <Ctrl><A>.*

Summary

In this chapter you learned how to run **Draft** and examine and modify work conditions. You also learned how to capture an initial macro and have it automatically set the work conditions each time you run **Draft**.

The next chapter gives you instructions for capturing the schematic for the clock oscillator circuit. In later chapters, you build on the knowledge you gain while learning more about **Schematic Design Tools 386+**.

Capturing the clock oscillator schematic

This chapter shows you the processes used to create a basic schematic. In this chapter, you learn how to:

- ❖ Get and place library parts
- ❖ Draw wires
- ❖ Place junctions
- ❖ Place labels and text

Running Draft

Figure 3-1 shows the schematic diagram of the clock oscillator circuit you create in this chapter. Refer to this figure for placement and orientation information while capturing the clock oscillator schematic.

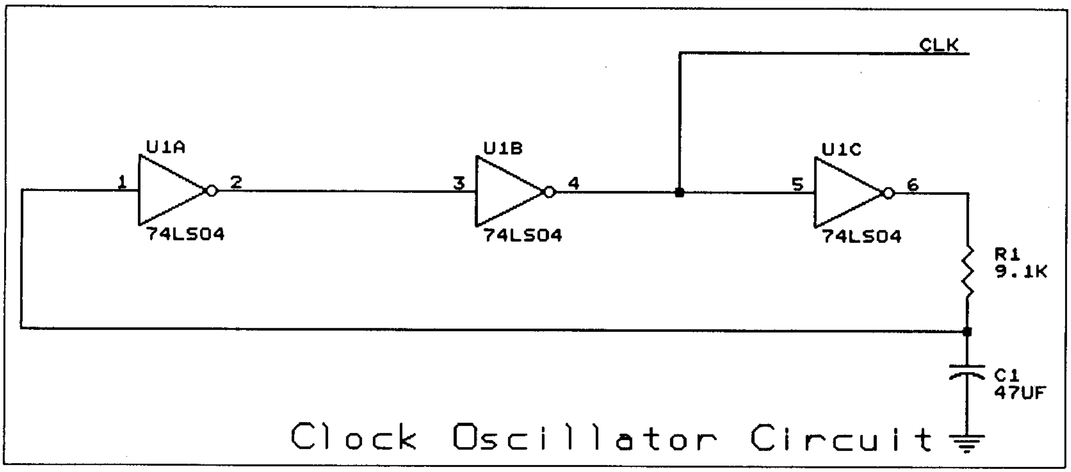


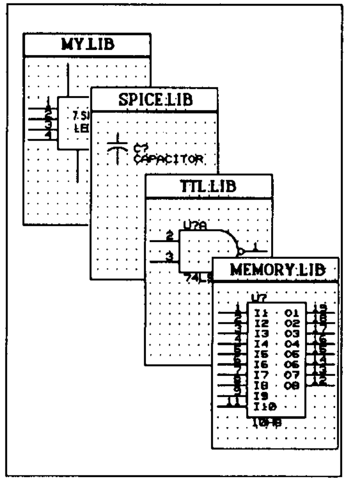
Figure 3-1. Clock oscillator circuit schematic.

About symbols

The first step in building a schematic diagram with **Draft** is to place symbols for the parts on the worksheet. The symbols can represent basic logic functions (such as AND gates), individual parts (such as capacitors), or blocks of circuitry to be designed later. The symbols can represent parts that use different technologies, such as TTL or CMOS.

About libraries

Symbols representing parts are stored in libraries. For **Draft** to get a symbol and place it on a schematic, the library containing it must be configured on the **Configure Schematic Design Tools** screen.



Parts libraries.

As shown in the illustration at right, library filenames typically end with the extension .LIB.

To build the clock oscillator, you need the following symbols:

- Three inverters
- One resistor
- One capacitor

The examples in this tutorial use TTL technology for the inverters.

Where to start

If you are continuing from chapter 2, the **Schematic Design Tools** screen is displayed. Follow these steps if it is not displayed:

1. If the operating system prompt is displayed, enter **ORCAD**.

△ *NOTE: In chapter 1, you set the start-up design to be TUTOR. Check to be sure that "TUTOR Design" is displayed in the middle of the top line of the screen. If it is not, go into **Design Management Tools** and change to the TUTOR design. This process is described in detail in chapter 1.*

2. On the main screen, select **Schematic Design Tools** and then select **Execute**.

Configure library files

Follow these steps to configure the libraries that are needed for the tutorial:

1. On the **Schematic Design Tools** screen, select **Draft**.
2. Select **Configure Schematic Tools.** The **Configure Schematic Design Tools** screen displays.
3. Scroll down until you can see the **Library Options** area.

   As shown in figure 3-2, **Library Options** shows **Available Libraries** on the left, and **Configured Libraries** on the right.

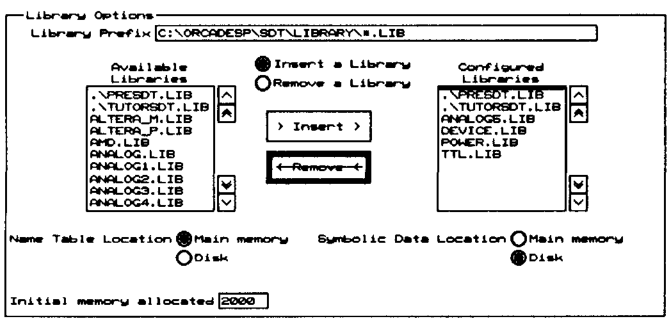


*Figure 3-2. **Library Options** area of **Configure Schematic Design Tools** screen.*

Draft loads and maintains libraries in your computer's memory in the order in which they are listed in the **Configured Libraries** box. This is important when retrieving parts while creating schematics. When you tell **Draft** to get a certain part name, it searches the libraries in the order listed during configuration and gets the *first* part it finds with a matching name.

4. For this chapter, **Draft** needs the library files TUTOR.LIB and PCBDEV.LIB. The library TUTOR.LIB must be listed first in the **Configured Libraries** box.

   Scroll the **Available Libraries** list box up and down by clicking the up and down arrow keys to the right of the list box. The double-arrow keys scroll the list a full box at a time.

△ *NOTE: Libraries that reside in the current design (instead of in the configured library directory) are preceded with ".\" and are listed at the beginning of the* **Available Libraries** *list. In the steps in this section, .\TUTOR.LIB refers to a library in the TUTOR directory.*

5. Locate .\TUTOR.LIB in the **Available Libraries** list box and select it. Select **Insert** to place it in the **Configured Libraries** list box.

6. Repeat step 5, but this time select PCBDEV.LIB. Notice that PCBDEV.LIB is added to the list at the position of the green highlight bar, which is located after the library you just inserted (.\TUTOR.LIB).

△ *NOTE: If any libraries other than .\TUTOR.LIB and PCBDEV.LIB are listed in the* ***Configured Libraries*** *list box, remove them. To do this, select* ***Remove a Library****, select the library to remove, and then select* ***Remove****.*

7. Scroll to the top of the configuration screen and click **OK** to return to the **Schematic Design Tools** screen.

8. Select **Draft**, and then select **Execute**.

   Draft runs the initial macro you captured in chapter 2. This macro sets the viewing scale to full size and causes the X,Y coordinates, grid references, and visible grid dots to be displayed. When the macro finishes running, a blank worksheet displays.

Placing parts

Follow these steps to get part symbols from part libraries:

1. Select the **GET** command from the main menu. The "Get?" prompt displays.

2. Press <Enter> to display a menu listing the configured libraries.

   The menu at right shows the libraries configured for the TUTOR design.

   ```
   Get?
   .\TUTOR.LIB
   PCBDEV.LIB
   ```

3. Be sure the highlight is on .\TUTOR.LIB and press <Enter>. A list of the parts stored in TUTOR.LIB displays, as shown at right.

   ```
   Get?
   22V10
   4SW SPST
   74LS04
   BATTERY
   CAP
   GND
   LM7805
   R
   SW PUSHBUTTON
   ```

4. Select the 74LS04 inverter. An image of the part displays on the worksheet and a command line displays across the top of the screen. When you move the part, the image simplifies temporarily—only the object's outline displays. When you stop moving the part, details redisplay.

5. Move the part to its general location on the worksheet. Refer to the grid reference bars at the left and top edges of the display and use the mouse to move the image to region A-3.

6. To move the part to its precise location, refer to the X,Y grid display at the upper right of the screen and move the image until the display shows that it is at location (12.80, 11.80). You can use the arrow keys to position the part. The part's upper left corner is its reference point for positioning.

7. Press <Enter> and select **Place** from the menu that displays. **Draft** places the part on the worksheet and creates another movable image of the part.

8. Since you need two more copies of the inverter, place copies of the part at locations (14.80, 11.80) and (16.80, 11.80).
9. When you have placed all three parts, press <Esc> to end the operation.

Shortcuts for getting parts

If you know the full name of a part you want to get from a library, you don't have to work your way through the menus. Simply type the complete part name at the "Get?" prompt. For example, if you enter **R** in response to the "Get?" prompt, **Draft** searches through the libraries and displays a resistor.

Place the remaining parts

Follow these steps to add a resistor, a capacitor, and a ground symbol to the clock oscillator circuit:

1. Press <G>, <R>, and <Enter> to select the resistor part from the TUTOR.LIB library. An image of the resistor displays on the worksheet.
2. Move the image to location (17.60, 12.30) and press <P> (or press <Enter> and select **Place**) to place the resistor.
3. Press <Esc>.
4. Press <G> to display the "Get?" prompt. Enter **CAP**. An image of the capacitor displays on the worksheet.
5. Place the capacitor at location (17.60, 13.00).
6. Press <Esc>.
7. Press <G> to display the "Get?" prompt. Enter **GND**. An image of the ground symbol displays on the worksheet.
8. Place the ground symbol two grid spaces below the bottom of the capacitor symbol, and then press <Esc>.

You have now placed all the parts and symbols for the clock oscillator circuit on the worksheet. The next step is to place the wires.

Drawing wires

Compare your worksheet with figure 3-1 at the beginning of this chapter. Your worksheet should contain the parts shown in figure 3-1, but not the wires. Most of the remaining tasks in this chapter establish signal connections between the parts you placed on the worksheet.

1. Select **PLACE** from the main menu. The **PLACE** menu displays.
2. Select **Wire**. The **PLACE Wire** command line displays.
3. Move the pointer until it rests at the free end of the output pin of the leftmost inverter. This is location (13.50, 12.00).
4. Select **Begin**, then move the pointer right to the input pin of the next inverter.
5. Select **End**. The wire segment is completed.
6. To complete the wiring, draw wires between the remaining parts as shown in figure 3-1.

   You can speed up wire drawing two ways:

   - Select **New** instead of **End** for each wire except the last one.
   - Instead of using the mouse to select the required commands, press <P>, <W>, <B>, <N>, and <E>.

△ *NOTE: When drawing wires, be sure to begin and end each wire segment at the end of a part pin, not within the body of the pin. Also be sure that the end of a wire does not overlap a pin. If you accidentally overlap wires on pins or part bodies, error messages result when you use certain **Schematic Design Tools** processors.*

Placing junctions at intersections

Wires that cross one another do not represent a connection. To tell **Draft** that the crossing wires are connected, you must define the intersection as a wire junction. You do this by placing a junction at the intersection. If two wires (or a wire and a part pin) are connected end to end, however, a junction is not necessary.

The connection between the capacitor and the input of the leftmost inverter requires a junction. No junction is necessary for the connection between the resistor and the capacitor because they connect end to end. A junction is also required where the wire labeled CLK (figure 3-1) connects, between the middle and right-hand inverters.

Place junctions

1. Select **PLACE Junction.**
2. Put the pointer on one of the wire intersections and select **Place.** A junction displays.
3. Place a junction at the other intersection by putting the pointer on it and selecting **Place.**
4. Press <Esc> to dismiss the **Place** command line.

You aren't finished with this circuit yet. You still have to assign values to the resistor and capacitor, add a signal label, and assign reference designators to all the parts. These steps are described in the next sections.

Editing part fields

Each part in **Schematic Design Tools** has ten reserved data areas called *part fields* for holding and displaying additional information. For example, you might use part fields to record part numbers on the schematic—making it easier to track and order parts—or to specify the multiple-element part to which a particular part belongs.

Two of the ten part fields are reserved for particular types of data:

❖ The **Reference** field is reserved for holding reference designator values, such as "U1A" and "Q1."

❖ The **Part Value** field is reserved for holding part names, such as "74LS04," or values relevant to the part, such as ohm (Ω) values for resistors.

To be processed correctly by **Schematic Design Tools 386+**, every part *must* have data in the **Reference** field and in the **Part Value** field.

The other eight fields are named **1st Part Field** through **7th Part Field,** and **Module Value.**

The data in a part field can be up to 128 characters long. You can edit the contents of these fields and make them visible or invisible on the schematic using the **EDIT** command.

In this chapter, you learn how to edit part fields one at a time. You can also automate part field editing using the **Update Field Contents** tool, as described in *Chapter 7: Using other Schematic Design Tools.*

For the inverters

The three 74LS04 inverters that you placed are actually three of the six inverters that make up the multiple-element part called 14DIP300. In the steps below, you will give each of the inverters the name of the multiple-element part that it belongs to by entering 14DIP300 in the **Module Value** field, and you will use the **Which Device** command to associate each of the six inverters with one of the six elements in the 14DIP300 multiple element part.

```
Edit part
Reference
Part Value
1st Part Field
2nd Part Field
3rd Part Field
4th Part Field
5th Part Field
6th Part Field
7th Part Field
Module Value
SheetPart Name
Orientation
Which Device
```

1. Select **EDIT** from the main menu.
2. Put the pointer on the part you want to edit, in this case the leftmost inverter.
3. Select **Edit**. The **Edit part** menu at right displays.
4. Select **Module Value**. The menu at right displays.

```
Module Value
Name
Location
Visible
```

5. Select **Name. Draft** displays:

```
Module Value?
```

6. Enter **`14DIP300`**. The information displays below the inverter symbol.
7. Select **Which Device** from the **Edit Part** menu.

   The prompt "Which device?" and a list of suffix letters (A through F) displays. A through F represent the six 74LS04 inverters in the 14DIP300 multiple-element part.
8. Select **A** from the list and press <Esc>.
9. Repeat steps 2 through 8 for the other inverters you placed. Since they are from the same multiple-element part, enter **14DIP300** for each, and assign suffix letters **B** and **C** to them.
10. Press <Esc> again to remove the **EDIT** command line from the screen.

About reference designator assignments

Notice that the suffix letters of the second and third reference designators you just modified changed to U?B and U?C, respectively. U?A is the first part of the multiple-element part, U?B is the second part, and U?C is the third part. When you run the **Annotate Schematic** tool on this schematic, all of the question marks for this multiple-element part are changed to a common number, such as 4. The parts will then be labeled U4A, U4B, and U4C.

Annotate Schematic automatically updates part reference designators and pin numbers associated with the reference designators in multiple-element parts. The **Annotate Schematic** tool is described in *Chapter 7: Using other Schematic Design Tools.*

For the resistor and capacitor

Follow these steps to edit the part fields for the resistor and capacitor:

1. Select **EDIT** from the main menu.
2. Put the pointer on the resistor.
3. Select **Edit**. The **Edit Part** menu displays.
4. Select **Part Value** and **Name**. **Draft** displays:

```
Value? R
```

5. To change the value, backspace over the present value and enter `9.1k`.
6. Press <Esc> to dismiss the **Edit Part** menu.
7. Put the pointer on the capacitor.
8. Repeat steps 3 through 6, entering `47uF` for the part value of the capacitor, measured in microfarads (uF).
9. Press <Esc> again to dismiss the **EDIT** command line.

You are nearly finished with the schematic for the clock oscillator circuit. In the next section, you learn to connect a wire in your circuit using a label. The label allows another circuit on the worksheet to behave as though it is directly connected to the output of this circuit.

Specifying connections with labels

Sometimes you may want to connect wires far apart on the worksheet. To keep the worksheet from looking cluttered, you'd like to do so without having to place a line representing the wire connecting them. You can do this by assigning a label with the same name to both wires.

1. Select **PLACE** from the main menu.
2. Select **Label**. At the "Label?" prompt, enter **CLK**. The label displays.
3. Position the label image so the pointer rests on the unconnected output wire of the clock oscillator circuit. Labels must be placed with the leftmost point of the label name touching the bus or wire.
4. Select **Place**. The "Label?" prompt redisplays.
5. Press <Esc> to dismiss the "Label?" prompt.

Schematic Design Tools 386+ treats all wires on this sheet labeled "CLK" as connected, just as if you had drawn the wire from the clock oscillator circuit directly to the other area of the schematic that uses it. You will refer to this wire label in a later chapter of this guide.

Placing comment text

You may often want to leave notes or descriptive text on a schematic diagram. Such text helps you and others understand the functions being performed or documents some aspect of circuit operation. Follow these steps to add a title:

1. Select **PLACE** from the main menu, and then select **Text**.
2. The prompt "Text?" displays. Enter **`Clock Oscillator Circuit`**.
3. To use the next larger type size for the text, select **Larger**. The image of the text becomes larger.
4. Move the text image so it is centered immediately below the circuit diagram and select **Place**. The "Text?" prompt redisplays.
5. Press <Esc> to dismiss the "Text?" prompt.

△ *NOTE: You may wish to use the* ***ZOOM Center*** *command to center the circuit before placing this text.*

Updating the file

This circuit is now complete. To save your work and exit **Draft**, follow the same steps you took earlier. Select **QUIT**, then **Update file**, to save the worksheet in TUTOR.SCH. Select **Abandon Edits** to exit from **Draft** and return to the **Schematic Design Tools** screen.

Summary

You just completed the schematic diagram for the clock oscillator circuit of the digital clock. In the next chapter, you capture the schematic of the power regulator circuit.

Capturing the power regulator schematic

In this chapter you use the processes you have already learned and also learn how to:

- ❖ Move a group of parts
- ❖ Delete a part
- ❖ Undo a delete
- ❖ Rotate a part
- ❖ Place a power symbol
- ❖ Capture and use a macro
- ❖ Set a tag
- ❖ Jump to a tag or reference location
- ❖ Print the worksheet

Figure 4-1 shows the schematic diagram of the power regulator circuit you create in this chapter. Refer to this figure for placement and orientation information while performing the exercise.

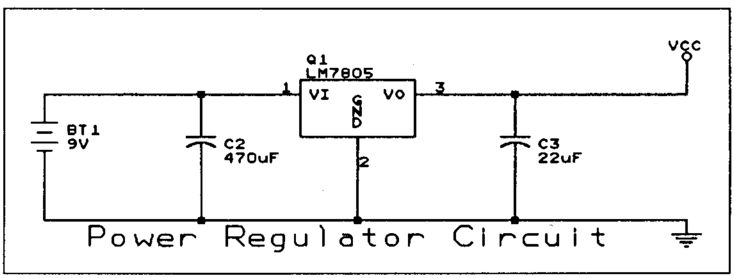


Figure 4-1. Power regulator circuit schematic.

Continuing schematic capture

If you did not abandon edits at the end of chapter 3, you can skip to the next section. Otherwise, follow these steps:

1. From the **Schematic Design Tools** screen, select **Draft**.
2. Select **Execute**. The last active view of the TUTOR.SCH schematic displays.

Moving a group of objects

Although you could just move your viewpoint over to another area of the worksheet to begin working on the power regulator schematic, now is a good time to learn about **BLOCK Move**.

Move the clock oscillator circuit to another place on the worksheet

Before beginning a **BLOCK Move**, zoom out so you can see all of the objects you are moving, as well as the beginning and ending points of the move. Follow these steps to zoom out and then move the clock oscillator circuit:

1. Select **ZOOM Select 5** to change the scale from one to five. The entire worksheet displays.
2. Select **BLOCK** and then select **Move**.
3. Place the pointer above and to the left of the clock oscillator circuit, and select **Begin**.
4. Move the pointer below and to the right of the circuit. As you move the pointer, a rectangle expands and contracts.
5. When the rectangle encloses the entire circuit, select **End**. The rectangle locks onto the circuit.
6. Move the outline of the circuit until it is centered in the B-2 region of the worksheet.
7. Select **Place** to move the clock oscillator circuit. The circuit moves to the new location.
8. Select **ZOOM Select 1** to return to a one-to-one scale.
9. Place the pointer in the A-2 area of the worksheet and select **ZOOM Center**. **Draft** moves the view of the worksheet so that the pointer displays in the center of the screen. You are now ready to capture the schematic for the power regulator circuit.

Building the power regulator circuit

To build the power regulator circuit, you need the following parts:

- An LM7805 IC regulator
- Two capacitors
- A nine-volt battery
- Power (V_{CC}) and ground (GND) symbols

As in chapter 3, the digital clock parts library (TUTOR.LIB) contains the parts you need to construct the power regulator circuit.

Get library parts and place them on the worksheet

Follow these steps to get the necessary parts for the power regulator circuit:

1. Select **GET** from the main menu. The "Get?" prompt displays.
2. Press <Enter> then select **.\TUTOR.LIB**.
3. The parts menu displays. Select an LM7805 (an IC regulator) and place it at location (15.00, 12.50).
4. By now you should be experienced at placing parts. Get the capacitor and place one on each side of the regulator, as shown in figure 4-1.
5. Now get the ground symbol (**GND**) and place it in the location shown in figure 4-1.

Deleting parts from the worksheet

If you place a part and then decide you don't need it after all, **Draft's DELETE** command lets you remove any object placed on the worksheet.

If you delete an object by mistake, you can undo your action.

Delete an object

To familiarize yourself with the delete process, delete the capacitor on the right side of the IC regulator.

1. Select **DELETE** from the main menu. The **DELETE** menu displays, as shown below.
2. Select **Object**. The **DELETE Object** command line displays.
3. Put the pointer on the rightmost capacitor.

```
Delete
Object
Block
Undo
```

4. Select **Delete. Draft** deletes the capacitor from the worksheet.

   Because of the way **Draft** deletes things, some dots may remain on the screen where the deleted object was.They are not really on the worksheet.

5. Press <Esc>. **Draft** redraws the screen. Any extra dots disappear.

Recover a deleted object

Follow these steps to recover the deleted capacitor:

1. Select **DELETE** again.
2. Select **Undo**. The capacitor reappears.

Rotating parts before they are placed

Now you are going to try something a little different. A battery symbol exhibits polarity, so even though you know that the negative terminal goes to ground, the symbol may end up backwards on the schematic if you are not careful. You may have to rotate the part to get the polarity correct. Follow these steps to familiarize yourself with this process:

1. Get the battery part (**BATTERY**) from the TUTOR.LIB library. Once the part is selected, the **Get Part** command line displays:

   ```
   Place Rotate Normal Up Over Down Mirror Find
   ```

2. Select **Rotate** twice and see the effect this has on the battery symbol.

   Before placing the part, experiment with the other **Place** commands to see their effect on the part orientation.

3. If you look closely at the part, you'll notice that one end of the part has a long heavy line as part of the border. The other end has a shorter heavy line as part of the border. The long heavy line indicates the positive terminal of the battery. As shown in figure 4-1, you want the positive terminal up, so rotate the symbol to this orientation (**Down**), and place it on the worksheet.

You have now placed all the parts and symbols, except for the V_{CC} power symbol associated with the power regulator circuit on the worksheet. Next you draw the wires for the power regulator circuit.

Drawing multisegment wires

Compare your worksheet with figure 4-1. Notice that several wires are missing from your worksheet.

In this section, you learn how to draw multisegment wires in one operation. A multisegment wire is a single wire that changes direction several times.

1. Select **PLACE Wire**. The **PLACE Wire** command line displays.
2. Move the pointer to the negative terminal of the battery, and select **Begin**.
3. Move the pointer down approximately three grid spaces.
4. Move the pointer to the right until it is directly under the first capacitor.
5. Select **Begin** and move the pointer to the end of the capacitor pin.
6. Select **End** or **New**. When you draw multisegment wires, remember to start and turn corners with **Begin** and cut the wire with **End** or **New**.
7. Now, connect wire segments between the remaining parts as shown in figure 4-1. Be sure to **Begin** and **End** each wire segment at the end of a part pin, not within the body of the part.
8. Using the **PLACE Junction** command, place junctions in the circuit at the five locations shown in figure 4-1.

△ *NOTE: If you cut a wire with* ***New****, the* ***PLACE Wire*** *command line remains displayed. You don't need to select* ***PLACE Wire Begin*** *to start a new wire. You only need to select* ***Begin****.*

More macros

You could continue drawing wires using keyboard or menu commands, but it's a repetitious process. Every time you begin drawing a wire, you must enter three commands in sequence, **PLACE**, **Wire**, and **Begin**.

You can do this by pressing the first letters of each command, <P><W><B>. You can also use **Draft**'s macro feature to make it even easier by combining these three keystrokes into one keystroke. You were introduced to macros when you developed the initial macro that sets up the work conditions each time **Draft** runs.

The following is a simple example of how to capture a macro. You can extend the principle to create complex macros, automating long command sequences.

Capture a macro to begin a wire

1. Select **MACRO**. The **MACRO** menu displays.
2. Select **Capture**. The "Capture macro?" prompt displays.
3. Press <F1> to assign a keystroke to this macro. "F1" displays at the "Capture macro?" prompt.
4. Press <Enter>. The message "<macro>" displays to remind you that you are capturing a macro and that any commands you select are added to the list of commands being stored in the macro.
5. Type the commands required to begin a wire by pressing <P> <W> <B>.
6. Press the key combination <Ctrl><End> to end the macro definition. The message "<<<MACRO END>>>" displays.

The macro you captured is now stored in the computer's memory and can be run by simply pressing the key you specified—in this case, <F1>. If you turn your computer off, however, the macro will be lost. You must save the macro to a file.

Save the macro

1. Select **MACRO Write**. The "Write all macros to?" prompt displays.
2. Enter the following filename for this macro:

```
Write all macros to? tutor.mac
```

3. **Draft** displays "Overwrite file?" and a short menu. Since the macro buffer contains all the macros in TUTOR.MAC (because they were read into the macro buffer when you ran **Draft**) *and* the macro you just created, select **Yes**. This will save *all* the macros in the macro buffer to TUTOR.MAC, thus overwriting the existing file.

You just saved this macro—and the macros that were already in TUTOR.MAC—to the macro file that automatically loads each time you start **Draft**. You can add more macros to this file as you capture them.

Placing the power symbol

Follow these steps to place the power symbol in the power regulator circuit:

1. Select **PLACE Power**. An image of the power symbol displays, with the value V_{CC} above it. The **PLACE Power** command line displays:

   ```
   Place Orientation Value Type Find Jump Zoom
   ```

   In this example, the power symbol is connected at the top of the wire. However, there may also be cases in which you need to turn the power symbol around.

2. Select **Orientation** to change the power symbol's orientation. The **Orientation of Power** menu shown below displays. The image of the power symbol disappears until you make a selection from this menu.

3. Practice changing the orientation of the power symbol. When you finish, select **Top** orientation.

   ```
   Orientation of Power
   Top
   Bottom
   Left
   Right
   ```

   See the *Schematic Design Tools 386+ Reference Guide* for detailed information about the display options available for the power symbol.

4. Now move the image of the power symbol until it rests on the end of the wire, as shown in figure 4-1, and select **Place**.

5. Press <Esc> to dismiss the **PLACE Power** command line.

Dragging wires

You may often want to move parts without having to replace the wires connected to the parts. Use **BLOCK Drag** to do this. Follow these steps to familiarize yourself with this process:

1. Select **BLOCK Drag**. Draft displays:

```
Begin  Find  Jump  Zoom
```

2. Move the pointer above and to the left of the power regulator circuit and select **Begin**.
3. Move the pointer so the rectangle encloses all of the power regulator circuit, except the ground symbol and the bottom wire of the circuit.
4. Select **End**. The circuit changes color.
5. Move the selected circuitry up approximately two grid spaces.
6. Select **Place**. Notice that the lower wires grow and remain connected to the ground wire.

Editing part fields

For the parts in the power regulator circuit, you need only specify the correct part values for the capacitors and battery. After all of the schematic is captured, you use the **Annotate Schematic** tool to update the other fields.

1. Place the pointer on the left capacitor and select **EDIT Edit Part Value Name.** The "Value?" prompt displays. Change the part value to 470uF.
2. Place the pointer on the right capacitor and change the part value to 22uF.
3. Place the pointer on the battery and change the part value to 9V.

Placing comment text

A title isn't necessary for a circuit, but it is helpful when someone new needs to understand what a portion of circuitry does.

1. Select **PLACE Text** command and enter `Power Regulator Circuit`.
2. Select **Larger** to use the type size that is one step larger than the part labels.
3. Center the text immediately below the schematic diagram and select **Place.**

Changing viewpoints

You have now captured two separate schematics on the same worksheet. At times, you may want to quickly change your viewpoint from one area of the worksheet to another. You use the **JUMP** command to do this.

Jump to new coordinates

Follow these steps to move around the worksheet:

1. Select **JUMP**. The **JUMP** menu displays, as shown below. You can move around the worksheet three ways:

   ```
   Jump
   A tag
   B tag
   C tag
   D tag
   E tag
   F tag
   G tag
   H tag
   Reference
   X location
   Y location
   ```

   - Using **X location** and **Y location**, specify the number of grid steps to add or subtract from the current pointer coordinates.
   - Using **Reference**, specify a new pointer location using grid reference regions, such as "A3."
   - Using the tags, move to a pointer location you defined earlier using the **TAG** command.

2. Select **X location**. The prompt "Jump X" displays. Note the current pointer coordinates.

3. At the prompt, enter `+5`. The pointer moves five grid spaces to the right (in the positive direction) and the X reference coordinate reflects a change of 0.50 inches (since each grid space is 0.10 inches).

   - To move left, enter a negative X value.
   - To move to an exact X reference, enter a value without a positive or negative sign. For example, to move to X reference .5, enter `5`; or to move to X reference 5.0, enter `50`.
   - To move up and down, use the **Y location** command.

4. Experiment for a moment with these commands and positive, negative, and unsigned **JUMP** values.

Tag and jump to specific locations

The **TAG** and **JUMP Tag** commands are useful when you need to return again and again to a particular location on the worksheet. The **TAG** command assigns a tag to a location on the worksheet. The **JUMP Tag** command moves the pointer to a tagged location. Follow these steps to practice assigning tags and jumping to them:

1. Place the pointer on the power regulator circuit.
2. Select **TAG** from the main menu. The **Tag set** menu displays, listing eight tag names you can use.
3. Select **A tag**.
4. Move to the clock oscillator circuit. Put the pointer in the middle of the center inverter, and repeat step 2.
5. Select **B tag**.
6. Select **JUMP**, and then select **A tag**. The pointer jumps to the middle of the power regulator circuit, where you assigned the A tag.
7. Now jump to the B tag.

Making a quick print

The last thing to do before ending this chapter is to print out a copy of the worksheet. While **Schematic Design Tools 386+** includes the **Print Schematic** and **Plot Schematic** tools for making copies of entire designs, **Draft** also has a quick way to get a high-quality print: the **HARDCOPY** command.

To do this, your computer must be connected to a printer. **HARDCOPY** does not work for plotters. The correct printer driver must be installed along with your other **Schematic Design Tools 386+** software.

Update the file

Follow these steps to save your work before you print the schematic:

1. Select **QUIT** and **Update file. Draft** updates the file TUTOR.SCH to reflect the current state of the worksheet.
2. Press <Esc> to dismiss the **QUIT** menu.

Make a hardcopy of the worksheet

1. Make sure the printer is connected to your computer, powered on, and on line.
2. Zoom out until you can see both circuits.
3. Select **HARDCOPY** from the main menu. The **HARDCOPY** menu displays.
4. Select **Width of Paper**. Choose the correct paper width for your printer. Select **Narrow** for paper 8.5 inches wide; select **Wide** for paper 13 inches wide.

   After you specify width, **Draft** redisplays the **HARDCOPY** menu.
5. Select **Scale**. Select **C** to reflect the size of your schematic. The **HARDCOPY** menu displays again.

6. Select **Block Print**. The **Block** menu displays.
7. Move the pointer above and to the left of the clock oscillator circuit, and select **Begin**.
8. Move the pointer to define a block that encloses both the clock oscillator and the power regulator circuits, and select **End**. The worksheet is sent to the printer.

For more information on sending designs to printers and plotters, see the **HARDCOPY** command and the **Print Schematic** tool and the **Plot Schematic** tools in the *Schematic Design Tools 386+ Reference Guide.*

Ending a Draft work session

After you save your design and make a hardcopy, you are finished with chapter 4. You need to exit **Draft** to perform steps in the next chapter.

Since you already saved your work, just select **QUIT** and then **Abandon Edits. Draft** exits and the **Schematic Design Tools** screen displays.

Summary

You have completed the schematic diagram for the power regulator circuit of the digital clock. In the next chapter, you use **Edit Library** to create a custom part to use in the display area of the digital clock schematic.

Creating a custom part

Although **Schematic Design Tools 386+** provides extensive libraries containing over 20,000 parts, you may occasionally need a part or symbol not in any library. You use the **Edit Library** tool to modify an existing part or create an entirely new part.

In this chapter, you learn how to:

- Run **Edit Library**
- Reconfigure **Edit Library**'s work conditions
- Draw a part body
- Draw special shapes
- Use shading and fills
- Add pins to the part body
- Add pin names
- Save the new part in a library

Running Edit Library

Edit Library performs a variety of tasks for creating and modifying custom parts and libraries. In this chapter, you create a completely new part to add to an existing library file. For detailed discussions of **Edit Library** commands, see the *Schematic Design Tools 386+ Reference Guide.*

Configure Edit Library

Before running **Edit Library,** you must configure it to open the library file called TUTOR.LIB.

1. Select **Edit Library** from the **Schematic Design Tools** screen.
2. Select **Local Configuration** from the menu that displays and then select **Configure LIBEDIT. Edit Library's** configuration screen displays.
3. Select .\TUTOR.LIB in the **Files** list box. The path and filename .\TUTOR.LIB displays in the **Source** entry box:

```
Source  .\TUTOR.LIB
```

4. Select **OK** to save the configuration.

Run Edit Library

From the **Schematic Design Tools** screen, select **Edit Library** and **Execute.**

The **Edit Library** screen displays. Initially it is blank, except for pointer coordinates displayed at the upper right of the screen.

Setting up the work conditions

As with **Draft**, you set up certain work conditions in **Edit Library**. You adjust two features: one governs visibility of the outline of the part body. The other governs visibility of the grid dots in the work area.

Make part body border and grid dots visible

1. Press <Enter> to display **Edit Library's** main menu.
2. Select **SET** from the main menu. The menu shown below displays.
3. Select **Show Body Outline.** "Show Body Outline?" displays.
4. Select **Yes.**
5. Select **SET Visible Grid Dots Yes.** Grid dots display in the work area.

```
Set
Auto Pan             YES
Backup File          YES
Error Bell           YES
Left Button          NO
Macro Prompts        YES
Power Pins Visible   NO
Show Body Outline    NO
Visible Grid Dots    NO
Repeat Parameters
Cursor Style
```

Beginning a new part

To modify or create a part, you use the **GET PART** command. When you create a new part, choosing **GET PART** initiates a sequence of queries about the type of part you want to create. You will create a seven-segment LED named TIL309.

Open a part editing pad

Follow these steps to begin a new part:

1. Press <Enter> to display the main menu.
2. Select **GET PART**.

   The prompt "Get?" displays.
3. Enter `TIL309`, the name of the part you plan to create.

   The prompt "TIL309 – New Part?" displays.
4. Select **Yes**.

   The prompt "Sheet Path?" displays. This is relevant when you create a hierarchical design and want the part to refer to another schematic worksheet.
5. Press <Enter> to signify that the TIL309 part does not refer to a schematic.

   The **Kind of Part?** menu displays. You use **Block** for simple rectangular parts with no graphics, **Graphic** for parts of any shape with graphics, and **IEEE** for IEEE/ANSI drawing-standard parts.
6. Select **Graphic**; the LED display includes graphics.

   Edit Library asks if this part is a grid array. A grid array part is a single-element graphic part with alphanumeric pin numbers.
7. Select **No** because you do not need alphanumeric pin numbers for this part.

   Edit Library displays a menu with the numbers 0 through 16 and prompts you for the number of elements in the part.
8. Select **1** because the seven-segment LED display is a single-element part.

   Edit Library asks whether the part has a convert (that is, whether you will also create a DeMorgan equivalent of the part you are creating).

9. Select **No.**

   The part editing pad displays, bordered by a solid line. Within the pad, a dotted border displays with the name you assigned the new part, **TIL309**. The pointer displays at the bottom right corner of the dotted border. The command line displays **Place** and **Escape.**

   The dotted border defines the size and shape of the region within which you create the part body. Pins you attach to the part are created outside this region, with their connection points on the part body border.

   You can adjust the size and shape of the dotted border by moving the pointer. Try it.

10. Move the pointer to location (+12.0, +12.0).

    This changes the part body border to a square shape. Figure 5-1 shows the part editing pad when the pointer is at location (+12.0, +12.0).

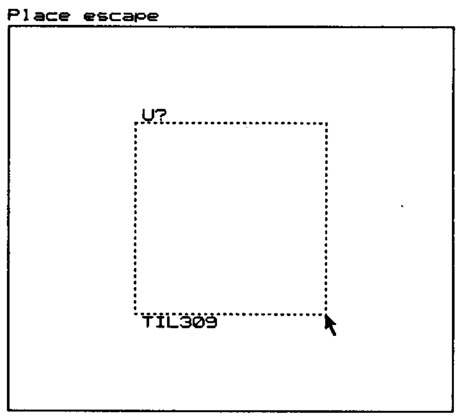


Figure 5-1. Part editing pad.

△ *NOTE: Depending on your computer's monitor, the border may not look square due to the proportions of the screen display.*

11. Select **Place** to set the size of the editing pad.

    The **BODY <Graphic>** menu displays.

Drawing the body outline

1. Select **Line**. The **BODY Line** command line displays.
2. Move the pointer to the upper left corner of the body, at (.0, .0), and select **Begin**.
3. Move the pointer to the next corner, at (+12.0, +0), and select **Begin** again.
4. Move the pointer to the next corner, at (+12.0, +12.0), and select **Begin** again.
5. Move the pointer to the next corner, at (+.0, +12.0), and select **Begin** one last time.
6. Move the pointer to the first corner, at (+.0, +.0), and select **End**. The **BODY <Graphic>** menu displays.
7. Press <Esc> to dismiss the **BODY <Graphic>** menu.

Changing the reference designator

Edit Library automatically puts a placeholder reference designator at the upper left of the part body border. The default class letter is the letter U and the default number is a question mark. The "?" serves as a placeholder for the values to be supplied when you use the part in a schematic and run **Annotate Schematic**. Because U conventionally designates IC parts, you need to change the class letter to D.

1. Select **REFERENCE** from the main menu. The prompt "Initial Reference Designator? U" displays, indicating that U is the current value.
2. Backspace over the U and enter **D**.

Creating a part body

Now you are ready to create the part itself, in this case, a seven-segment LED display. The first step is to create the part body. It consists of seven rectangular objects arranged in the shape of a numeric display, and a circle for the decimal point, as shown in figure 5-2.

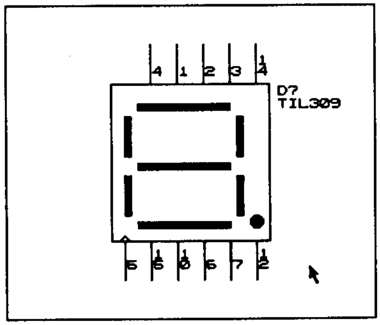


Figure 5-2. The part body you will create.

Zoom in to gain finer pointer control

Like **Draft, Edit Library** can display the part you are working on at several levels of detail. At the lowest level, level 1, the pointer snaps to grid points.

At either of the two higher magnification levels, you can move the pointer to any of 10 intermediate locations between the grid points. You need this fine control to draw the thin LED segments.

1. Select **ZOOM In**. The image doubles in size.
2. Move the pointer between the grid points. Notice that the pointer no longer snaps to grid points.

Draw a rectangle to represent an LED

Follow these steps to draw segment *a*, the top LED segment:

1. Select **BODY** from the main menu. The **BODY <Graphic>** menu displays, as shown below.
2. Select **Line**. The **BODY Line** command line displays.
3. Place the pointer at location (+2.0, +1.5).
4. Select **Begin**. **Edit Library** is now in line-drawing mode.
5. Move the pointer to location (+9.0, +1.5). A line stretches behind the pointer to show the line segment you are creating.

```
Body <Graphic>
Line
Circle
Arc
Text
IEEE Symbol
Fill
Delete
Erase Body
Size of Body
Kind of Part
```

6. Select **Begin**. The line you drew changes color, showing that it is completed.
7. Move the pointer to location (+9.0, +2.0). This forms the right side of the rectangular shape. A line stretches from the first line to the pointer.
8. Select **Begin** again to complete this segment and begin another.
9. Move the pointer to location (+2.0, +2.0). This forms the bottom segment of the rectangle.
10. Select **Begin**.
11. Move the pointer to location (+2.0, +1.5), the starting point, to complete the rectangle.
12. Select **End** or **New** to end the last segment and complete the rectangle. Press <Esc> to dismiss the **BODY <Graphic>** menu.

△ *NOTE: For drawing intricate library parts, you may wish to use the arrow keys rather than the mouse.*

Draw six more segments

You can repeat this process for each of the remaining six rectangles, which represent the other LED segments. Each row of table 5-1 shows the coordinates for one rectangle.

To save some time, capture the commands to draw one of the horizontal rectangles as a macro, and run the macro for each of the remaining horizontal segments. Then capture the commands for one of the vertical rectangles, and run that macro for each of the remaining vertical segments. Be sure to use the same corner as the starting point each time you run the macro.

	Top left	*Top right*	*Bottom right*	*Bottom left*
Segment g	(2.0, 6.0)	(9.0, 6.0)	(9.0, 6.5)	(2.0, 6.5)
Segment d	(2.0, 10.5)	(9.0, 10.5)	(9.0, 11.0)	(2.0, 11.0)
Segment f	(1.0, 2.5)	(1.5, 2.5)	(1.5, 5.5)	(1.0, 5.5)
Segment b	(9.5, 2.5)	(10.0, 2.5)	(10.0, 5.5)	(9.5, 5.5)
Segment c	(9.5, 7.0)	(10.0, 7.0)	(10.0, 10.0)	(9.5, 10.0)
Segment e	(1.0, 7.0)	(1.5, 7.0)	(1.5, 10.0)	(1.0, 10.0)

Table 5-1. Coordinates for rectangular LED segments. All coordinates are positive (+) values.

Add the decimal point

In addition to the seven rectangular LED segments, the display unit also has a circular LED at the lower right to represent a decimal point.

1. Select **BODY Circle** to draw the circle.
2. Place the pointer at the location where you want the center of the circle, in this case, location (+11.0, +10.5).
3. Select **Center.** More commands display, one of which is **Edge.** Edge means the edge of the circle being defined. When you move the pointer, a circle expands and contracts.
4. Move the pointer to any location five pointer steps from the center point. For example, put the pointer at location (+11.5, +10.5).
5. Select **Edge. Edit Library** places the circle.
6. Press <Esc> to return to the **BODY <Graphic>** menu.

Shading closed shapes

When you create a part, you may want to shade certain objects to make them stand out. To do this you can use the **BODY Fill** command.

1. Select **Fill** from the **BODY <Graphic>** menu. The **Fill** command line displays.
2. Put the pointer within one of the LED shapes.
3. Select **Fill. Edit Library** fills in the shape.
4. Repeat steps 2 and 3 for all the LED shapes.
5. Press <Esc> twice to dismiss the **Fill** command line and the **BODY <Graphic>** menu.

After drawing the LEDs, you are ready to add pins so the part can be electrically connected when you place it in a schematic. Because this is a representation of an existing part, you want to add the pins corresponding to the standard version of the part.

Adding pins to a part

Edit Library's **PIN** command is used to add pins. Pins must terminate on the border of the part body. The dotted line around the part is the part's border. If the edge of a part body coincides with this border, pins can terminate directly on the part body. But if the part body is inside this border, you must make a connection between the part body and the border using the **BODY Line** command.

Add a clock pin

1. Select **PIN** from the main menu. The **PIN** command line displays.
2. Move the pointer around. You'll find it is restricted to the part body border.
3. Put the pointer at a location on the border where you want to place the first pin. For this example, put it at coordinates (+1.0, +12.0).
4. Select **Add**. "Pin Name?" displays. The pin name is an identifier that **Draft** uses to identify particular pins. The pin name does not display on the graphic representation of a part.
5. Enter the name `STROBE`. The **Edit Library** tool assigns the name. The prompt "Pin Number?" displays.
6. Enter `5`. The **Pin Type** menu displays. The STROBE pin conducts a clock signal to the internal logic of the part. It should be defined as an input pin type.
7. Select **Input**. The **Pin Shape** menu displays.
8. Select **Clock**. **Edit Library** places the pin and displays the pin number you entered.

Add a reset pin

1. Place the pointer at the coordinates (+11.00, +12.00).
2. Select **Add**. The prompt "Pin Name?" displays.
3. Enter the name **DPIN**. The **Edit Library** tool assigns the name. The prompt "Pin Number?" displays.
4. Enter **12**. The **Pin Type** menu displays. The DPIN pin conducts a reset signal to the internal logic of the part. It should be defined as an input type pin.
5. Select **Input**. The **Pin Shape** menu displays.
6. Select **Line. Edit Library** places the pin and displays the pin number you entered.

Add the remaining pins

1. Put the pointer at a location where you want to place a pin. For this example, put it at coordinates (+3.00, +12.00).
2. Select **Add**. The prompt "Pin Name?" displays.
3. Enter the name **QAIN**. The **Edit Library** tool assigns the name. The prompt "Pin Number?" displays.
4. Enter **15**. The **Pin Type** menu displays. The QAIN pin conducts a signal to an LED segment. It should be defined as a passive type pin.
5. Select **Passive**. The **Pin Shape** menu displays.
6. Select **Line**.

7. Repeat these steps for the pins connected to the other LED segments. Table 5-2 lists the coordinates, names, numbers, types, and shapes to use for all the pins on this part. You already defined the first three pins, so start with the fourth pin.

Coordinates	*Name*	*No.*	*Type*	*Shape*
(1.0, 12.0)	STROBE	5	Input	Clock
(11.0, 12.0)	DPIN	12	Input	Line
(3.0, 12.0)	QAIN	15	Passive	Line
(5.0, 12.0)	QBIN	10	Passive	Line
(7.0, 12.0)	QCIN	6	Passive	Line
(9.0, 12.0)	QDIN	7	Passive	Line
(3.0, 0.0)	QAOUT	4	Passive	Line
(5.0, 0.0)	QBOUT	1	Passive	Line
(7.0, 0.0)	QCOUT	2	Passive	Line
(9.0, 0.0)	QDOUT	3	Passive	Line
(11.0, 0.0)	DPOUT	14	Passive	Line

Table 5-2. Pins for the TIL309 library part. All coordinates are positive (+) values.

8. Press <Esc>.

When you are finished, you should have 11 pins on the LED. The next step is to add the part to the library.

Saving a new part

Saving a new part involves two operations:

- Copying the part displayed on the screen to the part library currently loaded in the computer's internal memory. This is done using **LIBRARY Update Current.**
- Writing the modified library file in the computer's internal memory to disk. This is done using **QUIT Update file** or **QUIT Write to file.**

Save the part to the current library

1. Select **LIBRARY.**
2. Select **Update Current.** The part currently displayed is written to the library now loaded in memory.

Write the current library to a disk file

1. Select **QUIT Update file. Edit Library** updates the library with the edits you performed during this session and then redisplays the **QUIT** menu.
2. To confirm that the part TIL309 has been stored in a library named TUTOR.LIB, select **Initialize.** The prompt "Read Library?" displays.
3. Enter `.\TUTOR.LIB`.
4. Select **LIBRARY List Directory Screen.** TIL309 should be in the list of parts in TUTOR.LIB.
5. To dismiss the directory, press any key.

Get the new part

1. Select **GET PART.** When the prompt "Get?" displays, press <Enter>. The **Get** menu displays containing the name of the part you created, TIL309.
2. Select the TIL309 part. It displays in the edit pad.
3. Select **QUIT Abandon Edits** to leave **Edit Library** and return to the **Schematic Design Tools** screen.

Summary

Using the **Edit Library** tool, you created a new part and saved it on disk in an existing library. In *Chapter 2: Introducing Schematic Design Tools,* you configured **Draft** to load the TUTOR.LIB parts library. By adding the TIL309 part to this library, you made the new part available in **Draft** for use while capturing schematics.

Capturing the logic and display circuit schematic

This final schematic diagram for the digital clock circuit contains the logic and display circuit. This circuitry is more complex than the smaller schematics that you captured in the earlier chapters. The tasks you complete in this chapter progress naturally from the processes that were introduced in earlier chapters.

In this chapter you learn how to:

- Draw a repeatable portion of the schematic
- Make and place multiple copies of a schematic block
- Use repeat parameters to place wires and labels

Figure 6-1 shows the portion of the schematic you capture in this chapter.

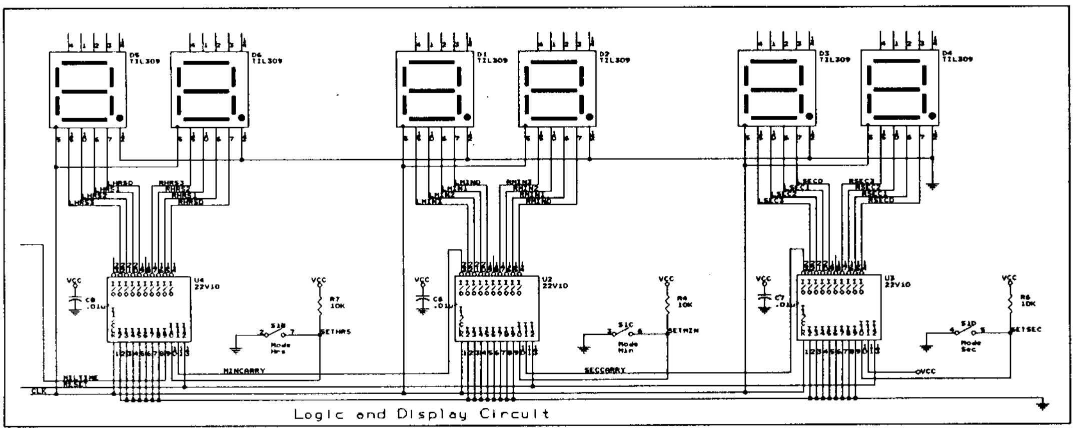


Figure 6-1. The logic and display circuitry.

Choosing parts

To build the rest of the digital clock schematic, you need the following parts:

- 22V10s
- TIL309 LED displays
- Resistors
- Capacitors
- Two switch types (SPST and pushbutton)
- Power (V_{CC}) and ground (GND) symbols

About TIL309 LED display chips

TIL309 display chips were selected in order to keep the chip count for the design down. These displays are capable of accepting binary-coded decimal input. Using TIL309s eliminates the need for decoder circuits. Six TIL309s are required: two each for seconds, minutes, and hours.

About 22V10 PALs

The schematic requires enough pins to drive the six TIL309 display chips and to transfer the "carry" signals. Once again, to reduce the total chip count for the design, 22V10s were chosen to drive the TIL309s rather than individual parts. Since the TIL309s are divided into pairs for seconds, minutes, and hours, you use one 22V10 per pair: three 22V10s altogether.

When deciding to use the 22V10s, the following factors were considered: number of inputs and outputs needed, complexity of the logic that the part needs to handle, cost, and availability. The 22V10s were chosen because they have enough inputs and outputs to accommodate fairly complex logic, are readily available from several manufacturers, and are not extremely expensive.

As in the previous chapters, the clock parts library (TUTOR.LIB) contains the parts you need to construct this circuit. In chapter 5, you added the seven-segment display part (TIL309) to the parts library.

Running Draft again

Select **Draft** from the **Schematic Design Tools** screen. Draft displays the last active view of the TUTOR.SCH schematic.

Drawing a portion of the schematic

As you look at the schematic of the logic and display circuitry in figure 6-1, it becomes apparent that three regions are nearly identical—seconds, minutes, and hours. Take advantage of this duplication by creating the schematic for the minutes area (figure 6-2), and copying it to the other areas.

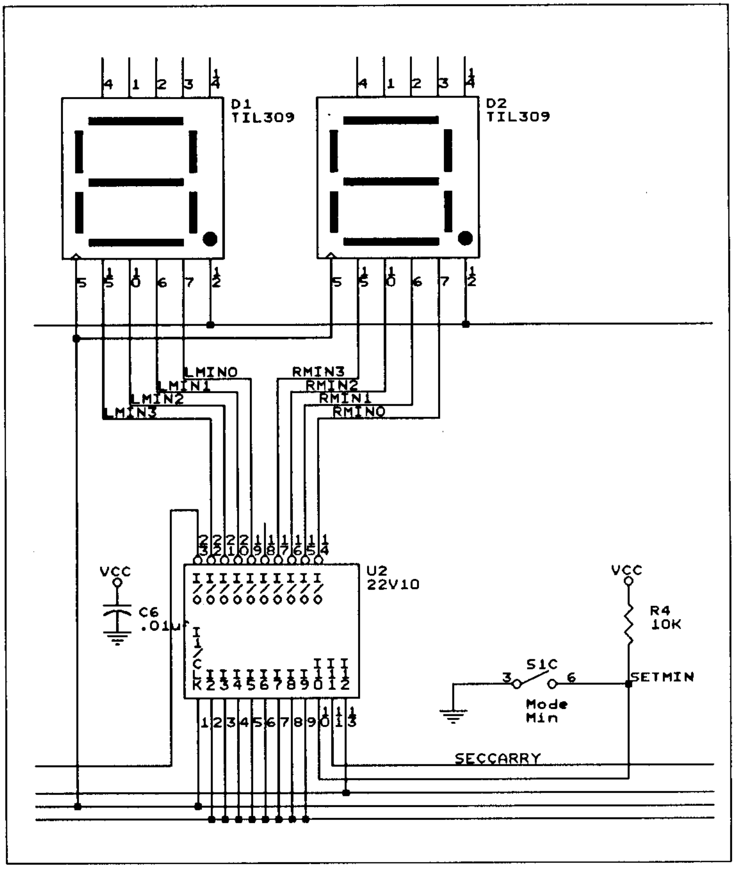


Figure 6-2. The minutes circuit.

Change viewpoint to a clear area

Follow these steps to move to an area of the worksheet with enough room to add the display and logic circuitry:

1. Select **ZOOM Select 2** to change the scale to two-to-one. This scale will work better for the tasks outlined in the next steps.
2. Select **JUMP Reference C 4** to bring the center of the worksheet into view.

The clock oscillator and power regulator circuits you captured earlier are in the lower right area (grid references A-2 and B-2) of the worksheet. The entire upper half of the worksheet is still vacant, so you can use it for this portion of the schematic.

The display and logic circuit shown in figure 6-1 contains so much detail that your immediate task of capturing the minutes area may seem more difficult than it actually is. Figure 6-2 shows only the parts and wires associated with the minutes area of the schematic. By comparing figure 6-2 with figure 6-1, you see the similarities in each of the areas. The steps in the next section describe how to build the minutes circuit.

Place the parts

Follow these steps to place parts for the minutes circuit:

1. Select **GET**. The prompt "Get?" displays.
2. Press <Enter>. A list of the part libraries specified in the **Schematic Design Tools** configuration displays.
3. Select **.\TUTOR.LIB**. A list of the parts in TUTOR.LIB displays.
4. Select **22V10**. The part and a command line display. The part's orientation is not correct for this schematic, so you will have to rotate the part.
5. Select **Rotate** to change the part orientation to match the part orientation shown in figure 6-2.
6. Position the part at coordinates (10.50, 6.00) and select **Place**.
7. Get a **TIL309** and place copies at (9.60, 2.50) and (11.50, 2.50).

8. Get a resistor, **R**, and place it at (13.70, 6.30).
9. Get a capacitor, **CAP**, and place it at (9.90, 6.30).
10. Finally, get a switch, **4SW SPST**, and place it at (13.00, 6.80).

At this point you have placed all parts and only need to place wires, nets, and the power and ground symbols. You will enter part values after all items are placed.

Draw the first wire

1. Select **ZOOM In**. You need a close-up view of the schematic to perform the next steps.
2. Referring to figure 6-2, move the pointer to the bottom of the resistor symbol and select **Place Wire Begin** to start drawing a wire.
3. Draw the wire so that it is three grid spaces below the lower pins of the 22V10 part at (13.80, 7.60).
4. Select **Begin.**
5. Draw the wire to (11.50, 7.60), and select **Begin.**
6. Continue the wire so that it connects to pin 10 on the 22V10 part (11.50, 7.30).
7. Select **End** to end the wire.

Run the macro to draw the other wires

The <F1> macro you captured earlier to start placing a wire should still be active. Follow these steps to use the macro to draw the rest of the wires:

1. Referring to figure 6-2, draw the wires between the 22V10 part and the right-hand seven-segment display, as shown. Instead of pressing the <P>, <W>, and <B> keys, just press <F1>, and then proceed as usual.
2. Continue using the macro to draw the wires between the 22V10 part and the left-hand seven-segment display, as shown in figure 6-2.

The <F1> macro saves some time, but there are other things you can do to save even more time. One time-saver is the **REPEAT** command.

Define REPEAT parameters

REPEAT duplicates the last entered object, label, or text string and places it on the worksheet. Follow these steps to define the **REPEAT** parameters:

1. Select **SET Repeat Parameters.**
2. Select **X Repeat Step**. The prompt "X Repeat Step?" displays. Enter `1`.
3. Repeat step 1 and select **Y Repeat Step**. The prompt "Y Repeat Step?" displays. Enter `0`.

REPEAT is now set to place a new object exactly one grid space to the right of the pointer when you select **REPEAT**.

Change viewpoint to speed wire placement

The wire placements in the next steps work better if you center the display first:

1. Move the pointer to the end of pin 2 at the bottom of the 22V10.
2. Select **ZOOM Center** to change your viewpoint to center pin 2 on the worksheet.

Use REPEAT to speed wire placement

Follow these steps to quickly draw more wires:

1. Draw a wire seven grid spaces long extending down from pin 2 of the 22V10 PAL. Press <F1> to begin the wire, and press <E> to end it.
2. Select **REPEAT** from the main menu and observe the wire that **Draft** places on pin 3 of the 22V10 PAL. If you usually use the mouse to select commands, try pressing <R> when you select the **REPEAT** command.
3. Select **REPEAT** six more times to place the remaining wires of this length as shown in figure 6-2.
4. Place a single horizontal wire along the bottom of these wires from below pin 2 to below the resistor.
5. Select **PLACE Junction**, and then **Place** to put a junction at the leftmost intersection of the wires placed in the prior steps. Press <Esc>.
6. Press <R> seven times to place wire junctions at each of the other wire intersections.

It takes longer to describe how to use the **REPEAT** command than to use it. It's a good idea to plan your schematics to take advantage of **REPEAT**.

Place the remaining parts of the minutes circuit

You have some more wires and junctions and the power and ground symbols to place before you are finished with this portion of the circuit. Because we intend to copy this circuit, it doesn't make sense to edit part labels or comment text yet. Follow these steps to finish placing objects:

1. Select **PLACE Power Place** to put power symbols above the resistor and capacitor symbols, as shown in figure 6-2.
2. Select **GET**, then enter GND.
3. Place ground symbols below the capacitor, and below and to the left of the switch symbol.
4. Place the remaining wires shown in figure 6-2.
5. Place junctions at the remaining locations shown in figure 6-2.
6. Examine your worksheet and carefully compare it with the schematic in figure 6-2. The exact position of objects is not as important as the presence or absence of these objects.
7. Correct any problems you find before going to the next exercise.

Copying a block

So far in this chapter, we have been careful to capture only the portions of the schematic that are repeated in several areas. Because three portions of the schematic are nearly identical, you should be able to finish this portion of the schematic in about a third of the time by copying the circuit you just drew.

1. Select **ZOOM Out** twice or **ZOOM Select 5** to change to the five-to-one scale. This way you can see all of the objects you are working with.
2. Select **BLOCK Copy. Draft** displays this command line:

```
Begin Find Jump Zoom
```

3. Move the pointer above and left of the minutes circuit, and select **Begin.**
4. Move the pointer so the rectangle encloses the minutes circuit.
5. Select **End.** The minutes circuit is highlighted and the **Place** command line displays:

```
Place Find Jump Zoom
```

6. Look at the Y coordinate on the screen. Carefully move the copy to the right of the original, keeping the copy at the same Y coordinate. When the block is positioned correctly, select **Place.**

△ *NOTE: Be sure the copy is horizontally aligned with the original and that there is enough space between the two to allow more wires to be drawn.*

7. After you place the copy of the circuit, the outline displays again so you can continue placing copies.
8. Next, place a copy of the circuit to the left of the original. Again, be sure that the copy is at the same Y coordinate as the original.
9 Press <Esc> to dismiss the highlighted copy of the block and the **Place** command line.

It's been a while since you had a look at the schematic you're duplicating. Figure 6-3 is another copy of the logic and display circuit schematic. Your worksheet should look similar to the worksheet shown in figure 6-3.

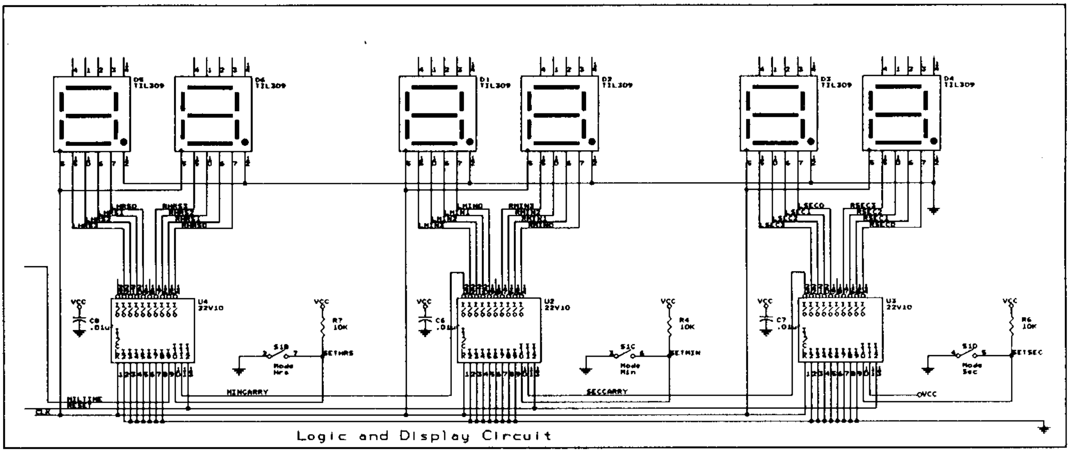


Figure 6-3. The logic and display circuitry.

Finishing the wiring

Figure 6-3 shows how the clock logic will look once you place the wires to connect the seconds, minutes, and hours circuits. The following sections describe how to do this. As you follow the steps in each of these sections, refer to the callouts—①, ②, ③, and so on—in each figure. These callouts correspond to the numbered steps that follow the figure.

Wire the seconds circuit

The following steps correspond to the callouts in figure 6-4. Before you begin, move the pointer to the rightmost 22V10, and select **ZOOM Select 1.**

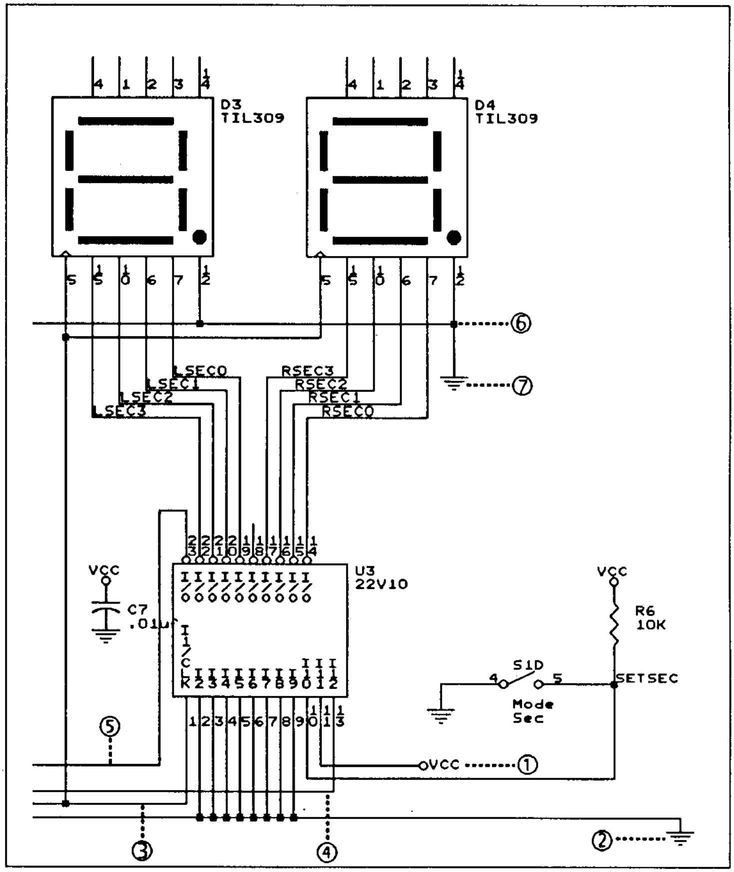


Figure 6-4. Seconds area of the clock logic. The callouts refer to the step numbers in this section.

1. The horizontal wire from the 22V10's pin 11 should be shortened. Referring to the ① in figure 6-4, delete the wire and redraw it so that it is only six or seven grid spaces long.

   Once the wire is the correct length, select **PLACE Power** to get a power symbol. This symbol must be turned before it is placed on the schematic, so select **Orientation Right** and then place it at the end of the wire you just drew.

2. The bottom horizontal wire must have a **GND** symbol added to it.

   Draw a wire that extends two grid spaces down from the right end of this wire. Get a **GND** symbol from the library TUTOR.LIB and place it at the end of this wire.

   Draw a wire that connects the left end of this wire to the bottom horizontal wire of the center, or minutes, circuit.

3. The second-from-bottom horizontal wire needs to be shortened so that it doesn't run as far to the right. Delete and redraw this wire so that it starts at the end of the wire connecting to the 22V10's pin 1 and goes left to connect to the minutes circuit. The junction at the end of the pin 1 wire is no longer needed. Delete it.

4. The third-from-bottom horizontal wire needs to be shortened so that it stops at the wire that connects to the 22V10's pin 13. Delete this wire and redraw it so that it starts at the end of the wire connecting to the 22V10's pin 13 and goes left to connect to the minutes circuit. Since the junction at the end of the pin 13 wire is no longer needed, delete it.

5. Connect the wire that comes from the 22V10's pin 23 to the minutes logic.

6. Delete the horizontal wire that is immediately below the seconds display and redraw it so that it starts at the wire that comes from the rightmost TIL309's pin 12 and goes left to connect to the minutes circuit.

7. Return to pin 12 of the rightmost TIL309 and extend that wire down. Get a **GND** symbol from the TUTOR.LIB library and place it at the end of the extension.

The seconds circuit is now complete and connected to the minutes circuit. Next you complete the minutes circuit.

Wire the minutes circuit

Before working on the minutes circuit, move the pointer to the middle 22V10 and select **ZOOM Center.** Then follow the steps below to complete the wiring for the minutes circuit (figure 6-5).

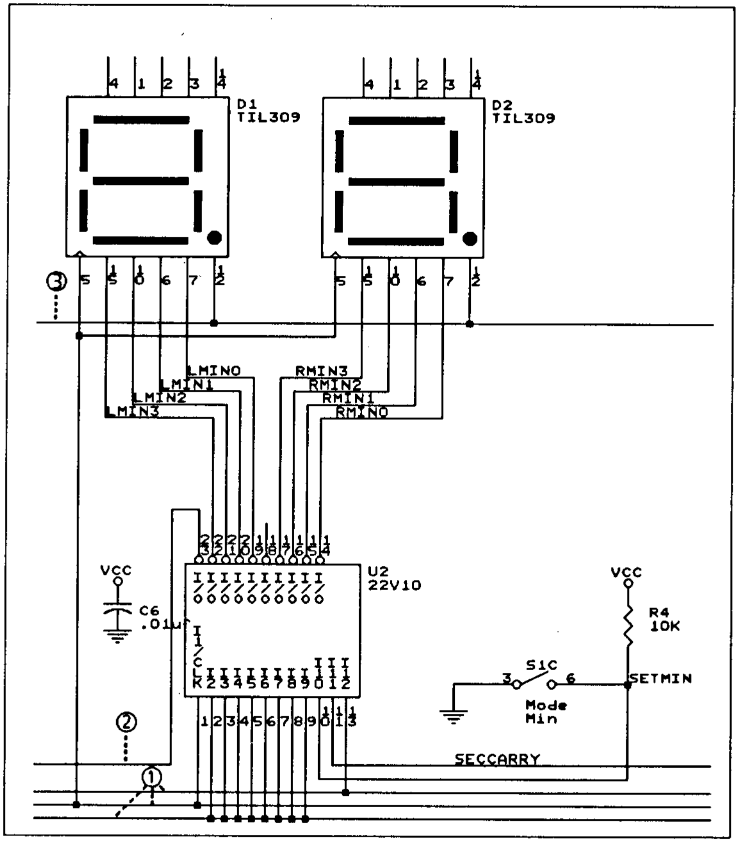


Figure 6-5. Minutes area of the clock logic. The callouts refer to the step numbers in this section.

1. Connect the bottom three horizontal wires to the hours logic.
2. Connect the wire from the 22V10's pin 23 to the hours logic.
3. Connect the horizontal wire that runs just below the minutes display to the hours logic.

The minutes circuit is now complete and connected to the hours circuit. Next you complete the hours circuit.

Wire the hours circuit

Before working on the hours circuit, move the pointer to the leftmost 22V10 and select **ZOOM Center.** Then follow the steps below to complete the wiring for the hours circuit (figure 6-6).

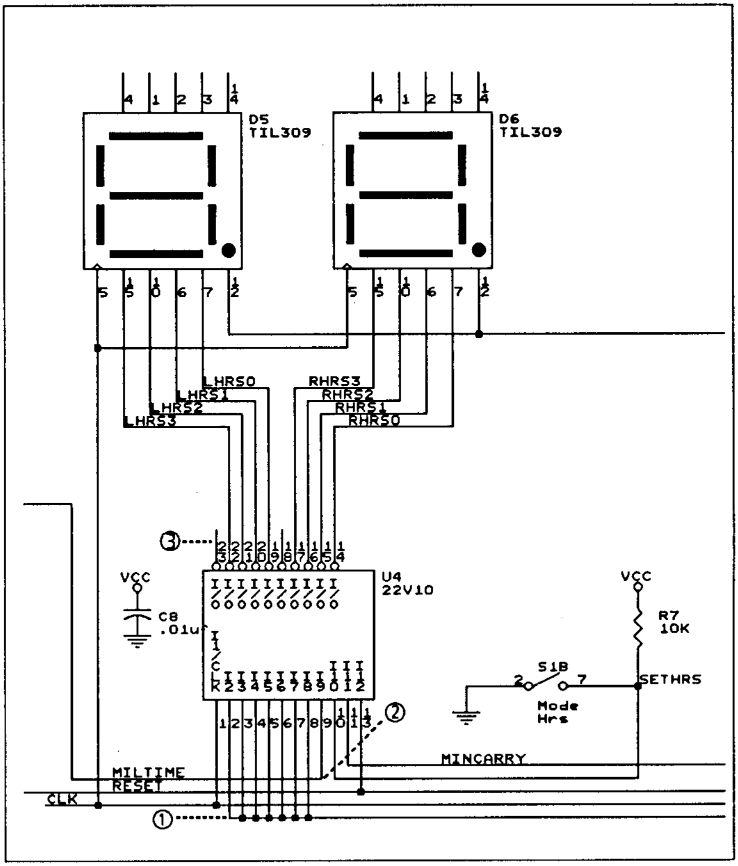


Figure 6-6. Hours area of the clock logic. The callouts refer to the step numbers in this section.

1. The bottom horizontal wire should end at the wire that extends from pin 2 of the 22V10. Delete the junction at the end of the pin 2 wire.

2. The vertical wire from pin 9 of the 22V10 should change so that it doesn't connect to the bottom horizontal wire. Delete this wire and its junction.

   Draw the wire again so it comes down from pin 9, turns left, and extends as far to the left as the other wires.

3. Delete the wire that comes from pin 23 of the 22V10. Be sure to delete all segments of this wire.

4. The horizontal wire that runs just below the hours display should stop at the wire that extends from pin 12 of the leftmost numeric display. Delete this wire and redraw the portion to the right of pin 12. Delete the junction at the end of the pin 12 wire.

View clock logic

You have now connected all of the wires in the logic and display portion of the schematic. Select **ZOOM Select 5** to view the entire schematic. It should look like figure 6-3. Note that the clock oscillator circuit and the power regulator circuit on your schematic do not show in figure 6-3.

Figure 6-7 shows how the schematic will look when you complete the remaining steps in this chapter.

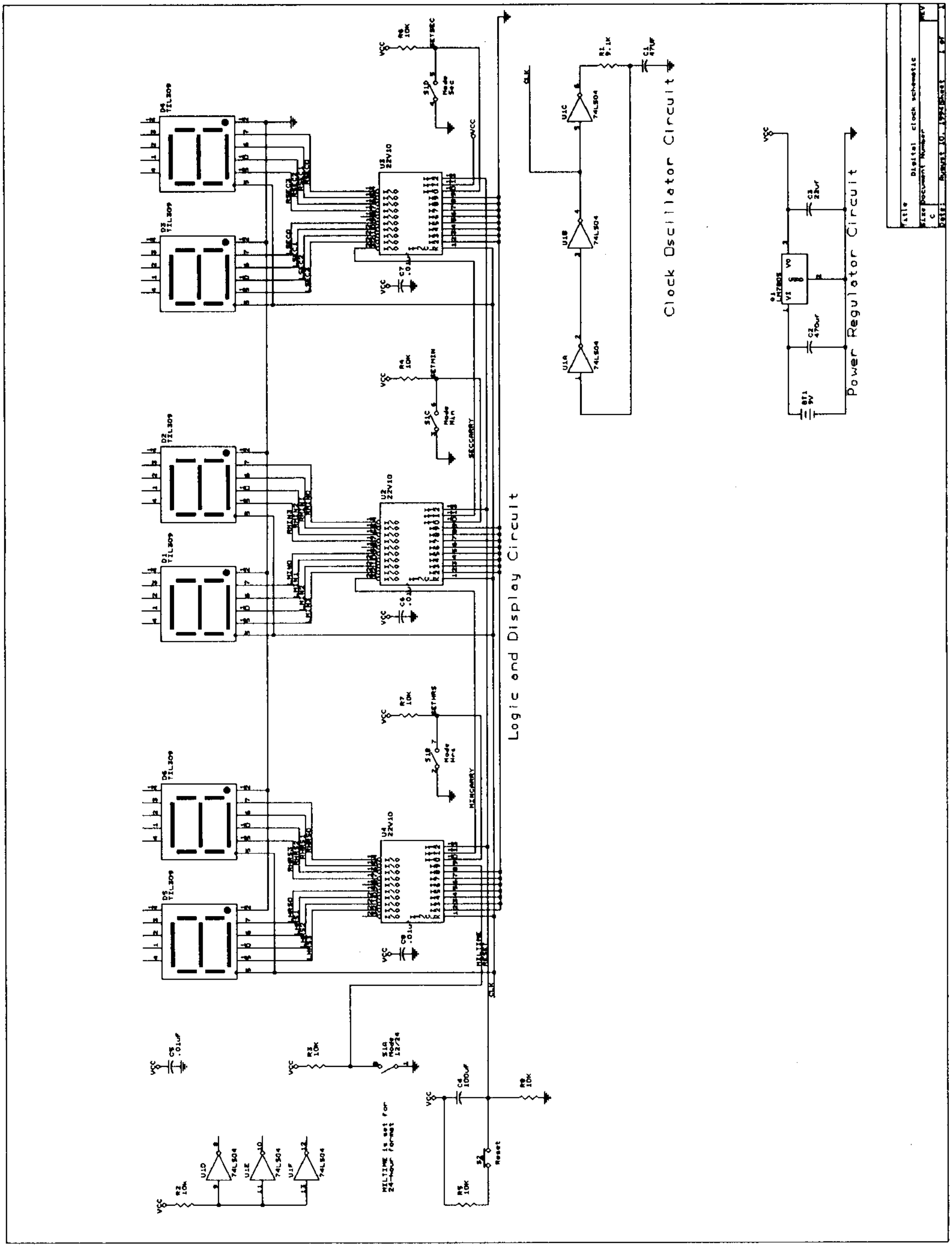


Figure 6-7. Completed TUTOR.SCH schematic.

Finishing the clock schematic

Compare the schematic in figure 6-7 with the schematic you have captured so far. Notice that you need to add a few parts and draw a few more wires to have a functional circuit. You also need to edit the labels and other text.

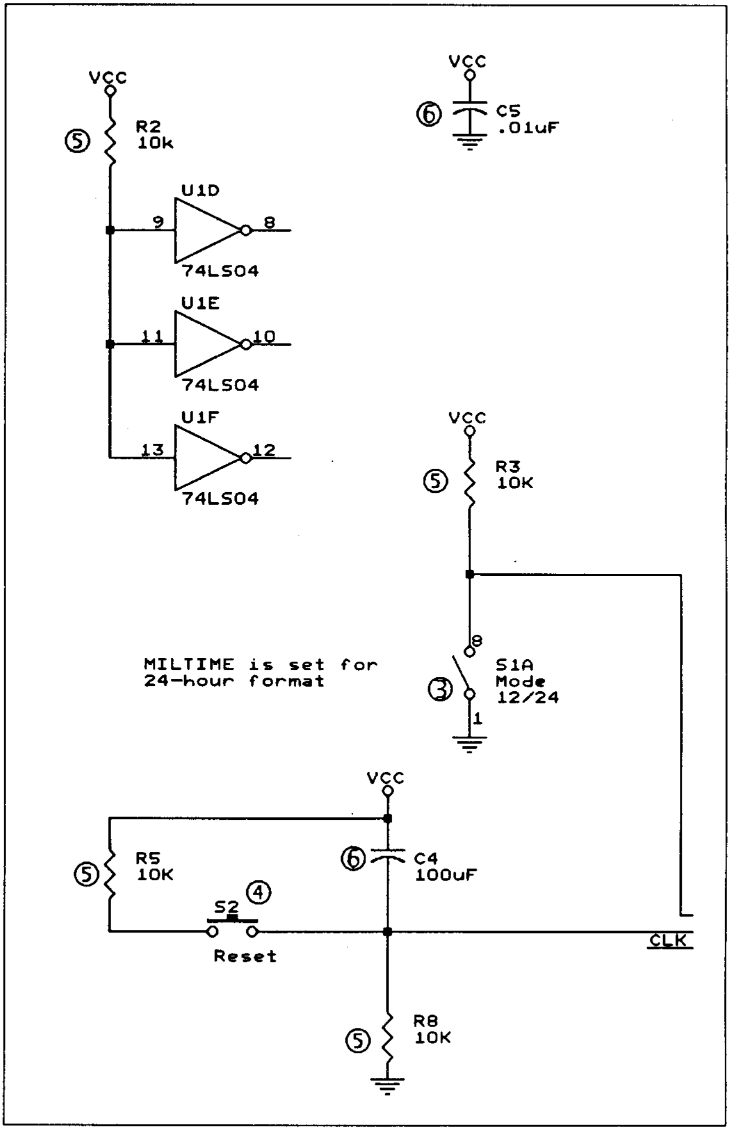


Figure 6-8. Switches, resistors, and other parts to be placed. The callouts refer to the step numbers in this section.

Four resistors, three inverters, two capacitors, two switches, and several power and ground symbols are needed to complete the logic and display circuit schematic. Figure 6-8 shows the arrangement of these parts.

Refer to figure 6-8 as you follow these steps:

1. Select **ZOOM In,** or **Zoom Select 2** to change the scale to two-to-one.

   Change your view of the worksheet so grid reference C-7 is visible.

2. Get the **4SW SPST** switch from TUTOR.LIB.

   The orientation of the 4SW SPST is not correct for this schematic.

3. With the part selected and the outline showing on the screen, select **Rotate** to turn the part so the orientation matches that shown in figure 6-8. Move the part to location (2.70, 6.00) and place it.

4. Get the **SW PUSHBUTTON** part from TUTOR.LIB and place it at location (1.20, 7.60).

5. Get a resistor (**R**) and place four copies at locations (.50, 2.70), (2.70, 4.80), (.50, 7.10), and (2.20, 8.10).

6. Select a capacitor and place it in locations (2.70, 2.60) and (2.20, 7.10).

Place the extra parts

There are also some leftover parts (from multiple-element parts) to be placed on the schematic. Figure 6-9 shows the arrangement of these parts.

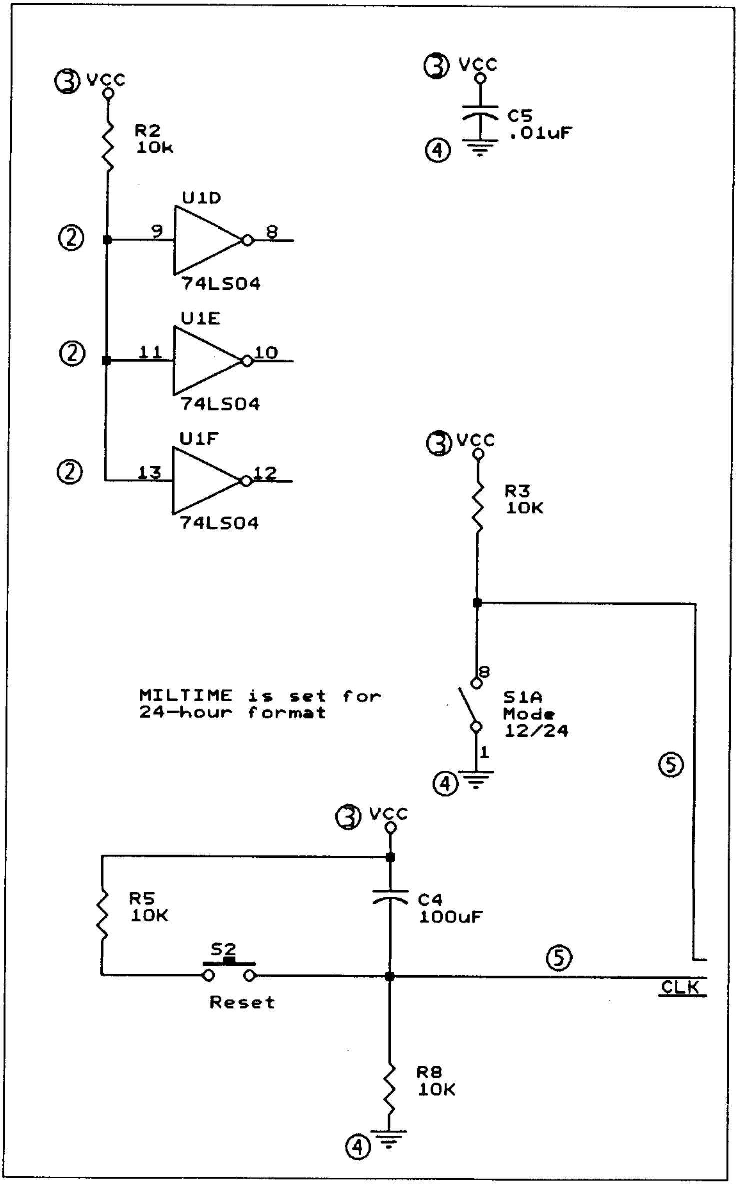


Figure 6-9. Inverters, power symbols, ground symbols, and wires to be placed. The callouts refer to the step numbers in this section.

Refer to figure 6-9 as you follow these steps:

1. Use **ZOOM** to change the scale to one-to-one. Move to reference grid D-8.
2. Get the **74LS04** inverter from TUTOR.LIB and place three copies at locations (1.00, 3.20), (1.00, 3.90), and (1.00, 4.60), as shown in figure 6-9.
3. Place four power symbols at locations (0.60, 2.60), (2.80, 2.50), (2.80, 4.70), and (2.30, 6.80).
4. Get a ground symbol from TUTOR.LIB and place three copies at locations (2.70, 2.80), (2.70, 6.50), and (2.20, 8.50).
5. Place wires to connect the remaining parts, as shown in figure 6-9. Be sure to connect the wires to the logic and display circuit at the two places shown in figure 6-9.
6. Inspect the wire intersections and use the **Place Junction** command to add junctions where required, as shown in figure 6-9.

Edit the part values

1. Put the pointer on the capacitor located at (2.20, 7.20).
2. Select **EDIT Edit**. The **Edit Part** menu displays.
3. Select **Part Value Name**, and change the default value CAP to 100uF.
4. Using the same procedure as in steps 2 and 3, assign the values shown in figure 6-7 to the all parts on the schematic. Table 6-1 gives a list of the values to edit (you have already edited the first item in this table). Notice that some of the parts require that you enter information into the **1st Part Field**.

Part	*Approximate Location*	*Old Part Value*	*New Part Value*	*New 1st Part Field*
Capacitor	(2.20, 7.20)	CAP	100uF	
Capacitor	(2.80, 2.60)	CAP	.01uF	
Resistor	(0.60, 2.80)	R	10k	
Resistor	(0.60, 7.30)	R	10k	
Resistor	(2.30, 8.30)	R	100k	
Resistor	(2.80, 4.80)	R	10k	
Switch	(2.80, 6.10)	4SW SPST	Mode	12/24
Switch	(1.40, 7.80)	SW PUSH-BUTTON	Reset	
Capacitor	(4.90, 6.40)	CAP	.01uF	
Capacitor	(9.90, 6.40)	CAP	.01uF	
Capacitor	(15.30, 6.40)	CAP	.01uF	
Resistor	(8.70, 6.40)	R	10k	
Resistor	(13.80, 6.40)	R	10k	
Resistor	(19.20, 6.40)	R	10k	
Switch	(7.40, 6.90)	4SW SPST	Mode	Hrs
Switch	(13.10, 6.90)	4SW SPST	Mode	Min
Switch	(18.30, 6.90)	4SW SPST	Mode	Sec

Table 6-1. Part value fields to edit.

Add labels to the wires

1. Select **PLACE Label** from the main menu. The "Label?" prompt displays.
2. Enter CLK. This label corresponds to the CLK label you assigned to a wire in the clock oscillator circuit schematic in chapter 3.
3. Move the pointer to the end of the unconnected wire at the left side of the logic and display circuit, and place the CLK label. Remember, when placing a label on a wire, the leftmost point of the label name must be placed next to the wire.

   The clock signal from the clock oscillator circuit is now logically connected to the wire to which you attached the CLK label (figure 6-7).
4. The "Label?" prompt returns each time you place a label. Label the following wires: MILTIME, RESET, SETHRS, MINCARRY, SETMIN, SECCARRY, and SETSEC. Refer to figure 6-7 for the location of these wires. Press <Esc> to dismiss the "Label?" prompt.

You still need to add labels to the wires between the 22V10s and the seven-segment display parts. You could continue placing labels as with the previous steps, but **Draft** allows you to take a shortcut when labeling with repeated text.

Set repeat text parameters

Follow these steps to set the parameters for quickly adding labels to the wires between the 22V10s and the LEDs:

1. Move the pointer to grid reference C-6. You want to look at the area where the labels will be placed.
2. Select **SET Repeat Parameters.** The menu shown below displays.
3. Set **X Repeat Step** to **+2.**
4. Set **Y Repeat Step** to **–1** (equal to the wire spacing).
5. Set **Label Delta** to **–1.**

```
Set Repeat Parameters
X Repeat Step      +0
Y Repeat Step      +1
Label Delta        +1
Auto Increment     NO
```

△ *NOTE: Depending on the spacing between wires, you may have to adjust the X and Y values. Try it and see what works for your worksheet.*

These **Repeat Parameters** cause labels to be placed two grid spaces to the right and one space up, and cause the number in the text to be decremented by one count each time you run the **REPEAT** command.

Place labels with repeat text

Follow these steps to quickly add labels to the wires between the 22V10s and the LEDs:

1. Select **PLACE Label** from the main menu. The prompt "Label?" displays.
2. Enter `LHRS3`.
3. Move the pointer to the bottom wire directly below the TIL309 on the left, and place the label as shown in figure 6-6.
4. Press <Esc> to dismiss the "Label?" prompt.
5. Select **REPEAT** three times.

   The labels LHRS2, LHRS1, and LHRS0 should be placed in the proper relative locations on the worksheet.
6. If the labels are not in the proper location, delete the out-of-position labels, adjust the **Repeat Parameters** to correct the problem, and repeat steps 1 through 5.
7. See figure 6-7 and place labels for the remaining TIL309s (LMIN*n* and LSEC*n*) by repeating steps 1 through 5.

Place the remaining repeat labels

The labels for wires going to the right displays slant in a different direction than those of the left displays, but otherwise the placement procedure is unchanged.

1. Select **SET Repeat Parameters**.
2. Set the **X Repeat Step** to **–1**, the **Y Repeat Step** to **–2** (again, these values may vary depending on your wire spacing), and the **Label Repeat Delta** to **+1**.
3. Select **PLACE Label**. The prompt "Label?"displays.
4. Enter `RHRS0`.
5. Move the pointer to the bottom wire for the right hours display, and place the label as shown in figure 6-7.
6. Press <Esc> to dismiss the "Label?" prompt.

7. Select **REPEAT** three times.
8. See figure 6-7 and place labels for the remaining right displays (RMIN*n* and RSEC*n*) by repeating steps 3 through 7.

Add comment text

1. Select **PLACE Text**. The "Text?" prompt displays.
2. Enter `Logic and Display Circuit`.
3. Select **Larger** from the **PLACE Text** menu to use a larger type size for the text. The image of the text becomes larger.
4. Center the text below the schematic diagram (at approximately 9.20, 8.40). Press <P> to place the text.
5. The "Text?" prompt redisplays. Enter `MILTIME is set for`.
6. Select **Smaller** from the **PLACE Text** menu until the text size is the same size as the part and wire labels.
7. See figure 6-7 and place the text to the left of the 12/24 switch, at approximately (0.80, 6.10).
8. The "Text?" prompt redisplays. Enter `24-hour format`.
9. See figure 6-7 and place the text under the text you placed in step 7.

Editing the title block

The title block is located in the lower-right corner of the worksheet. You use the title block to provide standard types of information on the schematic, such as a title for the sheet, date, and reference number.

Jump to the title block

You can use the mouse to move the pointer to the title block region of the worksheet, or move there quickly by using the **JUMP** command.

1. Select **JUMP Reference.** The **JUMP to Reference** menu displays.
2. Select **A**, and then **1**. The pointer jumps to region A-1 of the worksheet, and the title block is in view.

   Notice that the title block contains the information entered in chapter 2.

Edit the title block

1. Select **EDIT**. The **EDIT** command line displays.
2. Put the pointer somewhere within the title block. Select **Edit.** The **Edit title block** menu displays.
3. Select one of the types of information listed in the menu. For example, Select **Organization name.** The "Organization name?" prompt displays.

```
Edit title block
Revision code
Title of sheet
Document number
Sheet number
Number of sheets
Organization name
1st Address line
2nd Address line
3rd Address line
4th Address line
```

4. Since you already entered the name of your organization in chapter 2, you can either leave it as it is or delete it and enter a new name.

   Once you press <Enter>, **Draft** stores the information and redisplays the **Edit title block** menu so you can specify other types of information.

*NOTE: Once you change a field in the title block, the information entered in the **Worksheet Options** area of the **Configure Schematic Tools** screen is no longer used for the changed fields.*

5. Following the procedures in steps 3 and 4, fill in or change other title block information. Filling in the boxes is optional for this tutorial.
6. When you are finished, press <Esc> twice to dismiss the **Edit title block** menu and the **EDIT** command line.

Updating the file

The digital clock design schematic is now complete. Save your work and exit by selecting **QUIT**, then **Update file**, then **Abandon Edits**. The **Schematic Design Tools** screen displays.

Summary

In the past five chapters, you learned several ways to quickly create circuits using **Draft**. In the next chapter, you learn to use some of the other tools.

Using other Schematic Design Tools

In this chapter you learn how to use some of the other **Schematic Design Tools**. These tools are normally used after the schematic is complete. The tools covered in this chapter are:

Annotate Schematic	Automatically updates part reference designators (and the associated pin numbers, in multiple-element parts).
Update Field Contents	Loads information into the **Part Value** field and the **1st Part Field** through **8th Part Field**.
Check Electrical Rules	Checks for conformity to basic electrical rules.
Create Netlist	Creates a netlist and general wire list in one of over thirty standard formats.
Back Annotate	Updates part reference designators by using a list of old and new reference designators.
Create Bill of Materials	Creates a summary list, sorted by reference designator, of all the parts used.
Plot Schematic	Sends a plot of a single schematic or all schematics in a design to a plotter or a printer; supports scaling.

Housekeeping

Before proceeding with the tutorial, you should perform a few housekeeping tasks. You have completed quite a bit of work up to this point, so it's a good idea to back up your design files.

In case you were unable to complete the exercises in chapters 1–6, OrCAD has provided copies of the TUTOR schematic and library files. These files are called TUTOR2.SCH and .\TUTORSDT.LIB. By using these files you can perform the remaining exercises with predictable results. Once you back up your design, you copy these two files to TUTOR.SCH and .\TUTOR.LIB.

Backup Design

Use the **Backup Design** tool to back up onto floppy disks or to another part of your hard disk all the files belonging to a design. To conserve disk space, back-up files are stored in a condensed format. To restore the files to their normal format, use the **Restore Design** tool, which is described in the *ESP Design Environment User's Guide.*

Follow these steps to back up a design:

1. On the **Schematic Design Tools** screen, click the title bar or any place that is not a button. The **Design Management Tools** menu displays.

```
Design Management Tools
+----------------------------+
|Design Management Tools     |
|Suspend to System           |
|Vendor Selection            |
|Show Hot Key Assignments    |
|Exit                        |
+----------------------------+
```

2. Select **Design Management Tools.** The **Design Management Tools** screen displays.
3. Select **Backup Design.** The screen shown in figure 7-1 displays.

△ *NOTE: The* **Backup Design** *tool does not copy any files that reside outside the specified design directory. See* Chapter 9: Tips and techniques *for a list of the files to include when you transfer a design to another user or to OrCAD Technical Support.*

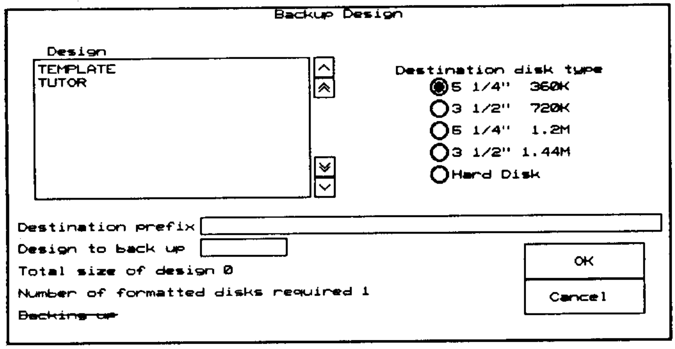
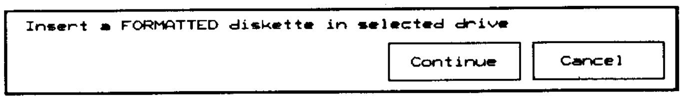


Figure 7-1. ***Backup Design*** *screen.*

4. Select the TUTOR design in the **Design** list box.
5. Select one of the **Destination disk type** radio buttons.
6. Move the pointer to the **Destination prefix** entry box and press <Enter>.
7. Enter the path to use for the backup. To back up the design on a floppy disk, enter the destination prefix `A:`. The message shown below displays.



8. Insert a properly formatted disk in drive A and select **Continue.** Select **Cancel** if you want to cancel the backup for the time being.
9. Select **OK**. The design environment makes a back-up copy of the selected design onto the disk or into the directory specified.

   Once the design is backed up, the message "Backup successfully completed" displays along with an **OK** button.
10. Select **OK** and then **Cancel**. The **Design Management Tools** screen displays.

Copy File

Follow these steps to copy the TUTOR2.SCH and TUTORSDT.LIB files to TUTOR.SCH and TUTOR.LIB, respectively:

1. The **Design Management Tools** screen should still be displayed. Make sure that the **Files wildcard** entry box shows `*.*` and select **File View** at the top of this screen. The **File View** screen displays.
2. Select **Copy File**. The **Copy File** screen displays.
3. Select the TUTORSDT.LIB file from the **Files** list box.
4. Move the pointer to the **Destination** entry box and press <Enter>.
5. Enter the new name for the file, **`TUTOR.LIB`**, and then select **OK**.
6. Because TUTOR.LIB exists, a message box displays. Select **OK** to overwrite TUTOR.LIB.
7. Select the TUTOR2.SCH file from the **Files** list box.
8. Move the pointer to the **Destination** entry box and press <Enter>.
9. Enter the new name for the file, **`TUTOR.SCH`**, and then select **OK**.
10. Because TUTOR.SCH exists, a message box displays. Select **OK** to overwrite TUTOR.SCH.
11. Select **Cancel** to return to the **File View** screen. Select **Cancel** again to return to the **Schematic Design Tools** screen.

Now that your files are backed up and you have copied TUTOR2.SCH and TUTORSDT.LIB to TUTOR.SCH and TUTOR.LIB, you are ready to continue learning about **Schematic Design Tools 386+**.

Running Annotate Schematic

Annotate Schematic scans schematic designs and automatically updates part reference designators. It also updates the pin numbers associated with the reference designators in multiple-element parts.

When you first place a part, a default reference designator—such as **U?** for a single-element part and **U?A** for a part of a multiple-element part—displays above the part. **Annotate Schematic** changes the default values to unique values for each part in a specified design.

For example, suppose the specified design contains three occurrences of a particular two-element part with the reference prefix "U." When each of the six parts is placed, **Draft** assigns it the default reference designator, **U?A.** **Annotate Schematic** updates these designators to **U1A, U1B, U2A, U2B, U3A,** and **U3B.** For each of these six parts, the number identifies the two-element part of which the part is an element, and the "A" and "B" suffix letter designates the part's "slot" in that two-element part.

You should run **Annotate Schematic** before running the other tools. Other tools report information about the worksheet file, and if you run **Annotate Schematic** first, you ensure that reported information includes the updated reference designators.

Annotate Schematic modifies the worksheet file, but it also creates a back-up file containing the original worksheet file.

You can also assign values of your choice using **Draft's EDIT** command, but assigning values using **Annotate Schematic** guarantees unique values. Unique reference designator values are necessary for some other processes, such as producing a netlist.

Run Annotate Schematic on TUTOR.SCH

Follow these steps to configure and run **Annotate Schematic**:

1. Select **Annotate Schematic** from the **Schematic Design Tools** screen. The menu at right displays.

```
Annotate Schematic
Execute
Local Configuration
Assign Hot Key
Show Version
Configure Schematic Tools
Help
```

2. Select **Local Configuration** and then select **Configure ANNOTATE**. The **Configure Annotate Schematic** screen displays (figure 7-2).

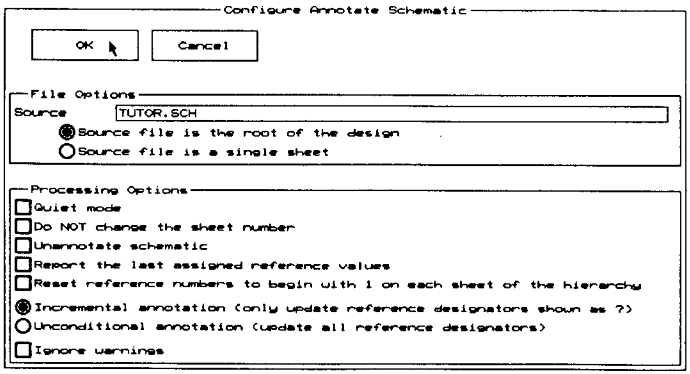


*Figure 7-2. The **Configure Annotate Schematic** screen.*

Notice that the **Source** entry box contains the filename TUTOR.SCH. The **Source** name is initially set to the name in the **Root Sheet** entry box of the **Design Management Tools** screen.

3. Under **File Options**, select **Source file is a single sheet.**

△ *NOTE: When you run **Annotate Schematic** on a multiple-sheet design, select **Source file is the root of the design.***

4. Select **OK.**

5. Select **Annotate Schematic** and then **Execute.**

   As it processes, **Annotate Schematic** scrolls status messages three lines at a time in the monitor box at the bottom of the **Schematic Design Tools** screen (figure 7-3).

```
Annotation - OrCAD/SDT 386+ V1.20   19-AUG-94
(C) Copyright 1991,1992,1993 OrCAD, Inc.      ALL RIGHTS RESERVED.
Reading Configuration Data File: sdt.bcf
```

*Figure 7-3. Status messages display at the bottom of the **Schematic Design Tools** screen.*

   When **Annotate Schematic** is finished, the monitor box disappears and the full **Schematic Design Tools** screen displays.

6. Run **Draft** and examine the TUTOR.SCH worksheet. Note the reference designators and pin numbers. Your reference designators may differ slightly from those shown in this tutorial. This is because **Annotate Schematic** assigns reference designators in the order in which you placed or edited the parts.

Notice the updated reference designators on the 74LS04 inverters near the upper-left corner of the worksheet. The U?A changed to U1D, U1E, and U1F. All six inverters are from the same multiple-element part. Also notice that the inverters' pin numbers changed.

Running Update Field Contents

Each schematic part has ten part fields —**Part Value**, **Reference**, and **1st Part Field** through **8th Part Field** — available for storing up to 128 characters of information. You can alter this information by editing individual parts in **Draft** as you did in the *Editing part fields* section of chapter 3. This is the only way to change the **Reference** field, which is protected, but you can use **Update Field Contents** to change the other nine fields. This is especially convenient when you want to edit the information for several parts.

You run **Update Field Contents** once for each part field you want to change. Before running **Update Field Contents** for a single part field, you must:

- Tell **Update Field Contents** which part field to update. You do this in the Local Configuration screen.
- Tell **Update Field Contents** which part fields to use as a key field. You do this in the **Configure Schematic Design Tools** screen. **Update Field Contents** compares the character strings in the key field to character strings in your update file in order to match the new information to a specific part.
- Tell **Update Field Contents**, for each schematic part to be updated, what character string is presently associated with the part (in the key field), and what character string is to be loaded. You do this by creating an *update file*.

After you complete these tasks for one part field field, run **Update Field Contents**. For each part field you want to update, repeat the procedure and run **Update Field Contents** again.

In the following sections, you update the **8th Part Field** (also called the **Module Value** field) of schematic parts on the TUTOR.SCH worksheet. The values you load into the **8th Part Field** will be used as module names when you create a netlist later in this chapter.

△ *NOTE: See* Chapter 1: Configure Schematic Tools *and* Chapter 14: Update Field Contents *in the* Schematic Design Tools 386+ Reference Guide *for more information about key fields and **Update Field Contents** and for a description of the update file format.*

Configure Update Field Contents

1. Select **Update Field Contents** on the **Schematic Design Tools** screen, and then select **Local Configuration** and **Configure FLDSTUFF**. The **Configure Update Field Contents** screen displays (figure 7-4).

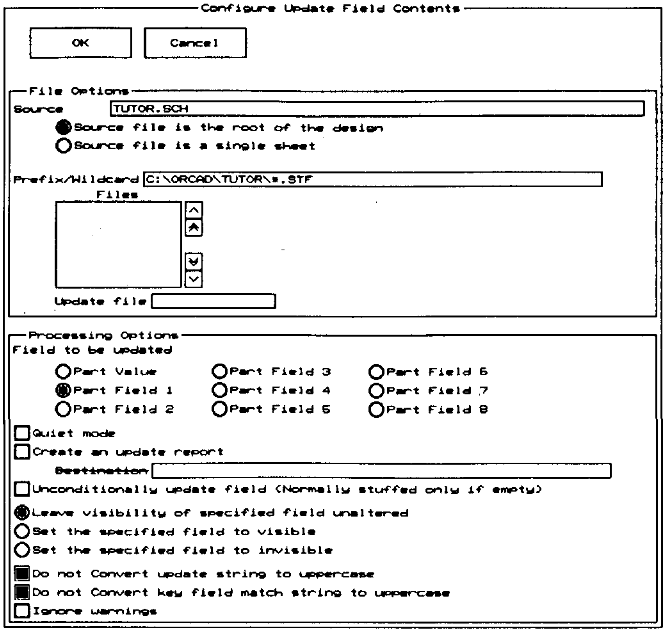


*Figure 7-4. The **Configure Update Field Contents** screen.*

2. Make sure the **Source** entry box contains the name of the worksheet, TUTOR.SCH.

3. Enter `TUTOR.STF` in the **Update file** entry box. This is the name of the update file you create later in this chapter.

4. Under **Field to be updated**, select **Part Field 8**. This tells **Update Field Contents** which part field to update.

5. For this tutorial, select **Unconditionally update field** and **Set the specified field to visible**. (You can make the field invisible later to keep it from cluttering your worksheet.)

6. Select **OK**. The **Schematic Design Tools** screen displays.

Define the key field

1. Select **Update Field Contents** on the **Schematic Design Tools** screen, and then select **Configure Schematic Tools.**
2. Scroll to the **Key Fields** area of the **Configure Schematic Design Tools** screen (figure 7-5).

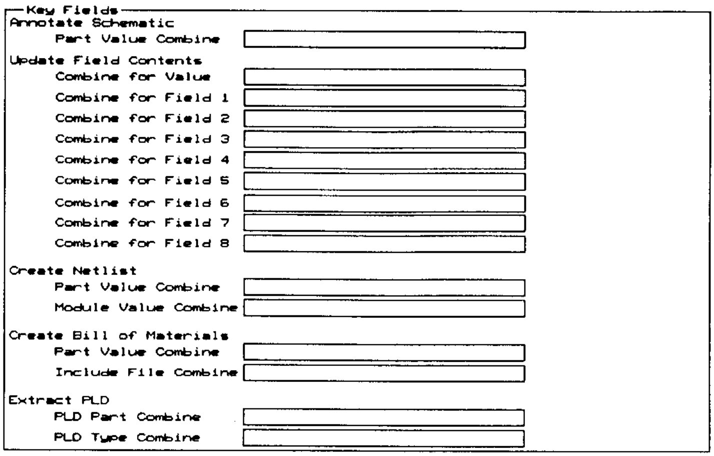


*Figure 7-5. **Key Fields** area of the **Configure Schematic Design Tools** screen.*

3. Enter **V** (it *must* be an uppercase "V") in the **Combine for Field 8** entry box. This tells **Update Field Contents** to compare the contents of the **Part Value** field with the *match strings* in the update file.
4. Select **OK.** The **Schematic Design Tools** screen displays.

Create the update file

The update file contains pairs of strings: a match string followed by an associated update string. Each string is enclosed in single quotation marks. For readability, place one pair on each line of the file.

1. Create a text file using **Edit File**. For this exercise, call the file TUTOR.STF.

   See the *ESP Design Environment User's Guide* for more information about the text editor that comes with the design environment or to learn how to configure the design environment to use another text editor.

2. Include the information shown at right, capitalized as shown, in the text file. Press <Tab> or use spaces to separate paired items.

3. Save the text file.

```
'9V'      '9VBAT'
'.01uF'   'CK06'
'22uF'    'CK15'
'100uF'   'CK15'
'470uF'   'CK15'
'47uF'    'CK06'
'LM7805'  'TO220'
'9.1k'    'RC06'
'10k'     'RC06'
'100k'    'RC06'
'Mode'    '8DIP300'
'Reset'   'TJACK200'
'22V10'   '24DIP300'
'74LS04'  '14DIP300'
```

△ *NOTE: Be sure to save this file as text only. Any special formatting inserted by your text editor will cause* ***Update Field Contents*** *to fail.*

Update Field Contents compares each of the match strings in the left column with the contents of the **Part Value** field of each part. This comparison is case-sensitive. If it finds a match, **Update Field Contents** loads the corresponding update string into the part's **8th Part Field**. By default, lowercase characters in the update string (the right column) are converted to uppercase when they are loaded into the field; select **Do not convert update string to uppercase** to prevent this.

Update the fields

1. Select **Update Field Contents** on the **Schematic Design Tools** screen, and then select **Execute.**

   As it processes, **Update Field Contents** scrolls status messages three lines at a time in the monitor box at the bottom of the **Schematic Design Tools** screen.

2. Run **Draft** on TUTOR.SCH to see the updated fields on the worksheet.

Each part with a **Part Value** field that matched one of the match strings in TUTOR.STF now displays a **8th Part Field.** Each **8th Part Field** contains the update string associated with the match string that matched the part's **Part Value** field.

Hide the fields

To minimize clutter on your worksheet, use **Select Field View** to hide the **8th Part Field** for all the parts.

1. Select **Select Field View** on the **Schematic Design Tools** screen, and then select **Local Configuration** and **Configure FLDATTRB.** The **Configure Select Field View** screen displays (figure 7-6).

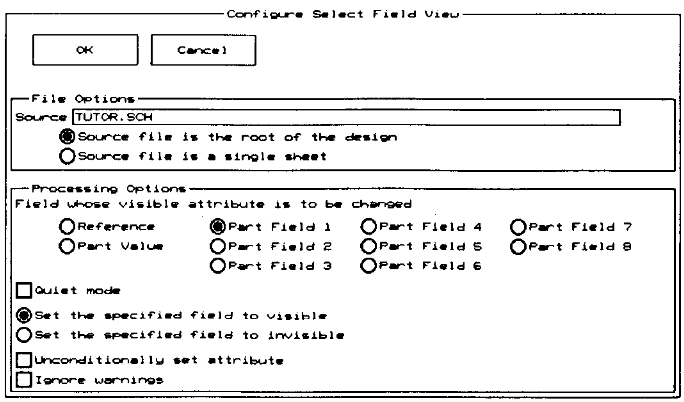


Figure 7-6. The Configure Select Field View screen.

2. Make sure the **Source** entry box contains the name of the worksheet, TUTOR.SCH.

3. In the **Processing Options** area, select **Part Field 8** and **Set the specified field to invisible.**

4. Select **OK.** The **Schematic Design Tools** screen displays.
5. Select **Select Field View** on the **Schematic Design Tools** screen, and then select **Execute.**

   As it processes, **Select Field View** scrolls status messages three lines at a time in the monitor box at the bottom of the **Schematic Design Tools** screen.
6. Look at TUTOR.SCH in **Draft**, and notice that the **8th Part Field** on all the parts is not displayed.

△ *NOTE: You can also control the visibility of individual part fields with the **EDIT** command in **Draft**.*

Running Check Electrical Rules

Check Electrical Rules performs a general electrical rules check. It issues warnings for unused inputs on parts, unlabeled wires connected to a bus, and invalid connections.

△ *NOTE: Always use **Check Electrical Rules** on your designs before going on to **Digital Simulation Tools** or **PC Board Layout Tools**. If any errors are reported, correct them before trying to simulate the design or the simulation results will be inaccurate.*

Follow these steps to configure and run **Check Electrical Rules**:

1. Select **Check Electrical Rules, Local Configuration,** and then **Configure ERC**. The **Configure Check Electrical Rules** screen displays.

   Notice that the **Source** entry box contains the filename TUTOR.SCH.

   Notice that the **Destination** entry box contains the filename TUTOR.ERC. **Check Electrical Rules** stores the report it creates in a text file with the worksheet name and the .ERC extension.

   You can also specify a path to another directory and another file, but for the purposes of this exercise, you should place the report in TUTOR.ERC.

2. In the **File Options** area, select **Source is a single sheet.**
3. Select **OK.**
4. Select **Check Electrical Rules** and then **Execute.**

   As it processes, **Check Electrical Rules** scrolls status messages three lines at a time in the monitor box at the bottom of the **Schematic Design Tools** screen.

5. After **Check Electrical Rules** is finished running, a message box displays. You may select **View Output** to display the report, or select **OK** to dismiss the message box. Select **OK.**
6. Examine the file TUTOR.ERC with **Edit File.** The contents of the file should be similar to figure 7-7.

```
TUTOR.SCH

Electrical Rules Check Report
Digital clock schematic  Revised:      September 2, 1994
          Revision:
OrCAD, Inc.
9300 SW Nimbus Ave
Beaverton, Oregon  97008-7137
(503) 671-9500

WARNING: Single node net. Net at:  X :=   4.50,Y :=   2.20
WARNING: Single node net. Net at:  X :=   4.70,Y :=   2.20
   .
   .
   .
WARNING: Single node net. Net at:  X :=  11.10,Y :=   5.70
WARNING: Single node net. Net at:  X :=  16.40,Y :=   5.70
```

Figure 7-7. The TUTOR.ERC file.

The report lists "single node nets," which are unconnected signals at each of the locations cited. Since the reported pins were intentionally not connected, you can ignore these warnings. If desired, you can examine the schematic and locate these pins.

View errors

Now use **Draft** to view the schematic. Notice a circle at each location where an error is reported by **Check Electrical Rules**. These are *error markers*. To display the associated error message, place the pointer in the center of an error marker and select **INQUIRE** from the main menu. The error message displays at the top of the screen. Repeatedly selecting **INQUIRE** cycles through all of the error messages for a particular error marker.

Select **QUIT Abandon Edits** when you are done looking at the schematic.

△ *NOTE: If you save the schematic file by selecting* ***QUIT Update File,*** *the error markers are erased.*

Running Create Netlist

Create Netlist creates a connectivity database, or netlist, in a number of possible formats. To create a proper netlist, you must carefully deal with labels, module ports, and power objects. The general guidelines are:

- ❖ Place labels in the correct format on all buses.
- ❖ Place labels in the correct format on all signals connecting to a bus.
- ❖ Place module ports in the correct format on all signals going off the worksheet.
- ❖ Don't put blank spaces in labels or between prefixes and suffixes in bus and module port names.
- ❖ Do not overlap wires or buses with other wires, buses, or object pins.

For a more detailed discussion of these guidelines, see *Chapter 3: Guidelines for creating designs* in the *Schematic Design Tools 386+ Reference Guide.*

Specify where to get the module value

1. Select **Create Netlist**, and then select **Configure Schematic Tools.**

2. Scroll to the **Key Fields** area of the **Configure Schematic Design Tools** screen (figure 7-8).

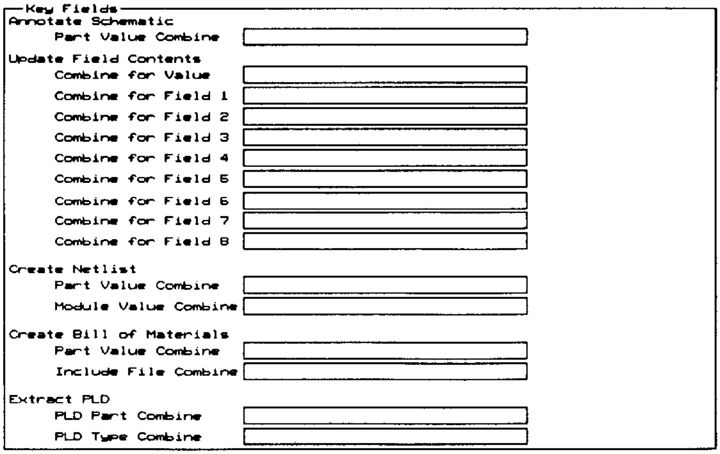


*Figure 7-8. **Key Fields** area of the **Configure Schematic Design Tools** screen.*

3. Enter 8 in the **Module Value Combine** entry box to tell **Create Netlist** to use the contents of the **8th Part Field** as the module name. (You loaded the module names into the **8th Part Field** when you ran **Update Field Contents** earlier in this chapter.)

4. Select **OK**. The **Schematic Design Tools** screen displays.

△ *NOTE: For any part that does not have a value in the field you specify, **Create Netlist** will substitute the contents of the **Part Value** field. In the section* Running Update Field Contents, *for example, you did not specify an **8th Part Field** for the TIL309 parts, because the module name is the same as the **Part Value** for these parts.*

Create a netlist in WIRELIST format

1. Select **Create Netlist** and then **Local Configuration** to configure **Create Netlist**.

   ```
   Select Configuration
   Configure INET
   Configure ILINK
   Configure IFORM
   INET on
   ILINK on
   IFORM on
   ```

   The menu shown at right displays. This menu has three processes to configure: INET, ILINK, and IFORM. INET is the compiler. ILINK is the connectivity linker, and IFORM is the netlist formatter. Each of these processes must be turned on to create a netlist. For more information on each of these processes, see the *Schematic Design Tools 386+ Reference Guide.*

2. Select **Configure INET**. The **Configure Incremental Netlist** screen displays (figure 7-9).

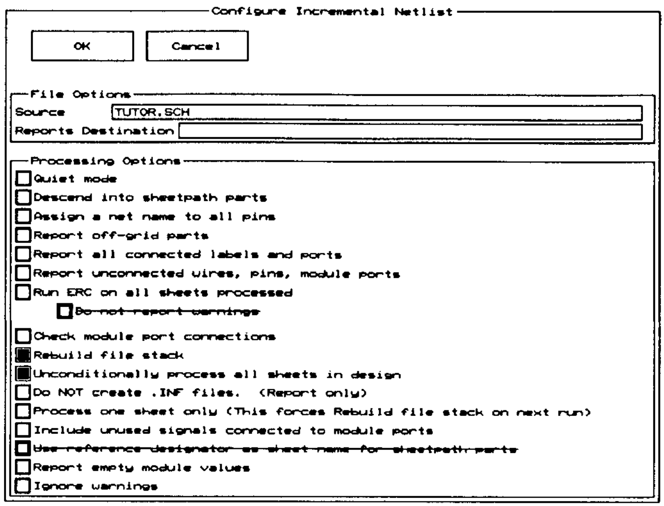


*Figure 7-9. The **Configure Incremental Netlist** screen.*

3. In the **File Options** area of the screen, check to be sure that the **Source** entry box contains the filename TUTOR.SCH. The design environment initially sets the **Source** entry box to the name in the **Root Sheet** entry box of the **Design Management Tools** screen.

4. Select **Cancel** to leave the configuration screen without making any changes. The **Schematic Design Tools** screen displays.

5. Now display **ILINK**'s local configuration screen. Notice that the **Source** entry box contains the filename TUTOR.INF. Select **Cancel**.

6. Now display **IFORM**'s local configuration screen. **IFORM** is the netlist formatter that converts the connectivity database that has been linked by **ILINK** into the format specified in this configuration.

   The **Source** entry box should contain the name TUTOR, showing that you will format the TUTOR files created by **ILINK**.

   The **Destination 1** entry box should contain the filename TUTOR.NET.

7. The **Format prefix/wildcard** should contain the following pathname and filter:

   ```
   Format prefix/wildcard
   C:\ORCADESP\SDT\NETFORMS\F*.EXE
   ```

   The **Netlist format** list box contains a number of files. Edit the **Format prefix/wildcard** entry box by inserting a "W" before the *, so that it becomes:

   ```
   Format Prefix/Wildcard
   C:\ORCADESP\SDT\NETFORMS\FW*.EXE
   ```

   The list box now contains far fewer filenames. Select FWIRELIS.EXE. The selected netlist format filename displays in the **Selected format** entry box:

   ```
   Selected format: FWIRELIS.EXE
   ```

8. Select **OK** to save all of the configuration changes.

9. Select **Create Netlist** and then select **Execute** to run **Create Netlist.**

   As it processes, **Create Netlist** scrolls status messages three lines at a time in the monitor box at the bottom of the **Schematic Design Tools** screen.

10. Using **Edit File,** look at the file generated by **Create Netlist,** TUTOR.NET. It should look like the wirelist-format netlist file shown in figure 7-10.

```
Wire List

Digital clock schematic                            Revised:      September 2, 1994
                                                   Revision:

OrCAD Inc.
9300 SW Nimbus
Beaverton, OR  97008
(503) 671-9500


<<< Component List >>>
.01UF                         C5        CK06
.01UF                         C6        CK06
.01UF                         C7        CK06
.01UF                         C8        CK06
100K                          R8        RC06
100UF                         C4        CK15
10K                           R2        RC06
10K                           R3        RC06
10K                           R4        RC06
10K                           R5        RC06
10K                           R6        RC06
10K                           R7        RC06
22UF                          C3        CK15
22V10                         U2        24DIP300
22V10                         U3        24DIP300
22V10                         U4        24DIP300
470UF                         C2        CK15
47UF                          C1        CK06
74LS04                        U1        14DIP300
9.1K                          R1        RC06
9V                            BT1       9VBAT
LM7805                        Q1        TO220
MODE                          S1        8DIP300
RESET                         S2        TJACK200
TIL309                        D1        TIL309
TIL309                        D2        TIL309
TIL309                        D3        TIL309
TIL309                        D4        TIL309
TIL309                        D5        TIL309
TIL309                        D6        TIL309

<<< Wire List >>>

  NODE  REFERENCE     PIN #   PIN NAME          PIN TYPE      PART VALUE

[00001] N00001
        R2               2       2              Passive       10K
        U1               9       I_D            Input         74LS04
        U1              11       I_E            Input         74LS04
        U1              13       I_F            Input         74LS04

[00002] CLK_1 (local to sheet 1)
        U1               4       O_B            Output        74LS04
        U1               5       I_C            Input         74LS04
        D6               5       STROBE         Input         TIL309
        D5               5       STROBE         Input         TIL309
        U4               1       I1/CLK         Input         22V10
        D4               5       STROBE         Input         TIL309
        D3               5       STROBE         Input         TIL309
        D2               5       STROBE         Input         TIL309
```

Figure 7-10. The wirelist-format netlist (continued on next page).

```
          D1              5      STROBE          Input       TIL309
          U2              1      I1/CLK          Input       22V10
          U3              1      I1/CLK          Input       22V10
          .
          .
          .
[00040]   GND
          C2              2      2               Passive     470UF
          BT1             2      2               Passive     9V
          Q1              2      GND             Input       LM7805
          C3              2      2               Passive     22UF
          C1              2      2               Passive     47UF
          R8              2      2               Passive     100K
          U4              3      I3              Input       22V10
          U4              2      I2              Input       22V10
          U4              4      I4              Input       22V10
          U4              5      I5              Input       22V10
          U4              6      I6              Input       22V10
          U4              7      I7              Input       22V10
          U4              8      I8              Input       22V10
          U2              2      I2              Input       22V10
          U2              3      I3              Input       22V10
          U2              4      I4              Input       22V10
          U2              5      I5              Input       22V10
          U2              6      I6              Input       22V10
          U2              7      I7              Input       22V10
          U2              8      I8              Input       22V10
          U2              9      I9              Input       22V10
          U3              2      I2              Input       22V10
          U3              3      I3              Input       22V10
          U3              4      I4              Input       22V10
          U3              5      I5              Input       22V10
          U3              6      I6              Input       22V10
          U3              7      I7              Input       22V10
          U3              8      I8              Input       22V10
          U3              9      I9              Input       22V10
          S1              4      1_D             Passive     MODE
          S1              3      1_C             Passive     MODE
          S1              2      1_B             Passive     MODE
          U3              12     GND             Power       22V10
          U2              12     GND             Power       22V10
          U4              12     GND             Power       22V10
          C7              2      2               Passive     .01UF
          C6              2      2               Passive     .01UF
          C8              2      2               Passive     .01UF
          S1              1      1_A             Passive     MODE
          D6              12     DPIN            Input       TIL309
          D5              12     DPIN            Input       TIL309
          D1              12     DPIN            Input       TIL309
          D2              12     DPIN            Input       TIL309
          D3              12     DPIN            Input       TIL309
          D4              12     DPIN            Input       TIL309
          U1              7      GND             Power       74LS04
          C5              2      2               Passive     .01UF
          D4              8      GND             Power       TIL309
          D3              8      GND             Power       TIL309
          D2              8      GND             Power       TIL309
          D1              8      GND             Power       TIL309
          D6              8      GND             Power       TIL309
          D5              8      GND             Power       TIL309
```

Figure 7-10. The wirelist-format netlist (continued from previous page).

Running Back Annotate

If you want to change the reference designator values assigned by **Annotate Schematic** (or the values you manually assigned), you need not reopen the worksheet and edit the reference designators one by one.

Back Annotate lets you change as many reference designators as you want in a single operation. You create a text file containing the current and new values (called a WAS/IS file) and then run **Back Annotate**, specifying the worksheet name and the WAS/IS filename.

You can run **Back Annotate** on a single worksheet or on an entire design.

For example, consider the TUTOR.SCH worksheet. Currently, the six LED parts in TUTOR.SCH have reference designators of D1, D2, D3, and so on. Suppose you decide you want the values to be A1, A2, A3, and so on. In this example, you will run **Back Annotate** on the schematic, TUTOR.SCH.

Change reference designator values

1. Create a text file using **Edit File**. Enter a filename in the **File to Edit** entry box. For the purposes of this exercise, enter `TUTREF`.

   See the *ESP Design Environment User's Guide* for more information about the text editor that comes with ESP, or to learn how to configure ESP to use another text editor.

2. Include the information shown at right in the text file. Use <Tab> or blank spaces to separate the paired items.

```
D1   A1
D2   A2
D3   A3
D4   A4
D5   A5
D6   A6
```

3. Save the text file.

△ *NOTE: Be sure to save this file as text only. Any special formatting inserted by your text editor causes* ***Back Annotate*** *to fail. In addition, some text editors may attach an extension to the TUTREF file. If it does, be sure to enter the extension when running* ***Back Annotate****.*

4. Return to the **Schematic Design Tools** screen and select **Back Annotate.** Select **Local Configuration** and then **Configure BACKANNO.** The screen shown in figure 7-11 displays.

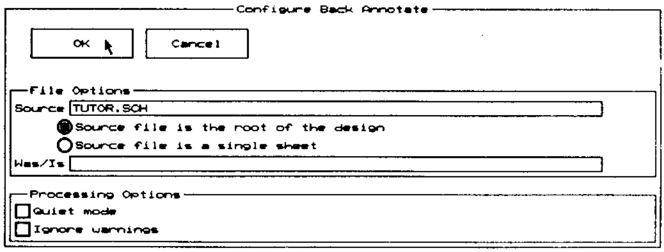


*Figure 7-11. The **Configure Back Annotate** screen.*

5. Enter the name of the file where **Back Annotate** gets the back annotation information—in this case `TUTREF`—in the **Was/Is** entry box.
6. Select **OK.** The **Schematic Design Tools** screen displays.
7. Run **Back Annotate.** As it processes, **Back Annotate** scrolls status messages three lines at a time in the monitor box at the bottom of the **Schematic Design Tools** screen.

   Back Annotate modifies the schematic file, TUTOR.SCH, to reflect the new reference designator values found in the WAS/IS file, TUTREF.
8. Run **Draft** on TUTOR.SCH to confirm that **Back Annotate** modified the reference designators on the schematic.

Running Create Bill of Materials

Create Bill of Materials creates a text file listing all parts in a single worksheet or an entire design.

Make a parts list

1. Select **Create Bill of Materials, Local Configuration,** and then **Configure PARTLIST**. The **Configure Create Bill of Materials** screen displays (figure 7-12).

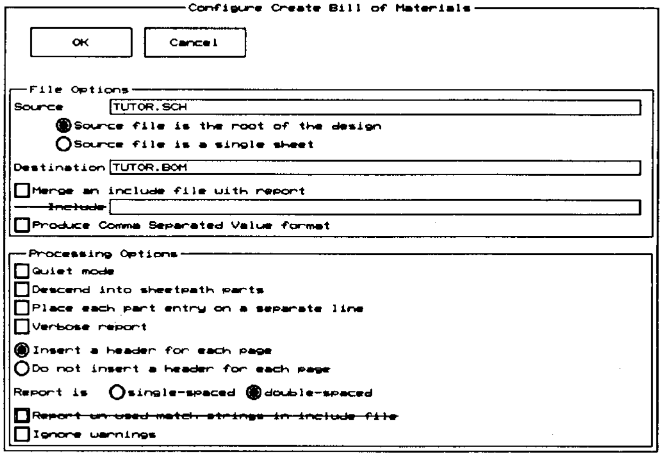


*Figure 7-12. **Configure Create Bill of Materials screen.***

2. In the **File Options** area of the **Configure Create Bill of Materials** screen, there are two filenames:

   ❖ In the **Source** entry box, the name of the worksheet from which the bill of materials is produced: TUTOR.SCH.

   This entry box tells **Create Bill of Materials** to use the worksheet file TUTOR.SCH to get the correct reference designator values.

   ❖ In the **Destination** entry box, the name of the file where **Create Bill of Materials** stores the report: TUTOR.BOM.

3. Select **Cancel**. The **Schematic Design Tools** screen displays.

4. Select **Create Bill of Materials** and then select **Execute**.

   As it processes, **Create Bill of Materials** scrolls status messages three lines at a time in the monitor box at the bottom of the **Schematic Design Tools** screen.

   The contents of TUTOR.BOM are shown in figure 7-13. Use **Edit File** to look at this file.

```
Digital clock schematic                     Revised:      September 2, 1994
                                            Revision:
Bill Of Materials             September    6, 1994     10:56:40     Page     1
```

Item	Quantity	Reference	Part
1	6	A1,A2,A3,A4,A5,A6	TIL309
2	1	BT1	9V
3	1	C1	47uF
4	1	C2	470uF
5	1	C3	22uF
6	1	C4	100uF
7	4	C5,C6,C7,C8	.01uF
8	1	Q1	LM7805
9	1	R1	9.1k
10	6	R2,R3,R4,R5,R6,R7	10k
11	1	R8	100k
12	1	S1	Mode
13	1	S2	Reset
14	1	U1	74LS04
15	3	U2,U3,U4	22V10

Figure 7-13. The TUTOR design bill of materials.

Running Plot Schematic

The last task in this chapter is to plot the design you have created so far.

Use **Plot Schematic** to send designs to a plotter or, optionally, to a printer using the **Send output to printer** button.

△ *NOTE: This section focuses on running* ***Plot Schematic*** *and assumes you have configured* ***Schematic Design Tools*** *and connected your printer or plotter correctly. Many variables affect plotting. As with other mechanical processes, make sure your equipment—paper, pens, and so on—is in good working order and is set up properly.*

See the Appendix E: Plotter information *in the* Schematic Design Tools 386+ Reference Guide *for more details about using your plotter with OrCAD tools.*

Follow these steps to configure and run **Plot Schematic**:

1. Select **Plot Schematic, Local Configuration,** and then **Configure PLOTALL.** The **Configure Plot Schematic** screen displays.

   If you are using a printer instead of a plotter, select **Send output to printer.**

△ *NOTE: When plotting a multiple-sheet design,* ***Plot Schematic*** *plots every worksheet in the design unless you select the* ***Source file is a single sheet*** *option. Worksheets of sheetpath parts are plotted only if you select* ***Descend into sheetpath parts.***

2. If your plots are too large or too small, you can change the scale. Select **Manually set scale factor and/or X, Y offsets**, and the **Set Scale factor** entry box becomes accessible.

   Enter the scale factor, expressed in the form *n.nnn*. For example, if the plot is larger than the paper, you might run **Plot Schematic** at half scale by entering the number `0.500`.

3. Select **OK**.
4. Select **Plot Schematic** and then select **Execute** to run **Plot Schematic**.

Summary

In this chapter you learned how to use **Annotate Schematic, Update Field Contents, Select Field View, Check Electrical Rules, Create Netlist, Back Annotate, Create Bill of Materials,** and **Plot Schematic**. The next chapter describes three design structures and shows how to create and use them.

Structuring your design

In this chapter you look at three types of design structures: a flat design, a simple hierarchy, and a complex hierarchy.

A flat design

A flat design is one in which all of the worksheets are linked at the same level. Use a flat design for relatively small designs with no more than ten worksheets. Since you must manage all of the interconnections between the worksheets of a flat design by the names assigned to the module ports, large designs consisting of many worksheets or repetitive logic are more easily managed using hierarchical structures.

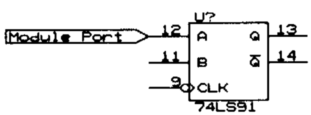
Flat designs are linked using module ports and the |LINK (read as "pipe-link") command.

The |LINK command

```
|LINK
|OTHER
```

You use the |LINK command on the root worksheet to inform the various schematic tools which worksheets are in a flat design. In figure 8-1, the |LINK command links the root worksheet, PROJECT.SCH, to OTHER.SCH. The filename of the root worksheet consists of the name of the design and a .SCH extension.

Module ports



Module ports are graphic objects that indicate where signals are conducted between worksheets. Module ports that have identical names are considered to be electrically connected. In figures 8-1 and 8-2, CLEAR, LOAD, and RCO are connected; Hi[0..3] and Lo[0..3] are not connected.

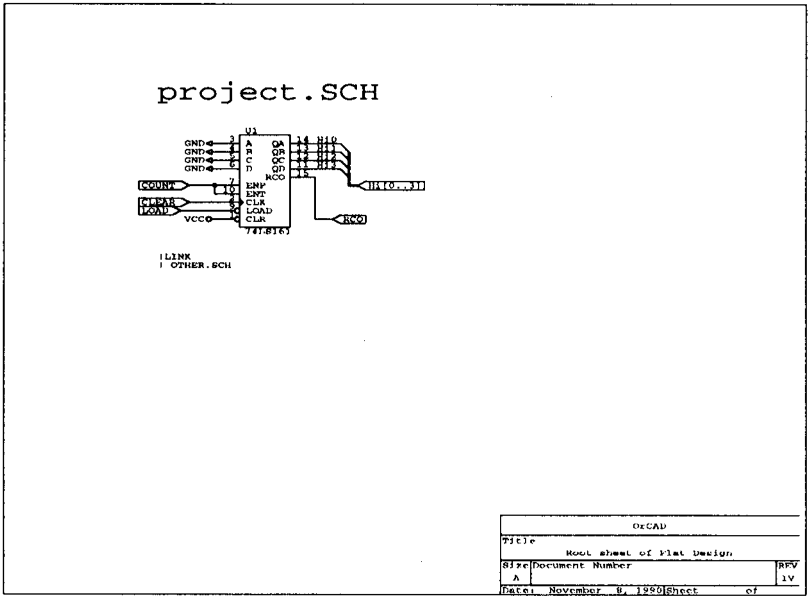


Figure 8-1. Root worksheet of flat design.

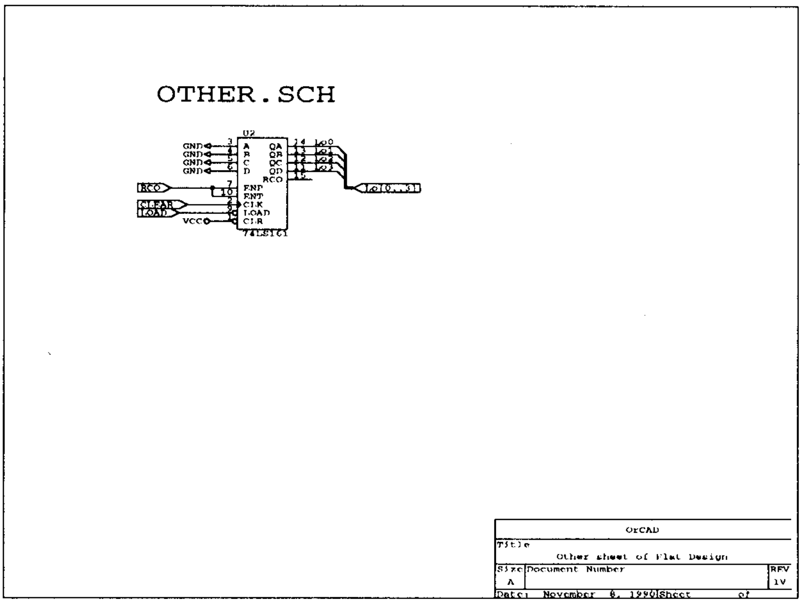


Figure 8-2. Other worksheet of flat design.

Creating new designs

The remainder of this chapter covers simple and complex hierarchies. Example files are included in the TUTOR design. Before you proceed through the rest of this chapter, you need to create two new design directories: CMOSCPU and 4BIT. You will copy the example files from the TUTOR design into these new design directories.

1. On the **Schematic Design Tools** screen, click the title bar or any place that is not a button. The **Design Management Tools** menu displays.
2. Select **Design Management Tools.** The **Design View** portion of the **Design Management Tools** screen displays.
3. Select **Create Design.** The **Create Design** screen displays. Make sure that **Copy all files** is selected.
4. Enter **CMOSCPU** in the **New design name** entry box. Select **OK.** The prompt "Working . . ." and several messages display at the top left corner of the screen. After the new design name is created, the **Create Design** screen is dismissed and the design name, CMOSCPU, appears in the **Design** list box.
5. Repeat steps 3 and 4, but this time create a new design named 4BIT.
6. Select **TUTOR** from the **Design** list box. The files you need to copy are in the TUTOR design.
7. Select **File View** to see the TUTOR files.
8. Select **Copy File.** The **Copy File** screen displays.
9. Using the information in table 8-1, select the source file from the **Files** list box, enter the destination in the **Destination** entry box, and then select **OK.** Repeat this procedure for each file in the table.

Source file name	*Destination*
CMOSCPU.SCH	..\CMOSCPU\CMOSCPU.SCH
MEMORY.SCH	..\CMOSCPU\MEMORY.SCH
POWER.SCH	..\CMOSCPU\POWER.SCH
4BIT.SCH	..\4BIT\4BIT.SCH
FULLADD.SCH	..\4BIT\FULLADD.SCH
HALFADD.SCH	..\4BIT\HALFADD.SCH

Table 8-1. Files to be copied and their destinations.

10. When you have copied all the files to their appropriate destinations, select **CANCEL**. The **Copy File** screen is dismissed.
11. Select **Design View**. Reset the current design to CMOSCPU and then select **OK**. The main screen displays.

A simple hierarchical design

The layered arrangement created by placing worksheets inside other worksheets is called a hierarchical design. This section describes the structure of a simple, three-worksheet hierarchical design.

Sheet symbols represent other worksheets in a hierarchical design. Each sheet symbol represents a subsheet. Sheet symbols may be placed at any level of the hierarchy.

The following example is a simple hierarchy because the two sheet symbols refer to different worksheets. In complex hierarchies, any number of sheet symbols can refer to the *same* worksheet.

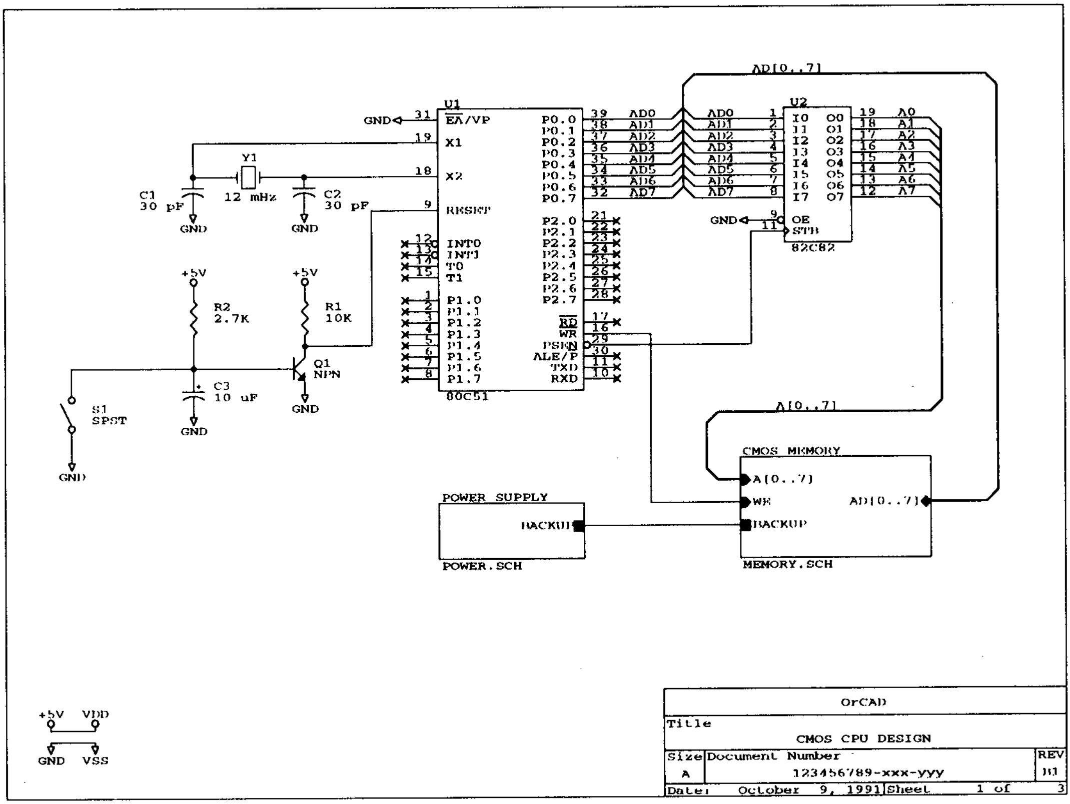


Figure 8-3. The root worksheet of the CMOS CPU design.

Libraries

The CMOS CPU design uses the following libraries:

- ANALOG2.LIB
- DEVICE.LIB
- INTEL.LIB
- MEMORY.LIB
- TTL.LIB

All of these libraries must be configured. Check the **Configured Libraries** list box in the **Library Options** area of the **Configure Schematic Design Tools** screen to determine their status. Follow these steps to configure a library:

1. From the **Schematic Design Tools** screen, select **Draft** and select **Configure Schematic Tools.** The **Configure Schematic Design Tools** screen displays.
2. Pan down to the **Library Options** area.
3. If any of the above mentioned libraries do not appear in the **Configured Libraries** list box, configure them now. Select the desired library from the **Available Libraries** list box, and then select **>Insert>**.
4. Select **OK** to save any configuration changes.

The root worksheet CMOSCPU.SCH

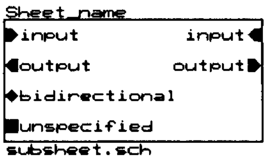
Figure 8-3 shows the root worksheet of this simple hierarchy. The design is called CMOS CPU. The root worksheet filename is CMOSCPU.SCH.

The root worksheet contains a descriptive name in the title block. A title helps identify a worksheet, but it is not required. The title is independent of the filename. A worksheet is identified by **Draft** and the operating system by the worksheet's filename.

The root worksheet of the CMOS CPU design contains:

- ❖ An 80C51 and an 82C82 part
- ❖ Discrete analog parts: a transistor, capacitors, resistors, and so on
- ❖ Two sheet symbols: POWER SUPPLY and CMOS MEMORY
- ❖ Power and ground symbols
- ❖ Wires and buses connecting the parts

Sheet symbols



Each sheet symbol has a *name* and a *filename*. The name and filename are separate. The sheet symbol name displays above the sheet symbol; the filename displays below the sheet symbol. The sheet symbol filename has to be identical to the name of the file containing the associated worksheet.

The two sheet symbols in figure 8-3 were placed on the root worksheet using the **PLACE Sheet** command. When you place a sheet symbol, **Draft** automatically assigns it a unique filename generated from the date and time of day on your computer. You can accept this unique (but not very descriptive) filename or change it to a filename of your choice. To change the filename, place the pointer inside the sheet symbol and select **EDIT Edit Filename.** The prompt "Filename?" followed by the sheet symbol's current filename displays. Enter the new filename.

In this example, the CMOS MEMORY sheet symbol was assigned the filename MEMORY.SCH, and the POWER SUPPLY sheet symbol was assigned the filename POWER.SCH. MEMORY.SCH refers to the worksheet on which the circuit's memory is located. POWER.SCH refers to the worksheet on which the system's power supply is located.

Sheet nets

The CMOS MEMORY sheet symbol contains four sheet nets: A[0..7], WE, BACKUP, and AD[0..7]. These sheet nets were placed into the sheet symbol using the **PLACE Sheet Add-NET** command. (Sheet nets are *not* module ports. See *Using sheets and parts to point to another worksheet* in *Chapter 9: Tips and Techniques* for more information.)

The A[0..7] sheet net is connected to a bus with eight members. The bus members are labeled A0 through A7.

While sheet nets do not need to have the same *prefixes* as the bus labels to which they connect ("A" in this example), the sheet nets *must* specify the same *ranges* as the bus labels to which they connect ("[0..7]").

For sheet nets and module ports, there should be no space between the prefix and suffix portions of the names.

Power objects

Power objects represent connections from the outside world to the pins in a part. Unless otherwise specified, power objects are global in scope; they connect to all other signals of the same name.

On the root worksheet a power object named +5V connects to a power object named V_{DD}. This connects the +5V supply to the V_{DD} pins of the 80C51 and the 82C82 parts. Likewise, a power object named GND connects to a power object named V_{SS}. This connects power ground to the V_{SS} pins of the 80C51 and the 82C82 parts.

Nested schematic worksheets

Once the sheet symbols for the nested logic are completed, you then create the worksheets to which these sheet symbols refer.

△ *NOTE: You don't have to create the root worksheet of the hierarchy before creating the nested worksheets; however, a top-down design methodology is recommended.*

Display the CMOS MEMORY worksheet

1. Run **Draft** and select **QUIT Enter Sheet.**
2. Place the pointer inside the CMOS MEMORY sheet symbol and select **Enter. Draft** displays the CMOS MEMORY worksheet (figure 8-4).

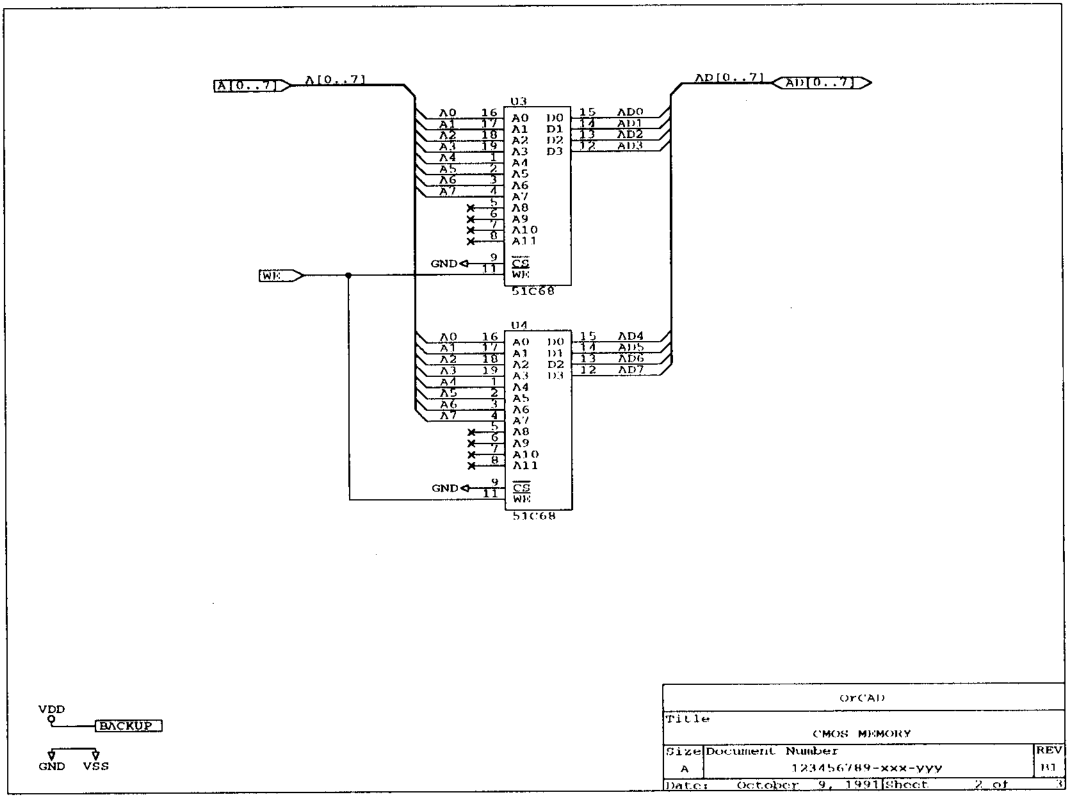


Figure 8-4. CMOS MEMORY worksheet.

The CMOS MEMORY worksheet is the schematic for the circuit's memory.

Notice the four module ports: A[0..7], AD[0..7], WE, and BACKUP. They connect to identically named sheet nets located in the CMOS MEMORY sheet symbol on the root worksheet.

Notice the buses. Buses are automatically connected to module ports with labels having the same name and range—module port A[0..7] connects to a bus labeled A[0..7].

Finally, notice the power objects. The V_{DD} power object connects to the module port named BACKUP. This isolates the power in the worksheet from any other V_{DD} power objects in the design.

The GND power object connects to the V_{SS} power object. This shorts the GND power net and the V_{SS} power net together. This allows any pin with a GND symbol to be connected to the V_{SS} power net.

When you are finished reviewing the CMOS MEMORY worksheet, return to the root worksheet by selecting **Leave** from the **QUIT Enter Sheet** command line (if it is displayed at the top of the screen) or by selecting the **QUIT Leave Sheet** command.

Display the POWER SUPPLY worksheet

1. If the **QUIT Enter Sheet** command line is not displayed at the top of the screen, select the **QUIT Enter Sheet** command.
2. Place the pointer inside the POWER SUPPLY sheet symbol and select **Enter**. Figure 8-5 shows the POWER SUPPLY worksheet.

The POWER SUPPLY worksheet is the schematic for the power supply circuitry.

Notice the module port named BACKUP. The BACKUP module port makes the *logical* connection to the sheet net named BACKUP. The BACKUP sheet net is in the POWER SUPPLY sheet symbol on the root worksheet of the CMOS CPU design (see figure 8-3).

The BACKUP module port makes the *electrical* connection to the BACKUP module port on the CMOS MEMORY worksheet (see figure 8-4).

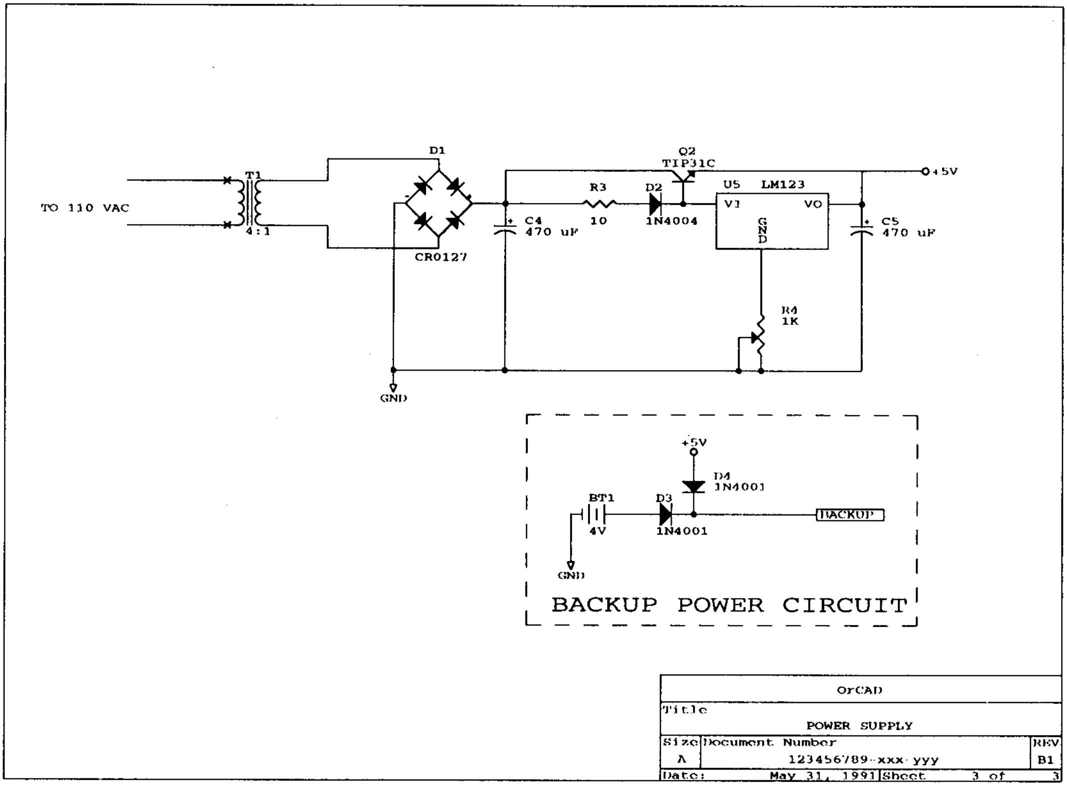


Figure 8-5. POWER SUPPLY worksheet.

When you are finished reviewing the POWER SUPPLY worksheet, return to the root worksheet by selecting **Leave** from the **QUIT Enter Sheet** command line, and then pressing <Esc> to dismiss the command line. (If the **QUIT Enter Sheet** command line is not displayed at the top of the screen, select the **QUIT Leave Sheet** command.) Select **QUIT Abandon Edits** to return to the **Schematic Design Tools** screen.

To complete a simple hierarchical design, you will use a variety of tools. The next sections in this chapter discuss the following tools: **Annotate Schematic, Check Electrical Rules, Show Design Structure,** and **Create Bill of Materials.**

Using Annotate Schematic on a simple hierarchy

Once a design is complete, you run **Annotate Schematic** to assign unique values to the part reference designators.

Follow these steps to annotate the simple hierarchy represented by the worksheet, CMOSCPU.SCH:

1. Select **Annotate Schematic** on the **Schematic Design Tools** screen. The menu at right displays.

```
Annotate Schematic
Execute
Local Configuration
Assign Hot Key
Show Version
Configure Schematic Tools
Help
```

2. Select **Local Configuration** and then **Configure ANNOTATE**. The **Configure Annotate Schematic** screen displays.
3. Check to make sure that the **Source** entry box contains the filename CMOSCPU.SCH. If not, enter **CMOSCPU.SCH** in the **Source** entry box.
4. Check to make sure that **Source file is the root of the design** is selected.
5. Select **OK** to save the configuration.
6. Select **Annotate Schematic** and then select **Execute.**

   As it processes, **Annotate Schematic** scrolls status messages three lines at a time in the monitor box at the bottom of the **Schematic Design Tools** screen.

When **Annotate Schematic** is done, the reference designators for each part in the worksheet have new, unique values. You may want to use **Draft** to view the new reference designator values on the CMOS CPU worksheet.

Using Check Electrical Rules on a simple hierarchy

After the worksheets are annotated, the design should be checked for electrical rule violations. **Check Electrical Rules** checks for several problems associated with a design, including open input pins, shorts, and bus contention.

Run **Check Electrical Rules** to check for any electrical rules violations in the design. For instructions, refer to the earlier discussion of how to run **Check Electrical Rules.** After you run **Check Electrical Rules,** you may review the error report with **Edit File.**

Figure 8-6 shows how the error report looks for the CMOSCPU.SCH design.

Warnings

Check Electrical Rules flags certain conditions possibly overlooked when your design was created. These *WARNINGS* are not critical errors. The warnings in this example are acceptable, because the power supplies were intentionally connected in the design.

Errors

Normally, if **Check Electrical Rules** reports *ERRORS* in a design, you should correct them before running other tools. In this example, however, all warnings are acceptable and other tools may be run.

```
CMOSCPU.SCH

Electrical Rules Check Report
CMOS CPU DESIGN  Revised:    October  9, 1991
123456789-xxx-yyy          Revision: B1
OrCAD




WARNING: POWER Supplies are CONNECTED GND <-> VSS
WARNING: POWER Supplies are CONNECTED VDD <-> +5V

MEMORY.SCH

Electrical Rules Check Report
CMOS MEMORY  Revised:   October  9, 1991
123456789-xxx-yyy          Revision: B1
OrCAD




WARNING: POWER Supplies are CONNECTED VSS <-> GND

POWER.SCH

Electrical Rules Check Report
POWER SUPPLY  Revised:       May 31, 1991
123456789-xxx-yyy          Revision: B1
OrCAD
```

*Figure 8-6. The error report produced by **Check Electrical Rules** for* CMOSCPU.SCH.

Using Show Design Structure on a simple hierarchy

Use **Show Design Structure** to obtain a text file listing the worksheets in a hierarchy. This tool is helpful for organizing a hierarchy containing many worksheets. Follow these steps to run **Show Design Structure:**

1. Select **Show Design Structure** on the **Schematic Design Tools** screen, and then select **Local Configuration** and **Configure TREELIST.** The **Configure Show Design Structure** screen displays.
2. Check to make sure that the **Source** entry box contains the filename CMOSCPU.SCH.
3. Check to make sure that the **Destination** entry box contains the filename CMOSCPU.TWG.

   Show Design Structure is now configured to create a schematic structure list for CMOSCPU.SCH and save the results in a text file, CMOSCPU.TWG.
4. Select **OK** to save any changes to the configuration.
5. Select **Show Design Structure** and then select **Execute.**

   As it processes, **Show Design Structure** scrolls status messages three lines at a time in the monitor box at the bottom of the **Schematic Design Tools** screen.

   Use **Edit File** to examine the CMOSCPU.TWG text file. The figure below shows the information stored in CMOSCPU.TWG.

```
<<<ROOT>>>
[CMOSCPU.SCH]    October  9, 1991
  CMOS MEMORY
  [MEMORY.SCH]    October  9, 1991
  POWER SUPPLY
  [POWER.SCH]        May 31, 1991
```

All worksheet filenames are enclosed within brackets, as in [*filename*]. Next to the filename is the date the worksheet was last modified. **Show Design Structure** lists sheet symbol names above the filenames of the worksheets to which they refer.

In this example, the root worksheet filename is CMOSCPU.SCH. Below the root worksheet filename are sheet symbol names and the filenames of the worksheets to which they refer. The sheet symbol named CMOS MEMORY refers to the worksheet with the filename MEMORY.SCH. The sheet symbol named POWER SUPPLY refers to the worksheet with the filename POWER.SCH.

Using Create Bill of Materials on a simple hierarchy

Create Bill of Materials creates a list of parts for all types of design structures. In this example, **Create Bill of Materials** is used on the simple hierarchy, CMOSCPU.SCH. Follow these steps to run **Create Bill of Materials:**

1. Select **Create Bill of Materials** on the **Schematic Design Tools** screen and then select **Local Configuration** and **Configure PARTLIST**. The **Configure Create Bill of Materials** screen displays.
2. Check to make sure that the **Source** entry box contains the filename CMOSCPU.SCH.
3. Check to make sure that **Source file is the root of the design** is selected.
4. Check to make sure that the **Destination** entry box contains the filename CMOSCPU.BOM.
5. Select **OK** to save any changes to the configuration.
6. Select **Create Bill of Materials** and then select **Execute.**

   As it processes, **Create Bill of Materials** scrolls status messages three lines at a time in the monitor box at the bottom of the **Schematic Design Tools** screen.

   To examine the text file created by **Create Bill of Materials,** use **Edit File.** Figure 8-7 shows the information stored in the text file CMOSCPU.BOM.

```
CMOS CPU DESIGN                                   Revised:   October  9, 1991
123456789-xxx-yyy                                 Revision: B1
Bill Of Materials                     June   12, 1992     11:40:57    Page   1

 Item  Quantity  Reference                 Part
___________________________________________________
   1       1      BT1                      4V
   2       2      C1,C2                    30 pF
   3       1      C3                       10 uF
   4       2      C4,C5                    470 uF
   5       1      D1                       CR0127
   6       1      D2                       1N4004
   7       2      D3,D4                    1N4001
   8       1      Q1                       NPN
   9       1      Q2                       TIP31C
  10       1      R1                       10K
  11       1      R2                       2.7K
  12       1      R3                       10
  13       1      R4                       1K
  14       1      S1                       SPST
  15       1      T1                       4:1
  16       1      U1                       80C51
  17       1      U2                       82C82
  18       2      U3,U4                    51C68
  19       1      U5                       LM123
  20       1      Y1                       12 mHz
```

Figure 8-7. Bill of materials for CMOSCPU.SCH.

A complex hierarchical design

This section describes a three-sheet complex hierarchy. Complex hierarchies are very useful when designing common logic blocks that are repeated. Figure 8-8 shows the root worksheet for the 4-bit adder design.

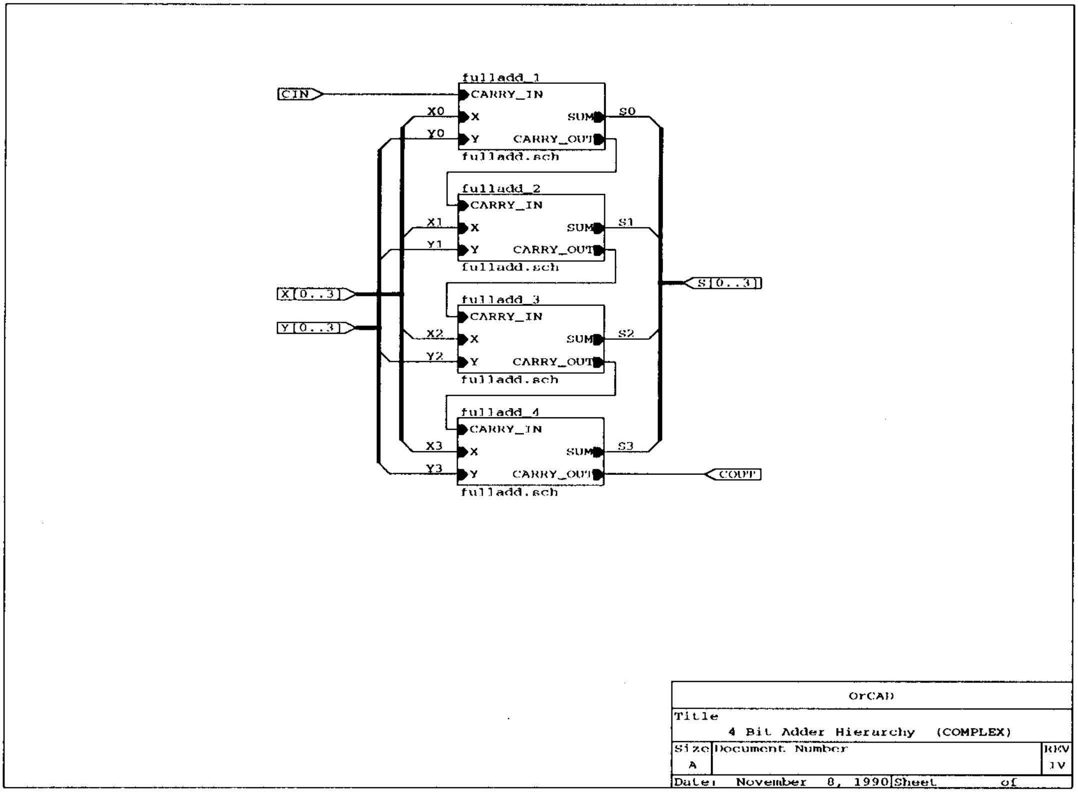


Figure 8-8. The root worksheet of the 4-bit adder design.

The 4-bit adder root worksheet

1. On the **Schematic Design Tools** screen, click the title bar or any place that is not a button, and then select **Design Management Tools.** The **Design View** portion of the **Design Management Tools** screen displays.
2. Select **4BIT** from the **Design** list box.
3. Select **OK.** The main screen displays.
4. Select **Schematic Design Tools** and then select **Execute.** The **Schematic Design Tools** screen displays.
5. Select **Draft** and then select **Configure Schematic Tools.** The **Configure Schematic Design Tools** screen displays.
6. Go to the **Library Options** area, configure TTL.LIB, and select **OK.**
7. Select **Draft** and then select **Execute.** The 4BIT root worksheet displays (figure 8-8).

The 4-bit adder design is a three-sheet complex hierarchy. The root worksheet contains four identical sheet symbols: fulladd_1, fulladd_2, fulladd_3, and fulladd_4.

These sheet symbols refer to four identical full adders. Because they are identical, it is not necessary to have a separate worksheet for each one, but it is necessary to assign the full-adder worksheet's filename to all four sheet symbols. Notice that all of the sheet symbols on the root worksheet have the filename FULLADD.SCH.

The full-adder worksheet

Follow these steps to display the full-adder worksheet.

1. Select **QUIT Enter Sheet.**
2. Place the pointer inside one of the sheet symbols and select **Enter. Draft** displays the worksheet referred to by the selected sheet symbol. Figure 8-9 shows the full-adder worksheet.

The full-adder worksheet contains two sheet symbols: halfadd_A and halfadd_B. These sheet symbols refer to two identical half adders. Because they are identical, it is not necessary to create a separate worksheet for each. Just as the four full-adder sheet symbols in the root worksheet refer to the full-adder worksheet for their logic, the two half-adder sheet symbols refer to a single half-adder worksheet for *their* logic.

Each module port in the FULLADD.SCH worksheet is named to connect to the sheet nets in the 4BIT.SCH worksheet, one level up in the hierarchy.

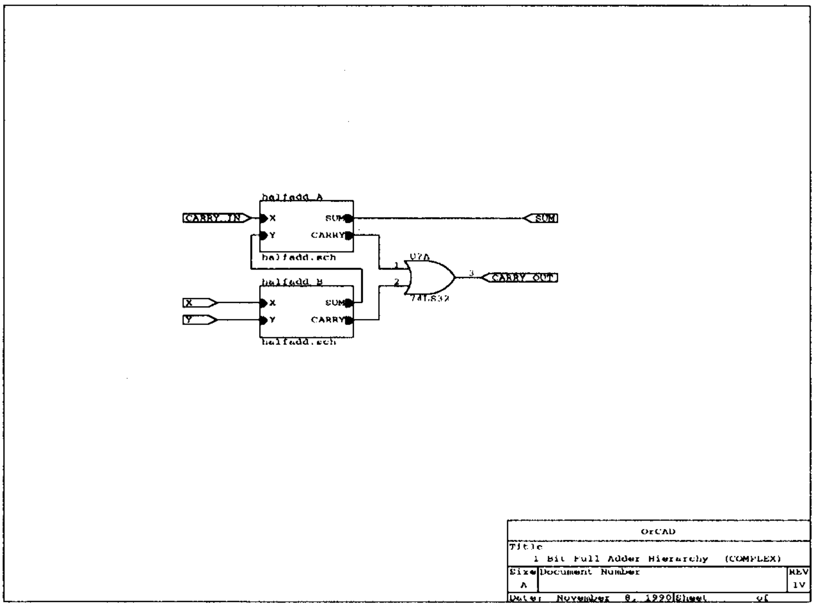


Figure 8-9. Full-adder worksheet.

The half-adder worksheet

Follow these steps to display the half-adder worksheet:

1. If the **QUIT Enter Sheet** command line is not displayed at the top of the screen, select the **QUIT Enter Sheet** command.
2. Place the pointer inside one of the sheet symbols and select **Enter. Draft** displays the worksheet referred to by the selected sheet symbol. Figure 8-10 shows the half-adder circuit.

Each module port in the HALFADD.SCH worksheet is named to connect to the sheet nets in the FULLADD.SCH worksheet, one level up in the hierarchy.

When you are finished reviewing the half-adder worksheet, return to the root worksheet by selecting **Leave** from the **QUIT Enter Sheet** command line, and then pressing <Esc> to dismiss the command line. (If the **QUIT Enter Sheet** command line is not displayed at the top of the screen, select the **QUIT Leave Sheet** command.) Select **QUIT Abandon Edits** to return to the **Schematic Design Tools** screen.

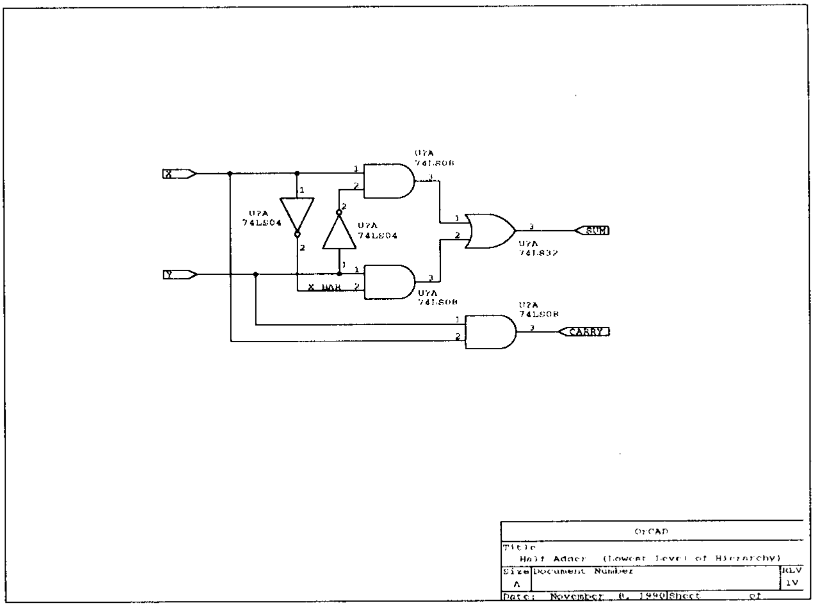


Figure 8-10. Half-adder worksheet.

Using Show Design Structure on a complex hierarchy

Use **Show Design Structure** to obtain a text file listing the worksheets in a hierarchy. This tool is helpful for organizing a hierarchy containing many worksheets.

Follow the steps given in *Using Show Design Structure on a simple hierarchy* earlier in this chapter, but substitute the filename 4BIT for CMOSCPU.

Use **Edit File** to examine the 4BIT.TWG text file. The figure below shows the information stored in 4BIT.TWG:

```
<<<ROOT>>>
[4BIT.SCH]   November  8, 1990
  fulladd_1
  [FULLADD.SCH]   November  8, 1990
    halfadd_A
    [HALFADD.SCH]   November  8, 1990
    halfadd_B
    [HALFADD.SCH]   November  8, 1990
  fulladd_2
  [FULLADD.SCH]   November  8, 1990
    halfadd_A
    [HALFADD.SCH]   November  8, 1990
    halfadd_B
    [HALFADD.SCH]   November  8, 1990
  fulladd_3
  [FULLADD.SCH]   November  8, 1990
    halfadd_A
    [HALFADD.SCH]   November  8, 1990
    halfadd_B
    [HALFADD.SCH]   November  8, 1990
  fulladd_4
  [FULLADD.SCH]   November  8, 1990
    halfadd_A
    [HALFADD.SCH]   November  8, 1990
    halfadd_B
    [HALFADD.SCH]   November  8, 1990
```

Notice that there are a number of references in this report to FULLADD.SCH and HALFADD.SCH. The 4-bit adder design, a complex hierarchy of only three worksheets, expands to thirteen worksheet references. Again, the advantage of complex hierarchical design structures is that, during the design phase, all of the repeated logic needs to be drawn only once.

Converting a complex hierarchy to a simple hierarchy

While a complex hierarchy is very useful in the design and simulation phase, it is not practical for some aspects of the design cycle. This is especially true when a design is to be turned into a printed circuit board. All of the design must then be simplified—converted to a simple hierarchy. This is necessary because all of the parts in the design must be assigned unique reference designators. It would be quite difficult to have a number of parts labeled U17 on the board and have to refer to the complex hierarchical schematic to determine which was which.

Design Management Tools includes the tool **Complex to Simple.** This tool creates a new design and builds a new version of the complex hierarchy, a version in which each sheet symbol refers to a unique file.

Follow these steps to run **Complex to Simple**:

1. On the **Schematic Design Tools** screen, click the title bar or any place that is not a button, and then select **Design Management Tools.** The **Design View** portion of the **Design Management Tools** screen displays.
2. Select **Complex to Simple.** The **Complex to Simple** screen displays.
3. Check to make sure that the **Source sheet** entry box contains the name 4BIT.
4. Enter `S4BIT` in the **Destination design** entry box.
5. Select **OK. Design Management Tools** builds the new design directory and converts the 4BIT design to S4BIT. As it processes, **Design Management Tools** displays "Working . ." and several messages at the top left corner of the screen. Then, **Design Management Tools** scrolls status messages three lines at a time in the monitor box at the bottom of the **Schematic Design Tools** screen.
6. Select **Cancel** when the process is complete.
7. Notice that S4BIT is now the current design. Select **OK.** The main screen displays.

Viewing the S4BIT design

Select **Draft** and then **Execute.** Notice that the four occurrences of the filename FULLADD.SCH are changed to FULLADD.SCH, FULLADDA.SCH, FULLADDB.SCH, and FULLADDC.SCH. Also notice that the eight occurrences of the filename HALFADD.SCH are changed to HALFADD.SCH and HALFADDA.SCH through HALFADDG.SCH.

Running Annotate Schematic on the S4BIT design

As with any new design, you should run **Annotate Schematic** before running the other tools. Other tools report information about the worksheet file, and if you run **Annotate Schematic** first, you ensure that reported information includes the updated reference designators.

The following figures show the simplified design after it is annotated using **Annotate Schematic.**

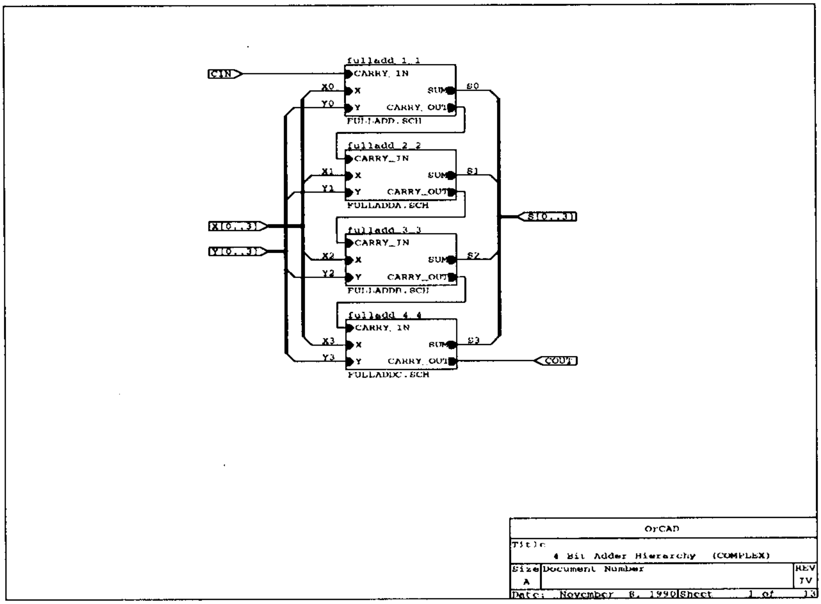


Figure 8-11. 4BIT.SCH schematic.

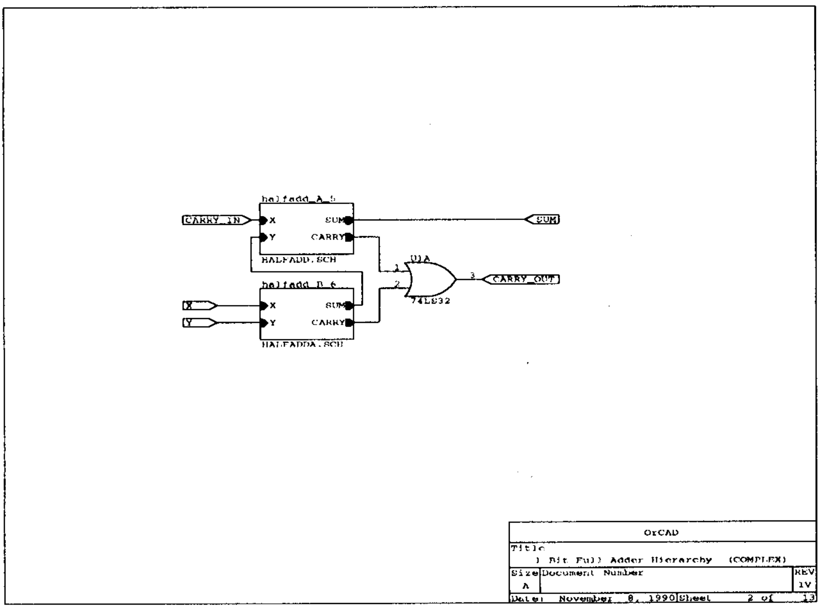


Figure 8-12. FULLADD.SCH schematic.

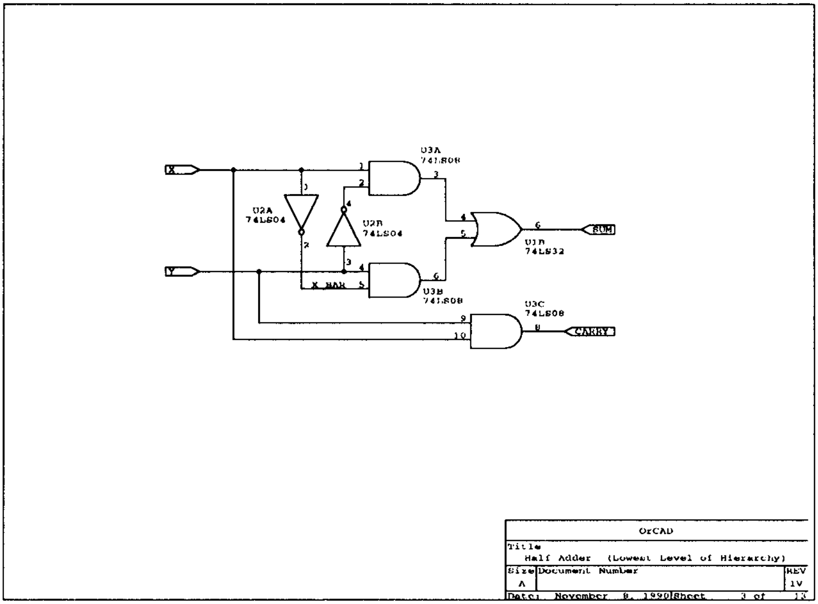


Figure 8-13. HALFADD.SCH schematic.

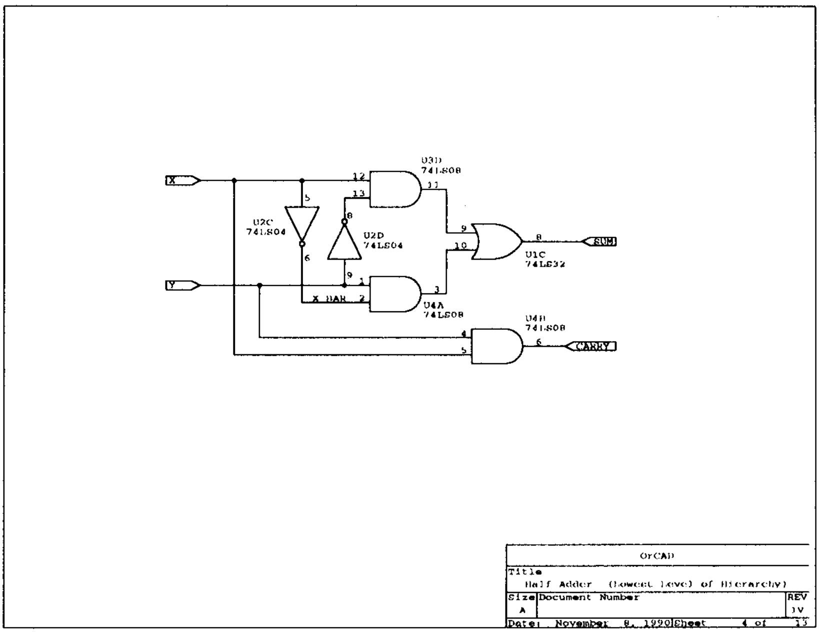


Figure 8-14. HALFADDA.SCH schematic.

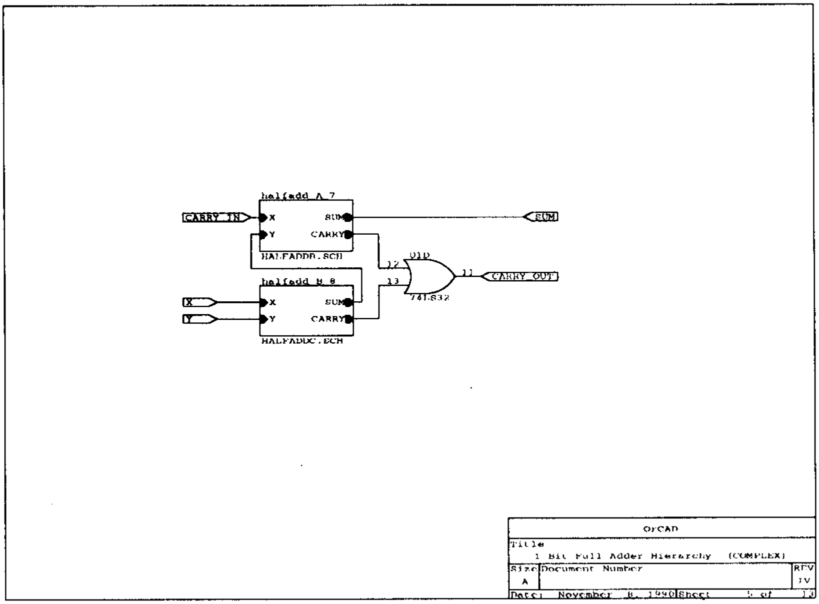


Figure 8-15. FULLADDA.SCH schematic.

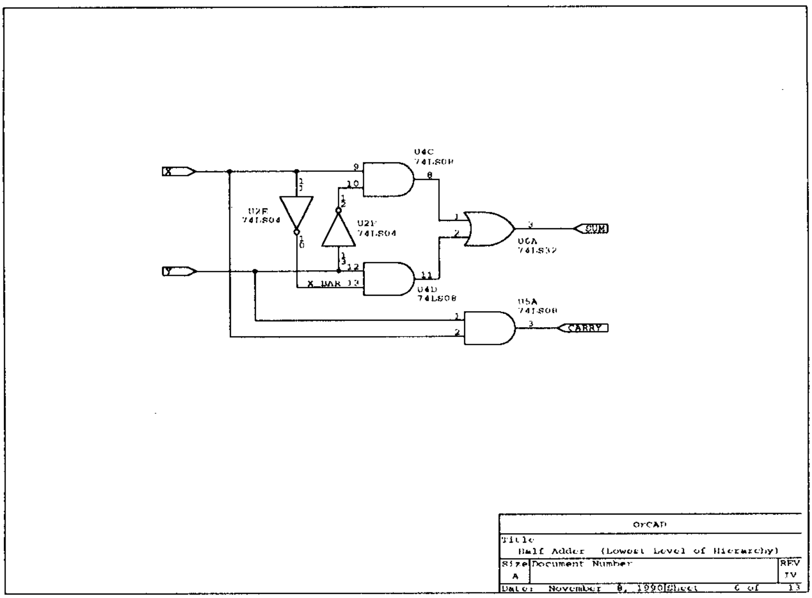


Figure 8-16. HALFADDB.SCH schematic.

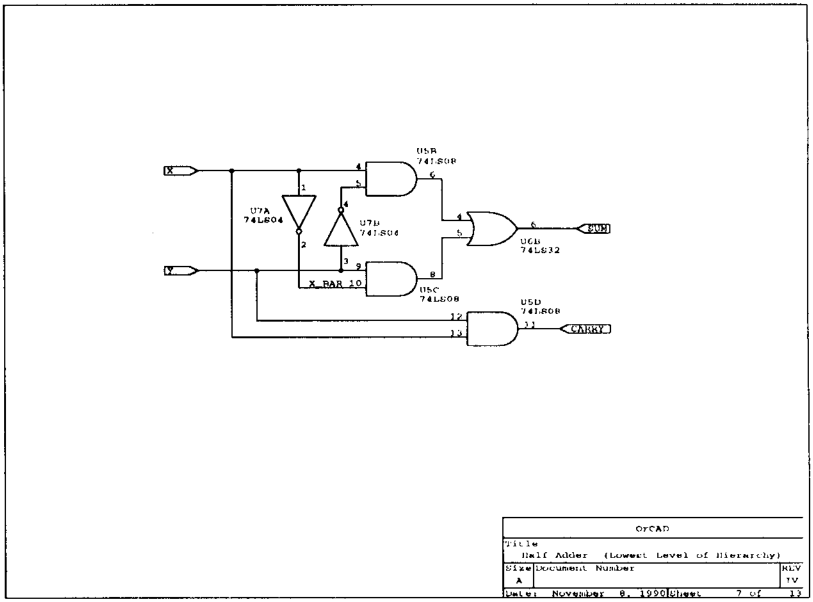


Figure 8-17. HALFADDC.SCH schematic.

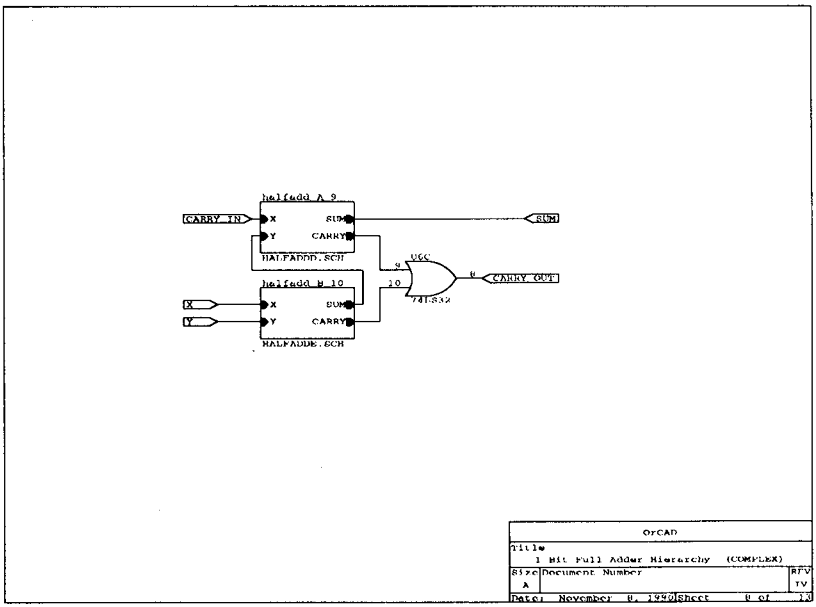


Figure 8-18. FULLADDB.SCH schematic.

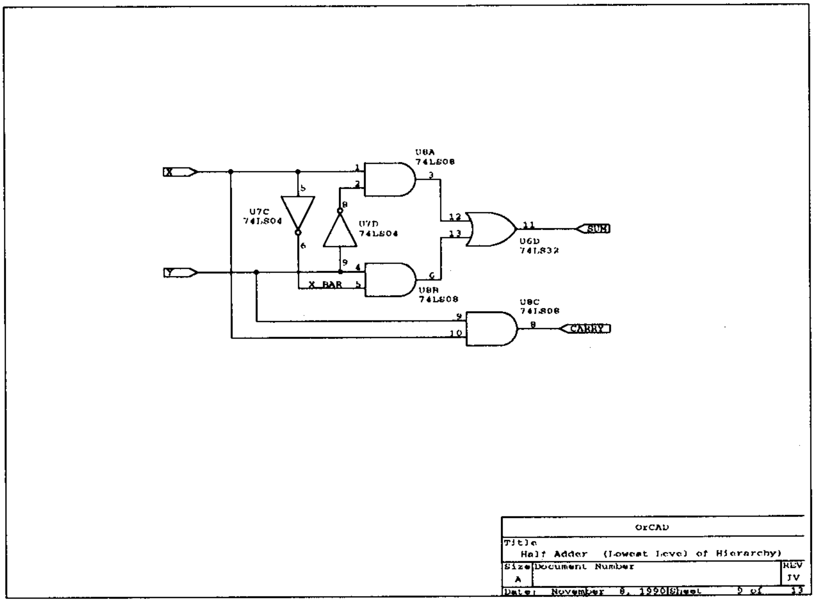


Figure 8-19. HALFADDD.SCH schematic.

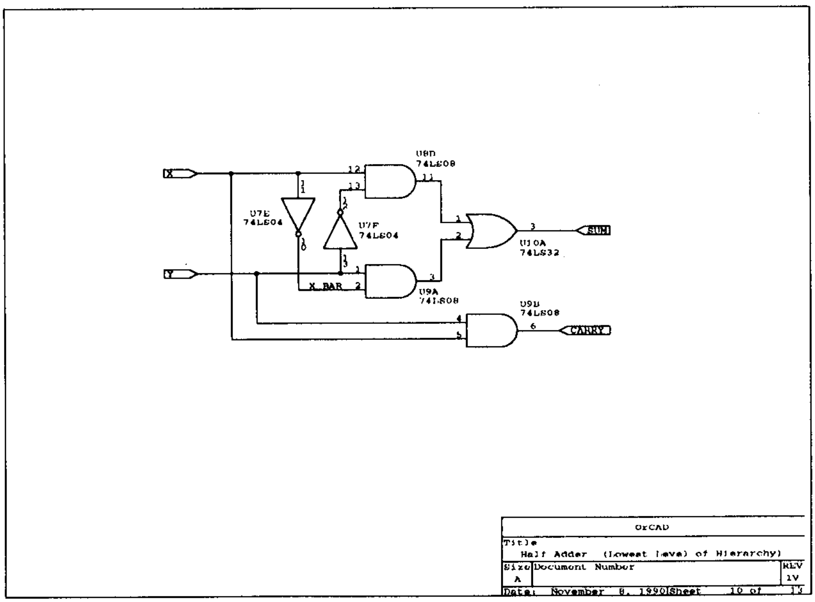


Figure 8-20. HALFADDE.SCH schematic.

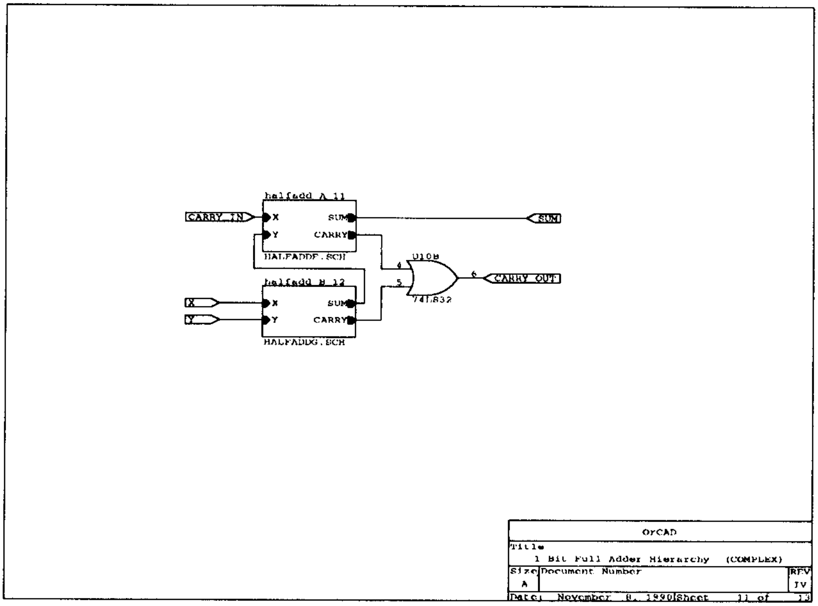


Figure 8-21. FULLADDC.SCH schematic.

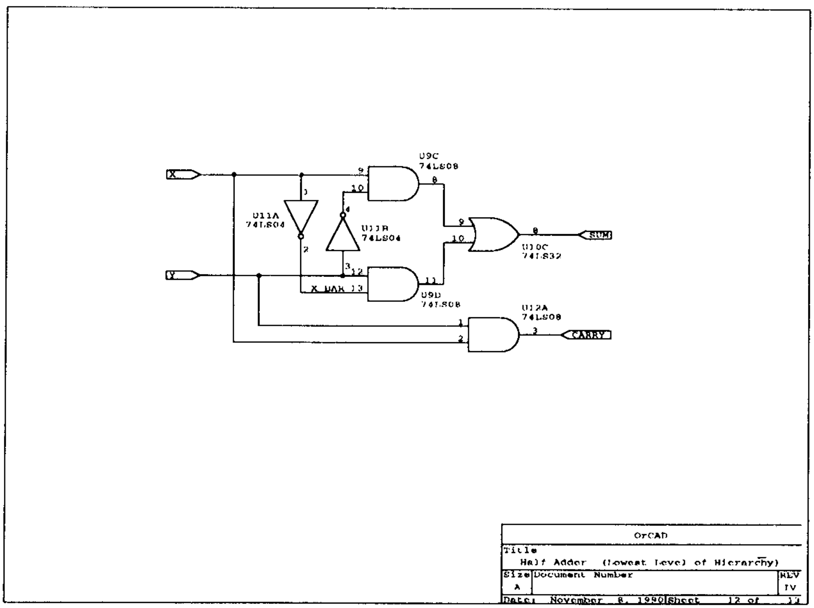


Figure 8-22. HALFADDF.SCH schematic.

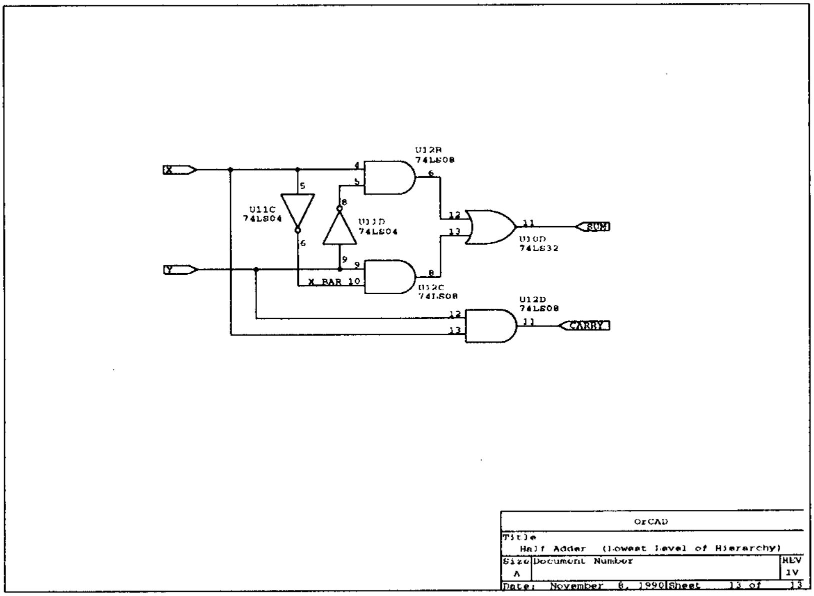


Figure 8-23. HALFADDG.SCH schematic.

Tips and techniques

This chapter is a collection of tips and techniques to help you better use **Schematic Design Tools 386+**. It includes information about the title block, nonconnective objects, and other topics.

Unlike the other chapters in this guide, this chapter is not tutorial in nature. For additional information about any of the commands, menus, or options described in this chapter, see the chapter for the corresponding tool in the *Schematic Design Tools 386+ Reference Guide.*

Converting complex hierarchies

When you use **Complex to Simple** to convert a complex hierarchy that contains any sheetpath parts, be sure not to use the same filename for schematics in the design directory and schematics in the library directory. If you do, **Complex to Simple** uses the schematic in the design directory instead of the schematic in the library directory.

When **Complex to Simple** is copying and naming schematics used more than once in the design, it checks the design directory for duplicate filenames before naming new files. For example, if a complex design includes schematics named INPUT.SCH and INPUTA.SCH, each of which is referred to twice, the simplified design contains these four files:

- INPUT.SCH
- INPUTA.SCH
- INPUTAA.SCH
- INPUTAB.SCH

Title block tips

Schematic Design Tools 386+ provides a great deal of flexibility with the title block on a worksheet. You can use **Draft**'s standard title block, an ANSI title block, or a custom title block you create. If your paper has a preprinted title block, you can use the **SET Title Block** command of **Draft** to not plot the title block. You would then place text to line up with your preprinted title block. This section describes ways that title blocks can be manipulated.

OrCAD's title block

The **Schematic Design Tools 386+** schematic editor, **Draft,** creates a title block that looks like the one shown in figure 9-1. You define all of the information on the title block except the date and size. **Draft** automatically enters the size and modification date of the worksheet.

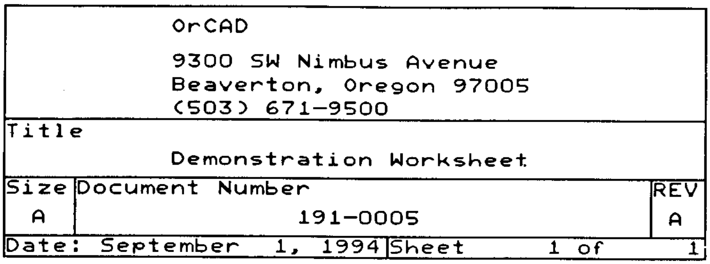


Figure 9-1. Sample OrCAD title block.

ANSI title block

You can configure **Schematic Design Tools** so that **Draft** creates an ANSI title block like the one shown in figure 9-2.

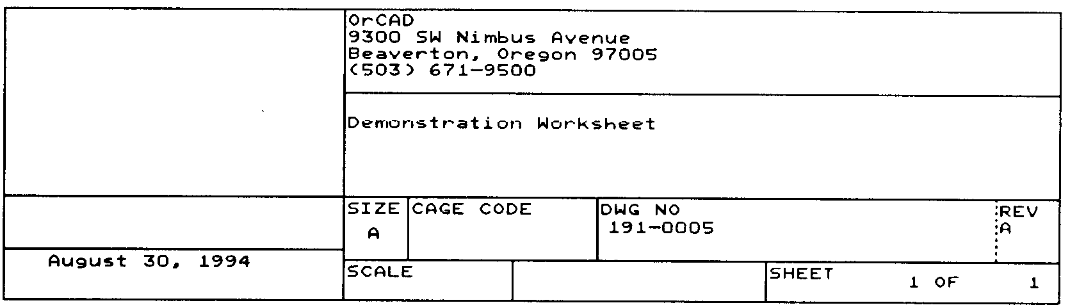


Figure 9-2. Sample ANSI title block.

The ANSI title block conforms to the guidelines given in ANSI Standard Y14.1-1980. As you can see in figure 9-2, the ANSI title block is larger than the default OrCAD title block. On an A-size drawing, it takes up a large amount of the drawing area.

See *On the Configure Schematic Design Tools screen* in this section for instructions on how to create an ANSI title block.

△ *NOTE: If you use an ANSI title block, you may want your worksheet to have ANSI standard dimensions. These dimensions are given in tables 1-2 and 1-6 in the* Schematic Design Tools 386+ Reference Guide. *Since most, if not all, PC-compatible printers and plotters cannot print as close to the edge of the page as specified in the ANSI standard, OrCAD's default worksheet dimensions are reduced. These reduced dimensions are given in tables 1-4 and 1-5 in the* Schematic Design Tools 386+ Reference Guide. *If your printer or plotter can print closer to the edge of the paper, adjust the worksheet size in the* ***Template Table*** *area of the* ***Configure Schematic Design Tools*** *screen.*

Defining title block information

You can define title block information either in **Draft** or on the **Configure Schematic Design Tools** screen.

Extended ASCII characters

You can use extended ASCII characters in all title block fields except **Sheet Size**. To type extended ASCII characters, make sure that the NumLock function on the keyboard is enabled, hold down <Alt> and type the appropriate number on the numeric keypad. Typing numbers on the keyboard will not produce characters.

▲ *CAUTION: These extended ASCII characters are not permitted: 128 (null), 141 (carriage return), and 167 (single quote). If you use one of these extended ASCII characters, you can expect unpredictable results when you try to open your worksheet.*

In Draft

To define title block information in **Draft**, place the pointer in the title block and select **EDIT**. The menu shown at right displays. Select the field to edit and answer the prompts that display.

```
Edit title block
Revision code
Title of sheet
Document number
Sheet number
Number of sheets
Organization name
1st Address Line
2nd Address Line
3rd Address Line
4th Address Line
```

For more information, see the **EDIT** command description in *Chapter 2: Draft*.

△ *NOTE: If you are using an ANSI title block (figure 9-2), you must use the **PLACE Text** command to place text in the **CAGE CODE** and **SCALE** boxes. The **EDIT** command does not contain menu items to edit this information.*

On the Configure Schematic Design Tools screen

To define title block information, display the **Configure Schematic Tools** screen and pan to the **Worksheet Options** area (figure 9-3).

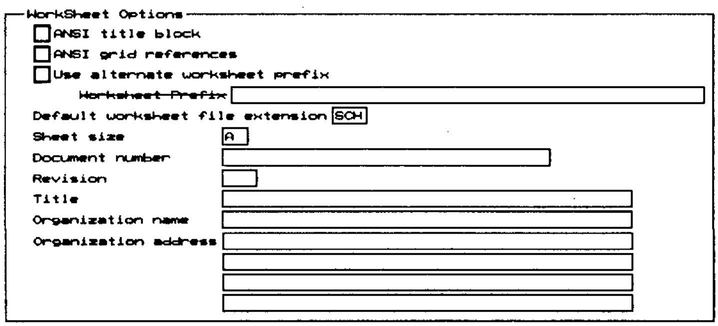


*Figure 9-3. **Worksheet Options** area of the **Configure Schematic Design Tools** screen.*

Information entered here automatically displays in the title block of each schematic created after the information is defined.

To use an ANSI title block (pictured in figure 9-2), select the **ANSI title block** option on this screen.

See *Worksheet Options* in chapter 1 for more information.

➤ *Helpful hint . . .*

Consider defining title block information such as organization name and address on the **Configure Schematic Design Tools** screen in your TEMPLATE directory. That way, each new design you create will be set up with your company name and address.

Suppressing title block elements

You can suppress the title block's lines, text, or both, as described here.

Lines

To suppress title block lines and leave title block text on the worksheet, display the **Configure Schematic Design Tools** screen. Pan down to the **Color and Pen Plotter Table** area.

Click in the **Pen** entry box to the right of **Title Block**. Enter `99` to tell **Plot Schematic** not to plot the title block.

When you open a worksheet in **Draft**, notice that the title block lines still display. However, they do not appear on the plot, as shown in figure 9-4.

```
                    OrCAD

                    9300  SW  Nimbus  Avenue
                    Beaverton  Oregon  97008
                    (503)  671-9500
Title
                    Demonstration  Worksheet
Size     Document   Number                                REV
  A                            191-0005                    A
Date:        September 1, 1994         Sheet    1    of    1
```

Figure 9-4. Plot of a title block with lines suppressed.

△ *NOTE: This also turns off the border around the drawing area during printing.*

Text

To suppress title block text and leave title block lines on the worksheet, display the **Configure Schematic Design Tools** screen. Pan down to the **Color and Pen Plotter Table** area.

Click in the **Pen** box to the right of **Title Text**. Enter **99** to tell **Plot Schematic** not to plot the title block's text.

When you open a worksheet in **Draft**, notice that the title block text still displays. However, it does not appear on the plot, as shown in figure 9-5.

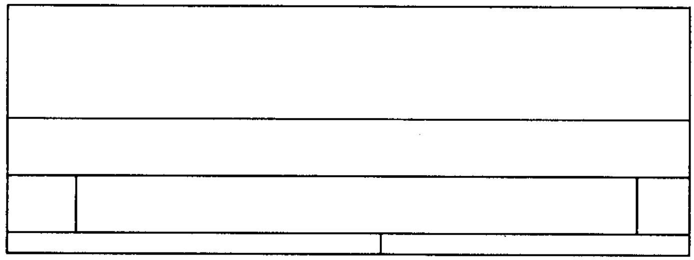


Figure 9-5. Plot of a title block with text suppressed.

Lines and text

There are two ways to suppress both title block lines and text:

- Use **Draft's SET Title Block No** command. If you use this method, the title block and its text do not display on the screen or appear on a print or a plot.
- Set both the title block and title text to a pen of 99 on the **Configure Schematic Design Tools** screen. If you use this method, the title block and its text plus the border around the drawing area all display on the screen, but do not appear on a plot.

➤ *Helpful hint . . .*

If you are plotting on paper that has your title block preprinted on it, suppress the title block and its text as described previously. Use **Draft's PLACE Text** command to place text in the correct position so that when you plot your schematic the text prints in the correct place in your title block.

Creating a custom title block

You can create a custom title block using a library part made to look like a title block, or by using wires without labels.

Using a library part

To create a library part that looks like a title block, use **Edit Library** to make a library part that looks like a title block. Be sure that the part is nonconnective. This means it must be a zero-element part, and cannot have any pins. For more information, see *Nonconnective objects* in this chapter.

In **Draft**, suppress the title block using the **SET Title Block No** command, as described previously.

Place the library part on each new schematic.

Using wires

To use wires to create a title block, suppress the title block using **Draft's SET Title Block No** command, as described previously.

Draw the title block using the **PLACE Wire** command.

If text is required, be sure to place text, not labels. The netlist tools ignore the wires only if they have no labels and no pins attached.

➤ *Helpful hints . . .*

If you would like wider lines in your title block, draw buses rather than wires.

You may want to use a combination of both of the methods described above. For example, you can create a logo as a library part, then draw the title block with wires and place the logo in your title block.

Using your custom title block in each design

Once you create a custom title block, you can easily duplicate it on each new design using one of the methods described here.

Create a template schematic

Use one of the methods described in *Creating a custom title block* to create a worksheet that contains only the title block. Keep this worksheet in the TEMPLATE directory. It will be copied into each new design. If it has a name of TEMPLATE.SCH, it will always be given the name of the new design with an extension of .SCH.

Create a macro

Create a macro that draws the title block with wires. This macro can also include any text that must be part of the title block.

Place the macro in a macro file in your TEMPLATE directory. It will become a part of each new design. Run the macro in each new design.

Archiving parts

To achieve more efficient use of memory, run **Archive Parts in Schematic** on each design, turning both LIBARCH and COMPOSER on. This creates a library containing only the parts used in your design.

Doing this protects the design from changes in standard libraries, simplifies moving the design, and results in efficient memory use because you configure only one library instead of several.

Nonconnective objects

You may want a design to contain objects that have no electrical connectivity and are not processed by **Create Netlist** or **Create Hierarchical Netlist**. These objects can be:

- Mounting holes.
- Mechanical hardware such as screws and washers.
- A physical representation of the device you are designing.
- Floating or unconnected pins that can connect to an option, such as unused pins on a serial port.
- Flowchart symbols.

Libraries

Schematic Design Tools 386+ includes special libraries containing nonconnective objects that can be used to customize your schematic worksheet but are not included in the netlist.

These nonconnective objects have no reference designators, values, or pins. They are found in the libraries listed in table 9-1.

Library	*Contents*
ASSEMBLY.LIB	Part outlines for assembly drawings to specify position of devices for board placement.
FLOWCHT.LIB	Programming flowchart symbols.
SHAPES.LIB	Generic library containing circles, squares, 90° arcs, and diamonds.

Table 9-1. Libraries that contain nonconnective parts.

See technical note #30: *Nonconnective objects in Schematic Design Tools* for illustrations of some of the parts found in these libraries.

Placing nonconnective objects on your schematic

Follow the steps below to place nonconnective objects on your worksheet:

1. Use **Edit Library** to create an object that looks like an actual device but has no electrical connectivity. You can use objects from any of the libraries listed in the table above. Use the objects as they are, or use them to create a new object.

   To be nonconnective, an object must be a zero-element part and cannot have any pins. Since IEEE parts are always single-element parts, they cannot be used. Only block and graphic parts can be nonconnective.

2. Place the part in a custom library. You can use it whenever you need it.

Uppercase letters in key fields

Lowercase letters entered in the **Key Fields** area of the **Configure Schematic Design Tools** screen are handled as literals. To use the special characters "V" for **Value** and "R" for **Reference** and have them interpreted correctly, enter them in uppercase.

Duplicate sheet names

Sheet symbols in a design must have unique names. If your design has two sheet symbols with the same name, you receive this message when you run **Create Netlist:**

```
Duplicate Sheet Names
```

For example, two sheet symbols named `SHEET` cause this message to display, but two sheet symbols named `SHEET1` and `SHEET2` do not.

Changing netlist formats

Create Netlist and **Create Hierarchical Netlist** create netlists incrementally. When you run a netlist, only the items that have *changed* are included. A new file date or time denotes change. If you configure your netlist for one format, run a netlist, then configure it for a different format and run a netlist again, the netlist does not change.

To produce a netlist with a new format from an unchanged design, run IFORM with the **Force IFORM to always create a formatted netlist** option selected.

Reporting unused match strings

When printing a bill of materials report using **Create Bill of Materials** and an include file, you may want to find out which strings in your include file do not have a corresponding match string in the design.

To do this, make the following settings on the local configuration screen for **Create Bill of Materials:**

- Specify your include file by selecting the **Merge an include file with report** option and entering the include file name in the **Include** entry box.
- Select the **Report unused match strings in include file** option.

Copying parts from one library to another

Follow these steps to copy parts from one library to another:

1. Use **Edit Library**'s **EXPORT** command to copy a part from the original library to a temporary text file.
2. Run **Edit Library** on the target library and **IMPORT** the temporary file.
3. Edit the part if necessary.
4. Select **LIBRARY Update Current.**
5. Select **QUIT Update File.**

Encapsulated PostScript

This section describes how to take an EPS file from an OrCAD application into two widely-used Microsoft programs, Word for Windows (WINWORD) version 2.0 and Word 5.0 for the Macintosh.

Creating EPS

Follow the instructions here to create an EPS file that can be incorporated into any application that accepts EPS.

Configure the tool set

Display the **Configure Schematic Design Tools** screen. Select one of the four EPS drivers. (To install the drivers on your system, run the INSTALL program from the Install disk.) Most illustrations in OrCAD manuals use EPS1.DRV.

Locally configure the Plot Schematic tool

Display **Plot Schematic**'s local configuration screen. Select the option **Send output to a file**, and the **Create a plot file** option below it. Some applications, such as WINWORD, require that EPS filenames end with .EPS, so enter **EPS** in the **File Extension** entry box. Then select **Automatically scale and set X,Y offsets for specified sheet size** and the **A** size option below.

Plot the schematic to disk

Once the tools are configured, you make an EPS file by executing **Plot Schematic**. You can now bring the EPS file into other applications.

Placing EPS in WINWORD

In WINWORD version 2.0, EPS support is built into NORMAL.DOT. Make sure your EPS filename ends with .EPS (required by WINWORD's EPS filter). Open NORMAL.DOT, select **Insert**, then **Picture**, and enter the name of the EPS file. In a moment, the placeholder for the EPS image appears.

That is all there is to it. The schematic does not display on your screen, but it will print on a PostScript printer. Earlier versions of WINWORD worked differently, so be sure to read the README.DOC file that accompanies version 2.0.

Placing EPS into Word 5.0

On the Macintosh, as in Word for Windows, you can place the EPS graphic into many applications. The following paragraphs describe how to insert an EPS graphic into Word 5.0 for the Macintosh.

Transfer the plot file from the PC to the Macintosh

Use one of many possible techniques to transfer the plot file from the PC to the Macintosh. For example, use MacLink Plus™ V5.01 (from DataViz Inc.) with file translation set to "EPS to EPS."

Open a Word document

Use Word's **Picture** command on the **Insert** menu to place the image in a Word document. If necessary, size or crop the image to fit the space available.

Error objects

If you run **Check Electrical Rules** and it flags errors on your schematic, remove them using **Draft's QUIT Update File** command.

Using sheets and parts to point to another worksheet

To create a hierarchy, you place an object on your worksheet that points to another worksheet file. This object can be a sheet symbol, a sheet part, or a sheetpath part. Their similarities and differences are discussed in detail in this section.

About sheets and parts

Before discussing sheet symbols, sheet parts, and sheetpath parts, it is important that you understand the difference between a *sheet* and a *part*. In trying to understand these differences, think of the function of the object rather than the object's physical appearance.

Parts are graphic objects that you place on the worksheet to represent the electronic devices in your design. Among other characteristics, they have part fields, are annotated, and may have power pins. If you print a bill of materials report, all parts on your worksheet are listed.

Sheets are objects that you place on a worksheet that point to another worksheet file. Sheets do not have part fields, are not annotated, and cannot have power pins. Sheets are not listed in a bill of materials report.

Table 9-2 compares parts and sheets.

Part	*Sheet*
Has part fields	Does not have part fields
Is annotated	Is not annotated
Can have power pins	Cannot have power pins
Appears in BOM report	Does not appear in BOM report
Appears in incremental connectivity database as a part	Appears in incremental connectivity database as an instance of a child

Table 9-2. Characteristics of parts and sheets.

Sheet symbol

A sheet symbol is a rectangle that you place on a worksheet using **Draft's PLACE Sheet** command. **Draft** automatically assigns it a random filename, such as 91G8F06#.SCH. You can tell it to point to the file of your choice by selecting **PLACE Sheet Filename.**

A sheet symbol functions as a sheet, and has all the sheet characteristics described in table 9-2.

The worksheet file that a sheet points to should be stored in the current design directory unless a path is specified as part of the sheet filename.

Sheet part

A sheet part is a library part that you change to a sheet. You place a library part on a worksheet using **Draft's GET** command. To change it to a sheet part, you use **Draft's EDIT SheetPart Name** command to tell it to point to a worksheet file.

Once you give a library part a sheet part name, it no longer functions as a part. It functions as a sheet, and has all the sheet characteristics described in table 9-2.

The worksheet file that a sheet part points to should be stored in the current design directory unless a path is specified with the **SheetPart Name**.

Sheetpath part

A sheetpath part is similar to a library part in that it is stored in a library. It may function as a sheet *or* as a library part. When you get a sheetpath part from a library, it *already* points to a worksheet file.

If you select the **Descend into sheetpath parts** option on a tool's local configuration screen, a sheetpath part functions as a sheet and has all the sheet characteristics described in table 9-2. If you don't select this option, the sheetpath part functions as a part and has all of the part characteristics described in table 9-2.

OrCAD libraries don't contain any sheetpath parts. You must create them and add them to a library. The worksheet file that a sheetpath part points to should be stored in the directory specified in the **Library Prefix** entry box in the **Library Options** area of the **Configure Schematic Design Tools** screen.

Conclusion

As described previously in this section, be sure to think of the function of an object rather than its physical appearance. Table 9-3 summarizes the different type of objects, and tells whether they function as a part or a sheet.

Type of object	*Functions as a sheet*	*Functions as a part*
Library part	No	Yes
Sheet symbol	Yes	No
Sheet part	Yes	No
Sheetpath part	If **Descend into sheetpath parts** option is selected	If **Descend into sheetpath parts** option is *not* selected

Table 9-3. Functionality of objects.

Moving designs

When you want to give a design to another user or send it to OrCAD technical support, you must include the following files:

- All associated schematic files
- An archived library file, or all custom library files which may be located in your library directory.
- Any custom netlist format files which may be located in your netlist directory
- The **Schematic Design Tools** configuration file for the design (SDT.CFG), unless you have included an archived library file.
- For OrCAD technical support, a readme file with name, phone number, a description of the problem, and instructions for reproducing the problem

Because **Backup Design** copies only those files and libraries that reside in the specified design directory, be sure to include any files that are located in other directories.

Installing new drivers

To add new display, printer, or plotter drivers once you have installed **Schematic Design Tool**, run the INSTALL program from your installation disk. You don't have to reload all the software—just the drivers that are found on the installation or upgrade disk.

1. Insert the disk labeled "Install" into your computer's floppy drive.
2. At the DOS prompt, enter the name of the drive the disk is in. For example, if you placed the installation disk in drive A, type `A:` and press <Enter>.
3. Type `INSTALL` and press <Enter>.
4. Follow the instructions on the screen, selecting the drivers you want to install. After you select drivers, INSTALL displays a menu of OrCAD tool sets. Don't select anything; just press <Enter>. This ends the installation.

Removing error objects

Check Electrical Rules puts temporary error markers, or *error objects*, on your worksheet to show you the locations of the errors. Select **QUIT Update** from **Draft**, or run **Cleanup Schematic** with the **Remove error objects from schematic sheet(s)** option selected to remove the error objects.

Converted part forms

A part's converted form *must* have the same pin numbers as the normal version of the part. Therefore, all pins must be declared in the normal version so they can be used in the converted form.

For example, if the normal version of a part has pins numbered 1 and 3, and the converted form has pins numbered 0 and 3, you will have errors when you create a netlist. To eliminate this problem, just be sure that the pin numbers used in the converted form of the part also appear in the normal version of the part. The normal part in the example should include a pin numbered 0.

Scaled printing

Plot Schematic can automatically scale your prints to the desired size. You do not have to do any calculations—**Plot Schematic** does it for you. **Plot Schematic** also automatically rotates the image for the best fit on the paper. Follow these steps:

1. Display **Plot Schematic**'s local configuration screen.
2. Enter the filename of the schematic you want printed in the **Source** entry box.
3. Select the **Send output to printer** option.
4. Select the **Automatically scale and set X, Y offsets for specified sheet size** option.
5. Select the size of paper to print your schematic on: **A, B, C, D,** or **E.** Or, if your software is set up to measure in **Millimeters** in the **Template Table** on the **Configure Schematic Design Tools** screen, select **A4, A3, A2, A1,** or **A0.**
6. Select **OK** to save the configuration and display the **Schematic Design Tools** screen.
7. Run **Plot Schematic.**

Creating global macros

If you have macros that you use with all your designs, try putting them all in a global directory. Follow these steps:

1. Create a directory called \ORCADESP\SDT\MACROS.
2. Copy the macro files you want to be global macro files to the \ORCADESP\SDT\MACROS directory and delete the files from the designs in which they originated.
3. Configure the **Draft Macro File** and **Edit Library Macro File** in the **Macro Options** area of the **Configure Schematic Design Tools** screen with the correct path and filename of the macro file in the global macro directory.

A

analog ▪ Circuitry in which both voltage and frequency output vary continuously as a function of the input.

annotation ▪ Assigning reference designators to parts in a schematic.

area ▪ A section of a screen containing related buttons or configuration options. Most areas are bordered and named. Examples include the **Editors** area on tool set screens and the **File Options** area on local configuration screens.

ASCII ▪ An acronym for *American Standard Code for Information Interchange*; a seven-bit code used to represent letters of the alphabet, the ten decimal digits, and other simple printable characters, as well as certain basic printer instructions such as Backspace, Carriage Return, and Line Feed.

B

bulletin board system ▪ A computer dedicated to maintaining messages and software and making them available over telephone lines. People *upload* (contribute) and *download* (gather) messages by calling the bulletin board from their own computers. Abbreviated BBS.

button ▪ A pushbutton-like image that you click to start an *action*. The *action* runs a single tool or a series of processes.

byte ▪ A piece of computer data composed of eight contiguous bits that are stored and typically interpreted as a single unit.

C

CAE ▪ An acronym for *computer aided engineering*.

check box ▪ A small square: ❑. Check boxes are used in lists of options when more than one option can be selected at a time. Compare *radio button*.

complex hierarchy ▪ A design in which two or more sheet symbols refer to a single worksheet. Compare *simple hierarchy*. See also *hierarchical design*.

configuration ▪ The information a button or tool set uses to operate. Configurations can be tailored to your needs. The configuration for a tool set applies to all tools in the set. See also *local configuration*.

connectivity database ▪ The incremental connectivity database (created by INET). The connectivity database describes the connectivity of a design, and is used to transfer a design to **Digital Simulation Tools**. See *incremental connectivity database*.

cursor ▪ A marker inside an entry box showing where characters typed on the keyboard will display. The cursor for insert mode is a heavy underline, and the cursor for overtype mode is a square. Compare *pointer*.

D

default ■ A preselected parameter.

design ■ A set of plans for electronic circuitry.

design cycle ■ The process of conceiving, developing, testing, and producing a circuit.

digital ■ Circuitry in which data in the form of digits are produced by binary (on-off or positive-negative) electronic signals.

download ■ The process of retrieving a file from another computer.

E

EDA ■ An acronym for *electronic design automation.*

editor ■ A tool used to create or modify a design file.

entry box ■ A box in which text or numbers can be entered using the keyboard: .

F

flat design ■ A schematic structure in which output lines of one sheet connect laterally to input lines of another sheet through graphical objects called *module ports.* Flat designs are practical for small designs of three or fewer sheets. See *module port, schematic,* and *hierarchical design.*

H

hierarchical design ■ A schematic structure in which sheets are interconnected vertically and laterally in a tree-like pattern. At least one sheet, the root sheet, contains symbols representing other sheets, called subsheets. See also *complex hierarchy, simple hierarchy, root sheet,* and *flat design.*

I

incremental connectivity database ■ Two or more files (.INX and .INF) produced by INET. The .INX file lists every sheet referred to in the design; a .INF file for each sheet describes connectivity for the sheet. ILINK uses the *incremental connectivity database* to create an incremental netlist. See also *connectivity database* and *incremental netlisting.*

incremental netlisting ■ A method of creating a netlist in which only changed worksheets are processed each time **Create Netlist** or **Create Hierarchical Netlist** is run. See also *netlist.*

initial macro ■ A macro that runs automatically whenever you run **Draft** or **Edit Library**. For the initial macro to work, you must configure **Schematic Design Tools** to load a macro file containing the desired macro definition.

intermediate netlist structure ■ Three files (.INS, .RES, and .PIP) produced by ILINK for a design. The .INS (instance) file, the .RES (resolved) file, and the .PIP file (pipe link commands) are used by IFORM to create a netlist in one of over 30 formats.

K

K ■ An abbreviation for *kilobyte*. 1K is equal to 2^{10} (1024) bytes. The "K" is taken from the metric system, where it stands for "kilo," or 1000.

key field ■ A list of the part fields to combine and compare. Key fields are defined on the **Configure Schematic Design Tools** screen.

L

library ■ A collection of standard, often-used part symbols stored as templates to speed up the design process.

librarian ■ A tool used to manage and create library parts.

list box ■ A box on local configuration screens and in windows that lists files in specific designs or directories. You move through the list using scroll buttons next to the list box. On local configuration screens, you can specify a wildcard so the list box contains files that match the criteria you specify.

local configuration ■ Configuration settings for a particular button. If the button runs several processes, each process can be configured locally. One tool can have different configurations in different buttons. For example, **Annotate** is configured differently under the **To Layout** button and under the **To Digital Simulation** button.

M

MB ■ An abbreviation for *megabyte*. See *megabyte*.

macro ■ A series of commands you can execute automatically at the touch of a single key. Macros dramatically reduce the number of keystrokes required to perform complex or repetitive actions.

megabyte ■ One megabyte is equal to 2^{20} (1,046,576) bytes. The prefix "mega" is taken from the metric system, where it stands for "one million." Abbreviated *MB*.

module port ■ A graphical object that conducts a signal between schematic worksheets. See also *flat structure*.

N

net ■ A graphical object that conducts a signal into or out of a sheet symbol, much as a module port conducts signals between schematic worksheets.

netlist ■ An ASCII file that lists the interconnections of a schematic diagram by the names of the signals, modules, and pins connected on a PCB. The nodes in a circuit. See also *incremental netlisting*.

P

pan ■ To change the portion of the worksheet being viewed by dragging the pointer from one location on the worksheet to another location. As you drag the pointer, the worksheet *pans* across the screen.

part ■ A schematic symbol that represents an object. The object can be either a part or another worksheet.

part field ■ A data area which holds and displays information about a part. Each part has ten part fields; two are reserved for part value and part reference, one is recommended for module value, and seven can be used to store other useful information. See also *key field*.

PCB ■ An acronym for *printed circuit board*.

PLD ■ An acronym for *programmable logic device*. See *programmable logic device*.

pointer ■ An arrow on the screen that moves as you move the mouse: ↖ . Compare *cursor*.

processor ■ A tool that subjects a design file to a specific process.

programmable logic device ■ A type of integrated circuit that contains fuses that can be blown, eliminating certain logical operations in the device and leaving others intact, giving the device one of many possible logical architectures or logical configurations.

prompt ■ A query from a program asking you to enter specific information.

R

radio button ■ A small circle: O. Radio buttons are used in lists of mutually exclusive options: only one button can be active at a time. Compare *check box*.

reporter ■ A tool that creates a report, but does not modify design data.

root directory ■ The main directory on your computer; the directory that the computer boots from.

root sheet ■ The worksheet from which references to other worksheets are made in a flat or hierarchical design. Each design has only one root worksheet. See also *flat design* and *hierarchical design*.

S

schematic ■ A graphical representation of a circuit using a standard set of electronics symbols. See *flat design*, *hierarchical design*, and *root sheet*.

scroll buttons ■ Buttons used to move a directory in its window so that a different part is visible. The four scroll buttons are:

- Line Up
- Page Up
- Page Down
- Line Down

sheet symbol ■ A block-shaped symbol representing another worksheet. Signals are conducted into and out of sheet symbols by nets. See *net*.

simple hierarchy ▪ A one-to-one correspondence between sheet symbols and the schematic diagrams to which they refer. Each sheet symbol represents a unique subsheet. Compare *complex hierarchy*. See also *hierarchical design*.

syntax ▪ The formal structure of a language. Syntax includes the rules for making statements in the language, but excludes the meanings of the statements.

T

tag ▪ A marked or saved location on a schematic or layout. You can use the **JUMP** command to go to a tag.

template ▪ A set of patterns used to create new designs. The template is *not* meant to be an actual working design.

text export ▪ The process of copying text from a schematic worksheet to a text file.

text import ▪ The process of copying text from a text file to a schematic worksheet.

TTL ▪ An acronym for *transistor-transistor logic*.

tool ▪ A computer program that performs some useful task. OrCAD tools are grouped into five categories: editors, processors, reporters, librarians, transfers.

tool set ▪ A collection of tools designed to perform a set of electronic design automation tasks. OrCAD tool sets include: **Schematic Design Tools, Programmable Logic Design Tools, Digital Simulation Tools,** and **PC Board Layout Tools.**

transfer ▪ A tool that transfers design information from one tool set to another tool set. Also runs whatever processes are necessary to go from one tool set to another.

U

upload ▪ The process of sending a file to another computer.

user button ▪ A button that you can program to perform whatever combination of functions you find useful (such as executables or batch files). User button programs are saved with the design files, so you can create design-specific buttons and not worry about overwriting user button programs for other designs.

W

wildcard ■ A series of characters you specify in a **Wildcard** entry box to filter the files that display in a list box. For example, *.* allows all files to be displayed in a list of files.

worksheet ■ The sheets of drafting paper on which schematics are drawn. Worksheets appear on the computer screen as a rectangular area in which you can place parts and draw wires.

Z

zoom ■ To change the view on the screen by making the objects appear larger or smaller.

A

B

C

H

I

J

K

L

M

N

O

P