OrCAD®
Schematic Design Tools User's Guide

Schematic Design Tools

User's Guide

OrCAD®

Electronic Design Automation Tools

Schematic Design Tools

User's Guide

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

Document Number: OR9062B 3-31-91

OrCAD®

3175 NW Aloclek Drive
Hillsboro, Oregon 97124-7135
U.S.A.

Sales & Administration	(503) 690-9881
Technical Support	(503) 690-9722
24-Hour Bulletin Board System	(503) 690-9791
FAX	(503) 690-9891

CONTENTS

CHAPTER 1

Welcome to OrCAD Schematic Design Tools

Welcome to practical electronic engineering. You now own **OrCAD Schematic Design Tools,** a powerful, yet straightforward design entry tool set with the power of an engineering workstation. Using **Schematic Design Tools,** complex design tasks can be done in a fraction of the time it takes by hand.

Developed specifically to run on personal computers, **Schematic Design Tools** supports most popular graphics boards, printers, and plotters.

Finding the information you need

Five manuals accompany **Schematic Design Tools.** They are:

- *Installation & Technical Support Guide*
- *OrCAD/ESP Design Environment User's Guide*
- *Stony Brook M2EDIT Text Editor User's Guide*
- *Schematic Design Tools User's Guide*
- *Schematic Design Tools Reference Guide*

Installation

Before you begin to explore **Schematic Design Tools,** take a few minutes to install the tool set and register for technical support. Just follow the instructions in the *Installation & Technical Support Guide.*

Project-oriented design environment

Schematic Design Tools is one part of a fully integrated *Electronic Design Automation* (EDA) system. The design environment is structured to allow you to focus on what's important: the design. Designs are organized on a project-by-project basis, with all the design files—schematics, netlists, parts lists, simulation results, and board layouts—stored together.

The *OrCAD/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 **Design Environment** screen, even if you only have one tool 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 Schematic Design Tools,* and 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 have mastered the basics, refer to the *Schematic Design Tools 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 **Schematic Design Tools'** commands and concepts, and tells how to transfer a design between OrCAD applications. It is designed to be a continuing source of instruction and reference as you use **Schematic Design Tools.**

What's new in the design environment?

Schematic Design Tools is one part of a fully integrated electronic design automation environment. The graphical design environment lets you:

- 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** screen.

These functions are described briefly on the pages that follow. The explanations assume 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 7 of this guide. Advanced concepts are fully explained in the *Schematic Design Tools 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.

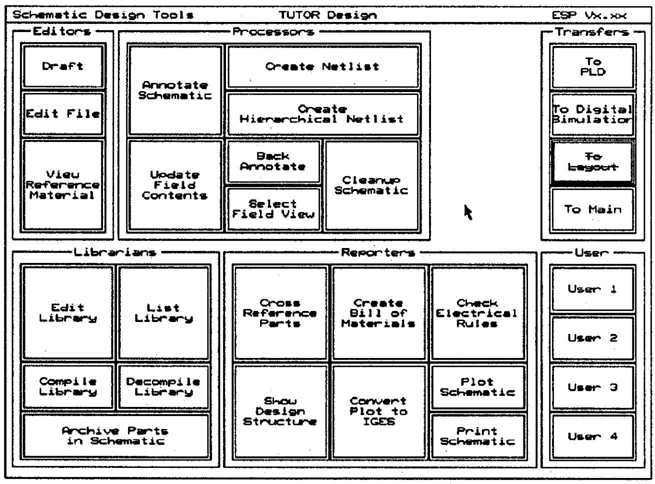


Figure 1-1. Schematic Design Tools screen.

Editors

Editors are used to create or modify design files. **Schematic Design Tools** contains three editors:

- **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 Material.** This tool allows you to review reference material supplied with **Schematic Design Tools** using a text editor. 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. **Schematic Design Tools** includes six processors:

- **Annotate Schematic.** This tool scans schematic designs and automatically updates part reference designators (such as U?, R?). It also updates the pin numbers associated with the reference designators in multiple-parts-per-package devices. **Annotate Schematic** can handle very large, complex, and multiple worksheets. It can update incrementally (leaving previously assigned reference designators alone) or unconditionally.
- **Create Netlist.** A netlist is a text file listing the logical interconnections between signals and pins. When the design becomes a real circuit board, the netlist turns into patterns of physical connections called tracks and nets. **Create Netlist** generates a netlist in one of over 30 different formats. Refer to *Appendix B: Netlist formats* in the *Schematic Design Tools 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 Reference Guide* for instructions.

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

- **Create Hierarchical Netlist.** This tool operates similar to the **Create Netlist** tool, only it uses a hierarchical design. Hierarchical designs are discussed later in this chapter.
- **Update Field Contents.** Every library part has ten data fields used to hold text or data associated with the part. One data field holds reference designator values, such as "U1A" or "Q1." Another holds the part's name, such as "74LS04" or values relevant for the part, such as Ohm (Ω) values for resistors. The other eight data fields can store any information you might find useful: part tolerance, vendor name, part number, and so on. **Update Field Contents** changes information in a data field for parts in a schematic, based on the contents of a match file. You create the match file using **Edit File**'s text editor.
- **Back Annotate.** This tool updates part reference designators in your design. A list of old and new reference designators—called a Was/Is file—is used to update your schematic worksheets. You create the Was/Is file using **Edit File**'s text editor.
- **Cleanup Schematic.** This tool checks a design for wires, buses, junctions, labels, module ports, and other objects that are placed on top of each other.
- **Select Field View.** The **Select Field View** tool makes the contents of a data field either visible or invisible on the schematic.

Librarians

Schematic Design Tools includes part libraries containing more than 20,000 devices. The libraries contain parts representing TTL, IEEE, CMOS, memory, ECL, discrete, analog, microprocessor, and peripheral devices.

In addition to the libraries, there are tools for managing and creating library parts. The **Librarian** tools are:

- ❖ **Edit Library**. This tool is a graphical editor for creating or modifying library components. With this editor, you use commands similar to **Draft**'s to build or modify a part and add it to a library.
- ❖ **List Library**. This tool lists all the parts in a library.
- ❖ **Archive Parts in Schematic**. This tool scans a set of schematics, collects all the library parts used, and makes a library file containing only the parts used in those schematic files.

Parts can also be created or modified using a text editor, such as the one available using **Edit File**. If you prefer to create or modify parts in this manner, you will find the following tools very useful:

- ❖ **Compile Library**. This tool converts a text file containing library source code into a compressed library object file, the form usable by the other **Schematic Design Tools**.
- ❖ **Decompile Library**. The inverse of the **Compile Library** tool, this tool converts a library object file to a text-only library source code file. You can then edit the source code file using **Edit File**.

Reporters

Reporters are tools that produce human-readable reports, but do not modify design data in any way. **Reporters** include:

- ❖ **Cross Reference Parts.** This tool scans the schematic files, gathers information for all parts used in the schematic files, and creates a cross reference reporting each part's location in the design.
- ❖ **Create Bill of Materials.** This tool lists all the parts used in a single schematic or in the entire design, sorted by reference designator. You can also merge additional information into the report using an *include* file.
- ❖ **Check Electrical Rules.** This tool checks a design for conformity to basic electrical rules. It checks for shorts, inputs with no driving source, unconnected pins, bus contention, and other common electrical hook-up problems.
- ❖ **Show Schematic Structure.** This tool scans a hierarchical organization of sheets to display the structure, sheet names, and sheet path names of the hierarchy.
- ❖ **Convert Plot to IGES.** This tool translates a plot file (created by the **Plot Schematic** tool) to the data format given Initial Graphics Exchange Specification (IGES). 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®).

Plotting and printing

There are two basic types of output devices that can be used with **Schematic Design Tools**: plotters and printers. These devices are categorized by the type of input they require.

If a device accepts *vector* commands, it is considered to be a plotter. A vector is a series of points with a specific function defined. For example, a line has a beginning point and an ending point. A circle has a center and a radius.

The device needs to know what the vector information is but does not need every point along the vector.

If a device accepts *raster* commands, it is a printer. A raster is an array of dots. When you draw a line to a raster device, you must specify each and every dot.

- **Plot Schematic**. This tool plots a single schematic or an entire design. It produces high-resolution, high-quality plots of your designs.
- **Print Schematic**. This tool prints a single schematic or an entire design. It produces rough draft-quality printouts of your designs.

Transfers

Transfer tools perform the steps needed to tell a design database that the design may be viewed by another OrCAD tool set. During the design process, the design database created in one tool set (such as **Schematic Design Tools**) is not useable by other tool sets (such as **Digital Simulation Tools**) for much of the design process. This is because the design is not complete, it is being designed. The transfer is how the design database is updated so that the other tool may have access. The **Transfer** tools take care of intermediate steps so that you don't have to. The four transfer tools in **Schematic Design Tools** are:

- **To PLD**
- **To Digital Simulation**
- **To Layout**
- **To Main**

For example, the **To Digital Simulation** tool does these intermediate steps:

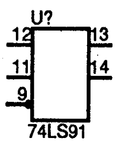
- Runs the **Annotate Schematic** tool
- Runs the incremental netlist compiler (INET)
- Runs the ASCTOVST process
- Transfers control to **Digital Simulation Tools.**

Graphic objects

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

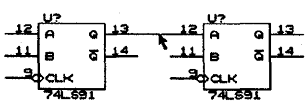
- Parts
- Wires
- Buses
- Junctions
- Power Objects
- Module Ports
- Sheet Symbols
- Labels
- Text
- Title Block
- Stimuli
- Trace
- Test Vectors
- Layout Directives

Parts



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

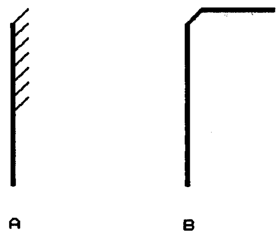
Wires



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

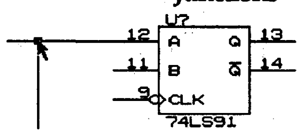
Buses

Buses are graphic objects used to represent an array of signals as a single unit on your worksheet.



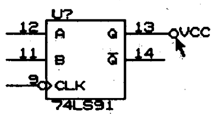
Junctions

Junctions are graphic objects that indicates a physical connection between wires, busses, and nodes. Junctions look like small square boxes.



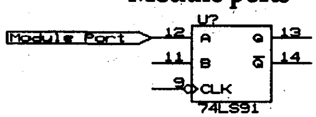
Power objects

Power objects are graphic objects that indicate a connection to a power source.



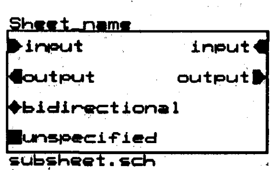
Module ports

Module ports are graphic objects that conduct signals between schematic worksheets.

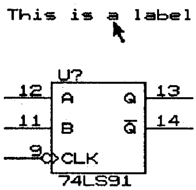


Sheet symbols

Sheet symbols are block-shaped symbols representing another worksheet. Each sheet symbol represents a subsheet.

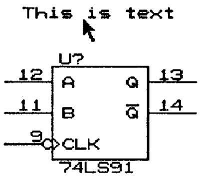


Labels



Labels are identifiers placed on a schematic that can physically connect signals together without actually showing the connection on the schematic.

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; and drawing title, number, size, and revision.

Stimuli

OrCAD's **Digital Simulation Tools** uses *Stimuli* to determine if a circuit performs as desired. A stimulus is an algorithmic function of the signal to be applied to a circuit.

Test vectors

A *Test vector* is similar to a stimulus, except it is a stream of signal values, which may or may not be algorithmic in pattern.

Trace

A *Trace* is used to tell **Digital Simulation Tools** which signals to trace.

Layout directives

A *Layout directive* is used to tell **PC Board Layout Tools** information about particular signals such as track width, via size, routing layer, etc.

The design process

As its name suggests, **Draft** is designed to be analogous to the schematic design tools with which you are already familiar: drafting board, pencil, sheets of paper, standard logic symbols and symbol templates, and so on.

In addition, **Draft** is designed to support the complete design *process* from general concepts of a design to the final sets of detailed schematic diagrams.

How does **Draft** represent these tools and processes?

The computer screen represents the drafting table. The pointer does what a pencil does, and more. Drawing (and erasing) are done using **Draft** commands.

Draft calls the sheets of drafting paper on which the schematics are drawn *worksheets*. Worksheets appear on the computer screen as a rectangular area in which you can place parts and draw wires.

When you save the work you have done on a worksheet, **Draft** stores the information on the computer's disk as a data file. The name of the worksheet is the name of the file in which it is saved. Worksheets are stored inside designs. A design is a directory that contains all of the files (including the worksheet) that are part of the design process. All designs are contained in the \ORCAD directory.

Draft saves the worksheet in the design in which you are working. The worksheet can have the design name and an extension of .SCH, or you may give it different name.

For example, if you have a design called TUTOR, the path and filename for the TUTOR schematic is \ORCAD\TUTOR\TUTOR.SCH.

Design structures

Some designs are small enough to be represented entirely on a single schematic worksheet. Draft's standard page sizes correspond to the five standard sheet sizes for plotters and printers (A through E for English, and A4 through A0 for Metric). You can also create custom page sizes up to 65 inches by 65 inches.

But a design may be too large to fit entirely on even the biggest sheet. And even if a very complex design could fit on one sheet, there are good reasons for dividing it up:

- 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.

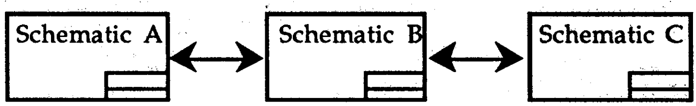
Draft offers two ways of handling multiple sheet designs:

- Flat designs
- Hierarchical designs

Each type offers advantages for certain designs. You can choose whichever way suits your design best.

Flat designs

Best suited for small designs no more than five to ten sheets in size, flat designs connect the output signals laterally from one schematic to the input signals of another. All files in the design are equal in importance to the others, as shown below.



Module ports

Flat designs are linked together by adding a brief text notation to the root schematic of the design. The root of the design is the schematic that has the same name as the design and a .SCH extension. In flat designs, the output signals from one schematic can connect to the input signals of any other. The signal connections that connect to other sheets are represented by graphical objects called *module ports*. Module ports that have identical names on both schematics are considered 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 **COUNT, CLEAR, LOAD,** and **RCO.** The module ports named **Hi[0..3]** and **Lo[0..3]** don't connect to each other.

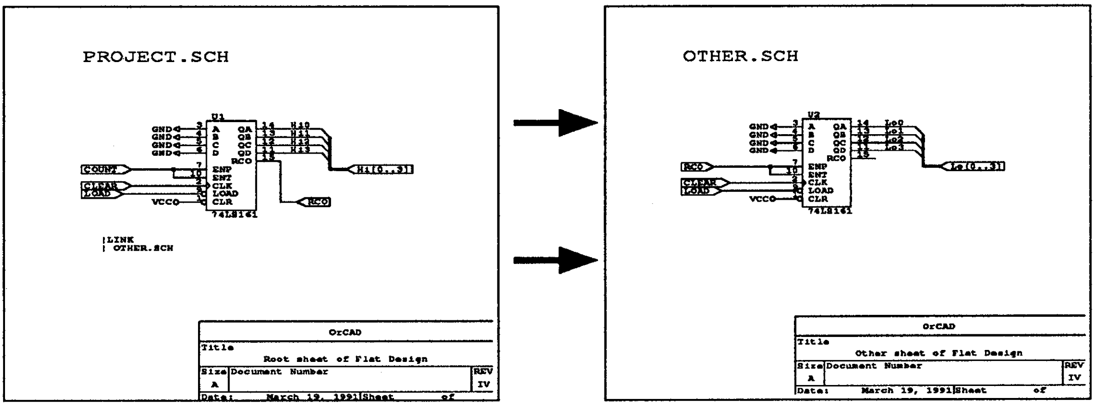


Figure 1-2. Module ports used to link one schematic to another.

Notice the |LINK command (pronounced "pipe link") on the PROJECT.SCH worksheet in figure 1-2. This command is used to tell which worksheets the module ports link to. It is described on the next page.

Figure 1-2 shows only input and output module ports, and connections between single wires. **Draft** has two other types of module ports: bidirectional and unspecified. You can use module ports to connect buses, as well as single wires. All four types of module ports are shown on the next page.

Input_name	Input module port
Output_name	Output module port
Bidirectional	Bidirectional module port
Unspecified	Unspecified module port

| LINK command

Module ports indicate the names of the signals to connect but do not specify which schematics are to be included in the design. Therefore, flat designs must have one other component: a list of the worksheets in the schematic. This list appears on the root schematic, and consists of the "pipe" character (the vertical bar on your keyboard) followed by the keyword "LINK", followed by subsequent lines containing the pipe character and the filenames of the worksheets to link to the root sheet.

The example below shows text as it would appear on a schematic that has module ports that link to schematics called SCHEM1.SCH, SCHEM2.SCH, AND SCHEM3.SCH. This text can appear anywhere on the worksheet.

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

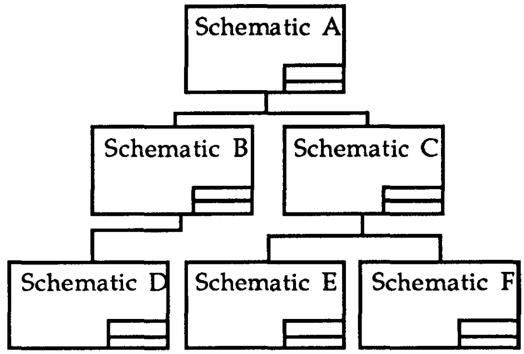
△ *NOTE: For details about module ports, see the* ***PLACE Module Port*** *command in the* Schematic Design Tools Reference Guide. *For details about placing text on a worksheet, see the* ***PLACE Text*** *command in the* Schematic Design Tools Reference Guide.

When to use a flat design

A flat design is best suited for small designs. The major limitation of a flat design is that the user must manage all of the interconnections between the sheets by the names assigned to the module ports. When the design becomes large, the number of names can be quite extensive.

Hierarchical designs

Instead of using a flat design, you can draw 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 corporate organizational chart or a schematic design—has "higher" and "lower" levels.



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

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

How signals enter and leave sheet symbols

Just as signals are conducted between schematics through module ports, they are conducted into and out of sheet symbols through graphical objects called *sheet nets*. These are the small black objects shown on the borders of the sheet symbols in figure 1-3.

You place sheet nets using **Draft's Add Net** command, which becomes available when you select the **PLACE Sheet** command.

The sheet nets on a sheet symbol correspond to module ports on the associated schematic. To associate a particular sheet net with a particular module port, assign them the same name.

The bracketed notation shown on the module ports and nets [m..n] designates the number of signals being carried by a bus. So [0..3] indicates four signals, 0 through 3.

In real designs, buses must have a label or a module port with similar bracket notation to indicate the number of signals they carry, and wires connected to buses must have labels identifying the signal they carry. These details are shown in figure 1-3.

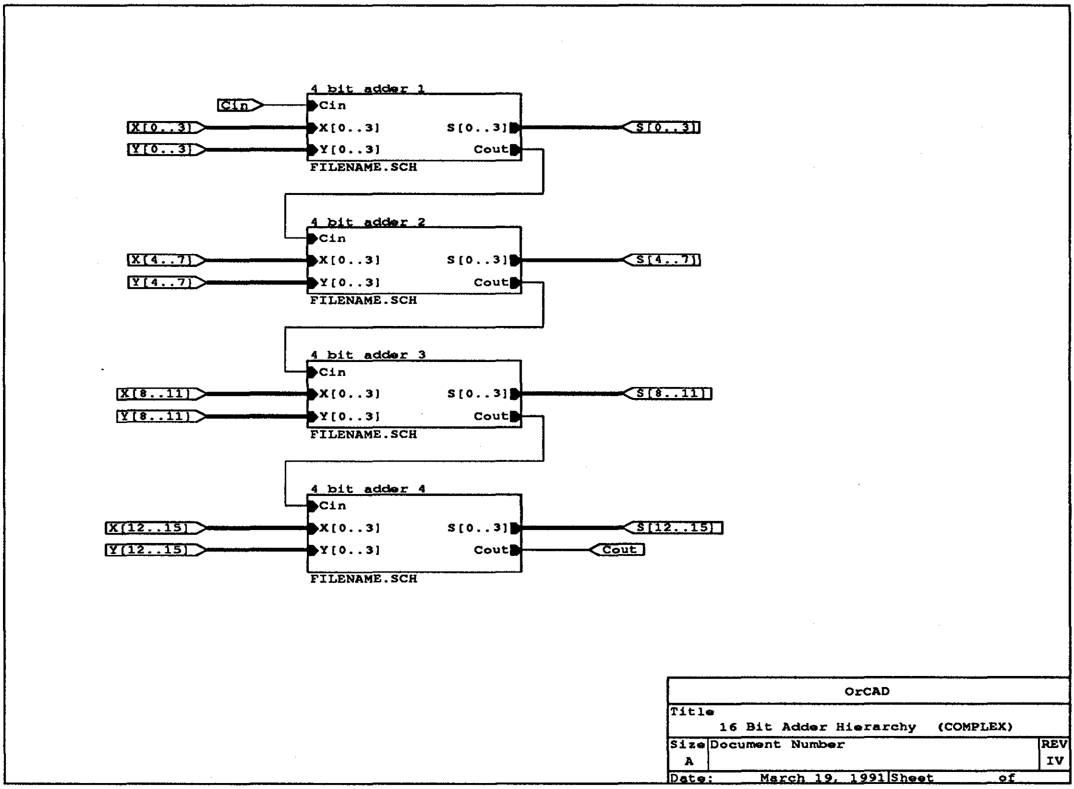


Figure 1-3. Simple hierarchical structure.

Hierarchies can access the same logic repetitively

The diagram shown in figure 1-3 shows a one-to-one correspondence between sheet symbols and the schematic diagrams they reference. This structure is called a *simple* hierarchy.

But what if you have a design in which the logic from a particular schematic must be used in several places? Is each sheet symbol representing the logic required to reference a separate schematic, even though they are identical?

Schematic Design Tools can reference a single schematic from more than one sheet symbol. All you have to do is mark (in the schematic) all the desired sheet symbols with the same filename, the filename of the schematic to reference. This structure is called a *complex* hierarchy.

How sheet symbols reference schematic logic

To get a sheet symbol to access the logic of a particular schematic, you "mark" the sheet symbol with that schematic's filename. This mark displays at the bottom of the sheet symbol, as shown in figure 1-3.

You "mark" the sheet symbols using the **Filename** command, which becomes available when you select the **PLACE Sheet** command.

In addition to their filename markers, sheet symbols also have names of their own. They are used to identify them on their own schematic. 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.

Moving between levels in a hierarchy

Draft makes it easy to move up and down in hierarchies, from sheet symbol to associated schematic and back again.

To go from a sheet symbol to the associated schematic, put the pointer on the sheet symbol, and select **QUIT Enter Sheet**. To go from the schematic back to the schematic in which it is referenced by a sheet symbol, select the **QUIT Leave Sheet** command.

More about hierarchical design structures

The schematic represented by a sheet symbol can itself have a sheet symbol within it. This means hierarchies many levels deep can be created, each level containing progressively more detail.

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

Another advantage offered by hierarchical structure is the ability to use sheet symbols to repeatedly reference "stock" schematics containing common circuit functions. This is used in gate array and FPGA designs.

△ *NOTE: Designing a deep hierarchy is much more efficient than designing a wide hierarchy. A wide hierarchy, while not a flat design, has many of the limitations in organization, presentation, and structure that flat designs have. A deep hierarchy lets the functional nature of the design be represented and presented more clearly.*

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

Learning Schematic Design Tools

The remainder of the *Schematic Design Tools 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 of the remaining chapters builds on the skills and concepts from the previous chapter. As you complete each chapter, you create a series of working files.

The summary below describes the design concepts and skills you learn in each chapter.

Chapter 2: Introducing Schematic Design Tools

This chapter introduces **Draft**, the **Schematic Design Tools** schematic editor. You learn how to run **Draft**, change default work conditions settings, select sheet size, change view and display options, and save your schematic.

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 components, how to draw 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 hardcopy of the schematic.

Chapter 5: Creating a custom component

In this chapter you use the **Edit Library** tool to define a custom component (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 the **Annotate Schematic** tool, the **Check Electrical Rules** tool, the **Create Netlist** tool, the **Back Annotate** tool, the **Create Bill of Materials** tool, and the **Plot Schematic** tool.

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 sheets are linked together is also reviewed.

Introducing Draft

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

- Run **Draft**, the schematic editor
- Change default configuration settings
- Change view and display options
- Define and save an initial macro
- Save your work
- Structure circuit designs in different ways

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 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

Other keys (such as <End>, <F1>, <F2>, etc.) can be shown in angle brackets. In addition, single characters or numbers are also shown in angle brackets (for example, <A> or <1>).

Keyboard input

Text for you to enter is shown in two ways:

- As bold text in typewriter font. For example, "enter **`tutor.sch`**"
- As bold text in typewriter font enclosed in a box. For example,

```
tutor.sch
```

or

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

In the examples above, you only enter the characters shown in bold. The non-bold characters show what is displayed on the screen.

Operating system command prompt

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

```
C:>
```

Filenames

Filenames can be from one to eight characters long. If desired, a filename can be followed by a period and up to three characters for an extension. You can use either uppercase or lowercase letters when entering a filename, but the operating system converts all the letters to upper case. Most of the instructions in this manual use lowercase file names.

Filenames usually contain only letters and numbers. You can use additional characters supported by the operating system. For best results, use letters (A-Z) and numbers (0-9) and limit special characters to under-score (_), pound sign (#), and at sign (@) for compatibility with OrCAD's environment.

Most OrCAD software works with any characters your operating system supports. Some applications used in conjunction with OrCAD software support a more limited character set than what the operating system supports. These include Spice programs, some PCB layout programs, and some text editors.

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 be working on a design called "TUTOR." All of the files for this design are contained in the directory called "TUTOR." Files pertaining to this design are given the name "TUTOR" and an extension to indicate the type of file. For example, the TUTOR schematic worksheet that you create in chapters 1 through 6 is named TUTOR.SCH.

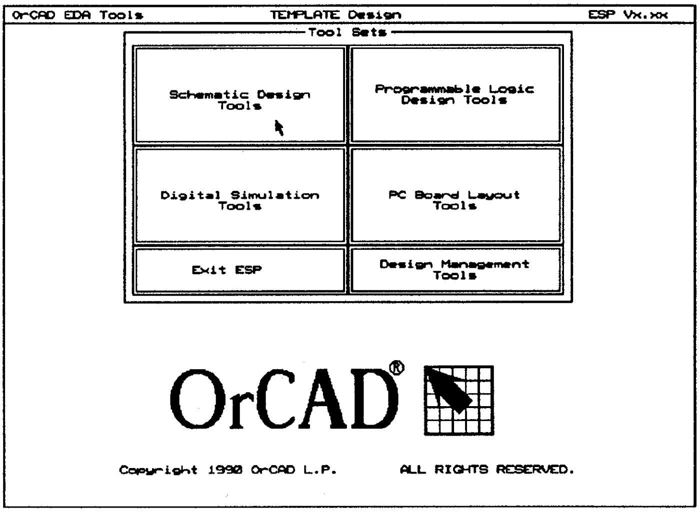
Running ESP

To run an OrCAD tool, you must first display the design environment screen. To do this, follow these steps:

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 design environment screen displays:



Design environment work screen.

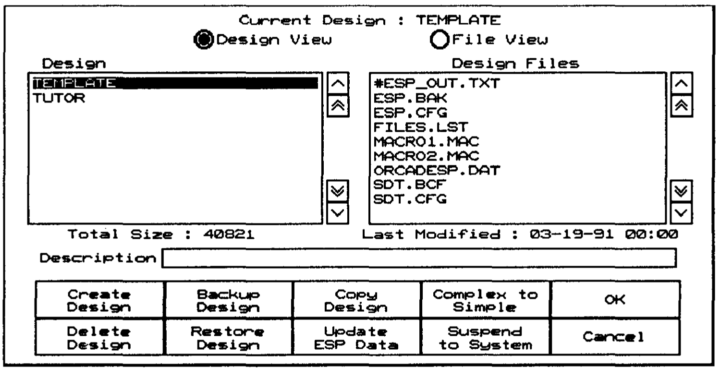
Changing to the TUTOR design

Before you do any work with any of the tools accessed from the design environment 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.

1. Place the pointer on the title bar at the top of the work screen and click the left mouse button.
   The menu shown above displays.

```
Design Management Tools
Design Management Tools
Suspend to System
Exit
```

2. The **Design Management Tools** command is highlighted. Click the left mouse button again. This selects the **Design Management Tools** command. The screen shown below displays.



Design management tools screen.

3. Place the pointer on the design named TUTOR and click the left mouse button to select the TUTOR design.

4. Click **OK** to return to the design environment screen. Notice the heading in the upper center of the screen has changed to **TUTOR Design**.

△ *NOTE: Refer to the* OrCAD/ESP Design Environment User's Guide *for instructions on how to use the features of the Design Management Tools.*

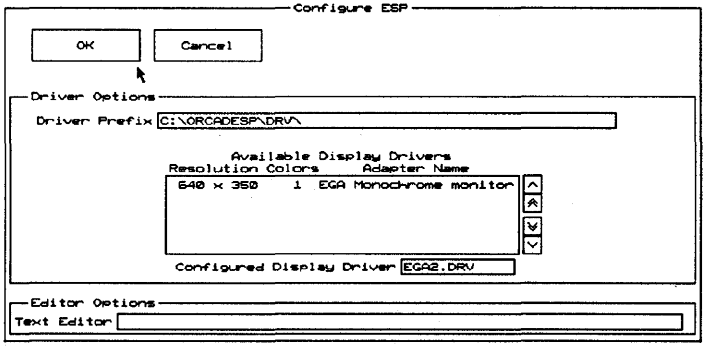
Change the start up design

The design environment is configured to display the TEMPLATE design directory each time you run OrCAD tools. Since you will be working in the TUTOR design throughout this guide, you need to change the start-up design to TUTOR.

1. Click on any of the tool buttons that display on the design environment screen. The menu at right displays.

```
Schematic Design Tools
Execute
Local Configuration
Configure ESP
Help
```

2. Select **Configure ESP**. The screen below displays:



First part 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** section.

```
Design Options
Startup Design  TEMPLATE
```

4. Place the pointer in the **Startup Design** entry box and click the left mouse button. Use the <Backspace> key to delete `TEMPLATE`. Enter **TUTOR** as the startup design.

```
Design Options
Startup Design  TUTOR
```

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

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

△ *NOTE: Refer to the* OrCAD/ESP Design Environment User's Guide *for detailed instructions on how to configure ESP.*

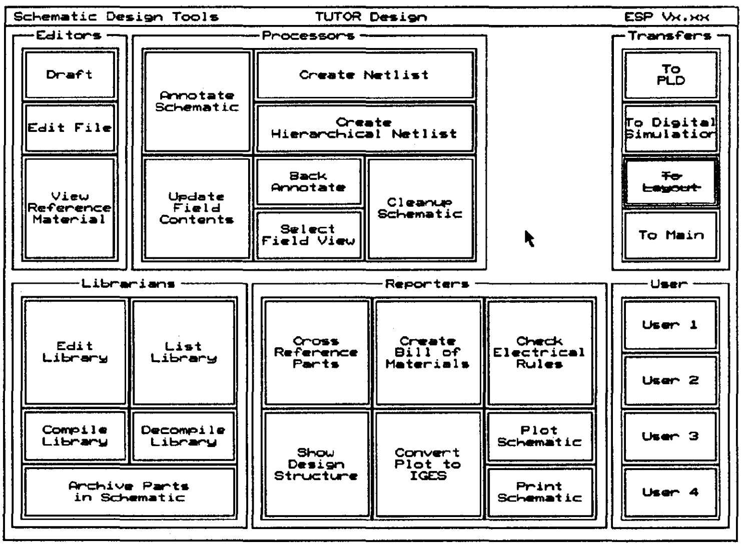
Running Schematic Design Tools

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

1. Point to the **Schematic Design Tools** button and click the left mouse button. The menu at right displays.

```
Schematic Design Tools
Execute
Local Configuration
Configure ESP
Help
```

2. Select the **Execute** command. The **Schematic Design Tools** screen displays:



Schematic Design Tools screen.

Defining title block information

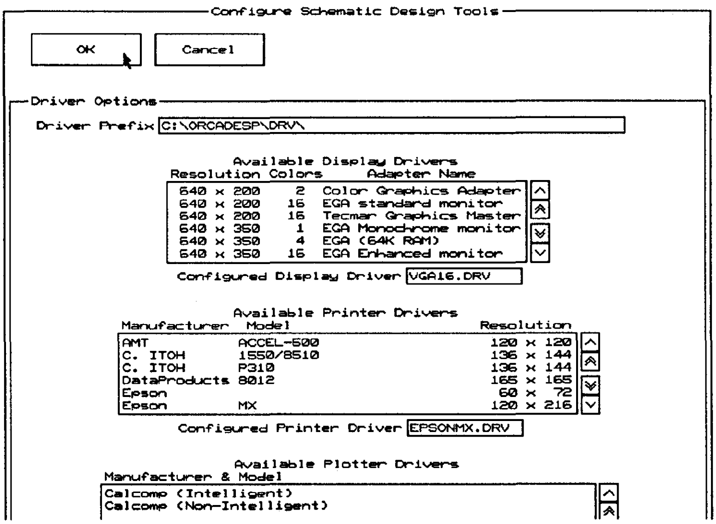
Before you run the schematic editor **Draft**, take a few minutes to define the information to appear in the title block of the worksheet you will create. To do this, you must display the **Schematic Design Tools** configuration screen.

View Schematic Design Tools' configuration

1. From the **Schematic Design Tools** screen, click the **Draft** button.
2. Select **Configure Schematic Tools** from the menu that displays.

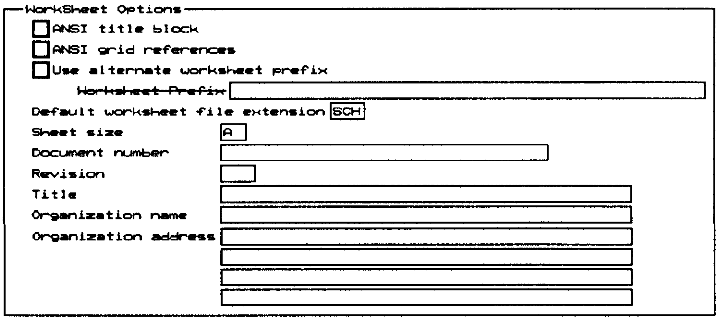
```
Draft
Execute
Local Configuration
Show Version
Configure Schematic Tools
Help
```

The figure below shows the first part of the **Configure Schematic Design Tools** screen. The parameters you see may vary, 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 Reference Guide.*



First part of ***Configure Schematic Design Tools*** *screen.*

3. Pan to the **Worksheet Options** area (shown below).



***Worksheet Options** area of **Configure Schematic Design Tools** screen.*

4. 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.

   Position the pointer within the **Title** entry box and press <Enter> or click the left mouse button. The pointer becomes a cursor inside the entry box. Enter the title: **`Digital clock schematic.`**

   Press <Tab> to move to the next entry box, in this case, **Organization name.**

   Press <Enter>.

5. Enter the name and address of your organization in the **Organization name** and **Organization address** entry boxes.

6. To update the configuration and return to the **Schematic Design Tools** work screen, move the pointer to the top of the screen and click the **OK** button.

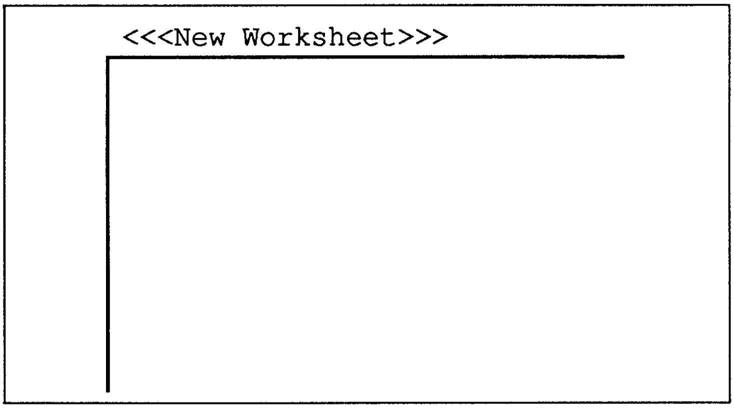
7. The **Schematic Design Tools** screen displays.

Running Draft

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

1. Click the **Draft** button. The **Draft** menu displays.
2. Select **Execute**.

Draft is now running. The top and left edges of the sheet are displayed. 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.



*New worksheet in **Draft**.*

OrCAD basics

Pop-up menus written in plain English guide you from step to step in OrCAD software. **Draft** organizes commands using menus and command lines. You can select a command by either clicking the mouse or pressing a key. (For complete command descriptions, refer to the *Schematic Design Tools Reference Guide.*)

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, whichever you prefer.
- 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, whichever you prefer.

Display the main menu

To display **Draft's** main menu, (shown at right) press <Enter> or press the left mouse button. To remove the main menu from the screen, press <Esc> or click the right mouse button.

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

Follow the steps below to display and remove the main menu.

1. To see the main menu, press <Enter>.
2. To remove the main menu from the screen, press <Esc>.
3. To see the main menu, click the left mouse button.
4. To remove the main menu from the screen, click the right mouse button.

Commands

To use a command, first you select it, then tell **Draft** to perform the task.

There are several ways to select and use 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 select a command	Use the arrow keys on the key-board to place the highlight over the command name.	With the mouse, slide the high-light over the command name.
To use a command	Press <Enter>.	Click the left mouse button.
To select and use a command	Press the capitalized letter in the command name.	

Table 2-1. Using the keyboard or mouse to select a command.

Selecting commands

Draft responds to a command by either performing the command's function or 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.

1. Press <Enter> to display the main menu.
2. Select the **BLOCK** command. Notice that another menu displays. The **BLOCK** menu is shown at right.
3. Press <Esc> to return to the main menu.

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

Command lines

Command lines are a series of command names 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. Press <Esc> or the right mouse button to return to the menu or command line that called the command line.

1. Press <Enter> to display the main menu.
2. Select the **EDIT** command.

   Notice a command line displays across the top of the screen. The **EDIT** command line is shown below.

   ```
   Edit  Find  Jump  Zoom
   ```

3. Press <Esc> to return to the main menu.

How command names are shown in this guide

In this guide, main menu command names are shown in uppercase letters. Other command names are shown with just the first letter capitalized. When you are asked to select a command, usually both the main menu command name and other command name are specified.

For example, the statement, "Select the **PLACE Wire** command" means, "Select the **PLACE** command from the main menu, and then from the resulting **PLACE** menu, select the **Wire** command."

Sometimes, when the context is clear, the main menu command is not specified. For example, if the **PLACE** menu is already displayed, and you are asked to select the **Wire** command, the instruction is simply, "Select the **Wire** command."

Return 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. At this point, the main menu appears if you press <Enter>.

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

1. Press <Enter> to see the main menu.
2. Select **SET** from the main menu. The **SET** menu appears, 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	

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

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

Auto Pan

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.

Pan across the schematic

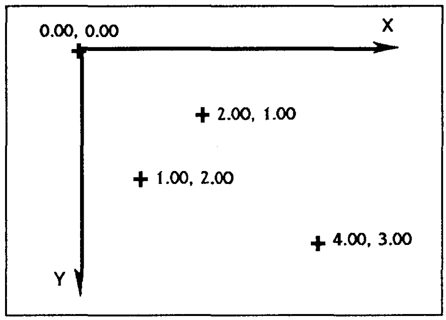
1. Press <Esc> to remove the **SET** menu from the screen . **Auto Pan** remains set to **Yes**.
2. Move the pointer to the lower right corner until the title block appears. The screen pans to keep up with the pointer. Notice the title block information that you entered earlier in this chapter.
3. Move the pointer toward the upper left-hand 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**.

X,Y Display

A coordinate system is used 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**.

Set X,Y display to YES

1. Select **X,Y Display**.

   "Display X,Y Coordinates of Cursor?" and a short menu appear.

2. Select **Yes**. The menu disappears.
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 are inches on the printed schematic. The upper left corner is (.00, .00) and the lower right corner is (9.50, 7.00). So, on a sheet 8.5 inches by 11 inches, the drawing area is 7 inches by 9.5 inches to allow borders around the drawings.

Worksheet size

The **Worksheet Size** command selects one of five sizes for your schematic.

Select worksheet size

1. Display the main menu, then select **AGAIN**. The **SET** menu displays.
2. Select **Worksheet Size**. A menu displays the five options available for the size of a worksheet, as shown at right.
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 dimensions of the worksheet's borders. On a C-size sheet (actually 22 inches by 17 inches), the drawing area is 20 inches by 15 inches.

```
Set Worksheet size
(Area inside borders)
A    9.50 x  7.00
B   15.00 x  9.50
C   20.00 x 15.00
D   32.00 x 20.00
E   42.00 x 32.00
```

△ *NOTE: If* ***Schematic Design Tools*** *is configured to use metric dimensions, the* ***Set Worksheet size*** *menu displays the International Standards Organization paper sizes: A4 through A0. For information about configuring* ***Schematic Design Tools*** *to use metric dimensions, refer to* Chapter 1: Configure Schematic Design Tools *in the* Schematic Design Tools 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 finished schematic.

ZOOM in and out

To zoom out and see more of the worksheet on the screen at one time, follow these steps:

1. Move the pointer to lower right corner until the title block appears.
2. Select **ZOOM** from the main menu. The **ZOOM** menu at right appears.
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 in the figure below.

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

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

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

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** commands on the **SET** menu let you set up some of these visual cues. **Grid Parameters** contains the three commands 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 grid guides divide the worksheet from 8 to 1. Vertically, the guides divide the worksheet from D to A. For example, the title block (lower right corner) is located at A-1. You use **JUMP Reference** to move to specific locations using these map-like coordinates.

1. Select **SET** from the main menu to change the grid display.
2. Select **Grid Parameters.**
3. Select **Grid References,** then select **Yes.** The grid reference bars appear at the top and left edges of the display.

△ *NOTE: Schematic Design Tools can be set up to use ANSI Y14.1 drawing standards. Refer to the* Schematic Design Tools 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: Stay on grid unless you have a compelling reason to be off-grid. Anything placed off-grid—such as text and labels—may be hard to select later if you want to edit it.*

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.

1. Select **SET** and **Grid Parameters** again.
2. Select **Visible Grid Dots,** then select **Yes**. Grid dots appear on the worksheet. You may want to adjust the intensity on your monitor to make the grid dots brighter or dimmer.

Updating the worksheet

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

Update the file

To save the worksheet without changing its filename, follow these steps:

1. Select **QUIT** from the main menu.

   Draft displays the filename and the **QUIT** menu, as shown at right.

```
Quit TUTOR.SCH
Enter Sheet
Leave Sheet
Update file
Write to file
Initialize
Suspend to System
Abandon Edits
```

2. Select **Update File. Draft** saves the file.
3. Press <Esc> or click the right mouse button to return to the main menu level.

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. Earlier in this chapter, you used the **SET** command to change work conditions parameters. To capture these commands in a macro that can be repeated each time you press <Ctrl> <A>, follow these steps:

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

1. Select **MACRO** from the main menu. The **MACRO** menu is shown at right.
2. Select **Capture** to record a macro. The prompt "Capture macro?" appears. You can assign a number of keys and key combinations to run macros. Single keys that can run macros are the function keys (<F1> - <F10>) and keys in the numeric keypad with text on them such as <Home>, <Page Up>, and <Page Down>. Key combinations that can run macros include:

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

   If you choose a prohibited key combination, **Draft** rejects it.

3. To assign a keystroke to this macro, press <Ctrl><A>. **^A** appears at the "Capture Macro?" prompt.
4. Press <Enter>. The message "<macro>" appears on the screen to remind you that you are defining a macro. Any commands you select while "<macro>" displays are added to the list of commands being stored in the macro.

5. Type the commands in the left column below:

<Enter>	Responds to the "load file?" prompt
`S X Y`	Commands to see the X,Y coordinates
`S G G Y`	Commands to see grid references
`S G V Y`	Commands to see the grid dots
`Z S 1`	Commands to set the viewing scale to full size ("zoom in" to the most detail).

6. Press <Ctrl><End> to end the macro definition. **Draft** confirms the macro definition is complete by displaying:

```
<<<MACRO END>>>
```

The macro is now stored in the computer's memory. You can run it anytime by pressing the key combination you specified, <Ctrl><A>.

△ *NOTE: Some keyboards have two keys labeled <End>. On a few of these, you must use the <End> key in the numeric keypad.*

Save the macro

Now, save the macro in a file.

1. Select **MACRO Write. Draft** displays:

```
Write all macros to?
```

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. Draft** displays:

```
Read all macros from?
```

4. Enter `tutor.mac`.
5. To test the macro you just saved, change some of the work conditions parameters and press <Ctrl><A> to restore the work conditions.

Exiting Draft

You are nearly done with this chapter. To exit **Draft**, follow these steps:

1. Select **QUIT** from the main menu.

   Draft displays the filename and the **QUIT** menu shown at right.

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

2. Select **Update File. Draft** saves the file.
3. Leave **Draft** by selecting **Abandon Edits. Draft** exits to the **Schematic Design Tools** screen.

Setting up automatically

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

View the configuration

To see the **Schematic Design Tools** configuration screen, perform the following steps:

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. Pan to the **Macro Options** portion of the **Configure Schematic Design Tools** screen.
4. To specify the name of the macro file you created earlier in this chapter, position the pointer within the **Draft Macro File** entry box and press <Enter> or click the mouse button. The pointer becomes a cursor inside the entry box. Enter the macro path and filename **`\orcad\tutor\tutor.mac`**.

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

5. Notice that the **Draft Initial Macro** entry box became highlighted once you made an entry in the **Draft Macro File** entry box.

   To define an **Initial Macro** that automatically runs when you run **Draft**, position the pointer within the **Draft Initial Macro** entry box and press <Enter>.

6. Enter the keystrokes used to execute the initial macro: <Ctrl> <A>. However, instead of pressing the <Ctrl> key, simultaneously press <Shift> and <6> to enter the "caret" symbol and press <A>. The caret symbol (^) is used to represent the <Ctrl> key.

```
Draft Initial Macro [^A                    ]
```

7. To update the configuration and return to the **Schematic Design Tools** screen, move the pointer to the top of the screen and click **OK**.

△ *NOTE: Once the ^A macro is defined and configured in the initial macro entry box on the **Configure Schematic Design Tools** screen, it runs automatically each time you run **Draft**. You can also run it at any time by pressing <Ctrl> <A>.*

Summary

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

The next chapter gives you detailed 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**. Now, it's time to start capturing schematic diagrams.

Capturing the clock oscillator schematic

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

- ❖ Get and place library components
- ❖ Draw and place 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.

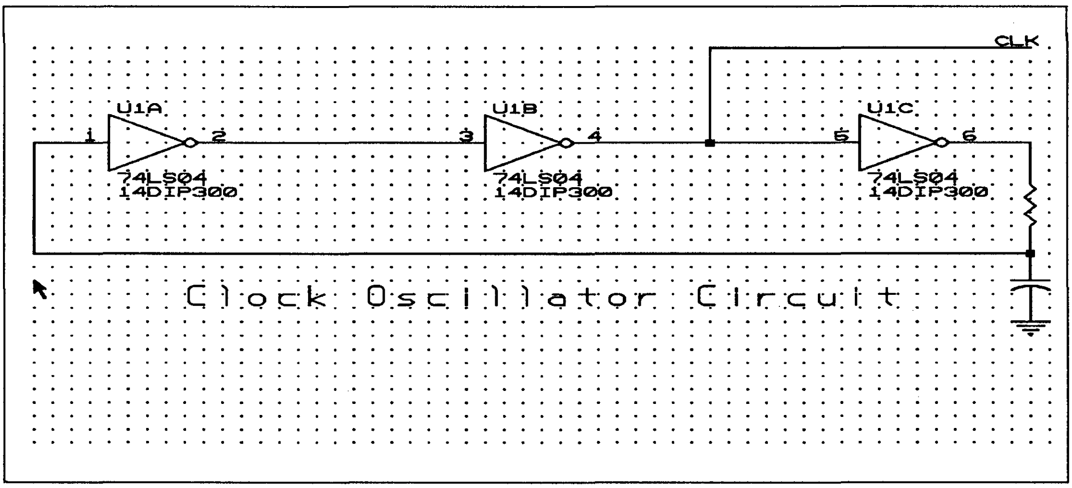


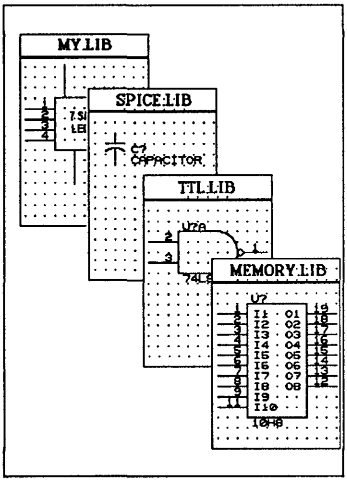
Figure 3-1. Clock oscillator circuit schematic.

About symbols

The first step in building a schematic diagram with **Draft** is to place symbols for the components on the worksheet. The symbols can represent basic logic functions (such as AND gates), discrete components (such as capacitors), or blocks of circuitry to be designed later. The symbols can represent components 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. If it is not displayed, follow these steps.

1. If the operating system prompt is displayed, type **ORCAD** <Enter>.

△ *NOTE: In chapter 1, you set the startup 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 design environment screen, click the **Schematic Design Tools** button and then select **Execute.**

Check library files

1. On the **Schematic Design Tools** screen, click the **Draft** button.
2. Select **Configure Schematic Tools.**

   The **Configure Schematic Design Tools** screen displays. Pan down until you can see the **Library Options** portion of the screen.

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

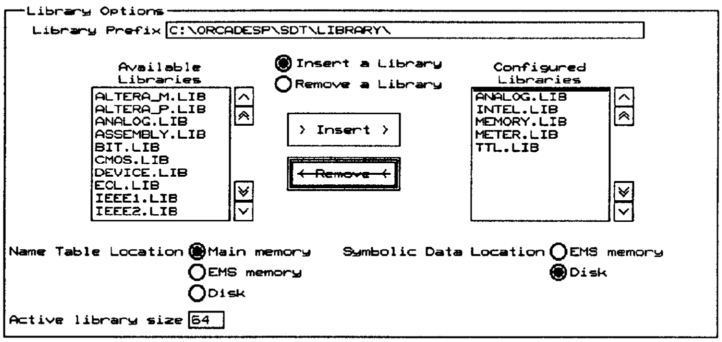


Figure 3-2. ***Library Options*** *portion of* ***Configure Schematic Design Tools*** *screen.*

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.

3. For this chapter, **Draft** needs the library files PCBDEV.LIB and .\DCLOCK.LIB. PCBDEV.LIB must be the first library listed in the **Configured Libraries** box.

   Scroll the **Available Libraries** list box up and down by clicking on 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.

4. Locate PCBDEV.LIB in the **Available Libraries** list box and click on it to select it. Click the `>Insert>` button to place PCBDEV.LIB in the **Configured Libraries** list box.
5. Repeat step 4, but this time select .\DCLOCK.LIB.

△ *NOTE: If any libraries other than PCBDEV.LIB and .\DCLOCK.LIB are listed in the **Configured Libraries** list box, remove them by clicking the **Remove a Library** button, selecting the library to remove, and then clicking the* `<Remove<` *button.*

6. Pan to the top of the configuration screen (or press the <Home> key) and click **OK** to return to the **Schematic Design Tools** screen.
7. Click the **Draft** button, and select **Execute** from the menu that displays.

   Draft plays the initial macro you defined in chapter 2. This macro causes X,Y coordinates, grid references, and grid dots to be displayed, and the viewing scale to be set to full size. When the macro is done playing, a blank worksheet displays.

Placing parts

To get symbols from part libraries, follow these instructions:

```
Which Library?
PCBDEV.LIB
.\DCLOCK.LIB
```

1. Select the **GET** command from the main menu. "Get?" appears.
2. Press <Enter> to display the **Which Library?** menu. The menu above shows the libraries defined for the TUTOR design.

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

3. Move the highlight to .\DCLOCK.LIB and press <Enter>. A list of the parts stored in the .\DCLOCK.LIB file displays, as shown in the menu at right.
4. Select the 74LS04 inverter. An image of the part appears on the worksheet and a command line appears across the top of the screen.

   When you move the part, the image *simplifies* temporarily. This means that only the object's outline appears. When you stop moving the part, details reappear.
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 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. This places the part

   Draft places the part on the worksheet and creates another movable image of the part. To add a second part of this type to your schematic, you move and place the new image elsewhere on the sheet.

   When you don't need another copy of the part you just placed, you can press <Esc> to end the operation.

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). A quick way to place the part is to press <P>.
9. When you have placed all three parts, press <Esc>.

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

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

1. Enter <G><R> to select the resistor component from the PCBDEV.LIB library. An image of the resistor appears on the worksheet.
2. Place the resistor at location (17.60, 12.30).
3. Press <Esc>.
4. Press <G> to display the "Get?" prompt. Enter **CAP**. An image of the capacitor appears 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 appears on the worksheet.

8. Place the ground symbol two grid spaces below 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.

Placing wires

Compare your worksheet with figure 3-1. Your worksheet should contain the parts shown in figure 3-1, but not the wires.

Most of the remaining tasks in this chapter have to do with establishing signal connections between the parts you placed on the worksheet.

Place wires

To place wires on your schematic, follow these steps:

1. Select **PLACE** from the main menu. The **PLACE** menu appears.
2. Select **Wire**. The **PLACE Wire** command line appears.
3. Move the pointer until it rests at the free end of the output pin of the left-most 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, place wires between the remaining components as shown in figure 3-1.

   You can speed up wire placement two ways:

   ❖ Select **New** instead of **End** for each wire except the last one.

   ❖ Instead of using menu selections, use the keyboard commands <P> <W> <B> <N> and <E>.

△ *NOTE: When placing wires, be sure to begin and end each wire segment at the end of a component pin, not within the body of the pin. Also be sure that the end of a wire does not overlap a pin.*

Placing junctions at intersections

Crossing wires do not represent a connection. To tell **Draft** the crossing wires are connected, you must define the intersection as a wire junction. You do this by placing a junction at the intersection. However, if two wires (or a wire and a component pin) are connected end-to-end, a junction is not necessary.

The connection between the resistor-capacitor junction and the input of the left-most inverter requires a junction. This is not necessary for the connection between the capacitor and the ground wire since they connect end-to-end.

A junction is also required where the wire connects between the middle and righthand inverters and ends at a point above and to the right of the rightmost inverter and is labeled CLK in figure 3-1.

Place junctions

To place a junction, follow these steps:

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

You aren't finished with this circuit yet, because you still have to assign values to the resistor and capacitor, add a signal label, and assign reference designators to all the parts.

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 want to record part numbers on the schematic to make it easier to track and order parts from manufacturers. Or you may want to specify the physical package 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" or "Q1."
- The **Part Value** field is reserved for holding part names, such as "74LS04" or values relevant for the part, such as Ohm (Ω) values for resistors.

The other eight fields are named **1st Part Field** through **8th Part Field.**

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

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. Alternatively, you can automate part field editing using the **Update Field Contents** tool. You will learn how in chapter 7.

Edit part fields

To specify the package type of the inverters, follow these steps:

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 appears. This menu is shown at right.

```
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
8th Part Field
Orientation
Which Device
SheetPart Name
```

4. Select **1st Part Field**. The menu shown at right appears.

```
1st Part Field
Name
Location
Visible
```

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

```
1st Part Field?
```

6. Enter **14DIP300**. After you do, this information displays below the inverter symbol.
7. Select **Which Device** from the **Edit Part** menu.

   A list of letters (A through F) displays. The library contains the number of devices in each TTL package type, and so **Draft** knows there are six inverters in the 74LS04 type, and therefore displays six letters from which to choose. If there is only one device per package, the **Which Device** menu item does not display.
8. Select **A** from the list and press <Esc>. By default, all devices in the package are initially "device A."
9. Repeat steps 2 through 8 for the other inverters you placed. Since they are from the same type of package, enter **14DIP300** for each, and assign device letters **B** and **C** to them.
10. Press <Esc> twice to remove the menus from the screen.

About reference designator assignments

Notice the reference designators change to U?B and U?C, respectively. U?A is the first part in the package, U?B is the second part in the package, and U?C is the third part in the package. When you run the **Annotate Schematic** tool on this schematic, it changes all of the question marks for this package to a common number, such as 4. The parts will then be labeled U4A, U4B, and U4C. The **Annotate Schematic** tool is described in *Chapter 7: Using Schematic Design Tools.*

You also can edit the reference designator and part values displayed for a part, but doing so prevents **Annotate Schematic** from performing this task. **Annotate Schematic** automatically assigns device numbers and reference designators to the parts on the schematic.

Edit part fields for the remaining parts

To edit the part fields for the resistor and capacitor, follow these steps:

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

```
Value? R
```

5. To change the value, backspace over the present value and enter `91K`.
6. Press <Esc> twice to clear the menus from the screen.
7. Using these six steps, change the part value of the capacitor, measured in microFarads (**uF**), to `47uF`.

You are nearly finished with the schematic for the clock oscillator circuit. In the next section, you learn to specify a connection to the unconnected wire in your circuit using a label. The label allows another remote circuit on the same schematic 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 draw a line representing the wire connecting them. You can do this by assigning a label with the same name to both wires.

Add a label

To add a label to a wire, follow these steps:

1. Select **PLACE** from the main menu.
2. Select **Label.** At the "Label?" prompt, enter `CLK`. The label appears.
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 next to the bus or wire.
4. Select **Place**. The "Label?" prompt reappears.
5. Press <Esc>.

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

Placing comment text

You may often want 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.

Add a title

To add a title to this circuit that tells its function, follow these steps:

1. From the main menu, select **PLACE**, then **Text**. "Text?" displays.
2. 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**.
5. Press <Esc>.

△ *NOTE: You may wish to use the* ***ZOOM*** *Center commands 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**, then **Abandon Edits**. When you select **Update file**, the file is saved in TUTOR.SCH. **Draft** exits and the **Schematic Design Tools** screen displays.

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

This chapter assumes you completed chapter 3. 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
- Define 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.

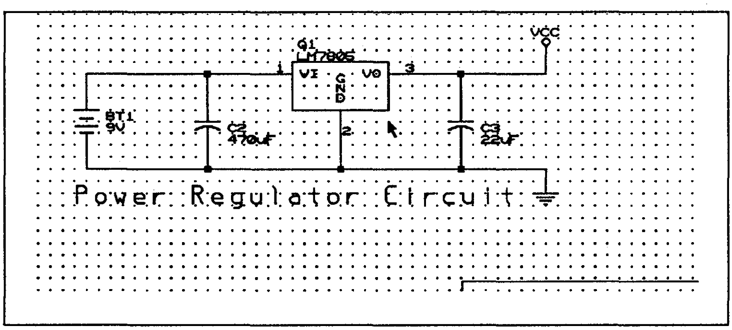


Figure 4-1. Power regulator circuit schematic.

Continuing schematic capture

If you did not abandon edits at the end of chapter 3 you do not need to re-run **Draft**, so skip steps 1 and 2 below.

1. From the **Schematic Design Tools** work screen, click the **Draft** button.
2. Select **Execute**. The worksheet view of the TUTOR.SCH schematic that was last active 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.

1. To change the scale from one to five, select **ZOOM Out** twice, or **ZOOM Select 5**. The entire worksheet appears.
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 to the right and below 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. Use **ZOOM** to return to a one-to-one scale. 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 components:

- 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 (.\DCLOCK.LIB) contains the parts you need to construct the power regulator circuit.

Get library parts and place them on the worksheet

1. Select **GET** from the main menu.
2. Press <Enter> at the "Get?" prompt, then select **.\DCLOCK.LIB**.
3. The parts menu displays. Select an LM7805 (an IC regulator) and place it at location (15.00, 12.50), as shown in figure 4-1.
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

For practice, delete the capacitor on the right side of the IC regulator.

1. Select **DELETE**. The **DELETE** menu appears, as shown below.
2. Select **Object**. The **DELETE Object** command line appears.
3. Place 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. Press <Esc> to tell **Draft** to redraw the screen, and the extra dots disappear.

Recover a deleted object

1. Select **DELETE** from the main menu again.
2. Select **Undo**. The capacitor reappears as it was before it was deleted.

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.

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

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

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 the pin 1 end of the part has a long heavy line. The pin 2 end has a shorter heavy line. 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 place the wires for the power regulator circuit.

Placing wires

Compare your worksheet with figure 4-1. In this section, you learn how to draw multi-segment wires in one operation. A multi-segment wire is a single wire that changes direction several times.

Draw a multi-segment wire

1. Select **PLACE Wire.** The **PLACE Wire** command line appears.
2. Move the pointer to the negative terminal of the battery, and select **Begin.**
3. Move the pointer down approximately three grid spaces.
4. Select **Begin** again and move the pointer to the right until it is directly under the first capacitor.
5. Select **Begin** again and move the pointer to the end of the capacitor pin.
6. Select **End** or **New.** When you draw multi-segment 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 components as shown in figure 4-1. Be sure to **Begin** and **End** each wire segment at the end of a component pin, not within the body of the component.
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>.

Or, you can 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.

This is a simple example that shows how to create a macro. You can extend the principle to create complex macros, automating long command sequences.

Write a macro to begin wires

1. Select **MACRO**. The **MACRO** menu appears.
2. Select **Capture**. The "Capture macro?" prompt appears.
3. Press <F1> to assign a keystroke to this macro. "F1" appears at the "Capture macro?" prompt.
4. Press <Enter>.

   The message "<macro>" appears to remind you that you are defining 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>>>" appears.

The macro you defined is now stored in the computer's memory and can be run at any time by simply pressing the key you specified, in this case, <F1>.

Save the macros

By the time your schematic capture session ends, you may have a set of macros you have defined to help you do your job. By writing these macros to a file, you can reuse them in a later session without having to redefine them.

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

```
tutor.mac
```

You just saved this macro to the macro file that automatically loads each time you start **Draft**. You can add more macros to this file as you define them.

Placing power symbol

Now place the power symbol in the power regulator circuit.

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

```
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. To change the power symbol's orientation, select **Orientation.** The **Orientation of Power Value** menu appears (shown below).

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

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

See the *Schematic Design Reference Guide* for detailed information on the display options available for the power symbol.

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

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.

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** to move the circuit. Notice that the lower wires grow and remain connected to the ground wire.

Next you assign values and reference designators to all the parts, and name this portion of the schematic before continuing on to chapter 5.

Editing part fields

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

Edit part values for the capacitors and battery

1. Place the pointer on the leftmost capacitor and change the part value to `470uF`. To review, the commands to enter are **EDIT, Edit, Part Value,** and **Name.**
2. Place the pointer on the rightmost 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

As in the previous chapter, a title isn't necessary for a circuit, but it is helpful when someone new needs to understand what a portion of circuitry does. Add a title to this circuit to describe its function.

Add a title

1. Select the **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. Place it.

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. The **JUMP** command is used to do this.

Jump to new coordinates

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

   - 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.

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

Follow these steps to practice using **X location.**

2. Select **X location**. The prompt "Jump X" appears. 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>. To move up and down, use the **Y location** command.

   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 is used to assign a tag to a location on the worksheet. Then the **JUMP Tag** command is used to move the pointer to that location.

1. Place the pointer on the power regulator circuit.
2. Select **TAG** from the main menu. The **TAG Set** menu appears, 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 assign it **B** tag.
5. Select JUMP, and from the JUMP menu, select **A** tag. The pointer jumps to the middle of the power regulator circuit, where you assigned the **A tag.**
6. Now jump to the **B tag.**

Making a draft-quality print

The last thing to do before ending this chapter is to print out a copy of the worksheet. While **Schematic Design Tools** includes the **Print Schematic** and **Plot Schematic** tools for making copies of entire designs, **Draft** also has a quick way to get a draft-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 program must be installed along with your other **Schematic Design Tools** software.

Update the file

1. Before you print the schematic, save your work using the command **QUIT Update file. Draft** updates the file TUTOR.SCH to reflect the current state of the worksheet.
2. Press <Esc> to return to the main menu.

Make a hardcopy of the worksheet

1. Make sure the printer is connected to your computer, powered on, and online.
2. From the main menu, select **HARDCOPY**. The **HARDCOPY** menu appears.
3. 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** returns to the **HARDCOPY** menu.
4. Select **Make Hardcopy. Draft** sends the display to the printer.

△ *NOTE: The size of the printed image depends on the printer driver* ***Draft*** *uses. With* ***HARDCOPY*** *(and the* ***Print Schematic*** *tool),* ***Draft*** *always produces an image at a resolution of 100 dpi (dots per inch). If the printer driver used prints at some other resolution, the image printed is changed by a fixed scale factor (100 dpi divided by the printer driver resolution). If the printer driver resolution is greater than 100 dpi, the printed image is smaller; if the driver resolution is less than 100 dpi, the printed image is larger.*

For example, if the printer driver you are using prints at a resolution of 300 DPI, the image printed on the paper is reduced in size by a factor of 100/300, or 1/3X. If the driver prints at 75 DPI, the image printed is enlarged by a factor of 100/75, or 1.33X.

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 Reference Guide.*

Ending a Draft work session

After you save your design and make a hardcopy of it, you are done with chapter 4. You can go on to chapter 5 or stop for the present. 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 the **Edit Library** tool to define a custom component to use in the display area of the digital clock schematic.

Creating a custom component

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

In this chapter, you learn how to:

- Run the **Edit Library** tool
- Redefine **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. Because this is an introduction, 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 Reference Guide.*

Configure Edit Library

Before running **Edit Library,** you must configure it to open the library file called .\DCLOCK.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**. The **Configure Edit Library** screen displays.
3. Look for the .\DCLOCK.LIB file in the **Files** list box. Click on it to select it. The name .\DCLOCK.LIB displays in the **Source** entry box:

```
Source  .\DCLOCK.LIB
```

4. Click the **OK** button to finish the configuration.

Run Edit Library

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

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

Setting up the work conditions

Like **Draft, Edit Library** lets you define certain work conditions. You adjust two features: one governs visibility of the border defining the part body. The other governs visibility of the grid 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. This displays the menu shown below.
3. Select **Show Body Outline.** "Show Bitmap Body Outline?" appears.
4. Select **Yes.**
5. Select **SET Visible Grid Dots Yes.** Grid dots appear in the work area.

Set

AutoPan	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

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

1. Press <Enter> to display the main menu.
2. Select **GET PART**. "Get?" appears.
3. Enter `TIL309`, the name of the part you plan to create.

   Edit Library displays "TIL309 - New Part?" and a short menu.
4. Select **Yes**.

   "Sheet Path?" appears. This is relevant when there is a schematic worksheet you want the part to reference when placed in a design.
5. Press <Enter> since the TIL309 part does not reference a schematic.

   The **Kind of Part?** menu appears. You use **Block** for simple rectangular parts, **Graphic** for more complex shapes, and **IEEE** for IEEE/ANSI drawing standard parts.
6. Select **Graphic** since the LED display is complex.

   "Number of Parts per Package" and a menu appear.
7. Select **1** since the seven-segment LED display has only one part per package.

   "Does Graphic Part have CONVERT?" appears. This tells **Edit Library** whether you will also create a DeMorgan equivalent part for the part you are creating.
8. 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 appears at the bottom right corner of the dotted border. The command line displays the choices **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)
11. To set the size of the editing pad, select **Place.** The **BODY** <**Graphic**> menu appears.

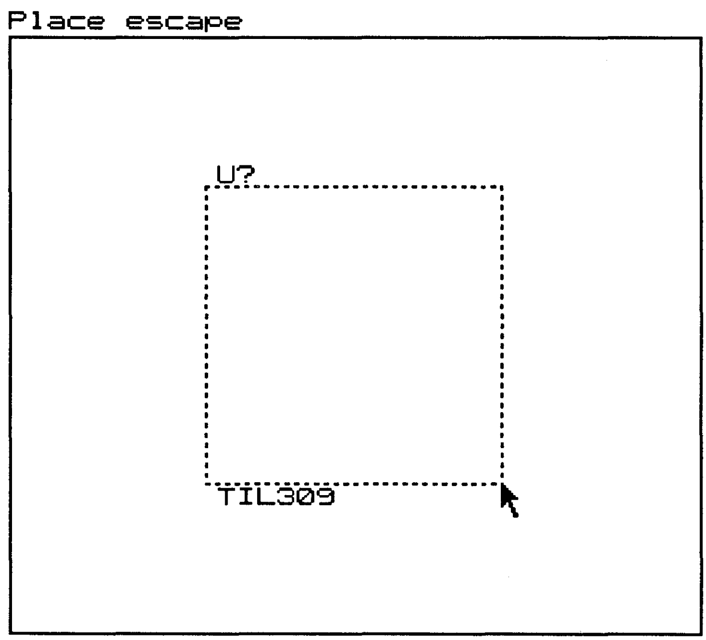


Figure 5-1. Part editing pad.

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

Drawing the body outline

1. Select **Line**. The **BODY Line** command line appears.
2. Move the pointer to the upper left corner of the body (location +.0, +.0).
3. Select **Begin**.

   Move the pointer to the next corner (location +12.0, +0).
4. Select **Begin** again.

   Move the pointer to the next corner (location +12.0, +12.0).
5. Select **Begin** again.

   Move the pointer to the next corner (location +.0, +12.0)
6. Select **Begin** one last time.

   Move the pointer to the first corner (location +.0, +.0).

   Select **End**. The **BODY <Graphic>** menu appears.
7. Press <Esc>.

Changing the reference designator

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

Change reference designator prefix to 'D'

1. Select **REFERENCE** from the main menu. The prompt, "Initial Reference Designator? U" appears. U is the current value.
2. Backspace over the U and enter <D>. The reference designator reflects the change immediately.

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.

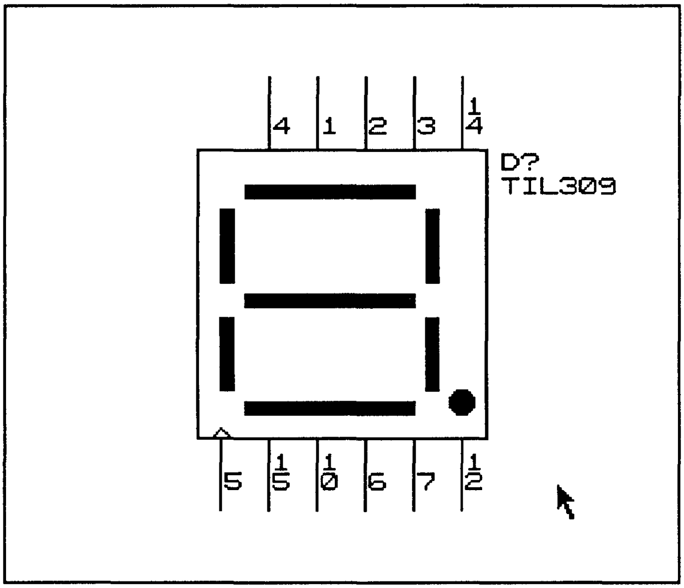


Figure 5-2. The part body you will create.

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. Try moving the pointer between the grid points.

Draw a rectangle to represent an LED

1. Select **BODY** from the main menu. This displays the **BODY <Graphic>** menu, as shown below.
2. Select **Line.** The **Line** command line appears.
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 Body
```

6. Select **Begin.** The line you drew changes color, showing 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. The **BODY** menu reappears.

Draw six more segments

Now, repeat this process for the remaining six rectangles that represent the LED segments. To ensure your LED comes out right, use the coordinates shown in table 5-1 to draw the remaining six rectangles. Each line of coordinates shown defines one rectangle, starting with the leftmost coordinate and then drawing each segment to the next coordinate shown.

	Top left	*Top right*	*Bottom right*	*Bottom left*
Segment 2	(2.0, 6.0)	(9.0, 6.0)	(9.0, 6.5)	(2.0, 6.5)
Segment 3	(2.0, 10.5)	(9.0, 10.5)	(9.0, 11.0)	(2.0, 11.0)
Segment 4	(1.0, 2.5)	(1.5, 2.5)	(1.5, 5.5)	(1.0, 5.5)
Segment 5	(9.5, 2.5)	(10.0, 2.5)	(10.0, 5.5)	(9.5, 5.5)
Segment 6	(9.5, 7.0)	(10.0, 7.0)	(10.0, 10.0)	(9.5, 10.0)
Segment 7	(1.0, 7.0)	(1.5, 7.0)	(1.5, 10.0)	(1.0, 10.0)

Table 5-1. Coordinates for rectangular LED segments.

You can capture the commands for one rectangle as a macro, and then run it for each rectangle of the same size and orientation you want to draw, only in different locations.

To do this, move the pointer to the coordinates for the top left portion of a rectangle. Then select **MACRO Capture** from the main menu. When the "Capture macro?" prompt appears, enter the key to be used to start the macro (such as <F2>). Now go ahead and draw the rectangle as described above. When the rectangle is complete, press <Ctrl> <End> to end the macro. To draw the next rectangle, move the pointer to the coordinates for the top left portion of the new rectangle and press the key you assigned to the macro (such as <F2>).

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 (11.00, 10.50).
3. Select **Center**. More commands appear, 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**. **Draft** 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** menu. The **Fill** command line appears.
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 return to the main menu level.

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 appears.
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?" appears. The pin name is an identifier not visible on the graphic representation of a part, but which **Draft** uses to identify particular pins on the part.
5. Enter the name `STROBE`. The **Edit Library** tool assigns the name. "Pin Number?" displays.
6. Enter <5>. The **PIN Type** menu appears. This pin conducts a clock signal to the internal logic of the part. It should be characterized as an input pin type.
7. Select **Input**. The **PIN Shape** menu appears.
8. Select **Clock** for pin shape. **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**. "Pin Name?" appears.
3. Enter the name `DPIN`. The **Edit Library** tool assigns the name. The prompt "Pin Number?" appears.
4. Enter `12`. The **PIN Type** menu appears.

This pin conducts a reset signal to the internal logic of the part. It should be characterized as an input type pin.

5. Select **Input** for pin type. The **PIN Shape** menu appears.
6. Select **Line** for pin shape. **Edit Library** places the pin and displays the pin number you assigned.

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**. "Pin Name?" appears.
3. Enter the name `QAIN`. The **Edit Library** tool assigns the name. The prompt "Pin Number?" appears.
4. Enter `15`. The **PIN Type** menu appears.

   This pin conducts a signal to an LED segment. It should be characterized as a passive type pin.

5. Select **Passive** for pin type by putting the highlight bar on this menu item, not by pressing <P>. This is because another menu item begins with "P" (**Power**) and appears in the menu before **Passive**; just pressing <P> selects **Power**, not **Passive**.

   When you have specified the pin type, the **PIN Shape** menu appears.

6. Select **Line** for pin shape.
7. Repeat these steps for the pins connected to the other LED segments. The table on the next page lists the coordinates, names, pin numbers, pin type and pin shape to use for the other pins. You have already defined the first three pins. Go ahead and start with the fourth pin.

Coordinates	Pin Name	Pin No.	Pin Type	Pin 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.

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 part involves two operations:

- Copy 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**.
- Write the modified library file in the computer's internal memory to disk. Use either **QUIT Update file** or **QUIT Write to file.**

Save the new part

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

Write the library in memory to a file on disk

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 .\DCLOCK.LIB, select **Initialize.** "Read Library?"displays.
3. Enter `.\DCLOCK.LIB`.
4. Select **LIBRARY List Directory Screen.** TIL309 should be in the list of parts in .\DCLOCK.LIB.
5. To leave the directory, press any key.

Get the new part

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

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 .\DCLOCK.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 are a natural progression from the processes that were introduced in the 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.

Choosing components

To build the rest of the digital clock schematic, you need these components:

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

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.

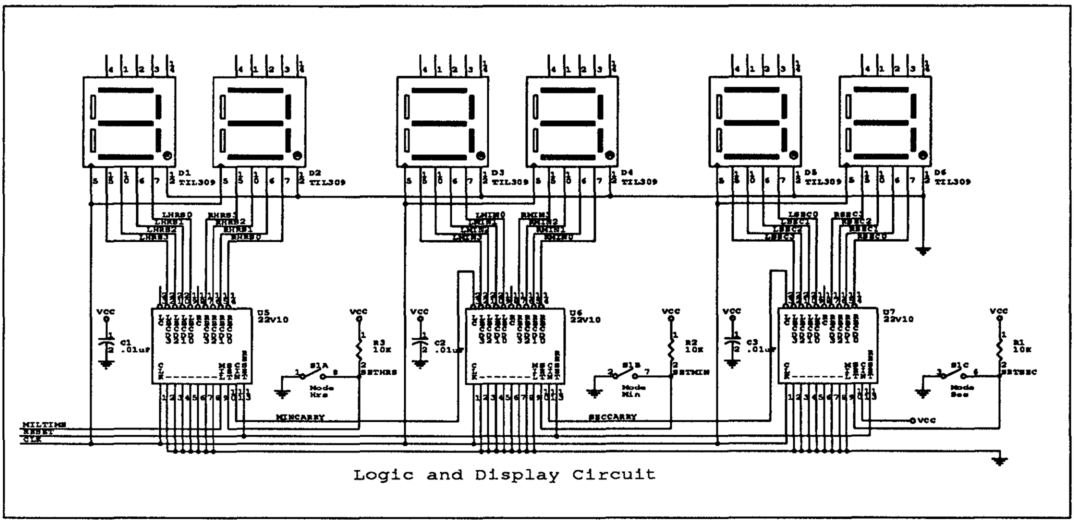


Figure 6-1. The logic and display circuitry.

The schematic requires enough pins to drive the six TIL309 display chips plus transfer carry signals. Once again, to keep the total chip count for the design down, 22V10s were chosen to drive the TIL309s rather than discrete components. 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 device 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 (.\DCLOCK.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.

Re-running Draft

Start **Draft** from the **Schematic Design Tools** screen. The worksheet view of the TUTOR.SCH schematic that was last active displays.

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 create the other areas.

First, move to an area of the worksheet with enough room to add the display and logic circuitry.

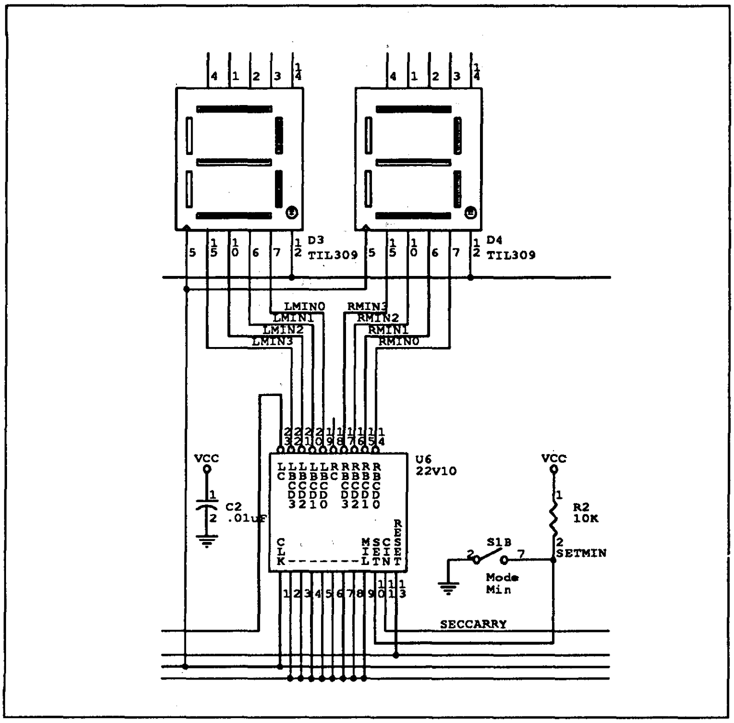


Figure 6-2. The minutes circuit.

Change viewpoint to a clear area

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 move the display window near the center of the worksheet.

The clock oscillator and power regulator circuits you captured earlier are in the lower right area (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 seems more difficult than it really is. Figure 6-2 shows only the components and wires associated with the Minutes area of the schematic. Compare figure 6-2 with figure 6-1 to see the similarities in each of the areas and then follow the steps below to build the minutes circuit.

Place the components

1. Select **GET**. The "Get?" prompt appears.
2. Press <Enter>. A list of the part libraries specified in **Schematic Design Tools'** configuration are displayed in a menu.
3. Select **.\DCLOCK.LIB**.

   A list of the parts in .\DCLOCK.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. Place the component at coordinates (10.50, 6.00).
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 components and only need to place wires, nets, and the power and ground symbols.

Place the wires

In order to perform the next steps, you need a close-up view of the schematic.

1. Select **ZOOM In.**
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 straight down so it is three grid spaces below the lower pins of the 22V10 component (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 component (11.50, 7.30).
7. Select **End** to end the wire.

Run the macro to place wires

The <F1> macro you defined earlier to start drawing a wire should still be active. Use the macro to place the following wires:

1. Referring to figure 6-2, place the wires between the 22V10 component and the right-hand seven segment display as shown. To begin a new wire, instead of issuing the <P><W><B> commands, just press the <F1> key. Then proceed as usual.
2. Continue using the macro and place the wires between the 22V10 component and the left-hand seven segment display as shown in figure 6-2.

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

Define REPEAT parameters

REPEAT duplicates the last entered object, label, or text string and places it on the worksheet.

1. To define where the duplicate will be placed, select **SET Repeat Parameters.**
2. Enter **1** for the **X Repeat Step**.
3. Enter **0** for the **Y Repeat Step**.

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

Change viewpoint to speed wire placement

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

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 the area you will be using on the worksheet.

Use REPEAT to speed wire placement

1. Place 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 shown in figure 6-2.
4. Draw a single horizontal wire along the bottom of these wires as shown in figure 6-2.
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 a lot longer to describe how to use the **REPEAT** command than it takes to use it. It's a good idea to plan your schematics to take advantage of the **REPEAT** placement capabilities of **Draft.**

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 done 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. Finish placing objects as follows:

1. Using **PLACE Power** and **Place,** 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 share common areas, there should be approximately a three-to-one time saving when you copy the circuit.

Save a schematic block

Before defining a block top copy, zoom out so you can see all of the objects you are working with.

1. Select **ZOOM Out** twice or **ZOOM Select 5** to change to the five-to-one scale.
2. Select **BLOCK Save. 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. Draft** saves the enclosed area in memory and returns to the main command level.

Copy a circuit

Now you can retrieve and place a copy of the minutes circuit.

1. Select **BLOCK Get**. An outline of the minutes circuit and a command line displays:

```
Place  Find  Jump  Zoom
```

2. 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 placed.*

3. After you place the copy of the circuit, the outline reappears so you can continue placing copies.
4. 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.

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.

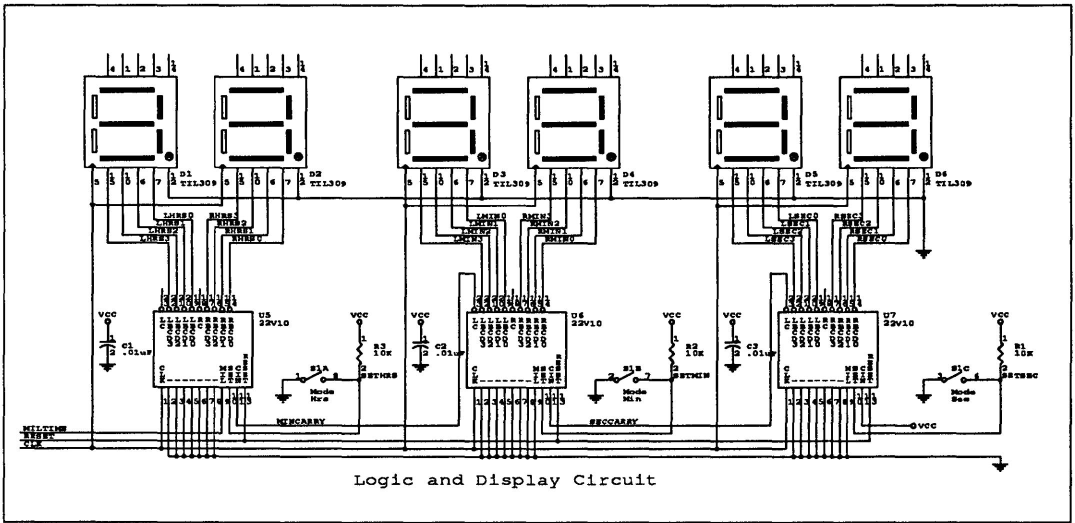


Figure 6-3. The logic and display circuitry.

Finish the wiring

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

1. Before beginning, move the pointer to the rightmost 22V10.
2. Select **ZOOM Select 1.**

Seconds circuit

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.

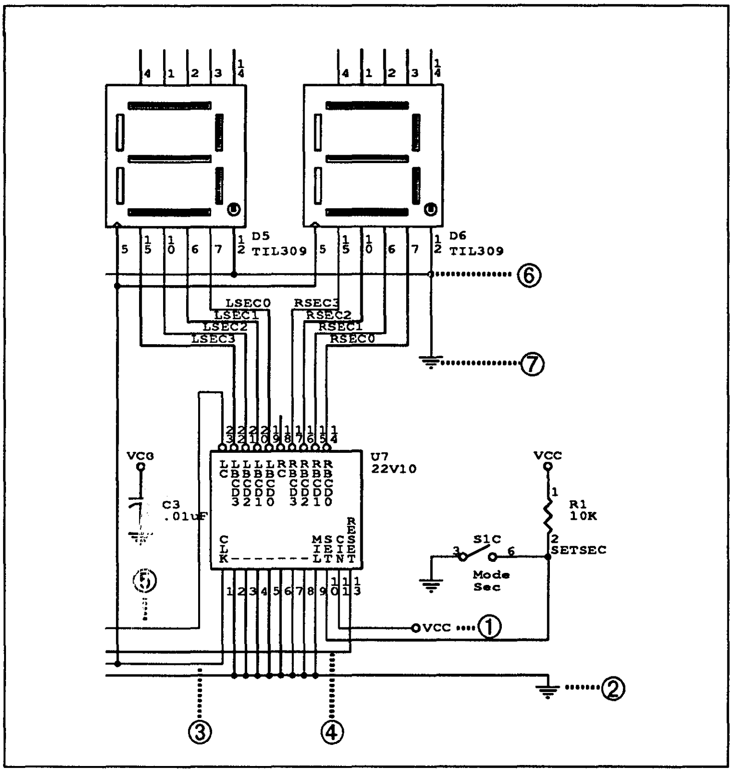


Figure 6-4. Seconds area of the clock logic. The callouts refer to the step numbers in this section.

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 end of this wire. Get a **GND** symbol from the library .\DCLOCK.LIB and place it at the end of this wire.

Place a wire at the left end of this wire to connect it to the 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. Extend the wire that comes from the rightmost TIL309's pin 12. Get a **GND** symbol from the .\DCLOCK.LIB library and place it at the end of this wire.

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

Minutes circuit

Before working on the minutes circuit, move the pointer to the middle 22V10 and select **ZOOM Center.**

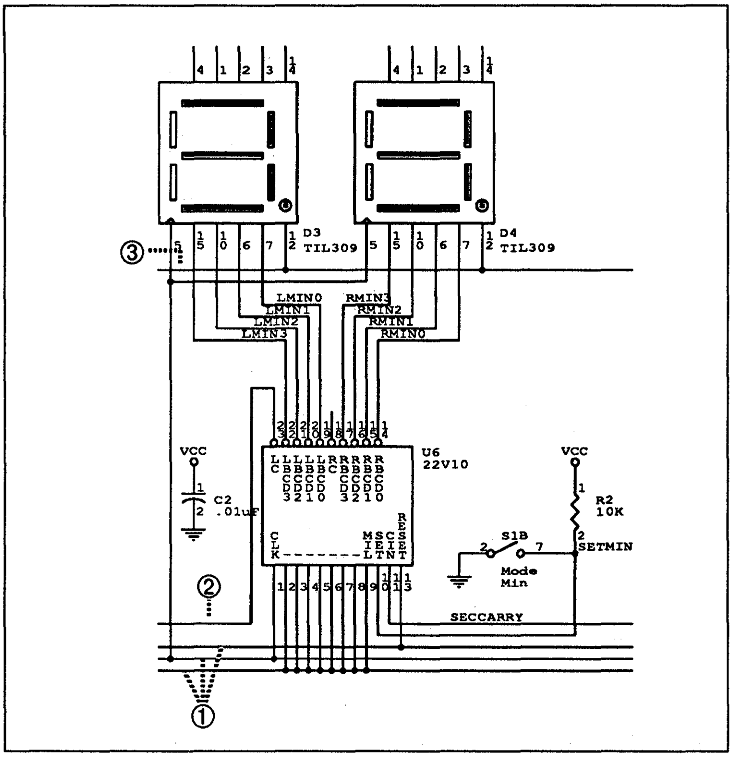


Figure 6-5. Minutes area of the clock logic. The callouts refer to the step numbers in this section.

1. Connect the bottom three horizontal wires to the hours logic (figure 6-5).
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.

Hours circuit

Before working on the hours circuit, move the pointer to the leftmost 22V10 and select **ZOOM Center.**

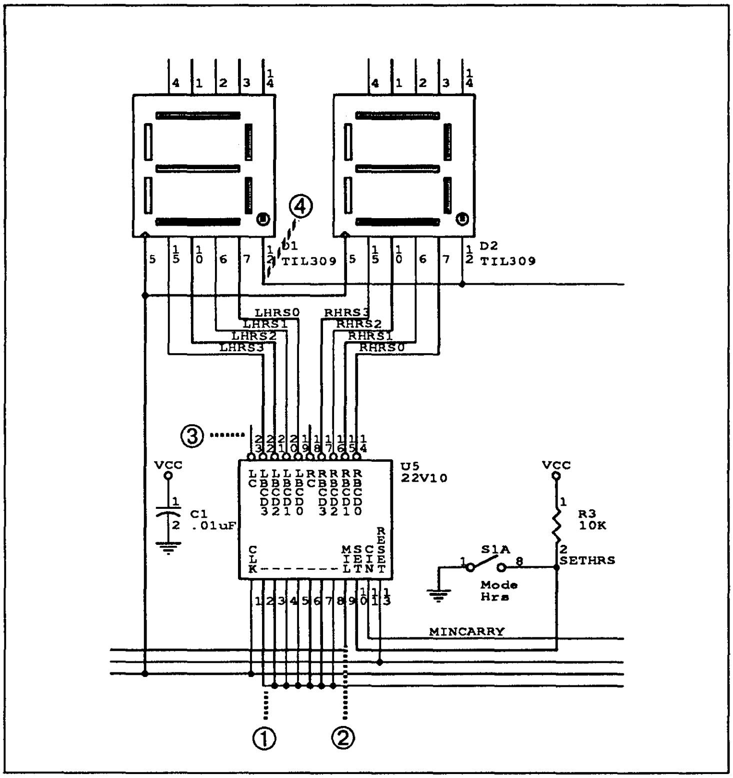


Figure 6-6. Hours area of the clock logic. The callouts refer to the step numbers in this section.

1. The bottom horizontal wire should end at the wire that extends from pin 2 of the 22V10 (figure 6-6).

   Delete this wire and redraw it so that it ends at the wire from pin 2 of the 22V10. Delete the junction at the end of the pin 2 wire also.

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 goes 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. Remove the portion of this wire that is to the left of pin 12. Delete the junction at the end of the pin 12 wire also.

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 on the next page shows how the schematic will look when you are through with the remaining steps in this chapter.

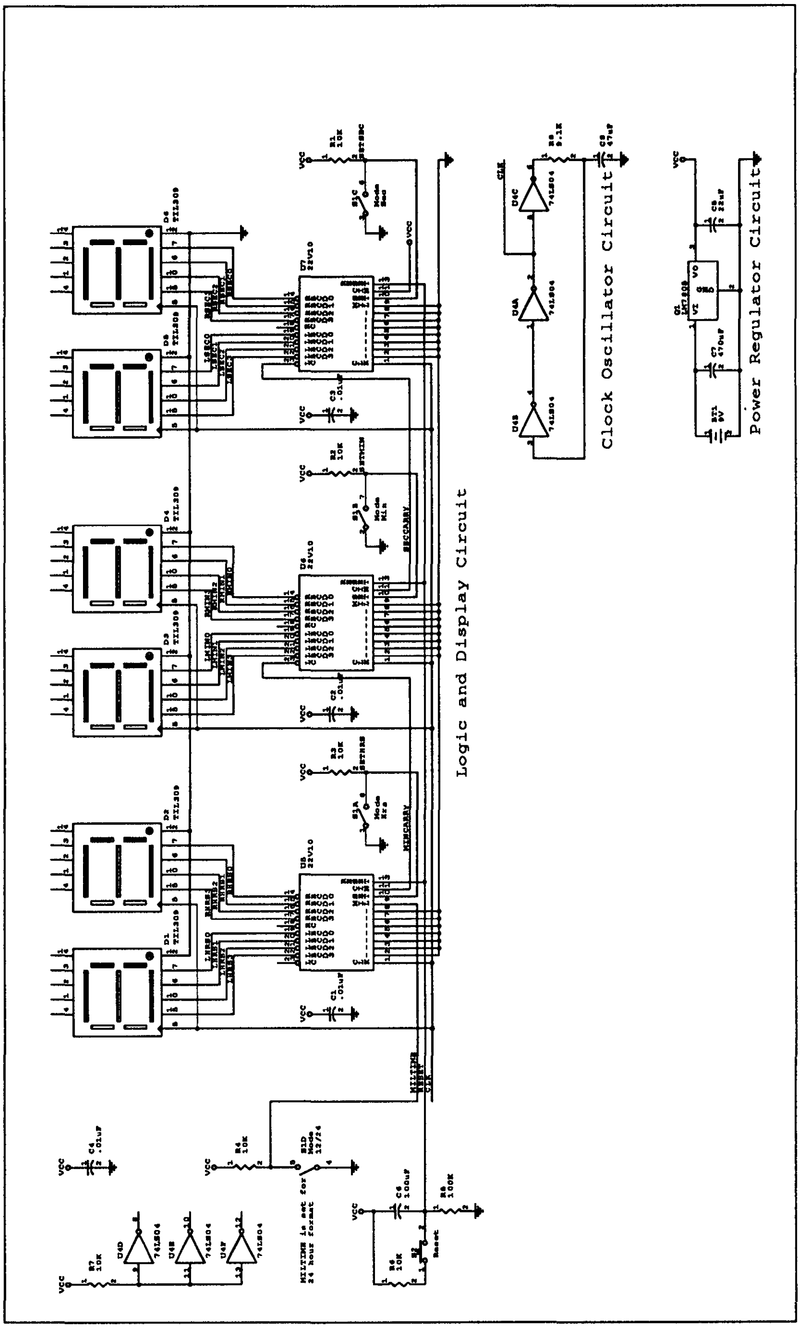


Figure 6-7. Completed TUTOR.SCH schematic.

Finishing the clock schematic

When you compare the schematic in figure 6-7 with the schematic you have captured so far, you can see you only need to add a few components and place a few more wires to have a functional circuit. You also need to edit the labels and other text in the schematic.

Place the remaining schematic parts

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

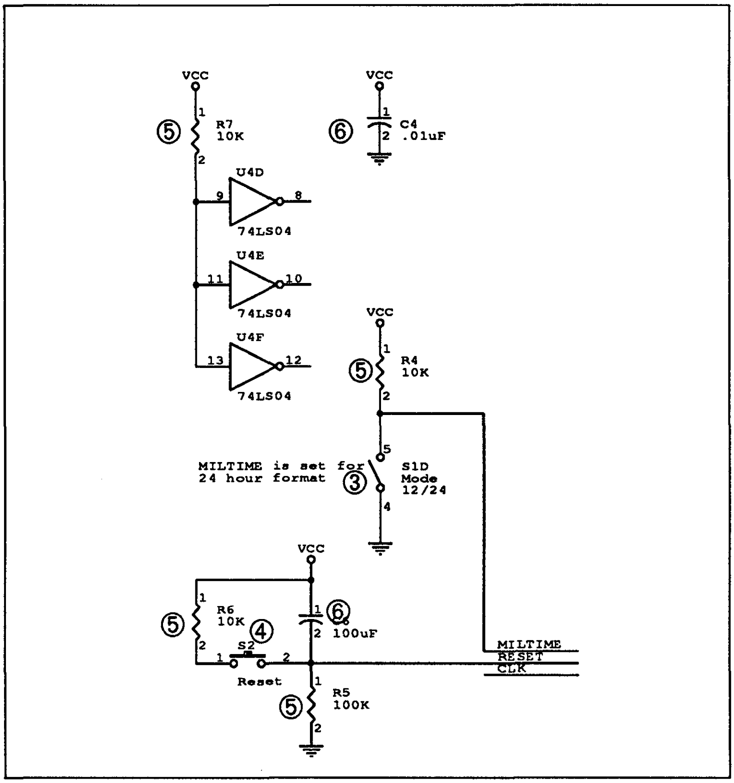


Figure 6-8. Switches, resistors, and capacitors to be placed. The callouts refer to the step numbers in this section.

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 .\DCLOCK.LIB.

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

   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. Now get the **SW PUSHBUTTON** component from .\DCLOCK.LIB and place it at location (1.20, 7.70).

5. Next get a resister (**R**) and place four copies at locations (.50, 2.70); (2.70, 4.80); (.50, 7.20); and (2.20, 8.20).

6. Finally, select a capacitor and place it in locations (2.70, 2.60) and (2.20, 7.20).

Place the extra parts

There are also some leftover parts (from multi-part packages) to be placed on the schematic. Figure 6-9 shows the leftover parts for this design.

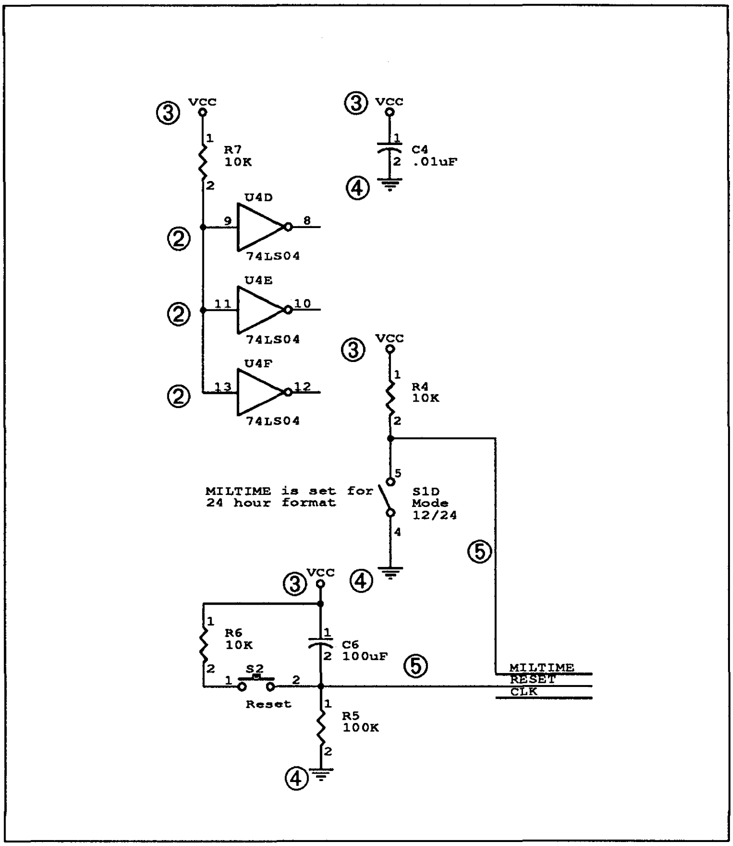


Figure 6-9. Inverters, power symbols, ground symbols, and wires to be placed. The callouts refer to the step numbers in this section.

1. Use **ZOOM** to change the scale to one-to-one. Move to reference grid D-8.

2. Get the **74LS04** inverter from .\DCLOCK.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.

   Place wires and junctions to connect the three inverters and resistor.

3. Place four **Power** symbols at locations (0.60, 2.60); (2.80, 2.50); (2.80, 4.70); and (2.30, 6.90).
4. Get a ground symbol from .\DCLOCK.LIB and place three copies at locations (2.70, 2.80); (2.70, 6.50); and (2.20, 8.60).
5. Place wires to connect the remaining components, 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 6-8.

Editing remaining text

Now you assign and edit labels, edit part values, and add comment text to complete the digital clock schematic.

Edit the part values

1. Put the pointer on the capacitor added at position 2.20, 7.20.
2. Select **EDIT Edit**. The **Edit Part** menu appears.
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 new 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 **1st Part field.**

Part	*Approximate Location*	*Old Part Value Name*	*New Part Value Name*	*New 1st Part field*
Capacitor	2.20, 7.20	CAP	100uf	
Capacitor	2.80, 2.60	CAP	.01uf	
Resistor	.60, 2.80	R	10K	
Resistor	.60, 7.30	R	10K	
Resistor	2.30, 8.30	R	10K	
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. "Label?" 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. (see figure 6-7).
4. The "Label?" prompt returns each time after 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 stop placing labels.

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

Set repeat text parameters

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

```
Set Repeat Parameters
X Repeat Step           +0
Y Repeat Step           +1
Label Repeat Delta      +1
Auto Increment Place    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 schematic.*

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.

Placing labels with repeat text

1. Select **PLACE Label** from the main menu. "Label?" displays.
2. Enter LHRS3.
3. Move the pointer to the bottom wire directly below the leftmost clock segment, and place the label as shown in figure 6-7.
4. Press <Esc> to return to the main menu level.
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 do steps 1 through 5 again.
7. See figure 6-7 and **PLACE** labels for the remaining left displays (**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 **2**, the **Y Repeat Step** to **1** (again, these values may vary depending on your wire spacing), and the **Label Repeat Delta** to **+1**.

3. Select **PLACE Label** from the main menu. "Label?" displays.
4. Enter **RHRS0**.
5. Move the pointer to the top wire for the right hours display, and place the label as shown in figure 6-7.
6. Press <Esc> to return to the main menu level of operation, and press <R> three times.
7. See figure 6-7 and **PLACE** labels for the remaining right displays (**RMIN***n* and **RSEC***n*) by repeating steps 3 through 6.

Add comment text

1. From the main menu, select **PLACE**, then **Text**.
2. At the "Text?" prompt, 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). Type <P> to place the text.
5. At the "Text?" prompt, enter:

```
MILTIME is set for
```

6. From the **PLACE Text** menu, select **Smaller** 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 (.80, 6.10).
8. When the "Text?" prompt returns, 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.

To get to the title block, use the mouse to move the pointer to the title block region of the worksheet. Another way to move there quickly is to use the **JUMP** command.

Jump to the title block

1. Select **JUMP** from the main menu. Then, select **Reference.**
2. In the **JUMP Reference** menu, 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

To add or change information in the title block, use the EDIT command.

1. Select **EDIT** from the main menu. The **EDIT** menu commands display.
2. Put the pointer somewhere within the title block. Select **Edit** from the **EDIT** menu. The **Edit Title Block** menu appears.
3. Select one of the types of information listed in the menu. For example, Select **Organization name.** A corresponding 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 you can delete the name and enter a new name.

   Once you press <Enter>, **Draft** stores the information and displays the **Edit Title Block** menu again, 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 procedure 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 done, press <Esc>. The title block displays the information you entered.

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**. **Draft** exits and the operating system prompt 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 other schematic 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	Assigns reference designators to parts in a schematic.
Check Electrical Rules	Checks for electrical rules violations.
Create Netlist	Generates a netlist and general wire list for a schematic in any of a number of standard formats.
Back Annotate	Updates part reference designators of parts in a schematic, based on a list of old and new reference designators.
Create Bill of Materials	Creates a list of all the parts used in a schematic or group of schematic sheets.
Plot Schematic	Plots a schematic or group of schematics in a batch mode. This tool 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 before completing the remaining exercises.

In addition, OrCAD has provided completed copies of the TUTOR schematic and library that you created in chapters 1 through 6. These files are called TUTOR2.SCH and .\DCLOCK2.LIB. Once you back up your design, you will copy these files to TUTOR.SCH and .\DCLOCK.LIB, respectively. This will allow you to use the tools in the remaining exercises with predictable results.

Backup Design

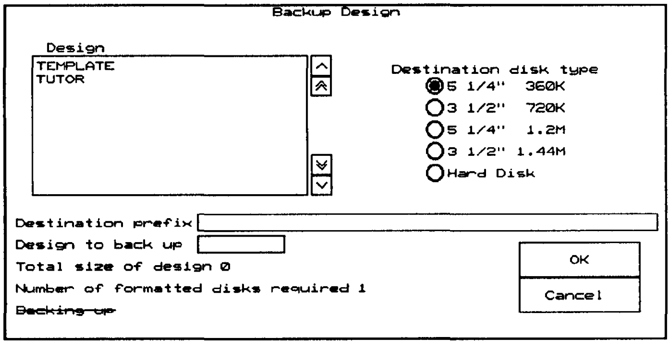
Use the **Backup Design** tool to back up all the files belonging to a design onto floppy disks or to another part of your hard disk. To conserve disk space, back-up files are stored in a condensed format. To restore the files to their normal format, you use the **Restore Design** tool, which is described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

To back up a design, follow these steps:

1. Click on the title bar. The menu shown at right displays.

```
Design Management Tools
Design Management Tools
Suspend to System
Exit
```

2. Select the **Design Management Tools** option. The **Design Management Tools** screen displays.
3. Click the **Backup Design** button. The screen shown on the next page displays.



Backup design screen.

4. Select the TUTOR design by clicking on its name in the **Design** list box.

5. Move the pointer to the **Destination prefix** entry box and press <Enter> or click the left mouse button.

6. Enter the path to use for the backup. To back up the design on a floppy disk, type the destination prefix `A:` and press <Enter>. The message shown below displays:

Insert a properly formatted disk in drive A and select **Continue**. Select **Cancel** if you want to cancel the backup for the time being.

7. Select **OK** from the **Backup design** screen. The environment makes a backup copy of the selected design in the disk or directory specified.

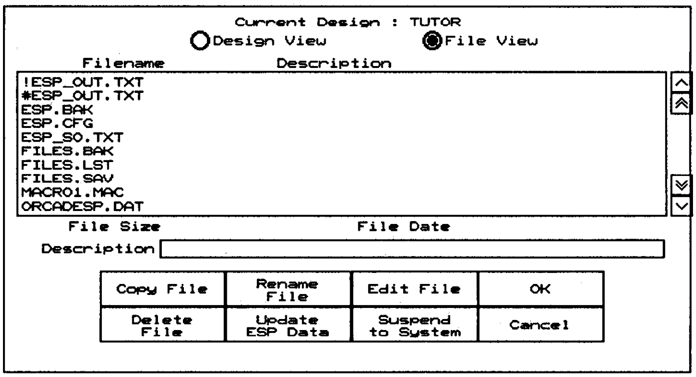
8. Once the design is backed up, the message "Backup successfully completed" displays along with an **OK** button. Click this **OK** button.

9. Click **Cancel** to return to the **Design Management Tools** screen.

Rename files

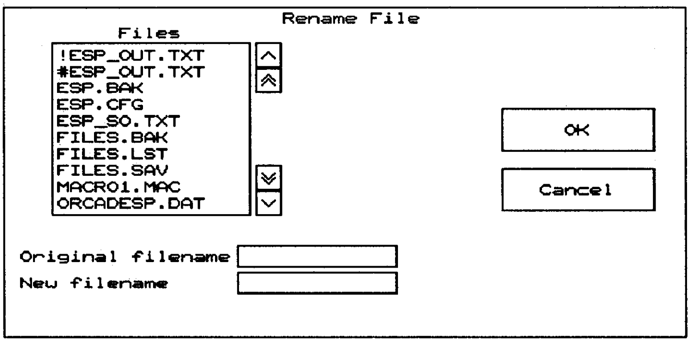
To rename the TUTOR2.SCH and .\DCLOCK2.LIB files to TUTOR.SCH and .\DCLOCK.LIB, follow these steps.

1. The **Design Management Tools** screen should still be displayed. Click on the **File View** radio button at the top of this screen. The screen shown below displays.



File view screen.

2. Click the **Rename File** button. The screen shown below displays.



Rename file screen.

3. Select the TUTOR2.SCH file from the **Files** list box. You will have to scroll the list to see this file name.

4. Move the pointer to the **New filename** entry box and press <Enter> or click the left mouse button.

5. Enter the new name for the file, **`TUTOR.SCH`**.

6. Select **OK** to rename the file.
7. Select the DCLOCK2.LIB file from the **Files** list box.
8. Move the pointer to the **New filename** entry box and press <Enter> or click the left mouse button.
9. Enter the new name for the file, `.\DCLOCK.LIB`, and then select **OK.**
10. 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 renamed TUTOR2.SCH and .\DCLOCK2.LIB to TUTOR.SCH and .\DCLOCK.LIB, you are ready to continue learning about **Schematic Design Tools.**

Running the Annotate Schematic tool

The **Annotate Schematic** tool updates worksheets with specific values for the reference designators and pin numbers of parts on the worksheet.

When you first place a part, a default reference designator value appears above the part, such as **U?** or **U?A. Annotate Schematic** changes the default values to unique values for each part in a specified design. Unique reference values are necessary for some other processes, such as producing a netlist. **Annotate Schematic** updates reference designators in the order in which they were placed on the worksheet.

Reference designator values are customarily used to designate which parts are to be grouped together in the same physical package.

For example, suppose the specified design contains three occurrences of the same part, and this particular part is manufactured with two parts per package. **Annotate Schematic** assigns values such as U1A, U1B and U2A. When layout of physical packages is performed, parts U1A and U1B would be referenced from a physical package identified as "U1." The "A" and "B" portions of the two values designate the unique identity of each part and its "slot" in the physical package. The U2A part would be referenced from a second physical package identified as "U2."

You can assign values of your choice using **Draft's EDIT** command, but assigning values using **Annotate Schematic** guarantees unique values.

Similarly, **Annotate Schematic** assigns appropriate, unique pin numbers to the pins of multiple instances of a part located in the same physical package.

Annotate Schematic modifies the worksheet file; but it creates a backup file containing the original worksheet file.

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 the information is reported in terms of the updated reference designators.

Run Annotate Schematic on TUTOR.SCH

1. From the **Schematic Design Tools** work screen, select **Annotate Schematic**. The menu at right displays.

```
Annotate Schematic
Execute
Local Configuration
Show Version
Configure Schematic Tools
Help
```

2. Select **Local Configuration** and then **Configure Annotate**. The **Configure Annotate Schematic** screen displays (figure 7-1).

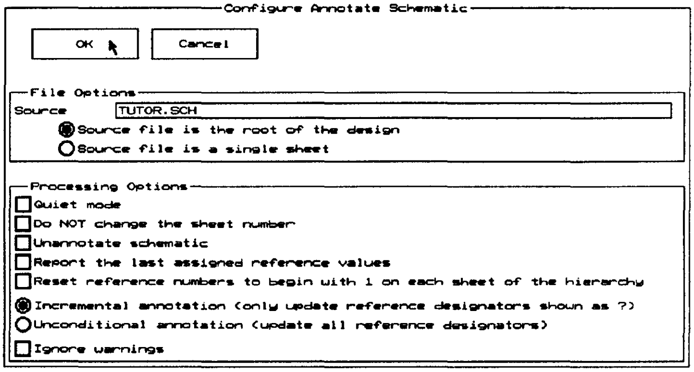


*Figure 7-1. **Configure Annotate Schematic** screen.*

3. Notice that the **Source** entry box contains the filename TUTOR.SCH. The design environment automatically sets the source to the design name and the default worksheet file extension found on the **Configure Schematic Tools** screen.

4. Now, click the **Source file is a single sheet** radio button.

5. Click the **OK** button.

6. 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.

4. Now, click the **Source file is a single sheet** radio button.
5. Click the **OK** button.
6. 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.

```
Annotation"   V4.00 01-DEC-90"
(C) Copyright 1985,1986,1987,1988,1989,1990 OrCAD L.P    ALL RIGHTS RESERVED.
Loading "\ORCADESP\SDT\LIBRARY\DCLOCK.LIB"
```

Status messages display at the bottom of the ***Schematic Design Tools*** *work screen.*

When it is finished, the text window disappears and the full **Schematic Design Tools** screen displays.

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

7. Run **Draft** and examine the TUTOR.SCH worksheet. Note the reference and pin numbers. Your reference designators may be slightly different than those shown in this tutorial. This is because **Annotate Schematic** assigns reference designators to parts in the order in which you placed them on the worksheet.

Notice the updated reference designators on the devices with multiple parts per package. For example the U?A on the 74LS04 inverters changed to U2A, U2B, and U2C. They are all parts of the same package, and their pin numbers changed accordingly.

Running the Check Electrical Rules tool

The **Check Electrical Rules** tool 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 take your designs through* ***Check Electrical Rules*** *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.*

Before running **Check Electrical Rules** on TUTOR.SCH, review its local configuration.

1. Display the **Check Electrical Rules Local Configuration** screen.

   Notice that the **Source** is TUTOR.SCH. You must also specify a **Destination**. When a destination file is specified, **Check Electrical Rules** stores the messages it creates in a text file with this name. You can specify a path to any directory and filename, but you really should place the file in the TUTOR directory.

   Storing the **Check Electrical Rules** tool report in a file is a useful practice, because then you can examine the output with **Edit File** or print it for reference as you examine your design.

2. Notice the filename **TUTOR.ERC** in the **Destination** entry box.

△ *NOTE: If you don't enter a destination name,* ***Check Electrical Rules*** *displays messages on your screen, instead of sending them to a file. If you like, try it both ways.*

Check Electrical Rules lets you know it is working by displaying a sequence of asterisks (*) and periods (.) on the screen.

After **Check Electrical Rules** is finished running, examine the file TUTOR.ERC with **Edit File**. The contents of the file should appear similar to figure 7-2.

```
Time Stamp - 14-FEB-1990     9:43:18

 "SHEET\TUTOR.SCH"

 LABEL REPORT
  (power) VCC
  (power) GND
     .
     .
     .
  MILTIME
  RESET
  CLK

 UNCONNECTED REPORT
  X=  1.90,Y=  1.20 Output        U1D,O
  X=  1.90,Y=  1.80 Output        U1E,O
     .
     .
     .
  X= 11.10,Y=  5.70 I/O           U3,RC
  X= 16.30,Y=  5.70 I/O           U4,RC

 Check Electrical Rules Report

 Digital Clock Schematic
 Revised: February 14, 1990
 Revision:
```

Figure 7-2. The TUTOR.ERC file.

The **Unconnected Report** shows some pins are unconnected. For example, consider the statement below:

```
X= 3.40 Y= 2.00 Output  U1,RCO
```

This means there is an unconnected signal at location (3.40, 2.00). It is further identified by its pin name, RCO, and by the reference designator of the part on which it is found, U1. Since the reported pins were intentionally not connected, you can ignore this information. 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 view the error message, place the pointer in the center of the circle and select the **INQUIRE** command from the main menu. Repeatedly selecting **INQUIRE** at the same location cycles through all of the error markers. The error message is displayed at the top of the screen.

When you are done looking at the schematic, select **QUIT Abandon Edits.** If you save the schematic file, the error markers are erased.

Running the Create Netlist tool

The **Create Netlist** tool generates a connectivity database and formats the interconnections in a number of possible formats. The format is specified when you configure the **IFORM** process of the **Create Netlist** processor.

To create a proper netlist, you must deal carefully 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 requirements, see the *Chapter 3: Guidelines for creating designs* in the *Schematic Design Tools Reference Guide.*

Generate a netlist in WIRELIST format

1. Configure the **Create Netlist** tool by selecting the **Create Netlist** button, then **Local Configuration.**

   The **Configuration** menu three has options to configure INET, ILINK, and IFORM. When creating a netlist, each of these process is used. INET is the incremental net connectivity database builder. ILINK is a connectivity linker, and IFORM is the netlist formatter. For more information on each of these processes, refer to the *Schematic Design Tools Reference Guide.*

2. Select **Configure INET**. The **Configure Incremental Netlist** screen (figure 7-3) displays.

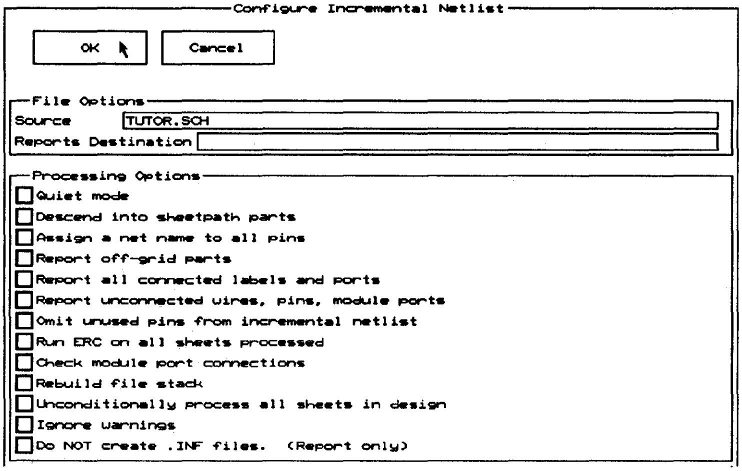


*Figure 7-3. **Configure Incremental Netlist** screen.*

3. In the **File Options** portion of the screen, check to be sure that the **Source** entry box contains the filename `TUTOR.SCH`. Again, this is automatically configured to be the root schematic file of the design.

4. Click **Cancel** to leave the configuration screen without making any changes and return to the **Schematic Design Tools** work screen.

5. Now display **ILINK**'s local configuration screen. Notice that the source is set to TUTOR.INF. Click **Cancel**.

6. Now display **IFORM**'s local configuration. **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** should already be set to TUTOR, showing that you will format the TUTOR database.

   Set the **Destination** to TUTOR.OUT.

7. The **Format Prefix/Wildcard** is set to:

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

   or something similar if you chose a different drive or path when you installed **Schematic Design Tools.** The **Netlist Format** list box contains a number of files. Edit the **Format Prefix/Wildcard** entry box and insert a "W" before the *, so that it becomes:

   ```
   Format Prefix/Wildcard
   C:\ORCADESP\SDT\NETFORMS\W*.CF
   ```

   The list box now contains far fewer filenames. Select WIRELIST.CF.

8. Press **OK** to accept all of the changes.

9. Now, run **Create Netlist** by selecting **Create Netlist** and then **Execute.**

   The **Create Netlist** tool lets you know it is working by displaying a sequence of asterisks (*) and periods (.). Figure 7-4 shows a wirelist format netlist of TUTOR.SCH.

10. Using **Edit File** or a word processor, look at the file generated by the **Create Netlist** tool. It should look like the file shown Figure 7-4.

```
Wire List

Digital Clock Schematic                    Revised:  November 1, 1990
                                           Revision:

<<< Component List >>>
.01UF                          C5         .01UF
.01UF                          C6         .01UF
.01UF                          C7         .01UF
.01UF                          C8         .01UF
100K                           R4         100K
100UF                          C4         100UF
10K                            R2         10K
10K                            R3         10K
10K                            R5         10K
10K                            R6         10K
10K                            R7         10K
10K                            R8         10K
22UF                           C3         22UF
470UF                          C2         470UF
47UF                           C1         47UF
74LS04                         U1         74LS04
9.1K                           R1         9.1K
9V                             BT1        9V
HRS                            U2         HRS
LM7805                         Q1         LM7805
MINSEC                         U3         MINSEC
MINSEC                         U4         MINSEC
MODE                           S1         MODE
RESET                          S2         RESET
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
        R8              2       2          Passive         10K
        U1              9       I_D        Input           74LS04
        U1             11       I_E        Input           74LS04
        U1             13       I_F        Input           74LS04
[00002] LHRS3
        D6             15       QAIN       Input           TIL309
        U2             22       LBCD3      BiDirectional   HRS
[00003] LHRS2
        D6             10       QBIN       Input           TIL309
        U2             21       LBCD2      BiDirectional   HRS
[00004] LHRS1
        D6              6       QCIN       Input           TIL309
        U2             20       LBCD1      BiDirectional   HRS
[00035] CLK
        U1              4       O_B        Output          74LS04
        U1              5       I_C        Input           74LS04
        D5              5       STROBE     Input           TIL309
        D6              5       STROBE     Input           TIL309
        U2              1       CLK        Input           HRS
```

Figure 7-4. Wirelist-format netlist (continued on next page).

```
        D4          5      STROBE   Input      TIL309
        U3          1      CLK      Input      MINSEC
        D1          5      STROBE   Input      TIL309
        D2          5      STROBE   Input      TIL309
        U4          1      CLK      Input      MINSEC
[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
        U1          7      GND      Power      74LS04
        R4          2      2        Passive    100K
        U2          3      -        Input      HRS
        U3          2      -        Input      MINSEC
        U3          3      -        Input      MINSEC
        U3          4      -        Input      MINSEC
        U3          5      -        Input      MINSEC
        U3          6      -        Input      MINSEC
        U3          7      -        Input      MINSEC
        U3          8      -        Input      MINSEC
        U3          9      MIL      Input      MINSEC
        U4          2      -        Input      MINSEC
        U4          3      -        Input      MINSEC
        U4          4      -        Input      MINSEC
        U4          5      -        Input      MINSEC
        U4          6      -        Input      MINSEC
        U4          7      -        Input      MINSEC
        U4          8      -        Input      MINSEC
        U4          9      MIL      Input      MINSEC
        S1          4      1_D      Passive    MODE
        S1          3      1_C      Passive    MODE
        S1          2      1_B      Passive    MODE
        U4          12     GND      Power      MINSEC
        U3          12     GND      Power      MINSEC
        U2          12     GND      Power      HRS
        C7          2      2        Passive    .01UF
        C6          2      2        Passive    .01UF
        C5          2      2        Passive    .01UF
        S1          1      1_A      Passive    MODE
        D5          12     DPIN     Input      TIL309
        D6          12     DPIN     Input      TIL309
        D4          12     DPIN     Input      TIL309
        D3          12     DPIN     Input      TIL309
        D2          12     DPIN     Input      TIL309
        D1          12     DPIN     Input      TIL309
        D1          8      GND      Power      TIL309
        D2          8      GND      Power      TIL309
        D3          8      GND      Power      TIL309
        D4          8      GND      Power      TIL309
        D5          8      GND      Power      TIL309
        D6          8      GND      Power      TIL309
        C8          2      2        Passive    .01UF
```

Figure 7-4. Wirelist-format netlist (continued from previous page).

Running the Back Annotate tool

If you don't like the reference designator values assigned by the **Annotate Schematic** tool (or that you assigned manually), you need not re-open the worksheet and edit the reference designators one by one. There's a faster way.

The **Back Annotate** tool lets you change as many reference designators as you want in a design, all at once. 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 really want the values to be A1, A2, A3, and so on. In this example, you will run **Back Annotate** on the design schematic sheet, TUTOR.SCH.

Change reference designator values

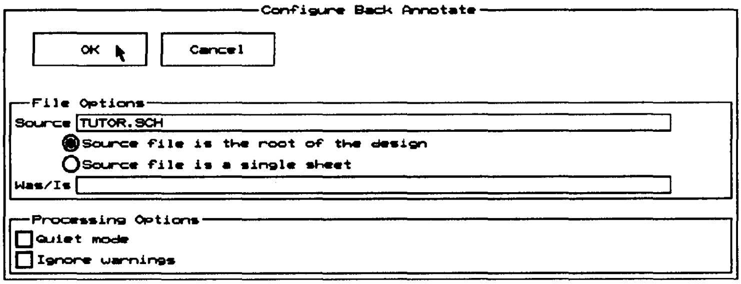
1. Create a text file using **Edit File.** Name the file NEWREF. Click the **Edit File** button. See the *ESP Design Environment User's Guide* for more information about the editor that comes with ESP, or to learn how to configure ESP to use another editor.
2. Make the text file contain the information shown at right. Use <Tab> or blank spaces to separate the two items in a pair.

```
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 the* ***Back Annotate*** *tool to fail. In addition, some text editors may attach an extension to the NEWREF file. If it does, be sure to enter the extension when running* ***Back Annotate.***

4. Return to the **Schematic Design Tools** work screen and enter **Back Annotate's Local Configuration** screen.



Configuration Screen for Back Annotate.

5. In the **File Options** portion of the **Configure Back Annotate** screen, in the **Was/Is** entry box, enter the name of the file where **Back Annotate** gets the back annotation information, in this case **NEWREF**.

   Back Annotate modifies the schematic file, TUTOR.SCH to reflect the new reference designator values found in the WAS/IS file, NEWREF.

6. Select **OK** to return to the **Schematic Design Tools** work screen and run **Back Annotate** by selecting the **Back Annotate** button and **Execute.**

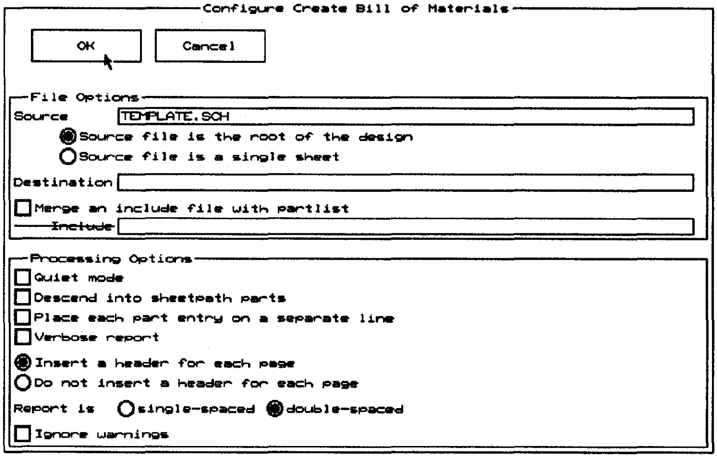
7. Run **Draft** on TUTOR.SCH to confirm that **Back Annotate** modified the reference designators on the displays.

Running the Create Bill of Materials tool

The **Create Bill of Materials** tool creates a text file listing all parts in a single sheet or an entire design.

Make a parts list

1. Configure the **Create Bill of Materials** tool by selecting the **Create Bill of Materials** button, then **Local Configuration.**
2. Select **Configure PARTLIST.** The **Configure Create Bill of Materials** screen displays.



Configure Create Bill of Materials screen.

3. In the **File Options** portion of the 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 the **Create Bill of Materials** tool to use the worksheet file TUTOR.SCH to get the correct reference designator values.

   ❖ In the **Destination** entry box, enter the name of the file where **Create Bill of Materials** stores the parts list, in this case `TUTOR.BOM`.

4. Click **OK** to save all of the changes.
5. Run **Create Bill of Materials** by selecting **Create Bill of Materials** and then **Execute**. The contents of TUTOR.BOM are shown in the TUTOR design Bill of Materials figure on the next page. USE **Edit File** to look at this file.

```
Digital Clock                          Revised:   November 1, 1990
                                                       Revision:
Bill Of Materials   November 1, 1990   16:17:16           Page  1

 Item  Quantity  Reference              Part
------------------------------------------------------------------

   1       1      BT1                      BATTERY
   2       1      C1                       47uF
   3       1      C2                       470uF
   4       1      C3                       22uF
   5       1      C4                       100uf
   6       4      C5,C6,C7,C8              .01uf
   7       6      A1,A2,A3,A4,A5,A6        TIL309
   8       1      Q1                       LM7805
   9       1      R                        9.1K
  10       1      R1                       R
  11       6      R2,R3,R4,R5,R6,R7        10k
  12       1      S1                       Mode
  13       1      S2                       Reset
  14       1      U1                       74LS04
  15       1      U2                       Hrs
  16       2      U3,U4                    Minsec
```

TUTOR design Bill of Materials.

Running the Plot Schematic tool

The last task in this chapter of *Learning Schematic Design Tools* is to plot the design you have created so far.

The **Plot Schematic** tool is used to send designs to a plotter, or optionally, to a printer using the **Send output to printer** radio button.

△ *NOTE: This section focuses on running the* ***Plot Schematic*** *tool and assumes you have configured the* ***Schematic Design Tool*** *and connected your printer or plotter correctly. There are many variables affecting plotting. As with other mechanical processes, make sure your equipment, paper, pens, and so on, are in good working order and set up properly.*

1. Configure the **Plot Schematic** tool by selecting the **Plot Schematic** button and **Local Configuration.**
2. Select **Configure PLOTALL.** The **Configure Plot Schematic** screen displays.

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

△ *NOTE: When there are multiple worksheets in a design,* ***Plot Schematic*** *plots every worksheet comprising the design.*

3. If the plot produced is too large or too small, you can change the scale by re-running the **Plot Schematic** tool and selecting the **Automatically scale and set X, Y offsets for specified sheet size** radio button and the **Set scale factor** check box. The **Set Scale factor** entry box is highlighted.

   Enter the scale factor, expressed in the form n.nnn. For example, if the plot is larger than the paper, you might run the **Plot Schematic** tool at half scale by entering the number: `0.500`.
4. Click **OK.**
5. Run **Plot Schematic** by selecting **Plot Schematic** and then **Execute.**

Structuring your design

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

A simple hierarchical design

This section describes a simple hierarchical design, discusses labeling, module ports, nets, sheet symbols, and other aspects of the design, and reviews how to execute some schematic design tools.

The design discussed is a three sheet *simple hierarchy.* In a *hierarchy,* schematic worksheets are *nested* in other worksheets; the nested schematics are symbolized and referenced by block-shaped *sheet symbols.* Sheet symbols may be placed at any level of the hierarchy.

The example design is a *simple* hierarchy because each sheet symbol in the root worksheet references a separate schematic worksheet. In a *complex hierarchy,* multiple copies of a sheet symbol reference a single schematic worksheet.

Before you begin this exercise, you need to create two new design areas in which you will place examples:

1. Enter the **Design Management Tools** area, and use the **Create Design** button to make a design called CMOSCPU and a design called 4BIT.
2. The files you need are in the TUTOR design area, so select TUTOR as the current design.
3. Switch to **File View** by selecting the **File View** button.
4. Select the **Copy File** button.
5. Now copy six schematic files to the two design areas. Table 8-1 shows the six files to be copied and the destinations.

Source	*Destination*
CMOSCPU.SCH	..\CMOSCPU\CMOS.CPU.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.

6. Select the source file from the scroll window and enter the destination in the entry box. Press **OK**. Repeat this procedure for each file in the table.
7. When you have copied all the files to the appropriate destinations, press **CANCEL** to leave the **Copy File** screen and reset the current design to CMOSCPU.
8. Press **OK** to return to the main work screen.

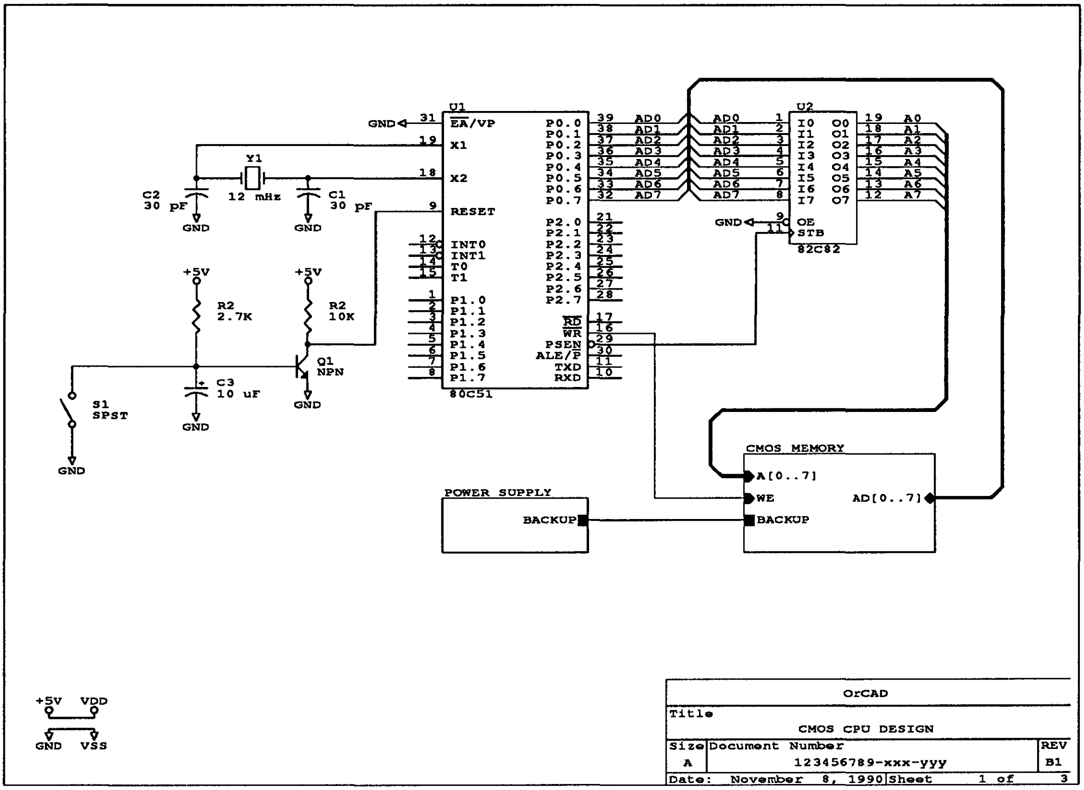


Figure 8-1. CMOS CPU DESIGN root sheet.

The root sheet CMOSCPU.SCH

Figure 8-1 shows the root sheet of this simple hierarchy.

The design is called CMOSCPU. The root sheet filename is CMOSCPU.SCH. You can also give the worksheet a more descriptive name in the title block, as the figure 8-1 shows. The descriptive name in the title block is independent of the filename by which **Draft** (and the operating system) identify the worksheet. A descriptive title helps identify the worksheet, but is not required.

In this example, the root sheet contains:

- Two component packages: an 80C51 and an 82C82.
- Discrete analog components: a transistor, capacitors, resistors, and so on.

- Two sheet symbols: POWER SUPPLY and CMOS MEMORY.
- Power and Ground symbols.
- Wires and buses connecting the components.

Sheet symbols

The two sheet symbols were placed in the worksheet using the **PLACE Sheet** command. The CMOS MEMORY sheet symbol references the worksheet in which the system's memory is located. The POWER SUPPLY sheet symbol references the worksheet in which the system's power supply is located.

Sheet symbols are associated with filenames. **Draft** uses the filename associated with a sheet symbol to find the schematic worksheet to be nested in the root sheet.

The *filename* assigned to the sheet symbol is separate and distinct from the *name* of the sheet symbol, which displays over the sheet symbol in the root sheet. The filename displays below the sheet symbol.

When a sheet symbol is created, **Draft** automatically assigns it a unique filename generated from the date and time of day. You can accept this unique (but not very descriptive) filename or change it to a filename of your choice.

In this example, the CMOS MEMORY sheet symbol was assigned the filename MEMORY.SCH, because this is the filename we plan to give the schematic it references. Similarly, we assigned the POWER SUPPLY sheet symbol the filename POWER.SCH, because this is the filename we plan to give the schematic it references.

The CMOS MEMORY sheet symbol contains four nets: A[0..7], WE, BACKUP, and AD[0..7]. These nets were placed into the sheet symbol using the **PLACE Sheet** command called **Add Net**. These nets are *not* module ports.

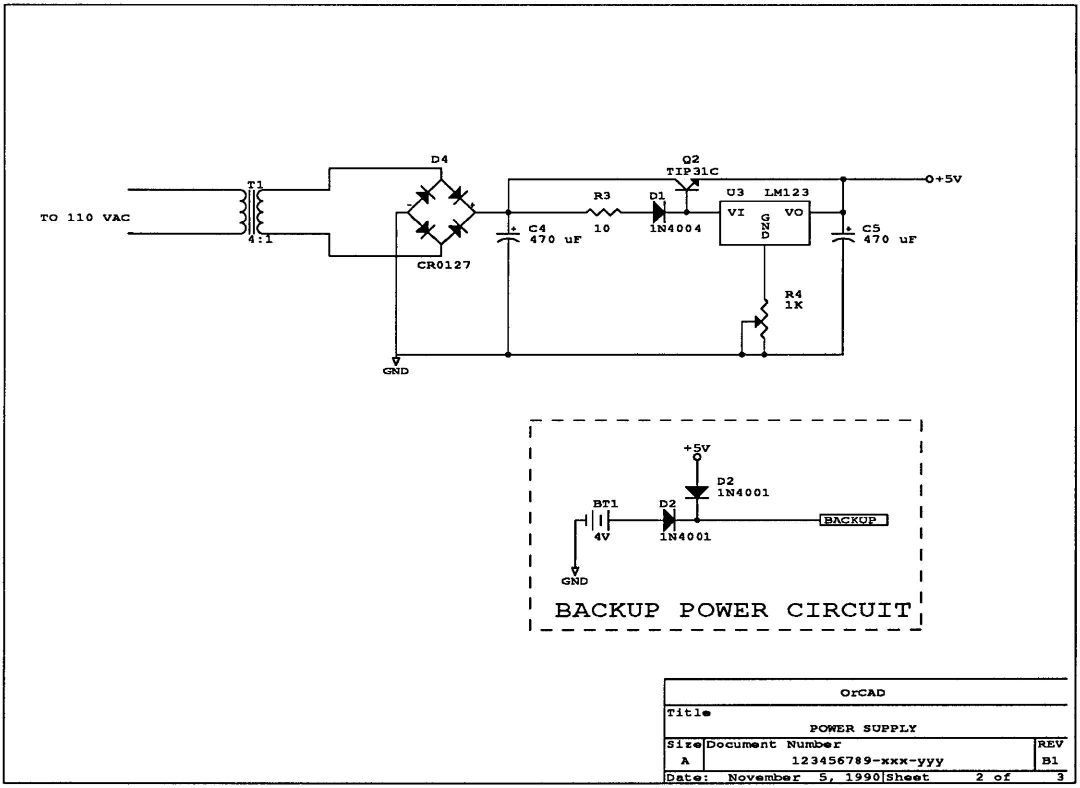


Figure 8-2. POWER SUPPLY worksheet.

Connected to the A[0..7] and AD[0..7] sheet nets are buses with labels placed on them indicating the name of the net they connect to.

While bus labels do not need to have the same *prefixes* as the sheet nets to which they are connected ("A" and "AD" in this example), the labels *must* specify the same *ranges* as the sheet nets to which they are connected ("[0..7]").

Connected to the WE sheet net is a wire going to the PSEN signal on the 80C51. A sheet net named BACKUP connects to a net in the POWER SUPPLY sheet symbol having the same name.

For labels and module ports, there should be no space between the prefix and suffix portion of the names.

Finally, in the root sheet is a power object named +5V connecting to a power object named VDD. This connects the VDD pins of the 80C51 and the 82C82 to the + 5 volt supply. Likewise, a power object named GND connects to a power object named VSS. This connects the VSS pin of the 80C51 and the 82C82 to power ground.

After the root worksheet is completed, save your work using **QUIT Update**. In this example, the root worksheet is saved with the filename CMOSCPU.SCH.

You do not need to create the root sheet of the hierarchy before creating the nested worksheets. A top-down design methodology is a useful approach, however. We follow this approach in this example.

Once the sheet symbols for the nested logic have been created, the next step is to enter the sheet symbols in the root sheet and create the schematic worksheets the sheet symbols reference.

Nested schematic worksheets

Start with the POWER SUPPLY sheet symbol. To enter it and display a worksheet it references, select **QUIT Enter Sheet**. Then place the pointer inside the POWER SUPPLY sheet symbol and select **Enter. Draft** displays a worksheet on which you can see the circuitry for the power supply. Figure 8-2 shows the completed POWER SUPPLY schematic.

Inside the POWER SUPPLY worksheet is the design for the power supply circuitry. Notice the module port named BACKUP. This makes the logical connection to the sheet net named BACKUP in the POWER SUPPLY sheet symbol in the root sheet, shown in Figure 8-1. Electrically, the BACKUP module port connects to a sheet net of the same name inside the CMOS MEMORY worksheet.

In operation, the CMOS MEMORY sheet only receives power via the module port named BACKUP. Power is isolated in the CMOS MEMORY worksheet, because the power is transferred through the module port.

When the review of the power supply design has been completed, leave the nested worksheet and return to the root sheet using **QUIT Leave Sheet.**

Back at the root level, the next step is to enter the CMOS MEMORY sheet symbol and review its circuitry. To do this, follow the same steps described above for entering POWER SUPPLY. Figure 8-3 shows the completed CMOS MEMORY schematic.

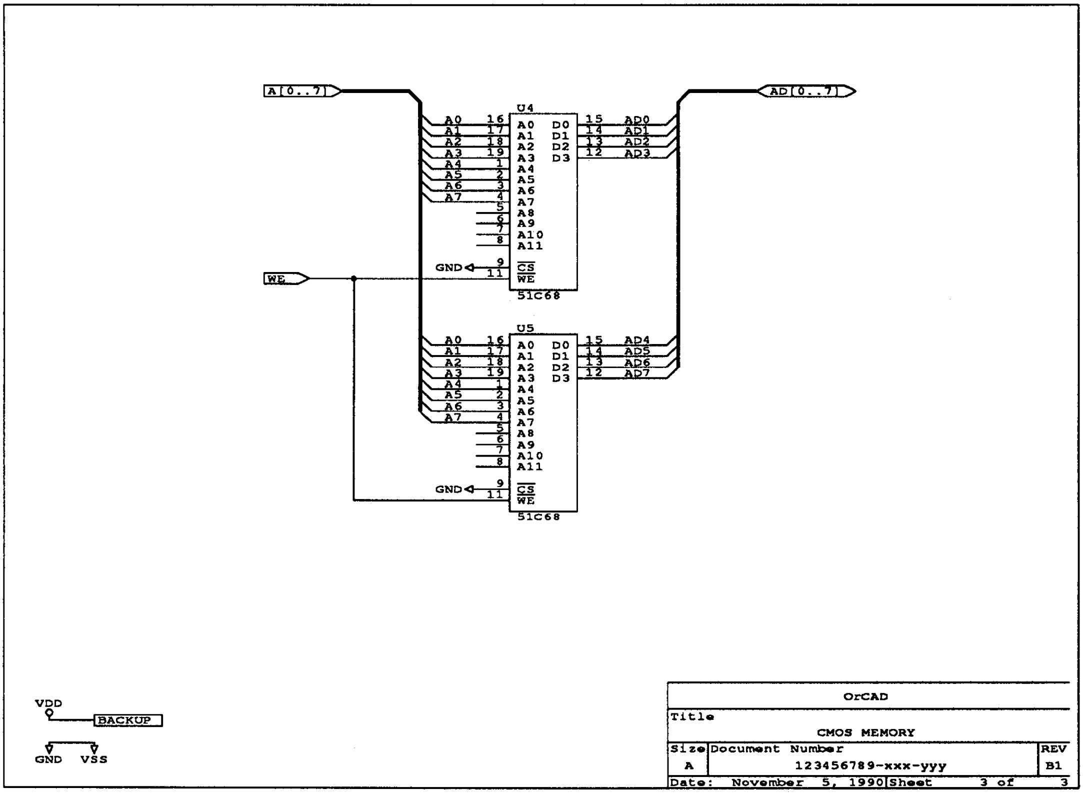


Figure 8-3. CMOS MEMORY worksheet.

In this worksheet are four module ports: A[0..7], WE, BACKUP, and AD[0..7]. They connect to identically named nets located in the CMOS MEMORY sheet symbol in the root sheet shown in Figure 8-1.

Buses are automatically connected to module ports with labels having the same name and range as the module ports to which they connect. In this case, module port A[0..7] automatically connects to a bus labeled A[0..7]. Module port AD[0..7] automatically connects to a bus labeled D[0..7].

Also required in a hierarchy, labels placed on signals connecting to a bus are given the same prefix name as the bus they connect to. For example, labels D0 through D7 correspond to the prefix of the bus labeled D[0..7] (the prefix is a "D" in this case). Likewise, labels A0 through A7 correspond to the prefix of the bus labeled A[0..7] (the prefix is an "A" in this case).

Power is supplied to the CMOS MEMORY worksheet through the module port named BACKUP. Power is isolated in this worksheet since the VDD power object connects to the module port named BACKUP.

Finally, a power object named GND connects to a power object named VSS. This connects the VSS pins of the 51C68 memory devices to the power ground symbol the CS pins are connected to.

When design work is completed on the CMOS MEMORY worksheet, update it before returning to the root sheet or exiting **Draft**.

Design guidelines for simple hierarchies

1. Read the discussion on the **Create Netlist** tool in the *Schematic Design Tools Reference Guide* carefully.
2. Place Labels in the correct format on buses.
3. Place Labels in the correct format on signals connecting to a bus.
4. Place module ports in the correct format, on all signals going off the worksheet.
5. Do not put a blank space in any label or module port name.
6. When placing sheet symbols, use "sheet nets," not module ports to connect to other sheet symbols.
7. Do not overlap wires or buses with other wires, buses, or object pins.

After creating the simple hierarchical design, you may run the **Draft** tool, the **Annotate Schematic** tool, the **Check Electrical Rules** tool, the **Show Schematic Structure** tool, the **Create Netlist** tool, the **Create Bill of Materials** tool, or any other tools in the ESP environment on this design.

Using Annotate Schematic on a simple hierarchy

After the design is complete, run the **Annotate Schematic** tool. **Annotate Schematic** assigns unique values to the reference designators of all library parts placed in the design.

To annotate the simple hierarchy represented by the worksheet, CMOSCPU.SCH, perform these tasks:

1. From the **Schematic Design Tools** work screen, select **Annotate Schematic**. The menu at right displays.

```
Annotate Schematic
Execute
Local Configuration
Configure Schematic Tools
Help
```

2. Select **Local Configuration** and **Configure ANNOTATE.** The **Configure Annotate Schematic** screen displays.
3. Verify that the **Source** name is CMOSCPU.SCH. If the **Source** name needs to be edited, place the pointer in the **Source** entry box in the **File Options** portion of the screen, and click once. Enter the filename of the file to be annotated. In this case, enter the following:

```
CMOSCPU.SCH
```

4. Now, click the Merge annotation information into schematic radio button.
5. Select the **OK** button to save all of the changes.
6. Run **Annotate Schematic** by selecting the **Annotate Schematic** button and selecting **Execute** from the menu that displays.

   Annotate Schematic displays some messages and the **Schematic Design Tools** work screen appears.

When **Annotate Schematic** is done, the reference designators for each part in the worksheet have new, unique values.

Using the Check Electrical Rules tool on CMOSCPU.SCH

Next, run the **Check Electrical Rules** tool to check for any electrical errors in the design. **Check Electrical Rules** runs the same for this design as earlier when you ran it on the TUTOR design. Refer to the earlier discussion of how to run **Check Electrical Rules** for instructions.

Every design should be checked for electrical rule violations using **Check Electrical Rules** after the worksheet is annotated and cleaned up. After configuration, if any, and execution, you may review the ERC report with the **Edit File** editor.

Check Electrical Rules checks for several problems associated with a design, including: open input pins, shorts, and bus contention.

Warnings

Check Electrical Rules flags certain conditions possibly overlooked when your design was created. These *WARNINGS* are not critical errors. In this example, most warnings inform you of inputs with no driving source. This is perfectly acceptable, if these pins are intentionally left open in the design. The connected power supply warnings are also acceptable, since they were intentionally connected in the design.

Errors

If **Check Electrical Rules** reports *ERRORS* in a design, you should correct them before continuing on and running other tools.

In this example, all warnings are acceptable and other tools may be run.

```
"cmoscpu.sch"
UNCONNECTED REPORT
 X=  4.50  Y=  1.90 I/O                U1,P2.0
 X=  4.50  Y=  2.00 I/O                U1,P2.1
 X=  2.60  Y=  2.10 I/O                U1,INT0
 X=  4.50  Y=  2.10 I/O                U1,P2.2
 X=  2.60  Y=  2.20 I/O                U1,INT1
 X=  4.50  Y=  2.20 I/O                U1,P2.3
 X=  2.60  Y=  2.30 I/O                U1,T0
 X=  4.50  Y=  2.30 I/O                U1,P2.4
 X=  2.60  Y=  2.40 I/O                U1,T1
 X=  4.50  Y=  2.40 I/O                U1,P2.5
 X=  4.50  Y=  2.50 I/O                U1,P2.6
 X=  2.60  Y=  2.60 I/O                U1,P1.0
 X=  4.50  Y=  2.60 I/O                U1,P2.7
 X=  2.60  Y=  2.70 I/O                U1,P1.1
 X=  2.60  Y=  2.80 I/O                U1,P1.2
 X=  4.50  Y=  2.80 I/O                U1,RD
 X=  2.60  Y=  2.90 I/O                U1,P1.3
 X=  4.50  Y=  2.90 I/O                U1,WR
 X=  2.60  Y=  3.00 I/O                U1,P1.4
 X=  2.60  Y=  3.10 I/O                U1,P1.5
 X=  2.60  Y=  3.20 I/O                U1,P1.6
 X=  4.50  Y=  3.20 I/O                U1,TXD
 X=  2.60  Y=  3.30 I/O                U1,P1.7
 X=  4.50  Y=  3.30 I/O                U1,RXD

WARNING: POWER Supplies are CONNECTED GND <-> VSS
WARNING: POWER Supplies are CONNECTED VDD <-> +5V

"POWER.SCH"
UNCONNECTED REPORT
 X=  1.10  Y=  1.10 Passive            T1,AA
 X=  1.90  Y=  1.30 Passive            T1,BCT
 X=  1.10  Y=  1.50 Passive            T1,AB

"MEMORY.SCH"
UNCONNECTED REPORT
 X=  4.30  Y=  2.40 Input              U4,A8
 X=  4.30  Y=  2.50 Input              U4,A9
 X=  4.30  Y=  2.60 Input              U4,A10
 X=  4.30  Y=  2.70 Input              U4,A11
 X=  4.30  Y=  4.50 Input              U5,A8
 X=  4.30  Y=  4.60 Input              U5,A9
 X=  4.30  Y=  4.70 Input              U5,A10
 X=  4.30  Y=  4.80 Input              U5,A11

WARNING: INPUT has NO Driving Source U4,A8
WARNING: INPUT has NO Driving Source U4,A9
WARNING: INPUT has NO Driving Source U4,A10
WARNING: INPUT has NO Driving Source U4,A11
WARNING: INPUT has NO Driving Source U5,A8
WARNING: INPUT has NO Driving Source U5,A9
WARNING: INPUT has NO Driving Source U5,A10
WARNING: INPUT has NO Driving Source U5,A11
WARNING: POWER Supplies are CONNECTED VSS <-> GND
```

Figure 8-4. The error report produced by ***Check Electrical Rules*** *for CMOSCPU.SCH.*

Using the Show Design Structure tool on a simple hierarchy

To obtain a text file listing the sheets in a hierarchy, use the **Show Design Structure** tool. This program is helpful for organizing a hierarchy containing many worksheets. To tell **Show Design Structure** the name of the file you would like a listing of, follow these steps:

1. Click the **Show Design Structure** button. The **Show Design Structure** menu displays.
2. Select **Local Configuration.** Select **Configure TREELIST.** The **Configure Design Structure** screen displays.
3. Under **File Options,** in the **Source** entry box, enter **CMOSCPU.SCH.**
4 Select the **Source file is the root of the design** radio button.
5. Enter **CMOSCPU.TRE** in the **Destination** entry box under **File Options. Show Design Structure** is now configured to run a schematic structure list for **CMOSCPU.SCH** and save the results in **CMOSCPU.TRE.**
6. Click the **OK** button to leave the **Local Configuration** screen and save your changes.

To execute **Show Design Structure** on the simple hierarchy **CMOSCPU.SCH,** click the **Show Design Structure** button, and select **Execute.**

CMOSCPU.SCH is the name of the root worksheet of the hierarchy. To examine the output file, use **Edit File.** The figure below shows the Schematic Design report stored in **CMOSCPU.TRE:**

```
<<<root>>>
[CMOSCPU.SCH]        November 8, 1990
   CMOS MEMORY
   [MEMORY.SCH]      November 5, 1990
   POWER SUPPLY
   [POWER.SCH]       November 5, 1990
```

All worksheet file names are enclosed within brackets [*filename*]. Next to the sheet name is the date the worksheet was last modified. **Show Schematic Structure** lists sheet symbol names above the file names of the worksheets they reference.

In this example, the root file is named CMOSCPU.SCH. Below the root are sheet symbols and the file names of the worksheets they reference. The sheet symbol named POWER SUPPLY references the worksheet file, POWER.SCH. The sheet symbol named CMOS MEMORY references the worksheet file, MEMORY.SCH.

Using the Create Bill of Materials tool on a simple hierarchy

The **Create Bill of Materials** tool creates a list of parts for all types of design structures in a text file. In this example, **Create Bill of Materials** is used on a simple hierarchy, CMOSCPU.SCH (shown in Figure 8-1 at the beginning of this chapter).

1. Configure **Create Bill of Materials** by selecting the **Create Bill of Materials** button and **Local Configuration.**
2. Select **Configure** PARTLIST. The **Configure Create Bill of Materials** screen displays.
3. In the **File Options** portion of the screen, you enter two filenames:
   - ❖ In the **Source** entry box, verify that the name of the worksheet from which the bill of materials is to be produced is `CMOSCPU.SCH`.
   - ❖ In the **Destination** entry box, enter the name of the file where Bill-of-Materials report is to be saved, in this case `CMOSCPU.BOM`. You can specify the pathname to any directory, but normally you would not set a path so that the design data all remains together.
4. Now, select the Source file is the root of the design radio button.
5. Exit the **Configure Bill of Materials** screen.

6. Now, run **Create Bill of Materials** by selecting **Create Bill of Materials** and **Execute.** To examine the output text file, use **Edit File.** Figure 8-11 shows the Bill of Materials stored in the text file CMOSCPU.BOM.

```
CMOS CPU DESIGN
Revised:  November  8, 1990
123456789-xxx-yyy
Revision: B1
Bill Of Materials          November    8, 1990
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	1N4004
6	2	D2,D3	1N4001
7	1	D4	CR0127
8	1	Q1	NPN
9	1	Q2	TIP31C
10	1	R2	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	1	U3	LM123
19	2		51C68

Figure 8-5. Bill of Materials for CMOSCPU.SCH.

A complex hierarchical design

In this section, you examine a *complex hierarchy*. Complex hierarchies are designs in which more than one sheet symbol references the same worksheet.

The design discussed in this section is a three-sheet complex hierarchy. In a hierarchy, schematic sheets are nested inside other worksheets The nested schematics are symbolized and referenced by block-shaped sheet symbols. Sheet symbols may be placed at any level of the hierarchy.

The example design is a complex hierarchy because schematic files are referenced by multiple sheet symbols. This is very useful when designing common logic blocks that are repeated.

Figure 8-6 shows the root worksheet, 4BIT.SCH.

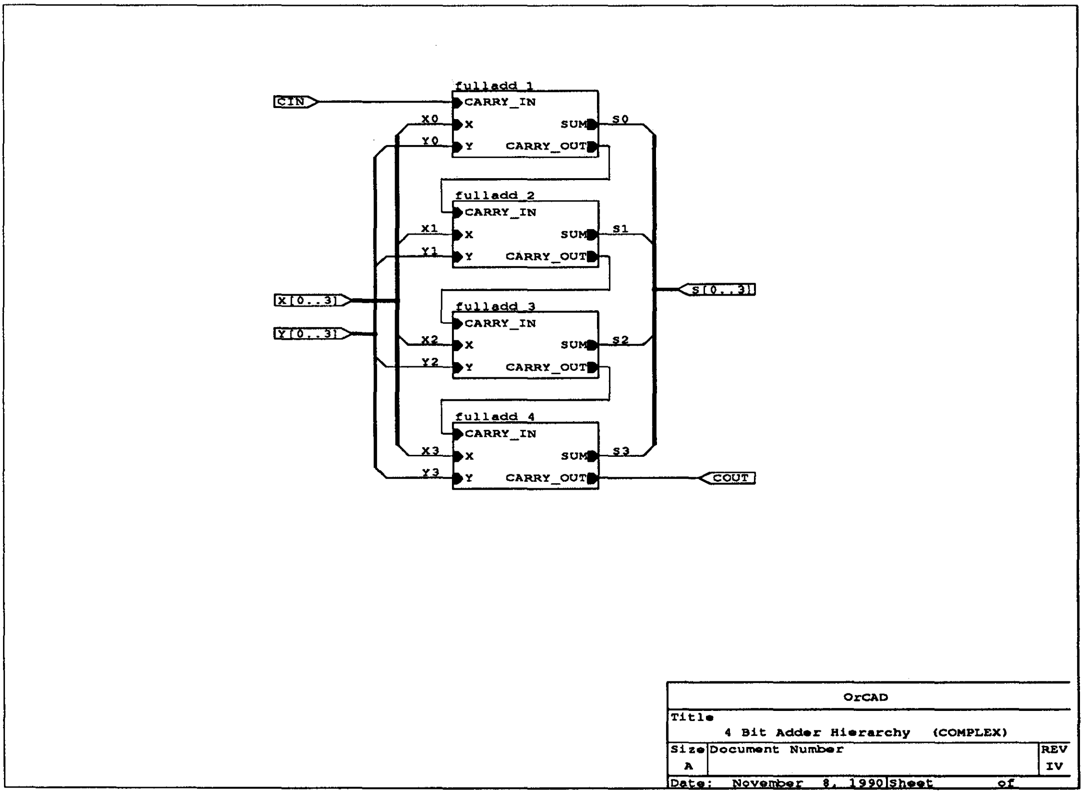


Figure 8-6. 4BIT ADDER root sheet.

The root sheet, 4BIT.SCH

The root worksheet contains four identical sheet symbols, named fulladd_1, fulladd_2, fulladd_3, and fulladd_4. The 4BIT Adder has module ports to connect to a level above it. In this example though, a subset of a very large design is presented to show the principles of a complex hierarchy.

Since all four full adders are identical in their design, it is not necessary to create a separate worksheet for each one. Instead, create just one, and cause all four sheet symbols to reference it by assigning the worksheet's filename to all four sheet symbols. Notice that all of the sheet symbols have the filename of FULLADD.SCH.

Use **QUIT Enter Sheet** to enter any one of the four full adder sheet symbols. **Draft** displays a new worksheet. In this worksheet, you can see the schematic for the circuitry referenced by the full adder sheet symbols in the root sheet.

Figure 8-7 shows the FULLADD.SCH worksheet.

This worksheet contains two new, identical sheet symbols, named HALFADD_A AND HALFADD_B. 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.

Just as the four full adder sheet symbols in the root sheet can reference the FULLADD.SCH worksheet for their logic, the two half adder sheet symbols in FULLADD.SCH can, in turn, reference a single HALFADD.SCH worksheet for *their* logic.

To view the HALFADD.SCH schematic, move the pointer onto one of the sheet symbols, then enter **QUIT Enter Sheet**. The half adder worksheet now appears.

Figure 8-8 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.

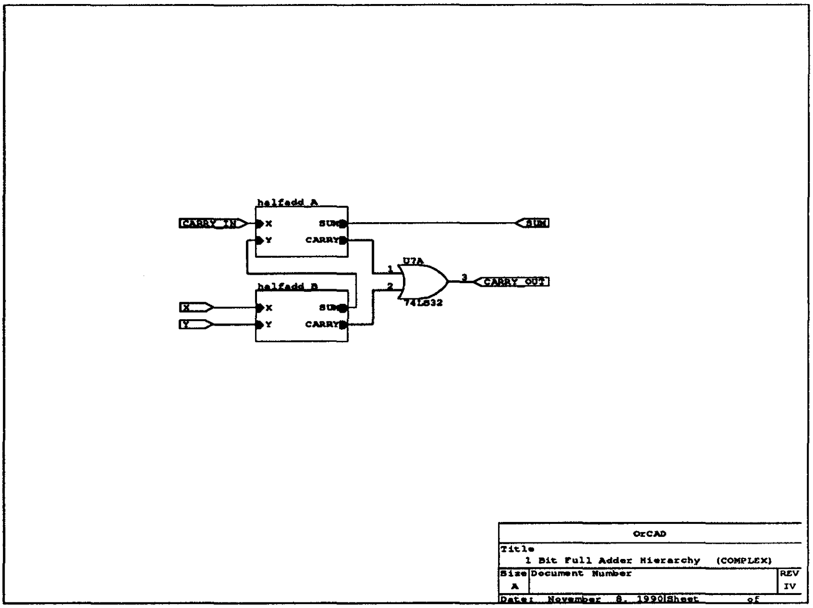


Figure 8-7. FULL ADDER worksheet.

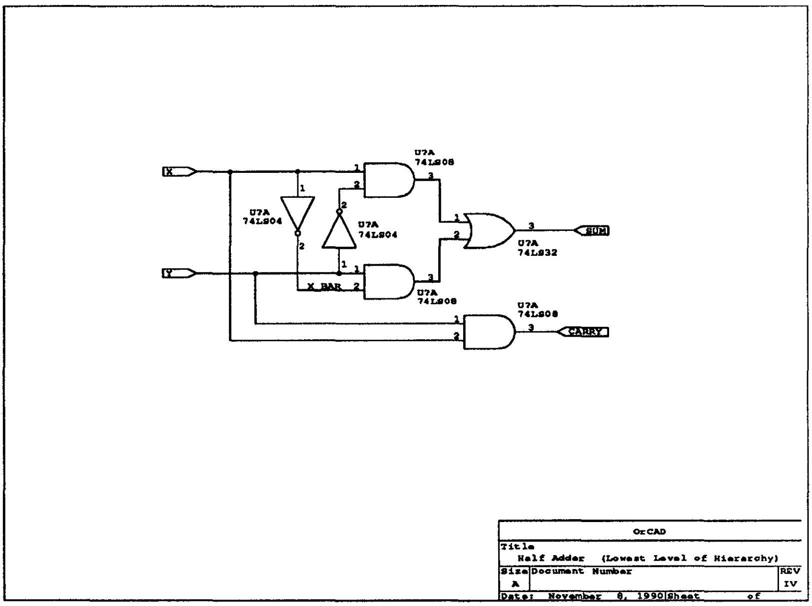


Figure 8-8. HALF ADDER worksheet.

Using the Show Design Structure tool on a complex hierarchy

To obtain a text file listing the sheets in a hierarchy, use the **Show Design Structure** tool. This tool is helpful for organizing a hierarchy containing many worksheets. To tell the **Show Design Structure** tool the name of the file you would like to examine, follow these steps:

1. Click the **Show Design Structure** button. The menu at right displays.

```
Show Design Structure
Execute
Local Configuration
Show Version
Configure Schematic Tools
Help
```

2. Select **Local Configuration**, then **Configure TREELIST.** The **Configure Show Design Structure** screen displays.
3. Enter `4BIT.SCH` in the **Source** entry box under **File Options.**
4. Enter `4BIT.TRE` in the **Destination** entry box under **File Options.** The **Show Design Structure** tool is now configured to run **Show Design Structure** on `4BIT.SCH` and save the results in `4BIT.TRE`.
5. Click the **OK** button to leave the **Local Configuration** screen and save your changes.

To execute the **Show Design Structure** tool on the simple hierarchy `4BIT.SCH`, click the **Show Design Structure** button, and select **Execute.**

4BIT.SCH is the name of the root worksheet of the hierarchy. To examine the output file, use **Edit File.** The figure below shows the schematic design report stored in **4BIT.TRE**:

```
<<<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 in the above report that there are a number of references to fulladd.sch and halfadd.sch, and that there are thirteen file references. The schematic structure, a complex hierarchy of only three sheets in this design, expands to thirteen referenced sheets. Again, the advantage of complex hierarchical design organization is that all of the repeated logic can be drawn once during the design phase.

Converting a complex hierarchy to a simple hierarchy

While a complex hierarchy is very useful in the design phase, it is not practical for some aspects of the design cycle. Particularly when a design is to be turned into a printed circuit board, all of the design must 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 figure out which was which from the complex hierarchy schematic.

In the design management tools area is a process called **Complex to Simple.** This process creates a new project and builds a new version of the complex hierarchy, a version in which each sheet symbol refers to a unique filename.

1. Enter the design management tools area.
2. Click the **Complex to Simple** button.
3. Select 4BIT from the **Designs** scroll window and notice that the text 4BIT appears in the **Source design** entry box.
4. Select the **Destination design** entry box, and enter S4BIT.
5. When this is complete, click **OK**. ESP builds the new design area and converts the 4BIT design to S4BIT.
6. Select **CANCEL** when the process is complete.
7. Select S4BIT as the current design.

Notice that 4BIT.SCH is now S4BIT.SCH, FULLADD.SCH is FULLADDA.SCH, FULLADDB.SCH, and FULLADDC.SCH and HALFADD.SCH are now eight files: HALFADDA.SCH through HALFADDH.SCH. If you use **Draft** to examine the design, you will see the filenames of the sheet symbols are now all unique.

The following figures show the simplified design after it is annotated using **Annotate Schematic.**

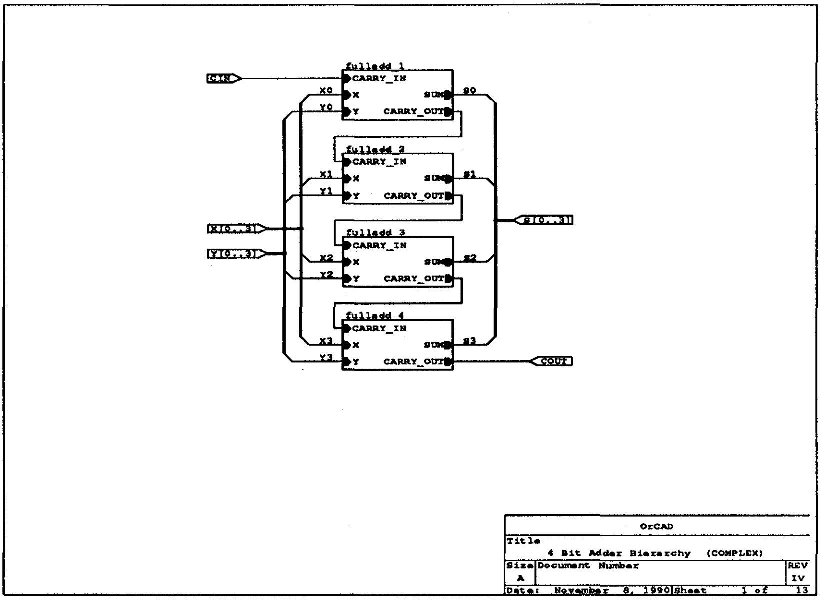


Figure 8-9. 4BIT.SCH worksheet.

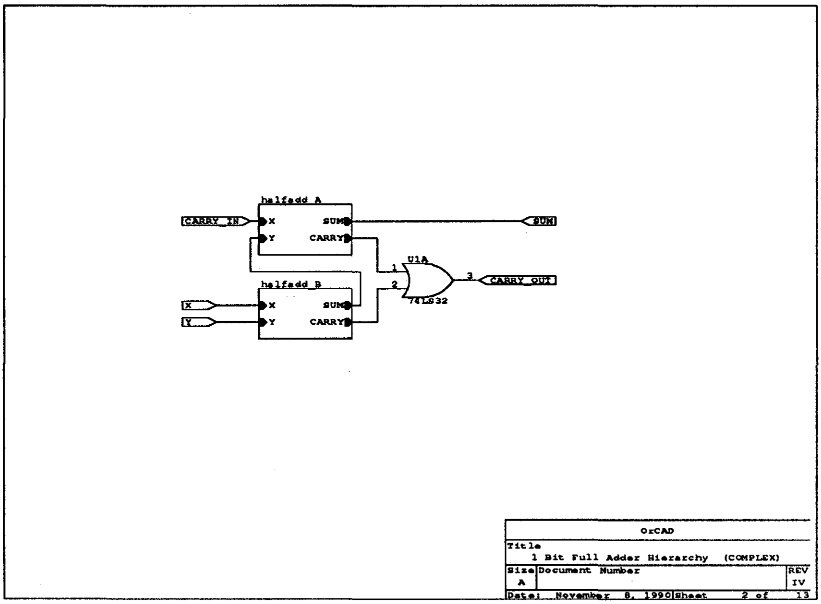


Figure 8-10. FULLADDA.SCH worksheet.

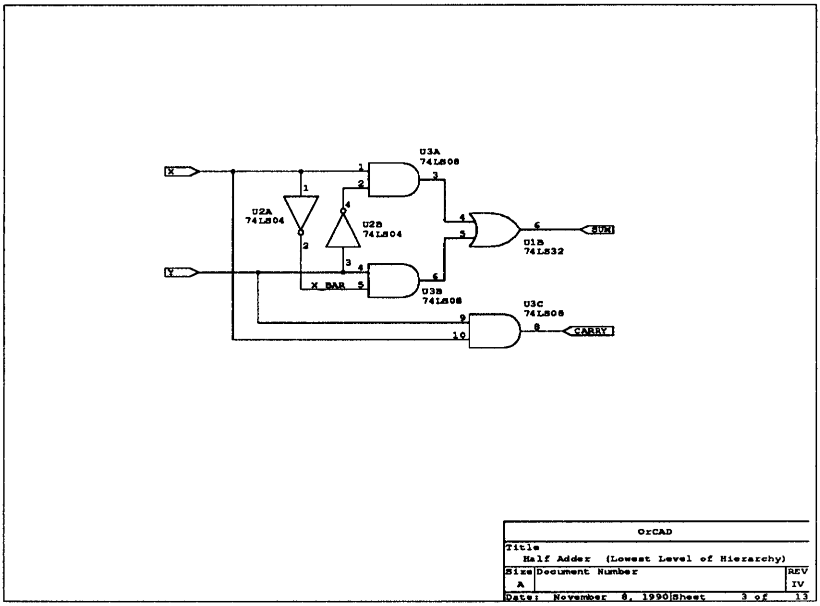


Figure 8-11. HALFADDA.SCH worksheet.

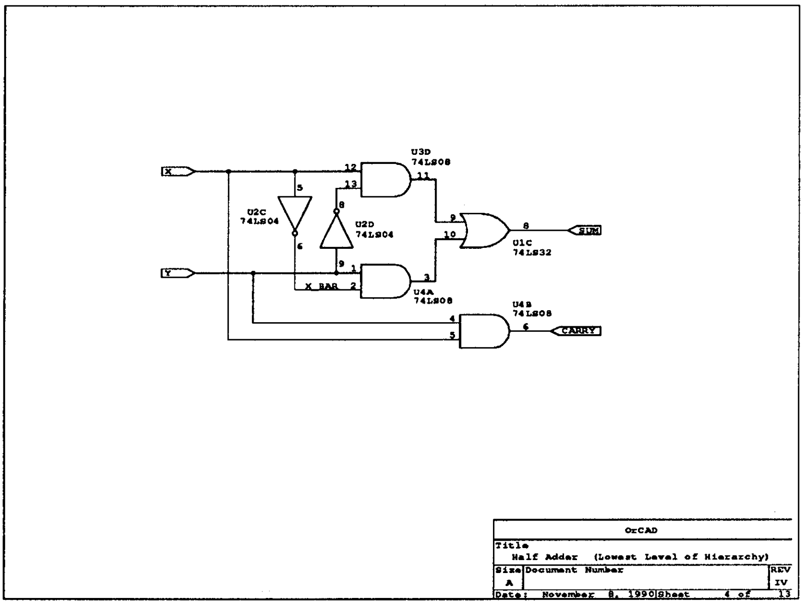


Figure 8-12. HALFADDB.SCH worksheet.

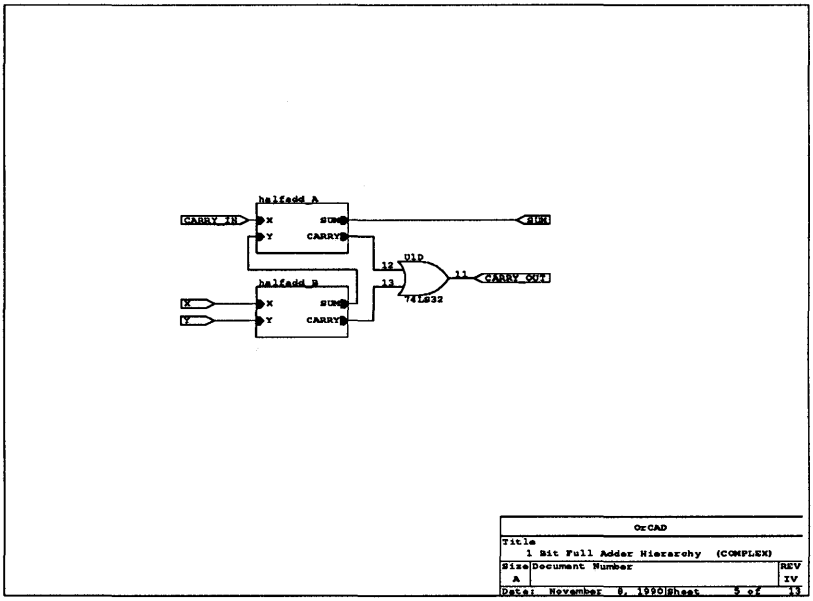


Figure 8-13. FULLADDB.SCH worksheet.

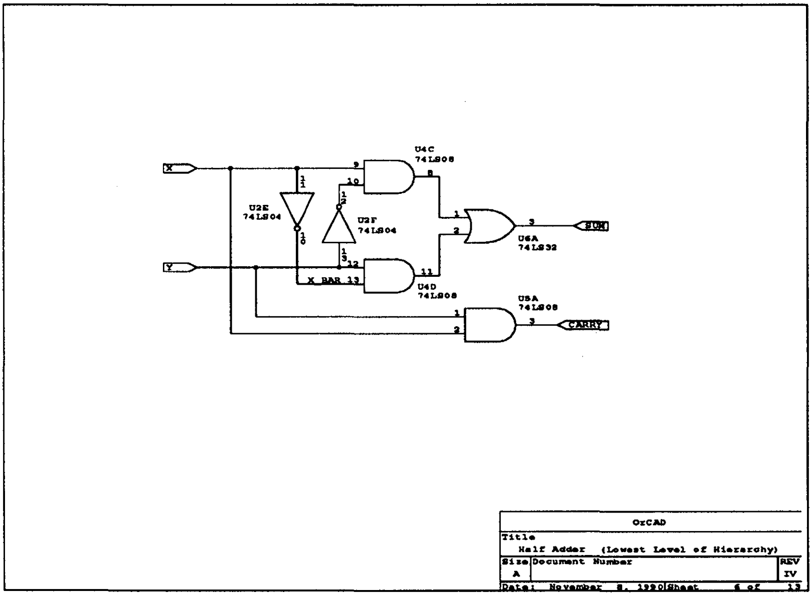


Figure 8-14. HALFADDC.SCH worksheet.

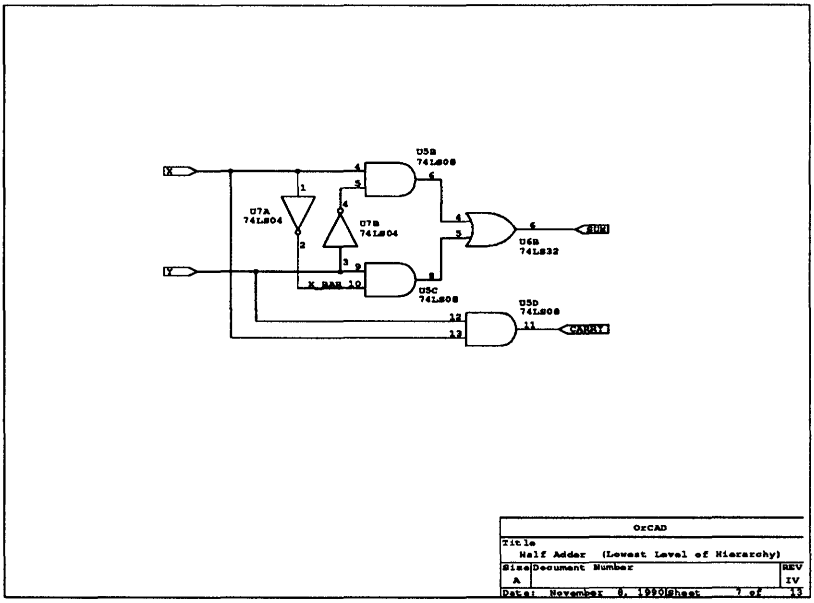


Figure 8-15. HALFADDD.SCH worksheet.

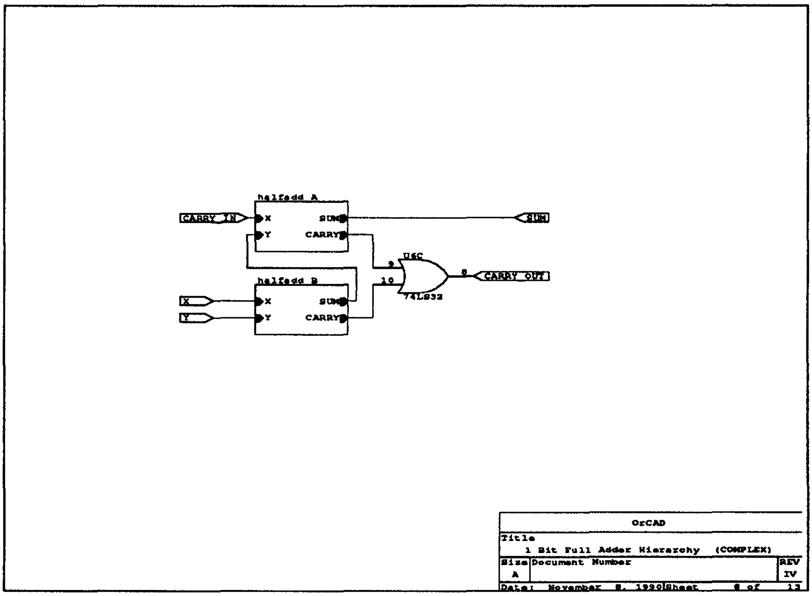


Figure 8-16. FULLADDC.SCH worksheet.

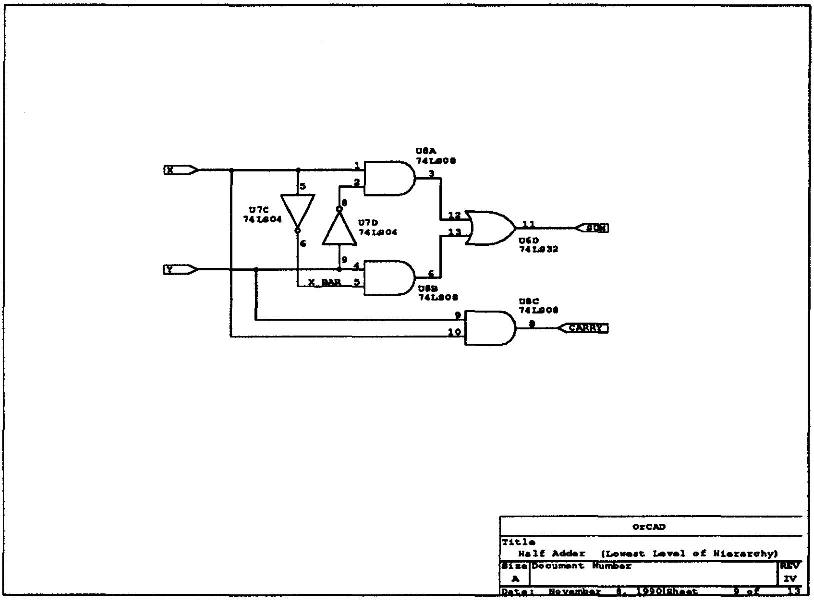


Figure 8-17. HALFADDE.SCH worksheet.

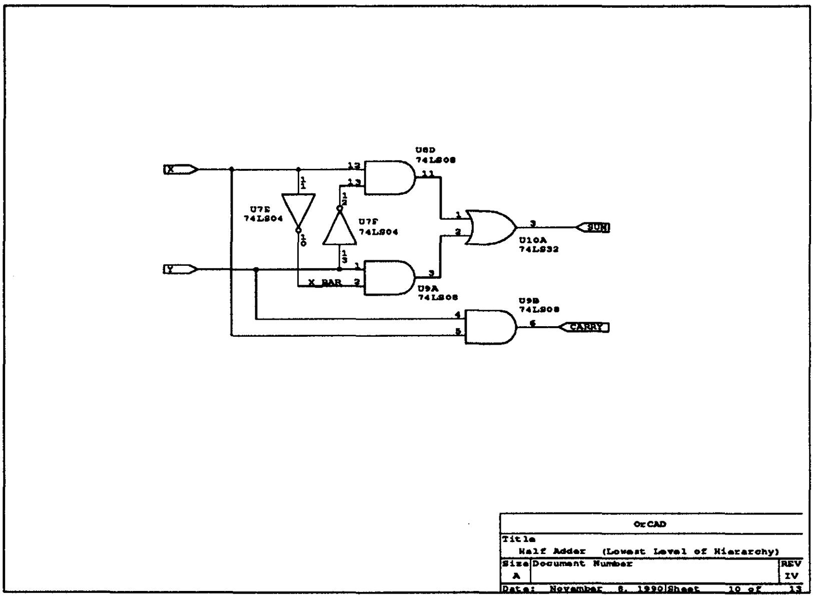


Figure 8-18. HALFADDF.SCH worksheet.

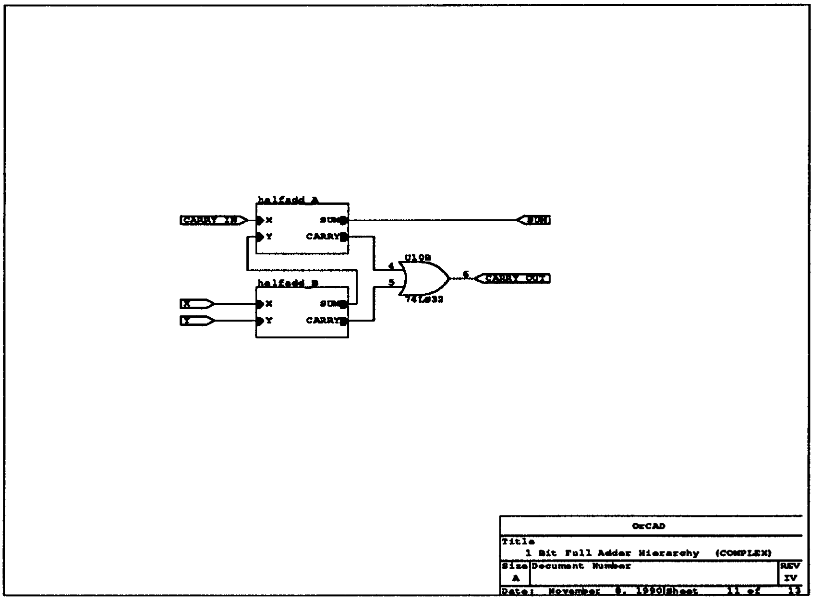


Figure 8-19. FULLADDD.SCH worksheet.

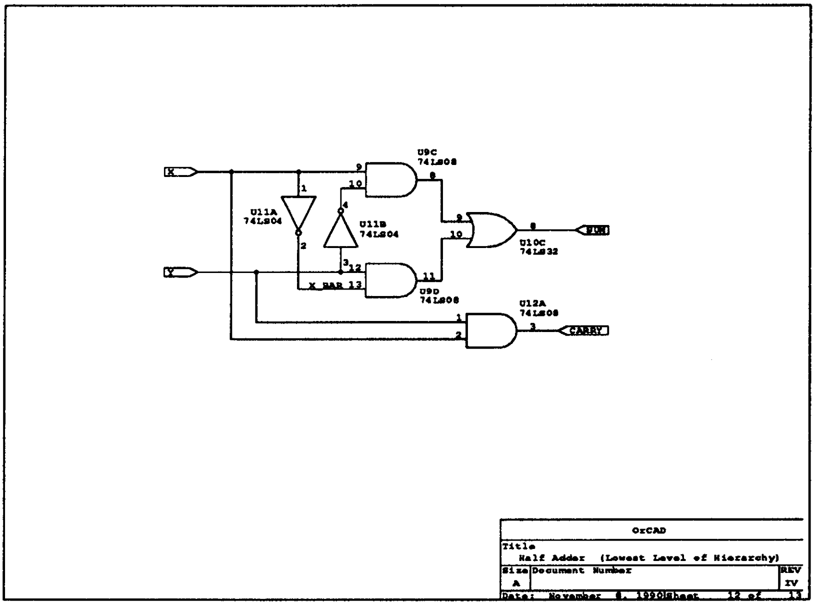


Figure 8-20. HALFADDG.SCH worksheet.

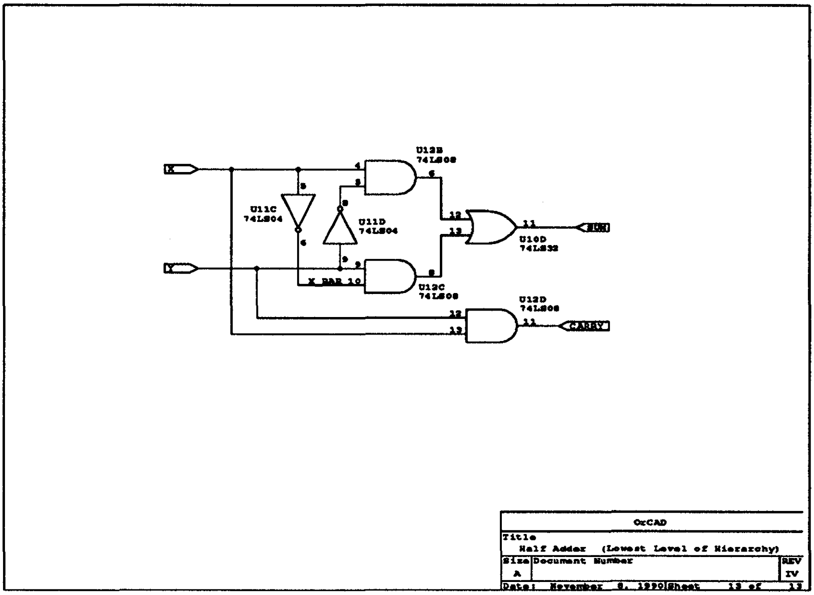


Figure 8-21. HALFADDH.SCH worksheet.

A flat design

A flat design is one in which all of the worksheets are linked together at the same level. There is not a representation of other schematic worksheets, but rather module ports are used to link the design. The advantage of flat designs is that for small designs of few worksheets, it is an easy and relatively productive way to design. The disadvantage is that for large designs of many sheets or repetitive logic, the management of the design and all of the inter-connections between the sheets can be difficult and time consuming.

Figures 8-22 and 8-23 are an example of a flat design. The module ports on each sheet that have the same name are connected together. In this design, COUNT, CLEAR, LOAD, and RCO are connected together,Hi[0..3] and Lo[0..3] are not connected.

The mechanism that informs the various schematic tools that particular schematics are in a flat design is the | LINK command. In this example we are linking to OTHER.SCH. As with hierarchies, the root of a flat design has the same name as the project. It is the | LINK command that informs the tools that the design is flat and gives the filenames of the schematics to link together.

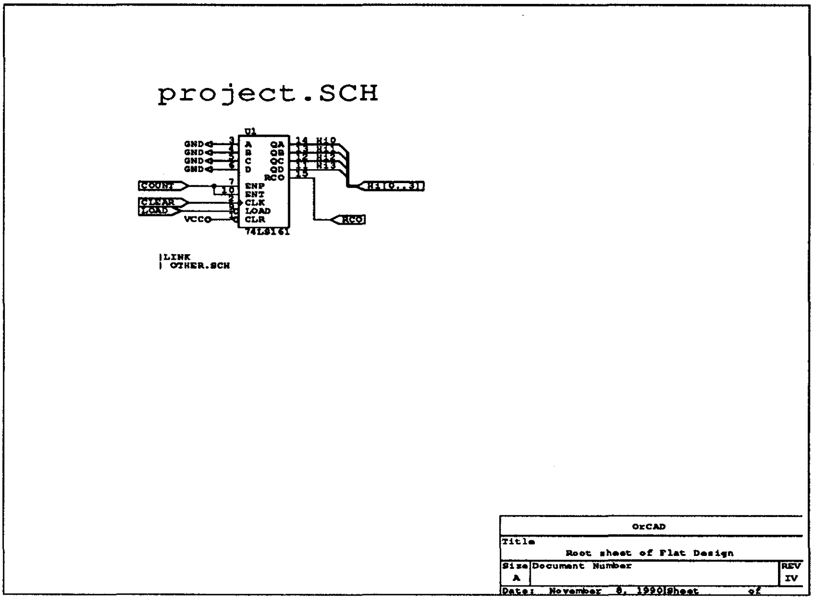


Figure 8-22. Root sheet of flat design.

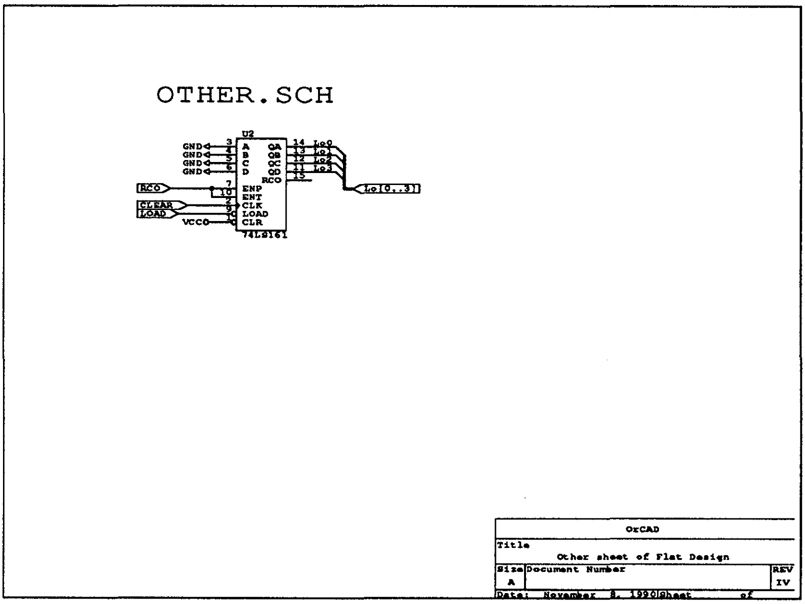


Figure 8-23. Other sheet of flat design.

A

Analog ▪ Circuitry where both voltage and frequency output vary continuously as a function of the input.

Annotation ▪ Assigning reference designators to components in a schematic.

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 instructions used to edit text on a computer, such as Backspace, Carriage Return, Line Feed, etc.

B

Bulletin board system ▪ A computer system for sending and receiving bulletins, messages, and files over telephone lines.

Button ▪ A pushbutton-like image that you click to initiate an action.

Byte ▪ A piece of computer data composed of 8 contiguous bits that are grouped together as a single unit.

C

CAE ▪ An acronym for *computer aided engineering*.

Check box ▪ A small square button: ❑. Check boxes are used in lists of options when more than one option can be active at a time.

Complex hierarchy ▪ A design in which two or more sheet symbols reference a single worksheet. Compare with *simple hierarchy*.

Configuration ▪ The information a program uses to operate. The configuration can be tailored to your needs.

Connectivity database ▪ The *connectivity database* consists of the incremental connectivity database (created by INET) and the linked connectivity database (created by ILINK). It describes the connectivity of a design, and is used to transfer a design to **Digital Simulation Tools** or **PC Board Layout Tools.** See *incremental connectivity database* and *linked connectivity database*.

Cursor ▪ A square marker inside a text field showing where characters typed on the keyboard will appear: ■
See *pointer*.

D

Default ▪ A preselected parameter.

Design cycle ▪ The process of conceiving, developing, testing, and producing a circuit.

Digital ▪ Circuitry where data in the form of digits are produced by binary on and off or positive and negative electronic signals.

E

EDA ▪ An acronym for *electronic design automation*.

Editor ▪ A tool used to create or modify a design file.

Entry box ▪ A box indicating that something (text or numbers) should 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, hierarchical structure.*

H

Hierarchical design ▪ A schematic structure in which sheets are interconnected in a tree-like pattern vertically and laterally. At least one sheet, the root sheet, contains symbols representing other sheets, called subsheets.

I

Incremental connectivity database ▪ INET produces the *incremental connectivity database.* It consists of an incremental connectivity database file (.INF) for each sheet in the design and an .INX file. The .INF file is a description of connectivity on each sheet. The .INX file lists each sheet referenced in the design. The *incremental connectivity data base* is used by ILINK to create an incremental netlist. See *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.

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 ▪ ILINK produces the *incremental netlist structure.* This consists of the .INS (instance) file, the .RES (resolved) file, and the .PIP file (contains pipe link commands). These files are used by IFORM to create a netlist in one of over 30 formats.

K

K ▪ A unit of measurement. 1K byte is equal to 1024 bytes. The "K" is taken from the metric system, where is stands for "kilo," or 1000. 1024 is 2^{10} and is close to 1000.

Key field ▪ To tell **Draft** and other tools which fields you want to combine and compare, *key fields* are used. A key field lists the part fields to combine and compare. Key fields are defined on the **Configure Schematic Tools** screen.

L

Library ▪ A collection of standard, often-used part symbols stored as templates to speed up design work on the system.

Librarian ▪ A tool used to manage or create library parts.

Linked connectivity database ▪ ILINK can optionally be configured to create the *linked connectivity database.* This ASCII file has an extension of .LNF and is used to transfer to **PC Board Layout Tools.**

Local configuration ▪ Configuration settings for a particular button. Roughly synonymous with *command line switches.* The same tool can have different configuration in different places in the same design. For example, **Netlist** is configured differently under the **To Layout** button and under the **To Simulate** button.

M

MB ▪ An abbreviation for *megabyte.* See *megabyte.*

Macro ▪ 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 ▪ Slightly more than one million bytes; 10 megabytes equals 10 million bytes. A megabyte is equal to 2^{20} bytes (1,048,576). "Mega" is taken from the metric system, where it is a prefix meaning one million.

Module port ▪ Graphical objects that conduct signals between schematic worksheets. See *flat file.*

N

Net ▪ Just as signals are conducted between schematic worksheets through module ports, they are conducted into and out of sheet symbols through graphical objects called *nets.*

Netlist ▪ An ASCII file that lists the interconnections of a schematic diagram by the names of the signals, modules, and pins connected together on a PCB. The nodes in a circuit. See *incremental netlisting.*

P

PCB ▪ An acronym for *printed circuit board.*

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 field ▪ A slot for holding text or data to be associated with a part. Each part has two part fields reserved for part value and part reference. It has eight other part fields that can be used to store other useful information. See *key fields.*

Pointer ▪ An arrow on the screen that moves as you move the mouse: ↖
See *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 round button: ○. Radio buttons are used in lists of mutually exclusive options: only one button can be active at a time.

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 at the top of a multiple-sheet 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:

- ⇞ Page Up
- ▲ Line Up
- ⇟ Page Down
- ▼ Line Down

Sheet symbol ▪ Block-shaped symbols representing other worksheets. Signals are conducted into and out of sheet symbols by graphical objects called nets. See *nets.*

Simple hierarchy ▪ A one-to-one correspondence between sheet symbols and the schematic diagrams they reference. Each sheet symbol represents a unique subsheet. See *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.

Text export ▪ The process of copying text from a schematic worksheet to an ASCII file.

Text import ▪ The process of copying text from an ASCII file to a schematic worksheet.

TTL ▪ An acronym for *transistor transistor logic.*

Tool ■ A tool is a computer program you can use to do some useful task. Tools are grouped into five categories: editors, processors, reporters, librarians, transfers.

Tool set ■ A collection of tools designed to perform a suite 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

Worksheet ■ **Draft** calls the sheets of drafting paper on which the schematics are drawn *worksheets*. Worksheets appear on the computer screen as a rectangular area in which you can place parts and draw wires.

Z

Zoom ■ The ability to change the view on the screen by making the objects appear larger or smaller.

NOTES

NOTES

NOTES

OrCAD®

01-5008 4/91 WCP 12K