Chapter

3   CapFast Tutorial

The CapFast tutorial is divided into two sections: The JumpStart tutorial and the Advanced Tutorial. Both tutorials are presented in a step–by–step format to minimize the time you need to spend and to give you a basic introduction to the software.

JumpStart. The JumpStart Tutorial shows you how to create a simple schematic and how to use many of the basic CapFast commands. It is a hands–on introduction to the CapFast schematic editor and the process of creating schematics. We start at the very basic level, showing you how to enter commands and work the menu hierarchies. From there you create your design.

Advanced. The CapFast Advanced Tutorial picks up where the JumpStart leaves off. We show you how to create your own symbols, how to create a multi–page design and how to use properties to pass information to netlisters. When you have finished the Advanced Tutorial, you will have a very good understanding of the CapFast software.

After finishing the Tutorials. When you finish the tutorials, you might want to examine the chapter(s) that refers to your special interest. Here's a list of places in the *CapFast User's Guide* that you might look at.

Extracting PCB Netlists – Chapter 8.

SPICE Analog Simulator Interface – Chapter 9.

HILO & SUSIE Digital Simulator Interfaces – Chapter 10.

PLD Tools – Chapter 11.

You will also want to review the EDIF 200 Translator's Guide if you are interested in performing translations between EDIF and CapFast formats. EDIF is a standard format used by many popular EDA programs, and is used for translation to EPICS databases.

1 ➤

➤ 2

file:///C:\CAPFAS~1\SCH_WIN2.BMP

3 ➤

4 ➤

➤ 5
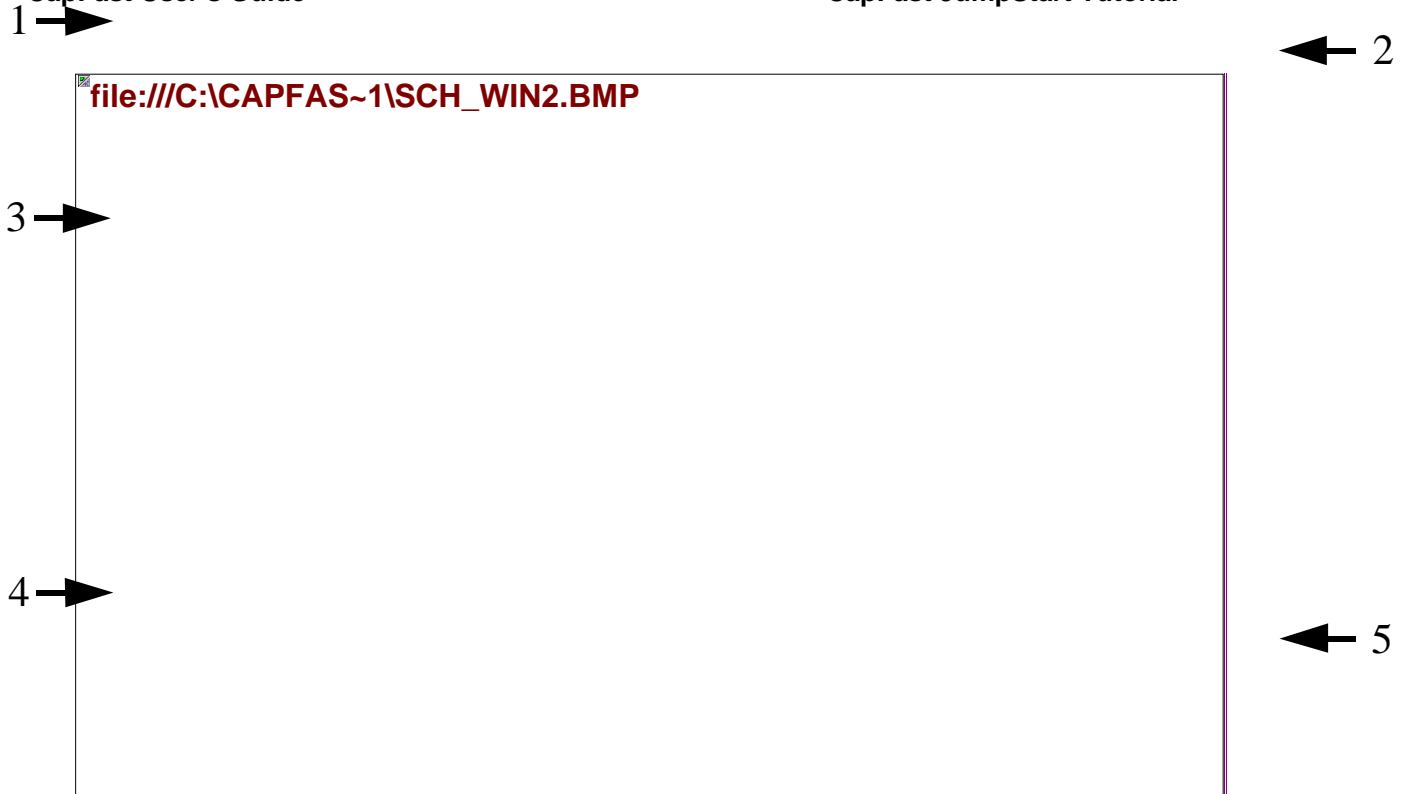
Figure 3–1. The CapFast *Schematic Editor (schedit)* window in Windows 95.

3.1   CapFast JumpStart Tutorial

This tutorial lets you jump right in and get your hands dirty. You'll create a simple schematic with CapFast's *Schematic Editor (schedit)*. Here's what you'll learn how to do.

- Get and place library components.

- Select, copy, move, and delete components.

- Get and place input, output, Vcc, and Gnd connectors.

- Draw wires.

- Assign reference identifiers and device pin numbers.

- Write a title and your name on the drawing.

- Do an electrical rules check on a drawing.

- Plot a drawing.

3.1.1 Setting Up a Page

Start CapFast as described in the installation section for your computer (Personal Computer or UNIX systems). Soon the design area, the menu bar, and the mouse button area appear on the screen as shown in the illustration below. Your screen may look slightly different, depending on what operating system you are running. Our screen shots are from Windows 95.

The *Schedit* window contains five areas.

1. *Menu bar*: Located at the top of the window, the menu bar contains the top level CapFast menus. You can select menus with the mouse or by using the keyboard.

2. *Mouse button area*: This area describes the current functions of the mouse buttons. In the left corner are the x,y coordinates of the last point you placed. (We'll show you how to place points later). In the right corner, the current x,y, cursor position appears.

3. *Design area*: This area contains the grid and is where you create schematics.

4. *Status area*: Located almost at the bottom of the window. This area provides information about the editing environment.

5. *Command and response area*: Located at the bottom of the window. This area displays prompts and feedback. If the computer beeps, look in this area for any error, warning, or informative messages.

3.1.2 Cursor and Mouse

In , the cursor is the pointer located in the center of the window. When you move the mouse, the cursor moves in the direction that the mouse moves. Move the mouse in several directions to get a feel for the cursor and how to control it.

Observe the mouse button area along the top of the screen. This area tells you what each mouse button does when you press it. Mouse functions will change depending upon cursor location and the command you are executing.

Figure 3–2. The CapFast *Schedit* menu commands

3.1.3 Entering Commands

You can enter commands for schematic entry in a variety of ways, one of which is to use the mouse. The first step in entering a command with the mouse is to move the cursor over the desired menu in the menu bar and press the left mouse button. A menu appears with additional menus or commands. Now move the cursor over the desired command and select it with the left mouse button.

Another way to enter commands or menus is to use the keyboard. You can enter commands or menus by typing the underlined letter of that command's or menu's name. For example, press **F** to show the `File` menu. This tutorial mostly uses the keyboard method, but you are encouraged to experiment with the combination you like the best.

For reference, *Figure 3–2* shows a listing of the *Schedit* commands in the hierarchy of menus. The bold entries show the menus in the menu bar and the indented text represents commands or menus within the hierarchy. The underlined letters represent the shortcut keys to access that command or menu from the previous level.

There are also some commands that are not available from the default menus. For more information, see the section on Internal Commands in *Chapter 5: The Command Reference Guide*.

3.1.4 Example showing how to enter commands

6. Select a menu from the menu bar by using the mouse, by using the cursor (arrow) keys, or by typing the underlined letter of the menu's name. If a command does not have an underlined letter, such as some commands in the `Part` menu and some custom menus, you can type the first letter of the command. Note that if more than one command starts with that letter, each time you type it, the highlight moves to the next item that starts with that letter.

7. When a menu appears, move the cursor on to any command and press the left mouse button. Or, press the key of the underlined letter in the desired command.

8. If another menu appears, follow the previous step to select a command.

9. If the command requires you to enter text, such as a name, type the required text and press **<Enter>**.

10. If you bring up a menu and decide you don't want it or you find you are in the wrong menu, just press **<Esc>** to get back to the previous level. A way to get back to the main level is to move the mouse outside of the menu and press the left

Figure 3–3. Tutorial schematic `spike1.sch`

mouse button. This will mark a point, which you can erase by pressing the right
mouse button.

3.1.5 Starting Your Schematic Drawing

11. The first step in starting a new drawing is to specify a file name for the drawing.
To do this, press **F** to get to the `File` menu. When the menu pops up, press **O** to
select `Open....` Remember that you can also use the mouse to select most
commands.

In the file requester, you are prompted for the schematic file name. This is a
standard file requester, which you can use to navigate through your directory
structure to find files. For this tutorial go to the directory where you want to store
your schematic files and type **spike1**, in the `File name` box. Then press
<Enter> or click on `OK`, even though the file does not exist yet.

This creates a new schematic named `spike1`. You can also wait until you exit
the program to create a file name. In this situation your schematic will be auto–
saved to `tmpsch.sch` while you are working on it, just in case you have a power
failure or computer crash.

12. Before you can place any parts, you need to tell *Schedit* what parts library to use.
To do this, press **C** for `Configure` and **L** for `Select Parts Library`. For
this tutorial, press **U** for the `SUSIE Library`.

13. The next step is to define the page size by putting a border on your drawing. To
do this, press **P** to bring up the `Parts` menu, and **B** for the `Border` menu. When
the Border menu appears, select `Complete border (with title + rev.`
`blocks)`. When the Border size list appears, select `A size`. Your A–size
border should appear on the screen, complete with title block and other elements.

Note for Windows 95 users: A "feature" of Windows 95 is how the menus seem
to be magnetically attracted to the mouse pointer. That is, the highlight follows
the mouse pointer instead of waiting for a mouse click. The result of this is that
menus will disappear if you move the mouse after using the keyboard shortcuts,
keeping you from selecting a menu item using the mouse. To select parts in
Windows 95, you must use only the mouse, only the cursor keys, or obtain and
use a utility such as *Old Mouse Mode* or *TweakUI*. For more information on

these utilities, visit *http://www.creativelement.com/win95ann/* and look for
Magnetic menus in the *User Interface* section.

14. Press the left mouse button to place the border.

3.1.6 Getting and Placing Library Parts

Now you're going to draw a schematic exactly as the one shown in *Figure* below. First
we'll place the library parts. Here are the steps.

15. Press **P** to bring up the P̲art menu. Then press **1** or select the 1)
    TTL54/74xxyyy (y:00-73) entry and press the left mouse button. When the
    parts list appears, move the cursor to the 74LS10 entry and press the left mouse
    button.

16. When the 74LS10 appears on the screen, use the mouse to move it to the place
    you want it on the page. Don't worry about precise positioning at this point.
    Later we will show you how to move parts around on the page.

17. Press the left or middle mouse button to fix the symbol in position.

18. Use the procedure described in steps 1 through 3 to get and place the rest of the
    gate symbols that you need for the drawing.

3.1.7 Selecting, Moving, Copying, and Deleting Components

Suppose that after you get a component placed, it is not where you want it, so you decide
to move it. To move, copy, or delete a component you must first select it.

Selecting

To select a component, place the cursor over it and press the left mouse button. This
places a numbered point on the part. You can make the point disappear by pressing the
right mouse button. Press **S** to bring up the S̲elect menu. Note that in the S̲elect
menu, you can select a single item, add items to those already selected, select a region of
components, or clear selections. For now, press **I** to choose the I̲tems command. All
parts that had points placed on them change color, indicating that they are now selected.

If you choose any of the S̲elect commands without first placing points, you will be
prompted to select an item or define a rectangle, depending on what command you
choose. Now you can select items one at a time or with a box and see the results as you

go. A component will change color as you move the pointer over it at the point where it would be selected if you click the left mouse button. The advantage to this method is that you can see if a part is selected as you go along, and if the wrong part gets selected, you can deselect it by pressing the right mouse button. Press the middle mouse button or **<Space>** or **<Esc>** when you are finished selecting components.

Moving

Press **E** to bring up the E̲dit menu, and **M** to execute the M̲ove command. When you move the mouse, an image of the selected components will appear and follow the motions of the mouse around the design area. Use the mouse to move the image to the desired position and press the left mouse button to fix it there. Note that you can pan the display but you cannot change the zoom level while moving a component.

Copying

To copy selected components, you press **E** for E̲dit, then **C** for C̲opy. Now, when you move the mouse, an image of the selected component should appear on the screen and follow the movements of the mouse. Position the copy in the lower right corner of your drawing and press the left mouse button to fix it there. Press the middle or right mouse buttons or **<Esc>** to terminate the copy command. In the next step, we will show you how to delete this extra gate.

Deleting

To delete components, select the components if they are not already selected, and then press **E** to bring up the E̲dit menu. When the E̲dit menu appears, press **D** to D̲elete the selected components.

Undoing

If you accidentally delete something you want to keep, immediately type **E U** to execute the E̲dit®U̲ndo command. Try this to bring back the gate you have just deleted. Then type **E U** to delete it again.

3.1.8 Adding External Connectors And Vcc Pullups

19. To get the external connector symbols, press **C** to bring up the C̲onfigure menu, **L** for Select Parts L̲ibrary, then **P** for P̲CB Library. This changes the parts library in use from SUSIE which we were using previously to PCB. Now press **P** for the P̲arts menu, **L** for the Linear menu, and **C** to choose the Connectors/switches menu. When the Connectors/switches menu appears,

move the cursor over the conni entry to highlight it. Then press the left mouse button. When the input external connector appears on the screen, position it to the left of the top 74LS00 NAND gate (U1A as shown in *Figure 3−3*). Then press the left mouse button to fix the connector in position.

20. *Schedit* automatically assigned a name to the connector you have just placed, but you want to change the name to correspond to the name of the external signal. Assuming the component is already selected (shown in red), press **T** to bring up the T̲ext menu and press **R** to select R̲elabel. When prompted, type **A** and press **<Enter>**. The letter A should appear next to the input connector.

21. To move the connector name to the left of the connector as shown in *Figure 3−3*, press **T** to bring up the T̲ext menu and press **M** to select M̲ove. Use the mouse to move the image of the letter to the desired position and press the left mouse button to fix it there.

22. Use the procedure described in steps 1−3 to get, place, and label the rest of the input connectors on the page.

23. Get an output external connector, conno, from the Connectors/switches menu, place it on the output of the circuit, and relabel it with a Y.

24. To get a Vcc symbol, choose P̲ower in the P̲arts menu. When the power sources menu appears, press **V** to choose Vcc. When the Vcc symbol appears on the screen, use the mouse to place it in the desired location and press the left mouse button to fix it there. Get and place the other two Vcc connectors you need for the drawing.

3.1.9 Drawing Wires

The next step is to draw wires which connect the symbols together.

25. Press **W** to bring up the W̲ire menu and **W** again to execute the W̲ire command. You will see a prompt in the lower left hand corner of the window asking you to enter points. The points define the path of the wire.

26. For your first wire, put the cursor on the tiny box at the right end, output, of the center 74LS10 and press the left mouse button. Make sure you don't overlap the wire section that is already part of the NAND symbol, or the wire will not connect to the NAND symbol.

27. Now a line will follow the cursor as you move it. Move the cursor to the center input of the output NAND gate and press the left mouse button. If you make a mistake, press the right mouse button to back up to the last point you placed. When you have finished routing the wire, press the middle mouse button or **<Space>** to end the wire command. When you are prompted for the name of the wire, enter a name of your choice or press **<Enter>**, causing *Schedit* to automatically assign a name to the wire. Wires must have names so that you can generate a netlist for the circuit. Netlists can then be used in simulators or other tools.

28. We'll place the next wire using a different approach. Place a point on the output of the top 74LS10 by pressing the left mouse button. Move the cursor right three grid dots and press the left mouse button. This leaves another point. Note that the points are numbered. Each point defines where the wire will be routed on the design area.

29. Continue placing as many points as you need until the last one is placed on the top input of the rightmost 74LS10. Now press **W** twice and a wire will be routed through all the points you have just placed, in numerical order. Then press **<Enter>** if you want to accept the default wire name. If you missed an output or an input with either wire end, a small circle will be placed around the offending end. If this happens, it may be easier to delete the wire by pressing **E** then **D**, and then redraw it. The alternative is to reroute the wire, which is explained in a later section.

30. Repeat steps 1−5 as needed to draw all the required wires, including those which go to the input connectors, output connectors, and Vcc pullups.

3.1.10 Adding, Changing and Moving Text

In this section, you will learn how to write text in the title block as shown below. Of course, the text that you place can be different.

31. To start, use the Zoom Area command we described earlier to expand the title block so that it fills as much of the screen as possible.

32. The first text you will add is the name of your company to the top box of the title block. Move the cursor to the point in the box where you want the <u>bottom</u> of the first letter and press the left mouse button. This defines a point marked by a + and labeled 0. This point is your reference point. Remember, if you get a point in the wrong position, simply press the right mouse button to remove that point.

33. Press **T** to bring up the T̲ext menu, then press **A** to select A̲dd. A prompt appears at the bottom of the screen asking you to enter text.

Figure 3–4. A title block

34. Type in the desired text and press **<Enter>**. The text should appear at the position where you put the point. The text is red, which means that it is selected. Since the text is selected, you can do other operations on it if you want, such as M̲ove.

35. The default scale for text is 100 percent, which is too small for your company entry. Press **T** to bring up the T̲ext menu and **S** to select S̲cale. When prompted for the new scale, type in **200** and press **<Enter>**. The enlarged text should appear in the box. If you do not like the result produced by the scale factor you chose, you can just change the scale until you get the size you want. Likewise, if the text is not positioned quite where you want it in the box, you can choose the M̲ove command from the T̲ext menu and then use the mouse to move the text where you want it in the box. If you move the text and the box becomes obliterated, type **V R** to redraw the screen.

36. Repeat steps 2–5 to write the other text entries in the title block as shown in *Figure 3–4*. Insert your own name instead of the author's, and in the empty boxes insert any comments you want. If you do not mark a point for the text location before you bring up the T̲ext menu, you will position the text afterward. Incidentally, to speed up the text entry process, you can mark several points, and then just type in each text string when prompted by the T̲ext®A̲dd command.

3.1.11 Doing an Electrical Rules Check on the Drawing

The *Schedit* Electrical Rules Checker (ERC) allows you to check your design for potential errors, such as floating inputs, unconnected outputs, a gate output connected to its input, multiple outputs connected to the same point, and outputs connected to Vcc or GND.

37. To run the ERC on your design enter the command **:qcheck** (note the colon).

38. When prompted Full ERC on schematic page (e), or only delete ERC props (d)?, press **<Enter>**. This defaults to a full ERC on the schematic page. For more detailed information on ERC, see its entry in *Chapter 5: The Command Reference Guide*.

Figure 3–5. The Properties Window in *Schedit*. Properties may not appear in the order shown.

39. When the requester for the errorfile name appears, just press **<Enter>** or click on OK.

40. If the ERC finds no errors, rejoice and skip the next step.

41. If ERC finds any errors, the component(s) containing the errors appear in red on the screen, indicating that they are selected. If you can't determine the error from just looking at the selected components, use the Zoom Area command to expand the region containing the error(s) so you can see more detail. If you still can't determine the error, you can display the errorfile on the screen to get some hints. Here's how you do this.

3.1.11.1 Displaying the Errorfile

Open *Notepad*, *vi*, or your favorite text editor or viewer. You can minimize the *Schedit* window first if you wish. For more information on how to switch between applications in Windows, see the Help in Windows.

Next, open the file spike1.erc in the directory where your schematic is stored. This file contains the error messages from the ERC on your schematic. You can write some notes about the error messages or if desired, print a copy of the messages.

After you correct any errors you have found, run the ERC again. If you want to find symbols or wires reported in the ERC errorfile, press **S S**, for the Select®Symbol by Name... command, or **S W**, for the Select®Wire by Name... command. A prompt appears asking you for the instance name of the element you are searching for. Type in the instance name you want, **n#25** for example, and the symbol or wire will become selected, changing color.

Repeat the ERC–fix cycle until the error file shows 0 errors or warnings found at the end of it. (No parts will be highlighted *if* all selections were cleared *before* starting ERC.)

3.1.12 Adding References and Pin Numbers

If you are going to simulate or physically build a circuit, you need to assign references to gates, such as U1, U2A, and U3, and assign package pin numbers to each device in the drawing. These and other characteristics of a device are referred to as its *properties*, so you will need to work in the Properties window to do these operations.

42. Start *Schedit* and open spike1.sch if you haven't done so already.

43. When the page appears on the screen, use the V̲iew®Zoom A̲rea command (typing **V A** on a selected area) to adjust your view so that the gate section fills the screen. Then select the top 74LS00 as described earlier in this tutorial.

44. Press **R** to bring up the P̲roperties menu. Press **E** to E̲dit... properties. This will bring you to the Properties window. The buttons in the Properties window tells you the commands you can perform on properties. The large box contains a list of the properties defined for the selected device and the value currently assigned to each of those properties. You can scroll down and across the list using the scroll bars, or if one of the properties is highlighted, you can use the arrow keys and **<Page Up>** and **<Page Down>**.

    One property on the list is the comment property. For this device, the comment property has a value of quad 2-input pos-NAND gate. This of course means that a 74LS00 package contains 4 NAND gates. To get an idea of the different types of properties defined for a device model, read through the list.

45. Continue to work in the Properties window where we will make some changes. The first property you want to change the value of is the ref property, so highlight the ref:*opt* line in the properties list. Notice that the property name, ref, appears in the box next to N̲ame: at the top of the window, and the value, *opt*, appears in the box next to V̲alue:.

46. Click in the V̲alue: box and replace *opt* with **U1A**. Then click on A̲dd/Modify. The new ref value for the selected device should appear to the right of the ref: in the properties list.

47. The next step is to change the display mode for the property so that it will be displayed on the drawing. Click on D̲isplay... to bring up the Property Display window. There are three display options for a property, shown in the box labeled Visi̲ble: All, Value Only, and Nothing. All displays both the property name (ref in this case) and the value (U1A in this case). Value Only displays only the property value, and Nothing displays nothing. Click on Value Only.

    Also note the O̲rientation, S̲cale, and Justification options. You can experiment with these, or see *Text Menu* in *Chapter 5: The Command Reference Guide*. Click on OK to apply the changes to the Property Display, or Cancel to not apply them.

48. The pinnumber(A) property shows you the possible choices for one of the pin inputs on a 74LS00 quad NAND gate. pinnumber(B) shows you the possible choices for the other input on a 74LS00 quad NAND gate, and the

pinnumber(Y) property shows you the possible choices for the output pin number. These numbers are in order according to the 74LS00 chip package, so for example, for one of the NAND gates in the package, pin 1 is one input, pin 2 is the other input, and pin 3 is the output.

Select the pin(A) line and change the value of the property to **1**. Then click on Add/Modify.

49. Click on Display... and change the Property Display for pin(A) so that the value is displayed. Then click on OK.

50. Next, change the value of pin(B) to **2** and the value of pin(Y) to **3**. You can change the display option for both of them at the same time by highlighting both properties and then proceeding to the Display Properties window as before.

51. Repeat steps 2−6 to assign reference identifiers and pin numbers to your drawing as shown in *Figure 3−3*.

3.1.13 Saving the Drawing

Schedit automatically makes a backup of your drawing about every 15 minutes (the time is configurable under Options®Auto-backup Interval), but before quitting or printing, you need to save the latest version on disk.

To save your drawing to a file, press **F** to bring up the File menu, then press **S** to Save the file with the current name.

If you want to save the drawing to a different file name, type **F A** for File®Save As.... Then type in the desired file name, navigate to the desired directory, and click on OK. If the file already exists, *Schedit* will ask you if you want to overwrite the existing file. Press **Y** or click on Yes to overwrite the file, or press **N** or just **<Enter>** to not overwrite.

3.1.14 Plotting Your Drawing

3.1.14.1 PC Windows Users

If you are using Windows versions of CapFast, you should set the printer up before printing or plotting. To do this, select File®Print Setup.... Make sure you have chosen the correct printer or plotter and that the settings are correct. You will probably

want to select <u>L</u>andscape rather than Po<u>r</u>trait under Orientation since most drawings are wider than they are tall.

To print or plot your drawing, select <u>F</u>ile®<u>P</u>rint... and click on OK (there aren't many options here). You can enter the printer setup here as well, by clicking on <u>S</u>etup....

3.1.14.2 UNIX Users

To print your drawing on a LaserJet printer that lpr knows about, in a shell type:

```
schplot spike1 | ljii -x 150 | lpr
```

Note: The *ljii* device driver has to create the image on a bit–by–bit basis, so it will take some time to produce the plot. If you leave the -x 150 option out of the schplot command line, it will plot much faster, but the resolution will be only 75 dots per inch.

For more detailed information on plotting/printing, see *Chapter 7: Plotting Schematic Hardcopies*.

3.1.15 Quitting *Schedit*

After you have written your file, press **F** to get to the <u>F</u>ile menu, and then **X** to E<u>x</u>it *Schedit*.

Figure 3−6. Interrupt controller

3.2 CapFast Advanced Tutorial

The CapFast Advanced Tutorial gives a much more in−depth look at the CapFast schematic and symbol editors than the JumpStart tutorial does. The Advanced Tutorial shows you how to create complex circuits in a hierarchical layout. You'll learn how to create schematics targeted for PCB board layout. The schematic layouts contain logic for a simple interrupt controller, seven−segment displays, and decoders to drive the displays. In the process, you will create primitive and hierarchical symbols using the CapFast *Symbol Editor (symed)*.

Here's a summary of what the Advanced Tutorial will show you. You'll find out how to:

- Design a hierarchical symbol

- Route signals using buses

- Make a multi−page schematic

- Manipulate properties

- Create symbols as needed

- Use a wider variety of commands than you did in the JumpStart tutorial.

The CapFast Advanced Tutorial assumes that you are familiar with basic CapFast commands and that you know how to place and wire components in the design area. If you have not already done so, you are encouraged to go through the JumpStart Tutorial where these commands are explained.

3.2.1 Creating a Hierarchical Design

In this section, you will design a small schematic and then create a symbol to represent it. You can then include this symbol in other schematics. This feature is handy when you find yourself using the same elements over and over again in different schematics. We will begin by designing a simple interrupt controller to a microprocessor. The controller receives an interrupt and passes it on, then waits for the microprocessor to strobe an acknowledge line twice before the interrupt vector address is placed on the bus.

If you haven't already done so, start *Schedit* and select F̲ile®O̲pen.... When the file requester appears, enter **int** for the file name and click on OK. This file will contain the schematic that we are going to create for this tutorial. Ultimately, your schematic should look like the one shown in *Figure 3−6*.

3.2.2 Getting and Placing a Part

In creating a schematic, your first step is to get and place parts. You can do this in two ways. To get a J–K flip–flop (74LS112) you can either:

52. Type **:get 112 <Enter>** (remember to type the colon).

53. Or, enter the P̲art menu hierarchy to get a 74LS112 part. To do this, first make sure the parts library in use is the PCB Library (type **C L P**). You only need to do this one time after you start *Schedit*, unless you switch between libraries. Then, select the 74LS112 from P̲art®TTL/Logic®Standard DIP®1) 54/74xxyyy (y:00-124)®112 (type **P T S 1** and select 112). The **:get** command is handy if you know the exact name of the symbol you want, but if you don't, use the menus. Most of the parts in this tutorial will be obtained with the **:get** command, but you could also use the P̲art menu.

Once you get the part, place it by pressing either the left or middle mouse button.

3.2.3 Making Default Part Names Visible

We're going to name parts as you place them. You'll find it makes the schematics easier to read and this tutorial easier to follow. *Schedit* has assigned a default name to the J–K flip–flop that you have just placed. This name is not visible but can be made so by doing the following:

54. Enter the O̲ptions®O̲bject Visibility menu by typing **O O**.

55. Then press **D** to toggle Default Names Visible to ON.

56. You will be asked if you want to save the change (to your schedit.rc file). Saving this change to your schedit.rc file keeps this change in effect for future editing sessions.

57. You will have to redraw the window by typing **V R** for V̲iew®R̲edraw for the names to appear.

You have now made default wire and symbol names visible. This is a handy way to see default names assigned by *Schedit*. If you do not want to see the default names, just repeat these steps, toggling this option to OFF.

3.2.4 Naming Parts

Name the first part **JK1** by selecting the J–K flip–flop and using the T̲ext®R̲elabel command, and entering **JK1** at the prompt.

3.2.5 Copying and Placing a Part

Copy the flip–flop symbol first. Press **E C** for E̲dit®C̲opy. Since the JK1 symbol was already selected, you did not have to select which part to copy.

Place the new part to the right and slightly above the other J–K flip–flop. Don't worry about placement. If parts are too close to each other, you can move them. Click the middle or right mouse button or press **<Esc>** to end the C̲opy command. Rename this part **JK2** by using the T̲ext®R̲elabel command.

The C̲opy command copies the part with no instance properties. What this means is that if you made any changes in the property window, those changes do not appear in the copied part. The new copy, therefore, contains no instance properties.

To copy properties from one part to another, first select the part that you want to copy the properties from. Then type **R E** for P̲roperties®E̲dit. When the Properties window appears, select the property you want to copy and click on C̲opy in the lower right corner of the window. When the Properties window disappears, click on the part you want to copy the properties to.

3.2.6 Getting, Naming, and Placing More Parts

Next we'll get a D flip–flop to latch the interrupt signal. Type:

```
    :get 74
```

to get a 74LS74 part and place this about seven grid units below the first J–K flip–flop. Name this part **DF1**.

Let's move the view to the right to make it easier to place more parts. Press the right arrow key to pan the view to the right. If the screen pans too little or too much for you, CapFast allows you to set the panning increment to an amount that you like.

3.2.7 Setting panning increments

You can set the panning increment by doing the following:

58. Type **O V** to bring up the O̲ptions®V̲iew Options menu.

59. Press **P** to change the P̲an Amount. Set the pan amount to what you want, in windowfuls. Default panning is set to half (0.5) of the display window. The zoom in and zoom out factors can be set here as well.

Let's get some more parts for the schematic. Type:

```
    :get 245
```

to get a 74ALS245 transceiver. Place the part to the right of the second J–K flip–flop, JK2. Name this part **TR**. Type **5** or **V V** to see everything you have placed so far. Now type:

```
    :get 00
```

to get a 74LS00 NAND gate. Place it about four grid points before the G input of the 74ALS245 transceiver. Name this part **NAND1**.

Now type **E C** to make copies of the NAND gate. Drag the first copy so that it is a few grid points after the Q output of the 74LS74 flip–flop. Press the left mouse button to place the copy.

Continue making copies. Drag the cursor and place a copy of the NAND gate right after the one you just placed. Place another copy of the NAND gate under JK1 and to the right. Finally, press the middle or right mouse button to end the copying process.

Name the NAND gates you have just copied so that the first NAND gate you copied is NAND2, the second one is NAND3, and the third one is NAND4. You can do this in one of the following ways.

> 60. By selecting each gate individually and renaming them one by one, or

> 61. By selecting all three of them at once and entering the Text®Relabel command. This would highlight the name of each gate, one by one, allowing you to enter the new name.

If you want NAND4 to look like the example schematic, select it and enter the Edit®Flip Y command to flip it across a vertical line.

At this point, you have placed almost all the components necessary for the design. You will find it is easier to place all the components before you do any wiring because later you'll probably want to move parts around.

3.2.8 Getting and placing hierarchical connectors

The last parts you need to get are hierarchical connectors. Here are the steps to get the connectors.

> 62. Press **P** for Parts, and then **C** for Schematic Connectors.

> 63. Select Hierarchical Input.

> 64. Place this input connector to the left of JK1, and in line with its CLK input.

65. You will be prompted for an instance name, which is the name you want this
    connector to have. Type:

     INTA~

66. Repeat steps 1–2 to select another hierarchical input connector and place this one
    to the left of the CLK input of the 74LS74. Name this connector INT.

67. Repeat step 1 and select Hierarchical Output. Place this connector to the right
    and slightly above the 74ALS245 transceiver. Label this connector **BUS[0–7]**.

68. Place another output connector to the right of the two NAND gates and label this
    connector **INTR**.

You have now defined outputs and inputs to the symbol that describe your schematic,
one of which is a bus output. Other than placing Vcc and GND symbols, all the symbols
have been placed and you can now begin wiring.

3.2.9 Wiring a Bus

To begin wiring a bus, you need to get a bustap. Here are the steps.

69. Type **V A** to enlarge the area that has the 74LS245 transceiver and the output
    connector labeled BUS[0–7].

70. Type **W** to enter the Wire command, then **T** to get the Bustap Symbol.
    Position the tap just to the left of the output connector and press the left mouse
    button to place it.

71. Type **E** to enter the Edit menu, then **Y** for the Flip Y command. This flips the
    bustap about the y–axis, orienting the bustap in the right direction. Your goal is
    to have the top right–hand corner pointing towards the output connector, and the
    bottom–left corner of the bustap pointing towards the transceiver.

72. Type **E C** to enter the Edit®Copy command. Place seven copies of the bustap
    below the first one with one grid point space between each one. You may want to
    use Select®Area by typing **S A**, to select all the bustap inputs and then move
    them all at the same time, making them level with the transceiver outputs, if they
    aren't already level.

73. Place a point on the nose of the output connector using the left mouse button.
    Place points on each of the upper right boxes of the bustaps, in order from top to
    bottom.

74. Type **W B** to enter the Wire®Draw Bus command. A wire segment goes from
    the output connector through the bustaps and a prompt appears asking for an
    instance name. Label this wire after the output connector, **BUS[0–7]**. When the
    program warns that the name is being used by a connector and asks if you want to

select a different name, click on <u>N</u>o to keep the bus the same name as the connector.

Note that you can use the <u>O</u>ptions®<u>W</u>ire Options®<u>N</u>ame Wires After Connectors command to automatically label wires with the same names as connectors.

3.2.10 Making Wire Names Invisible

Note that the wire name appears eight times on the design. This can make your design look cluttered, especially if you leave default names visible. If this is offensive, you can remove the names by doing the following:

75. Type **S C** to clear all the selections.

76. Place points on the wire segments between the bustaps. Do not place a point on the wire segment between the top bustap and the output connector.

77. Type **S I**, and all the selected wire segments, along with their respective labels, become highlighted.

78. Type **O T C** for <u>O</u>ptions®<u>T</u>ext options®<u>C</u>ommands Affect All Text. This option will toggle from NO to YES.

79. Now type **T D** for <u>T</u>ext®<u>D</u>isplay. You will see a prompt at the bottom of the screen:

```
New text visibility (on or off)? [on] *
```

Go ahead and type **off <Enter>** to turn off all the selected labels. You might want to go back into the <u>O</u>ptions®<u>T</u>ext Options menu and toggle the <u>C</u>ommands Affect All Text option back to NO.

3.2.11 Wiring Bustaps to the Transceiver

Next we will place wires from the bustaps to the outputs of the 74ALS245 transceiver. Here are the steps.

80. Type **W N** to toggle the <u>W</u>ire®Auto-<u>n</u>ame Bus Wires feature ON, or type **:bus**. Enter **BUS 0** for the name BUS beginning with bus index 0. Until you toggle <u>W</u>ire→Auto-<u>n</u>ame Bus Wires to OFF, all wires are named BUS and are subscripted by an array index that is automatically incremented by *Schedit*.

81. Begin by placing a point on the B8 (pin 11) output of the transceiver and another one on the bottom bustap.

82. Now type **W W** and *Schedit* automatically draws a wire with the subscripted wire name BUS[0]. If you choose not to use the <u>W</u>ire®Auto-<u>n</u>ame Bus Wires

feature, you must manually label all wires from bustap segments to components. Make sure that these names have the same name as the bus and have unique subscripts that are within bus array boundaries.

Since wiring can be a rather tedious chore, you might find it easier to assign this to a macro function key.

3.2.12 Creating a Macro

83. Type **M** to enter the Macro menu, then **R** for Record....

84. Press the **<F1>** function key for the macro name. Notice the prompt telling you that *Schedit* is recording a new command Func1.

85. Place points on the B7 output from the transceiver and on the second bustap from the bottom.

86. Type **W W** to enter the Wire®Wire command. The second wire will be automatically named BUS[1] if you are using Auto-name Bus Wires....

87. Select Stop from the Macro menu to exit the macro command definition mode.

3.2.13 Running a Macro

88. Now place a point on the B6 output of the transceiver and another one on the third bustap from the bottom.

89. Press the **<F1>** function key. *Schedit* automatically draws the wire and gives it the name BUS[2].

90. Continue doing this until all the bustaps are wired, and then toggle the Wire®Auto-name Bus Wires option to OFF by pressing **W N**.

3.2.14 Completing the Schematic

Now we get to the final stages of your drawing. The rest is pretty straightforward. Most of it is wiring.

3.2.14.1 Wiring Inputs and Outputs

91. Place a wire from NAND3 to the INTR hierarchical output connector. Then place a wire from NAND1 output to the G input of TR. You can press **<Enter>** to accept the default name assigned by *Schedit*, or enter your own. If you enter a name, make sure that it is not the same as another wire that is not attached to the one you are naming.

Figure 3–7. The *Symed* window

92. Wire the output from NAND2 to both inputs of NAND3. Since the middle mouse button lets you repeat the last command you entered, you can repeat the wiring process by placing points and then pressing the middle mouse button.

93. Place a wire from the Q output of DF1, the D flip–flop, to one of the inputs of NAND2.

94. Wire the Q output from JK1 to the J input of JK2. Place a wire from the Q~ output of JK1 to the other input of NAND2.

95. Wire one input of NAND1 to the Q output of JK2. The other NAND1 input goes to the Q output of JK1.

96. Now place a wire from the INTA~ input to the CLK inputs of JK1 and JK2. You will want to place a wire from the INTA~ input to one input on NAND4. The other input on NAND4 is wired to the Q~ output of JK2.

97. Now wire the output of NAND4 to the CLR input of all three flip–flops. Place a wire from the INT hierarchical input connector to the CLK input to the D flip–flop.

3.2.14.2 VCC and GND

Next, you need to get power and GND symbols.

98. The K inputs to the J–K flip–flops should go to GND, and the PRE input of all flip–flops should go to Vcc.

99. The J input to JK1 should go to Vcc as should the D input of DF1. You can place any vector address you want on the 74ALS245 transceiver, the one shown is 04H.

3.2.14.3 Adding a Border

The only thing left now is the border around your schematic. Select a B size complete border from the Part®Border menu. Place it around your schematic. Then zoom in on the title block and, using the Text®Relabel command, enter your company information and your name if you want.

You are now done drawing your schematic. Save it to a file and exit *Schedit*.

3.2.15 Creating a Hierarchical Symbol

To complete the hierarchical process, you must create a symbol with the same name as your hierarchical schematic. You create symbols with the symbol editor called *Symbol Editor (symed)*.

3.2.16 Starting *Symed*

To start *Symed*, double click on the *Symbol Editor* icon in the CapFast for Windows group or, for UNIX users, type **xsymed<Enter>**.

The *Symed* screen is very similar to *Schedit*'s but has a few different features. You'll see that *Symed* has many of the same commands and the general layout of the screen is also the same. Remember though, that *Symed* is strictly used for symbol creation.

3.2.17 Creating a Symbol

100. Begin by typing **P F** or **:import** to enter the P̲orts®F̲rom Schematic... command. This will allow you to import into *Symed* the hierarchical connectors you placed on your schematic.

101. At the prompt, type:

    int

    and after a while you will see the connectors come up on the display area. Place them using the left mouse button. *Symed* preserves the distance between connectors that it found on the schematic. These are bound to be too far apart.

102. Type **S A** to enter the S̲elect®A̲rea commands and enclose the two input connectors. Move them close to the output connectors and place them using the left mouse button.

103. Now select one of the input connectors and move it to within two grid points below the other one. Do the same thing for the output connectors. Magnify the area where the connectors are by typing **V V**. Notice that the connectors have the same names as those from the int schematic.

104. Type **D R** to enter the D̲raw®R̲ectangle command. Create a box that is seven grid points high and four grid points wide. Place the two input connectors two grid points to the left of the box, and then place the two output connectors two grid points to the right of the box.

105. Place a point on the INTA~ connector and another one on the box. Press **D V** to enter the D̲raw®V̲ector command. A line appears from the connector to the box. In a similar fashion, draw lines for the remaining connectors. You have now defined the input and output ports to the symbol describing your schematic.

3.2.18 Adding Properties to the Symbol

At this point you may want to put properties on the symbol you just created. Which properties you want to define and give values to depends upon which netlister you plan

Figure 3–8. A simple synchronous controller.

to use after you have finished you design. We will only put a `comment` property on this symbol. Here are the steps.

106.Type **R E** to call up the `Properties` window. It contains no defined properties. Look over the window for a minute and familiarize yourself with it.

107.In the `Name:` box, type in **comment**, and in the `Value:` box, type **INTERRUPT**. Then click on `Add/Modify` to add the property.

108.Click on `Display...` and make the `comment` property's value, `INTERRUPT`, visible. Then click on `OK`.

109.Click on `OK` to get out of the `Properties` window. The property name appears somewhere around the symbol you have just created. To place it inside the symbol, do the following.

110.Type **R E** to call up the `Properties` window, then select the `comment` property.

111.Click on `Move`. The `Properties` window disappears and the comment `INTERRUPT` becomes highlighted. Move the comment so that it is within the symbol box and press the left mouse button to place it.

You may want to add other properties to the symbol. This will depend upon what you plan to use the symbol for, such as PCB layout, SPICE extraction, or digital simulation. Please read the tutorials that pertain to each of these for more detailed information about properties. Also, see the *Chapter 6: Properties* for a chart that lists the properties associated with each netlister.

3.2.19 Mapping Schematic Connectors to Symbol Ports Created in *Symed*

Although we began the Advanced Tutorial by having you create the basic schematic in *Schedit* and then do your wiring, we could have first had you create a symbol in *Symed*. After creating the symbol, you could import its ports into *Schedit*. If you prefer this method, starting in *Symed*, then you must know how to import your symbol into *Schedit*. You do this by mapping the connectors into *Schedit*. We'll show you how with a simple symbol.

112.Bring up *Symed* and draw a box.

113.Type **P V** for `Port®Add Visible Port`.

114.Select `Input` for the inputs to the box and `Output` for outputs from the box. Give the ports names.

115.Draw vectors from each port to the box.

116.Save the symbol, exit *Symed*, and start *Schedit*.

117.When you are in *Schedit*, type **F H M** or **:import** to import the symbol ports. For the file name, enter the symbol file name that you just saved.

The inputs and outputs from the symbol will now be mapped into the design area for use with your schematic.

Note: You can use the <u>F</u>ile®<u>H</u>ierarchy menu to edit a symbol that has an underlying schematic. Just select the symbol and type **F H D** to <u>D</u>escend in the schematic's hierarchy, to edit the schematic that represents the symbol. Now you can see the schematic that defines the symbol and even edit it if you want to. If the symbol is a primitive symbol, which means that no schematic defines it, you will see a design area with no schematic in it. To return to the top level schematic, type **F H A** to <u>A</u>scend in the schematic's hierarchy.

3.2.20 Multi–page Schematics

Creating a multi–page schematic is essentially like creating any other schematic. The main difference is in which type of connector you use. For multi–page schematics you use offpage connectors, the lozenge–shaped connectors shown in *Figure 3–8*, in addition to hierarchical ones. For example, if you wanted to make the circuit called interrupt into a page in a design, you need to change all the input hierarchical connectors to input offpage connectors, and all the output hierarchical connectors to output offpage connectors. To get an input offpage connector, for example, you would type **P C** to enter the <u>P</u>arts®Schematic <u>C</u>onnectors menu, and select Offpage Input. You then would not need a symbol to represent the circuit.

The design approach you use, design pages or hierarchical, depends upon your preference and the nature of the design. If you choose to make a multi–page design, you will have to keep track of offpage connectors. If your design is quite large, you can use the utility program *Design Cross Reference Generator* (*schxref*), to do this. See *Chapter 12: Utilities* for a description of this utility.

3.2.21 Creating Page One

118.At this point in the tutorial you should be fairly comfortable with getting symbols and wiring them together. Using the PCB parts library, draw the schematic shown in *Figure 3–8*, placing everything but the resistors, transistors, and wires.

The design in *Figure 3–8* takes inputs from two parallel ports. One input port is decoded to drive the common anode of one of six seven–segment displays. The other port sends an already decoded value to the selected seven–segment display.

119.Get a resistor by typing **P L A R** to select Res from the <u>P</u>art®<u>L</u>inear®<u>A</u>nalog menu.

120. Place the resistor and type **R E** bring up the `Properties` window.

121. Select the `value` property, and in the `Value:` box, enter **1K ohm**. Click on `Add/Modify` to enter the change. Then turn on the display of `Value Only` of the value property. Click on `OK` to exit the `Properties` window.

122. Now get a transistor by typing **P D B 1** for `Part®Discrete Semiconductors®Bipolar transistors®1) 2N696 - 2N3504,` and select a `2N2907A` pnp transistor. Place this transistor so that the base is just to the right of the resistor.

123. Wire the resistor to the base of the transistor and the collector of the transistor to Vcc. Then type **S A** to enter the `Select®Area` command and select the two wires, resistor, transistor, and Vcc symbol because you're going to make of copy of this grouping. Press **E C** to enter the `Edit®Copy` command. Place copies of the grouping so that you reasonably match the schematic in *Figure* . Now wire the parts together.

124. This is an example where you want to replicate an instance property, such as the value property of the resistor. To give the other resistors a value of `1K ohm`, select the first resistor, with the `1K ohm` already entered, and open the `Properties` window. Then select the `value` property and click on `Copy`. The `Properties` window will disappear and you will be prompted to select a symbol which you want to copy the property to. Click on one of the other resistors, and the `Properties` window will re–appear. Note that although the `value` property is not displayed by default, it was copied. Click on `Copy` again for each of the other resistors. Then close the `Properties` window by clicking on `OK`.

125. To turn on the display of the `value` property's value, select all the resistors that don't have it displayed. Then open the `Properties` window, select the `value` property, and turn its display on. Click on `OK` and the next resistor's `Properties` window will appear. Repeat the process to turn on the display for the value property of each resistor.

This completes the first page in a three page schematic. Save the schematic to a file, such as `address`.

3.2.22 Creating Page Two

The second page of the schematic will contain six seven–segment displays. Since wiring six seven–segment displays can be very tedious, you will be asked to wire only three seven–segment displays. A sample schematic, `sample3.sch`, containing all the displays is provided in the `wcs/examples` directory of the CapFast software.

Since there is no seven–segment symbol currently available in the symbol library, you will have to create your own. Here's how.

126. Start *Symed*. When the *Symed* window comes up, press **P V** to enter the <u>P</u>ort®Add <u>V</u>isible Port menu. Then select an Input and place it on the drawing area. Name it **CA** (common anode).

127. Use either the <u>E</u>dit®<u>C</u>opy command or the <u>P</u>ort menu to place eight more of these underneath the first one, one grid space apart. Name these ports as you go, from top to bottom, **A**, **B**, **C**, **D**, **E**, **F**, **G**, **DP** (decimal point). If you use the <u>E</u>dit®<u>C</u>opy command, you will have to name them by using the <u>T</u>ext®<u>R</u>elabel command after you have placed them.

128. Type **D V** and draw vectors 2 grid points long to the right of each input port. You can draw one and then use the <u>E</u>dit®<u>C</u>opy command to make the rest.

129. Type **D R** and draw a rectangle two grid points to the right of the input ports. Make it 8 grid points wide by 12 grid points high.

130. Type **D V** and draw the 8 that customarily appears on a seven–segment display. Then type **D C** for the <u>D</u>raw®<u>C</u>ircle/Arc command and draw the decimal point. If the decimal point seems too large and you can't make it smaller, try using a different snap grid size by:

3.2.22.1 *Changing the* `Snap Grid Size`

131. Type **C S** to enter the <u>C</u>onfigure®<u>S</u>nap Grid Size command.

132. Then enter a number such as **8** or **4** for the new snap grid size.

3.2.23 About Properties

You will now want to put some properties on this symbol. Properties are used by translation programs, called netlisters, to extract information about circuit elements, such as symbols. For example, you can create netlists for SPICE and HILO if you're going to do analog or digital simulation, or for PCB layout, as well as other purposes. The properties you will place in this example will be meaningful to a netlister that's used for printed circuit board design.

3.2.23.1 *Placing Properties*

In the following instructions, you will be instructed to enter properties in the form

```
property-name: property-value
```

Each time, enter property-name in the <u>N</u>ame: box and property-value in the <u>V</u>alue: box. Then click on <u>A</u>dd/Modify.

133. Type **R E** to get the Properties window up. A good property to begin with is the comment property. Enter the following property:

```
comment: 7 segment common anode display
```

Figure 3–9. Three seven–segment displays.

This property becomes the first property on the property list. It will be displayed, both the name and the value, unless you change its display mode.

134.The next property you add will have a qualifier in front of it. Enter the property:

    `(ASC)primitive: dip14`

All CapFast netlisters are case–sensitive so it is important that the qualifier contains all upper–case letters. The PCB netlister uses the qualifier to differentiate the property from other properties of the same name. For example, enter the property:

    `(!ASC)primitive: Senior 9A SEA 311ORE`

The PCB netlister will ignore this property. Other netlisters may either accept or ignore this property depending upon their default qualifiers and whatever options you use when you invoke them. See the netlister's manual and the appropriate chapter in the *CapFast User's Guide* for the netlister you plan on using.

135.Using the methods described above, add the following properties into the symbol:

```
ref: *opt*
section: A
reftype: *opt*
pin(A): 1
pin(B): 13
pin(C): 10
pin(D): 8
pin(E): 7
pin(F): 2
pin(G): 11
pin(CA): 3
pin(DP): 6
```

Notice the optional property values indicated by `*opt*`. They can be assigned values either by you or by a packaging program, such as the CapFast packaging program *Component Packager* (*pkgr*). A symbol with a `*req*` property must be assigned a value by the user, since programs like *pkgr* can not touch them. Netlisters that look for these properties generate error messages if the `*req*` value is still there.

136.You probably don't want all the properties displayed, so select the ones you don't want displayed by pressing **`<Ctrl>`** while clicking on the properties, or pressing the left mouse button on a property and dragging the pointer over the other properties. Then change the display mode to what you desire.

137.Save the symbol you have just created to a file, such as `7seg`, and exit *Symed*.

3.2.23.2 `ref` *Property*

A brief explanation of the `ref` property and how it is different from an *instance name* might be in order here.

When you bring up a symbol in *Schedit*, either you or *Schedit* must assign a unique instance name to that symbol. If you bring up a 7400 NAND gate, for example, *Schedit* might name it something like `00#23`. A 7400 chip package has four of these NAND gates in it. If you placed four NAND gates on the display screen, each one would have its own instance name but no idea that they somehow must be packaged into the same chip. This is what the `ref` property does. Each NAND gate would have its `ref` property edited so that its value would be the same, such as `U12`, for example. For a further explanation of properties, see *Chapter 6: Properties*.

3.2.24 Completing Page Two

Completing page two requires that you place the seven−segment symbol you have just created, as well as placing offpage and bustap connectors. Refer below to *Figure 3−9* to see what page two should look like.

138. Start *Schedit* and open the file you want to edit, such as segments. Then configure the P̲art menu to use the PCB library using C̲onfigure®Select Parts L̲ibrary.

139. Now bring in the seven−segment symbol you just created by typing **:get 7seg**. Note that you can do the same thing by typing **P G** to enter the P̲art®G̲et By Name command and then typing **7seg** at the prompt for the symbol name.

140. Make copies of the symbol and place them in a row, so that you have three seven−segment displays. Place two offpage connectors on the schematic as shown in *Figure 3−9*. Label the top offpage connector **ON[0−5]**. Label the bottom offpage connector **DATA[0−7]**. These are the same names as two of the offpage connectors placed on the first schematic page, `address.sch`.

141. Place the bustap connectors used by each bus and wire them to their appropriate connectors, as shown in *Figure 3−9*. You might want to make sure the N̲ame Wires After Connectors option is ON. To check this, type **O W** for O̲ptions®W̲iring Options, and see if N̲ame Wires After Connectors is ON. If it is, *Schedit* will automatically make your wire into a bus and give it the correct name.

142. Now toggle ON the Auto-n̲ame Bus Wires feature by typing **W N**, and enter the name and starting subscript of the bus. Wire up the schematic as shown in *Figure* .

Figure 3−10. Completed design.

Using the commands you've learned so far, complete the schematic as shown above in *Figure 3−9*, and save it to a file named segments.

3.2.25 Creating Page Three

Page three is the final page of the design. On this page, you will place the hierarchical symbol you created for the interrupt circuit, as well as offpage connectors that correspond to the offpage connectors P1[0-7] and P2[0-2] found on page one. In *Schedit*, type **F O** to enter the File®Open... command. Then enter **all** for the filename, click on OK, and a new design area will appear.

143. Type **:get int** and place the symbol representing your int schematic by pressing the left mouse button.

144. Get two offpage connectors by typing **P C** to enter the Part®Schematic Connector command, and select Offpage Output. Place them as shown in *Figure* . Label the top one **P1[0−2]** and the bottom one **P2[0−7]**.

145. Type **P L C** to enter the Part®Linear®Connectors/switches command. Then select a db25m twenty−five pin male connector. Place the components as shown in *Figure 3−10*.

146. Place bustaps and rotate or mirror them as required. Note that you will need bustaps for the bus that is coming from the hierarchical connector on the symbol interrupt.

147. Wire the connections as shown, using the Auto−name Bus Wires..., Name Wires After Connectors, and other commands already discussed.

3.2.26 Design Files

For different netlisters to be able to access all the schematics in a multi−page schematic, you need to create a design file using a text editor. The file should end with a .dsn extension. The design file for the multi−page schematic used in this tutorial should look like this:

```
[design]
segments
all
address
```

If you named your file board.dsn, this would specify that it is a design file composed of the schematics segments, all, and address.

3.2.27 Netlisters

Although netlisters are a complex subject, here's is a very basic explanation of what a netlister is and how it works. If you want to simulate your schematic or use it for input to a PCB layout tool, then you need to create a netlist. A netlist contains some or all of the electrical, physical and simulation characteristics of a schematic. When you create a netlist, you are extracting all or some of these characteristics and putting them into an information file, which you can then use as input to the simulator or the PCB layout tool, or any other tool that requires a netlist.

Properties are very important to the netlister because the schematic's electrical, physical and simulation characteristics are passed on to the netlister through property values that you have assigned when you created your schematic. Therefore, if you are going to be creating a netlist, you must be aware of which property values that particular netlister requires and which are optional. A chart appears in *Chapter 6: Properties* in this manual, showing you each CapFast netlister and the properties you must assign values to so that the netlister operates correctly.

3.2.27.1 Running Netlisters

For this tutorial, we're going to run two netlisters, *Component Packager* (*pkgr*) and *PADS−PCB Interface* (*sch2pcb*). *pkgr* automatically assigns pins and reference properties. *sch2pcb* creates a netlist that you can input to PADS−PCB, a PCB layout tool.

3.2.27.1.1 Qualifiers

Properties are the way in which you transmit information from your schematic to the netlister. Each netlister, by default, extracts information from certain properties and ignores other properties entirely. You can, however, tell a netlister to look at properties it normally ignores by placing a qualifier in front of that property. For example, if you want the CapFast packaging utility, *pkgr*, to look at a property it normally passes over, you must place ASC in front of that property. Then, when you run *pkgr*, you must also put the ASC qualifier on the command line to tell *pkgr* to add ASC−qualified properties to its list of properties from which it extracts information. Here's an example. Run *pkgr* with the command line

```
pkgr -Q ASC board
```

pkgr is further explained in a section below.

Some netlisters, by default, look for certain qualifiers. For example, *sch2pcb* looks for properties that have the ASC qualifier in front of them. In this situation, you do not need to put the ASC qualifier on the command line when you start *sch2pcb*, because it automatically will search out these ASC−qualified properties.

Remember, when you create your schematic, you must know which netlisters you're going to use so that you can qualify properties correctly within your schematic.

3.2.27.2 If you plan to use Design Cross Reference Generator

If you are planning to use the utility program *Design Cross Reference Generator* (*schxref*), discussed earlier in this tutorial, you will want to place borders around each schematic used in the multi−page design. Once you place a border, bring up its `Properties` window and give a value to the `page` property. The value is the page number. *Schxref* looks for this property and will print out its value. Before using *schxref*, however, you need to make a design file describing your multi−page schematic.

3.2.27.3 Running Component Packager

You'll find *pkgr* a convenient utility to use. It automatically generates `ref` and `pin` properties for components that don't have these already assigned, such as the NAND gates used in the interrupt schematic. When you run *pkgr*, you want it to look at properties that have an `ASC` qualifier. To do this, you must type `ASC` on the command line when you start *pkgr*. For example, for the design file `board.dsn`, type:

```
pkgr -Q ASC board
```

You can use *pkgr* on each schematic individually, but it is better to run it on the entire design to avoid duplication of reference designators on separate pages of the design. *pkgr* also annotates your schematics so that you can see what the reference designators are.

3.2.27.4 Running sch2pcb

sch2pcb creates a netlist that you can input to PADS−PCB. Run *sch2pcb* with the design file name. For example:

```
sch2pcb board
```

You don't need to specify the `ASC` qualifier on the command line since *sch2pcb* uses this qualifier as its default qualifier. Be sure that the properties in the schematic that *sch2pcb* needs to look at have the `ASC` qualifier. This netlister will package your design so that it can be imported into the PADS−PCB design environment.