

DesignWorks™ Professional Schematic Capture for Windows

Hands-On Demo Guide

Version 4.2 February 3, 2003

1641 Lonsdale Ave, Suite 603
North Vancouver, B.C., Canada, V7M 2J5

©2003 Capilano Computing Systems Ltd.

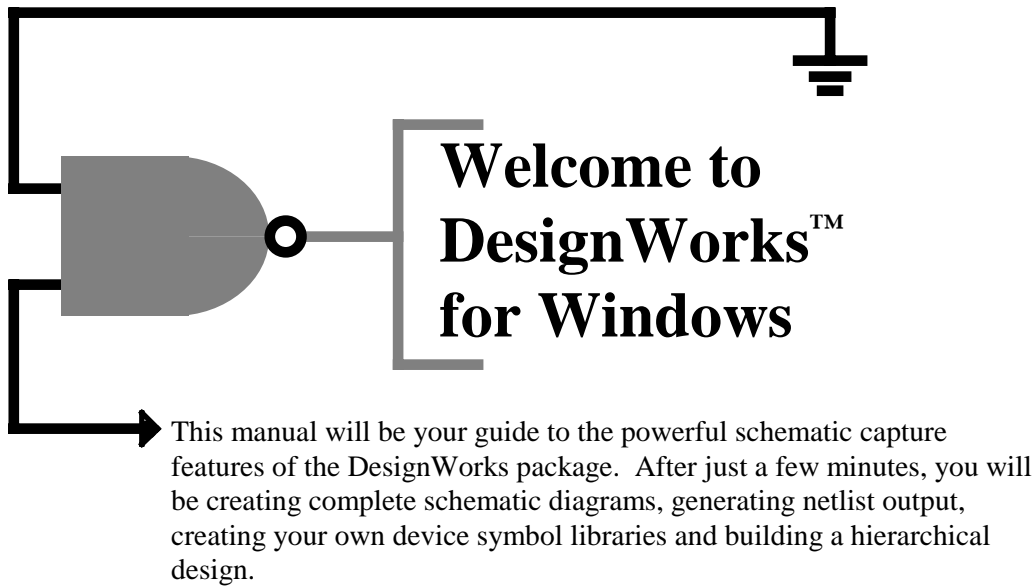
www.capilano.com

Table of Contents

INTRODUCTION	1
THE DESIGNWORKS SCREEN	2
MANUAL FORMAT	3
THE 5-MINUTE SCHEMATIC	4
Starting the Program.....	4
Creating a New Design.....	4
Choosing a Library.....	5
Selecting a Part	5
Placing a Device	6
Moving Devices	7
Using Undo/Redo	7
Part Selection by Name	7
Automatic Pin Connection	8
Wire Editing.....	9
Power and Ground Connections	10
Connector Devices	11
Connecting Signals by Name	11
Confirming Connections	12
Discrete Components.....	13
Setting Component Value	13
More on Power and Ground.....	14
Generating a Netlist.....	16
ADVANCED SCHEMATIC EDITING	17
Starting the Program.....	17
Navigating Around the Schematic.....	18
Deleting a Group of Objects	19
Creating a Bus.....	19

Table of Contents

Using the Clipboard.....	21
Adding a Page	22
Setting Sheet Size	24
Adding Text Notations	25
Creating Attribute Fields	26
Using the Browser Tool.....	27
Device Packaging	28
 DEVICE SYMBOL EDITING AND HIERARCHICAL DESIGN	 30
Starting the Program.....	30
Creating a New Library	30
Creating a Device Symbol	31
Setting Default Part Attributes.....	32
Entering Pin Names and Numbers	34
Saving and Using the Part.....	34
Auto-Creating a Symbol.....	36
Creating a Hierarchical Block	37
 USING DESIGNWORKS WITH SPICE-BASED SIMULATORS.....	 39
Starting the Program.....	39
Creating a SPICE Design	39
Placing SPICE Devices	40
Entering SPICE parameters	40
Creating a SPICE Netlist	42
 APPENDIX A - OTHER DESIGNWORKS FEATURES.....	 43
Libraries	43
Find and ErrorFind Tools.....	44
Report Generation	44
Custom Sheet Settings	44
Attribute Operations	44
Hierarchy and Part Type Operations	45
Back Annotation	45

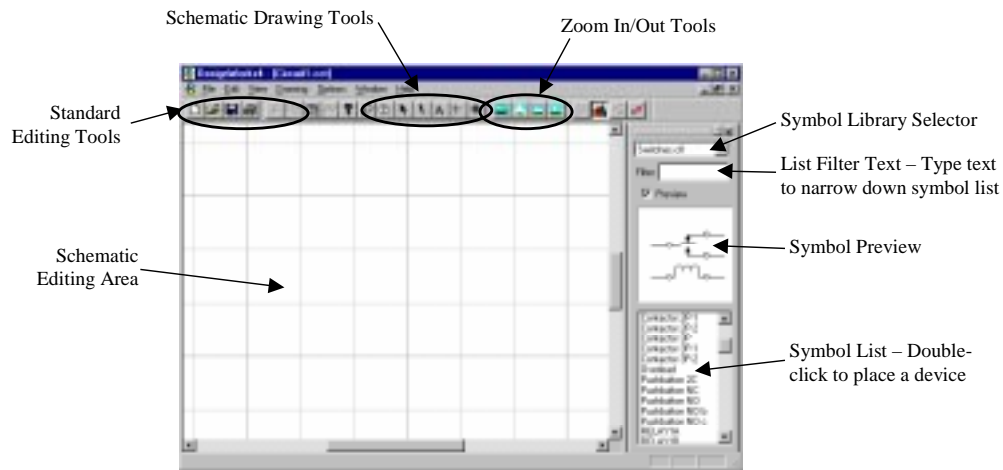


To allow you to get a quick overview and then delve into specific topics of interest, this manual is divided into the following sections:








- The 5-Minute Schematic
- Advanced Schematic Editing
- Creating Device Symbols and Hierarchical Design
- Using DesignWorks with SPICE-based Simulators
- Other Schematic Features

Note: All sections build on the information presented in the 5-Minute schematic tutorial. It is recommended that this section be reviewed first.

The DesignWorks Screen



Schematic Drawing Tools

Symbol	Tool	Description
	Pointer	Used to select or drag objects and extend signals.
	Text	Used to name a signal or device to name, or to place text notations on the schematic.
	Zap	Used to remove single objects. Press the button to remove whatever the tip of the cursor is pointing at.
	Draw Bus	Used to create a new bus line or extend an existing bus.
	Draw Sig	Used while creating a new signal line or extending an existing signal.
	Magnify	Used to zoom in and out. Clicking on a point or dragging down and right zooms in, dragging up and left zooms out.
	Attribute Probe	When the "dot" portion of the cursor is clicked on a device, pin or signal, the contents of the attribute fields for that object are displayed.

Manual Format

In all of the following sections, action instructions are shown

- **bold face like this**,
whereas explanatory text is shown in normal typeface.

Mouse Instructions

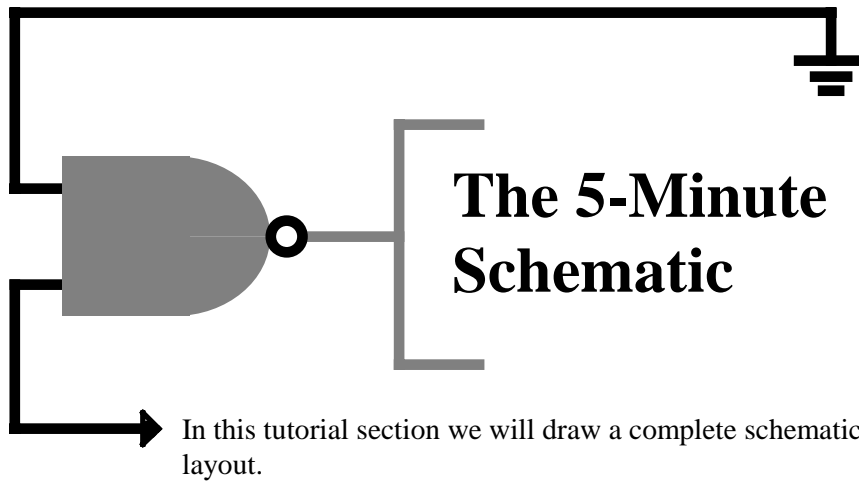
In this manual, the following terms will be used for mouse operations:

Click	Press and release the left mouse button with the mouse held in the same position.
Double-click	Press and release the left mouse button twice in quick succession.
Click and Hold	Press the left button and hold it pressed until a subsequent instruction tells you to release it.
Click and Drag	Press the left mouse button and hold it pressed while moving it to a new location. Release the button when the cursor is positioned at the desired final location.
Right-click	Press and release the right mouse button to display a pop-up menu. A menu item may then be selected with the left mouse button.

Keyboard Usage

The following special keys on the keyboard are commonly used in DesignWorks editing operations:

Arrow (cursor) Keys	The arrow keys have two uses: <ul style="list-style-type: none">• They set the orientation of device symbols or clipboard scraps while in placement mode.• They can be used to “nudge” selected items.
Spacebar	As long as no text entry is active, the spacebar resets the cursor to the normal pointer mode.



Starting the Program



- If it is not already running, double-click on the **DesignWorks Professional 4** icon, or select the **DesignWorks Professional 4** item in the Start menu to start the program.

After loading the program and opening libraries, you will see the “Welcome to DesignWorks” dialog box.

Creating a New Design



This box allows you to select a design template suitable for your application. Each selection enables the group of options most commonly used for the application type. All options can be changed later, if desired.

- Select the **"Generic PCB"** template.
- Click on the **Create** button.

We have selected a mode which enables PCB auto-packaging functions and assumes a flat (i.e. non-hierarchical) design.

You will now see an empty schematic drawing window ready for editing.

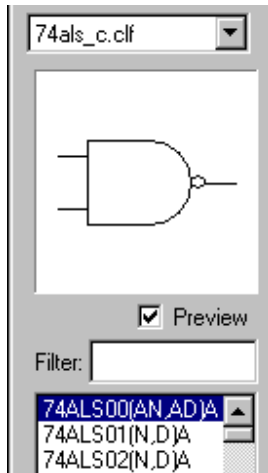


Choosing a Library

- Move the cursor to the library selection drop-down list and choose the "74als_c.clf" library.

The contents of this library will now be displayed in the part selection list.

Note: If you are using the demonstration version of DesignWorks, the exact parts listing may vary from what is shown here.

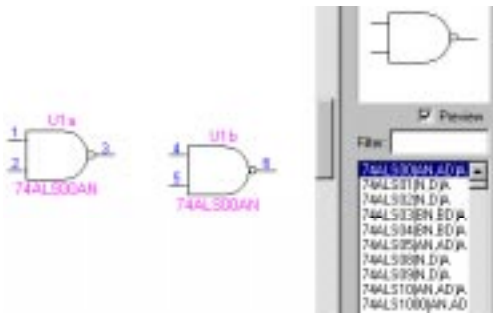


Selecting a Part

- Move the cursor to the part selection list and click once on the item "74ALS00(AN,DN)A" to select it. You will see a preview image appear in the the box above the list.
- Double-click on the same item to place it in the drawing. Move the cursor into the schematic drawing area.

An image of the selected part will follow the cursor movement. This part does not become a permanent part of the schematic until you click the mouse button.

Placing a Device

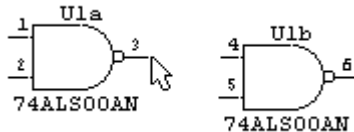


- **Move the cursor to an area just left of the parts palette and click on the schematic area to place a device.**
- **Move and click again to place a second device as shown. Order of placement is not important.**

You can continue to place devices of the same type just by clicking on the desired locations. Notice that each device is automatically labeled with its part type and package assignment. This automatic assignment can be disabled if desired.

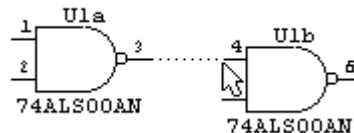
- **Return to the normal pointer by pressing the spacebar.**

Wiring Pins



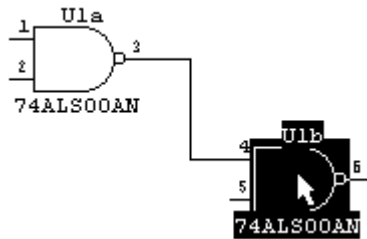
- **Move the pointer cursor exactly to the end of the output pin on the left-hand device, as shown, then click and hold the mouse button.**

Wiring Pins (cont'd)



- **With the mouse button still held down, move the cursor to the right, away from the output pin, so that it is positioned exactly over the end of the upper input pin of the right-hand device. Release the mouse button.**

You will see the signal line flash briefly indicating that a connection has been made.



Moving Devices

- Click and drag on the right-hand device and move it to a new position, as shown.

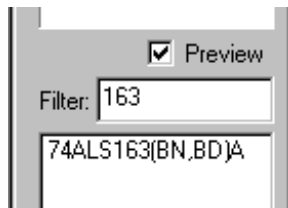
Notice that the connected signal line stays attached and stretches to follow the device movements.



Using Undo/Redo

- Select the Undo command from the Edit menu.
- Select the Undo command again, and repeat until all items you have placed have disappeared.
- Now select the Redo command and repeat it until all edits are redone.

Most schematic editing operations can be undone and redone up to 10 levels. Major structural changes (like adding a page or hierarchy level) or any operation involving a dialog box cannot be undone.



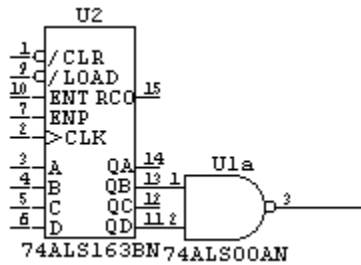
Part Selection by Name

- Click on the part selector text box in the Parts Palette.

You will notice a text cursor starts to flash in this box.

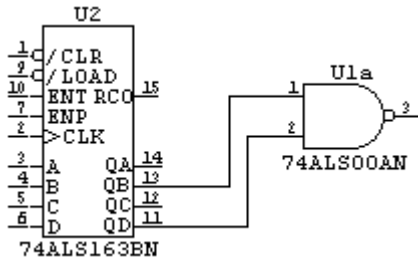
- Enter the value 163.
- Double-click on the part "74ALS163(BN,BD)A" in the list.

Automatic Pin Connection



- Move the cursor so that you can place the 74ALS163 device exactly as shown so that the QB and QD pins just touch the two inputs on the NAND gate.
- Click the mouse button to place the device at this point. (Depending on the size of your screen, you may need to use the scroll bar at the bottom of the schematic window to make room on the left.) You will notice the two pins flash to indicate a connection.
- Press the spacebar to return to the normal pointer.

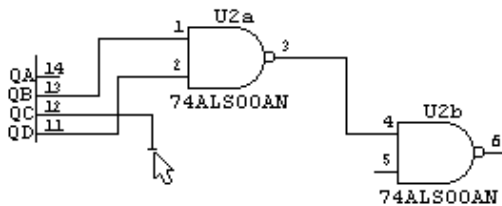
Automatic Pin Connection (cont'd)




- Click and drag the 74ALS163 device to the left and notice that right-angle lines maintain the connections between the two devices.

This auto-connection feature can be a convenient way of making connections between large devices, such as 8-bit registers. Just place them so the two sets of pins touch, then drag them apart and all the connections are made automatically.

Wire Editing

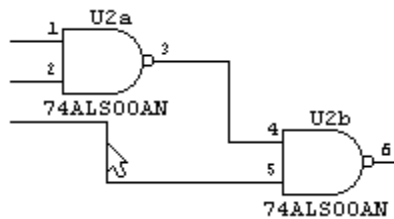


- Using the pointer () tool, click and hold on the end of output pin QC on the 74ALS163.
- Move to the location shown and release the mouse button. (You may have to modify this procedure slightly depending on the exact positions of your symbols.)


Notice that a small perpendicular mark is placed at the end of the signal line. All unconnected line ends are marked this way automatically to simplify checking for missed connections.



- Click and hold on the end of the line just completed and connect it to the lower pin on the NAND gate.

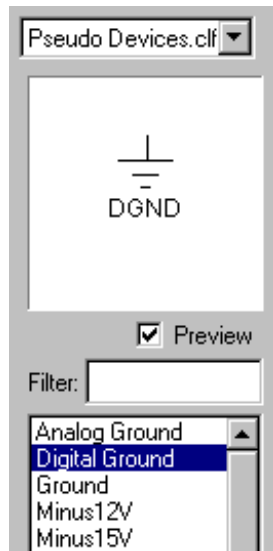
Wire Editing (cont'd)



Note: If you're using a small screen, you can hide the Parts palette at any time by double-clicking in the top left corner of the window. To re-display it, select Parts Palette in the Window menu.

- Using the pointer () tool, click and hold at a point midway along the vertical line (or any line) just created.

Notice that you can drag this line segment sideways. With the pointer () tool, clicking at the end of a pin or line segment or at an intersection allows you to extend the signal. Clicking in the middle of a segment allows you to move that segment. The signal drawing tool () can be used to draw from any point.

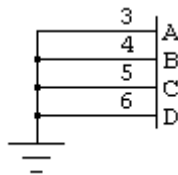


Power and Ground Connections

- If necessary, use the scroll bar at the bottom of the schematic window to expose some drawing area to the left of the 74ALS163 device.
- Go to the library drop-down list and select the "Pseudo Devices" library.

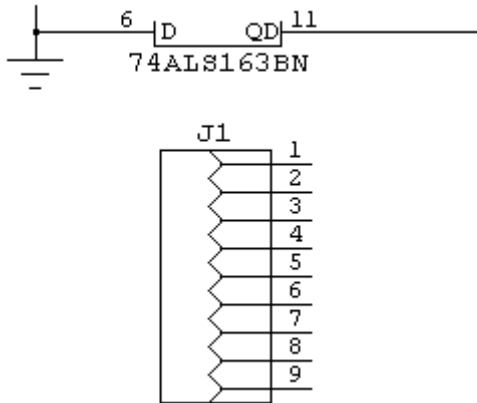
The term "pseudo device" is used in DesignWorks to refer to symbols that are edited like devices on the schematic, but are actually symbols used to modify signal connections. Examples of pseudo-devices are power and ground symbols, page connectors and bus breakouts. These items will be discussed in more detail later.

Power and Ground Connections (cont'd)



- Select the Ground item in the parts palette, then place one as shown.
- Click on the signal tool (+) in the tool palette and wire it to the 74ALS163 device as shown.

The Ground symbol automatically names the attached net "Ground" and causes it to be logically connected to all other ground nets in the circuit.



Connector Devices


- Select the "Connectors" library and place a DB9F connector symbol below the 74ALS163, as shown. (Depending on the size of your screen, you may need to scroll the schematic window downwards.)

Connectors can be treated as a single unit, as in this case, or broken up into multiple symbols each with 1 or more connector pins on them. In the netlist, these will be treated as a single device.

Connecting Signals by Name

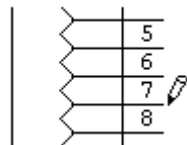
- Click on the text (A) tool in the tool palette.



This tool is used to name devices and signals, edit device pin numbers, edit attribute text or create miscellaneous text notations, depending on where it is clicked. The cursor will initially take on a pencil shape () allowing you to point accurately at the item to be named.

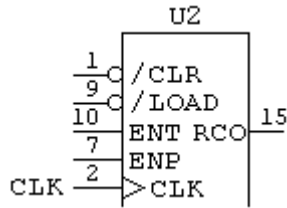
Connecting Signals by Name (cont'd)

- Click on the end of connector pin 7 using the pencil tool.



When you release the button a default name is displayed. Unless auto-naming is disabled, every signal is assigned a unique name as it is created. This name is normally only displayed on the schematic when explicitly requested.

- Enter the name "CLK" and press the Enter key to terminate text entry.

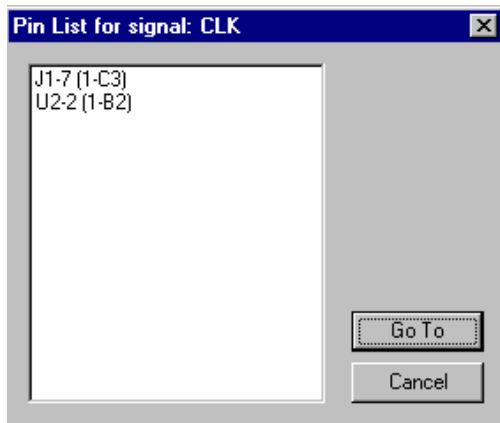


Connecting Signals by Name (cont'd)

- Move to the CLK input on the 74ALS163 device and repeat the same procedure, naming this pin "CLK" as well.

Notice that when you press the Enter key this time, the signal flashes to indicate a connection has been made with the other CLK label.

Confirming Connections



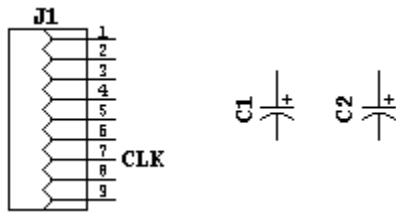
- Press the space bar to return to Point mode.
- Click on the name CLK that was just typed so that the name itself and the device pin is highlighted.
- Select the Get Info command from the Options menu. Click on the Pin List button.

(A shortcut for this procedure is to right-click directly on the signal and select the Pin List command.) The Pin List dialog shows you all device pins that are linked by this signal, along with their page number and grid reference.

- Double-click on any item in this list to display it on the screen.

The Pin List dialog will locate items on any page in the circuit, whether connected by signal line, by name, by off-page connector or by bus.

Discrete Components



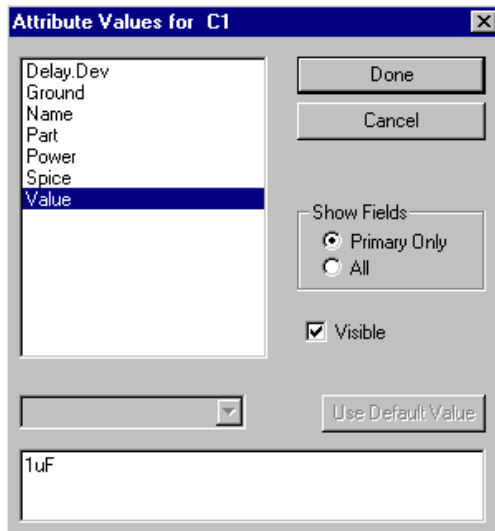
- Using the library selection pop-up menu, select the "Discretes" library. Double-click on the item CAP POL a (polarized capacitor) and move the cursor into the schematic area.
- Press the arrow keys (or, if you don't have arrow keys, use the Orientation command in the Options menu and select a new orientation) to orient the symbol vertically.
- Place two capacitors as shown.

Devices can be rotated to one of 8 orientations (the 4 compass points plus mirrored versions of each). Device text notations can optionally be rotated to match the device.



Setting Component Value

- Click on one of the capacitors just placed, so that it is highlighted.
- Choose the Get Info command from the Options menu, then click on the Attributes button.



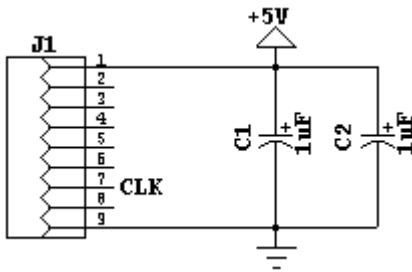
Setting Component Value (cont'd)

This dialog allows you to view and edit text "attributes" of a device. The list at the left shows the available field names. Clicking on one of these will display the associated value in the text box.

- Click on the item **Value** in the field list at left. Enter the value **"1uF"**.
- Make sure that the **Visible** box is checked, then click on the **Done** button then the **OK** button.
- Repeat this procedure to assign the same value to the other capacitor.

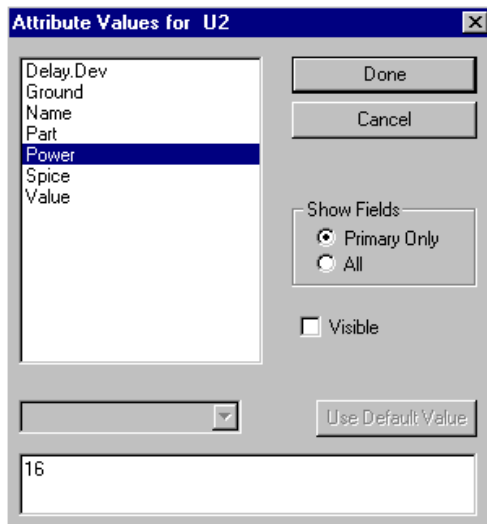
The component value just entered will now appear adjacent to the device. It can be moved around independently, if desired. We will see in a later tutorial how to edit, rotate, hide and set text style for this text.

More on Power and Ground



- Go back to the "pseudo.clf" library. Place a **Ground** and **Plus5V** symbols.
- Wire them to the capacitors and connector, as shown. (Note: You may have to use the arrow keys again to return to normal orientation.)

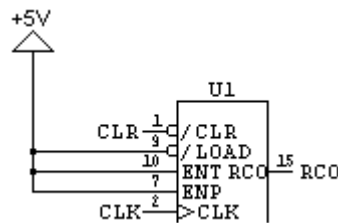
The Ground and Plus5V symbols are a special class of pseudo-device known in DesignWorks as a "signal connector". They cause all like-named nets to be connected together, even across multiple pages. You can customize your own signal connectors for other types of common connections.



More on Power and Ground (cont'd)

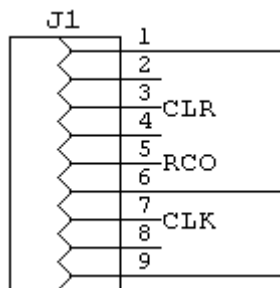
- Click on the 74ALS163 device so that it is highlighted.
- Select the Get Info command from the Options menu, then click on the Device Attributes button.
- Select the field Power in the attribute field list and note the field value.
- Select the field "Ground".

You will notice that the fields named "Power" and "Ground" contain the numbers of the power supply pins for this device. You can add other pins to this list, if needed, separated by commas. This allows you to create power and ground connections without showing them explicitly on the schematic. The standard power and ground connections are included in all integrated circuit parts in the DesignWorks libraries.



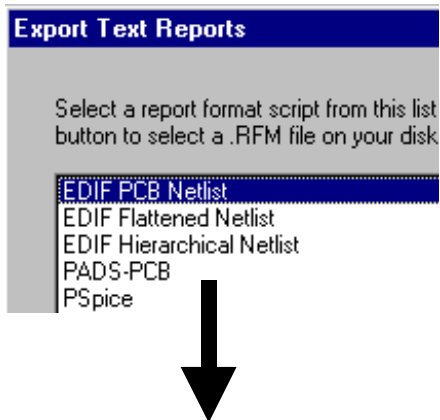
More on Power and Ground (cont'd)

- To complete the counter wiring:
- Add a Plus5V symbol and wire it as shown.
- Apply names to the CLR and RCO counter pins, as shown.



More on Power and Ground (cont'd)

- To complete the connector wiring:
- Using the text (T) tool, apply names CLR and RCO to the connector pins shown.
- Using the (+) tool, draw a wire from the unconnected NAND gate output to connector pin 6.



```
(EDIF
(rename D_058_092_PROGRAM_032_FI
(EDIFVersion 2 0 0)
(EDIFLevel 0)
(keywordMap
(keywordLevel 0))
(status
(written
(timestamp 2000 3 8 14 26
```

Generating a Netlist

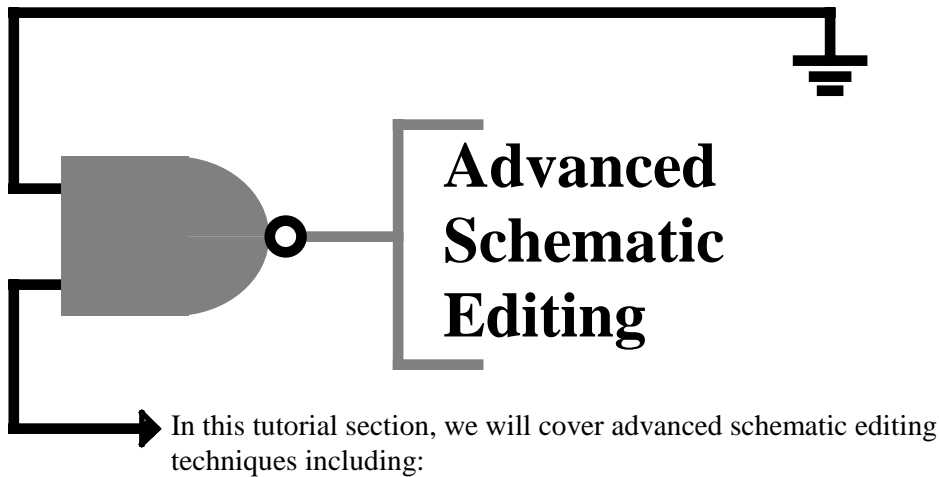
We have now completed our simple schematic and the last step is to generate a netlist for PCB layout purposes.

- **Select the Export command from the File menu.**
- **Click on the Text Formats button.**
- **Click Next.**
- **Select the EDIF PCB Netlist item in the list of available formats.**
- **Click the Finish button.**

You have now saved a text report file that contains a connection list (netlist) for your circuit in the standard EDIF format. Appendix B at the back of this manual describes more features of the report generation tool.

This completes the tutorial section “The 5-Minute Schematic”.

- **You may wish to use the Save As command from the File menu to save the completed example at this point**



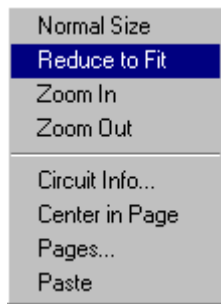
- More Editing Techniques
- Pop-Up Menus
- Busses and Breakouts
- Multi-Page Schematics
- Sheet Size Settings
- Text Notations and Graphics
- Creating Attribute Fields
- Device Packaging

Starting the Program



- If it is not already running, double-click on the DesignWorks icon to start the program.
 - Use the Open Design command to open the demonstration file “5 Minute Demo.cct” provided in the Demo directory OR open the version you created using the instructions given earlier in this manual.
-

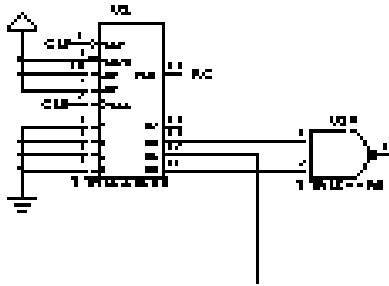
Navigating Around the Schematic




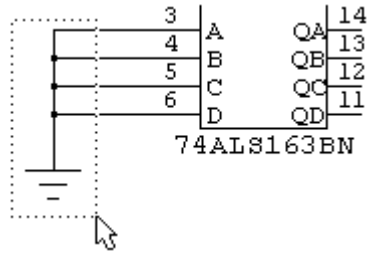
- Press the right mouse button (Right-click) in an empty area of the schematic (i.e. not on a device or signal line).
- In the pop-up menu that appears, select the command **Reduce to Fit**.

The screen display will be zoomed out to fit the entire schematic in the window. Right-clicking in the schematic displays a pop-up menu containing short-cut editing commands. Right-clicking on a device, signal, pin or attribute field will display a special menu for each type of object.


Navigating Around the Schematic (cont'd)

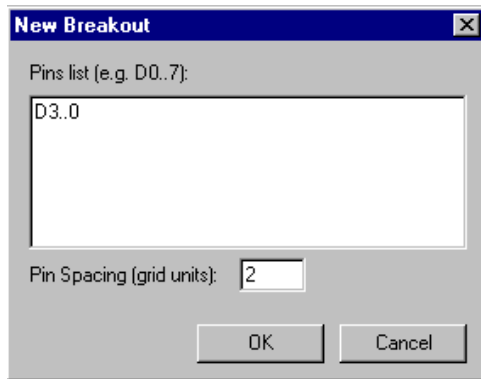


- Click on the magnify () tool in the tool palette and then click near the A B C D input pins of the 163 counter.
- Clicking the magnifying glass tool zooms in on that part of the schematic. Some less obvious uses of this tool are:
- Clicking and dragging down and right causes the display to zoom so that the area swept over just fits in the window.
 - Clicking and dragging up and to the left a small amount causes the display to zoom out one step.
 - Clicking and dragging up and to the left a large amount (more than 1/2 the screen) causes the display to do a Reduce to Fit.



Deleting a Group of Objects

- Using the pointer () tool, click and drag from above and to the left of the ground symbol to below and right of it.
- Make sure that the ground symbol itself and the attached signal line are highlighted and no other items.
- Press the Delete key to remove these items.

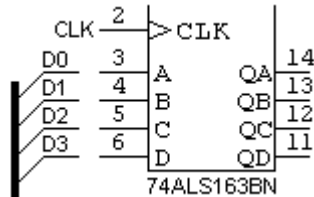


Creating a Bus

- Select the New Breakout command from the Options menu. In the signal list area of this dialog, enter "D3..0".
- Click on the OK button.

We used sequentially numbered signals in this case, but any collection of names can be used, for example "CLK CLR SIZ0 SIZ1". "0..3" places the highest numbered signal at the top, "3..0" places it at the bottom.

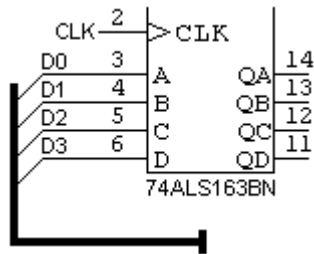
Creating a Bus (cont'd)




- Place the "breakout" symbol adjacent to the 163 device, as shown. Use the arrow keys or tool palette if necessary to orient the breakout symbol.
- If not done already, connect the 4 breakout pins to the inputs of the 163 device.

A "breakout" is a pseudo-device symbol that ties any collection of signal lines into a single bus line. You can extend a bus line away from either end of the "spine" of the breakout symbol.

Creating a Bus (cont'd)

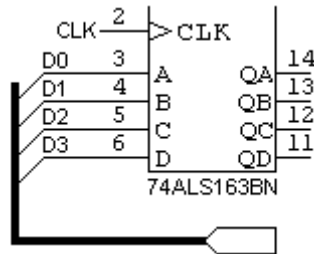


- Using the pointer () tool, click and hold exactly at the lower end of the "backbone" of the breakout symbol.
- Drag down and right from this point, then release the mouse button. This will have created a bus line as shown.
- Right-click on the bus line. In the pop-up menu, select the Signal Info command.

When the Signal Info command is selected for a bus, a list of the internal signals is displayed.

- Click on the OK button.

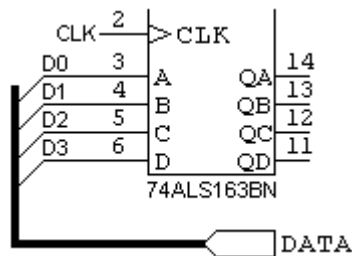
Creating a Bus (cont'd)



- In the Parts palette, select a "Page Conn (Bus)" device in the "pseudo.clf" library.
- Place this symbol as shown so that it connects to the bus that was created above.

The Page Connector symbol indicates that this bus will be connected to other busses with the same name on other pages of the schematic. Without this symbol, connections by name occur only within a single page.

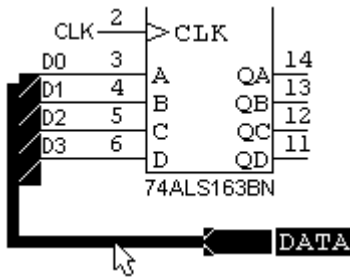
Creating a Bus (cont'd)




- Select the text (**T**) tool in the tool palette.
- Click and hold on the Page Connector symbol.
- Still holding the mouse button, move to a point just right of the symbol, then release the button.
- Enter the name "DATA" then press the Enter key.

We have now named the bus "DATA". This has no effect on the names of the internal signals, which still retain the names that were assigned in the New Breakout dialog.

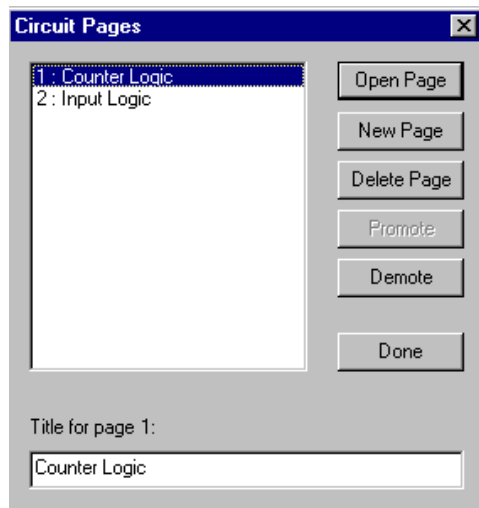
Using the Clipboard



- Return to the pointer () tool and click in an unused area of the schematic to ensure that no items are selected.
- Click on the breakout symbol (i.e. on the diagonal line area in the middle of the symbol), to select it.
- While holding the Shift key, click on the bus line and then on the Page Connector symbol. All three of these items should now be highlighted.

Using the Shift key, you can select any collection of items on one page. These items can then be operated on in subsequent menu commands.

- Select the Copy command from the Edit menu.



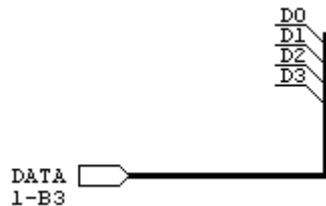
Adding a Page

- Select the Pages command from the Drawing menu.
- Click on the New Page button.
- Enter the title "Input Logic".
- Click on the item "1" in the page list and enter the title "Counter Logic".
- Click on the Done button.

We have now added a second page to the circuit, which can be edited in its own window. The second page is still logically part of the same circuit and will be stored in the same file. A single circuit can have up to 1000 pages.

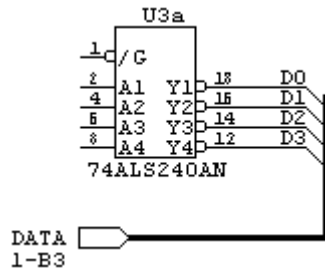
Adding a Page (cont'd)

- Select the Paste command from the Edit menu. A flickering image of the bus lines copied from the first page will now appear.
- Using the arrow keys, orient this entire group so that the Page Connector is on the left and the breakout is on the right.
- Click in the middle of the page to place the circuit segment.



You will notice that a new notation appears next to the Page Connector symbol. This is an automatic page reference which indicates the page and grid position of other page connectors attached by name to this one.

Adding a Page (cont'd)

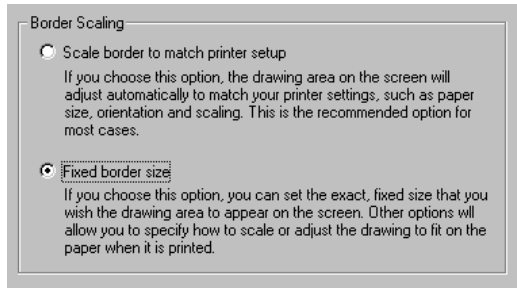


- Select a 74ALS240 device from the "74als_c.clf" library and place it so the its outputs connect with the breakout.
- Drag this new device away from the breakout to expose some signal lines between the two.
- Right-click on one of these lines.
- In the pop-up menu, select the Pin List command.

The Pin List dialog will now indicate that the pin on the 240 device is connected via the bus and page connector to the 163 device on Page 1.

- Double-click on a pin on U2 to return to the first page.

Setting Sheet Size

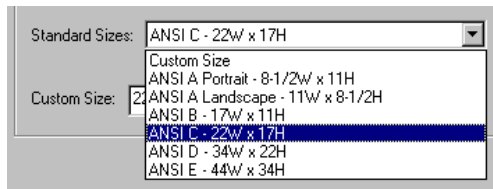


- Select the Sheet Size Wizard command from the Drawing menu.
- Click on the “Adjust all pages in the current circuit” box, if it isn’t checked already.
- Click on the Next button.
- Click on the Fixed border size button, if it isn’t selected already.
- Click on the Next button.

There are two general methods for setting sheet sizes in DesignWorks. The “use printer setup” mode automatically adjusts the size of the sheet you see on the screen to match your current printer settings. This is generally the most convenient in that it guarantees that the border you see on the screen will fit exactly on the paper.

The “fixed sheet size” mode is preferable if you will be switching to a variety of printers and want the border size to stay fixed at a specified size.

Setting Sheet Size (cont’d)

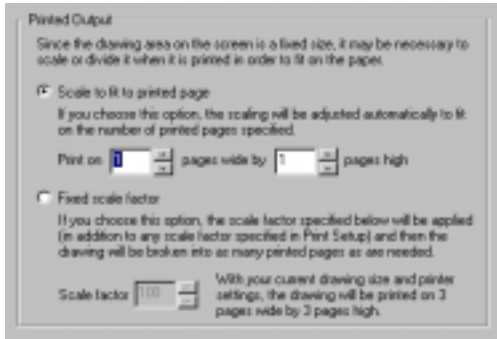


- Select the standard sheet size ANSI C – 22W x 17H from the popup list of standard sizes.

Note that the list of standard sheet sizes can be customized using settings in the initialization file, so the list of settings you see may not match the ones shown here. If the one mentioned above is not in the list, pick any one that suits you at this point.

- Click on the Next button.

Setting Sheet Size (cont'd)



- Select the “scale to fit printed page” option, if it is not already.
- Set the width and height settings to 1 and 1.

These settings will produce a large printed area on the screen and cause the program to scale the page automatically when printed to provide the best fit on the paper available.

- Click on the **Finish** button to apply the new settings.
- Select the **Reduct to Fit** command in the **Drawing** menu to observe the new sheet size settings.

Adding Text Notations

All capacitors 10% tol
unless otherwise noted.

- Move to an unused corner of the schematic page.
- Select the text (A) tool in the tool palette.
- Click in an open area of the schematic (i.e. not on an existing device, signal or text item). You will see a text entry rectangle.
- Enter "All capacitors 10% tol unless otherwise noted", or any other notation that suits you. Carriage returns can be used in text notations, if desired.
- Click outside the text entry rectangle.

Text notations created in this fashion are stored with the circuit, but are not associated with any device or signal and will never appear in a netlist.

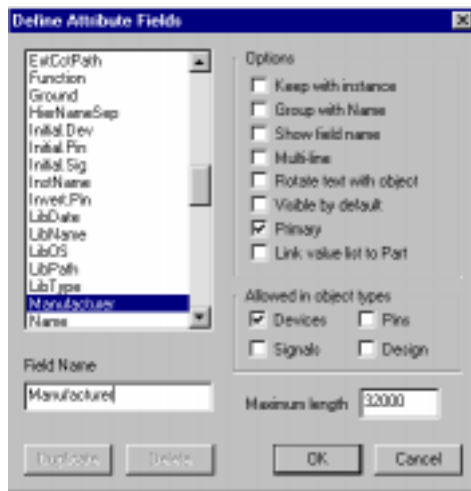
Adding Text Notations (cont'd)



- Right-click on the text item that was just created.
- Select the Draw Frame option.
- Click on the Font Specs button.
- Use the controls in this dialog to set any desired text style.
- Click on the OK button in both dialogs to return to the schematic.

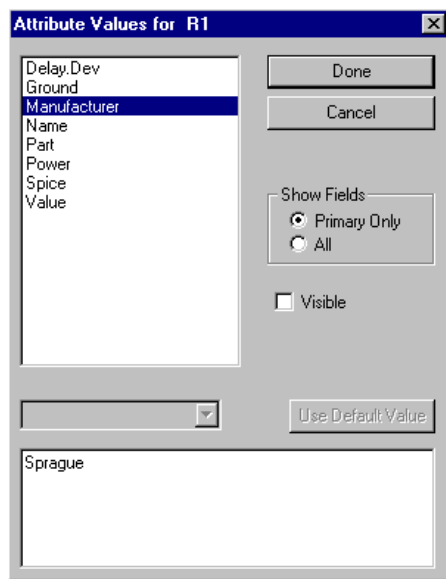
Text style can be set individually for random text notations such as this one. Other forms of text such as attribute and pin numbers have global text style settings, but cannot be set individually. See the Design Preferences command from the Drawing menu.

Creating Attribute Fields



- Select the Define Attribute Fields command from the Options menu.
- Enter the name "Manufacturer" in the Field Name box.
- Click on the Devices box under "Allowed in object types".
- Click on the OK button.

You have now defined a new attribute field that will be associated with devices. A separate value can be entered for each device in your design. The other options in this dialog determine how the value is stored and displayed on the schematic.



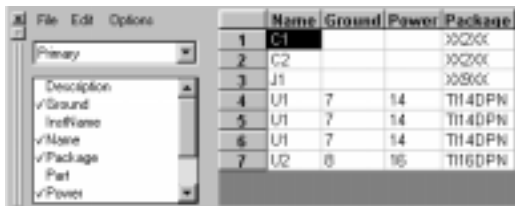
Creating Attribute Fields (cont'd)

- **Right-click on any device in the schematic.**
- **In the pop-up menu, select the Attributes command.**
- **Enter a value for the Manufacturer field in the text box.**
- **Click on the Done button.**
- **Repeat this for each device.**

Attribute values can be entered for devices, signals, pins and for the design itself. These values can be displayed on the schematic and extracted in netlists and bills of materials for external use.

Using the Browser Tool

- **Click in an unused area of the schematic to ensure that no devices or signals are selected.**
- **Select Browser from the View menu.**

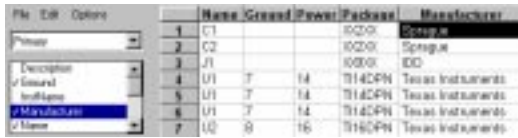


The Browser tool provides a spreadsheet view of the schematic. The Browser displays the selected type of object in a spreadsheet format. This allows you to easily locate items and edit attributes without having to search for them on the schematic sheet.

- **Click on any device name in the list and note that the corresponding item in the schematic is selected.**

The Browser can also view Signals or Pins (see the Options menu in the Browser panel).

Using the Browser Tool (cont'd)



The screenshot shows a software window titled 'Browser' with a menu bar (File, Edit, Options) and a list of components. The components are organized in a table with columns: Name, Symbol, Power, Package, and Manufacturer. The 'Manufacturer' column is highlighted, and the 'Texas Instruments' entry is selected.

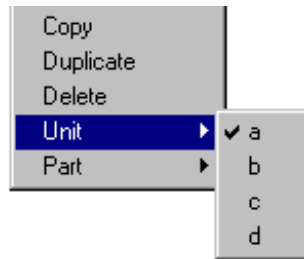
	Name	Symbol	Power	Package	Manufacturer
1	C1			8000	8000
2	C2			8000	8000
3	J1			8000	8000
4	U1	7	14	7814C4N	Texas Instruments
5	U1	7	14	7814C4N	Texas Instruments
6	U1	7	14	7814C4N	Texas Instruments
7	U2	9	16	7814C4N	Texas Instruments

- Click on the attribute type control and select the Primary attributes.
- In the field list box, double-click on the Manufacturer field, or any other desired field, to add it to the Browser window.
- Double-click on any box in the "Manufacturer" column and entered any desired values.
- Use the Enter key to move down to other cells in the same column.

Changes made to data in the Browser cause the Schematic to be updated immediately. These changes cannot be canceled or undone!

- Close the Browser by clicking in the close box (X) at the top right corner.

Device Packaging



- Right-click on one of the NAND gate devices on page 1.
- Move down to the Unit sub-menu, near the bottom.
- Select Unit "c" in the Unit sub-menu.

The Unit sub-menu contains a list of all gate units available for this part type. Selecting a different unit will update the pin numbers to correspond.

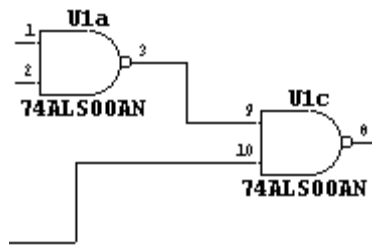
Device Packaging (cont'd)



- Right-click on the same device as in the previous section and select the **Device Info** command.
- Click on the **Packaging Options** button.
- Select the **"Lock and check package and unit"** option.
- Click on the **OK** button in both dialogs.

The Packager has now been instructed to keep this device name unchanged but check it for conflicts in future packaging operations.

Device Packaging (cont'd)

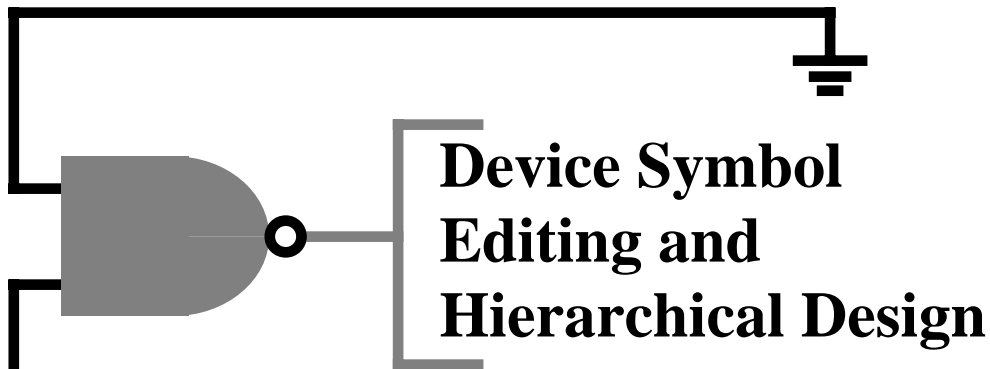


- Select the **Device Naming and Packaging Options** command from the **Packaging** sub-menu of the **Options** menu.
- Enter the prefix **"XX"** and click on the **OK** button.
- Select the **Repackage Design** command from the **Packaging** sub-menu of the **Options** menu.
- Click on the **OK** button in the confirmation dialog.

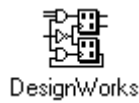
Notice that new names have now been assigned to all devices except the one that we marked as "Lock and Check". Names are assigned starting in the "A1" grid of the schematic, working toward the opposite corner.

This completes the tutorial section "Advanced Schematic Editing".

- The remaining tutorials deal with specific areas of the program and do not make further reference to this file.



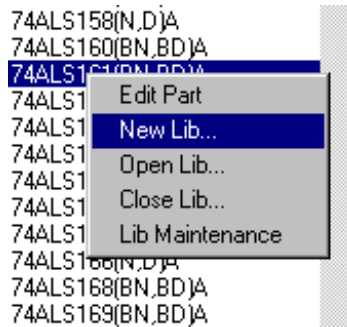
In this tutorial section, we will cover some of the DesignWorks features for creating and editing device symbols as well as associating internal circuits with them.



Starting the Program

- If it is not already running, double-click on the DesignWorks icon to start the program.

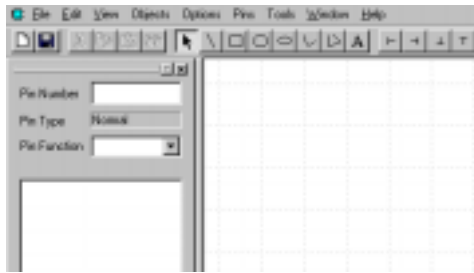
Creating a New Library



- Right-click on the Parts palette's parts list and select the New Lib command from the pop-up menu.
- Create a new library called "MyLib.cdf" in the Libs directory.

Device library files hold collections of part symbols along with associated pin function information, default attribute values and external circuit references. A single library can contain from one to thousands of part definitions, to suit your needs.

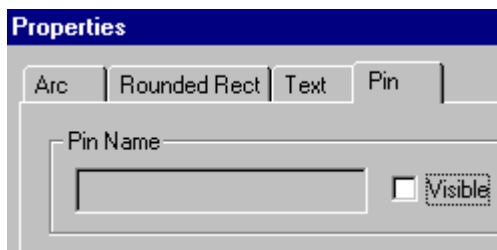
Creating a Device Symbol



- Select New command in the File menu.
- Select Device Symbol option in the document type list and click OK.

The DevEditor window contains a drawing area for your symbol, a tool palette and a pin list. The tool palette includes standard drawing tools plus special items for normal, inverted and bus pin placement.

Setting Pin Name Visibility

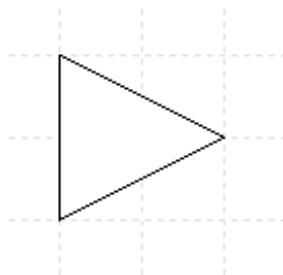



By default, the symbol editor displays the name of each pin next to the pin stub. For this example, we want to turn this off. To do this:

- Make sure no graphic items are selected in the drawing area, then select the Properties command in the Options menu.
- Click on the Pin tab and turn OFF the Visible switch in the Pin Name area.
- Click OK

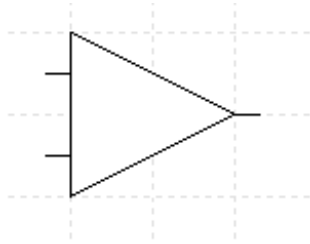
Setting items in the Properties command with no objects selected sets the default properties when future objects are created.

Creating a Device Symbol (cont'd)



- Click on the polygon () tool in the tool palette.
- Draw a symbol similar to the one shown (Note: double click on the last point to terminate polygon drawing).

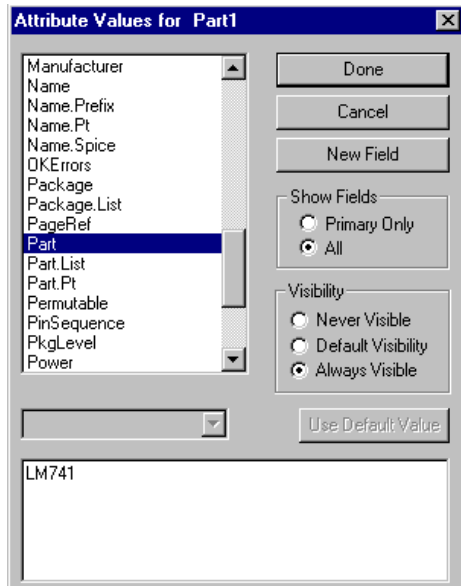
Creating a Device Symbol (cont'd)



- Select the (⇐) pin tool.
- Place input pins on the symbol by clicking at the positions shown.
- Select the (⇒) pin tool.
- Place an output pin as shown.

Note: The crossbar portion of the T pin tool only appears during placement and dragging for alignment purposes.

Setting Default Part Attributes



- Return to the pointer (☞) tool.
- Select the Part Attributes command from the Options menu. (Note: there will be no attributes in the list if there is no active design).
- Select the Part field in the list.
- Enter the value "LM741".
- Select the Package field in the list.
- Enter the value "NAT8DPN" or other package code.

These attribute values will appear as the defaults when this part is used on a schematic. If desired, these values can be overridden for each individual device.

Setting Default Part Attributes (cont'd)



- Click on the New Field button in the attributes dialog.
- Enter the field name "Plus12V" then press the Tab key to terminate this value.
- Select the "New Field" item at the top of the list again.
- Enter the field name "Minus12V".
- Click on the OK button.

If you create multiple devices in succession, you only need to perform the "New Field" operation once since the DevEditor keeps the same field list for all devices.

Setting Default Part Attributes (cont'd)

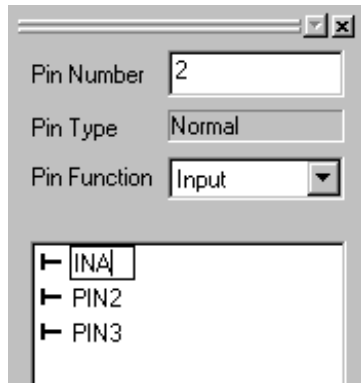


- Enter the value "4" for the new Minus12V field.
- Enter the value "7" for the "Plus12V" field.

These entries will determine the default power connections for this device in a netlist.

- Click on the Done button to close the attributes dialog.

Entering Pin Names and Numbers



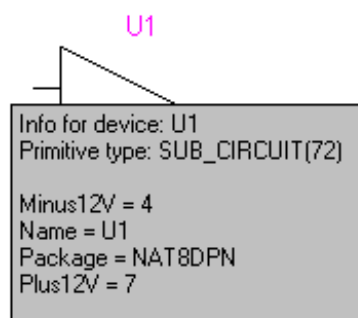
- Double-click on the PIN1 item in the pin list.
- Enter the pin name "INA" then press the Enter key.
- Enter the pin number 2 in the Pin Number box.
- Double-click on PIN2 and enter the pin name "INB" and the pin number 3.
- Similarly, change "PIN3" to "OUT" with pin number 6 then select Output from the Pin Function pulldown box.

We have now entered default values for the pin numbers that will appear in a netlist. These can be edited on the schematic for individual pins, if desired.

Saving and Using the Part



- Select the Save Part As command from the File menu.
- Enter the part name "LM741" or any other desired name.
- Click on the "MyLib.clf" library in the list to select a destination and press "Save".
- Close the symbol editor window.



Saving and Using the Part (cont'd)

- If it is not already selected, select the "MyLib.clf" library in the drop-down library list in the Parts palette.
- Double-click on the newly-created part and place one in the schematic.

Note: A warning dialog will be displayed indicating that new attribute fields are being defined.

- Select the Attribute Probe (?) tool in the schematic tool palette.
- Click on the new device to verify that the default attribute values appear.

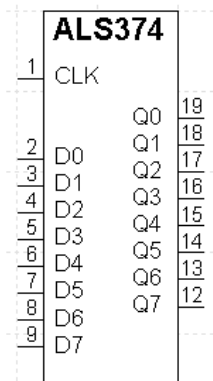
Auto-Creating a Symbol

For standard types of rectangular symbols, the Auto Create feature will generate a symbol for you in seconds.



- Select New in the File menu.
- Select the Device Symbol option and click OK.
- Select the Auto Create Symbol command from the Options menu.
- In the Part Name box, enter "ALS374", or any other desired symbol name.
- In the "Left Pins" box, enter the text "D7..0(9..2),,CLK(1)".

"D7..0" will generate a set of 8 pins named D7, D6, etc. "(9..2)" are the corresponding pin numbers. The three commas indicate that we want extra space between these pins. "CLK(1)" creates a single pin called CLK with pin number 1. The pin numbers can be omitted, if desired.
- In the "Right Pins" box, enter the text "Q7..0(12..19)".
- Click on the Generate Symbol button.

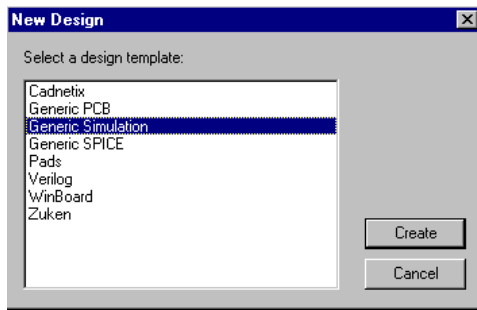


Auto-Creating a Symbol (cont'd)

The auto-generated symbol should now display the pins and pin numbers entered above. These items can be edited using the drawing tools and Pin Info dialog, if desired.

- Select the Save Part As item in the File menu and save the new part to the "MyLib.clf" library.
- Close the symbol editor window.

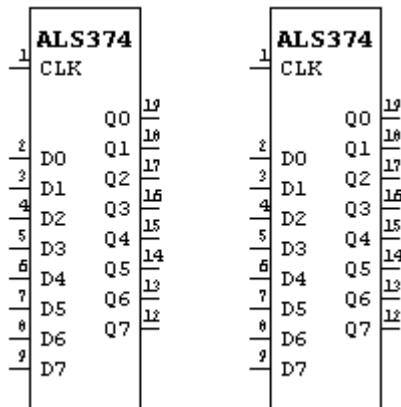
Creating a Hierarchical Block



- Select the New command from the File menu.
- Select the Design option and click OK.
- Select the Generic Simulation design template.

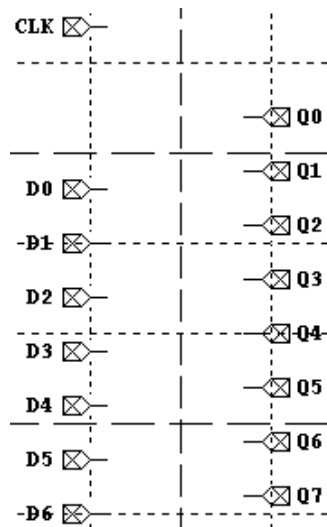
The Generic Simulation template is preconfigured to allow hierarchical design, which will allow us to create nested circuit blocks inside device symbols.

Creating a Hierarchical Block (cont'd)



- In the Parts palette, double-click on the ALS374 device that was just created.
- Place two of these devices on the schematic, as shown.

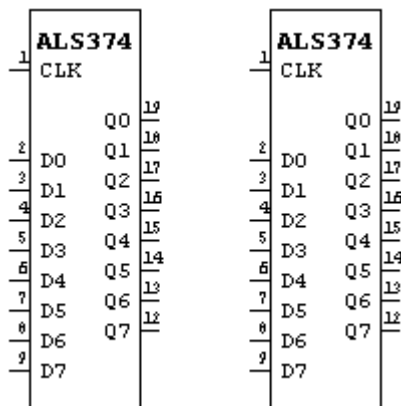
When new symbols are made in the symbol editor, they are set by default to be "Sub-Circuit" devices, meaning that an internal circuit can be created in them, even if none is provided initially.



Creating a Hierarchical Block (cont'd)

- Double-click on either of the ALS374 devices that were just placed.
- Click on the OK button in the warning dialog to create an internal circuit.

A new window will now appear showing the internal circuit for this block. When a new internal circuit is created, "port connector" symbols are automatically laid out to match the pins on the parent symbol. Any signal connection made to a port connector becomes logically connected to the pin with the same name on the parent symbol.



Creating a Hierarchical Block (cont'd)

- Close the internal circuit window.

This will cause the definition of all devices of the same type to be updated throughout the design. Double-clicking on any other device of the same type will now reveal the same internal circuit.

Note: Unless you are an experienced user of hierarchical design concepts, please refer the chapter "Before Starting a Major Design" in the DesignWorks Reference Manual before using these features.

This completes the tutorial section "Device Symbol Editing and Hierarchical Design".

- More information on hierarchical design features is found in Appendix A.



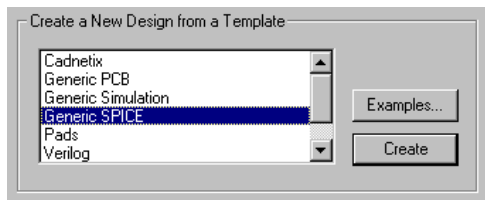
In this tutorial section, we will create a simple SPICE circuit and write out the complete netlist. You can then try out this netlist with your SPICE package.



Starting the Program

- If it is not already running, double-click on the DesignWorks icon to start the program.

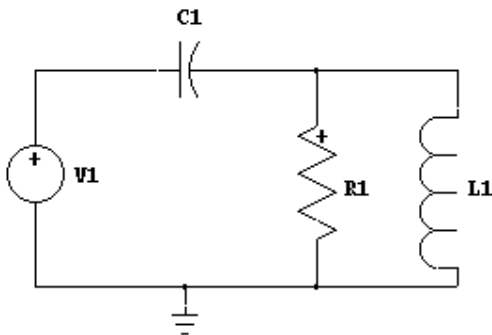
Creating a SPICE Design



- When the Welcome dialog appears, Select the "Generic SPICE" template.
- Click the Create button.

This template enables an option that auto-names devices using a SPICE prefix supplied in the library. In the SPICE library provided with DesignWorks, this field contains the standard prefixes for SPICE models.

Placing SPICE Devices



- Select the "spice.clf" library in the library selection list in the Parts palette.

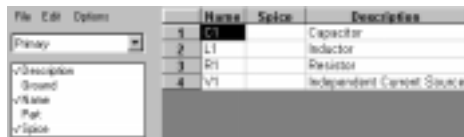
The SPICE library contains symbols that have the standard SPICE prefix and pin order predefined so that appropriate names are assigned and netlist pin order is correct.

You can use any symbol from any library in a SPICE design, but you will need to ensure that an appropriate name is assigned and that the pin order in the netlist matches the requirements of your particular SPICE-based simulator.

- Using the devices Ind Volt. Src, SPICE Ground, SPICE Res, SPICE Cap and SPICE Ind, create the circuit shown at left.

Some of the passive components in this library, such as the SPICE resistor, are marked to ensure consistent polarity for current-measuring purposes.

Entering SPICE parameters



- Click in an unused area of the schematic to ensure that no devices or signals are selected.
- Select Browser from the View menu.
- If the "Spice" attribute field is not already displayed in the spreadsheet area, click on the Attr Type control and select the Primary attributes, then, in the Attr List Box, double-click on the Spice field to add it.

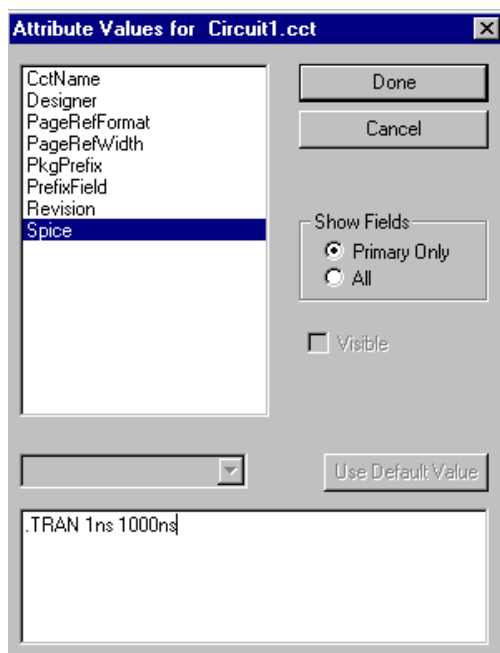
All text entered in the Spice attribute will be included in the netlist for with the associated device.

Entering SPICE parameters (cont'd)

	Name	Spice
1	C1	1000pF
2	L1	100nH
3	R1	1K
4	V1	PULSE(0V 10V 50NS 10NS 10NS

- **Double-click in the cell under Spice adjacent to the device C1 in the Name list.**
- **Enter the value “1000pF”**
- **Enter values for the remaining devices as follows:**
L1 100nH
R1 1K
V1 PULSE(0V 10V 50NS 10NS 10NS
 100NS 1US)
- **Press the Enter key to ensure that the last value to be sure the schematic is updated.**
- **Close the Browser by clicking in the close (X) control at the upper right corner of the window.**

Entering SPICE Parameters (cont'd)

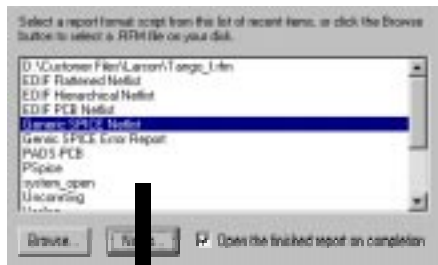


- **Select the command Design Attributes in the Options menu.**
- **Select the Spice field in the list.**
- **Enter the text ".TRAN 1ns 1000ns". If you are using PSpice, you may wish to press the Enter key and enter a ".PROBE" line to generate output for the Probe display program.**

Any text entered in the design's Spice field will appear at the beginning of the SPICE netlist. This can be used to enter SPICE commands, model definitions, etc. Multiple lines can be entered if desired using the Enter key. You can also use the Ctrl-V (paste) to paste larger numbers of lines into this box.

- **Click on the Done button to close the attributes dialog.**

Using DesignWorks with SPICE-based Simulators



Design2 Thursday, October 22, 1993 11:52 AM

.TRAN Ins 1000ns

C1 SIG1 SIG4 1000pF

L1 SIG4 0 100nH

R1 SIG4 0 1K

V1 SIG1 0 PULSE (0V 10V 50NS 10NS 10NS 100NS 1US)

.END

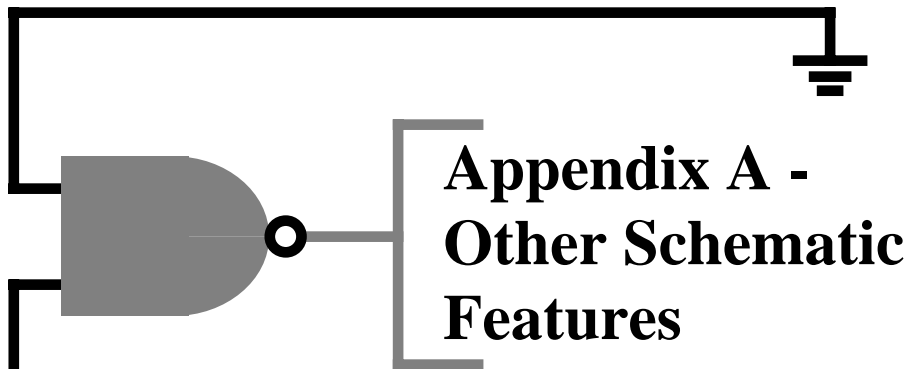
Creating a SPICE Netlist

- Select **Export** from the **File** menu.
- Select the **Text Formats** option and click **Next**.
- Select the script **Generic SPICE Netlist** in the format list.
- Click on the **Finish** button.
- Select an appropriate **output file name** and click on the **Save** button.

The netlist file just created can now be fed directly to your simulator.

This completes the tutorial section “Using DesignWorks with SPICE-based Simulators”.

- **DesignWorks can also directly create hierarchical SPICE netlists with .SUBCKT definitions. Hierarchical block symbols can be nested to any desired depth.**



DesignWorks has numerous additional features that are not directly mentioned in the tutorial sections. This section provides an overview of some of these features.

Libraries

DesignWorks™ is shipped with an extensive set of over 12,000 device symbols. Digital components are provided in Compact, ANSI (rectangular) and IEEE formats. Gate devices are provided in monolith (i.e. one package = one symbol) and with DeMorgan equivalents. All digital devices have full gate packaging information.

Family	Compact	ANSI	IEEE	Comments
Connectors	X	X		General connector symbols
Diodes	X			Various standard diode types
Discretes	X			Common discrete components
National Linear	X	X		Linear ICs
Transistors	X			Various standard transistor types
Spice Lib	X			For SPICE-based simulators
Primitives	X			DesignWorks simulation primitives
54 TTL	X	X	X	Includes 54, 54S, 54LS, 54AS, 54ALS, 54AC, 54ACT, 54F, 54HC, 54HCT, 54HCU
74 TTL	X	X	X	Includes 74, 74S, 74LS, 74AS, 74ALS, 74AC, 74ACT, 74F, 74HC, 74HCT, 74HCU
75 TTL	X	X	X	Includes 75ALS
CMOS	X	X	X	GE/RCA CMOS
ECL	X	X	X	National ECL
Memory	X	X	X	Motorola, Texas Instruments (TI) and Toshiba
MicroProcessors	X	X	X	Intel, Motorola and Zilog
Miscellaneous	X	X	X	
PLD	X	X	X	Note: When used with MacABEL, PLD symbols are auto-generated from the MacABEL library.

Appendix A - Other Schematic Features

Find Tool

The Find tool allows you to interactively locate devices, pins or signals by name, attribute value or by complex conditions defined by a script. Matching items can be viewed one at a time, or all items can be selected at once and then accessed with another tool. This scripting feature can be used to scan a design for possible design errors. Any located errors can be flagged as OK so they are not located again.

Report Generation

DesignWorks includes a powerful Custom Report Generator tool for netlist and text report generation. The report format is driven by a "form file" which contains formatting commands and constant text. Form file features allow you to control:

- Overall report structure, e.g. netlist formats by signal or by device, listings by device for bills of materials, etc.
- Design, device, signal and pin attributes to be included.
- Selection of devices or signals to include in the report, by name or attribute value.
- Selection of attribute fields to be used as sources for power and ground connections.
- Sorting and merging of lines within the file.
- Format of each pin connection entry, each line and each page.
- Hierarchy format, including flattened and hierarchical netlists.
- Error checking, including checking for missing attribute fields.

The package includes 20 or more form files for standard industry formats. These can be used as-is or used as a guide for creating your own formats.

Custom Sheet Settings

DesignWorks supports full customization of sheet borders, grids and graphics. The default border which is created for each new design can be updated from a "sheet template" file with a single operation. Template files are really just design files with custom settings for border size, text font, grid spacing, company logo, etc. Templates for the standard ANSI sizes A - E and other common sizes are included.

Attribute Operations

DesignWorks™ allows text attributes to be associated with any device, signal or pin in a design, and with the design itself.

Attribute Editing - Attributes can be entered individually on each object through a standard select and Get Info operation. Any attribute that is displayed on the schematic can be edited right on the schematic.

Attribute Display Options - Attributes can be displayed on the schematic adjacent to the associated signal, device or pin. Each field can be moved and rotated individually. A default display position can be specified for device attributes. Field names can be optionally displayed with each item. Text style for attribute display is set globally for all attributes in the design.

Attribute Global Operations - Attributes are defined centrally for a design. Global Duplicate, Merge and Delete operations allow all data associated with a field to be modified throughout the design. User-defined attribute fields can be added at any time.

Report Generation - All attribute fields can be used in custom report formats.

Hierarchy and Part Type Operations

DesignWorks™ provides a number of powerful operations for working with device or hierarchical block symbols and internal circuit definitions.

Hierarchy Mode - Three hierarchy modes are available to accommodate different design requirements. "Flat" mode provides the simplest possible structure for smaller designs. "Pure" mode implements a pure hierarchical structure, i.e. each part type is defined only once and all instances of a type are identical. "Physical" mode allows full hierarchical operation, but allows separate attribute data to be associated with each physical instance of a part.

Hierarchy Operations - Hierarchical designs can be created "top-down" or "bottom-up". Attach and Detach commands allow separately-created designs to be merged into a single hierarchy, or vice-versa. Internal circuit definitions can be saved to a separate circuit file. Part definitions can be saved from a schematic sheet to a library. Devices in a schematic can be updated from their original library or from any selected new source.

Back Annotation

The DesignWorks Back Annotation allows device package assignments to be automatically updated from a "was-is" text file created by an external system. Formats supported include Pads, Cadnetix (SCICARDS), Douglas, Racal-Redac RINF and XCAD. Pin swaps, gate swaps and component renaming are supported in flat or hierarchical designs.