EMA | Design Automation™

OrCAD Capture Essentials

17.4

EMA Education Services

Classroom, Live-Online, and e-Learning

Table of Contents

Overview / Using This Manual

Course Agenda

Lesson 1: Getting Started with OrCAD Capture
Lesson 2: Setting up Your Environment
Lesson 3: Creating Part Libraries
Lesson 4: Creating Homogeneous Parts
Lesson 5: Creating Heterogeneous Parts
Lesson 6: Building a Simple Schematic
Lesson 7: Processing a Schematic Design
Lesson 8: Building a Multi-Sheet Schematic
Lesson 9: Working with the Property Editor
Lesson 10: Using Search/Find, Edit Find and Browse
Lesson 11: Processing a Design for Cadence OrCAD and Allegro® PCB Editor

Related EMA Courses

All classes offered by EMA may be viewed at our website:

https://www.ema-eda.com/learning

Here you'll see a list of courses that are offered in a variety of delivery modes: Classroom based; Live-Online; e-Learning. Custom classes, tailored to your specific company needs, may also be requested.

Formatting Conventions

The following formatting conventions are used throughout this training manual:

When lab procedures instruct you to click a dialog box button, tab, option, or toolbar icon, the item is formatted in *bold, italic* text.

When lab procedures instruct you to select a file name, folder, or schematic page in either the OrCAD Capture Project Manager window or a directory tree, the name of the file, folder, or page is formatted in courier text.

When lab procedures instruct you to access a directory path, the path is formatted in courier text. When you are instructed to select a menu option, the option is formatted in *bold, italic* text.

When you are instructed to select a series of menu options, the primary and secondary menu options are separated by the *(>)* symbol.

When lab procedures instruct you to press a key on your PC keyboard, the name of the key is enclosed in brackets. For example: *<key>*.

When lab procedures instruct you to use *<LMB>* you are expected to press and hold the *<Ctrl>* key on your keyboard while you click design objects using your left mouse button. This technique if often used to select multiple design objects.

Throughout this manual *(LMB)* refers to your "**L**eft-**M**ouse- **B**utton". The instructor may also refer to this as the "***Control, Select"*** method of choosing objects.

Throughout this manual *(RMB)* refers to your "**R**ight-**M**ouse- **B**utton".

When lab procedures instruct you to select a design name or object, the name or object is formatted in courier text.

Lesson 1: Getting Started

Lesson Objectives

- Discuss design flow using OrCAD Capture
- Learn how to start OrCAD Capture
- The OrCAD Capture "Start Page"
- Open an existing Project
- Explore the user interface
- Describe project structure
- Select and edit objects

Cadence Solutions Overview
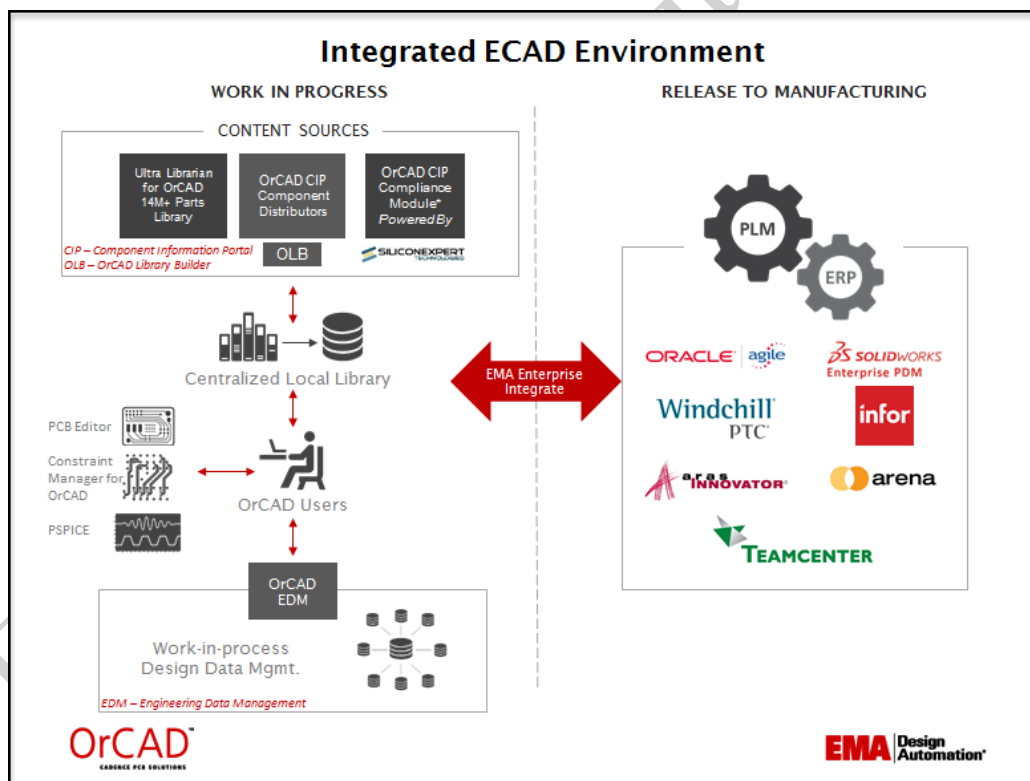
Successful PCB Data Management requires that several areas to be available within the design environment. **OrCAD Capture** schematic entry is the central starting point of the overall solution to data flow.

Project & Data Management

OrCAD EDM (Engineering Data Management): Having native PCB support puts a management solution directly into the CAD environment. EDM is a collaborative environment that enables project managers and users to track work in progress, check in and check out designs, track changes and manage entire projects from a comprehensive dashboard.

OrCAD Capture CIS (Component Information System) enables full database control over schematic symbols, PCB footprints, and all associated part parametrics and manufacturing information. Engineers can confidently choose parts for their designs from the validated, approved database, ensuring that their parts are accurate and available.

OrCAD CIP (Component Information Portal) enables users to completely manage the parts database. It provides access through assigned privileges, enabling teams to acquire new database parts directly from distributors like Digikey, Newark, Mouser, Premier-Farnell, Arrow, and RS Components. It also provides capabilities for viewing the schematic symbol for placement, as well as the PCB footprint, and the pcb step model. Additionally, there is a direct link to launch OrCAD Library Builder within CIP.

CIP Compliance Module

The **OrCAD CIP Compliance** module enables you to start part selection in the earliest stage of design. You'll no longer have to spend wasted time after the fact in part substitution because of obsolescence or because a part is found to be non-compliant, or due to sourcing issues.

Part Acquisition

Ultra Librarian for OrCAD (UL) provides a comprehensive, cloud-based library of millions of components eliminating the need for manual building and maintenance. Ultra Librarian for OrCAD offers engineers the option to search, preview and place components quickly without leaving the native CAD design environment saving time and eliminating errors.

OrCAD Library Builder (OLB) is the most advanced PDF data extraction tool available for quickly generating schematic symbols, PCB footprints, and PCB step models. It is easily accessible within the OrCAD environment.

OrCAD Capture Highlights

- Extensive schematic entry capabilities and productivity features ensure easy, fast, intuitive design capture
- Hierarchical, reuse, and variant design capabilities streamline the creation of complex designs
- **OrCAD CIS** accelerates the design process and lowers project costs by promoting preferred parts and optimizing part selection while OrCAD CIP further streamlines the process by providing a ready-made database with the ability to portal direct to the distributor to retrieve parts.
- Integrated signal integrity, analog/mixed-signal design, and simulation support enable circuit exploration, constraint development, and verification to help achieve first-pass success
- Bi-directional integration with the **OrCAD PCB SI** and the **OrCAD PCB** design products
- CAD vendor netlist interfaces for most PCB packages extends OrCAD Capture into multivendor flows

Schematic Entry Features

Schematic Editor

The OrCAD Capture schematic editor builds on the legacy of OrCAD providing fast and easy schematic editing for the simplest to the most complex designs. It combines an intuitive interface with all the features and functionality you need to speed through design tasks and circuit creation.

Productivity and Ease of Use

The schematic editor provides numerous features and functionality that enhance usability and speed for accomplishing design tasks and publishing design data. Context-sensitive menus, OLE support, custom colorization of wires, nets, and parts, and a tabbed and dock-able interface all provide a better user experience.

For larger, more complex designs, OrCAD Capture supports circuit reuse and hierarchical circuit blocks. Such designs are easy to traverse with navigation aids and OrCAD Capture ensures that all connections are maintained accurately throughout the design.

Intelligent PDF

The Intelligent PDF export in OrCAD Capture enables you to create content-aware PDFs of your schematic design. More than just a .pdf file with searchable text, the content-aware PDF retains intelligent design information, allowing you to query signals, display properties, navigate hierarchical blocks, and more. These PDFs eliminate the need for other team members to have additional software licenses or specialized viewers for design reviews, increasing the ease of collaboration and communication.

FPGA Co-design

OrCAD Capture, together with the OrCAD FPGA System Planner product, addresses the challenges you encounter when designing large-pin-count FPGAs for the PCB: creating the initial pin assignment, integration with the schematic, and ensuring that the device is routable on your PCB. By replacing manual, error-prone processes with automatic pin-assignment synthesis, this unique placement-aware solution helps eliminate unnecessary physical design iterations that shorten design time.

PSpice simulation

The OrCAD PSpice analog/mixed-signal circuit simulator solution is seamlessly integrated within OrCAD Capture to boost productivity and allow you to use the same schematic for both simulation exploration and PCB layout, reducing rework and errors. Even if you're not creating a circuit for use in the PCB flow, the integration allows for easy setup, model placement, circuit creation, and simulation, as well as cross-probing of simulation results.

Signal integrity

Tightly integrated to provide a bi-directional schematic entry and signal integrity flow, OrCAD Capture and the OrCAD PCB SI product allow you to perform circuit topology exploration, constraint development, and signal integrity analysis from the schematic during design entry. The associated Electrical Constraint set (Electrical CSet) as well as the complete topology file is embedded in the schematic database.

PCB Design

Seamless bi-directional integration with OrCAD PCB Editor in the OrCAD PCB Designer products enables synchronization and cross-probing/placing between the schematic and the board.

The interface makes synchronizing connectivity and design rules easy and straightforward and automates back-annotation of engineering change orders (ECOs) including layout changes, gate/pin swaps, and changes to component names or values.

The OrCAD Capture Design Flow
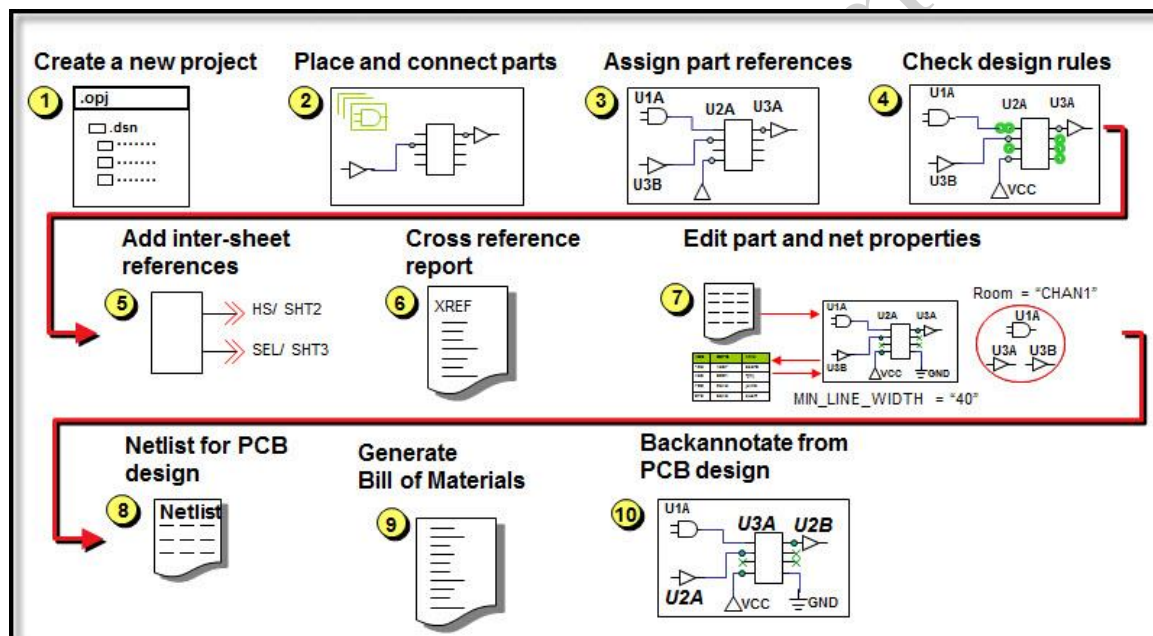
This diagram shows the basic steps in the OrCAD Capture design flow. Steps for FPGA design, analog or digital simulation, and OrCAD and Allegro PCB Editor are not shown, and are not covered in this class.

Starting OrCAD Capture

There are a couple of ways you can open the OrCAD Capture application:

- Select *Start > Cadence PCB 17.4-2019 > Capture CIS 17.4*
  - You may see a list of licenses appear where you can select *OrCAD Capture*
- Double click the OrCAD Capture shortcut icon located on your desktop

The OrCAD Capture menu paths displayed from the Start button are automatically defined during software installation. The OrCAD Capture desktop icons are manually created.

The OrCAD Capture "Start Page"

When you start the OrCAD Capture software, the main session window appears with the "Start Page" as shown in the following graphic.
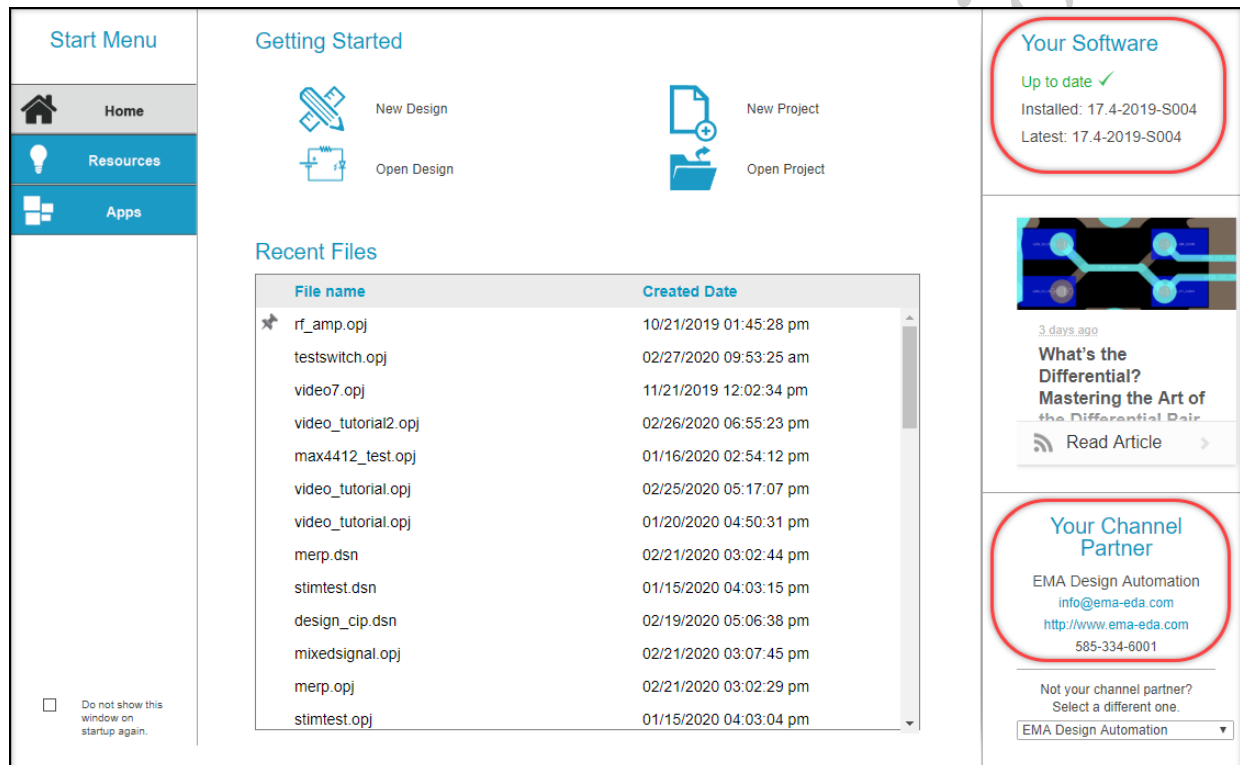
The "Start Page" tab appears in the open Session Window. If you leave this active, it is from here you can start a New Project (or Design), Open an existing Project (or Design), open the Capture Tutorial, or open Capture's Documentation. You can also access these commands from the toolbars at the top.

In the upper, right corner, the current version you are running will be displayed.

Below that cadence shows articles about interesting topics or ways to better use your Cadence tools.

In the lower, right hand corner you can select your current Vendor or Channel Partner and "Home" page will now appear giving you access to Support information, Upcoming Events, and Design Solutions.
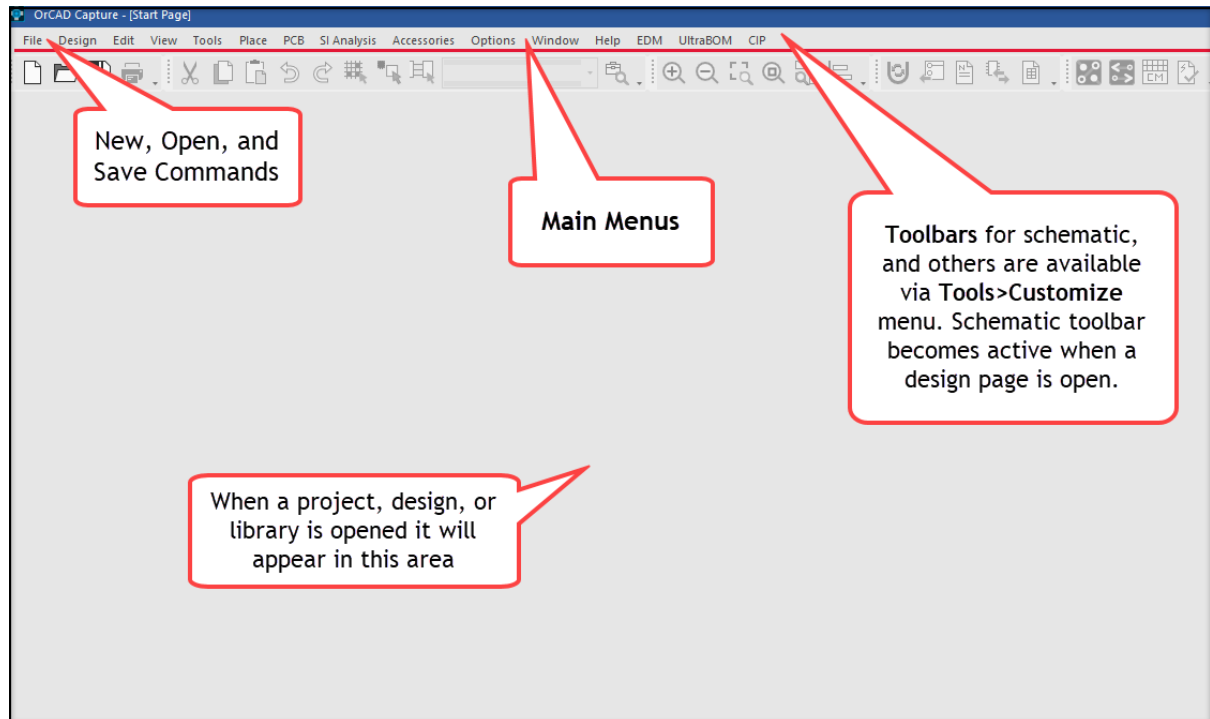
Use the *File* menu to create new projects and libraries, open existing ones, save design edits and print schematics.

Use the *Options* menu to set user preferences or to create a design template. User preferences and design templates are covered later in this lesson.

The *Session Log* is a window that displays messages and errors. You can save the contents of the Session Log window to a file. This window does not create a log file that can be replayed. It will be displayed in a tab below the schematic window. It can also be accessed from View in the main menu.

Click on session log and all the seven menus: File, View, Tools, Edit, Options, Window and Help should be available.

OrCAD Capture Help

The *Help* button ⑦ located on the far right of the main session window toolbar allows access to the online Help system and an online tutorial. Many of the topics included are similar to topics covered in the OrCAD Capture training class.

Demo Designs

The Cadence installed hierarchy contains more than 150 demo designs to help you understand Capture, Capture CIS, and the Capture -> Pspice flow. With this release these designs can be accessed via the **File > Open > Demo Designs** menu command as shown in the following graphic.
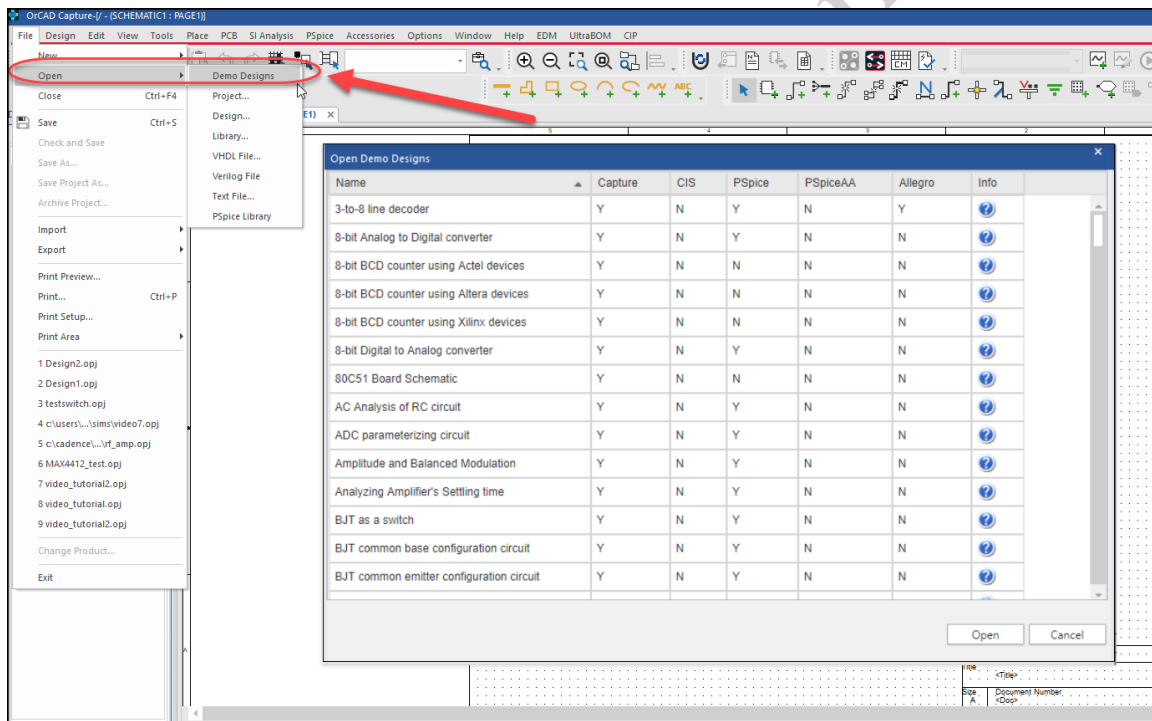
EMA Customer Support

The EMA Resource Center gives you access to downloads, FAQ's, Application Notes, and more. Access the EMA Resource page using the following link.

http://support.ema-eda.com/

Opening an Existing Project

Use the Open Project window to navigate to and select an existing project *.op*j file.
OrCAD Capture reads the project file and opens the Project Manager window. Projects
can also be accessed from the Start Page.
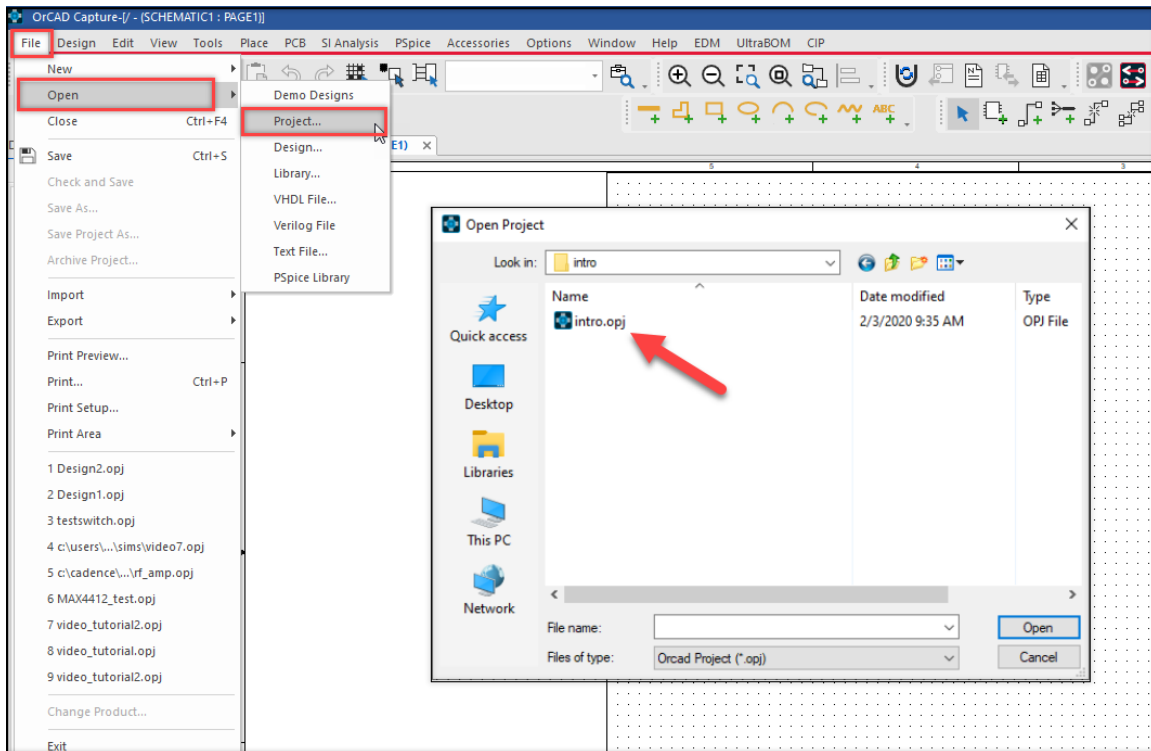
The Project Manager

The Project Manager is a window that displays a directory tree of all files related to your design or project.
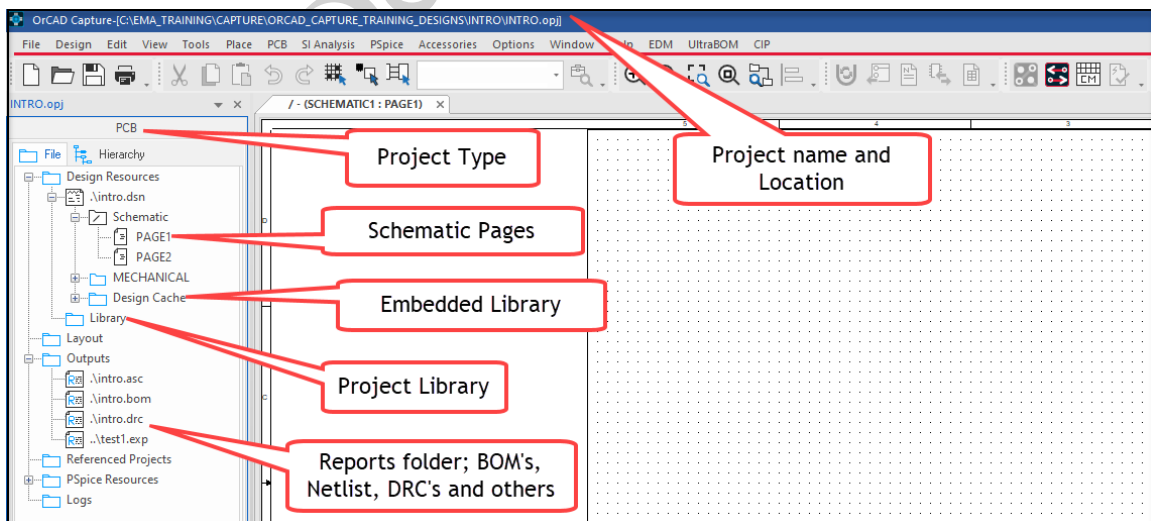
The project name and project type are shown at the top of the Project Manager window. (Project types will be covered in Lesson 8.)

You can use the Project Manager to open schematic pages and view reports, schematic pages, or other project files which open when you double click a page or file name. You can also use the Project Manager with the right mouse pop-up menu to add, delete, or rename schematic pages.

You can open multiple projects at the same time, and copy designs or schematic pages between Project Manager Windows. You can also use the Project Manager to add libraries to your project setup.

When you open a project, the setting preferences last used become active in the open project. For example, if a schematic page was open when you saved and exited the project, that schematic page will automatically display when you reopen the project.

A Simulation Resources folder (not shown in the previous graphic) may also be present for use with PSpice A/D.

The OrCAD Capture User Interface

OrCAD Capture is a menu driven application. There is no command line entry. All OrCAD Capture commands are contained in pull-down menus, icon toolbars, right mouse pop-up menus, and keyboard shortcuts (or Hot-Keys).

OrCAD Capture commands are context sensitive (see the graphic below). Accesses to commands vary with different active windows such as the Project Manager or schematic page. When either window is selected, buttons and menu commands not relevant to the active window are greyed out or missing. Tasks you perform must always be

relevant to the current or active window. You can apply a process to a specific page of a design by pre-selecting it before you perform the task. Menu choices are also affected by project type.
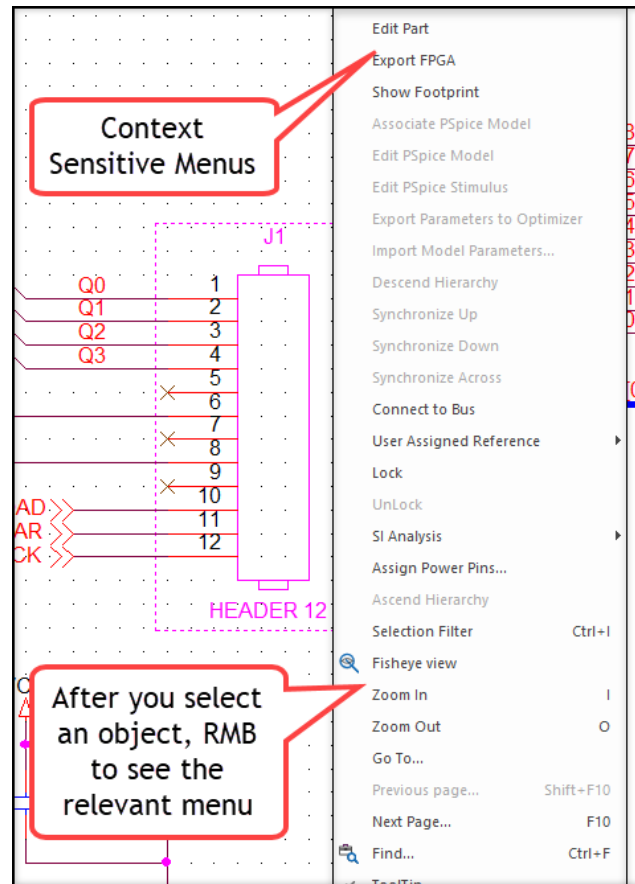
The toolbars default to the top and right side of the OrCAD Capture session window, but you can move them to any edge of the session window, make them float free of the session window, and change their shapes.

The Main Toolbar

The main toolbar appears at the top of the session window. If you place the cursor over a button in the toolbar (without clicking), a brief description appears.
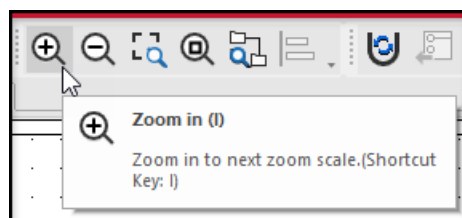
The Schematic Toolbar

The schematic toolbar provides a quick way to execute schematic editing tasks. This toolbar appears, by default, on the right edge of the design window. It can be moved to any convenient viewing location. The Toolbar is active only when a schematic page window is selected.

Tools to place parts and connect with wires

Tools to place symbols

Non-electrical Tools to draw shapes

Complete command descriptions for the Schematic Toolbar can be found on page 86. Key board shortcuts are also available and can be found on page 45of this document.

OrCAD Capture Essentials   Version 17.4

Cross Hair Mode

Cross Hair (full cursor)
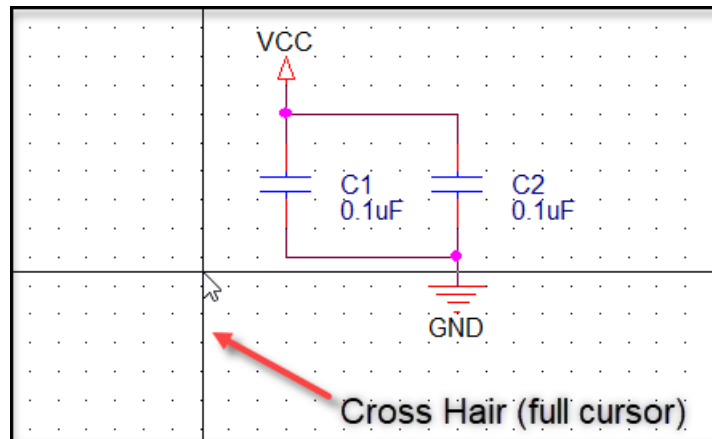
When placing a part on a very dense design, it is not always easy to find your way around the design. To make the task more manageable, you can turn on the Cross Hair Mode (full cursor). This will place a Cross hair at the edge of the mouse pointer. As you move your cursor across the page the cross hair will make it easier to locate object like pin and nets or very small parts in your design.

Use the F6 key to toggle this mode on or off.

Lab 1-1:   Opening Existing Project

NOTE: Please observe the appropriate mode of training you are completing at this time (Classroom, Live-Online, or e-Learning) and follow the corresponding steps to ensure access of the training lab material.

Classroom

This training generally provides <u>laptop</u> computers which have been pre-set with the Capture directories where the training files are stored. If you find that your training material is not in the directory indicated, consult your training instructor, and you may also create the proper directory to house the training files for the duration of the course. The directory where your class files are stored will be: **C:\EMA_Training\Capture_Essentials_17.4**

Live-Online

This training uses the AWS (Amazon Work Space) environment. In this scenario EMA generates an AWS for each student. You are informed prior to the course how to connect to your AWS machine, which is already pre-loaded with necessary software and training designs. Each AWS machine has been pre-set with the Capture directories where the training files are stored. If you find that your training material is not in the directory indicated, consult your training instructor, and you may also create the proper directory to house the training files for the duration of the course. The directory where your class files are stored will be: **C:\EMA_Training\Capture_Essentials_17.4**

e-Learning

This training is self-paced, not instructor led. It is dependent on having the software installed on a machine you can access in order to complete the training labs that are part of this material. On the machine you will be using to complete the labs, you will want to create a directory named: **C:\EMA_Training\Capture_Essentials_17.4** and then download the training materials from the **E-Learning portal** for the Capture class to this directory. Refer to your **E-Learning Welcome Email** to review the steps for this task.

Lab Objectives

After you complete this lab you will be able to:

- Start OrCAD Capture
- Open a project
- Zoom and pan
- Close a project

Start OrCAD Capture and Open a Project

1. Select *Start > Cadence PCB 17.4-2019 > Capture CIS 17.4*
2. You may see the Product Choices Window appear where you can select *OrCAD Capture*. Optionally you can double-click the Capture icon located on the desktop.

The OrCAD Capture session window appears.

The startup path and directory name may vary with different versions of the application installation CD.

3. Select *File > Open > Project*.
4. The Open Project window appears.
5. Navigate to the C:\EMA_Training\Capture_Essentials\Designs directory. This directory is your lab database. All your lab files are stored under this folder. Please refer to the **TRAINING FILES NOTE** above that describes the training format you are taking and how to create the EMA_Training… directory, if it is not already created.
6. Select and open the Sample directory and choose sample.opj and click *Open*.

   The Project Manager window opens. Observe that the name of the project file is displayed at the top of the Project Manager window.

Opening a Schematic

If the schematic is not already open:

1. In the Project Manager window, double click .\sample.dsn.

2. Double click **SCHEMATIC1**, then double click PAGE1.
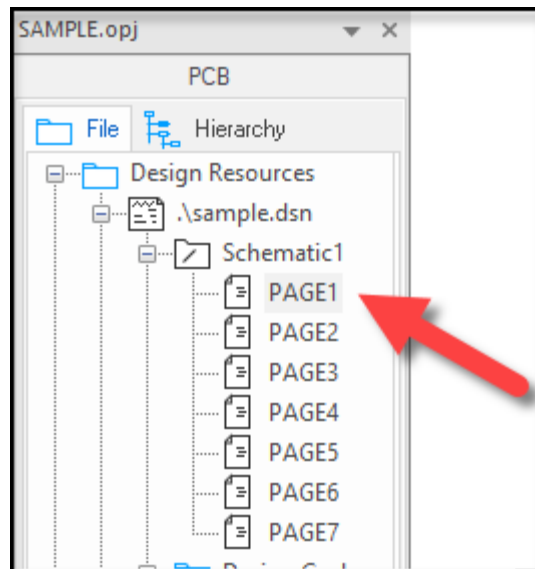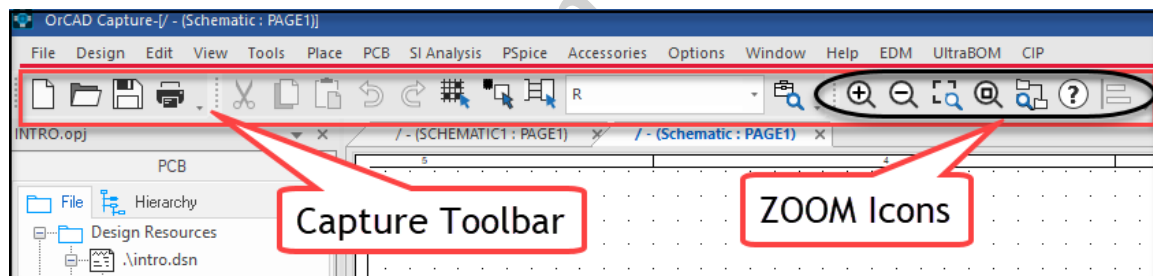
4. The image has been reduced by a factor of 2 to 1. The zoom scale factor is a user preference. We will show you how to set user preferences in lesson 2.
5. Press the **<I>** key to zoom in and the **<O>** key to zoom out. When zooming in or out, observe that the cursor remains at the center of the view. These commands are "cursor location dependent".
6. Place your cursor in a different area and press the **<I>** and **<O>** keys again.
7. With your view zoomed in, place your cursor to the right or left side of your view. Press the **<C>** key to center the view.
8. Hold the **<C>** key down (do not touch the mouse buttons) and move the mouse. Notice the "auto scroll" feature.
9. In the main toolbar, click the **Zoom to Region** icon.
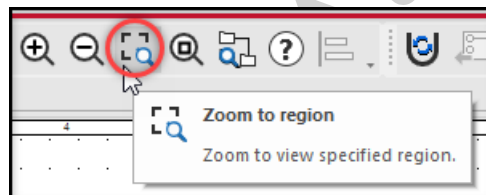
Observe that the cursor shape turns into a magnifying glass.

   a.   Click and drag to define the area into which you want to zoom.

   b.   Right-click and select **End Mode** from the pop-up menu. You can also use the **<Esc>** key to exit a command.

Zoom commands are also available in the **View** pull-down menu.

Panning

1. Click the **Zoom to All** icon, followed by the **Zoom In** icon.
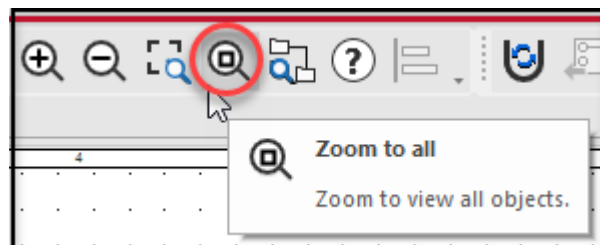
2. Place your cursor on a part (do not click the mouse).
3. Press the **<C>** key on your keyboard.
4. The location of your cursor becomes the new center-point of the view. Try this on another part.
5. To pan, press **<Shift>** and mouse scroll wheel. Or hold down **<C>** and move your mouse back and forth (do not click the mouse).

Additional keyboard shortcuts and function key settings are documented on page 45 of this workbook. You cannot create your own custom keyboard shortcuts.

Panning with the Right Mouse Button (Optional)

1. Zoom into an area of the page.

2. Click the middle-mouse-button once. A panning cursor appears.
3. Now simply move your cursor slowly around on the page. The "cursor" will attempt to follow your mouse movement. When you are done, click the middle-mouse-button again.

Closing the Project

1. To close the schematic page, select *File > Close* or press the "**X**" on the tab next to the schematic page name.
2. To close the project, select *File > Close.*

The Project Manager window closes, but the main session window remains open.

Lab 1-2: Using the Help System

Lab Objectives

After you complete this lab you will be able to access and use:

- Online Help
- Online tutorials
- Online manuals and product notes

Online Help

1. In the OrCAD Capture session window depending on the OrCAD Capture tool you opened, select either *Help > OrCAD Capture Help*
2. The Cadence Help window appears displaying the topic *OrCAD Capture*. Click on the *Documentation browser* icon in the top toolbar to expand and display the contents found under the *OrCAD Capture User Guide* in a panel to the left of the original.
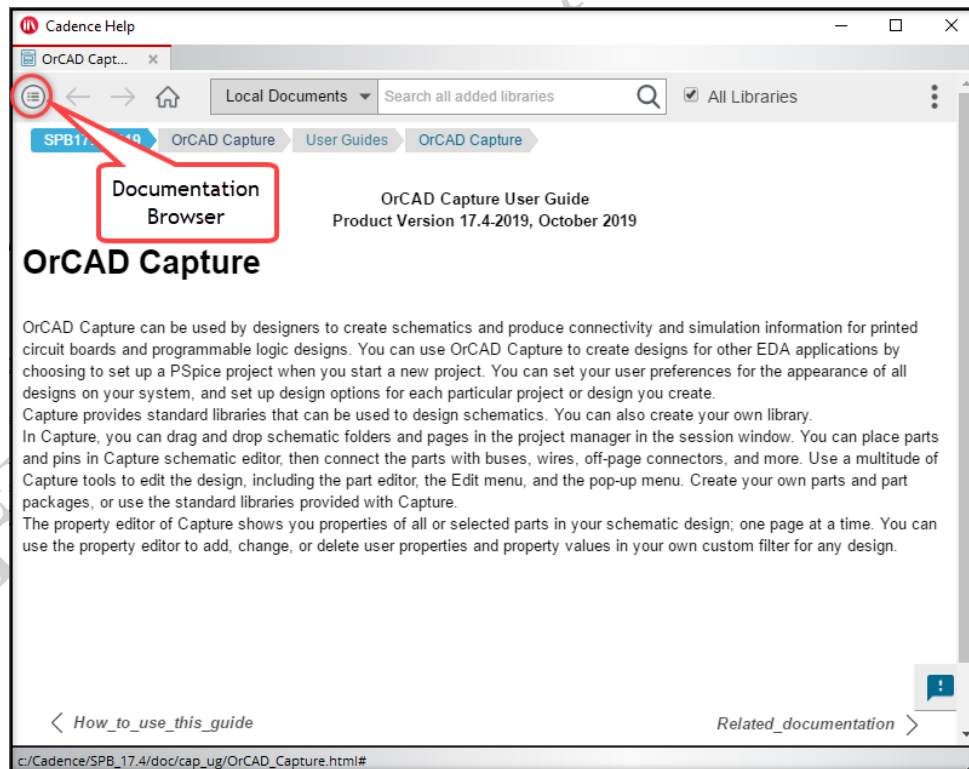
Online Tutorial

1. In the OrCAD Capture Start Page, on the left side are icons, one of which is the *Learning Resources* icon. Click this and notice the other areas you can select, including the *Capture Flow Tutorial*.
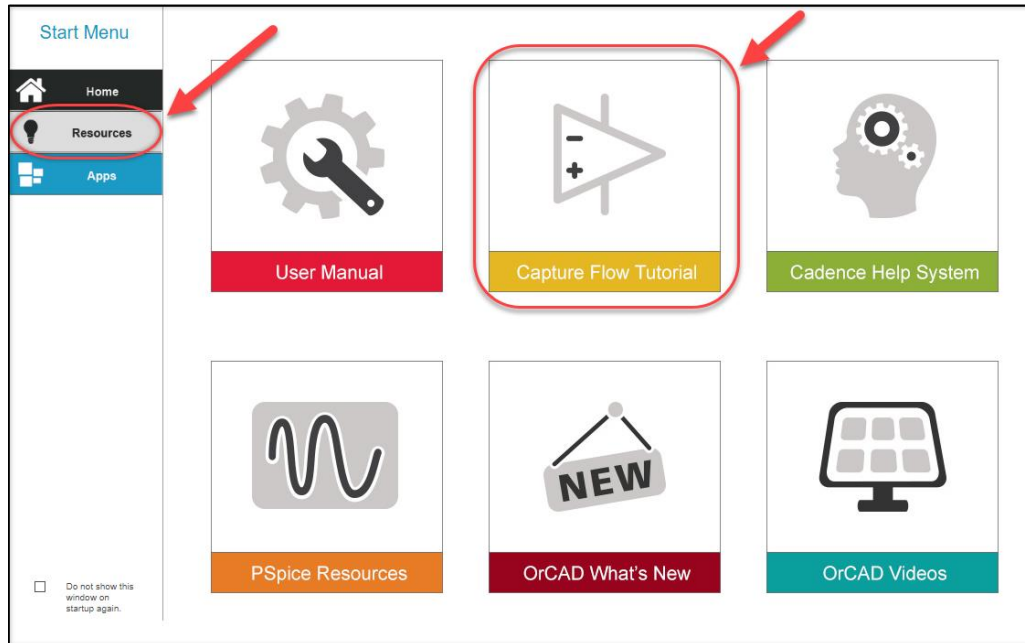
2. Click on *Capture Flow Tutorial* link. Once you do, notice the items that are presented, under the *What's New* image, you will find tutorials Tutorials.
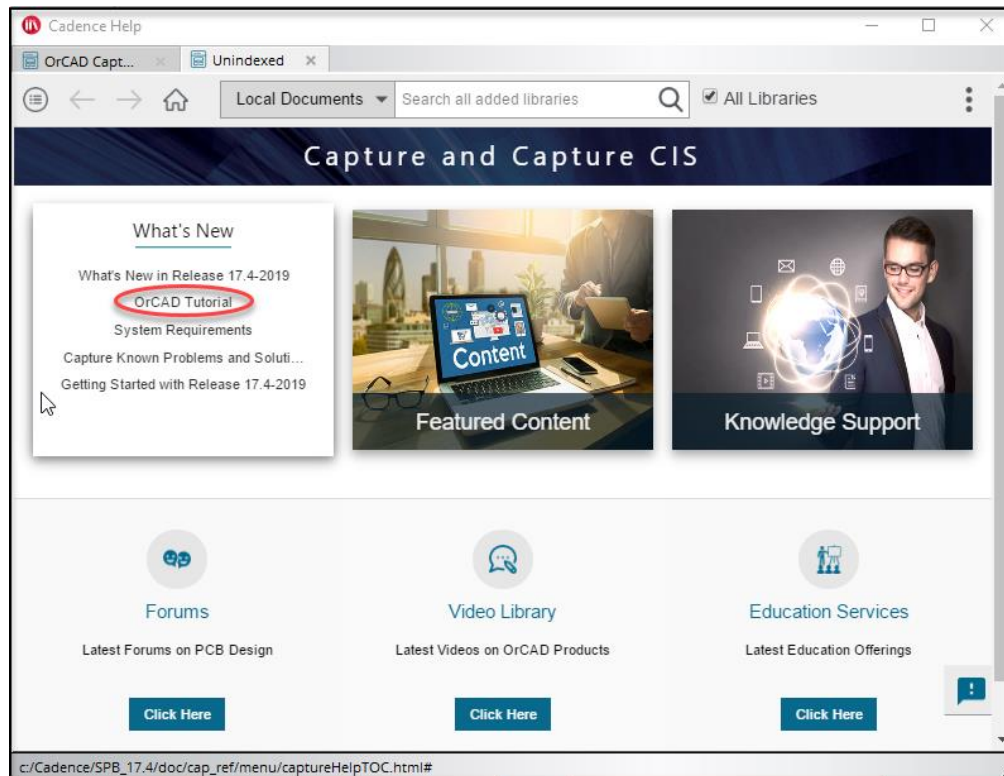
You will not be using the tutorials for this training class. For future reference you can access these as training aids once you have Capture installed in your own work environment.

3. Close the Help windows.

File Structure

Project Manager

The Project Manager structures design resources into a 'virtual' hierarchy of folders and files. However, this hierarchy exists only within the Project Manager window (not on disk).

The graphic that follows illustrates the directory structure of folders and files in the Project Manager.
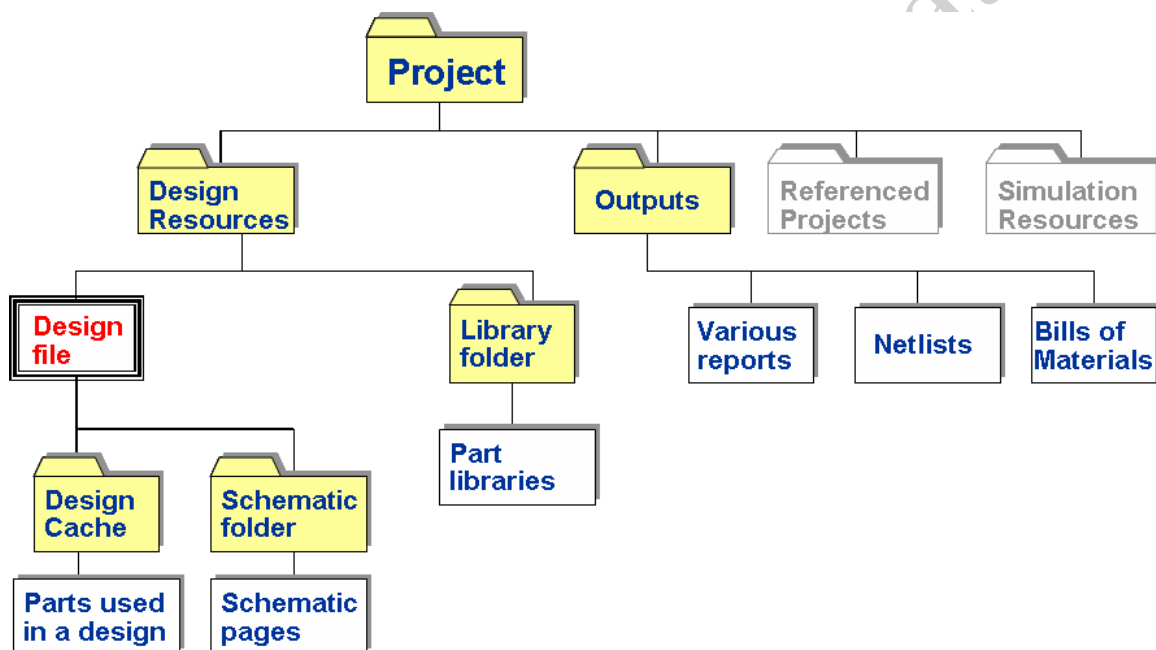
The Referenced Projects folder is typically used to reference timing and functionality data from an FPGA project.

The Simulation Resources folder is present when the project type is Analog or Mixed A/D, and you plan to use the PSpice A/D simulator.

System Files

Some of the structures shown in the Project Manager Window relate to OrCAD Capture system files. Although many project files map to system files, many do not. All files related to a project should be contained in a top-level project folder.
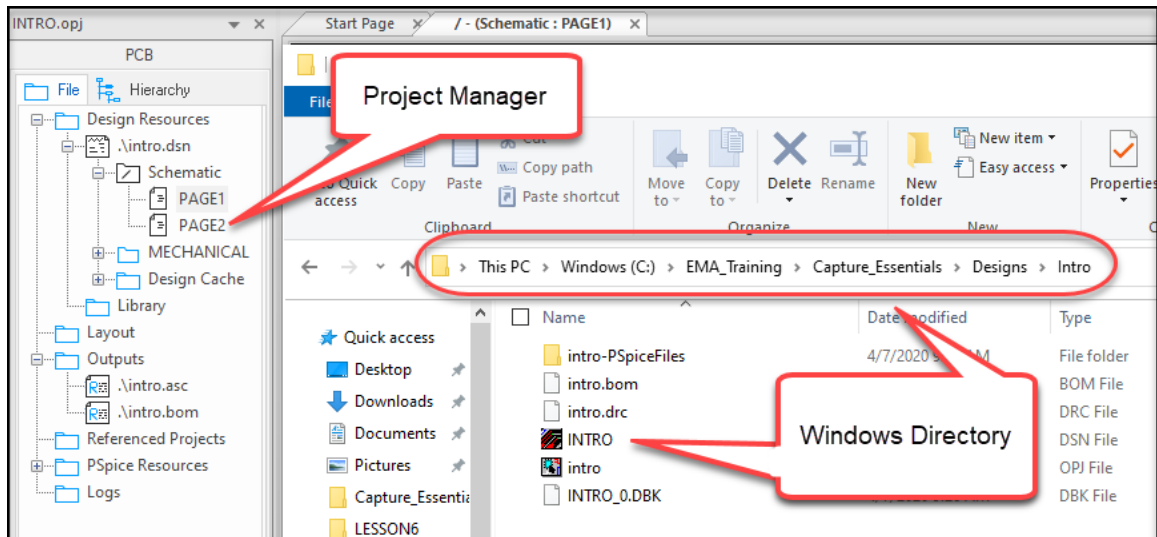
File Extensions

The table that follows lists OrCAD Capture files and extensions.

File Type	Extension
OrCAD Capture project file *	.OPJ
OrCAD Capture design file *	.DSN
OrCAD Capture part library	.OLB
Property update file	.UPD
Design rule check file	.DRC
Bill of Materials file	.BOM

Export properties file	.EXP	
OrCAD and Allegro PCB Editor back annotation file	.SWP	
OrCAD and Allegro PCB Editor netlist files	.DAT (3 files)	
VHDL source file	.VHD or .VHO	
EDIF netlist or back annotation file	.EDF or EDN	
Cross-reference report	.XRF	
Other netlist files	.NET or .ASC	

In the above table, files with an asterisk are necessary when handing the design off to another person for archiving your design.

NOTE: If you have a greater need to manage OrCAD files because you are working in a team environment that may share work-in-progress you may want to look into OrCAD EDM. **OrCAD EDM (Engineering Data Management)** is a collaborative design management tool that enables you to organize and manage designs from a project management level. With EDM you can: Track the history of each change; Control who has access to projects: and control and store all of the related project files; and more. Visit EMA Design's website to see how OrCAD EDM can assist you with project data management.

Each time you save an OrCAD Capture design, a <design>.dbk backup file is created.

Automatic backup enables you to specify the backup frequency of the DSN file. This feature also lets you specify the number of backup versions kept. You can save between one and ten backup versions.

Selecting and Deselecting Objects

The selection and deselection of OrCAD Capture objects controls your ability to edit, move, copy, delete, or edit properties. The graphic that follows shows a selected part, wire, pin and text.
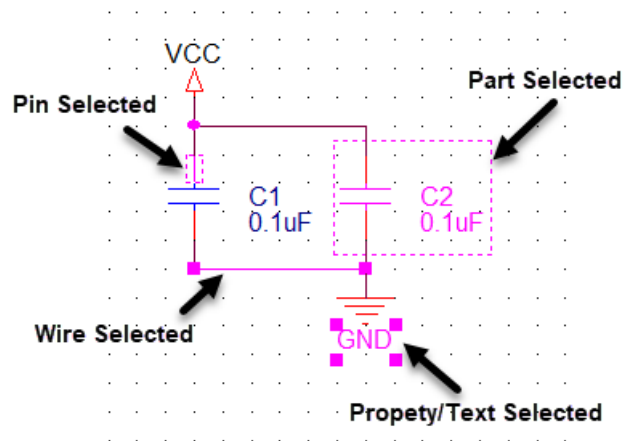
When selecting parts, be sure you see the rectangular selection box around the entire symbol, not just one of its elements, such as a pin or text.

Selection Filter

Controlling the selection of objects on a schematic page during a block selection is done by using the Selection Filter dialog box. It provides check box options to select or exclude Parts, Nets, Power/Gnd, Title Blocks, etc. so that only specific objects are selected when you perform the window or block-select operation.
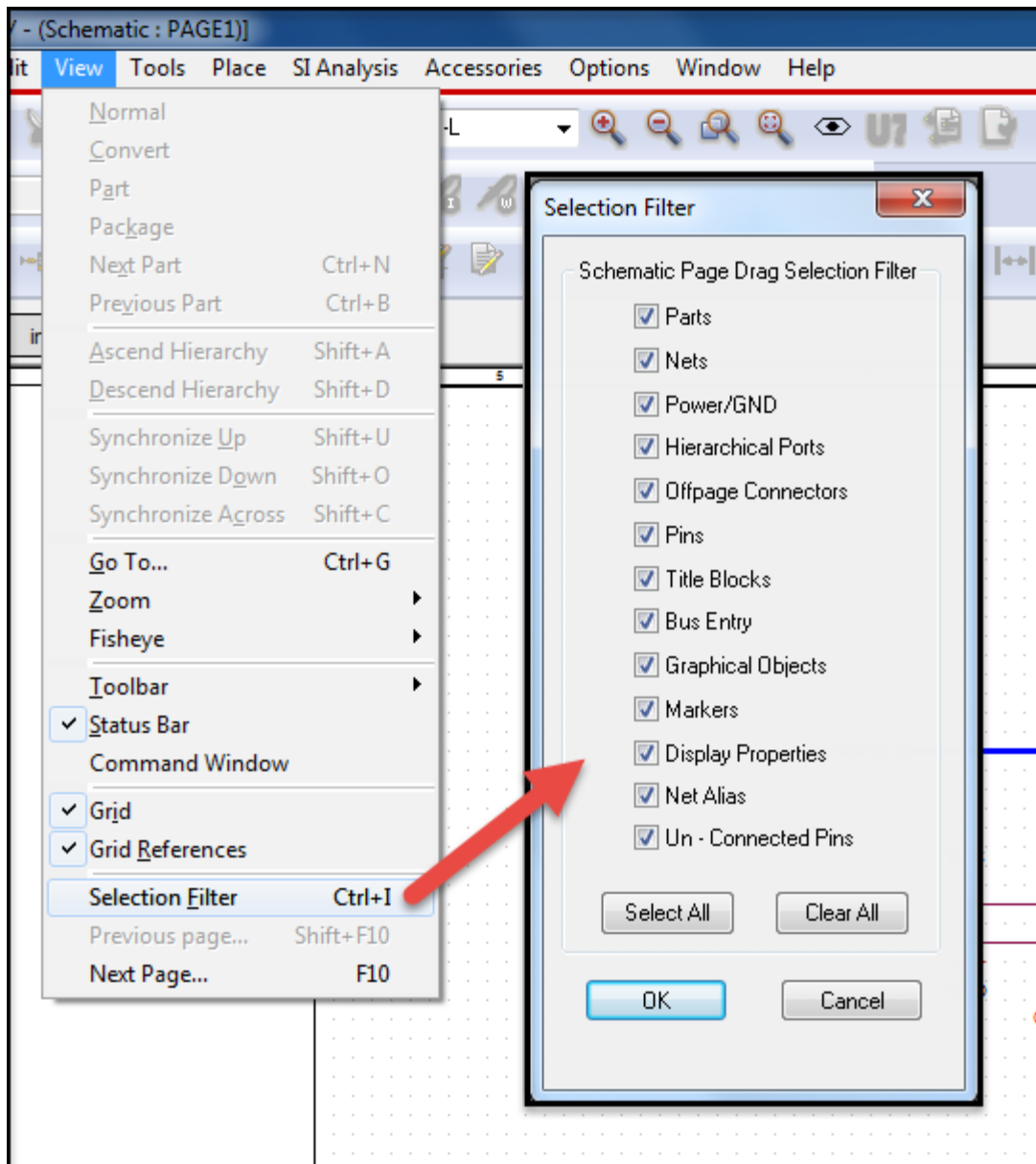
Editing Objects

Frequently used editing commands are accessible from the right mouse pop-up menu.

Although editing options may vary with the selection of different object types, the graphic that follows shows one example of options available through the pop-up menu.

Mirror Horizontally		
Mirror Vertically		
Mirror Both		
Rotate		
Edit Properties...		
Edit Part		
Export FPGA		
Show Footprint		
Associate PSpice Model		
Edit PSpice Model		
Edit PSpice Stimulus		
Export Parameters to Optimizer		
Import Model Parameters...		
Link Database Part	Ctrl+L	
View Database Part..	Ctrl+D	
Descend Hierarchy		
Synchronize Up		
Synchronize Down		
Synchronize Across		
Connect to Bus		
User Assigned Reference	▶	
Lock		
UnLock		
SI Analysis	▶	
Add Part(s) To Group..	Ctrl+Shift+A	
Remove Part(s) From Group..	Ctrl+Shift+R	
Assign Power Pins...		
Ascend Hierarchy		
Selection Filter	Ctrl+I	
Fisheye view		
Zoom In	I	
Zoom Out	O	
Go To...		
Previous page...	Shift+F10	
Next Page...	F10	
Find...	Ctrl+F	
ToolTip		
Cut	Ctrl+X	
Copy	Ctrl+C	
Delete	Del	
More...	▶	

[R 12 Ref: J1]

Pop-up Menus are entity dependent.

For example - after selecting a component RMB click to view this Menu.

Lab 1-3: Selecting Objects

Lab Objectives

After completing this lab you will be able to:

- Start OrCAD Capture and open a project
- Open a schematic
- Select an object
- Select multiple objects

Starting OrCAD Capture

1. If OrCAD Capture is still open from the previous labs move to step 2 in "Opening the Project". If you closed Capture after the last labs then: Select *Start > All Programs > Cadence PCB 17.4-2019 > Capture CIS 17.4*. Optionally you can double-click the Capture icon located on the desktop. Choose Capture, then the OrCAD Capture session window appears.

Opening the Project

1. Select *File > Open > Project*.
   The Open Project window appears.
2. Navigate to the C:\EMA_Training\Capture_Essentials\Designs\intro directory.
3. Select intro.opj and click *Open*.

Opening the Schematic

1. In the Project Manager, double click on .\intro.dsn.
2. Double click on *Schematic or* press the ⊞ next to the schematic folder.
3. Double click on *PAGE1*.
4. Enlarge the schematic window and click the *Zoom to All* icon. ⊕
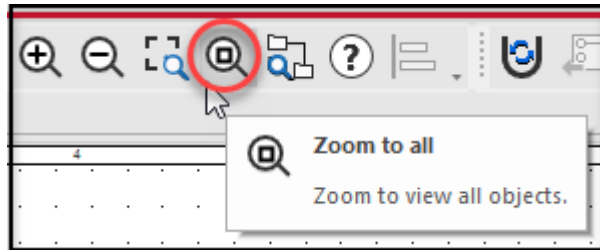
Selecting Objects

1. Select a part by clicking on it with the left mouse button.
   Observe the dashed boundary around the part.
2. Click to select a wire. Observe that the wire segment highlights, and selection boxes appear at its endpoints.
3. Zoom in and select a text string (for example, a net name or reference designator).

   Observe that the text string highlights, and selection boxes appear around it.

Selecting Multiple Objects

1. Click on capacitor **C1**. Hold the *<Ctrl>* key down and click on capacitor **C2**.
   Both caps are now selected.
2. Hold the *<Ctrl>* key down and click on capacitor **C2** again.
   Capacitor **C2** is now deselected.
3. Click left anywhere in the page (away from parts or wires).
   This deselects all objects in the page.
4. Press and hold the left mouse button, and drag a rectangle around capacitors **C1** and **C2**. (Be sure to include the power/gnd symbols and wires.)
   All parts, wires, and text within the rectangle are now selected.
5. Press the *<Esc>* key to deselect all objects.

Filter Selection

1. There are three ways to get to the *Selection Filter* command.

   a.   From the *View* pull-down menu, select *Selection Filter*.
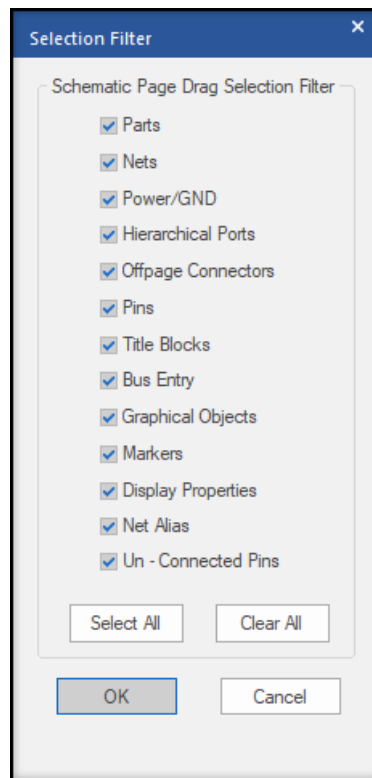
b.   Place your cursor in the schematic page window. From the right mouse
        button select *Selection Filter***.**

c.   The third method is to simply use the hot-keys: ***<CTRL+I>***

2. When the Selection Filter dialog box appears, select Clear All.
3. Now place a check mark in only the ***Nets*** box and click on ***OK.***
4. Draw a box around everything on the current page. Notice that only the wires or
    Nets are selected.
5. Click once in an open area to deselect the items.
6. Return to the Selection Filter dialog box and reset to ***Select All***.

The following table is included as a reference for selecting and deselecting objects.

To select …	Do this…
A single object	Left click on the object (the selected object will highlight). *Tip*: While selecting, press and hold the ***<Tab>*** key to toggle between overlapping objects.

Multiple objects	Press and hold the *<Ctrl>* key while selecting individual objects.
All objects in an area	Click and drag to draw a selection rectangle around an area.
All objects on a schematic page	Select *Edit > Select All* from the pull-down menu.
All objects of a specified type	Choose *Edit > Find*, and specify search criteria for the type(s) of objects you want to select.

To deselect …	Do this…
All selected object(s)	Click anywhere else other than on the selected object(s).
One of a group of selected objects	Hold the *<Ctrl>* key and click over the desired object. Continue this procedure to deselect other objects individually.

Lab 1-4: Editing Objects

Lab Objectives

After completing this lab you will be able to:

- Move individual parts and wires
- Move groups of parts and wires
- Delete
- Undo
- Copy parts

Moving Individual Parts and Wires

1. Place your cursor on a part, press and hold the left mouse button, and drag the part to a new location. Observe that the attached wires stretch with the part.
2. Place your cursor on a wire, press and hold the left mouse button, and drag the wire segment to a new location. Observe that the wire segment remains connected.
3. Use the *Undo/Redo* toolbar icon to return the parts to their original locations.
4. Hold the *<ALT>* key and drag **C2** to the right.
   The part is disconnected from the wires. You can now move the parts without attached wires.

Moving Groups of Parts and Wires

1. Press and hold the left mouse button and drag a rectangle around capacitors **C1** and **C2**. (Be sure to include the power/gnd symbols and wires.)
2. Place your cursor on a part or wire, and notice that the cursor changes to a cross hair.
3. When the cursor appears as a crosshair shape, press and hold the left mouse button, and drag the selected group to a new location.
4. Press *<Esc>*.

Delete

1. Click to select a part or wire.

   a) Right-click and select *Delete* from the pop-up menu.
   b) Press the *<Delete>* or *<Backspace>* key.

2. Click on capacitor **C1**. Hold the <*Ctrl*> key down and click on capacitor **C2**. (This is referred to as the "control, select" or <***CTRL+LMB***> method). Both caps are now selected.

     a) Press the <***Backspace***> *or* <***Delete***> key to delete both capacitors at once.

3. Press and hold the left mouse button and drag a rectangle around both capacitors (include the power/gnd symbols and wires).
4. Delete the selected parts and wires, and then undo the deletion.
5. Select and delete C1, then select and delete C2.
6. Follow directions in the next section to undo both deletions.

Undo

You can undo the deletion of both **C1** and **C2** because of the "Multiple Un-do's" in OrCAD Capture.

1. Press <***Ctrl+Z***>

2. Click the ***Undo*** icon in the main toolbar .

Copying Parts

1. Place your cursor on a part, press and hold the <***Ctrl***> key and left mouse button, and drag a copy of the part to a new location.
2. Press <***Ctrl+Z***> to undo the copy.
3. Click on the part again.

    a.   Press <***Ctrl+C***> to copy.

    b.   Press <***Ctrl+V***> to paste.

    c.   Click to place a copy of the part.

    d.   Press <***Ctrl+Z***> to undo the copy.

You will also find ***Copy, Paste*** on your pop-up menus as well as the 3 icons on the tool bar for ***Cut, Copy, and Paste***.

4. Press and hold the left mouse button and drag a rectangle around a capacitor (include the power/gnd symbols and wires).
5. Press and hold the <***Ctrl***> key and left mouse button and drag a copy of the circuit to a new location.
6. Click left in a blank area to unselect all objects.

Move with Disconnect

1. Select C1 (or C2) again. Attempt to drag it to another location to the right or left.
   Notice that wires WILL stay connected to the pins of a part.
2. Move the part back to the previous location or use *Undo*.
3. Select the part again. This time hold the *ALT* key down as you drag the part to a new location.
   Notice that this time the part is unconnected from the wires.
4. Leave the design open to continue to the next lesson.

Select *File - Close* and *No* to discard all changes if you wish to close the design at this time.

The following table lists additional editing commands.

To …	Select the object and do this…
Move an object	Press and hold the left mouse button and drag the object to a new location. *Tip*: Hold the *<Alt>* key to split an object away (for example, to disconnect a part from existing wires).
Delete an object	Press *<Del>* or *<Backspace>* or Choose *Edit > Delete*.
Copy an object	Press and hold the *<Ctrl>* key and drag a copy of the selected part to a new location. or press *<Ctrl+C>* or Choose *Edit > Copy*.
Cut an object	Press *<Ctrl+X>* or Choose *Edit > Cut*.
Paste an object	Press *<Ctrl+V>* or Choose *Edit > Paste*.

Undo the last action	Press the ↺ button or press *<Ctrl+Z>* or Choose *Edit > Undo*
Reverse an Undo	Press the ↻ button or Press *<Ctrl+Y>* or Choose *Edit > Redo*.
Repeat the last command	Press *<F4>* or Choose *Edit > Repeat*.
Rotate an object(s) 90 degrees counter-clockwise	Press *<R>* or Choose *Edit > Rotate*.
Mirror an object vertically and/or horizontally	Choose *Edit > Mirror*.
Resize an object	Drag a resize handle on the highlighted object.
Group selected objects so you can manipulate them as a set	Choose *Edit > Group*.

Observe that the main toolbar also contains *Copy, Cut*, and *Paste* icons.

Lesson 2: Setting up Your Environment

Lesson Objectives

- Set User Preferences
- Create a Design Template

Setting User Preferences

OrCAD Capture, Version 17.2, has the following user preferences:

- Theme
- Color assignment
- Grid display
- Pan and zoom
- Area selection
- Line and fill style
- Autobackup
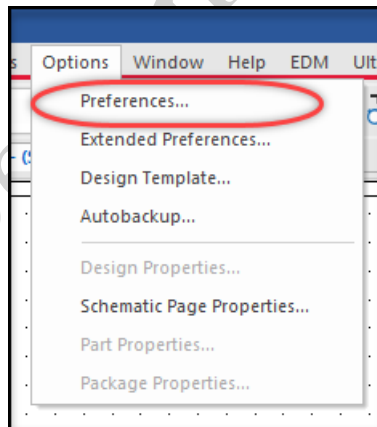
User preferences are workstation-specific. For example, if you copy a design from another person and open it on your machine, your user preferences (color assignments, grid display) are used.

User preferences are stored in the Capture.ini file (not in the design database). Changes to user preferences take effect immediately.

Theme

Use the **Options > Preferences** to set your preferred theme. Themes can be set to Light, dark, or Legacy.
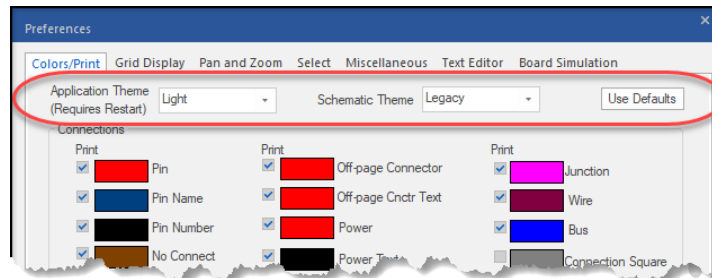
Below are examples of the dark theme and the light theme.
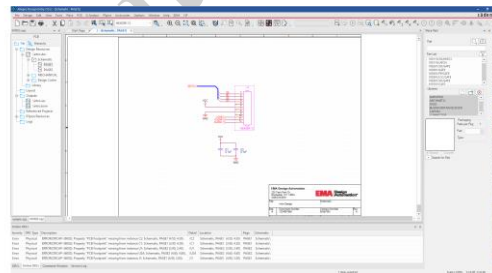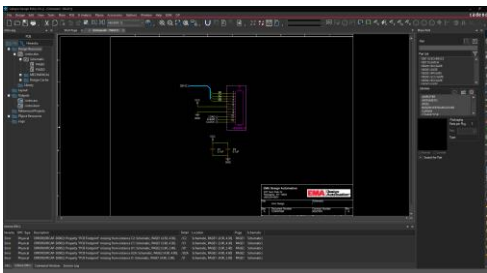
Color Assignments

Use the *Colors/Print* tab to assign colors to design objects and to control visibility of design objects on hard copy prints.
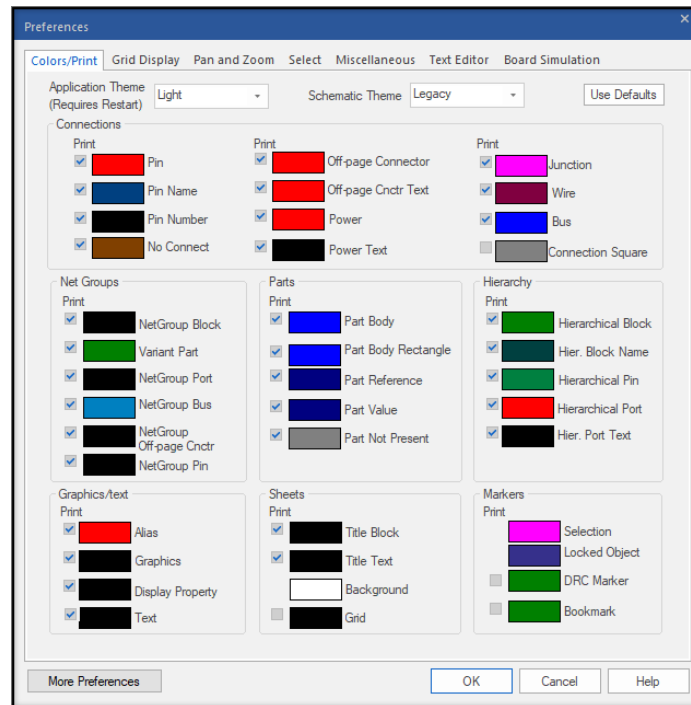
Variant Settings (Used in OrCAD Capture CIS only)

The *Default for Variant* color setting is only used by OrCAD Capture CIS to display on the schematic page any part(s) that are marked in the Variant BOM section (green) and which parts are marked with a "Do not stuff" property (gray).

Grid Display

Use the *Grid Display* tab to turn the grid on or off, or to specify grid style.

OrCAD Capture uses a 100-mil grid. The *Pointer Snap to Grid* option forces all parts, wires and text to snap to this grid. A *Snap to Grid* icon is also available on the main toolbar.

When you are adding parts and wires, the *Pointer Snap to Grid* option should be on (the default). You should turn this option off when adjusting the location of text or properties only.

The *View > Grid* command can also be used to toggle the grid on or off. You can set preferences to display either a dot or line grid system. You can also specify whether you want placed objects to automatically snap to the grid.
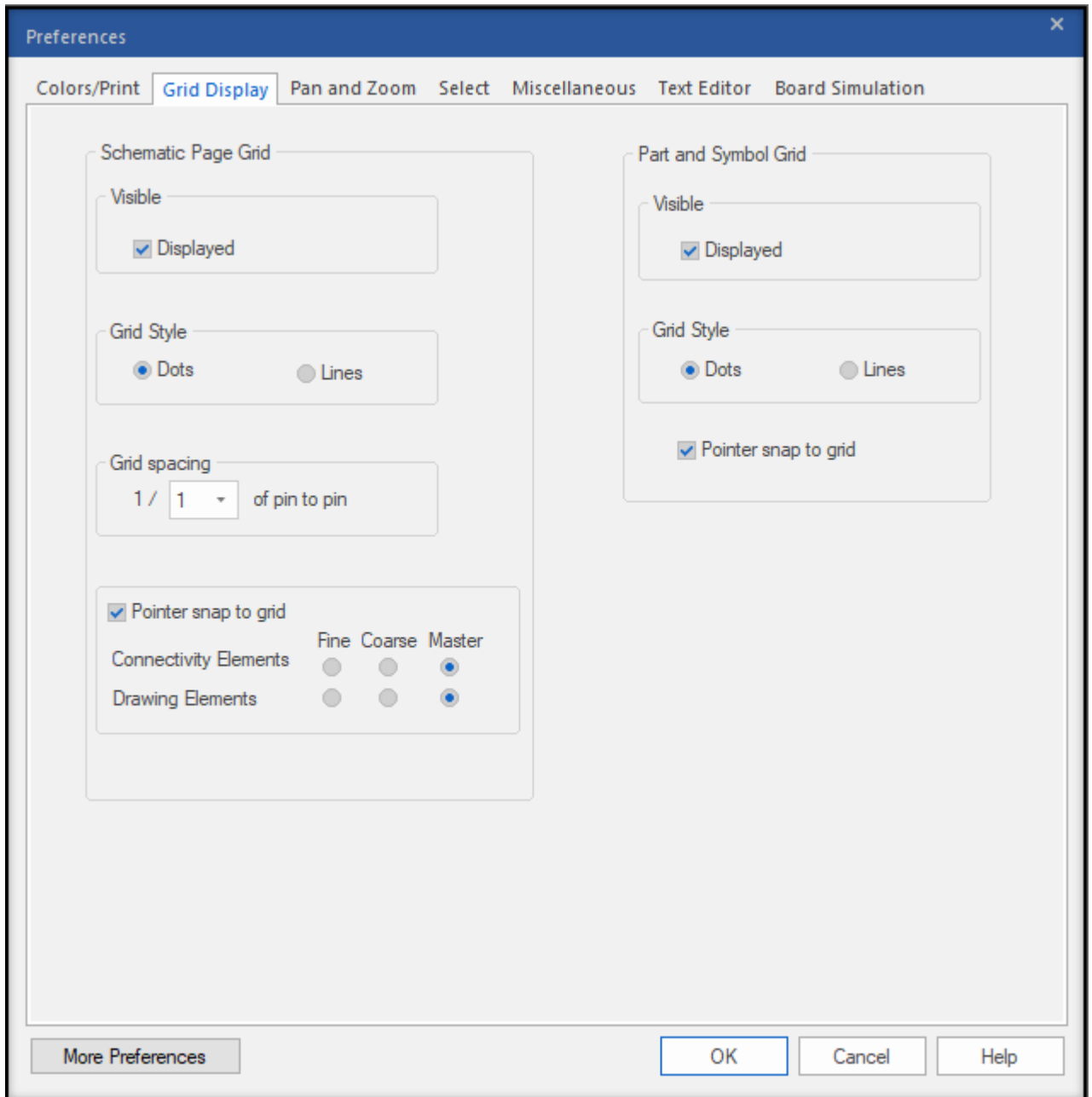
It is recommended that you set *Fine* for Drawing Elements such as text, lines, rectangles, etc. This will allow movement of objects at almost a "gridless" setting. Set *Coarse* for Connectivity Elements such as parts and wires to allow these objects to move using the grid spacing setting. The setting for *Master* will let both object types behave in the default manner.

Pan and Zoom

When you use the Zoom In or Zoom Out icons in the main toolbar, OrCAD Capture refers to this setup menu to determine how far to zoom in or out. Similarly, the amount of zoom performed by the *View > Zoom* menu options or the *<I> (Zoom In)* and *<O> (Zoom Out)* keyboard shortcuts is also defined here.

The *Auto Scroll Percent* option controls how fast the schematic page scrolls when you drag an object into the border area of the schematic window. It is an old feature has been replaced by the action of selecting the <C> key, holding it down, and moving the mouse.

The zoom factor must be an integer between 2 and 10 (no fractions or decimals).
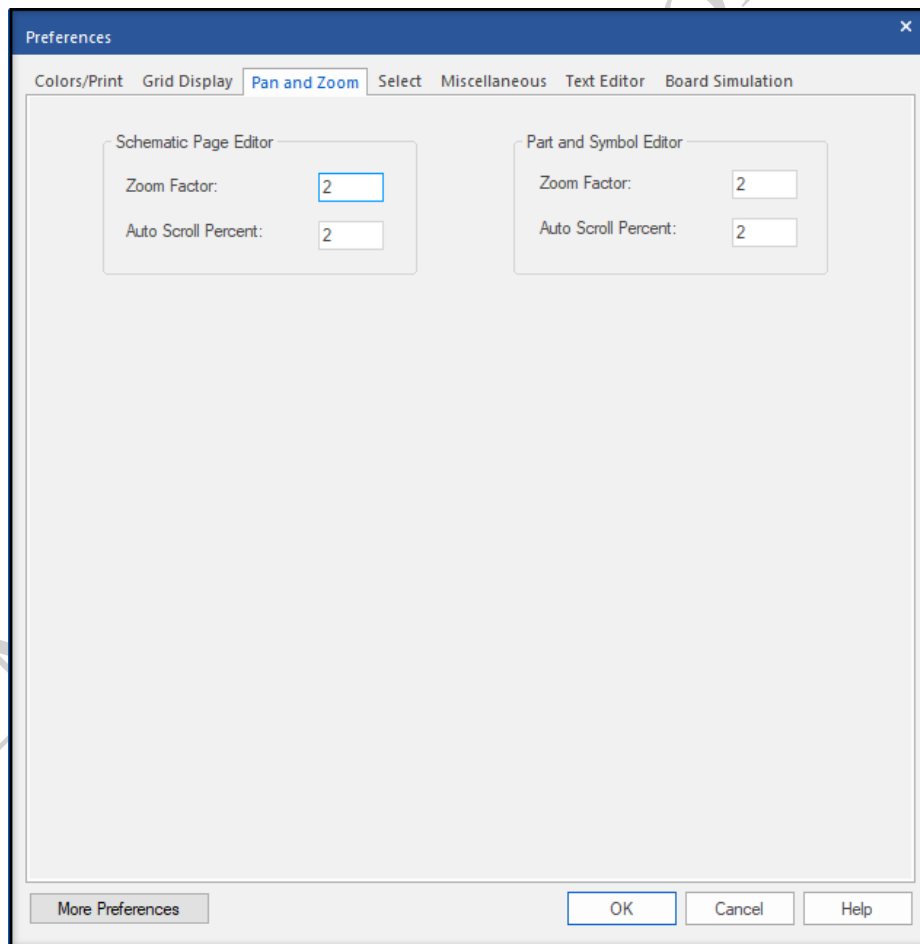
The new function of the *<C>* key replaces the use of the Scroll settings.

Area Selection

To select multiple objects, you can use your left mouse button to drag or draw a rectangle around the objects you want to select. All objects intersected by the rectangle are selected by default. However, if you prefer you can alter the settings in the *Options > Preferences > Select window* to select only objects fully enclosed.

The graphic that follows shows a selection rectangle and objects selected when the *Intersection object* is activated as a preference as well as objects selected when the *Fully > enclosed* option is activated. Both of these settings now have an icon on the main tool bar to allow you to switch between the settings without the need to set them here. (Refer to Lesson 1and page 2-11 of this lesson.)

Selection Preferences

When you drag a rectangle to select multiple objects, all objects intersected by the rectangle are selected by default. As introduced in the previous sub-topic, you can modify preference settings so that only objects fully enclosed within the rectangle are selected.

Use the *Select* tab to set your area selection preference. After selecting multiple objects within an area, you may plan to move them. Use the options on this tab to specify the maximum number of objects that are visible at high resolution while moving the selected group. When you drag a number of objects greater than this value, a simple box replaces the symbol graphic for each part.
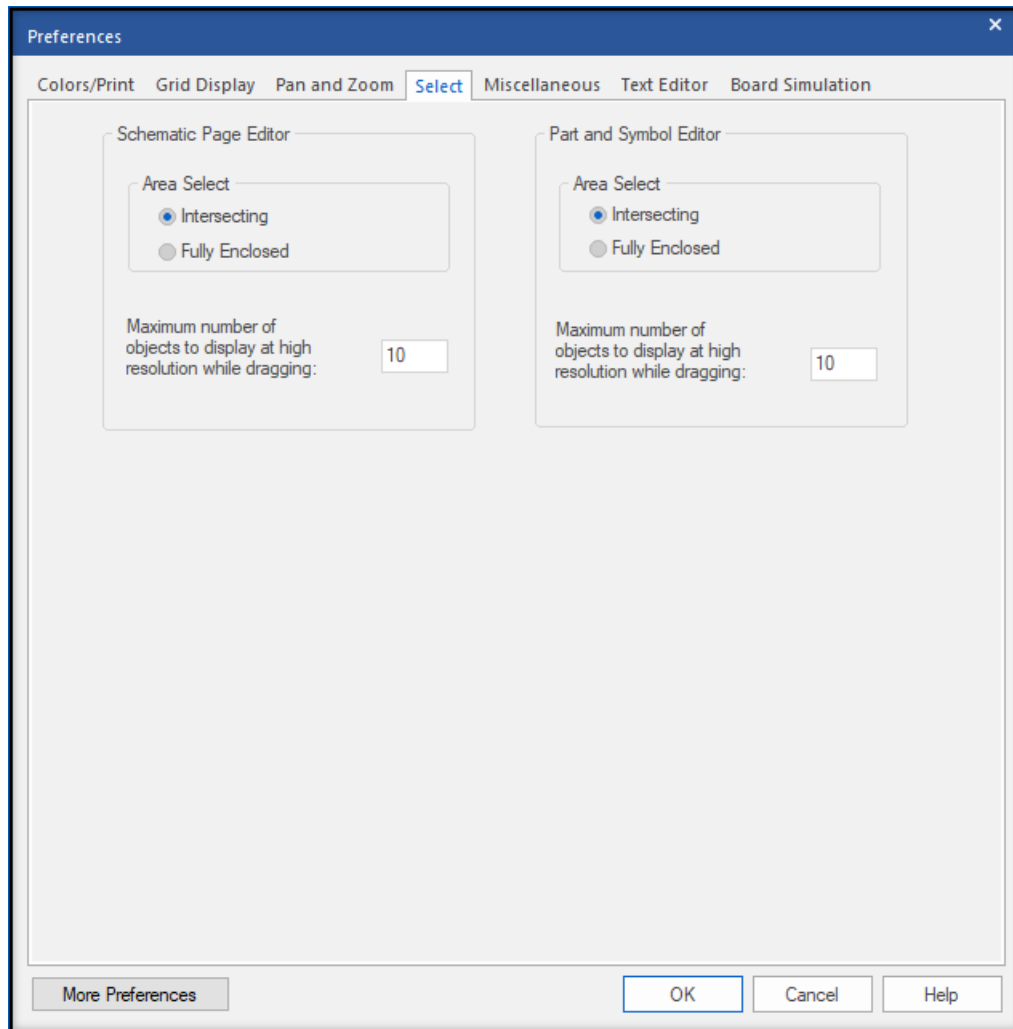
The **"*Maximum Number of Objects to display."*** message is an older setting that, with the advent of the high resolution graphics cards, is no longer a setting worth using. Simply leave it set as is.

Miscellaneous Preferences
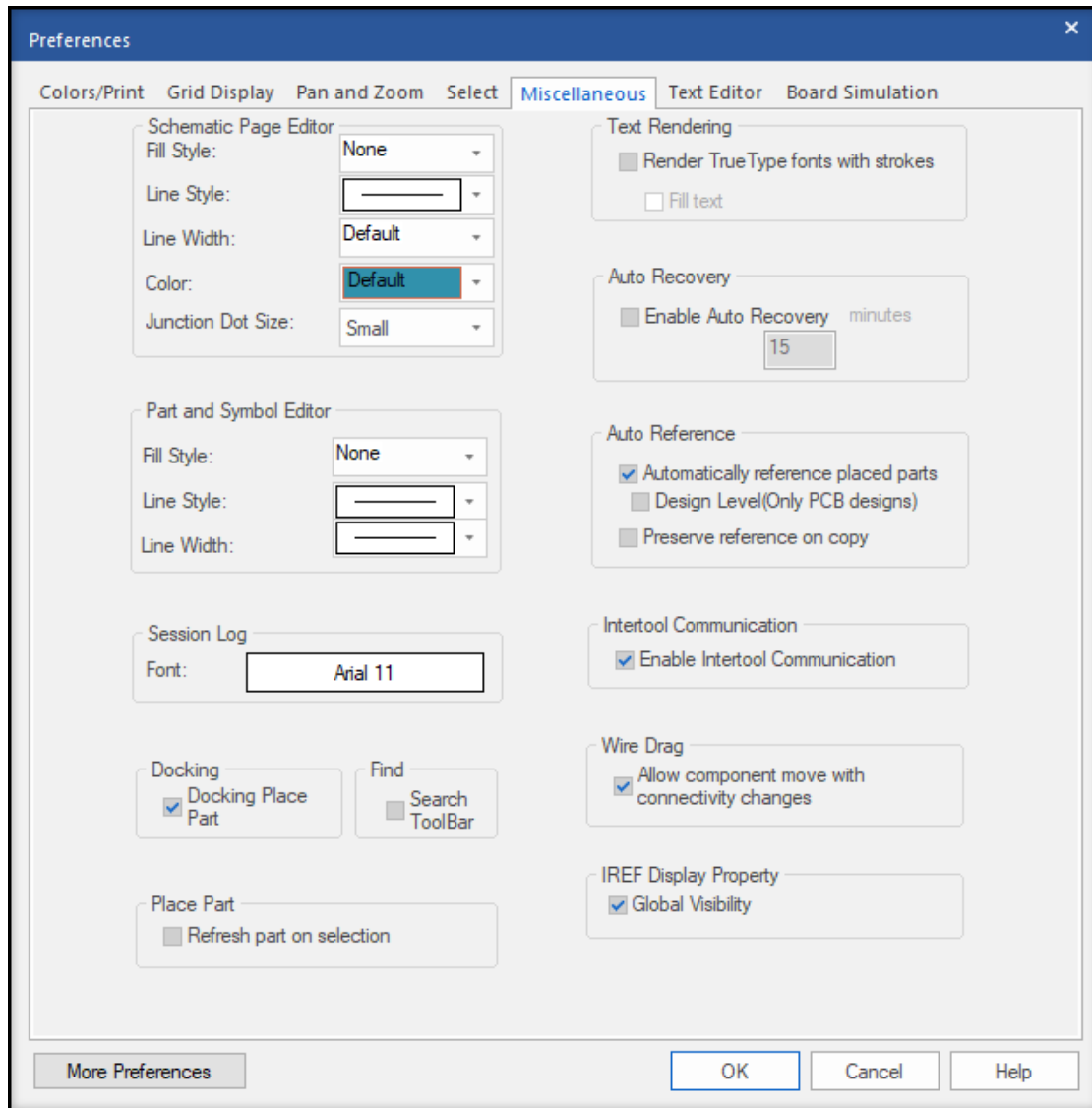
Use the *Miscellaneous* tab to define the characteristics of non-electrical graphics such as polygons and lines. You can also choose to render text as small line strokes rather than text fonts.

Schematic Page Editor

Set the default line draw mode for the schematic page by selecting *Schematic Page Editor > Line Fill* and *Line Style and Width*.

Junction Dot Size will let you set the size of the Junction Dot to *Small* (Default), *Medium, Large*, and *Extra Large*.

Docking will allow you to open and dock the *Place Part* command box and leave it open as you find and place parts on the schematic page.

Enabling the Search Toolbar

Find will allow you to open and dock the find command or *Search Toolbar*.

Auto Recovery

The *Auto Recovery* option protects you from loss of work due to system crash or power failure. OrCAD Capture automatically saves design changes at the end of each Auto Recovery interval (in minutes). These backup files are saved to the "current working directory", and are automatically deleted when you exit normally. If no changes have occurred since the last save, no auto recovery (backup) is performed.

Auto Referencing

The *Auto Reference* option will automatically give you the next reference designator in a sequence. If you wish to use the *Annotate* command, do not use this function.

Enable part references to be preserved while pasting a part to a schematic page. When you copy a part and paste it on a schematic page, the part will retain the same reference designator as that of the copied part. But, if you place a new part on a schematic page, OrCAD Capture will assign the reference designator found in the library. For example: U?A or J?P.

Intertool Communication

The *Intertool Communication* option lets OrCAD Capture interact with the OrCAD and Allegro PCB Editor tools. For example, you can cross-probe parts and nets in the schematic and correspond them to parts and nets in the PCB design.

Wire Drag

The *Wire Drag* option (ON) allows you to drag and place the selected part or wire on the schematic, even if it results in connectivity changes. Also, OrCAD Capture flags a warning with a changed cursor and will show the temporary markers.

(OFF) then the selected part or wire attaches to the cursor and does not get placed on the schematic, if it results in connectivity changes. Also, OrCAD Capture flags only a warning with a changed cursor and does not show the temporary markers.

IREF Display Property

The *IREF Display Property* setting will display the Intersheet Reference text after it is generated.

Extended Preferences

Clicking on More Preferences (located at the bottom of the Preferences window) will launch a subsequent window where you can select from the list provided to set specific preferences. For example, in the sample below, if you select Schematic you can choose from the options provided.
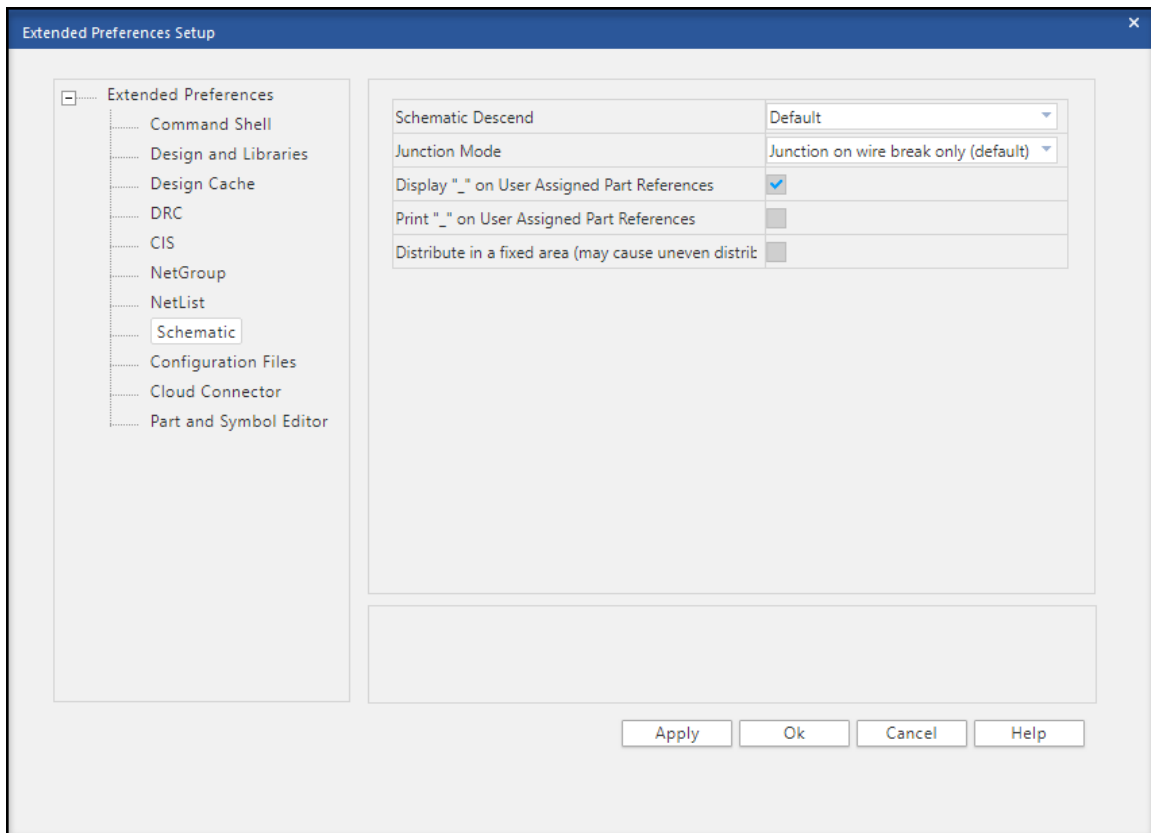
Text Editor

The **Text Editor** Tab is used to set up color, font, and highlighting for Verilog or VHDL keywords, comments, and quoted strings. These settings are applied to a text editor window that is used to create behavioral models. The **Tab Setting** option in the Text Editor tab can also be used to align the columns or data in the Bill or Materials report.
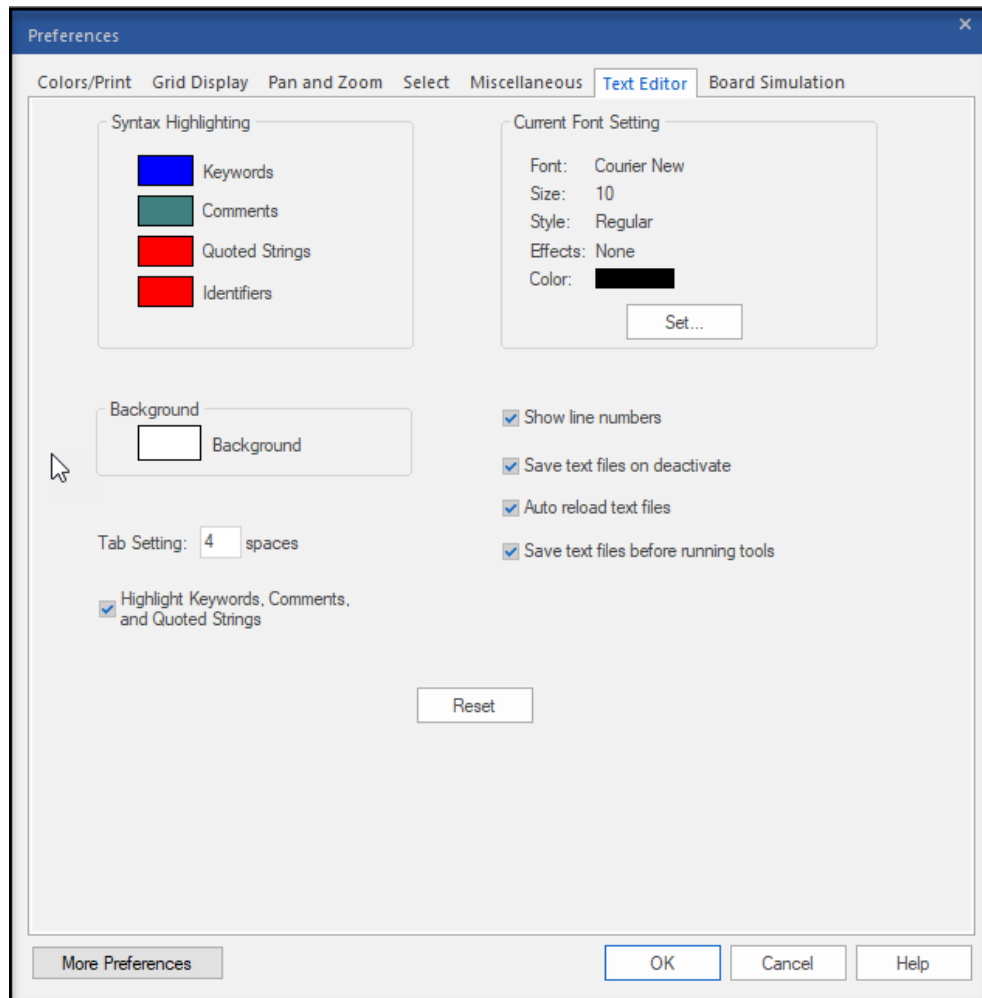
Board Simulation

The *Board Simulation* tab is used to specify which hardware description language will be used for simulation.

Preferences

Colors/Print   Grid Display   Pan and Zoom   Select   Miscellaneous   Text Editor   Board Simulation

Simulation flow

○ Verilog flow   ● VHDL flow

More Preferences   OK   Cancel   Help
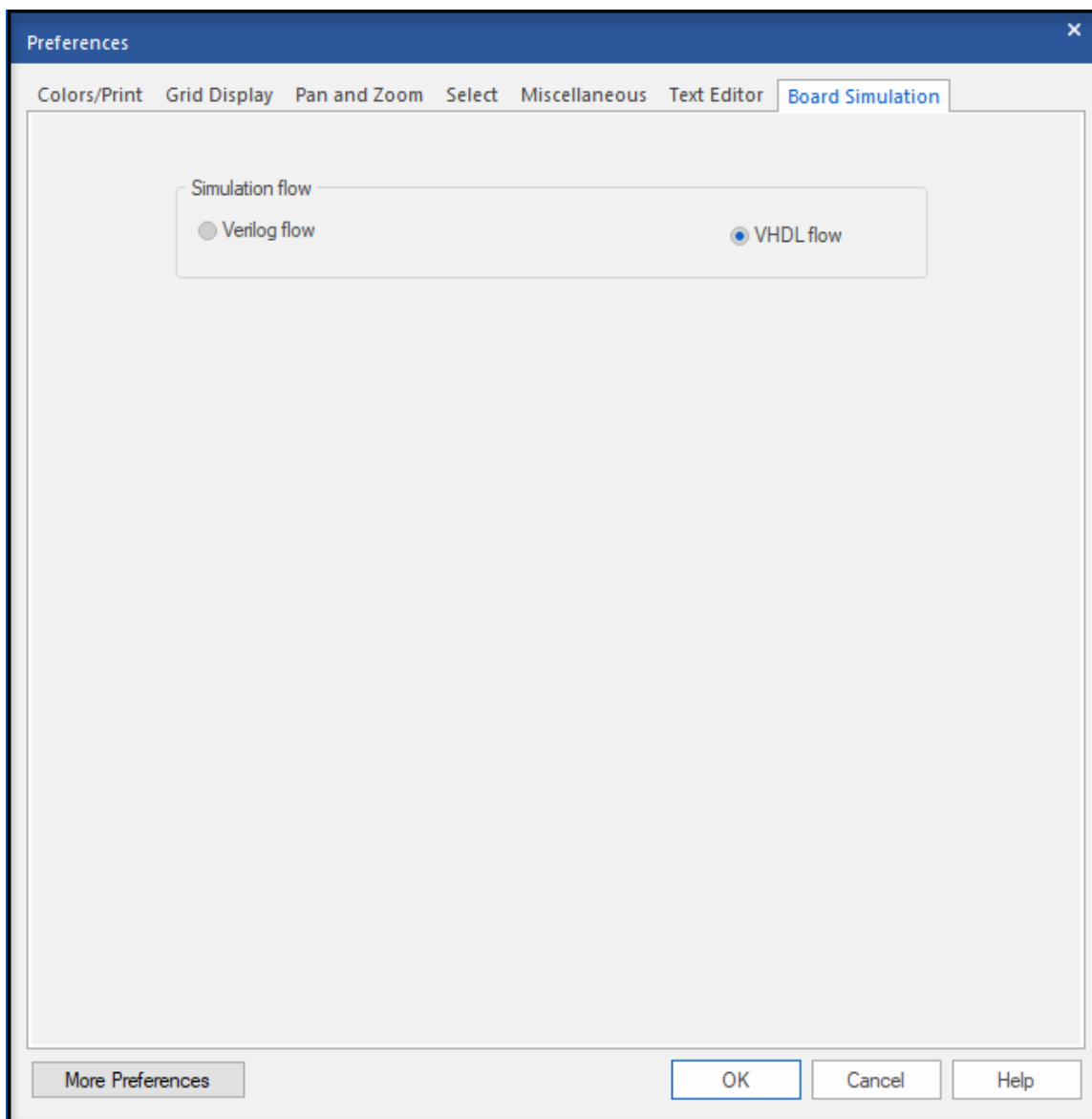
Lab 2-1: Setting up Preferences

Lab Objectives

After completing this lab you will be able to:

- Define a theme
- Set color assignments
- Set the grid display
- Pan and zoom
- Select an area
- Set selection preferences
- Set miscellaneous preferences
- Explore extended preferences

Choosing Theme

Capture can be used in legacy light theme or the newly introduced dark theme.

1. In the main toolbar choose **Options > Preferences**

2. In the top of the Preferences window you will see application theme; here you can choose which theme you would like to continue the class in.
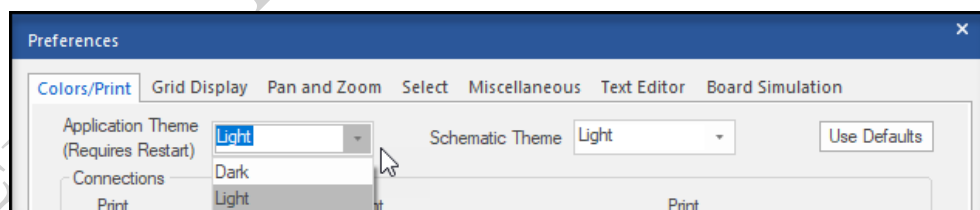
*Note you must restart your capture program to change the theme

On the next page examples of the dark and light theme are shown.
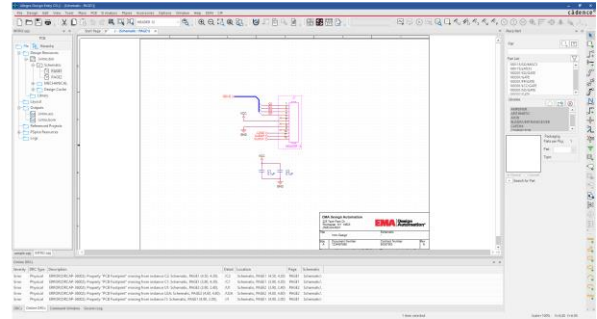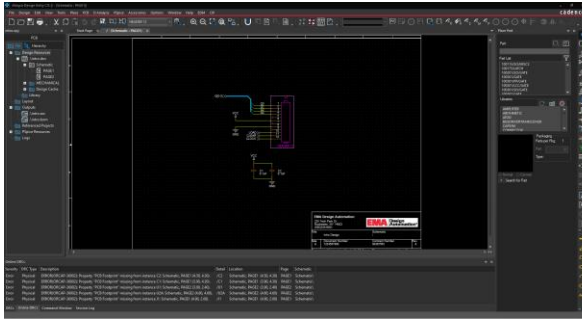
Assigning Colors

If you closed the design in the last lab, reopen intro.dsn.

1. In the Project Manager window of the, double click **PAGE1**.
2. Enlarge the schematic window and zoom in to capacitors **C1** and **C2**.

   Observe the current color settings.

3. Select *Options > Preferences*
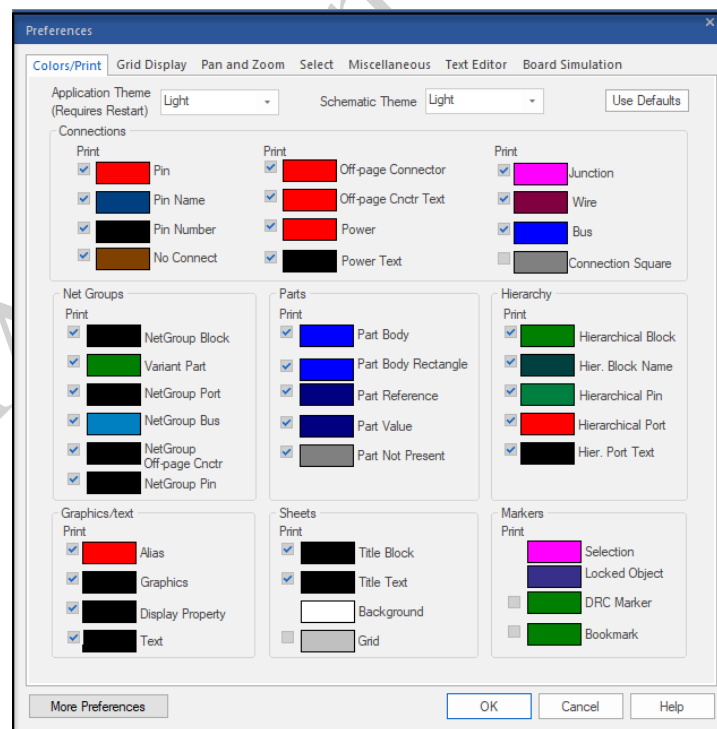
4. The *Colors/Print* tab is the first to be displayed.
5. Click the color box next to *Part Body*, click a different color from the color palette, and then click *OK*.
6. Next change the color assignments for *Part Reference*, *Part Value*, and *Wire* also.
7. Change the *Grid* color to black so you can see it easily on the screen.
8. Click *OK*.

The color assignments take effect immediately. These preferences are machine specific (any design opened on your system will use these color settings).

If your grid is not visible, use the *View > Grid* command to turn it on.

Using Default Colors (Optional)

1. Select *Options > Preferences*.
2. Click the **Use Defaults** button in the lower right corner of the Colors/Print tab and click *OK*.
3. The schematic displays the original color assignments.
4. Try several more color assignments (for example, *Alias*, *Background*, *Junction*, *No Connect*, *Pin*, *Power*, *Power Text*, and *Title Block*).

Assigning colors to some of these objects helps you prepare for the design entry tasks covered in the next lesson.

The Select Tab

1. Select Options > Preferences again.
2. Click the *Select* tab.
3. Change the Area Select setting for the Schematic Page Editor to *Fully Enclosed* and click *OK*.
4. Return to the Project manager window and open Page2 of the design.
5. Zoom in to U2A (74LS00).
6. Drag a selection rectangle across (but do not fully enclose) U2A.
7. Observe that the part is not selected.
8. Drag a larger selection rectangle and fully enclose U2A (including pin stubs).
9. Observe that the part is now selected.
10. Click left in a blank area to deselect all objects.
11. Select *Options > Preferences* and click the *Select* tab.
12. Change the Area Select setting back to *Intersecting* and click *OK*.

13. Drag a rectangle across a portion of U2A to select it.

One Icon on the OrCAD Capture main tool bar also performs the same task without having to go to the User Preferences.
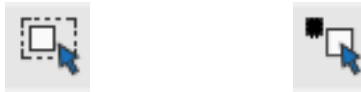
Fully Enclosed Intersecting

Miscellaneous Tab

1. Select *Options > Preferences*.
2. Click the *Miscellaneous* tab and do the following settings.

    a.   For the *Schematic Page Editor* set the *Line Style* to dashed.

    b.   Set *Auto Recovery* on and for 15 minutes.

    c.   Remove the "check" for *Auto Reference*, for now. This will enable us to use the *Annotate* command later in the labs.

    d.   Make sure there is a "check" in the *Wire Drag* option.

3. Make sure all your settings reflect what is displayed in the following image.
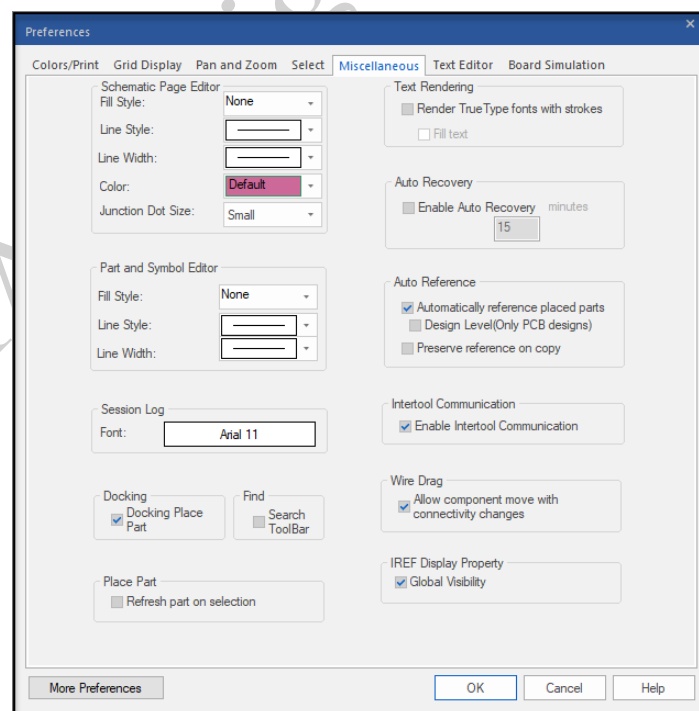
4. Click *OK*.

Exploring Extended Preferences

1. At the bottom of the Preferences window, select the More Preferences button.
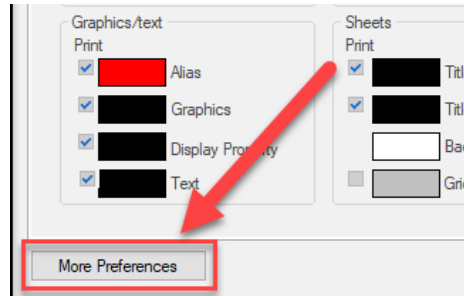
2. When the Extended Preferences Setup window opens, explore the options by selecting an item in the left pane, then observe the options that are available on the right. See the example below.
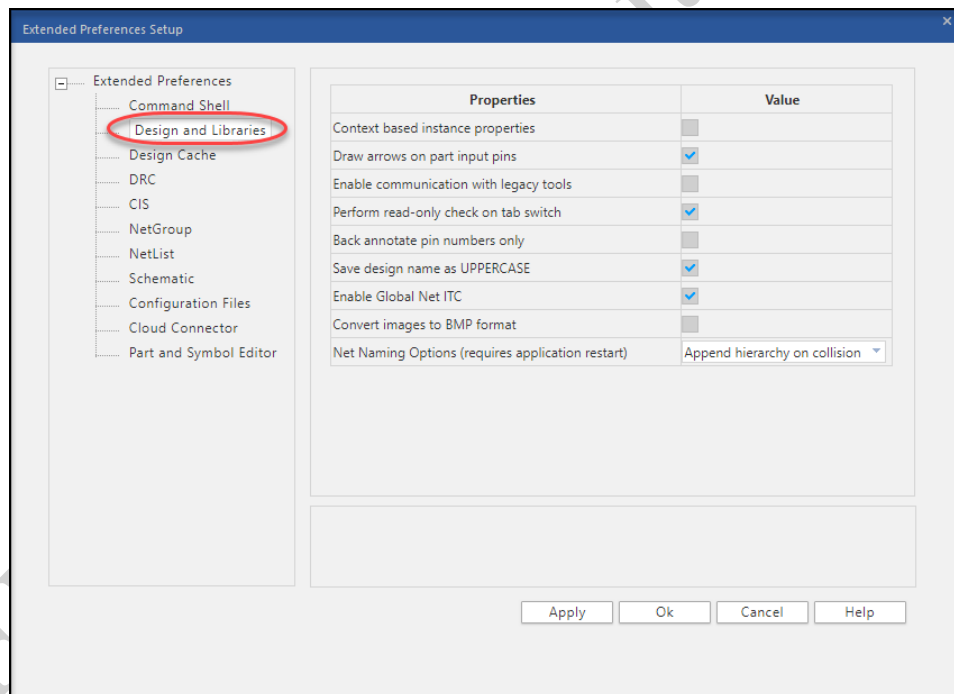
3. Close the window when you are finished browsing the options.

Set AutoBackup Preferences

1. Select *Options > Autobackup*.
2. When the "Multi-level Backup settings" window opens, select the Browse button.
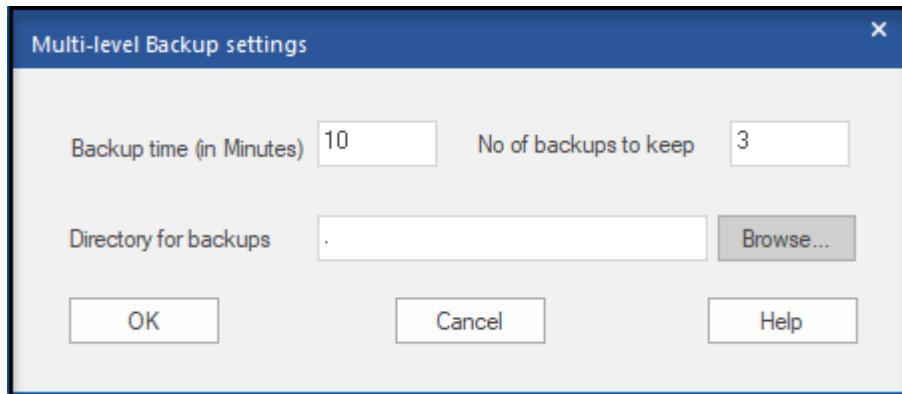
3. Find the C:\EMA_Training\Capture_Essentials\Designs*.*
4. Using the *Create Dir* button, enter the directory name for *Backup.*
5. In the Select Directory window, make sure you select the Backup directory under C:\EMA_Training\Capture_Essentials\Designs*.*
6. Select *OK*.
7. Enter the time interval you wish backup to occur and the number of backups you wish to save. Click *OK*.

Saving Copy of Project

1. Select *File > Close*.

   The schematic window closes.

2. Select *File > Save Project As*.
3. The *Save Project As* dialog will appear. With the Home tab active, enter the directory location.
4. Enter the Project Name and click OK.

The Project Manager window closes. The main session window is still running.

The Design Template
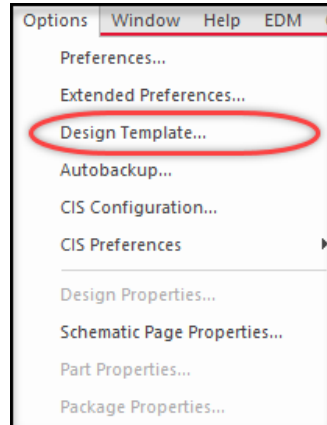
The design template is a collection of settings that are used whenever you create a new design. These settings are design specific and are retained even when the design is transferred to another workstation.

When you define settings of your design template, you define the following:

- Font type and size
- Title block
- Page size
- Grid settings

Design template settings are stored in the Capture.ini file. When you create a new design, the current settings are copied into the design file. When you change the design template, existing designs are not affected.

You can override text font settings at the design level using the
Options > Design Properties menu item, or override page size and border settings using the *Options > Schematic Page Properties* command.

Fonts Tab
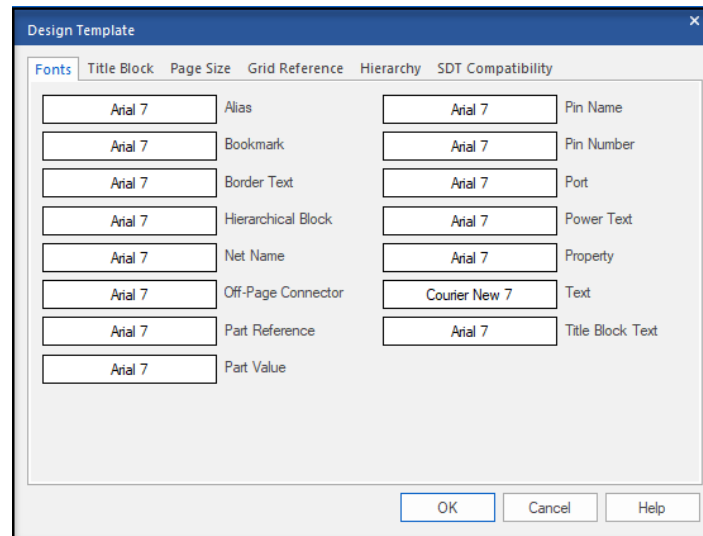
You can use the **Fonts** tab to control the appearance of various kinds of schematic text (for example, net names, reference designators, properties, and so on).

Title Block Tab
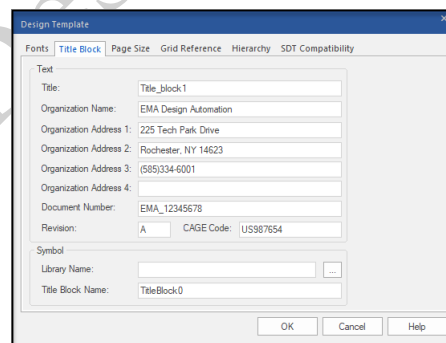
You can use the **Title Block** tab to specify which title block symbol you want automatically added to each new schematic page. You can also fill in your company name and address.

If you also supply design name and revision information, remember to change these settings before starting a new design.

Page Size Tab
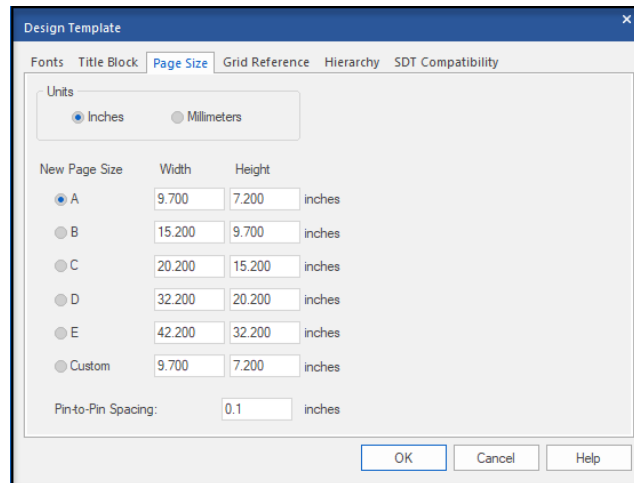
You can use the *Page Size* tab to set the default page size for new schematic pages. Also use this menu to specify the units of measure and the page dimensions.

OrCAD Capture always maintains one grid space between pins. The default grid spacing is 100 mils. The *Pin-to-Pin Spacing* field changes the distance between grid points (which changes the distance between pins).

Do not change the *Pin-to-Pin Spacing* field (always use a 100-mil spacing or "tenth of an inch").

For example, a *Pin-to-Pin Spacing* of 200 mils causes the size of parts to increase by a factor of 2 to 1, because the distance between adjacent pins is 200 mils. Thus, the *Pin-to-Pin Spacing* field scales the schematic.

Grid Reference Tab
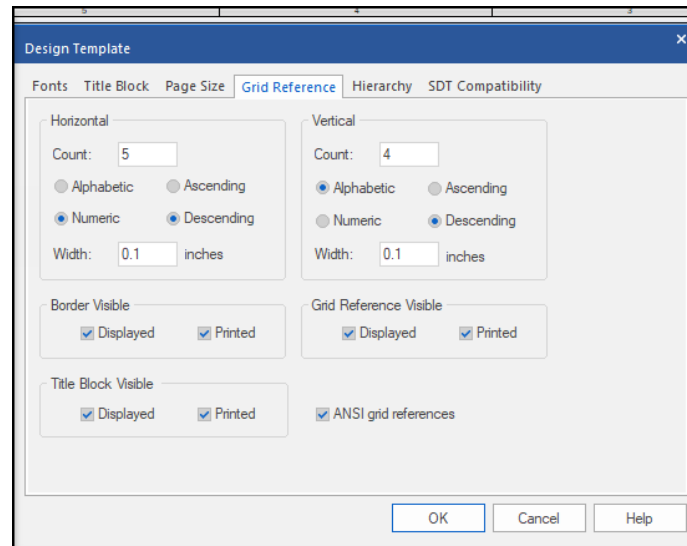
You can use the *Grid Reference* tab to define the grid zones across the top and left edges of schematic pages. Also use this menu to control visibility of the title block.

Hierarchy Tab

This tab lets you set the default setting for Hierarchical block symbols. We will cover more on Hierarchical designs in a later chapter.
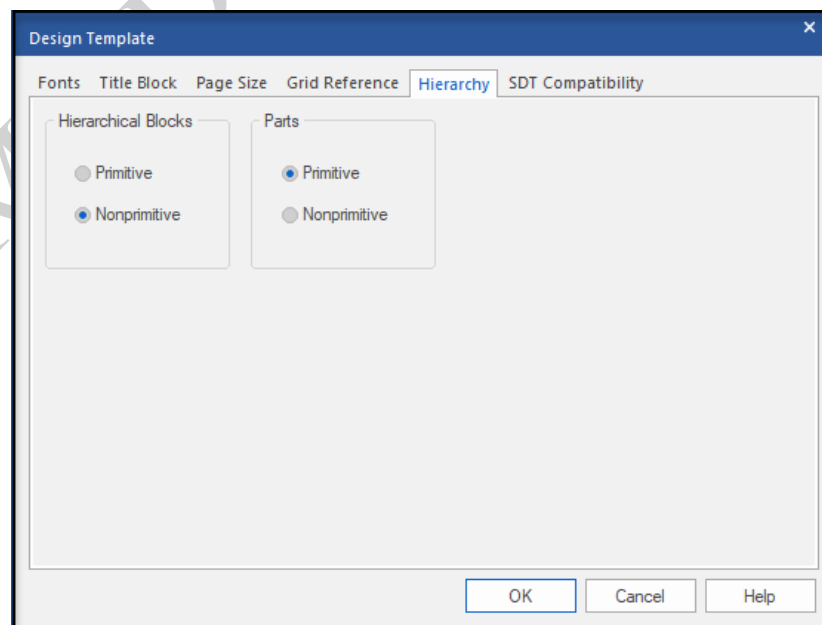
Lab 2-2: Setting Up the Design Template

Lab Objectives

After completing this lab you will be able to:

- Set up default options for your designs
- Add a title into the title block
- Add a name and address to the title
- Add a document number and revision number to the title block

Setting up Default template settings

1. Select *Options > Design Template*.
2. Select the Page Size tab at the top of the window that opens. Make sure the Page Size is set to A for this exercise.
3. Select the Grid Reference tab. The settings should match the following picture.
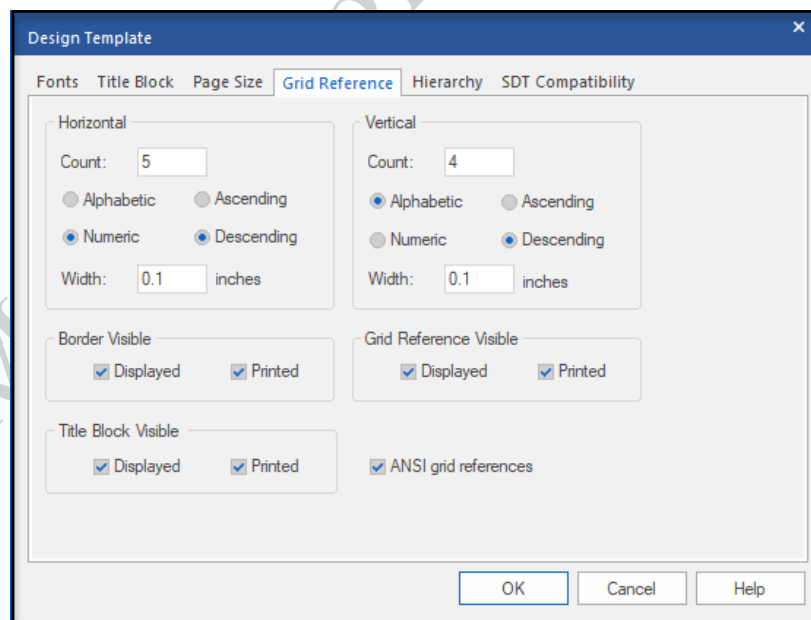
4. Leave all the other tab settings as is except for the *Title Block* tab. See the following instructions for this.

Adding a Title Line to the Title Block
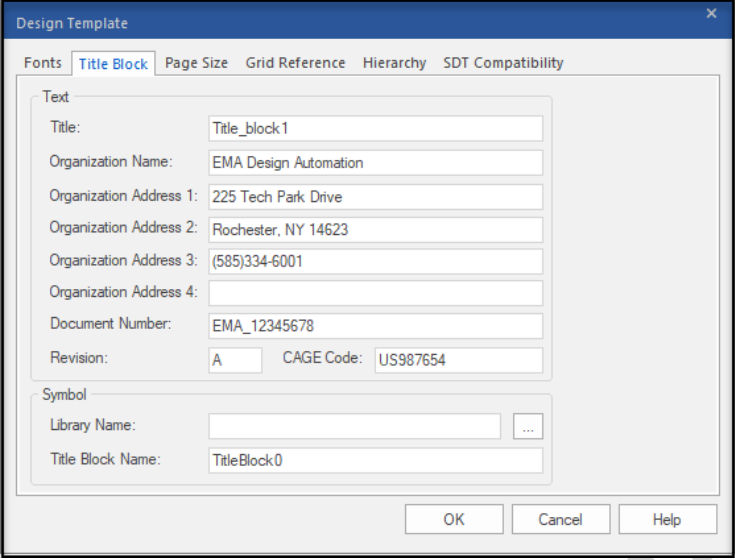
1. Click on the *Title Block* tab.
2. In the *Title* field, enter:

    Lesson6

Adding a Name and Address

1. In the *Organization Name* field, enter your company name.
2. In the *Organization Address1* field, enter your company street address.
3. In the *Organization Address2* field, enter city, state, and zip code.
4. Use the *Address3* and *4* fields ONLY if necessary.

Adding a Document Number/Revision Number

1. In the *Document Number* field, enter:

    EMA-12345678

2. In the *Revision* field, enter:

    A

3. **OPTIONAL** - Add a *Cage Code* IF you need the information on your design pages.

    US987654

4. Click the browser button to the right of the *Library Name* field, and navigate to your training directory, C:\EMA_Training\Capture_Essentials\Libraries.
5. Click on Training_lib.olb and click *Open*.

    The above path may vary, depending upon your training site and where the Cadence software has been installed.

6. In the *Title Block Name* field, enter:

    TitleBlock0

7. Compare your settings to the following example and click *OK*. (The following is an example only.)

This design template will take effect when you create a new schematic in Lesson 6. The default settings for page size, grid reference, and fonts will also be applied.

8. With the Project Manager active choose **File > Save**. Then choose **File > Close** to close the project.

Lesson 3: Creating Part Libraries

Lesson Objectives

After you complete this lesson you will be able to:

- Create a new part library
- Copy from one library to another

Starting a New Part Library

You can use the *File > New > Library* command to create a new Part Library. This creates a library "shell". Once the library is created, use the *Save As* command to rename the default library name and save it to the desired directory.

Location of Library Files

When you create a new library, its location defaults to a directory within the Cadence software hierarchy.

If you use the *File > New > Library* command while a project is open, the new library location defaults to the current project directory. This 'project' library will appear in the Library folder in the Project Manager window, and will be automatically accessible whenever you open the project.

The OrCAD Capture libraries supplied by Cadence are organized by part function (such as OPAmp, Discrete, Gate, and so forth). It is strongly recommended that you organize your new libraries in a similar fashion.

Never create or save your own libraries in the same directory where the OrCAD Capture libraries are loaded. Make sure you keep them in an entirely different location.

Renaming a Library

You can modify the name and/or location of a library by clicking on the library with the right mouse button and then selecting the *Save As* option from the pull-down menu. This will open a new form that will allow you to modify the name and/or location of the library.

As we begin the creation of libraries and designs, remember:

Field Name	Length	Acceptable Characters
Package Name	255	A to Z, 0 to 9, Dash(-) Underscore(_)
Device Type	30	All except ! And '
Function Designator	255	All except ! And '
Reference Designator	30	All except ! And '
Pin Number	30	All except ! And '
Pin Name	255	All except ! And '
Net Name	255	All except ! And '
Property Value	79	All except ! And '
Tolerance	1023	All except ! And '
User Part Number	1023	All except ! And '
Value	1023	All except ! And '

Lab 3-1: Creating a New Library

Lab Objectives

After you complete this lab you will be able to:

- Create a library
- Rename the library

Starting OrCAD Capture

1. If Capture is still open on the start page, move to step 2, named "Creating a Library". If Capture was exited in the last lab, choose ***Start > All Programs > Cadence PCB 17.4-2019 > Capture CIS 17.4***. Optionally you can double-click the Capture icon located on the desktop. Choose Capture from the selection window.

Creating a Library

2. In the main session window, select ***File > New > Library***.

The Project Manager window appears. A new library file is listed under the Design Resources folder.
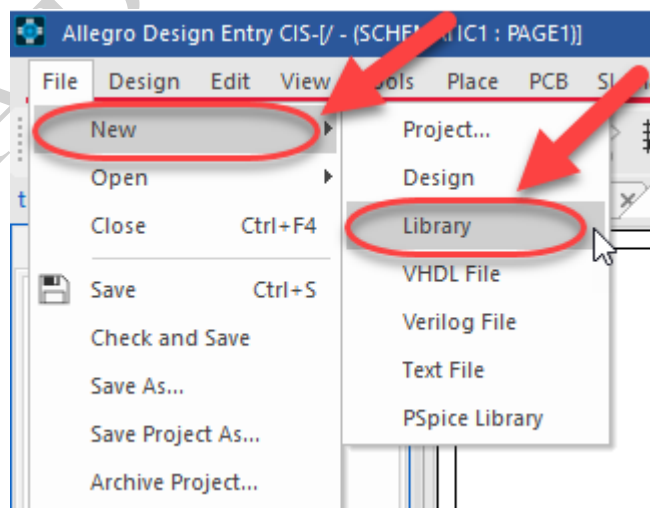
Renaming the Library

1. In the Project Manager window, click on the library file libraryn.olb.
2. (The letter *n* will be the number 1 or greater.)
3. Select *File > Save As*. (Trng.olb). See the figure above on how to save and rename the library.
4. To set the *Save in* field, navigate to the C:\EMA_Training\Capture_Essentials_17.4\Libraries directory.
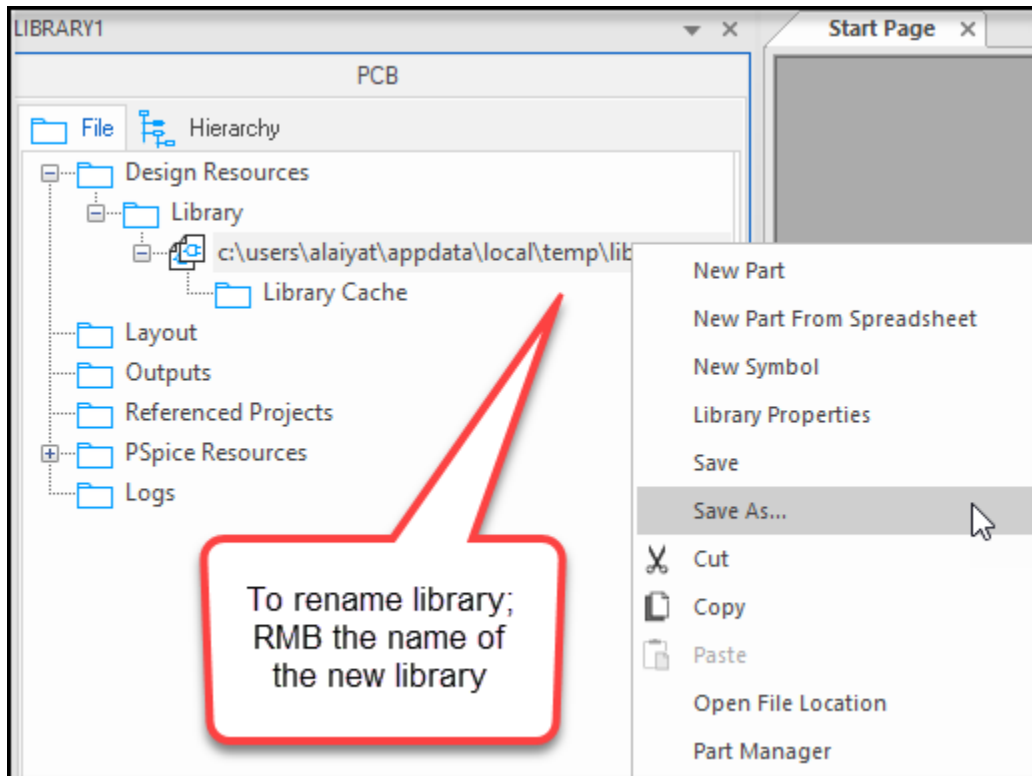
Copying and Renaming Parts and Symbols
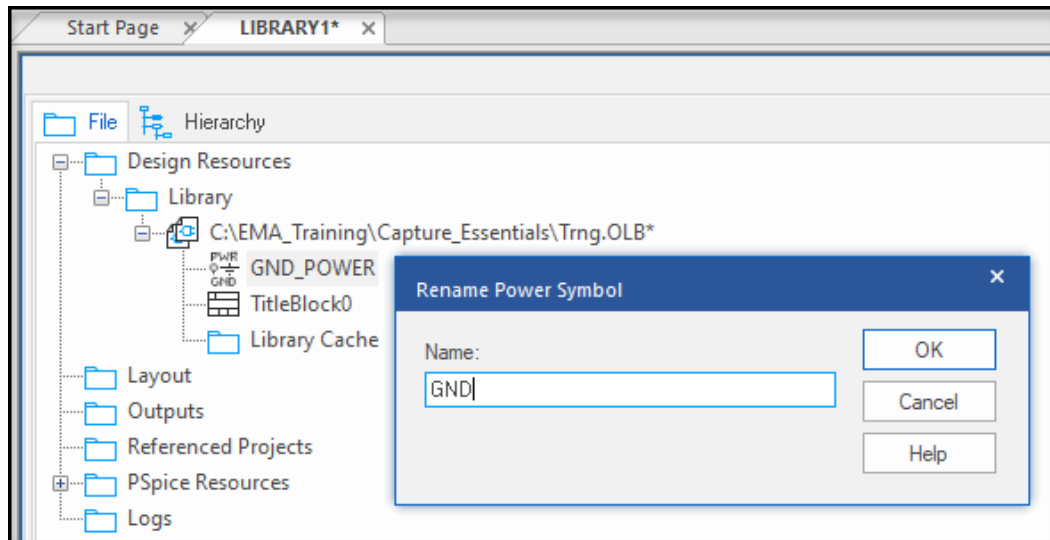
It is common to copy parts from a Cadence/OrCAD-supplied library and modify them for use in your own library.

NOTE: Never modify the default libraries supplied with the OrCAD Capture software. Instead, use them as source libraries.

To copy and paste a part from one library to another, both libraries must be open. Once you have selected the part or parts you want to copy in the source library, be sure to click on the name of the target library before using the *Edit > Paste* command.

You can also open a part in one library, and use the *File > Save As* command to save the part into another library.

To copy and rename a part in the same library, first select the part and use the *Edit > Copy* command. Then use the *Design > Rename* command to change the name of the original part. Next, paste the copied part back into the library. Last, open the renamed part and modify it.

Lab 3-2: Copying and Renaming Parts and Symbols

Lab Objectives

After you complete this lab you will be able to:

- Open an OrCAD Capture supplied libraries
- Copy parts from one library and add them to another

Opening an OrCAD Supplied Library

1. Select *File > Open > Library*.
2. Navigate to the C:\Cadence\SPB_17.4\tools\capture\library directory, as shown below.

The above path may vary with different installation locations of the OrCAD software.
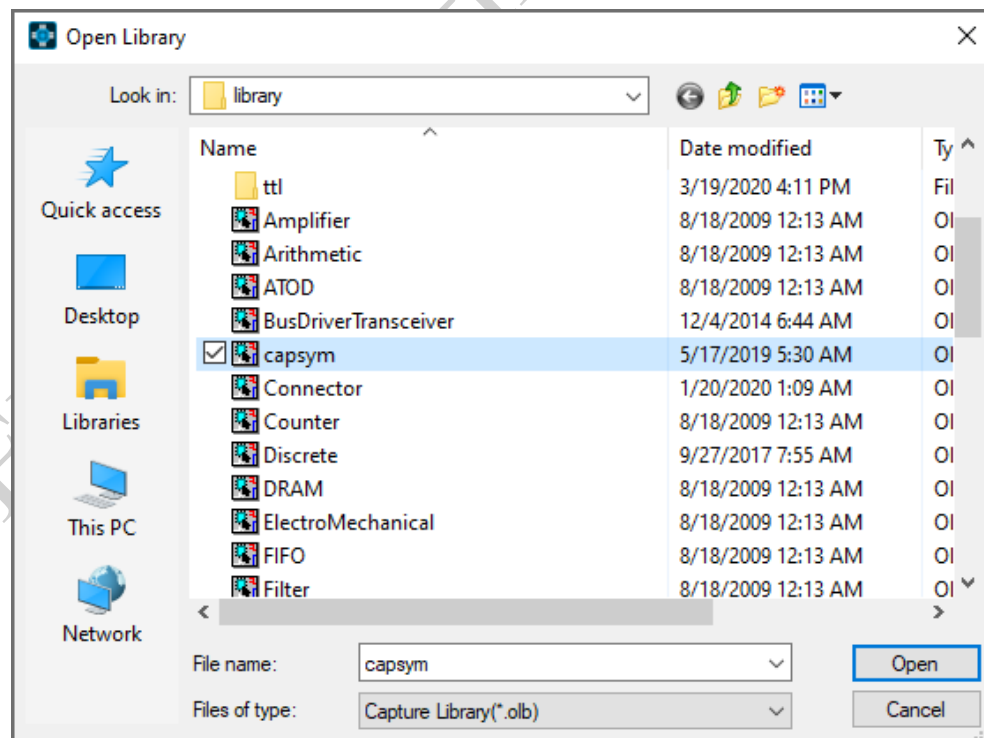
3. Click on capsym.olb and click *Open*.

4. Align the two Project Manager windows side by side.
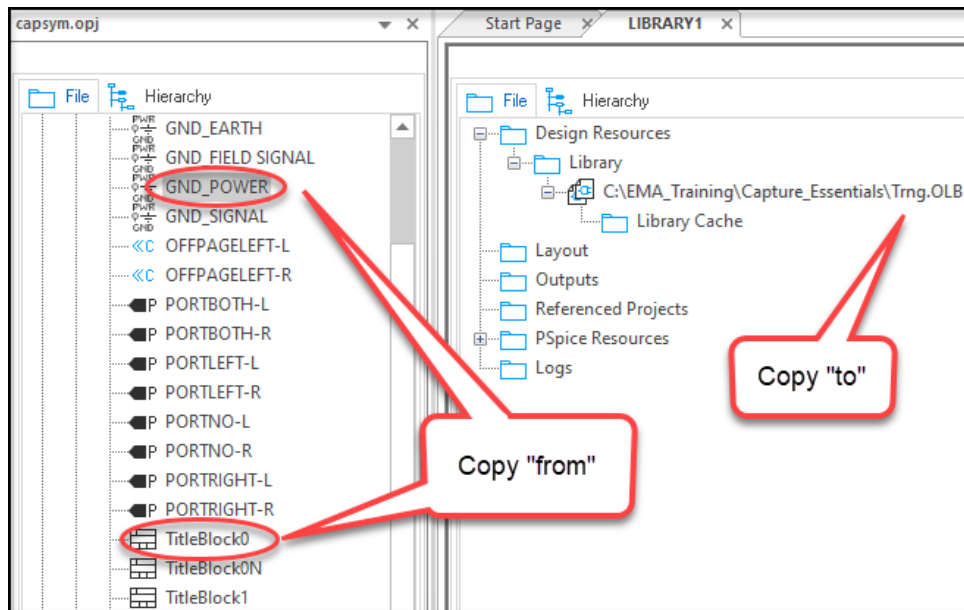
The library opens. You will copy parts from this source library into your TRNG library. When you open a new library it may open in a tab next to your other library. To separate this tab press the arrow in the top left of the project manager and choose Tabbed Document.
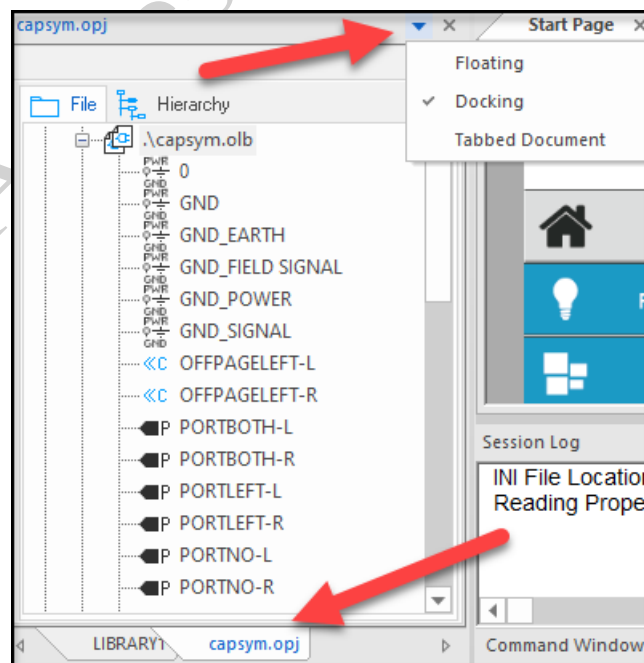
5. In the CAPSYM library window, use *<Ctrl+LMB>* to select both the *GND_POWER* and *TITLEBLOCK0* symbols.
6. Select *Edit > Copy* (or *CTRL+C*).
7. In the other library window, click on TRNG.OLB and select *Edit > Paste* (or *CTRL+V*).
8. The two selected symbols now appear in the TRNG library.
9. Close the CAPSYM library window.
10. In the TRNG library, select the *GND_POWER* symbol and then select *Design > Rename* or LMB and select *Rename* from the pop-up menu in the Library window.

    The *Rename Power Symbol* window appears.
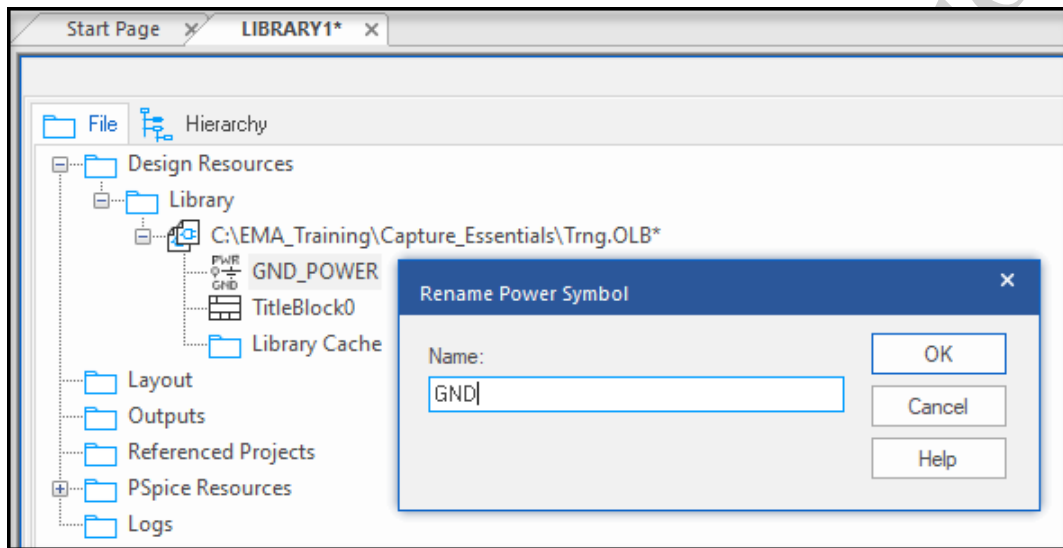
11. Change the symbol name to *GND* and click *OK* to close the form.
12. You no longer need the Capsym.olb library. Close the Capsym.olb

Lesson 4: Creating Homogeneous Parts

Lesson Objectives

After you complete this lesson you will be able to:

- Create a Homogeneous part

Homogeneous Parts

A homogeneous part is a part that has only one part graphic. If you define the part as having two or more parts per package, this means the package holds two or more identical logic parts, each with its own unique physical pin numbers.
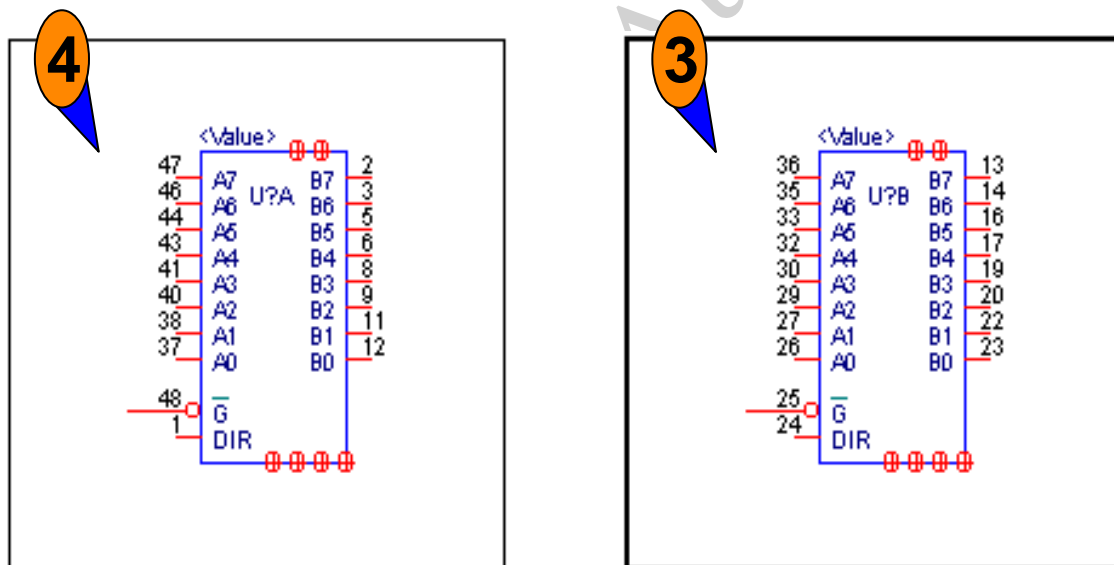
When you create a homogeneous part, you create one part graphic only. If the part graphic represents the entire package (called single-section), then *Parts per Pkg* is set to *1*. If the part graphic represents just a portion of the entire package (called multi-section), then *Parts per Pkg* is set to *2* or more.

What is a Package?
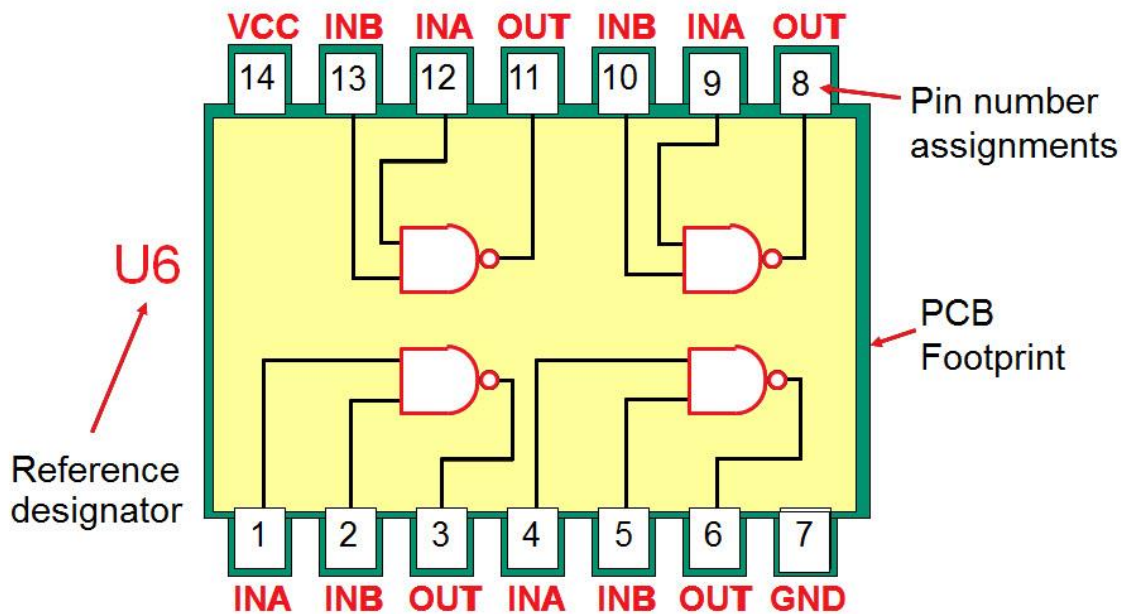
A package is a grouping of one or more circuit elements.

If a symbol represents the entire package, the part is said to be *single- section*. If the symbol represents only a portion of the physical package, the part is called *multi-section* (other terms are *multi-gate* or *multi-slot*).

A part has both logical and packaging related characteristics. For example, the symbol defines the part graphics and pin names. But packaging information is required to define the number of times the symbol fits into a package, and the logical-to-physical pin mapping required to annotate and netlist a schematic design. A package contains power, ground, and no-connect pins that also need to be considered during part building.

Creating a New Part
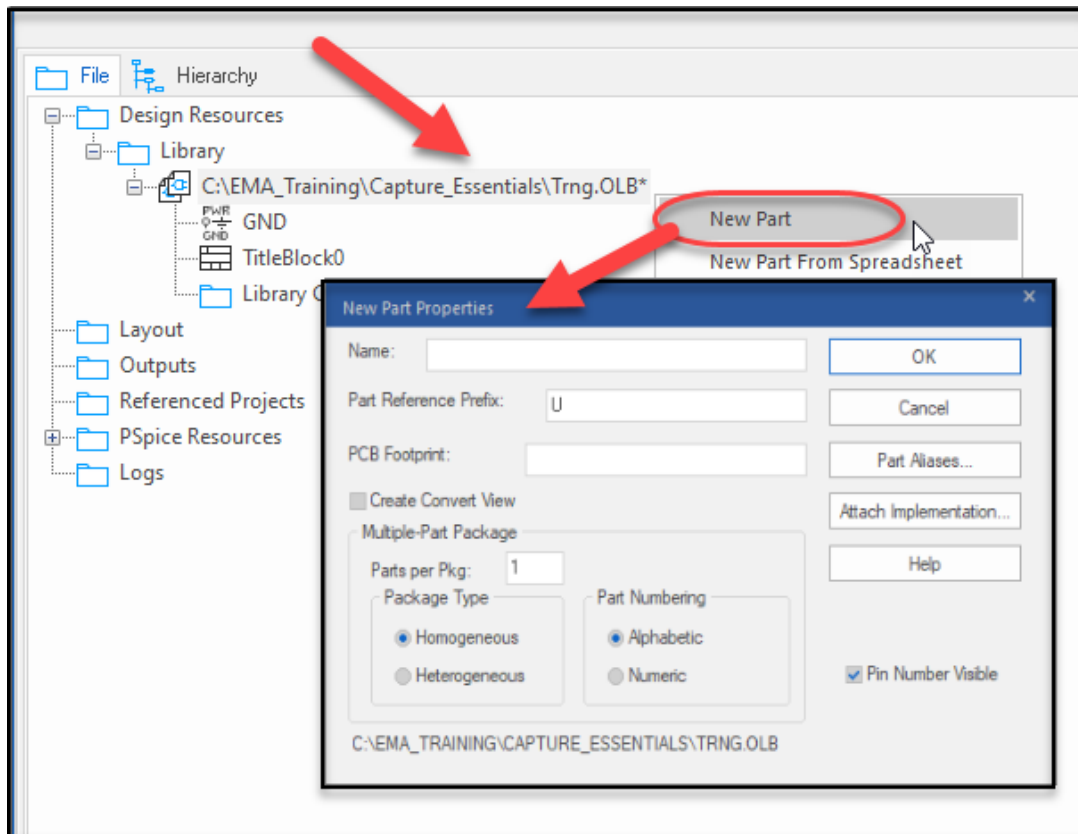
Highlighting the Library name, right click and choose the **New Part** command from the popup menu. Use the **New Part Property** dialog box to create a new part in a current or active library. The table that follows lists and describes each option and field within the New Part Properties dialog box. The following are some of the acceptable characters and character limits.

Pins	Length	Illegal Characters
Pin number	30	! : " ' , ~ * < > space
Pin name	255	Same as above

Dialog Box Option	Use Description
Name	Specifies the part's name. This is used as the default part value when the part is placed on a schematic page. Part names can be up to 31 characters long (for example, R1206, RN55).
Part Reference Prefix	Specifies the first letter of the reference designator ("C" for capacitor or "R" for resistor).
PCB Footprint	Specifies the name of the footprint pattern used during the PCB layout process (for example, SM_1206).
Create Convert View	Controls the creation of an alternate version. You might use the convert option to define a DeMorgan equivalent. A part with this option specified will have two views (a normal and a convert) that you can switch between once the part has been placed in the schematic.
Parts per Pkg	Specifies whether or not there are multiple parts in the package (for example, gates).
Package Type	If the part is a multi-gate package, specifies whether all the parts in the package are symmetrical (homogeneous) or asymmetrical (heterogeneous). This setting cannot be changed once the part has been created.
Part Numbering	If the part is a multiple-part package, specifies whether parts in the package are identified by letter or number. For example: U?A (alphabetic) or U?-1 (numeric).
Part Aliases	Add technology-specific aliases for the current part (such as 5400, 7400, 74LS00 and so on).
Attach Implementation	Specifies a behavioral model for simulation, or an underlying schematic view for a hierarchical block.

Once the Package Type has been set to either Homogeneous or Heterogeneous, it cannot be changed.

To access the *Part Aliases* button, highlight the *trng.olb* and select *Options > Package Properties*. Examples of different technologies are 75F00, 74LS00, 74ACT00, and 74HCT00. Some aliases indicate the use of a military style package versus one for commercial applications such as 5400.

These are parts that can use the same symbol graphics, logical pin names, physical pin mapping, and PCB footprint. Part aliases appear in the Project Manager window with a dashed line in the center of the symbol icon. When a part is copied from one library to another, its aliases are copied also.

The Parts in the OrCAD Capture Libraries were created using Part Aliases. If you require individual information in each part, i.e. Part Number, **do not** use Part Aliases if you have specific data, i.e. Part Numbers, to add to each part. **DO NOT** use Part Aliases if you are using OrCAD Capture CIS.

When you *OK* the New Part Properties dialog box, the OrCAD Capture tool opens the Part Editor.

The Draw toolbar is usually located on the right side of the OrCAD Capture window but can be moved to any desired location. The Part Editor activates only those tools used to create and edit part bodies. Those commands that are not used at this time are grayed out. You can also access the same commands from the *Place* pull-down menu.

The Place Pull-down Menu

Tools	Place	SI Analysis	Accessories	Optic
	Pin...	Shift+G		
	Pin Array...	Shift+J		
	Part...	P		
	PSpice Component...	▶		
	Parameterized Part...			BRARY1
	Wire	W		
	Auto Wire	▶		
	Bus	B		
	Junction	J		
	Bus Entry	E		
	Net Alias...	N		
	Power...	F		
	Ground...	G		
	Off-Page Connector...			
	Hierarchical Block...			
	Hierarchical Port...			
	Hierarchical Pin...			
	No Connect	X		
	IEEE Symbol...	Shift+X		
	Title Block...			
	Bookmark...			
	Text...	T		
	Line	Shift+L		
	Rectangle	Shift+R		
	Ellipse	Shift+F		
	Arc			
	Elliptical Arc	Shift+T		
	Bezier Curve	Shift+Q		
	Polyline	Y		
	Picture...			
	OleObject...			
	NetGroup...	U		

The following table describes each Draw toolbar buttons used in the Part Editor.

Toolbar Icon	Comments
	Selects objects. Default mode.
	Adds an IEEE symbol.
	Place Pin - Adds a single pin to the part.
	Place Pin Array - Adds a pin array to the part with a defined range.
	Place Line - Draws a line.
	Place Polyline - Draws a multiple segment line.
	Place Rectangle - Draws a rectangle. Hold <SHIFT> key to create a square.
	Place Ellipse - Draws an ellipse. Hold *<SHIFT>* key to create a circle.
	Place Arc - Draws an arc (1st edge, 2nd edge, radius).
	Place Elliptical Arc - Draws an Elliptical Arc.
	Place Bezier - Draws a Bezier curve.
	Place Text - Adds comment text. (non-intelligent)

These commands are also located in the *Place* pull-down menu.

The following are the illegal pin characters:

Pins	Length	Illegal Characters
pin number	30	! : " ' , ~ * <> Space
pin name	255	same as above

Adding Graphics
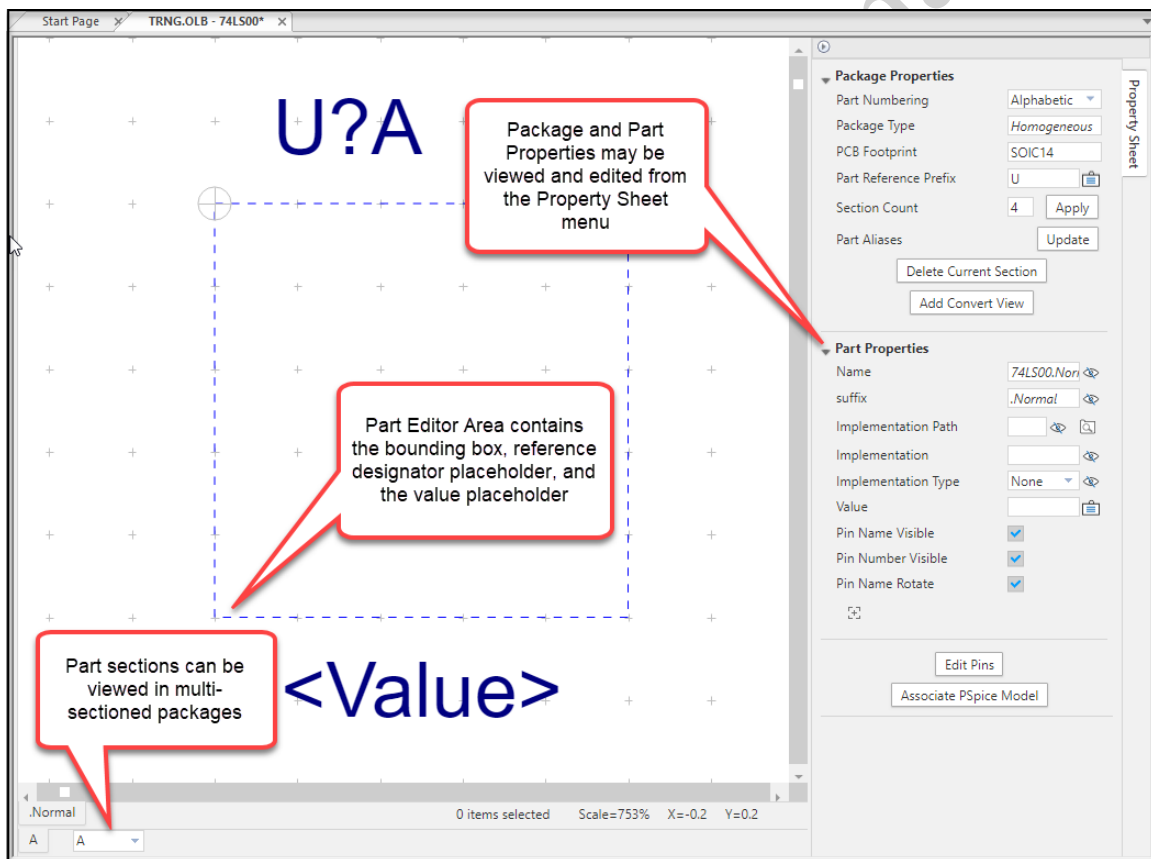
Initially, the Part Editor contains a dotted-line part boundary box and placeholder properties for the reference designator and part value.

Editing Part Graphics

All graphics that make up the part must fit within the dotted-line boundary box (with the exception of pins and pin graphics).

The boundary box also defines the selection area for the part when used in the schematic.

The dotted boundary box automatically enlarges to surround the part graphics you create. If the part graphics are smaller than the boundary box, the box size must be reduced manually.

All part graphics snap to a 100-mil grid in the Part Editor window. If you do not want to use the snap feature, click the **Snap-to-Grid** icon in the main toolbar to create off-grid graphics. This is also helpful when adding or relocating text and properties. DO NOT, however, turn off the **Snap-to-Grid** when placing the pin of the part. They MUST fall on a grid.

Adding Pins
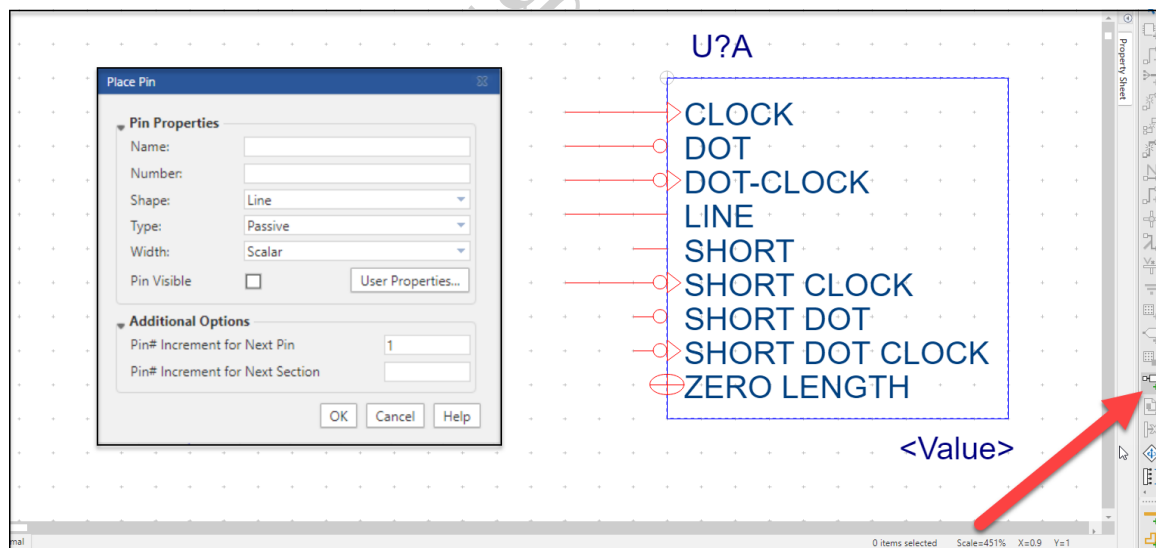
To access the Place Pin dialog box, click the **Place pin** icon in the Part Editor tool palette. Pins must be placed on the boundary box edges only and must be placed on a 100-mil grid.

The table that follows lists and describes each option and field within the *Place Pin* dialog box.

The following are the illegal pin characters:

Pins	Length	Illegal Characters
pin number	30	! : " ' , ~ * <> Space
pin name	255	same as above

Dialog Box Option	Use Description
Name	Specifies the pin name. If the name ends with a digit (0 - 9), each pin is incremented by one every time you place a pin. You can create a pin name with an overbar by adding a back-slash (\) after every letter in the pin name.
	You will get errors if you create duplicate pin names on the same symbol. The only duplicate pin names on a symbol should be power pins (see *Adding Power Pins* later in this lesson).
	By default, all pin names (except power) are visible. Pin name text cannot be moved.
Number	Specifies the pin number. The pin number can be alphanumeric. If it ends in a number, it is incremented by one after each pin is placed.
	If the Pin Number field is left blank, no error occurs. During annotation, no pin number data is added to the schematic. A pin number is required by the OrCAD and Allegro PCB Editor netlister.
	OrCAD Capture will flag duplicate pin number assignments on the same section only. For example, in a multi-section part, you can map a shared enable pin to the same pin number as long as it is in a different section.
	By default, all pin numbers (except power) are visible. Pin number text cannot be moved except in the Edit Part command at the schematic page level.

Width	Specifies whether the pin connects to a bus or a wire. If bus is specified, the pin must connect to a bus; otherwise, it must connect to a wire. Bus pins cannot be used directly as netlisting pins. Their main purpose is to make it possible to use non-primitive parts more easily by connecting large numbers of signals to a child schematic folder.
Shape	Select the pin shape from the list of six pin shapes.
Type	Select the pin type from the list of eight pin types: 3 State, Bidirectional, Input, Open Collector, Open Emitter, Output, Passive, and Power. Be careful to enter the correct pin type, as this will affect design rule checking and simulation. If a part has no voltage pins, then its pins should be defined as Passive.
Pin Visible	Specifies the pin visibility on the schematic page. All pins (except power pins) are visible (the Pin Visible toggle is greyed out). Power pins are invisible by default, but can be visible if you choose. (The Pin Visible toggle is accessible for power pins only.)
User Properties	Displays the User Properties dialog box so you can add and edit the pin's properties.

Pins cannot be added to the corner of the part boundary box. If you need to place a pin on a corner, enlarge the boundary box before adding the pin, and then shrink the box so the pin is on the corner.

Editing a Single Pin
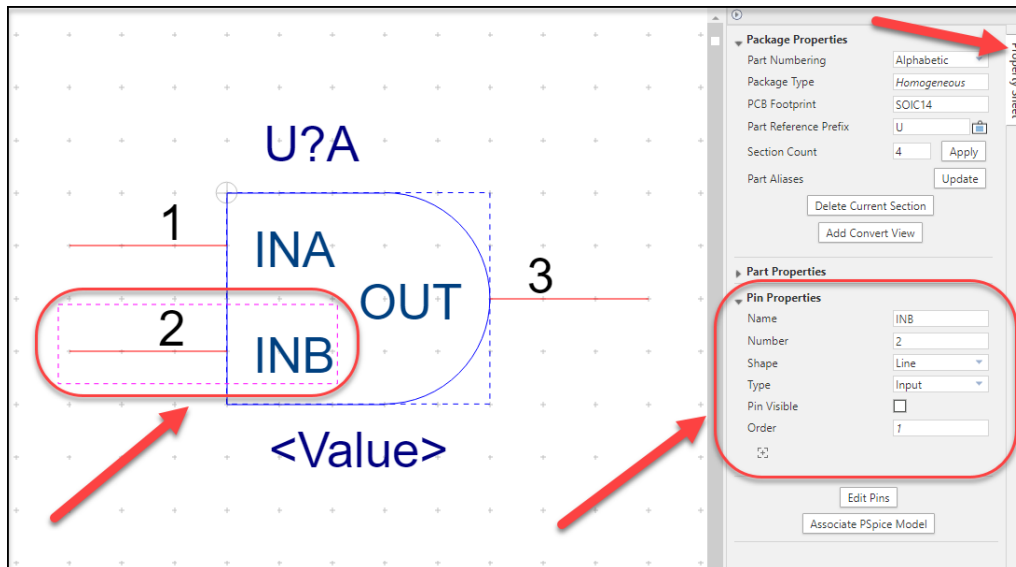
You can edit properties such as the pin name, pin number, shape, or pin type of a single pin from the *Property Sheet* menu after the pin has been selected.

Editing Multiple Pins
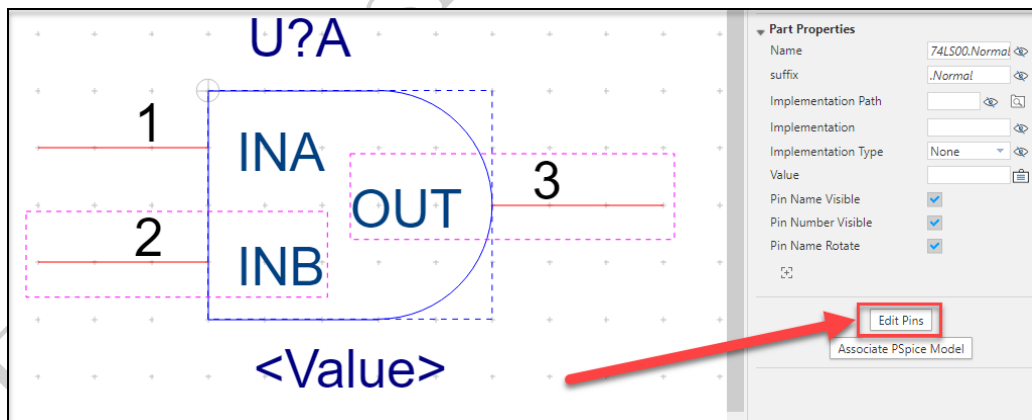
When multiple pins are selected; you can access the **Edit Pins** area from the Property Sheet menu. The **Edit Pins** area enables you change pin name, number, type, or specify pin graphics (for example, clock wedge or inversion dot) for multiple pins.

Editing Pins at Package Level

When no pins are selected, you can access the **Edit Pins** area of the Property Sheet menu that enables you change pin name, number, type, or specify pin graphics (for example, clock wedge or inversion dot) of all sections of the package. You can also access the **Edit Properties** window by **RMB** edit pins or **<Shift+H>**.

Adding Power Pins
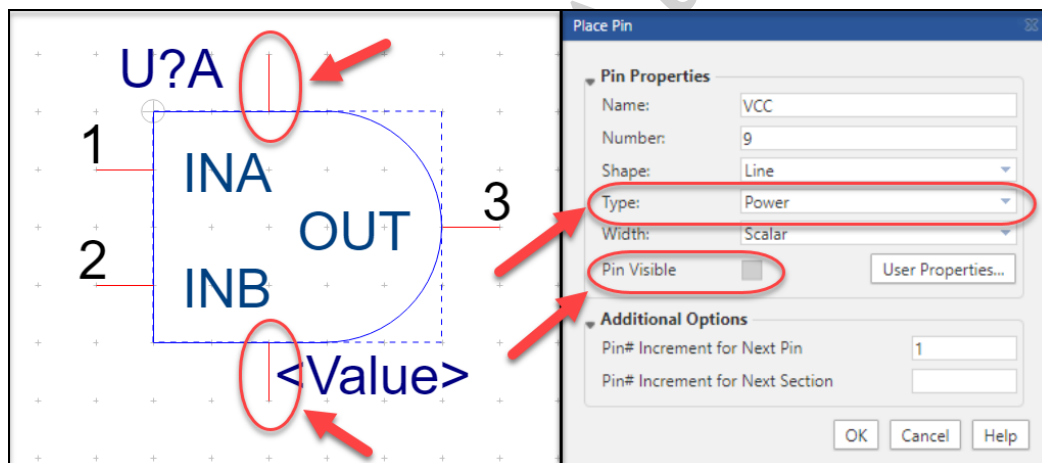
To add a power pin, in the **Place Pin** window set the *Type* field to *Power*. The name of the power pin is used as the name of the power net.

When you set the pin type to Power, the Pin Visible field is activated (normally greyed out for all other pin types). You now have the option available, by default the option is checked. When the box is unchecked the pins are no longer visible in the schematic however, they are automatically added to the netlist.

If you want to override this default signal defined for a power pin, edit the part properties in the design and make its power pin visible. Then connect it to an

appropriate voltage symbol in the schematic. If you check the Pin Visible option when building the part, then the power pin is always visible in the schematic and will always need to be explicitly tied to a voltage symbol or wire in order to appear in the netlist.

It is not a good idea to use overbars above power pin names. If you do, any netlists that you create will have invalid power pin names.

It is common to add power pins using the Zero Length pin shape.

Multiple power pins tied to the same voltage should be added with duplicate pin names (for example, pin name +5V) because the pin name is used as the net name. As long as the Pin Type is Power, no duplicate pin errors will occur when you save the part.

Edit Pins

Pin Number ☑ Pin Group ☑ Pin Ignore ☑ Order ☑ Pin Type ☑ Pin Shape ☑
Section [All ▼]

Normal View: Pin Name	Section: A Pin Num...	Section: A Pin Ignore	Section: B Pin Num...	Section: B Pin Ignore	Section: C Pin Num...	Section: C Pin Ignore	Section: D Pin Num...	Section: D Pin Ignore	Order	Pin Group	Normal View: Pin Shape	Normal View: Pin Type	Normal View: Pin Visible
INA	1	No	4	No	10	No	13	No	0		Short	Passive	Yes
INB	2	No	5	No	9	No	12	No	1		Short	Input	Yes
OUT	3	No	6	No	8	No	11	No	2		Short Dot	Output	Yes
GND	7	No		Yes		Yes		Yes	3		Short	Power	No
GND	14	No		Yes		Yes		Yes	4		Short	Power	No

[OK] [Apply] [Close] [Help]

The Ignore field causes a pin to be ignored during design rules check.

Pin Swapping

Use the PinGroup column to define pin swapping. This field must contain an integer. For example, INA and INB are equivalent inputs. Assigning the same pin group value to each of these pins lets them pin swap during PCB layout.

Edit Pins

Pin Number ☑ Pin Group ☑ Pin Ignore ☑ Order ☑ Pin Type ☑ Pin Shape ☑
Section [All ▼]

Normal View: Pin Name	Section: A Pin Num...	Section: A Pin Ignore	Section: B Pin Num...	Section: B Pin Ignore	Section: C Pin Num...	Section: C Pin Ignore	Section: D Pin Num...	Section: D Pin Ignore	Order	Pin Group	Normal View: Pin Shape	Normal View: Pin Type	Normal View: Pin Visible
INA	1	No	4	No	10	No	13	No	0		Short	Passive	Yes
INB	2	No	5	No	9	No	12	No	1		Short	Input	Yes
OUT	3	No	6	No	8	No	11	No	2		Short Dot	Output	Yes
GND	7	No		Yes		Yes		Yes	3		Short	Power	No
GND	14	No		Yes		Yes		Yes	4		Short	Power	No

[OK] [Apply] [Close] [Help]

Adding User Properties

In the **Part Properties** area you can make Pin names and numbers visible, or rotate pin names. You can also add new properties by clicking on the plus sign just below the **Part Properties** area.
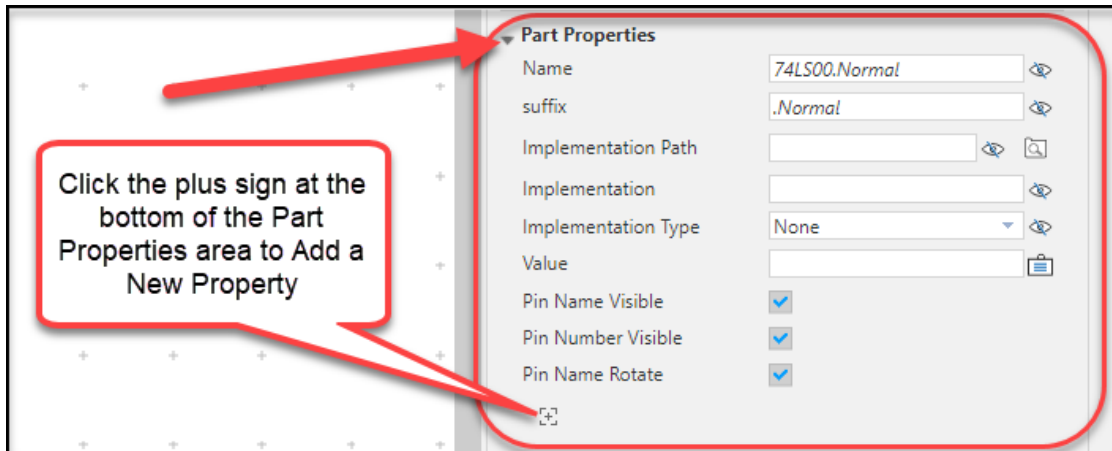
Once you have added a new property you can choose to make it visible by clicking the Display Property icon next to the new property entry.
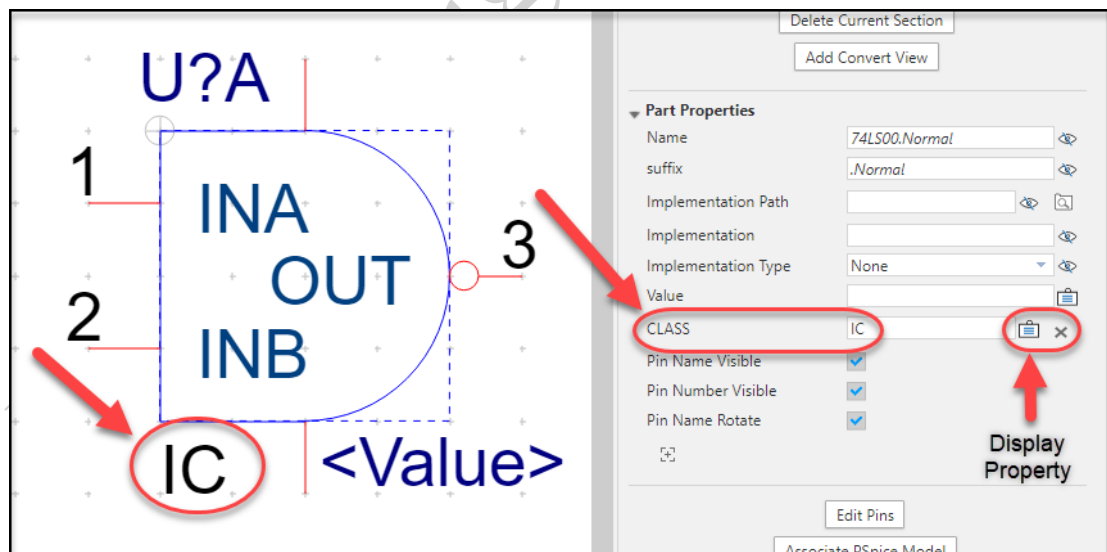
NOTE: The property known as CLASS is a required property for OrCAD and Allegro PCB Editor, which gets used in netlisting and the setup of rules.

The *Part Properties* section enables you to add new properties and edit existing ones. It also contains some default part properties. For example, various implementation properties let you define relationships between the symbol and a behavioral simulation model or lower level schematic in a hierarchical design. There are also pin properties to control visibility and rotation of pin names and numbers. The table that follows identifies and describes each option and field within the Part Properties section of the Property Sheet menu. Deleting a visible property in the *Part Editor* only makes it invisible again.

Default Part Property	Use Description
Pin Names Rotate	When True, pin names will be horizontal on left and right edges, and vertical on top and bottom edges.
Pin Names Visible	Some parts do not need logical pin names visible (for example, simple gates or discretes).
Pin Numbers Visible	Some parts do not need pin numbers visible (for example, discrete resistors and bypass capacitors).

For parts with Passive pin types, you can select the part in the schematic, edit its properties, and control visibility for pin names and numbers. For all other pin types, visibility of names and numbers must be set at the library level.

Adding Properties for OrCAD and Allegro PCB Editor

When applicable, **NC_PINS** (no connect) pins may be added as a user property for the OrCAD and Allegro PCB Editor.
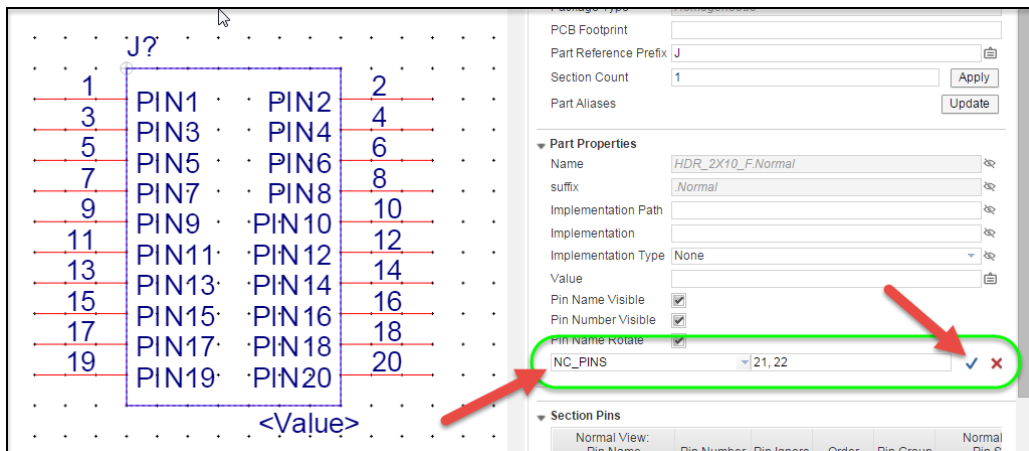
No connect pins are not added to the symbol, but are defined with the **NC_PINS** user property. The value of the **NC property** is a comma-separated list of physical pin numbers.

When transferring the design to the OrCAD and Allegro PCB Editor, the **NC_PINS** property is required to account for all package pins not present on the schematic symbol.

Yet another way to handle package pins such as connector mounting holes is to create these pins as "mechanical" pins when creating the footprint in the OrCAD and Allegro PCB Editor tools. This is covered in the OrCAD PCB Editor training class.

All parts, for the advanced features of the OrCAD and Allegro PCB Editor, MUST have a property assigned to them, called **CLASS**. The value for this Property is one of three: **IC**, **IO**, of **DISCRETE**. This property is used by the OrCAD and Allegro PCB Editor to group types of components for placement.
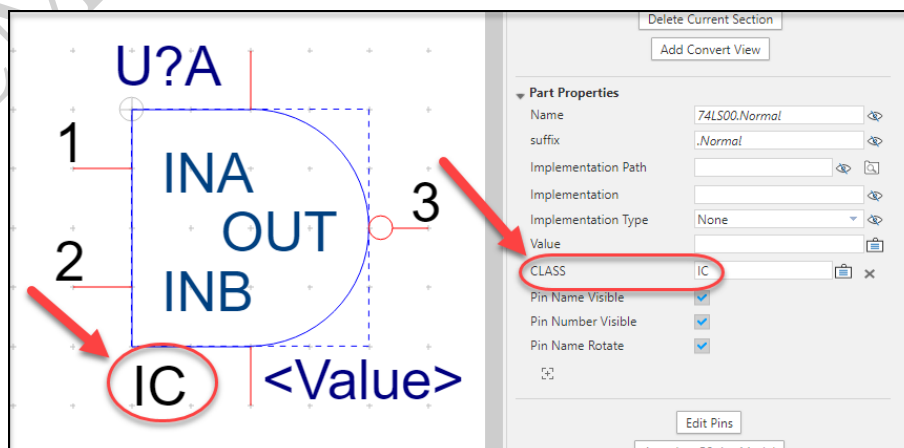
Lab 4-1: Creating a Homogeneous Part

Lab Objectives

After you complete this lab you will be able to:

- Open the New Part Properties window
- Create part graphics
- Add pins and power pins
- Modify pin properties
- Assign pin numbers to all gates in a package
- Set up pin swapping for the OrCAD and Allegro PCB Editor
- Add user properties
- Control property display
- Add part aliases (optional)

Open the New Part Properties Window

1. In the Project Manager, click (highlight) TRNG.OLB.
2. From the *RMB* pop-up menu select *New Part*.
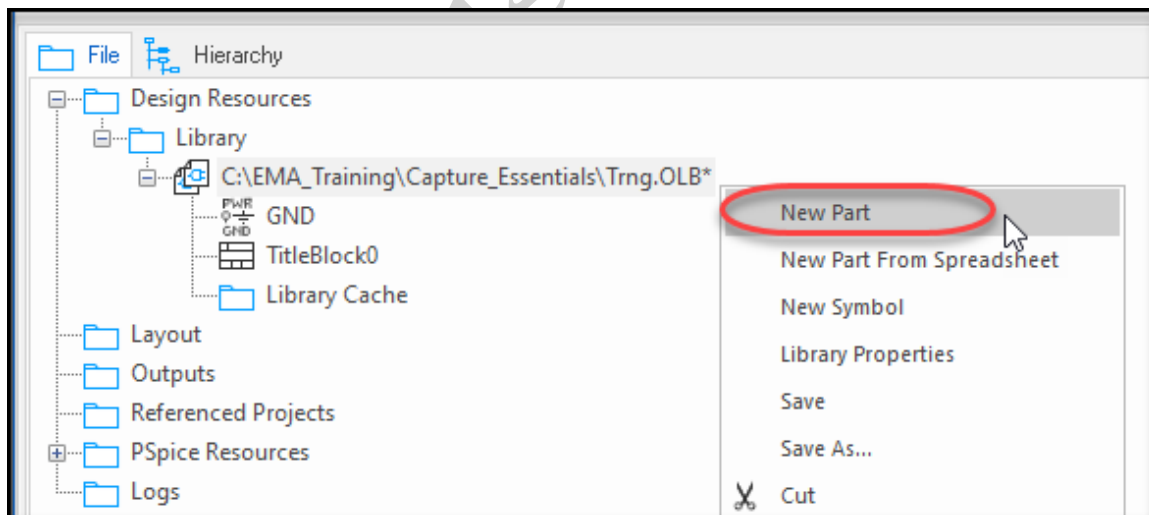
3. The New Part Properties dialog box appears.
4. Complete the form as shown in the graphic that follows.
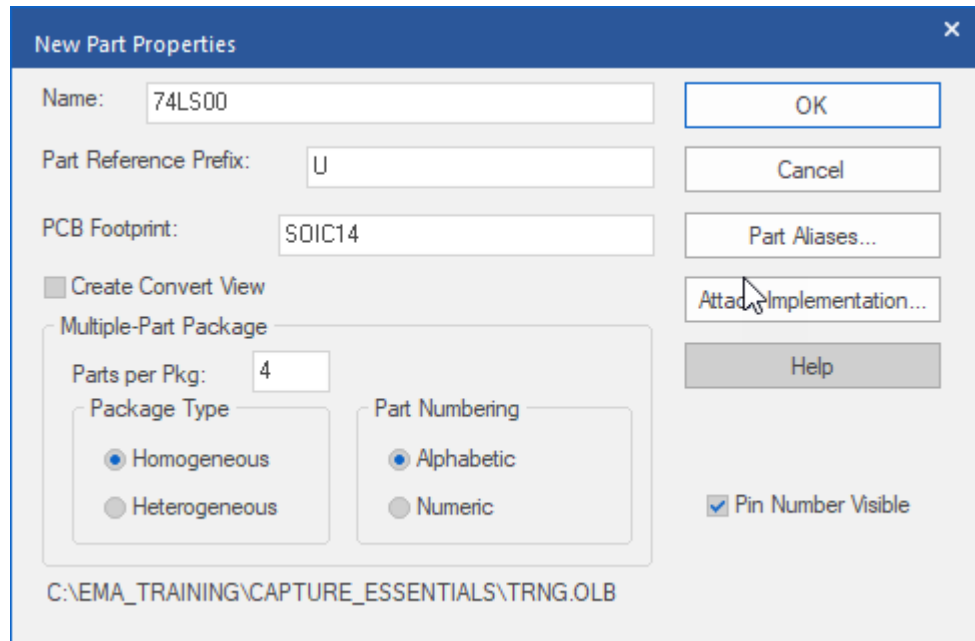
5. Click **OK**.

When finished, the TRNG library window would look like the example below if you copied the 2 symbols in the previous lab.
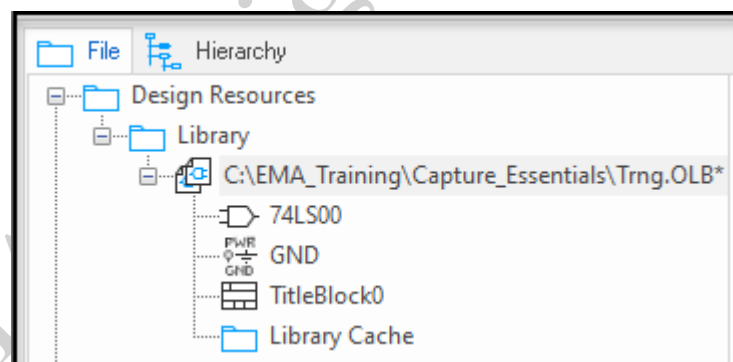
Creating the Part Graphics

The **Part Editor** automatically displays a part boundary box five grid spaces square. All part graphics must fit within the dotted boundary, but the boundary can be expanded by selecting it and sliding the corners to the desired size for the symbol graphic.

1. Maximize the Part Editor window, then select **View > Zoom > All** (or click the **Zoom to all** icon in the toolbar 🔍).

2. Next, adjust the boundary to encompass 4 grids by 4 grids. Do this by selecting the dotted line boundary and dragging the corner(s) to be a 4X4 square. This will accommodate the size of the gate you are drawing.

Drawing Line Segments

Next you will begin placing the line segments.

1. Select *Place > line* (or click the *Place line* icon ⬚) and add lines to complete the polygon, as shown in the next figure.
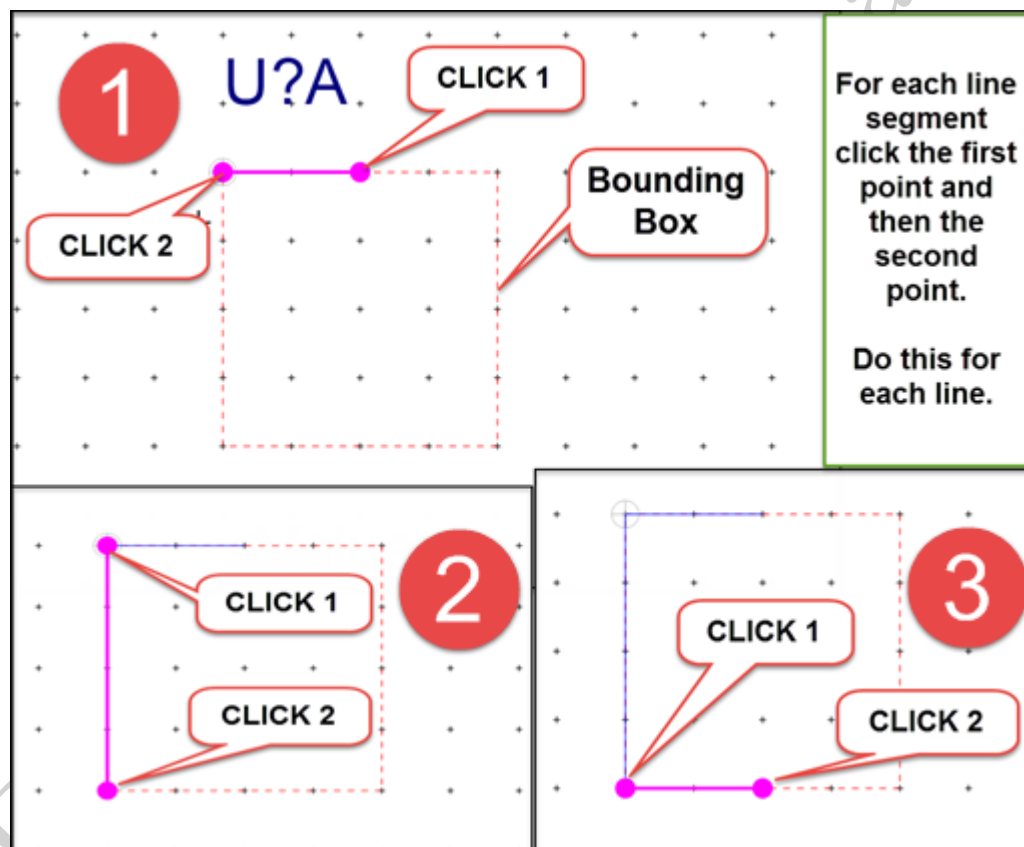
2. Right-click and select *End Mode* from the pop-up menu or press the *<ESC>* key to exit polyline mode.

Placing an Arc

Next you will draw the arc to complete the symbol graphic. Arcs are generally drawn using the following method:

Step 1- click the first "edge"

Step 2 - click the next outer "edge"

Step 3 - expand the circle to meet the desired size of the arc;

Step 4 - finally – drag the arc to the desired shape.

Follow the steps outlined below.

1. Select the **Place Arc** command from the **Place** pull-down menu or the icon   in the **Draw Toolbar**.

2. Examine the "dots" on the screen. Your first click should be what would be the "below edge" of an imaginary "circle, indicated by the '1' in the figure below.

3. Now move the cursor up four grids and click to designate the top edge of the circle (indicated by the '2' in the figure below) and click a second time.

4. Now, move your cursor two grid points to the right in the center and notice a circle appears that you can stretch as you move the cursor (indicated by the '3' in the figure below). Click on the 3rd point indicated in the graphic below to mark the circle.
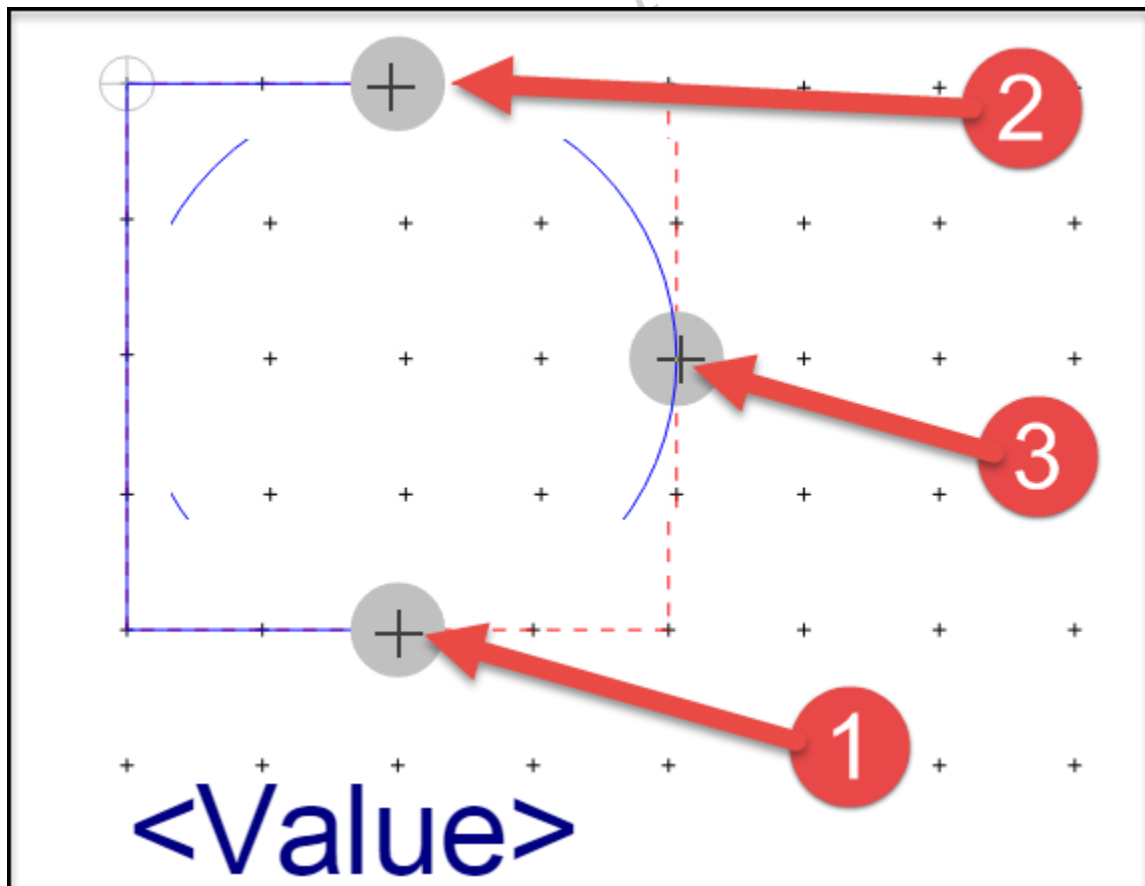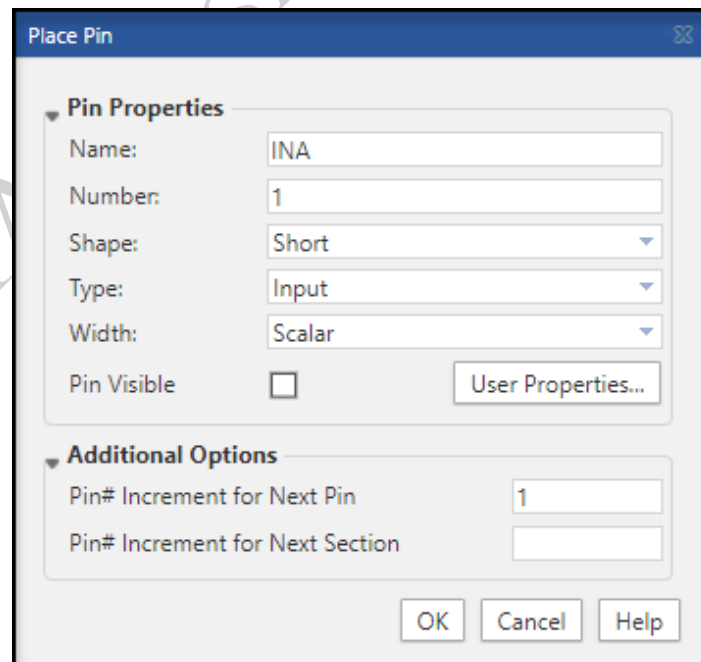
5. Finally, as you move your cursor you can see the arc forming from bottom to top. **Double click** the final point to complete the arc.
6. Once your arc has been completed it will still be highlighted. Right click (RMB) in an area away from the symbol and choose **End Mode** to end 'Place Arc' mode.
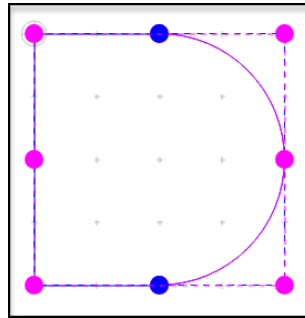
Adding Pins

Next you will be placing pins on the part.

1. Click the *Place pin* icon (or select the *Place > Pin* command from the pull-down menu).
2. Enter the information shown in the following picture.

3. Click *OK*.

   The pin image attaches to the boundary box. As you move your cursor, the pin snaps to 100 mil grid points along the boundary edges.

4. Position it as shown in the following picture, and click to place it.

   Do not double click when adding a pin, as this places two pins at the same location.
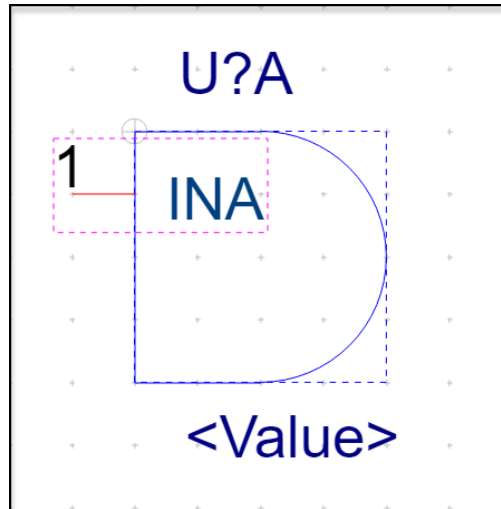
5. Move the cursor and notice that a second pin is now attached and ready for placement.
6. Click to add a second pin two grids below the first pin.

   Notice that the pin number automatically increments. You will change the pin name later.

7. Place a third pin on the right side, as shown in the graphic that follows.
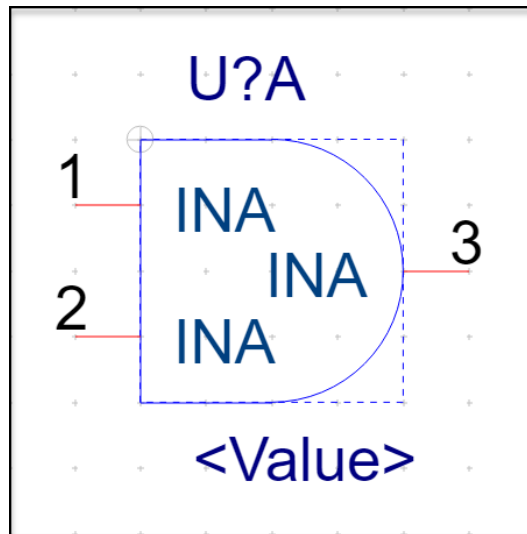
8. Press the *<ESC>* key twice.

Modifying Pin Properties

Next you will edit the pin properties of 2 of the placed pins. You can edit each pin individually or you can select both pins to edit. When editing a single pin – after you select it notice the **Pin Properties** area appears in the **Property Sheet** menu (to the right of the work area). In this way you could edit each pin. Optionally, you can select multiple pins and then use the **Edit Pins** area to make the desired changes. You will be using the multiple-pin select method in the following steps.

1. Use *<Ctrl+LMB>* to select both pins 2 and 3.
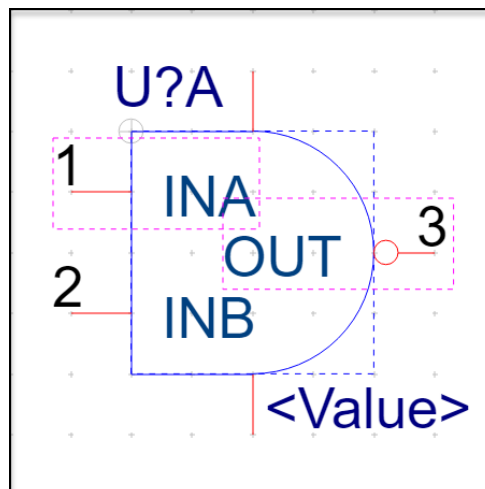
2. **RMB** somewhere on the screen and select **Edit Pins** or press **<Shift+H>**. This will open the Edit Pins window. Here you can edit the pin properties.

Edit Pins								

Pin Number ☑   Pin Group ☑   Pin Ignore ☑   Order ☑   Pin Type ☑   Pin Shape ☑

Normal View: Pin Name	Section: A Pin Num...	Section: A Pin Ignore	Order	Pin Group	Normal View: Pin Shape	Normal View: Pin Type	Normal View: Pin Visible
INA	1	No	0		Short	Passive	Yes
OUT	3	No	2		Short Dot	Output	Yes

OK   Apply   Close   Help
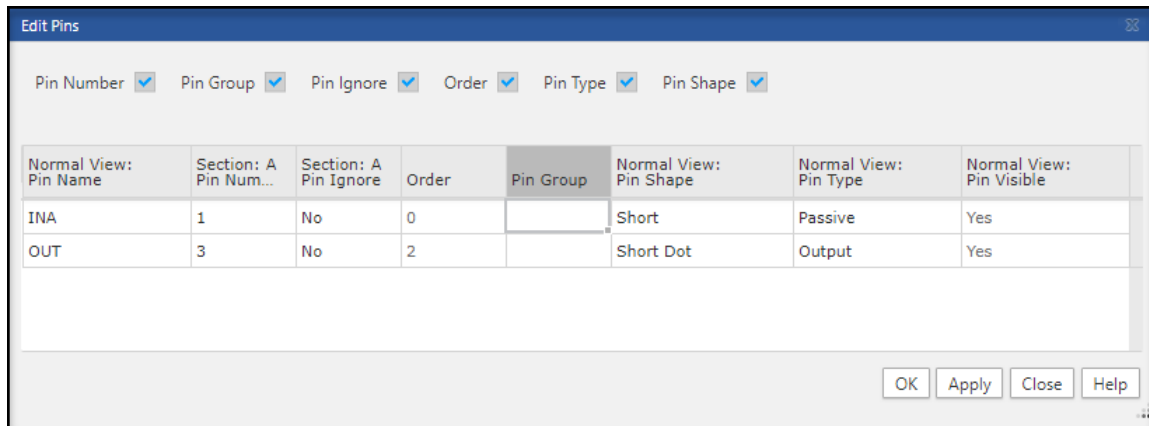
3. Click in the **Name** field for pin **2** and change the pin name to **INB**.
4. Change the **Name** for pin **3** to **OUT**, set the **Type** to **OUTPUT**, and check the **Short Dot** checkbox.

5. Click **Apply**.
6. Press **<ESC>** to deselect all pins.

   Your part should now resemble the following picture:
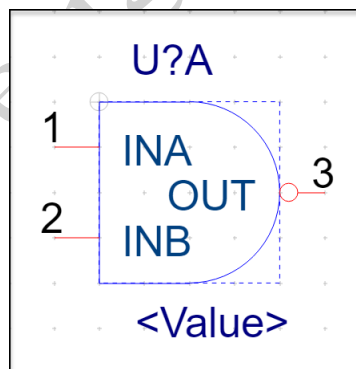
Adding Power Pins

You will now add power pins, as shown in the following example.
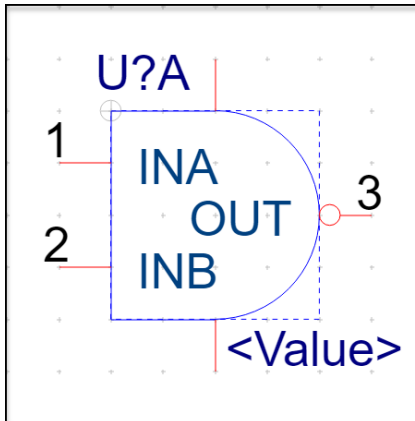
1. Click on the *Place pin* icon ⌐⊃, and enter the values shown in the following picture.
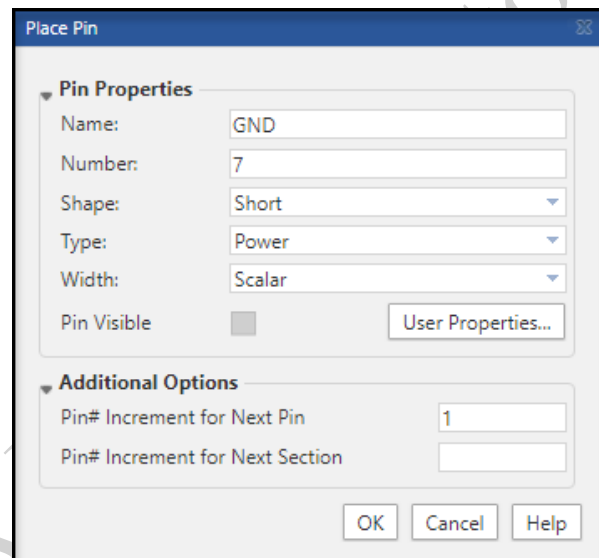
2. Uncheck the *Pin Visible* setting.

Whether power pins are made visible or not visible they still are interpreted as global nets during netlisting. With power pins visible it may be easier to see the power if you wanted to tie it to a different voltage, such as 3V.

If you choose to keep power pins visible, the text can be adjusted for size as well as rotated to make it fit better within the symbol graphic. You will experiment with editing text size and rotation in a later step.

3. Click *OK*.
4. Place the GND pin on the bottom of the graphic.
5. Remain in pin placement mode and place another power pin at the top of the graphic. Then press *<ESC>*.
6. If necessary, select and drag the Part Reference *U?A*, and *<VALUE>* so they do not overlap the power pins.
7. Select the power pin you placed at the top of the graphic and change the *Name* to *VCC* in the **Pin Properties** area of the **Property Sheet** menu.

   Note that the pin number was automatically incremented to 8.

8. Change the *Number* to *14*.
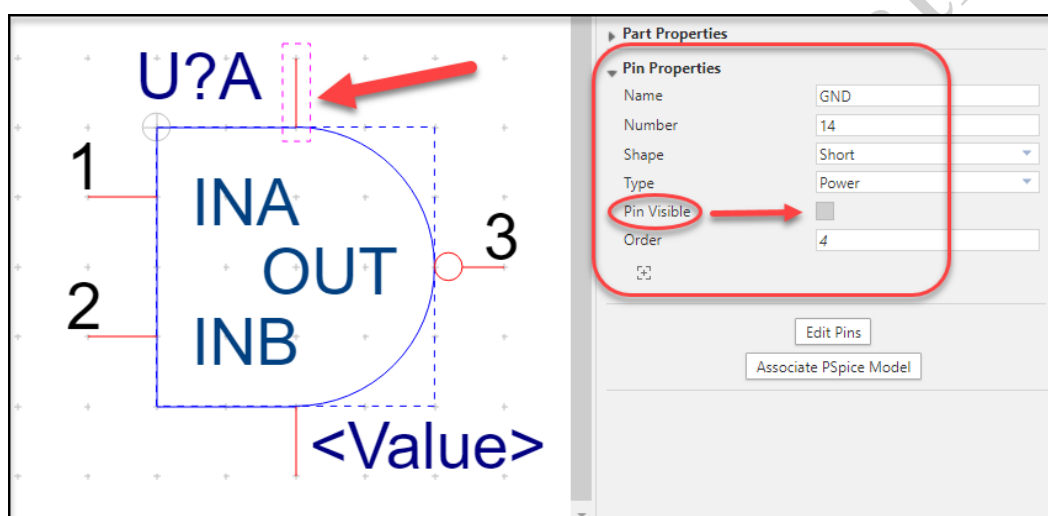
Assigning Pin Numbers to All Package Gates

As this is a homogeneous part with four parts per package, the part graphic represents one of four identical sections. The easiest way to assign pins to the remaining sections is to use the **Edit All Pins** dialog. You will use this method in the following steps.

1. Scroll down in the **Property Sheet** menu and click on the **Edit Pins** button (make sure you do not have any pins selected.
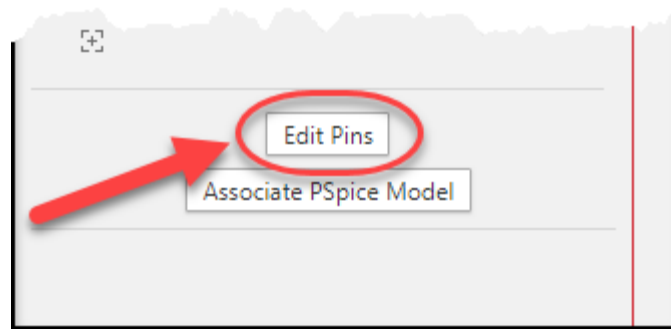
The Edit All Pins dialog appears. Towards the top are the names of the items you can edit: Pin Number; Pin Group; Pin Ignore; Order; Pin Type; Pin Shape. The first column shows the Normal View: Pin Name and then you can see the rest of the columns as they pertain to each section of the part. In this case the part has 4 sections: A, B, C, and D.

Notice the scroll bar towards the bottom of the window enables you to scroll to the right to see more available information.

2. Assign pin numbers to sections **B, C**, and **D** as shown in the next picture.

The **GND** and **VCC** pins map to pins **7** and **14** respectively, for all sections in the package. Choose ignore and leave the pin number
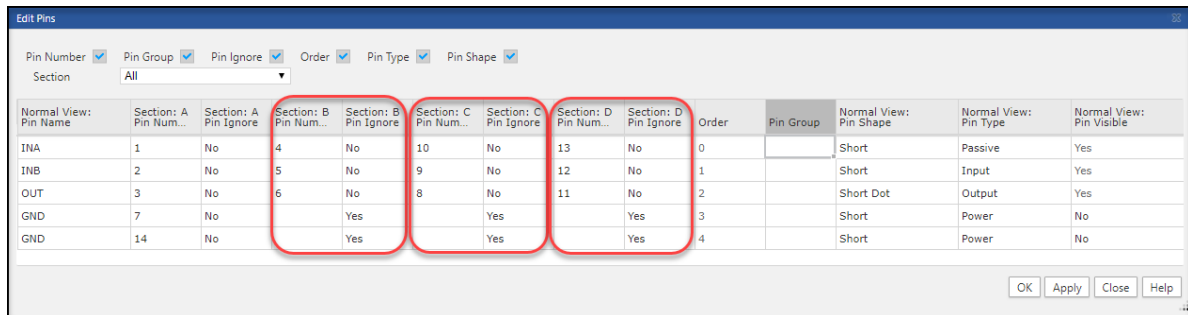
3. Click **Apply**. Do not close the dialog yet.

Setting Up Pin Swapping for the OrCAD and Allegro PCB Editor (Optional)

In order to perform pin swapping in the OrCAD and Allegro PCB Editor you must set pin swap properties in the OrCAD Capture library.

1. In the **Edit All Pins** dialog (should still be open) use the scrollbar at the bottom to scroll to the right to locate the **Pin Group** column and the rest of the available information..
2. In the **Pin Group** column - for the input pins (**INA** and **INB**), assign the number **1**.
3. In the **Pin Group** column - for the output pins (**OUT**) assign the number **0**. (zero)
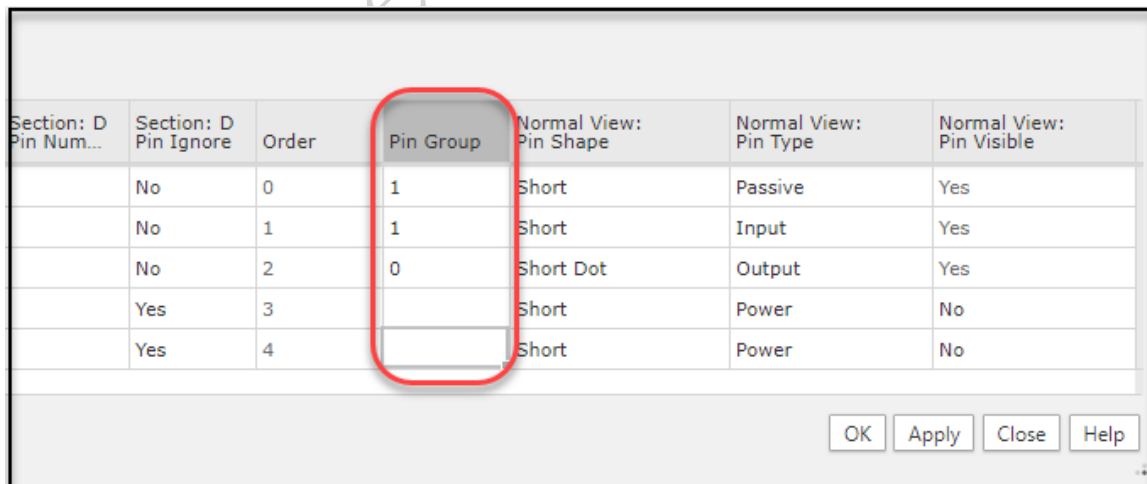
Using a '1' indicates that the input pins are swappable within each gate. Using a zero indicates that the pin is not swappable. Leave the PinGroup field empty for pins **GND**, and **VCC**.

4. Click **OK** to apply the settings.
5. Select **File > Save** to save the part.

Adding User Properties & Controlling Display

1. From the Property sheet menu, expand the part Properties area and select the **Add New Property** button at the bottom.
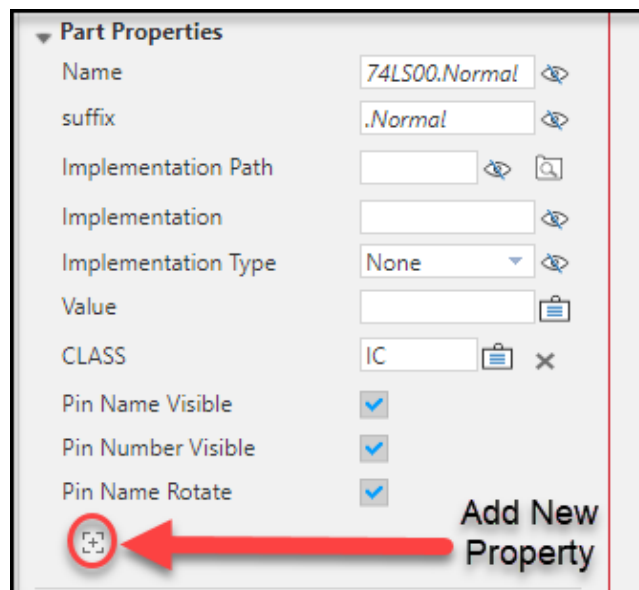
2. Add 2 new properties (CLASS, and Part_Number) as shown in the Part Properties area below.
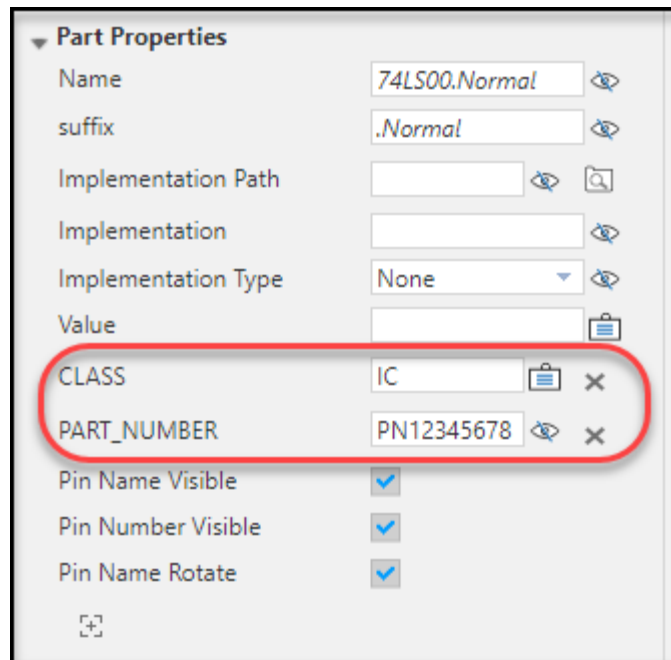
3. Next, for each new property select the display icon and choose to display the Value Only.
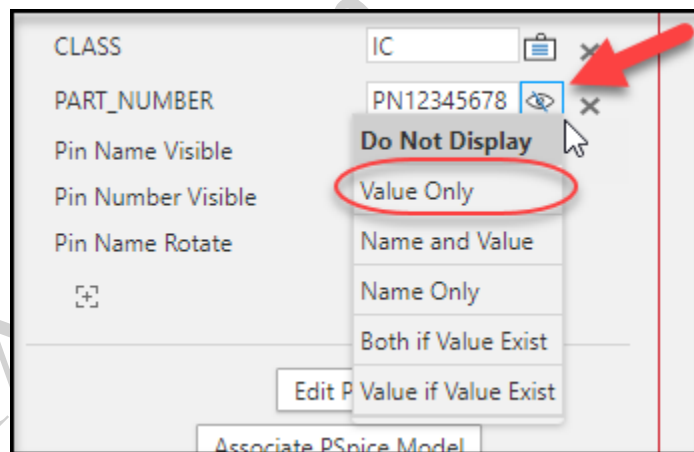
Notice that the part now displays those 2 values. While they are set to display at the symbol level they can be made invisible on the schematic if desired. You can also move the text to a more desirable location away from pins.

4. Save and Close the part.
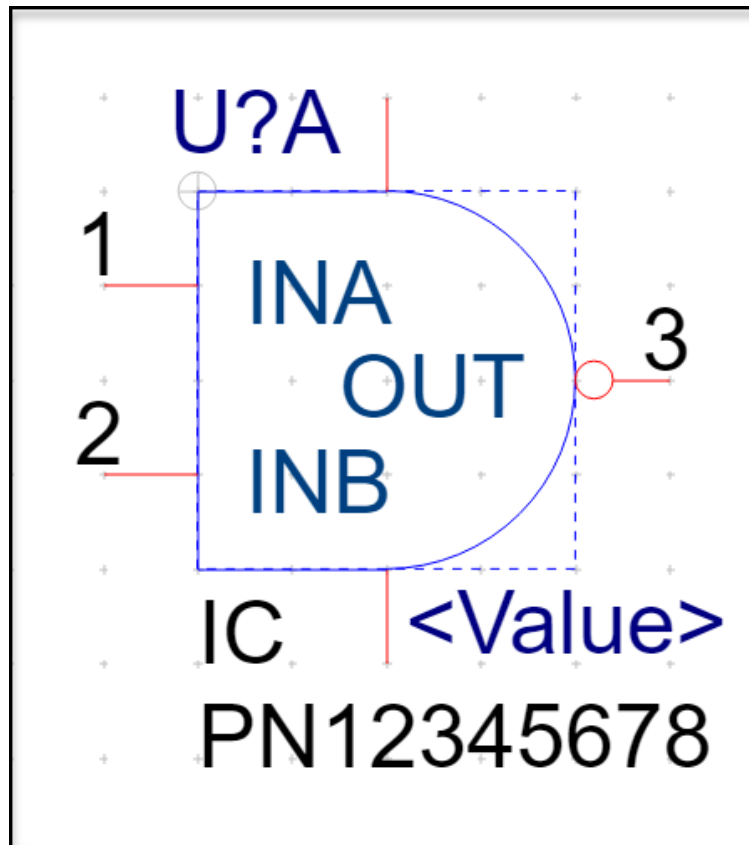
Lesson 5: Creating Heterogeneous Parts

Lesson Objectives

After you complete this lesson you will be able to:

• Create a Heterogeneous part

Heterogeneous Parts

A heterogeneous part is a part that has two or more unique part graphics or pin count. It could have a different number of pins in each "gate" or part. Any part with a pin common to all the other gates must also be defined as a 'Heterogeneous' part.

The following part displays each gate with a different number of pins in the gates as Power & GND are displayed on only the first gate.
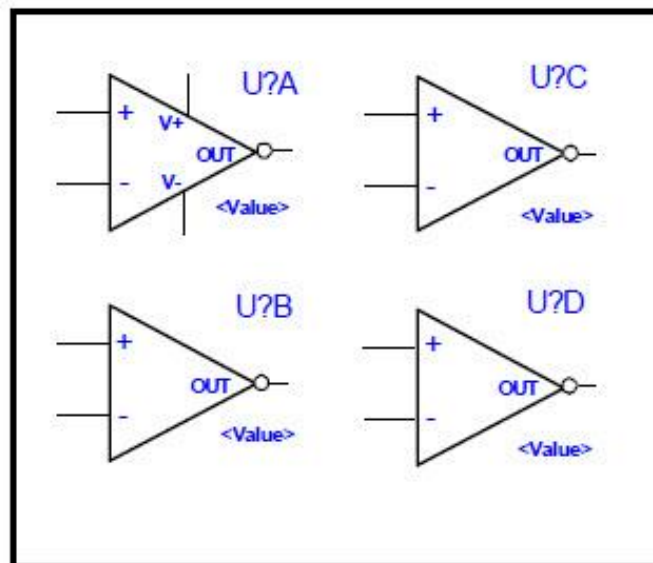
A heterogeneous part must be defined as multi-section (**Parts per Pkg** is set to two or more). Each part graphic represents a different portion of the entire package.

For example, an OPAMP may have 4 gates but only the first gate contains the Power and/or Ground pins. You define this heterogeneous part as having four parts per package. In the Part Editor, use the **View > Next Part** command to create three unique part graphics, as shown in the previous graphic.

Annotation

Auto Annotating Parts

You can have Capture preferences set to automatically assign the next available reference designator in that category by making sure the auto-annotate option is check on. By so doing, you will not have to be concerned about managing reference designators manually. See the graphic below.
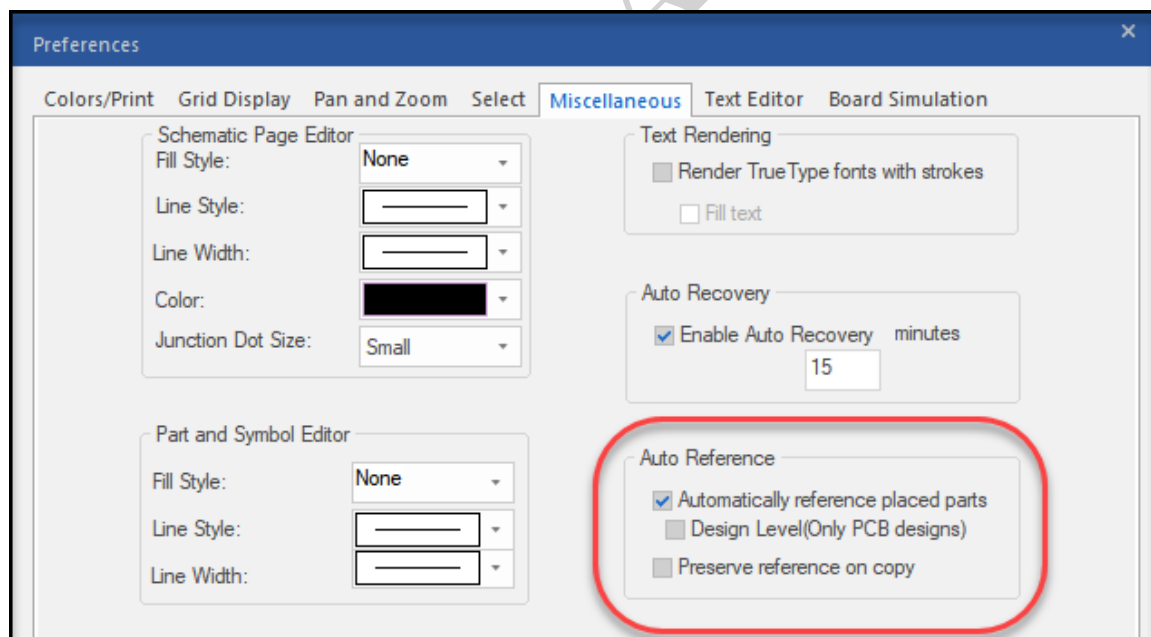
Lab 5-1: Creating a Heterogeneous Part

Lesson Objectives

After you complete this lab you will be able to:

- Create an heterogeneous part

1. If a library is not still open in OrCAD Capture, select and open the TRNG.OLB library you were working with.
2. Highlight the library name and from the right-mouse button pop-up menu select *New Part*.
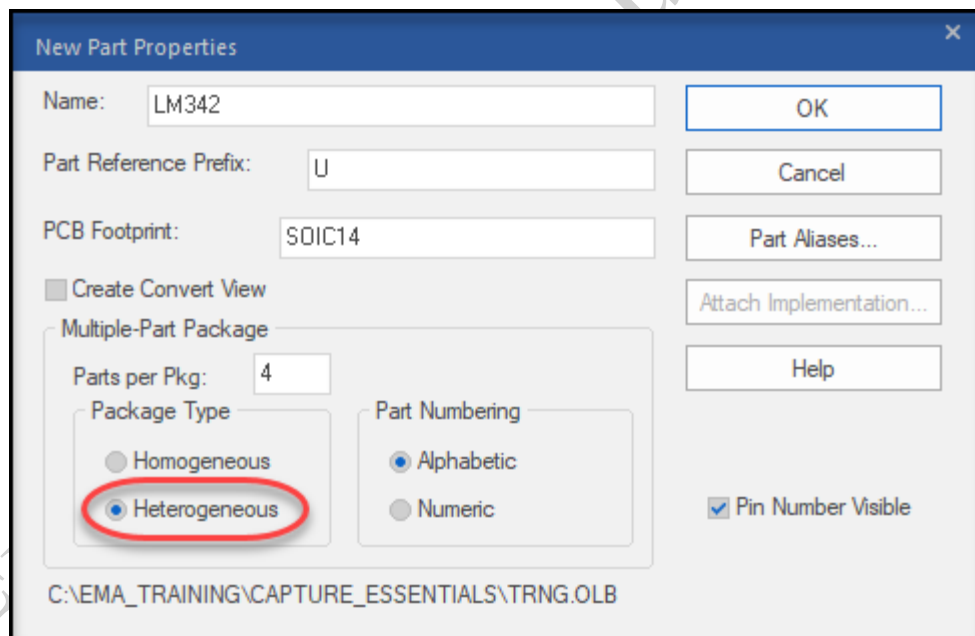
3. Enter the information as shown above. Be sure and select **Heterogeneous** for *Package Type*.
4. Click *OK*.

Creating the Symbol for the first gate

1. Set your window to full, then select *View > Zoom > All* (or click the button in the toolbar) to adjust the view.

2. Using the **Place Polyline** command ⬚ ; and **draw a triangular graphic** as shown in the next image. To draw diagonal lines press **<Shift>** then draw line.

3. After the triangle is drawn adjust the boundary to the edges of the triangle.
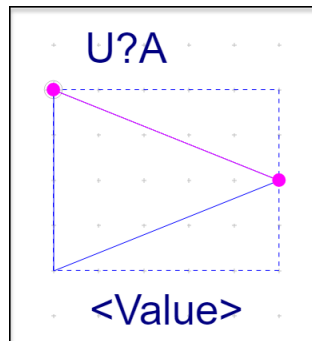
Adding Pins for the first gate

1. Click the *Place > Pin* button.
2. Enter the information shown in the following picture, then click **OK**.
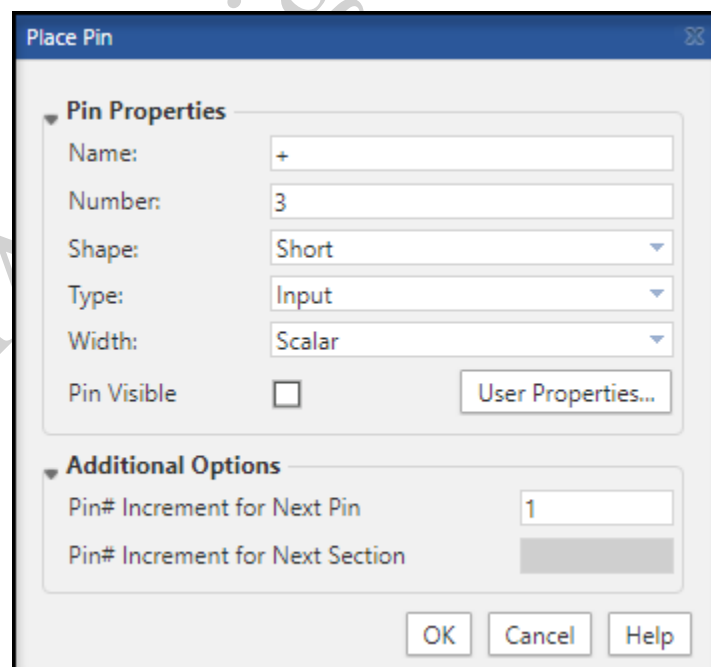
3. The pin image appears, attached to the cursor. Position it as shown in the following picture and click to place it.

4. Place the two additional pins. Next you will edit the additional pins.

Editing the Placed Pins

1. To edit the other pins press <Shift+H> or RMB and select Edit Pins

Edit Pins

Pin Number ☑	Pin Group ☑	Pin Ignore ☑	Order ☑

Section [All ▼]

Normal View: Pin Name	Section: A Pin Num...	Section: A Order	Section: A Pin Group	Sec...: D Pin ...up	Pin Ignore	Normal View: Pin Shape	Normal View: Pin Type	Normal View: Pin Visible
+	3	0			No	Short	Input	Yes
-	2	1			No	Short	Input	Yes
OUT	1	2			No	Short	Output	Yes

[OK] [Apply] [Close] [Help]
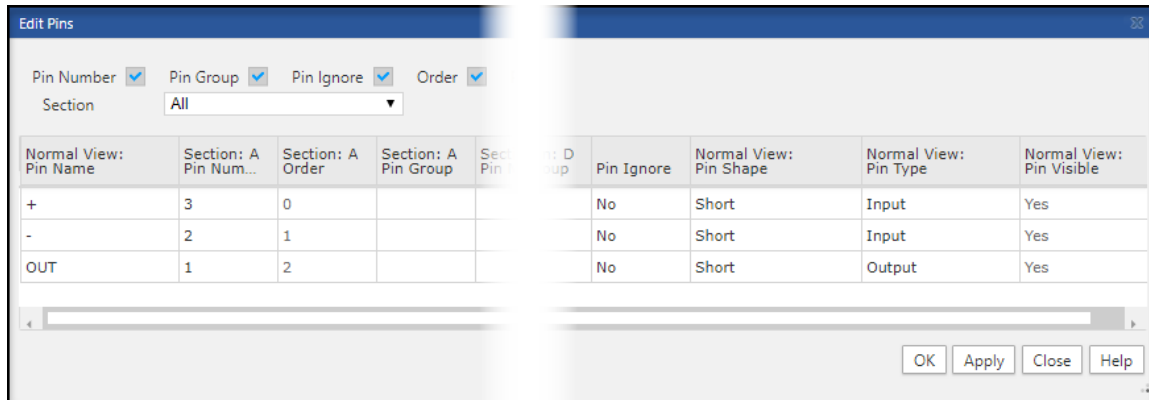
2. Using the *Edit Pin* area, edit the newly added pins to achieve the follow image.

Next pin:	Name (-)	**Next pin:**	Name (OUT)
	Number (2)		Number (1)
	Type (Input)		Type (Output)

3. When you have finished editing the pins compare it to the graphic provided below. The pins names, types, and numbers should match the graphic.
4. If you have not yet adjusted the part boundary, be sure and do so at this time. See the graphic below.
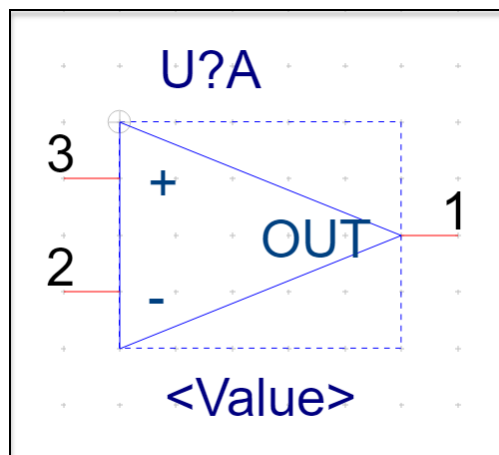
Place the Power pins

1. **Make sure Snap to Grid is on.** **Pins** should always be placed on a part using *Snap to Grid* to ensure accurate connections to wires on the schematic.

2. Select *Place > Pin*. Use the Place Pin window to add the following properties for the first power pin (see the following graphic)
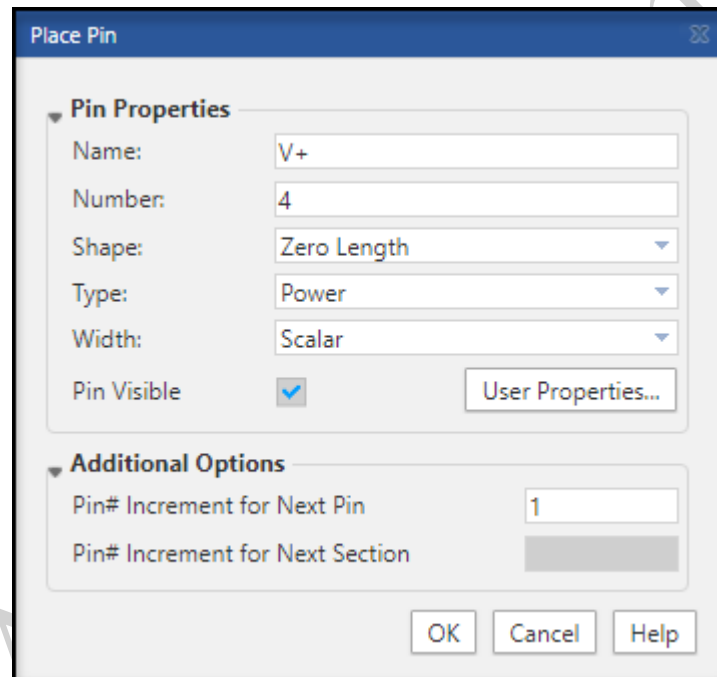
3. Once you click okay the pin will attach to the cursor. Adjust it to be placed on the top of the symbol as shown in the graphic below.
4. Follow the same steps (as in Step 2 & 3 above) to add another power pin (pin number = 11, name = V-) and add it to the bottom of the symbol as shown in the graphic below. In steps later on you will learn how to adjust and rotate the text.
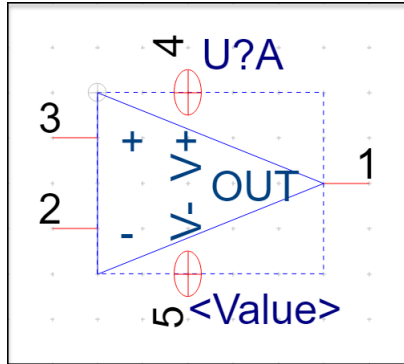
Drawing a Stub Line on the Graphic

Because the gate is a triangle the power pins would not be able to be placed directly against the body of the gate. Instead, you can use a graphic *line* to connect from where the pin would connect to the image body. For this step you'll be drawing a line.

You will also want to **turn off** the *Snap to Grid* in order to place the line accurately. But – when you have completed drawing the line be sure and turn *Snap to Grid* back on.

1. Use the line graphics command, or *Place > Line*, to draw a short "stub" from the edge of the triangle graphics to the edge of the bounding box. See the following graphic.
2. Make sure you turn off *Snap to Grid* to do this. Turn it back on when you finish drawing the line stubs.
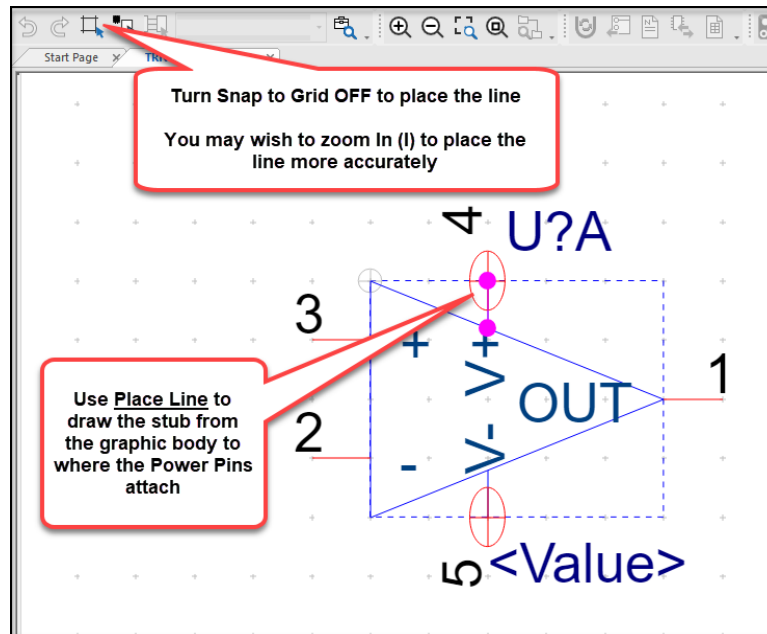
Place the Power pins

1. **<u>Be sure to turn Snap to Grid back on</u>** - Pins should always be placed on a part using **Snap to Grid** to ensure accurate connections to wires on the schematic.
2. Select **Place > Pin**. Use the Place Pin window to add the following properties for the first power pin (see the following graphic)
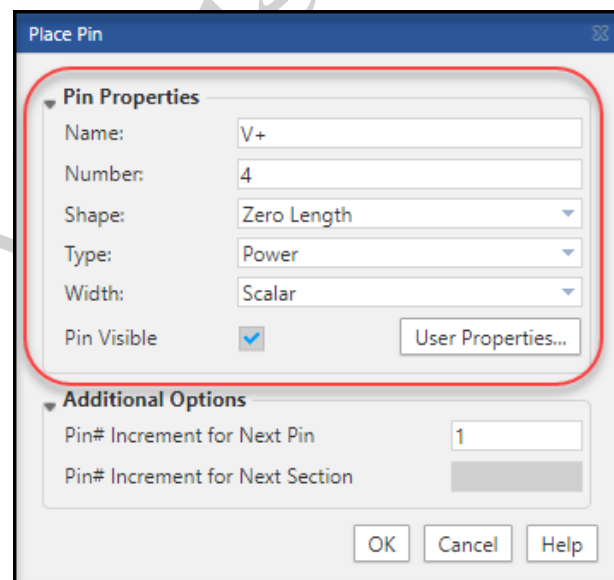
3. Once you click okay the pin will attach to the cursor. Adjust it to be placed on the top of the symbol as shown in the graphic below.

4. Follow the same steps (as in Step 2 & 3 above) to add another power pin (pin number = 11, name = V-) and add it to the bottom of the symbol as shown in the graphic below. In the next steps you will learn how to adjust and rotate the text.
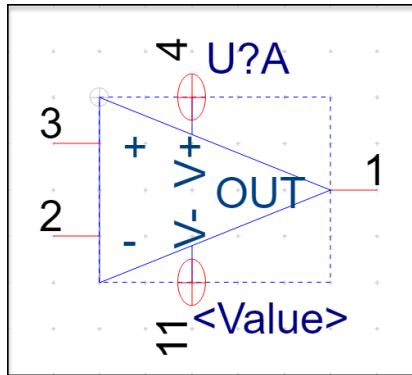
Adjusting the Text Size and Placement

1. Select the pin name text (V+) for **Pin 4**. Notice that there is now a section of the **Property Sheet** menu called **Basic Attributes**. Here you could change attributes like font and font size.
2. With pin name **V+** still selected, RMB and choose **Rotate > Rotate 270**.
3. Follow the same step to rotate the pin name text (V-) for **Pin 11**.
4. Next, use the same method to rotate the power pin numbers (4 & 11).
5. Next, you may wish to resize the pin name text to be smaller so that it fits within the part. Select the pin name **V+** and, using the Basic Attribute area, select the Font Size dropdown and change the font to size **5**. See the graphic below.
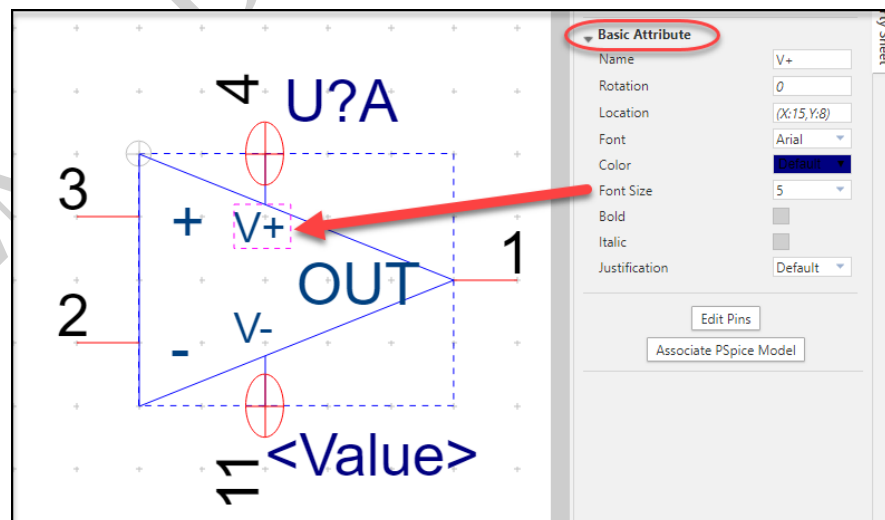
6. Now you can grab the pin name and move it within the body of the part so that it fits without colliding into other text. You may want to turn off Snap to Grid to maneuver the text more easily inside the symbol body. Be sure to turn Snap to Grid back on when you are finished.
7. Do the same for the power pin named **V-**.

   The part should look like the following picture.
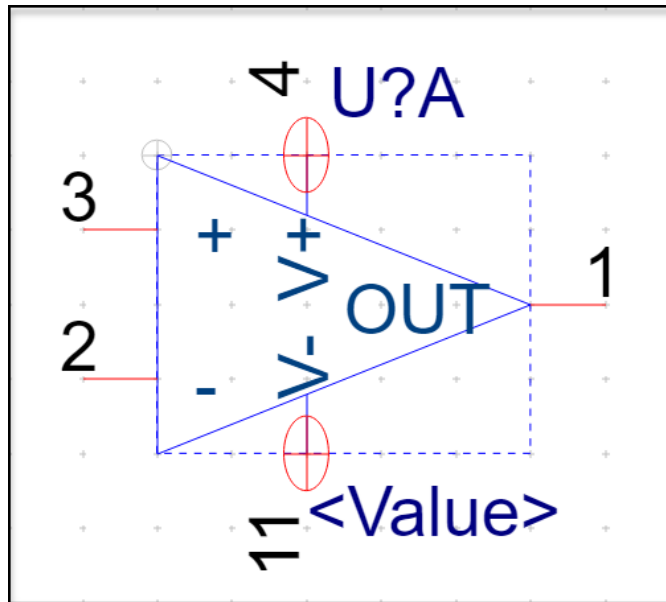
Creating the Symbol for Section Two, Three, and Four

1. Choose **View > Next Part**. The next symbol window will appear.
2. Follow the steps you used to create the first symbol graphic, only this gate will NOT have the power pins.
3. Edit the pins to match the following graphic.
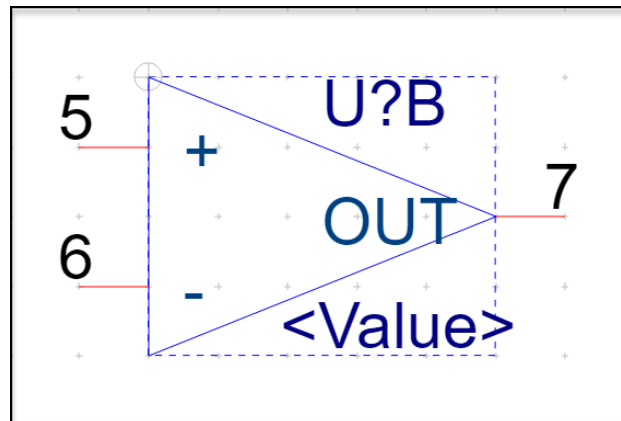
4. Choose **View > Next Part**. The next symbol window will appear.
5. Edit the pins to match the following graphic.
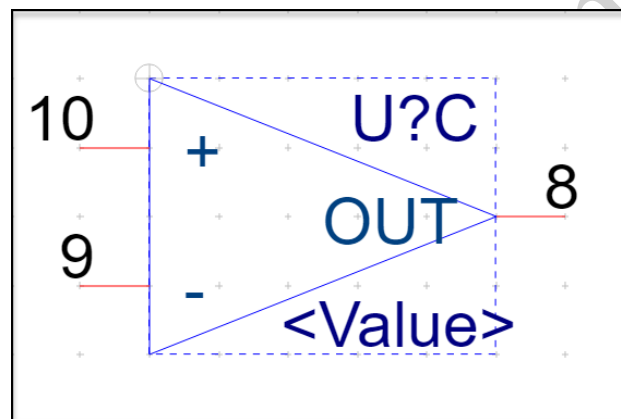
6. Choose **View > Next Part**. The next symbol window will appear.
7. Edit the pins to match the following graphic.
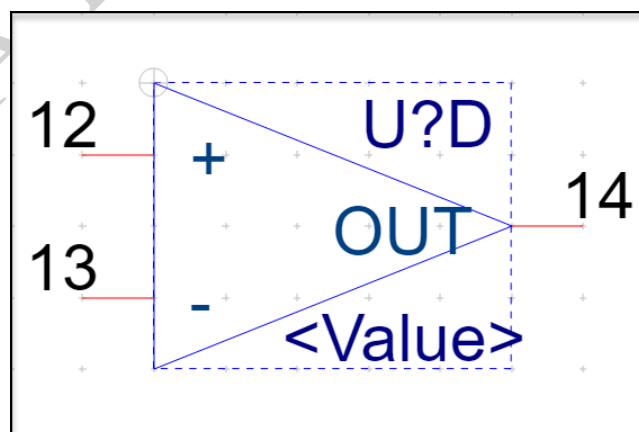
8. *Save* the library part, then *Close* the part window.

9. You can also edit all pins at once by **RMB** and select **Edit Pins** to edit all numbers at once

Normal View: Pin Name	Section: A Pin Num...	Section: A Order	Section: A Pin Group	Section: B Pin Num...	Section: B Order	Section: B Pin Group	Section: C Pin Num...	Section: C Order	Section: C Pin Group	Section: D Pin Num...	Section: D Order	Section: D Pin Group	Pin Ignore	Normal View: Pin Shape	Normal View: Pin Type	Normal View: Pin Visible
+	3	0											No	Short	Input	Yes
-	2	1											No	Short	Input	Yes
OUT	1	2											No	Short	Output	Yes
V+	4	3											No	Zero Length	Power	Yes
V-	11	4											No	Zero Length	Power	Yes
+				5	0								No	Short	Input	Yes
-				6	1								No	Short	Input	Yes
OUT				7	2								No	Short	Output	Yes
+							10	0					No	Short	Input	Yes
-							9	1					No	Short	Input	Yes
OUT							8	2					No	Short	Output	Yes
+										12	0		No	Short	Input	Yes
-										13	1		No	Short	Input	Yes
OUT										14	2		No	Short	Output	Yes

10. **File > Close** the library.

Lesson 6: Building a Simple Schematic

Lesson Objectives

As you create a single-page schematic in this lesson, you will:

- Create a new Project
- Find and Place parts
- Add and name wires

Creating a New Project

In OrCAD Capture, you can create four different types of projects using a Project Wizard. When you open the Project Wizard from the *File > New > Project* menu, the first dialog box that displays presents the different options you can choose.
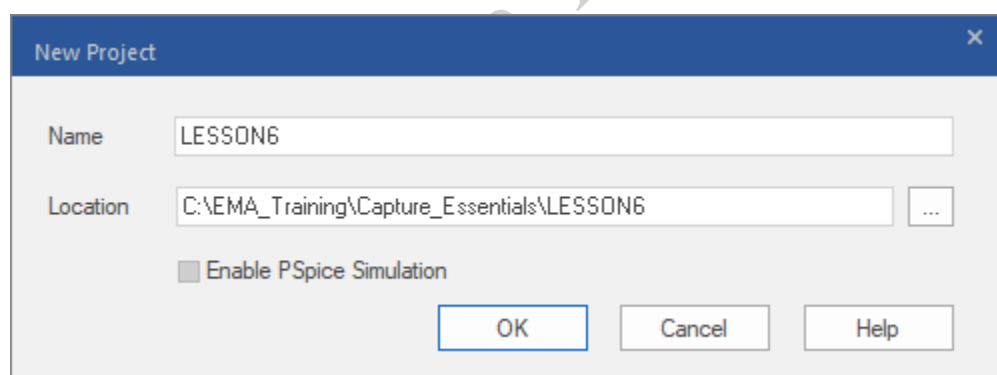
If you plan to use PSpice A/D to simulate the schematic, you must check Enable Pspice Simulation when you create a new project

The project name you specify is used to name both the project and design files. For example, if you name the project *mydesign*, the project filename will be mydesign.opj and the design filename will be mydesign.dsn.

It is important to create a design directory for each new project. This structure keeps all the files for each design together in one place. Name the design directory after the project name.

OrCAD Capture includes the option to change the project type of any existing OrCAD Capture project to a different one.

You can change a capture project into a pspice simulation by creating a simulation. Once you create the simulation, capture will prompt you to select a pspice license.
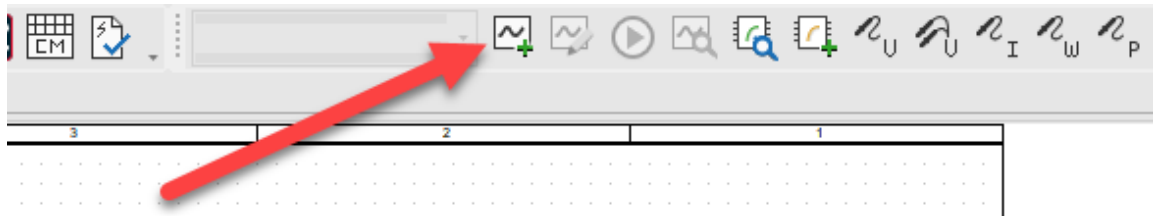
Enabling a Simulation Project

Select the *Enable PSpice Simulation* checkbox if you intend to have simulation capabilities in your design.
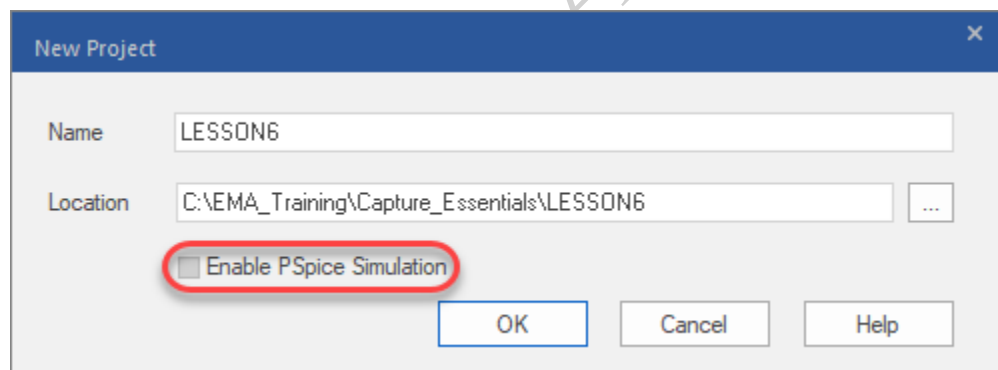
The New Design Project
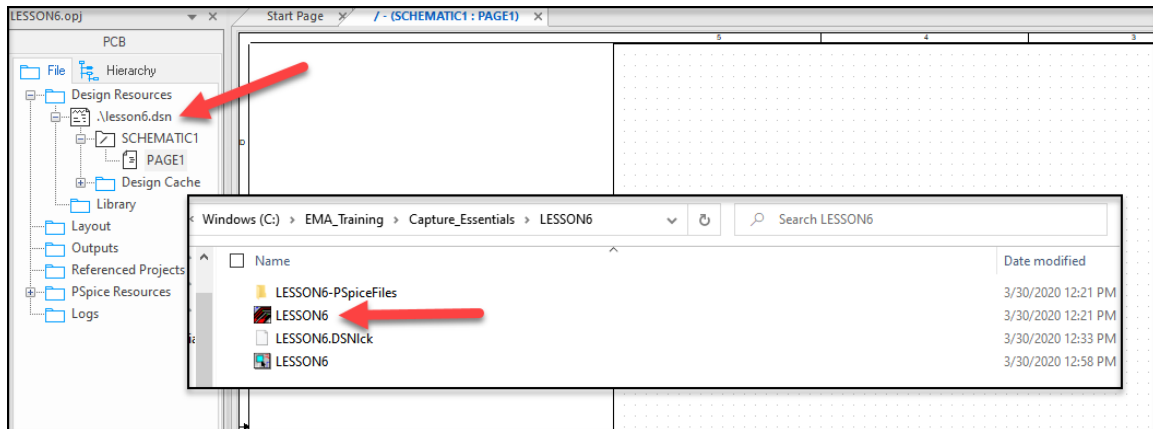
Once project setup is complete, OrCAD Capture opens the new project and displays the design resources in a Project Manager window. A default schematic folder named **SCHEMATIC1** is added to the design file. OrCAD Capture also adds a default schematic page named **PAGE1** and opens it for edit.

Notice that the Title block displays the setting you added in a previous lab when you set up the Design Template.

Placing Parts
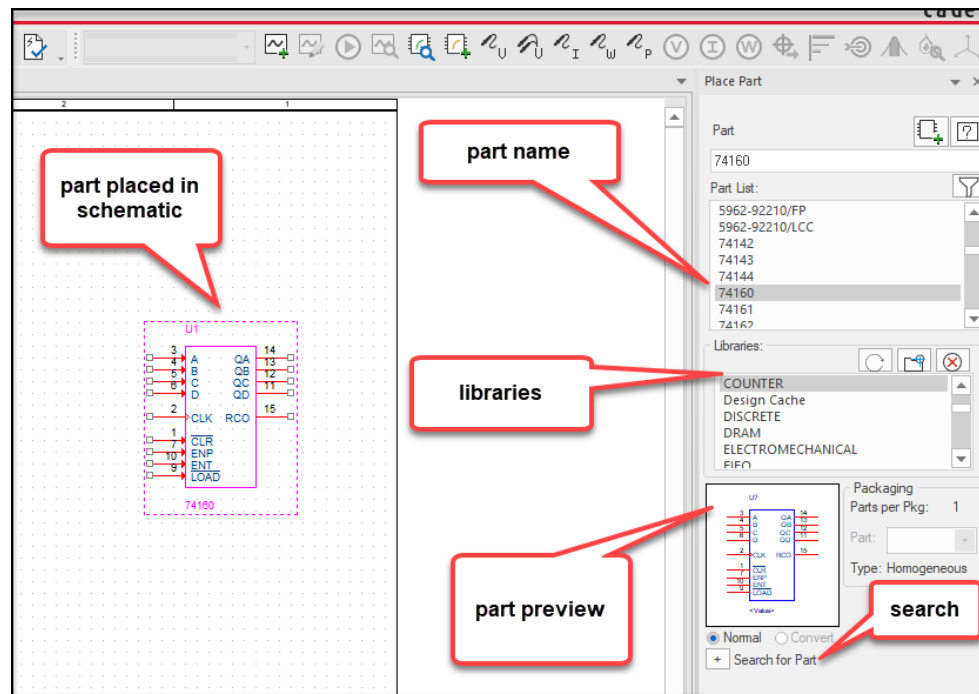
To open the place part menu, press the icon ⬚ in the schematic toolbar or open through the main menu **place>part**.

Before you can add a part to your design, you need access to the library where it resides. If this library was not configured during initial project setup, then you need to press the *Add Library* button to add it to the Place Part window before you can select it for use in the design.
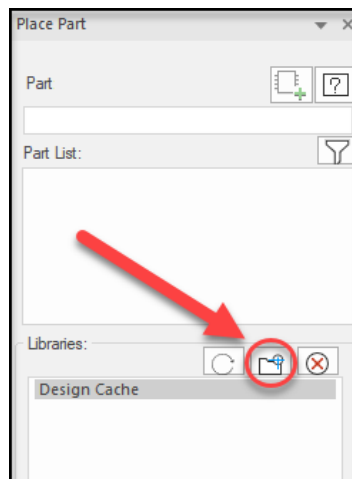
Once the library has been added, you can select it in the Libraries list. The parts in the selected library are listed in the Part List. When you select a part in the *Part List*, you can view a graphic of the selected part.

Rather than scroll through the *Part List*, you can type the name of the part you want to use into the *Part* field. As you enter each character, the data is matched against the parts in the selected library, and the *Part List* automatically scrolls to show the current matches.

More Information

Once a library has been added to the Place Part window, this setup is stored in the Capture.ini file located in the <install_dir>\tools\Capture directory. It will auto load into the *Place Parts* window during subsequent OrCAD Capture sessions. Press the *Remove Library* button to delete the library name from the Place Part window and the Capture.ini file.

Cadence supplies many libraries for use with OrCAD Capture. These libraries are located at <install_dir>\tools\capture\library, where <install_dir> is the location of the **OrCAD** software.

If you are planning to perform a PSpice A/D simulation, you must place parts from special PSpice A/D libraries. The PSpice A/D libraries are located at <install_dir>\tools\capture\library\pspice.

You can automatically assign a part reference designator to each part you add. If there is more than one part in the package (see the *Parts per Pkg* field in the previous graphic), this option should be turned off. Select the *Options > Preferences > Miscellaneous* command to set this user preference.

When more than one library is selected in the Libraries list, the Part List displays both the part and library name for each part in the selected libraries (for example, **COUNTER/74160**).

The Place Part Menu
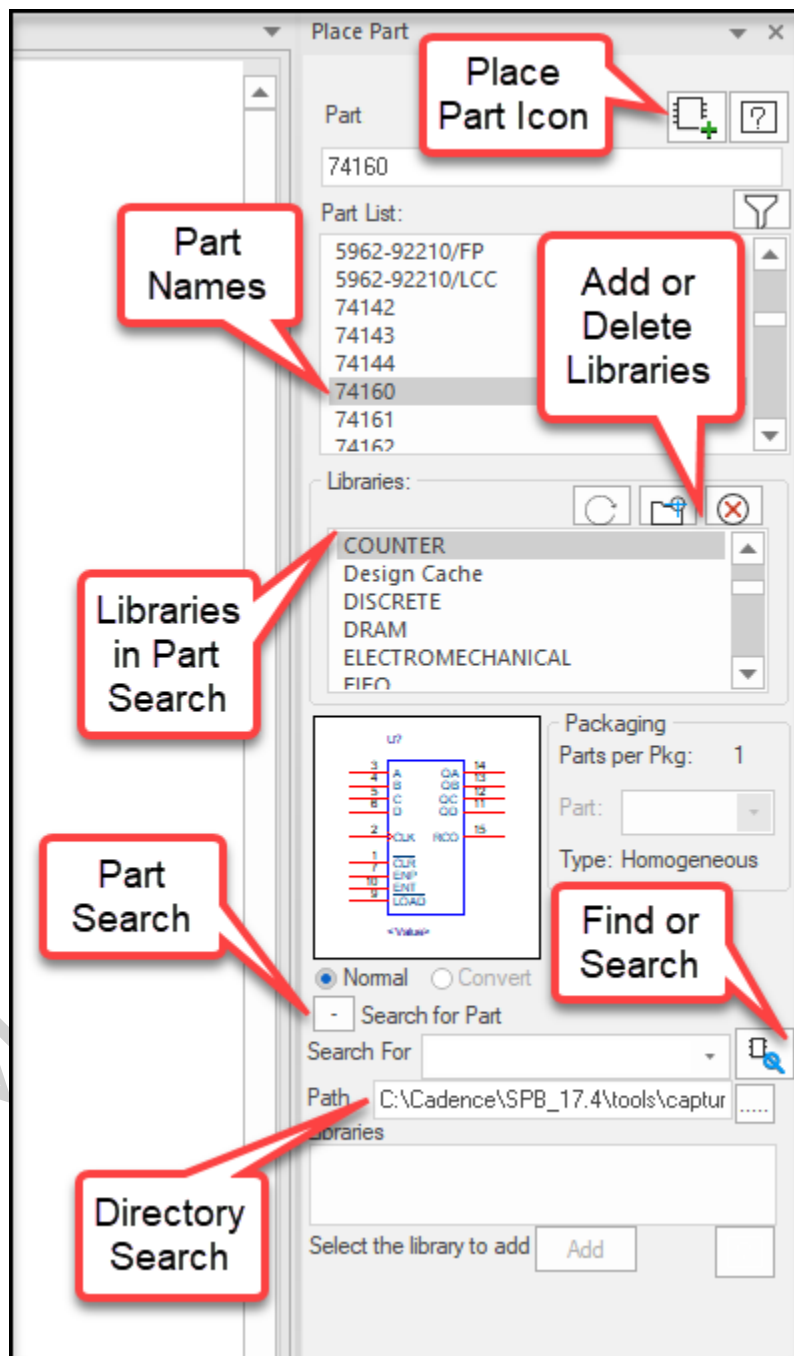
Searching OrCAD Capture Libraries
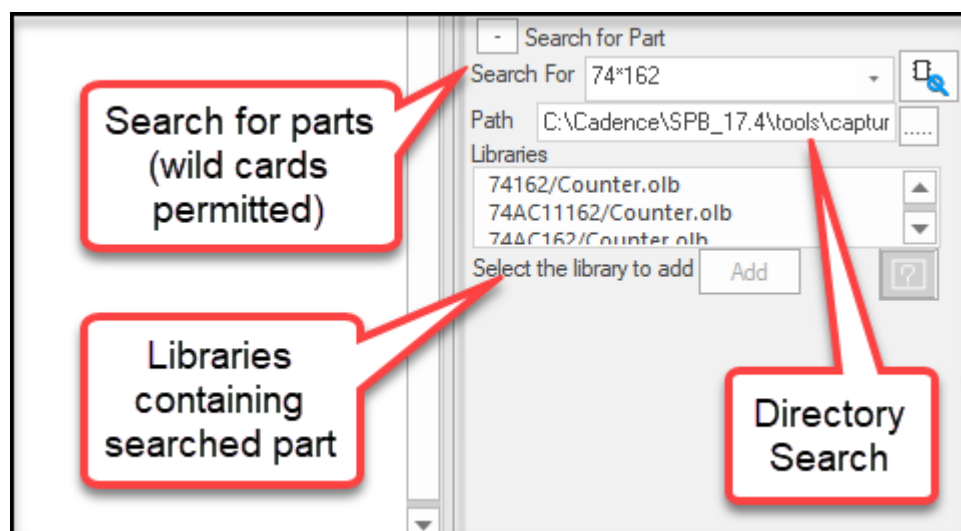
If you want to add a part but don't know which library it is in, you can locate the part by searching all OrCAD Capture libraries. The **Search for Part** area on the **Place Part** window lets you use wildcard characters (*, ?). Once a part has been located, its associated library is automatically added to the Place Part window, and you are ready to select the part for use in the design.

Use the **Library Path** field in the **Search for Part** section of the window to specify the path to a directory containing the OrCAD Capture library files.

Adding and Naming Wires
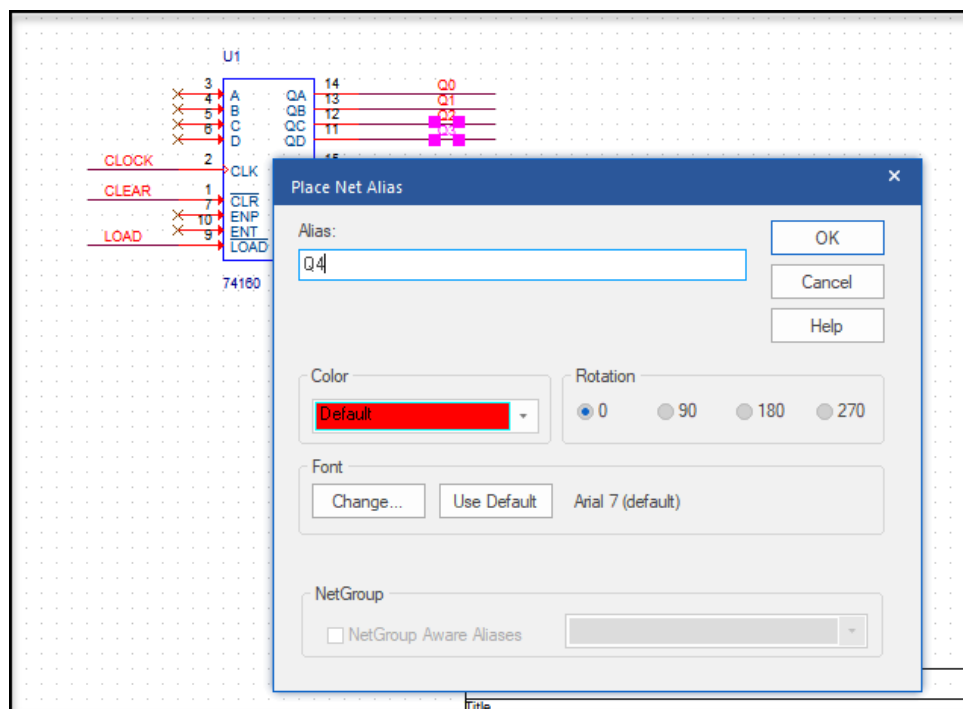
You can use wires and net names (known as *aliases*) to define connectivity in the design.

When you place a wire, it is assigned a system-generated net name (for example, N12345). You can replace the default net name with a net alias [icon].

Two wires with the same alias on the same page are connected by name, even though the two wires may not be physically connected to each other.

More Information

If the last character of a net alias is a number (for example, DATA7), after the net alias has been placed on the schematic, OrCAD Capture automatically increments the last character of the net alias (for example, to DATA8) so you can easily place successive aliases in sequence. If the last character is not a number, then the same net name is assigned with each mouse click.

Net aliases can contain any alphanumeric character, including dashes and underscores. The use of periods, slashes, and spaces in net names is not recommended. The percent sign (%) is a reserved PSpice A/D character. The asterisk (*) is considered a wildcard character by some downstream tools (for example, try not to use asterisks when sending a netlist to the OrCAD and Allegro PCB Editor).

It is possible to add two or more unlike aliases to the same wire, or to connect two wires with unlike aliases together. Such errors can be found later during a design rules check, but will not be flagged by OrCAD Capture when they occur (more on design rules checking later in the course).

Here are some things to remember when adding wires.

- Leave **Grid Snap** enabled (located on the main toolbar above the work area).
- When you are drawing a wire, OrCAD Capture alerts you with a red warning dot

  whenever the wire endpoint comes into contact with another wire or pin (indicating that if you click, you will create a connection). If the proposed wire path will create a problem (for example, short two pins together), OrCAD Capture adds a triangular warning symbol.

- Wire junctions are marked with junction dot symbols . The absence of the junction dot means the wires are not connected. To remove a junction dot, use the **Junction** icon in the schematic toolbar to place another junction dot on top of it.

- When you connect a wire to a pin, the connection square on the end of the unconnected pin disappears. If not, the pin is not connected.
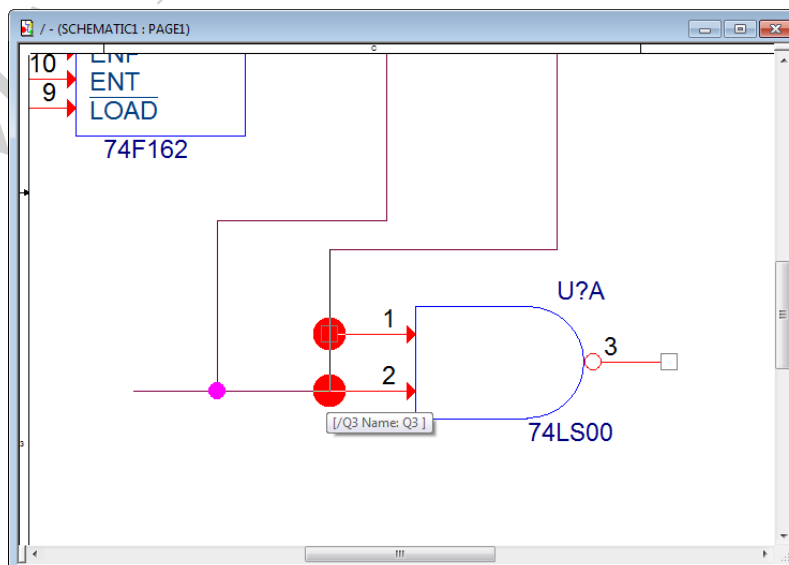
Design Cache

As you begin to add components to the schematic page, the Design Cache will record each part, as shown in the sample graphic below.
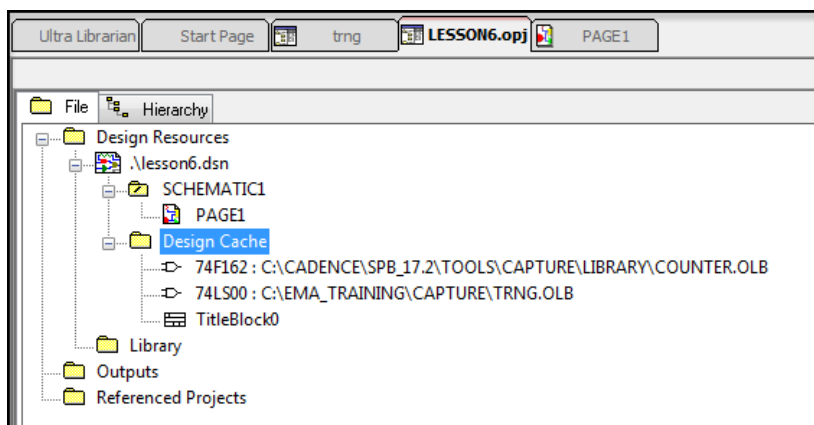
When you add a part to a design, OrCAD Capture automatically stores a copy of the part in the *Design Cache* folder. This makes the OrCAD Capture design completely self-contained (not dependent on any supporting libraries), because copies of all the parts are included in the design file.

More Information

Once a part has been copied into the Design Cache folder, it is used as a comparison tool. For example, you place a part in a design, and later that part is modified in the library. If you return to the library and try to place another instance of the same part, OrCAD Capture will compare that part to the original copy stored in Design Cache, and issue a warning saying the part you are adding is different from the one currently in your design.

You can use the ***Design > Update Cache*** menu item to update the part in the Design Cache to match the edited version in the library, or you can decide against this, and instead place a copy of the part from the Design Cache (using the *MRU* list box).

When you delete a part from the design, a copy of that part remains in the ***Design Cache***. Use the ***Design > Cleanup Cache*** menu item to remove parts from the Design Cache folder that no longer exist in the design. You can also use the ***Design > Replace***

Cache menu item to replace a part in the cache with a completely different part (such as a global part replacement).

Saving and Closing the Schematic

Use the *File* pull-down menu to save the design and close the schematic window.
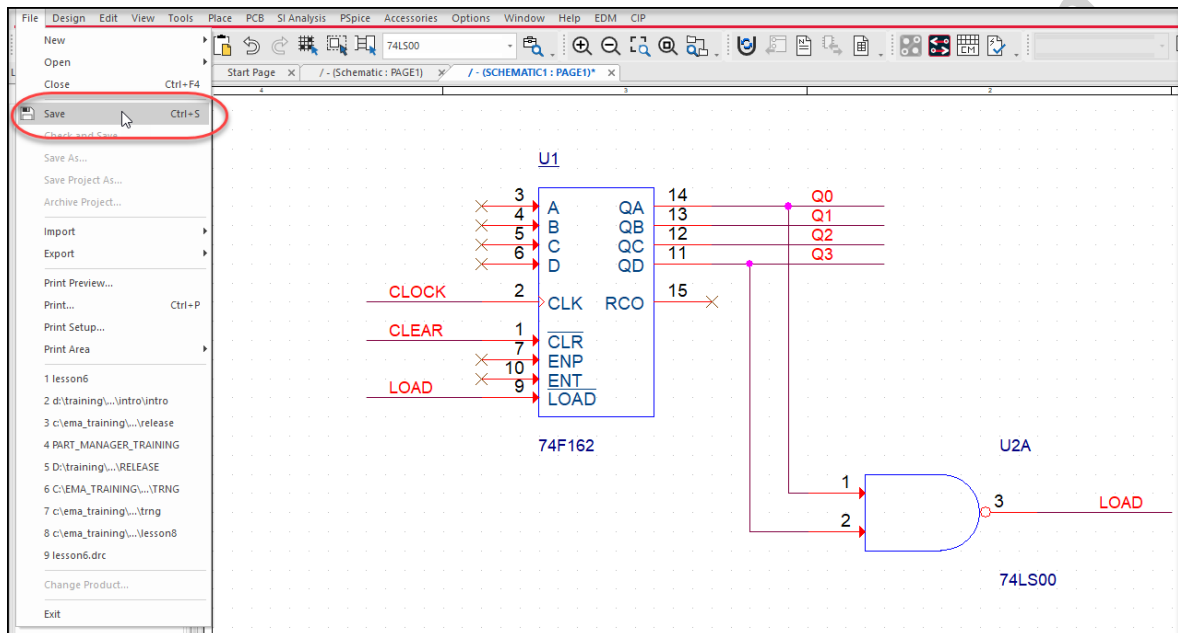
You may also close and save a schematic page by selecting the X in the upper right corner of the schematic page or selecting the Save icon on the tool bar.

Closing the Project and Exiting OrCAD Capture
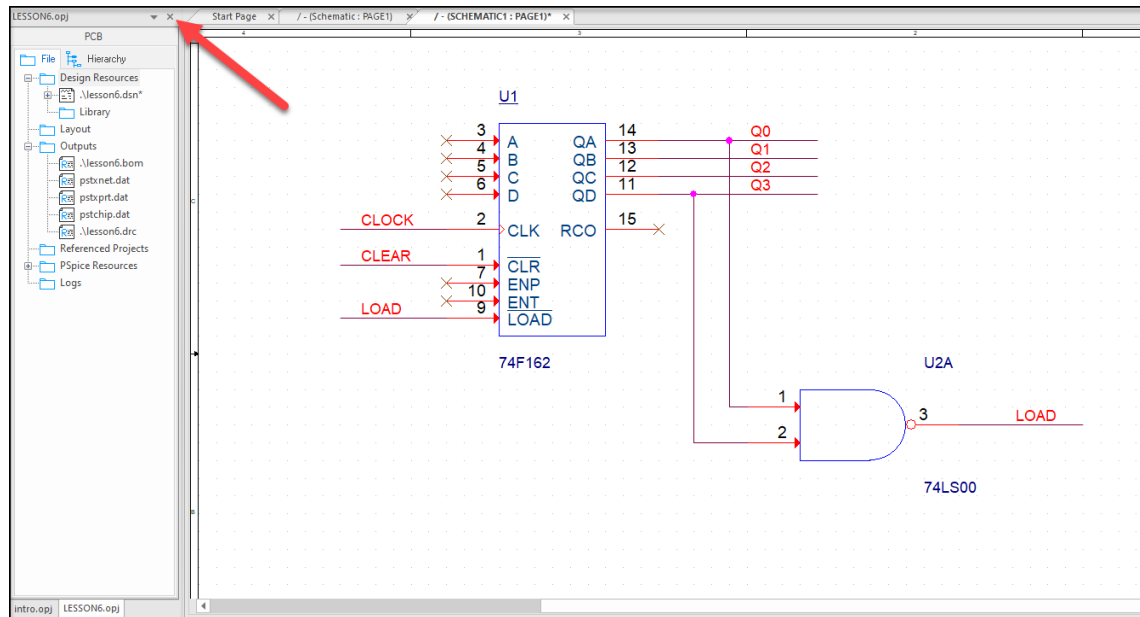
Use the *File* pull-down menu to close the project.

You can also close the project by clicking the X in the corner of the Project Manager Window. If you click in the Project Manager window while a schematic window is open and close the project, the Project Manager records the window arrangement. The next time you open the project, the schematic window is restored to its previous size and location.

Lab 6-1: Creating a New Project

Lab Objectives

After completing this lab you will be able to:

- Identify the project location
- Name a project
- Identify a project type
- View the title block

Create the Project Directory

1. On your computer, navigate to C:\EMA_Training\Capture_Essentials\Designs
2. If the \Designs directory does not exist go ahead and create it.
3. Next, create a new directory named **LESSON6.** This is where you will store your LESSON6 files in the upcoming lab.
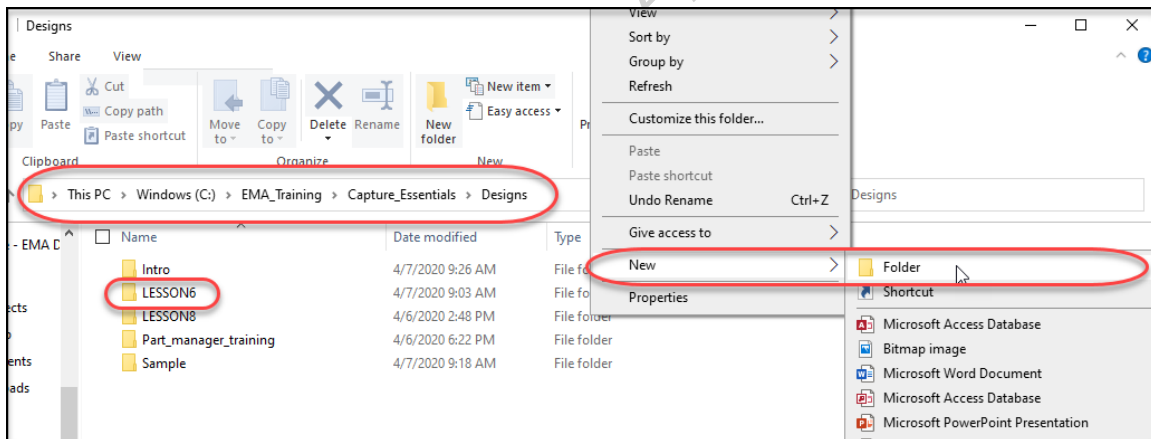
Project Name

In the next steps you will create a new project that resides in the new design directory you just created, located at: C:\EMA_Training\Capture_Essentials\Designs\LESSON6

1. In the OrCAD Capture session window, select **File > New > Project**.

   The New Project window appears.

2. In the **Name** field, enter: LESSON6

3. In the location field, browse to the directory you created earlier:
   C:\EMA_Training\Capture_Essentials\Designs\LESSON6
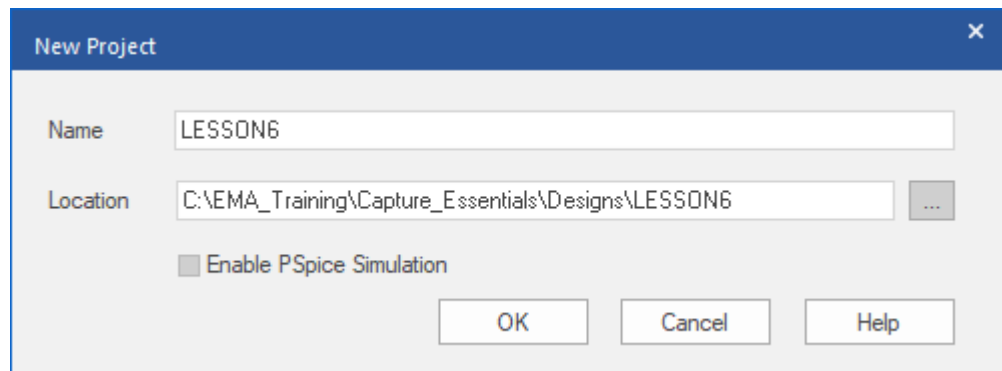
Making the Schematic Page Active

1. Click on the schematic page tab to make it active.
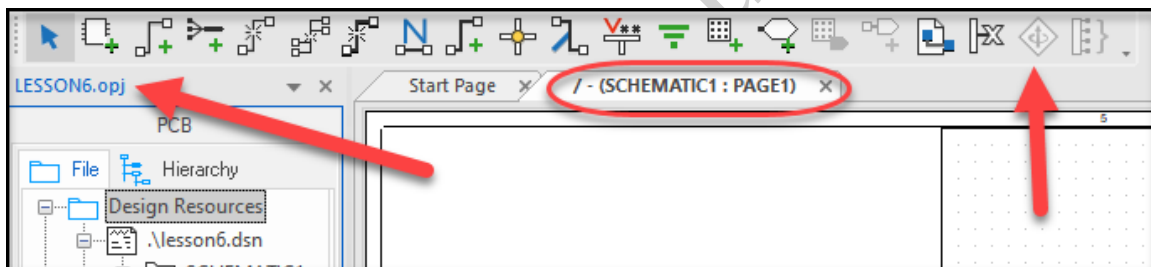
Observe the schematic toolbar – it will be either along the top or along the right edge of the OrCAD Capture session window. Also observe the main toolbar and pull-down menus across the top of the OrCAD Capture session window.

2. Click in the Project Manager tab to make the LESSON6 Project Manager active.
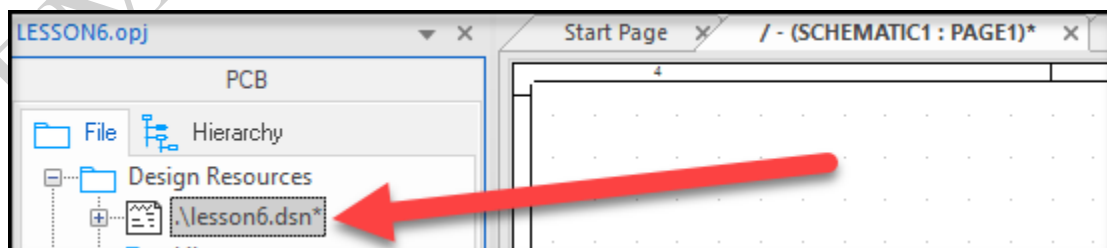
Observe that some icons in the schematic toolbar are greyed out when in the project manager and vice versa in the schematic page. The main toolbar and pull-down menus change as well.

3. Click in the schematic window to make it active again. Resize this window (make it larger) if necessary.

4. Click the *Zoom to all* @ icon. This fits the contents of the schematic in the window. All you currently have in this design is a title block.

Observe that the data in the title block reflects the information you entered into the design template

Title					
	Lesson6				
Size	Document Number				Rev
A	EMA-12345678				A
Date:	Monday, March 30, 2020		Sheet	1 of 1	

5. Click the *Zoom out* icon.

You are ready to begin your first design.

Lab 6-2: Find and Place Parts

Lab Objectives

After completing this lab you will be able to:

- View the reference schematic
- Find and place parts from a library

Viewing the Reference Schematic

1. Take a moment to examine the following schematic. Please use this example as a guide.
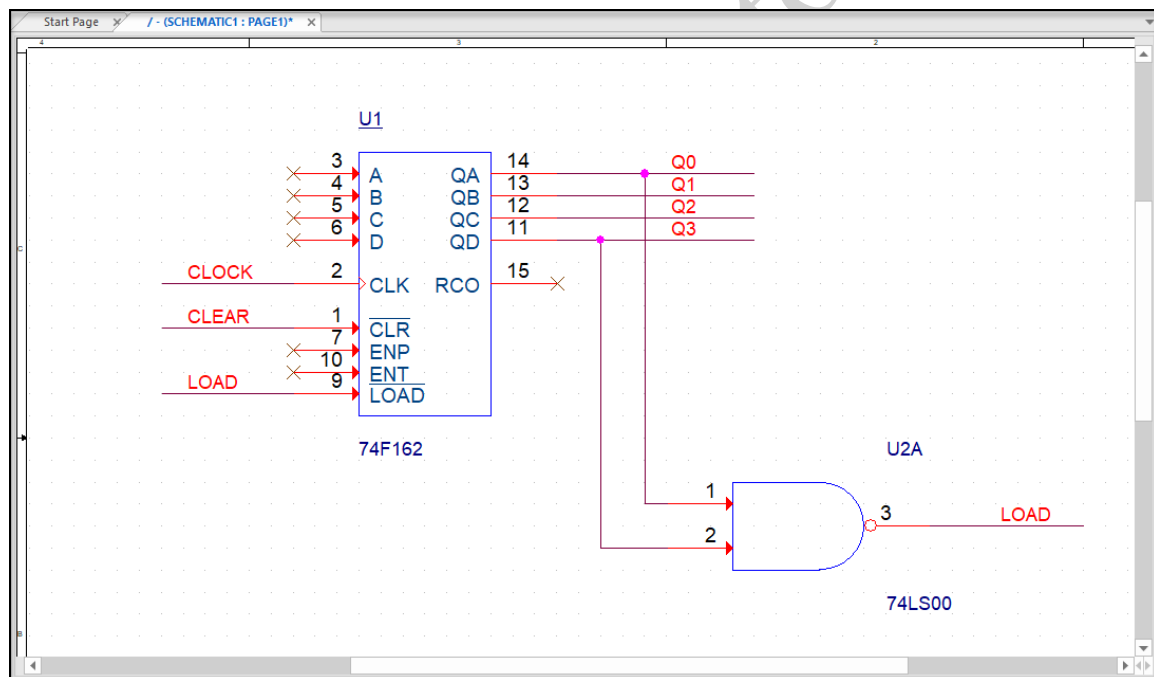
Adding a 74F162

1. Make sure the schematic window is active.
2. Select **Place > Part** from the pull-down menu on the menu bar.

    The Place Part window appears to the right of the Icon toolbar.

3. Click the **Add Library** button.
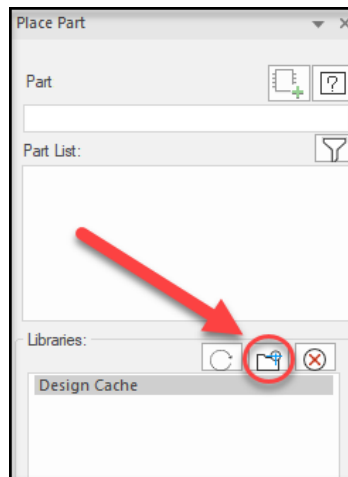
The Browse File window appears.

4. Set the *Look in* field to C:\Cadence\SPB_17.4\tools\capture\library.
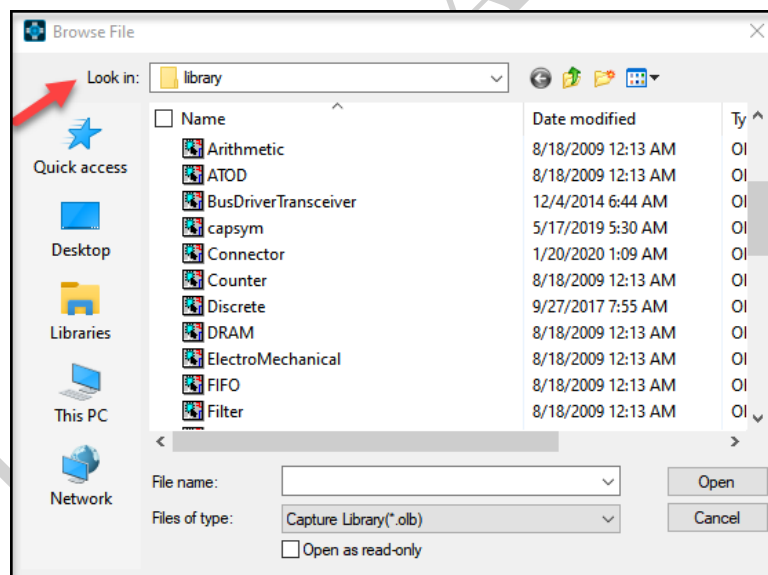
This is where all the OrCAD Capture libraries are stored. The above path may vary, based upon your training site and where the Cadence software is installed on your system.

5. Select the Counter.olb library file and click *Open*.

The COUNTER library is added to the Libraries list in the *Place Part* window, and all the parts in this library are listed in the *Part List* above.

6. In the *Part* field, enter:

74F162

Observe how each character you enter is used to search the Part List for all matching entries. A picture of the 74F162 part is shown in the lower right corner of the Place Part window.
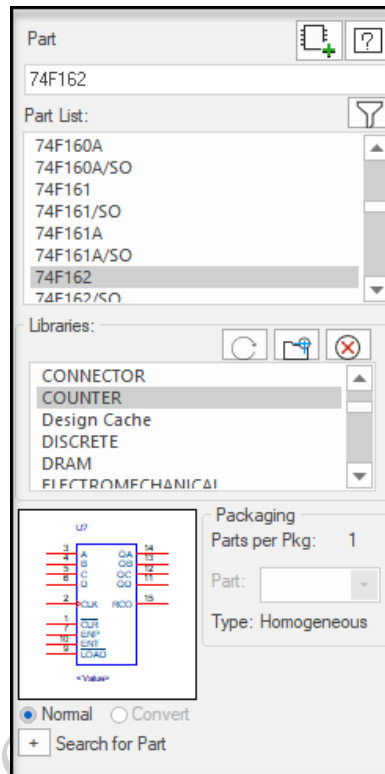
7. At the top right of the Place Part window, click on the **Place Part** Icon.

    The part silhouette attaches to the cursor.

8. With the part "floating" on the cursor, select a location on the schematic page and click **(LMB)** to place the part on the page.

    The part silhouette remains attached to the cursor, ready to place more instances of it.

9. Right-click and select **End Mode** from the pop-up menu or press <**Esc**>.
10. Left-click anywhere in the page to deselect the newly added part.
11. Select **File > Save**.

The 74F162 part was in the COUNTER library. This library was added to the Libraries list. This library list is stored in the Capture.ini file.

Adding a 74LS00

1. Make sure the schematic window is active and return to the **Place part** window*.*

    If the window is not open press the **<P>** key on the keyboard.

2. In the Place Part window click on the plus sign (+) to the left of **Search for Part**.
3. In the **Search For** field, enter:
        74LS00
4. The library **Path** field (at bottom) should already be set to
    C:\Cadence\SPB_17.4\tools\capture\library
    If necessary, use the **Browse** button to set it. See the following example.
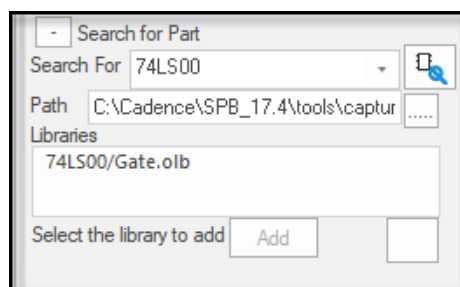
Depending on your software installation, the above path to the library may differ.

If the filter gets set, enter the * symbol into the Place Part box and then press the **<Enter>** key. This resets the setting.

5. Click the **Part Search** icon.
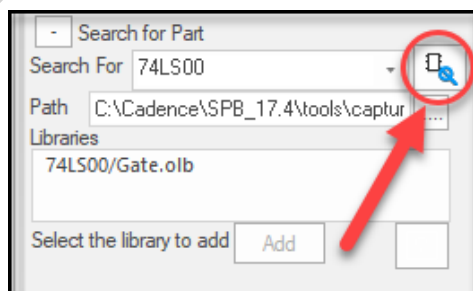
The search results are shown in the Libraries list (for example, 74LS00/Gate.olb). This means that the 74LS00 part was found in the GATE library.

6. Double-click on the 74LS00/Gate.olb entry in the Libraries list.

    The GATE library is added to the Libraries list in the Place Part window, and the 74LS00 part is automatically selected and "floating" on the cursor.

7. Add this part to the schematic.
8. To exit from Place Part mode, press <***Esc***>.
9. Click left anywhere in the page to deselect the newly added part.
10. Select ***File > Save***.
11. At this point the schematic page should look something like the following picture.
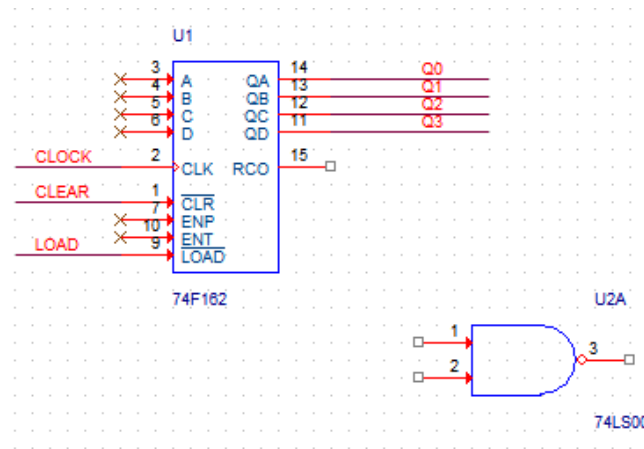
The 74LS00 part was in the Gate library. Since you did not know which library this part was in, you searched all OrCAD Capture libraries before adding the part. Once the 74LS00 was found, the Gate library was added to the Libraries list.

Add the PCB Footprint Names to the Parts

This property will be required to correctly netlist the design. Each schematic symbol must have an accompanying pc board footprint in order to correctly establish the pcb layout.

1. Select both of the parts on the current page. Use the <CTRL> LMB (left mouse button) to select both parts.
2. RMB (right mouse button) click, and from the pop-up menu select ***Edit Properties***.
3. When the Property Editor appears, find the column or row for PCB Footprint and enter the following values:

    For the 74LS00 make it SOIC14

    For the 74F162 make it SOIC16

Location Y-Coordinate	220	570
Name	INS294	INS605
Part Reference	U1	U2A
PCB Footprint	SOIC16	SOIC14
Power Pins Visible		
Primitive	DEFAULT	DEFAULT
Reference	U1	U2
Source Library	C:\CADENCE\SPB_17.4 ...	C:\CADENCE\SPB_17.4 ...

If you get an **Undo Warning** select **Yes**. This warning is letting you know that this action will change information on the schematic and once completed you cannot undo previous actions using undo button.

4. If your Property Editor does not show the properties in columns, try clicking on the **Pivot** button, which should alter the view as shown in the graphic below.
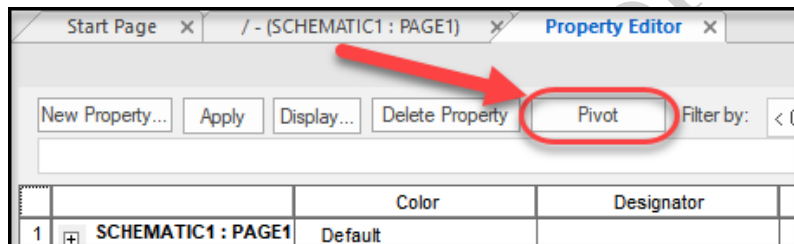
5. Close the Property Editor by clicking X at the edge of the tab.
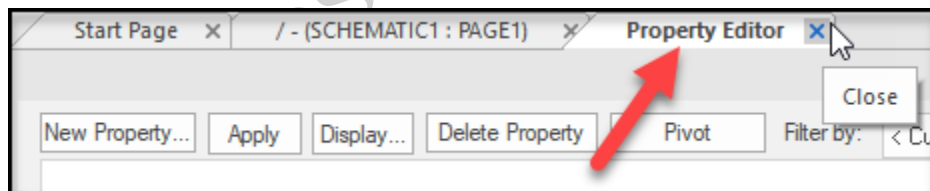
We will go into much more detail on using the Property Editor in a later lesson of this manual.

Lab 6-3: Adding and Naming Wires

Lab Objectives

After completing this lab you will be able to:

- Add wires
- Name wires
- Complete the schematic

Adding Wires

1. Make sure that the schematic window is active and select *Place > Wire*.

    You can also use the *Place > Wire* icon in the schematic toolbar (or press the *<W>* key on the keyboard).

2. Click pin **14** of part **74F162**. Move the cursor to the right a short distance, then double click to end the wire. Notice that the wire is still highlighted but that you are no longer in Draw Wire mode.

3. With the new wire still selected, press the *<F4>* key three times.

    The wire segment is repeated each time, one grid down from the original.

4. Press the *<Esc>* key twice.
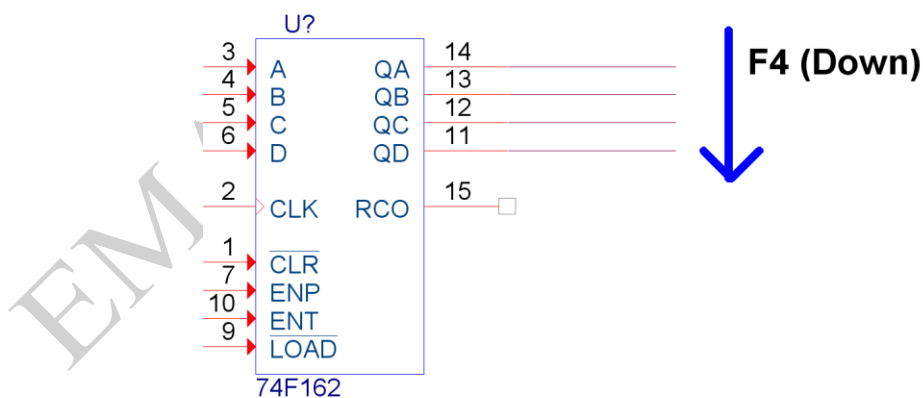
Naming Wires

1. Select *Place > Net Alias*.

    You can also click the *Place > net alias* icon , or type *<N>* on your keyboard.

2. In the Place Net Alias window, enter **Q0** and click *OK*.
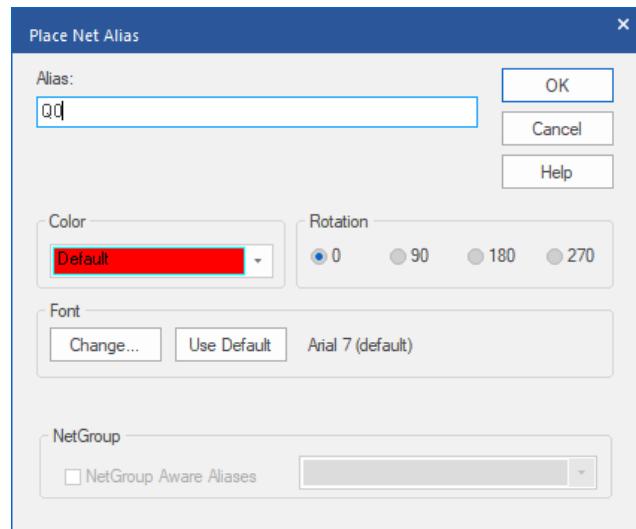
A net name box is attached to the cursor.

3. Click the wire attached to pin 14 of the **74F162** part.

Net name **Q0** is assigned to the selected wire. You are still in Place Net Alias mode.
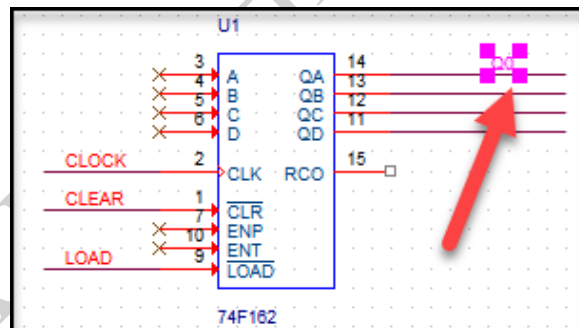
4. Successively click on the wires attached to pins **13, 12**, and **11**.

OrCAD Capture automatically increments the net names to **Q1, Q2** and **Q3**, respectively.
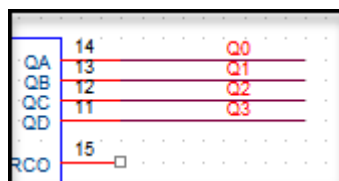
5. Press the *<Esc>* key twice to exit Place Net Alias mode.

Pressing *<F4>* to repeat the prior action does not work in *Place > Net Alias* mode. Also, if the net alias ends in a non-numeric character, the name remains the same when you place successive aliases.
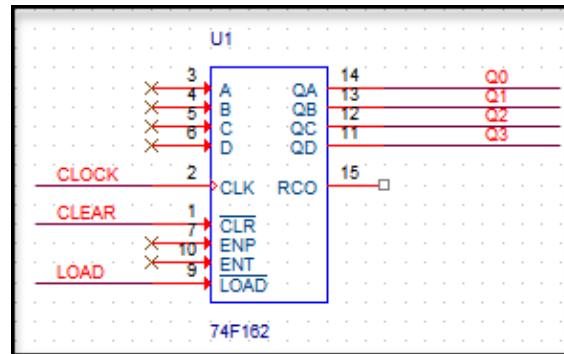
Completing the Schematic

1. Use the *Place > Wire* and *Place > Net Alias* icons to complete the schematic and connect the 74LS00 gate. (Please refer to the example shown at the beginning of this lab.)
2. Use *Place > No Connect* or the icon to add No Connect symbols to all unconnected pins of the 74F162 part (except pin 15). The schematic page should look something like the picture below.
3. Save the design.

To remove a "No Connect" symbol, click the *Place > No Connect* icon, and click over any existing "No Connect" symbol already set on a pin in the schematic.
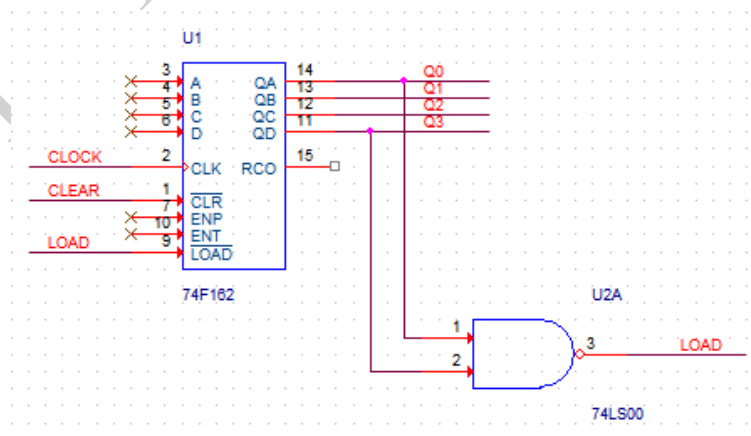
Auto-Wire (Between 2 points) (Optional)

1. In another part of the page, add 2 more 74F162 parts.

2. From the *Place* pull-down menu *Auto>Wire > Two Points* or using the icon.
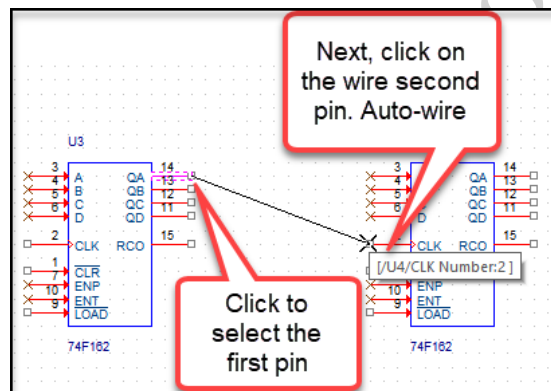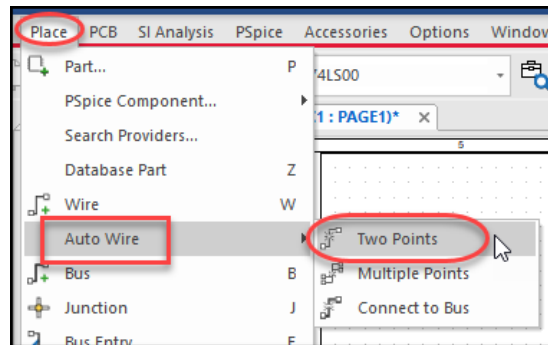
3. Select 2 pins to connect and from the pop-up menu
4. A wire will automatically connect between the pins selected. Notice the following example.
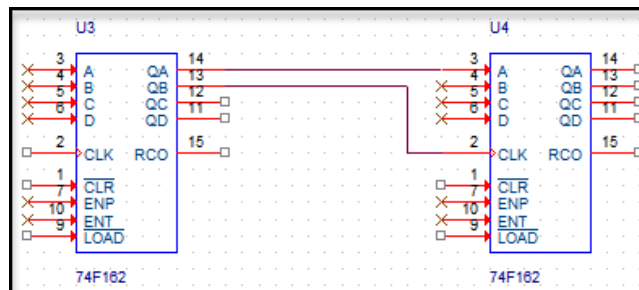
5. When you have finished experimenting with this new command, delete the 2 new 74F162 parts and wires. They will not be used in the upcoming design.
6. Save the design.

Lesson 7: Processing a Schematic Design

Lesson Objectives

After you complete this lab you will be able to:

- Assign reference designators
- Check the design for errors
- Create a netlist for OrCAD and Allegro PCB Editor or other board layout tools
- Generate a Bill of Material (BOM)

Assigning Part References

Each part in your design must have a unique reference designator. In OrCAD Capture you can assign a reference designator manually or automatically.

Each part requires a unique part reference. Part references must be assigned before you can check the design for errors or create a netlist for PCB design.

The following graphic shows reference designators on **74F162** and **74LS00** set with **"?"**.
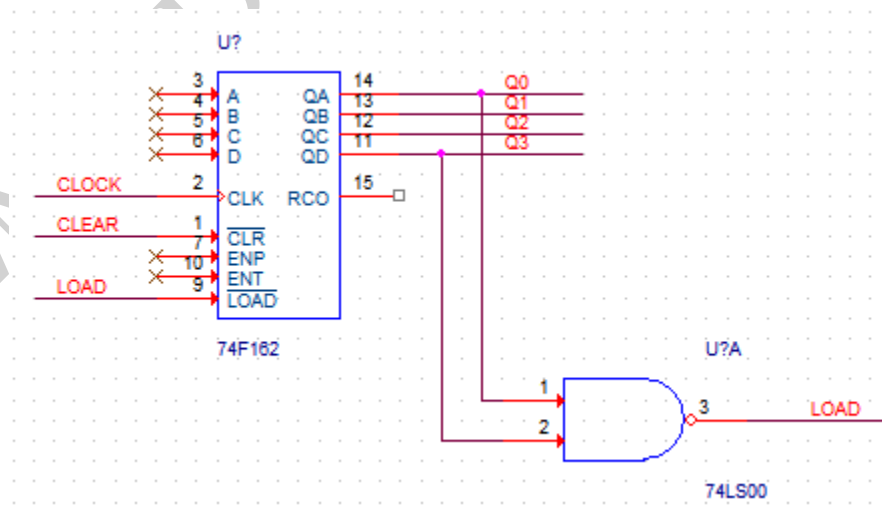
Manual Assignment after Placement
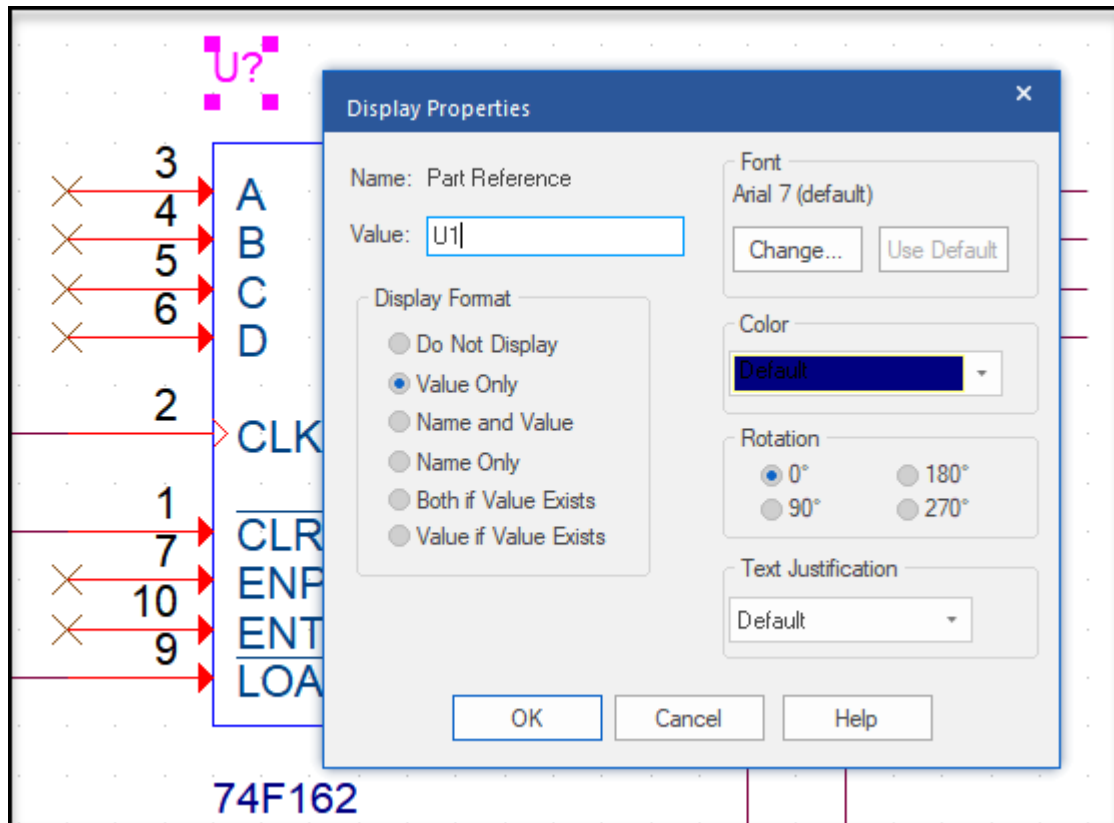
If the part has already been added, and it currently has no assigned part reference, double-click on its temporary part reference and edit its value (as shown in the above illustration). You can also use this technique to change an existing part reference.

Here are some things to remember about manual part reference assignments:

- All parts come into a design with pin outs assigned.
- All parts come into a design with a temporary part reference (for example, U? is single-section, or U?A is multi-section). You can override this temporary part reference before or after placing the part (as previously described).

Single-Section versus Multi-Section Parts
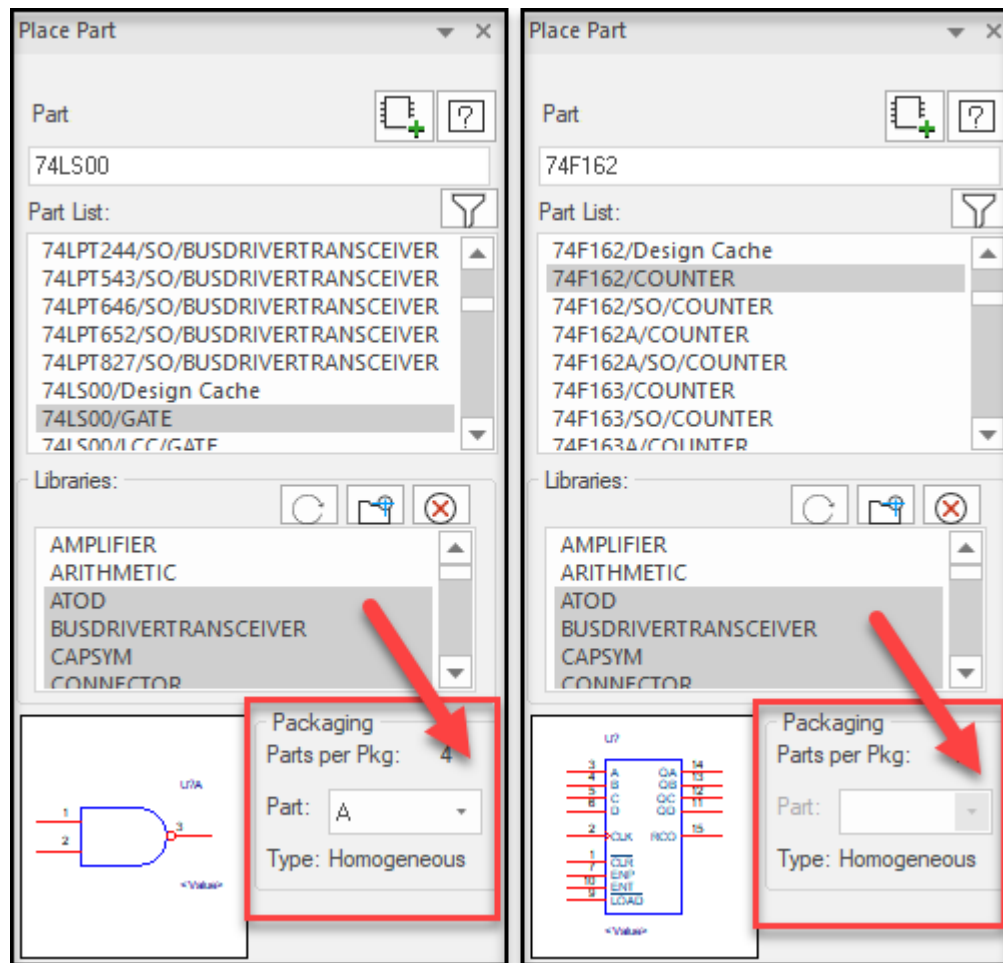
A single-section part is a part that represents the entire physical package. A multi-section part is a part that represents only a portion of the physical package. The Packaging section of the Place Part form reflects the number of sections (or slots) in the physical package.

For a multi-section (gate-level) part:

- If the Packaging section of the **Place Part** form was not used, the temporary part reference and pin outs will default to section A (repetitively).
- If the Packaging section of the **Place Part** form was used, then the temporary part reference and pin outs will coincide with the selected section. Pinots always match the section.

- If the part is homogeneous, you can manually change the section once the part is placed (for example, from U?A to U?B). This causes the pin outs to change automatically. You cannot manually assign a section that does not exist.
- If the part is heterogeneous, all you can do is change the question mark (?) to show grouping into a specific package. The pin out is locked at placement based on the section you selected in the **Place Part** form.
- When you manually edit a multi-section part reference, OrCAD Capture does not let you remove the section indicator.
- When you use the Packaging section of the **Place Part** form to select a specific section, the Annotate program does not maintain the selected section (even in Incremental mode) unless you also supply a part reference.

Automatic Assignment during Placement
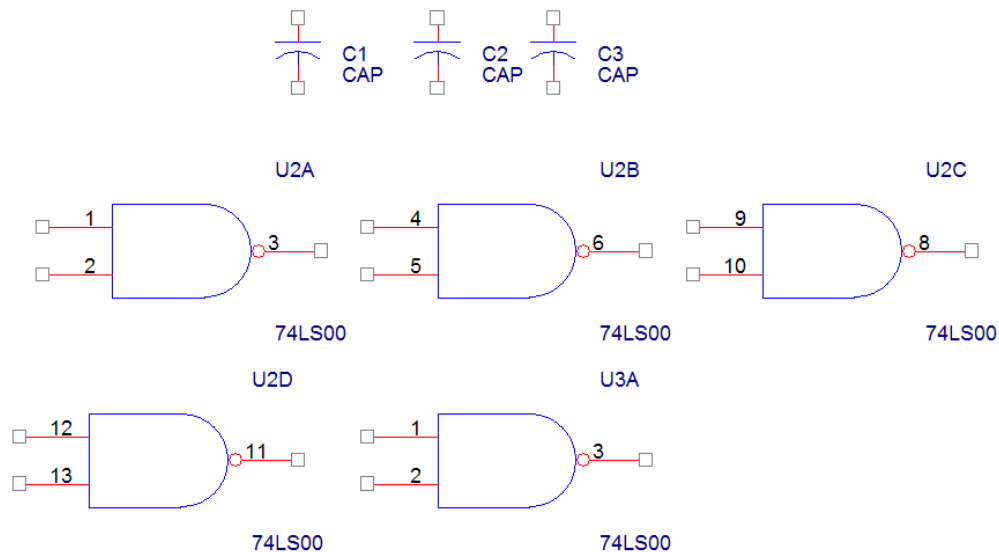
You can set your user **Preferences** to automatically assign a part reference as each part is placed in the design. This is set in the **Auto Reference** option.
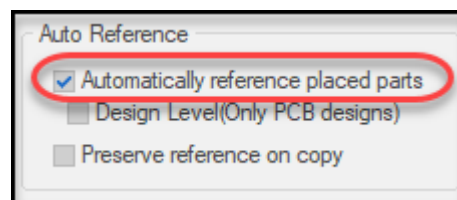
This works well for single-section parts (for example, capacitors, discrete resistors, and most digital logic devices), as well as for multi-section parts. See the example above.

For multi-section parts, OrCAD Capture does not assign part references in a way that optimizes physical package counts in the design, only the next number available in the sequence. For this example, U2A, U2B, U2C, U2D represent parts or gates of the same physical package. These less-than-optimal part reference assignments could be manually edited, or automatically overwritten (see the next section, *Automatic Assignment after Placement*).

Automatic Annotation
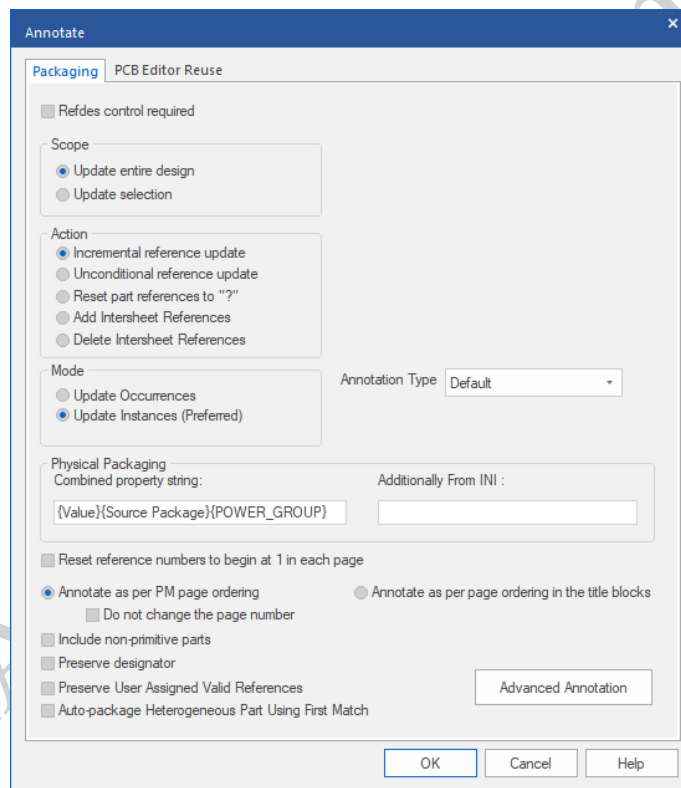
NOTE: When Annotating Heterogeneous parts, use the option labeled: **Auto-package Heterogeneous Part Using First Match**. This will be the most efficient way of packaging the heterogeneous symbol.

To automatically assign part references after placing parts, use the ***Annotate*** icon in the main toolbar , or the ***Tools > Annotate*** command.

In Incremental mode, the Annotate program will not change a manually or auto assigned part reference, even if duplicate part references exist. Incremental mode only processes parts that have no assigned part references. Incremental mode starts with the next highest part reference. (For example, if the design already has a U100, then the next part reference will be U101.)

In Unconditional mode, the Annotate program overwrites all existing part references, regardless of how they were assigned. Unconditional mode optimizes all multi-section parts into a minimum number of physical packages. Unconditional mode starts with U1, R1, C1, and so forth, and works its way through the sequence, processing parts in the order they appear in the schematic page (left to right, top to bottom). This mode eliminates all duplicate part references in the design.

The option **Annotation Type** allows you to select the general direction on the schematic pages for Annotation.

To automatically annotate heterogeneous parts, these parts must have an attached user property to show package grouping (for example, PARTGROUP). This property must be entered in the Combined Property String box at the bottom of the Annotate window.

Cadence Allegro Reuse

(This option is NOT currently available for OrCAD PCB Editor.) The options in this tab are related to the OrCAD Capture/Cadence Allegro design flow. You can create and reuse modules in both the OrCAD Capture and Allegro environments. This means you can store physical placement and signal routing data created in Allegro together with the OrCAD Capture schematic information and reuse it in another design. This can be very helpful when different designs share functional logic blocks or when a design contains redundant circuitry.

Design Rules Check (DRC)
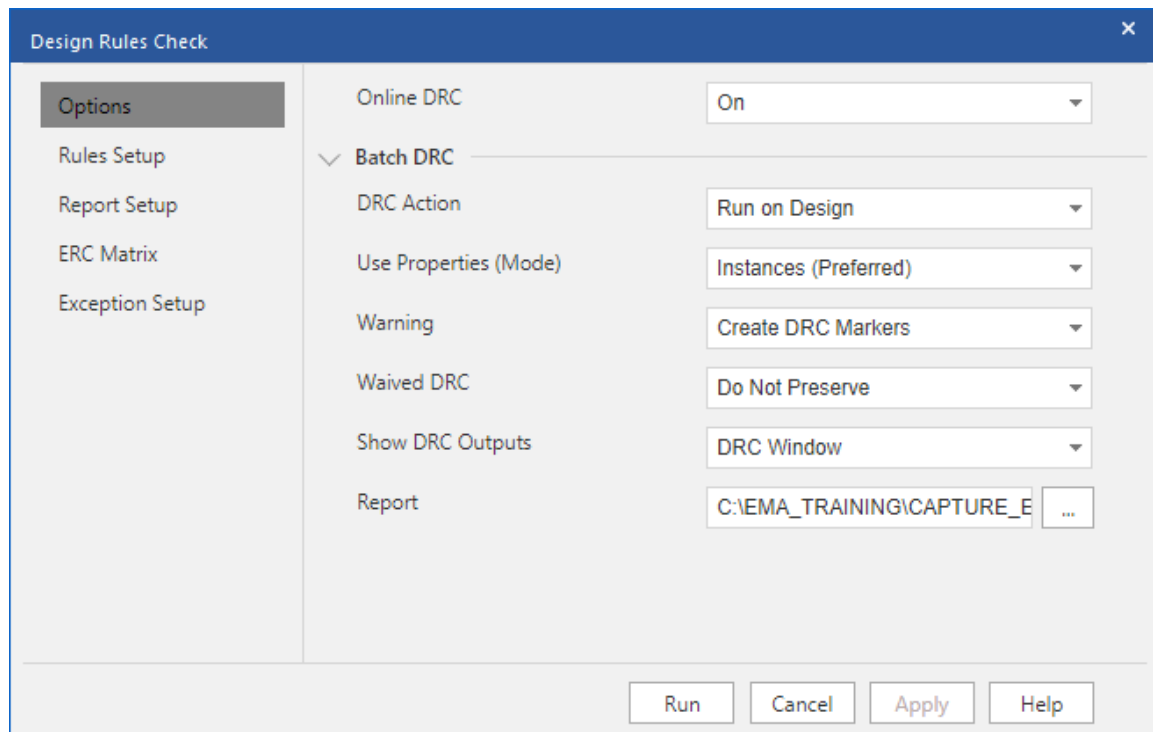
To check the design for errors, use the **DRC** icon in the main toolbar, or the **Tools > Design Rules Check** command. You must use the DRC program to flag the connectivity errors in your design.

OrCAD Capture does not prevent you from assigning more than one signal name to the same wire, or physically shorting two different nets together.

The DRC program flags nets with no driving source, and nets with fewer than two connections. When two nets have been shorted together, OrCAD Capture simply uses the net that has the lowest alphanumeric value. No error or warning message is generated by the DRC program. However, to find shorted nets, look at the list of net names created by the **Report All Net Names** option. OrCAD Capture does not prevent you from manually assigning duplicate or invalid part references. The Annotate program ignores these errors in Incremental mode (or resolves them in **Unconditional** mode). You must use the DRC program to flag manual annotation errors.

For example, the DRC program flags invalid packaging (such as unlike device types assigned to the same physical package), as well as duplicate part references.

The design needs to be annotated before you can run **Design Rules Check**.

When the **Design Rules Check** window opens there a series of tabs across the top of the window for all the settings that are available to run with a DRC check.
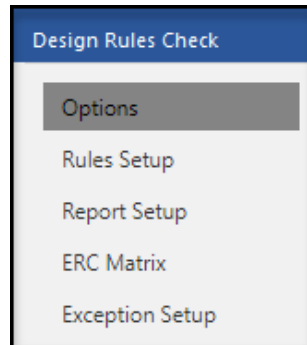
Design Rules Options

The first tab is shown in the window above. Here you decide if you want to:

- Add DRC markers for warnings besides the errors
- Run this DRC against a "flat" design (Instance), or a hierarchical design (Occurrences)
- Choose to preserve waived DRCs
- You can also flag it to show you the output of the report as soon as it finished. (**View Output**).

Rules Setup

The Electrical Rules drop down allows you to set checks for:

- Single node nets
- Pin conflicts and net with no driving source
- matching off-page connections and hierarchical ports
- unconnected bus nets
- off grid objects
- misleading tap (bus) connections

The Physical Rules drop down allows you to set checks for:

- Power pin visibility
- missing PCB Footprint
- missing pin numbers
- unused part packages
- identical part references
- invalid packaging

The Custom Rules drop down allows you to set checks for:

- Device Pin mismatch
- Hanging wires
- Overlapping wires
- Part reference mismatch
- Port pin mismatch
- invalid packaging

ERC Matrix
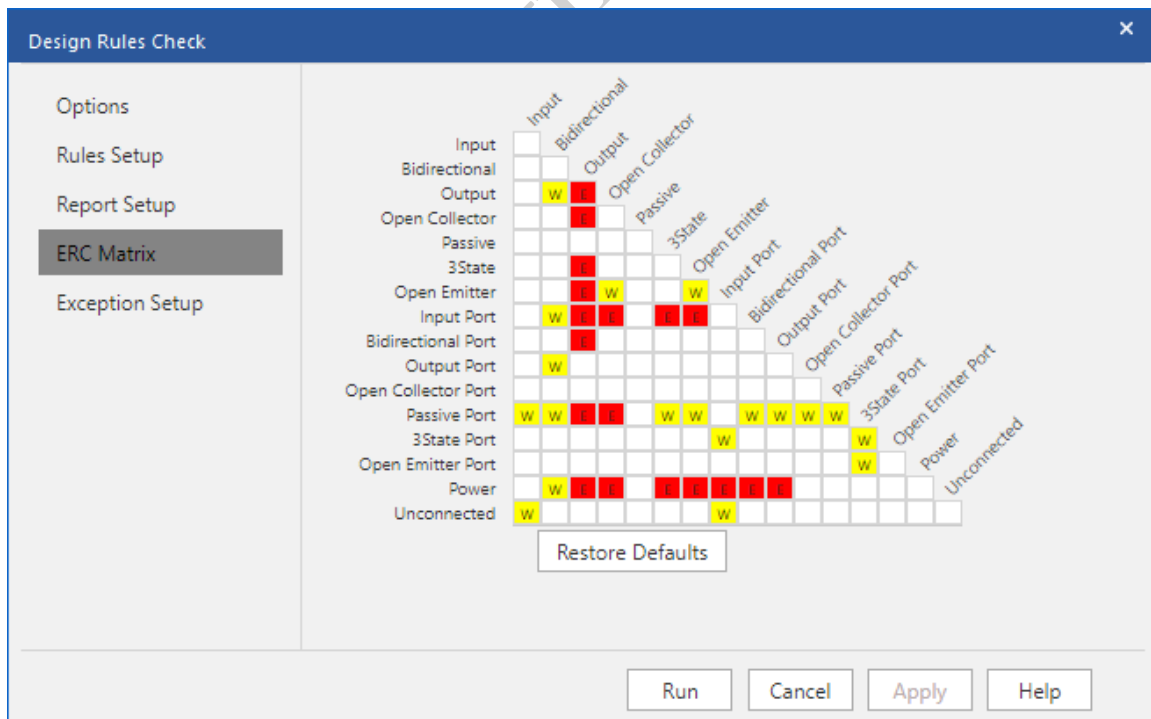
The *Electrical Rules Check* flags invalid pin-to-pin function connections found in the design. The ERC Matrix tab shows a matrix of all possible pin-to-pin combinations. Each pin-to-pin combination can be defined as valid (no error or warning), or invalid (error or warning flag generated).

In addition, the matrix includes a row used to define the error condition for various types of unconnected pins.

The matrix buttons have three toggle positions as identified in the following table.

Icon	Meaning
	The pin combination represented by this matrix position will not be flagged (the pin-to-pin connection is logically acceptable).
W	The pin combination represented by this matrix position will be flagged as a warning. Turn on the *Create DRC Markers for Warnings* option in the DRC window to see these flags in the schematic.
E	The pin combination represented by this matrix position will be flagged as an error.

The severity of the ERC Matrix settings has no effect on your ability to create a netlist. The ERC Matrix settings are stored in the Capture.ini file. Use the *Restore Defaults* button to reset the matrix.

At this point in time, anytime you see **Occurrences** or **Instances** in any of the windowed selections use the following as a guideline:

- Instances = "flat" designs
- Occurrences = hierarchical designs

The DRC Report
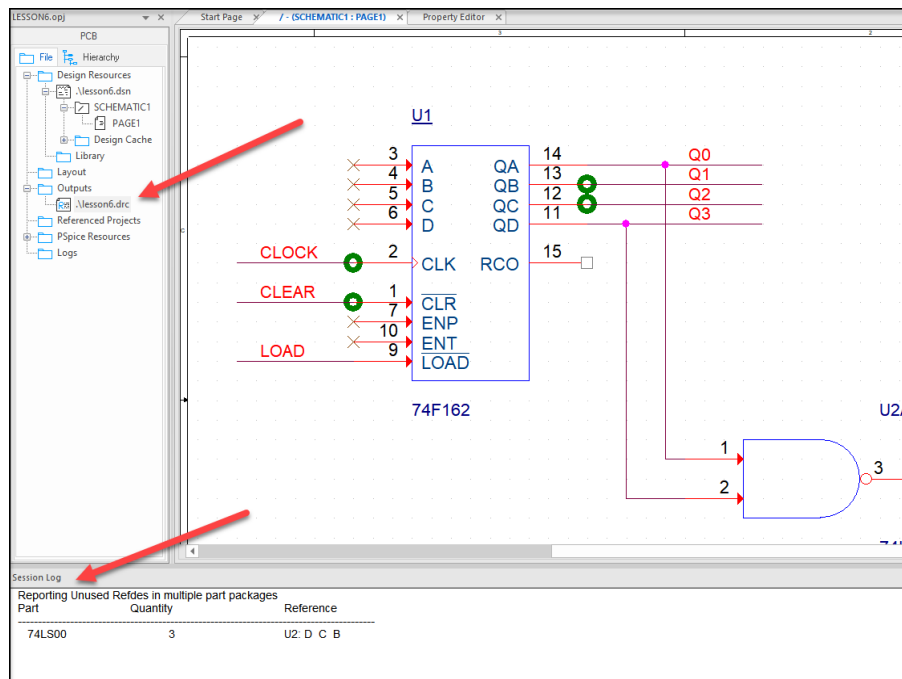
The **Design Rules Check** generates a report listing all the objects checked, and any violations encountered. This report is listed in the Outputs folder of the Project Manager window. (Double click on the file name to view the report in a new tab.)

The results of a DRC check are also displayed in the OrCAD Capture Session Log.

Error Markers
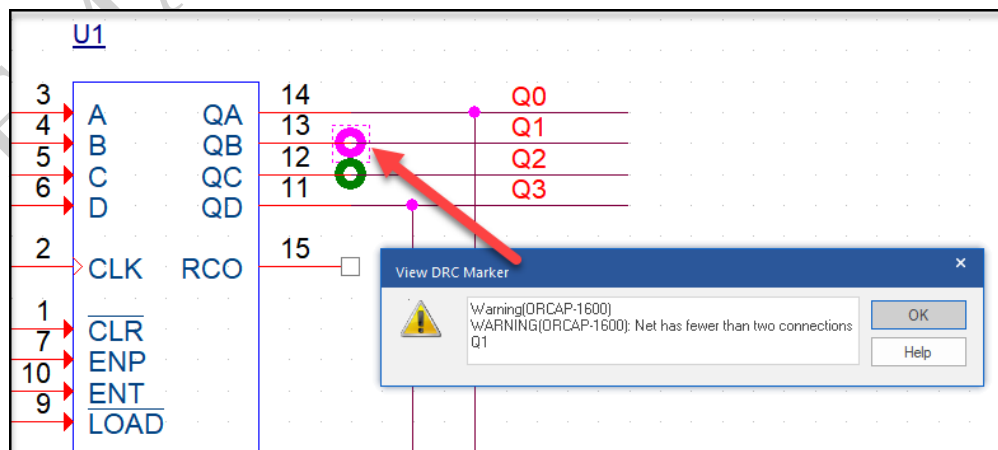
The **Design Rules Check** marks schematic pages with warning and error markers. You can double click on a DRC error marker to see a description of the problem.

If you save the design, the DRC markers are also saved. Each time you start the DRC program, it automatically clears existing error markers and creates new flags. (The design must be saved to retain the new markers.)

You can delete the DRC markers interactively or delete them automatically by using the **Delete Existing DRC Markers** option in the DRC setup window.

OrCAD and Allegro PCB Editor Netlist
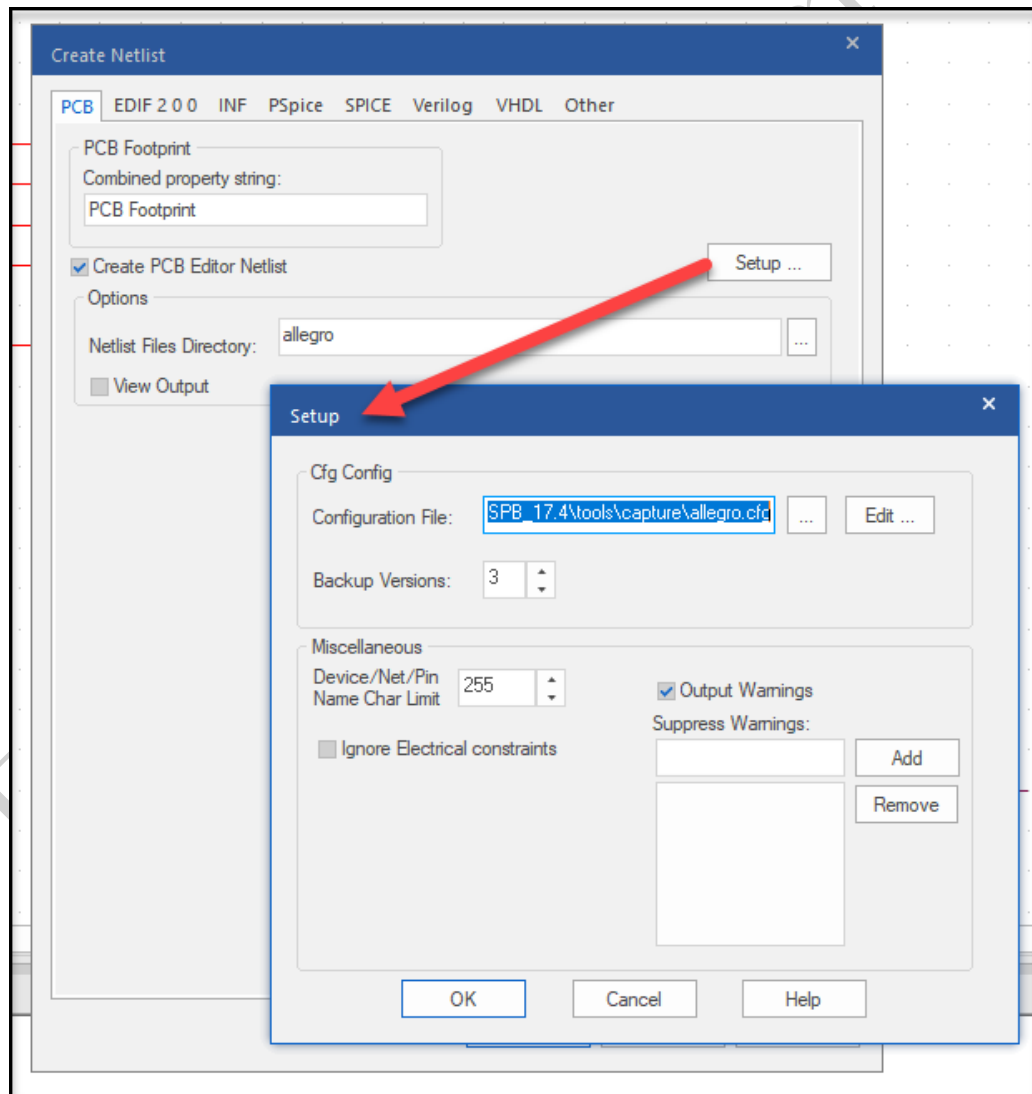
OrCAD Capture uses a configuration file to control part association with net properties. This configuration information which is included in the OrCAD and Allegro PCB Editor netlist is stored in the application installation directory by default. However, you may use the *Setup* button located in the *Tool > Create Netlist > PCB Editor* tab of the dialog box to specify an alternate path to this file.

As you create an OrCAD and Allegro PCB Editor netlist, three netlist files are created and stored in the directory location you specify in the Netlist Files Directory field. In the above illustration, the netlist files are created in the *allegro* subdirectory within the current project.

The OrCAD and Allegro PCB Editor routing process creates a SPECCTRA.dsn file that should be stored separately from the <design>.dsn file associated to the OrCAD Capture schematic. It is important to take note of the separate directory locations.

The three files created when you create an OrCAD and Allegro PCB Editor netlist are listed in the Project Manager window within the *Outputs* folder.

- pstchip.dat: This file contains the library definitions of all the devices in the OrCAD Capture design.
- pstxnet.dat: This file defines all the connections in the design. This file also contains net properties.
- pstxprt.dat: This file is a parts list. All part properties are sent to the OrCAD and Allegro PCB Editor in this file.

The Create Netlist window also lets you load the three netlist files into an OrCAD and Allegro PCB Editor design.

You can use an alias in the configuration file to map an old Allegro PCB Editor property into a new one. If you have a previous Allegro design that uses the *DELAY_RULE* property, you can map this property to another property name (for example, *DELAY_RULE = PROPAGATION_DELAY*).

Common Issues and Errors

The netlister has an Allegro PCB Editor-specific error checking mechanism that generates a list of Allegro PCB Editor formatting errors and halts the netlist generation. The error messages are written to the session log. A netlist.log file is also written to the current working directory.

The following table identifies and describes common problems associated with creating or loading an OrCAD and Allegro PCB Editor netlist. For additional information, please see **Help > Known Problems and Solutions** within the OrCAD Capture application.

Error Message	Description
Missing PCB Footprint	Every part in the design must have an assigned PCB Footprint property to ensure operation of the netlister. If you attempt to run the netlister before PCB footprints are assigned, the process will abort.
	The footprint property can be defined through either the library or parts in the design.
No Connect Pins	A pin with a No Connect symbol placed on it will be tied to net name NC in the netlist. These are unconnected pins, as opposed to No Connect pins.
	No Connect pins are non-electrical pins on the physical package. No Connect pins are usually not present on the OrCAD Capture symbol. However, these pins must be defined, or else errors occur when you load the netlist into the OrCAD and Allegro PCB Editor. Use the NC property to supply a list of No Connect pin numbers for a part.
	No connect pins can be added to the OrCAD Capture symbol, but they must have unique pin names (for example, NC1, NC2, and so on). Pins that appear on the symbol should not be included in any NC property.
	OrCAD and Allegro PCB Editor package symbols can contain mechanical pins, or pins without numbers (for example, the mounting holes on a connector). A part in the OrCAD Capture library would not need an NC property to account for these mounting holes.
Duplicate Pin Names	If two or more visible pins have the same pin name, the netlister will abort. However, if the pin type is POWER, the netlister will automatically alter the duplicated pin names to make them unique.
Illegal Characters	The netlist does not allow the use of single quotes or exclamation marks in any pin, part, or net names. It is best to use A-Z, 0-9, and the underscore character as the only punctuation.
Missing Pin Number	Every pin must have an assigned pin number. The netlister will abort if you attempt to use it before completing pin number assignments.

Creating an OrCAD and Allegro PCB Editor Netlist

You can create an OrCAD and Allegro PCB Editor netlist using the OrCAD Capture commands *Tools > Create Netlist > PCB Editor* or by using the icon 📄. Then import the three default OrCAD and Allegro PCB Editor net files into OrCAD and Allegro PCB Editor.

In your <install directory>, in the OrCAD Capture directory, is a configuration file that lists the properties that the Netlister will extract from the OrCAD Capture Schematic for both Parts and Nets. This file is the Allegro.cfg file. These are also the properties that you will find listed under the *Cadence-Allegro* property filter in the OrCAD Capture schematic editor. (See how to edit or create property filters in Lesson 9 of this manual.)

General Rules for Netlisting

The OrCAD and Allegro PCB Editor netlister extracts 3 ASCII files: PSTCHIP, PSTXNET, and PSTXPRT.

- Make sure that ALL parts have a *PCB Footprint* value.
- Pin counts of the OrCAD Capture part must match the physical symbol (footprint).
- ALL pins must have a pin name and number in the OrCAD Capture part.
- To account for UNUSED pins (such as component mounting holes) use the *NC* part property to list them or make the pins in the physical symbol (footprint) mechanical pins.
- There are attribute length limits and character limits for the OrCAD and Allegro PCB Editor. See the following chart.

Field name	Length	Acceptable characters
package name *	255	A to z, 0 to 9, dash (-) and underscore (_)
device type	30	All except ! and '
function designator *	255	All except ! and '
reference designator	30	All except ! and '
pin number	30	All except ! and '
pin name *	255	All except ! and '
net name *	255	All except ! and '
property value	79	All except ! and '
tolerance	1023	All except ! and '
user part number	1023	All except ! and '
value	1023	All except ! and '

* In the OrCAD and Allegro PCB Editor the default maximum number of characters is 30. You can set the initial length for new designs to a maximum length of 255 by using the *allegro_long_name_size* environment variable (Choose *Setup > User Preferences*). You can change the size in existing designs by choosing *Set > design Parameters* and specifying a new maximum for the *Long Name Size* in the *Design* Tab.

Remember, it is recommended that you avoid the use of spaces in directory names, design names and net names.

Common Errors

View these error messages in the OrCAD Capture Session Log.

- PCB Footprint missing in OrCAD Capture
- Footprint does not match library (symbol) name
- Cannot find symbol - psm or pad path incorrect
- Pin mismatch - pin numbers or total pin counts do not march from Footprint to component symbol
- Pin name not allowed
- Pin Number missing in OrCAD Capture part.

Lab 7-1: Assigning Reference Designators

Automatic Assignment

1. In the Project Manager window, highlight .\LESSON6.dsn.

2. Select **Tools > Annotate** (or click the [icon] icon in the main toolbar).
3. In the Scope choose Update Entire Design
4. In Action, select "Incremental Reference Update".
5. In Mode, make sure it is set to "Update Instances (Preferred)".
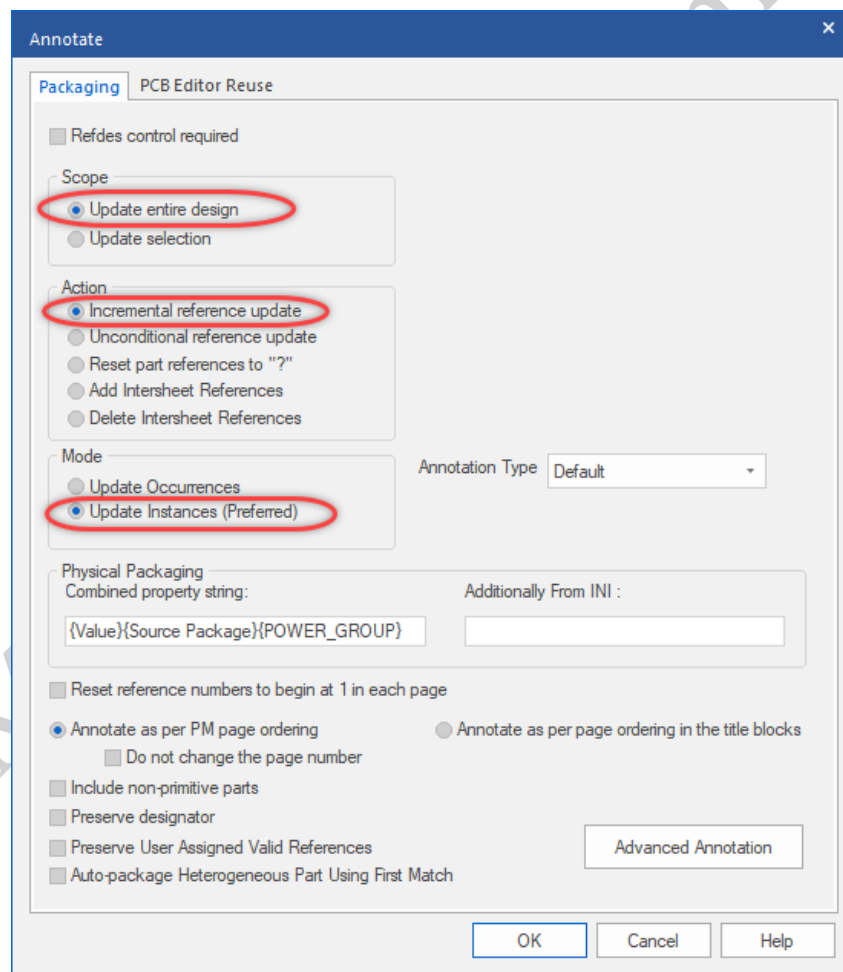
6. Click **OK**. When prompted with the **Undo Warning** window, click **OK** again to continue.
7. Your design is annotated and saved.

NOTE: If you had heterogeneous type parts in your schematic, it is recommended that you choose the option towards the bottom of the annotate window to: ***Auto-package Heterogeneous Part Using First Match.*** This option will utilize the most efficient packaging method for all the gates in a heterogeneous part.

View the auto-assigned reference designator (U2A). Observe that the manually assigned reference designator was not changed. This is because you ran the annotation program in Incremental mode.

The ***Options > Preferences > Miscellaneous*** tab contains an Auto Reference checkbox (If checked, this option auto-assigns reference designators as you place parts). Design Rules Check

Lab 7-2: Design Rules Check

Lab Objectives

After completing this lab you will be able to:

- Verify design compliance to design rules
- Correct warning on unconnected pins

Running the Design Rules Check

1. In the Project Manager window, click **.\LESSON6.dsn**.

2. Select **Tools > Design Rules Check** (or click the ⬚ icon in the main toolbar).

   The Design Rules Check window appears.

3. In Design Rules Options make sure the following are selected:
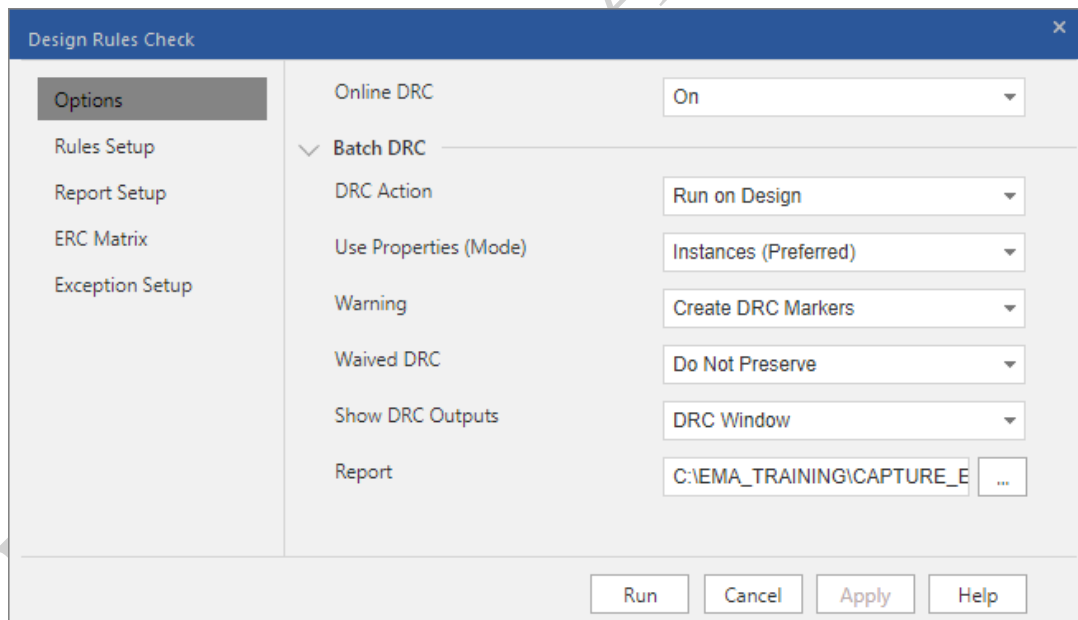
- DRC Action: Run on Design
- Use Properties (Mode): Instance (preferred) (for Flat-designs)
- Warning: Create DRC markers for warnings
- Waived DRC: Do not Preserve
- Show DRC Outputs: DRC Window
- Report saved in your EMA_training directory

4. Select the Rules Setup tab and set the following:
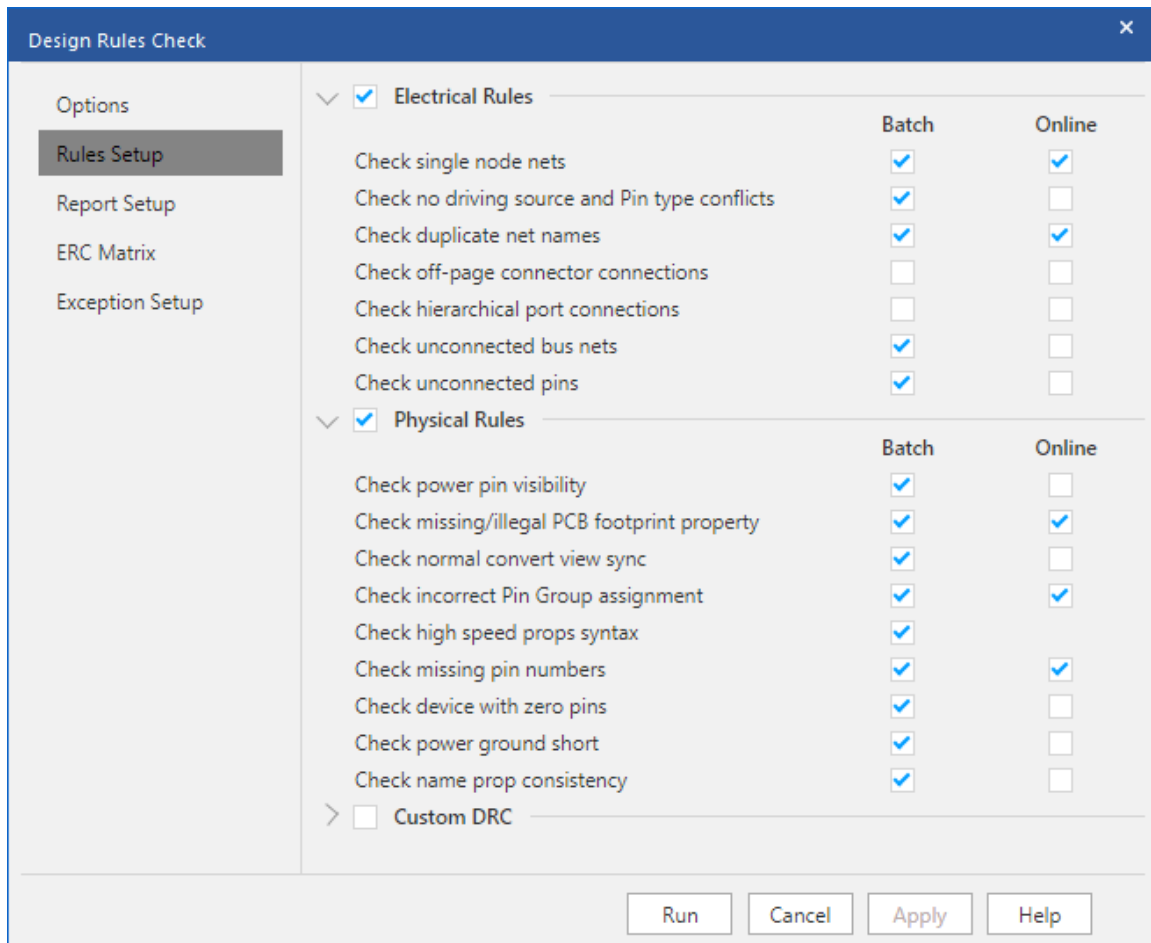
- Check single node nets
- Check no driving source and Pin type conflicts
- Check duplicate net names
- Check Power pin visibility
- Check missing/illegal PCB Footprint property
- Check missing pin numbers
- Check Power Ground short

Under Report Setup set:
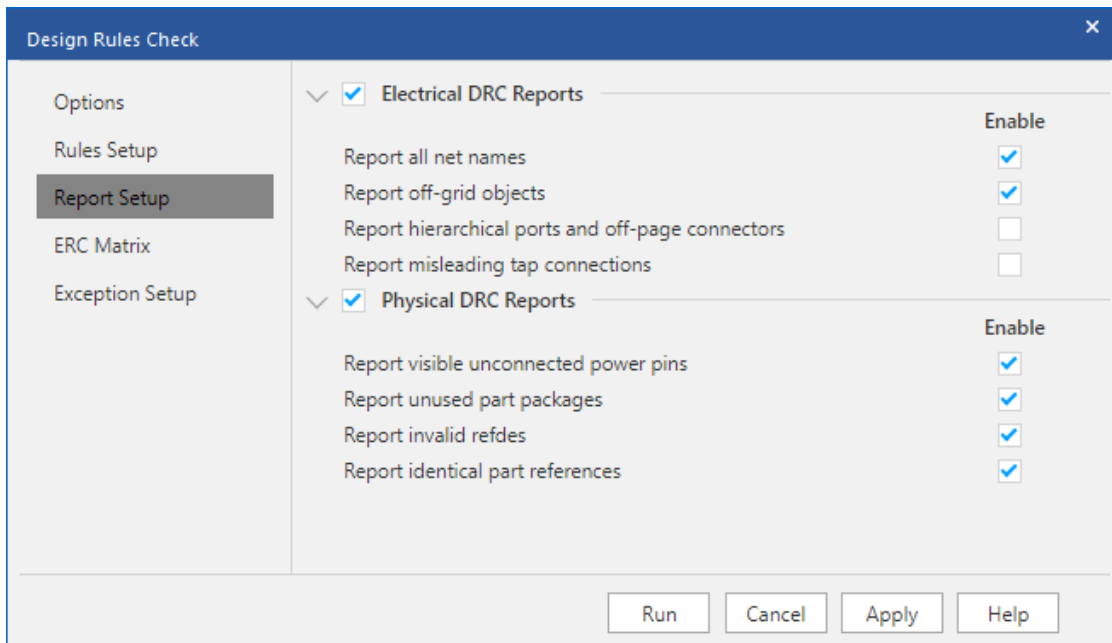
5. Click the **ERC Matrix** tab at the top of the window.
6. Set the matrix buttons along the bottom row to **W**, as shown below.
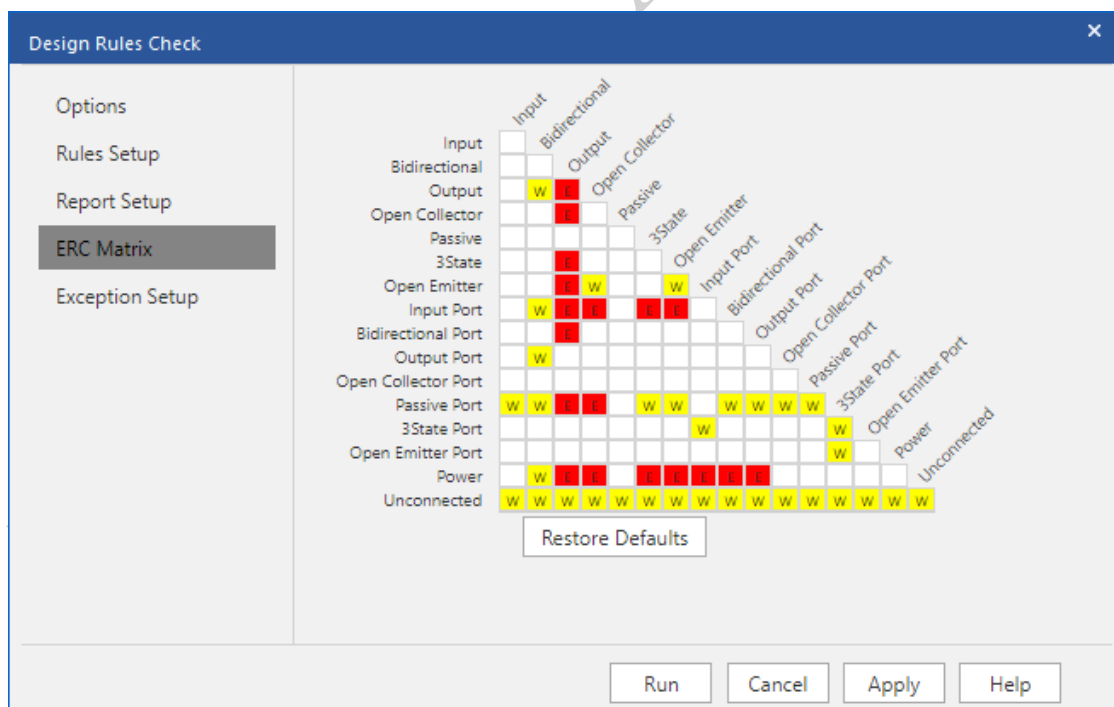
This causes all unconnected pins to be flagged as warnings.

7. Click **OK** to check the schematic.

The DRC report is created. The report contains warnings for an unconnected pin, nets with no driving source, and nets with fewer than two pins (one-pin signals) and missing Footprint names (if you forgot to set).

Severity	DRC Type	Description	Detail	Location	Page	Schematic	Waived
Warning	Electrical	WARNING(ORCAP-1600): Net has fewer than two connections	Q2	SCHEMATIC1, PAGE1 (5.20, 2.50)	PAGE1	SCHEMATIC1\	No
Warning	Electrical	WARNING(ORCAP-1600): Net has fewer than two connections	Q1	SCHEMATIC1, PAGE1 (5.20, 2.40)	PAGE1	SCHEMATIC1\	No
Warning	Electrical	WARNING(ORCAP-1600): Net has fewer than two connections	CLEAR	SCHEMATIC1, PAGE1 (4.00, 3.00)	PAGE1	SCHEMATIC1\	No
Warning	Electrical	WARNING(ORCAP-1600): Net has fewer than two connections	CLOCK	SCHEMATIC1, PAGE1 (4.00, 2.80)	PAGE1	SCHEMATIC1\	No
Warning	Electrical	WARNING(ORCAP-1608): Net has no driving source	CLEAR	SCHEMATIC1, PAGE1 (4.00, 3.00)	PAGE1	SCHEMATIC1\	No
Warning	Electrical	WARNING(ORCAP-1608): Net has no driving source	CLOCK	SCHEMATIC1, PAGE1 (4.00, 2.80)	PAGE1	SCHEMATIC1\	No

DRCs | Online DRCs | Command Window | Session Log

8. View the DRC report window.

   These errors are also flagged with DRC markers on the schematic page.

9. To get information about an error, double click on the DRC marker.

Alternate Method to Locate DRC Markers

1. Select the Project Manager window and highlight the design name.
2. From the *Edit* pull-down menu, select *Browse > DRC Markers*.

   A "Browse" window will open with a list of all the DRC markers, their locations, and the error.

3. Double click on the error message in the left-hand column for "Unconnected Pin".

   The page with the error marker will open and the error marker will be highlighted for you.

Fixing the Unconnected Pin Warning

1. Click the *Place no connect* icon and add a "no connect" symbol on pin **15** of the 74F162.
2. Save the design.
3. Check the design again (the design file must be selected in the Project Manager window).
4. Review the new DRC report and observe that the unconnected pin warning is gone.

   Use the **No Connect** symbol to notify the error checker when pins are intentionally left unconnected.

5. Close the report.
6. In the schematic, observe that the error marker on pin 15 of U1 has been cleared.

   Please do not fix the remaining errors. These problems are addressed in the next lesson: **Netlist for PCB Editor**

Lab 7-3: Netlist for PCB Editor

Lab Objectives

After completing this lab you will be able to:

- Create a netlist
- View the netlist
- Close the schematic and project
- View the system files

Creating a Netlist

1. With the design file highlighted in the Project Manager window, select ***Tools >***

   Create Netlist (or click the icon in the main toolbar ⬚).

   The Create Netlist window appears.

2. Explore the multiple choices available by selection some of the tabs in this window.
3. Click the tab for PCB Editor. The following window opens.
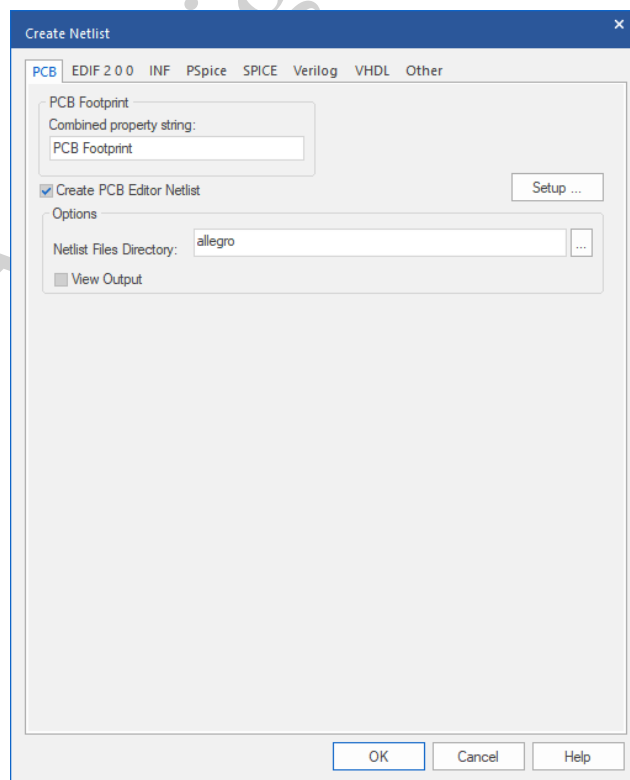
4. You do not need to change anything. The defaults are acceptable for this design. Click **OK**.

5. If asked, click **OK** when prompted about the design being saved prior to netlisting.

6. If prompted to create the "Allegro" directory, click **yes**.

OrCAD Capture creates a netlist for the OrCAD and Allegro PCB Editor in the "Allegro" sub-directory which consists of 3 files referred to as ".DAT" files.

pstxnet.dat

pstxchip.dat

patxprt.dat

Viewing the Netlist

1. In the Project Manager window, double click on one of the ".dat" files (under the Outputs folder).
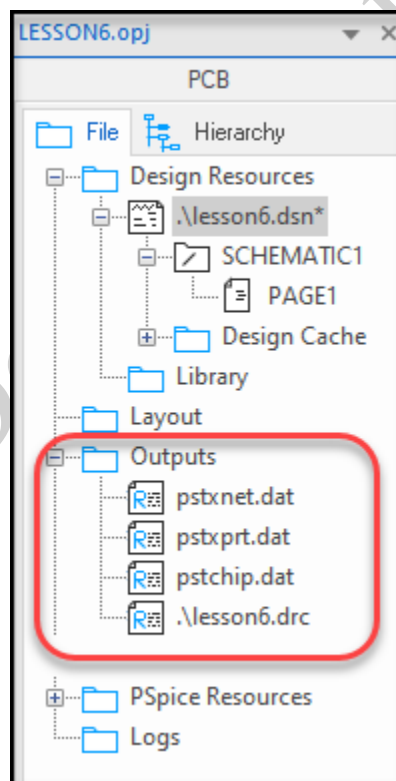
2. When you are finished, close the netlist window.

Closing the Schematic

1. Click in the schematic window to make it active, and select *File > Close* or click the **X** next tot the schematic page tab.
2. If it asks you to save, do so.

Closing the Project

1. Select *File > Close*.

   The main session window is still running.

2. A project can also be closed by clicking the **X** in the top right of the Project Manager Window

Viewing the System Files

1. Use Windows Explorer to navigate to the C:\EMA_Training\Capture_Essentials\Designs\LESSON6 directory.
2. Notice the files stored in your project folder, and their various extensions.

Lab 7-4: Creating Bill of Materials

After completing this lab you will be able to:

- Create a Bill of Materials

Create a Bill of Materials

1. Highlight the design name in the Project Manager window.
2. Select **Tools – Bill of Materials** or press the icon**.**
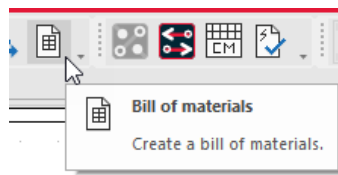
The Bill of Materials window appears.

3. Click **Place each part entry on a separate line** (checkbox **ON**).
4. Click View Output (checkbox ON).
5. (Alternate) Instead of selecting **View Output**, select the **Open in Excel** to have the BOM output open directly in Excel.

Bill of Materials

Scope
- ● Process entire design
- ○ Process selection

Mode
- ● Use instances (Preferred)
- ○ Use occurrences

OK

Cancel

Help

Line Item Definition

Header:

Item\tQuantity\tReference\tPart

Combined property string:

{Item}\t{Quantity}\t{Reference}\t{Value}

☑ Place each part entry on a separate line   ☐ Open in Excel

Include File

☐ Merge an include file with report

Combined property string:

{Item}\t{Quantity}\t{Reference}\t{Value}

Include file:

C:\EMA_TRAINING\CAPTURE_ESSENTIALS\   Browse...

Report

Report File: ☑ View Output

C:\EMA_TRAINING\CAPTURE_ESSENTIALS\D   Browse...

6. Click **OK**.

The Bill of Materials report is displayed. Observe that the report contains four columns of data; Item, Quantity, Reference, Part.

Close the report.

Lesson 8: Building a Multi-Sheet Schematic

Lesson Objectives

After you complete this lesson you will be able to:

- Create a new project
- Create a Bus and use off-page connectors
- Copy from one design to another

Multi-Sheet Connectivity

Two wires with the same net alias are automatically connected by name when they appear on the same page. But when two such wires appear on different pages, you must add **off-page connectors** to each page to establish connectivity between the two wires.

Off-Page Connectors *do not connect nets from one hierarchical folder to the next*. You must use a *Hierarchical Port* for this.
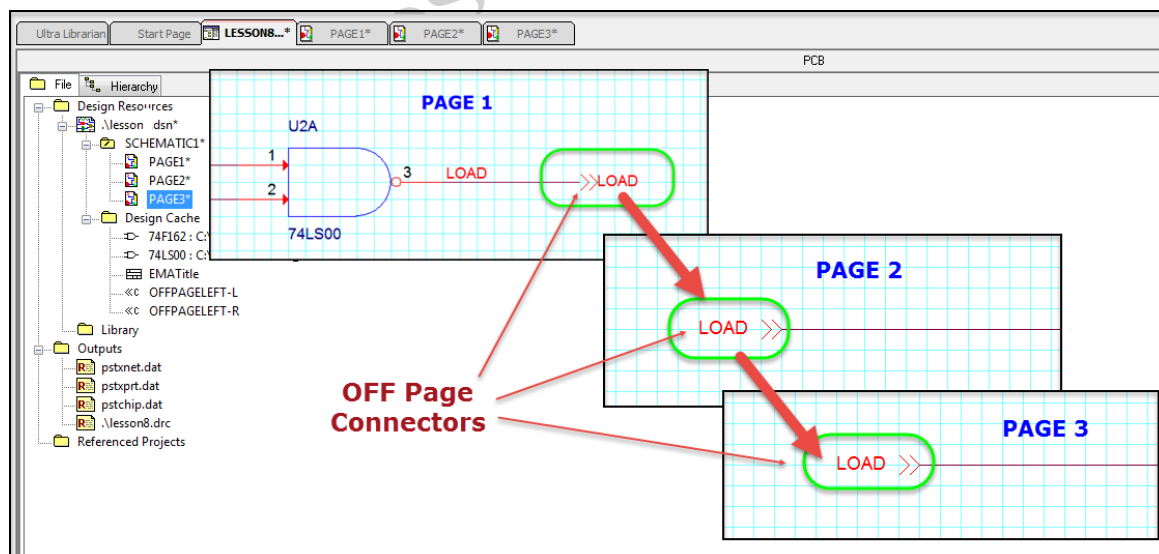
Each off-page connector you add must be named. This name identifies the net that needs to be connected to another page in the design. ("Any wire attached to a symbol, assumes the name of the symbol...")

The name of the off-page connector overrides the net alias of the wire to which it is attached. When an off-page connector is tied to an unlabeled wire, the wire will inherit the off-page connector name.

Off-Page Connectors
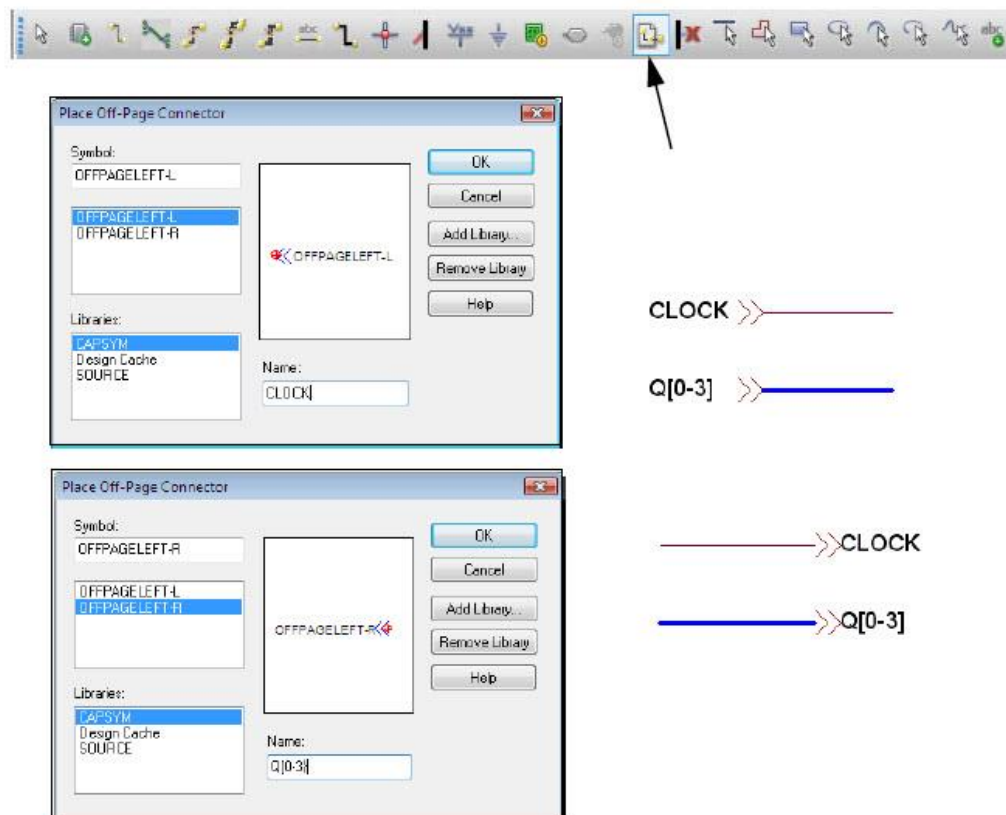
You must use the *Place off-page connector* icon in the schematic toolbar to add off-page connectors to the design. There are two off-page connector symbols located in the CAPSYM.OLB library.

When placing an off-page connector, you need to consider two things:

- The direction in which you want the off-page connector to point
- The location of the pin (connection point) on the off-page connector

The name of the off-page connector symbol embodies both direction and pin location information.

For example, the *OFFPAGELEFT_R* symbol in the CAPSYM.olb library is an off-page connector that points to the left, with a pin on the right. The other off-page connector in the CAPSYM library is called *OFFPAGELEFT-L* (it points to the left, and its pin is also on the left).

By default, both of the off-page connector symbols point to the left. If you draw a schematic with input signals on the left side of the page and output signals on the right side, you will probably rotate both these off-page connector symbols before placing them in the design (as demonstrated in the previous illustration).

From a logic standpoint, off-page connectors do not define any directional information about the nets to which they are attached; they just specify the net name only.

Power/Ground Symbols
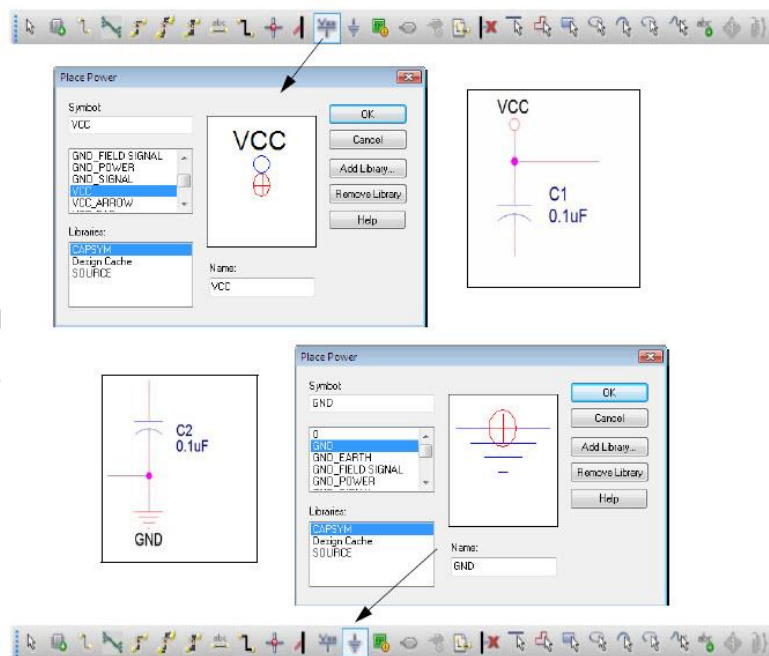
You can use the *Place power* and *Place ground* icons in the schematic toolbar to add power and ground symbols to the design. These symbols are contained in the CAPSYM.OLB library. All symbols in the CAPSYM.OLB library can be added using the schematic toolbar icons.

When you click on the *Place power* and *Place ground* icons, a dialog appears in which to enter parameters. Use the Name field in the dialog to specify the voltage name.

All power and ground symbol connections represent global nets. This means that any wires tied to voltage symbols having the same name will be connected together, regardless of which schematic page they are on, or in which schematic folder they appear in the hierarchy. (Hierarchical designs are further discussed later in this course.)

Another source of global net connections is the hidden voltage pins defined for a part in the library.

When you connect two voltage symbols together, the voltage symbol with the alphanumerically lowest net name takes precedence. For example, if you connect an AGND symbol to a GND symbol, all GND connections will appear in the netlist as AGND.

Use the PSpice A/D library, *Source.olb*, to properly designate a high or low signal during simulation. Alternately, set the net name on the CAPSYM library symbol to 0 (zero).

Buses and Bus Entries

Use Bus Entry Symbols to connect
a single bit wire to a bus

Use special syntax to
name bus

Q[0-3]

Q0
Q1
Q2
Q3

Use Bus command to draw Bus
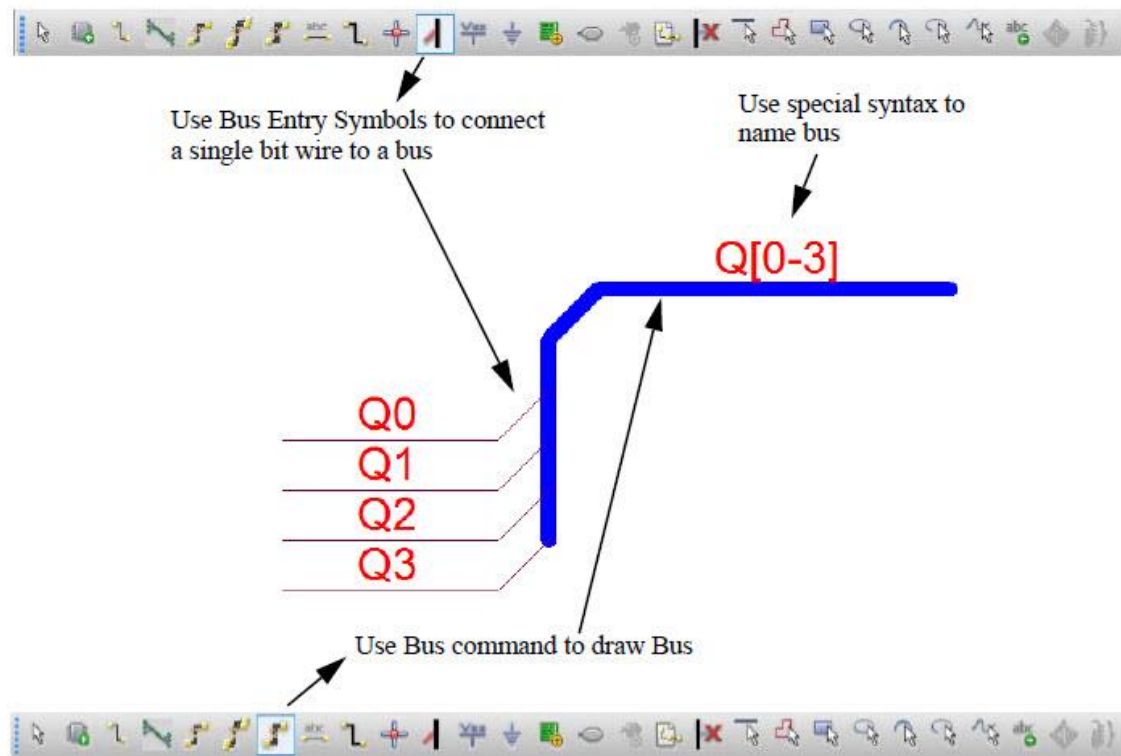
A bus entry is a short diagonal symbol used to connect wires to buses. Use the *Place Bus Entry* icon in the schematic toolbar to add bus entries. (You may need to rotate the bus entry symbol to the proper position before placing it.)

A bus wire is a vectored (multiple bit) wire used to bundle a group of scalar (single bit) wires together. A bus wire is graphically thicker than a regular wire. Use the *Place Bus* icon in the schematic toolbar to add bus wire connections.

A bus wire must be labeled to reflect the scalar nets contained within it (otherwise, it will be assigned a default system-generated net name like any other unlabeled wire in the design). If a bus wire is mislabeled, the DRC program will flag any connectivity errors that may result. (For example, Q[0-3] has no driving source, or has fewer than two connections, and so on.)

Use the *Place Net Alias* icon in the schematic toolbar to assign a net alias.

Use the following special syntax when labeling bus wires.

BASENAME[n-m]   i.e. BA[0-7]
or
BASENAME[n..m]   i.e. BA[0..7]
or
BASENAME[n:m]   i.e. BA[0:7]

The variable [n-m] represents the range of signals carried by the bus (n can be greater or less than m). The brackets must be square brackets; otherwise, OrCAD Capture does not let you attach the alias to the bus wire. Whichever convention you choose, use it consistently.

You can use a bus entry symbol to merge or split like-named buses. When a bus entry is used in this way, it appears thicker than a normal (single bit) bus entry.
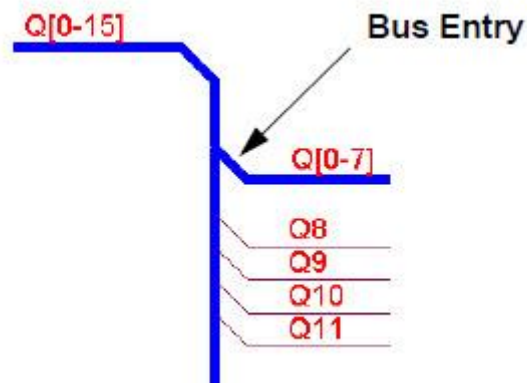
Manual Assignment during Placement (Option)
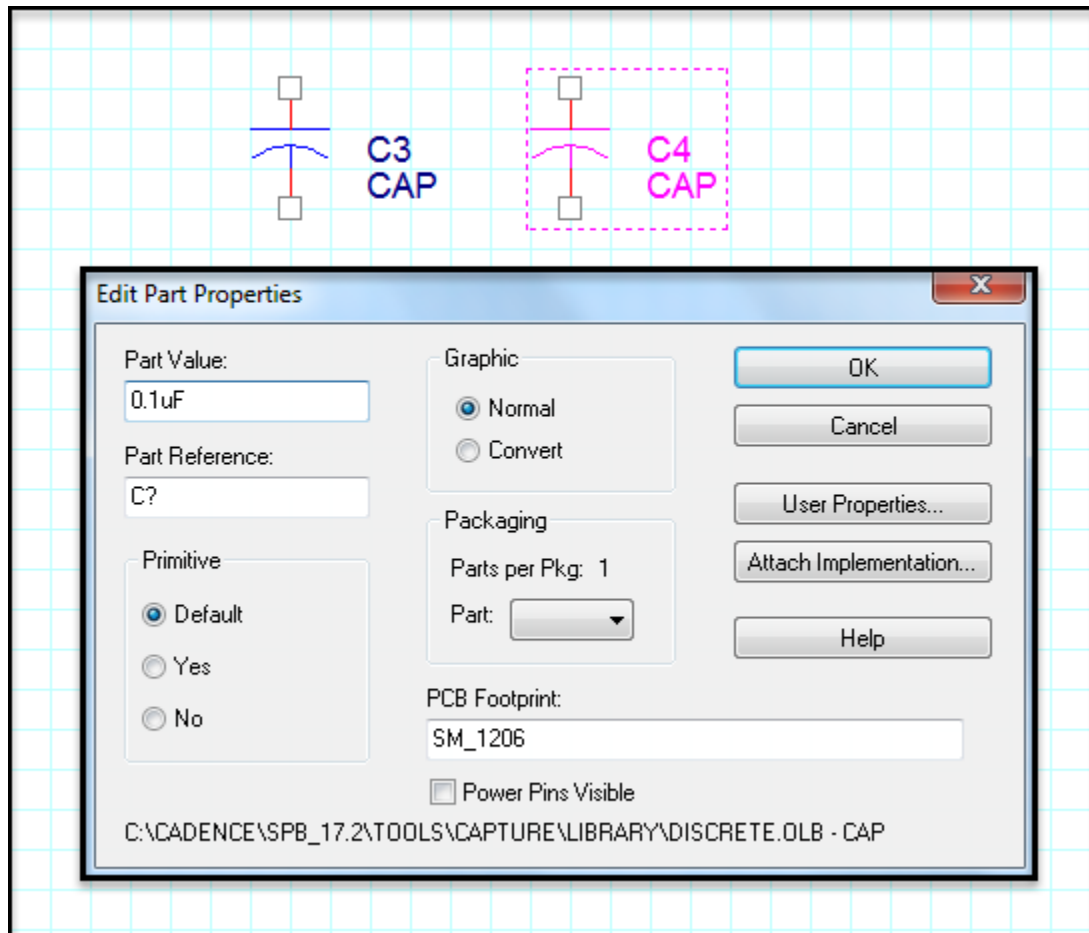
You can specify a part reference while adding a part to the design. After you select the part from the library, right-click and select *Edit > Properties* before you place it. This opens the Edit Part Properties window as shown above.

Use the **Part Reference** field to assign a reference designator manually.

Use the **Part Value** field to assign a value to the Part.

Use the **PCB Footprint** field to assign a Footprint pattern to the Part.

Select **User Properties** to assign any other Property you wish to the Part.

Lab 8-1: Creating a New Project

Lab Objectives

After completing this lab you will be able to:

- Edit a design template
- Assign a name to a project
- Assign a project "type" to a project
- View the title block
- View the system files and design resources

Create the Project Directory

1. On your computer, navigate to C:\EMA_Training\Capture_Essentials17.4\Designs
2. If the \Designs directory does not exist go ahead and create it.
3. Next, create a new directory named **LESSON8.** This is where you will store your LESSON8 files in the upcoming lab.

Editing the Design Template

1. Select *Options > Design Template*.
2. Click the *Title Block* tab, and in the *Title* field enter:
      Lesson8
3. Click *OK*.

Project Name

1. In the OrCAD Capture session window, select *File > New > Project*.

   The New Project window appears.

2. In the *Name* field, enter:
      Lesson8

Opening a New Project

1. If the start page is still open you can open New Project there, if not go to **File > New > Project…**

New Project

Name	LESSON8
Location	C:\EMA_Training\Capture_Essentials\Designs\LESSON8

☐ Enable PSpice Simulation

OK   Cancel   Help
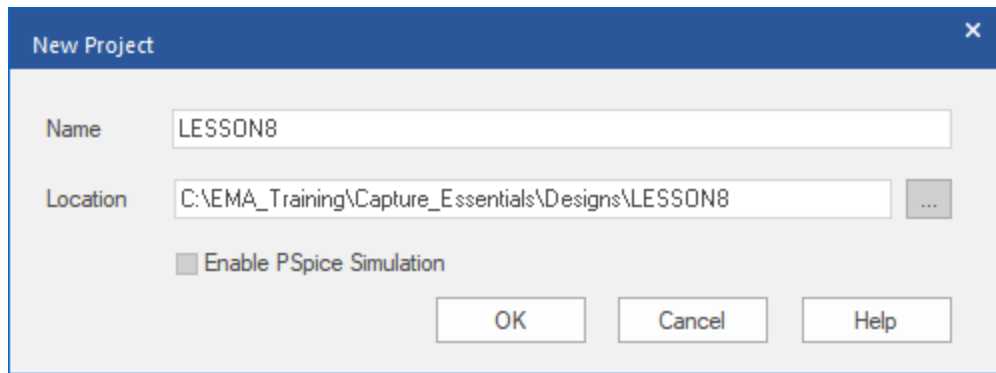
2. Name project and choose the LESSON8 directory that you created
3. Leave the **Enable Project Simulation** box unchecked and click *OK*.

Viewing the Title Block

1. Click in the schematic window to make it active. Resize this window (make it larger) if necessary.
2. Click the *Zoom to All* icon. ⊕

   Observe that the data in the title block reflects the change you made to the design template.

3. Click the *Zoom Out* icon.

Viewing the System Files

1. Using Windows Explorer, navigate to the directory:

   C:\EMA_Training\Capture_Essentials\Designs\Lesson8

2. Observe the following files in the project directory:
   Lesson8.dsn - binary design file.
   Lesson8.opj - ASCII project file.
3. Close the Windows Explorer.

Viewing the Design Resources

1. Click in the Project Manager window to make it active.
2. Click on the plus sign (+) to the left of .\Lesson8.dsn.

   The display expands to show the contents of the design file.
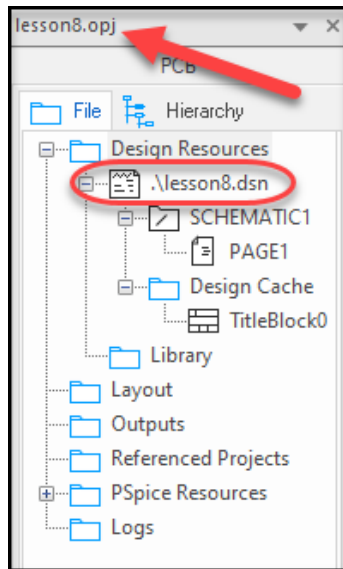
3. Click on the plus sign to the left of **SCHEMATIC1**.

    The display expands to show **PAGE1** of the schematic.

4. Click on the plus sign to the left of **Design Cache**.

    The display expands to show the parts that have been added to the design. The only symbol currently in the design is a title block.

Files within the Design Cache folder are binary files, which can only be viewed from the Project Manager window.

Lab 8-2: Create a Bus and use Off-Page Connectors

Lab Objectives

After completing this lab you will be able to:

- View a reference schematic
- Add a header connector
- Mirror the connector
- Add and copy decoupling capacitors
- Add wires and net aliases
- Add no connect and power/ground symbols
- Add bus entries and wires
- Add off page connectors
- View the design cache

Viewing the Reference Schematic

1. Take a moment to examine the following schematic. Please use this example as a guide as you complete the steps in this lab.
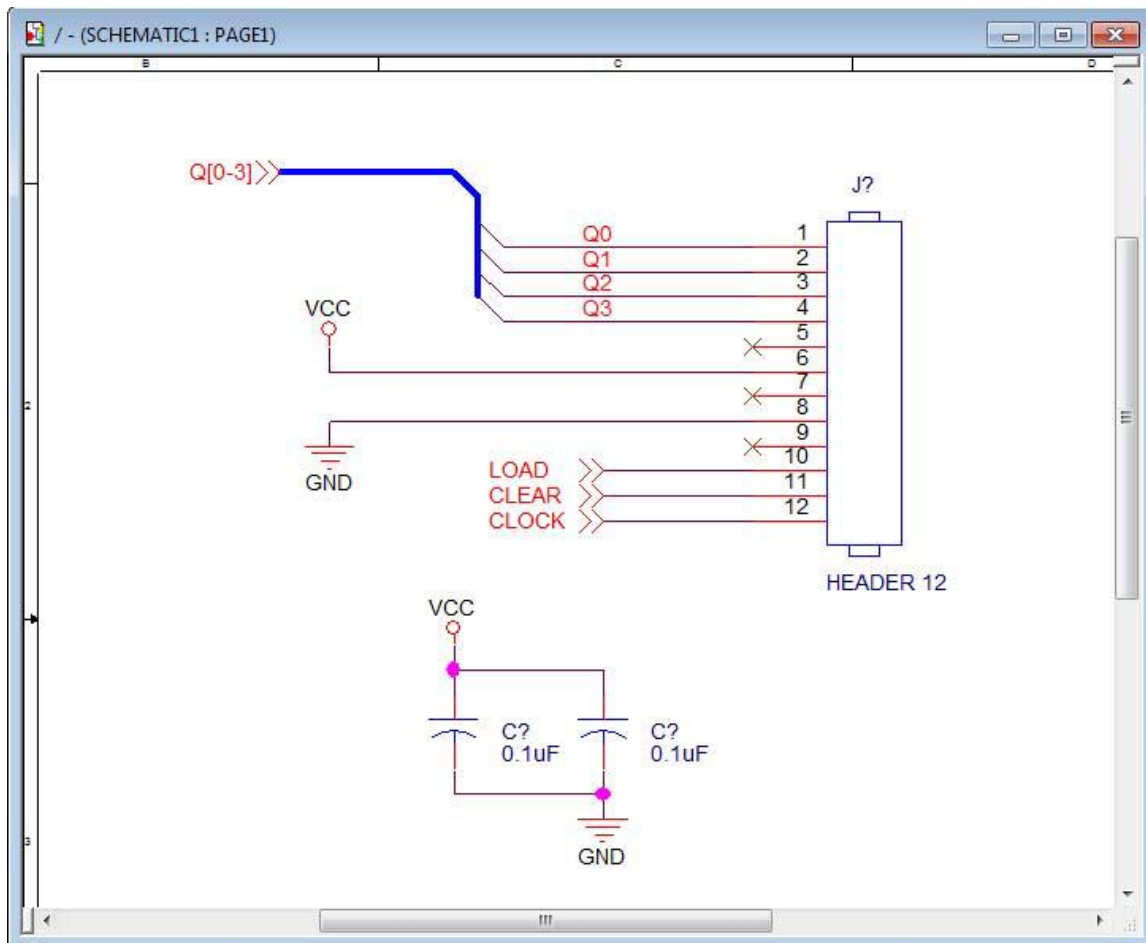
Adding a 12-Pin Header Connector

1. Make sure the schematic window is active, and click the *Place Part* icon in the schematic toolbar (or press the *<P>* key on the keyboard).

2. In the Place Part window, click on plus sign (+) to the left of *Search for Part*.

   The Part Search portion of the Place Part tab opens.

3. The library *Path* field (located at the bottom of the dialog box) should already show the installation path and library directory:

   C:\Cadence\SPB_17.4\tools\capture\library

   If not, use the *Browse* button to set this path.

4. In the *Search For* field, enter:
       header*

   OrCAD Capture searches all libraries for any parts that begin with the word "header".

5. Click the *Part Search* icon.

OrCAD Capture locates many different headers in a CONNECTOR library.

6. Scroll down and double-click on the HEADER 12/Connector.olb entry in the Libraries list, then click the ADD button to add the library to your list.

   The CONNECTOR library is added to the Libraries list in the Place Part window, and the HEADER 12 part is automatically selected and "floating" on your cursor.
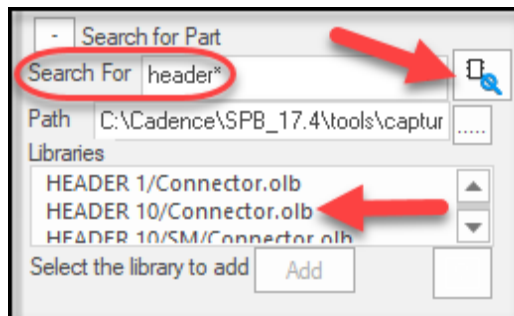
7. Do not add the part to your schematic yet.

Mirroring the Connector

1. If the part did not automatically attach to your cursor, select it now and go to Step 2.
2. With the connector still attached to your cursor, right-click, and select *Mirror Horizontally* from the pop-up menu.

   The connector pins should now be facing to the left.

3. Right-click again and select *Edit Properties* from the pop-up menu.
4. In the Edit Part Properties window, enter **J1** into the *Part Reference* field.
5. In the *PCB Footprint* field enter **CONN12** and click *OK*.
6. Click to place the connector in the design.
7. Press *<Esc>* to exit Place Part mode.
8. Press *<Esc>* again to deselect the connector.
9. Select *File > Save*.

   You mirrored the connector horizontally and assigned a reference designator to it before placing it in the design. You could also have done this after the part was placed.

Adding Decoupling Capacitors

1. If the Part Place tab is closed, click the *Place Part* icon in the schematic toolbar.
2. In the area of the *Place Part* window where it says Libraries, there are two icons. Hover over them with your cursor until you see *Add Libraries*. Click on the *Add Library* button.
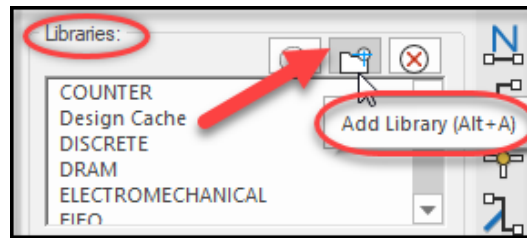
The Browse File window appears.

3. In the *Look in* field, navigate to the library directory:

C:\Cadence\SPB_17.4\tools\capture\library

This path will vary depending on your directory location of the Cadence software.

4. Select the DISCRETE.olb library file and click *Open*.

The DISCRETE.OLB library is added to the Libraries list in the *Place Part* window, and all the parts in this library are listed in the Part List.

5. In the *Part* field, enter:
    cap

Observe that each character you enter is used to search the *Part List* for all matching entries.

6. At the top right of the Place Part tab, click on the *Place Part* icon, but **Do Not** place it yet.

7. Right-click and select *Edit Properties* from the pop-up menu.
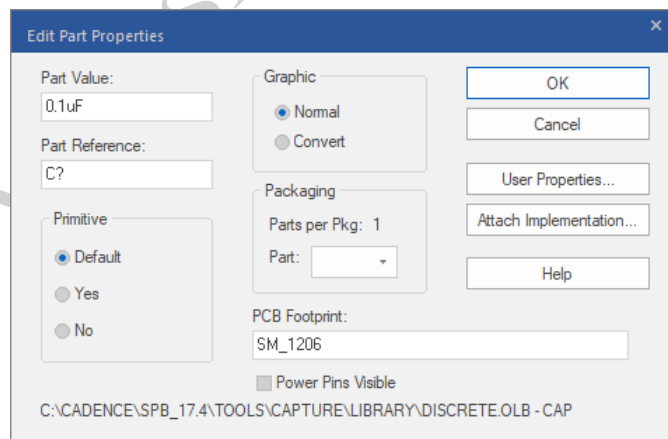
8. In the *Part Value* field, enter 0.1uF.
9. In the *PCB Footprint* field enter SM_1206.
10. Click *OK*.
11. Click to place the capacitor in the design. (You might want to place both capacitors or do the next step to copy. See "Copying the Capacitor on the next page.)
12. Press *<Esc>* to end the command, and *<Esc>* again to deselect all parts.

13. Select *File > Save*.

You assigned capacitance before placing the bypass cap in the design. You could also have done this after the part was placed.
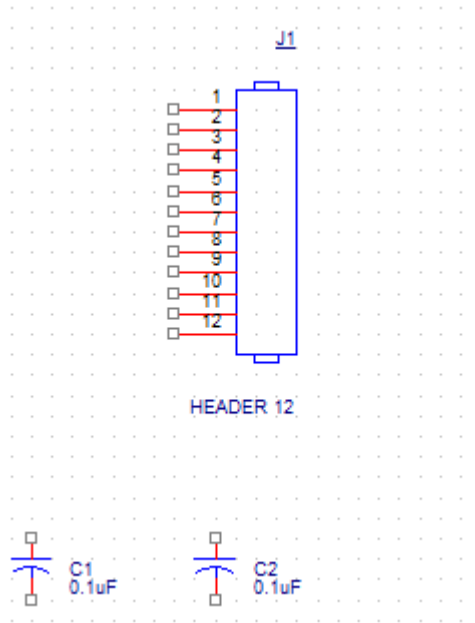
Copying the Capacitor

1. If you have already placed 2 capacitors move to the next section titled "**Draw BUS Using Auto Wire > Bus**"

2. Place the cursor on the capacitor, click to select, and press and hold *<Ctrl+LMB>*, and drag a copy of the capacitor to a new location.

   (LMB stands for Left Mouse Button.)

3. Press *<Esc>*.

Draw the BUS Using Auto Wire > BUS

1. The bus line will be connecting to pins 1 thru 4 of the Header, so make sure it extends down far enough to use a bus entry to pin 4. Choose **Place > Bus**

2. Before you begin drawing the bus line, hold down the Shift key. This will enable you to make the angled line in the bus. Draw the line using the following graphic as a guide.
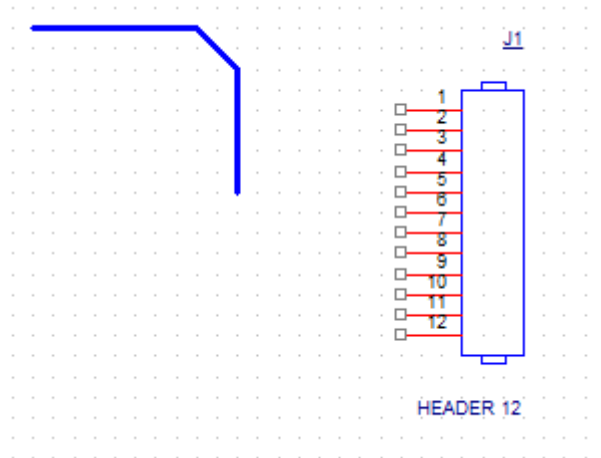
HEADER 12

3. Using the image above, draw a bus line to the left of the connector. Leave about 10-12 grid spaces for this example.

Connecting the Bus to the Header Pins
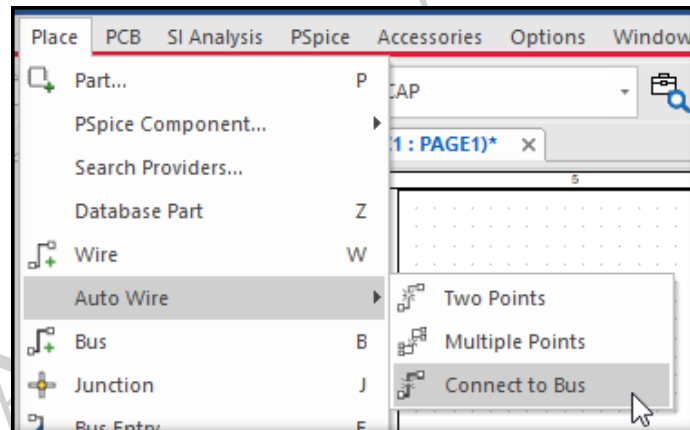
4. Select the *Auto Wire > Connect to Bus* command from the Place pull-down menu or the *Auto Connect to Bus* icon on the schematic toolbar. The "route" cursor will appear.

HEADER 12
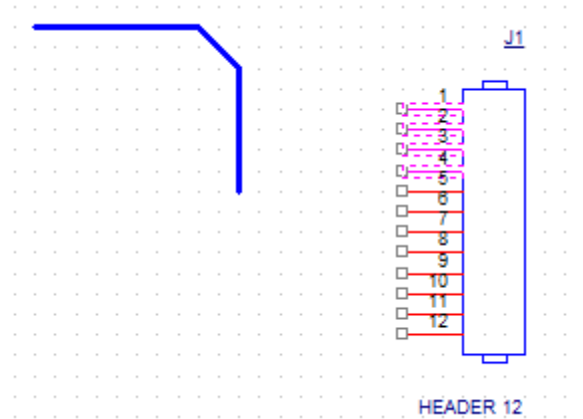
5. Click on pins 1-4 of the connector then click on the bus you drew to the left of the connector. The wires and bus entries will be drawn to the bus for you. See image above.
6. A small label window will appear. Enter the name of the bus as it includes all the bits of the bus...Q[0-3].
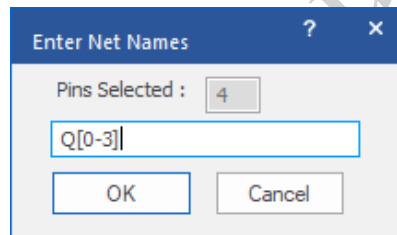
7. Select OK. The bus will be annotated with the bit numbers you entered. See following image.

HEADER 12
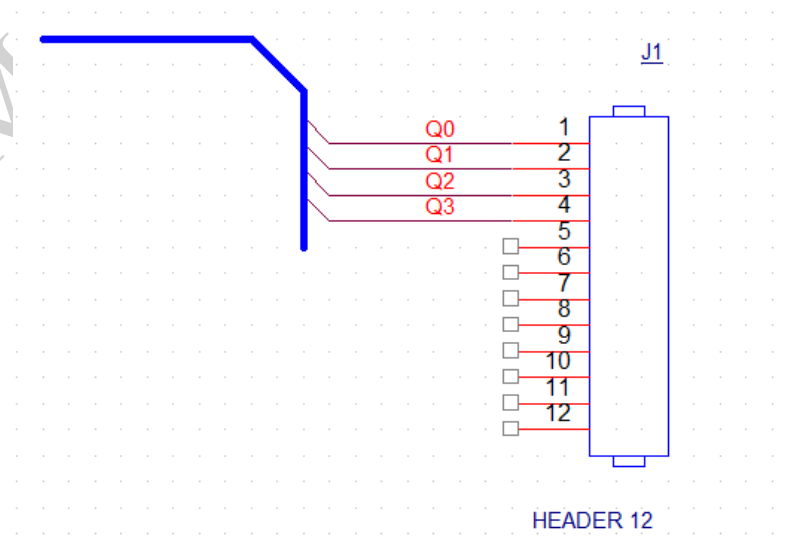
Alternate Method to Draw Bus Connections

An alternate method of drawing a bus would be to do it manually using the following commands:

- Place Bus
- Place Bus Entry
- Place Wire
- F4 (to easily repeat wires)
- Place Net Alias

In this case you have placed a bus wire on a schematic page that is not very dense, but if the schematic page were dense you may need to leverage the manual method in order to properly add the bus wire.

Adding No Connect Symbols

1. Use the *Place No Connect* icon to place No Connect symbols on all unconnected pins of the HEADER 12 part.
2. Press <*Esc*> to exit the command.
3. Save the design.

Adding Power/Ground Symbols

1. Click the *Place Power* icon .

   The Place Power window appears.

2. In the Libraries field, click CAPSYM.
3. Scroll down, select the **VCC** symbol, and click *OK*.
4. Add the **VCC** symbol to the schematic. You may place as many **VCC** symbols as required (or you can copy them later).
5. Click the *Place Ground* icon .
6. Select the **GND** symbol, click *OK*, and place as required.
7. Press <*Esc*> to exit the command, and <*Esc*> again to deselect all parts. Your caps should now look like the following once you've added the power symbols:
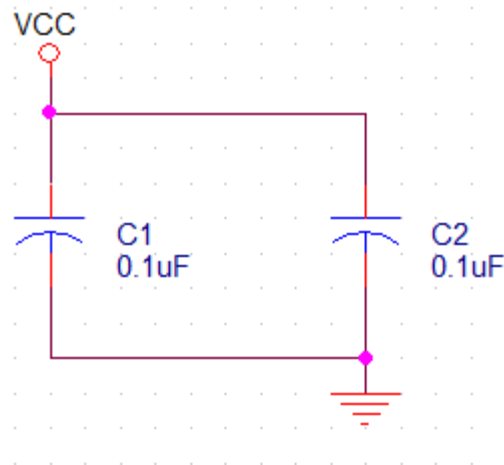
8. To move a power or ground symbol, press and hold the left mouse button and drag the symbol to a new location.
9. Add or adjust wiring to connect the power and ground symbols.
10. Save the design.

Making GND Symbols Names Visible (Optional)

1. Select the GND symbol, then RMB, and from the **RMB** pop-up menu select **Edit Properties**.
2. When the Property Editor window opens look for the Property called **Name** and select the column as shown below.
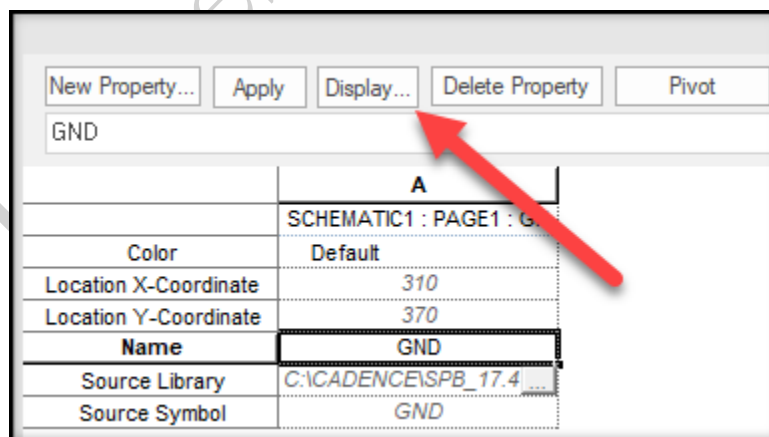
3. Just above the Property line you will see a "button" for **Display**. Click on it.

4. The Display Properties window will open. Select *Value Only* and click *OK*.
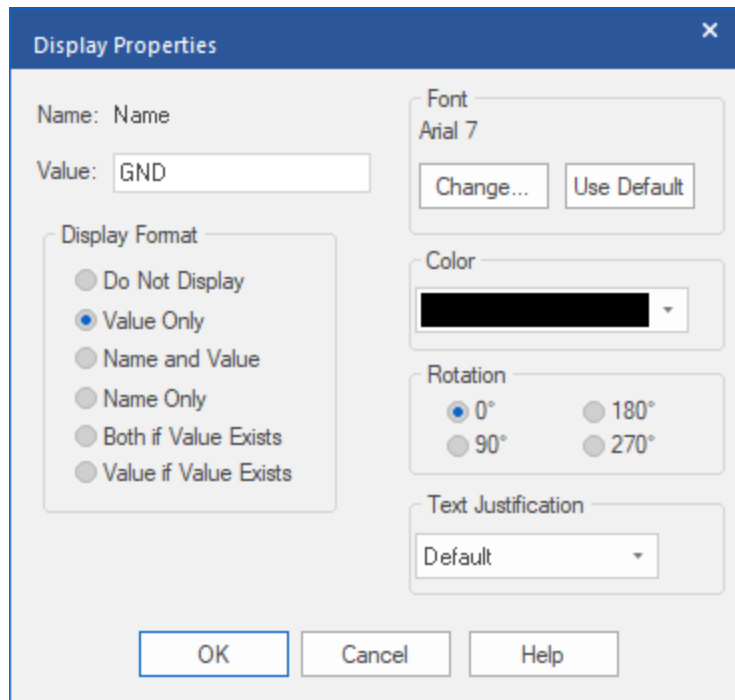
The Display Properties window will close.

5. Close the Property Editor window to apply the change. View the following picture.
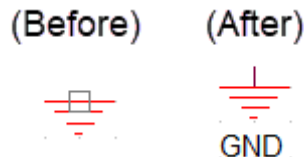
Adding Off-Page Connectors

Off page connectors must be used if a signal travels between pages. They tell Capture that there is more than one instance of a connection that resides on another schematic sheet.

1. Click the *Place Off-page Connector* icon.

The Place Off-Page Connector window appears.

2. In the Libraries list, click on CAPSYM.OLB, then select **OFFPAGELEFT - L** from the list above.
3. In the *Name* field (at the bottom), enter: **Q[0-3]**
4. Click *OK*.

The off-page connector attaches to your cursor.

5. Press the <*R*> key to rotate the symbol. Do this twice to spin it 180 degrees.
6. Click to attach the off-page connector to the end of the bus wire (as shown in the next graphic).
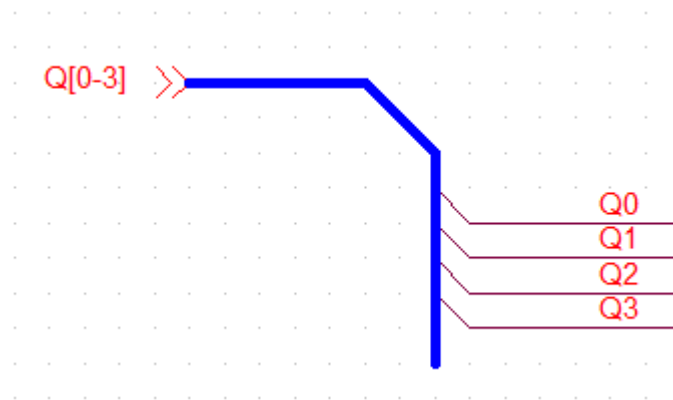
7. Press <*Esc*> to exit the command, and <*Esc*> again to deselect the off-page connector.

8. Click the *Place Off-page Connector* icon , and repeat this process to add off-page connectors to the **LOAD**, **CLEAR**, and **CLOCK** nets also (refer to the next graphic illustration).

An easy way to change the name of the next off-page connector you want to add is to right-click and select *Edit Properties* from the pop-up menu.

Save and Close the Design

1. Click the *Zoom to All* icon . Your finished page should resemble the following image.
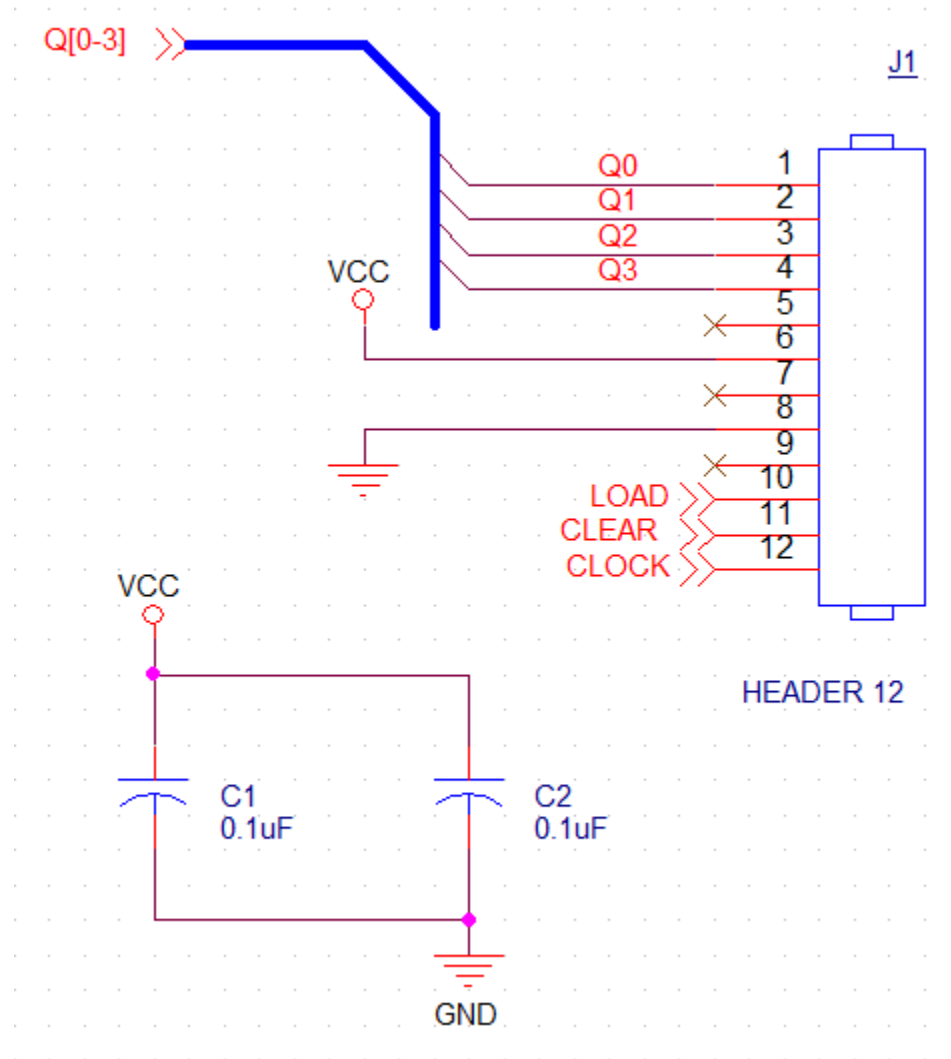
2. Select **File > Save**.
3. Select **File > Close**, and **No** to close the window.

Viewing Design Cache

1. In the Project Manager window, look at the list of parts under the Design Cache folder.

   This list represents all the parts that were added to page one of this design.
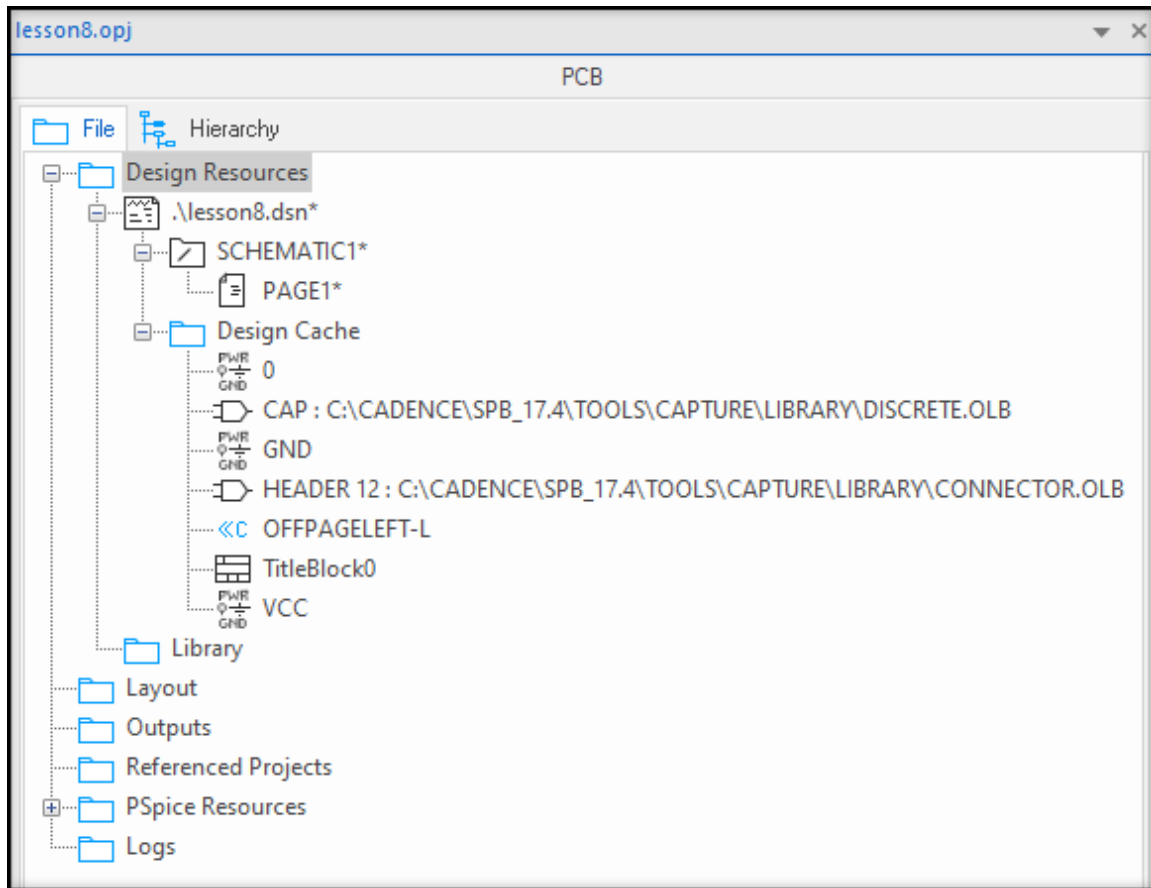
2. Click the minus sign **(-)** to the left of the Design Cache folder to collapse the display.

Lesson 9: Property Editor

Lesson Objectives

After completing this lesson, you will be able to:

- Work with Property Filters in the *Property Editor*
- Add part and net properties using the *Property Editor*
- Using other Property Editor Techniques

Part Properties
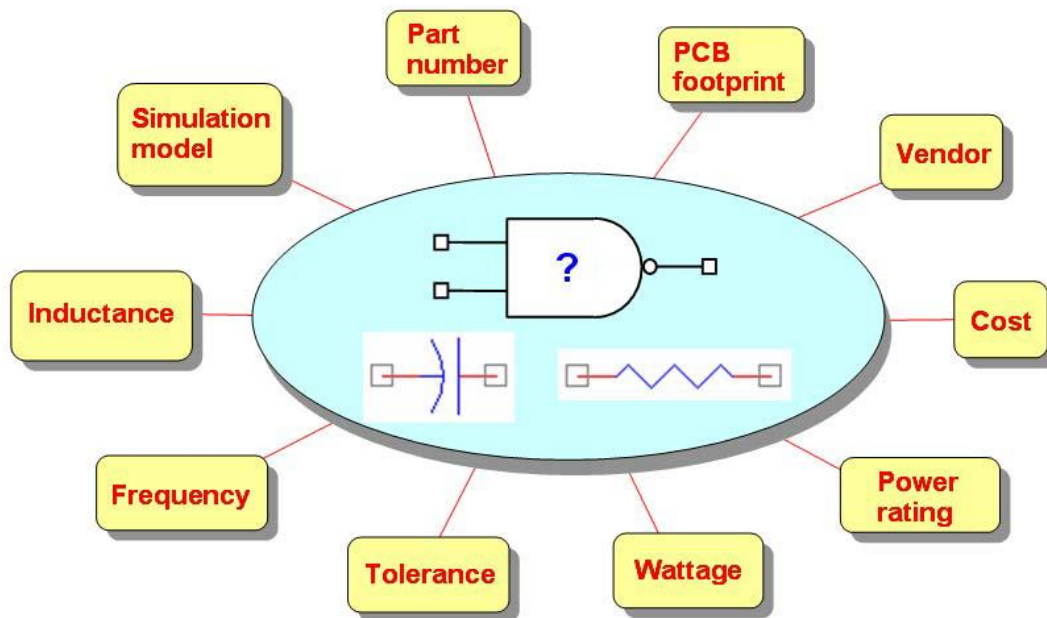
Each part in the design has special characteristics that need to be defined as properties. For example, every part needs a PCB footprint and a company part number. Depending upon the type of device (discrete, analog, digital IC), some parts may require more properties than others.

The Property Editor
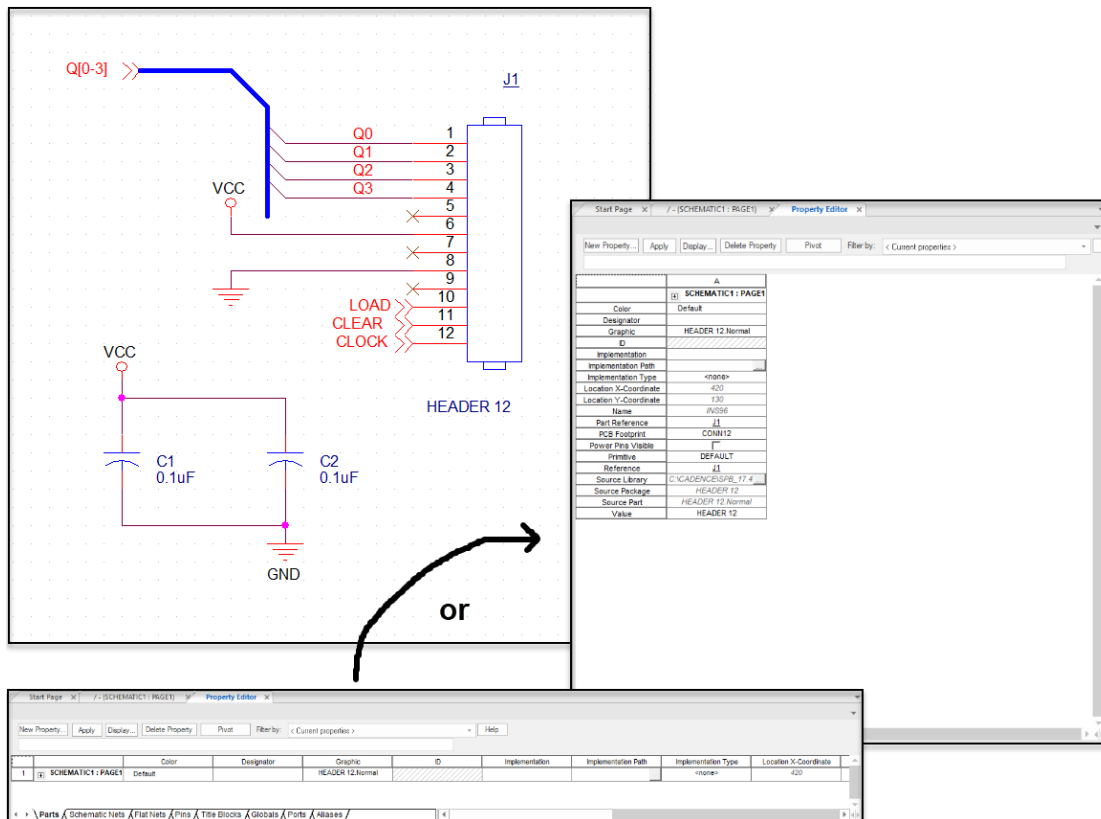
The Property Editor Window lets you add, delete, or modify the properties of parts, wires, and text. To access the Property Editor, click on an object (part, wire, or text), click the right mouse button, and select the ***Edit Properties*** command from the pop-up menu. You can also double click on the part, wire, or text.

Pivoting the Spreadsheet

The Property Editor displays a spreadsheet with selected object(s) along the left side, and a list of properties across the top. You can pivot the spreadsheet to display the property list along the left, and a list of selected objects across the top (as shown in the accompanying illustration).

When the spreadsheet is pivoted with property names on the left, the ***New Column*** button becomes the ***New Row*** button (see illustration).

Filtering the Spreadsheet

The Property Editor lets you filter the contents of the spreadsheet. Use the tabs along the bottom of the window to filter the contents by object type. For example, when you select a part, the Parts tab is selected by default, and the part properties appear in the property table. Click the *Pins* tab, and the spreadsheet displays the pin properties for all the pins on the part.

Use the *Filter by* field at the top of the Property Editor window to filter the spreadsheet using a property list. (Think of Property Filters as a list of specific properties that you wish to see at a given time.) Each property list is given a name and is known as a filter. All filter names are displayed in the Filter By field. Select a filter name to switch from one property list to another.

The properties in the Cadence-supplied filters target specific design tools. For example, there is a property list for the OrCAD and Allegro PCB Editor. The *<Current properties>* filter lists all properties currently assigned to the selected object(s).

You can create your own filters. For example, you can create a filter that just contains manufacturing-related part properties. When you select a part (or all parts), you can load, view, and edit these parts using a special property filter. Or you can create a list of high-speed routing properties to use when you're working with special nets.

The filter names and property lists are stored in the <install_dir>\tools\capture\prefprops.txt file.

It is strongly recommended that the prefprops.txt file **not** be edited. Also, copy it to another directory before you begin editing in the Property Editor. The only way to restore the original is to re-load OrCAD Capture.

Lab 9-1: Property Filters in the Property Editor

Lab Objectives

After completing this lab you will be able to:

- Open an existing project
- Filtering by object type
- View and select available property filters
- Pivot the editor spreadsheet
- Assign a PCB footprint
- Add a new property
- View and edit net properties

Opening an Existing Project

1. Select *File > Open > Project*.
2. Navigate to the C:\EMA_Training\Capture_Essentials\Designs\Intro directory.
3. Select the Intro.opj file, and click *Open*.
4. In the Project Manager window, click the plus signs *(+)* to display the contents of the design file, and double click on *PAGE1*.

   Page one of the schematic is opened.

5. Resize the schematic window, and zoom to fit.
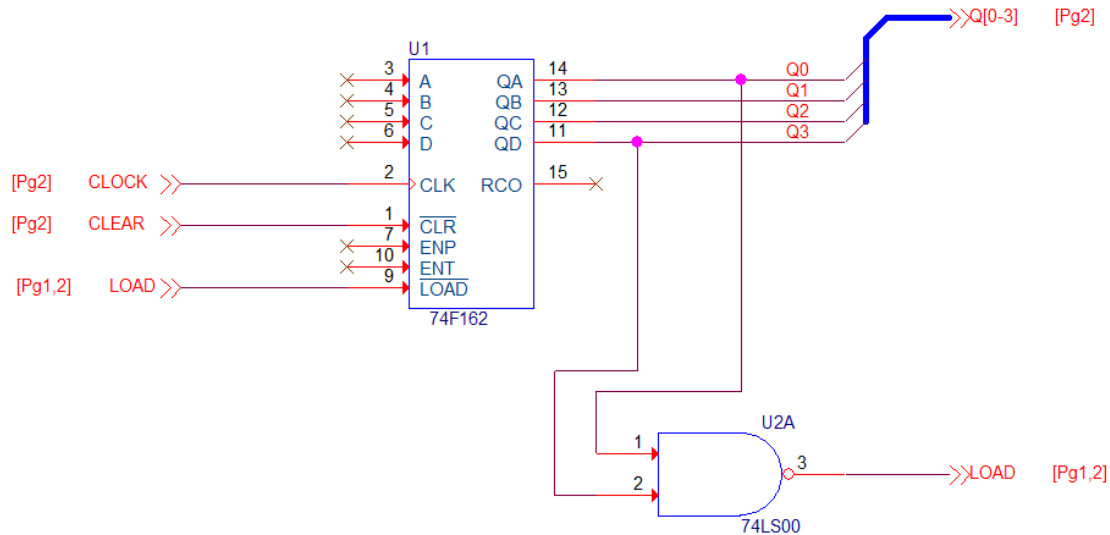
Selecting a Property Filter

1. Click the **74F162** part to select it, right-click, and select **Edit Properties** from the pop-up menu.

   The Edit Properties menu appears.

2. At the top of the Edit Properties window, click in the **Filter by** field.
3. Scroll through the list of filter names, and select the **OrCAD PCB Designer Standard** entry.
4. Close the Property Editor window.

Viewing Available Property Filters

1. Double click the **74F162** part, again, to select it.
2. At the top of the Edit Properties window, click in the **Filter by** field.
3. Now set the **Filter by** field to **Capture**.
4. Set the **Filter by** field to **<Current Properties>**.

   All properties currently assigned to the part are displayed (regardless of which tool the property is targeting).

Filtering By Object Type

Another way to control the contents of the Property Editor window is by object type.

1. Observe the tabs along the bottom edge of the Property Editor window.

2. Click the *Pins* tab or entity filter at the bottom of the window.

    All the pin properties for the *74F162* part are displayed.

3. Click the *Parts* tab or entity filter at the bottom of the window.

Pivoting the Editor Spreadsheet

1. Right click in the empty cell in the top left position of the spreadsheet, and select *Pivot* button
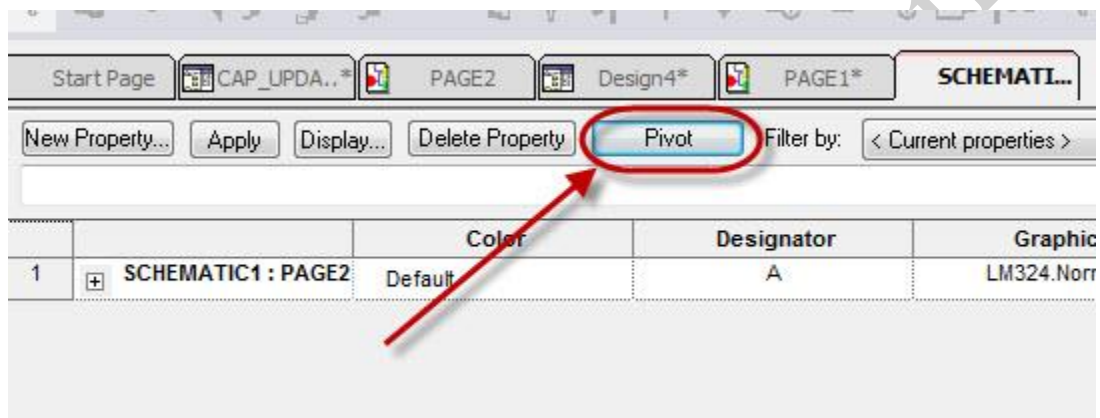
Observe that the spreadsheet now lists property names at the left (as shown below) instead of across the top. If you double click again in the top left cell, the property names will pivot back across the top of the spreadsheet (the default format).

Printing the Property Editor Spreadsheet

If you want to print out the Property Editor spreadsheet, do the following:

- Highlight all the Properties.
- Go to the main OrCAD Capture menu and do *Edit > Copy*.
- Open a text editor, such as Word or Notepad, and *Paste* into the editor
- Optionally you could do a **File > Export > PDF**

Lab 9-2: Using the Property Editor to add Part and Net Properties

Lab Objectives

After completing this lab you will understand how to edit properties for both parts and nets.

Assigning a PCB Footprint

1. With the property list pivoted along the left (as shown in the last view), click in the *PCB Footprint* field and enter:
   SOIC16
2. Click *Apply* and close the Property Editor window.
3. Press *<Esc>* to deselect the *74F162* part.
4. If the PCB Footprint has already been applied move to the next step.

Controlling Property Visibility (Optional)

1. Double click on the *74F162* part.

   The Property Editor window is displayed. Observe that the PCB Footprint property value is set to *SOIC16*.

2. Click on the *SOIC16* property value, and click the *Display* button at the top of the spreadsheet.

   The Display Properties window appears.

3. In the Display Format section of the window, click the *Value Only* option, and click *OK*.
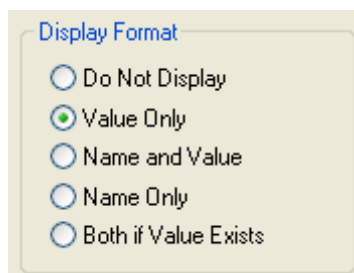
4. Close the Property Editor window.

   The PCB Footprint property value *SOIC16* is now visible in the schematic.

5. Press *<Esc>* to deselect the *74F162* part.
6. Select *File > Save*.

Adding a New Property

1. Double click on the *74F162* part.
2. In the Property Editor, click *New Row*.

    The Add New Row window appears.

3. In the *Name* field, enter:
        PART_NUMBER
4. In the *Value* field, enter:
        20-67890
5. Click *OK*.

    Notice the property name and value in the Property Editor window.

6. Close the Property Editor window.
7. Press *<Esc>* to deselect the *74F162* part.
8. Save the design.

Viewing and adding Net Properties

1. Double click on a wire with the name of *LOAD* to select it.

    Observe the net name in the Property Editor window. Also observe that the
    Schematic Nets tab is automatically selected based on the type of object you picked.

2. Toggle the *Filter by* field to *Cadence-Allegro*.

    This filter list the net properties you can use for the OrCAD and Allegro PCB Editor
    tool.

3. Using the above mentioned Filter, set the future trace width to 20.

        (In the *Cadence-Allegro* filter, look for *MIN_LINE_WIDTH*.)

4. Reset the *Filter by* field to *<Current Properties>*, and close the Property Editor
    window.
5. Press *<Esc>* to deselect the wire.

You can change a net name by editing the properties of a wire using the Aliases entity
filter at the bottom of the Property Editor. Or you can double click on the net alias text,
and use the Edit Net Alias window.

Lab 9-3: Using Other Property Editor Techniques

Lab Objectives

After completing this lab you will be able to:

- Edit multiple objects
- Add new properties
- Search a current page
- Browse a design

Multiple Object Editing

1. Press *<Ctrl+LMB>* to select both the *74F162* and *74LS00* parts.

Remember, throughout this manual "LMB" refers to your "**L**eft-**M**ouse-**B**utton".

2. Right-click and select *Edit Properties* from the pop-up menu.

   Observe that the properties for both parts are displayed in the Property Editor window.

3. Click in the *PCB Footprint* field for the *74LS00* part and enter:
   SOIC14

   a.   Click *Apply* to set the value.

   b.   Click the *Display* button.

   c.   Set the Display Format to *Value Only* and click *OK*.

   This makes the value of the PCB Footprint property visible in the schematic.

4. Click in the *PART_NUMBER* field for the *74LS00* part and enter:
   20-12345
   Click *Apply* to set the value.

5. Click in the *PCB Footprint* field for the *74F162* part and enter:
   SOIC16
   Click *Apply* to set the value.

6. Close the Property Editor window.

7. Press *<Esc>* to deselect both parts.

Multiple Object Property Display

1. Reselect both the *74F162* and *74LS00* parts.
2. Press *<Ctrl+E>* to display the Property Editor. (Or select *Edit Properties* from the pop-up menu.)
3. Click on the *PCB Footprint* property name box on the left (as shown in the following graphic).

	A	B
	⊞ SCHEMATIC1 : PAGE2 : U3	⊞ SCHEMATIC1 : PAGE2 : U4
Color	Default	Default
Designator		A
Graphic	74F162.Normal	74LS00.Normal
ID		
Implementation		
Implementation Path	[...]	[...]
Implementation Type	<none>	<none>
Location X-Coordinate	300	570
Location Y-Coordinate	100	310
Name	INS1667	INS1807
Part Reference	U3	U4A
PCB Footprint	SOIC16	SOIC14
Power Pins Visible	☐	☐
Primitive	DEFAULT	DEFAULT
Reference	U3	U4
Source Library	C:\CADENCE\SPB_17.2\TO [...]	C:\CADENCE\SPB_17.2\TOO [...]
Source Package	74F162	74LS00
Source Part	74F162.Normal	74LS00.Normal
Value	74F162	74LS00

Toolbar: New Property... | Apply | Display... | Delete Property | Pivot | Filter by: < Curre

All PCB Footprint property values are highlighted.

4. Click *Display*.

   a.   In the Display Properties window, click *Do Not Display*.

   b.   Click *OK*.

5. Close the Property Editor window and press *<Esc>* to deselect all parts.

   Observe that all PCB Footprint properties are now invisible.

Adding New Properties

1. Reselect both the *74F162* and *74LS00* parts.
2. Right-click and select *Edit Properties* (or press *<Ctrl+E>*).

3. At the top of the Property Editor window, click **New Row**.

   The Add New Row window appears.

4. In the **Name** field, enter:
   VENDOR

5. Click **OK**.

   The **VENDOR** property is added to the Property Editor window.

6. Click on the **VENDOR** property name box (on the left).
7. Right-click and select **Edit** from the pop-up menu.

   The Edit Property Values window displays.

   a.   Click in the **VENDOR** field and enter:

   Digi-Key

   b.   Click **OK**.

   The **VENDOR** name is applied to both parts.

8. Close the Property Editor window and press **<Esc>** to deselect all parts.

You can also use the **<Ctrl+C>** and **<Ctrl+V>** keys to copy and paste property values between cells in the Property Editor window.

Searching the Current Page

1. With the schematic page active, select **Find or Search** toolbar. It looks like the following figure.

2. To the right of the **Search** icon, there is a down-arrow. Select that. When the list appears, make sure you do **Deselect All** first. Then select it again and highlight **Nets**.
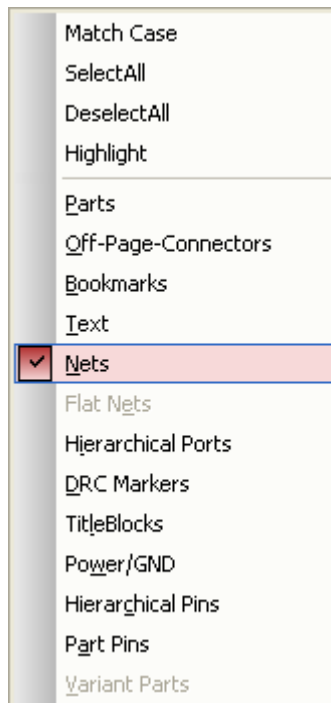
3. In the **Find** box, enter a ***** . This means **"all"**. Then click on the **Search** icon.

4. A list of all the Nets present on the open page will be displayed in a Browse window at the bottom of your design window.

Object ID	Net Name	Page	Page Number	Schematic	Part Pin
CLEAR	CLEAR	PAGE1	1	SCHEMATIC1	U1.1
CLOCK	CLOCK	PAGE1	1	SCHEMATIC1	U1.2
LOAD	LOAD	PAGE1	1	SCHEMATIC1	U1.9,U2.3
Q0(Wire Alias)	Q0	PAGE1	1	SCHEMATIC1	U1.14,U...
Q1(Wire Alias)	Q1	PAGE1	1	SCHEMATIC1	U1.13
Q2(Wire Alias)	Q2	PAGE1	1	SCHEMATIC1	U1.12
Q3(Wire Alias)	Q3	PAGE1	1	SCHEMATIC1	U1.11,U...
Q[0-3]				SCHEMATIC1	
Q[0-3]				SCHEMATIC1	
Q[0-3]	Q2	PAGE1	1	SCHEMATIC1	
Q[0-3]	Q3	PAGE1	1	SCHEMATIC1	
Q[0-3]	Q[0-3]	PAGE1	1	SCHEMATIC1	
U2A/7(PartPin)	GND	PAGE1	1	SCHEMATIC1	U2A.7
U2A/14(PartPin)	VCC	PAGE1	1	SCHEMATIC1	U2A.14

(context menu: Edit Properties (Ctrl+Shift+E), Save as HTML)

Nets

All nets on the current page are selected. This provides a convenient way to search for all objects of a desired type, without selecting everything else on the page.

5. Select a Net or group of Nets. (Use the **CTRL+LMB** technique.)

6. With your cursor within the browse page, right-click and select *Edit Properties*.
7. When the Browse Spreadsheet appears, select the *New* button and enter the information as shown.
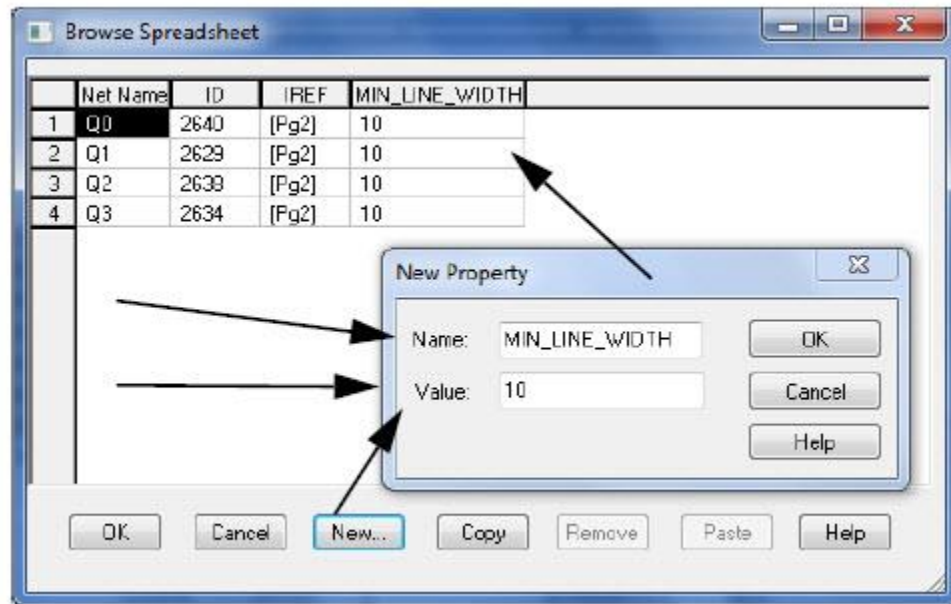
This illustrates adding a property to a wire(s). The Property for width in the OrCAD and Allegro PCB Editor is *MIN_LINE_WIDTH*. Although the *MIN_LINE_WIDTH* property is listed in the Cadence-Allegro property filter, we add it here as an example for adding a property to a wire "on-the-fly".

8. Click *OK* to apply the new property and its value.

More detail for using the *Find* and *Browse* commands are covered in Lesson 16.

Saving the Page

1. Select *File > Save* and *OK* to save all pages of the design.
2. Close the schematic page.

Custom Property Filters

In this lesson, we discussed the Property Editor, and the default property filters that come with OrCAD Capture. These included the OrCAD and Allegro PCB Editor filter that let you work with a list of part or net properties specific to that PCB tool, without having to view other unrelated properties at the same time. There are also several vendor-specific filters that apply to FPGA design.

The Property Editor lets you create custom property filters for different purposes. For example, you may want to view only a handful of properties required for your Bill of Materials report. Or you may want to check all parts for a valid footprint, and don't want to scan through a long list of properties to find just the PCB Footprint property name. In this case, you can create a custom filter that displays only a specific set of properties and associated values.

More Filter Commands

The Filter command pop-up menu contains several other useful commands. For example, you can use the **Hide** option to remove a property from the spreadsheet, but not from the filter itself.

Use the **Invert Filter** command to toggle the display between properties defined in the current filter or properties on the selected object that are not defined in the current filter (including hidden and optional properties).

The **Show Undefined** option toggles the display between properties defined in the current filter or properties defined in the filter plus any properties on the selected object not defined in the current filter. When you create a new filter, by default, the Show Undefined option is enabled. This causes all current properties on the selected object to display in the spreadsheet (because they are not yet defined as part of the current filter).

Use the **Show** option to add a current property to the current filter, or to re-display a hidden property.

Use the **Undefined** option to remove the selected property from the current filter.

Setting the status of a property to **Optional** causes a user-defined property to disappear from the spreadsheet if no value exists on the selected object. The Optional setting has no effect on reserved property names.

Null versus Blank Property Values

To set a property value to <null>, select the property value cell in the spreadsheet and click the **Delete Property** button. For user-defined properties, the cell then displays a pattern of diagonal lines to indicate a null value.

If you use the <Delete> or <Backspace> keys to clear a property value from a cell, the selected property will have a blank value also.

Lesson 10: Navigating the Schematic Design

Lesson Objectives

After completing this lesson you will be able to:

- Use the Search/Find toolbar
- Use the alternate Find command
- Use the Browse command
- Using Intelligent PDF

The Find Toolbar provides shortcuts for locating things in a schematic or on a schematic page(s).
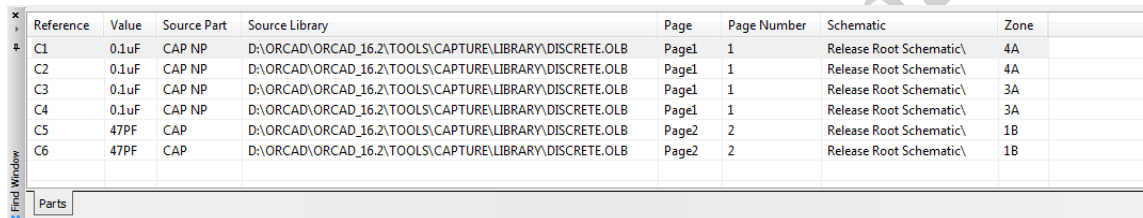
Use the Search/Find toolbar
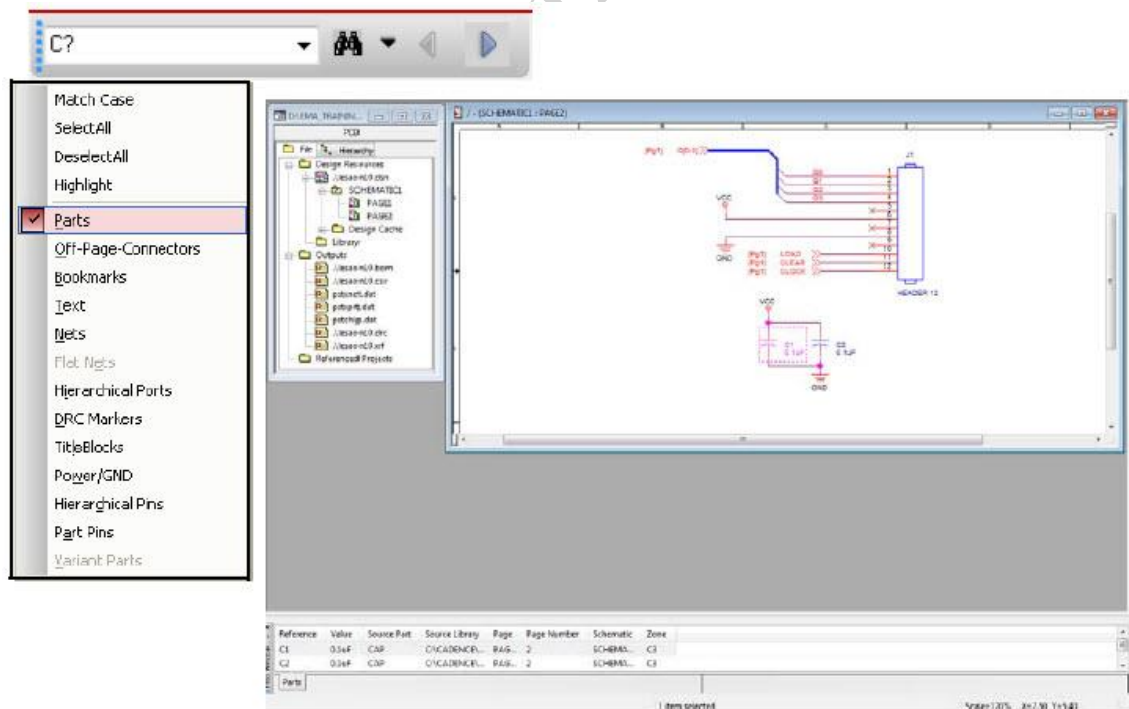
Search functions include ways to narrow down or expand your search to the types of objects on a given page, the schematic folder or the entire design.

Once the search is complete the Find window groups the search results by object type making it easy to find specific entities. You also have the option of saving the results to an HTML.

The Find Window is also "dockable" allowing you to conveniently position the window. By default it will appear at the bottom of your screen. Depending on what is highlighted from the Project Manager window, you can search at the design, folder or page level. You can also search the current open page.

Reference	Value	Source Part	Source Library	Page	Page Number	Schematic	Zone
C1	0.1uF	CAP NP	D:\ORCAD\ORCAD_16.2\TOOLS\CAPTURE\LIBRARY\DISCRETE.OLB	Page1	1	Release Root Schematic\	4A
C2	0.1uF	CAP NP	D:\ORCAD\ORCAD_16.2\TOOLS\CAPTURE\LIBRARY\DISCRETE.OLB	Page1	1	Release Root Schematic\	4A
C3	0.1uF	CAP NP	D:\ORCAD\ORCAD_16.2\TOOLS\CAPTURE\LIBRARY\DISCRETE.OLB	Page1	1	Release Root Schematic\	3A
C4	0.1uF	CAP NP	D:\ORCAD\ORCAD_16.2\TOOLS\CAPTURE\LIBRARY\DISCRETE.OLB	Page1	1	Release Root Schematic\	3A
C5	47PF	CAP	D:\ORCAD\ORCAD_16.2\TOOLS\CAPTURE\LIBRARY\DISCRETE.OLB	Page2	2	Release Root Schematic\	1B
C6	47PF	CAP	D:\ORCAD\ORCAD_16.2\TOOLS\CAPTURE\LIBRARY\DISCRETE.OLB	Page2	2	Release Root Schematic\	1B

Parts

While the search results display in the find window, the Session Log window also shows the details of the search like the total number of objects found and the resulting page the object(s) were found.

Lab 10-1: Using the Search/Find Toolbar

Lab Objectives

After completing this lab you will be able to:

- Activating the Search Toolbar
- Use the Search toolbar

Activating the Search *toolbar*

The following lab can be done ONLY if the Search Toolbar is NOT already active. If the Toolbar is active go to step 4.

1. A current method for search is to activate the *Search Toolbar*. From the **Tools > Customize**.
2. When the *Customize* window opens select the *Search toolbar* to enable it. See the following figure.
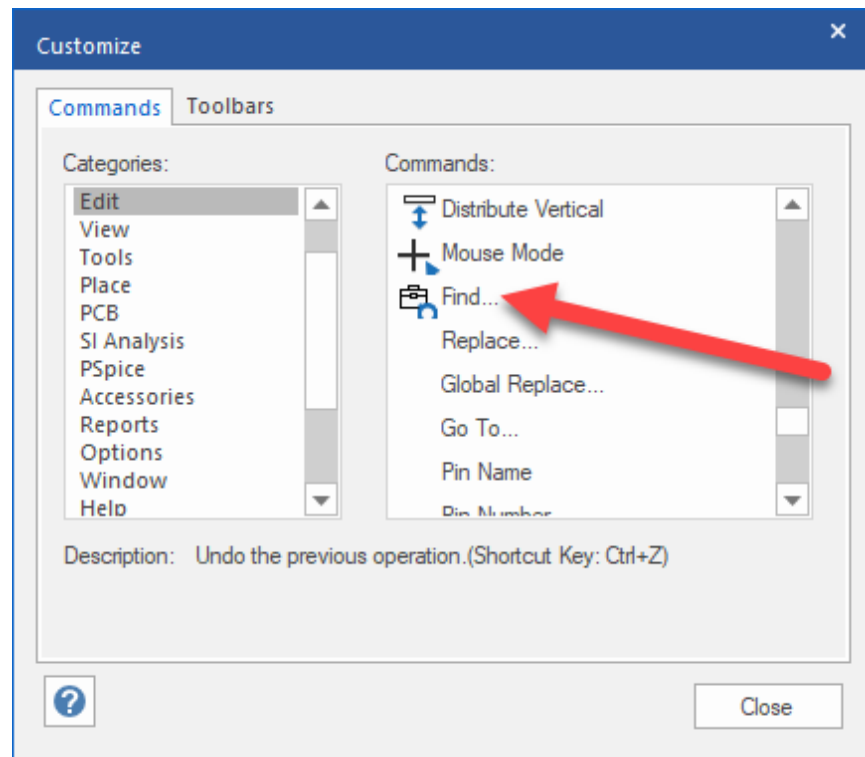
3. Select OK to close the Toolbar window and you should see the Toolbar now active along the top where other Toolbars are shown. Move it where it will be convenient for you. It will look like the figure below.
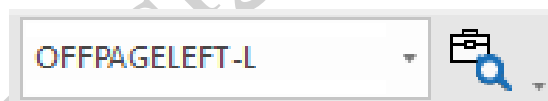
When this **Toolbar** is active the original method for search is disabled.

4. If a design is not still open, open either LESSON8 or the INTRO design. Do not open any pages yet.

5. There is also a Selection Filter as part of the Search Toolbar. To the right of the **Search icon (binoculars)** there is a small "down" arrow. Select it. A pull-down menu appears. All items on it are currently selected. Click on the **Deselect All** command.

6. In the **Search Toolbar**, again select the down-arrow to the left of the Search icon and enable only Parts.
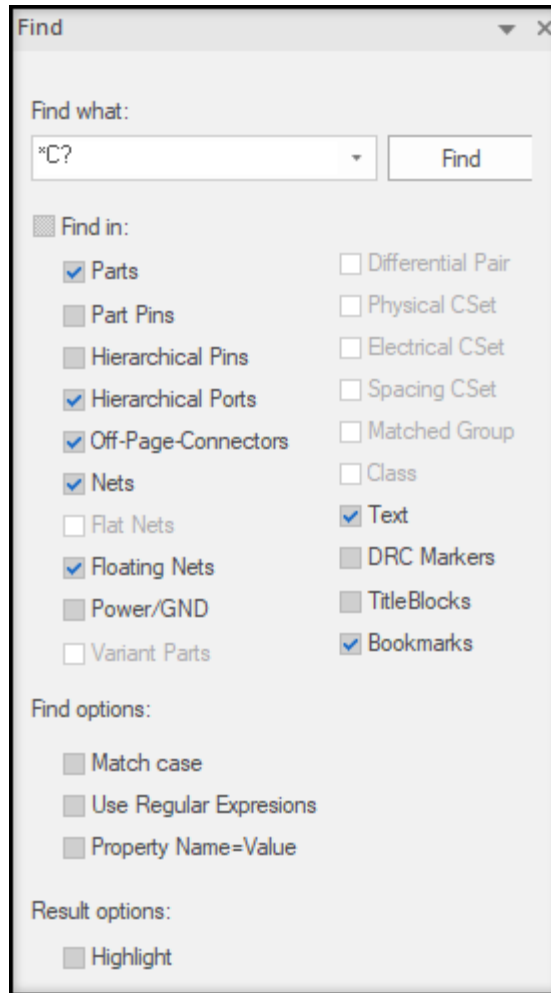
7. In the white search window enter **C1** and then select, the Search icon .

The **Session window** should now open displaying information about **C1** and the page containing the part will also open and be highlighted. Notice the next example figure.
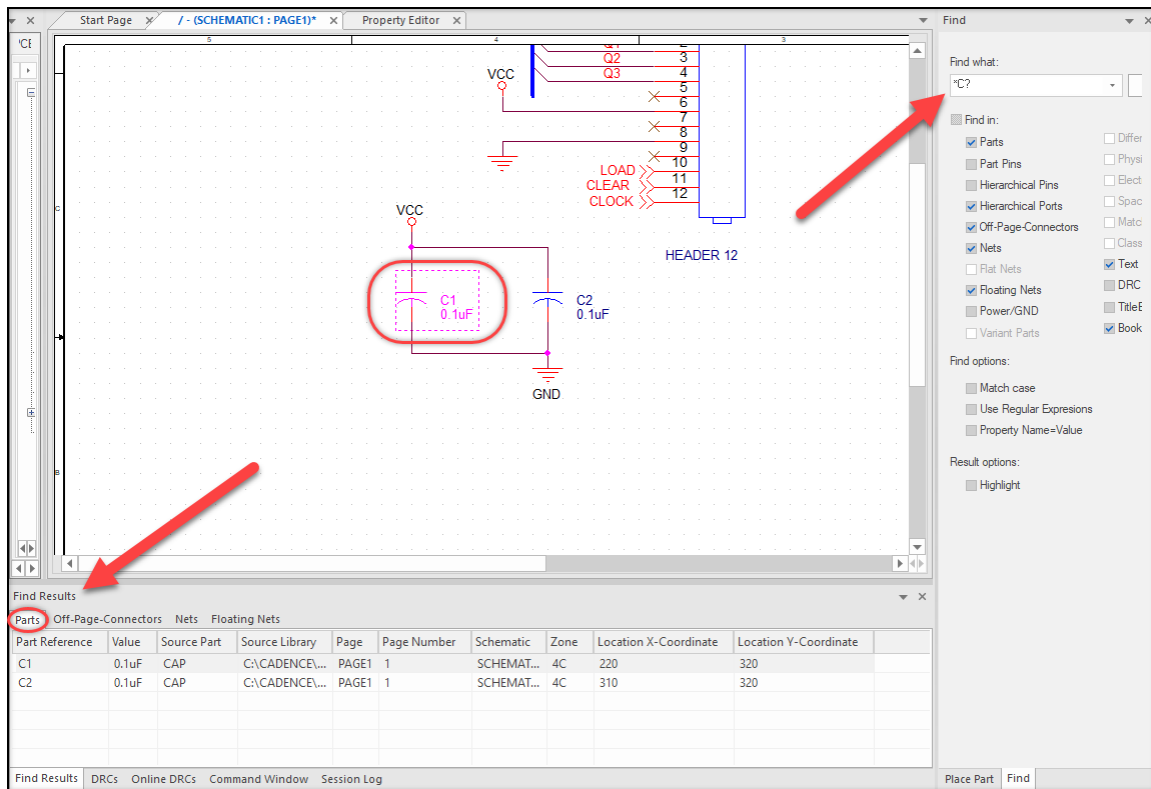
8. Select *File > Save*.
9. Click on the small "**X**" in the upper left of the Browse window to close the Browse Parts window.
10. Select *File > Save* to save the design

...another search...

1. Go back to the down-arrow again and select *Nets*.
2. Now in the *Search box* on the *Search Toolbar* enter *C** and all nets that begin with *"C"* are displayed in the Session Window.
3. Select one of the nets such as CLEAR and the net will be highlighted wherever it is connected on the open page. See the following figure.
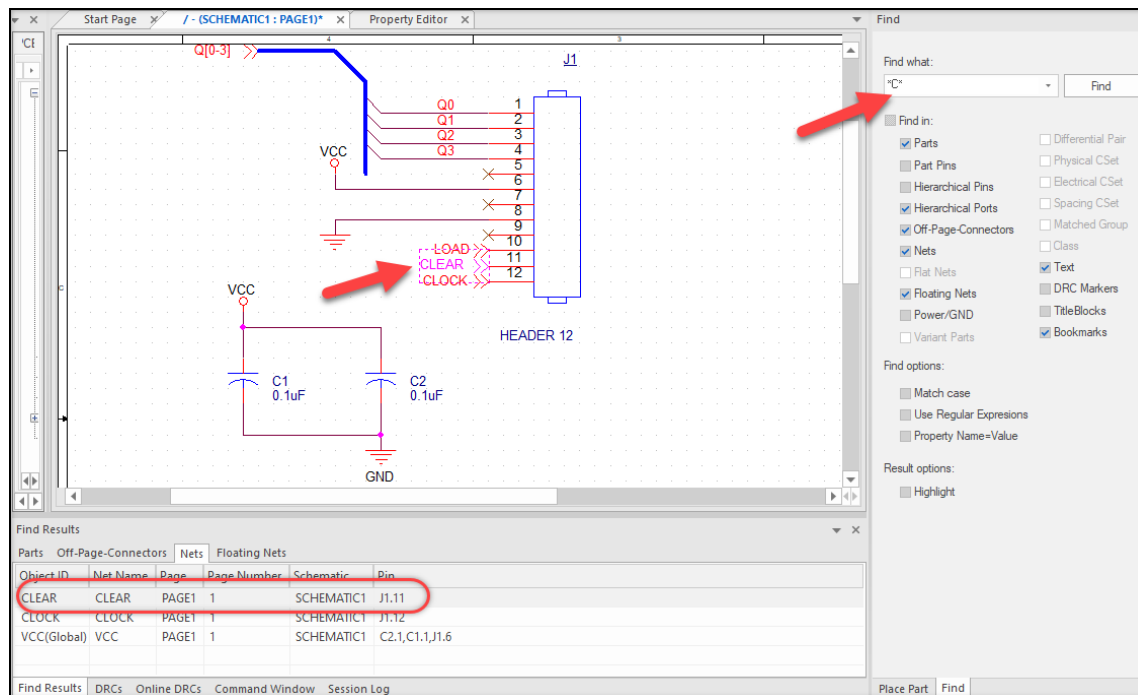
4. If you wish, experiment with these search options on your own.
5. When you have finished, close all pages and the design with "*no Save*".

The Schematic "Edit Find" Command (Alternate)
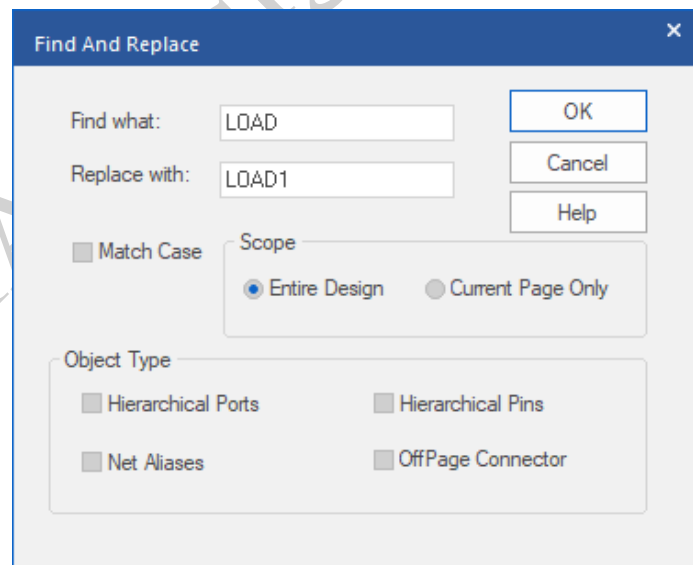
This Method was the original command to "Find" or "Browse" for items in the schematic. It is still present in the software but you must disable the new *Search Toolbar* before it will work.

When a schematic window is selected, the **Edit** pull-down menu contains the same editing commands as those available in the right mouse pop-up menu (for example, Cut, Copy, Paste, Delete, Mirror, and Rotate). However, the Edit pull-down menu contains commands such as the Find and Global Replace commands that are not available elsewhere.

The **Edit > Find** command can be used to select parts, nets, text, and other types of objects in the current page. When using this command, you can use wildcards to search for an object, or use the Find What field to select objects with specific properties. Once these objects are selected, you can open the Property Editor to edit them.

Use the **Edit > Global Replace** command to change the name of a net in the design. This command will not process off-page connectors. You cannot use this command to replace a part with another part. Wildcards are not supported in this menu.

The Project Manager "Edit Browse" Command
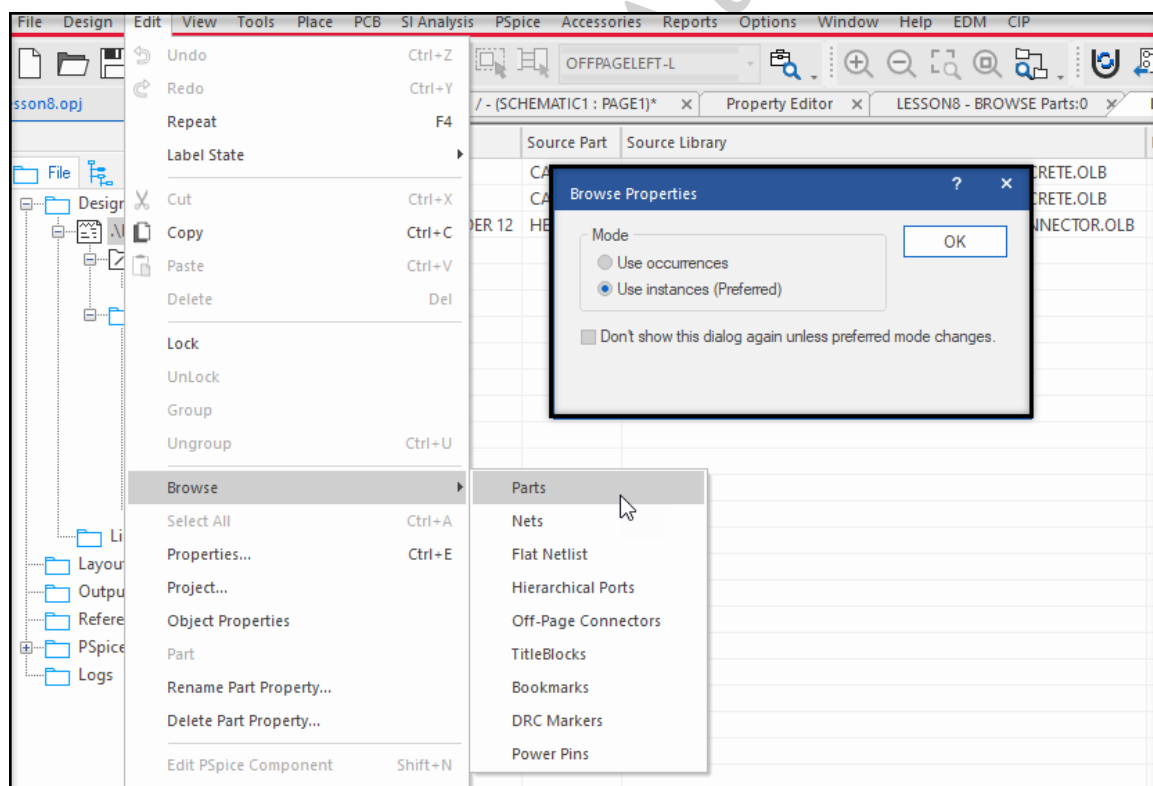
	Start Page ×	/ - (SCHEMATIC1 : PAGE1)* ×	**LESSON8 - BROWSE Parts** ×		

Part Reference	Value	Source Part	Source Library	Page	Schematic
C1	0.1uF	CAP	C:\CADENCE\SPB_17.4\TOOLS\CAPTURE\LIBRARY\DISCRETE.OLB	PAGE1	SCHEMATIC1
C2	0.1uF	CAP	C:\CADENCE\SPB_17.4\TOOLS\CAPTURE\LIBRARY\DISCRETE.OLB	PAGE1	SCHEMATIC1
J1		HEADER 12	HEADER 12 C:\CADENCE\SPB_17.4\TOOLS\CAPTURE\LIBRARY\CONNECTOR.OLB	PAGE1	SCHEMATIC1

When the Project Manager window is active, the *Edit* pull-down menu contains some commands not available elsewhere. For example, the *Edit > Browse* command lets you search by object type (for example, parts, nets, and so on). You can select specific schematic pages in the Project Manager to limit the search.

The results are displayed in a browser window. Use the browser window to select and locate any part or net in the list. (The Project Manager will automatically open the applicable page for you.) If you make multiple selections in the browser window, you can edit them all at once from the Browse Spreadsheet window.

With the Project Manager window active, you can also use the *Edit* pull-down menu to rename or delete a user-defined property in the design.

Lab 10-2: Using the Edit Find Command

Lab Objectives

- Using the *Edit Find* command

Setting up the Find method

Before you can use this method, you must de-activate the *Search Toolbar*.

1. From the *Tools* pull-down menu select *Customize*.
2. When the Customize window opens, remove the check-mark for the *Search*. Then click on *OK*.
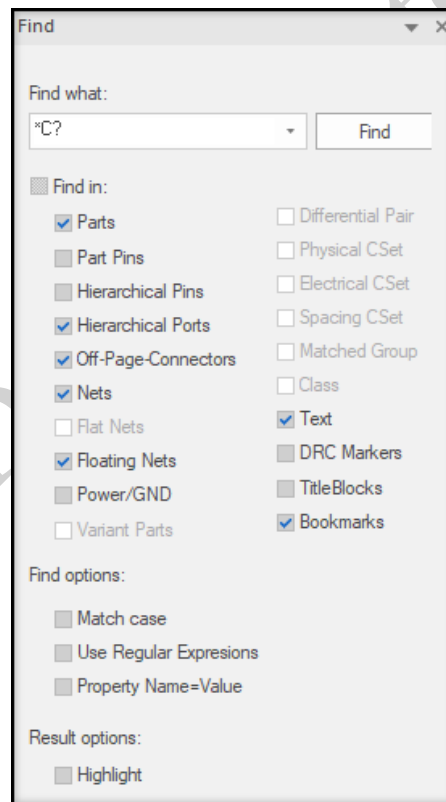
3. Now go the *Options > Preferences*, and select the *Miscellaneous* tab. When that opens, look on the lower left and remove the check-mark form *Find/Search Toolbar*. Select the *OK* to apply the setting.
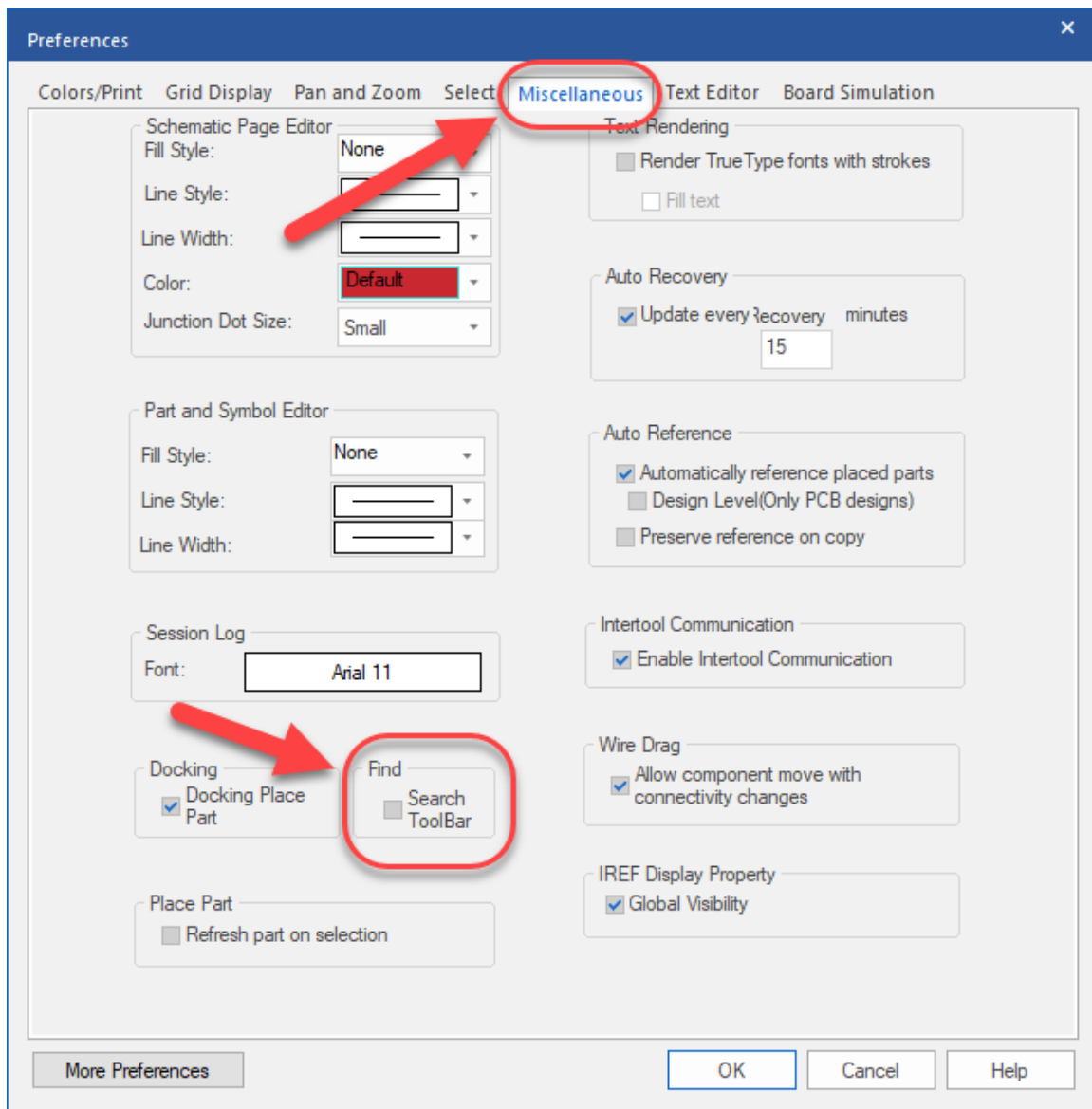
Searching the Current Page

1.   With the schematic page active, select **Edit > Find** or **<Ctrl+F>**.
2.   In the **"Find What"** box, make sure there is an " ***** ". This means **"all"**.
3.   Set the Scope to **Nets** and click **OK**.
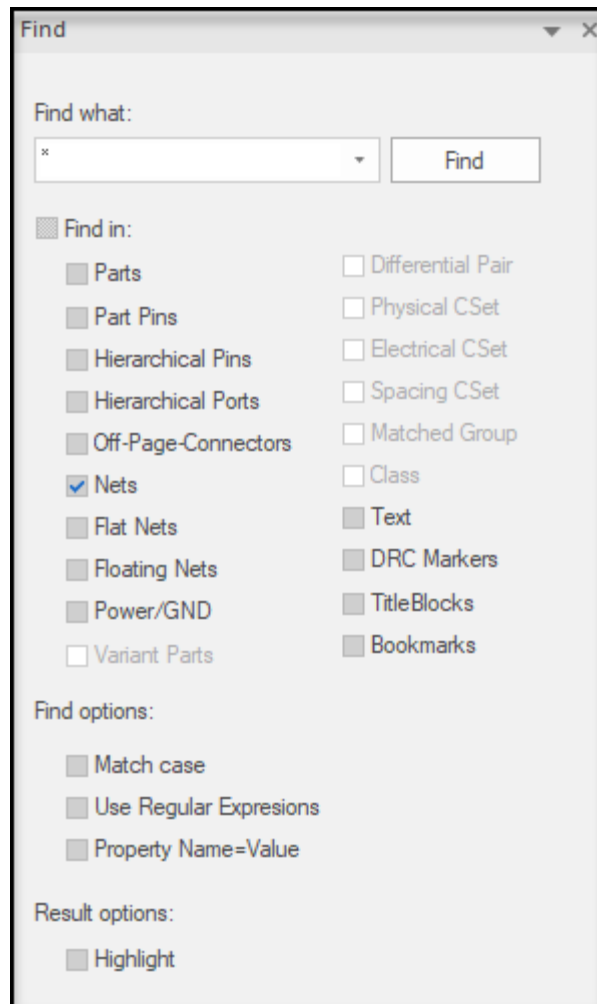
All nets on the current page are selected. This provides a convenient way to select all objects of a desired type, without selecting everything else on the page.

4. In the **Find Results** section select all nets.
5. RMB Edit properties or use keyboard command <Ctrl+Shift+E>
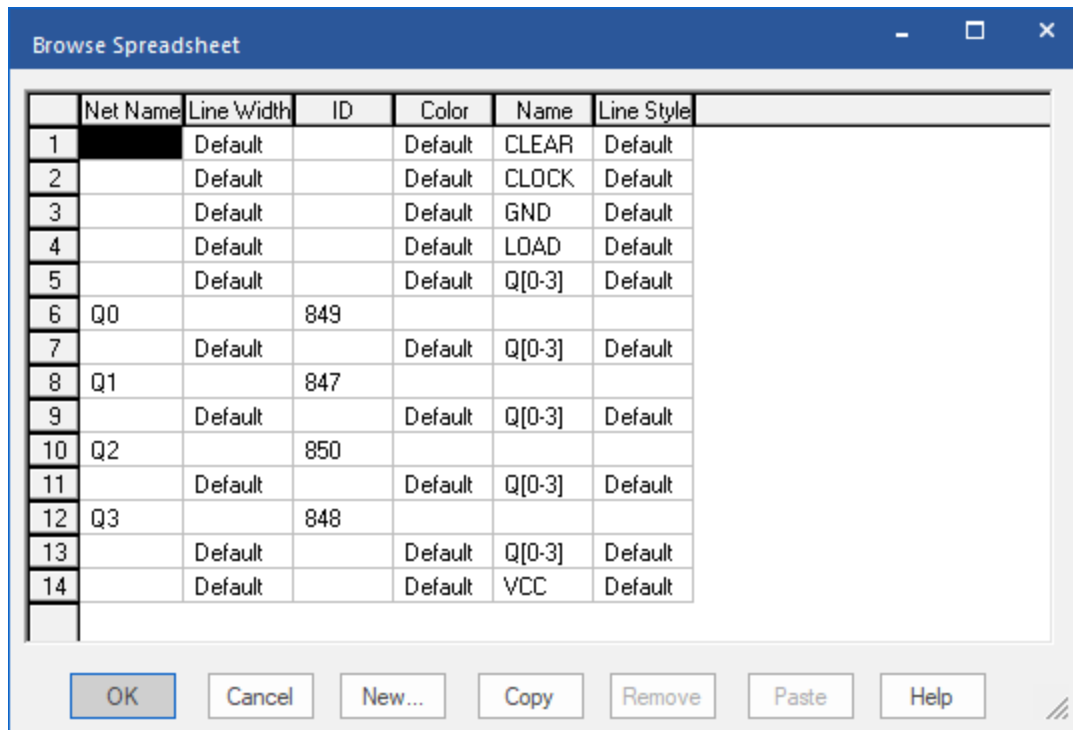
The Browse Spreadsheet will appear, from this window you can edit add or remove properties. The following exercise illustrates adding a property to a wire. The Property for width in the OrCAD and Allegro PCB Editor is *MIN_LINE_WIDTH*.
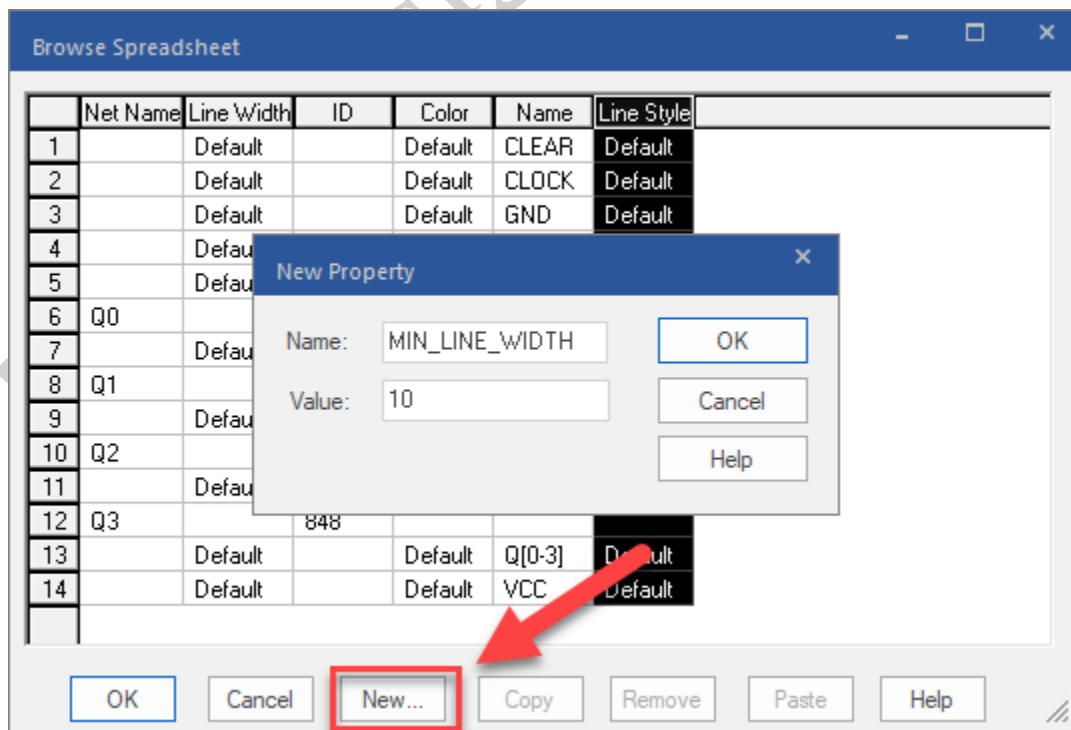
6. Give the New Property a Name and a Value. Although the MIN_LINE_WIDTH property is listed in the Cadence-Allegro property filter, we will add it here as an example for adding a property to a wire on the "fly".

Browse Spreadsheet

	Net Name	Line Width	ID	Color	Name	Line Style	MIN_LINE_WIDTH	
1		Default		Default	CLEAR	Defaul	10	
2		Default		Default	CLOCK	Defaul	10	
3		Default		Default	GND	Defaul	10	
4		Default		Default	LOAD	Defaul	10	
5		Default		Default	Q[0-3]	Defaul	10	
6	Q0		849				10	
7		Default		Default	Q[0-3]	Defaul	10	
8	Q1		847				10	
9		Default		Default	Q[0-3]	Defaul	10	
10	Q2		850				10	
11		Default		Default	Q[0-3]	Defaul	10	
12	Q3		848				10	
13		Default		Default	Q[0-3]	Defaul	10	
14		Default		Default	VCC	Default	10	

OK   Cancel   New...   Copy   Remove   Paste   Help

7. Press **OK** to close the Browse Spreadsheet window.

Saving the Page

1. Select **File > Save** and **OK** to save all pages of the design.
2. Close the schematic page.
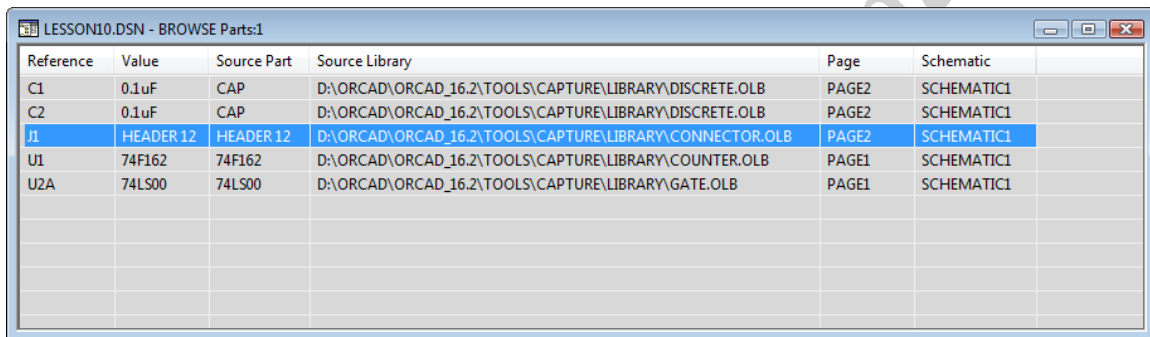
Lab 10-3: Using the Browse command

Lab Objectives

- Using the ***Browse*** Command

The **Browse** *command*

1. In the Project Manager window, click on .\Lesson8.dsn.
2. Select ***Edit > Browse >Parts*** and ***OK*** to list all parts in the design.
3. Double click part ***J1***.

Reference	Value	Source Part	Source Library	Page	Schematic
C1	0.1uF	CAP	D:\ORCAD\ORCAD_16.2\TOOLS\CAPTURE\LIBRARY\DISCRETE.OLB	PAGE2	SCHEMATIC1
C2	0.1uF	CAP	D:\ORCAD\ORCAD_16.2\TOOLS\CAPTURE\LIBRARY\DISCRETE.OLB	PAGE2	SCHEMATIC1
J1	HEADER 12	HEADER 12	D:\ORCAD\ORCAD_16.2\TOOLS\CAPTURE\LIBRARY\CONNECTOR.OLB	PAGE2	SCHEMATIC1
U1	74F162	74F162	D:\ORCAD\ORCAD_16.2\TOOLS\CAPTURE\LIBRARY\COUNTER.OLB	PAGE1	SCHEMATIC1
U2A	74LS00	74LS00	D:\ORCAD\ORCAD_16.2\TOOLS\CAPTURE\LIBRARY\GATE.OLB	PAGE1	SCHEMATIC1

Page two of the schematic is automatically opened, and the ***HEADER12*** connector selected.

4. Place your cursor in the schematic window, right-click, and select ***Edit Properties***.
5. In the Property Editor window, pivot the property names so that they appear in a list on the left.
6. Assign a ***PART_NUMBER*** and ***PCB Footprint***, as shown in the following graphic.

7. Close the Property Editor window and press **<Esc>** to deselect all parts.
8. Close and **Save** the design.

Lesson 11: Preparing the Schematic for the OrCAD and Allegro PCB Editor

Lesson Objectives

After completing this lesson, you will be able to:

- Add an OrCAD and Allegro PCB Editor part or net property to your design

Part Properties for OrCAD and Allegro PCB Editor

There many **OrCAD and Allegro PCB Editor** part properties you can add to your design. These are some of these properties are listed and described in the table that follows.

OrCAD and Allegro PCB Editor Part Properties	Description
* PCB Footprint	Defines the OrCAD and Allegro PCB Editor footprint pattern. All OrCAD and Allegro PCB Editor parts are required to have this property. The name of this property can be up to 255 characters long and may include the characters A-Z, 0-9, as well as the dash (hyphen) and underscore characters. Blank spaces within the name are not permitted.
ALT_SYMBOLS	Lists of alternate footprint patterns you can use apply to your part during PCB layout. Each alternate footprint must have the same pin count.
* CLASS	Classify the part as an IC, IO, or a Discrete device. This property is required in the OrCAD and Allegro PCB Editor to assist in the component type selection during part placement and is required if you are exporting your design to SPECCTRAQuest. The Discrete and IO part classes affect the model assignment process in SPECCTRAQuest.
ROOM	Assigns the part to a specific area or room of the PCB.

POWER_GROUP	Used to assign alternate voltages to parts. This property overrides the voltages specified by invisible power pins with the names of alternate voltages you want the part to use.

In the previous table, the * symbol before a Part Property indicates a required property.

To associate any of the properties shown in the above table to a part, select one or more parts, and set the *Filter by* field to *Cadence-Allegro* in the OrCAD Capture *Property Editor*.

Grouping Parts for PCB Design
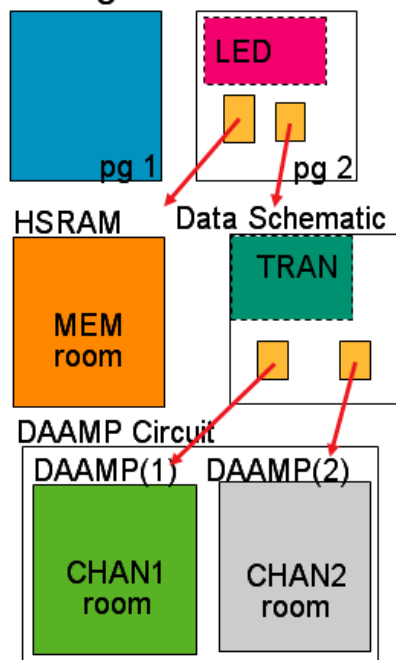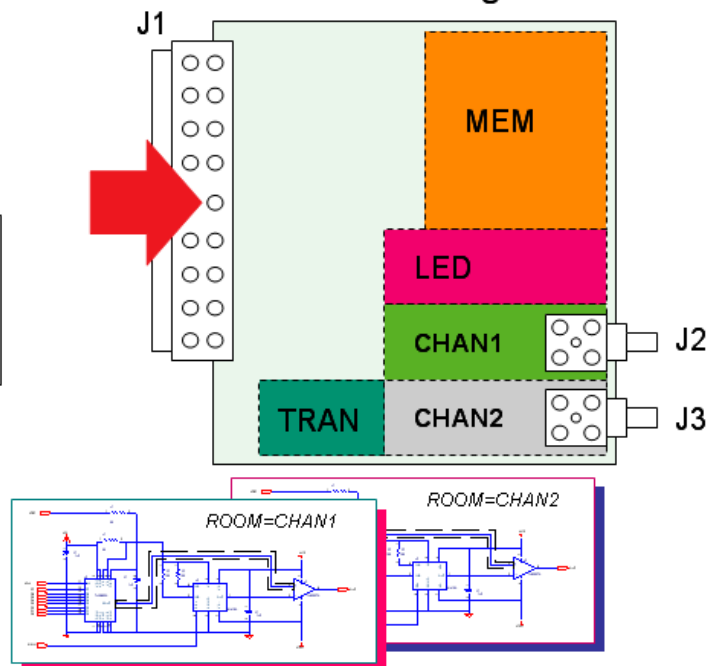
The above graphic illustrates how the *ROOM* property can be used to define areas in advance within the schematic. Next, when the PCB board is ready for component placement, the ROOMS property will make it easier to allocate the designated components into their respective areas on the board.

Net Properties for the ORCAD and Allegro PCB Editor

There are many different ORCAD and Allegro PCB Editor net properties you can add to your design. The table that follows describes five commonly used OrCAD Capture to OrCAD and Allegro PCB Editor net properties.

OrCAD and Allegro PCB Editor Net Properties	Description
MIN_LINE_WIDTH	This property attaches to a net and sets the minimum width of a net or connect line. Use this property to override the *Min line width* rule in a physical constraint set. It also overrides the line *Width* electrical constraint for a differential pair, if set. The value of this property is in design units.
MAX_LINE_WIDTH	This property attaches to a net and sets the maximum width of a net or connect line. Use this property to override the *Max line width* rule in a physical constraint set.
NET_PHYSICAL_TYPE	This property attaches to a net. This translates to the name of the **Physical** > **Net Class** to be applied in the OrCAD and Allegro PCB Editor.
NET_SPACING_TYPE	This property attaches to a net. This translates to the name of the **Spacing** > **Net Class** to be applied in the OrCAD and Allegro PCB Editor.
PROPAGATION_DELAY	This property defines the minimum and maximum propagation delay constraint between any pair of pins or rat-Ts in a net or an extended net (Xnet).
RELATIVE_PROPAGATION_DELAY	This property replaces MATCHED_DELAY in Allegro PCB Editor version 14.0. It is an electrical constraint attached to pin-pairs on a net. It specifies a group of pin-pairs that are required to have interconnect propagation delays matching a specified delta (offset) and tolerance. A RELATIVE_PROPAGATION_DELAY group has one or more reference pin-pairs against which all other pin-pairs in the group are compared.

To attach the properties shown in the list, select one or more nets, and set the *Filter by* field to *Cadence-Allegro* in the *Property Editor*. Then choose the corresponding net property from the list that displays.

Create Differential Pairs between flat nets

OrCAD Capture has the ability to allow you to create Differential Pairs between flat nets in your design for the Allegro PCB Editor. You can set the **DIFFERENTIAL_PAIR** property on selected nets using the **Property Editor**. See the Property filter for Cadence-Allegro.

There is also a command **Create Differential Pair** located on the Tools pull-down menu. In the **Create Differential Pair** dialog box you can view the nets defined in your design and create a Differentials Pair between selected nets.
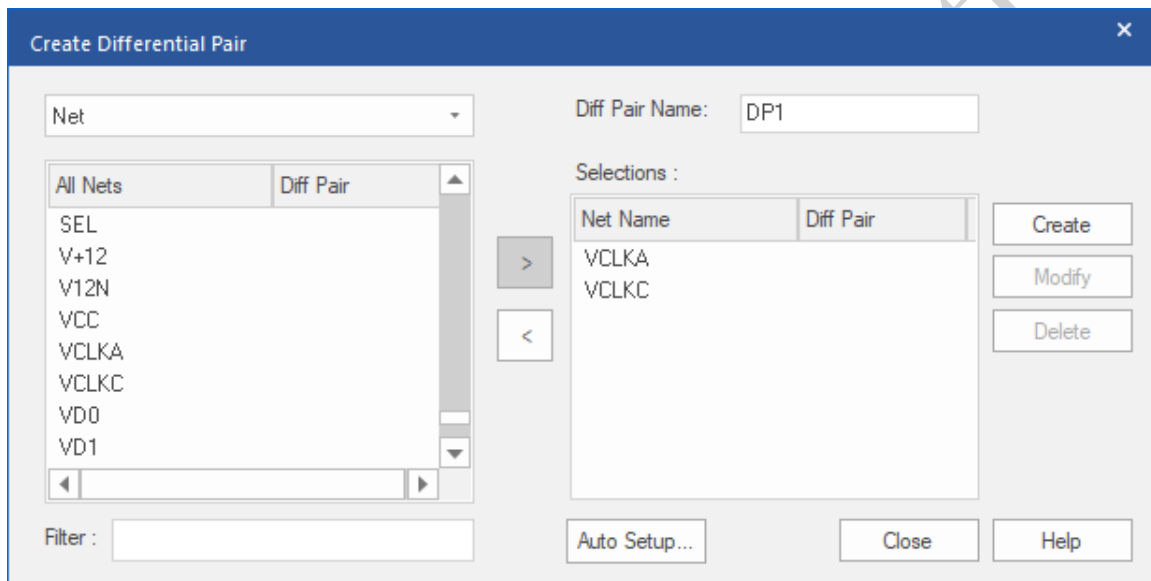
Back Annotating the Schematic
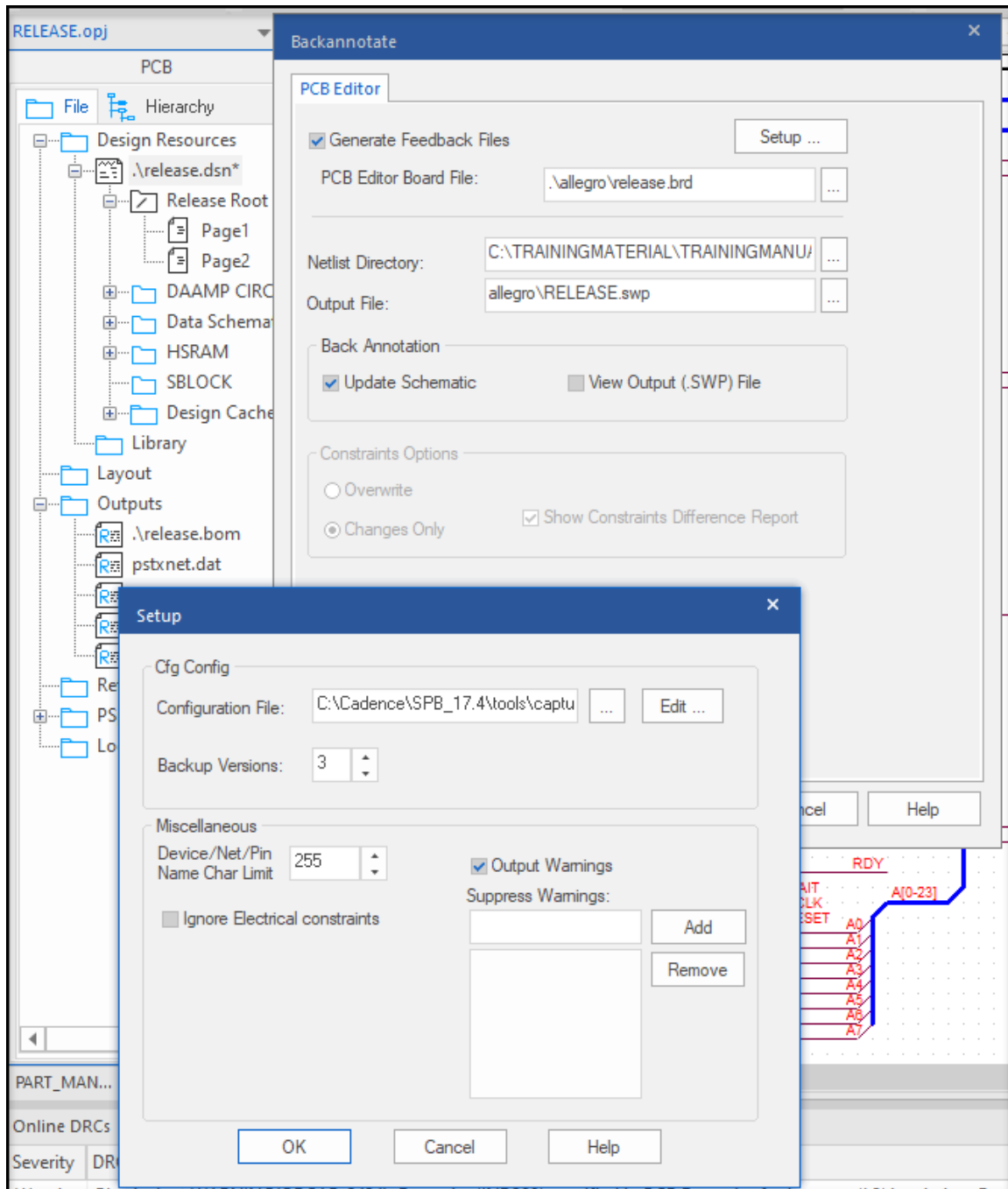
OrCAD Capture lets you backannotate schematics with design changes made in the OrCAD and Allegro PCB Editor. The OrCAD Capture *Back Annotate* command lets you:

- Change part references

- Change net properties
- Change part properties
- Swap gates and pins

When you use the OrCAD Capture back annotation command, you no longer need to export the net files from the OrCAD and Allegro PCB Editor.

In the OrCAD and Allegro PCB Editor, when you use the **File > Export Logic** command to send information back to the OrCAD Capture schematic, the OrCAD and Allegro PCB Editor produces four backannotation files:

- pinView.dat: Contains all pins and net connections
- netView.dat: Contains all net properties
- compView.dat: Contains all part properties
- funcView.dat: Contains all gate properties

These files can also be produced in OrCAD Capture using the **Tools > Back Annotate** command, and selecting the **Generate Feedback Files** option.

In OrCAD Capture, when you use the **Tools > Back Annotate** command, these four files are processed into a single backannotation file, called a **swap** file. The swap file is then used to update the OrCAD Capture schematic with new part references, as well as part and net properties.

All gate and pin swapping data is automatically included in the swap file. The backannotation of part and net properties is controlled by the Allegro.cfg file.

The **Export > Logic** files for the Back Annotation process must be in the same "Allegro" subdirectory as the original Net List files.

Lab 11-1: Adding OrCAD and Allegro PCB Editor Properties

Lab Objectives

After completing this lab you will be able to:

- Browse parts
- Add a PCB Footprint property
- Add a ROOM property

An alternate method for doing some of the following labs is to use the Property Editor and the *Edit Object Properties* method from the Project Manager window.

Browsing Parts

1. Choose *File > Open > Design* and navigate to C:/EMA_Training/Capture_Essentials/Designs/part_manager_training and select the Part_Manager_Training.dsn
2. In the Project Manager window, select .\Part_Manager_Training.dsn, then RMB click and select *Edit > Object Properties*. The *Property Editor* window opens displaying the current properties of all of the parts in the design.
3. Set Filter by to: *OrCAD PCB Designer Standard*.
4. Close the Property Editor window.
5. Select *Edit > Browse > Parts* from the main menu.
6. Click *OK* to continue.

   The *Browse Parts* window lists all the parts in the design.

Adding the PCB Footprint Property

1. Scroll down and double click on part reference *U2*.
2. When the schematic page opens note that U2 will be highlighted. Double click on U2 to open the Properties window.

   The *Browse Spreadsheet* displays properties currently attached to *U5*.

3. Locate the **PCB Footprint** column to verify a cell value of *SOIC32-600*.
4. If necessary, click in the cell, and enter:
      SOIC32-600
5. Click *Apply* to make the necessary change.

6. RMB on the Property Editor tab and close the window.
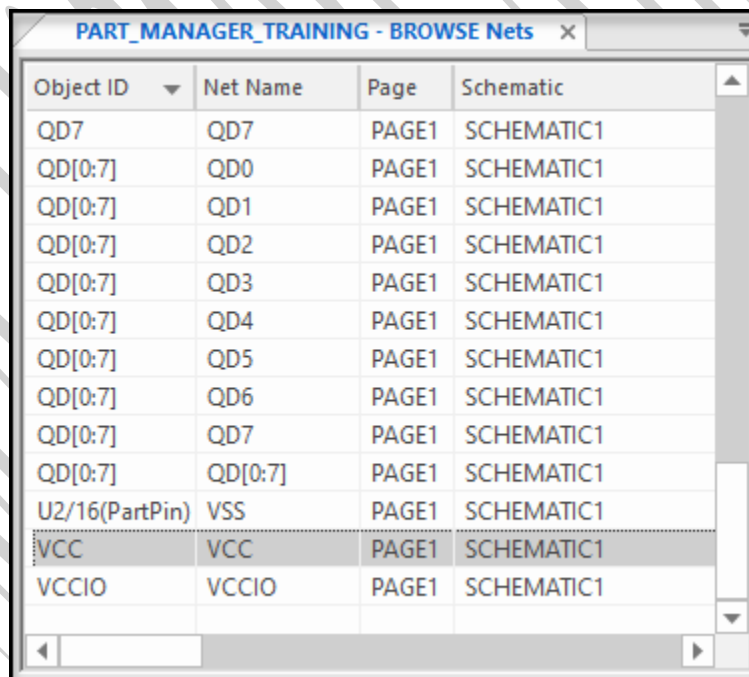7. Close the **Browse Parts** window, and save the design.

Browsing Nets

1. In the Project Manager window, click on .\Part_Manager_Training.dsn and select **Edit > Browse > Nets**.
2. Click **OK** to continue.

   The **Browse Nets** window lists all the nets in the design. You can also select the ObjectID column header to sort the nets.

Adding the MIN_LINE_WIDTH Property

The following is one method you can use to add net properties to wires.

1. You may wish to click in the Object ID field name to sort the net names for easier searching. Scroll down and locate VCC.

Object ID ▼	Net Name	Page	Schematic
QD7	QD7	PAGE1	SCHEMATIC1
QD[0:7]	QD0	PAGE1	SCHEMATIC1
QD[0:7]	QD1	PAGE1	SCHEMATIC1
QD[0:7]	QD2	PAGE1	SCHEMATIC1
QD[0:7]	QD3	PAGE1	SCHEMATIC1
QD[0:7]	QD4	PAGE1	SCHEMATIC1
QD[0:7]	QD5	PAGE1	SCHEMATIC1
QD[0:7]	QD6	PAGE1	SCHEMATIC1
QD[0:7]	QD7	PAGE1	SCHEMATIC1
QD[0:7]	QD[0:7]	PAGE1	SCHEMATIC1
U2/16(PartPin)	VSS	PAGE1	SCHEMATIC1
VCC	VCC	PAGE1	SCHEMATIC1
VCCIO	VCCIO	PAGE1	SCHEMATIC1

PART_MANAGER_TRAINING - BROWSE Nets ✕

2. Double click on the VCC line to open the schematic. Notice that the schematic page shows the highlighted net VCC.
3. RMB click and choose **Edit > Properties** from the pull-down menu (or **CTRL+E**).

The Property Editor opens and displays properties currently attached to the VCC net. Add the following values if they have not already been added.

4. In the **MIN_NECK_WIDTH** field enter 8
5. In the **MIN_LINE_WIDTH** field enter 15
6. Click **Apply** to continue.

MIN_BOND_LENGTH	
MIN_HOLD	
MIN_LINE_WIDTH	15
MIN_NECK_WIDTH	8
Name	VCC
NET_PHYSICAL_TYPE	
NET_SPACING_TYPE	

7. Close the Browse Nets window, and save the design.

Adding a ROOM Property

1. In the Project Manager window, double click on **PAGE1** to open it.
2. When PAGE1 opens select the components as shown in the following graphic: C1, C2, C3, C4, C5, C6, and U5. You may wish to use the **View > Selection Filter** to choose only **Parts** for selection.
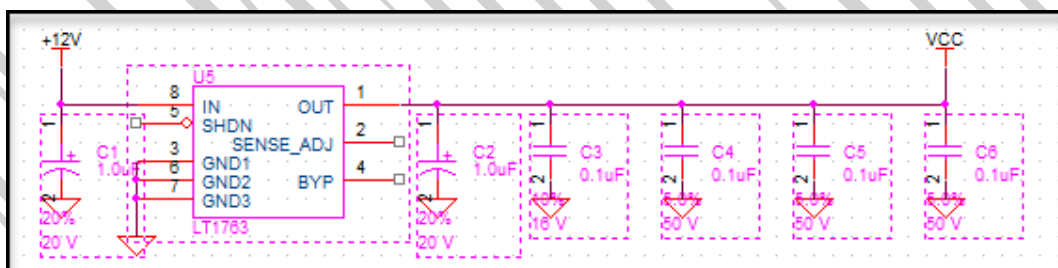
3. Next, RMB and choose **Edit – Properties**

4. Scroll down in the Property Editor and notice that there are no settings for the ROOM property for the selected parts.

5. In the ROOM field under C1 enter the value ROOM1.

6. Do this for each of the parts. Optionally you can select the first ROOM1 cell and drag (from the lower right corner of the selected cell) across the rest of the ROOM cells for all of the parts. This will copy the ROOM1 entry across all those cells.

Primitive	DEFAULT	DEFAULT	DEFAULT	DEFAULT	DEFAULT	DEFAULT	DEFAULT
RATED_MAX_TEMP							
REFERENCE	C1	C2	C3	C4	C5	C6	U5
ROOM	ROOM1	ROOM1	ROOM1	ROOM1	ROOM1	ROOM1	ROOM1
SIGNAL_MODEL							
Source Library	C:\EMA\CIP-E\SCHEMA	C:\EMA\CIP-E\SCHEMA	C:\EMA\CIP-E\SCHEMA	C:\EMA\CIP-E\SCHEMA	C:\EMA\CIP-E\SCHEMA	C:\EMA\CIP-E\SCHEMA	C:\EMA\CIP-E\SCHEMA

7. Click *Apply* to activate the change.
8. Close the Property Editor page.
9. Save the design.

Closing the Part_Manager_Training Project

1. Select *File > Close*.

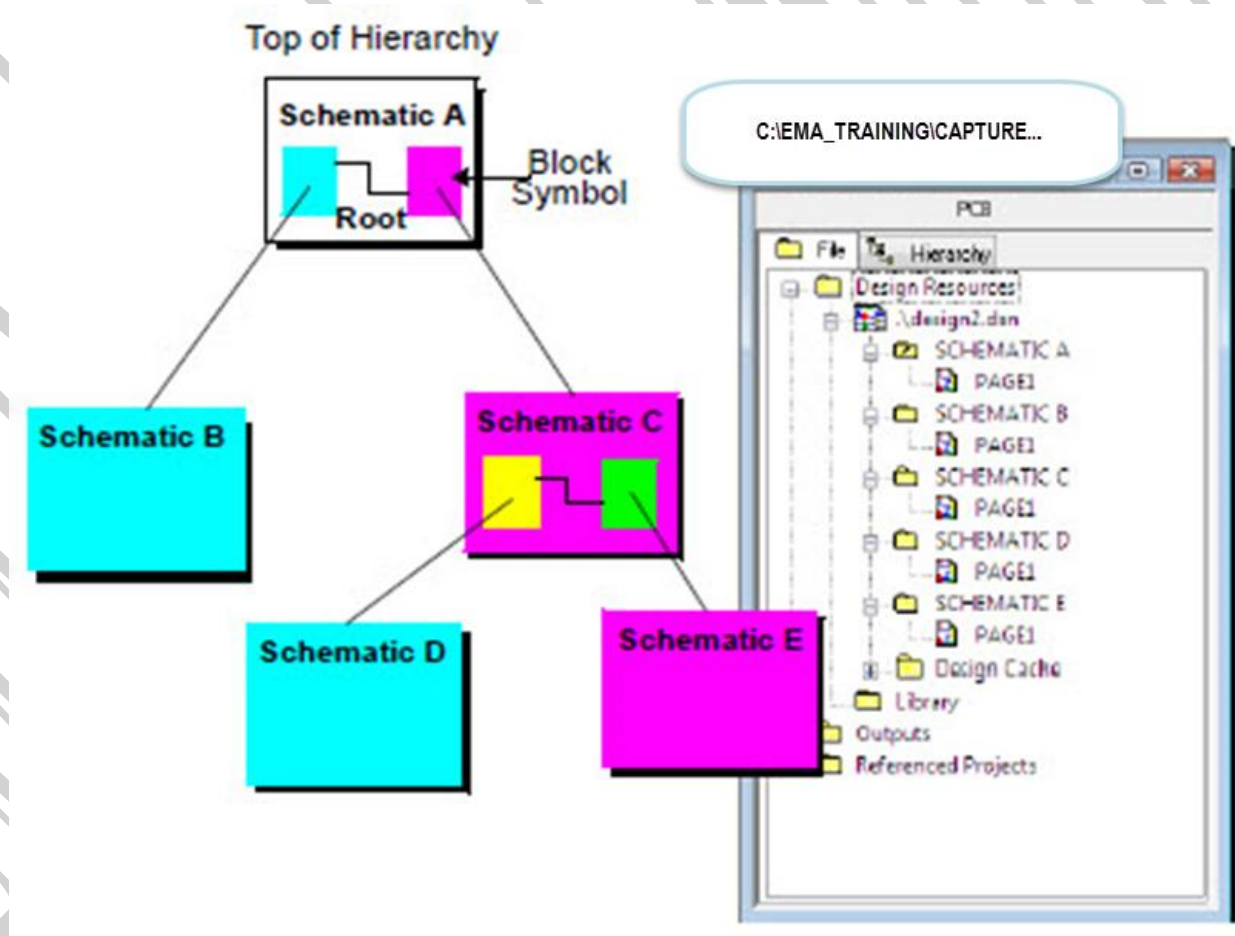This completes Capture Essentials training.

# Appendix A: Working with Hierarchy in Capture

Hierarchical Design

When you embed a schematic folder inside another schematic folder, you have created a hierarchical design. The process requires a block symbol, which represents a schematic or functional model. When the block symbol is used once in a schematic, the result is called a *simple* hierarchy (shown in the accompanying diagram). When the block symbol is used two or more times, the design is called a *complex* hierarchy.

More Information

The root schematic represents the top of the hierarchy and is displayed in the Project Manager window with a slash on the folder icon.

A hierarchical design has several advantages. Its block diagram structure clearly shows how the functional units interact, and each block can be reused in the same or future designs. When you edit a block, all instances of the replicated block are changed throughout the hierarchical design.

Complex Hierarchical Designs
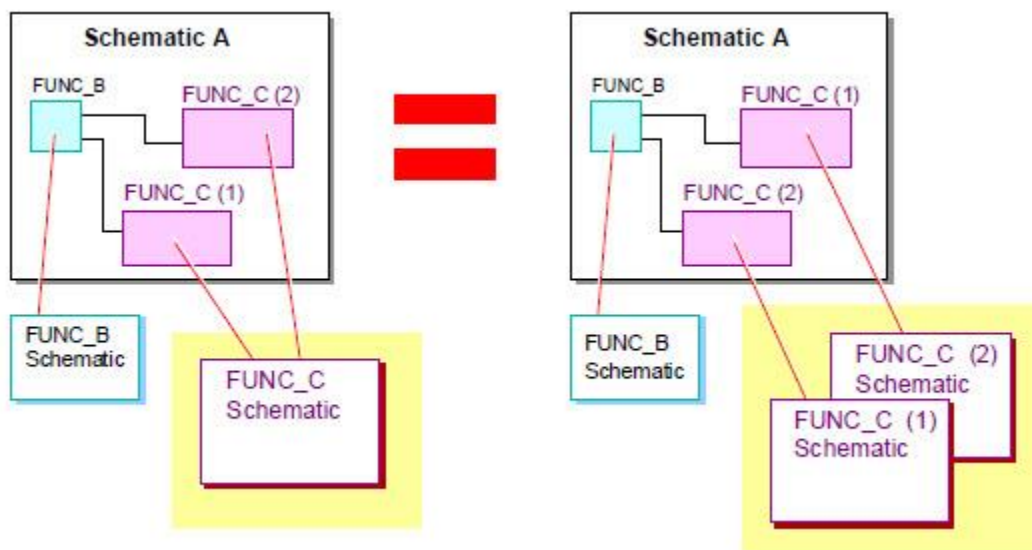
In a complex hierarchical design, OrCAD Capture automatically maintains multiple copies of the schematic in its database, one for each time it is referenced.
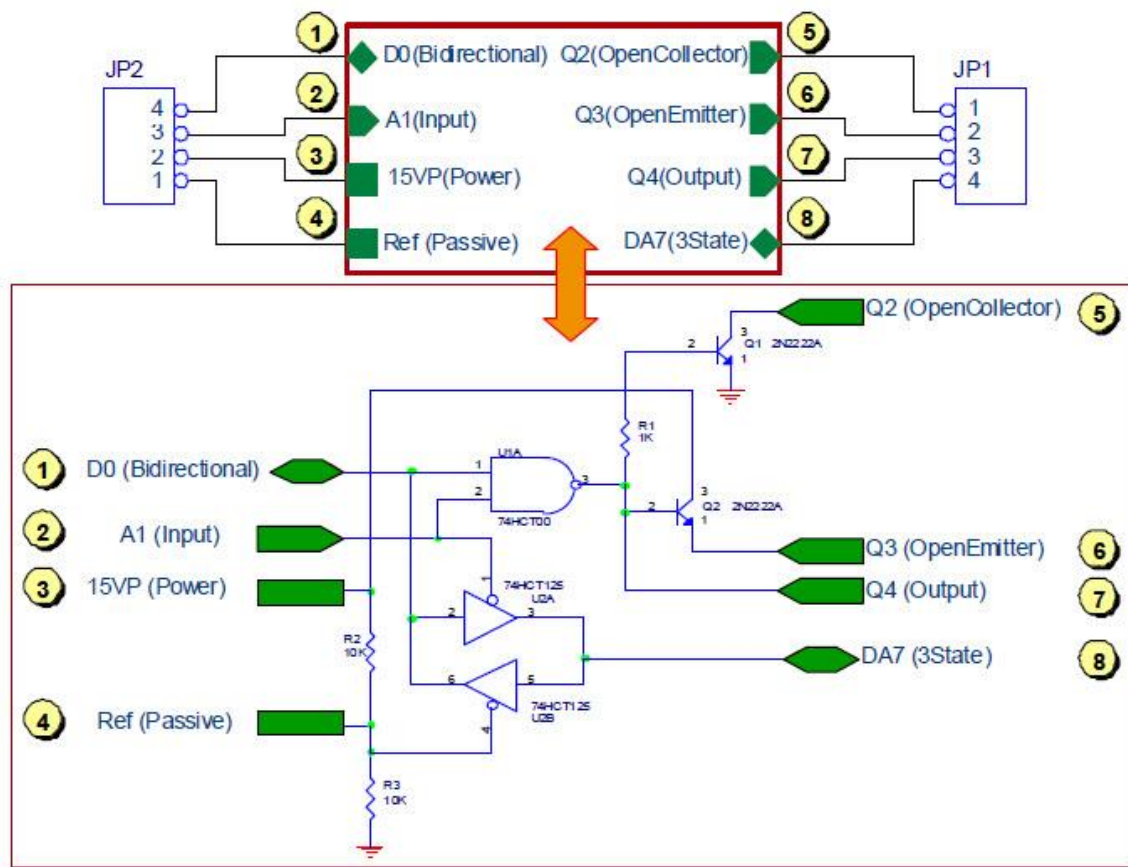
Connectivity in Hierarchical Designs
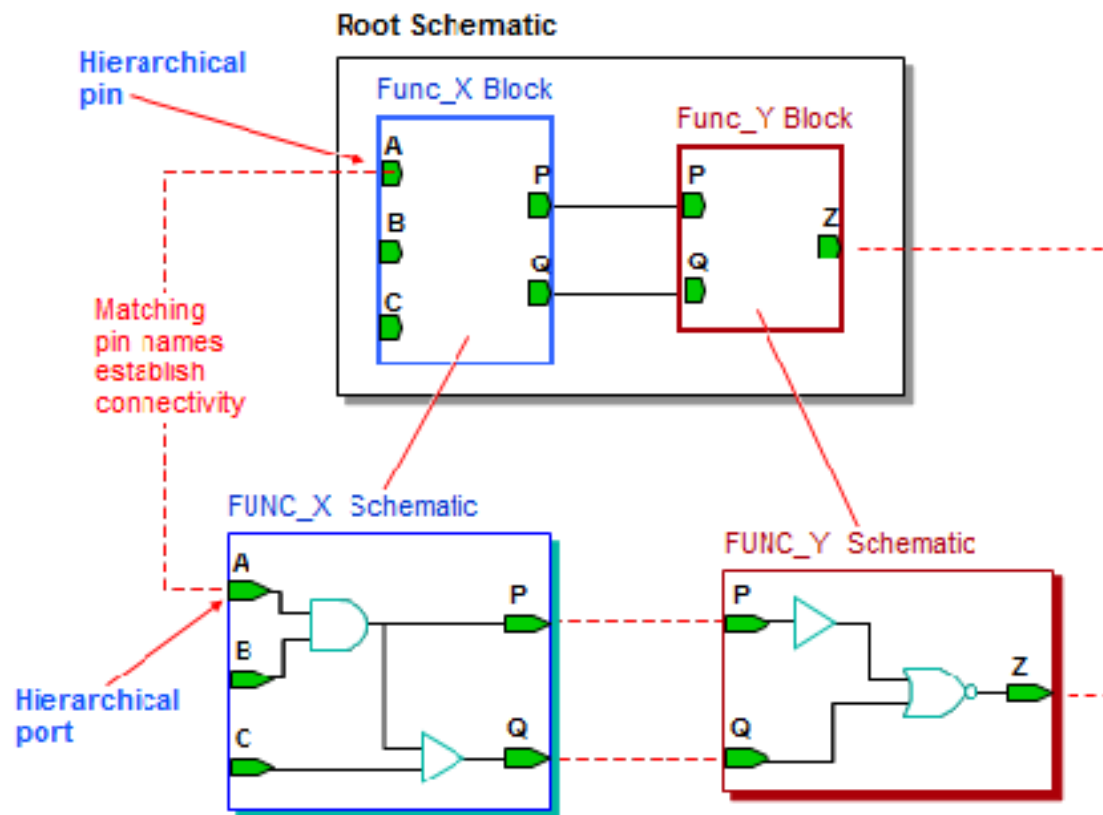
In flat designs, all the schematic pages are at the same level. You establish inter-page connectivity by terminating nets with off-page connectors.

By contrast, hierarchical designs have what might be called nested functionality embedded in schematic pages. Hierarchical block symbols on one schematic page reference more detailed design descriptions (other schematics or VHDL models).

More Information

The hierarchical pins on the block symbol and the hierarchical ports in the schematic (or VHDL file) are used to establish connectivity between the schematic folders in a hierarchical design.

Hierarchical Pins and Port Types

There are eight types of hierarchical ports, each with its own corresponding hierarchical pin type (as shown in the accompanying illustration).

To establish connectivity, a hierarchical pin must have a matching hierarchical port with the same Name and Type.

Adding a Port Symbol

In OrCAD Capture, there are eight port symbols to choose from. The symbol names relay information to the user about the direction in which the port is pointing (graphically), and the location of the pin (connection point). For example, **PORTLEFT-R** is a port symbol that points to the left, with a pin on the right. You may need to rotate port symbols before you place them.

More Information

You need to name the port to establish connectivity between the wire on the port, and the hierarchical pin on the block symbol. You can enter the name while placing the port, or change the name after you have placed it.

When adding a port, you must specify the port type. The port type provides a logical description of the net it represents (for example, input, output, or bidirectional).

Each of the eight symbols has a default type setting. You can select the port type while placing it, or change it after it has been added.

| Port symbols	Default type
PORTBOTH-L PORTBOTH-R	Bidirectional
PORTLEFT-L PORTLEFT-R	Output
PORTNO-L PORTNO-R	Passive
PORTRIGHT-L PORTRIGHT-R	Input

Hierarchical Design Methods
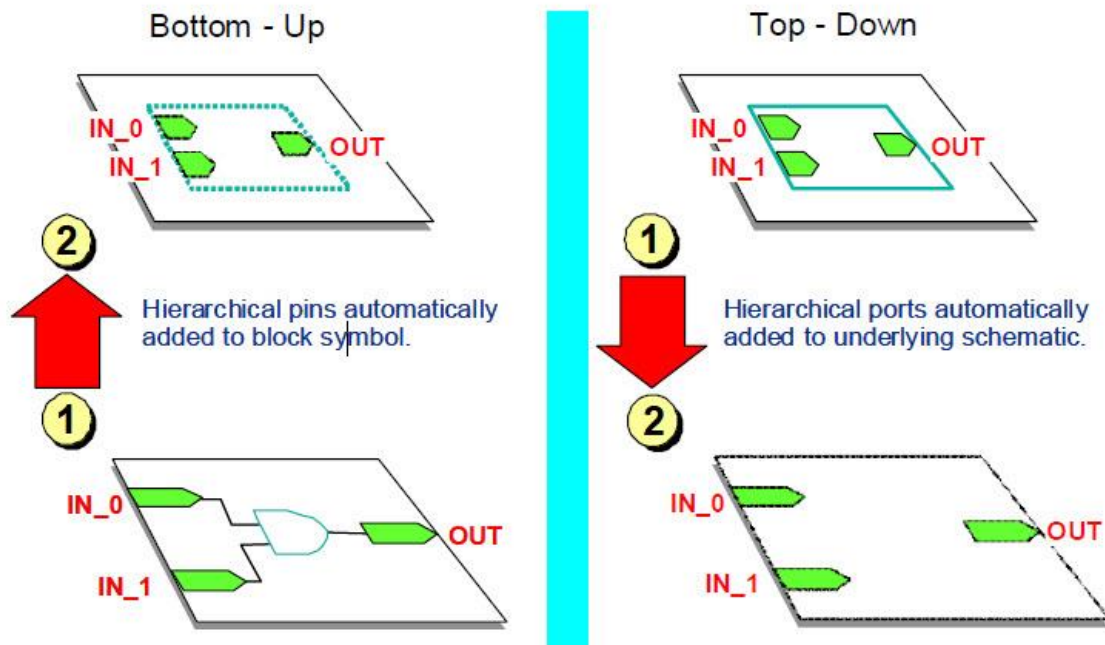
Some degree of automated design supports either a bottom-up or top-down design flow. If you create additional hierarchical ports in the schematic or additional pins in a hierarchical block symbol, use the *Synchronize Up* or *Synchronize Down* command to re-link the data.
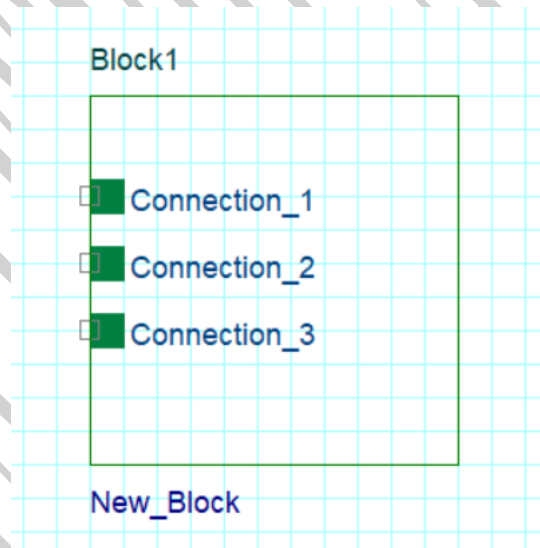
Drawing a Hierarchical Block

The following steps are used to generate a Hierarchical Block:

- Blocks may be drawn by using the **Place > Hierarchical Block** command from the Place menu, or by using the command icon.
- When the dialog opens give the block a Reference name, an Implementation name, and choose Schematic View for the Implementation Type, then click OK.
- Drag a rectangle to indicate the block area.

Block1

New_Block
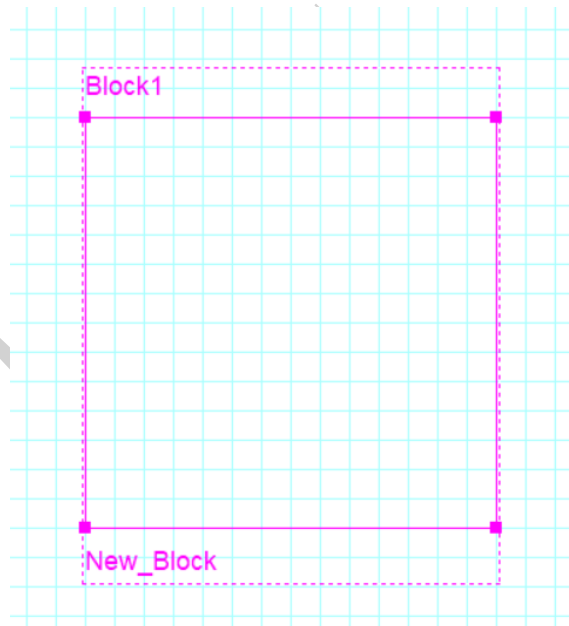
- Add hierarchical pins to the block by making sure that the block is selected, then choose **Place >> Hierarchical Pin** from the Place menu.

Block1

Connection_1

Connection_2

Connection_3

New_Block

Since a Hierarchical Block represents a schematic page, the block must be synchronized with the schematic page it will be representing.

- Therefore you will need to "push" into the block to add the schematic page and complete the synchronization between the block and its schematic page. Do this by *double-clicking* on the block.

The new schematic page will open, and the **Hierarchical Ports** will appear in the upper left corner of the schematic. These may be moved to a preferred location on the page.
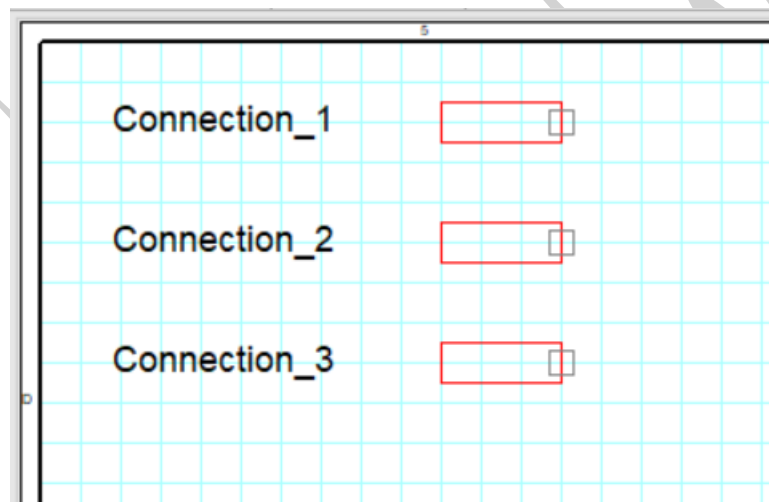
When making changes to either a hierarchical block, or to the schematic it represents, the changes must be synchronized so that both the page and the block connectivity aligns. This alignment process must occur at the block level, where the block is selected, then RMB and choose either **Synchronize Up** -or- **Synchronize Down**.

Synchronizing Up is when you have made changes at the schematic level, (for instance adding new hierarchical ports). The new ports must be made visible on the block to complete the connectivity between schematic page and block.

Synchronizing Down is when you have made changes at the block level (for instance adding H-pins). The new h-pins must be made visible on the schematic page to complete the connectivity between the block and the schematic page.