# OrCAD 'QUICK START' Guide #2

# Designing a Circuit with 'Capture CIS'

## Introduction

Everything starts with a project, in which you save one or more schematics and with which parts libraries, etc, are associated.

Once you open a project, you can start drawing a circuit schematic and selecting parts.

Start by launching **Capture CIS.**

## Step #1 – Creating a project folder

In the top menu bar in the application window, choose ***File – New – Project*** to create a new project.

In the **New Project** dialogue box that appears, as shown in Figure 1 below, specify a ***Name*** and a ***Location*** for your project.

Also select ***Analog or Mixed A/D*** to create analogue or mixed-signal designs to be used with PSpice later.

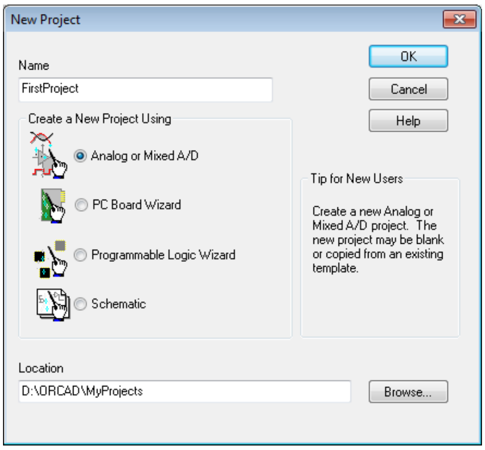



Figure 1. New Project Dialog Box

In the **Create PSpice Project** dialog box that appears, as shown in Figure 2, select ***Create a blank project*.**

**Note**: In Capture, you can either create a blank project (ie from scratch) or base the new project on an existing project (ie an existing schematic). To create a project based on an existing project, select ***Create based upon an existing project***. This will create a new project with the same name and files as the selected existing project.

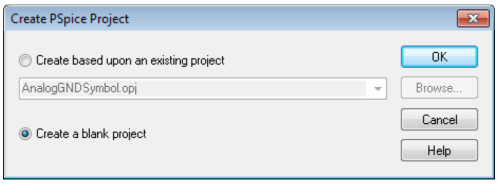



Figure 2. Create PSpice Project Dialog Box

## Step #2 – Entering your schematic

Once a new project is created, a blank schematic entry page with the name PAGE1 is displayed, as shown in Figure 3.

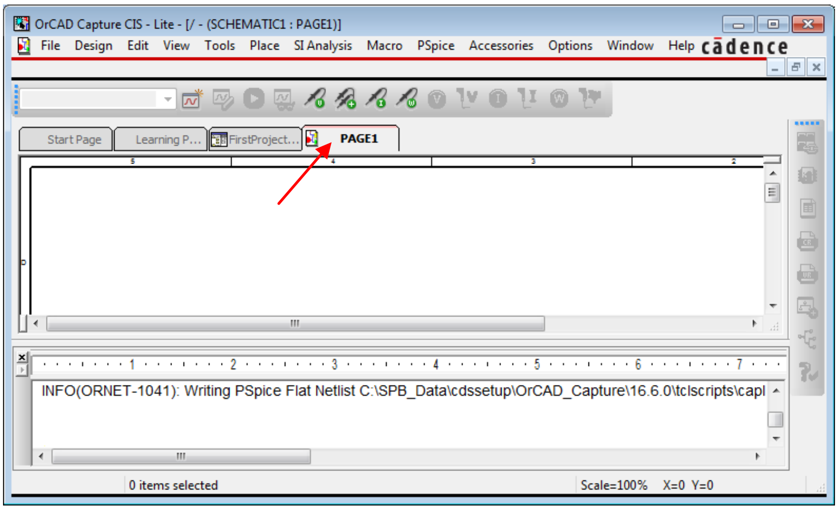



Figure 3. Capture Window with New Schematic Page

## Step #3 - Placing Parts

You can place parts **one of two ways**:

- place parts that are simulation-ready (ie that have PSICE behavioural models associated with them)

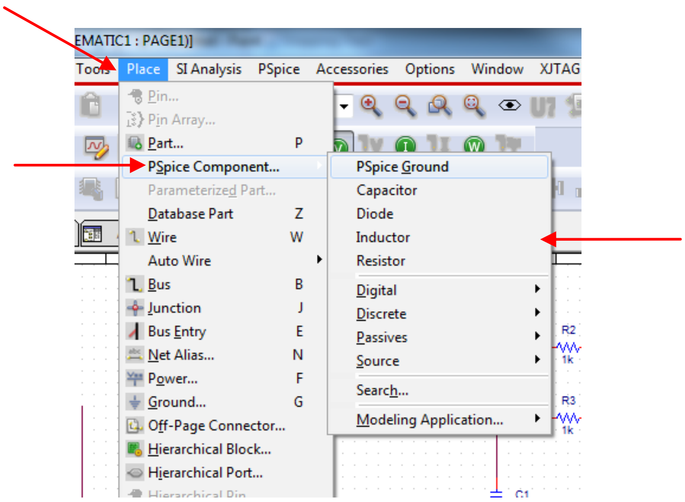
  ***Or***

- place parts that have no PSpice models associated with them, but to which you can add specification data later

To place **simulation-ready** parts (recommended to get you started):

From the TOP MENU use ***Place — PSpice Component***

- To place a resistor, choose *Place — PSpice Component — Resistor* and then click on the schematic canvas to place it.
- To place a capacitor: Choose *Place — PSpice Component — Capacitor*, right-click and choose *Rotate* to rotate the capacitor, and then click on the schematic canvas to place it.
- To place a DC voltage source, choose *Place — PSpice Component — Source — Voltage Sources — DC* and click on the schematic canvas to place it.
- To place a ground, choose *Place — PSpice Component — PSpice Ground* and click on the schematic canvas to place it.



To place **non-simulation-ready** parts (which you can edit later):

From the TOP MENU use ***Place — Part***

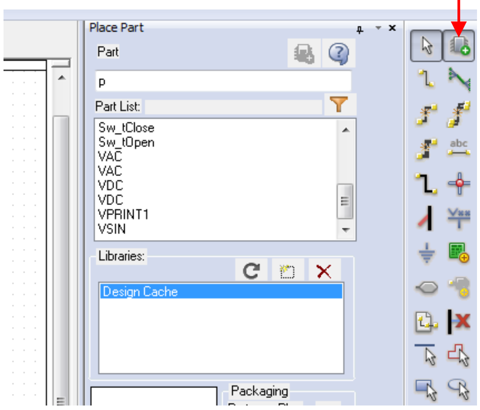
***Or***

Use the keyboard shortcut ‘P’ which opens the Part selector as shown below

***Or***

Click directly on the RIGHTHAND MENU icon shown here:

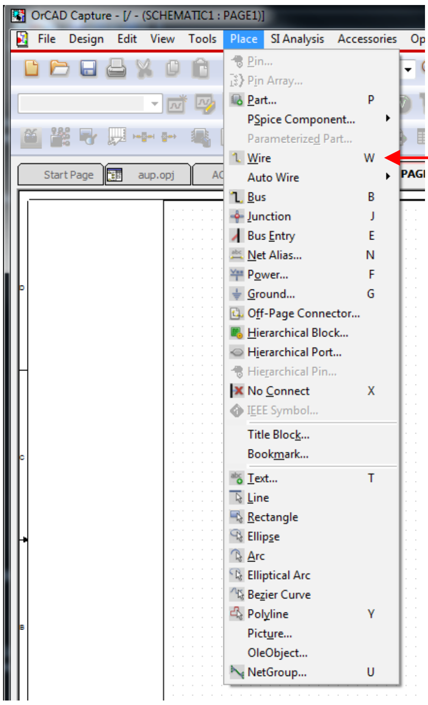
Until you’re ready to find specific parts,
we recommend using PSpice Components,
as above.




## Step #4 - Connecting Parts

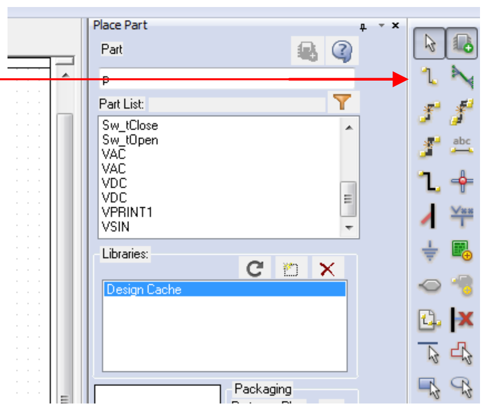
You can connect parts **one of three ways**:

- From the TOP MENU, choose *Place — Wire*. Then, click on the respective pins on the schematic to connect components. Press Escape when done.




***Or***

- From the RIGHTHAND MENU, choose the desired drawing function. The one shown here is the Wire function.




***Or***

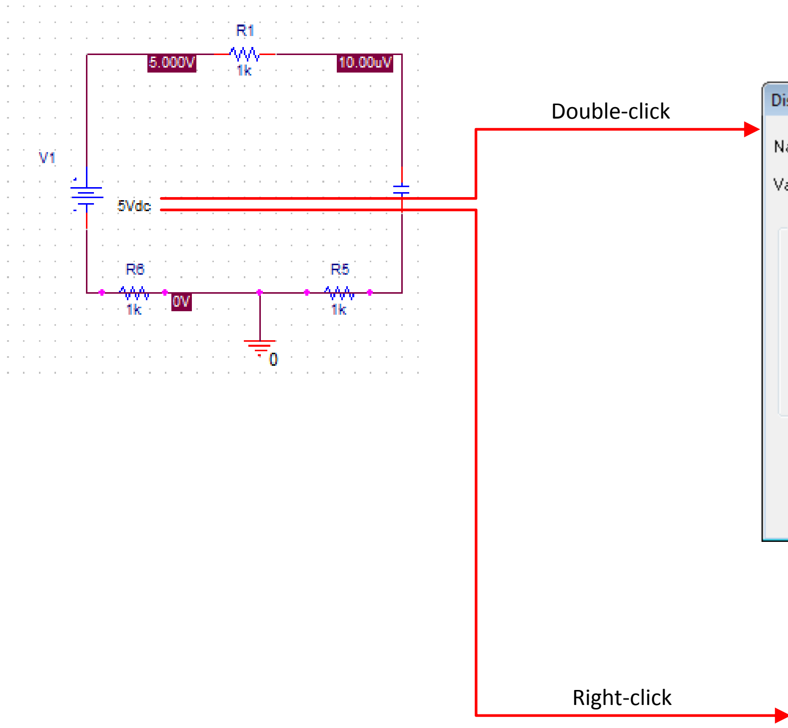
- Use the keyboard shortcuts W (= Wire)
  (Refer ‘Keyboard Shortcuts’ guide)

## Step #5 - Editing Part Properties

The placed parts have their own default properties and values. You can edit many of these properties, such as voltage, impedance, etc.

To edit a part's properties, put your cursor on the value you wish to change and double-click it. Any value, whether resistor values, source voltages, etc, can be edited.

This will open the **Display Properties** window, as shown in Figure 5. Here you can edit values as well as specify a *Display Format*, if needed.



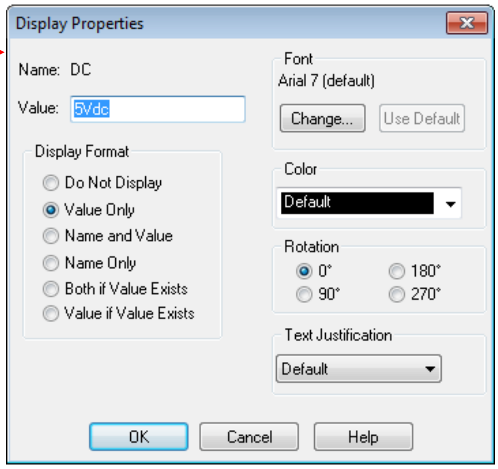
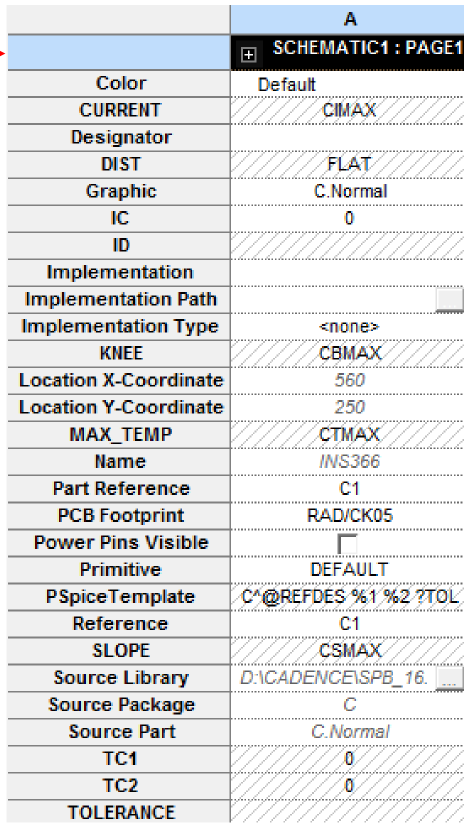




Figure 5. Display Properties Dialog Box

You can also select a part and **right-click** to choose *Edit Properties* to change property values. Open the **Edit Properties** window, as shown in Figure 6, for the capacitor.



| | A |
|---|---|
| | SCHEMATIC1 : PAGE1 |
| Color | Default |
| CURRENT | CIMAX |
| Designator | |
| DIST | FLAT |
| Graphic | C.Normal |
| IC | 0 |
| ID | |
| Implementation | |
| Implementation Path | |
| Implementation Type | <none> |
| KNEE | CBMAX |
| Location X-Coordinate | 560 |
| Location Y-Coordinate | 250 |
| MAX_TEMP | CTMAX |
| Name | INS366 |
| Part Reference | C1 |
| PCB Footprint | RAD/CK05 |
| Power Pins Visible | |
| Primitive | DEFAULT |
| PSpiceTemplate | C^@REFDES %1 %2 ?TOL |
| Reference | C1 |
| SLOPE | CSMAX |
| Source Library | D:\CADENCE\SPB_16. |
| Source Package | C |
| Source Part | C.Normal |
| TC1 | 0 |
| TC2 | 0 |
| TOLERANCE | |

Figure 6. Editing Part Properties

For example, click in the value column of the property *IC* and enter 0 to specify an initial condition of 0V. To display the property and value, click *Display* and then select *Name and Value*, as shown in Figure 7, to display the name of the property and its value.

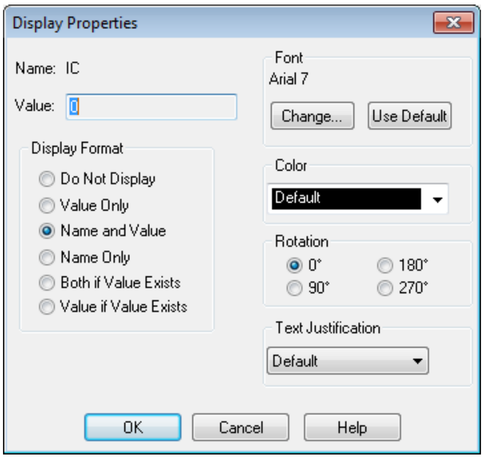



Figure 7. Displaying Property and Value

Click *OK* and then click *Apply*. Close the **Property Editor** window.

| **A Selection of Capture How-To Videos** | |
|---|---|
| **Cadence Help Tutorial** | https://www.youtube.com/watch?v=8hAjE80LE40&index=4&list=PLCgZ9VoSvNf1KC1wnrpO86Cp3M1IeA9LO |
| **Placing Parts** | https://www.youtube.com/watch?v=A5eTfZIlL7I&index=7&list=PLCgZ9VoSvNf1KC1wnrpO86Cp3M1IeA9LO |

**Ready to run a simulation?**

**GOTO: #3 Running a Simulation with PSpice A/D**