ESE380 : OrCAD Capture CIS Lite : Common Issue Summary

ISSUE: Eliminating the UWANTED underlining of edited reference designators

This issue arises when you change/edit a reference designator. For example, suppose the reference designator for a resistor is currently R27. Assume it needs to be R3 (as there are only three resistors on the schematic). Once changed to R3, you will most likely get R3, instead of R3.

So… How does one get rid of the underlining?

Solution #1: A first method of removing these unwanted *underlinings* is to click and select the **part** with the underlined reference designator (and NOT the reference designator is self). Then *right click the part*, and select **User Assigned Reference > Unset**. Click off the part, and the underline should vanish. (*Courtesy of J.N.*)
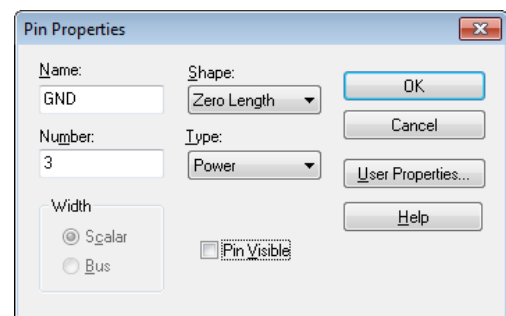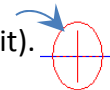
Solution #2: This method enables a removing all unwanted *underlinings* with one operation. Simply, select all, and cut the entire schematic (to clipboard). Next, paste it right back into the schematic page. The unwanted *underlines* should all be gone.

ISSUE: Missing power pin(s)... One or more Gnd/Vcc Pins are NOT Visible

The power pins are sometimes "missing" from certain parts found in the OrCAD/Cadence libraries. For example, certain voltage regulators and voltage references will have a library symbol that does NOT show the third (ground) pin . That is, the part symbol only has the Vin and Vout pins, but not Gnd. Generally, we prefer to use these symbols with all such pins explicitly shown and fully visible.

To make power pins visible, the following approach should work:

1. Right click on the part, and select **Edit Part**
2. Locate the missing/invisible pin (look for an oval, with a line thru it).

3. Click the center of the oval to highlight the invisible pin, right click on it, and select the **Edit Properties** option
4. The **Pin Properties** popup box should appear...
5. Select the **Pin Visible** check box option
6. Next, using the Shape pull-down menu change from **Zero Length** to **Line**
7. Now click the OK button
8. IF… the NOW visible pin is shown, but the *pin name* and *pin number* are still not shown, right-click on the pin, and select the **Reset Pin Name Location** option.

9. The pin should now be fully visible… Move/edit items (e.g., <Value>), as required.

ISSUE: Removing arrow-heads on pins of certain library provided parts:

NOTE: This is a personal preference issue, and is NOT a mandatory change/modification that must made with ESE380 schematics.

The arrow-heads seem to indicate that the pin is an input. They are more commonly found with logic and digital parts. However, the use or inclusion of arrow-heads does NOT seem to be consistently applied. **In addition, the use of arrow heads in input pins is not an industry standard (and are actually seen rarely).**

Solution: Use the following procedure to remove the arrow-heads from each input pin...

1. Select the part, and right-click once... The find and click on **Edit Part** option
2. Next select a pin with an arrow-head, and select **Properties...** The **Pin Properties** box will popup.
3. Using the **Type** pull-down menu, change the type from *Input* to *Passive*
4. Click OK
5. No more arrow-head!

ISSUE: Precisely Locating Textual Items & Grid Related Items :

IMPORTANT NOTE: At times, it may be necessary to TEMPORARILY go *Off-Grid*… to properly and precisely locate and place reference designators, part names/numbers, alias names, and so on. IF YOU DO THIS, once the textual items are located, *immediately* GO BACK ON-GRID!

NEVER, stay off-grid! NEVER "center" a drawing or place parts OFF GRID. Connecting wires is almost impossible to do off grid. An entire drawing may have to be re-done, if you make these mistakes.

_____

To make the grid visible (if not currently):
1. Click **Options** > **Preferences** … And select the **Grid Display** tab
2. Next in the **Visible** box, click the Displayed check box. Click OK.
3. The 0.1" grid should be visible. It is STRONGLY recommended NOT to change the grid spacing, and to leave it at a setting of 0.1".

To TEMPORARILY go *off-grid*:

1. Click **Options** > **Preferences** … And select the **Grid Display** tab
2. Next uncheck both "**Pointer Snap to**…" check boxes.
3. Click OK.

REMEMBER: Only go off-grid, AS NEEDED, to make as "fine" position changes when necessary. **Once the text items are located, go right back on-grid asap.**

To TEMPORARILY go back *on-grid*:

4. Click **Options** > **Preferences** … And select the **Grid Display** tab
5. Next, check both "**Pointer Snap to**…" check boxes.
6. Click OK.

===
===
===
===

Student Proposed "*Issue-Solution*" Addition:

Should you find an *Issue-Solution* item that you think should be added to this document, please forward the description to Scott Tierno via email at scott.tierno@stonybrook.edu

Thanks!