Schematic Pin Customization

Contents

- Access
- Customizing Position
- Customizing Font

Altium

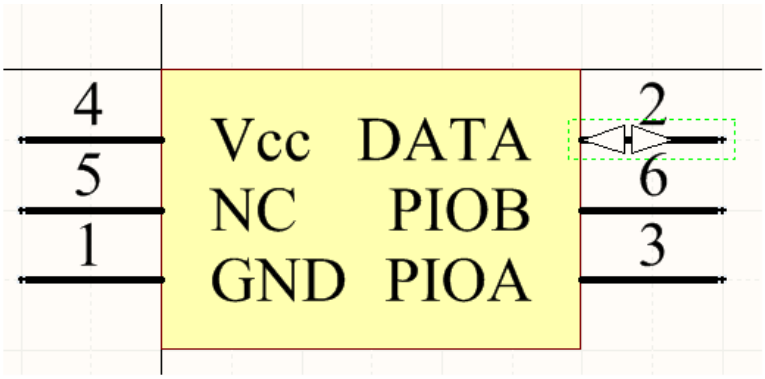
Related Videos

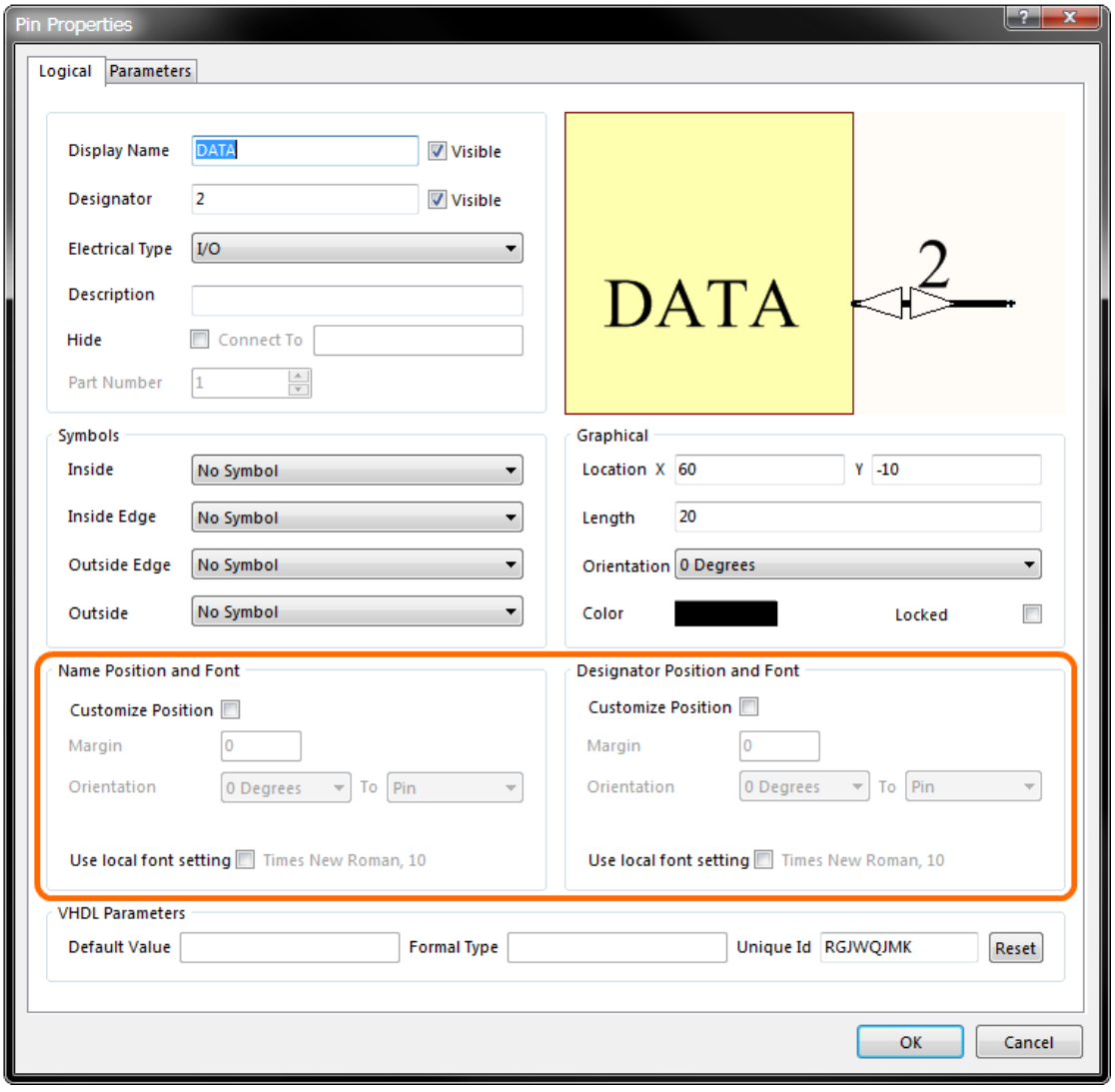
Schematic Enhancements

When it comes to the design of library components, the most important aspect is the electrical connectivity – provided courtesy of optimally placed pins. And with respect to pins, it is important to have control over the position of the pin's Name and Designator, in relation to the pin itself, as well as the font used. This can aide no-end in enhancing the readability of components once placed into designs. To support this, Altium Designer provides the ability to either run with default settings for these attributes, or to customize them individually, overriding at the per-pin level where needed.

Access

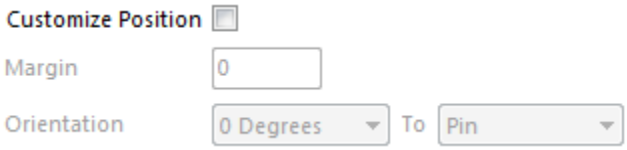
Controls for customization of the position and font for a pin's Name and Designator can be found on the **Logical** tab of the *Pin Properties* dialog, when editing a pin in either the Schematic Library or Schematic Editors. While the controls themselves are the same for both attributes, separate sets of controls allow them to be customized independently of each other.





Accessing controls to customize the position and font of the pin Name and Designator.

Customizing Position



Position-related controls.

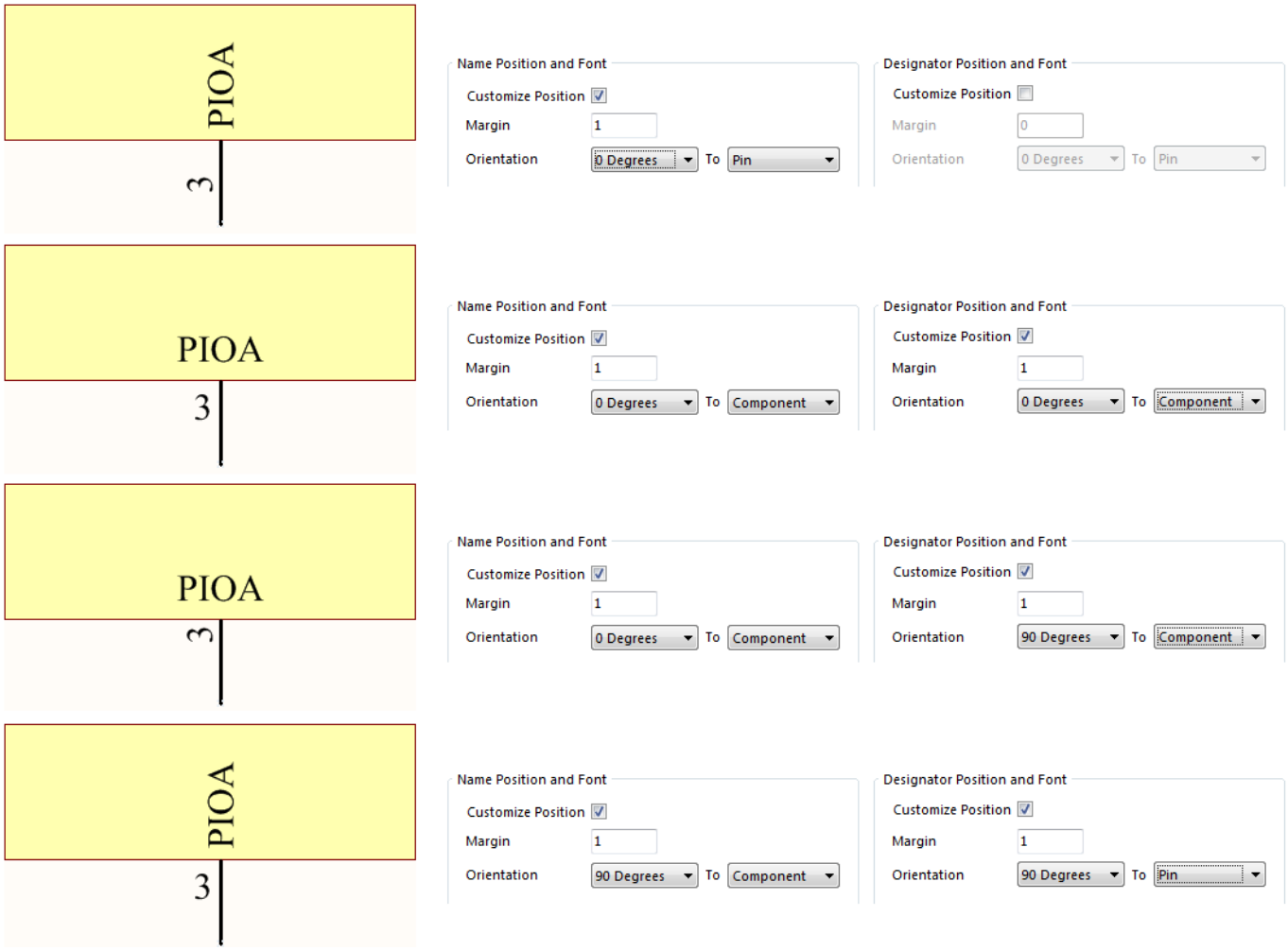
Use the **Customize Position** option to change from following the default settings for position, to an overriding, customized position.

With this option disabled (default), the positions of the pin Name and Designator follow the default settings for **Margin** and **Orientation**:

- *Pin Name*:
  - **Margin** = `5`
  - **Orientation** = `0 Degrees` To `Pin`.
- *Pin Designator*:
  - **Margin** = `8`
  - **Orientation** = `0 Degrees` To `Pin`.

> Values for the **Margin** are taken from the corresponding entries on the **Schematic – General** page of the *Preferences* dialog.

Once you enable the **Customize Position** option, you have access to define **Margin** and **Orientation** as required. For the **Margin**, simply enter a new value directly in the associated field. For the **Orientation**, use the drop downs to choose the angle (`0 Degrees` or `90 Degrees`) and the reference (`Pin` or `Component`). The preview window will display the result of a change, so you don't need to leave the dialog until you have things just the way you want them.



Example customized positions for pin Name and Designator.

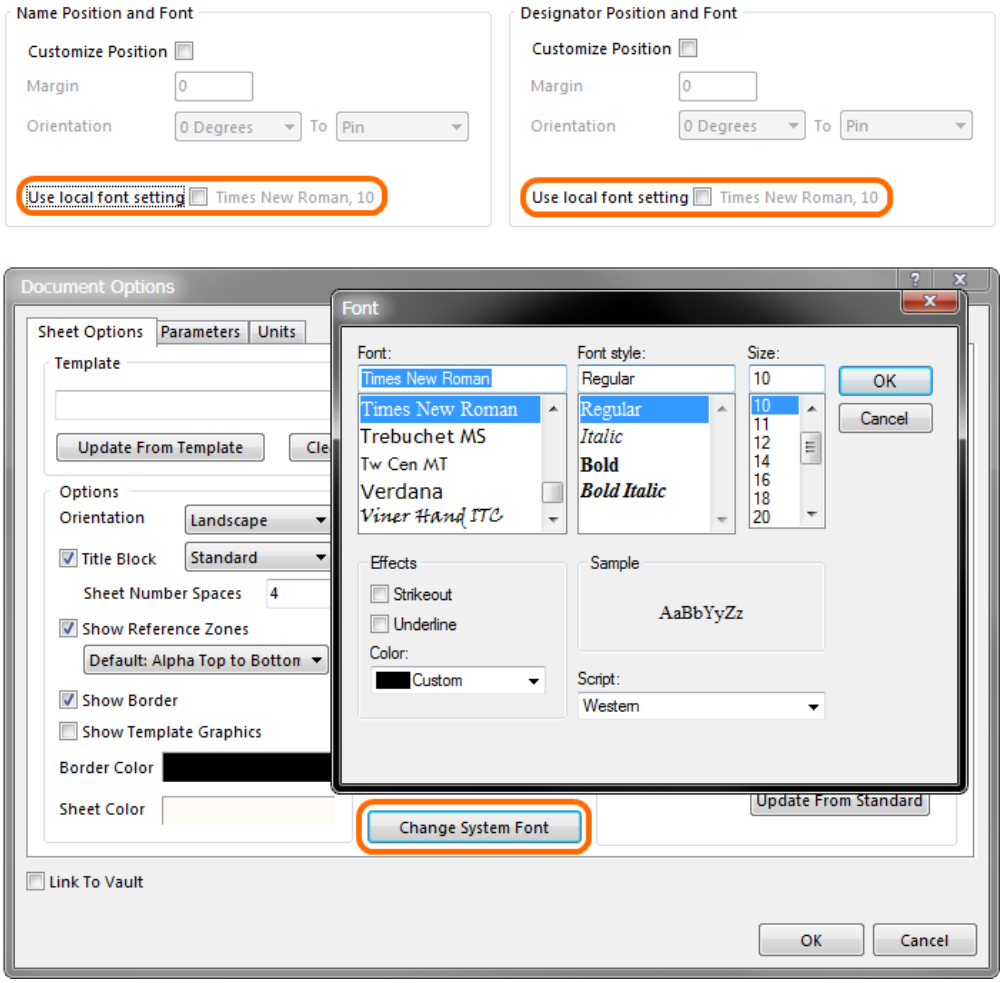
Customizing Font

Font-related controls.

Use the **Use local font setting** option to change from following the default font, to an overriding, customized font.

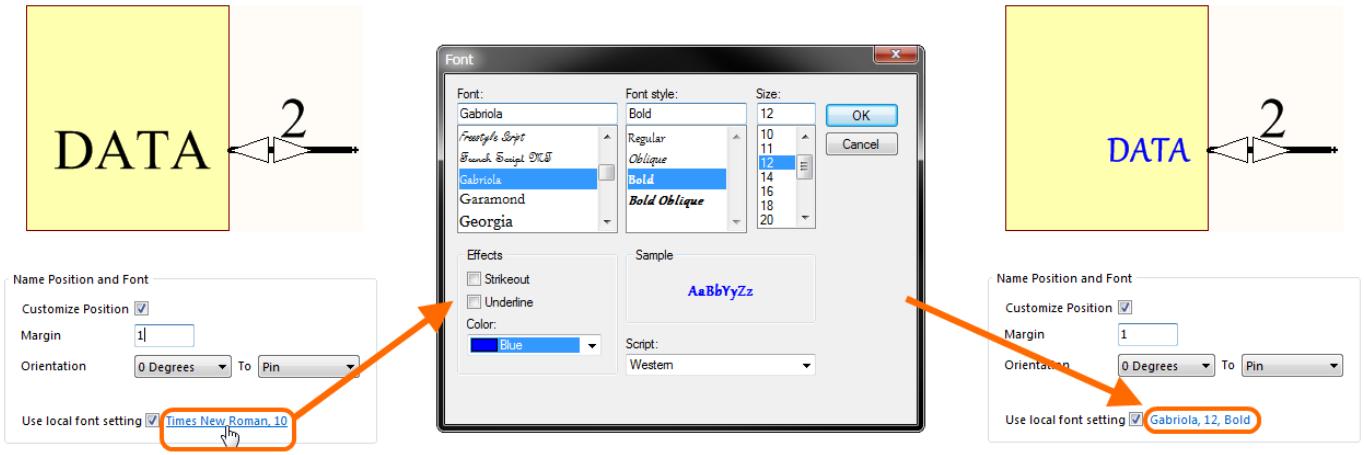
With this option disabled (default), the font used for the pin Name and Designator will be:

- *Schematic Library Document* – `Times New Roman, 10pt, Regular`. This is fixed. There is no ability to change the default font for a library document.
- *Schematic Document* – the font defined at the schematic document level – `Times New Roman, 10pt, Regular` being the default. This document-level font applies when a library component is placed onto a schematic sheet, and can be changed using the **Change System Font** button in the *Document Options* dialo g (**Design » Document Options**).



By default, pin Names and Designators will use the document-level font, when components are placed on the schematic sheet.

By enabling the **Use local font setting** for the Name and/or Designator, you can override the default font, replacing it with your own font customization. To do so, simply click on the font control to the right of the option to access the standard *Font* dialog. The control doubles as a notification for the font currently chosen, or 'in-force'.



Example customization of the font for a pin's Name.