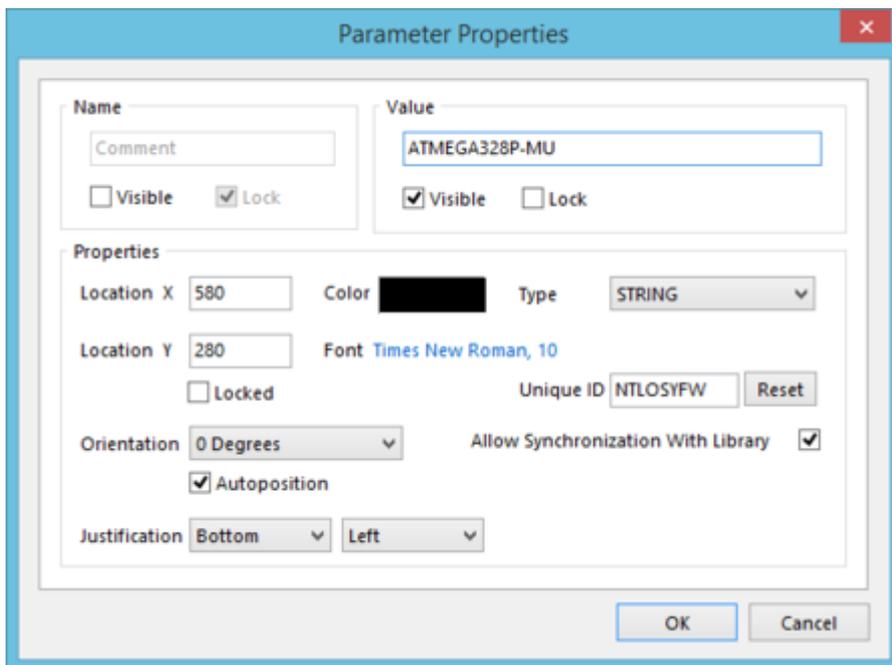# Parameter Properties

Modified by Jason Howie on Aug 7, 2018

Other Related Resources
[Parameter (Object)](#)

Parent page: [Sch Dialogs](#)

The *Parameter Properties* dialog.

Summary

This dialog allows the designer to specify the properties of a Parameter object. Parameters can be added in a variety of places to provide additional information. For example, parameters at the local document level can be used as the source for special string parameteric data to be added to schematic sheets - most notably as part of the title block in a schematic template. Parameters can also be added to specific design objects, such as components, sheet symbols, ports, and pins. These can be both standard parameters, giving additional useful information local to the object, or rule-based parameters. The latter enable design constraints to be specified on the schematic side, which will translate into PCB design rules with appropriate scope, once the design is passed to the PCB document.

Access

The *Parameter Properties* dialog can be accessed from the following places:

- In the Schematic Editor, clicking the **Add**, **Edit**, or **Add as Rule** button, on the **Parameters** tab

of the *Document Options* dialog (**Project | Content |** **Document Options**).

- In the Schematic Editor, clicking the **Add**, **Edit**, or **Add as Rule** button, on the **Parameters** tab of the *Sheet Symbol* dialog.
- In the Schematic Editor, clicking the **Add**, **Edit**, or **Add as Rule** button, on the **Parameters** tab of the *Port Properties* dialog.
- In the Schematic Editor, clicking the **Add**, **Edit**, or **Add as Rule** button, in the **Parameters** region of the *Properties for Schematic Component* dialog.
- In the Schematic Library Editor, clicking the **Add**, **Edit**, or **Add as Rule** button, in the **Parameters** region of the *Library Component Properties* dialog.
- In the Schematic Library Editor, clicking the **Add**, **Edit**, or **Add as Rule** button, on the **Parameters** tab of the *Pin Properties* dialog.

Options/Controls

Name

- **Name** - use this field to give a standard (non-rule) parameter a meaningful name, perhaps reflective of its purpose.
- **Visible** - enable this option to have the parameter's name displayed on the schematic.
- **Lock** - enable this option to prevent editing of the parameter's name, either through a dialog, or the *Inspector* panel.

> Visibility only applies to port, pin, and component parameters, and not to document-level parameters. For the latter, the **Visible** option will be unavailable.

> None of these options are editable for a parameter added as a rule - the **Name** field is set permanently to `Rule`, the **Visible** option is permanently disabled, and the **Lock** option is permanently enabled.

Value

- **Value** - use this field to give a standard (non-rule) parameter its value. For a rule-type parameter, this entry will initially be `Undefined`. The value can only be set/modified by defining the constraints of the chosen rule type. This is done by clicking the **Edit Rule Values** button.
- **Visible** - enable this option to have the parameter's value displayed on the schematic.
- **Lock** - enable this option to prevent editing of the parameter's value, either graphically in the workspace, through a dialog, or through the *Inspector* panel.
- **Edit Rule Values** - this button is only available for a parameter that has been added as a rule. If the value for the parameter is currently `Undefined`, clicking this button provides access to the *Choose Design Rule Type* dialog. Choose the required rule, and use the subsequent *Edit PCB Rule (From Schematic)* dialog to define its constraints. Once the rule is defined, subsequently clicking this button will access the latter dialog directly.

> Visibility only applies to port, pin, and component parameters, and not to document-level parameters. For the latter, the **Visible** option will be unavailable.

⚠️ On the sheet, the **Value** alone can be visible, or both the **Name** and the **Value** (appearing in the format `<Name>:<Value>`), but not the **Name** by itself.

Properties

> The following lists all possible properties for the dialog. Note that when accessing the dialog by clicking the **Edit** button on the **Parameters** tab of the *Document Options* dialog, many of these properties will not be present.

- **Location X/Y** - the current X (horizontal) and Y (vertical) coordinates for the bottom-left corner of the parameter's bounding rectangle. Edit these values to change the position of the parameter in the horizontal and/or vertical planes respectively.
- **Color** - click the color sample to change the color used for the parameter (both for its name and value), using the standard *Choose Color* dialog.
- **Type** - the type of parameter, which determines the valid entries that can be used for its value. Available types are: STRING, BOOLEAN, INTEGER, and FLOAT. For a rule-type parameter, this entry is always STRING.
- **Font** - this control serves two purposes. Firstly, it reflects the currently chosen font – applied to the text for the parameter's name and value - in terms of *Font Name*, *Font Size* and *Font Style*. Secondly, when clicked it provides access to the standard *Font* dialog, from where to change the font as required.

> Effects are also displayed when enabled (Strikeout, Underline). If Regular is used for the font's style, this will not be displayed visually in the control's string.

- **Locked** - enable this option to protect the parameter from being edited graphically.

> An object that has its **Locked** property enabled cannot be selected or graphically edited. Double click on the locked object directly and disable the **Locked** property, to graphically edit the object.

- **Unique ID** - the current unique identifier for the parameter. The Unique ID (UID) is a system generated value that uniquely identifies this current parameter. A new UID value can be entered directly into this field.
  - **Reset** - click this button to have the system generate a new UID for the parameter.

> ✅ When adding a parameter as a rule to an object, a unique ID is given to it. The same ID is given to the corresponding design rule that is created in the PCB. With this Unique ID, the constraints of the rule can be edited on either the schematic or PCB side and the changes pushed through upon synchronization.

- **Orientation** - specify the orientation of the parameter, counter-clockwise in relation to the horizontal. Options available are: 0 degrees, 90 degrees, 180 degrees, 270 degrees.
- **Autoposition** - this option is meaningful only for a component parameter. Enable it to have visible parameters positioned automatically every time the associated component is rotated. Disable this option to take manual control over parameter placement. The parameter will have a dot appear at its bottom-left corner to distinguish it as being a manual parameter.
- **Justification** - use the two buttons associated to this field to specify how the parameter is to be vertically (left button) and horizontally (right button) justified. This will depend on its current **Orientation** and **Location X/Y**.
  - **Vertical Justification** - the following justification options are available, using a visible parameter with 0 degrees orientation as an example:

- **Top** - the top edge of the parameter's bounding rectangle is against the *Y = Location Y* line.
- **Center** - the parameter's bounding rectangle is centered across the *Y = Location Y* line.
- **Bottom** - the bottom edge of the parameter's bounding rectangle is against the *Y = Location Y* line.
  - **Horizontal Justification** - the following justification options are available, using a visible parameter with `0 degrees` orientation as an example:
    - **Left** - the left edge of the parameter's bounding rectangle is against the *X = Location X* line.
    - **Center** - the parameter's bounding rectangle is centered across the *X = Location X* line.
    - **Right** - the right edge of the parameter's bounding rectangle is against the *X = Location X* line.
- **Allow Synchronization With Library** - this option is meaningful only for a component parameter. Enable it to maintain synchronicity with the parameter in the corresponding component in the relevant source library. If there is a change to the parameter in the source, an update of the schematics from that source will bring the change into the parameter of the placed component instance.