

## Pin Properties

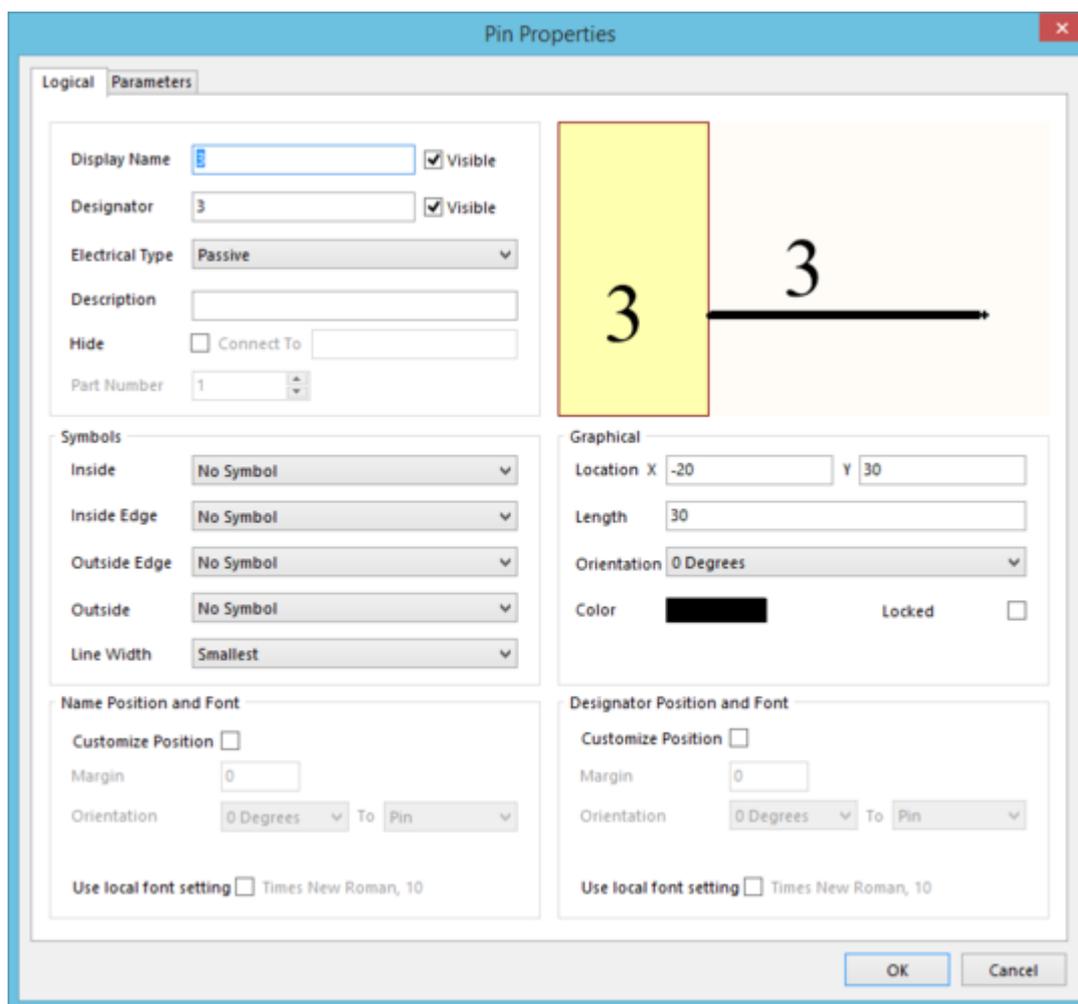
Modified by Admin on Oct 24, 2017

### Other Related Resources

[Pin \(Object\)](#)

[Component Pin Editor \(Dialog\)](#)

Parent page: [Sch Dialogs](#)



The *Pin Properties* dialog.

## Summary

This dialog allows the designer to specify the properties of a Pin object. A pin is an electrical design primitive. Pins give a component (part) its electrical properties and define the connection points on the part for the incoming and outgoing signals.

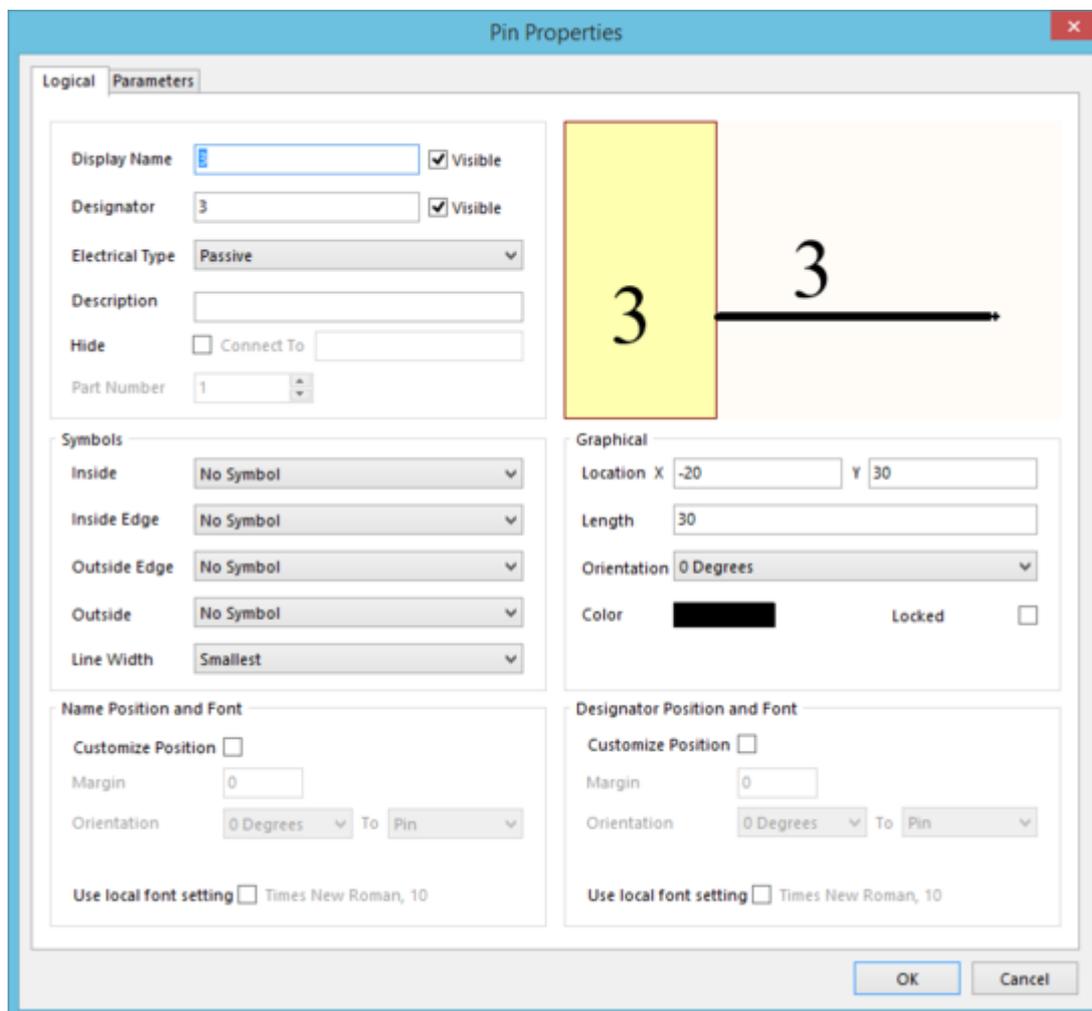
## Access

The *Pin Properties* dialog can be accessed during placement by pressing the **Tab** key.

After placement, the dialog can be accessed in one of the following ways:

- Double-clicking on the placed pin object.
- Placing the cursor over the pin object, right-clicking and choosing **Properties** from the context menu.

## Logical Tab



*Pin Properties* dialog - **Logical** tab.

Use the dialog's **Logical** tab to modify electrical and graphical properties of the pin object.

## Options/Controls

- **Display Name** - use this field to specify an optional display name for the pin. By default, a newly placed pin will be named using the designator value. Supplying a display name is particularly useful for IC-type components, where the provision of a meaningful name enables a designer to quickly see what the pin is being used for. Note that while the pin name is optional, it is required when the pin is going to be hidden. A hidden pin is automatically connected to other hidden pins with the same name, and to nets with the same name, when a net-list is created.
  - **Visible** - use this option to determine whether the Display Name for the pin is displayed (enabled) or hidden (disabled) when the parent part is placed on a schematic sheet.
- **Designator** - the numerical identifier of the pin. Each pin in a part must have a unique designator.
  - **Visible** - use this option to determine whether the Designator for the pin is displayed (enabled) or hidden (disabled) when the parent part is placed on a schematic sheet.
- **Electrical Type** - use this field to set the electrical type of the pin. This type is used when compiling a project or analyzing a schematic document to detect electrical connection errors (using the Electrical Rules Check feature). Available types are: Input, I/O, Output, Open Collector, Passive, HiZ, Open Emitter, and Power.
- **Description** - use this field to provide an optional description for the pin, perhaps a concise summary of its purpose.
- **Hide** - enable this option to hide the pin. This is typically the case for power pins of multi-part components, where their display would otherwise cause unnecessary clutter on the schematic sheet.
  - **Connect To** - use this field to specify the net to which the hidden pin is to be explicitly connected. This is typically a power net, such as VCC or GND.



Hidden pins for a component can be revealed on the sheet in the Schematic Editor or Schematic Library Editor, by enabling the **Show All Pins On Sheet (Even if Hidden)** option, in the associated *Properties for Schematic Component* dialog, or *Library Component Properties* dialog, respectively. Alternatively, in the Schematic Library Editor, the display of hidden pins can be toggled by clicking **Tools | Options |**  **Show Hidden Pins**, from the main menus.

- **Part Number** - this field is available when the pin is being added to a multi-part component. Use the associated up/down arrows of the field to specify the part to which the pin is to be associated. A multi-part component also includes a non-graphical part, *Part Zero*. Part Zero is used for pins that are to be included in all parts of the multi-part component, for example power pins.

For a multi-part component, the power net connections should ideally be assigned through use of Part Zero. For each pin that is required to connect to a power net in this way, simply enable the **Hide** option, leave the **Connect To** field blank, and set the **Part Number** field to .

- **Preview Window** - this area of the dialog provides instant visual feedback as you change various options, enabling you to adjust the look and feel of the pin to meet design

requirements.

## Symbols

Use this area of the dialog to add additional symbols to the pin. Such symbols can be used to visually enhance the component by showing, in a purely graphical way, the electrical characteristic of the pin.

- **Inside** - use this field to optionally add a symbol to the pin, on the inside of the component graphic. Choose from: No Symbol, Postponed Output, Open Collector, HiZ, High Current, Pulse, Schmitt, Open Collector Pull Up, Open Emitter, Open Emitter Pull Up, Shift Left, and Open Output.
- **Inside Edge** - use this field to optionally add a symbol to the pin, on the inside edge of the component graphic. Choose from: No Symbol and Clock.
- **Outside Edge** - use this field to optionally add a symbol to the pin, on the outside edge of the component graphic. Choose from: No Symbol, Dot, Active Low Input, and Active Low Output.
- **Outside** - use this field to optionally add a symbol to the pin, on the outside of the component graphic. Choose from: No Symbol, Right Left Signal Flow, Analog Signal In, Not Logic Connection, Digital Signal In, Left Right Signal Flow, and Bidirectional Signal Flow.

These symbols are purely graphical. The true electrical property of the pin is determined by the entry set for the pin's **Electrical Type**.

- **Line Width** - use this field to determine the width of the line used to draw the symbols. Choose from either Small or Smallest.

## Graphical

- **Location X/Y** - the current X (horizontal) and Y (vertical) coordinates for the non-electrical end of the pin (the end that is placed against the component symbol's outline). Edit these values to change the position of the pin in the horizontal and/or vertical planes respectively.
- **Length** - use this field to specify the length of the pin, in accordance with the currently defined units of measurement.
- **Orientation** - specify the orientation of the pin, counter-clockwise in relation to the horizontal. Options available are: 0 degrees, 90 degrees, 180 degrees, 270 degrees.
- **Color** - click the color sample to change the color of the pin, using the standard *Choose Color* dialog.

The **Color** option affects all graphical aspects of the pin - its line, its **Display Name**, its **Designator**, and any additional pin symbols (**Inside**, **Inside Edge**, **Outside Edge**, **Outside**). The **Display Name** and **Designator** colors can, however, be changed independently, using the respective **Use local font setting** control.

- **Locked** - enable this option to protect the pin 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.

## Name Position and Font

- **Customize Position** - enable this option to change from following the default settings for position of the pin's Display Name, to an overriding, customized position.
  - **Margin** - use this field to enter the required margin (space) between the pin's Display Name text and the edge of the component symbol outline.
  - **Orientation** - use this field to choose the orientation of the pin's Display Name text (0 Degrees or 90 Degrees).
    - **To** - use this field to set the reference point for orientation of the Display Name text. Choose from either Pin or Component.

With the **Customize Position** option disabled (default), the position of the pin's Display Name follows the default settings for **Margin** (5) and **Orientation** (0 Degrees **To Pin**). The value for the **Margin** is taken from the corresponding entry on the **Schematic - General** page of the *Preferences* dialog.

- **Use local font setting** - enable this option to change from following the default font, to an overriding, customized font, using the font control to the right. This control serves two purposes. Firstly, it reflects the currently chosen font - applied to the pin's Display Name text - 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.



By default, and for a component placed on a schematic sheet, the font used for the pin's Display Name text follows the document-level font, set on the **Sheet Options** tab of the *Document Options* dialog (**Project | Content | [Document Options](#)**).

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.

## Designator Position and Font

- **Customize Position** - enable this option to change from following the default settings for position of the pin's Designator, to an overriding, customized position.
  - **Margin** - use this field to enter the required margin (space) between the pin's Designator text and the edge of the component symbol outline.
  - **Orientation** - use this field to choose the orientation of the pin's Designator text (0 Degrees or 90 Degrees).
    - **To** - use this field to set the reference point for orientation of the Designator text. Choose from either Pin or Component.

With the **Customize Position** option disabled (default), the position of the pin's Designator follows the default settings for **Margin** (8) and **Orientation** (0 Degrees **To Pin**). The value for the **Margin** is taken from the corresponding entry on the **Schematic - General** page of the *Preferences* dialog.

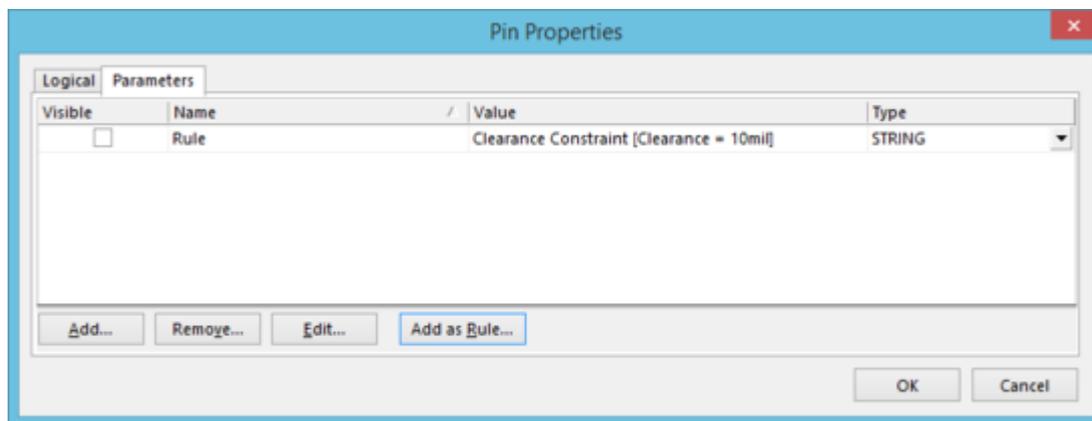
- **Use local font setting** - enable this option to change from following the default font, to an overriding, customized font, using the font control to the right. This control serves two purposes. Firstly, it reflects the currently chosen font - applied to the pin's Designator text - in terms of *Font 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.



By default, and for a component placed on a schematic sheet, the font used for the pin's Designator text follows the document-level font, set on the **Sheet Options** tab of the *Document Options* dialog (**Project | Content | Document Options**).

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.

## Parameters Tab



*Pin Properties* dialog - **Parameters** tab.

Use the dialog's **Parameters** tab to manage parameters attached to the currently selected pin object. You can also add rule-based parameters.

Adding a parameter (as a rule) to a component pin on the schematic, results in a PCB design rule being generated - when the design is transferred to the PCB document - with a scope that targets the corresponding Pad for that component.

## Options/Controls

- **Parameters Grid** - the main region of the tab lists all of the parameters currently defined for the pin, in terms of:
  - **Visible** - use this option to determine the visibility of the parameter's value in the workspace. Note that this does not relate to the visibility of the parameter's **Name**, which can be determined, for a standard (non-rule) parameter only, in the *Parameter Properties* dialog.
  - **Name** - the name of the parameter. For a rule-type parameter, this entry will be locked as Rule.
  - **Value** - the value of the parameter. For a rule-type parameter, the entry will reflect the rule type, along with a listing of its defined constraints.

- **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.

A standard parameter (non-rule) can be modified with respect to any of these attributes directly in the grid. However, attempting to change a locked **Name** and/or **Value** attribute will raise an error, and you will need to press **Esc** to abandon such changes.

A parameter added as a rule can not be edited directly in the grid with respect to its **Name, Value, or Type**. Its **Name** and **Type** are set to Rule and STRING respectively, and are always uneditable. Its **Value** can only be edited by changing the constraints of the rule. To do this, select and edit the parameter, and click the **Edit Rules Button** in the *Parameter Properties* dialog - this will give access to the *Edit PCB Rule (From Schematic)* dialog, from where the changes to the constraints can be made.

- **Add** - click this button to add a new parameter to the list. The *Parameter Properties* dialog will appear. Use this to define the parameter, especially its Name, Value, Type, and whether or not it's value is to be visible in the workspace.
- **Remove** - click this button to delete the selected parameter(s) from the list of parameters.
- **Edit** - click this button to modify the currently selected parameter. The *Parameter Properties* dialog will appear, with which to do so.
- **Add as Rule** - click this button to add a new design rule directive parameter to the list. The *Parameter Properties* dialog will appear, but this time will contain the **Edit Rule Values** button, which in turn gives access to the *Choose Design Rule Type* dialog, from where you can choose, and subsequently define, the constraints of the required rule type.

## Right-Click Menu

The Parameters Grid right-click menu offers the following commands:

- **All On** - use this command to quickly enable the **Visible** option for all parameters in the list.
- **All Off** - use this command to quickly disable the **Visible** option for all parameters in the list.
- **Selected On** - use this command to quickly enable the **Visible** option for all currently selected parameters in the list.
- **Selected Off** - use this command to quickly disable the **Visible** option for all currently selected parameters in the list.
- **Add** - use this command to add a new standard (non-rule) parameter to the list.
- **Remove** - use this command to remove the currently selected parameter(s) in the list.
- **Edit** - use this command to edit the currently selected parameter in the list.
- **Select All** - use this command to quickly select all parameters in the list.
- **Select None** - use this command to quickly deselect all parameters in the list.

---

**Source URL:**

[http://documentation.solidworkspcb.com/display/SWPCB/Sch\\_Dlg-PinPropertiesDialog\(\(Pin+Properties\)\)\\_PW](http://documentation.solidworkspcb.com/display/SWPCB/Sch_Dlg-PinPropertiesDialog((Pin+Properties))_PW)