Schematic Editor object properties are definable options that specify the visual style, content and behavior of the placed object. The property settings for each type of object are defined in two different ways:

- **Pre-placement settings** - most Pin object properties, or those that can logically be pre-defined, are available as editable default settings on the Schematic - Defaults page of the Preferences dialog (access from the button at the top-right of the workspace). Select the object in the Primitive List to reveal its options on the right.

- **Post-placement settings** - all Pin object properties are available for editing in the Properties panel when a placed Pin is selected in the workspace.
default settings in the
In the below properties listing, options that are not available as default settings in the Preferences dialog are noted as "Properties panel only".

### General Tab

#### Location *(Properties panel only)*

- **(X/Y)**
  - X (first field) - the current X (horizontal) coordinate of the reference point of the object, relative to the current workspace origin. Edit to change the X position of the object. The value can be entered in either metric or imperial; include the units when entering a value whose units are not the current default.
  - Y (second field) - The current Y (vertical) coordinate of the reference point of the object, relative to the current origin. Edit to change the Y position of the object. The value can be entered in either metric or imperial; include the units when entering a value whose units are not the current default.
- **Rotation** - use the drop-down to select the rotation. Choices are: 0 Degrees, 90 Degrees, 180 Degrees, and 270 Degrees.

#### Properties

- **Designator** - the numerical identifier of the pin. Each pin in a part must have a unique designator. Use or to determine whether the Designator for the pin is displayed or hidden (respectively) when the parent part is placed on a schematic sheet.
- **Name** - use 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 a meaningful name enables you to see quickly how the pin is being used. Note that while the pin name is optional, it is required when the pin is 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. Use or to determine whether the Name for the pin is displayed or hidden (respectively) when the parent part is placed on a schematic sheet.
- **Electrical Type** - use the drop-down to set the electrical type of the pin. This 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** - enter a meaningful description of the pin, if desired.
- **Pin Package Length** - enter the pin-package length. The unit will automatically be entered after you press Enter.
- **Pin Length** - use to specify the length of the pin in accordance with the currently defined units of measurement. Click the color box to edit the color of the pin.

#### Symbols

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

- **Inside** - use 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 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 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 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.

- **Line Width** - use this field to determine the width of the line used to draw the symbols. Choose from either Small or Smallest. This provides support for meeting GOST standards, which stipulates that these symbols should be of the same width as the line used to draw the component's symbol.

The **Line Width** setting will also apply to the automatic symbol used in relation to the pin's defined **Electrical Type**.

**Font Settings**

- **Designator**
  - **Custom Settings** - enable to access the **Font Settings** below to customize the font.
  - **Font Settings** - use the controls to configure the font, font size, color, and special settings such as bold and underlining.
  - **Custom Position** - enable to access the controls below to customize the position.
  - **Margin** - enter the desired margin.
  - **Orientation** - use the drop-down to select the orientation.
  - **To** - use the drop-down to select the desired object of the designator.

- **Name**
  - **Custom Settings** - enable to access the **Font Settings** below to customize the font.
  - **Font Settings** - use the controls to configure the font, font size, color, and special settings such as bold and underlining.
  - **Custom Position** - enable to access the controls below to customize the position.
  - **Margin** - enter the desired margin.
  - **Orientation** - use the drop-down to select the orientation.
  - **To** - use the drop-down to select the desired object of the name.

**Parameters Tab**
Parameters

Use this region to manage parameters attached to the currently selected pin object.

- **Grid** - this region lists all of the parameters currently defined for the pin. Use the icon to lock/unlock the associated parameter.
  - **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.
- **Font** - click to open a menu to select the desired font, font size, color, and attributes to bold, italicize, etc., if desired.
- **Other** - click to open a drop-down to change additional options:
  - **Show Parameter Name** - enable to show the parameter name.
  - **Allow Synchronization with Database** - enable to synchronize with the database.
  - **X/Y** - enter the X and Y coordinates.
  - **Rotation** - use the drop-down to select the rotation. Choices are: 0°, 90°, 180°, and 270°.
  - **Autoposition** - check to enable auto-positioning.
- **Add** - click to add a parameter. Use to delete the currently selected parameter.

Source URL: https://www.altium.com/documentation/display/ADES/Sch_Prop-Pin((Pin+Properties))_AD