# Schematic Symbol Guidelines

1. Making a New Schematic Symbol

1.1. Check the Master Libraries (ACTIVES.olb, ELECTRO-MECH.olb, PASSIVES.olb) to see if a symbol already exists for your part. If not, you'll have to make your own.

1.2. Create your symbol in a local library on your computer.

1.3. Make sure symbol meets all of the guidelines in this document.

1.4. Create an entry in the Symbol_Checklist.accdb form.

1.5. Copy symbol from your library to one of the editable libraries in the Master Library directory (NEW-A.olb, NEW-B.olb, NEW-C.olb).

2. How to Change Properties

2.1. **User Properties** box - Used to delete excess properties, toggle pin name/number visibility, and modify the "Value" property. The letter V under Attributes means the property is displayed.
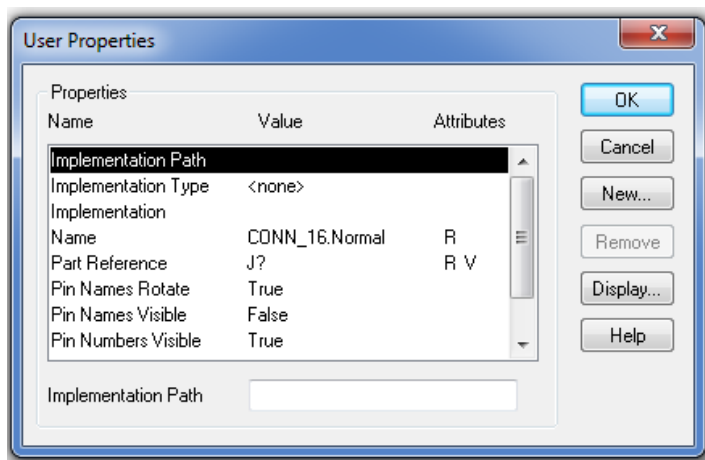
To open, go to Part View (View > Part), then double click anywhere outside the symbol.

2.2. **Pin Properties** box - Used to modify individual pin names, shapes, and types. Names and types can also be modified for all of the pins at once using the Package Properties box (2.4).
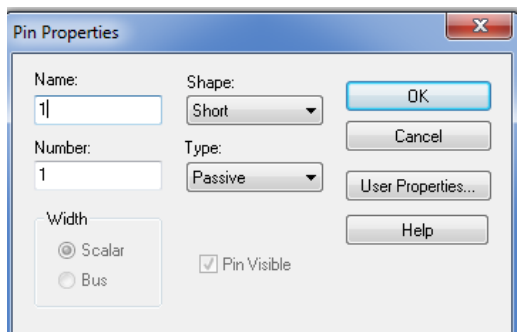
To open, double click on any individual pin.

2.3. **Edit Part Properties** box - Used to modify the name and reference designator for the symbol. Footprints are defined in the CIS database, so the PCB Footprint field should be blank.

To open, go to Package View (View > Package), then double click outside the thick black outline.

2.4. **Package Properties** box - Used to view all of the pin names and properties at once. Pin names and types can be modified here or individually in the Pin Properties box (2.2).

To open, go to Package View (View > Package), then Edit > Properties.

3. Schematic Symbol Naming

Guidelines for naming symbols can be found in "Symbol_Naming.docx", in the following directory:
J:\R&D\Functional_Groups\PCB_Design\Master_Library\Checklists\Symbols\Symbol_Naming.docx

4. Reference Designators

The symbol's reference designator must match the list of established Physio-Control reference designators and can be changed in "Edit Part Properties" (2.3).

Several documents define standard reference designators, including IEEE 315-1975, ASME Y14.44-2008, and IEC 113-2. These standards have been combined and voted on to establish a set of Physio-Control standard reference designators. The full list of Physio designators can be found here:
J:\R&D\Functional_Groups\PCB_Design\Master_Library\Checklists\Symbols\Reference Designators - Physio standards.xls

Physio-Control Approved Reference Designators (abbreviated list):

Battery	BT	Filter	FL	Resistor	R
Capacitor	C	Inductor	L	Shield	SH
Connector, Jack	J	IC	U	Switch	SW
Connector, Plug	P	LED	D	Thermistor	RT
Crystal/Oscillator	Y	Microphone	MIC	Transformer	T
Diode	D	Mounting Hole	MH	Transistor	Q
Ferrite Bead	FB	Relay	K		

5. Symbol Properties

5.1. **Value property** - The Value property should be blank and visible. It is very tempting to put the manufacturer's part number in this property, especially for an IC, but the CIS database will automatically populate it when the part is placed in your schematic.   Value can be cleared in the "User Properties" box (2.1). When it is blank and visible, it will display on the symbol as <Value>.

5.2. **PCB Footprint property** - This property should be blank, since the CIS database defines the footprints for each part. Modify this property in the "Edit Part Properties" window (2.3).

5.3. **Extra Properties** - When symbols are imported from previous schematics or online libraries, they often contain extra properties. These properties should be removed to prevent confusion. View and modify your symbol's properties in the "User Properties" box (2.1). Your symbol should contain only the following properties:

- Implementation Path
- Implementation Type
- Implementation
- Name
- Part Reference
- Pin Names Rotate
- Pin Names Visible
- Pin Numbers Visible
- Reference
- Value

6. Pin Properties

6.1. **Pin names**

6.1.1. Pins must be named and numbered to match the manufacturer's datasheet.

6.1.2. Labeling

6.1.2.1. Preferred: label the pins by making the pin names "visible".

6.1.2.2. Acceptable: If the pin names need to be displayed smaller in order to fit, then the pins can be labeled with plain text instead. This method introduces a small amount of risk because it places one piece of information in two fields. If one of those fields is changed but not the other, an error could result. Therefore, this method should be avoided but is acceptable when needed.

6.1.3. Uniqueness

Two pins cannot have the same name; add number suffixes to differentiate (NC1, NC2, etc).

6.1.4. Connectors

Connector symbols can often be reused by many connectors, so it is important that their pins are named generically. Connector pins should not be named to match the specific signals they are connecting. Symbol numbers should match the connector datasheet (1,2,3….or A1, B1, A2).

6.2. **Pin types**

6.2.1. Pin types (input, power, passive, etc.) should be assigned according to the manufacturer's datasheet.

6.2.2. Pin type can be changed individually in the Pin Properties box (2.2) or for all pins in the Package Properties box (2.4).

6.2.3. **Special pin functions**

6.2.3.1. **No connect** pins - pin name should contain NC

6.2.3.2. **Active low** - indicated by shape and/or name
- Shape: "dot" (draws a circle on the pin)
- Name: one of the following suffixes to the pin name: * or \

6.2.3.3. **Clock** - indicated by shape
- Shape: "clock" (draws a caret on the pin)

6.2.3.4. Pin shape can be set for individual pins in the Pin Properties box (2.2)

7. Symbol Drawing

7.1. **General**

7.1.1. The body and all accompanying lines should be drawn with thin lines.

7.1.2. Part Reference and Value should be displayed in the symbol drawing.

7.2. **Text**

7.2.1. The part reference and value must use the default font and size.

7.2.2. (For ICs) Part Reference should be placed above or in the middle of the body.

7.2.3. (For ICs) Value should be placed below or in the middle of the body.

7.2.4. Other text acceptable, and may be whatever size is appropriate for its placement.

7.3. **Grid**

7.3.1. Grid should be set to 0.1" x 0.1", as shown in the picture below. To see the grid settings, go to Options > Design Template.
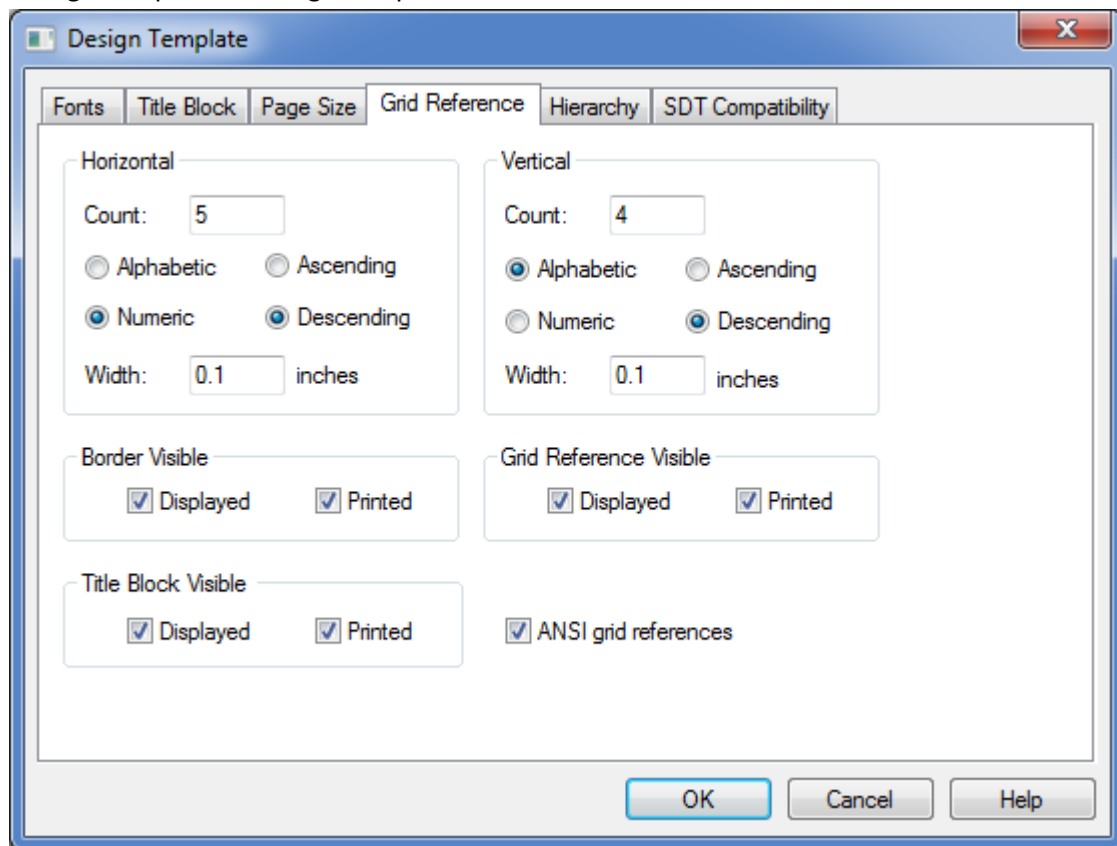
7.4. **Pins**

7.4.1. Pins must be on grid.

7.4.1.1. Preferred: pins spaced two gridlines apart.

7.4.1.2. Acceptable: pins spaced one gridline apart.

7.4.2. Pins should all connect to the body of the part if possible.

7.4.2.1. "No connect" pins may be either attached to the body of the part as regular pins, or hidden (pin shape= zero length). No floating regular-size pins (third symbol below).

GOOD   GOOD   UNACCEPTABLE