

Properties Glossary

Properties are divided into system properties (those which provide information that is critical to the part being handled correctly by the layout program) and user properties (properties for user reference). System property names are in all caps, and user property names are mixed case.

These are the only symbol properties that are editable in **PADS Maker Schematic**. All other properties attached to a symbol are searchable, but cannot be edited within the software or displayed on the schematic.

System Properties

ALT_PKG_LST

ALT_PKG_LST defines alternate footprints attributed to a symbol. Up to 16 alternate footprints can be assigned to ALT_PKG_LIST using the [Foot print Browser](#). This property is optional and may be added by following the instructions under [Additional Properties](#).

DEVICE

This property defines which physical component the symbol represents. If you do a Save As in **Symbol Editor**, it updates the DEVICE ID. The DEVICE property is interpreted by the PCB interfaces and utilities. In the **PADS Maker Schematic** to **Layout** flow, this property is an automatically generated GUID when a save-as is performed to a symbol, ensuring that each physical component type is unique (if the part has been imported from PartQuest, the DEVICE property will be equivalent to the Manufacturer Part Number). A user defined value is acceptable, if the value of this property is unique and is not shared with any other parts.

Note that the DEVICE property value must be identical for the same part with multiple schematic symbols—for example, where the **HETERO** property is required. In this instance the user must override the automatically generated value. See [Fractured Symbols](#) for specific details.

Note: The DEVICE property cannot exceed 40 characters and cannot contain the following characters: ~ * ' \ / @ " space

HEIGHT

The HEIGHT property describes the component's physical height. This property is optional and may be added by following the instructions under [Additional Properties](#).

HETERO

The HETERO property defines a heterogeneous package containing two or more symbols. It allows you to bundle two or more different symbols into one package. You must assign the HETERO property to all symbols you want to bundle into a single package.

The HETERO property value is a list of two or more symbol names (without file extensions) separated by commas. You can enclose each symbol with parentheses. The list itself provides the necessary symbol binding information. It is the presence or absence of the parentheses that determines the view that is "seen" by the layout system. The parentheses indicate that the view for the enclosed symbol is schematic only. You can change this property in **Symbol Editor**.

The HETERO device must use the SIGNAL property for power pins.

For examples of how the HETERO property is used, see [Fractured Symbols](#).

NC

The NC (No Connect) property identifies any unconnected physical package pins that are not graphically displayed on the symbol. To define unused pins with the property NC, place the NC property with no value on any unused pin. The property value is auto-populated with a list of these pin numbers. Change this property in the **Symbol Editor**. This property is optional and may be added by following the instructions under [Additional Properties](#).

NETNAME

NETNAME defines the signal to which this property is attached as global in scope. All signals with the same value are connected together throughout the design. The NETNAME property is only used with power and ground symbols. This property is optional and may be added by following the instructions under [Additional Properties](#).

PARTS

The PARTS property specifies the number of occurrences or slots this symbol has within the physical component. You need more than one symbol to create a part, and PARTS can only be changed in the **Symbol Editor**. This property is interpreted by the schematic editor, the PCB

interfaces, and the PCB utilities. You must have more than one symbol to create a part, and this property is managed in **Symbol Editor**.

The format of this property is PARTS=*n*.

If you specify a PARTS value greater than one, ensure that each # property attached to symbol pins has a value for each of the parts specified. The number of parts must match the number of pins listed for each # property on the symbol. If no PARTS property value is specified, the system defaults to 1.

For more information on PARTS, see [gate swap](#) .

Example

PARTS=4 specifies four functions or slots for a SN74LS00 because there are four independent 2-input NAND gates in this package. Ensure that each # property contains four values, such as #=1,4,9,12.

PINSWAP

The PINSWAP property identifies which pins can be swapped for other pins in a target CAD system. When part types are automatically generated, the PINSWAP property allows automatic pin swapping. Generating part types with missing PINSWAP data could inhibit optimal layout. This property can be changed in the **Symbol Editor**.

The PINSWAP property format allows you to group pins. Use parentheses to indicate swappable groups, and brackets to indicate groups that are not swappable.

Example

The property **PINSWAP=([A,B],[C,D])** indicates that pins A and B are not swappable and C and D also are not swappable, but the group A,B is swappable for the group C,D.

PINTYPE

The Pin Type property is used by **PADS Maker Schematic's** design rule check utility (DRC) when checking that all connections to the pin comply with its designation. This property is changed in the **Symbol Editor**.

PADS Maker Schematic has the following set of pre-defined pin types:

IN	Input
OUT	Output
BI	Bidirectional
ANALOG	Analog
OCL	Open Collector
OEM	Open Emitter
POWER	Power
GROUND	Ground
TERMINAL	Terminal Connector

Example

A pin with Pin Type **IN** must be connected in the schematic in such a way that it does not violate the conditions imposed by an input pin.

PKG_TYPE

The PKG_TYPE property specifies the footprint that represents this component in a board drawing or on any type of PCB CAD layout system. The PCB interfaces and utilities interpret this property. PKG_TYPE can be assigned using the [Footprint Browser](#). Note that the PKG_TYPE property value cannot contain a comma (,).

Example

If you specify a **PKG_TYPE=14PDIP** property on a 74LS00 symbol, the 14PDIP package symbol is represented on the board drawing in place of the 74LS00 component.

REFDES

The Ref Designator property is the physical reference for a device. This property is used to indicate to the packaging utility which symbols are to be put in the same component.

The REFDES property value has a unique prefix followed by a number, assigned or unassigned, as shown below.

Example

Assigned: **REFDES=U4**

Unassigned: **REFDES=U?**

An assigned value identifies the package with a unique number. An unassigned value uses a question mark (?) after the prefix to leave the assignment open to the value determined by the REFDES program.

SIGNAL

The SIGNAL property defines a signal, such as power or ground, connecting to a component's supply pin. If you use the SIGNAL property, the symbol must not contain a symbol pin that represents a supply pin. You can only define this in **Symbol Editor**, and may be overridden in Schematic.

The format of this property is `SIGNAL=<signal name>;<pin_number>,<pin_number>, ...`

Example

The property **SIGNAL=VDD;8,14** specifies that pins 8 and 14 of this component are attached to VDD.

Note that the SIGNAL property is best used for implicit power and ground setups. Explicit power and ground setups can be defined by a placed pin with the same pin number. This property is optional and may be added by following the instructions under [Additional Properties](#).

User Properties

Datasheet

Use the Datasheet property to link to a component's datasheet.

Description

You can enter a description of a component using this property.

Drawing Author

This is a border property that places the name of the design's author on the border of the schematic. This property is almost exclusively used with border symbols, but may be used with others. This property is optional and may be added by following the instructions under [Additional Properties](#).

Drawing Number

This is a border property that places the design number on the border of the schematic. This property is almost exclusively used with border symbols, but may be used with others. This property is optional and may be added by following the instructions under [Additional Properties](#).

Drawing Title

This is a border property that places the title of the design on the border of the schematic. This property is almost exclusively used with border symbols, but may be used with others. This property is optional and may be added by following the instructions under [Additional Properties](#).

Drawing Revision

This is a border property that places the revision number of the design on the border of the schematic. This property is almost exclusively used with border symbols, but may be used with others. This property is optional and may be added by following the instructions under [Additional Properties](#).

Manufacturer

This property contains the name of the part manufacturer.

ManufacturerPN

This property lists the manufacturer part number.

Part List Exclude

Part List Exclude lists all components that should not be included in the part list for the BOM. This property can be found in the drop-down box in **PADS Maker Schematic**. This property is optional and may be added by following the instructions under [Additional Properties](#).

PartNumber

This property contains the customer part number. This property is optional and may be added by following the instructions under [Additional Properties](#).

Part_ Color

This property specifies a part's color, such as LEDs. This property is optional, but it's supported by **PADS Maker**. This property is optional and may be added by following the instructions under [Additional Properties](#).

Photo

A link to a photo of a component can be included using this property.

RatedPower

RatedPower specifies the amount of power that a component can dissipate. This property is optional and may be added by following the instructions under [Additional Properties](#).

RatedCurrent

This property specifies the amount of current a part can consume. This property is optional and may be added by following the instructions under [Additional Properties](#).

RatedVoltage

RatedVoltage specifies voltage across a DC analog component. This property is optional and may be added by following the instructions under [Additional Properties](#).

Supplier

This property contains the name of the part supplier.

SupplierPN

This property contains the supplier part number,

Tolerance

This property contains the percent guaranteed on the value of the part. This property is optional and may be added by following the instructions under [Additional Properties](#).

Value

This property contains the inductance, resistance, or capacitance value. This property is optional and may be added by following the instructions under [Additional Properties](#).

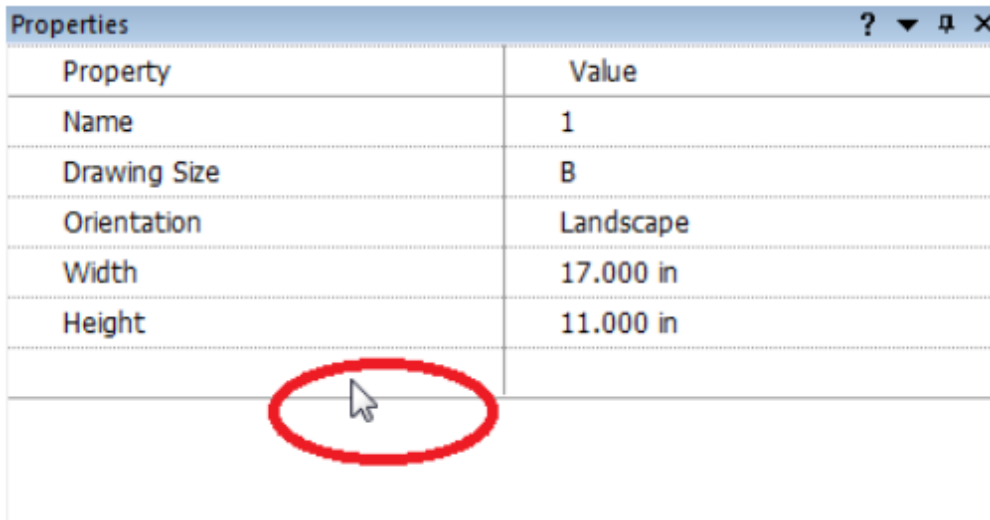
Xref Annotation

Xref annotation is reserved for future use. This property is optional and may be added by following the instructions under [Additional Properties](#).

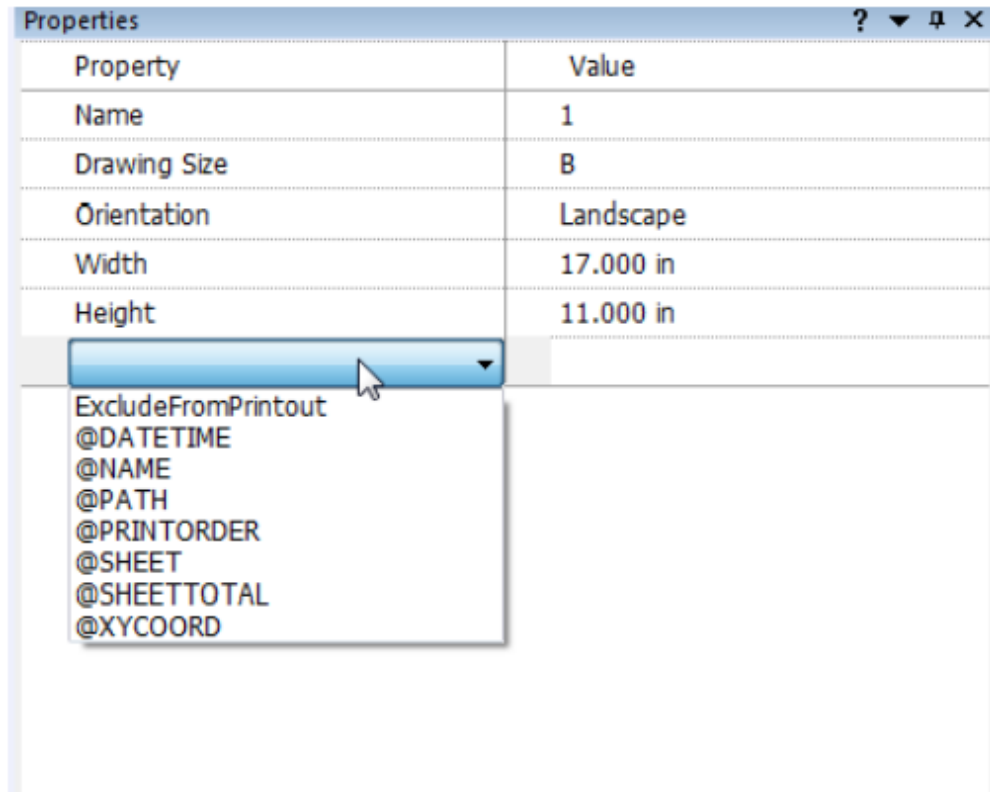
Additional Properties

Many optional properties are available to you besides those named in the **Properties** window in **PADS Maker Schematic**. To access these additional properties, go to the bottom of the Properties window and click on the last empty cell of the table. Clicking on that box should open up a drop down box from which to choose a variety of additional, optional properties. To change the properties of a particular part, click on its symbol in your schematic to view the part's properties, then go to the bottom of that Properties window list and follow the directions below.

Properties on the **Sheet** border in the **Schematic**.



Property	Value
Name	1
Drawing Size	B
Orientation	Landscape
Width	17.000 in
Height	11.000 in



Property	Value
Name	1
Drawing Size	B
Orientation	Landscape
Width	17.000 in
Height	11.000 in

- ExcludeFromPrintout
- @DATETIME
- @NAME
- @PATH
- @PRINTORDER
- @SHEET
- @SHEETTOTAL
- @XYCOORD

Click on any of the properties from the drop-down box. Once selected, the property will appear in your **Properties** table.

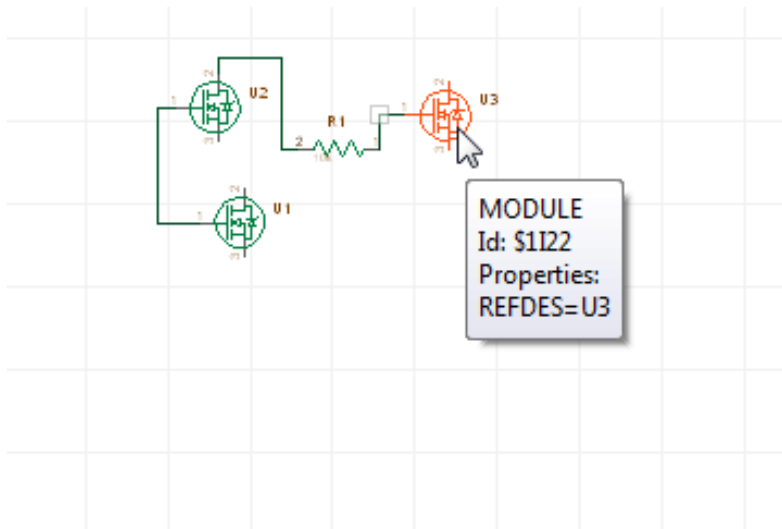
Properties	
Property	Value
Name	1
Drawing Size	B
Orientation	Landscape
Width	17.000 in
Height	11.000 in
<div style="border: 1px solid black; padding: 5px;"> ExcludeFromPrintout @DATETIME @NAME @PATH @PRINTORDER @SHEET @SHEETTOTAL @XYCOORD </div>	

Properties	
Property	Value
Name	1
Drawing Size	B
Orientation	Landscape
Width	17.000 in
Height	11.000 in
<input type="checkbox"/> @DATETIME	<input checked="" type="checkbox"/> 12/19/16

Part Properties

Part Properties may be edited in the schematic or in the **Symbol Editor**.

To change the property of a part in the schematic, select the part by double-clicking on the symbol. The **Properties Window** is displayed. Properties changed in the schematic exist only in that schematic. If you wish to save the property change to the part for future use in other schematics, change the part in **Symbol Editor** and save it to the **Library** where this part is stored.



In the **Properties** window, scroll down through the list of properties to the property you wish to edit and make the desired updates. To add an additional property, scroll down to the bottom of the Properties window list, and follow the same directions as above.

Property	Value
<input type="checkbox"/> Current - Continuous Drain (Id) @ 25degC	4A (1C)
<input type="checkbox"/> DEVICE	DIGIKEY_497_3177_6_ND
<input type="checkbox"/> Datasheet	http://www.st.com/web/en/resource/techn
<input type="checkbox"/> Description	MOSFET N-CH 60V 4A SOT223
<input type="checkbox"/> Drain to Source Voltage (Vdss)	60V
<input type="checkbox"/> FET Feature	Standard
<input type="checkbox"/> FET Type	MOSFET N-Channel, Metal Oxide
<input type="checkbox"/> Gate Charge (Qg) @ Vgs	9nC @ 5V
<input type="checkbox"/> Input Capacitance (Ciss) @ Vds	340pF @ 25V
<input type="checkbox"/> LeadFree	Lead Free
<input type="checkbox"/> Manufacturer	STMicroelectronics
<input type="checkbox"/> ManufacturerPN	STN3NF06L
<input type="checkbox"/> Mounting Type	Surface Mount
<input type="checkbox"/> NonStock	false
<input type="checkbox"/> Operating Temperature	-55degC ~ 150degC (TJ)
<input type="checkbox"/> PKG_TYPE	SOT-223
<input type="checkbox"/> Package / Case	TO-261-4, TO-261AA
<input type="checkbox"/> Packaging	Digi-Reel(R)
<input type="checkbox"/> PartID	DIGIKEY_497_3177_6_ND
<input type="checkbox"/> Photo	http://media.digikey.com/Renders/Diodes%2
<input type="checkbox"/> Power - Max	3.3W
<input checked="" type="checkbox"/> REFDES	U3
<input type="checkbox"/> Rds On (Max) @ Id, Vgs	100 mOhm @ 1.5A, 10V
<input type="checkbox"/> RohsCompliant	RoHS Compliant
<input type="checkbox"/> Supplier	digikey
<input type="checkbox"/> Supplier Device Package	SOT-223
<input type="checkbox"/> SupplierCategory	Transistors - FETs, MOSFETs - Single
<input type="checkbox"/> SupplierPN	497-3177-6-ND
<input type="checkbox"/> URL	http://www.digikey.com/product-detail/en/S
<input type="checkbox"/> UnitPrice	0.63
<input type="checkbox"/> Vgs(th) (Max) @ Id	2.8V @ 250uA

Property	Value
<input type="checkbox"/> ALT_PKG_LST	
<input type="checkbox"/> Drawing Author	
<input type="checkbox"/> Drawing Number	
<input type="checkbox"/> Drawing Revision	
<input type="checkbox"/> Drawing Title	
<input type="checkbox"/> HEIGHT	
<input type="checkbox"/> NC	
<input type="checkbox"/> NETNAME	
<input type="checkbox"/> Part List Exclude	
<input type="checkbox"/> PartNumber	
<input type="checkbox"/> Part_Color	
<input type="checkbox"/> RatedCurrent	
<input type="checkbox"/> RatedPower	
<input type="checkbox"/> RatedVoltage	
<input type="checkbox"/> SIGNAL	
<input type="checkbox"/> Tolerance	
<input type="checkbox"/> Value	
<input type="checkbox"/> Xref Annotation	
<input type="checkbox"/> @DATETIME	
<input type="checkbox"/> @NAME	
<input type="checkbox"/> @PATH	
<input type="checkbox"/> @PRINTORDER	
<input type="checkbox"/> @SHEET	
<input type="checkbox"/> @SHEETTOTAL	
<input type="checkbox"/> @XYCOORD	

NO FOOTPRINT

The check boxes next to the names of the **Properties** and their values are toggles to display/not display the property name and property value on the part.

You may add as many additional properties as you like from the drop-down box.

To change properties in the **Symbol Editor**, see the section on [Creating a New Part in Symbol Editor](#).

Searching PADS Maker Edition

Contents	
1	System Properties
1.1	ALT_PKG_LST
1.2	DEVICE
1.3	HEIGHT
1.4	HETERO
1.5	NC
1.6	NETNAME
1.7	PARTS
1.8	PINSWAP
1.9	PINTYPE
1.10	PKG_TYPE
1.11	REFDES
1.12	SIGNAL
2	User Properties
2.1	Datasheet
2.2	Description
2.3	Drawing Author
2.4	Drawing Number
2.5	Drawing Title
2.6	Drawing Revision
2.7	Manufacturer
2.8	ManufacturerPN
2.9	Part List Exclude
2.10	PartNumber
2.11	Part_Color
2.12	Photo
2.13	RatedPower
2.14	RatedCurrent
2.15	RatedVoltage
2.16	Supplier
2.17	SupplierPN
2.18	Tolerance
2.19	Value
2.20	Xref Annotation
3	Additional Properties
4	Part Properties