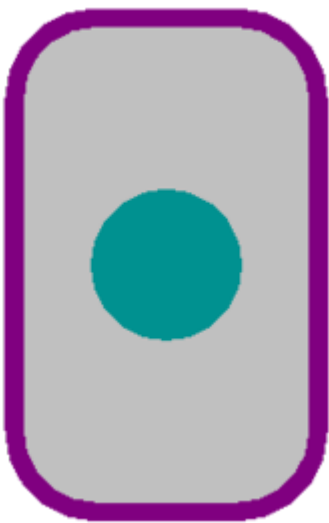


## Pad

Modified by Jason Howie on Sep 11, 2014

Parent page: [Objects](#)



Pads are used to provide both mechanical mounting and electrical connections to the component pins.

## Summary

A pad is a primitive design object. Pads are used for fixing the component to the board, and to create the interconnection points from the component pins to the routing on the board. Pads can exist on a single layer, for example as a Surface Mount Device pad, or they can be a 3 dimensional through-hole pad, having a barrel-shaped body in the Z-plane (vertical), with a flat area on each (horizontal) copper layer. The barrel-shaped body of the pad is formed when the board is drilled and through-plated during fabrication. In the X and Y planes, pads can have a circular, rectangular, octagonal or rounded rectangular shape. Pads can be used individually as free pads in a design, or more typically, they are used in the PCB Library editor, where they are incorporated with other primitives into component footprints.

## Availability

Pads are available for placement in both the PCB editor and the PCB Library editor:

### PCB Editor

- Click **Home** | **Place** |  Pad .

## PCB Library Editor

- Click **Home** | **Place** |  **Pad** .

## Placement

After launching the command, the cursor will change to a crosshair and you will enter pad placement mode.

1. Position the cursor and click or press **Enter** to place a pad.
2. Press the **Tab** key to access an associated properties dialog, from where properties for the region can be changed on-the-fly.
3. Continue placing further pads, or right-click or press **Esc** to exit placement mode.

A pad will adopt a net name if it is placed over an object that is already connected to a net.

Note that the attributes can be modified during placement, press **Tab** to bring up the *Pad* dialog. Changes made will affect the object being placed, and subsequent objects placed during the same editing session.

## Graphical Editing

Pads cannot have their properties modified graphically, other than their location.

- To move a free pad and also move the connected tracks, click, hold and move the pad. The connected routing will remain attached to the pad as it is moved.
- To move a free pad without moving the connected tracks, select the **Tools** | **Arrange** | **Move Object** command, then, click, hold and move the pad.



If you click and drag a selection rectangle around *component* pads, they will not select as they are actually child objects of the component. To sub-select just the pads, hold **Ctrl** as you click and drag the selection window.

If attempting to graphically modify an object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit. Double click on the locked object directly and disable the **Locked** property to graphically edit the object.

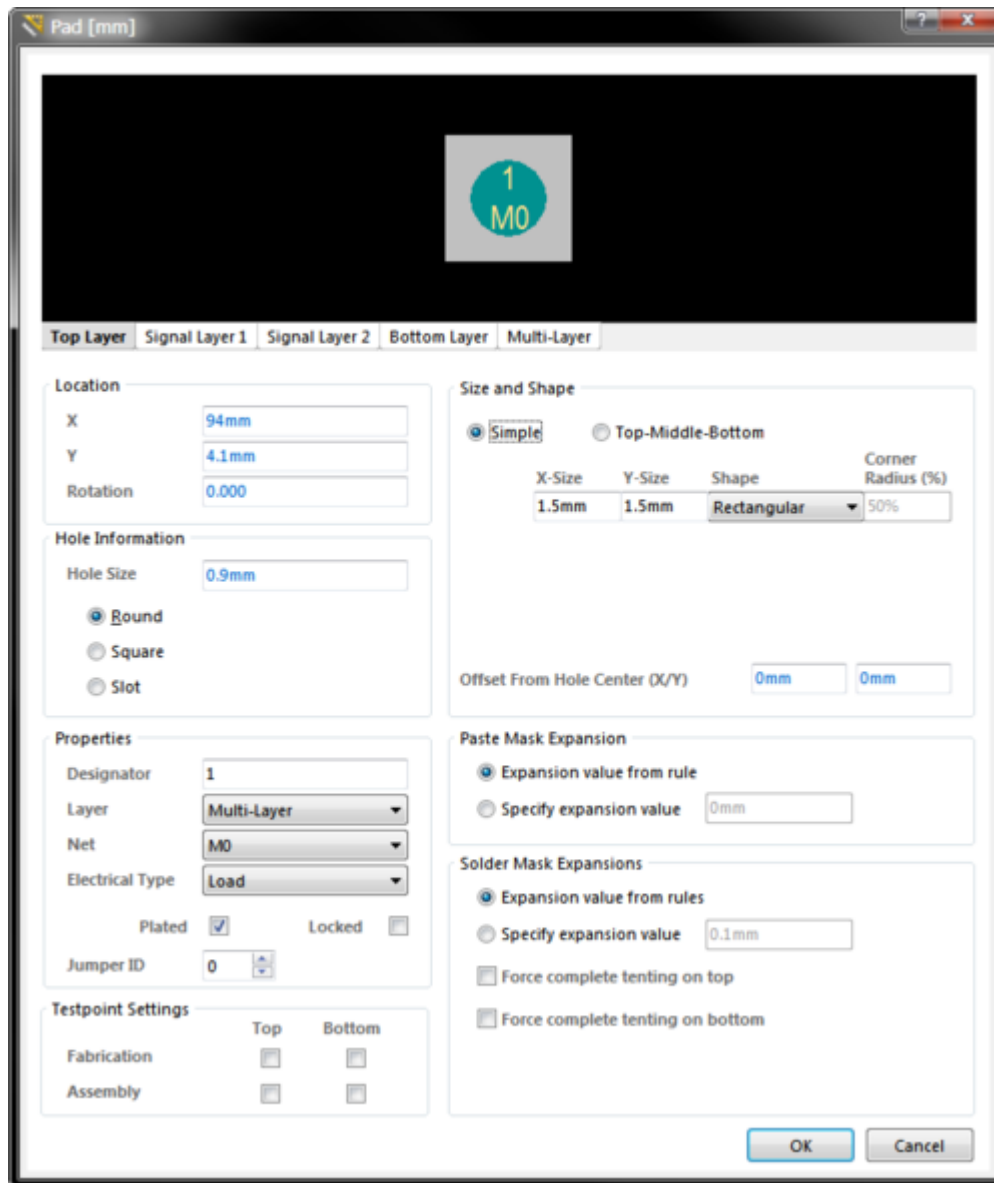
## Non-Graphical Editing

The following methods of non-graphical editing are available:

## Editing via an Associated Properties Dialog

Dialog page: [Pad](#)

This method of editing uses the [Pad dialog](#) to modify the properties of a pad object.



Edit the properties of the Pad in the dialog.

During placement, the dialog can be accessed by pressing the **Tab** key.

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

- Double-click on the placed object.
- Place the cursor over the object, right-click and choose **Properties** from the context menu.

## Setting the Hole Size

The hole can be round, square (punched), or a circular-ended slot. It can also be plated or unplated. Separate drill file (NC Drill Excellon format 2) are generated for each hole kind (**Round**, **Square** and **Slot**) as well as for plated and non-plated holes (as defined by the **Plated** checkbox). That means there can be up to six different drill files generated.



Use the **Hole Size Editor** (a mode of the *PCB* panel) to examine and rationalize the hole sizes used in a completed PCB design.

## Incrementing the Pad Designator

Each pad should be labeled with a designator, which is usually a component pin number. The designator can be up to 20 alphanumeric characters in length. Pad designators will auto-increment by 1 during placement if the initial pad has a designator ending with a numeric character. Press **Tab** to change the designator of the first pad prior to placement.

Multiple pads can share the same designator, if required.

## Setting the Layer

The layer drop-down is used to select the layer that the pad is present on. When designing a surface mount component footprint, set the **Layer** to Top Layer even if the component is intended to be used on the bottom side of the board. The software will automatically flip the pads' layer when the component is flipped to the bottom side of the board. For a standard, through-hole pad, set the **Layer** to Multi-Layer.

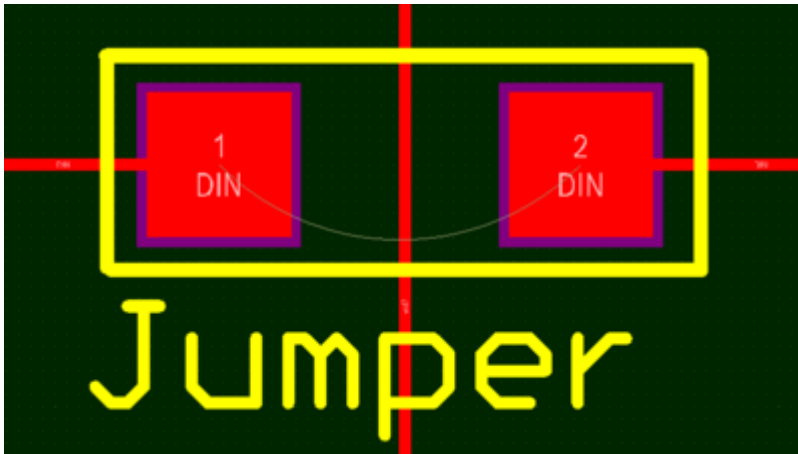
## Connecting to a Net

The **Net** field is used to define which net this pad is to connect to. Pads automatically connect to any internal power planes that are assigned to the same net. The pad will connect to the plane in accordance with the applicable [Power Plane Connect Style](#) design rule. If you do not want a pad to connect to power planes, add another Power Plane Connect Style design rule targeting the required pads, with a connection style of **No Connect**. Pads connect to polygon pours in accordance with the applicable [Polygon Connect Style](#) design rule.

## Setting the Jumper ID

If it is not physically possible (or desirable) to implement all connections as routing, for example on a single-sided PCB, then jumpers can be used. Within a component footprint, label the pads that are to be connected by a jumper with the same, non-zero, **Jumper ID** value. Pads that share the same **Jumper ID and Net** name tell the system that these pins will be connected by a physical jumper during PCB assembly, if that component also has a **Type** of Jumper.

Jumper connections are shown as curved connection lines in the PCB editor. The Design Rules Checker will not report jumper connections as unrouted nets.



Wire-link style jumpers can be defined by setting matching JumperID values in both pads.

## Testpoint Settings

Use the Testpoint Settings to define this pad as a testpoint for **Fabrication** and/or **Assembly** testpoint file generation. A testpoint is a location where a test probe can make contact with the PCB to check for correct function of the board. Any pad or via can be nominated as a testpoint, when this is done the component that the pad or via belongs to, automatically gets locked. Separate Fabrication and Assembly Testpoint files are generated from the **Outputs** tab of the Ribbon.

## Mask Expansions

An opening in the paste and solder mask is automatically created by the software, the same shape as the pad. This opening can be larger (a positive expansion value) or smaller (a negative expansion value) than the pad itself, as defined by the Mask Expansion settings. Typically paste mask openings are smaller than the pad, while solder mask openings are larger than the pad, but there are exceptions to this. The default is for the pad to use the Expansion value from rules (the [Paste Mask Expansion](#) rule and the [Solder Mask Expansion](#) rule), this can be overridden and local values defined directly in the *Pad* dialog, if required.

The term tenting means *to close off*. If a tenting option is enabled then the settings in the applicable [Solder Mask Expansion](#) design rule will be overridden, resulting in no opening in the solder mask on that solder mask layer for this pad. When this option is enabled, the **Expansion value from rules** and the **Specify expansion value** options are ignored.

Paste and Solder mask layers are shown in the negative, that is, when you see an object on one of those layers it is actually a hole or opening in that layer.

If your component needs paste or soldermask shapes that are different from the pad shape, then you can place design objects, such as fills, regions, tracks or arcs, directly onto the top solder or paste mask layer in the library editor, to create the required shape.

## Editing via an Inspector Panel

An *Inspector* panel enables the designer to interrogate and edit the properties of one or more design

objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

---

**Source URL:** [http://documentation.circuitmaker.com/display/CMAK/PCB\\_Obj-Pad\(\(Pad\)\)\\_CM](http://documentation.circuitmaker.com/display/CMAK/PCB_Obj-Pad((Pad))_CM)