

Getting started



Select objects



Viewable window



Zoom in



Zoom out



Pan window



Through hole pin

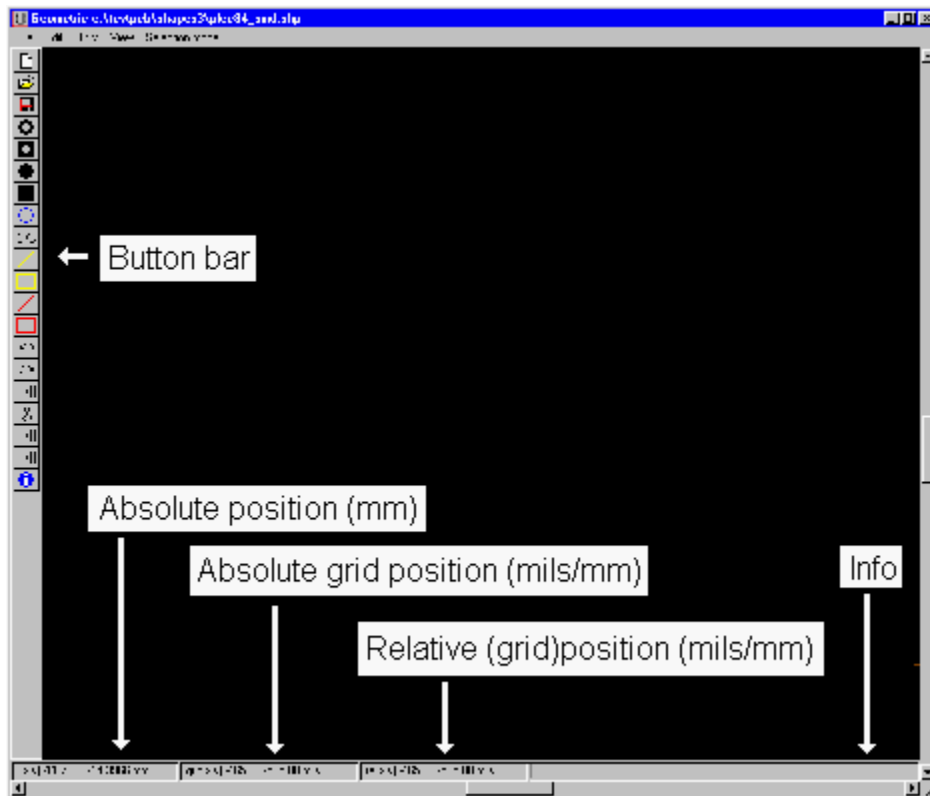


SMD_pad

Geometry editor PCB elegance

The geometry tool of PCB elegance is a tool to design geometries (shapes), to be used in the layout editor.

Viewable window



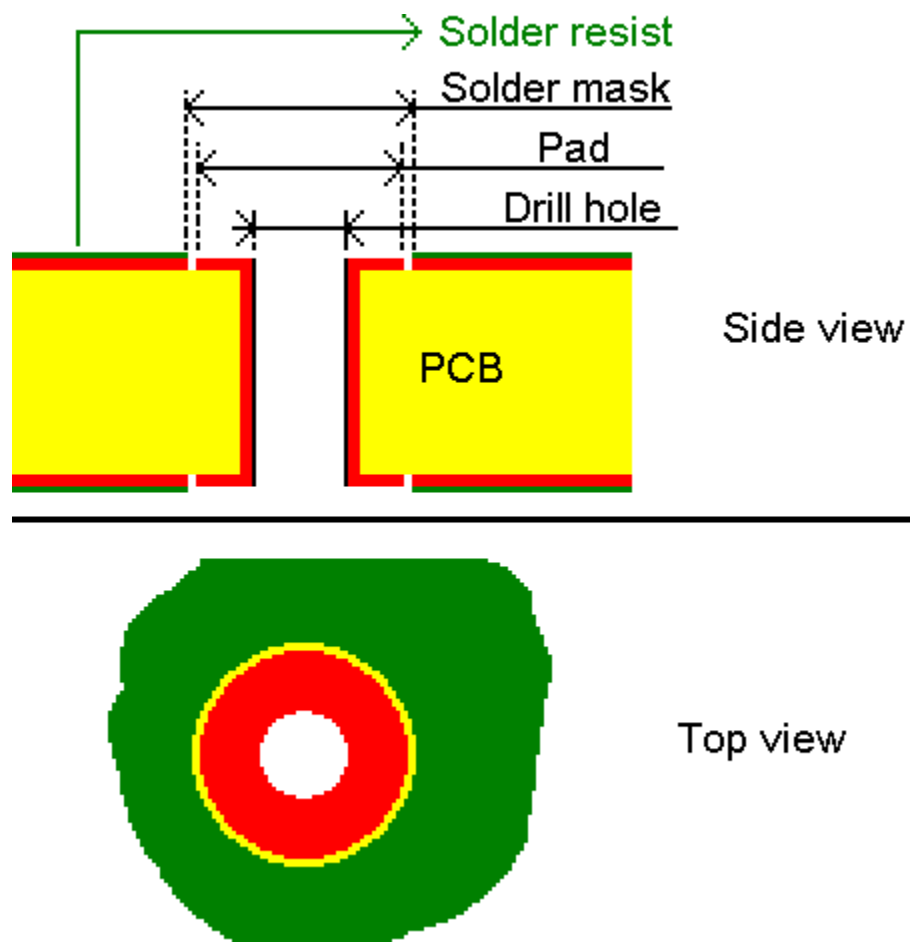
In the viewable window there are four bars visible who have a function. Those four bars are the **Button bar**, **Absolute position (mm)**, **Absolute grid position (mils/mm)**, **Relative (grid)position (mils/mm)** and **Info**.

The **Absolute grid position (mils/mm)** displays the current gridposition of the mouse cursor. The dimension depends on the current units, and is in mils or mm. The **Relative (grid)position (mils/mm)**



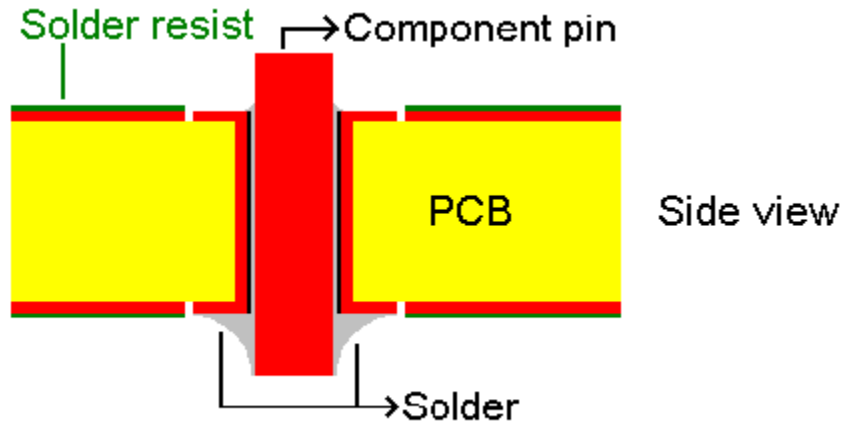
Zero relative cursor

Through hole pin



In the above figure a through hole pin is shown.

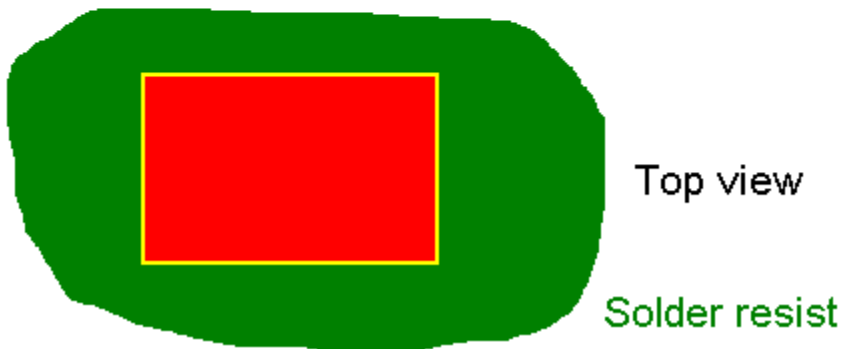
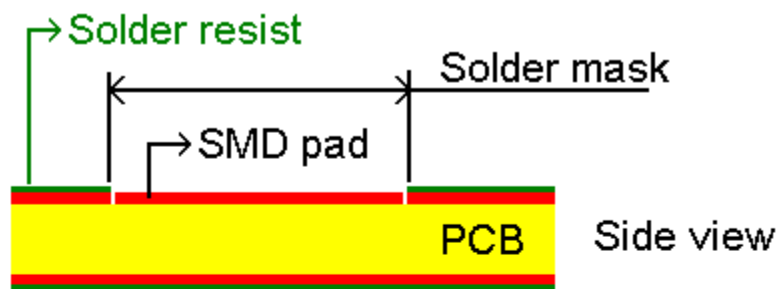
After soldering this through hole pin will look like



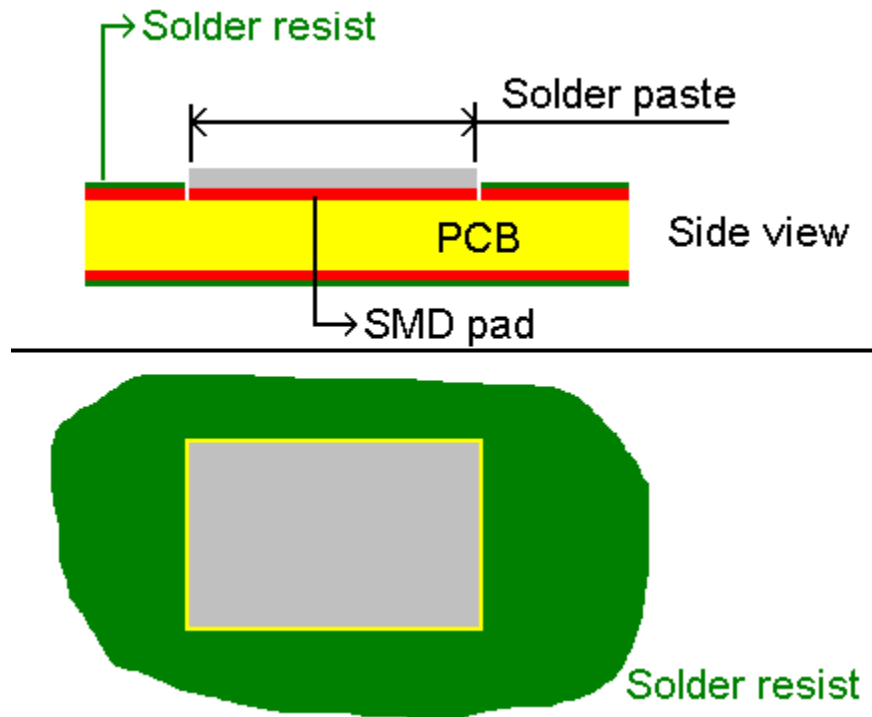
After the bottom side of the PCB has gone through a solder bath, all copper areas at the bottom of the PCB are soldered. The copper areas on the bottom PCB side which are covered with solder resist, are not soldered.

Usually the anti pad for the solder mask is 8 mil greater than the copper pad. The size of +8 mil for the solder mask, is because of tolerances.

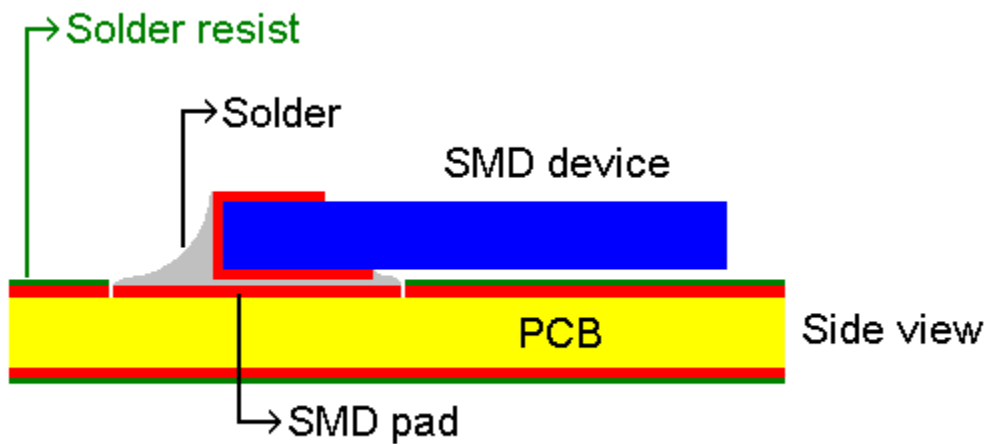
SMD pad



In the above figure a SMD pad is shown.



In the above figure a SMD pad with solder paste is shown.



In the above figure a SMD pad is shown after soldering. By applying heat on top of the PCB, SMD devices will be soldered using the solder paste on the pad as the solder.

Usually the anti pad for the solder mask is 8 mil greater than the copper pad, and the paste pad is the same size. The size of +8 mil for the solder mask, is because of tolerances.

Backup

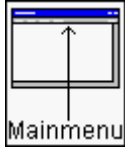


Backup

Make new geometry



Press **New geometry** button



Sub menu **File** menu item **New**

In the next dialogbox there is a listbox with six items.

Those items:



A new (empty) geometry will be made.



DIP



QUAD flatpack



BGA



PGA



SOIC

New DIP geometry

In the next dialogbox the parameters for a new DIP (Dual In Line) geometry can be entered. The DIP geometry is a device based on through holes.

The following parameters can be changed:

Nr pins

If the total amount of pins is two times the NrPins entered. The pinnumber counting starts with one, and increments with one for the following pins. The pin counting direction for the left column with pads is downwards, and for the right column with pads upwards.

Pad

Pad size solder mask

If the solder mask pad is not necessary, fill the parameter with zero.

Diameter anti power pad

The anti power pad will always be a circle.

If the anti power pad is not necessary, fill the parameter with zero.

Diameter inner pad

The inner pad will always be a circle.

If the inner pad is not necessary, fill the parameter with zero.

Drill hole

Distance

Clearance

The initial clearance is the clearance used for this geometry.

Pin 1 type

The first pin (1) can be a square or a circle. All the next pins will be circles.



Definition of a through hole pin

New QUAD flatpack geometry

In the next dialogbox the parameters for a QUAD flatpack geometry can be entered. The QUAD flatpack geometry is a SMD based device.

The following parameters can be changed:

Nr pins X,Nr pins Y

If the total amount of pins is two times the (Nr pins X + Nr pins Y) entered. The pinnumber counting starts with an optional string and a number. and increments with one for the following pins. The pin counting direction is counter clockwise.

Pad size X,Y

Pitch

Pad size solder mask

If the solder mask pad is not necessary, fill one of the parameters (X,Y) with zero.

Pad size paste mask

If the paste mask pad is not necessary, fill one of the parameters (X,Y) with zero.

Clearance

The initial clearance is the clearance used for this geometry.

Distance X,Y

Starting pin nr

The starting pin nr consists of two editboxes. The first editbox(optional) contains text or a number, and will not be changed. The second editbox contains a start number. This startnumber will be increased with one for the next pads.



Definition of a SMD pad

New BGA geometry

In the next dialogbox the parameters for a BGA (Ball Grid Array) geometry can be entered. The BGA geometry is a SMD based device.

The following parameters can be changed:

Nr pins

If the total amount of pins is (Nr pins X * Nr pins Y). The pinnumber counting starts with **A1** and increments with one for the following pins in the horizontal direction. In the vertical direction, counting is based on letter increments. For the first 23 rows the following letters will be used: **A,B,C,D,E,F,G,H,J,K,L,M,N,P,R,S,T,U,V,W,X,Y,Z**. Only the letters **I,O,Q** will not be used because of similarity with other characters. When more than 23 rows are necessary, two letters will be used. The 24th row will use the letters **AA**. The following rows will use the letters **AB,AC,AD, ... AZ,BA,BB ... BZ**, etc.

Pad

Pitch

Pad solder mask

If the solder mask pad is not necessary, fill one of the parameters (X,Y) with zero.

Pad paste mask

If the paste mask pad is not necessary, fill one of the parameters (X,Y) with zero.

Clearance

The initial clearance is the clearance used for this geometry.

Starting pin nr

The starting pin nr is **A1**, and the pad can be a circle or a square.



Definition of a SMD pad

New PGA geometry

In the next dialogbox the parameters for a BGA (Ball Grid Array) geometry can be entered. The BGA geometry is a device based on through holes.

The following parameters can be changed:

Nr pins

If the total amount of pins is (Nr pins X * Nr pins Y). The pinnumber counting starts with **A1** and increments with one for the following pins in the horizontal direction. In the vertical direction, counting is based on letter increments. For the first 23 rows the following letters will be used: **A,B,C,D,E,F,G,H,I,J,K,L,M,N,P,R,S,T,U,V,W,X,Y,Z**. Only the letters **I,O,Q** will not be used because of similarity with other characters. When more than 23 rows are necessary, two letters will be used. The 24th row will use the letters **AA**. The following rows will use the letters **AB,AC,AD, ... AZ,BA,BB ... BZ**, etc.

Pad

Diameter anti power pad

The anti power pad will always be a circle.

If the anti power pad is not necessary, fill the parameter with zero.

Diameter inner pad

The inner pad will always be a circle.

If the inner pad is not necessary, fill the parameter with zero.

Pitch

Drill

Pad solder mask

If the solder mask pad is not necessary, fill one of the parameters (X,Y) with zero.

Clearance

The initial clearance is the clearance used for this geometry.

Starting pin nr

The starting pin nr is **A1**, and the pad can be a circle or a square.



Definition of a through hole pin

New SOIC geometry

In the next dialogbox the parameters for a SOIC geometry can be entered. The SOIC geometry is a SMD based device.

The following parameters can be changed:

Nr pins

If the total amount of pins is two times the NrPins entered. The pinnumber counting starts with one, and increments with one for the following pins. The pin counting direction for the left column with pads is downwards, and for the right column with pads upwards.

Pad size X,Y

Pitch

Pad size solder mask

If the solder mask pad is not necessary, fill one of the parameters (X,Y) with zero.

Pad size paste mask

If the paste mask pad is not necessary, fill one of the parameters (X,Y) with zero.

Clearance

The initial clearance is the clearance used for this geometry.

Distance



Definition of a SMD pad

Initialisation file geom.ini

The initialisation file **geom.ini** is used to save some designs parameters. The file will be in the same directory as the geometries.

The following paramaters are used:

[ExeDirectory]

The next line contains the directory of the Executables directory.

[GeometryLibraryPath]

The next line contains the directory of the Geometries directory.

[Settings]

WindowWidth	The width of the windows
WindowHeight	The height of the windows
WindowStartX	Origin X of the windows (0,0 = left top)
WindowStartY	Origin Y of the windows
Units	(0 = mils, 1 = mm)
GridSize	The gridsize (10nm units)
DrawGrid	0 = FALSE, 1 = TRUE
DrawAreaFills	0 = FALSE, 1 = TRUE
DrawAreaFillWithHatches	0 = FALSE, 1 = TRUE
DrawClearances	0 = FALSE, 1 = TRUE
DrawCompOutline	0 = FALSE, 1 = TRUE
DrawConnections	0 = FALSE, 1 = TRUE
DrawDrills	0 = FALSE, 1 = TRUE
DrawInnerPads	0 = FALSE, 1 = TRUE
DrawTopPads	0 = FALSE, 1 = TRUE

DrawBottomPads	0 = FALSE,1 = TRUE
DrawCompPlacement	0 = FALSE,1 = TRUE
DrawSilkScreen	0 = FALSE,1 = TRUE
DrawObjects	0 = FALSE,1 = TRUE
DrawVias	0 = FALSE,1 = TRUE
DrawViaClearances	0 = FALSE,1 = TRUE
DrawCompReference	0 = FALSE,1 = TRUE
DrawCompValue	0 = FALSE,1 = TRUE
DrawTwoTryingTraces	0 = FALSE,1 = TRUE
SelectionMode	0 = replacement, 1= appending
Layer0	Draw bottom layer 0 (0 = FALSE,1 = TRUE)
Layer1	Draw layer 1 (0 = FALSE,1 = TRUE)
Layer2	Draw layer 2 (0 = FALSE,1 = TRUE)
...	...
Layer31	Draw layer 31 (0 = FALSE,1 = TRUE)
Grid0	Gridsize definition 0 (10nm units)
Grid1	Gridsize definition 1 (10nm units)
...	...
Grid29	Gridsize definition 29 (10nm units)
TraceWidth0	Trace width definition 0 (10nm units)
TraceWidth1	Trace width definition 0 (10nm units)
...	...
TraceWidth29	Trace width definition 29 (10nm units)
ClearanceWidth0	Clearance width definition 0 (10nm units)
ClearanceWidth1	Clearance width definition 1 (10nm units)
...	...
ClearanceWidth29	Clearance width definition 29 (10nm units)

BackColor	24 bit RGB color (Stored as 32 bit)
SilkScreenColor	24 bit RGB color (Stored as 32 bit)
CompOutlineColor	24 bit RGB color (Stored as 32 bit)
ShapePlacementOutLineColor	24 bit RGB color (Stored as 32 bit)
ShapePinsDrillColor	24 bit RGB color (Stored as 32 bit)
ShapePinsDrillUnplatedColor	24 bit RGB color (Stored as 32 bit)
ShapePinsCompSideColor	24 bit RGB color (Stored as 32 bit)
ShapePinsSoldSideColor	24 bit RGB color (Stored as 32 bit)
ShapeInnerPadColor	24 bit RGB color (Stored as 32 bit)
ShapePasteMaskCompSideColor	24 bit RGB color (Stored as 32 bit)
ShapePasteMaskSoldSideColor	24 bit RGB color (Stored as 32 bit)
ShapeSoldMaskCompSideColor	24 bit RGB color (Stored as 32 bit)
ShapeSoldMaskSoldSideColor	24 bit RGB color (Stored as 32 bit)
GeomNameColor	24 bit RGB color (Stored as 32 bit)
PowerPadColor	24 bit RGB color (Stored as 32 bit)
ClearanceColor	24 bit RGB color (Stored as 32 bit)
ButtonInfoColor	24 bit RGB color (Stored as 32 bit)
GridColor	24 bit RGB color (Stored as 32 bit)

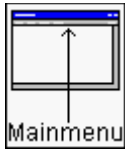
Zero relative cursor



Press **Ctrl z**

The relative cursor will be set to zero.

With the next menu item function the relative position will be on the grid or not.



Sub menu **View** menu item **Relative position on grid**

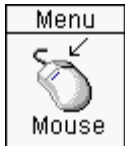
Undo



Press **Undo** button



Press **u**



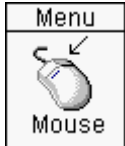
Undo

This function will undo almost all previous actions

Redo



Press **Redo** button



Redo

This function will redo previous undo actions.

Thickness line/clearance



Sub menu **Edit** menu item **Thickness line/clearance**

In the next dialogbox there are four items which can be changed:

- Trace thickness
- Clearance
- Line thickness component outline
- Line thickness silkscreen

Set origin point geometry



Sub menu **Edit** menu item **Move origin**

The origin of the geometry will be moved to the mouse position after pressing the **left mouse button**.

Set origin point geometry to center selected objects



Sub menu **Edit** menu item **Set origin to center selected objects**

The origin of the geometry will be set to the center of the selected objects. This center should first be marked with the function **Mark center selected objects**.



Move objects (special)

Set insertion point geometry



Sub menu **Edit** menu item **Set insertion point**

The insertion point of the geometry will be moved to the mouse position after pressing the **left mouse button**.

Set insertion point geometry to center selected objects



Sub menu **Edit** menu item **Set insertion point to center selected objects**

The insertion point of the geometry will be set to the center of the selected objects. This center should first be marked with the function **Mark center selected objects**.



Move objects (special)

Change geometry name



Sub menu **Edit** menu item **Change geometry name**

In the next dialogbox the geometry name (And also the filename) can be changed.

View topics



[Zoom in](#)



[Zoom out](#)



[Window based Zooming](#)



[Hide/view layers](#)



[Pan window](#)



[Window based panning](#)



[Return to previous view window](#)



[Repaint](#)



[View whole design](#)



[Change grid](#)

Hide/view layers



Press **Select layers** button



Press **Ctrl a**



Sub menu **View** menu item **Layers**

Change visibility layers.

The **hide/view layers** function can be used in every possible drawing/moving function.



Change grid

Zoom in



Press **z**



Sub menu **View** menu item **Zoom in**

The **zoom in** function can be used in every possible drawing/moving function.



Window based zooming

Zoom out



Press **Z**



Sub menu **View** menu item **Zoom out**

The **zoom out** function can be used in every possible drawing/moving function.



Window based zooming

Window based Zooming

To zoom in on a window, place the mouse cursor to the left top place of the window. Hold down the **Ctrl** key, than press and hold down the left mouse button. Move the mouse cursor in the right bottom direction of your window. After releasing the **Ctrl** key and the left mouse button zooming in will take place.

To zoom out, use the previous function, but now move the mouse cursor in the left top direction. The non changing rectangle visible is the border of your design. The changing rectangle is the zoom out window. After releasing the **Ctrl** key and the left mouse button, zooming out will take place.

The **window based zooming** function can be used in every possible drawing/moving function.

Pan window



Press $\leftarrow, \rightarrow, \uparrow, \downarrow$



Press **x**



Press **Shift** and move the mouse the window border

Window

Use the scrollbars

When pressing the **x** key, the window will be panned around the current mouse position, and the mouse position will be moved to the window center.

The **pan window** function can be used in every possible drawing/moving function.



Window based panning

Window based panning

There is a function available to view a different part of your design(special window for panning). To enter this function, hold down the **Ctrl** key, then press and hold down the **right** mouse button. The non changing rectangle visible is the border of your design. The changing rectangle is the viewable window. After releasing the **Ctrl** key and the **right** mouse button panning will take place.

The **Window based panning** function can be used in every possible drawing/moving function.

Return to previous view window



Press **v**



Previous view

Return to a previous view.

The **Return to previous view window** function can be used in every possible drawing/moving function.

Repaint



Press **F5**



Repaint

The whole window will be repainted.

The **Repaint** function can be used in every possible drawing/moving function.

View whole design



Press **Shift F8**



View whole design

The window view will be scaled that the whole design will fit.

The **View whole design** function can be used in every possible drawing/moving function.

Change colors



Sub menu **View** menu item
Change colors

The color settings can be modified in the next dialogbox. The color settings will be copied into the **geom.ini** initialisation file. This file is stored into the current shapes directory.

To use those colors for new designs, copy this **geom.ini** file to main directory. Whenever a new design is created this **geom.ini** file in the main directory will be copied to the **pcb\shapes** subdirectory of the new design.



[Initialisation file geom.ini](#)

Load default colors



Sub menu **View** menu item **Load
default colors**

The default color settings will be loaded.



Initialisation file geom.ini

Short cuts

The most important functions of the geometry editor have a short cut key (Accelerator). Those keys can be modified by editing the **geom.ini** file, section **[Keys]**.

Change units



Press **Ctrl u**



Sub menu **Units** item **mm/mils**

Changing the units (between mils/mm) is possible in every drawing/moving function.



Initialisation file geom.ini

Change grid



Press **Ctrl g**



Sub menu **View** menu item
Change grid

Changing the grid is possible in every drawing/moving function.

The grid settings in the dialogbox can be modified by changing the **geom.ini** settings.



[Initialisation file geom.ini](#)

View/hide grid



Press **g** (View/hide grid)

View/hide grid

Selection/deselection objects

To select an object, place the mouse cursor above the object, and press and hold the left mouse button. A rectangle will mark the selection window. There are two selection modes available. The first and default selection mode is the **replacement mode**, and the second selection mode is the **adding selection mode**.

The **replacement selection mode** means, every time a new selection rectangle is drawn the previous objects selected will be unselected. When pressing down the left shift key together with the left mouse button it is possible to use more than one selection at a time.

The other selection mode is the **adding selection mode**. In this mode every object which is selected stays selected, until the deselect all function is executed. To deselect an object press the left mouse and place the selection rectangle around this object again.

To change the selection mode use the **Replacement** or **Appending** in the **Selection mode** section of the menu.



Deselect all

Deselect all



Press **Deselect all** button



Press **F2**



Deselect all

Deselect all function.

Info on selected objects



Press **Info selected objects** button



Press **i**



Info

Displays some information about selected objects.

Add objects



Add rectange objects



Add circle objects



Add line objects



Add arc objects



Add text objects



Add trace



Add drill



Add rectangle SMD pads with solder and paste mask

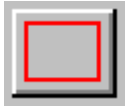


Add circle SMD pads with solder and paste mask

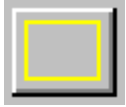


Add through hole pads with solder mask and drill hole

Add rectangle objects



Press **Add rectangle component outline** button



Press **Add rectangle placement outline** button



Press **r** (Placement outline)



Add pad -> Rectangle -> Select layer

Add rectangle -> Select layer

A rectangle object will be added. When the **spacebar** is pressed, an dialogbox will popup, and the rectangle parameters can be edited by hand. The first two parameters are the width, and height. The optional third and four parameter is the rectangle center. When the first character typed is a **@** the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).



Add rectangle SMD pads with solder and paste mask



Add through hole pads with solder mask and drill hole

Add pad -> rectangle

A rectangle (solid) pad can be added on the following layers:

- Pad top/bottom layer
- Solder mask top/bottom layer
- Paste mask top/bottom layer

Add rectangle

A rectangle (open) can be added on the following layers:

- Silkscreenlayer
- Component outline layer
- Placement outline layer

Add circle objects



Add pad -> circle -> select layer

Add circle -> select layer

A circle object will be added. When the **spacebar** is pressed, a dialogbox will popup, and the circle parameters can be edited by hand. The first parameter is the diameter. The optional second and third parameter is the circle center. When the first character typed is a **@** the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).



Add circle SMD pads with solder and paste mask



Add through hole pads with solder mask and drill hole

Add pad -> circle

A circle (solid) pad can be added on the following layers:

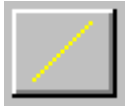
- Pad top/bottom layer
- Solder mask top/bottom layer
- Paste mask top/bottom layer

Add circle

A circle (open) can be added on the following layers:

- Silkscreenlayer
- Component outline layer
- Placement outline layer

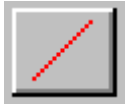
Add line objects



Press **Add line component outline** button



Press **L**



Press **Add line placement outline** button



Press **I**



Add line -> select layer

A line object will be added. When the **spacebar** is pressed, a dialogbox will popup, and the line parameters can be edited by hand. As many as 16 points (15 lines) can be edited. In addition, one point can be edited for the starting point of the line. When the first character typed is a **@** the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

A line can be added on the following layers:

- Silkscreen layer
- Component outline layer
- Placement outline layer

Add arc objects



Add arc -> select layer

An arc object will be added. When the **spacebar** is pressed, a dialogbox will popup, and the arc parameters can be edited by hand. The first parameters are the diameter. The optional second and third parameter is the arc center. The optional fourth and fifth parameter is the first radial ending point. The optional sixth and seventh parameter is the second radial ending point. When the first character typed is a **@** the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

A arc can be added on the following layers:

- Silkscreen layer
- Component outline layer
- Placement outline layer

Add text objects



Add text -> select layer

A text object will be added. In the next dialogbox the text can be entered. In addition the text height can be edited. After pressing the **OK** button the text can be placed. When the **spacebar** is pressed, a dialogbox will popup, and the text placement point can be edited by hand. When the first character typed is a **@** the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

Text can be added on the following layers:

- Silkscreen layer
- Component outline layer

Add trace



Add trace->

Copper top layer

Solder mask top layer

Paste mask top layer

Copper bottom layer

Solder mask bottom layer

Paste mask bottom layer

A trace with the current trace width and clearance and layer will be added.



Change trace width



Change clearance

Add drill



Press **Add unplated drill hole** button



Add drill -> plated

Add drill -> unplated

A drill hole (plated/unplated) will be added. When the **spacebar** is pressed, a dialogbox will popup, and the drill hole parameters can be edited by hand. The first parameter is the diameter. The optional second and third parameter is the drill hole center. When the first character typed is a **@** the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

Add rectangle SMD pads with solder and paste mask



Press **Add rectangle SMD pads** button



Add pad -> SIL SMD rectangle pads



Copy special -> Add SIL SMD based on selected objects

If a number of rectangular SMD pads (Pads, paste mask and solder mask) on a equal distance needs to be included, this function can do the job. In the next dialogbox all the necessary parameters can be entered. After pressing the **OK** button the pads can be placed. When pressing the **right mouse button** the pads will rotate 90 degrees counter clock wise. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change. When the **spacebar** is pressed, a dialogbox will popup, and the position of the first pad can be edited by hand. When the first character typed is a **@** the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

The function **Add SIL SMD based on selected objects** will do the same, but the dialogbox parameters will already be filled, with the parameters of a **selected** pad.

The following parameters can be changed:

Pad width and height

Pitch

Nr pads

Pad width and height solder paste

If the solder paste pad is not necessary, fill the X or Y parameter with zero.

Pad width and height solder mask

If the solder mask pad is not necessary, fill the X or Y parameter with zero.

Top/bottom layer

The pads will be placed on the top or bottom layer

Clearance

The initial clearance is the clearance used for this geometry.

Startpin

The startpin consists of two editboxes. The first editbox(optional) contains text or a number, and will not be changed. The second editbox contains a start number. This startnumber will be increased with **Increment** for the next pads.



Definition of a SMD pad

Add circle SMD pads with solder and paste mask



Press **Add round SMD pads** button



Add pad -> SIL SMD circle pads



Copy special -> Add SIL SMD based on selected objects

If a number of circular SMD pads (Pads, paste mask and solder mask) on a equal distance needs to be included, this function can do the job. In the next dialogbox all the necessary parameters can be entered. After pressing the **OK** button the pads can be placed. When pressing the **right mouse button** the pads will rotate 90 degrees counter clock wise. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change. When the **spacebar** is pressed, a dialogbox will popup, and the position of the first pad can be edited by hand. When the first character typed is a **@** the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

The function **Add SIL SMD based on selected objects** will do the same, but the dialogbox parameters will already be filled, with the parameters of a **selected** pad.

The following parameters can be changed:

Pad diameter

Pitch

Nr pads

Pad diameter solder paste

If the solder paste pad is not necessary, fill the parameter with zero.

Pad diameter solder mask

If the solder mask pad is not necessary, fill the parameter with zero.

Top/bottom layer

The pads will be placed on the top or bottom layer

Clearance

The initial clearance is the clearance used for this geometry.

Startpin

The startpin consists of two editboxes. The first editbox(optional) contains text or a number, and will not be changed. The second editbox contains a start number. This startnumber will be increased with **Increment** for the next pads.



Definition of a SMD pad

Add through hole pads with solder mask and drill hole



Press **Add round through hole** button



Press **Add square through hole** button



Add pad -> SIL through hole pads



Copy special -> Add SIL based on selected objects

If a number of circular through hole pads (Pads,paste mask) on a equal distance needs to be included, this function can do the job. In the next dialogbox all the necessary parameters can be entered. After pressing the **OK** button the pads can be placed. When pressing the **right mouse button** the pads will rotate 90 degrees counter clock wise. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change. When the **spacebar** is pressed, a dialogbox will popup, and the position of the first pin can be edited by hand. When the first character typed is a **@** the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

The function **Add SIL based on selected objects** will do the same, but the dialogbox parameters will already be filled, with the parameters of a **selected** pad.

The following parameters can be changed:

Pad size

Pitch

Nr pins

Pad size solder mask

If the solder mask pad is not necessary, fill the parameter with zero.

Diameter anti power pad

The anti power pad will always be a circle.

If the anti power pad is not necessary, fill the parameter with zero.

Diameter inner pad

The inner pad will always be a circle.

If the inner pad is not necessary, fill the parameter with zero.

Drill hole

Clearance

The initial clearance is the clearance used for this geometry.

Pintype

The pin can be a square or a circle.

Startpin

The startpin consists of two editboxes. The first editbox(optional) contains text or a number, and will not be changed. The second editbox contains a start number. This startnumber will be increased with **Increment** for the next pads.



Definition of a through hole pin

Change objects



Move objects



Move objects (special)



Delete objects



Copy objects



Copy on multiple coordinates



Copy objects to a different layer



Rotate objects



Mirror objects



Unselect objects



Select only



Assign objects to pin



Change circle objects



Change rectangle objects



Change trace width



Change text



Change text height



Change line width



Change clearance

Change circle objects



Change diameter circles

The diameter of **selected** circles can be changed into a new value typed in the following dialogbox.

Change rectangular objects



Change width/height rectangles

The width and height of **selected** rectangles can be changed into a new value typed in the following dialogbox.

Change text



Change text



Press t

The **selected** text can be changed in the following dialogbox.

Change text height



Change textheight

The textheight of **selected** texts can be changed into a new value typed in the following dialogbox.

Change trace width



Change trace width

The line width of **selected** objects can be changed into a new value typed in the following dialogbox.

Objects:

- Copper top/bottom layer
- Solder mask top/bottom layer
- Paste mask top/bottom layer

Change line width



Change line width

0 The line width of **selected** objects can be changed into a new value typed in the following dialogbox.

1 Objects:

- component outline layer
- silkscreen layer
- geometry name

Change clearance



Change clearance

The clearance of **selected** objects can be changed into a new value typed in the following dialogbox.

Delete objects



Press **Delete** button



Press **Del**



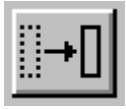
Delete

Delete **selected** objects.



Undo

Move objects



Press **Move** button



Press **m**



Move

Move **selected** objects. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change. When the **spacebar** is pressed, a dialogbox will popup, and the endpoint parameters can be edited by hand. The endpoint coordinates will be the center of the selected objects. When the first character typed is a **@** the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

Move objects (special)

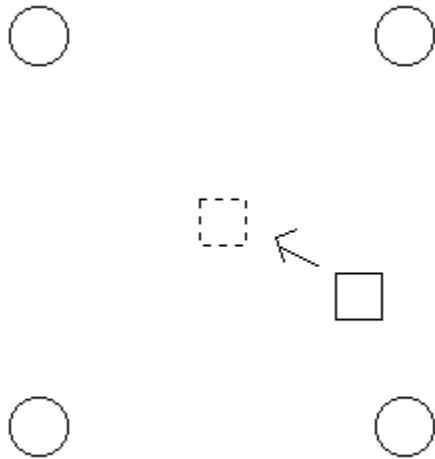


Special move/centering -> Mark center selected objects

Special move/centering -> Move objects centered to previous selected objects

Move **selected** objects on a special way.

An example:



Suppose the square has to be moved to the center of the four corner circles. This can be achieved by selecting the four circles. When the four circles have been selected use the function **Mark center selected objects**. After this execution of this function select the rectangle, and use the function **Move objects centered to previous selected objects**.

Rotate objects



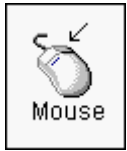
Press **R**



Rotate

Rotate **selected** objects 90 degrees counter clock wise.

Mirror objects

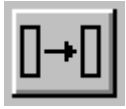


Mirror X

Mirror Y

Mirror **selected** objects in X or Y direction.

Copy objects



Press **Copy** button



Press **c**



Copy

With this function **selected** objects can be copied to a new location. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change. When the **spacebar** is pressed, a dialogbox will popup, and the endpoint parameters can be edited by hand. The endpoint coordinates will be the center of the selected objects. When the first character typed is a **@** the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

Copy objects to a different layer



Copy to other layer ->

Copper top (Only traces/rectangles/circles)

Copper bottom (Only traces/rectangles/circles)

Silkscreen (Only lines/circles/arcs/texts)

Component outline (Only lines/circles/arcs/texts)

Paste mask top (Only traces/circles/rectangles)

Paste mask bottom (Only traces/circles/rectangles)

Solder mask top (Only traces/circles/rectangles)

Solder mask bottom (Only traces/circles/rectangles)

Drill plated (Only circles)

Drill unplated (Only circles)

Inner pad (Only circles)

Power pad (Only circles)

With this function **selected** objects can be copied to the specified layer. Selected objects on the same layer as the specified layer will not be copied.

Copy on multiple coordinates



Copy on multiple coordinates

In the next dialogbox a maximum of 16 coordinates (x,y) can be typed. At every coordinate the **selected** objects will be copied. This can be handy when a range of the same pins should be added, on many different coordinates.

Unselect objects



Unselect -> Select object layer

Unselect objects.

Select only



Select only -> Select object layer

Select only objects.

Assign objects to pin



Press **Assign objects to pin** button



Press **a**



Assign objects to pin

Selected objects will be assigned to a pinnumber (pinname). In the next dialogbox the pinnumber(name) can be selected or edited.

It is possible to assign as many objects as necessary to a pinnumber(name).

The maximum length of a pinnumber(name) is 9 characters.



Press **A**

Selected objects will be assigned to a pinnumber which equals the previous assigned pinnumber + 1.

