



ALTium **365**

## **Altium Designer**

### **Essentials Course - Altium 365**

Component Creation for A365 Workspace

Software, documentation and related materials:

Copyright © 2024 Altium LLC

All rights reserved. You are permitted to use this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document are made. Unauthorized duplication, in the whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium LLC. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties.

ACTIVEBOM®, ActiveRoute®, A365™, Altium 365®, Altium Concord™, Altium Concord Pro™, Altium Designer®, AD™, Altium NEXUS®, Altium OnTrack™, Altium Vault®, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Common Parts Library™, Concord™, Concord Pro®, Draftsman®, Dream, Design, Deliver®, DXP™, Easytrax®, EE Concierge®, Fearless HDI™, Geppetto®, Gumstix®, Learn, Connect, Get Inspired™, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, OnTrack™, Overo®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™, Upverter®, X2®, XSignals® and their respective logos are trademarks or registered trademarks of Altium LLC or its affiliated companies. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.

# Table of Contents

<b>Component Creation for A365 Workspace</b>	<b>3</b>
<b>1.1 Purpose</b>	<b>3</b>
<b>1.2 Shortcuts</b>	<b>3</b>
<b>1.3 Preparation</b>	<b>3</b>
<b>1.4 Create a New Component</b>	<b>4</b>
1.4.1 Creating a New Component .....	4
<b>1.5 Component Editor</b>	<b>6</b>
1.5.1 Add Part Information: Parameters, Symbol and Footprint .....	6
<b>1.6 Creating a New Schematic Symbol</b>	<b>9</b>
1.6.1 Creating a New Schematic Symbol .....	9
1.6.2 Preparing the Design Space .....	10
1.6.3 Defining the Pins .....	10
1.6.4 Adding Symbol Body .....	12
<b>1.7 Creating a New PCB Footprint</b>	<b>15</b>
1.7.1 Footprint name and definition .....	16
1.7.2 Preparation for Footprint Creation.....	16
1.7.3 Adding Pads.....	18
1.7.4 Setting the Component Origin.....	19
1.7.5 3D Crystal Body.....	20
1.7.6 Adding Top Overlay Outline .....	23
<b>1.8 Save the component to the workspace</b>	<b>27</b>

# Component Creation for A365 Workspace

## 1.1 Purpose



In this exercise, you will learn to create components in Altium Designer. A component is a combination of a schematic symbol, a PCB footprint, and parameters containing all relevant information required for component procurement and board assembly.

The management of components used in a design is a fundamental element of all PCB designs. Altium Designer has three distinct ways in which it is possible to manage components, the simplest is Integrated libraries, and then there is the Database libraries, the best and most advanced way of managing your library components is, is the Altium 365 Workspace.

In the real-world, components that get mounted on the board are represented as schematic symbols during design capture, and as a PCB footprint for board design (optionally, with a 3D model for visualization, 3D clearance checking, and export to the mechanical CAD domain). A component can also include a simulation model for the circuit simulator and an IBIS model for signal integrity analysis. There is also an option to use *Part Choices* to allow you to specify different options for a single component, for example, different manufacturers for the same type of component.

In this exercise we will create a component in an Altium 365 workspace. The Workspace library is a collection of components that belong to your company's internal library, it contains components that can be used consistently across different PCB design projects.

## 1.2 Shortcuts



Shortcuts when working with Component Creation for A365 Workspace

**Ctrl+S:**                  Save Document

## 1.3 Preparation

1. **Close all existing projects and documents.**

## 1.4 Create a New Component

### 1.4.1 Creating a New Component

2. Select the **File » New » Component...** command from the main menu, see Figure 1. A new dialog *Create new component* opens, described at step 4, after we show you an alternative way to start the component creation.

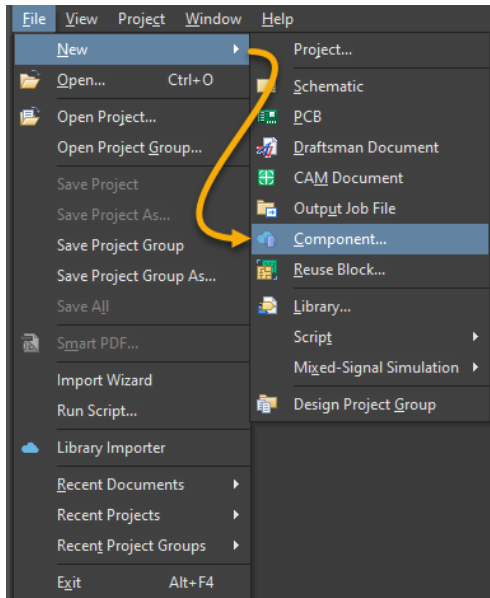


Figure 1. New Component Creation

3. Alternatively, select **File » New » Library...** from the main menus. In the *New Library* dialog that opens, select **Create Library Content » Component...** from the **Workspace** region of the dialog, Figure 2.

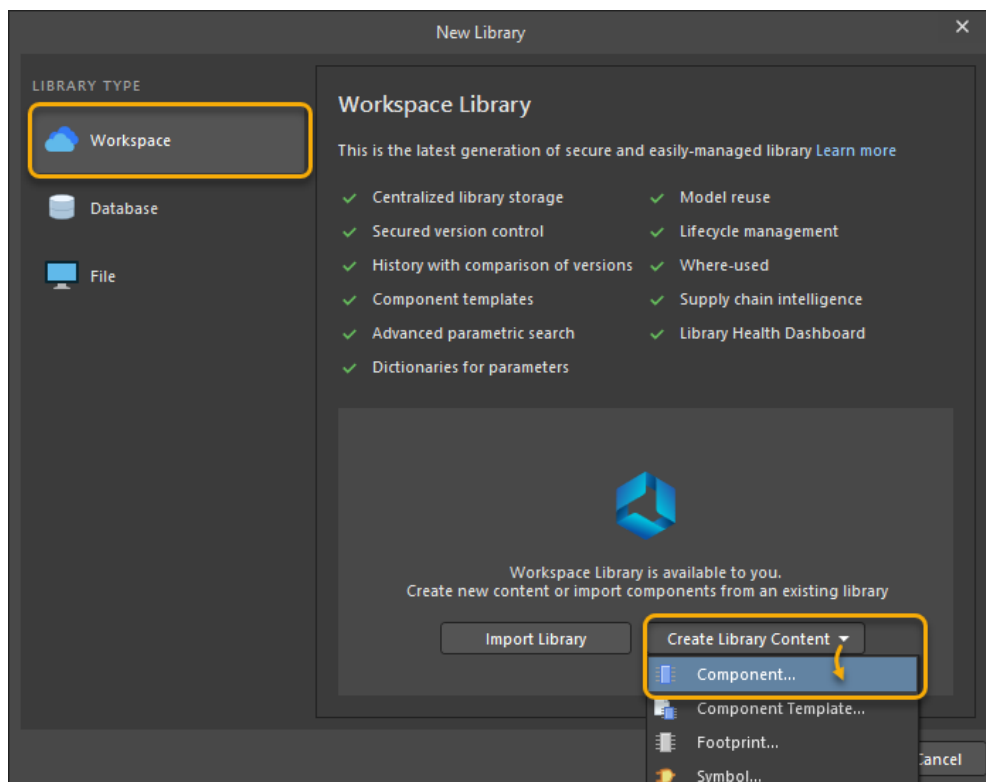


Figure 2. New Library - Create Component in Workspace

4. From the *Create new component* dialog that opens, choose a component type. If there is a component template linked to the selected component type, this template will be used to predefine the component parameters. See Figure 3

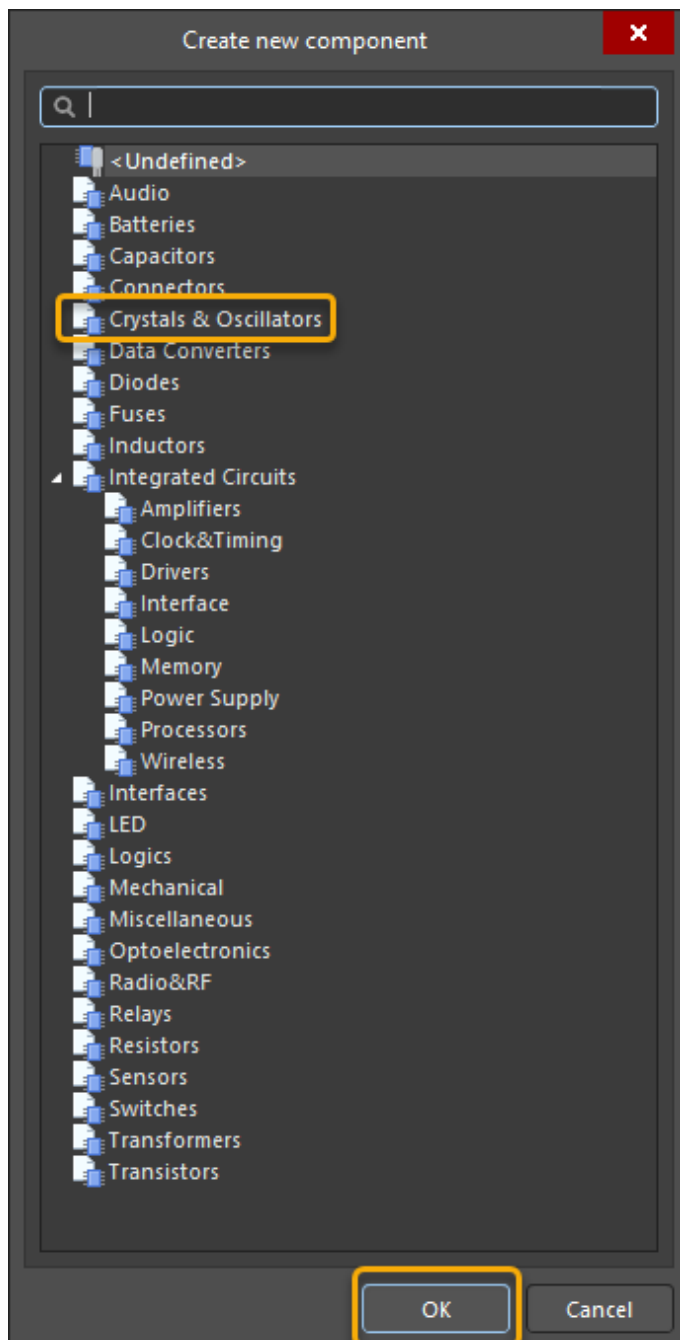


Figure 3. Component Type

## 1.5 Component Editor

### 1.5.1 Add Part Information: Parameters, Symbol and Footprint

5. After clicking **OK**, the *Component Editor* will open, as mentioned earlier. If a component template is linked to the component type, the information will be shown in the *Parameters* section; also, if there are predefined symbols and even footprints, these can also be linked to a template. However, in this exercise, we will create a new schematic symbol and a new PCB footprint. See Figure 4.

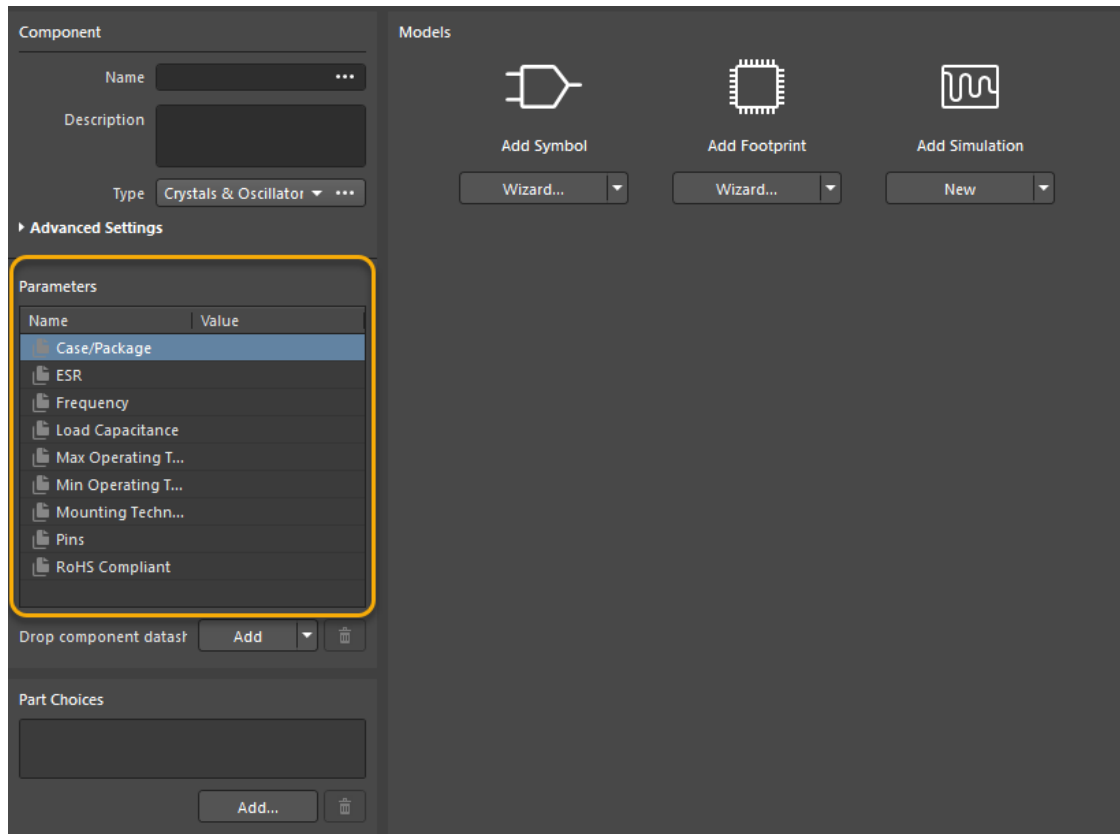


Figure 4. Component Editor

Next, we will add details such as the component's **Name** and **Description**, this information can be very important when using the component search options.

The **Advanced Settings** section is used to define or change some of the other component details such as the component's **Id**, this will be as defined in the workspace, and **Folder details** where the component will be placed in the target workspace.

The **Parameters** region of the Component Editor contains the electrical specification for the component as well as information required for procurement purposes, such as the component manufacturer, manufacturer part number, internal company stock-codes, URL links to website pages (for example, a manufacturer's website), and links to datasheets, and any other information required for reporting purposes.

In the **Part Choices** region of the Component Editor, it is possible to specify one or more Part Choices for the component that can be used.

In the **Models** region of the Component Editor, we can add links to existing symbols and footprints.

6. Using Figure 5 as reference, in the top of the *Component* section, add the following information:
- a) **Name:** 50MHZ\_HC49\_SMD
  - b) **Description:** 50MHz 20pF 20ppm SMD

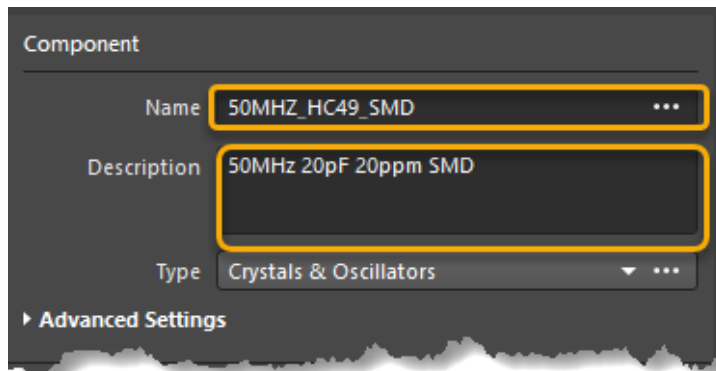


Figure 5. Component Name and Description

7. In the *Parameters* section using Figure 6 as reference, add the following values:
- a) **Frequency:** 50MHz
  - b) **Load Capacitance:** 20pF

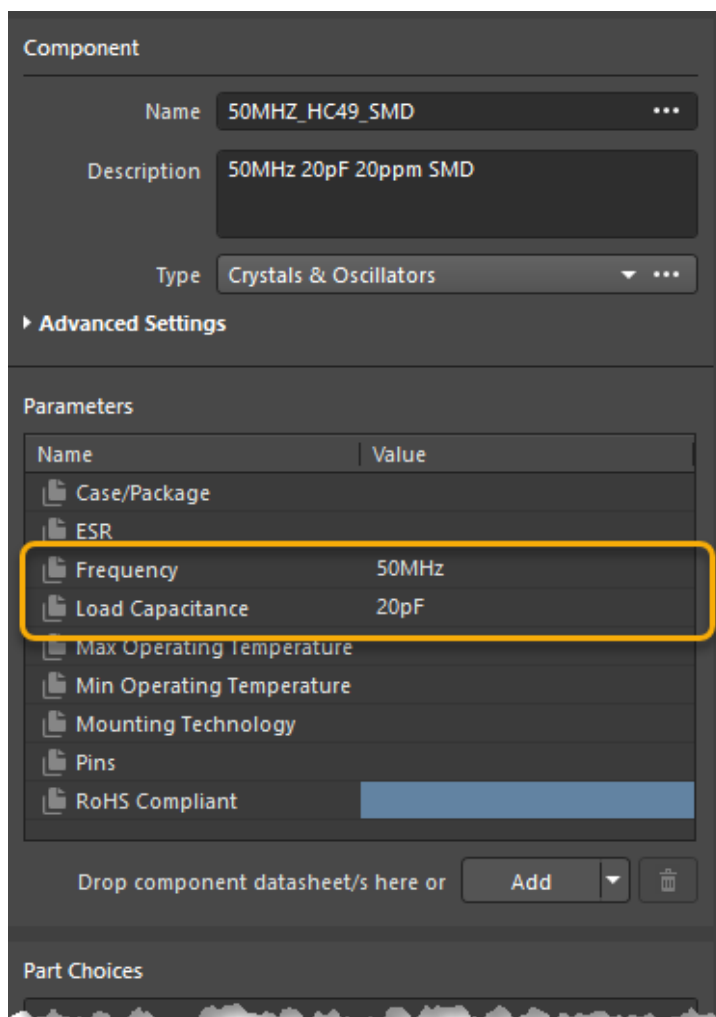


Figure 6. Component Parameters



8. Next, using Figure 7 as reference, add a new parameter and its value by selecting the button **Add » Parameter:**
- a) **Frequency Tolerance:** 20ppm

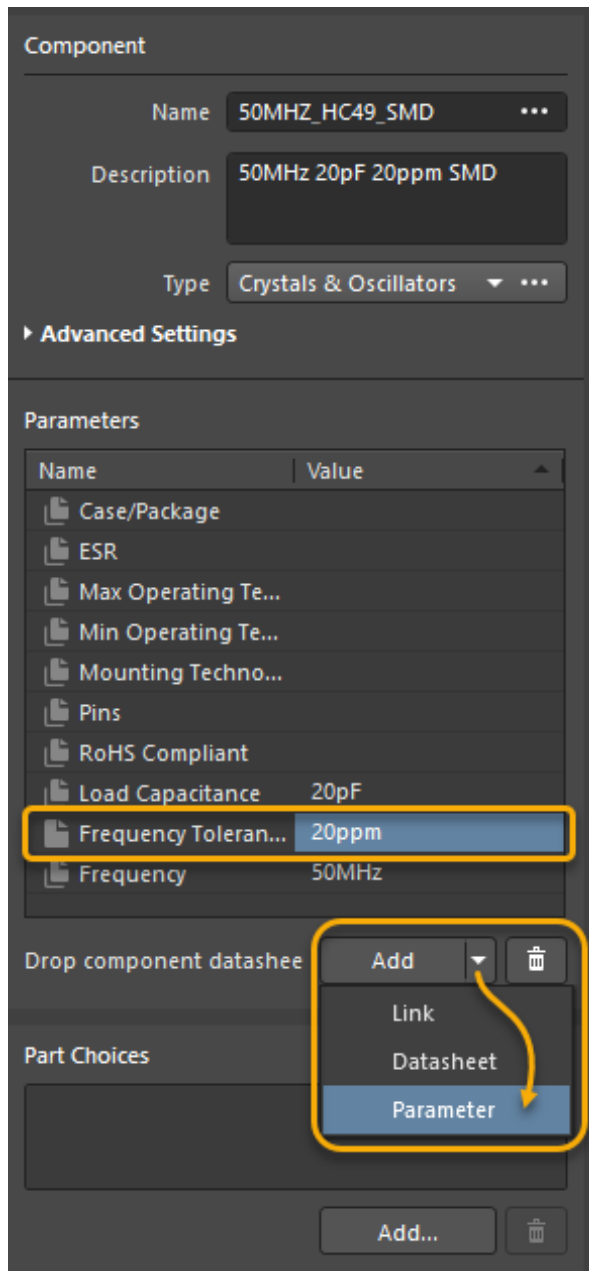




Figure 7. Add a new parameter



If a parameter is missing, it is also recommended to contact the Workspace Administrator, so that the template can be updated to include the missing parameter.

9. Feel free to save the modifications you have done so far  Save .

## 1.6 Creating a New Schematic Symbol

### 1.6.1 Creating a New Schematic Symbol

In this section we will learn how to create a new schematic symbol.

10. From the **Add Symbol** of the *Models* section, select **New** from the dropdown list.  
See Figure 8.

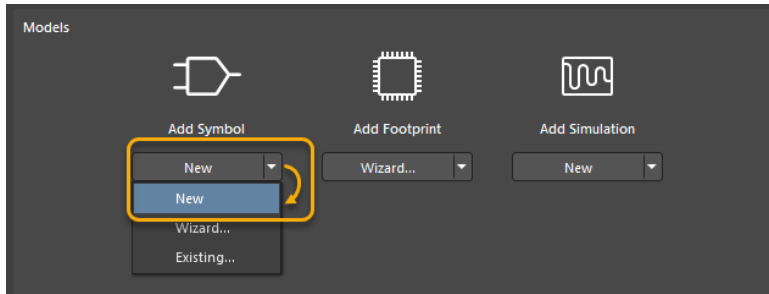


Figure 8. Creating New Symbol

11. A new schematic symbol document will be added to the document bar, Figure 9.

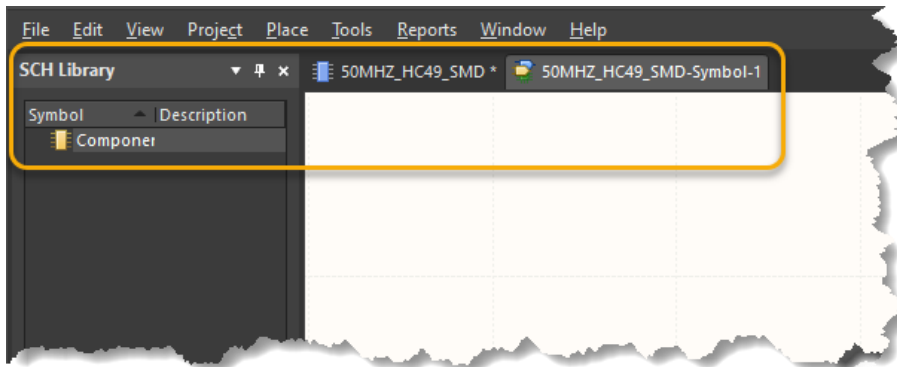


Figure 9. New Schematic Symbol Document

12. In the *Properties* panel enter the following information, Figure 10:

- a) **Designator:** X?
- b) **Name:** Crystal - Generic
- c) **Description:** General Crystal Symbol

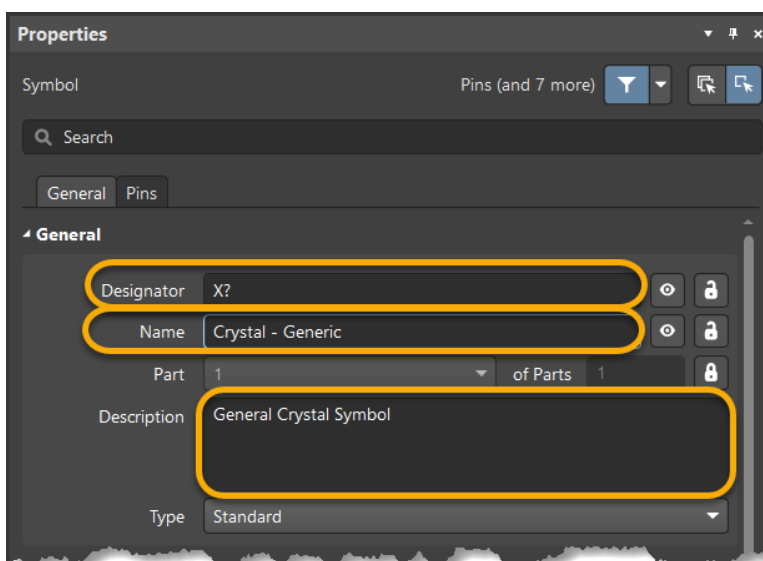


Figure 10. Symbol Properties

## 1.6.2 Preparing the Design Space

Schematic symbols are created by placing drawing objects to represent the component body, and pins to represent the physical pins on the actual component.

The default units for schematic and schematic library grids are imperial. Since all Altium components symbols are designed on this imperial grid, it is important to appreciate the impact of deciding to switch to a metric sheet grid, as it becomes difficult to correctly wire to symbols created on different grids. Note that imperial grids can be used with metric sheet sizes, such as A3, so it is not necessary to change to a metric grid when working with metric-sized sheets.

## 1.6.3 Defining the Pins

It is recommended to draw the component symbol close to the sheet origin (the centre of the sheet). If necessary, relocate the origin of the sheet to the centre of the design window by selecting **Edit » Jump » Origin** (shortcut **J, O**). Check the Status bar at the bottom left of the screen to confirm that you have the cursor at the origin.

A proper Snap Grid will help with the symbol creation. The common grid for placing pins is 50mil, 100mil or 200mil. The ideal setting for this exercise (because of the symbol body creation ) is a snap grid of 50mil

13. Go to **Tools » Document Options....** to open the *Properties* panel. Change the *Snap Grid* to 50mil as shown in Figure 11.

As an alternative, you can click anywhere in the schematic editor, and hit the **G** key to toggle between the preset grid values, 10mil, 50mil or 100mil.

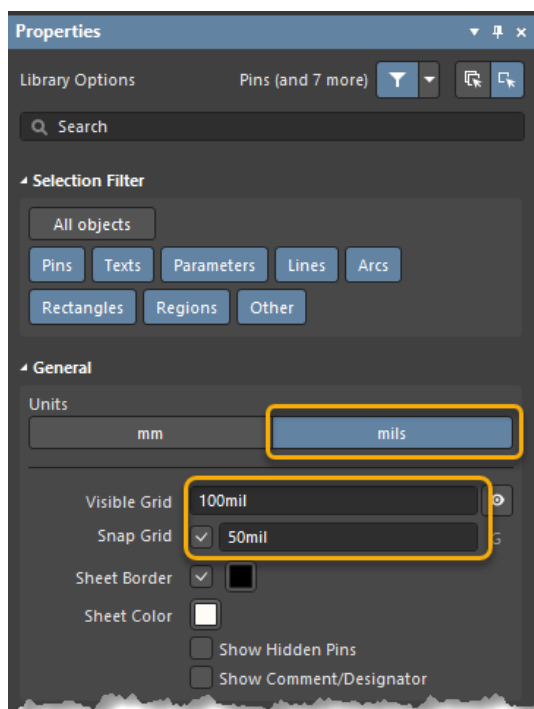


Figure 11. Grid Properties

14. Next, using the **Active Bar** start placing a **Pin**, hit **Tab** to open its properties before you place the pin.



Figure 12. Placing Pin

15. Configure the settings to match those shown in Figure 13. Hit the **Enter** key or select the **Pause** icon (⏸) after you've updated all the fields, continue the pin placement.

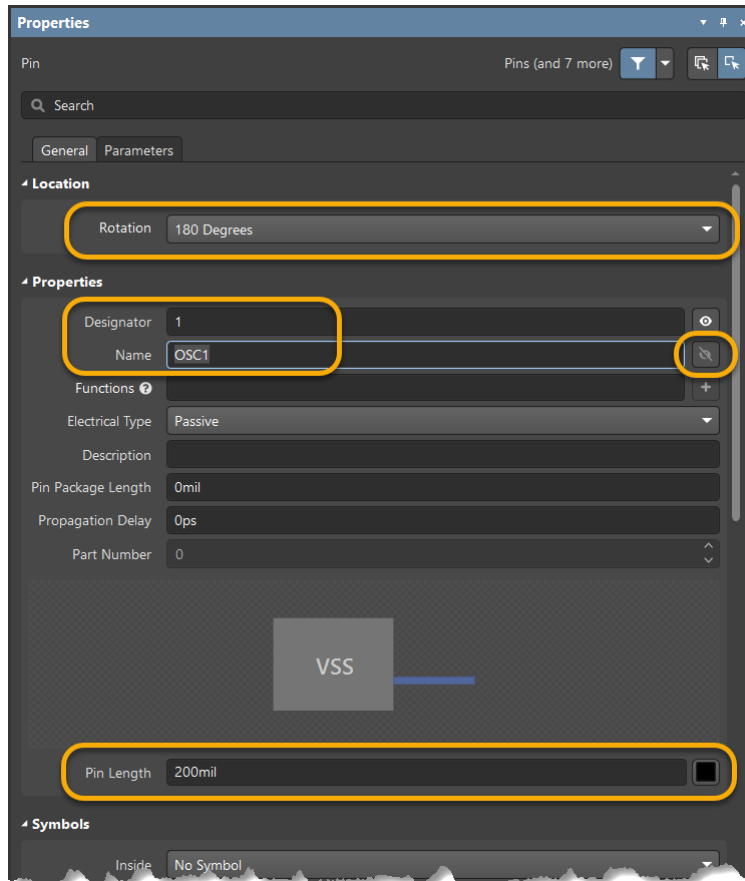


Figure 13. Pin Properties



Alternatively, it is possible to place a pin using the **Place » Pin** (or use the **P»P** shortcut keys).

16. Notice that one end of the pin has a small “X” as shown in Figure 14. This X crosshair defines its electrical connection or hotspot.
17. Use the **X** key to flip the pin horizontally or use the **Spacebar** to help with the vertical orientation if needed.

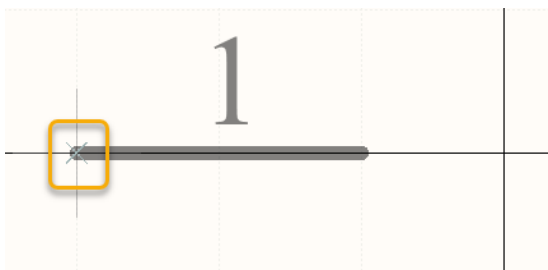


Figure 14. Pin Electrical Hotspot

18. Using *Figure 15* as reference,
- Place pin 1 to the left of the origin, at the coordinates X= (-350mm), Y= (0mil)
  - After placing Pin 1, another pin will appear on the cursor with the pin name and designator already incremented to 2.
  - For Pin 2, ensure that the electrical hotspot is on the right-side of the pin. Then, position it to the right of the origin at the coordinate X= (350mil) Y=(0mil) as shown in *Figure 15*.
  - Right-click** or hit **ESC** to exit pin placement mode.

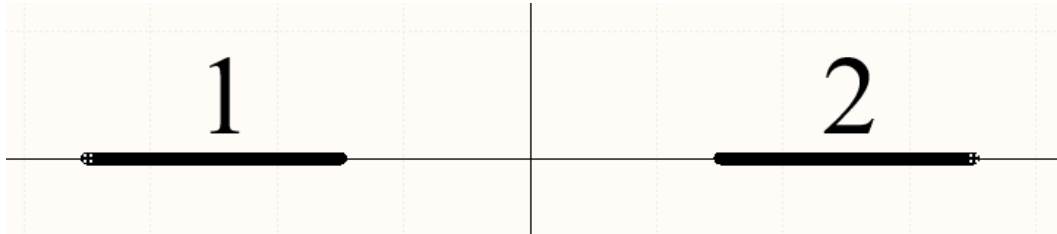



Figure 15. Position pin 2 to the right of pin 1

### 1.6.4 Adding Symbol Body

- To place the two vertical lines on the end of each pin, use the **Active Bar** or command **Place » Line**.
- Press the **Tab** key during placement to open the *Properties* panel.
  - Set the *Line Width* to **Medium** and choose a color of your choice.
  - Press **Enter** or the Pause icon  to continue the placement.
- Draw the 2 lines similar to what is shown in Figure 16.
- Right-click to terminate each line segment.
- Right-click a second time when you are finished to terminate the command.

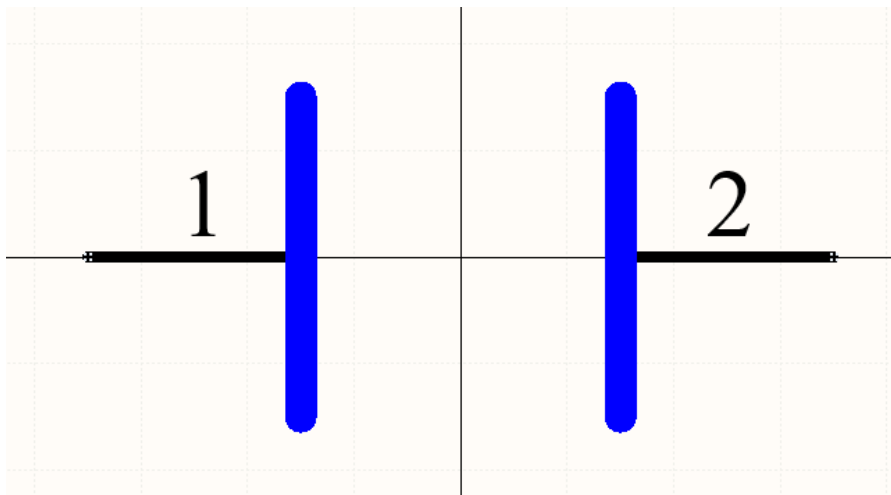



Figure 16. Add Lines to Crystal Symbol

24. Go to  **Preferences**, under the *Schematic* branch, and go to the *Defaults* page. Select the object *Round Rectangle* from the *Primitive* list and change both the *Corner X Radius* and *Corner Y Radius* to 50mil. See Figure 17.
25. Select **OK** to apply the modification as default.

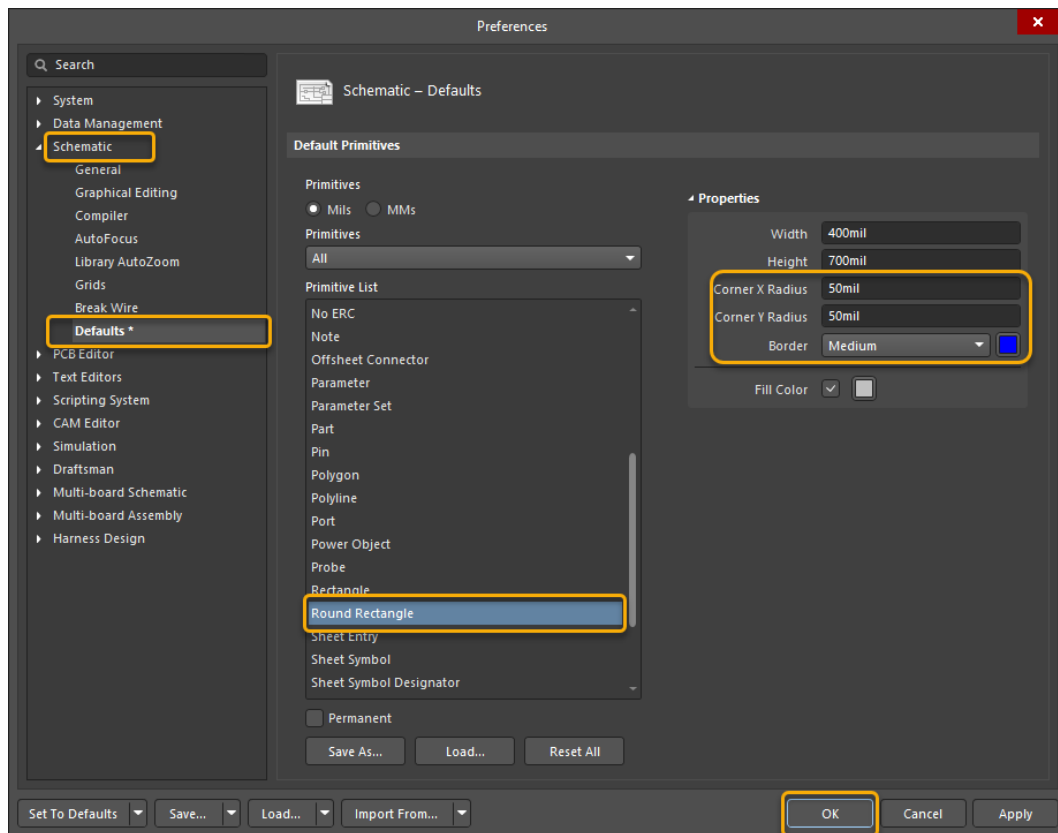


Figure 17. Rounded Rectangle Properties

26. Go to **Place » Round Rectangle** command to add the central body for the crystal.
27. Hit the **Tab** key to open its *Properties* panel. Change the *Border* to **Medium** and select your *Border* and *Fill* color of choice as shown in Figure 18.

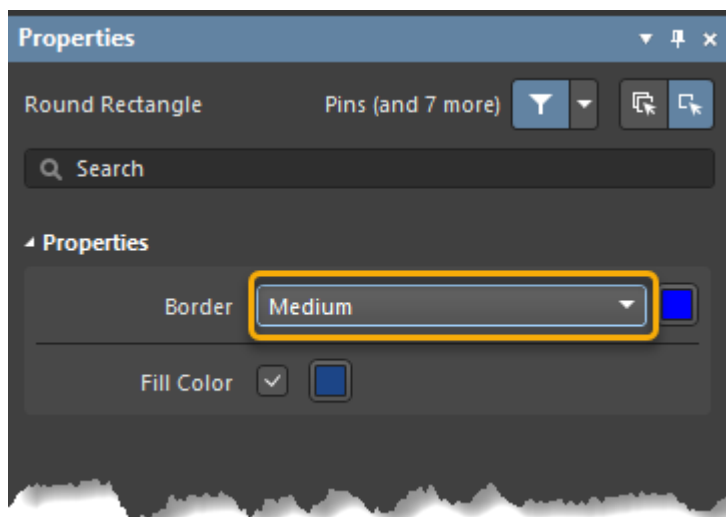


Figure 18. Rounded Rectangle Properties

28. Hit **Enter** or the Pause icon  to start placing the body.

29. Draw the rectangle in the center of the symbol as shown in Figure 19 by clicking in the top left corner, then the bottom right corner to complete the shape. Feel free to make adjustments as needed.
30. Right-click to terminate the command.

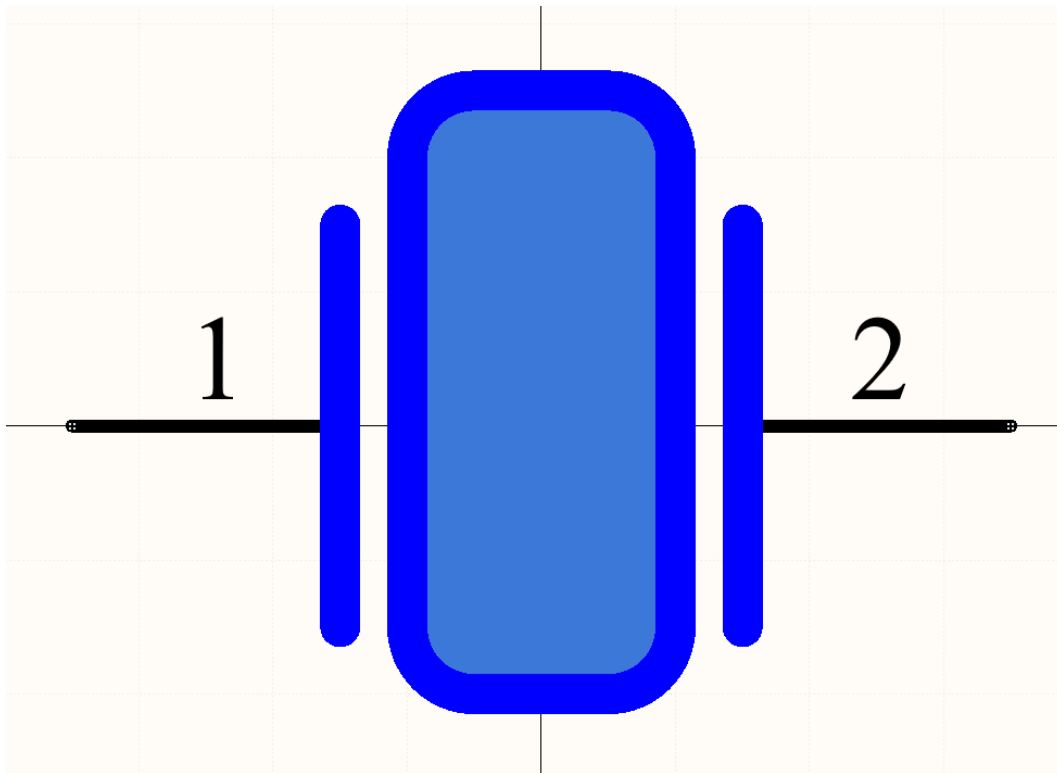



Figure 19. Crystal with Rectangle Body.

31. Save the library by going to **File » Save**  Save Ctrl+S.
32. Return to the *Component Editor* dialog.

## 1.7 Creating a New PCB Footprint

In this section we will learn how to create a new PCB footprint.

33. From the **ADD Footprint** of the Models section, from the dropdown list select **New**. See Figure 20

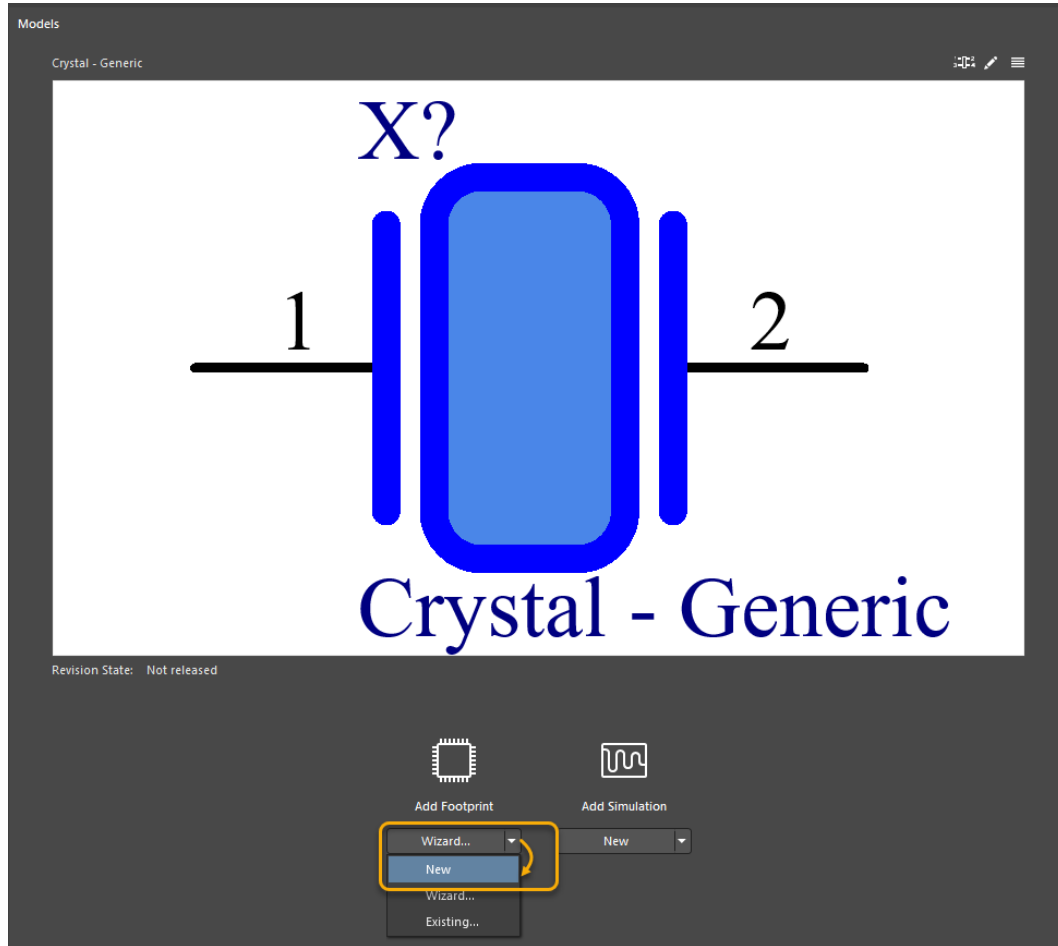


Figure 20. Creating New Footprint

After starting **Add Footprint**, a new footprint document will appear and a tab will be added to the **Document Bar**. Figure 21

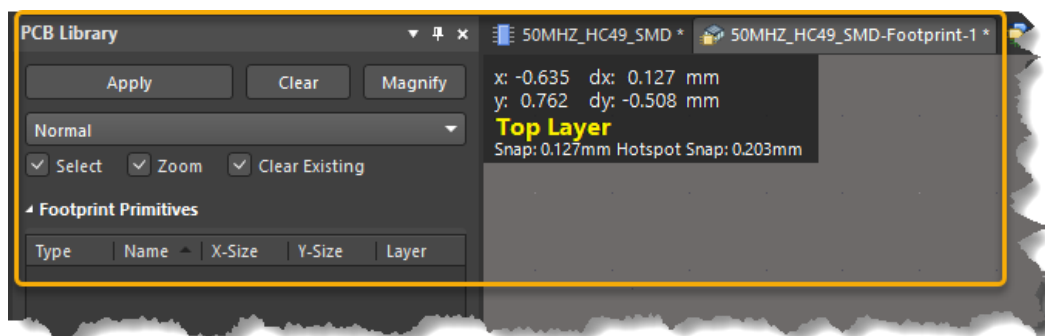


Figure 21. PCB Footprint Document



### 1.7.1 Footprint name and definition

34. Select **Tools » Footprint Properties** to open the *Properties* panel with the general Footprint configuration and add the following.
- a) **Name:** HC49SMD
  - b) **Description:** is HC49SMD-RoHS Compliant
  - c) **Height:** 4.5mm
  - d) **Area:** 128 sq.mm

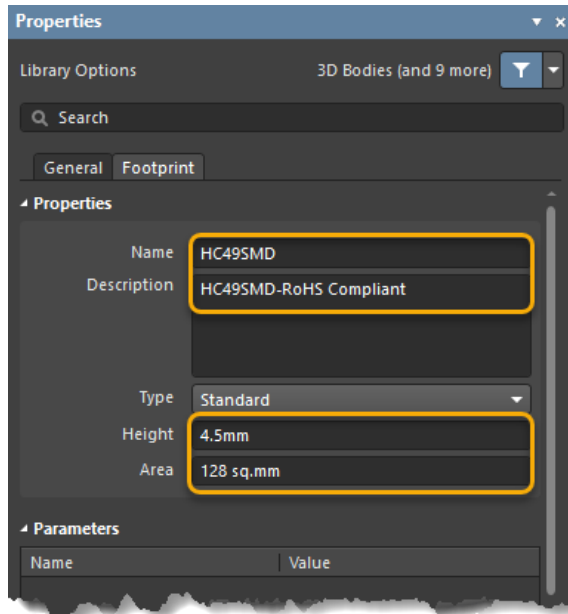


Figure 22. Footprint Name and parameters

### 1.7.2 Preparation for Footprint Creation

Below is a 2D and 3D preview of the footprint we will be creating in this exercise. Figure 23

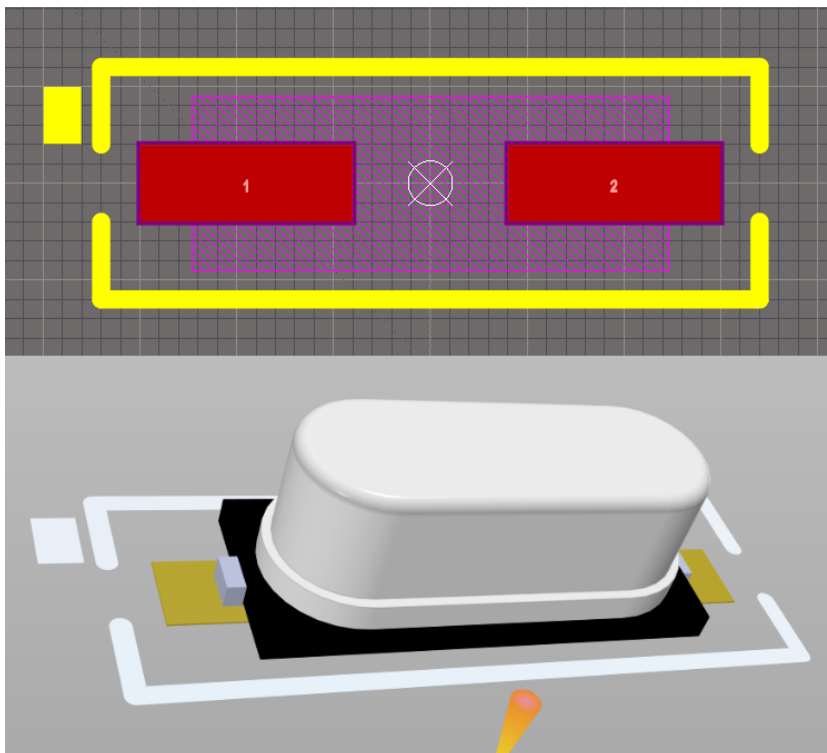


Figure 23. Footprint Preview

35. Ensure the *Properties* panel is visible, if not select it from the Panels list **Panels** or use **F11** function key.
36. Change the measurement units to **Metric**, by either using the **Q** key the when the editing space is active, or by scrolling down the *Properties* panel and selecting **mm**, Figure 24.

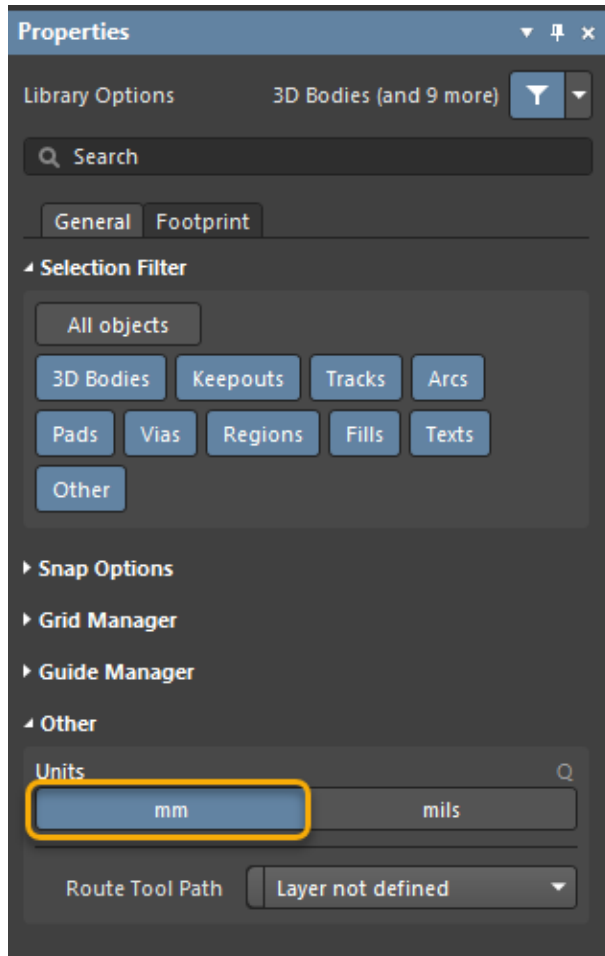


Figure 24. Measurement Units

37. Next change the grid to 0.5mm,
  - a) Select **Ctrl+G** keyboard combination to open the *Cartesian Grid Editor*.
  - b) In the *Cartesian Grid Editor* pop up window type in 0.5mm for **Step X**.
  - c) Change the *Display* option to **Lines**, see Figure 25.

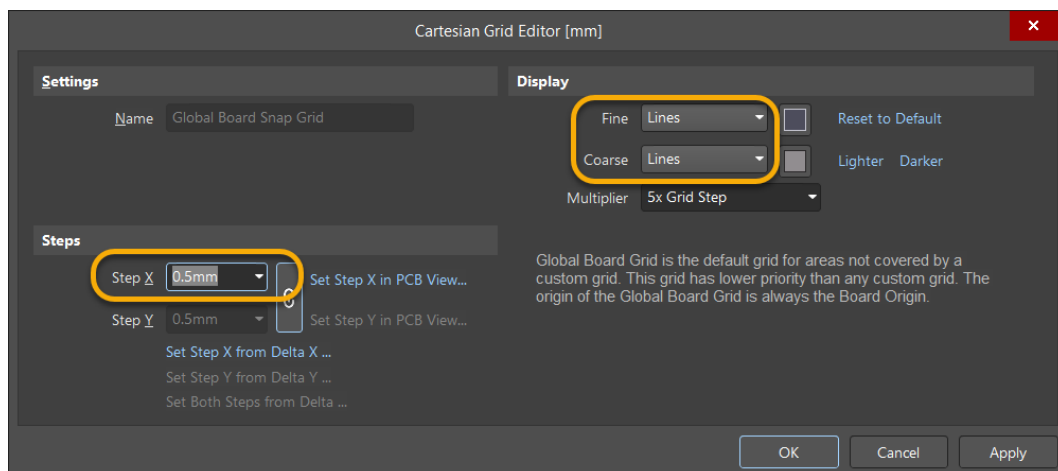



Figure 25. Grid Editor

### 1.7.3 Adding Pads

38. Select and place a **Pad** from the **Active Bar** , or alternatively use the menu **Place » Pad**, or use the **P » P** shortcut keys.
39. Hit the **Tab** key to open its *Properties* panel before placing it. You can use Figure 26 below as a reference for the following steps.
  - a) Change its *Designator* to 1 and ensure the **Top Layer** is selected in the *Layer* option.
  - b) In the *Pad Stack* region, change the *Shape* to **Rectangular**
  - c) Change the (X/Y) pad dimensions to X: 5.5mm Y: 2mm. You may need to scroll down the *Properties* panel to enter the pad dimensions, as shown in Figure 26.

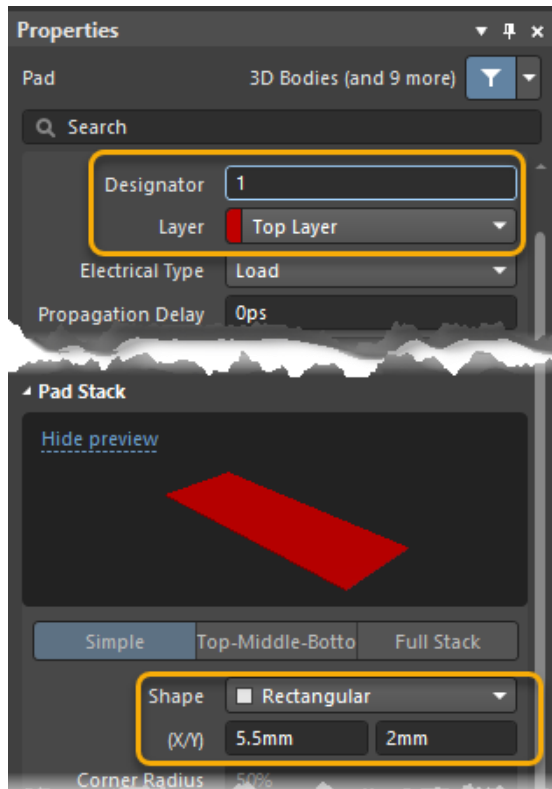




Figure 26. Pad Properties

- d) Hit **Enter** or select the pause symbol  to continue placement.
40. Left-click to place Pad 1 on the origin .
41. A second pad of the same shape is now attached to your cursor. The designator is also auto-incremented to 2.



If it is not currently visible, turn on the *Heads Up Display (HUD)* by hitting **Shift + H**.

42. Slowly move the position of Pad 2 until the *HUD* displays a
  - a) *dx* value of **9.5mm**
  - b) and a *dy* value of **0mm** as shown in Figure 27.
  - c) Left-click to place Pad 2 at that position.

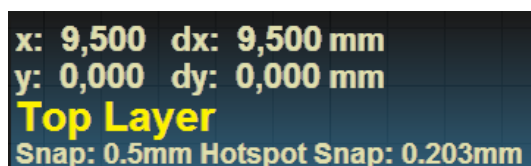


Figure 27. Heads Up Display showing incremental cursor position

### 1.7.4 Setting the Component Origin

There are three options to set an origin for a component: **Pin 1**, **Center** or **Location** as shown in Figure 28.

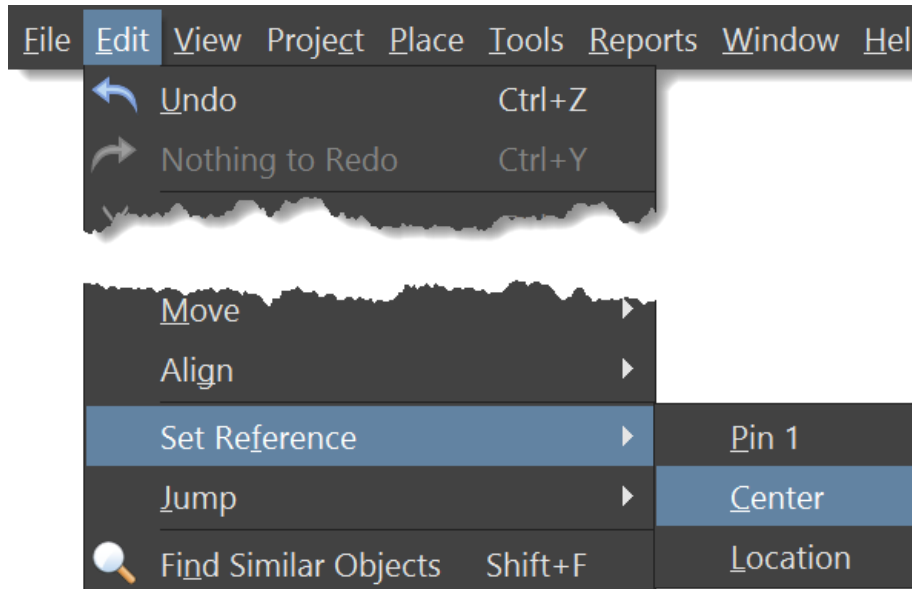


Figure 28. Setting Component Origin from the Edit menu

43. Assign the origin to the center of the existing pads by going to **Edit » Set Reference » Center**. The resulting circle and diagonal lines indicates the component origin as shown in Figure 29.

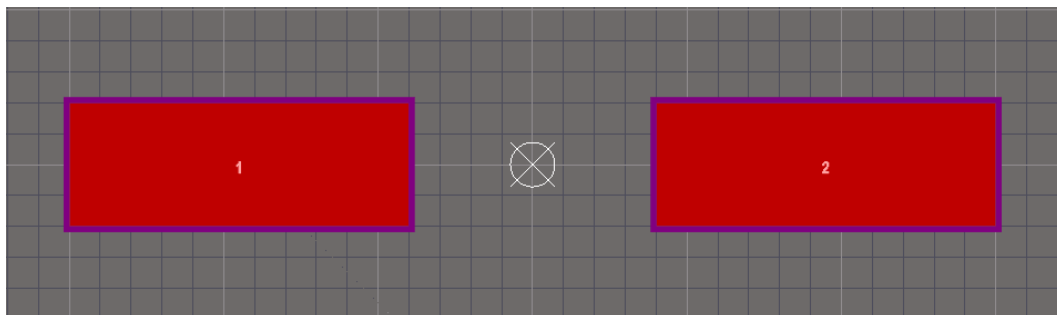


Figure 29. Component Origin set to Center

### 1.7.5 3D Crystal Body

In this section we will add a 3D STEP model for the crystal package for this component. First, we need to create the appropriate 3D mechanical layer.

44. Enable the *View Configuration* panel by selecting it from the **Panel** button **Panels** at the bottom right of the Altium screen, or by hitting the **L** key whilst the editing area is active.
45. **Right click** anywhere in the *View Configuration* panel and select **Add Component Layer Pair**, see Figure 30

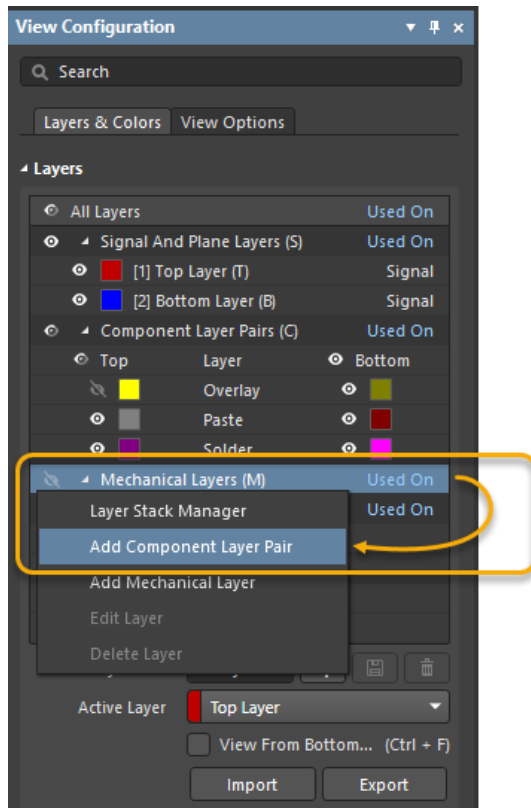


Figure 30. Add a Mechanical Layer Pair

46. Enter the information shown in Figure 31. Change *Layer Pair Name* to **3D Models**, and from the *Layer Type* drop down list select **3D Body**.

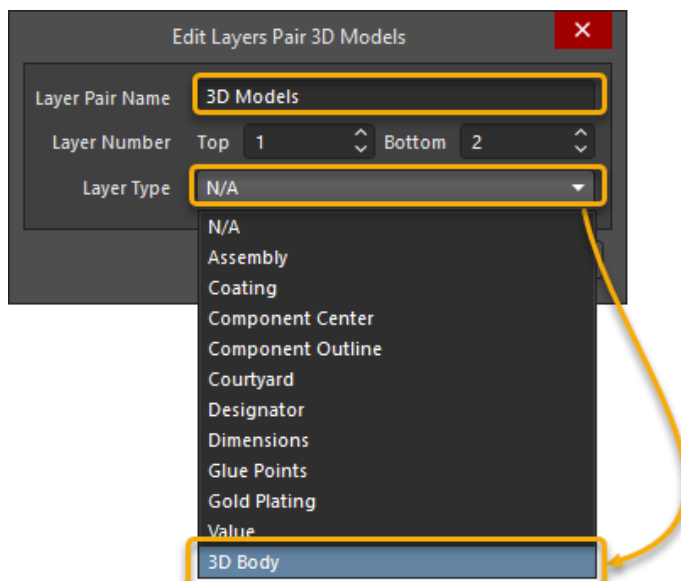


Figure 31. Configure Mechanical Layer Pairs

47. The new layer pair will be added to the *Component Layer Pair* section of the *View Configuration* panel, Figure 32.

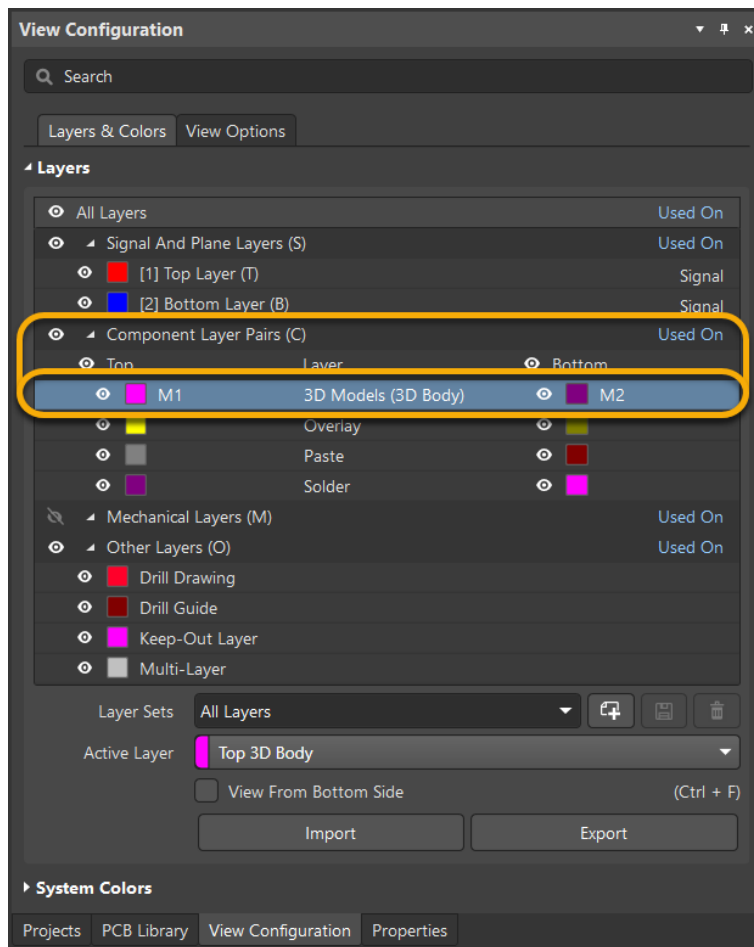


Figure 32. Layer Pairs

48. Go to **Place » 3D Body (P, O)** from the menu as shown in Figure 33.

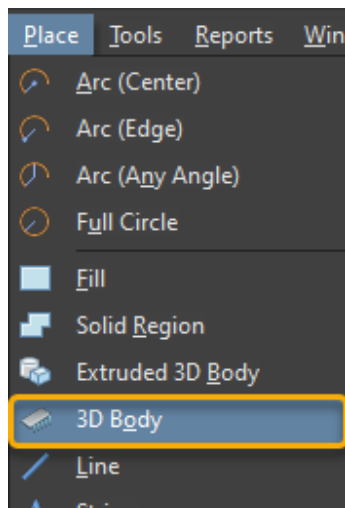


Figure 33. Place 3D Body Command



Warning: The Mechanical layer we use during this exercise are just for training purposes. Later check your company guidelines for Altium, to choose the proper Mechanical Layers for component creation.

49. In the *Choose Model* dialog navigate to the training folder `..\PCB Library` and select the STEP Model `HC-49_SMD.STEP`.



In the 2D environment, all 3D models, are seen as shapes with diagonal lines.

50. Before placing the 3D model, if not already the active layer, navigate to the **Top 3D Model** layer by either using the **+** or **-** key from the **Num** pad on your keyboard.
51. With the STEP model attached to your mouse, place the 3D Body in the center of your Footprint, as seen in *Figure 34*

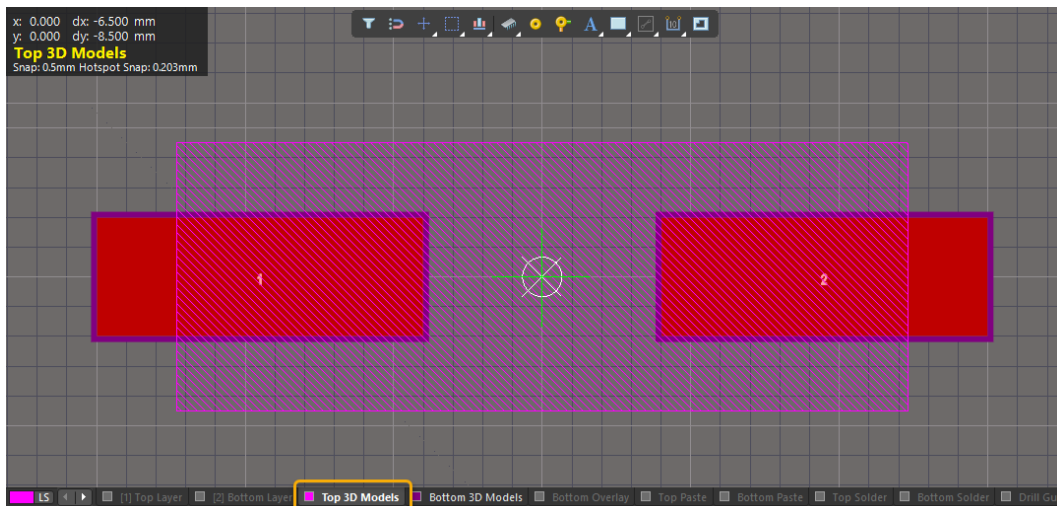


Figure 34. 3D Body Dialog in 2D Mode

52. With the 3D model still selected open the *Properties* panel and if activated, deactivate the option **Override Color**, as seen in *Figure 35*

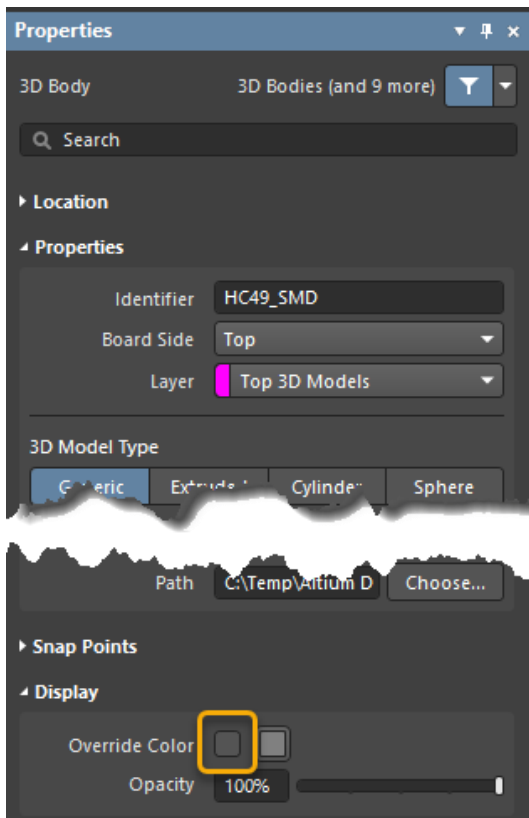


Figure 35. 3D Model Properties

Hit the **3** key to view the body in 3D mode as shown in Figure 36.

53. Hit the **8** key to see the 3D model as isometric view.

54. Hit the **2** key to go back into 2D mode.

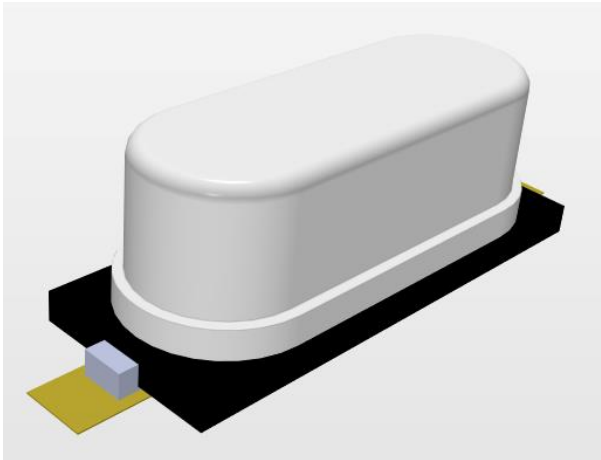


Figure 36. STEP 3D Crystal Body and Footprint

### 1.7.6 Adding Top Overlay Outline

55. Switch to the silkscreen layer by clicking on the **Top Overlay** layer tab at the bottom of the workspace.

56. Right click on the **Graphical Place** menu in the **Active Bar**, and select line, as shown in Figure 37. Alternatively, to place a line go to the **Place » Line (P,L)** command.

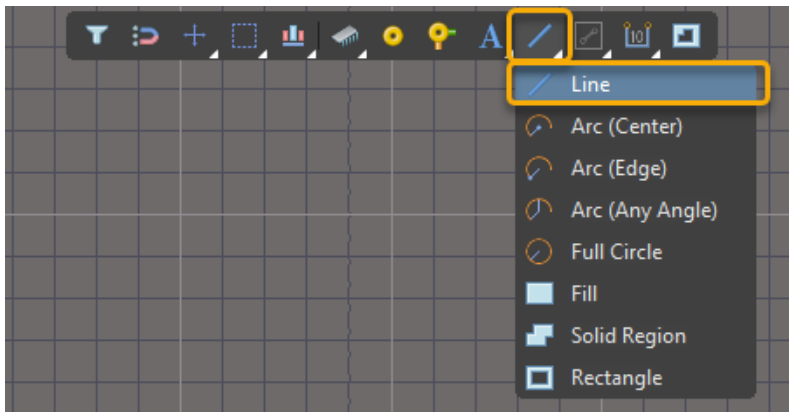



Figure 37. Placing a Line

57. Next hit the **Tab** key to open the *Properties* panel, set the Line Width to 0.5mm, see Figure 38. Press **Enter** or select the pause symbol  to continue the line placement.

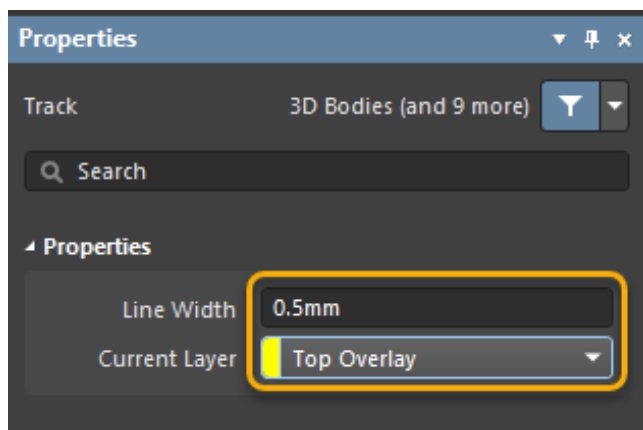


Figure 38. Line Properties



58. Now that we have a component origin, we can use the HUD to position the component outline precisely. Move the cursor to the component origin at the center of the component and note that the HUD reports an x and y value of 0.
59. The command **Edit » Jump » New Location (J-L)** with x:0 and y:0 or **...Jump » Reference (Ctrl+END)** could help to find the correct position.
60. Using the HUD to guide you, move the cursor to location x: -8.500mm y: 1.000 and click to set the starting point for the line.
61. Move the cursor up and to the right to see the corner angle mode of the line.
62. Press **Shift + Spacebar** to change the shape of the corner and **Spacebar** to change the orientation until the HUD displays **Line 90 EndPlace Line** as shown in Figure 39.

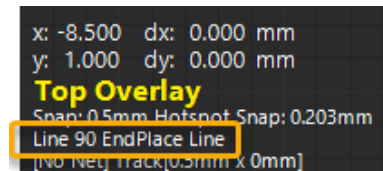


Figure 39. HUD displaying current line mode of 90 degrees

63. Move the cursor to the following positions and left click to set the vertices at each location:
  - a) (-8.5, 3.0)
  - b) (8.5, 3.0)
  - c) (8.5, 1.0)
64. Right-click to terminate the line and right-click again to terminate the command. The line should now be similar to what is shown in Figure 40.

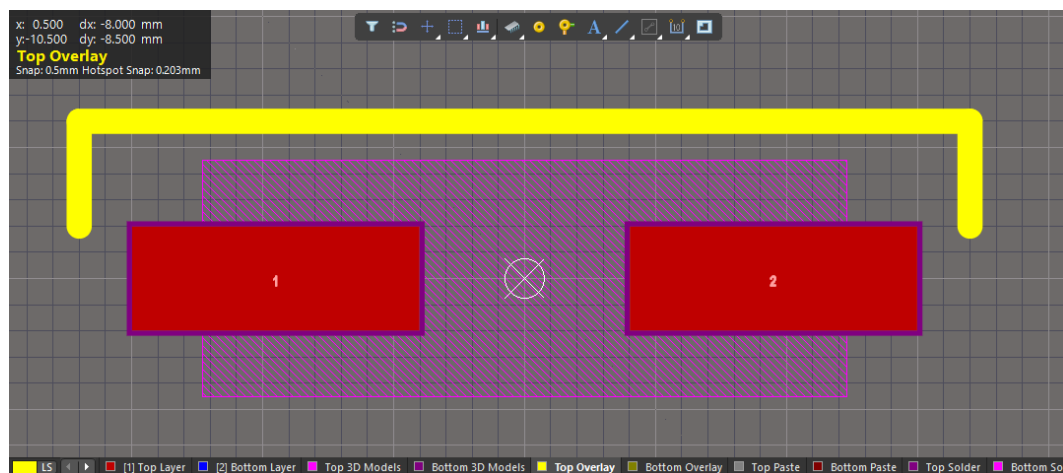


Figure 40. The top portion of the Overlay added to the component

65. Next we will copy and paste the lines from the top of the component to add them to the lower portion, below the pads. In the PCB editor, Altium Designer requires a reference point for the copied object so that it can assign a local origin.
66. Change the selection filter to **Tracks** only, see Figure 41

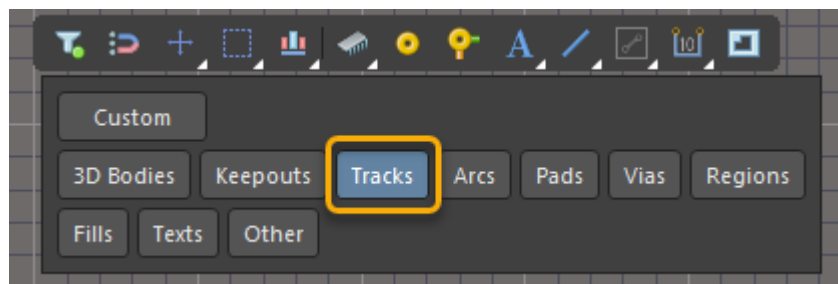


Figure 41. Filter Options

67. Drag a selection rectangle around the yellow lines.
  - a) Press **Ctrl + C** to copy it to the clipboard.
  - b) The cursor becomes a crosshair, and a message is displayed at the bottom of the screen: Select a reference point as shown in Figure 42.
  - c) Click on the Origin to set it as a reference point.

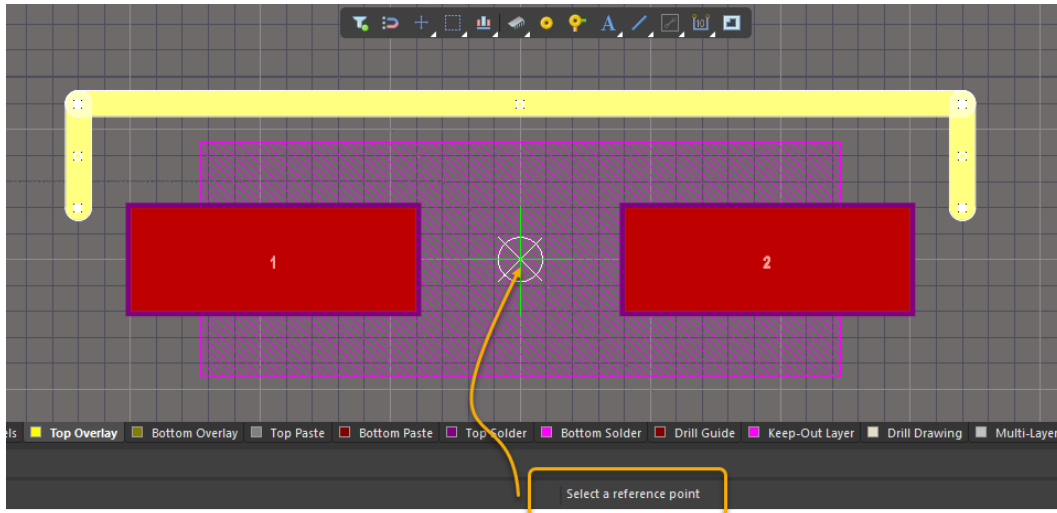


Figure 42. Copy the upper outline and set a reference point in preparation for pasting

- d) Hit **Ctrl + V** to paste a copy of the outline. Notice that the cursor is attached outside of the horizontal line, which is the reference point you just selected.
  - e) Hit the **Spacebar** twice to rotate the lines.
  - f) Then, left-click to position the cursor on the origin so that the lines are now symmetrical below the pads as shown in Figure 43.

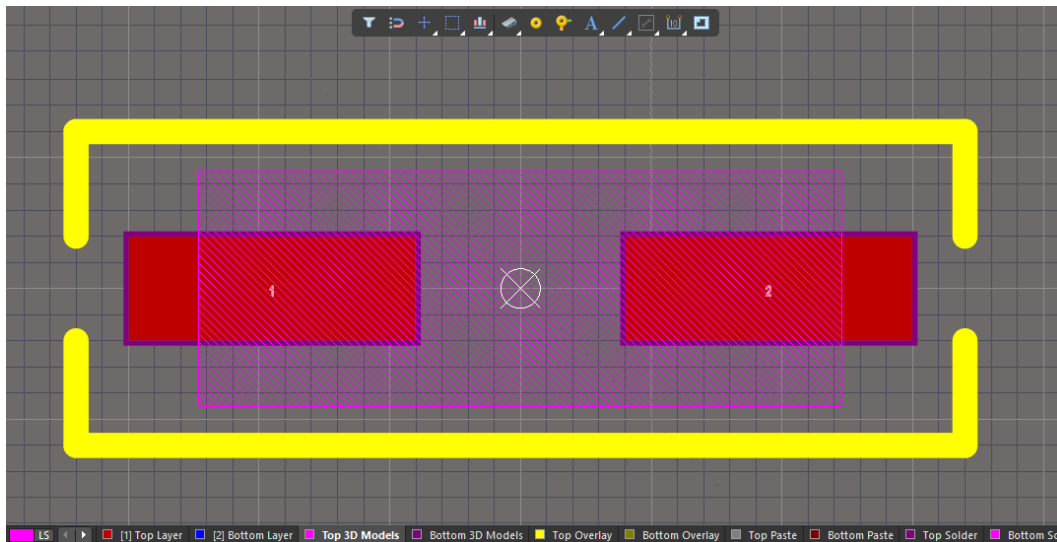


Figure 43. Copied outline

68. For soldering and debugging purposes, Pad 1 should also be visually marked. Use the **Active Bar** and right click on the **Graphics** menu and place a Fill near Pad 1, as shown in Figure 44.

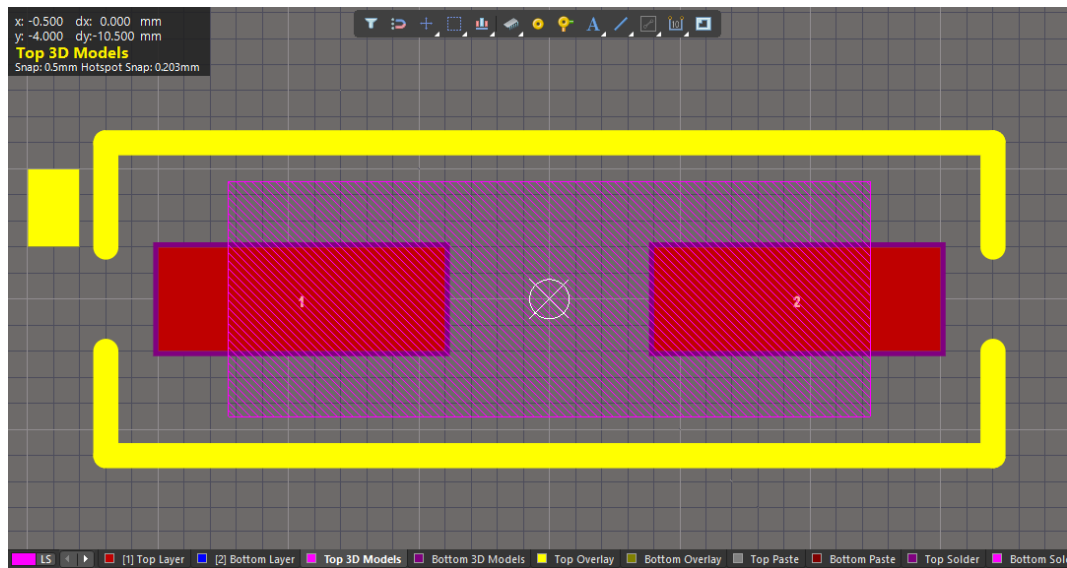



Figure 44. Footprint HC49SMD

69. Go to **File » Save**  **Save** **Ctrl+S** to save the library.
70. Return to the *Component Editor* dialog.

## 1.8 Save the component to the workspace

Now that we have finished component creation by adding parameters, symbol and footprint, it is a good time to check Pin Mapping, and save the Component to the workspace.

71. Select the **Edit Pin Mapping** command , as seen at Figure 45

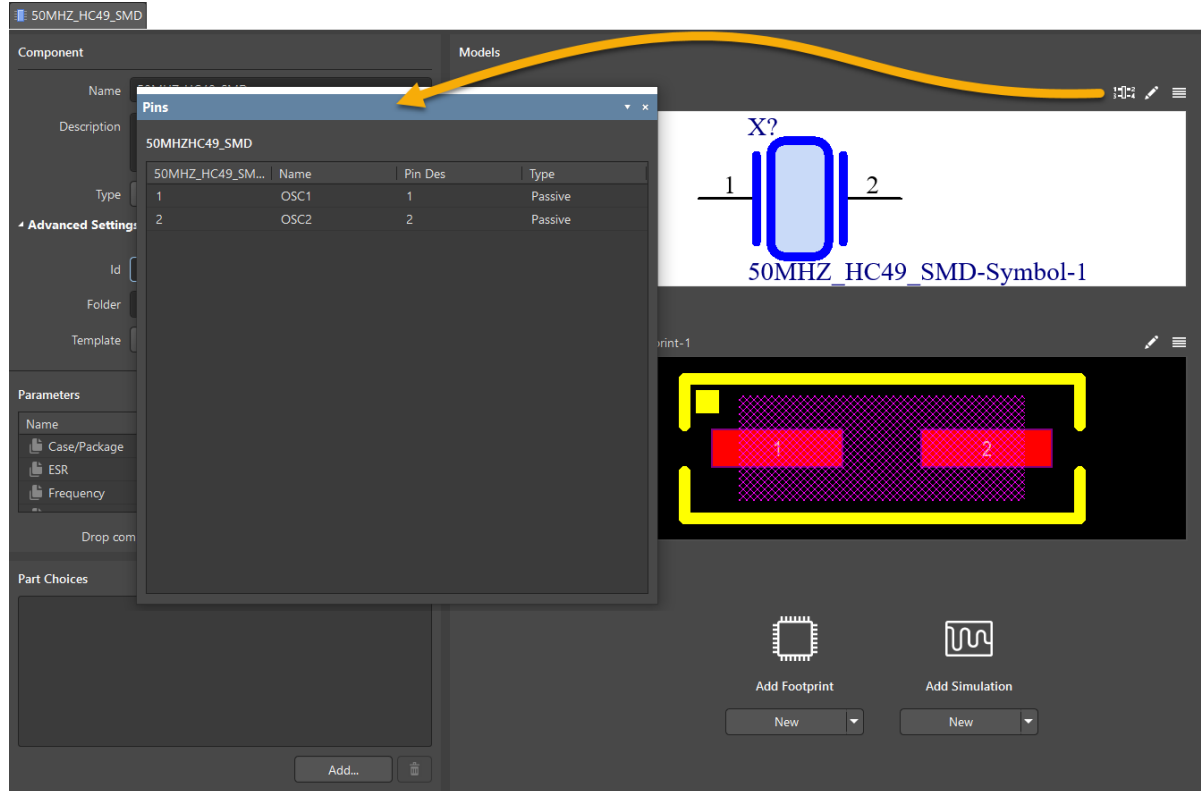



Figure 45. Pin Mapping

72. In most cases the pins and pads will match their corresponding numbers for e.g. Pin 1 to Pad 2; Pin 2 to Pad 2. Close the dialog after you checked the mapping.
73. Execute the command **File » Save to Server** or select at the *Project* panel **Save to Server**  to save your component.
74. At the *Edit revision for Item* dialog add a release note, e.g. Initial release for Crystal 50MHz. Select **OK** to continue.
75. If no error is detected the new crystal component is saved to the workspace, the component should now be available for placement using the Components panel.

**Congratulations on completing the Module**

Component Creation for A365 Workspace

**from the**

**Altium Designer Essential Course with A365**

**Thank you for choosing Altium Designer**