



Altium Designer

Essentials Training with Altium 365

Component Creation for A365 Workspace

Altium
TRAINING





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.

TRADEMARKS

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

Component Creation for A365 Workspace	3
1 Purpose	3
2 Shortcuts	3
3 Preparation	3
4 Create a New Component	4
4.1 Creating a New Component	4
5 Component Editor	7
5.1 Add Part Information: Parameters, Symbol, and Footprint	7
6 Creating a New Schematic Symbol	10
6.1 Creating a New Schematic Symbol	10
6.2 Preparing the Design Space	12
6.3 Defining the Pins	12
6.4 Adding Symbol Body	15
7 Creating a New PCB Footprint	18
7.1 Footprint Name and Definition	19
7.2 Preparation for Footprint Creation	20
7.3 Adding Pads	22
7.4 Setting the Component Origin	23
7.5 3D Crystal Body	24
7.6 Adding Top Overlay Outline	29
8 Save the Component to the Workspace	32





Component Creation for A365 Workspace

1 Purpose

In this exercise, you will learn how to create components in Altium 365 Workspace. 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 to do this: the simplest is Integrated libraries, then there is the Database libraries, and the best and most advanced way is the Altium 365 Workspace.

Components are represented as schematic symbols during design capture and as PCB footprints for placement during the board design. It helps tremendously with 3D visualization, clearance checking and export to mechanical CAD if the footprints have 3D models associated with them. 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.

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.

2 Shortcuts

Shortcuts used when working with Component Creation for A365 Workspace

Ctrl+S	Save Document
P» [Command]	Place (SCH or PCB)
G	Grid (SCH or PCB)
Spacebar	Rotate (SCH or PCB)
X, Y	Flip (SCH)

3 Preparation

1. Close all existing projects and documents.





4 Create a New Component

4.1 Creating a New Component

We will show you two methods of starting the process of creating a new component:

- a) **File » New » Component**
 - b) **File » New » Library**
2. For the first method, select the **File » New » Component...** command from the main menu, see Figure 1 below. A new dialog *Create new component* will open. The next action is described in Step 0.

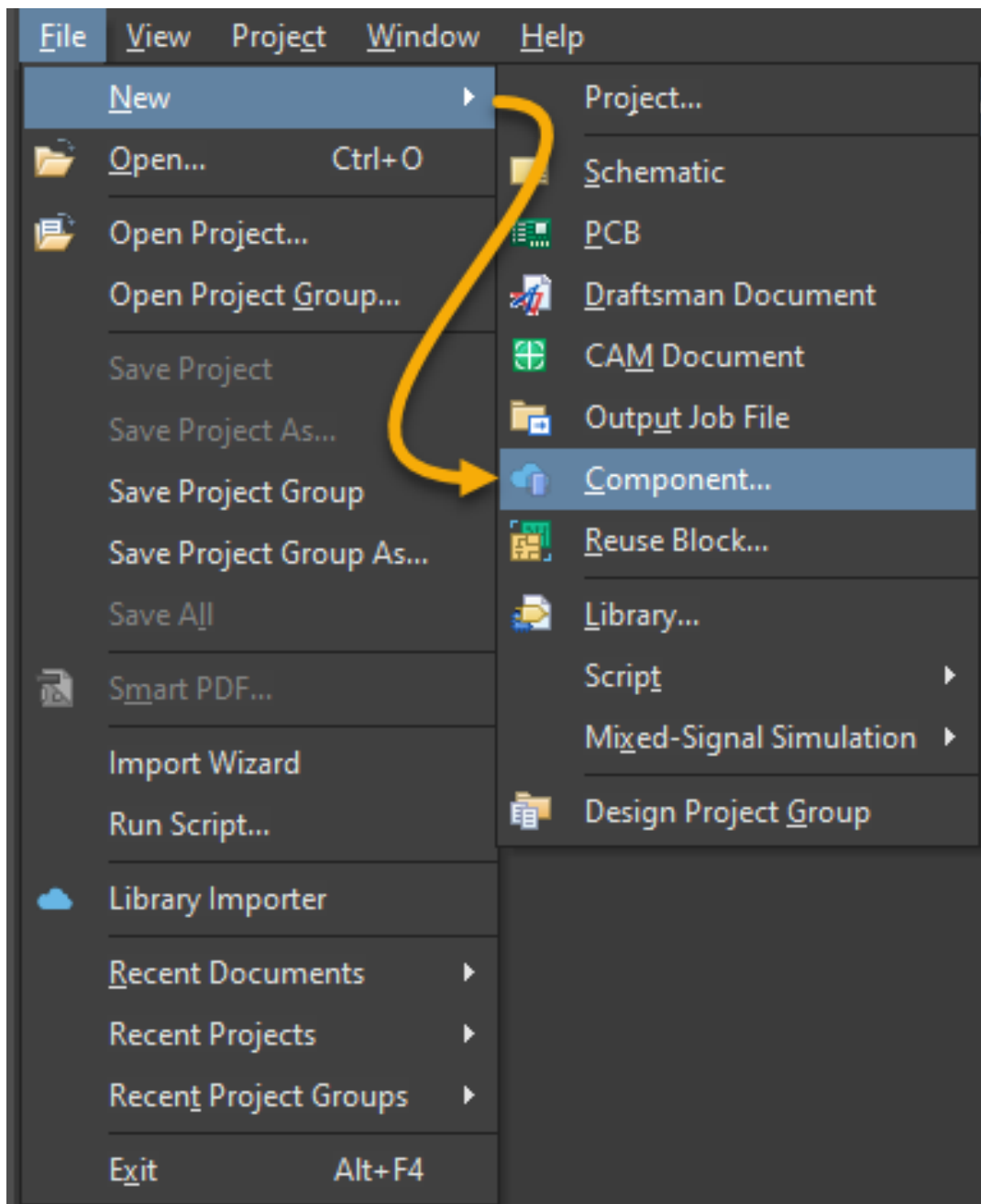


Figure 1. New Component Creation





- For the second method, select **File » New » Library...** from the main menus. In the **New Library** dialog that opens, select **Workspace** as the Library Type and then **Create Library Content » Component...** from the drop-down menu on the right, as shown in Figure 2 below. A new dialog **Create new component** will open.

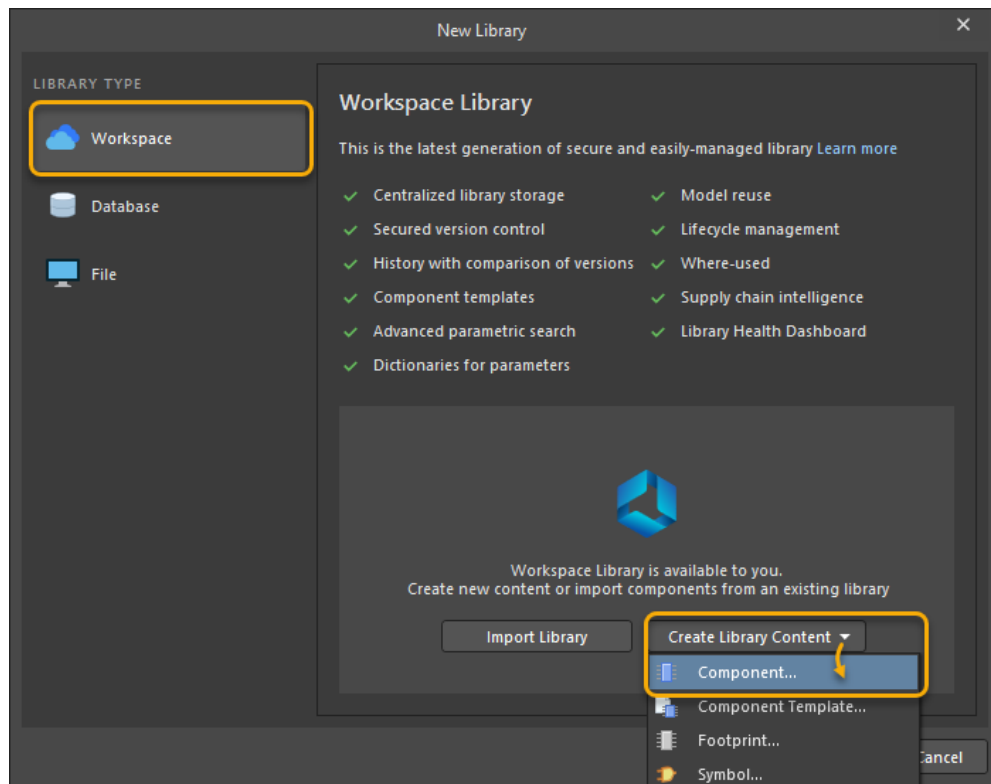
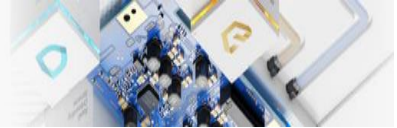


Figure 2. New Library - Create Component in Workspace





- The next steps are the same for both methods. Choose a component type from the list in the Create new component dialog, as shown in Figure 3 below. If there is a component template linked to the selected component type, this template will be used to predefine the component parameters.

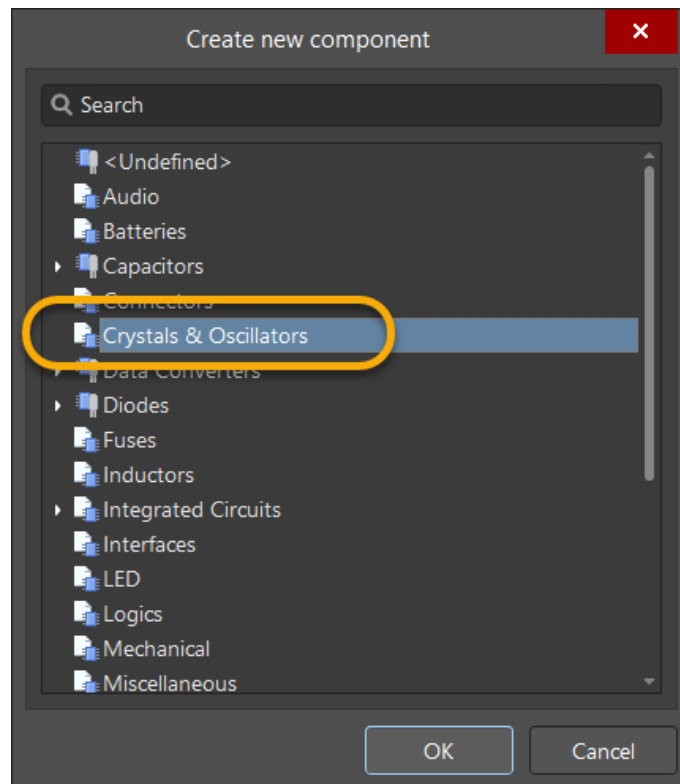


Figure 3. Component Type





5 Component Editor

5.1 Add Part Information: Parameters, Symbol, and Footprint

5. After clicking **OK**, the *Component Editor* will open, as mentioned earlier.
6. If a component template is linked to the component type, the information will be shown in the *Parameters* section. If the template has predefined symbols and footprints, these will also show here. See Figure 4 below.

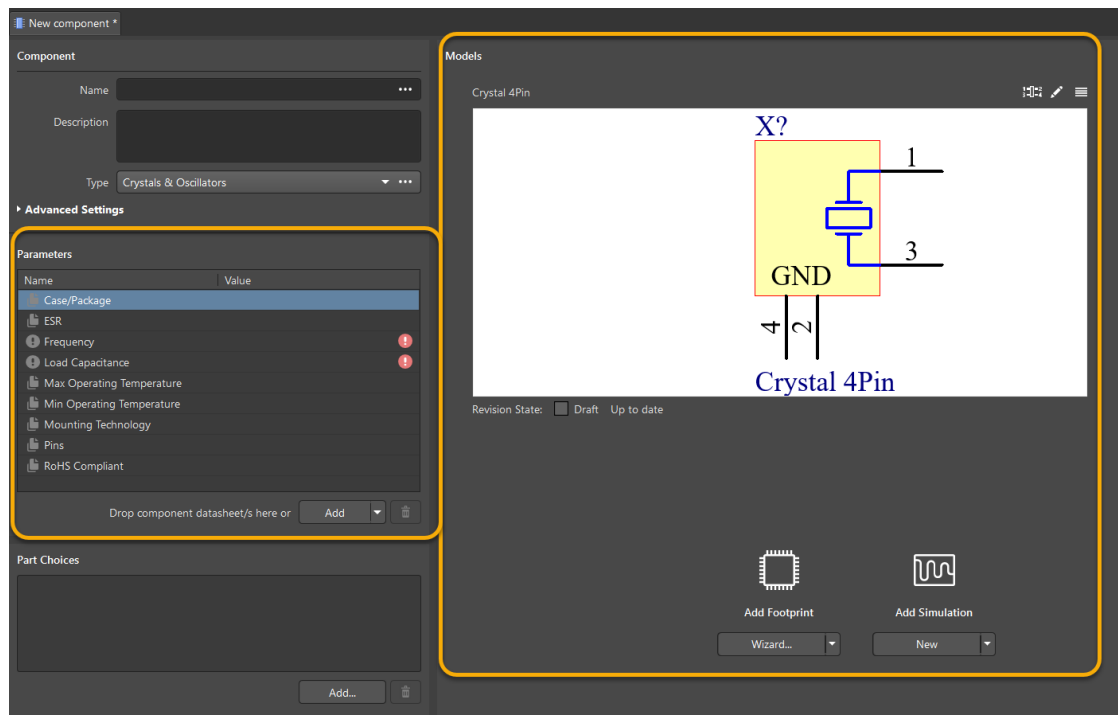


Figure 4. Component Editor

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.





7. At the top of the *Component* section (see Figure 5 below), add the following information:
 - a) **Name:** 50MHZ_HC49_SMD
 - b) **Description:** 50MHz 20pF 20ppm SMD

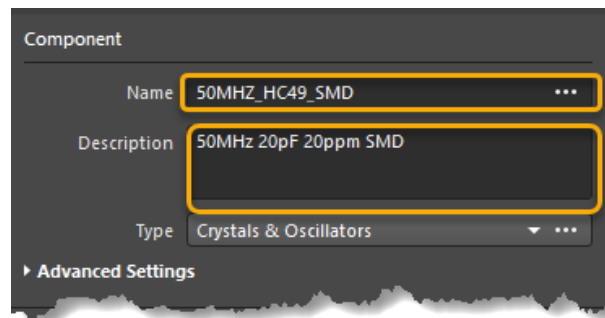


Figure 5. Component Name and Description

8. Under the *Advanced* section (see Figure 6 below), add your training folder.

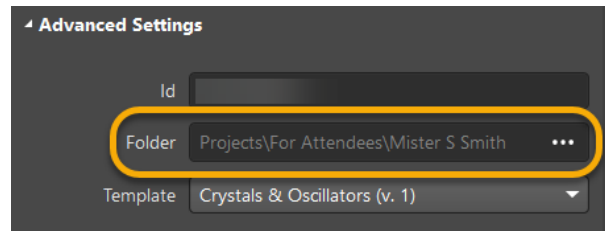


Figure 6. Folder for Component

9. In the *Parameters* section (see Figure 7 below), add the following values:
 - a) **Frequency:** 50MHz
 - b) **Load Capacitance:** 20pF

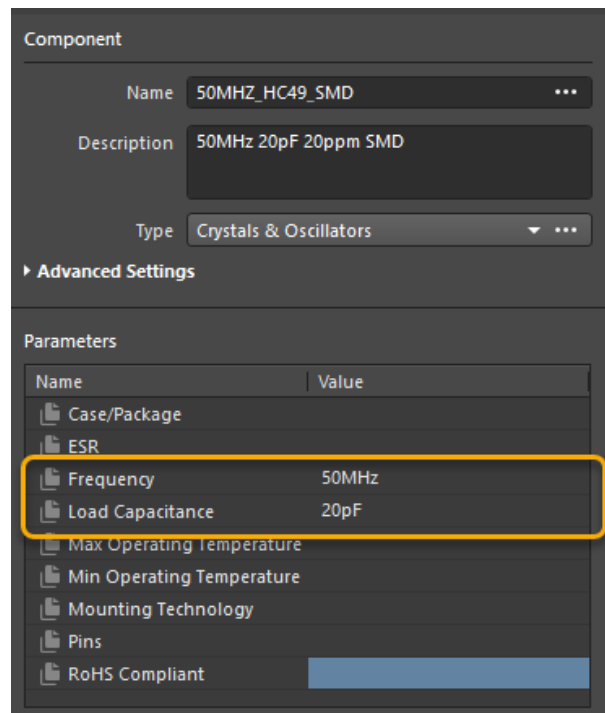


Figure 7. Component Parameters





10. Next, using Figure 8 as reference, add a new parameter and its value by selecting the button **Add » Parameter:**

a) **Frequency Tolerance:** 20ppm

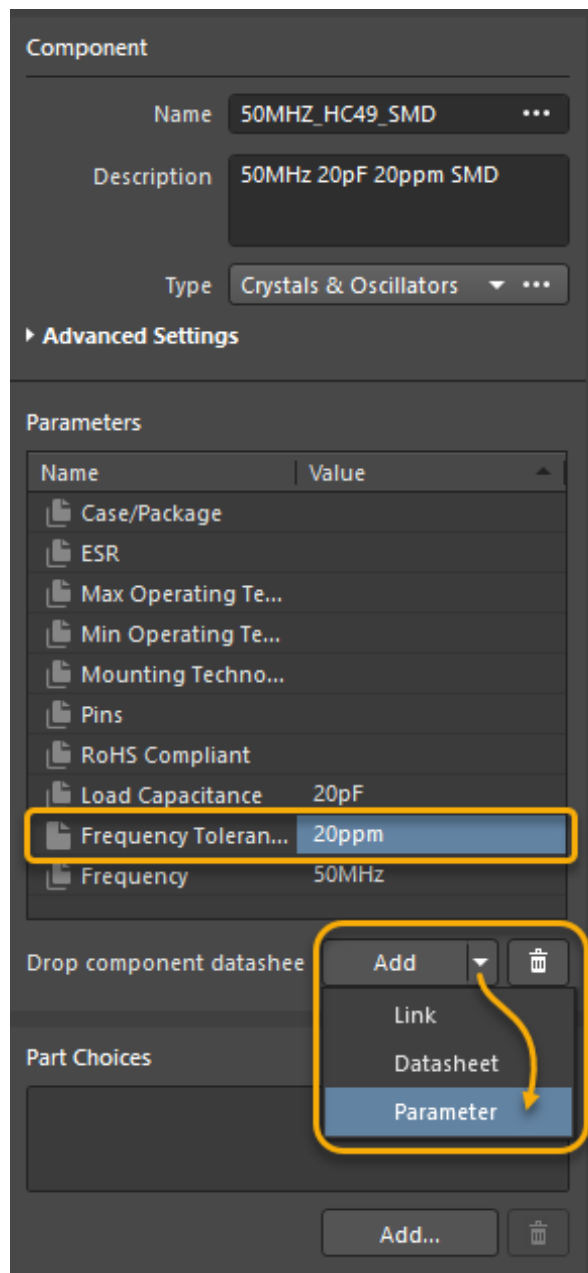




Figure 8. Add a new parameter

Hint: 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.

11. Feel free to save the modifications you have done so far  Save  Ctrl+S .






6 Creating a New Schematic Symbol

6.1 Creating a New Schematic Symbol

In this section you will learn how to create a new schematic symbol. The template has a default symbol with 4 pins, but the crystal for the training has only 2 pins, so we will remove the existing symbol and create a new symbol.

12. From the existing 4-pin crystal, right-click the four-bar menu button  and select **Remove**. See Figure 9 below.

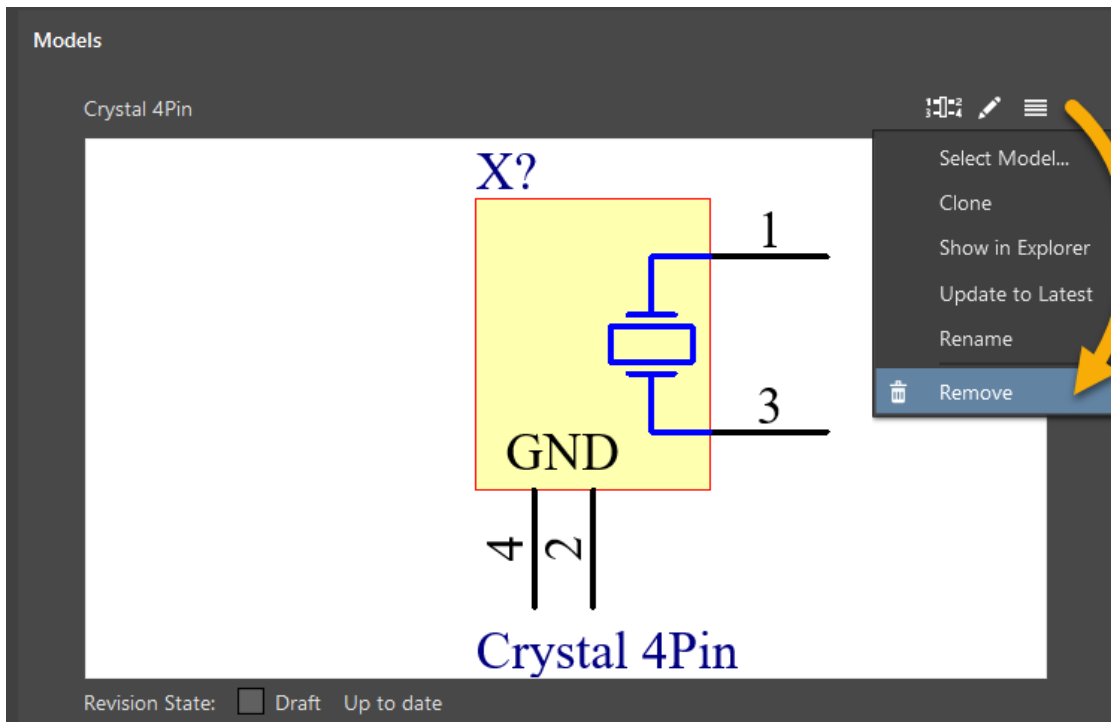


Figure 9. Remove existing 4-pin crystal symbol

13. Select **New** from the drop-down under **Add Symbol** in the Models section. See Figure 10 below.

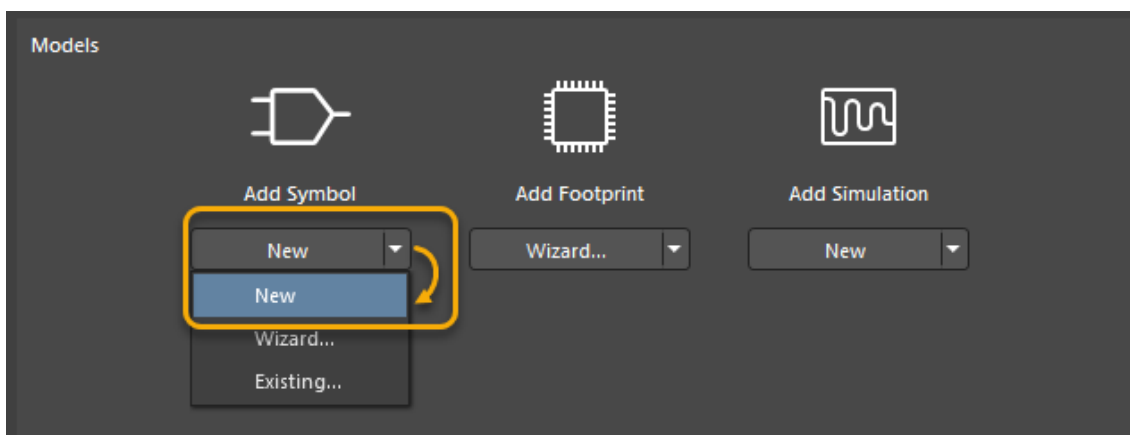


Figure 10. Creating New Symbol





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

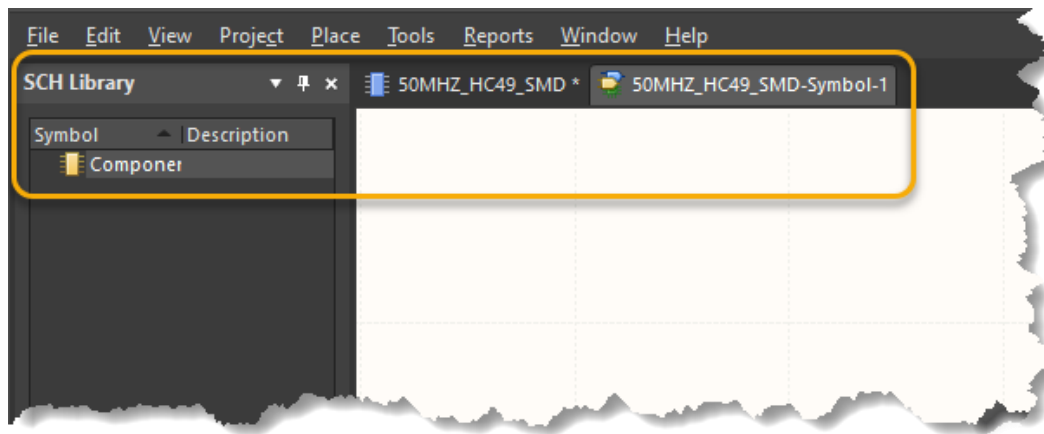


Figure 11. New Schematic Symbol Document

15. In the *Properties* panel, enter the following information shown in Figure 12 below.

Note: Add your name to the name of the crystal to avoid duplicate entries.

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

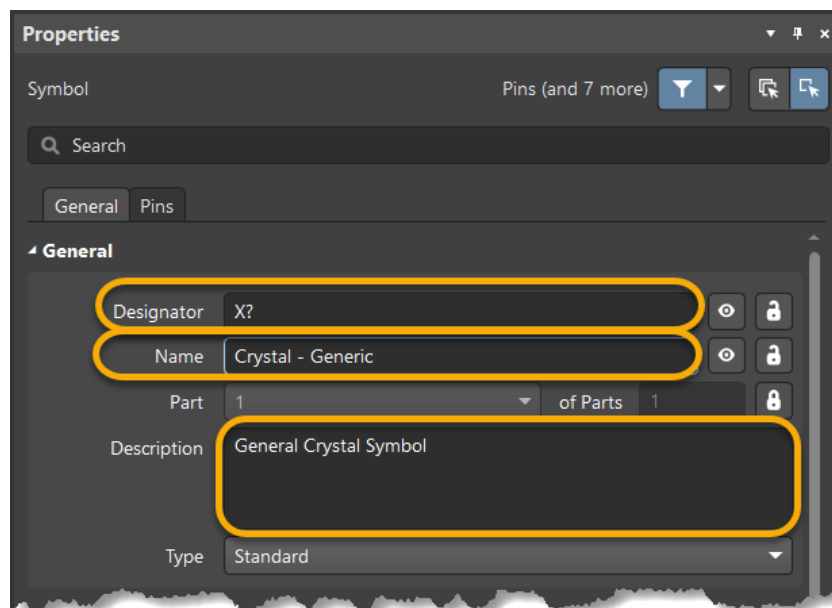


Figure 12. Symbol Properties





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.

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

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

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.

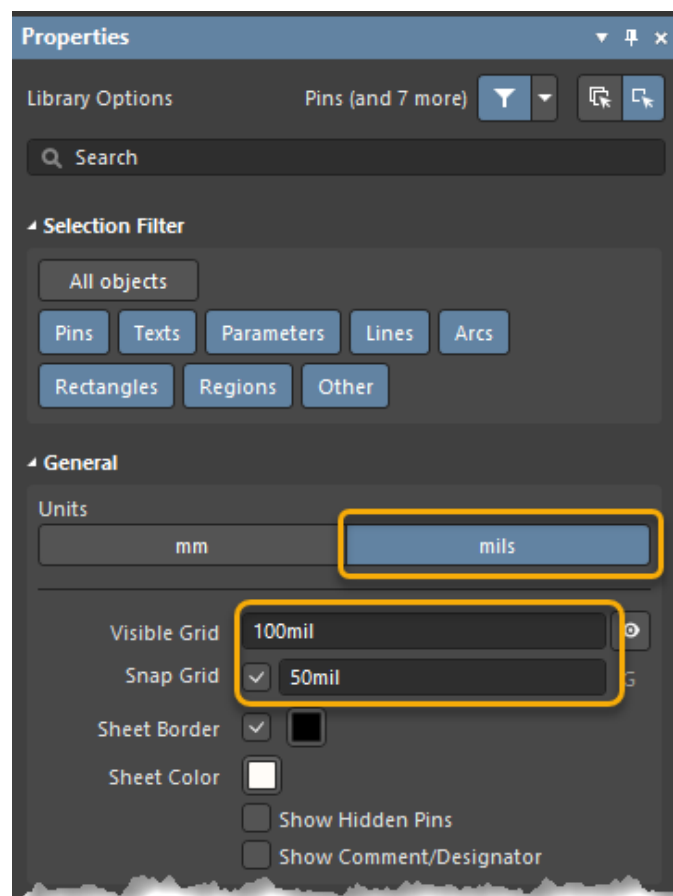


Figure 13. Grid Properties





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

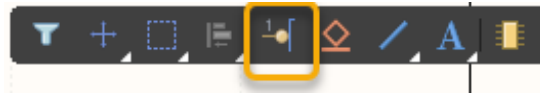


Figure 14. Placing Pin

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

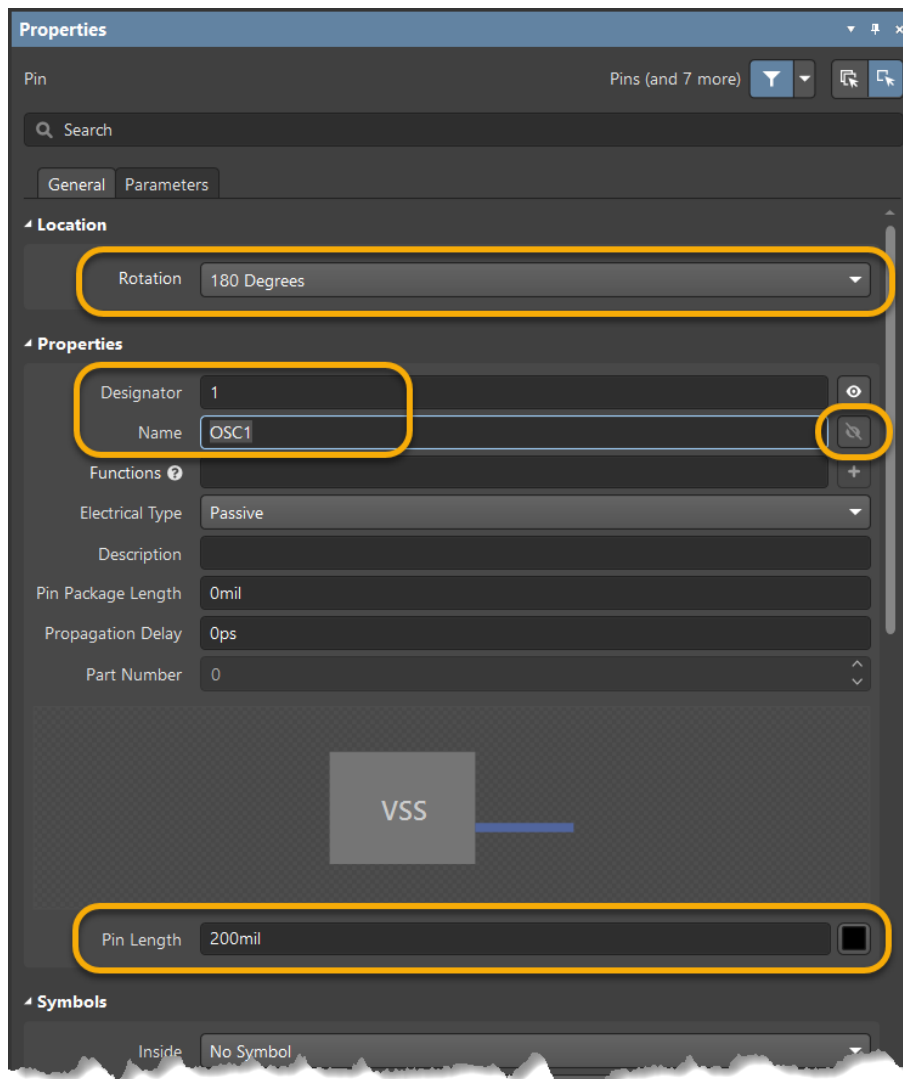


Figure 15. Pin Properties

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





19. Notice that one end of the pin has a small "X" as shown in Figure 16. This X, or crosshair, defines its electrical connection or hotspot.
20. Use the **X** key to flip the pin horizontally or use the **Spacebar** to rotate it, if needed.

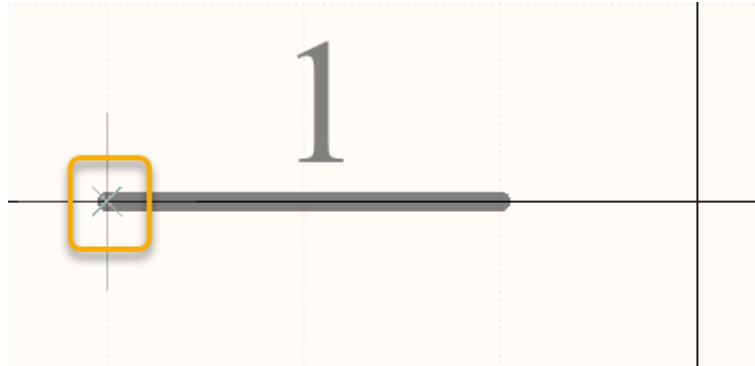


Figure 16. Pin Electrical Hotspot

21. Using Figure 17 as reference:
 - a) Place pin 1 to the left of the origin, at the coordinates X= (-350mm), Y= (0mil).
 - b) After placing Pin 1, another pin will appear on the cursor with the pin name and designator already incremented to 2.
 - c) 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 17.
 - d) **Right-click** or hit **ESC** to exit pin placement mode.

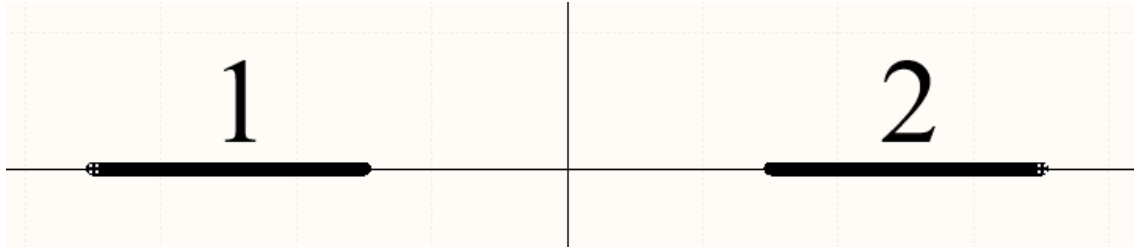



Figure 17. Position pin 2 to the right of pin 1





6.4 Adding Symbol Body

22. To place the two vertical lines on the end of each pin, use the **Active Bar** or command **Place » Line**.
23. Press the **Tab** key during placement to open the *Properties* panel.
 - a) Set the *Line Width* to **Medium** and choose a color of your choice.
 - b) Press **Enter** or the Pause icon  to continue the placement.
24. Draw the 2 lines similar to what is shown in Figure 18.
25. Right-click to terminate each line segment.
26. Right-click a second time when you are finished to terminate the command.

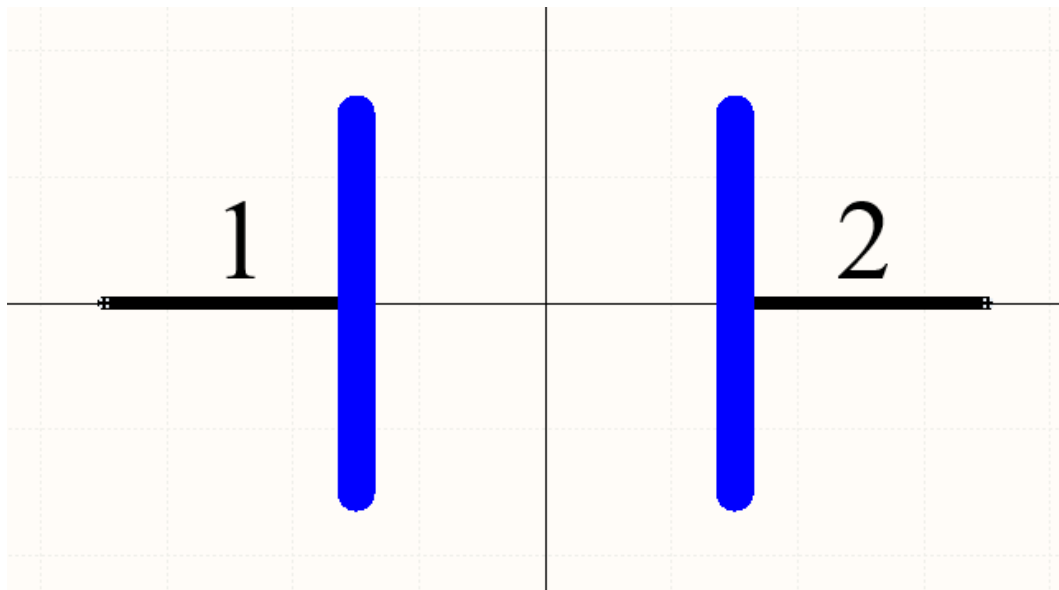



Figure 18. Add Lines to Crystal Symbol





27. 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 19.
28. Select **OK** to apply the modification as default.

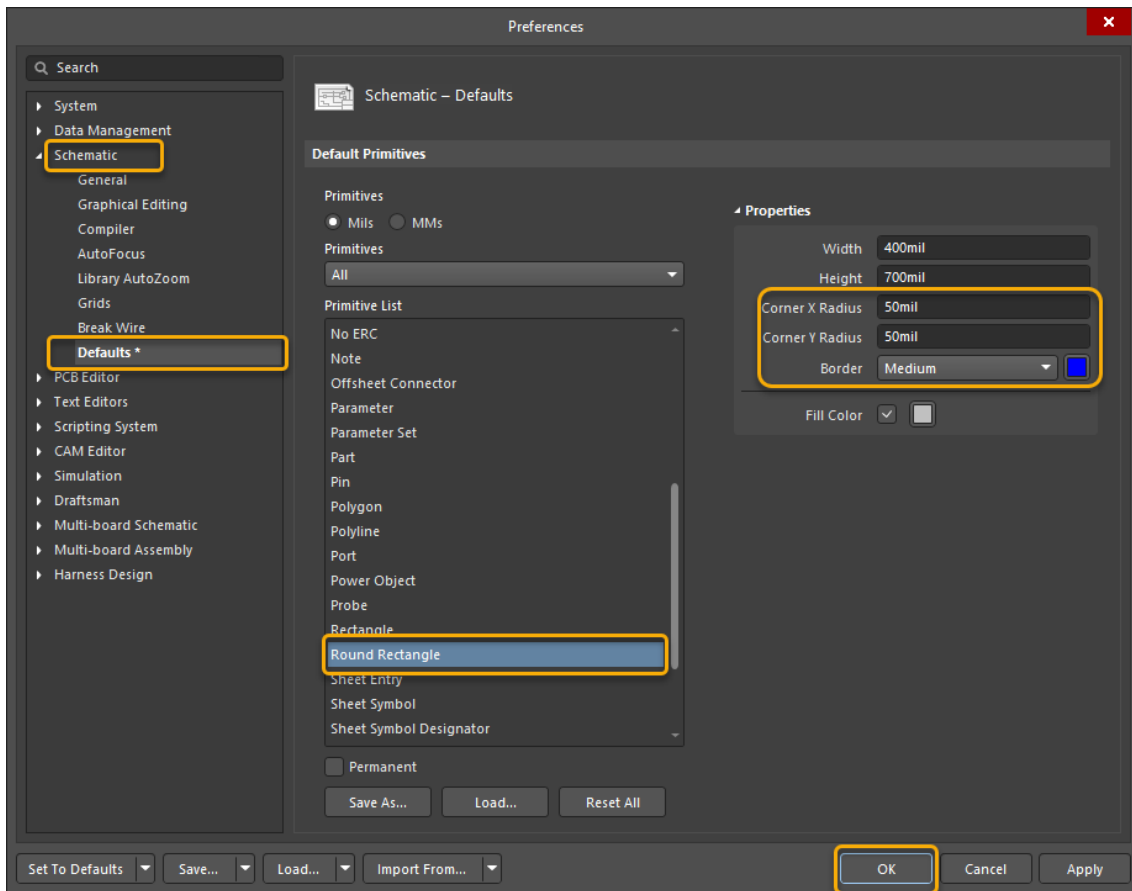


Figure 19. Rounded Rectangle Properties

29. Go to **Place » Round Rectangle** command to add the central body for the crystal.
30. Hit the **Tab** key to open its *Properties* panel. Change the *Border* to **Medium** and select the *Border* and *Fill* color of your choice, as shown in Figure 20 below.

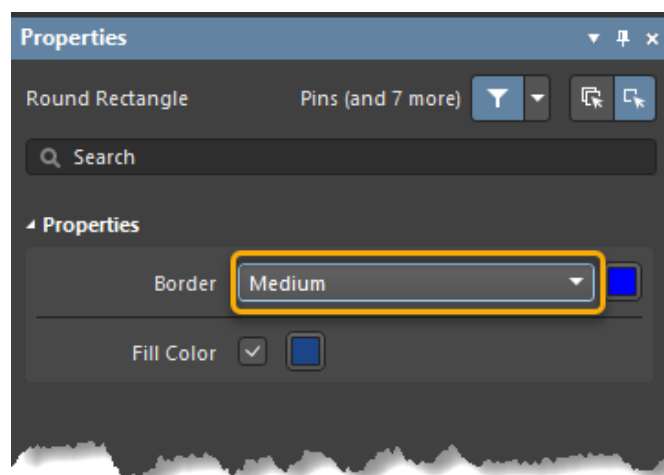



Figure 20. Rounded Rectangle Properties





31. Hit **Enter** or the Pause icon  to start placing the body.
32. Draw the rectangle in the center of the symbol, as shown in Figure 21, by clicking at the top left corner, then the bottom right corner to complete the shape. Feel free to make adjustments as needed.
33. Right-click to terminate the command.

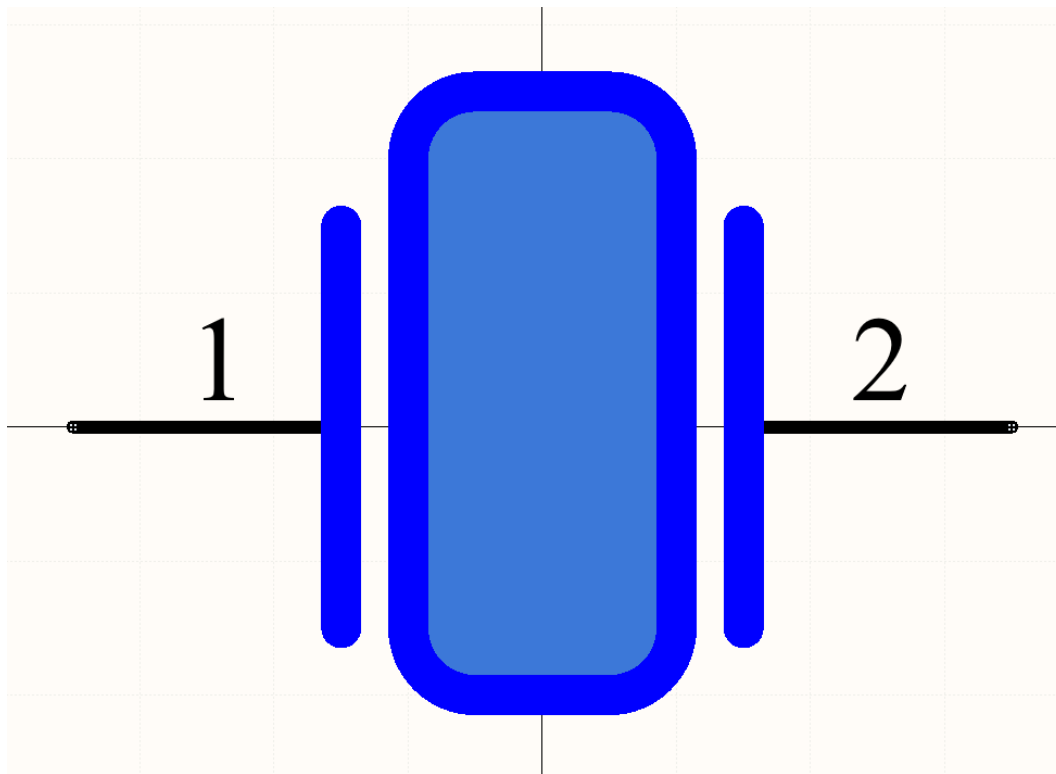




Figure 21. Crystal with Rectangle Body

34. Save the library by going to **File » Save**  **Save**  and close the symbol Editor.
35. Return to the *Component Editor* dialog.





7 Creating a New PCB Footprint

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

36. Select **New** from the drop-down under **Add Footprint** in the *Models* section. See Figure 22 below. A new footprint document will open and a tab will be added to the Document Bar – see Figure 23 below.

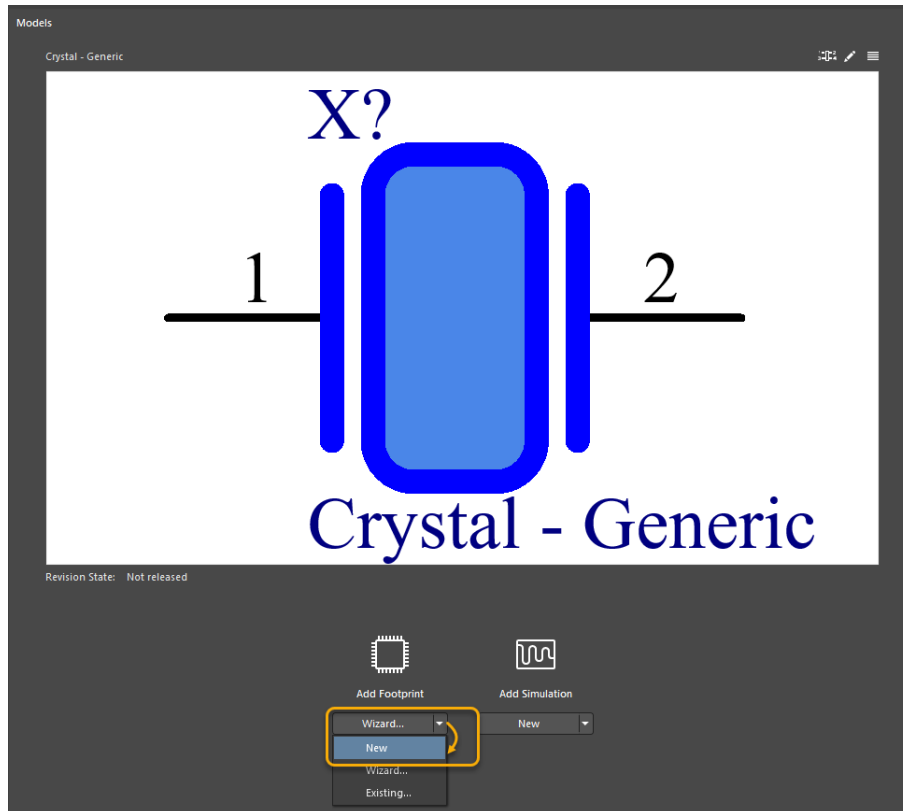


Figure 22. Creating New Footprint

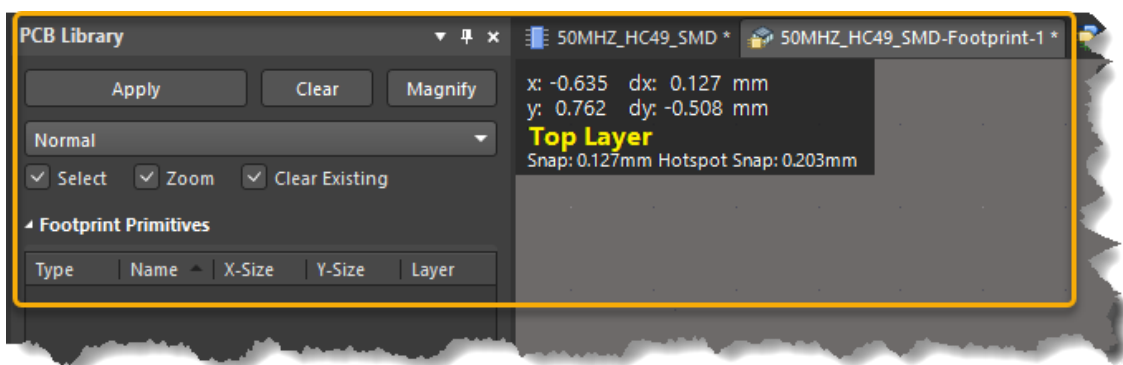


Figure 23. PCB Footprint Document





7.1 Footprint Name and Definition

37. 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

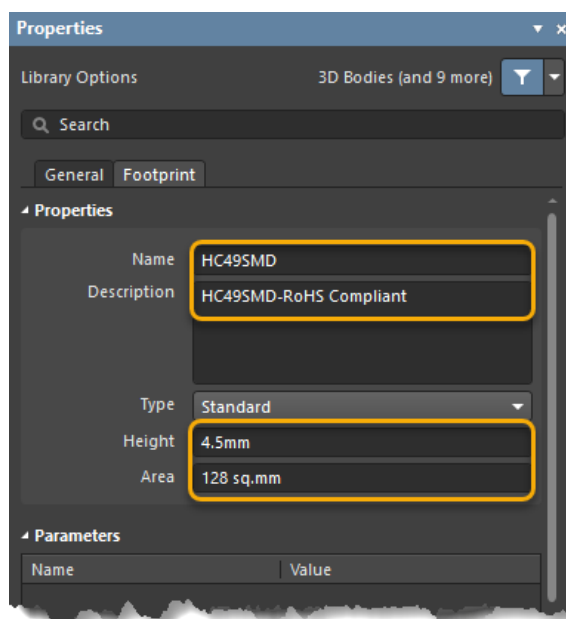


Figure 24. Footprint Name and parameters





7.2 Preparation for Footprint Creation

Below is a 2D and 3D preview of the footprint you will be creating in this exercise. See Figure 25 below.

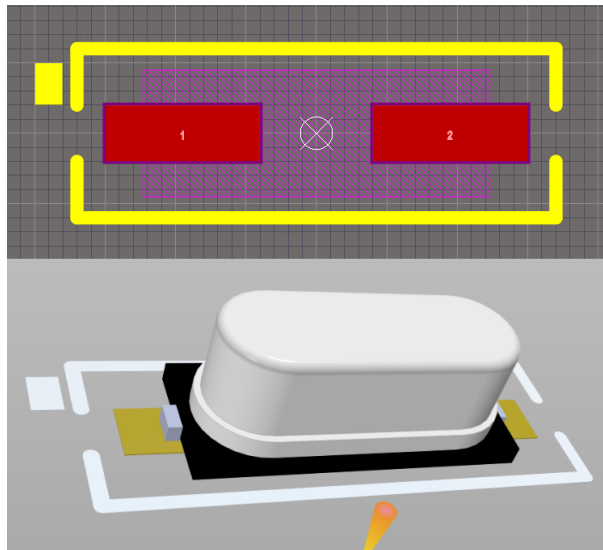


Figure 25. Footprint Preview

38. Ensure the *Properties* panel is visible. If not, select it from the Panels list **Panels** or use the **F11** function key.
39. 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**. See Figure 26 below.

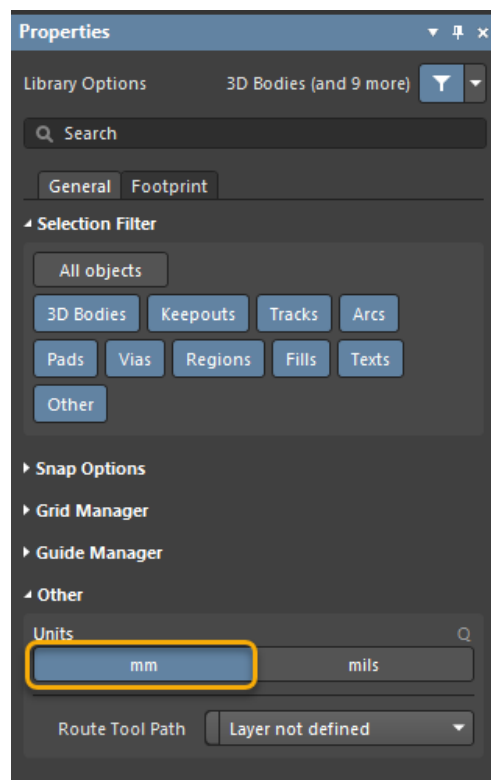


Figure 26. Measurement Units





40. Next, change the grid to 0.5mm:
- Press **CTRL+G** to open the *Cartesian Grid Editor*.
 - Enter 0.5mm as the **Step X** size.
 - Change the *Display* option to **Lines**, see Figure 27.

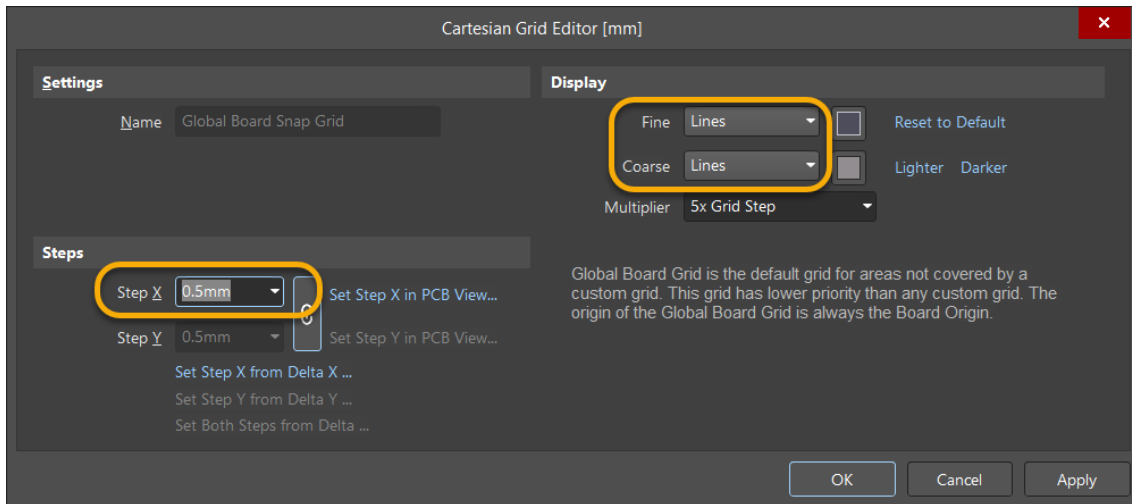



Figure 27. Grid Editor





7.3 Adding Pads

41. Select and place a **Pad** from the **Active Bar** , or alternatively use the menu **Place » Pad**, or use the **P » P** shortcut keys.
42. Hit the **Tab** key to open its *Properties* panel before placing it. You can use Figure 28 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 28.

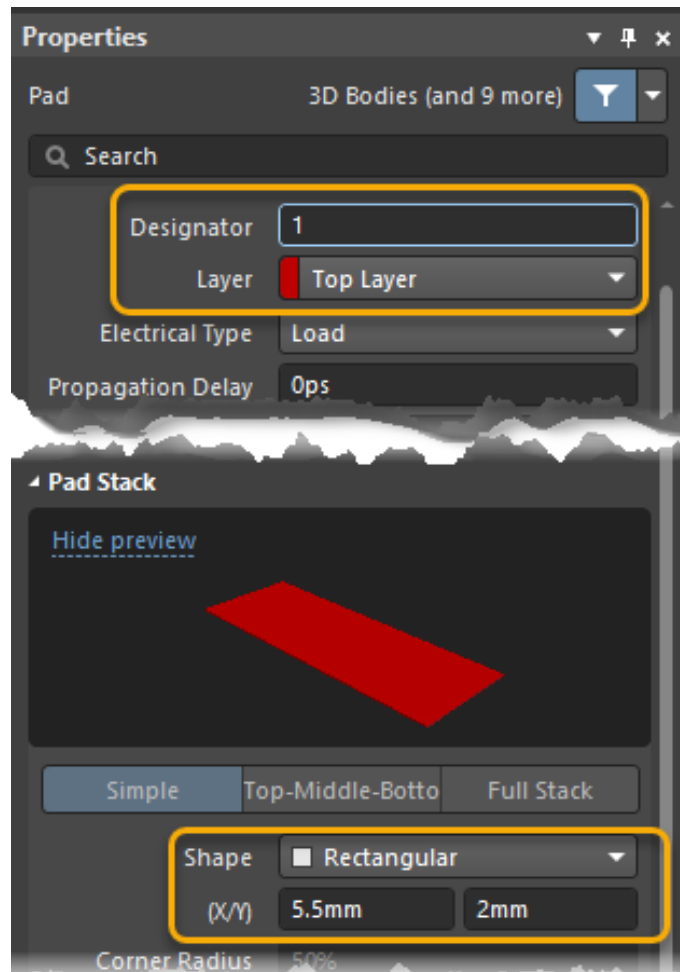




Figure 28. Pad Properties

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

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





44. Slowly move the position of Pad 2 until the *HUD* displays (Figure 29):

- a) A dx value of **9.5mm**
- b) A dy value of **0mm**
- c) Left-click to place Pad 2 at that position

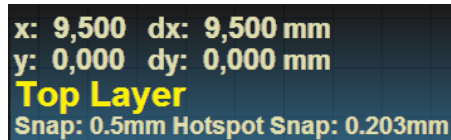


Figure 29. Heads Up Display showing incremental cursor position

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 30.

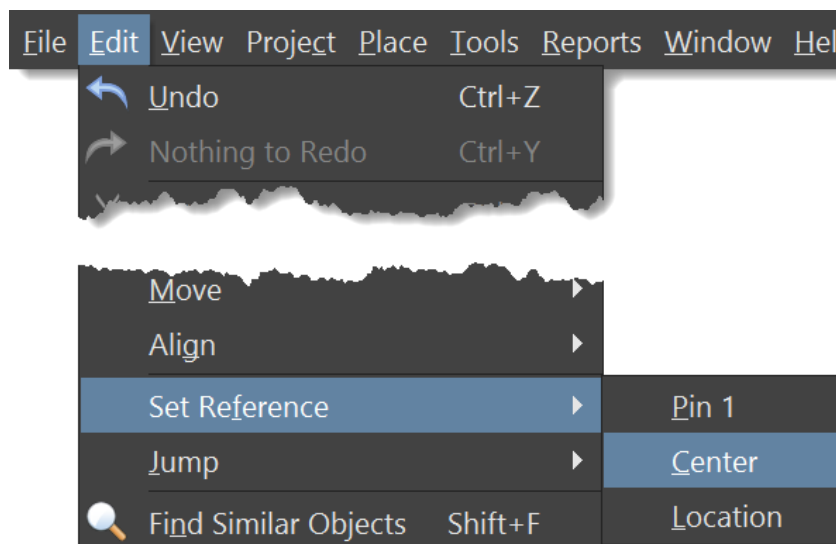


Figure 30. Setting Component Origin from the Edit menu

45. Assign the origin to the center of the existing pads by going to **Edit » Set Reference » Center**. The component origin, shown by the circle and crosshairs, will be placed at the geometric centre of the pads, as shown in Figure 31.

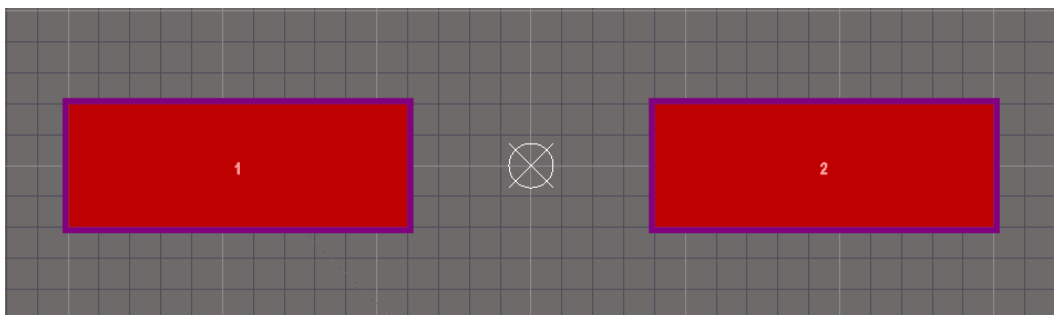


Figure 31. Component Origin set to Center





7.5 3D Crystal Body

In this section we will add a 3D STEP model of the package for this component. For the training a STEP model is available for download from the Workspace. We also need to create an appropriate 3D mechanical layer. For the training, a STEP model is available for download from the Workspace.

46. Open the *Explorer* panel and follow the instructions below as seen in Figure 32:

- a) Navigate to *Mechatronic 3D Models – Component 3D Models*.
- b) Select the item A3D-000001.
- c) Right-click on the Item and choose **Operations » Download...**
- d) Save the STEP file in a location you prefer, for example, Desktop or Downloads.

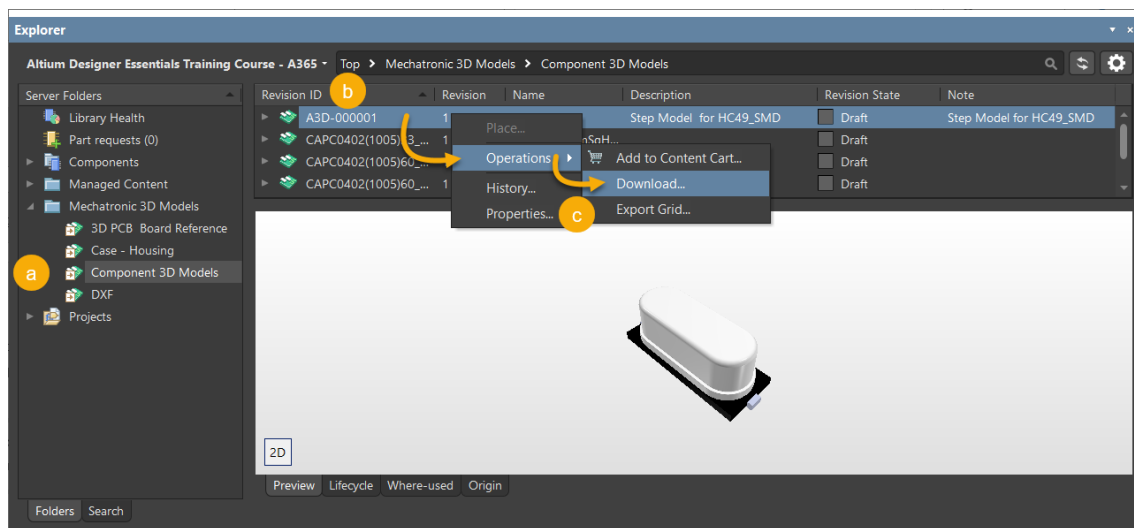


Figure 32. Download STEP Model

47. 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 while the editing area is active.





48. **Right-click** anywhere in the *View Configuration* panel and select **Add Component Layer Pair**, see Figure 33 below.

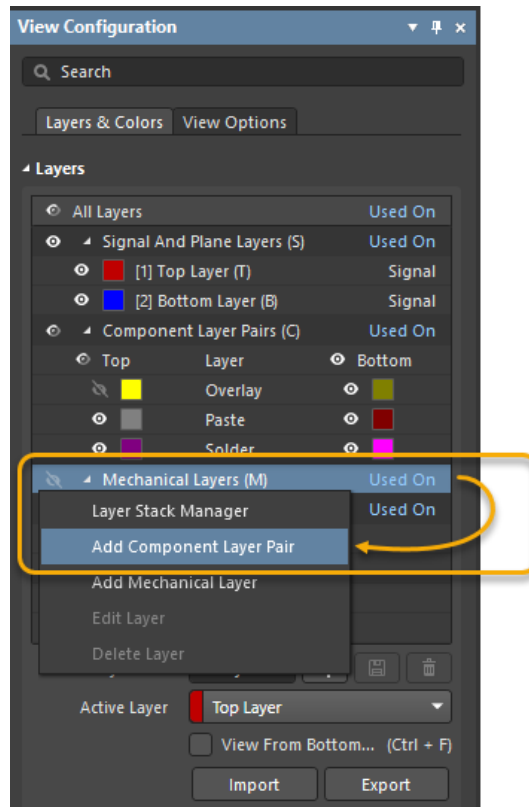


Figure 33. Add a Mechanical Layer Pair

49. Enter the information shown in Figure 34. Change *Layer Pair Name* to **3D Models**, and from the *Layer Type* drop down list select **3D Body**.

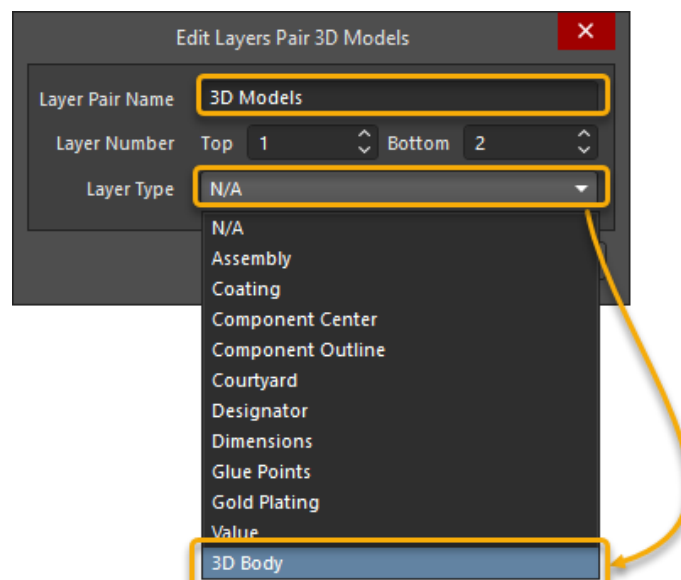


Figure 34. Configure Mechanical Layer Pairs





50. The new layer pair will be added to the *Component Layer Pair* section of the *View Configuration* panel, see Figure 35.

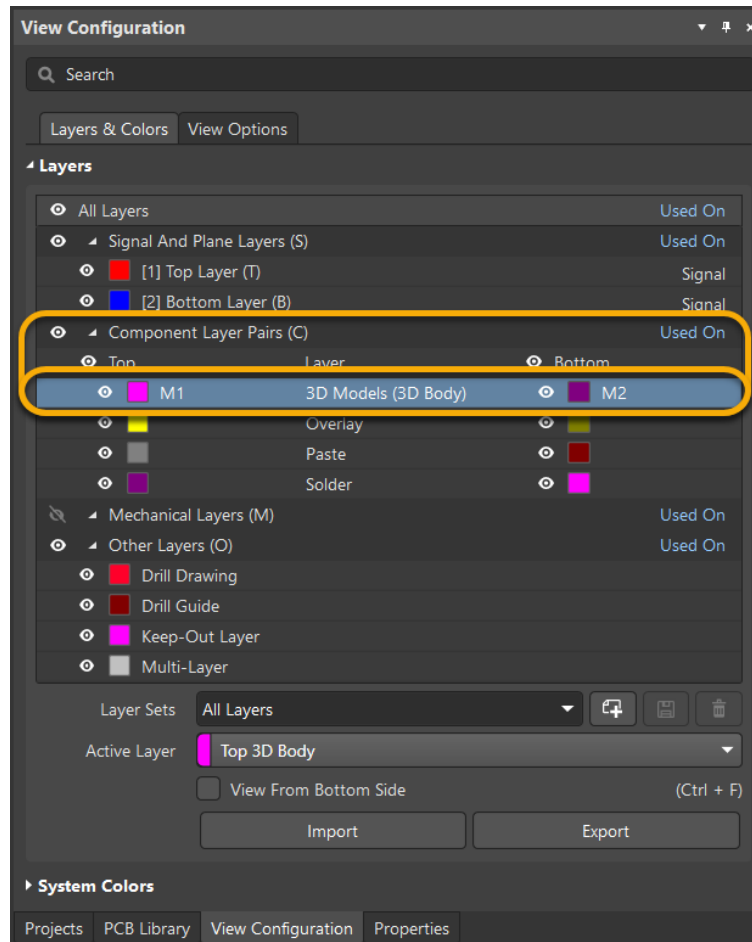


Figure 35. Layer Pairs

51. Go to **Place » 3D Body (P, O)** from the menu as shown in Figure 36.

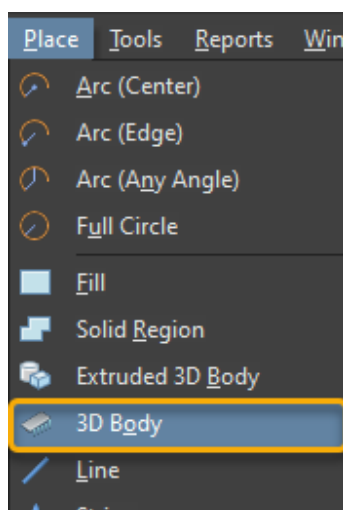


Figure 36. Place 3D Body Command





Caution: The mechanical layers we use during this exercise are just for training purposes. Check your company guidelines for Altium to choose the correct mechanical layers for component creation. If a layer *.stackup file is available, you can import the stackup through the **Tools >> Import Mechanical Layers...** menu option.

52. In the *Choose Model* dialog navigate to the local folder you selected during Step 46, for example, Desktop.
53. Open the Subfolder `... \A3D-000001 \Released` and select the STEP Model `HC-49_SMD.STEP`.

Hint: In the 2D environment, all 3D models, are seen as shapes with a diagonal hatch pattern.

54. Before placing the 3D model, ensure that you are on the **Top 3D Model** layer by either using the **+** or **-** keys from the **Num** pad on your keypad.
55. With the STEP model attached to your mouse, place the 3D Body in the center of your Footprint, as seen in Figure 37.

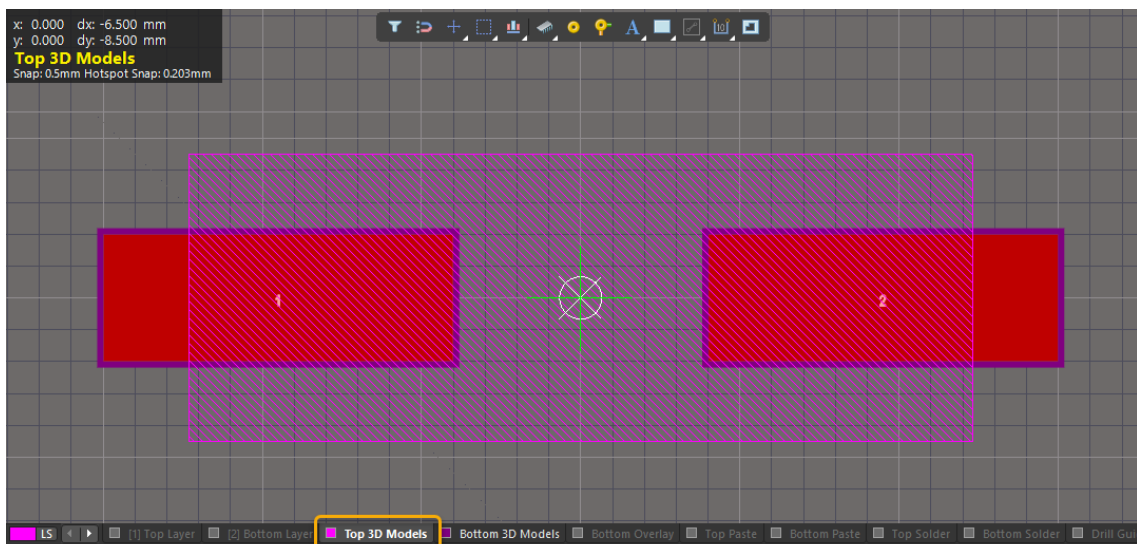


Figure 37. 3D Body Dialog in 2D Mode





56. With the 3D model still selected, open the *Properties* panel and make sure the *Override Color* option is not enabled, as shown in Figure 38.

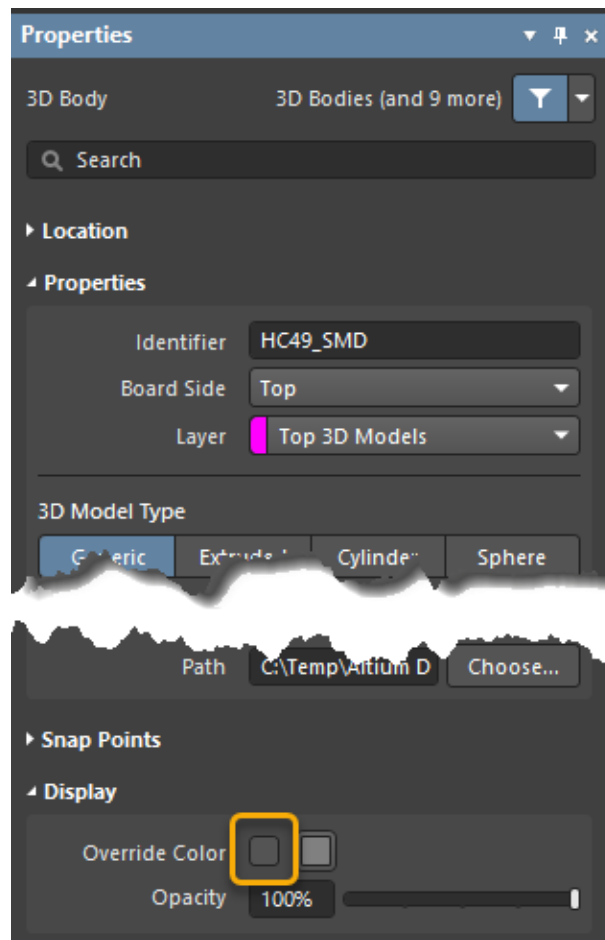


Figure 38. 3D Model Properties

57. Hit the **3** key to view the body in 3D mode, as shown in Figure 39 below.
 58. Hit the **8** key to see the 3D model in isometric view.
 59. Hit the **2** key to go back into 2D mode.

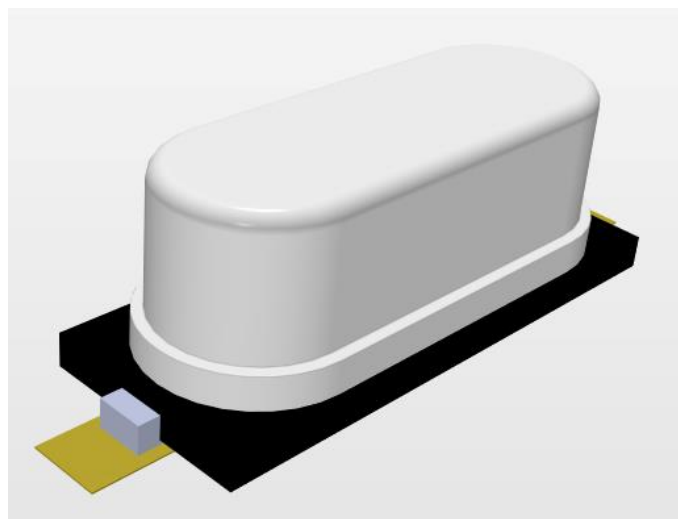


Figure 39. STEP 3D Crystal Body and Footprint





7.6 Adding Top Overlay Outline

60. Switch to the silkscreen layer by clicking on the **Top Overlay** layer tab at the bottom of the workspace.
61. Right-click on the **Graphical Place** menu in the **Active Bar**, and select line, as shown in Figure 40. Alternatively, to place a line go to the **Place » Line (P » L)** command.

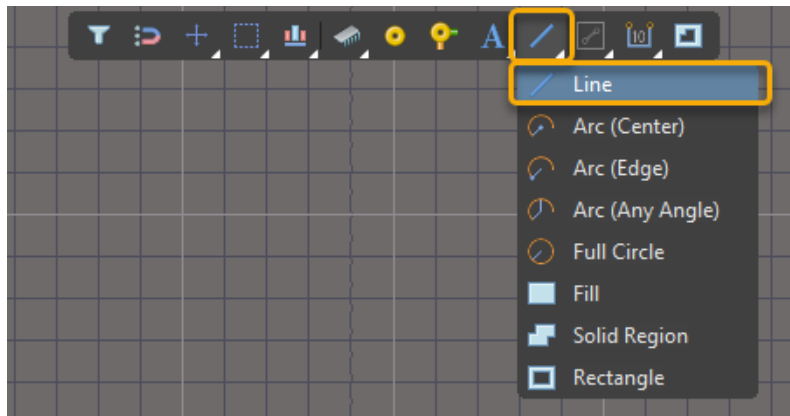



Figure 40. Placing a Line

62. Next, hit the **Tab** key to open the *Properties* panel, set the Line Width to 0.5mm, see Figure 41. Press **Enter** or select the pause symbol  to continue the line placement.

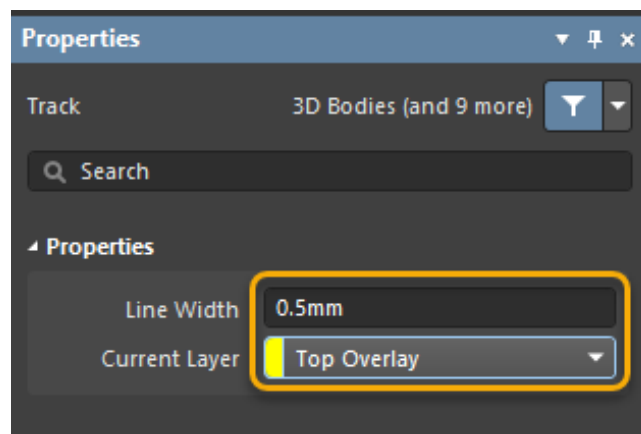


Figure 41. Line Properties

63. 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.
64. 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.
65. Using the HUD to guide you, move the cursor to location x: -8.500mm y: 1.000mm and click to set the starting point for the line.
66. Move the cursor up and to the right to see the corner angle mode of the line.





67. 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 42.

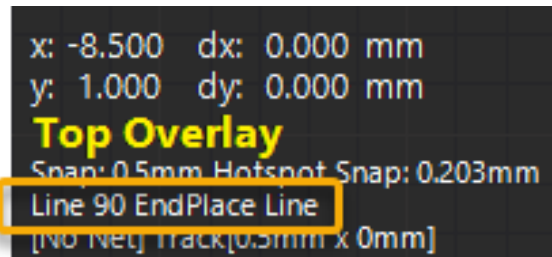


Figure 42. HUD displaying current line mode of 90 degrees

68. 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)
69. 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 43.

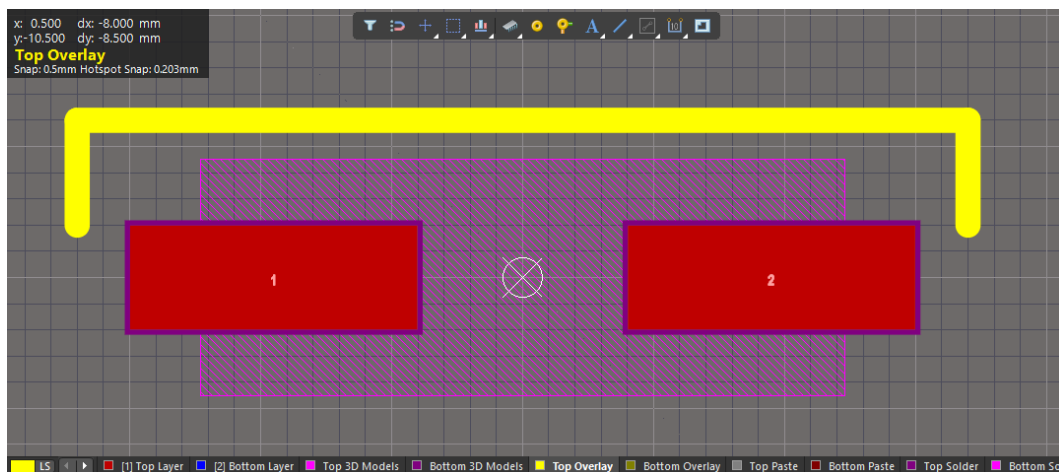


Figure 43. The top portion of the Overlay added to the component

70. Next, we will copy and paste the lines from the top of the component to the bottom part, 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.
71. Change the selection filter to **Tracks** only, see Figure 44.

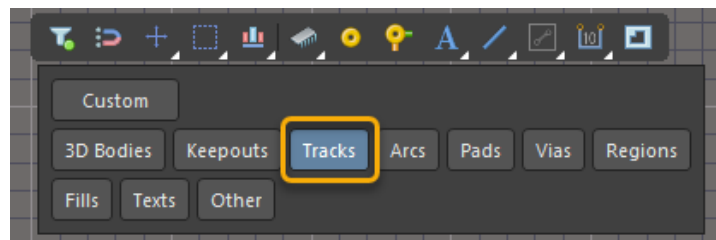


Figure 44. Filter Options





72. Drag a selection rectangle around the yellow lines.

- Press **Ctrl + C** to copy it to the clipboard.
- The cursor becomes a crosshair, and a message is displayed at the bottom of the screen: *Select a reference point* as shown in Figure 45.
- Click on the Origin to set it as a reference point.

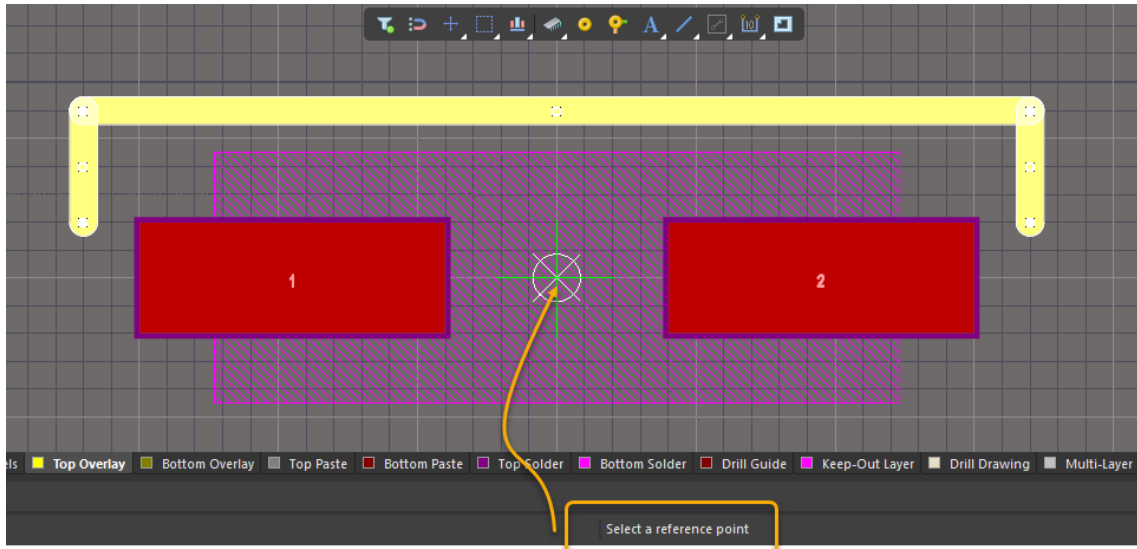


Figure 45. Copy the upper outline and set a reference point in preparation for pasting

- 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.
- Hit the **Spacebar** twice to rotate the lines.
- Then, left-click to position the cursor on the origin so that the lines are now symmetrical below the pads as shown in Figure 46.

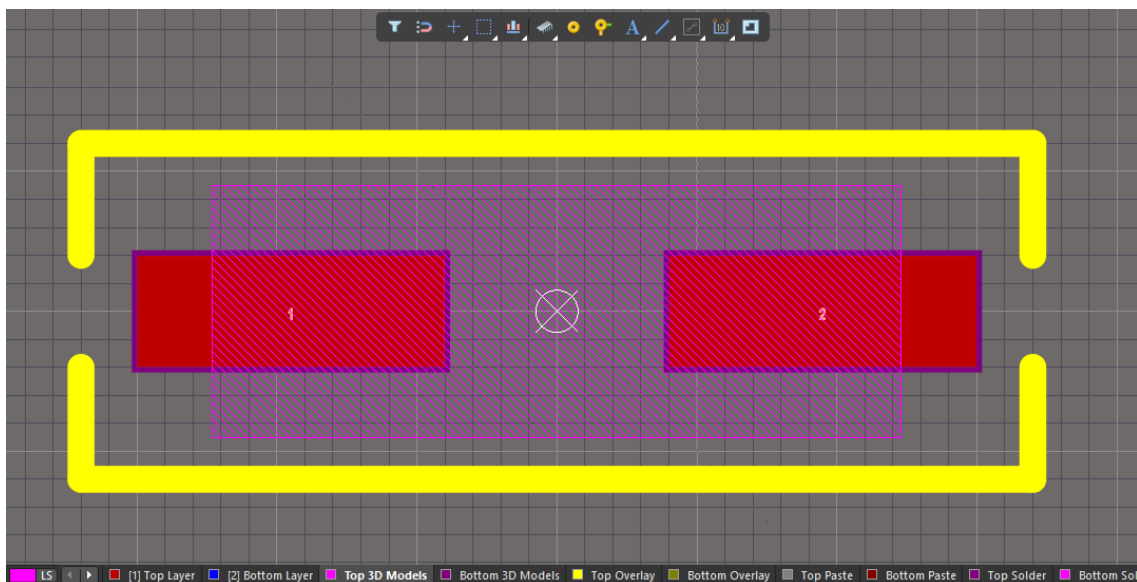


Figure 46. Copied outline

73. Go to **File » Save**  **Save**  to save the library.

74. Return to the *Component Editor* dialog.





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.

75. Select the **Edit Pin Mapping** command , as seen in Figure 47.

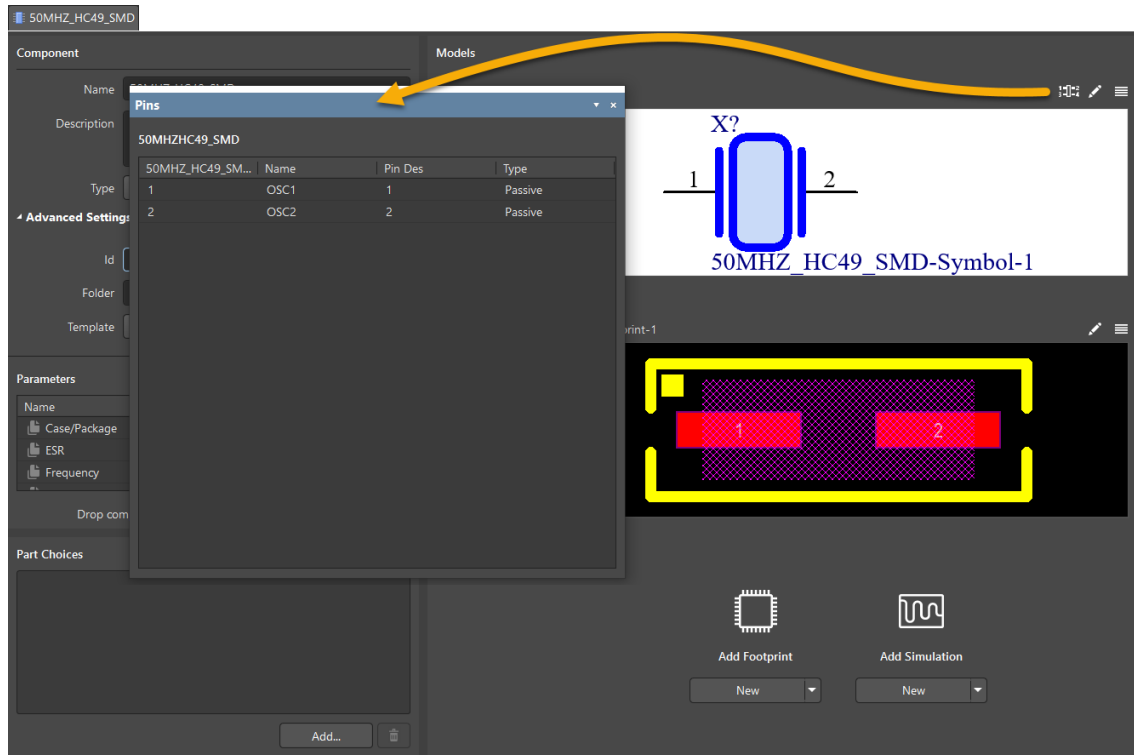


Figure 47. Pin Mapping

76. In most cases the pins and pads will match their corresponding numbers for e.g. Pad 1 to Pin 2; Pin 2 to Pad 2. Close the dialog after you checked the mapping.

77. Execute the command **File » Save to Server** or select at the *Project* panel **Save to Server**  to save your component.

78. At the *Edit revision for Item* dialog add a release note, for example, Initial release for Crystal 50MHz.

79. Select **OK** to continue. 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

**Altium Designer Essentials Training
with Altium 365**

Thank you for choosing **Altium Designer**

