

Creating a Custom Component in NI Ultiboard

Publish Date: jan 24, 2007 (<http://zone.ni.com/devzone/cda/tut/p/id/5631>)

Overview

NI Multisim and **NI Ultiboard** provide an integrated platform to design, simulate and lay out a complete Printed Circuit Board (PCB). The highly flexible database manager makes it easy to add a new SPICE simulation model, to a custom defined schematic symbol which can then transfer an accurate footprint to layout.

Creating a Custom Component in NI Multisim (<http://zone.ni.com/devzone/cda/tut/p/id/3173>) and **Creating a Custom Component in NI Ultiboard** are your resource on how to intuitively and quickly learn how to create your own custom components.

Table of Contents

1. Introduction
2. Step 1: Create a Database Group
3. Step 2: Edit the Grid Spacing
4. Step 3: Placing Landpattern Pins
5. Step 4: Changing Pin Names and Values
6. Step 5: Setting Reference ID and Value location
7. Step 6: Using the Ruler Bar
8. Step 7: Place the Landpattern Shape on the Silkscreen
9. Step 8: Creating a 3D Landpattern
10. Step 9: Saving the Landpattern
11. Step 10: Adding the Landpattern to a Multisim Database

1. Introduction

This tutorial is the second in a series of articles on component creation in **NI Multisim** and **NI Ultiboard**.

Part 1, entitled **Creating a Custom Component in NI Multisim**

(<http://zone.ni.com/devzone/cda/tut/p/id/3173>), explains how to utilize the Component Wizard to create a custom component with a schematic symbol and SPICE simulation model. In this part, a pre-defined database footprint, or landpattern, is associated with the newly created component.

This tutorial, **Part 2**, outlines how to create a custom Ultiboard landpattern for layout. This landpattern is created manually so as to precisely define the shape, size, and dimensions of a Surface Mount Device (SMD).

NI Multisim and **NI Ultiboard** are an integrated design platform. This means that the schematic symbol in Multisim is associated with an Ultiboard landpattern that is transferred to layout. These two tutorials form a complete, A to Z guide for component creation, from the schematic symbol and simulation model to the final PCB landpattern.

The following tutorial will outline the steps to create a custom 20-pin SMD landpattern that can be associated with the custom component created in the **Creating a Custom Component in NI Multisim** (<http://zone.ni.com/devzone/cda/tut/p/id/3173>) tutorial.

Steps in the creation process include:

- Creating a database group
- Defining a custom PCB part
- Setting the environment grid spacing
- Placing SMD landpattern pads
- Setting the Reference ID and Value
- Using the Ruler Bars for Object Positioning
- Defining the IC Package
- Creating the 3D Model
- Saving the landpattern to the Ultiboard and Multisim database
- Associating a landpattern with a Multisim symbol

Please note: The landpattern you are creating in this tutorial is not a commercial package. The pad and package dimensions and spacing are used primarily to showcase an effective design procedure in Ultiboard.

The term **landpattern** and **footprint** will be used interchangeably throughout this tutorial.

2. Step 1: Create a Database Group

Ultiboard and Multisim both share a common database structure which organizes the various components into logical groups. The creation task therefore begins in the Ultiboard Database Manager.

Tips: öppna eller skapa ett projekt för komma åt verktygsmenyerna

a.) Select **Tools >> Database >> Database Manager** to display the Database Manager (Figure 1).

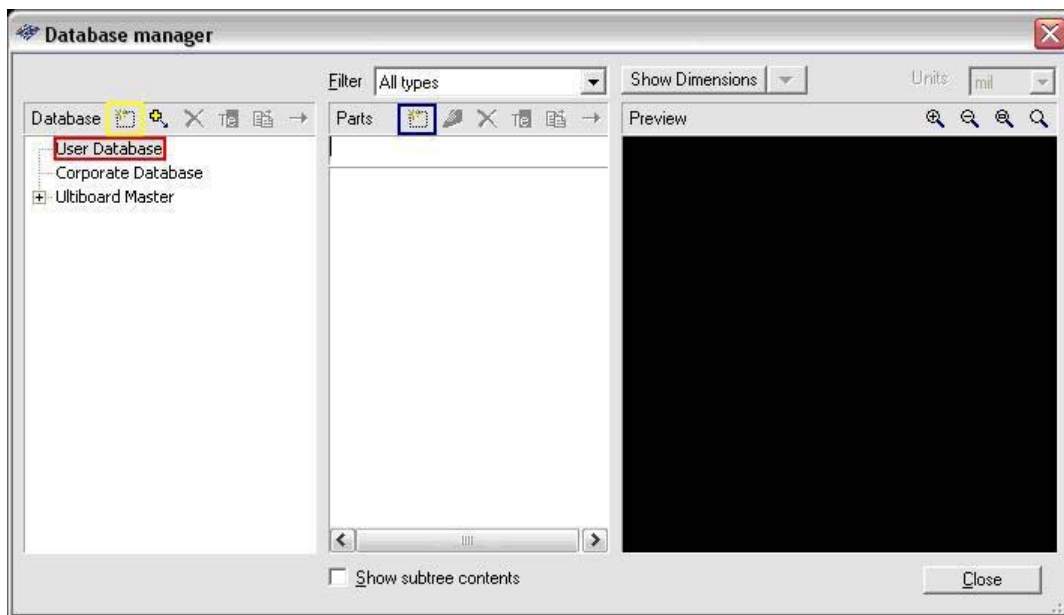


Figure 1 - Database Manager

b.) In the Database Manager select the **User Database** (the red square in Figure 1 above).

Custom groups can be created in the Database Manager to logically organize where to save the footprint. The database group does not have to be created for every new component. The groups are created to assist in organizing the database appropriately. If a group has already been created which your custom component can be saved to, you can skip directly to e.

c.) Select the New Database Group icon (the yellow square in Figure 1 above).

d.) A new branch will appear to signify a group in the User Database root. Enter the group name, **Custom SMD**.

e.) Select the **Create a new part** icon (the blue square in Figure 1 above). NI Ultiboard will display a dialog box displaying the options of the type of component to create as displayed in Figure 2.

f.) Select the PCB part option in the dialog box shown in figure 2.

The Ultiboard design area will now be in the 'Footprint edit mode' and display the **x??** symbol (Figure 3). **x?** represents the Reference Designator (RefDes) and **?** represents the Value of the component. These values will be edited later in the design process in Step 4.

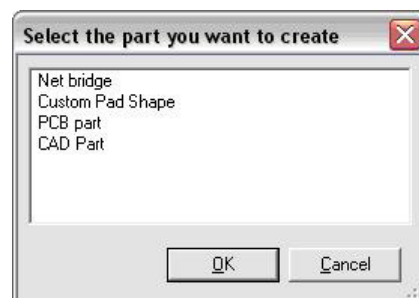


Figure 2 - Select Part

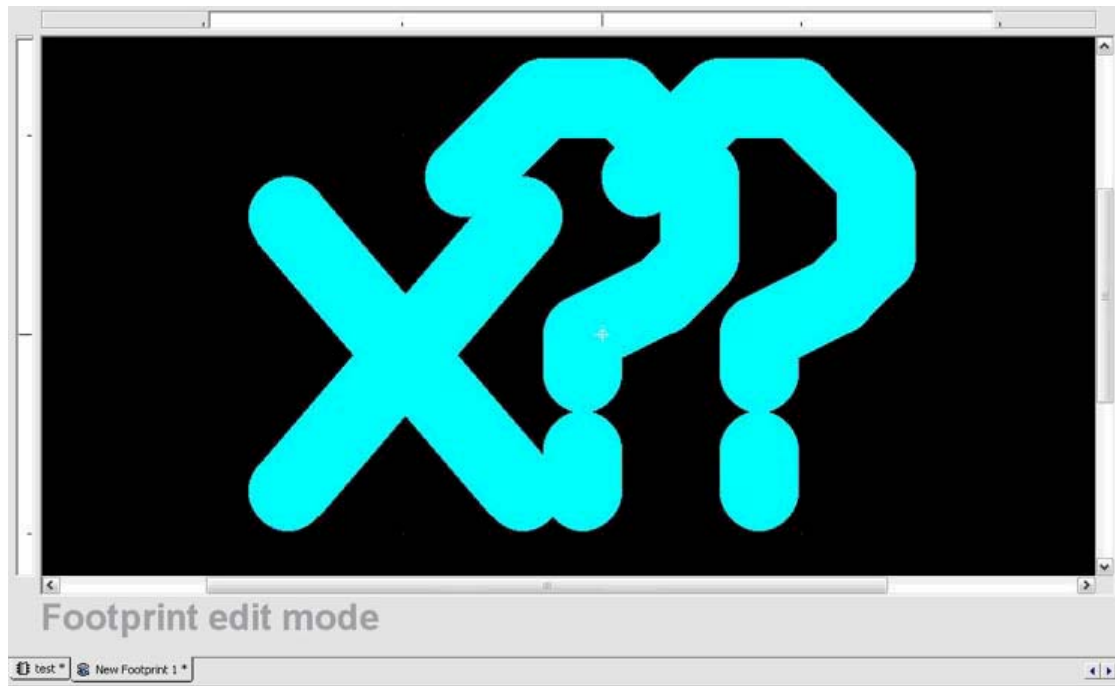


Figure 3 - Footprint Edit Mode

3. Step 2: Edit the Grid Spacing

It is important to begin the design process by setting the units of measurements for the landpattern design grid. When placing components, the design grid is what objects will snap to when they are being moved and/or placed. The grid spacing therefore defines the degree of precision with which objects can be placed in the work-area.

- a.) Select **Options >> PCB Properties**.
- b.) Click on the **Grid & Units** tab.
- c.) In the units section (Figure 4) set the **Design Units** as **mil**.

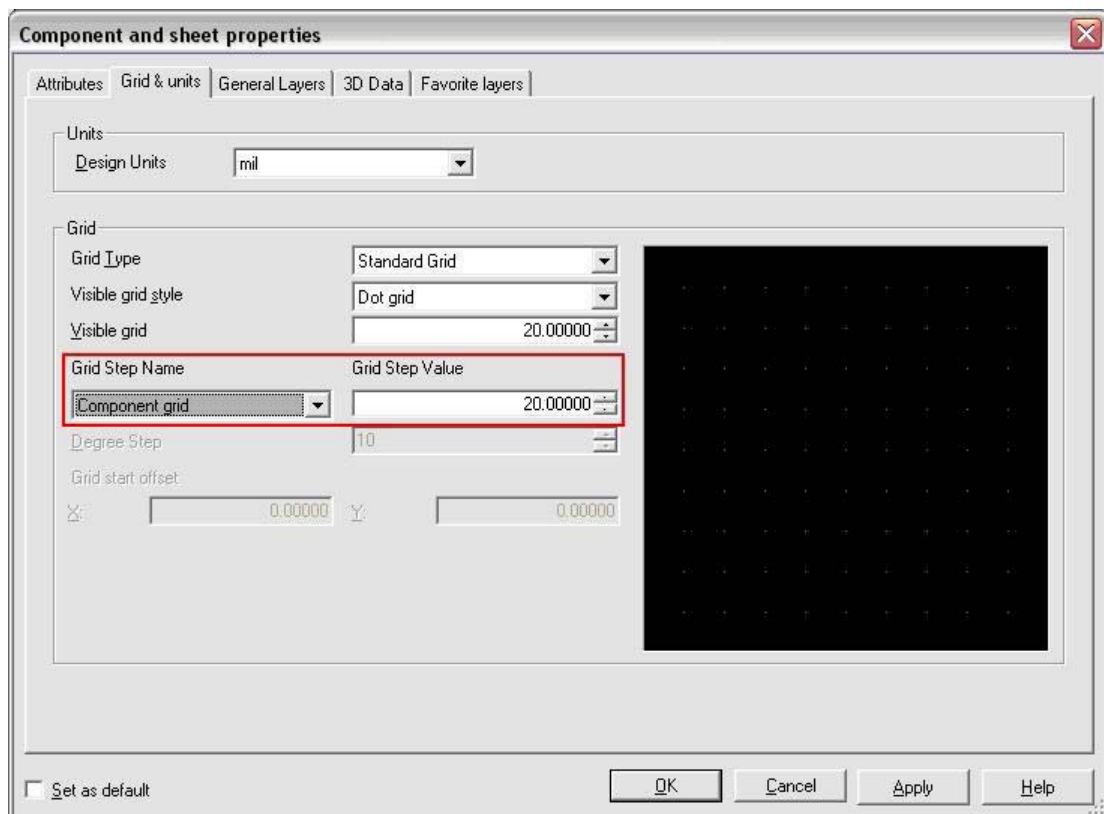


Figure 4 - Grid Properties

d.) To set the various Ultiboard design grids select **Component Grid** in the Grid Step Name combo box (the red box in Figure 4).

Tips: **Component Grid** har blivit omdöpt till **Part Grid**

e.) Set the **Grid Step Value** to 1 mil.

f.) Select the **Copper Grid** in the Grid Step Name combo box.

g.) Set the **Grid Step Value** to 1 mil.

With these settings both components and copper will be placed on a 1 mil by 1 mil design grid.

4. Step 3: Placing Landpattern Pins

Both THT (through-hole technology) pins and SMD (surface mount device) pads can be placed onto a custom landpattern. The following steps will outline how to add an array of SMD pads.

a.) Select **Place >> Pins**. This will display the Place Pins dialog box (Figure 5).

You will now define a 10 by 2 array of SMD pins for our custom landpattern.

b.) Select the SMD Pin radio button. The following values, such as Top, Pin Length, Pin Width and Pin Radius define the shape and physical dimension of the each pin. Set the following values in **Table 1** to define the pin properties (as in Figure 6);

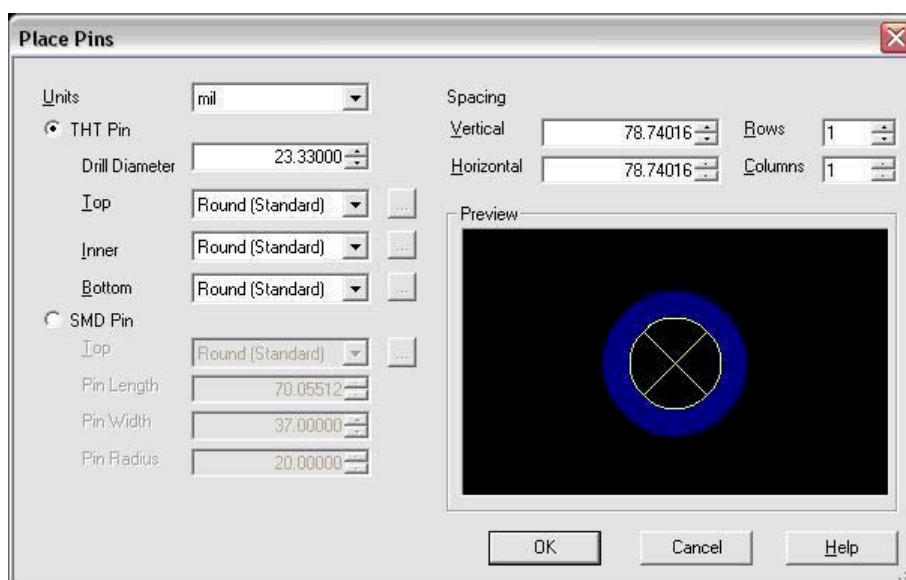


Figure 5 - Pin Properties Dialog

Table 1: Pin Dimensions

Field Name	Value	Field Description
Top	Rounded Rectangle	The top field described the shape of the pad. Other options include Round, Rectangle etc...
Pin Length	58.34	The pin length defines the vertical dimension of the pin
Pin Width	15	The pin width defines the horizontal dimension of the pin
Pin Radius	7.5	In the case of rounded objects, the pin radius defines the radius of the circular part of the object

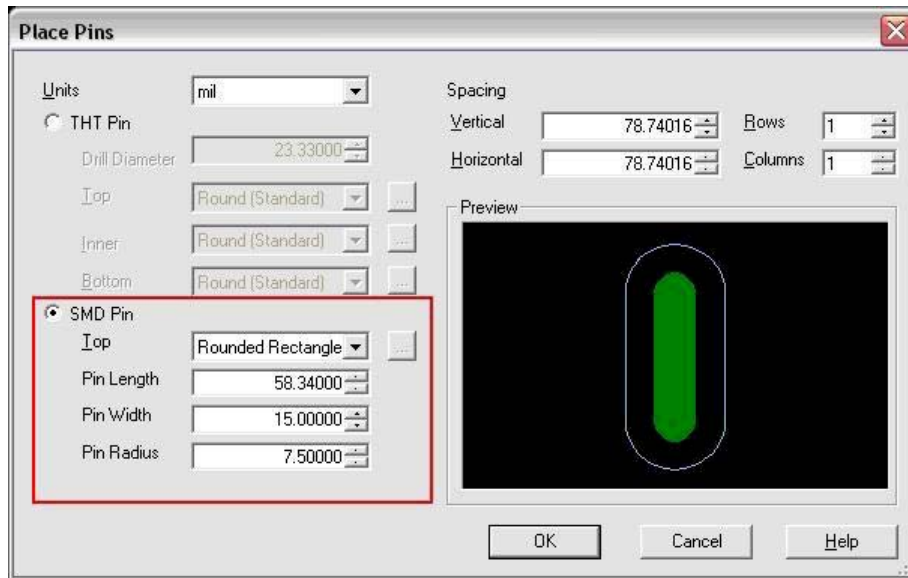


Figure 6 - Set SMD Pin Properties

c.) You will now create the 10 by 2 SMD pin array to place on the landpattern. The vertical and horizontal spacing that will now be set indicate the center-to-center distance between adjacent pins. Set the following spacing values in Table 2. The dialog box should look like Figure 7.

d.) Click on the **OK** button to place the pads onto the landpattern

Table 2: Pin Spacing

Field Name	Value	Field Description
Vertical	25	The center-to-center vertical spacing between adjacent pins.
Horizontal	200	The center-to-center horizontal spacing between adjacent pins.
Rows	10	The number of rows in the pin array
Columns	2	The number of columns in the pin array

Note: The above dialog settings can be changed to fit your specific landpattern. For example, if both **Rows** and **Columns** are set to **1**, you can place individual pins.

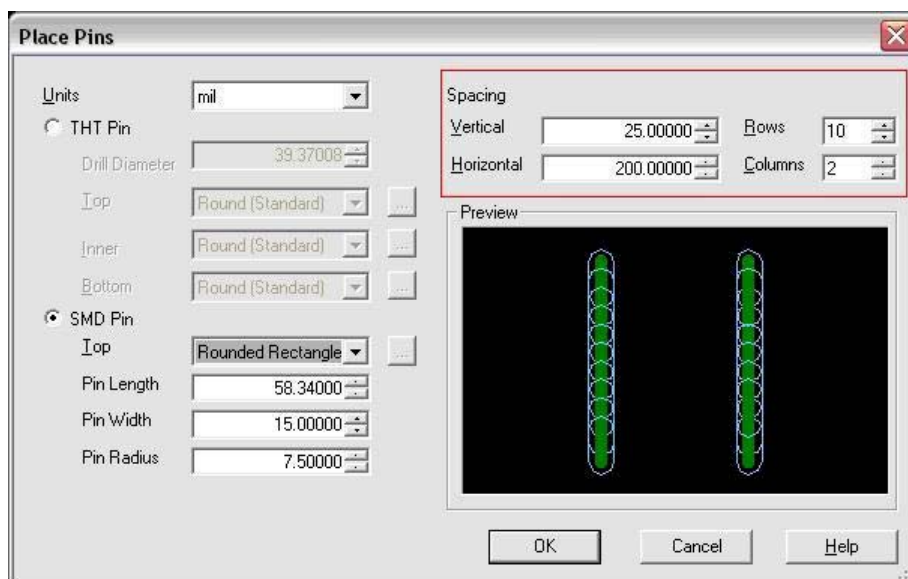


Figure 7 - Set SMD Pin Spacing

Pin 1 of the pin array is the upper-left pad and is attached to the mouse cursor. To accurately place the pad so that the reference point is in the middle of the pad press the **asterisk key (*)** on the right number pad (not shift + 8). Laptop users must press **FN + 0**.

Tips: ange X = 0 och Y = 0 i dialogrutan

The Ultiboard work area will now appear as in Figure 8. You will notice that our pins are oriented incorrectly (vertically, rather than horizontally) and must be rotated by 90 degrees.

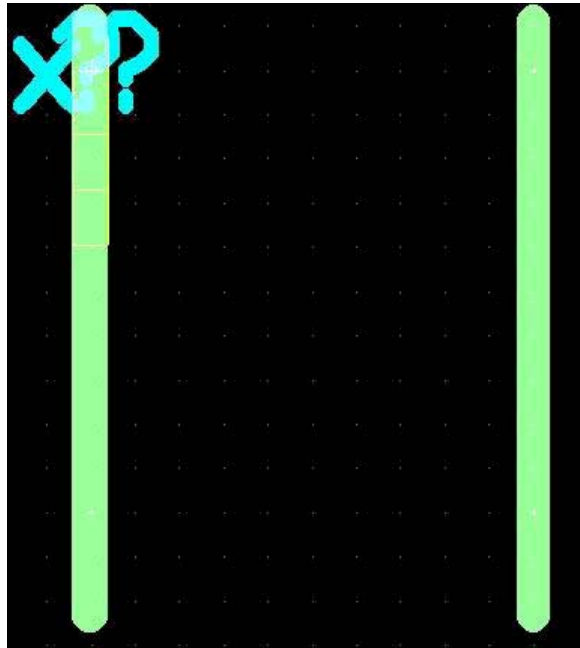


Figure 8 - Pin Setup

e.) In the Select toolbar, click the **Enable Selecting SMD pads** icon as shown in Figure 9.

f.) Drag the mouse across the design area to create a selection box, which selects all of the SMD pins

g.) Double-click on one of the selected pins. The SMT Pin Properties dialog box will appear. Select the General tab.

h.) Select the **Angle (degrees)** combo box and set to 90 degrees (as in the red box in Figure 10). Click on the **OK** button to save the change. You will notice that the SMD pins have been rotated and now look like Figure 11.

Tips: Angle har blivit omdöpt till Rotation



Figure 9 - Select Toolbox (SMD Pads)

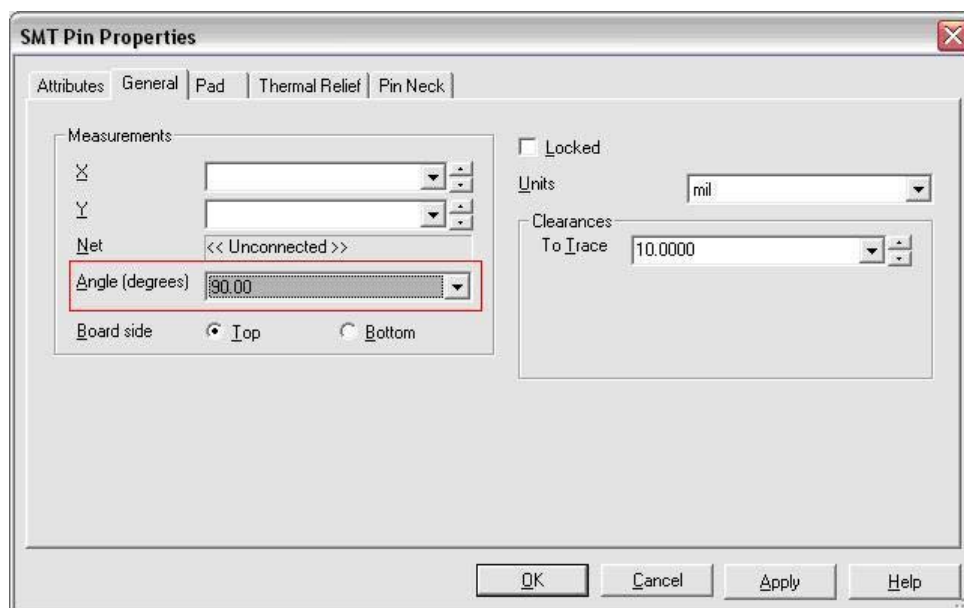


Figure 10 - Pin Angle

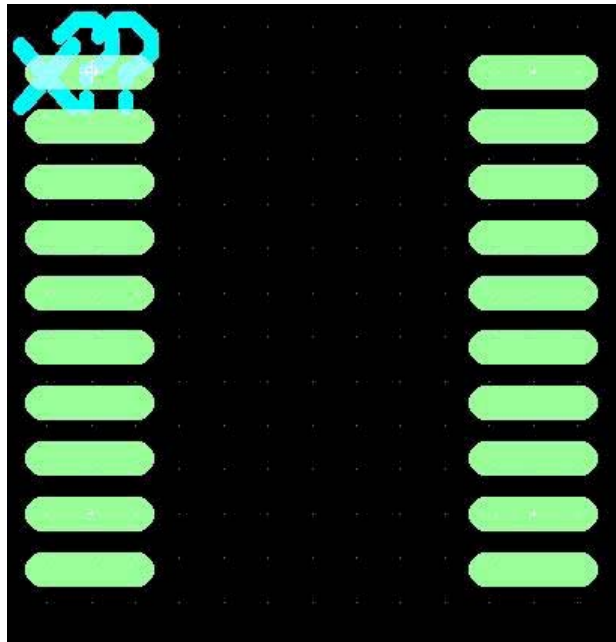


Figure 11 - Final SMD Pin Orientation

5. Step 4: Changing Pin Names and Values

During the creation of a new landpattern pins may be placed onto the design in a random order. It may therefore become necessary to edit the value associated with each pin so that they are numbered sequentially around a package.

- a.) In the Select toolbar, click the **Enable Selecting SMD pads** icon as shown in Figure 9.
- b.) Double-click on the pin in the bottom right-hand corner. The SMT Pin Properties dialog will appear. Select the Attribute tab.
- c.) The attribute list (Figure 12) displays the tag which stores the pin number (**NUMBER**), the pin value (**20**) and its visibility (**None**).
- d.) Select the **NUMBER** tag in the attribute list, which will activate the **Change** button (red box in Figure 12). Click the **Change** button.
- e.) In the attribute dialog box, changing the **Value** field (red box in Figure 13) will change the value of the pin.

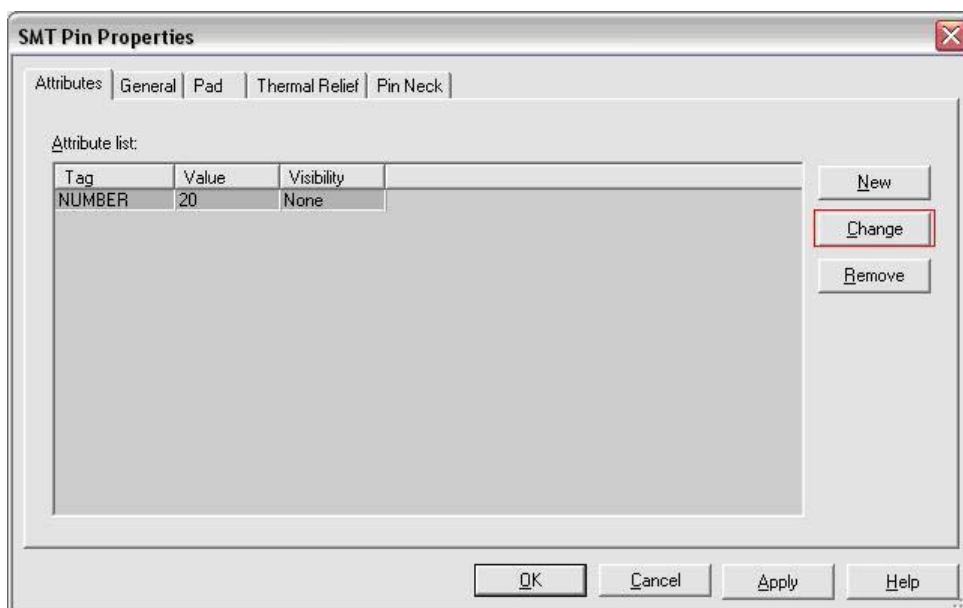


Figure 12 - SMT Pin Properties

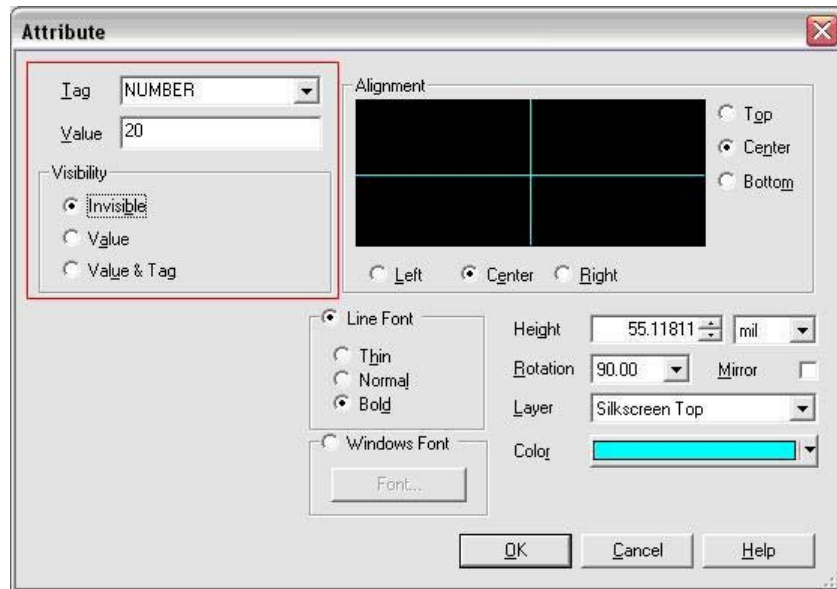


Figure 13 - SMD Pin Attributes

In general a package such as this has the pins numbered sequentially in a counter-clockwise direction. Therefore pin **10** should be in the bottom left-hand corner and pin **11** in the bottom right-hand corner. The final pin number configuration for this design should be as in Figure 14. Therefore to change the pin number;

f.) Change the **Value** field to 11 and click on the **OK** button. Click on the **OK** button in the SMT Pin Properties dialog box to save the changes.

g.) Repeat **d.)**, **e.)** and **f.)** for all the pins for the required configuration (as in Figure 14).

1	20
2	19
3	18
4	17
5	16
6	15
7	14
8	13
9	12
10	11

Figure 14 - Component Pin Setup Map

6. Step 5: Setting Reference ID and Value location

In Figure 11 you will continue to notice the **x??** symbol, which represents the Reference Designator and Value of the landpattern.

The **Reference Designator** is the key identifier of the component and is represented as **x?**. This **x** can be changed so that it corresponds with the type of component it is associated with. For example, if this landpattern was for a resistor this value could be set to R. The **?** represents a number to denote the different components in a design. So, returning to the example of a resistor, the resistors placed onto a design, would be named R1, R2, R3 etc -

Value, represented by the **? symbol**, is the physical value of the component. Taking the example of the resistor into consideration once more, the **?** could equal **20k** for 20 Kohms.

In this step you will select, move and alter the RefDes and value.

a.) In the Select toolbox, click the **Enable Selecting Attributes** icon as shown in Figure 15 (in the red box). This will allow you to refine the selection of components on the landpattern to be only the RefDes and Value attributes.



Figure 15 - Select Toolbox (Selecting Attributes)

The RefDes and Value are currently overlapping the landpattern placement as shown in Figure 11. b.) Click on the **X?** symbol and place it to the upper-right of the pad array.

c.) Click on the **?** symbol and place it to the upper-right of the pad array. d.) Double click on **X?** and select the Attribute tab

e.) In the Value field set the REFDES tag to **U?** as in Figure 16 (in the red box) f.) Click on **OK** to apply changes.

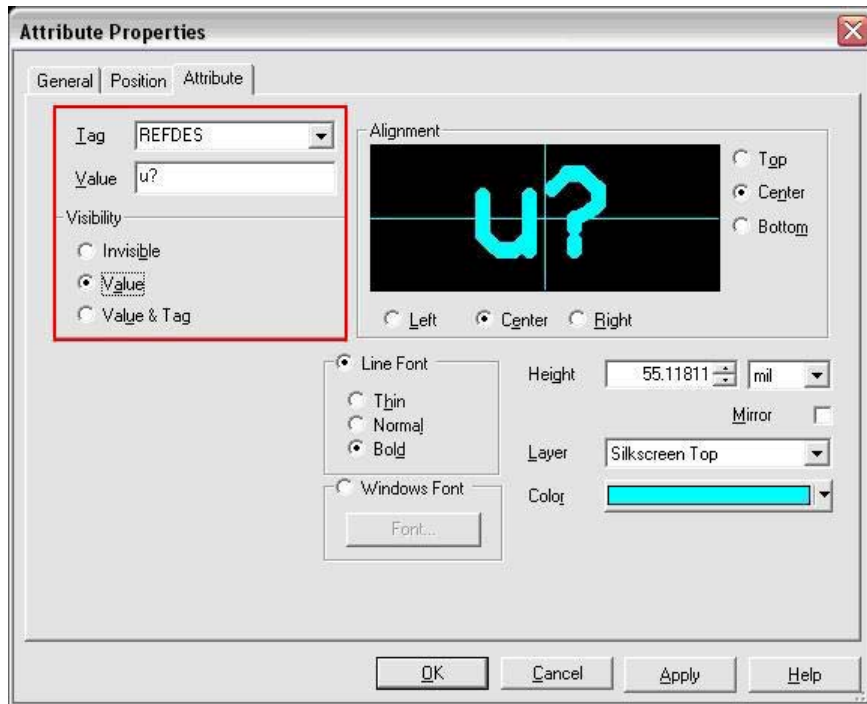


Figure 16 - REFDES Attribute Properties

When the custom component is now placed the RefDes for the landpattern will be U1, U2, U3 etc...

Generally the Value attribute, ?, is set to either visible or invisible depending on the need to highlight the value of the landpattern on the final design. In this example, it will be set to invisible.

g.) Double click on **?** and select the Attribute tab

Ändring g.): ? var X? i original

h.) In the Visibility section of the attributes set the Invisible radio button

7. Step 6: Using the Ruler Bar

Both along the top and left side of the Ultiboard design area are the ruler bars in Figure 11. If the ruler bars cannot be seen, go to **View >> Ruler Bars**. These ruler bars allow the positioning of dashed lines to align shapes from one object to the next, precisely within the design area. For the following steps refer to Figure 14 for the pin numbers.

a.) Left-click in the horizontal (top) ruler bar area for a small arrow marker to appear and move the marker immediately to the right of pin 1 (Figure 17).

b.) Left-click to place a marker, 5 mils to the right of the marker placed in a. You will notice that the distance between two markers is displayed in the ruler bar (Figure 17).

c.) On the vertical (left) ruler bar place another marker directly above pin 1 (Figure 17).

d.) Repeat a.), b.) and c.) to align ruler bar markers at the right and bottom of the SMD pad arrays as shown in Figure 17.

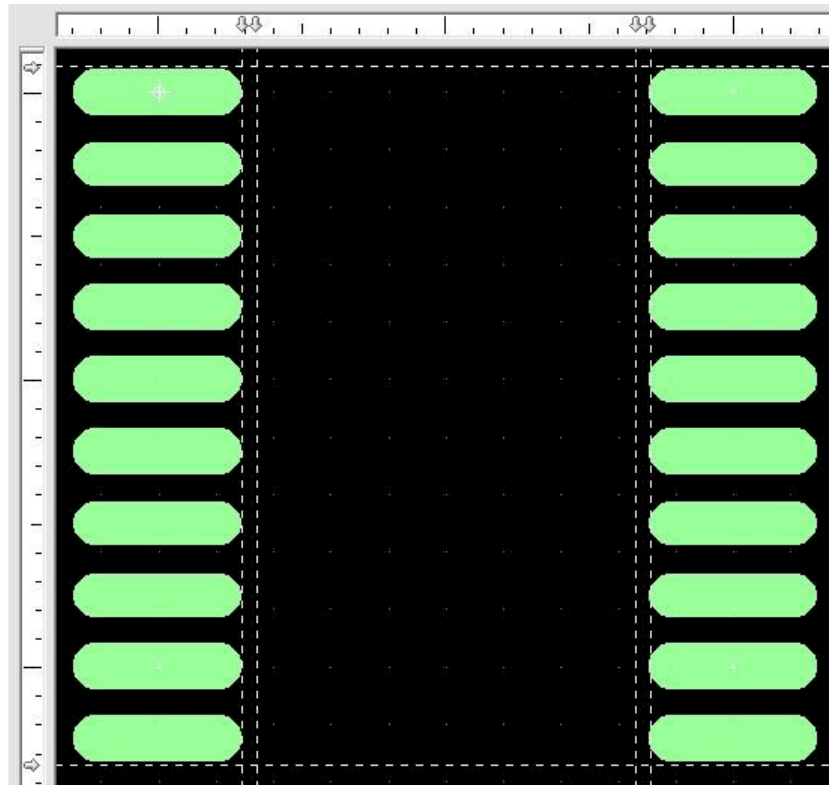


Figure 17 - The Ruler Bar

These markers will now act as a guide on where to begin the placement of the landpattern shape on the silkscreen layer in step 7.

If at any time markers need to be cleared, you can right-click on a specific marker and select **Clear**, or select **Clear All** to remove all markers.

8. Step 7: Place the Landpattern Shape on the Silkscreen

The placement of shapes on the silkscreen layer defines how the custom component will appear in your design. It represents the physical dimensions of the package. In this step you will utilize the ruler markers set up in step 4 to place a rectangular landpattern shape.

a.) In the Design Toolbox (Figure 18) double-click on the Silkscreen Top to make that particular layer active. This will highlight the layer in a red.

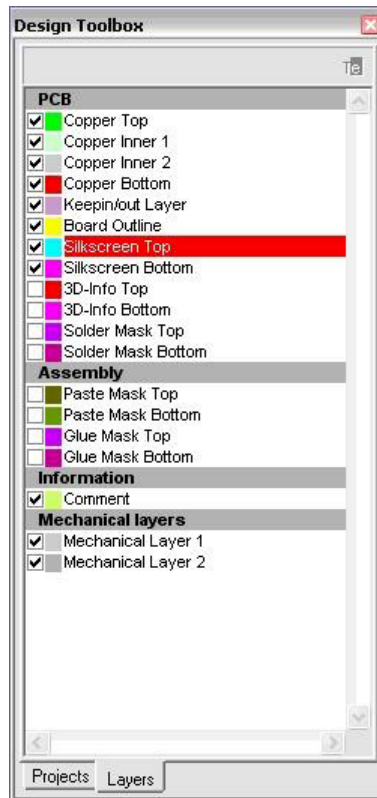


Figure 18 - Design Toolbox

- b.) Select **Place >> Shape >> Rectangle**. Tips: ska vara **Place >> Graphics >> Shape >> ...**
- c.) Ultiboard, now in a drawing mode is prepared to draw a rectangle. Line up the mouse to the upper left hand point (5 mils from pin 1 as indicated by the ruler markers in Figure 16).
- d.) Left-click and drag the mouse, drawing a rectangular shape within the inner rectangle of Figure 11.
- e.) The rectangle will appear as a solid, filled-in shape. Click on the **Enable selecting Other Objects** icon in the Select toolbar (Figure 19) and double-click on the solid rectangle.

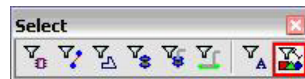


Figure 19 - Select Toolbox (Select Other Objects)

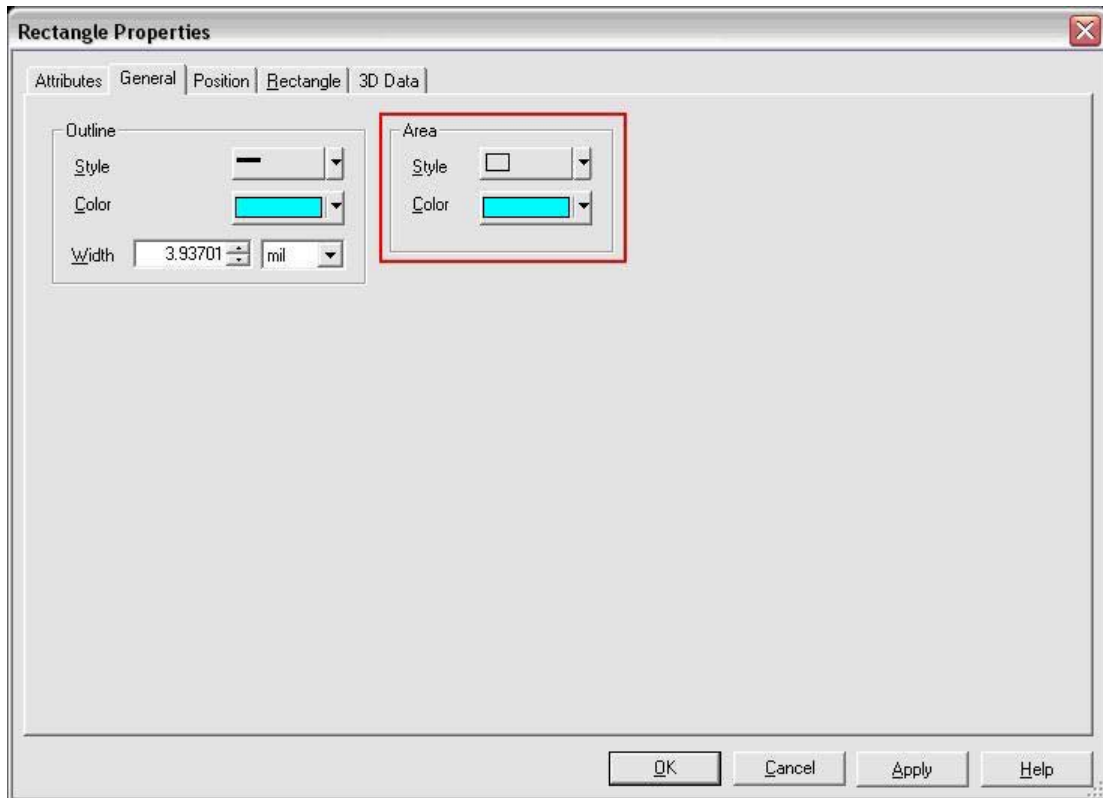


Figure 20 - Rectangle Properties

The landpattern will now appear as shown in Figure 21.

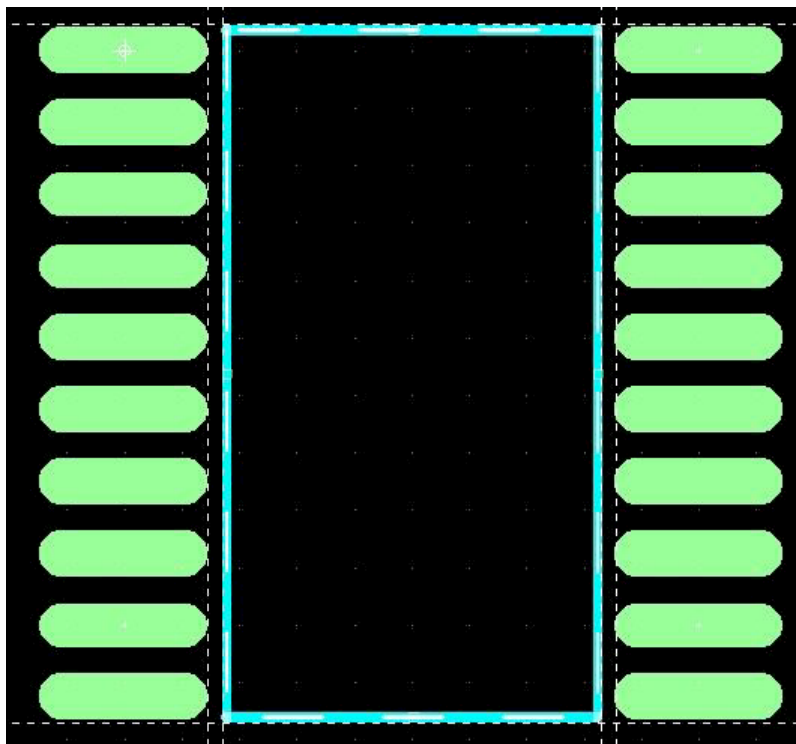


Figure 21 - Final Landpattern

9. Step 8: Creating a 3D Landpattern

In Ultiboard a design can be previewed in 3D to virtually represent the final design. In order to setup the custom landpattern for this 3D preview, the necessary information should be setup at the creation stage.

a.) In the Design Toolbox (Figure 18) double-click on the Silkscreen Top to make that layer active. This

will highlight the layer in a red.

b.) Click on the **Enable selecting Other Objects** icon in the Select toolbar (Figure 19) and right-click on the rectangle package shape. Select **Copy** in the pop-up menu.

c.) Select **Edit >> Paste** and place the ghosted image onto the Ultiboard work area to the right of the landpattern. d.) Double click on the copied rectangle and select the Position tab in the Rectangle Properties dialog box.

e.) In the Layer combo box select the **3D-info Top** option as shown in Figure 22. Click on **OK**.

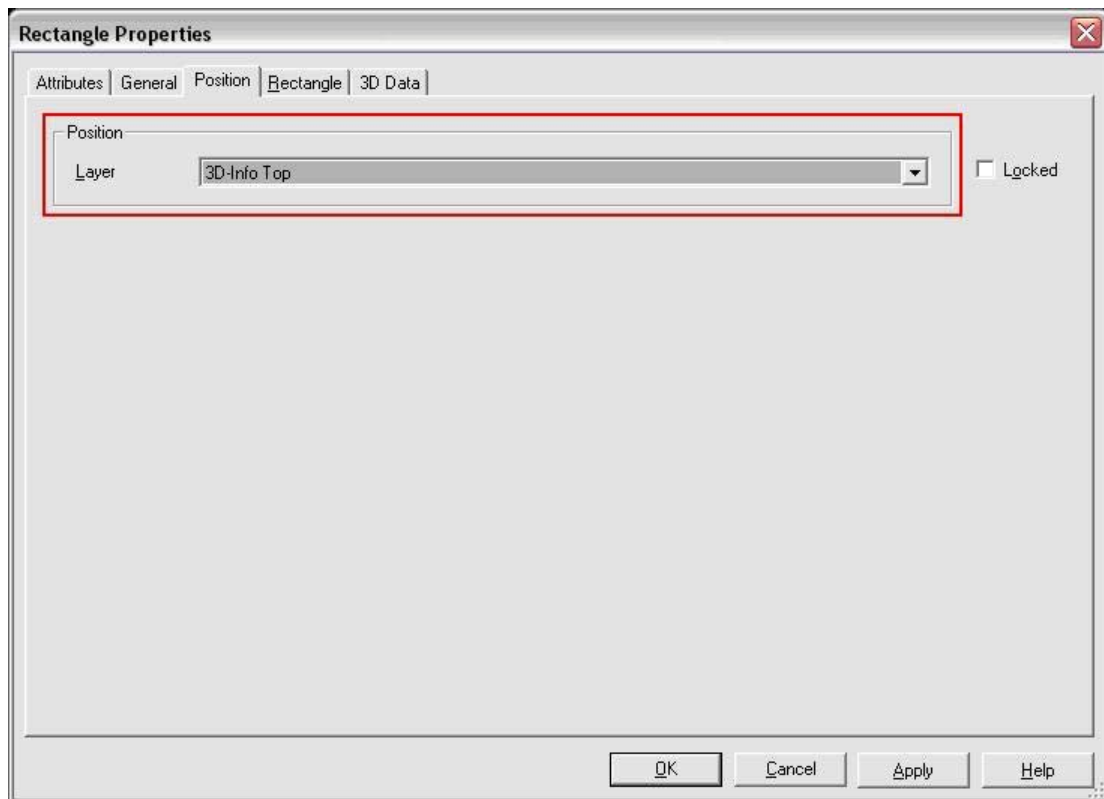


Figure 22 - 3D Rectangle Properties

f.) You may notice that the copied 3D box has now disappeared as it becomes associated with the 3D layer, which is not currently an active layer. To view the rectangle again, double-click on the **3D-InfoTop** layer in the Design Toolbox to make it active. The 3D rectangle will appear in red.

g.) Select the 3D info rectangle and move it so it is positioned directly on top of the original silkscreen shape.

h.) Double click the black work-area and select the tab for 3D data. Tips: följ h.) noga

i.) Set the properties for the 3D information (Table 3) and the dialog will appear as in Figure 23 (in the red box). Click the **OK** button.

Table 3: 3D Information

Field Name	Value	Field Description
Enable 3D for this object	Checked	Enables Ultiboard to render a 3D object
Height	45.27555	The height in mils of the 3D object
Offset	1.96850	The offset height from board
Use 2D data to create 3D shape	Checked	Uses the 2D shape on the silkscreen layer to create the package shape
Solid Shape	Selected	Creates a solid 3D representation

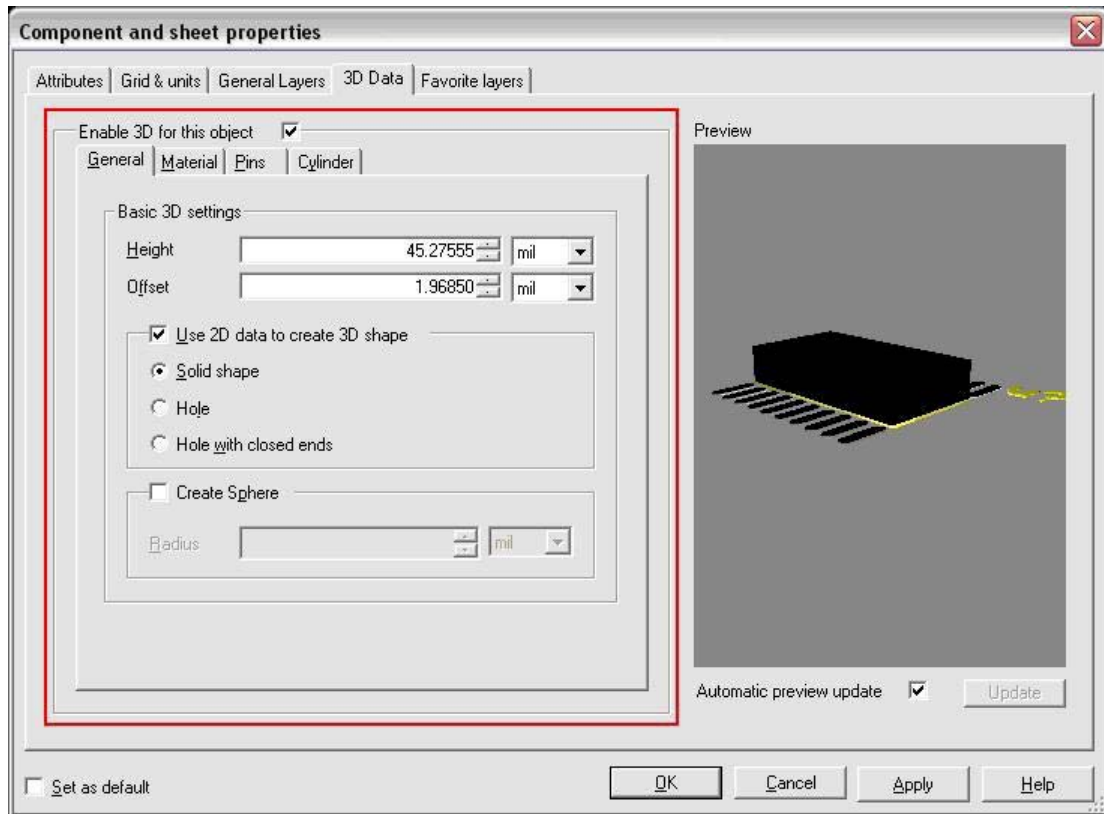


Figure 23 - 3D Settings

j.) From the 3D Data tab select the Pins section

k.) Set the properties for the 3D Pin information as in Table 4. Click on the **OK** button.

Table 4: 3D Pins

Field Name	Value	Field Description
Angle to Pad	90	Sets the angle at which Ultiboard will render the pin connections with respect to the package.
Pin Shape	Checked	Defines the shape of the 3D pin
Type	SMDPIN	Renders the specific type of pin

In the 3D Preview you should immediately notice that the pins on the left side of the component face away from the package. This is because by default, the leads on a horizontal SMD pin will be directed towards the left. Therefore the pins must be re-orientated towards the package by rotating them 180 degrees.

l.) In the Select toolbar, click the **Enable Selecting SMD pads** icon as shown in Figure 9.

m.) Hold down the **CTRL** key and select the left column of pins on the landpattern

n.) Double-click on one of the selected pads.

Ändring m.): left var right i original

o.) In the **SMT Pin Properties** dialog box select the General tab

Recall, in **step 3** that the pins were set to 90 degrees for a horizontal orientation.

Due to this 90 degree rotation, we must rotate an additional 180 degrees, in the above step so that the 3D leads are now directed towards the component. $90 + 180 = 270$ degrees.

p.) In the **Angle(degrees)** combo-box set the angle to **270**. q.) Click on the **OK** button.

If you double click in the black work-area and select the tab for 3D data you will notice that as in Figure 24, the pins are properly positioned with respect to the 3D package.

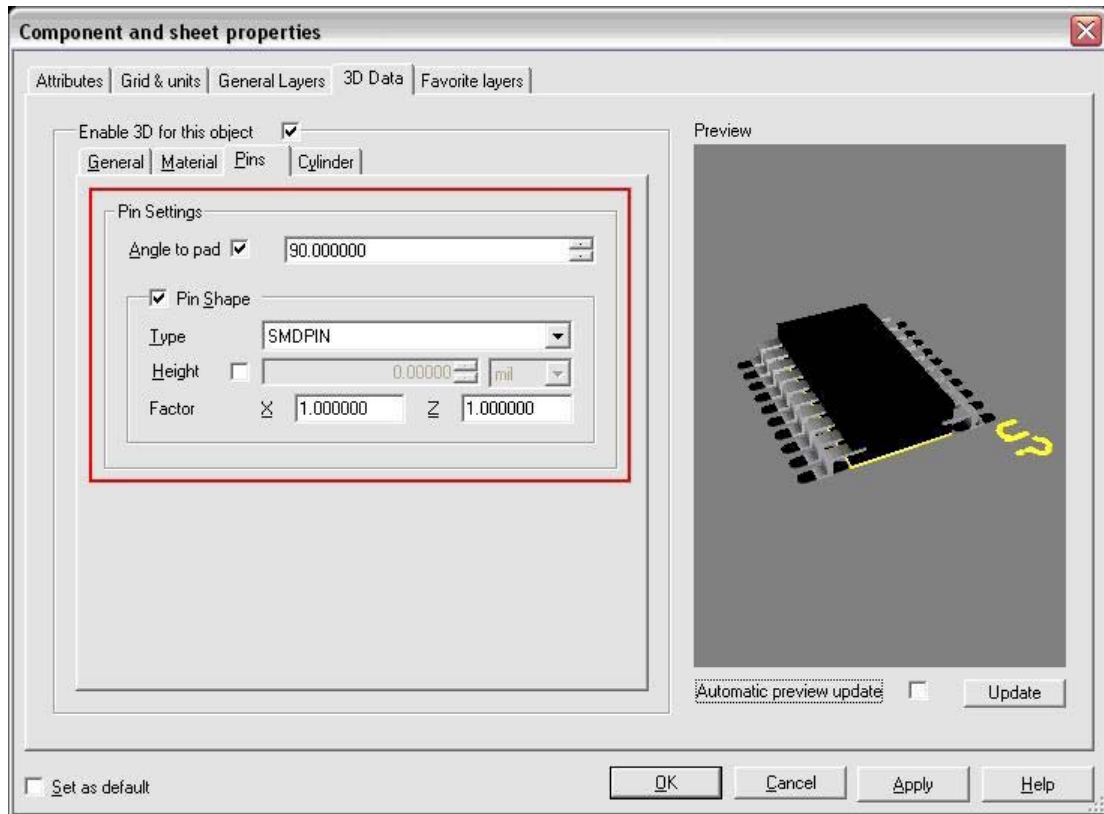


Figure 24 - 3D Pin Settings

10. Step 9: Saving the Landpattern

With the landpattern created all that remains is saving the to the Ultiboard database. a.) Select **File >> Save to Database**.

b.) In the database section select the **Custom SMD** group created in step 1 (Blue box in Figure 25).

c.) Name the component **SMD20** as in Figure 20 (red box).

d.) Click on the **OK** button.

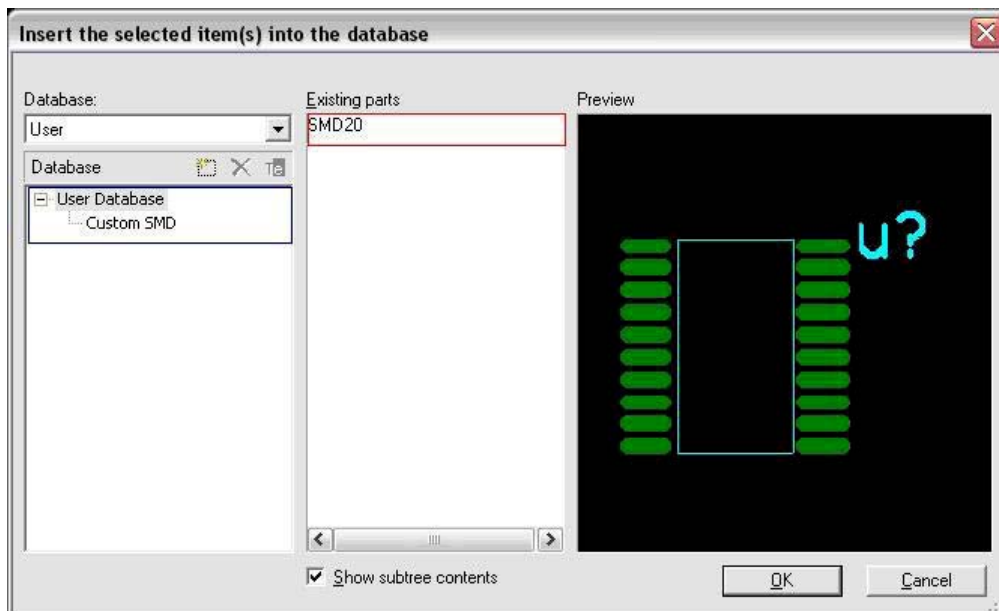


Figure 25 - Save Dialog Box

11. Step 10: Adding the Landpattern to a Multisim Database

Before a custom landpattern can be utilized in Multisim, it must be added to the database. Once added to the database, it can be seen as an available landpattern, or footprint, and added to a component.

In this step, the **SMD20** tutorial landpattern created will be associated with the symbol created in Part 1 of the tutorial. If you have not yet created a custom Multisim component, please click [here](#) to see Part 1 of the tutorial

a.) Open NI Multisim.

b.) Select **Tools >> Database >> Database Manager**.

c.) In the Database Manager select the Components tab. Under the Database Name, select the User Database. d.) In the Component List select the **THS7001** component and click on the **Edit** button.

e.) In the **Component Properties** dialog box click on the Footprint tab. f.) Click on the **Add from Database** button.

g.) In the **Select a Footprint** dialog box select the User Database (red box in Figure 22). You will notice that currently the User Database does not see the landpattern that has been created in Steps 1 to 9 of this tutorial.

h.) To add the landpattern click on the **Add** button (blue box in Figure 26).

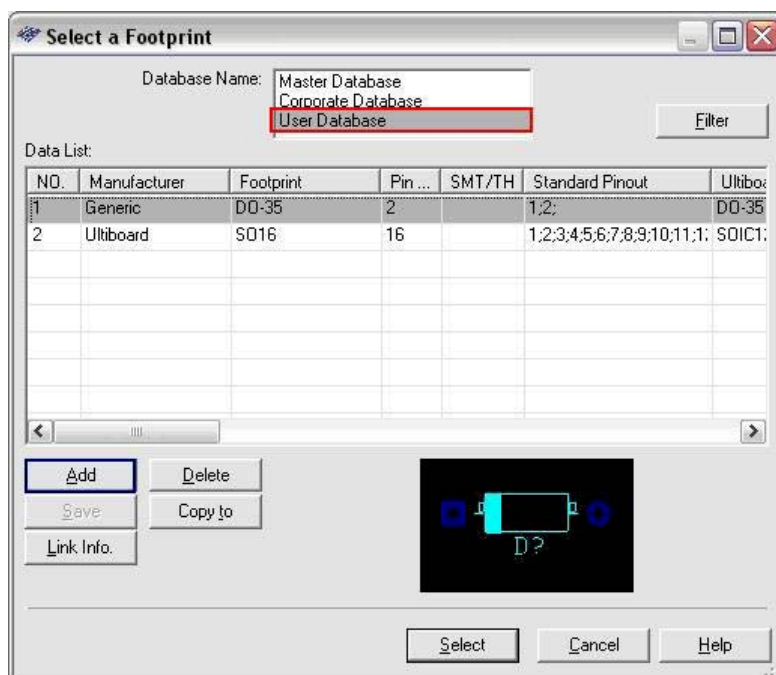


Figure 26 - Multisim Database Manager

i.) In the **Add a Footprint** dialog box set the following data as shown Table 5. The dialog box will appear as in Figure 27 (red box);

Table 5: Add a Footprint

Field Name	Value	Field Description
Database Name	User Database	The database in which you saved the landpattern in Ultiboard
Manufacturer	Generic	An additional manufacturer identifier
Footprint	SMD20	This is a CASE SENSITIVE field, and must identify the landpattern exactly as it is saved in Ultiboard
Ultiboard Footprint	SMD20	This is a CASE SENSITIVE field. While the Footprint field identifies the specific package name of the component it can be associated with the Ultiboard Footprint so that a generic footprint with the same dimensions can be transferred to layout
Number of Pins	20	Number of pins on the landpattern
SMT/TH	SMT	The type of pin technology Surface Mount or Through Hole

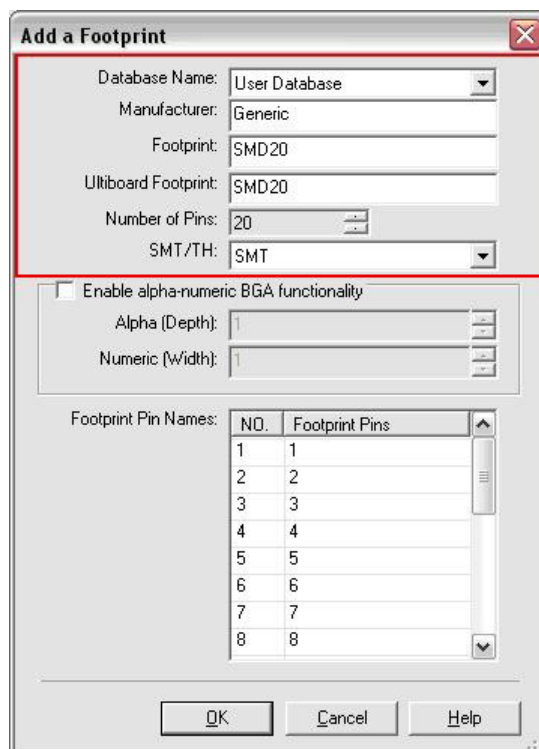


Figure 27 - Add a Footprint

- j.) Click on the **OK** button.
- k.) You will notice that the SMD20 can now be previewed within the dialog box. Click on the **Select** button to select the newly added SMD.
- l.) Click on the **OK** button to save the component settings.
- m.) Save the component to the appropriate database and group (as it was originally saved to) and click on the **OK** button. You will be prompted to overwrite the original component. Choose the **Yes** button.

Success!! You have created a custom Ultiboard landpattern, defined a 3D model and associated it to a Multisim symbol.