Creating a Component Using the Multisim Component Wizard

1) Start the component wizard process by selecting the “Create Component” button from within Multisim.

2) In “Step 1” or the “Component Information” portion of the Component Wizard, a new dialog box opens, shown in Figure 1.
   a) Provide the actual manufacturer’s component name along with a detailed description of the device
   b) Select the “Layout only (footprint)” option unless you are provided with a Multisim model from the manufacturer.

   ![Figure 1: Setting the general component information](image1)

3) In “Step 2” or the “Enter Footprint Information” portion of the Component Wizard select a footprint as close as possible to actual component footprint from the “Master Database” as shown in Figure 2.
   a) The number of component pins must also be set for the device.
   b) The device footprint is what will be used by Ultiboard for PCB layout

   ![Figure 2: Selecting the footprint for the new component](image2)

4) In “Step 3” or the “Enter Symbol Information” portion of the Component Wizard, the Multisim symbol used for schematic layout will be edited as shown in Figure 3.
a) Change shape (must first resize boundary box using the \( \square \) button).
b) Change the symbol artwork (lines, shapes, embedded pictures)
c) Change pin locations, spacing, and parameters using the bottom pin assignment table as shown in Figure 4.

Figure 3: Entering the symbol information for the new component

Figure 4: Multisim symbol editor for new component

5) In “Step 4” or the “Set Pin Parameters” portion of the Component Wizard, the pin names can be changed for the component’s associated symbol as shown in Figure 5.
   a) The pin types can also be changed according to their purpose (input, output, power, ground, etc.)
6) In “Step 5” or the “Set Mapping Information” portion of the Component Wizard, the symbol pins are mapped to the footprint pins as shown in Figure 6.

7) In “Step 6” or the “Family Tree” portion of the Component Wizard, the new component is assigned to the desired component database
   a) Select the User database to store new components
   b) A new “Family” must be added if one does not already exist for the new component
8) Select the “Finish” option to finalize new component.
   a) New component can now be selected like all standard components for inclusion in Multisim schematics.
   b) Recall that this component is only for layout purposes, and therefore if used in schematic the design can not be simulated.