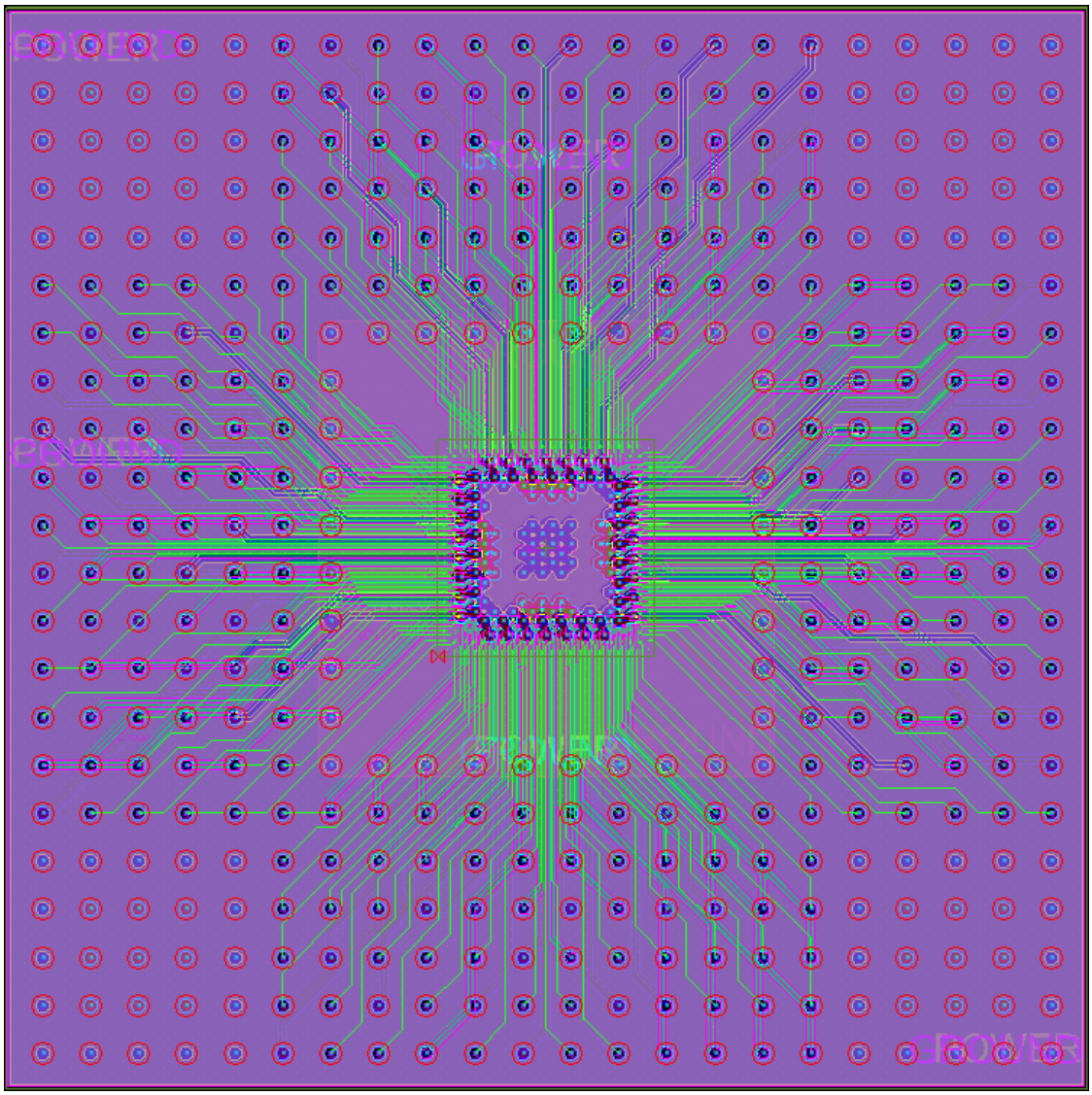
cadence®

ECO Process for Cadence SiP

Product Version SPB17.2
November, 2017



Copyright Statement

Contents

Purpose

A designer can do ECO within Cadence SiP Digital in several different ways. For example, the designer can move a cline or change the cline width. However, it is much more difficult to add or remove a pin to a BGA/LGA or a Die within an existing SiP design.

This application note will go through the flow and process needed to take a BGA with 420 pins and replace it with a component with 421 pins.

Audience

This document is intended for Cadence SiP Digital/Allegro Package Designer users who want to replace an existing component with another component that has a differing pin count.

Terms

- SiP System in Package
- BGA Ball Grid Array
- LGA Land Grid Array
- APD Allegro Package Designer
- ECO Engineering Change Order
- PCB Printed Circuit Board
- RMB Right Mouse Button

Overview

In this application note, you will learn to unplace a component from a design, and then delete the component from the parts list. Next, you will add a new component to the parts list and place the new component in the design. Finally, you will derive the net assignments.

The process flow to replace a component with another one that has a differing pin count is shown below:

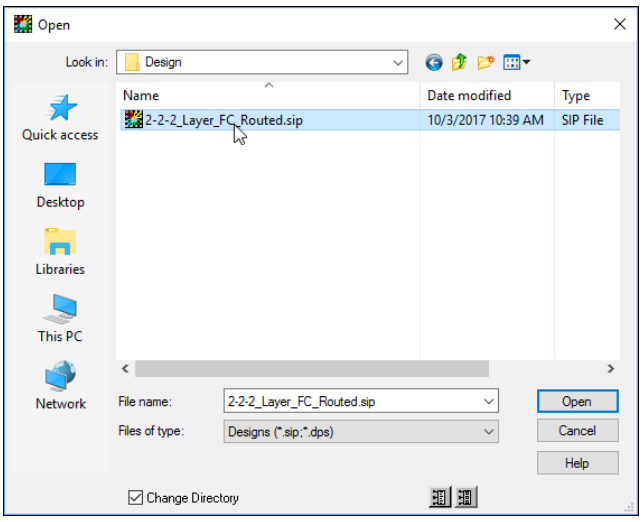
1. Open Database
2. Unplace Component
3. Delete Component from Parts List
4. Add New Component to Parts List
5. Place Component
6. Derive Net Assignments

Procedure

The following steps define the basic process for replacing a package in the design where the pin count has changed.

Open Database

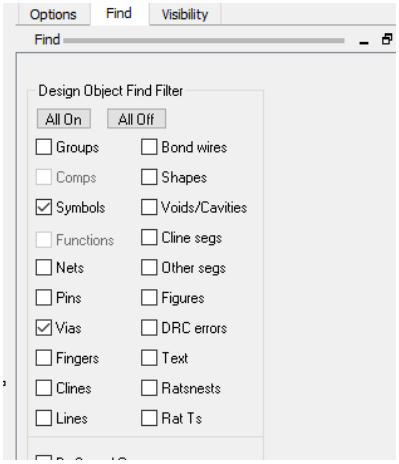
1. Open the database by selecting **File > Open** (C:\project\design\2-2-2_Layer_FC_Routed.sip).



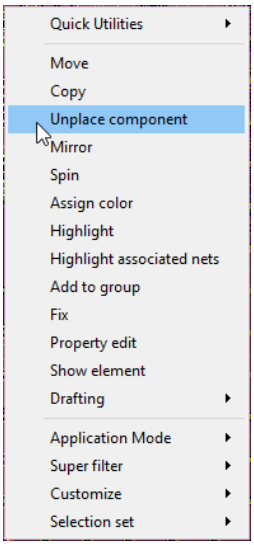
Unplace Component

1. Change Application Mode to General Edit. This is done by selecting **Setup > Application Mode > General Edit**.

2. Select the package by changing the **Find** filter to **Symbols** and selecting the package or die.

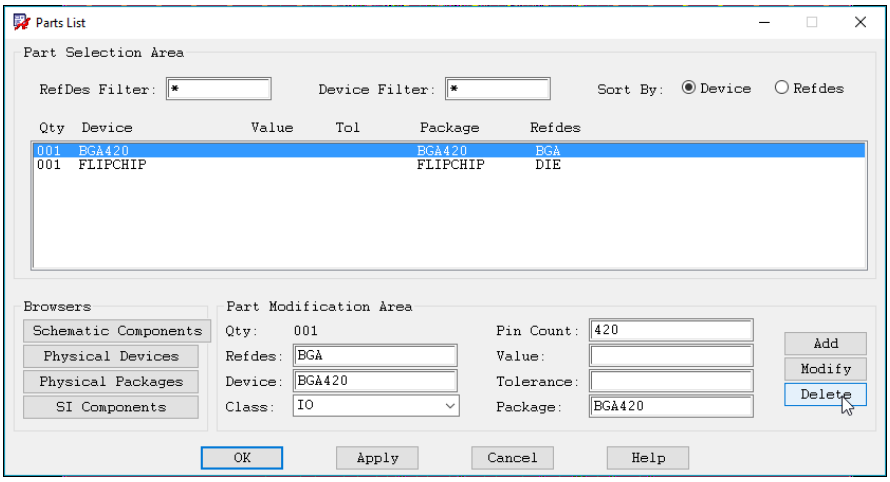


3. Then, hover over the component, **RMB**-select, and pick **Unplace component**. The part will be unplaced.



Delete Component from Parts List

1. Next, you will need to delete the part from the design. This is done by selecting **Logic > Edit Parts List**. The following dialog will display.

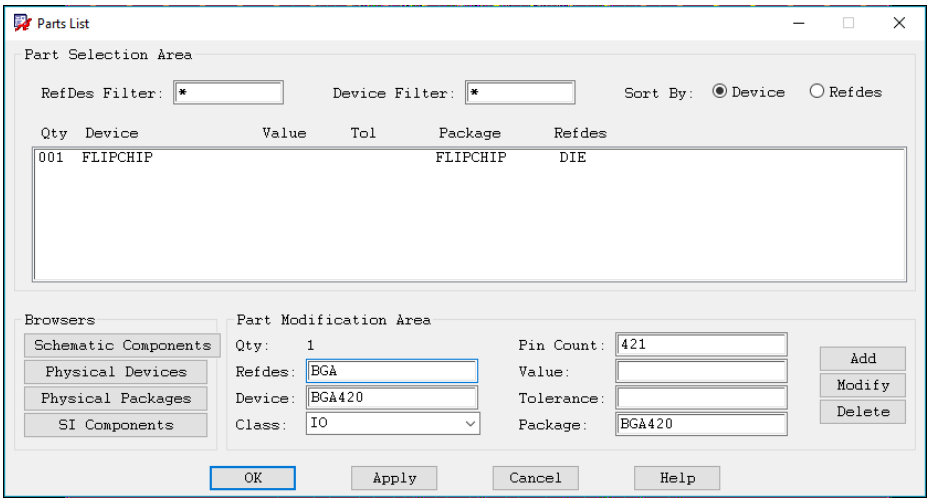


2. Select the part and select the **Delete** button as shown. Then, select **Apply** and **OK**.

   Note: Make sure that the Part.txt (BGA420.txt) file has been saved when the change is made. If the file is not saved, it will reflect the previous pin count.

Add New Component to Parts List

3. Browse for and select the package or the die that you are replacing and make sure that the pin count is different.

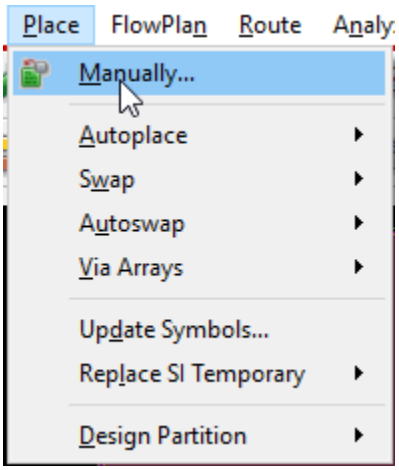


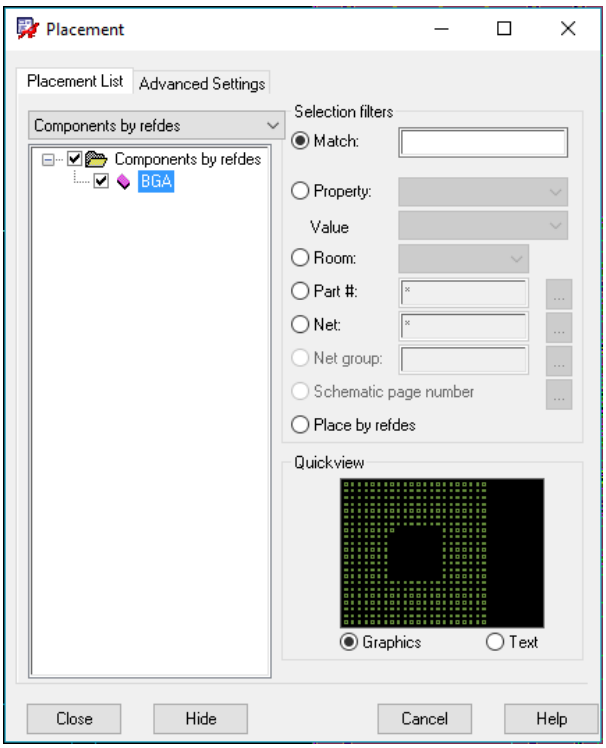
Then, select the **Add** button. Click **Apply** to commit the new part."

4. to commit the new part.

Place Component

1. The next step is to place the part. This is done by selecting in the menu **Place > Manually**.





2. Place the part at 0,0 by typing `x 0 0` at the command line.

Derive Net Assignments

1. Once you have placed the part, go to the menu and select **Logic > Derive Assignment**. Then, drag a window around the design. This will reassign the nets to the part.

2. Save the design by selecting **File > Save**.

   The part is replaced, with the new pin count.

Summary

In this application note, you learned to unplace a component from a design and delete it from the design. Then, you added a new component into the design, placed the new component, and derived the net assignments.

Support

Cadence Support Portal provides access to support resources, including an extensive knowledge base, access to software updates for Cadence products, and the ability to interact with Cadence Customer Support. Visit *https://support.cadence.com.*

Feedback

Email comments, questions, and suggestions to content_feedback@cadence.com.