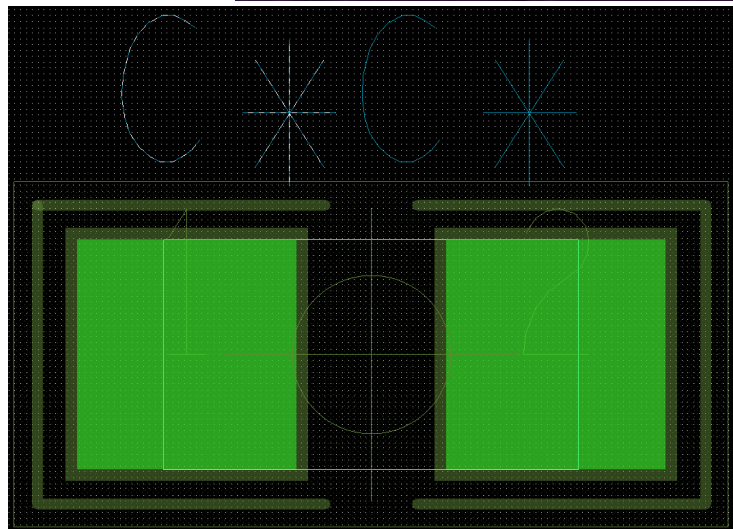


Creating a SMD Footprint in Cadence Allegro



Ricky Liu
998192101
EEC 134

Table of Contents

Abstract	3
Introduction.....	3
Tutorial.....	4
1. Getting Started.....	4
2. Create Pads in Pad Designer.....	4
3. Arrange Pads in PCB Editor.....	5
4. Add Layers/Labels.....	8
Tips and Tricks	11
Glossary	12
References.....	12
Appendix.....	12
Additional resources	12

Abstract

This minimanual presents a tutorial for creating a footprint for a surface mount device (SMD), specifically a surface mount capacitor. At relevant steps, there are explanations of the specific features of the footprint. The software package Cadence Allegro will be used for this tutorial.

Introduction

Printed circuit board (PCB) design is a skill that all electrical engineers obtain in their careers. One of the most important processes of PCB design is creating footprints, an arrangement of pads on a PCB used for attaching electronic components to the circuit board. The process can be broken down into three simple steps, as shown in figure 1.

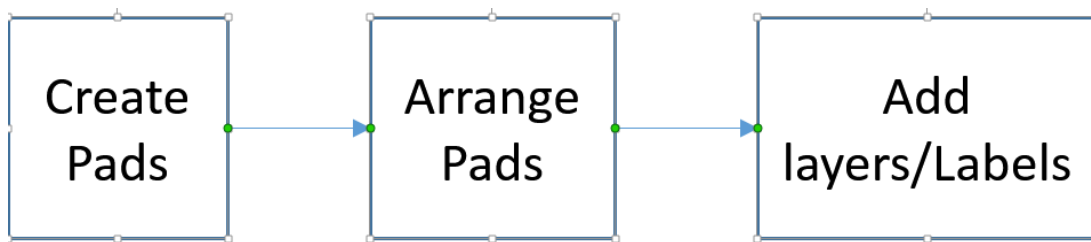

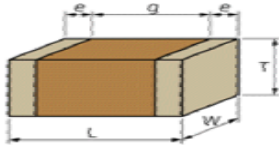


Fig. 1 Footprint Creation Steps

Footprints cannot be created using any arbitrary shapes and sizes. The design specifications, including pad dimensions, are in the component's datasheet. This tutorial uses a two pin SMD capacitor as an example. The dimensions are listed in figure 2.



L size	1.6 ±0.1mm
W size	0.8 ±0.1mm
T size	0.8 ±0.1mm
External terminal width e	0.2 to 0.5mm
Distance between external terminals g	0.5mm min.
Size code in inch(mm)	0603 (1608)

Fig. 2 SMD Capacitor Dimensions

Tutorial

1. Getting Started

The following programs must be installed onto your computer and opened before starting this tutorial:

- Pad Designer (Cadence SPB 16.5/16.6 -> PCB Editor Utilities -> Pad Designer)
- PCB Editor (Programs -> Cadence -> Release 16.6)

2. Create Pads in Pad Designer

1. Click the Units drop-down menu and select millimeters in the 'Units' section of the 'Parameters' tab.

NOTE: The unit mils (1/1000 of an inch) is another common unit of measurement.

2. Check the box next to 'Single Layer Mode' in the 'Layers' tab.
Checking this box will limit the number of layers between the top and bottom copper layers to one layer.

3. Select the 'BEGIN LAYER' row. Click the 'Geometry' drop-down menu and select 'Rectangle'. Input 0.2 in the 'Width' box and 0.8 in the 'height' box.

TIP: The begin layer is the top copper layer of the PCB.

4. Right click 'Bgn' button and click copy. Right click 'SOLDERMASK_TOP' row and click paste. Paste onto the 'PASTEMASK_TOP' row.

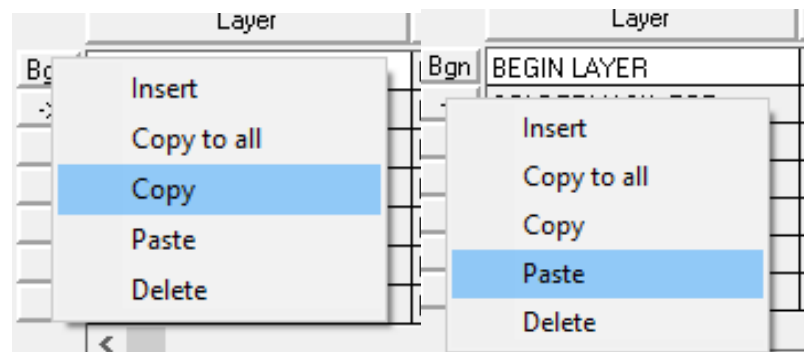


Fig. 3 Copy/Paste

TIP: The 'SOLDERMASK_X' and 'PASTEMASK_X' widths and heights can be 0.1 mm bigger than the 'BEGIN_LAYER'.

5. Save the pad and name it GRM188R61C474KA93.

The file should be named GRM188R61C474KA93.pad. The final pad configuration should look like Fig. 4.

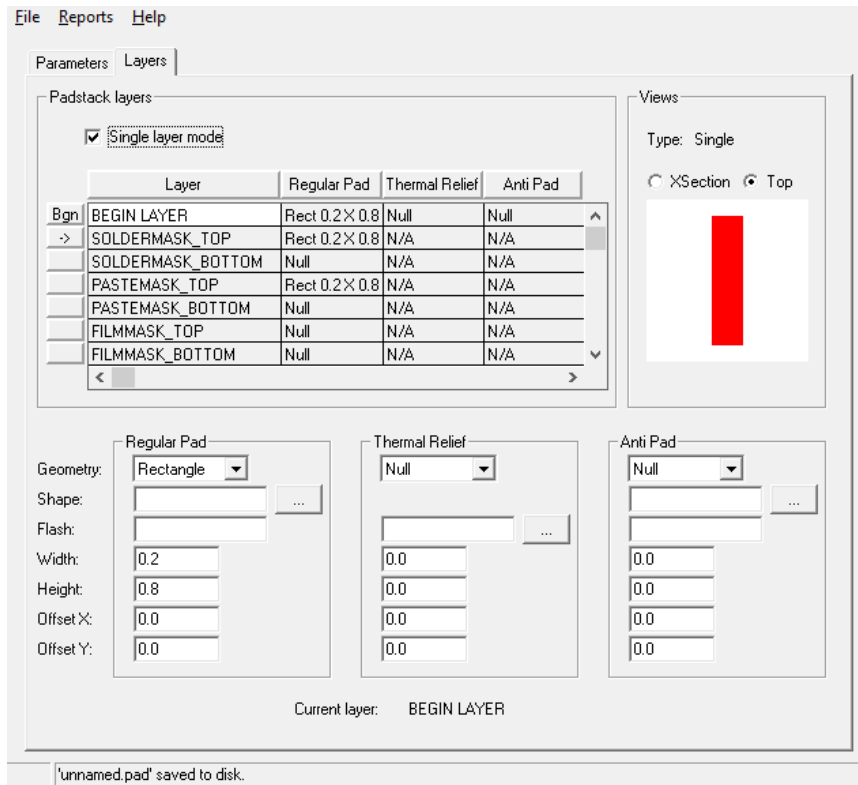


Fig. 4 Pad Configuration

3. Arrange Pads in PCB Editor

1. Click File -> New.
2. Choose 'Package Symbol', input 'cap0603' in the 'Drawing Name' box.
3. Set the folder that the footprint will be saved.

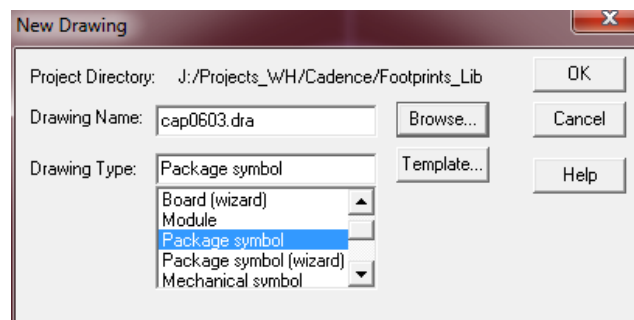



Fig. 5

4. Click Setup -> Design Parameters. Click the 'Design' tab. Change the 'User units' to millimeters.
5. Click the 'Display' tab and click  to set up the grids.
6. Change all 'x' and 'y' boxes to 0.0254.
Changing the 'x' and 'y' values makes each dot on the grid 0.0254 mm (1 mil) apart.
7. Click 'OK' to close the 'Define Grid' window. Close the 'Design Parameter Editor'.
8. Click Layout -> Pins. Click 'Options' on the right side of the window, as shown in Fig. 6.

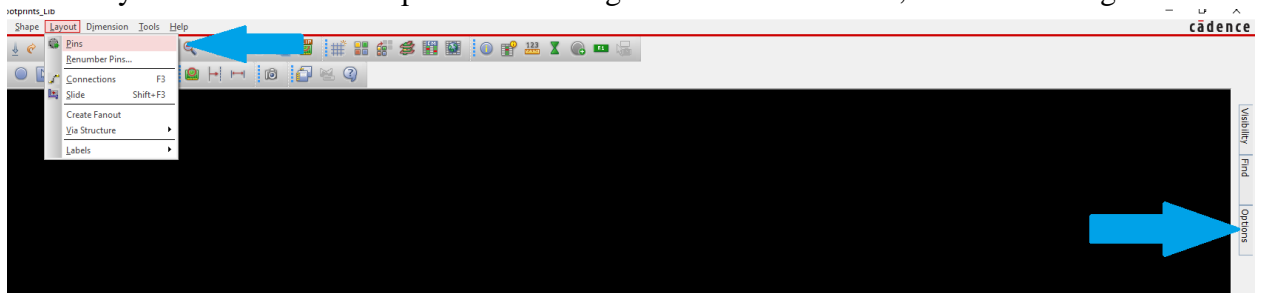


Fig. 6.

9. Choose GRM188R61C474KA93 for 'padstack', input 1.6 in the 'Spacing' box in the 'x' row, and 0.5 in the 'Spacing' box in the 'y' row, as shown in fig. 7.

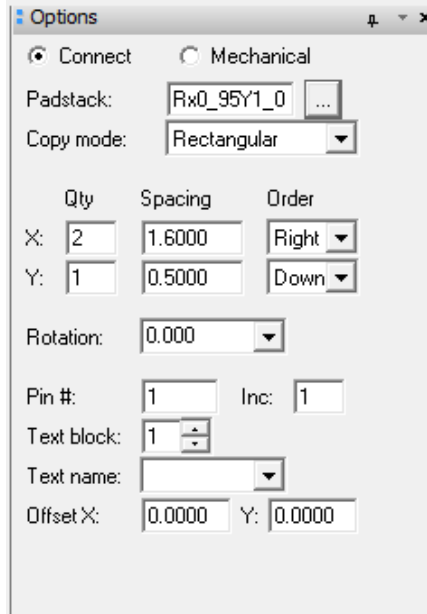


Fig. 7

10. Click anywhere on the workspace (black grid) to place the pads.

NOTE: The black grid is the area of the PCB Editor program where the footprint is placed.

11. Save the file as GRM188R61C474KA93.dra.

WARNING: Several windows will pop up warning you that the footprint is missing features. Close the windows and continue to the next section.

4. Add Layers/Labels

1. Click Shape -> Rectangular. Click Options on the right side of the window.
2. Choose 'Package Geometry' from the top drop-down menu and choose 'Place_Bound_Top' in the bottom drop-down menu, as shown in fig. 8.

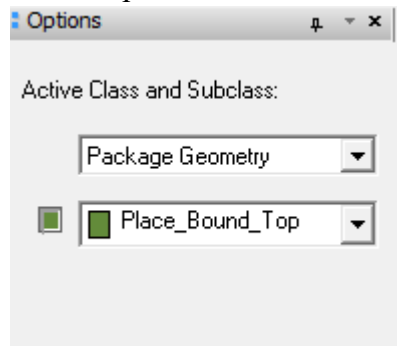


Fig. 8

3. Place the rectangle onto the workspace so that the rectangle is surrounding the pads, as shown in fig. 9.

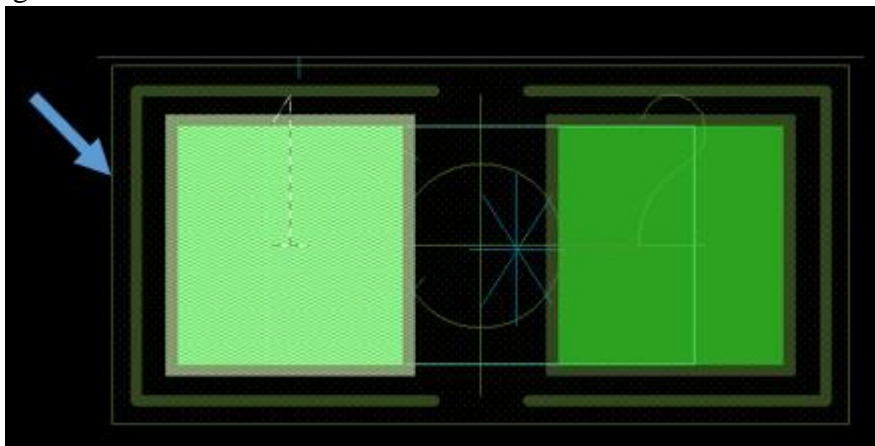


Fig. 9

4. Click Add ->Line. Click Options on the right side of the window.
5. Choose 'Package Geometry' from the top drop-down menu and choose 'Silkscreen_Top' from the bottom drop-down menu.
6. Set Line lock: line , 90
7. Set Line width: 0.05
8. Draw the lines around the pads, leaving 0.05 mm between the pad edge and the line, as shown in fig. 10.

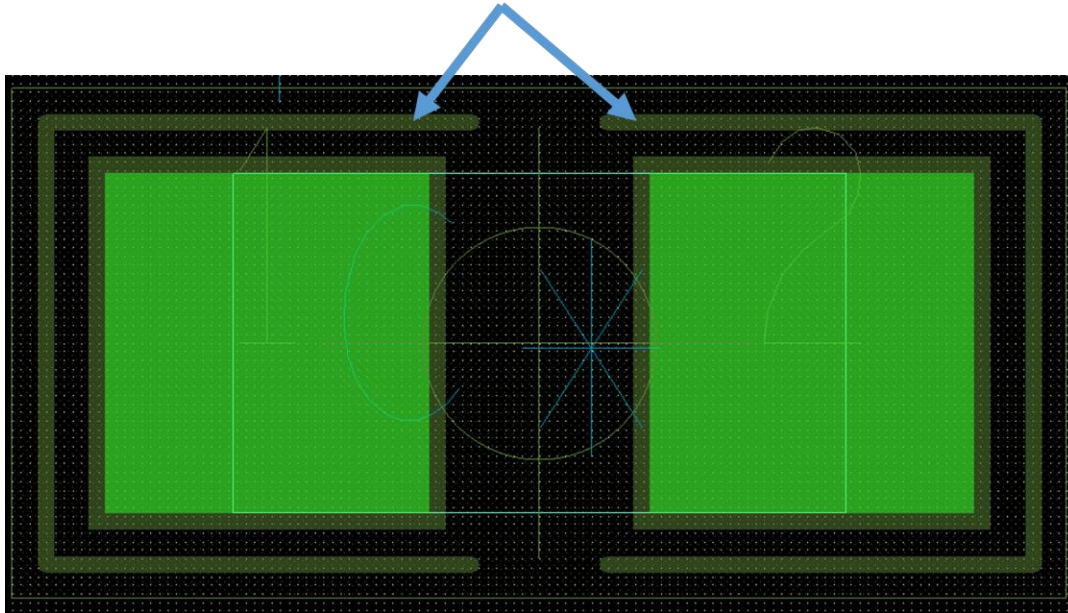


Fig. 10

9. Repeat steps 1 and 2, except choose 'Assembly Top' from the bottom drop-down menu.
10. In 'Shape Creation', Select 'Place Rectangle' and set the width and height to be 1.8 mm and 1 mm, respectively.

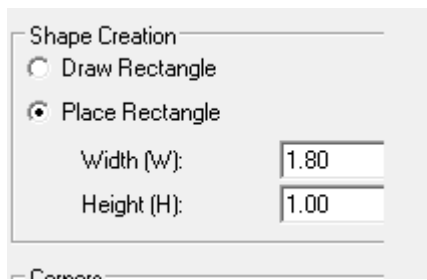


Fig. 11

11. Place the rectangle in the workspace as shown in fig. 12.

NOTE: The center of the rectangle should be aligned with the origin.

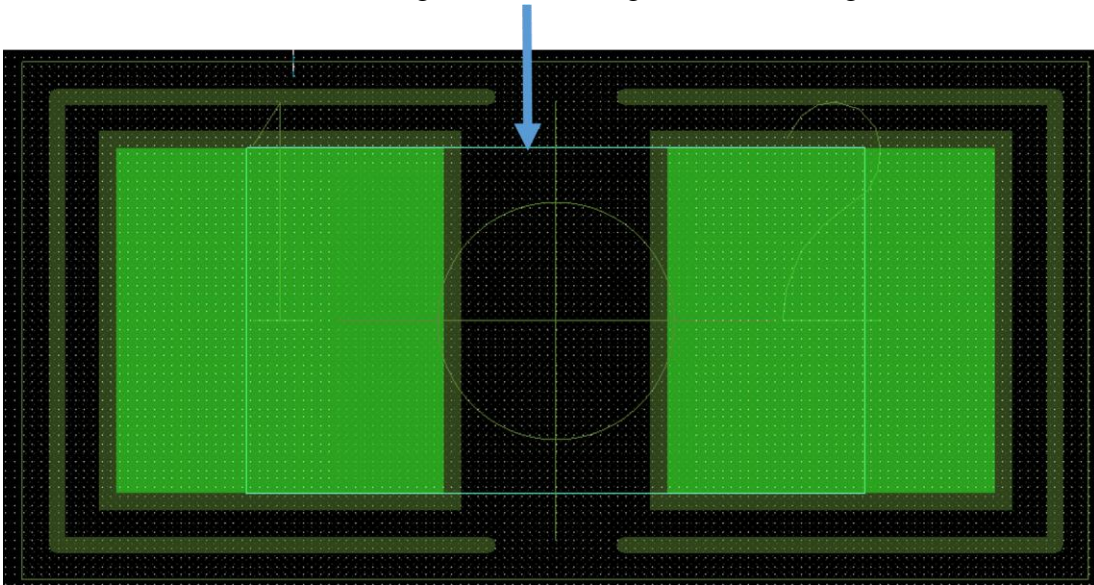


Fig. 12

12. Click Layout -> Labels -> RefDes

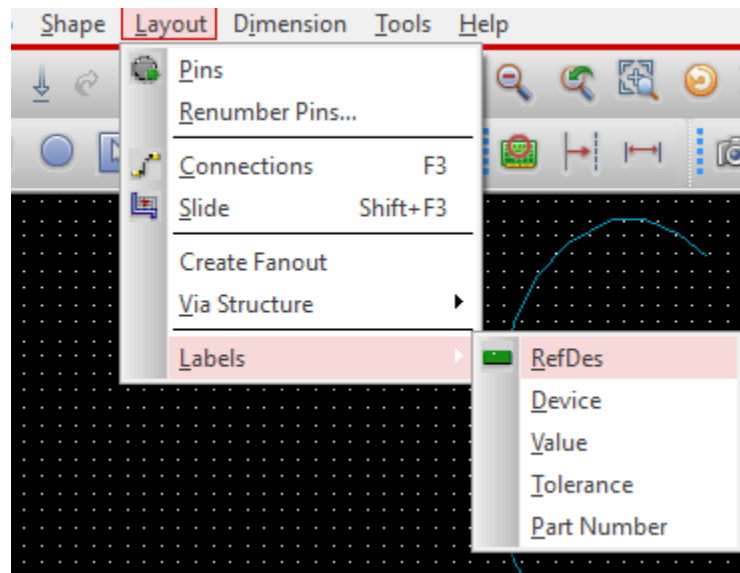


Fig. 13

13. In Options, select 'Assembly Top' from the drop-down menu.

14. Click anywhere in the workspace and type 'C*'.

15. Repeat steps 12-14 for 'Silkscreen Top'. The symbol should look like fig. 14.

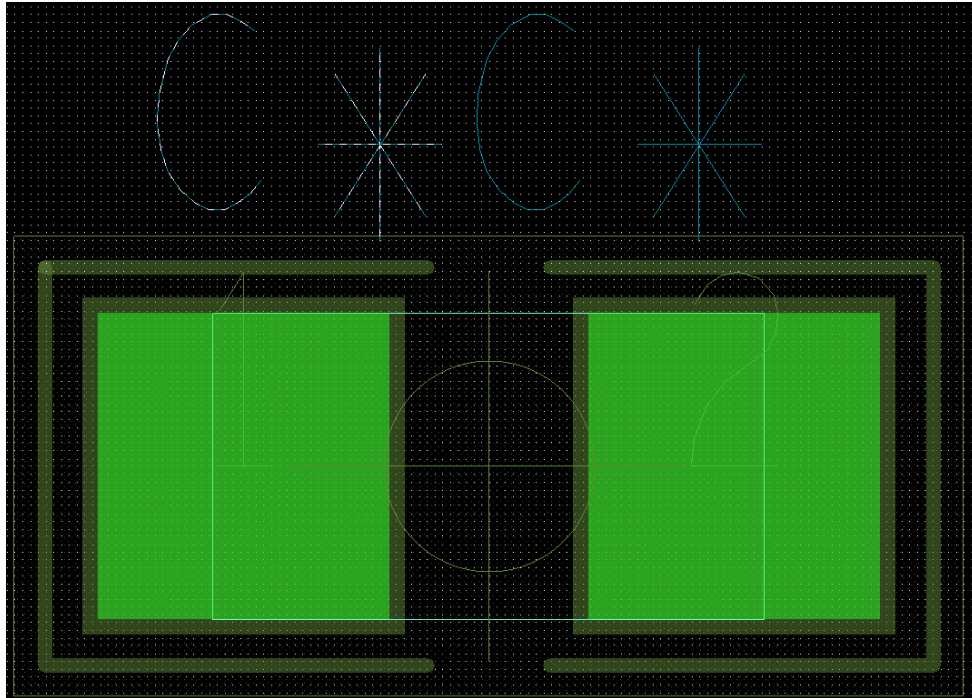


Fig. 14

16. Save the file.

NOTE: The footprint is finished when the following layers are present:

- Bound_Top
- Silkscreen_Top
- Assembly_Top
- RefDes Silkscreen
- RefDes Assembly Top

Tips and Tricks

- Practice undoing actions by right-clicking and pressing 'oops'.
- The black grid workspace can be moved by pressing and holding the middle button on the mouse and moving the mouse.
- Use the visibility tab on the right side of the screen to show and hide desired layers.
- The options tab can be pinned to the workspace by clicking the thumbnail button.



Glossary

Assembly Layer: layer that contains the information about the placement and orientation of all the components on the circuit board.

Bound Layer: layer that contains the outline of the footprint.

Footprint: graphical symbol composed of pads used for connecting electronic devices to a circuit board.

Pad: also known as pins; surface of the circuit board designated for soldering electronic components.

Pastemask Layer: layer that contains information about the places on the circuit board where there should be no pastemask.

Pins: leads on an electronic device for connecting to other circuit components; **pin** is also used interchangeably with **pad** when referring to the footprints.

Reference Designator (RefDes): A standardized method of identifying a device on a printed circuit board. Typical reference designators begin with a letter, followed by a number.

Silkscreen Layer: layer that contains the labels and outlines of every component.

Soldermask Layer: layer that contains the information about the places on the circuit board where there should be no soldermask.

References

“Capacitor Data Sheet”. GRM188R61C474KA93. Murata. April, 2016

“Complete PCB Design Using OrCAD Capture and PCB Editor”. Mitzner, Kraig. Elsevier, 2009. Print.

“GRM Series” *Murata* April 30, 2016. <<http://www.murata.com/en-us/products/capacitor/mlcc>>

Appendix

Additional resources

http://www.referencedesigner.com/tutorials/allegro/allegro_page_1.php

http://www.referencedesigner.com/tutorials/allegro/allegro_page_5.php