Title: DFA ((Dynamic) Design For Assembly)

Product: Allegro PCB Designer

Summary: This Application Note describes how to use the DFA functionality and the different settings.

Author/Date: Marco Waller / 19.11.2011
Update/Date: Pascal Willems / 12.12.2013

Table of Contents

1 Introduction .................................................................................................................. 2
2 Library Settings before using the DFA Functionality .................................................... 2
   2.1 How to add a DFA Boundary ................................................................................. 2
   2.2 How to categories several symbol automatically .................................................. 4
3 Manage the Package to Package distances .................................................................. 5
   3.1 Load the Symbols to the DFA Spreadsheet ......................................................... 5
   3.2 Add symbols to classes in the Spreadsheet Editor ................................................ 6
   3.3 Define the distance between the Symbols ............................................................. 7
4 Use of the DFA Functionality during component placement ....................................... 8
5 Additional settings / FAQ ............................................................................................ 9
   5.1 Controlling the resistance of the spacing bubble ................................................... 9
   5.2 The DFA DRC’s are not visible ............................................................................. 10
   5.3 Not seeing DFA circles? .................................................................................... 10
   5.4 Which edge is Package Side and End? ................................................................. 11
   5.5 Read-Only Version ............................................................................................. 11
6 Additional Information ............................................................................................... 12
1 Introduction

Miniaturisation is a leading trend in today’s PCB requirements. But you have some minimal spacing between components to respect during the placement. These limitations are defined by the type of soldering or placement process. Ask your print manufacturer or assembler for the minimum component spacing which you have to obtain. It also exist an IPC Guidline with the minimum spacing requirements.

This application note describes how you can use the DFA functionality in PCB Editor to optimize the component placement.

2 Library Settings before using the DFA Functionality

Each component has a placement boundary. When two components are overlapped placed, the placement boundary creates an error (DRC). Now for the DFA check you have to define an additional DFA boundary.

Maybe your footprints already have a DFA boundary. Otherwise you can add a DFA boundary automatically.

2.1 How to add a DFA Boundary

The DFA Update Tool adds a DFA boundary automatically to your component.
You find this utility in following folder:
Windows Start Button → Alle Programme → Cadence → SPB16.6 → PCB Editor Utilities → DFA Symbol Update

DFA place-bounds can be created on the DFA_BOUND_TOP and DFA_BOUND_BOTTOM layers of the PACKAGE GEOMETRY class, similarly to existing place-bounds. The dfa_update command adds DFA place-bounds to the DFA_BOUND_TOP layer only.
When selecting a single symbol, you can convert the selected symbol.

When you want to update the entire directory, you can replace the symbol name with an * asterisk. Then it looks like following printscreen:

After converting your footprints, you should verify and modify the new DFA boundary to meet your manufacturing requirements.
2.2 How to categories several symbol automatically

In the DFA spreadsheet you can further define the distance between all the different footprints. It makes it easier when you can define the DFA Distances within different classes. You can categories several symbols when adding the DFADEVICE_CLASS property to the symbols. To add this property to several components you can also use the DFA Symbol update utility. In the DFADEVICE_CLASS property field you can enter the name of the desired class.

When your symbols are named with a special characteristic, you can filter them. For example when you want to classifies all the symbols which starts with the name bga in a separate class, you can use following method:
3 Manage the Package to Package distances

After defining the DFA Boundary and the DFA_DEV_CLASS you can define the distance between the symbols in the DFA Spreadsheet Editor.

You can start the DFA Spreadsheet from:
- Windows Start Button → Alle Programme → Cadence → SPB16.6 → PCB Editor Utilities → DFA Spread Sheet Edit.
- Within the PCB Editor you can start the tool from:
  Setup → Constraints → DFA Constraints Spreadsheet or within the tool with following Icon:

3.1 Load the Symbols to the DFA Spreadsheet

After opening the DFA Spreadsheet editor you need to add the symbols or the classes to the spreadsheet. Afterwards you can define the distance between all the components.

DFA Spreadsheet Editor:
3.2 Add symbols to classes in the Spreadsheet Editor

You may have some symbols in the package folder, which are not categorized in a class. You can move a symbol to a class with following process within the spreadsheet editor:

1. Select the symbol
2. Click right mouse button
3. Chose “Cut selected items”
4. Select the class
5. Click right mouse button
6. Chose “Paste to class”

You can also define a new Class.

1. Enter the name of the new Class in the "new class name field"
2. Press the [Tabulator] key
3. Paste a part into the new class (As described above)
4. Click on the [update] button
3.3 Define the distance between the Symbols

With the following process you can define the distance between the symbols:
- Enter the distance-value in the default-field
- Select the cells on which you want to apply the distance
- Press apply to selected cells

You can also define the distances manually in each field.

After entering all the values, you can copy the values to the bottom side by clicking on the [Copy top table to bottom] button.
The values on the bottom side can be different from the top side values.
At the end the table will be stored in a *.dfa file. Click yes if you are asked if you want to Copy top to bottom table.

After defining or loading a spreadsheet, the DFA functionality needs to be enabled. Just turn the DRC mode to On (Only possible when DFA constraints window is started from PCB Designer):
4 Use of the DFA Functionality during component placement

For using the DFA feedback functionality, the place manually command is required. If you want to use the DFA functionality during a move command, you also have to use the place manually command or Place Manual -H. When placing a component close to another, a circle (spacing bubble) appears which represents the minimum distance between the components. The size of the circle spacing match’s with the value in the DFA spreadsheet. This allows you to place the component at the minimum spacing. When a component is placed to close, a DRC appear.

Minimum DFA distance: Distance between components is too small

![Diagram showing minimum DFA distance and when distance is too small.]
5 Additional settings / FAQ

5.1 Controlling the resistance of the spacing bubble

The resistance can be controlled by the “dfa_pause_level” variable located in user preference → Placement → DFA. Set to 0 for no resistance.

Set to 3 for maximum resistance. When a component is slowly approached to another it cannot get closer as the DFA Constraint Value.
5.2 The DFA DRC’s are not visible

Please activate in the Stack-Up folder the Package_Top or Package_Bottom DRC subclass to display the DFA DRC’s.

5.3 Not seeing DFA circles?

Enable the DFA DRC mode and use the Place Manual command:

Or use the Place Manual -H command:
5.4 Which edge is Package Side and End?

The software determines the package side and end based on the longer and shorter dimensions of the package symbol.

End is always the shorter dimension of the package symbol.

Attention: Take care, the DFA boundary is not the same as the component case size. It contains also the pins. In the following figure, an 8 pin SOIC shows the side where you normally might think the end is located.

![Package Side and End](image)

5.5 Read-Only Version

The stand-alone version of the DFA Spreadsheet Editor offers a setting to save the file in a read-only mode. When the PCB Designer opens the file, he will not be able to modify the contents. A read-only file can be opened by the development version of the spreadsheet editor and modified.
6 Additional Information

For more information please contact FlowCAD:

<table>
<thead>
<tr>
<th>Switzerland</th>
<th>Germany</th>
<th>Poland</th>
</tr>
</thead>
<tbody>
<tr>
<td>+41 (0)56 485 91 91</td>
<td>+49(89) 4563-7770</td>
<td>+48 (58) 732 74 77</td>
</tr>
<tr>
<td><a href="mailto:support@FlowCAD.ch">support@FlowCAD.ch</a></td>
<td><a href="mailto:support@FlowCAD.de">support@FlowCAD.de</a></td>
<td><a href="mailto:support@FlowCAD.pl">support@FlowCAD.pl</a></td>
</tr>
</tbody>
</table>