



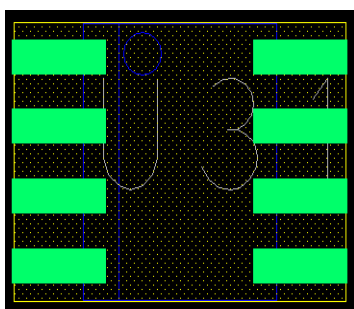
Introduction

DFA (Design for Assembly) technology available with Allegro PCB Designer offers real-time package-package clearance checking during interactive component placement. Driven from a 2-dimensional spreadsheet array of classes and package instances, real-time feedback in the form of a spacing circle provides minimum clearance requirements based on the package's side-side, side-end, end-end or end-side profiles. With thousands of components on today's boards, real-time DFA analysis feedback increases the Designer's productivity and efficiency by placing components to corporate or EMS (Electronic Manufacturing Services) guidelines and helps reduce the dependency on referencing specifications or interfacing with 3rd party CAM systems.

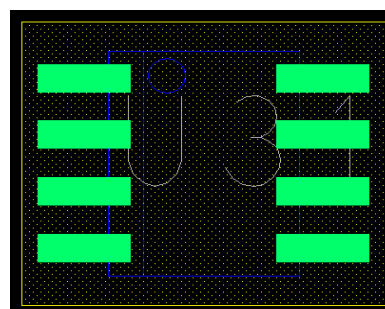
Adoption Considerations

The new DFA application is not considered "use out of the box" technology. Adoption, especially with larger companies, requires some planning between CAD and Manufacturing Groups. When considering the DFA application, a review of your package symbol placebound shapes against the clearance parameters is necessary. Questions to consider:

1. Are your current placebound shapes drawn tightly around the package extents not accounting for clearance requirements to other packages? If so, you may not require migration to the new DFA placebound shape.
2. Are your current placebound shapes drawn outside the extents of the pins or assembly outline to account for clearance requirements? If so, Cadence recommends incorporating the new DFA placebound shape which can be automatically generated with the utility, DFA_Update.



Compact placebound geometry



Extended placebound

Legacy placebound shapes may be drawn to include the space required for rudimentary clearance checking and may not integrate well with DFA Spreadsheet-driven rules. Although the DFA checker reverts to the legacy placebound shape if a DFA placebound is not found, it is estimated this model will be inaccurate for many companies because it lacks precision due to the static nature of a single placebound shape for multiple combinations of clearance requirements. Customers are cautioned to sample the output of DFA_Update to ensure conformance with their package spacing guidelines.

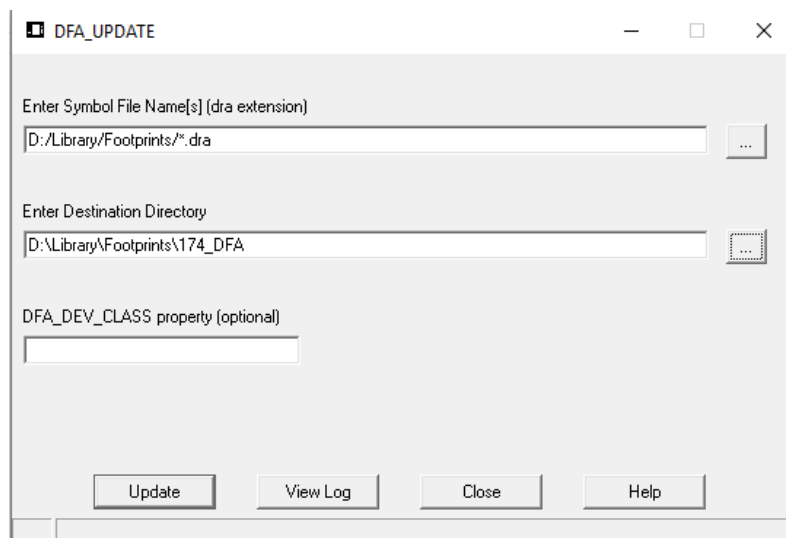
Uprev Package Symbol Library

DFA_Update provides an automated solution to uprev all or a family of package symbols requiring the addition of the DFA placebound shape and the property DFA_DEVICE_CLASS. The auto-generated rectangular placebound shape is drawn about the extents of the symbol's pins and assembly outline. The property, which may exist in your current library, allows package symbols to be classified into class-based objects. The utility can be run from the Start-Run command if on a PC. The benefit of applying the DFA_DEVICE_CLASS property to library-based symbols is that it allows the DFA Constraint Spreadsheet to be viewed hierarchically with the option of reducing the number of instances in the spreadsheet. The Purge Grouped Symbols control, located within the spreadsheet, removes symbol instances belonging to classes. Candidates for class-based entries include symbols with common spacing requirements. The class may be composed of a single family of packages, like all BGAs or heterogeneous combinations like SOICs and TSOPs.

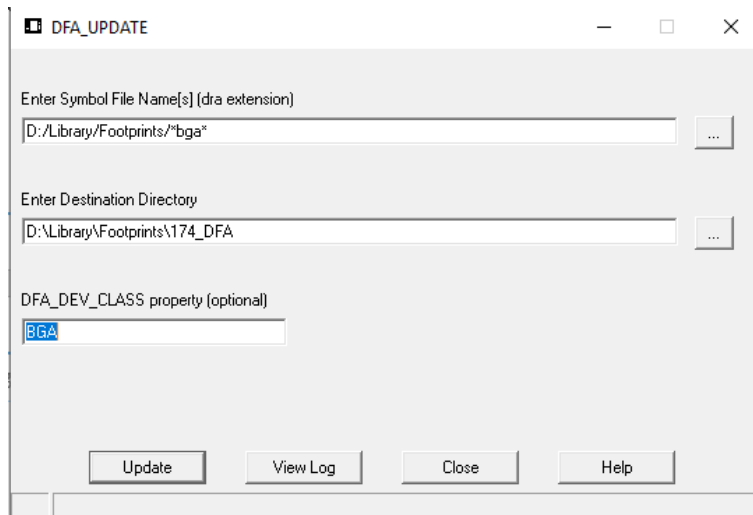
When using this utility, Cadence recommends reading symbols from your current library but writing to a new library. In the example below, the DFA_Update utility is used to update all .dra files in a 17.2 library. It adds the DFA placebound shape and writes the output to a 17.4 library. Properties are not recommended when performing a global update.

Steps:

1. From Windows 10 Start – Cadence PCB Utilities 17.4-2019 – DFA Symbol Update 17.4
2. In the Enter Symbol Name field, browse to the location of any .dra file in your current library.
3. Edit that symbol name reducing it to the suffix *.dra as shown in the following figure.
4. In the Enter Destination Directory field, browse to the location of the newly created 16.5 library name.
5. Leave the DFA_DEV_CLASS property field blank.
6. Click Update then review the log file.
7. Sample a variety of unique symbols for company compliance.

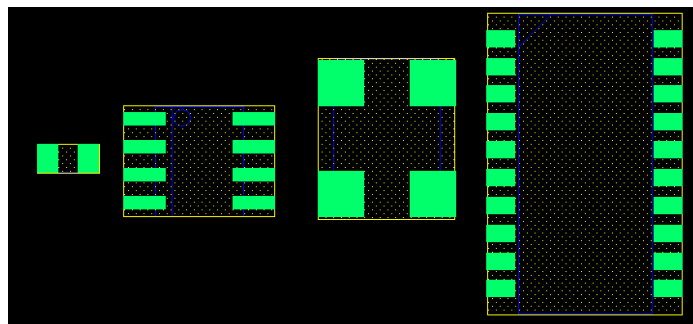


In the next figure, all BGA are updated and assigned a class property of BGA because they all have the same clearance requirement.

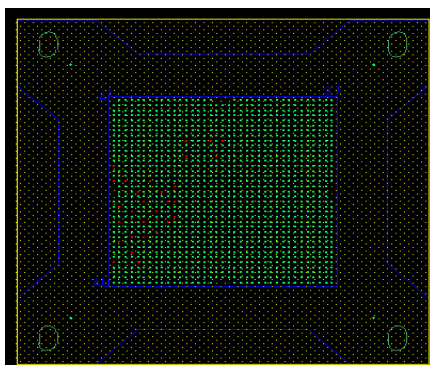


Sampling Auto-Generated DFA Placebounds

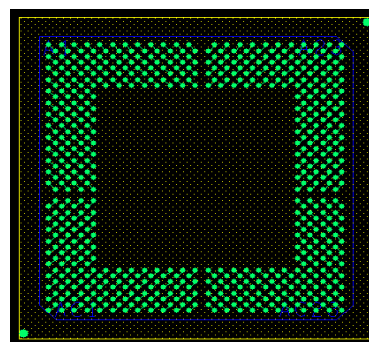
It is critical that you sample several but unique packages before releasing the library to production. The results from DFA_Update, in some cases, may not conform to your company guidelines. The following figures are examples of auto-generated DFA placebounds. The two new subclasses, dfa_bound_top and dfa_bound_bottom, are located in the Package Geometry Class.



DFA Placebound Shapes in Yellow, Assembly Outline in Blue



Complex Assembly Outline

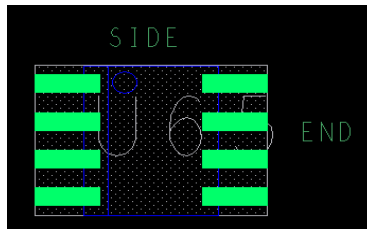


Package with Fiducials

Package Side and End

The software determines the package side and end based on the longer and shorter dimensions of the package symbol. If Allegro PCB Editor cannot determine the side or end, it uses the most conservative value of the string entered in the spreadsheet for that particular package to package requirement. For example, if a cell contains a string of 25:50:100:50, (mils) the value of 100 is considered the most conservative. Package symbols built to a square or with more than four sides would be examples where the software cannot determine between a side or an end.

Warning: Do not assume the leaded edge of a package is the side. In the following figure, an 8 pin SOIC shows the side where you normally might think the end is located.



Package Side and End

DFA Constraint Spreadsheet

Package symbol spacing rules for Side-Side, End-End, Side-End and End-Side profiles are entered into cells in the spreadsheet, which supports top and bottom side views. The spreadsheet has a *.dfa extension and can be opened by choosing Setup>Constraints>DFA Constraint Spreadsheet or as a stand-alone application. DFA files can be stored on disk and be controlled by the design path variable DFACNSPATH.

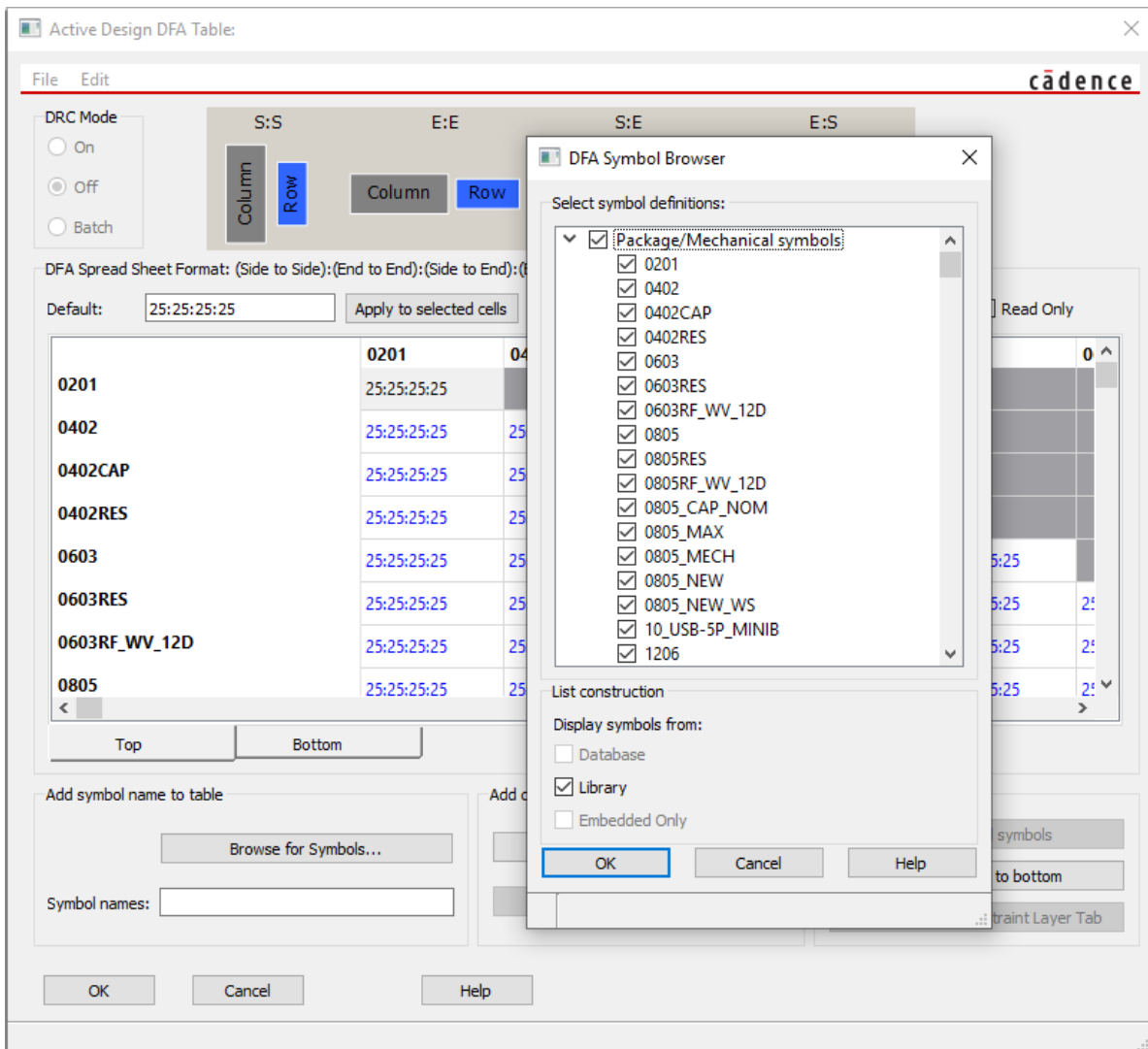
Purge controls help reduce the number of unused symbols or symbol instances belonging to class objects. The Show Symbol Classifications utility provides an alternative, yet more intuitive method for adding the DFA_DEV_CLASS property to package symbols. It also provides a viewing mechanism for the class structure.

A stand-alone, unlicensed version of the DFA Constraint Spreadsheet, shown in the following figure, is available provided the user has an Allegro PCB Editor environment installed. The stand-alone version is recommended for development and offers the ability to write class information to the library. These controls, found in the Show Symbol Classification user interface, are disabled in the PCB Editor version. The stand-alone DFA Spreadsheet can be launched from the Start menu.

Upon opening a blank spreadsheet, a two-dimensional array of package symbols can be quickly created using the following steps.

1. From Start – Cadence PCB Editor Utilities 17.2-2019 – DFA Spreadsheet Editor 17.4, launches the DFA development spreadsheet. Note all these tasks can be accomplished by using the spreadsheet within PCB Editor except for writing properties to the library.
2. Click the Browse for Symbols button then enable the box adjacent to Package/Mechanical symbols. At this time, all symbols in your PSMPATH are being read into the dialogue shown in the Display Symbols from Library Control. If you invoke the spreadsheet from PCB Editor, you can display symbols from either the active database or the Library.
3. Click the OK button and your table should be generated very quickly.
4. To apply a global spacing value, enter a value in the Default cell.

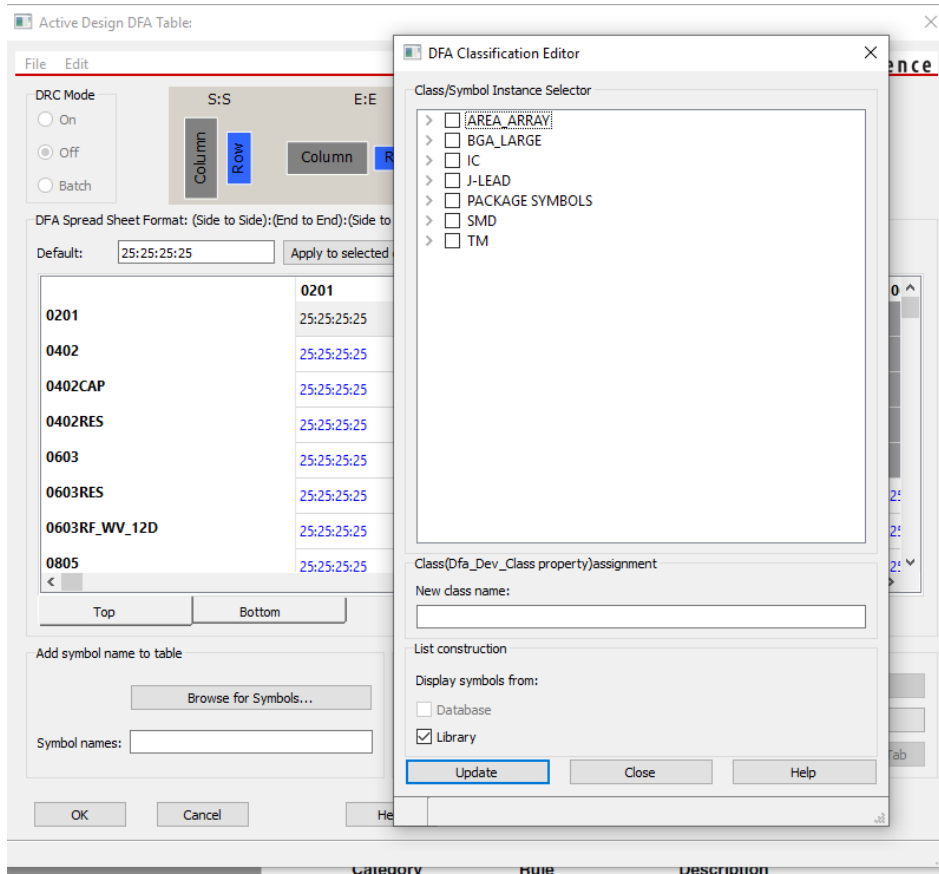
- Click on the grey cell in the upper left corner of the grid to highlight all cells. Alternatively, you can select cells similar to the Excel model clicking your left mouse button.
- Click on the Apply to Selected Cells button to apply the spacing value to all cells.



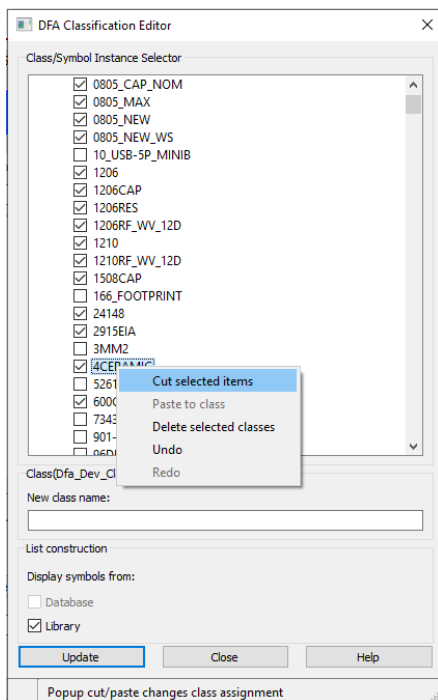
Class-Based Considerations

Table sizes based on sourcing libraries are expected to have hundreds or thousands of entries. Classifying package symbols into hierarchical structures can help reduce the size of the table significantly. As mentioned earlier in this paper, the DFA_Update utility can be used to add this property. Alternatively, the property can be assigned using the Show Symbol Classifications utility found in the DFA Spreadsheet as shown in the following figure.

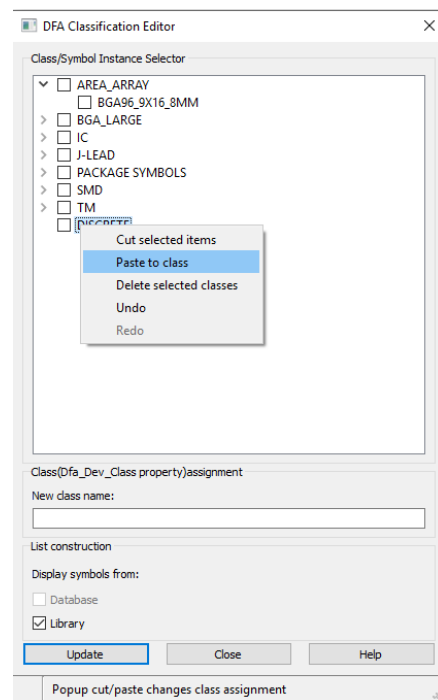
- Click on the Show Symbol Classification button.
- If your library symbols already have DFA_DEVICE_CLASS properties assigned, those values are shown in the UI along with the list of instance-based package symbols under the Package Symbol category.
- Select the class entry then click the UPDATE button to transfer them to the table as shown below.
- The class-based entries are located at the beginning of the table. A blue box surrounds the values. Note the values inherit the setting of the Default cell.



New classes and their associations can also be managed within the DFA Classification Editor. Enter a new class name in the New Class Name cell followed by clicking the TAB key. Package symbol assignments are done by selecting the specific instances followed by the cut/paste actions of the right mouse button (see following figures). Upon completion, clicking the Update button not only moves the new classes to the table but writes the DFA_DEVICE_CLASS to the library-based symbols. A pop-up prompt advises the user of this action.



Cut symbols



Paste into new Class

Overriding Cells

The method for editing cells is similar to that used in Microsoft Excel. Drag cells with the LMB or use the control or shift keys to extend the selection. In the figure below, the cell entry of 15:20:25:15 represents the syntax for S-E-E:S-E:S-E.

Default:	15:20:25:15	Apply to selected cells	Millimeter
	0201	0402	0402CAP
0201	25:25:25:25		
0402	25:25:25:25	25:25:25:25	
0402CAP	25:25:25:25	25:25:25:25	25:25:25:25
0402RES	15:20:25:15	25:25:25:25	25:25:25:25
0603	15:20:25:15	25:25:25:25	25:25:25:25
0603RES	15:20:25:15	25:25:25:25	25:25:25:25
0603RF_WV_12D	15:20:25:15	25:25:25:25	25:25:25:25
0805	25:25:25:25	25:25:25:25	25:25:25:25

Table Compression

Before releasing the table to production, the owner may elect to reduce the number of entries, (assuming class hierarchy exists), by removing symbol instances. This action is done from selecting the Purge Classified Symbols button. This step can be applied in the Allegro PCB Editor on a design instance basis.

Table Access from DesignPath Variable

DFA Spreadsheet files should be archived in a central location for the design team to access. A design path variable, DFACNSPATH is used to set the default location of these files. The PCB Designer can elect to use the file browser to navigate for .dfa files or the data browser which defaults to the location set by this variable.

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.

Loading Spreadsheet into Allegro PCB Editor

The PCB Designer can load only one spreadsheet into Allegro. If a change is made as to how the board is assembled or soldered, additional spreadsheets can be applied to overwrite what is stored in the database. From the Setup menu, select DFA Constraint Spreadsheet. To open a new spreadsheet, select one of the commands listed below:

- File - Open: standard file browser searching for .dfa files.
- File - Browse DFA Library: search path aligned with DFACNSPATH variable.

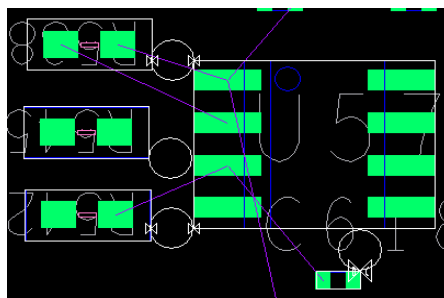
The spreadsheet will be stored inside the Allegro PCB Editor database assuming a save has been performed.

A new category PkgToPkg Spacing has been added in the Design for Assembly constraints. You can now maintain the DFA table within Constraint Manager and assign different DFA rules based on stack up technology, specifically within rigid-flex designs. All existing functionalities are still maintained from previous versions.

If the board drawing contains an existing DFA Constraint table, and you create a new Manufacturing > Design for Assembly > PkgToPkg constraint definition, a prompt is displayed to import the existing table into the new Manufacturing > Design for Assembly constraints. After import, perform all modifications only in the new Manufacturing > Design for Assembly constraints. Board drawing will ignore the prior version of the DFA table including modifications. There is no down rev option to return to the previous DFA Constraints format.

DFA Driven Interactive Placement

When placing or moving a component to a DFA rule, a circle that represents the clearance appears when the component being placed is at the minimum distance to adjacent components. A slight pause may be noticed when the circle appears. This allows you to instantiate the component at the minimum spacing. Pause levels can be controlled in the User Preference Editor.



Design Rule Check

The software determines the package side and end. This is based on classifying the longer edge as side; the shorter edge as end. If it cannot determine the side or end, it uses the most conservative value of the string entered in the spreadsheet for that particular package to package requirement. The size of the spacing circle that appears during placement aligns with the spacing value in the spreadsheet. The circle is the minimum requirement for the current placement condition. Violations are reported with a “D-C” marker and can be viewed in the DRC workbook in Constraint Manager.

Note: Not seeing DFA circles? Activate the DRC located in the DFA Spreadsheet.

Legacy Placebound Shape

The DFA check is designed to work with, not instead of, the legacy package-to-package check. Customers are advised not to disable the former mode because it is used to check height, collision, and keepout violations. Once your company is committed to the DFA implementation, boundary adjustments may be required on legacy placebound shapes to prevent false errors.

DRC's

DFA errors are itemized in the DRC Browser window (Tools > DRC Browser) under the heading of Design for Assembly PkgToPkg Spacing. The worksheet provides information on the type of error in terms of side or end, subclass, required DRC value, actual clearance, and violating objects listed as reference designators.

Design For Assembly Definition

DRC Browser

DRC > Design for Assembly > PkgToPkg Spacing > Package to package spacing

All Unread Review Filter by Area

DRC Location	DRC Subclass	Actual Value	Required Value	Waive DRC
*	*	*	*	*

Package to package spacing Errors:
Total = 0
Waived = 0

Show DRC Chart Options Close

User Preference Controls

The controls include:

DFA_Pause Level: Sets the pause level in manual placement command. If DFA DRC check is enabled, when a component reaches its minimal DFA spacing of another component, it pauses briefly so it can be placed as close as possible to the other component without DFA DRC violation.

Display_nodfa_drc_marks: If set, DRC markers for DFA violations do not display dynamically during interactive placement; only dynamic spacing circles are displayed.

The following are trademarks or registered trademarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134
Allegro®, Cadence®, Cadence logo™, Concept®, NC-Verilog®, OrCAD®, PSpice®, SPECCTRA®, Verilog®

Other Trademarks

All other trademarks are the exclusive property of their prospective owners.

NOTICE OF DISCLAIMER: Parallel Systems is providing this design, code, or information "as is." By providing the design, code, or information as one possible implementation of this feature, application, or standard, Parallel Systems makes no representation that this implementation is free from any claims of infringement. You are responsible for obtaining any rights you may require for your implementation. Parallel Systems expressly disclaims any warranty whatsoever with respect to the adequacy of the implementation, including but not limited to any warranties or representations that this implementation is free from claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.