Best Practices:
Working with Real-Time DFA Analysis

Product Version 23.1
October 2023

Contents

1
Using Real-Time DFA Analysis 5

Licensing 5

Adoption Considerations 5

Uprev of Package Symbol Library 6

Sampling Auto-Generated DFA Placebounds 9

Package Side and End 11

DFA Constraint Spreadsheet 12

Class-Based Considerations 14

Overriding Cells 16

Side-End vs. End-Side 17

When comparing two identical symbols, only the side-end (S-E) value is used and, end-side (E-S) is considered superfluous. 18

Table Compression 18

Table Access from DesignPath Variable 18

Read-Only Version 18

Loading Spreadsheet into Allegro PCB Editor 18

DFA-Driven Interactive Placement 19

Design Rule Check 20

DFA Constraint Resolution for Embedded Layers 21

Legacy Placebound Shape 21

Constraint Manager DRC Worksheet 22

User Preference Controls 22

1

Using Real-Time DFA Analysis

The Allegro PCB Editor 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 or end-end 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 guidelines and helps reduce the dependency on referencing specifications or interfacing with 3rd party CAM systems.

Licensing

DFA analysis is now available in the Allegro PCB Editor Base product. This feature was originally introduced in the 15.5 release. (XL license).
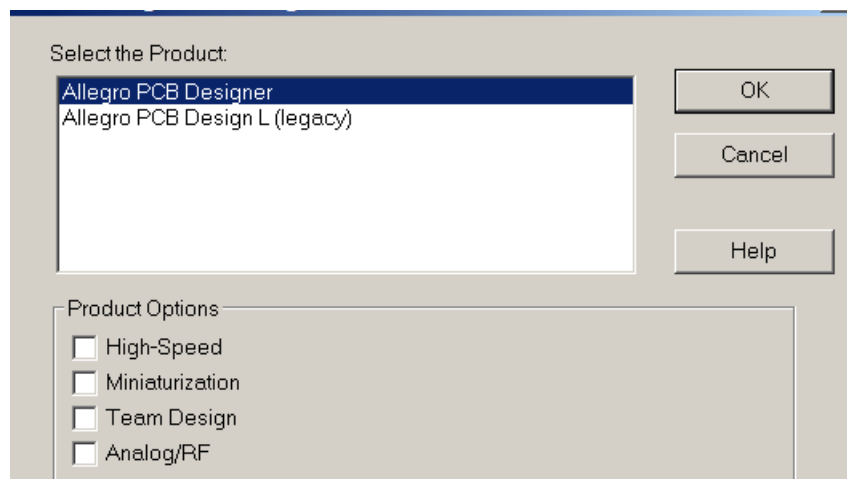
Adoption Considerations

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:

■ 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

■ 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*.

Figure 1-1 Compact placebound geometry (l) and extended placebound (r)
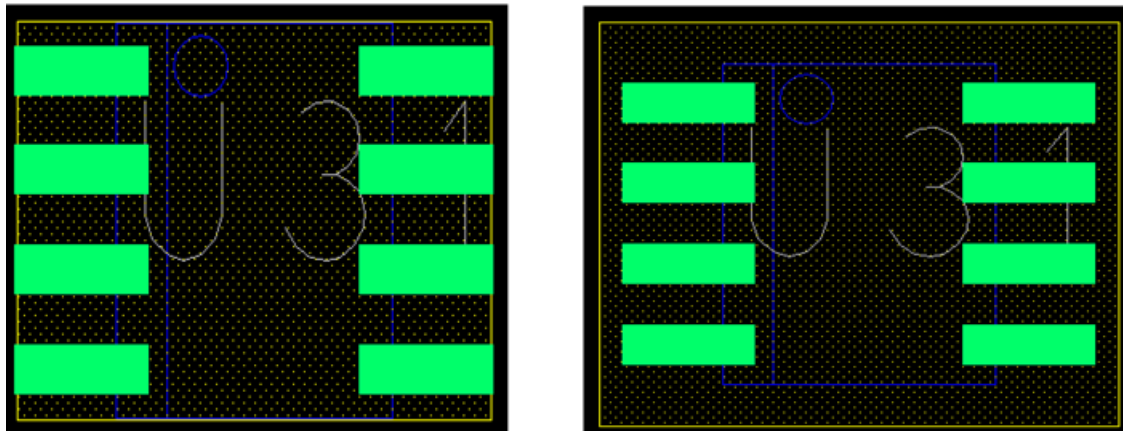
Legacy placebound shapes may be drawn to include the space required for rudimentary clearance checking and may not integrate well with DFA Spreadsheet-driven rules. Although the DFA checker reverts to the legacy placebound shape if a DFA placebound is not found, it is estimated this model will be inaccurate 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 of 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* menu 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. Symbols in Release 15.5 cannot be used with downrev software. In the example below, the *DFA_Update* utility is used to update all `.dra` files in a 15.2 library. It adds the DFA placebound shape and writes the output to a 15.5 library. Properties are not recommended when performing a global update.

1. From *Start – Run*, enter `dfa_update`.

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 .

4. In the *Enter Destination Directory* field, browse to the location of the newly created 15.5 library name.

5. Leave the DFA_DEV_CLASS property field blank.

6. Click *Update*.

7. Review the log file.

8. Sample a variety of particular symbols for company compliance.

Figure 1-2 Global update of all package symbols
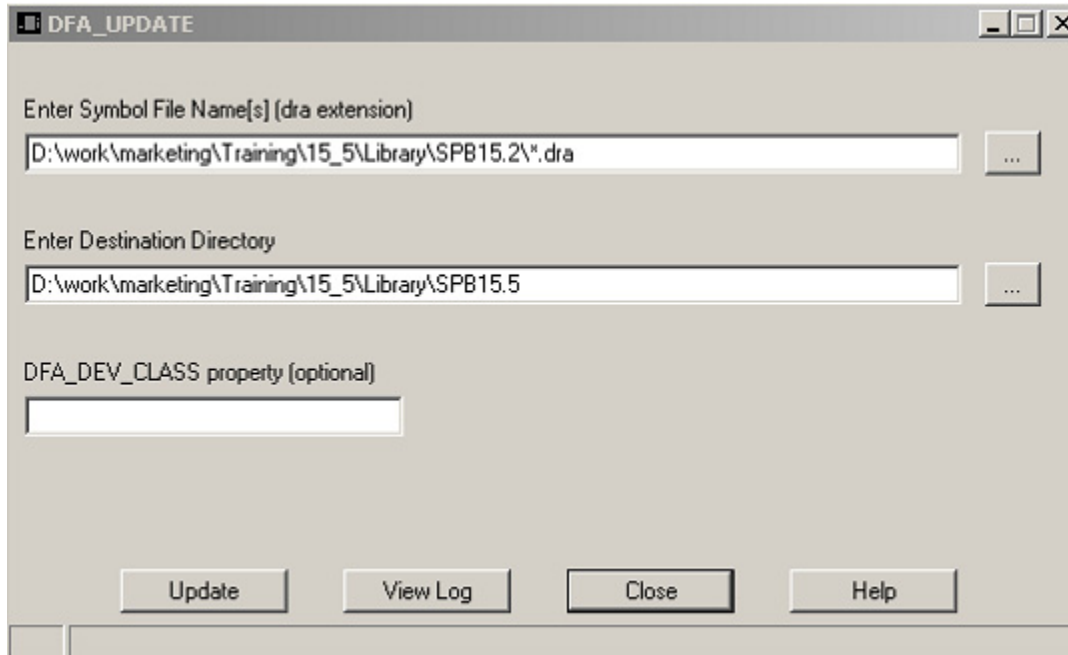
In Figure 1-3 on page 9, all BGA are updated and assigned a class property of BGA because their clearance requirements are identical.

Figure 1-3 All BGAs updated and assigned class property of BGA
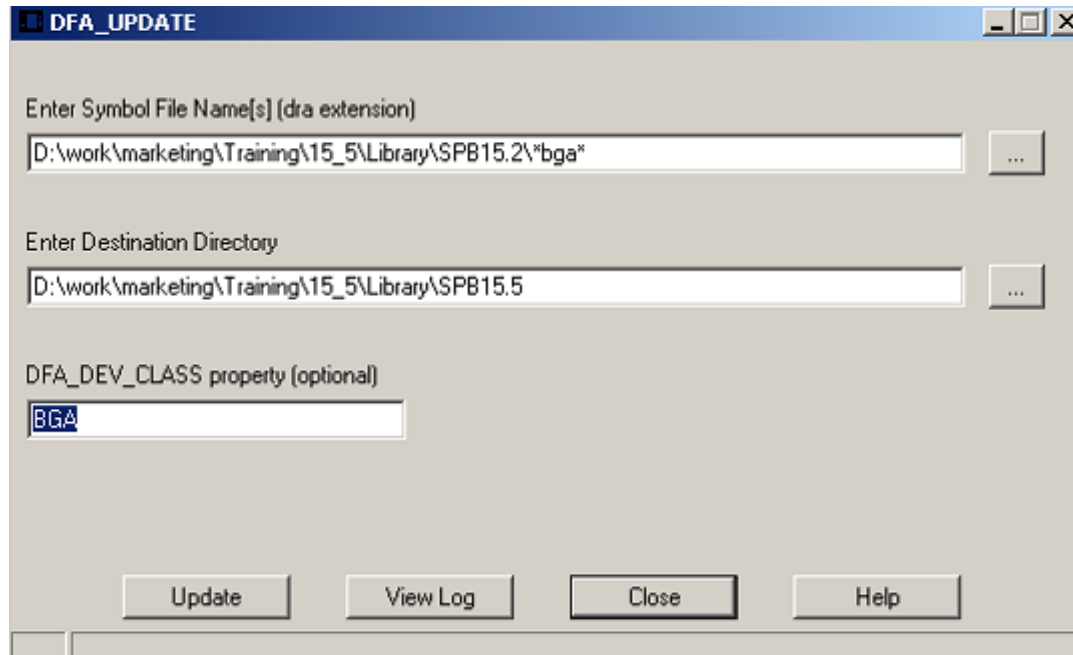
Sampling Auto-Generated DFA Placebounds

It is critical that you sample several but unique packages before releasing the library to production. The results from the *DFA_Update* utility, in some cases, may not conform to your company guidelines. Figure 1-4 on page 10 exemplifies auto-generated DFA placebounds, which display in yellow. The assembly outline displays in blue. The DFA_BOUND_TOP and DFA_BOUND_BOTTOM subclasses are located in the PACKAGE GEOMETRY class.

Figure 1-4 Auto-generated DFA Placebound Shapes
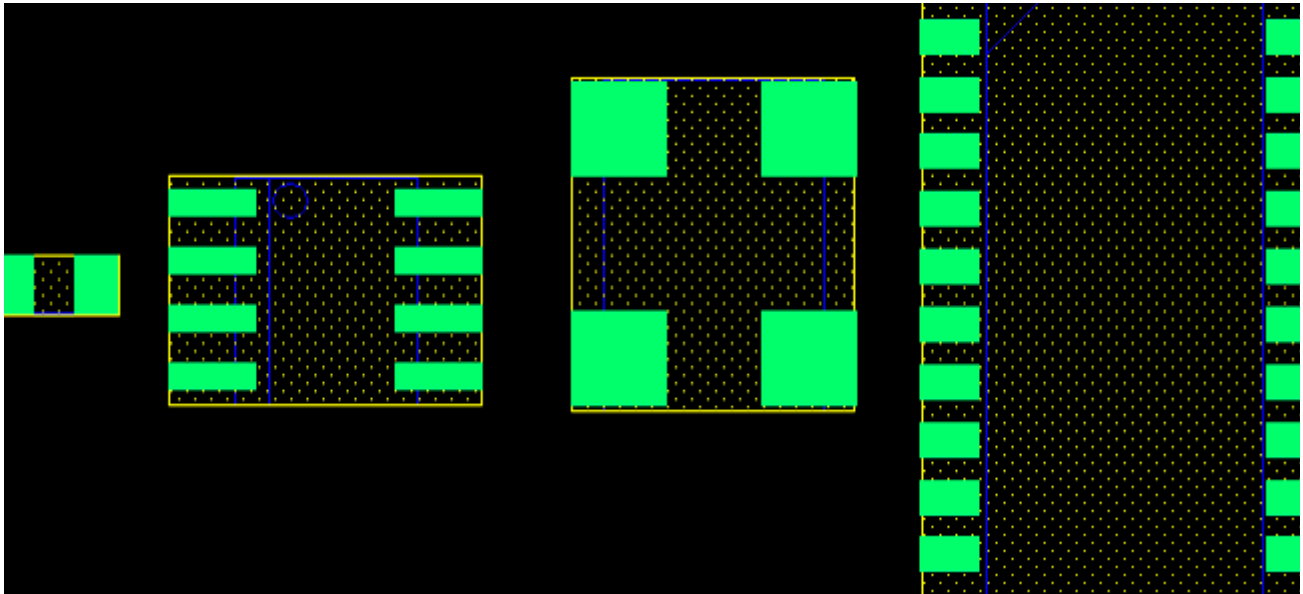
Figure 1-5 shows auto-generated DFA placebound shapes for a package with fiducials.

Figure 1-5 Auto-generated DFA Placebound Shapes for a Package with Fiducials
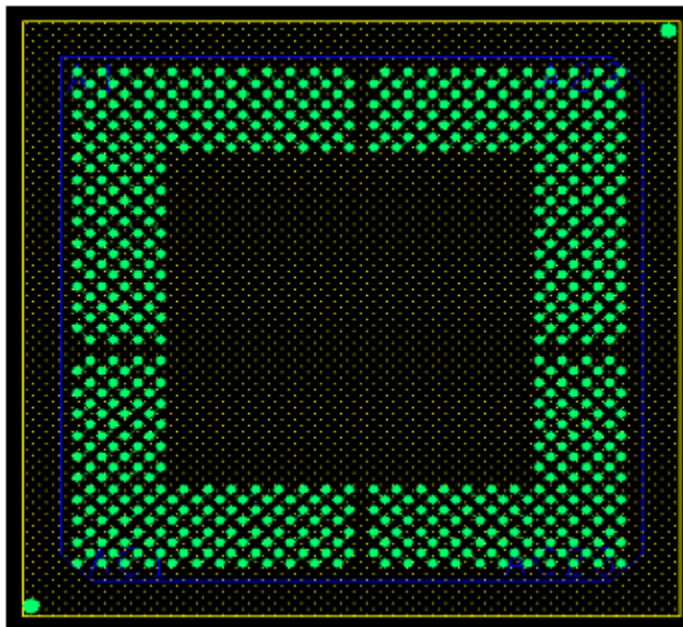
Figure 1-7 shows an example of auto-generated DFA placebound shapes for a complex assembly outline.

Figure 1-6 Auto-generated DFA Placebound Shapes for a Complex Assembly Outline
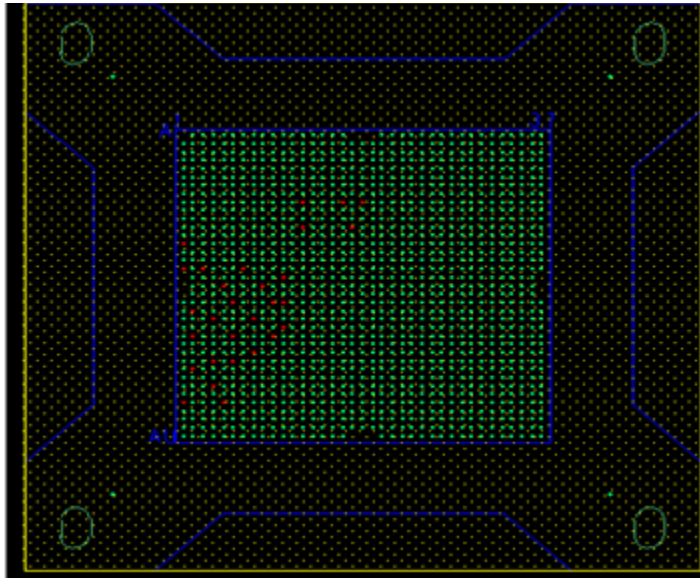
Package Side and End

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, 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.

⊘ *Caution*

> **Do not assume the leaded edge of a package is the side. In Figure 1-7, an 8-pin SOIC shows the side where you normally might think the end is located.**

Figure 1-7 Package Side and End
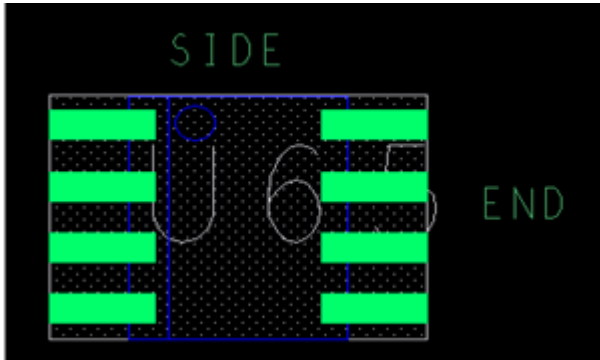
DFA Constraint Spreadsheet

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. You can define spacing rules for embedded components ia a separate tab.The spreadsheet has a `.dfa` extension and can be opened by choosing *Setup - 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 if an Allegro PCB Editor environment is installed. The stand-alone version is recommended for development and offers the ability to write class information to the library. These controls, in the *Show Symbol Classification* user interface, are disabled in Allegro PCB Editor XL. The stand-alone DFA Spreadsheet can be launched from the Start menu or with the command `dfa_dlg.exe`.

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

1. Do one of the following:

   a. Choose *Start – Programs – Cadence SPB – PCB Editor Utilities – DFA Spreadsheet Editor*.

   b. Click *Start – Run*. In the Open field of the Run dialog box that appears, enter `dfa_dlg` to launch the DFA Spreadsheet Editor.

> **Note:** These tasks can be accomplished by using the spreadsheet with Allegro PCB Editor, except for writing properties to the library.

2. To add spacing rules for embedded components, click Add *Embedded/Constraint Layers Tab*.

3. Select the MASTER_EMBEDDED or individual layers from *DFA Add Embedded Layers* dialog box.

4. Click *Browse for Symbols*.

5. Click *Package/Mechanical symbols* in the DFA Symbol Browser dialog box.

6. Click *Display Symbols from Library*. At this time, all symbols in your PSMPATH populate the DFA Symbol Browser dialog box.

7. Click *Embedded Only*, to display symbols that has EMBEDDED_PLACEMENT property*.

8. Click *OK* to quickly generate the table.

9. To apply a global spacing value, enter a value in the *Default* cell.

10. Click the grey cell in the upper left corner of the grid to highlight all cells. Alternatively, you can select cells as you do in Excel by clicking your left mouse button.

11. Click *Apply to Selected Cells* to apply the spacing value to all cells.

Figure 1-8 DFA Constraint Spreadsheet
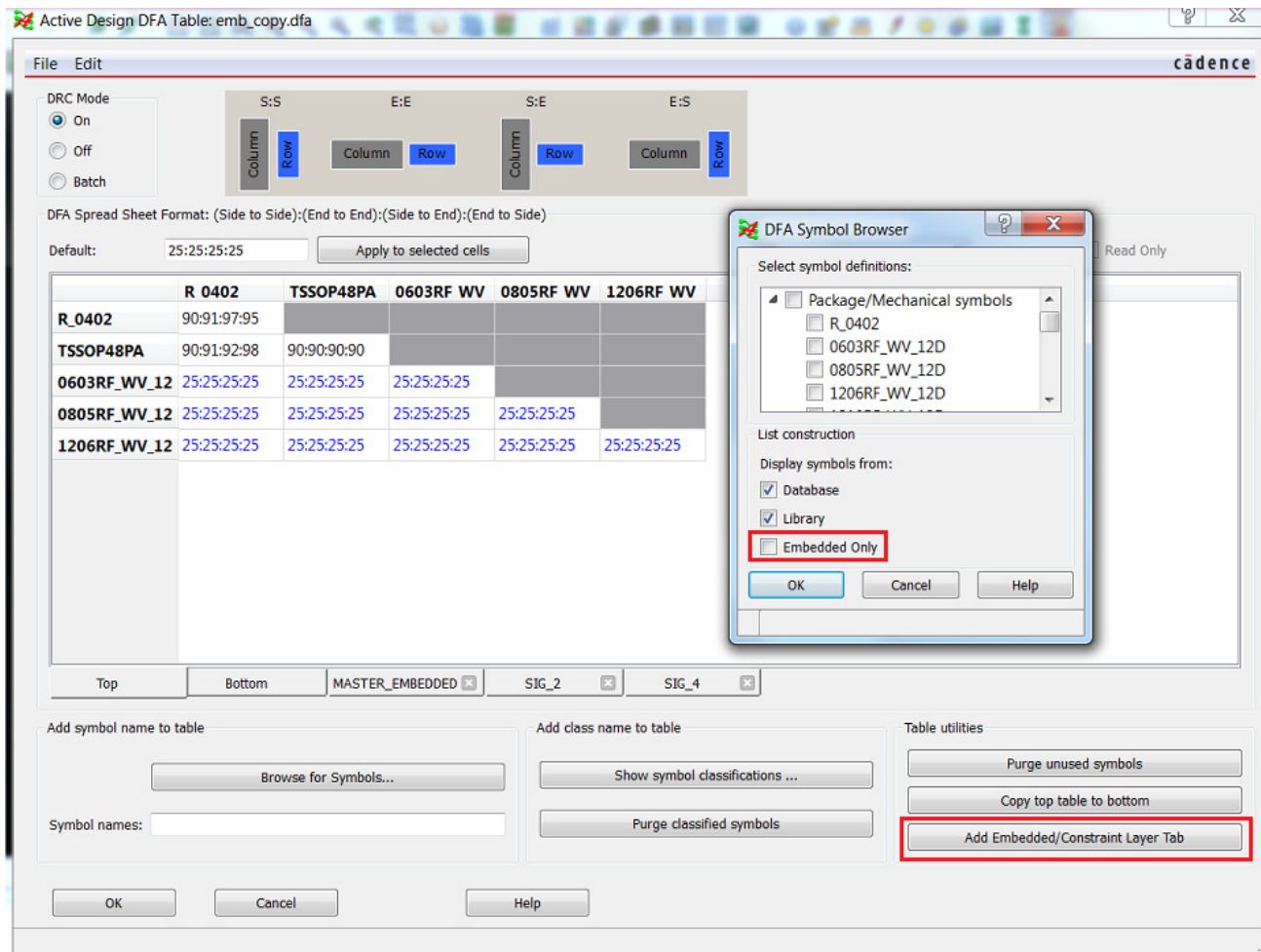
Class-Based Considerations

Table sizes based on sourcing libraries are expected have hundreds or thousands of entries. Classifying package symbols into hierarchical structures can help reduce the size of the table significantly. As mentioned earlier, 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.

1. Click on *Show Symbol Classification*.

2. If your library symbols already have DFA_DEVICE_CLASS properties assigned, those values are represented in the UI along with the list of instance-based package symbols under the *Package Symbol* category.

3. Select the class entry; click *Update* to transfer them to the table as shown in Figure 1-9 on page 15.

4. The class based entries occur at the beginning of the table. A blue box surrounds the values. Note the values inherit the setting of the *Default* cell.

Figure 1-9 Class-Based Entries
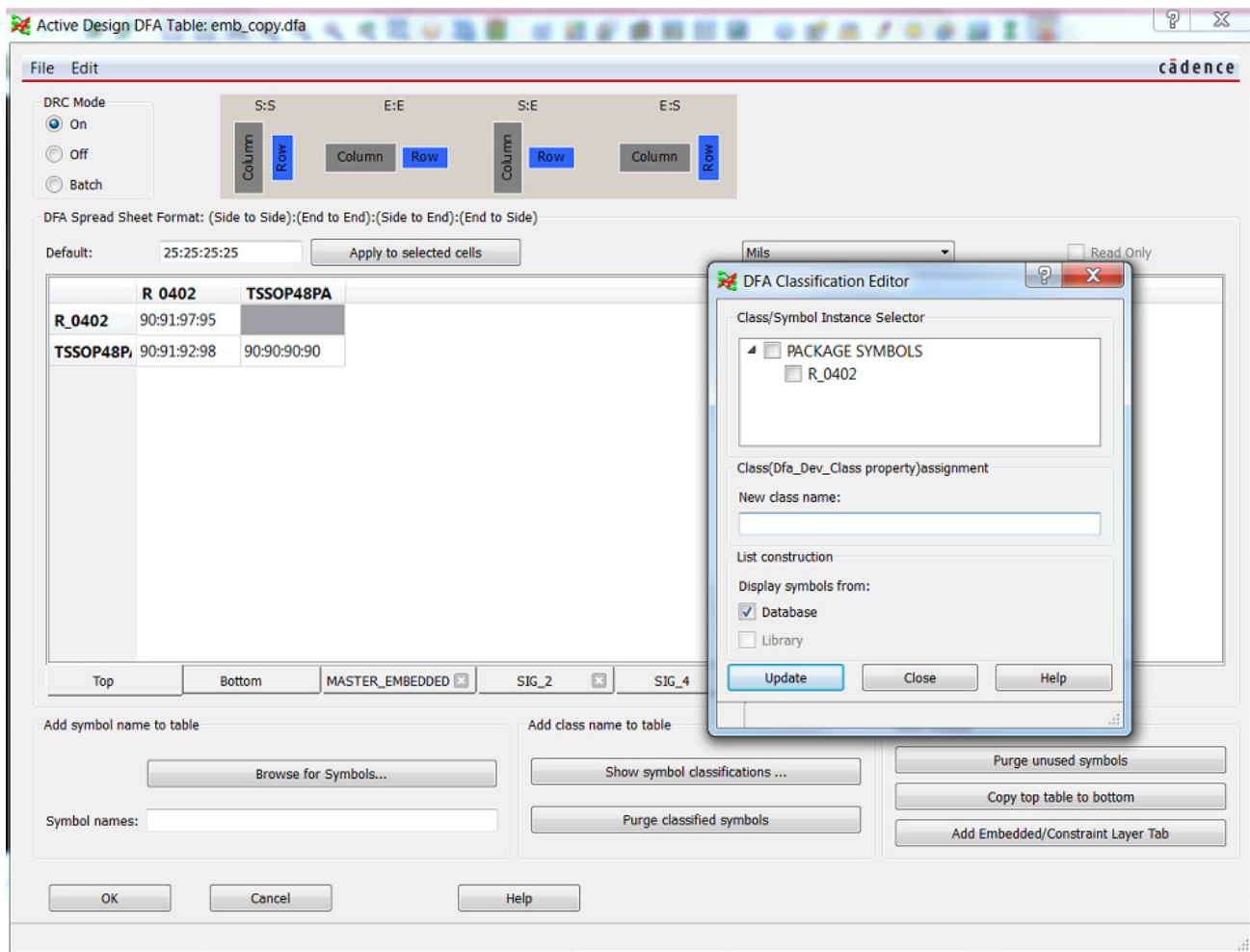
New classes and their associations can also be managed within the DFA Classification Editor. Enter a new class name in the *New Class Name* cell, then click the *Tab* key. Package symbol assignments occur by selecting the specific instances followed by the cut/paste actions of the right mouse button as in Figure 1-10 on page 16. Upon completion, clicking *Update* not only moves the new classes to the table but also writes the DFA_DEVICE_CLASS to the library-based symbols. A pop-up prompt advises the user of this action.

Figure 1-10 Cutting symbols and pasting into new class
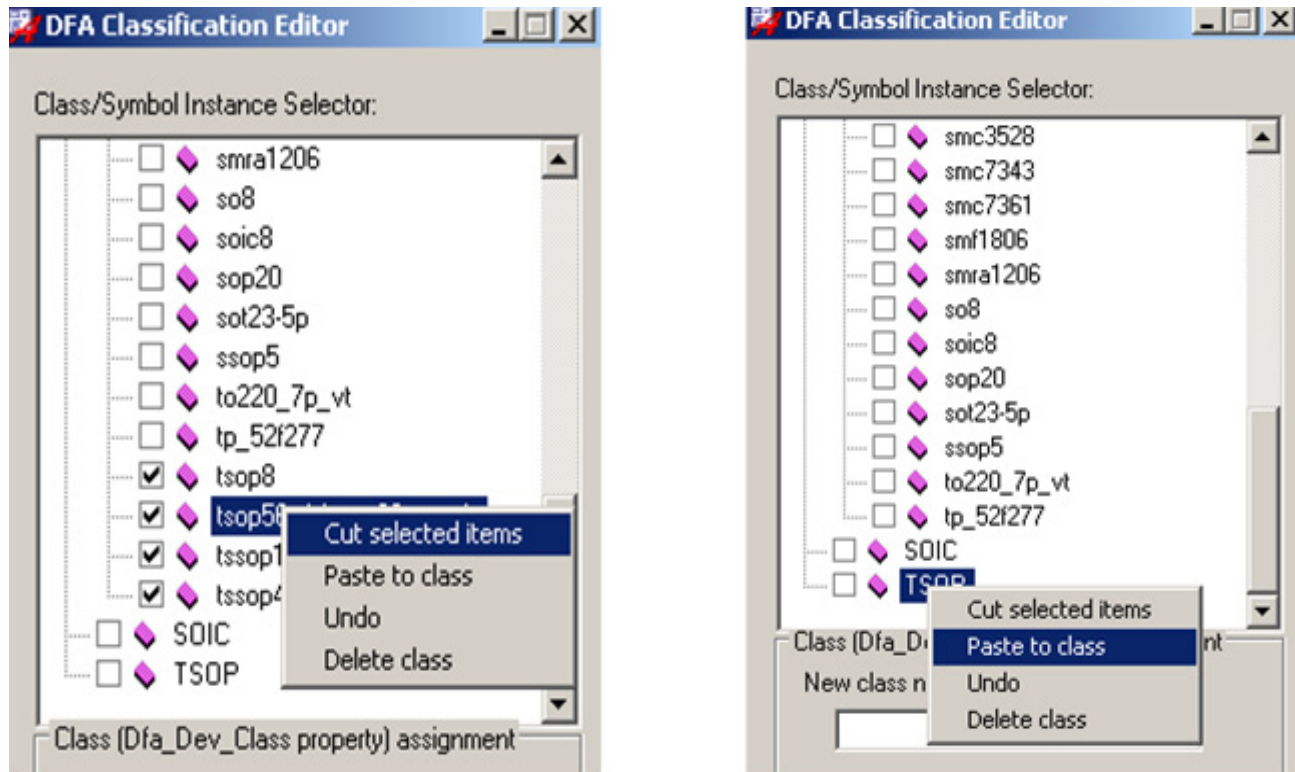
Overriding Cells

The method for editing cells is similar to that used in Microsoft Excel. Drag cells with the left mouse button or use the Ctrl or Shift keys to extend the selection. In Figure 1-11 on page 17, the cell entry of 25:20:25 represents the syntax for S-S:E-E:S-E.

Figure 1-11 Cell Editing

DFA Spread Sheet Format: (Side to Side):(End to End):(Side to End):(End to Side)							
Default:	25:25:25:25		Apply to selected cells				Mils

|	R 0402	TSSOP48PA	AC200100	AX1000X225	AX1000X30(	CPCYL1D400	CPCY
R_0402	90:91:97:95						
TSSOP48PA	90:91:92:98	90:90:90:90					
AC200100	25:25:25:25	25:25:25:25	25:25:25:25				
AX1000X225034	25:25:25:25	25:25:25:25	25:25:25:25	25:25:25:25			
AX1000X300037	25:25:25:25	25:25:25:25	25:25:25:25	25:25:25:25	25:25:25:25		
CPCYL1D400LS2000	25:25:25:25	25:25:25:25	25:25:25:25	25:25:25:25	25:25:25:25	25:25:25:25	
CPCYL1D400LS2250	25:25:25:25	25:25:25:25	25:25:25:25	25:25:25:25	25:25:25:25	25:25:25:25	25:25
CPCYL1D500LS2250	25:25:25:25	25:25:25:25	25:25:25:25	25:25:25:25	25:25:25:25	25:25:25:25	25:25

Top	Bottom	MASTER_EMBEDDED ☒	SIG_2 ☒	SIG_4 ☒

Side-End vs. End-Side

In the figure below, A and B both represent a side-end (S-E) condition where different values need to be applied:
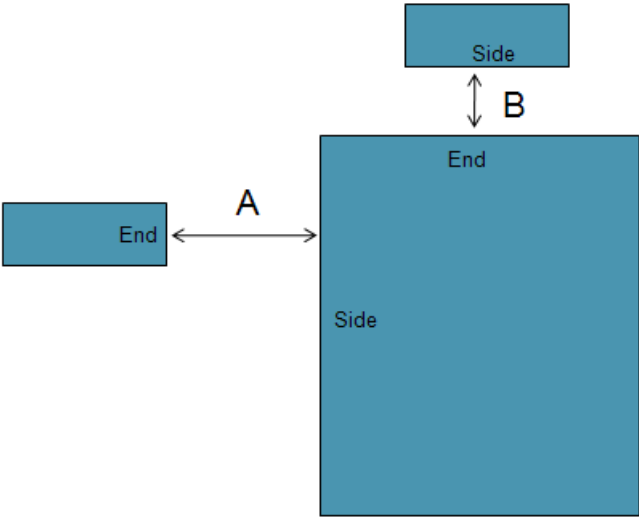
- The symbol considered the Reference Symbol is located in the column of the DFA spreadsheet.

- If end-side (E-S) value is not present, DRC uses side-end (S-E) value for both the conditions (pre 16.5 behavior).

When comparing two identical symbols, only the side-end (S-E) value is used and, end-side (E-S) is considered superfluous.

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 using *Purge Classified Symbols*. 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 centrally archived for the design team to access. The design path variable DFACNSPATH set the default location of these files. The PCB Editor can use the file browser to navigate for `.dfa` files or the data browser that 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 PCB Editor opens the file, its contents cannot be modified. 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 Editor can load only one spreadsheet into Allegro PCB Editor. If a change occurs as to how the board is assembled or soldered, additional spreadsheets can be applied to overwrite the database contents. Choose *Setup – Constraints – DFA Constraint Spreadsheet.* To open a new spreadsheet, select one of the following:

- *File – Open:* standard file browser searching for `.dfa` files.

- *File – Browse DFA Library*: search path aligned with DFACNSPATH variable.

The Allegro PCB Editor database stores the spreadsheet, assuming a save has been performed.

DFA-Driven Interactive Placement

When placing a component referencing the DFA table, a circle that represents the clearance to adjacent components appears when the component being moved is at the minimum distance. 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.

Figure 1-12 DFA Spacing Circles
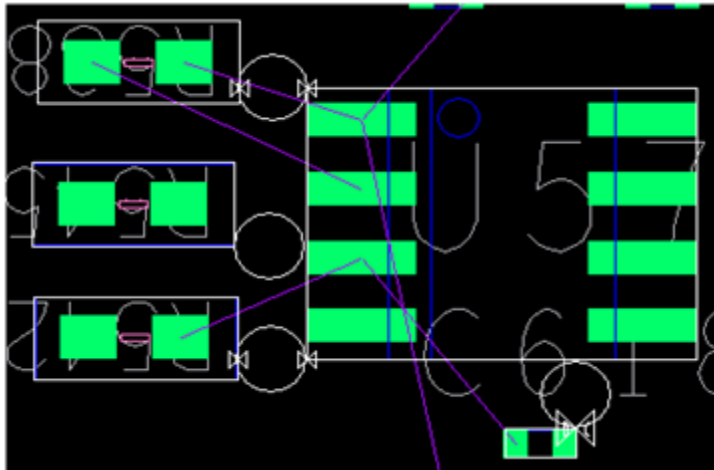
Similarly, on moving a group of components referencing the DFA table, circles are displayed when the group is at the minimum distance from the adjacent components. You can create the group using standard group selection methods such as window select, temp group, select by polygon, lasso or path.

Figure 1-13 DFA feedback during group move
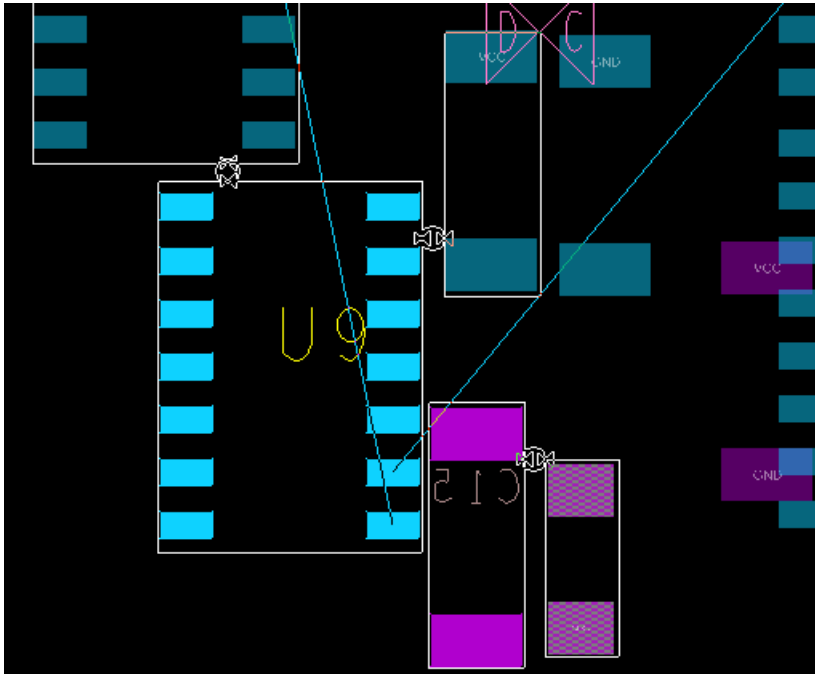
Note: The DFA feedback is not available when moving modules.

Design Rule Check

Allegro PCB Editor determines the package side and end 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.

Tip

If DFA circles do not appear, activate the DRC located in the DFA Spreadsheet.

### DFA Constraint Resolution for Embedded Layers

Spacing values for embedded components are drawn from the embedded layer tabs in the DFA table. If embedded layer and constraint type tabs are not present in the table then spacing values are derived from MASTER_EMBEDDED tab. If MASTER_EMBEDDED tab is also not added then spacing values are drawn from the Top layer tab.

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.

Constraint Manager DRC Worksheet

DFA errors are itemized in the Design Worksheet under the heading of Package to Package DFA 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.

User Preference Controls

The User Preference Editor features the category *Dfa_drc,* which includes:

dfa_pause_level	Sets the pause level in `place manual` command. If DFA DRC check is enabled when the component reaches the minimal DFA spacing of another component, it pauses briefly, so it can be placed as close as possible to the other component without a DFA DRC violation.
display_nodfa_drc_marks	If set, DRC markers for DFA violations do not display dynamically during interactive placement.
dfacnspath	Search path for DFA spreadsheet.