

DFM Add-Ons for EAGLE Initial Release, January 2008

By downloading and using these Sunstone specific rule sets you can have confidence that your board can be built by Sunstone without manufacturing difficulties or unnecessary delays.

Release Contents:

SSC-Eagle DRU_2 v1.0.0.3 *for 2 layer designs*

Last modification date: 12/12/07

Requirements: EAGLE (Easy Applicable Graphical Layout Editor) for Windows

SSC-Eagle DRU_4 v1.0.0.3 *for 4 layer designs*

Last modification date: 12/12/07

Requirements: EAGLE (Easy Applicable Graphical Layout Editor) for Windows

SSC-Eagle DRU_6 v1.0.0.3 *for 6 layer designs*

Last modification date: 12/12/07

Requirements: EAGLE (Easy Applicable Graphical Layout Editor) for Windows

Description:

DFM Add-Ons for EAGLE are EAGLE *Design For Manufacturing* (DFM) Rules files, written in accordance with Sunstone's manufacturing criteria for 2, 4, and 6 layer designs.

By using Sunstone's rules during design, and running the design check while in your CAD program, you can easily make any needed design changes while in the design stage, thus minimizing costly revisions.

Make sure to **save your project** after implementing any changes and after running the final design check with the Sunstone DFM Add-On rule deck.

The EAGLE .RUL files can easily be imported into EAGLE CadSoft to setup the design rules for you. For instructions on how to import the .RUL file(s), please refer to your EAGLE user documentation.

Functional Notes:

Clearance

Sunstone has a minimum trace/space width of 6 mils for our Quick Turn Products. This can be increased if needed as long as other manufacturing requirements are met. Please note as stated in the EAGLE help menu:

A polygon in the special signal named `_OUTLINES_` will be used to generate outlines data and as such will not adhere to these clearance values. Make sure that you always delete the `_OUTLINES_` polygon after generating the outlines data. Leaving this polygon in your drawing will cause short circuits since this special polygon does not adhere to the Design Rules!

Distance Setting ~ Copper/Dimension

Sunstone's DFM rules have a Copper/Dimension distance setting of 20 mils to ensure that there is adequate space between outermost copper feature and the board outline. The Dimension layer (Layer 20) will also have hole placement outlines that are located on the pcb. Sometimes DRC errors that occur when a pad is too close to one of these hole "outlines" may be ignored.

Please review the following information regarding Distance from EAGLE Help on *Design Rules*.

The Distance tab defines the minimum distance between objects in signal layers and the board dimensions, as well as that between any two drill holes. Note that only signals that are actually connected to at least one pad or smd are checked against the board dimensions. This allows edge markers to be drawn in the signal layer without generating DRC errors.

For compatibility with version 3.5x the following applies: If the minimum distance between copper and dimension is set to 0 objects in the Dimension layer will not be taken into account when calculating polygons (except for Holes, which are always taken into account). This also disables the distance check between copper and dimension objects.

Masks

Sunstone design rules have a minimum value for soldermask openings of 3 mils on the radius to ensure proper openings for pads, vias, and SMD's. If needed, this minimum value can be adjusted (as long as other manufacturing requirements are met) depending on your needs. And since EAGLE also automatically generates a solder stop mask for vias, you may elect to have the

vias covered by soldermask. Please review the help topic below located on the FAQ section on CadSoft's website. <http://www.cadsoft.de/faq.htm>

EAGLE generates by default a solder stop mask for each Via (also for Pads and SMDs, of course). This means the Via is free of coating material. The solder stop mask is drawn automatically in the layers 29, *tStop*, for the top side and 30, *bStop*, for the bottom side. The size of the solder stop mask can be determined in the Design Rule's *Mask* settings. See the values for *Stop*. By default the value is fixed to 4 mils. Minimum and Maximum are set to the same values therefore. If you want to have a diameter-dependent mask you could also define a certain percentage. The resulting value can be limited by a minimum and a maximum.

In order to have vias coated, EAGLE allows you to set the solder stop *Limit* in the Design Rules' *Mask* tab. Here you can define a value which is dependent on the drill diameter of the via. Let's assume you would like to set the Limit to 0.012 inch. Now all vias in the layout up to a drill diameter of 0.012 inch will be covered with coating material. All those vias that have bigger drills will stay uncovered.

In case you want to have some smaller drilled vias uncovered, you have the possibility to select vias out of those that are covered in order to uncover them. This can be done with the command *CHANGE STOP ON | OFF* in the Layout Editor. This also works for groups.

By the way: It is allowed to draw areas that should remain free of coating material in the *tStop/bStop* layers directly. But it is not possible to delete certain automatically generated solder stop symbols there. You have to deal with *Limit* and *CHANGE STOP ON | OFF* instead.

Drill Sizes

Please note: Our Quick Turn Products feature preset Drill sizes.

The original hole sizes are redefined during an in house scripting process. For instance, a hole size of 0.0179" (or 17.9 mils), will be drilled using a 0.0200" (or 20 mil) drill bit for a 14 mil finished hole size. Please refer to Table 1, below.

For our Quick Turn services you should adjust your hole sizes to match the *finished* sizes in the chart below for accurate results. This will ensure proper pad clearances on the finished pcb.

New Transcode Table DS (Finish sizes)		New Transcode Table ML (Finish sizes)		Bit Size	Finish Size	DRILL TRANSCODE TABLES
From	To	From	To			
N/A		0.0000	0.0159	0.0135	0.0080	
0.0000	0.0179	0.0160	0.0179	0.0200	0.0140	
0.0180	0.0249	0.0180	0.0249	0.0260	0.0200	
0.0250	0.0289	0.0250	0.0289	0.0310	0.0250	
0.0290	0.0319	0.0290	0.0319	0.0350	0.0290	
0.0320	0.0349	0.0320	0.0349	0.0390	0.0330	
0.0350	0.0380	0.0350	0.0380	0.0420	0.0360	
0.0381	0.0410	0.0381	0.0410	0.0465	0.0405	
0.0411	0.0449	0.0411	0.0449	0.0492	0.0432	
0.0450	0.0519	0.0450	0.0519	0.0520	0.0460	
0.0520	0.0599	0.0520	0.0599	0.0595	0.0535	
0.0600	0.0659	0.0600	0.0659	0.0670	0.0610	
0.0660	0.0759	0.0660	0.0759	0.0730	0.0670	
0.0760	0.0839	0.0760	0.0839	0.0860	0.0800	
0.0840	0.0899	0.0840	0.0899	0.0935	0.0875	
0.0900	0.0969	0.0900	0.0969	0.0995	0.0935	
0.0970	0.1059	0.0970	0.1059	0.1065	0.1005	
0.1060	0.1160	0.1060	0.1160	0.1160	0.1100	
0.1161	0.1329	0.1161	0.1329	0.1319	0.1259	
0.1330	0.1429	0.1330	0.1429	0.1470	0.1410	
0.1430	0.1529	0.1430	0.1529	0.1570	0.1510	
0.1530	0.1709	0.1530	0.1709	0.1730	0.1670	
0.1710	0.2019	0.1710	0.2019	0.1990	0.1930	
0.2020	10.0000	0.2020	10.0000	0.2570	0.2510	

Table 1: Quickturn preset drill sizes

Known Bugs/Problems/Incompatibilities:

None

Bug Fixes, Enhancements, Improvements since Last Release:

Initial release of DFM Add-Ons for EAGLE.

Release History:

V1.0	1/11/2008	Initial public release.
------	-----------	-------------------------