

## Release Notes

### ValueProto™ DFM Add-Ons for EAGLE v1.0.0.0 New Release, October 2011

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\_VP v1.0.0.0 for 2 layer ValueProto™ Designs  
Last modification date: 10/31/2011  
Requirements: EAGLE (Easy Applicable Graphical Layout Editor) for Windows

#### Description:

DFM Add-Ons for EAGLE are an EAGLE-format set of *Design For Manufacturing* (DFM) Rules files, written in accordance with Sunstone's manufacturing criteria for 2 layer ValueProto™ 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 CadSoft's EAGLE tool, to setup the design rules for you. For instructions on how to import the .RUL file(s), please refer to your EAGLE user documentation, or leave a comment on the Sunstone Forum and we will do our best to assist you.

## Release Notes

### Functional Notes:

Clearance ~ Sunstone has a minimum trace/space width of 7 mils for our ValueProto™ Service. This can be increased if needed as long as other manufacturing requirements are met. Please note the following, 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 DFM rules have a minimum value of 3 mils on the radius for soldermask openings, 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

## Release Notes

vias covered by soldermask. Please review the help topic below located on the FAQ section on CadSoft's website <http://www.cadsoftusa.com/training/faq/#13>.

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 *tStop*, for the top side and *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 ValueProto Products feature preset Drill sizes.

The original hole sizes in your design file will be 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 16 mil finished hole size. Please refer to Table below.

For our Quick Turn services you should adjust your hole sizes to match the *finished* sizes in the chart below for accurate results.

## Release Notes

<b>Value Proto Preset Tool Drill Table</b>			
<b>Range</b>		<b>Specs</b>	
<b>From</b>	<b>To</b>	<b>Bit Size Used</b>	<b>Finish Size</b>
0.0000	0.0179	0.0200	0.0160
0.0180	0.0249	0.0260	0.0200
0.0250	0.0289	0.0310	0.0250
0.0290	0.0319	0.0350	0.0290
0.0320	0.0349	0.0390	0.0330
0.0350	0.0380	0.0420	0.0360
0.0381	0.0410	0.0465	0.0405
0.0411	0.0449	0.0492	0.0432
0.0450	0.0519	0.0520	0.0460
0.0520	0.0599	0.0595	0.0535
0.0600	0.0659	0.0670	0.0610
0.0660	0.0759	0.0730	0.0670
0.0760	0.0839	0.0860	0.0800
0.0840	0.0899	0.0935	0.0875
0.0900	0.0969	0.0995	0.0935
0.0970	0.1059	0.1065	0.1005
0.1060	0.1160	0.1160	0.1100
0.1161	0.1329	0.1319	0.1259
0.1330	0.1429	0.1470	0.1410
0.1430	0.1529	0.1570	0.1510
0.1530	0.1709	0.1730	0.1670
0.1710	0.2019	0.1990	0.1930
0.2020	10.0000	0.2570	0.2510

### Known Bugs/Problems/Incompatibilities:

None

### Release History:

V1.0.0.0 10/2011 New Release

Sunstone Circuits Page 4