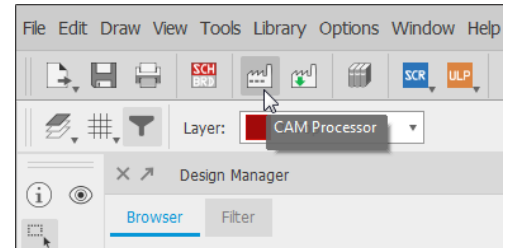


# Autodesk Eagle Gerber Export

You have the option to send your EAGLE \*.brd file using our Convert to Gerber service when you order your circuit boards. However if you prefer to convert your own files we have made our standard Eagle CAM files available for download on our website.

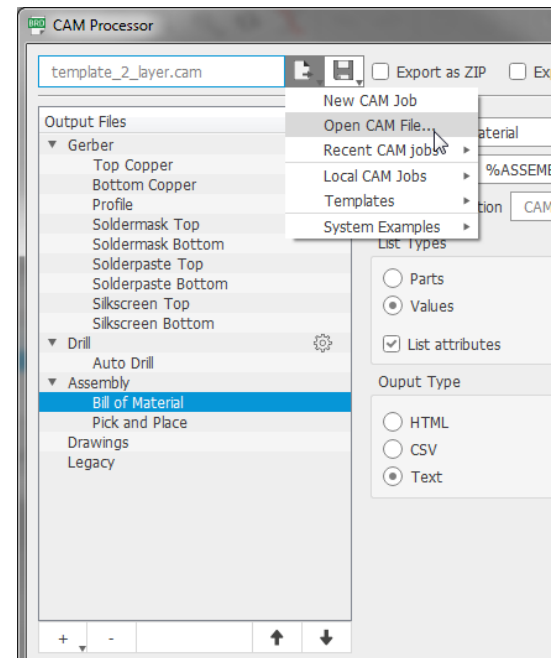
## Step 1: CAM Processor

With your Eagle BRD open click the *CAM Processor* button in the tool bar or click *File → CAM Processor*



## Step 2: Load A CAM File

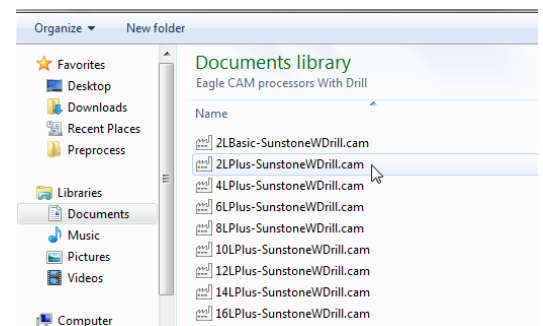
From the Eagle CAM Processor we need to load one of the predefined Sunstone CAM Files by clicking the *Load Job File* Button then clicking *Open CAM File...* Navigate to where you saved the Sunstone CAM files and select the CAM processor that matches your design. This depends on the specs of your board. Please see below for a full break down.



File	Board Spec
2LBasic-SunstoneWDrill.cam	Two Layer no mask or silk
2LPlus-SunstoneWDrill.cam	Two Layer with mask and silk
4LPlus-SunstoneWDrill.cam	Four Layer with mask and silk
6LPlus-SunstoneWDrill.cam	Six Layer with mask and silk
8LPlus-SunstoneWDrill.cam	Eight Layer with mask and silk
10LPlus-SunstoneWDrill.cam	Ten Layer with mask and silk
12LPlus-SunstoneWDrill.cam	Twelve Layer with mask and silk
14LPlus-SunstoneWDrill.cam	Fourteen Layer with mask and silk
16LPlus-SunstoneWDrill.cam	Sixteen Layer with mask and silk

## Step 3: Layer Selection

The Sunstone CAM definitions are based on standard Eagle settings. As such the copper layers assume the outer layers are 1 and 16 then work from the outside in (IE: the copper layers of a six layer board would be: 1, 2, 3, 14, 15, 16). Eagle generates a black and white preview of your data as you select each layer. Be sure to double check that everything appears correct.





## **Step 4: Export**

Once all of your layer settings have been double checked click the Process Job button. A save dialog will pop up allowing you to name your exported data and place it in an easily found location.

## **Step 5: Verification**

With any CAD suite we always recommend verifying your gerber data after export. Simply use your gerber viewer of choice and import your production data to verify that it appears correctly.