



Eagle PCB pdf export

Lorem ipsum dolor sit amet, consectetur adipiscing elit

Sakib Ahmed
18/04/2019

Introduction

A “how to” on [Autodesk Eagle](#) design export in pdf format for *DIY* or local PCB printing.

Layers to export

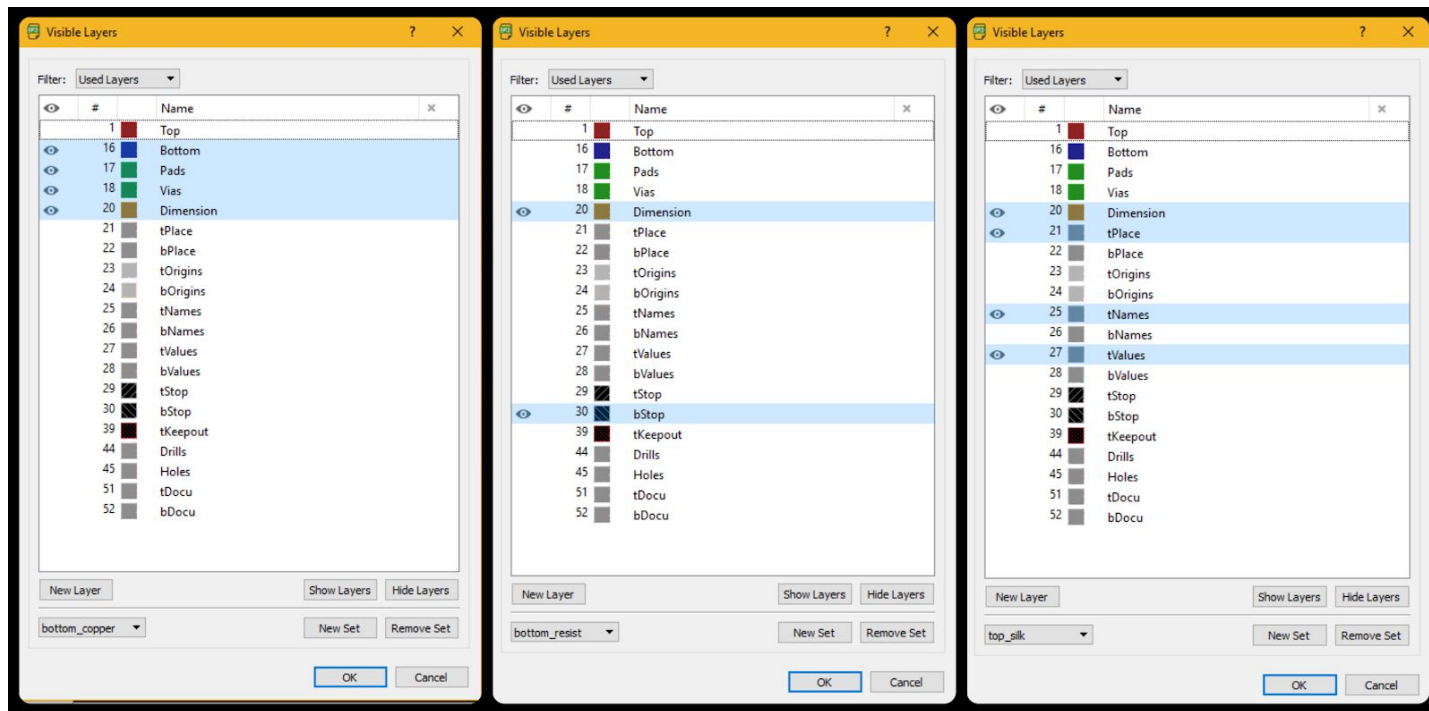
Single Layer Design

Needed files:

1. **Bottom Copper (Must)**
 - Dimension, Bottom, Pads, Vias
2. Bottom Resist/Mask (optional)
 - Dimension, bStop
3. Top Silk (optional)
 - Dimension, tPlace, tNames, tValues

Layers of interest:

(From left to right) Bottom Copper, Bottom Resist/Mask , Top Silk





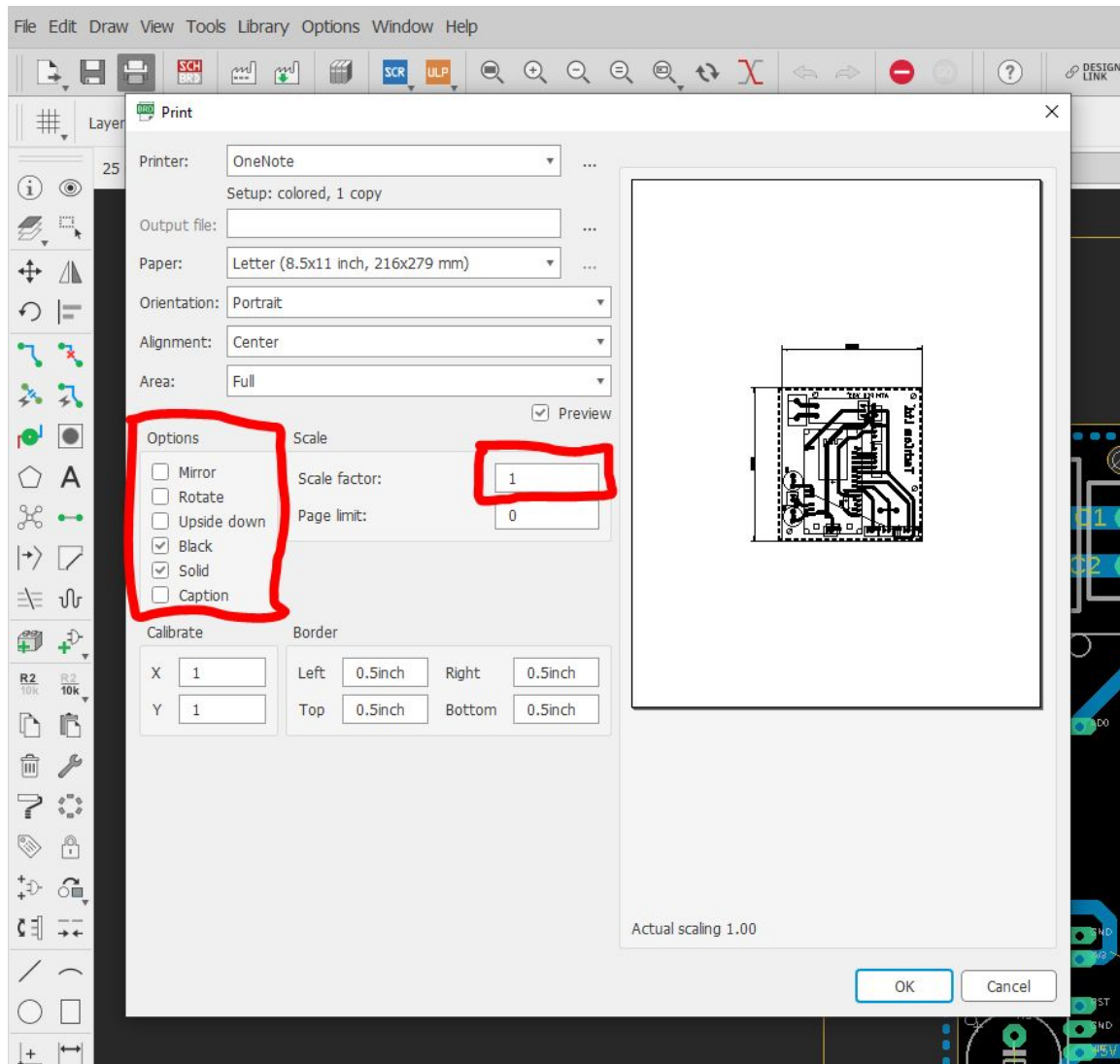
Double Layer Design

Needed files:

1. **Bottom Copper** (Must)
 - Dimension, Bottom, Pads, Vias
2. Bottom Resist/Mask (*Must for Bottom Silk*)
 - Dimension, bStop
3. Bottom Silk (optional)
 - Dimension, bPlace, bNames, bValues
4. **Top Copper** (Must)
 - Dimension, Top, Pads, Vias
5. Top Resist/Mask (*Must for Top Silk*)
 - Dimension, tStop
6. Top Silk (optional)
 - Dimension, tPlace, tNames, tValues

Export/Print (Save as pdf)

Note these marked parameters. We have to change them for different layer sets.



Remember the following rules:

1. Anything Bottom must be Mirrored
 - I.e. Bottom Copper, Bottom Silk, Bottom Resist
2. Every file should be **Black, Solid**
3. **Scale factor** must always be 1