How to Create Gerber Files for the Protomat S100 using Protel Altium Designer 2004

The goal of this document is to have you create 4 gerber files:
- project_name.gtl – top layer gerber data
- project_name.gbl – bottom layer gerber data
- project_name.txt – Excellon drill file
- project_name.gko – Keepout file gerber data

These files will be located in a subdirectory of the project folder in a folder named "Project Outputs for project_name". Only the four files listed above are necessary for Protomat S100.
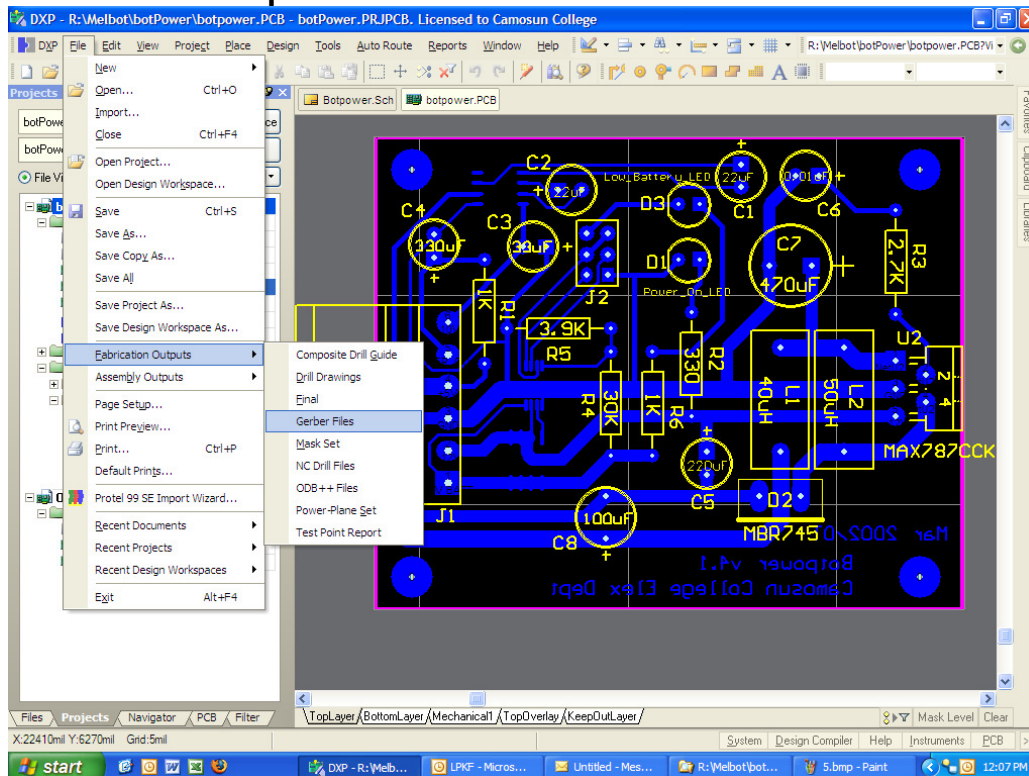
Gerber Files
- inches
- 2:4 format
- suppress leading zeroes
- reference to absolute origin
- use software arcs
- optimize change location commands
- plotter type: sorted (vector)
- no G54 data
- include top, bottom layer and keepout
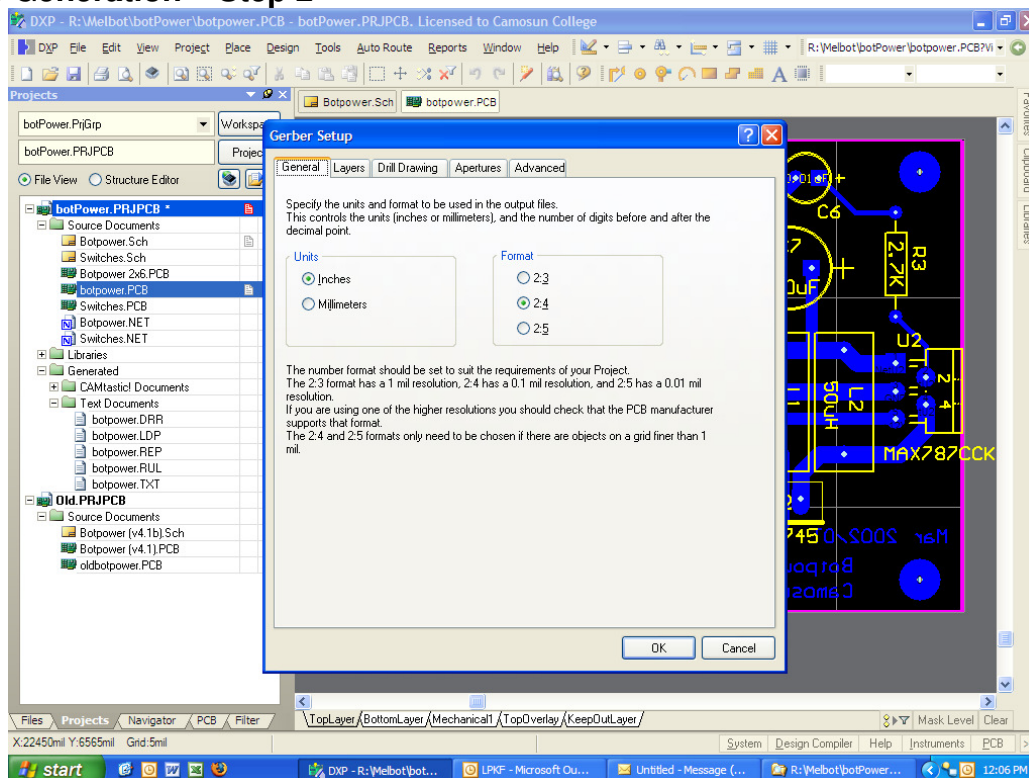- use embedded apertures (RS274X)

Drill File
- inches
- 2:4 format
- suppress trailing zeroes
- reference to absolute origin
- optimize change location commands

The following screenshots were taken from Protel 2004 SP2. You should create the gerber files in Protel the same way.
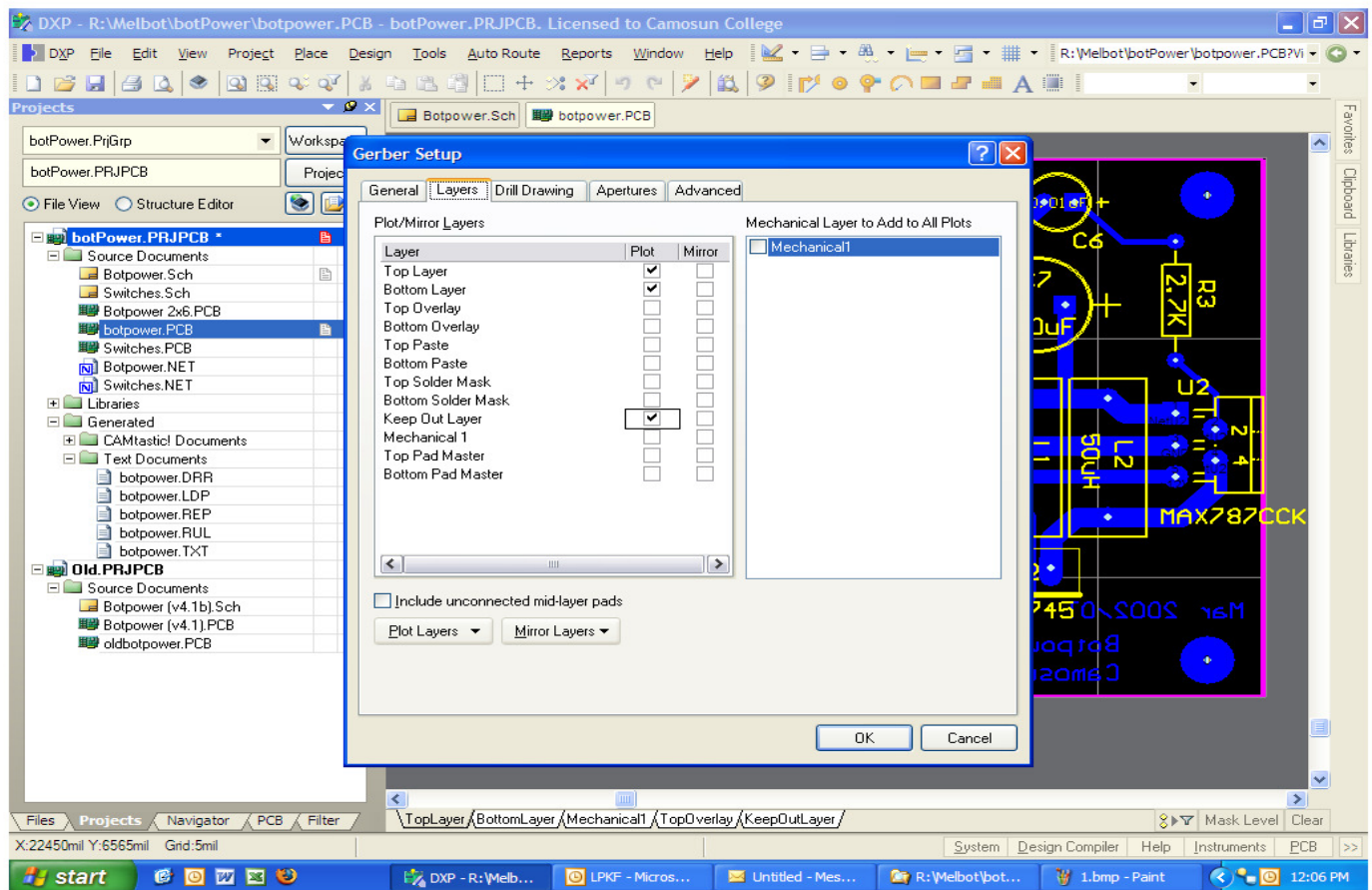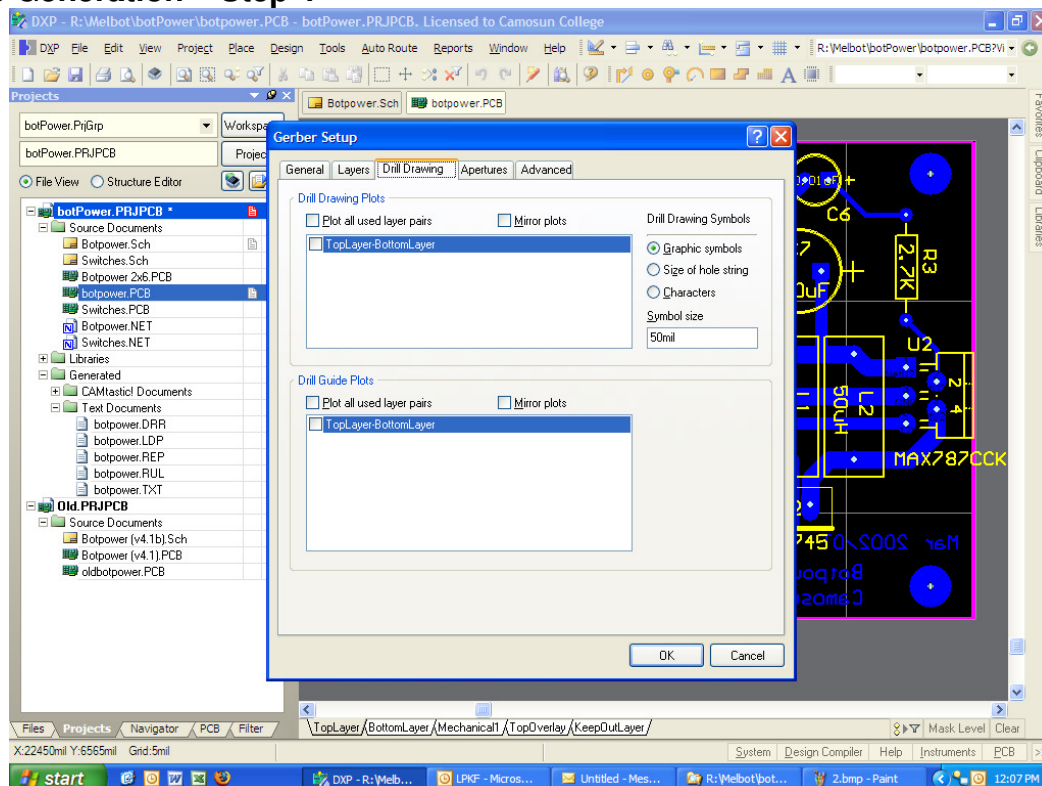
Gerber File Generation – Step 1

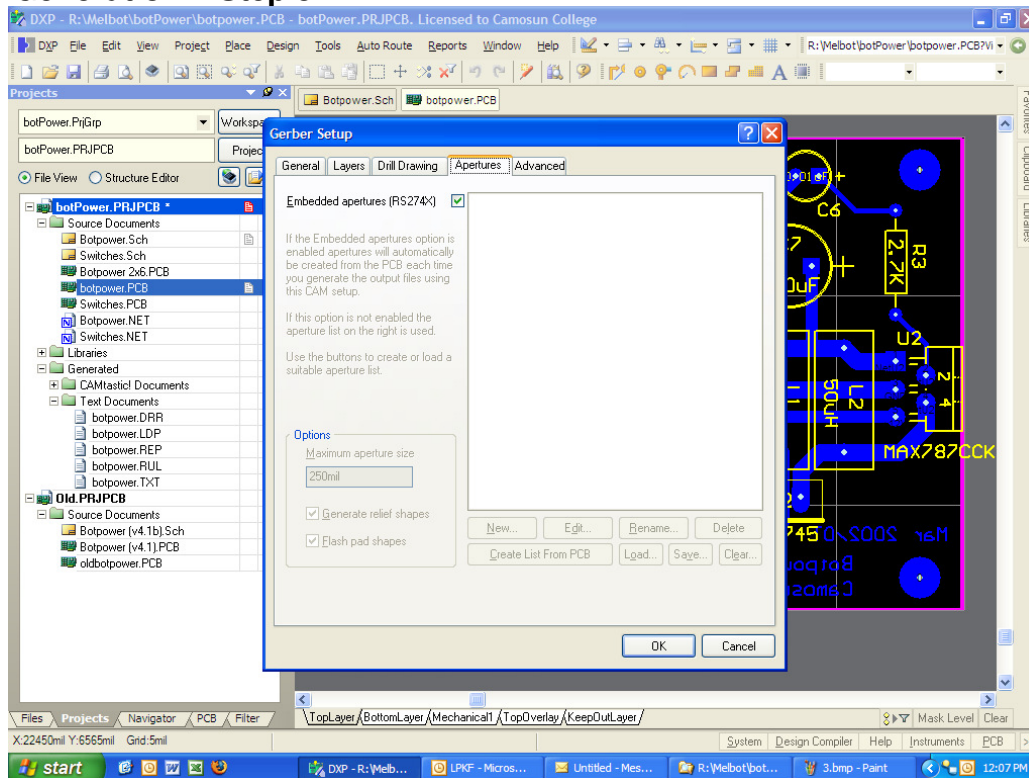## Gerber File Generation – Step 2

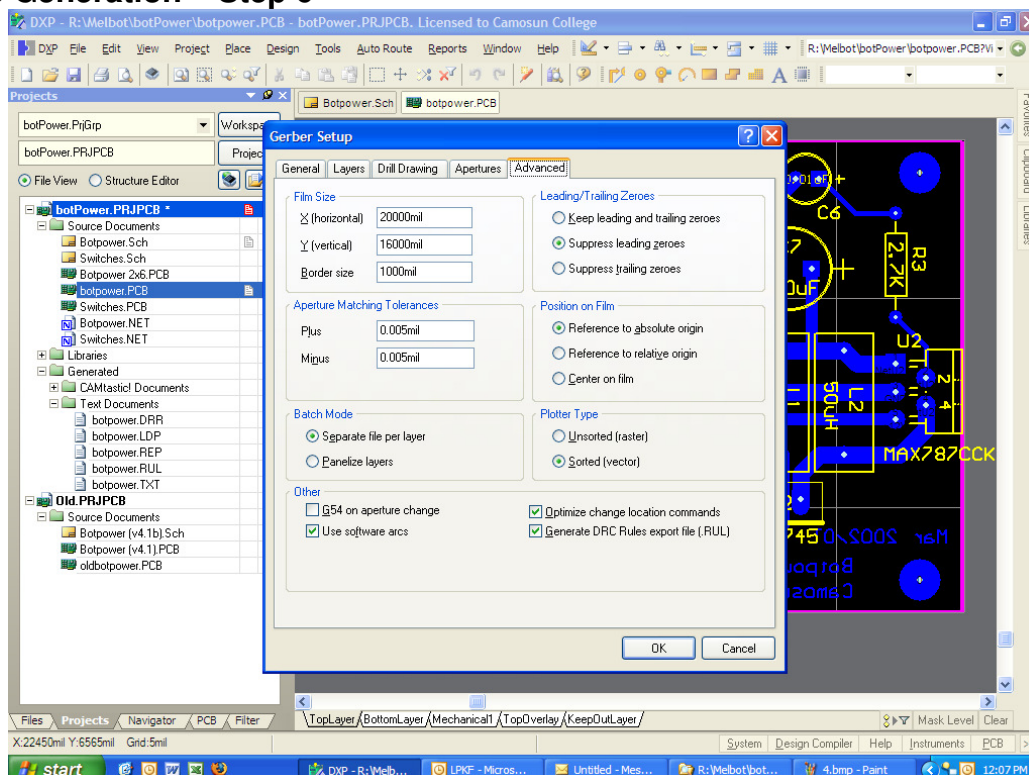# Gerber File Generation – Step 3

Gerber File Generation – Step 4

Gerber File Generation – Step 5

Gerber File Generation – Step 6

Drill File Generation – Step 1

Drill File Generation – Step 2