How to export Gerber files from Altium Designer (Protel) matching Olimex' PCB production

Design Setup from Altium

Contents

List of Figures

Version History

1.1	• Figure 20 changed to use "absolute origin" • Change format of document
1.0	• original

1. Clearance Setup

Before routing and placing anything be sure to setup clearance to minimum 10mill (Olimex can handle a minimum of 8mill but I had problems even though – a setting of 10 mill has solving the problems.)

This setting can be set in the menu: Design |Rules…

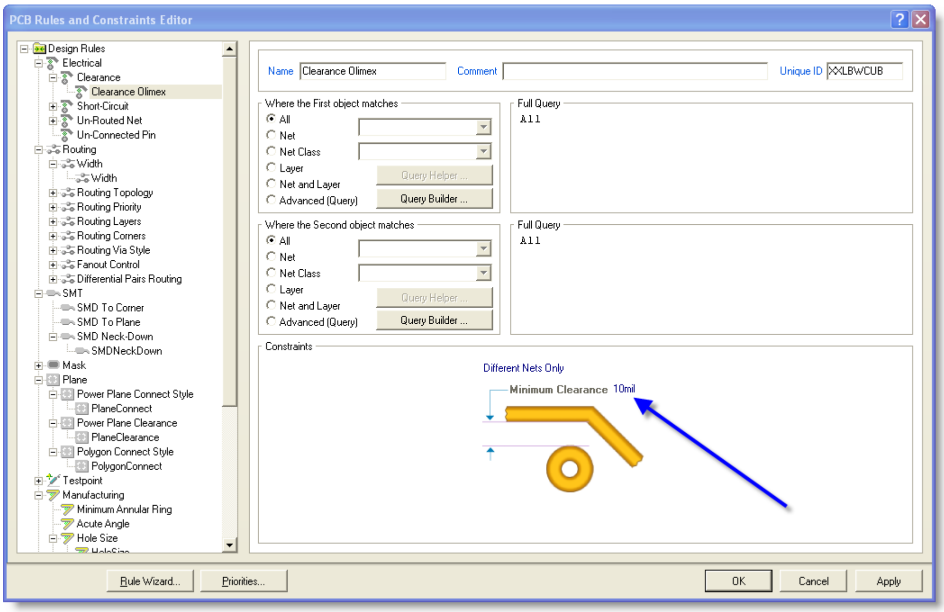

Figure 1 - Clearance Setup

Change this to 10mill as shown by the blue arrow

2. Routing Width

Be sure to use minimum 8 mill setting for the Routing Width.
This can also be changed in the menu: Design |Rules…

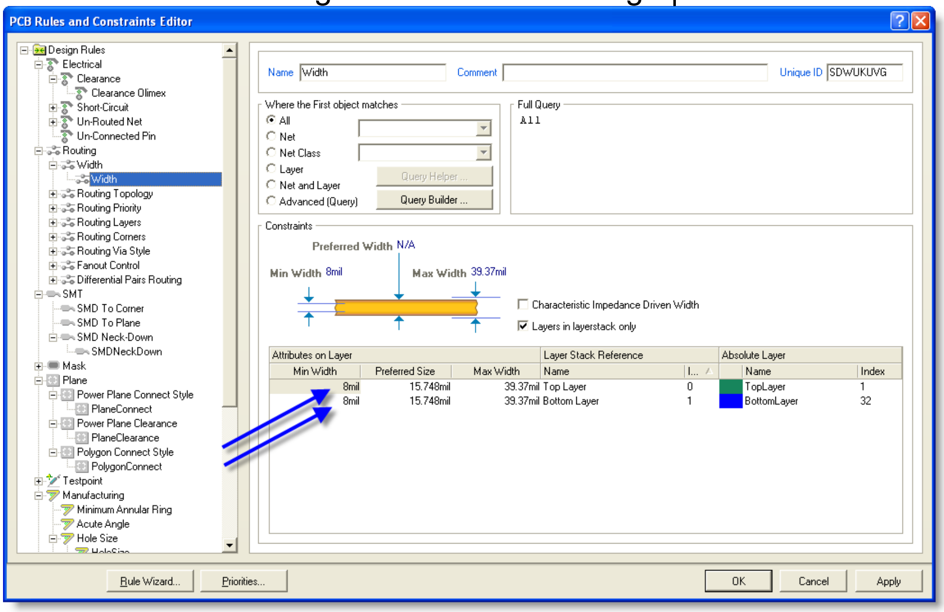

Figure 2 - Routing Width

3. Plane Connect

Go to menu: Design |Rules… and choose the setting for Plane | PlaneConnect
Also here be sure to use minimum 8 mill.

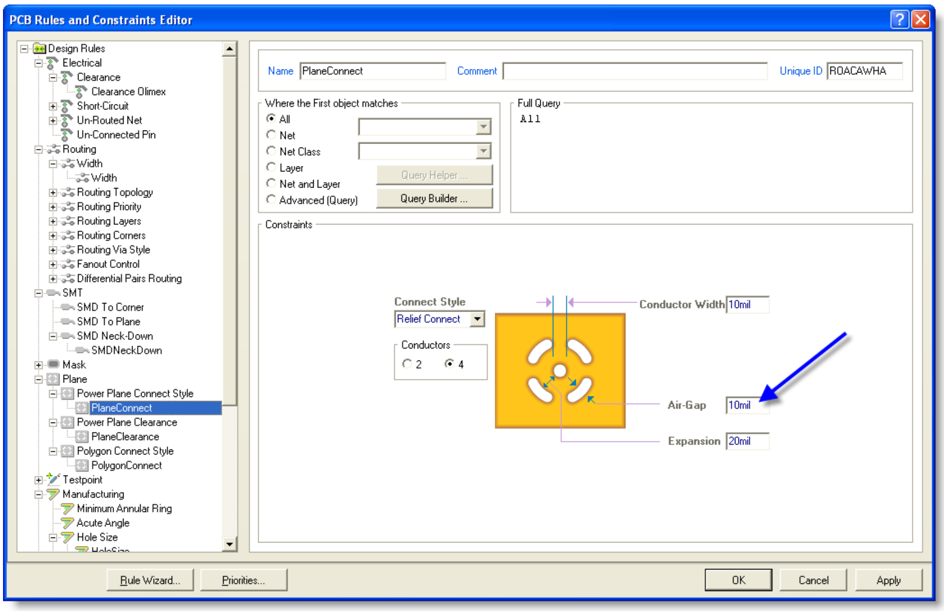

Figure 3 - Plane Connect

4. Plane Clearance

Go to menu: Design |Rules… and choose the setting for Plane | PlaneCleance
Also here be sure to use minimum 8 mill.

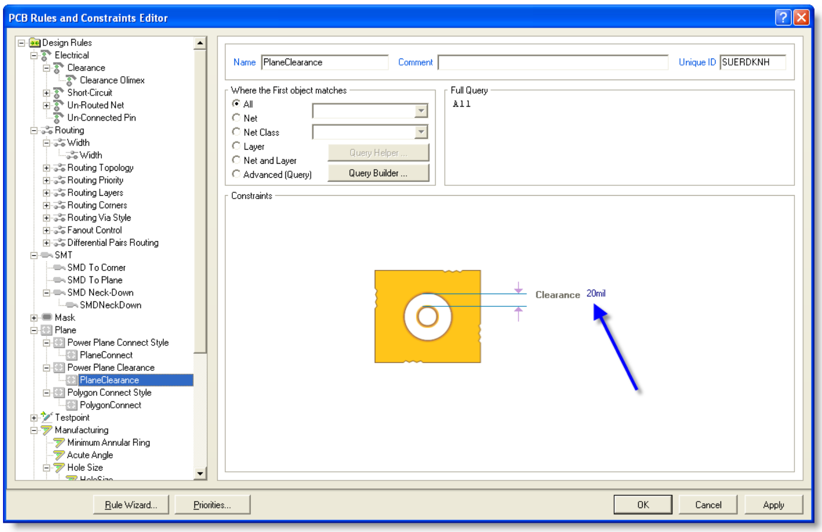

Figure 4 - Plane Clearance

5. Using Polygon Pour

When using a copper surface (Polygon Pours) also remember to use minimum of 8 mills here

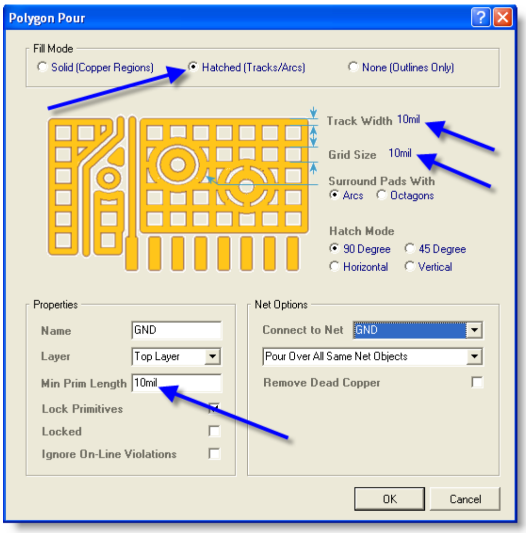

Figure 5 - Polygon Pour

NOTE ! The pour needs to be Hatched (made by Tracks and Arcs) instead of a solid copper area.

Remember to set Track Width, Grid Size, Minimum Primitive Length also to minimum 8 mill.
A setting of 10 mill works every time ☺ !

6. Design Rule Check

After finishing the design and before generating the Gerber files you should run a Design Rule check.

This check will use the setting and distances that you have already set up in the previous items following this tutorial.

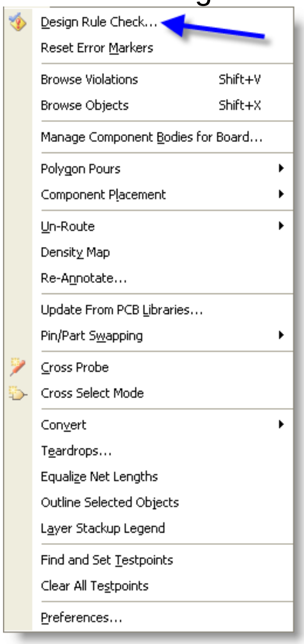

Figure 6 - Design Rule Check

Go to the menu: Tools | Design Rule Check….

Run the check and go no further with gerber files if errors are found.

Read the errors if any – correct them until no further errors are found.

7. Gerber Export

To make the Gerber files to the menu: Files | Fabrication Outputs and choose “Gerber Files”

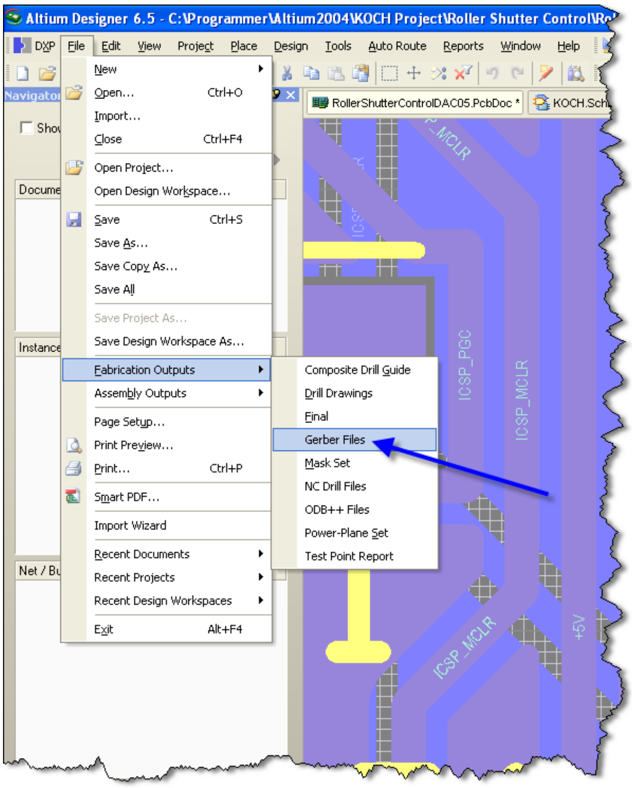

Figure 7 - Gerber Export - Tracks

You will now see 5 pages in the following dialog box

General Setting

In the General Setting set the precision to 2:4

(0,1 mill resolution)

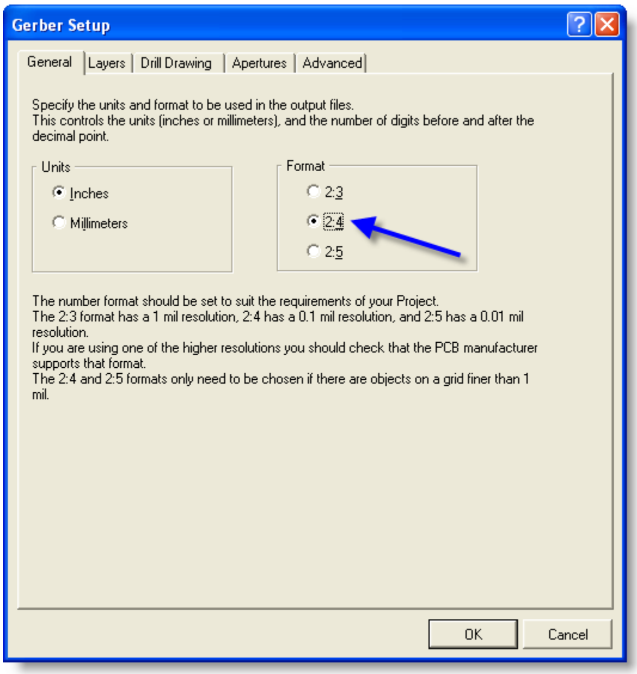

Figure 8 - Gerber Setup - General

Layers Setting

Include the layers that you want to export by marking these

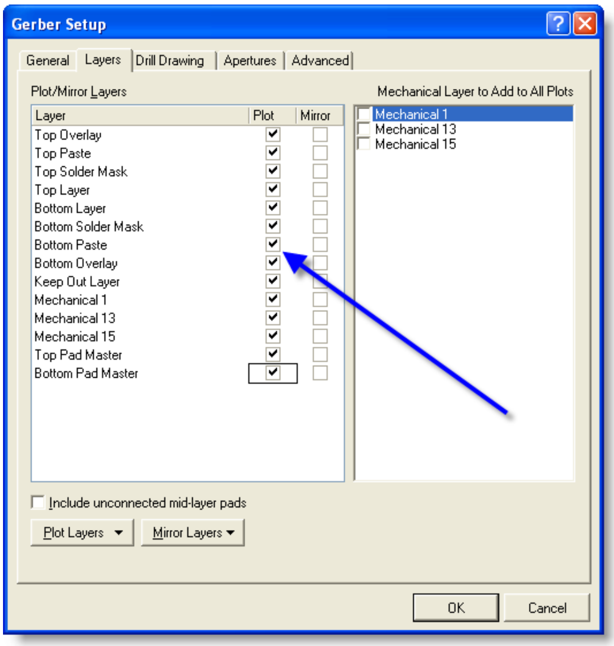

Figure 9 - Gerber Setup - Layers

Drill Drawing Setting

Mark both layers for Drill Drawing Plots

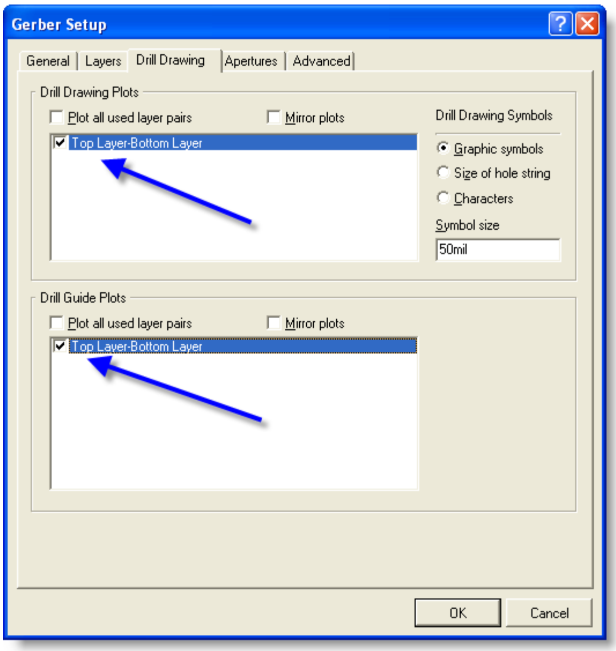

Figure 10 - Gerber Setup - Drill Drawing

Aperture Setting

Be sure to mark “Embedded apertures (RS274X)”

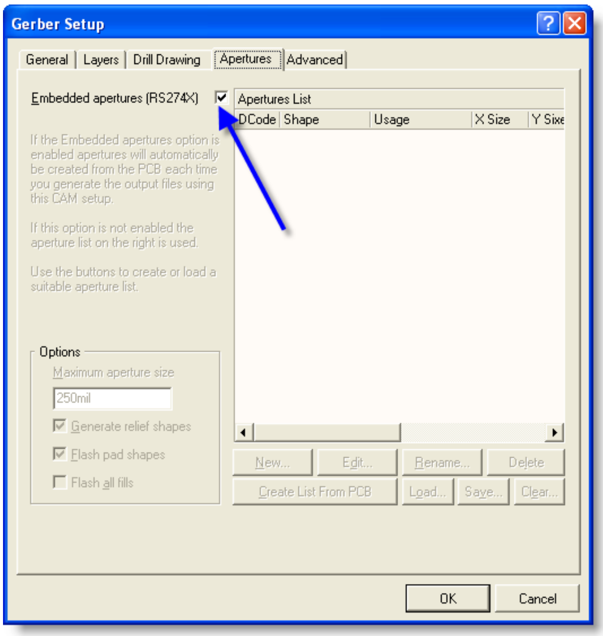

Figure 11 - Gerber Setup - Apertures

Advanced Setting

Be sure to set the Leading/Trailing Zeroes to: “Keep leading and trailing zeroes” and the Position on Film to: “Reference to absolute origin”

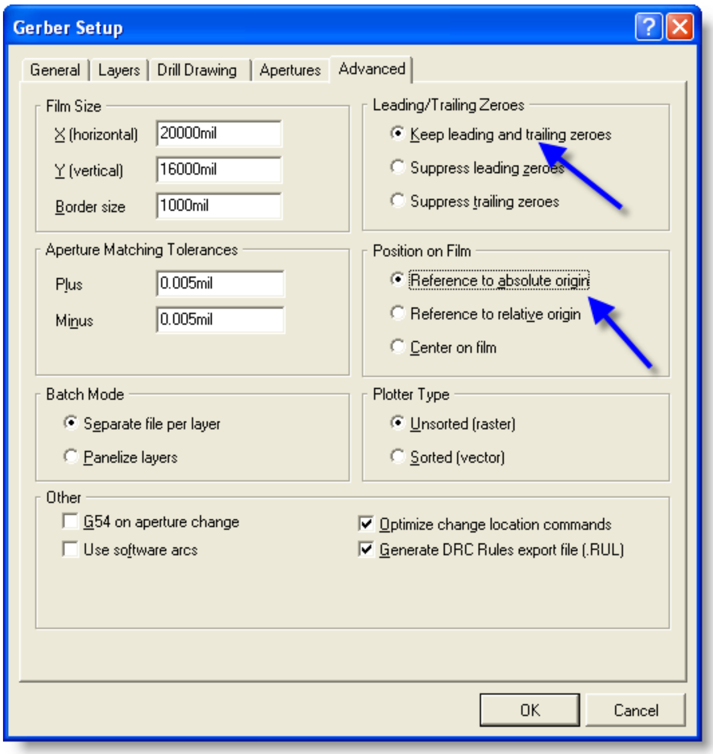

Figure 12 - Gerber Setup - Advanced

Now press the button "OK" to go further on.

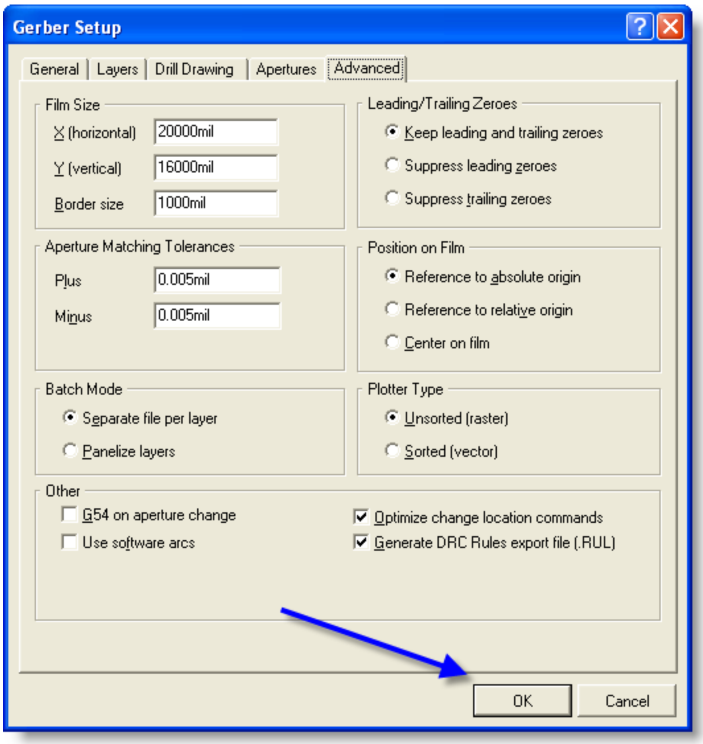

Figure 13 - Gerber Setup - OK

8. CAMtasticx.Cam file

A new page called CAMtasticx.Cam will now arrive showing your PCB.

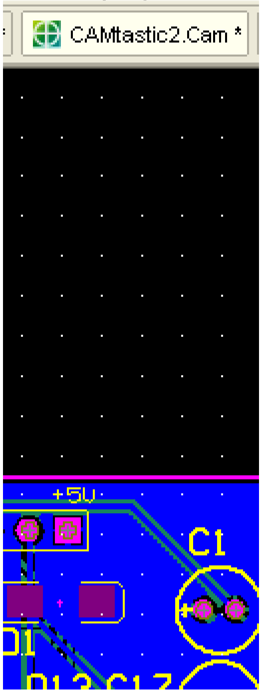

Figure 14 - CAMtasticx.Cam file

Go selecting menu: File | Export | Gerber…

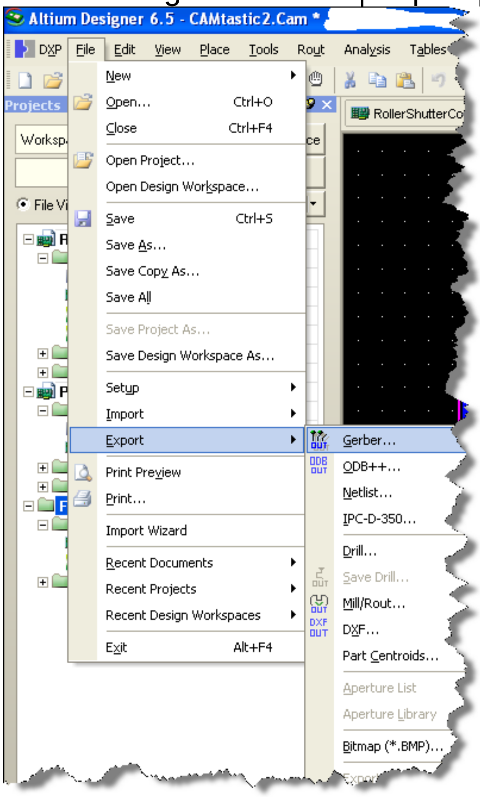

Figure 15 - Export Gerber

In the dialog box be sure to set these settings

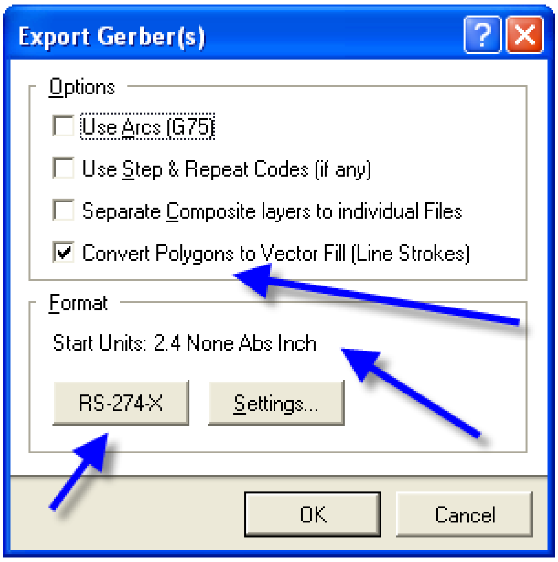

Figure 16 - Export Gerber - RS-274-X (extended Gerber)

Finally Press “OK” and you can select where to put your gerber files for each layer.

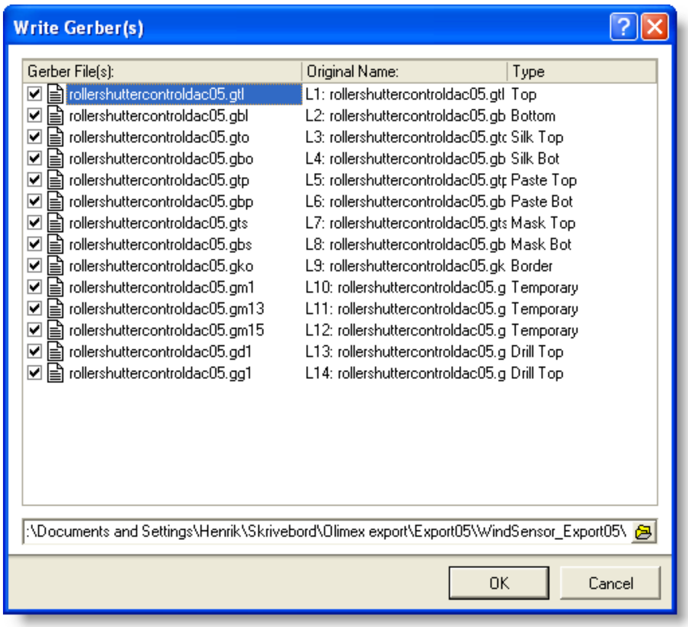

Figure 17 - Write Gerber(s)

9. Exporting your drill settings

Go back to your design PcbDoc file by pressing the Design page in top of your window

Figure 18 - Exporting Drills

Go to the menu: File | Fabrication Outputs and choose “NC Drill Files”!

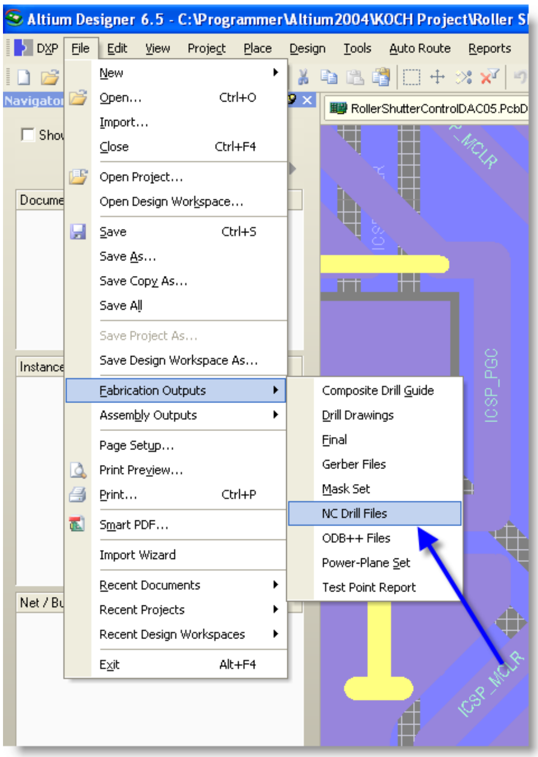

Figure 19 - NC Drill Outputs

Setup the following showed by the blue arrows

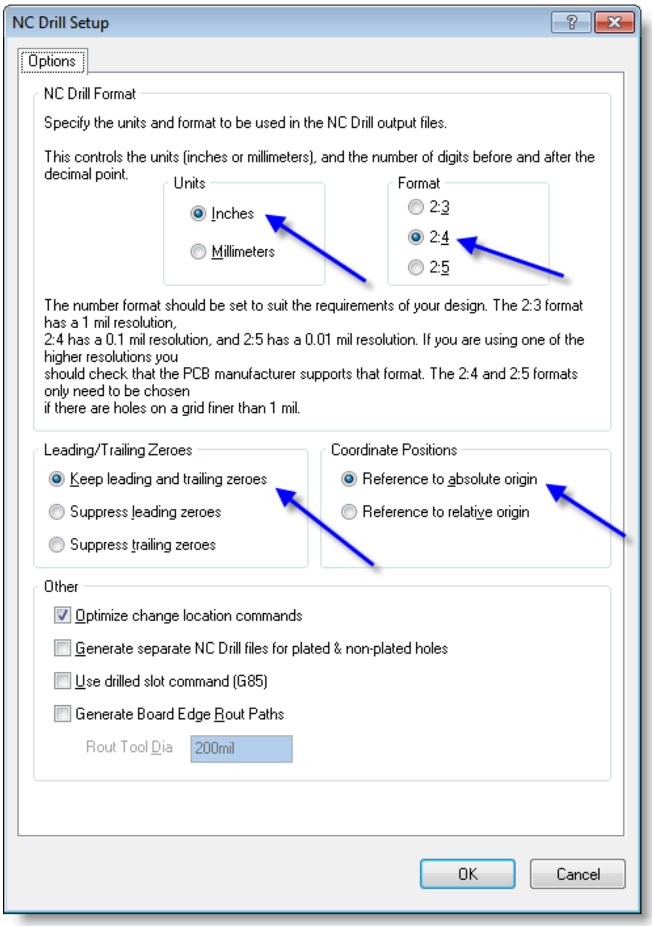

Figure 20 - NC Drill Setup (Altium Designer 2009)

Press OK and the following dialog box will be shown

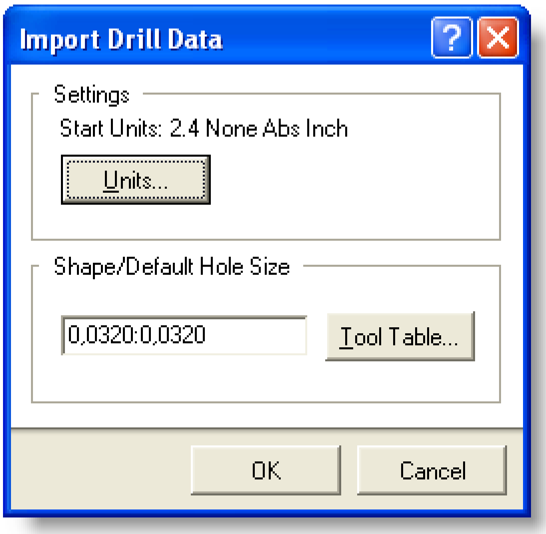

Figure 21 - Import Drill Data

Again press the button “OK”

And another CAM page with all your drillings will be shown. From within this new page
Go to the menu: File | Export and choose “Gerber”

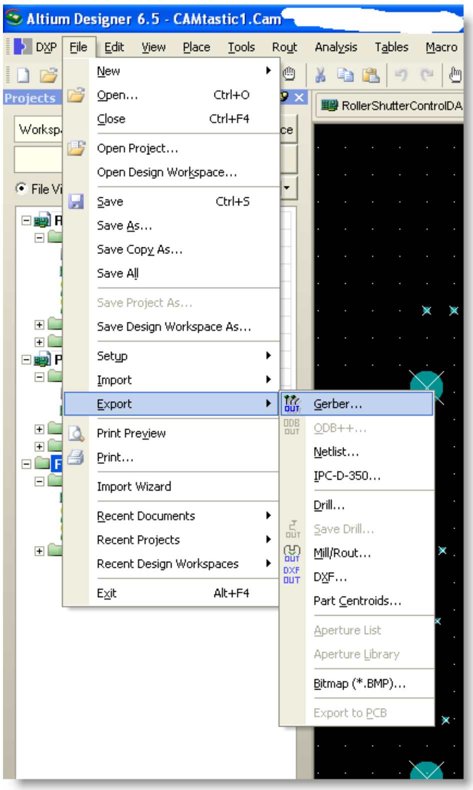

Figure 22 - Export NC Drill Files to Gerber

Now we are near the end

A new dialog box will be shown:

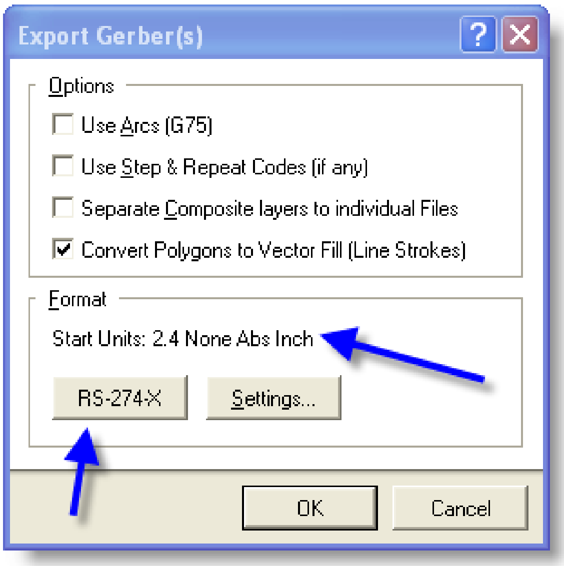

Figure 23 - Export Gerber(s) - RS-274-X

Check that this has been selected.

NB: Be sure that exactly RS-274-X has been selected !!!!!

(default is RS-274 which are not extended gerbers !!)

Finally press ‘OK’ save this gerber file and your are ready to email them to Olimex

Using the email: fastpcb@olimex.com

A happy user of Olimex

H.J. Koch 2009
henrik@koch-enginering.com