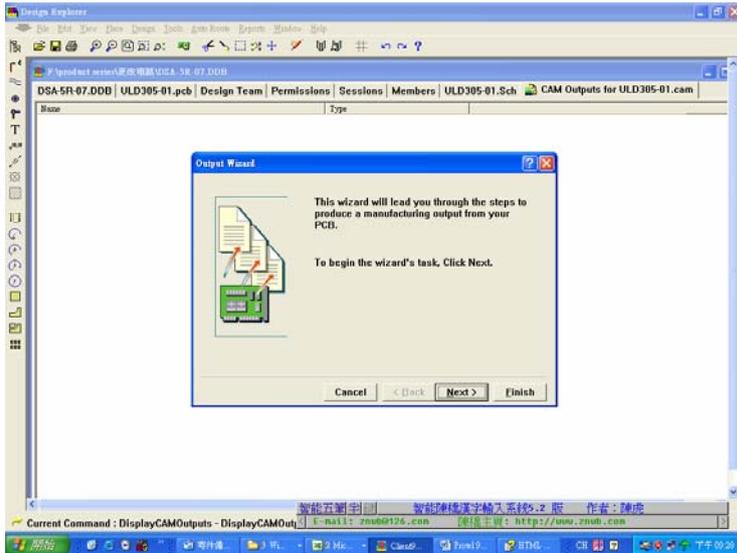
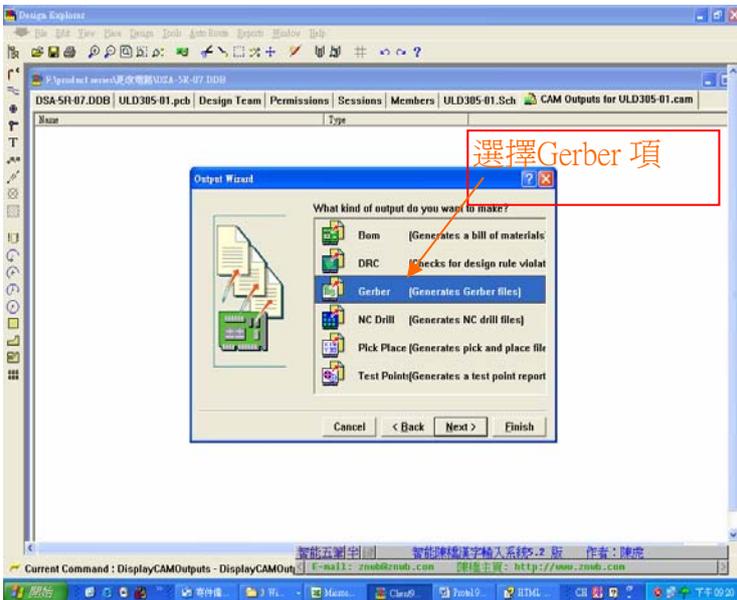


Protel 99SE 轉 GERBER 檔順序

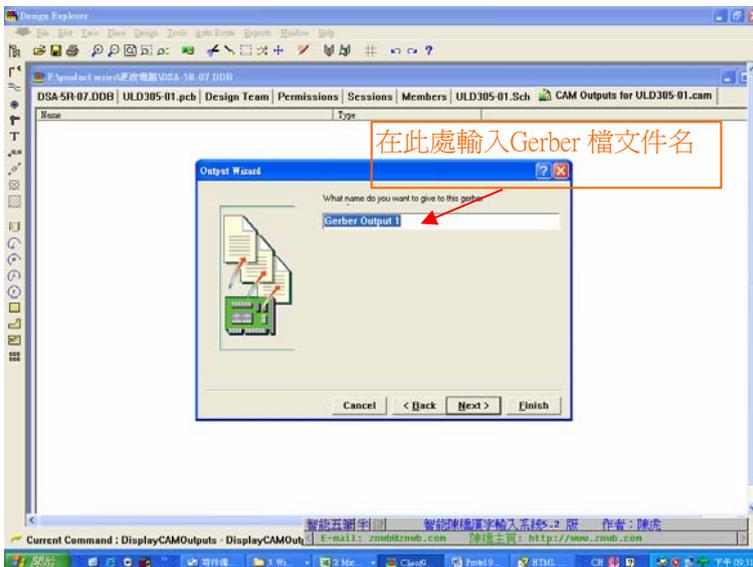
1、File- CAM Manager...



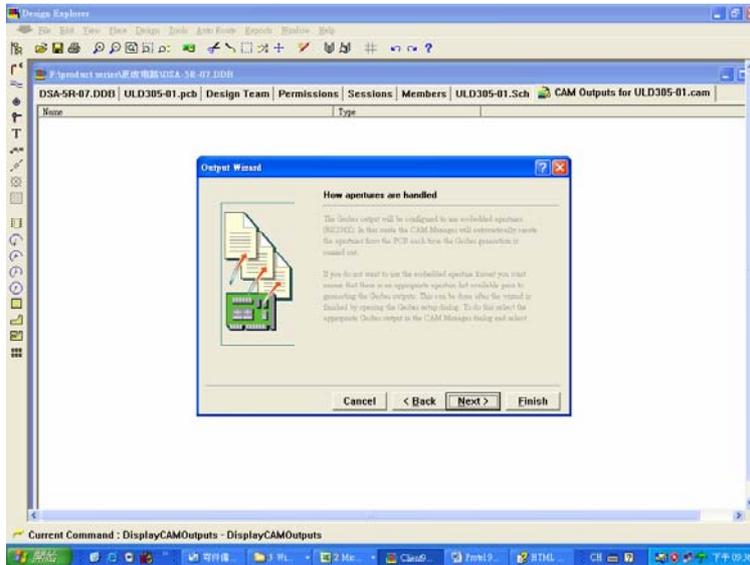
2、點擊" Next".



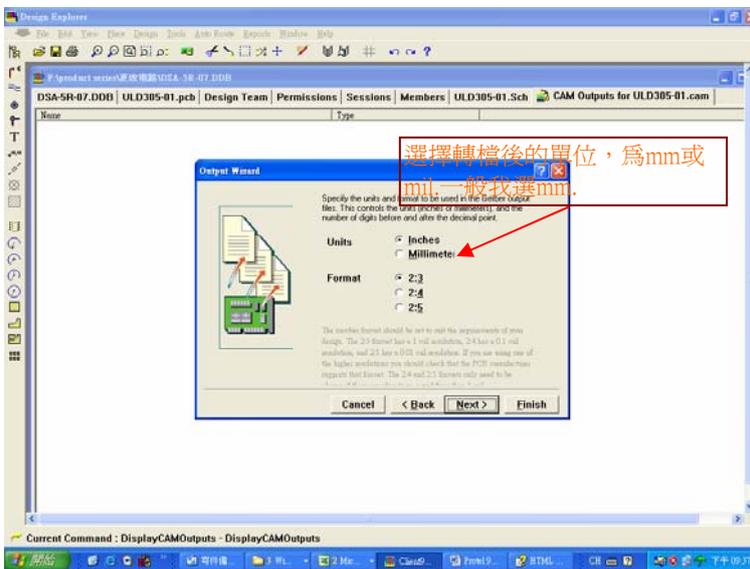
3、點擊 "Next".



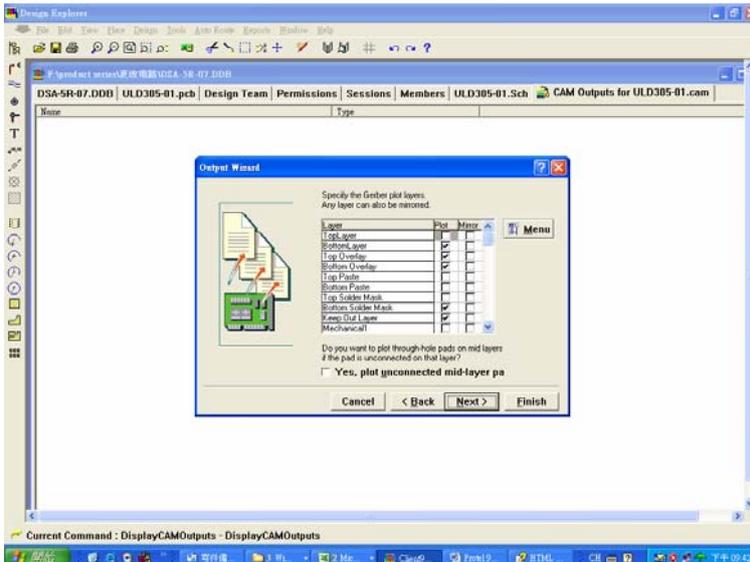
4、點擊“Next”。



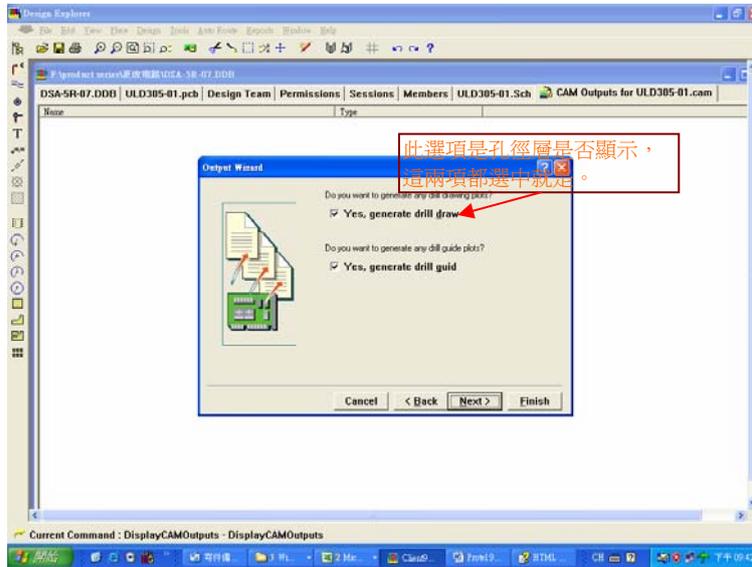
5、點擊“Next”。



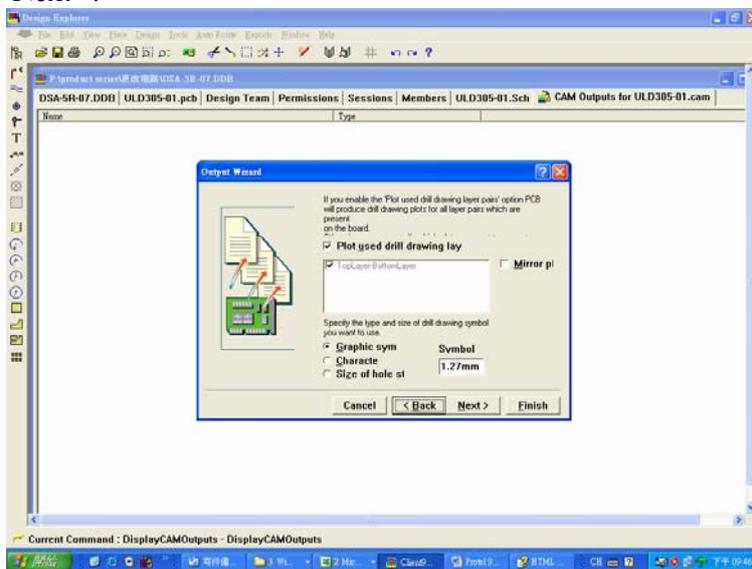
6、點擊“Next”。



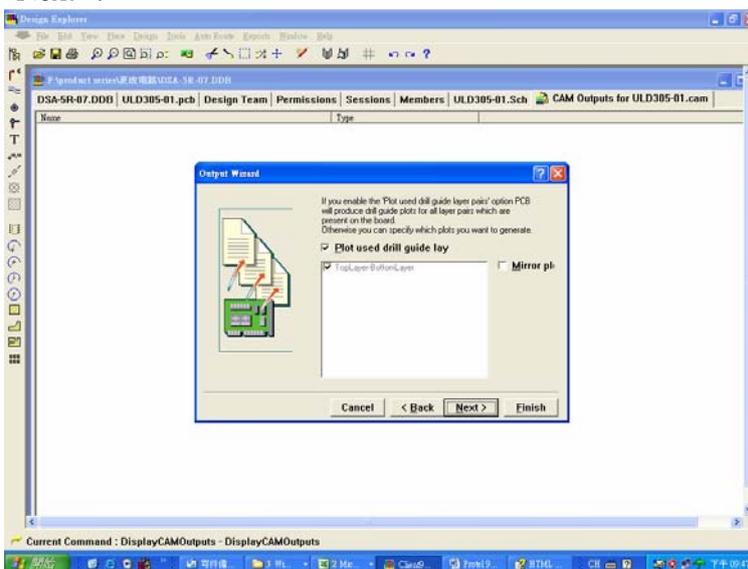
7、點擊“Next”。



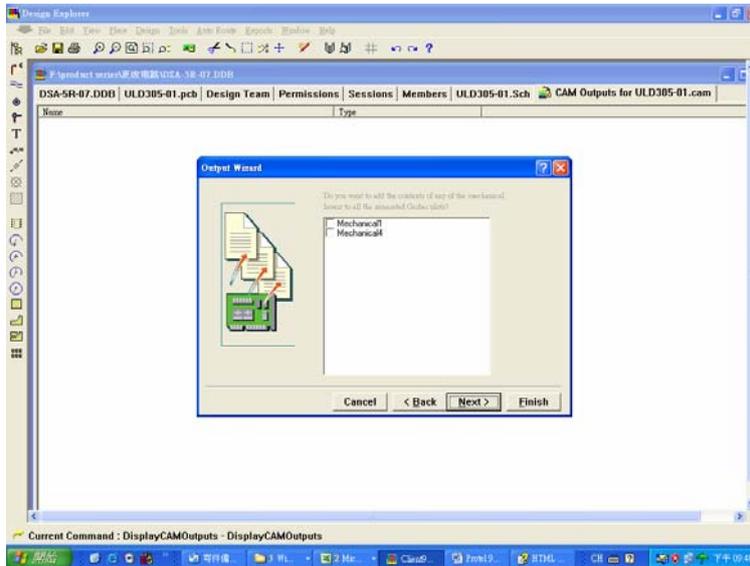
8、點擊“Next”。



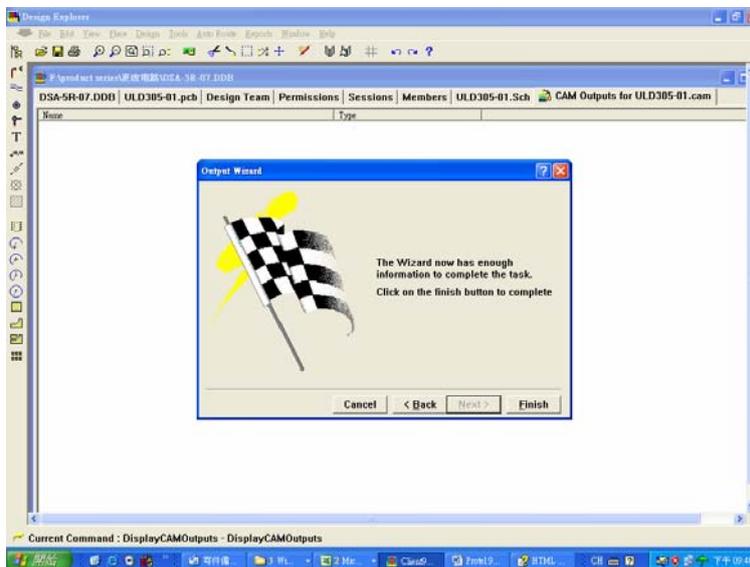
9、點擊“Next”。



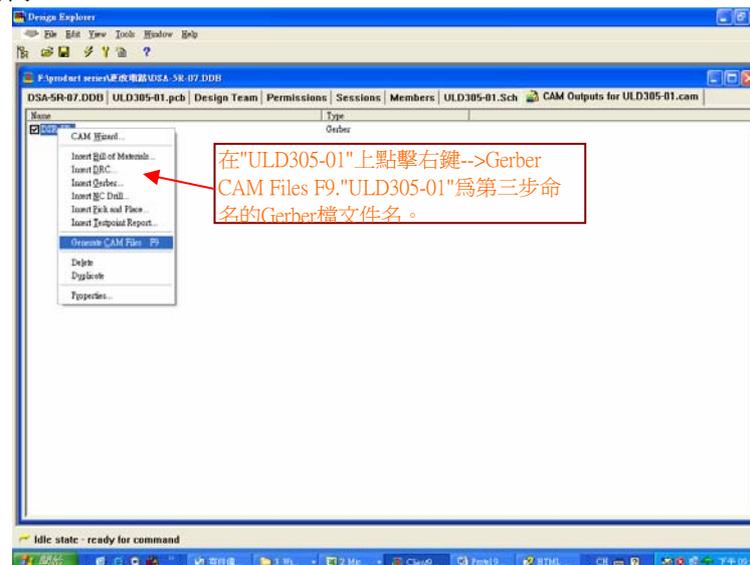
10、點擊“Next”。



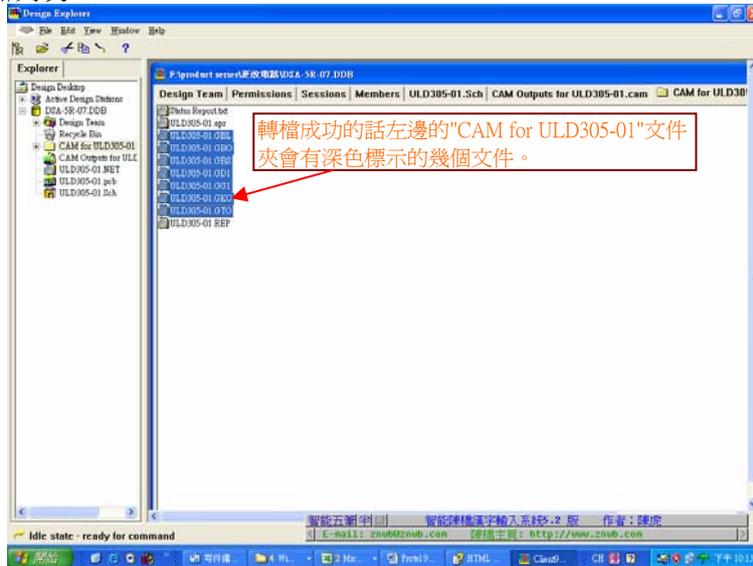
11、點擊“Finish”。



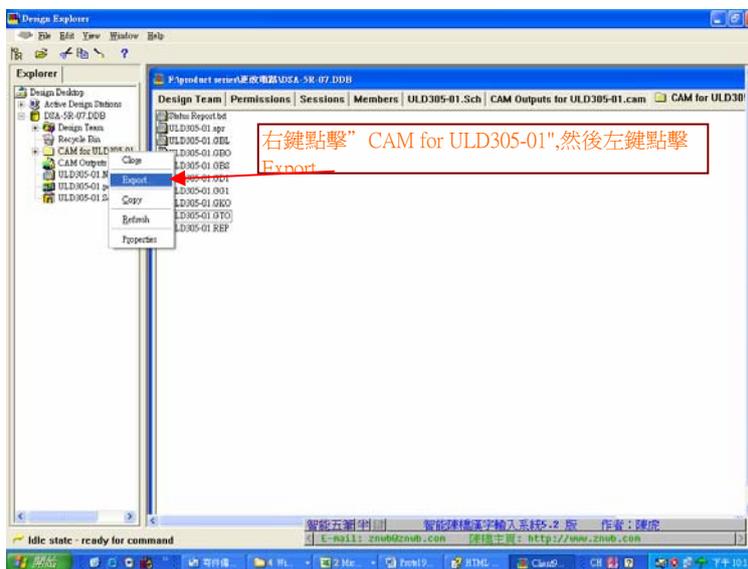
12、開始轉



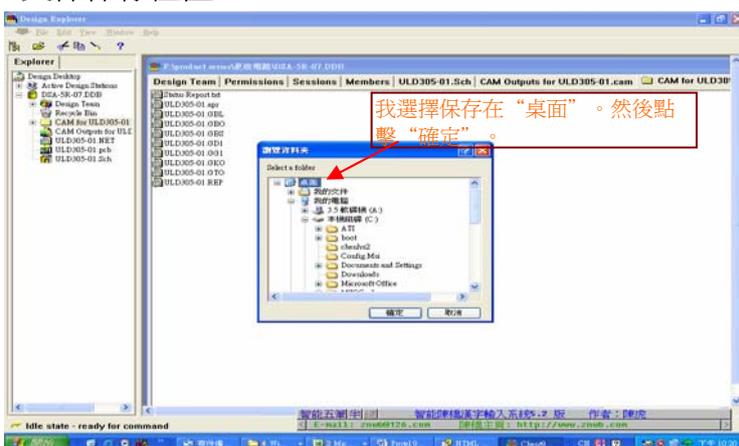
13、轉檔成功。



14、導出Gerber文件



15、Gerber文件保存位置



16、大功搞成，此檔就可以傳給電路板廠商打樣。



17、若是安裝有FabFactory v7.0軟件，就可看到如下打開的Gerber檔圖面。

