

# Generating Gerbers for ITEE PCBs from Altium

In order to generate gerber files for submission to the ITEE PCB manufacturing runs, You will need to first generate the graphic layers, and then the drill file.

## Generating the graphic layers

From your PcbDoc file, press File->Fabrication Outputs->Gerber Files. Ensure the units are set to “Inches”, and the format is “2:4”. In the Layers tab, ensure the following boxes are ticked in the Plot column: GTO, GTS, GTL, GBL, GBS, GBO, GKO. Press OK. A CAMtastic document should now be open in Altium.

In the CAMtastic document, press File->Export->Gerber. Ensure that the “Start Units:” in the window are set to “2.4 Trailing Abs Inch” and ensure that all of the check boxes are empty. The format button must be displaying “RS-274-X”, not “Fire9000” or “RS-274”; click on it to cycle through the options. Press OK and you can now save your gerber graphic layers. You may rename them to whatever you wish, but do not change the file extension.

It is possible that Altium will not generate a layer file if the layer is empty. At this time, we still require you to submit all of the layer files. You may need to add something inconsequential to the layer to force it to be generated.

## Generating the drill file

From your PcbDoc file, press File->Fabrication Outputs->NC Drill Files. Ensure the units are set to “Inches”, and the format is “2:4”. The Leading/Trailing Zeroes option should be set to “Suppress trailing zeroes”. The Coordinate Positions option should be set to “Reference to relative origin”. In the Other section of the window, ensure that only the “Optimize change location commands” check box is ticked. Press OK. A CAMtastic document should now be open in Altium.

**In the CAMtastic document, press File->Export->Save Drill. In the Select Layer box, select Layer 1 (there should only be one layer available). If there is more than one layer, you will need to merge the drill files using an external tool such as [gerby](#). Ensure that the units in the window are set to “2.4 Trailing Abs Inch”. Press OK and you can save your drill file. IMPORTANT: Some versions of Altium will save the drill file with the extension “.drl”. This must be renamed to “.txt” for submission.**