

PCB PLANET (India) LTD.

World Class PCB CAM Services



Ready to Use Scripts in UCAM

Quick Junction

This script will generate the report file in excel format or user can get direct its printout on the productivity of the job from the gerber data to quote for it.

Benefits :

- Increase performance with 100% Accuracy
- Saves Time
- Seamlessly integrates with UCAM, no additional system requirements

Before you start:

Load the layers, modify them and classify them correctly. Means choose class and subclass for each layer by choosing "Modify" from the "Layers menu" in the "Job Editor".

Inputs required by the script:

- Select outline of the PCB when dialog box asks for it. Give minimum value for job parameters, like pad-pad, pad-track and track-track spacing, in the appropriate dialog box when script asks for it.
- Fill certain informations on customer panel option, material, surface finish, board thickness, outer and inner copper start

Output of the script:

- Script will generate report file in excel format on the productivity of the job from the gerber data.
- Script will produce 3D graph for Tool no. --> Hole counts, Minimum Spacing / Tracing --> Error counts, OAR / IAR value --> Error counts.

