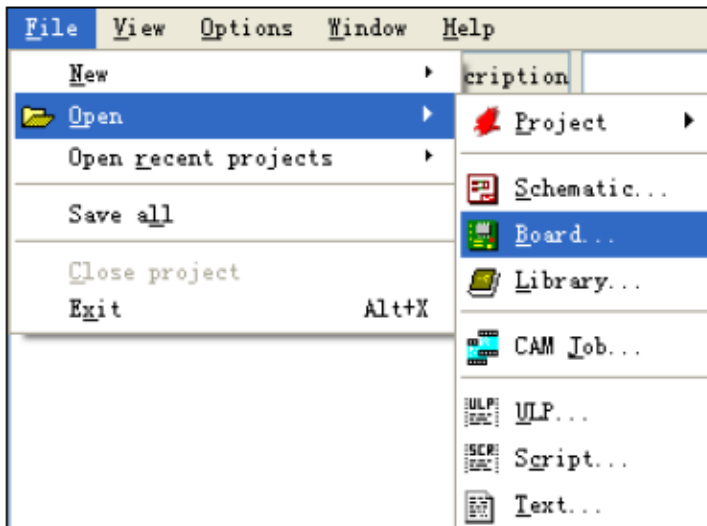
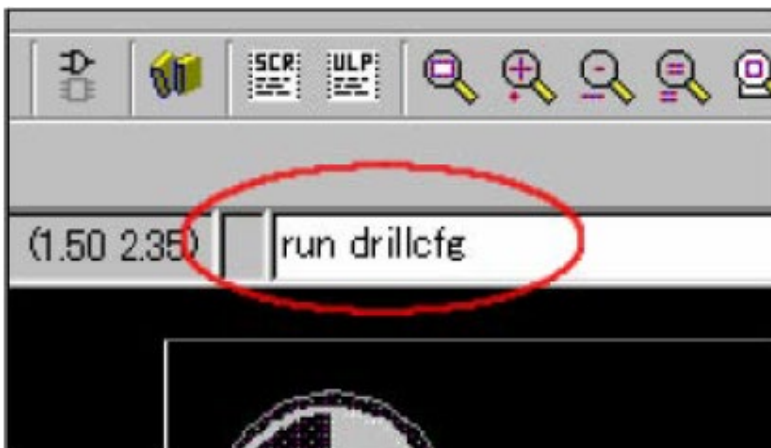


Convert your Eagle files to Gerber files:

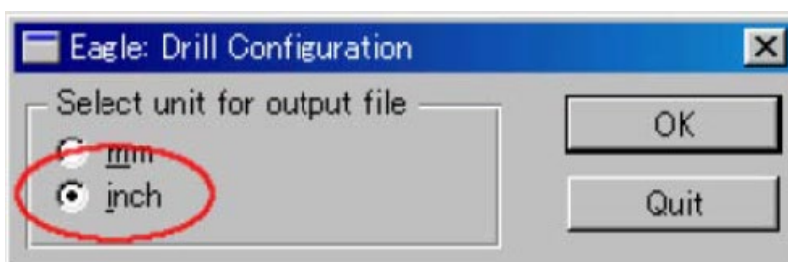
Follow all the steps below and Gerber files will be generated in the same directory as your .brd file. You can zip this Gerber files and upload the .rar or .zip file to www.pcbcore.com.



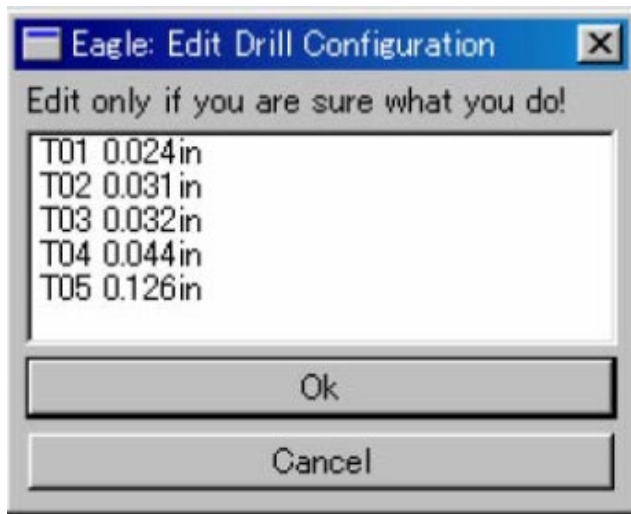
Open the .brd file in Eagle (board file).
Create a drill rack file



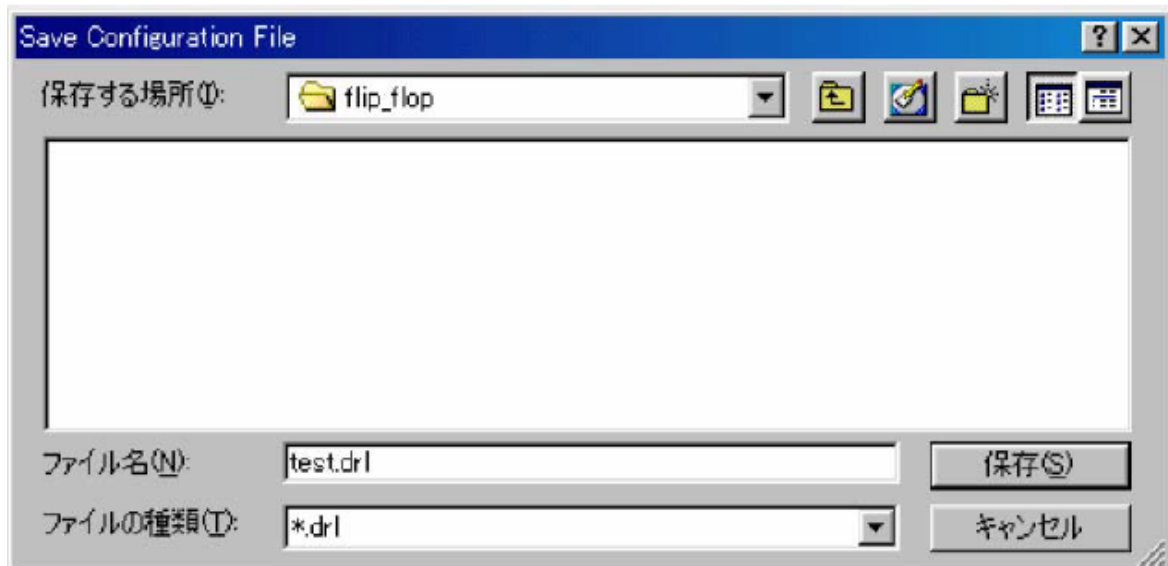
Type in "run drillcfg" to a command bar, and push an enter key.



Choose an inch as a unit, and push the OK button.

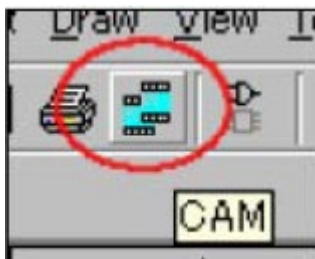


The list of the drill size used now is displayed. On this screen, push the OK button.

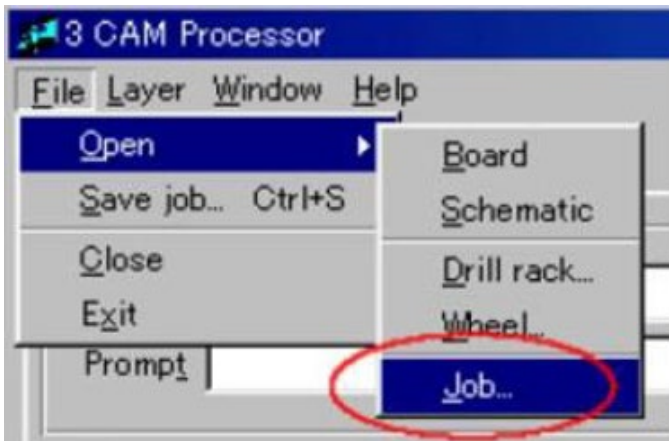
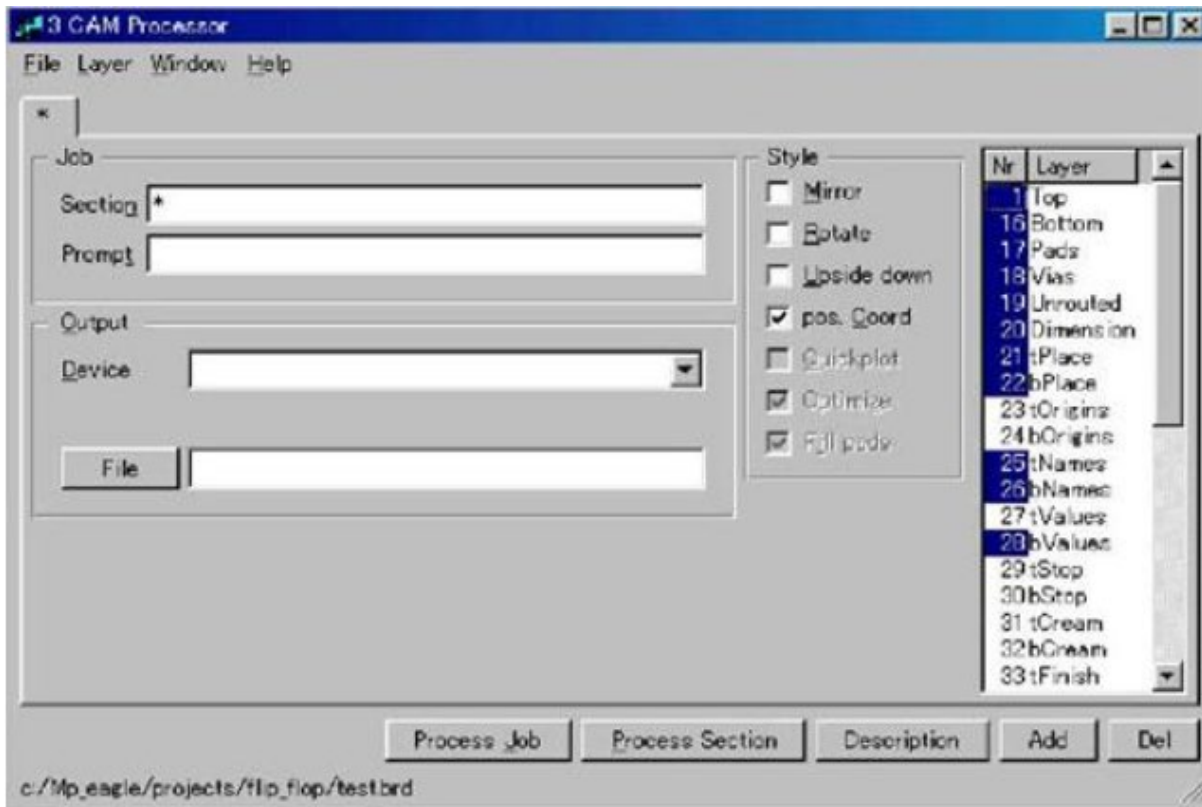


The dialog which saves a Drill Rack file (* . drl) is displayed. You have to save this file in the same folder as a board file (* . brd). The information written to the board file is used in CAM Processor

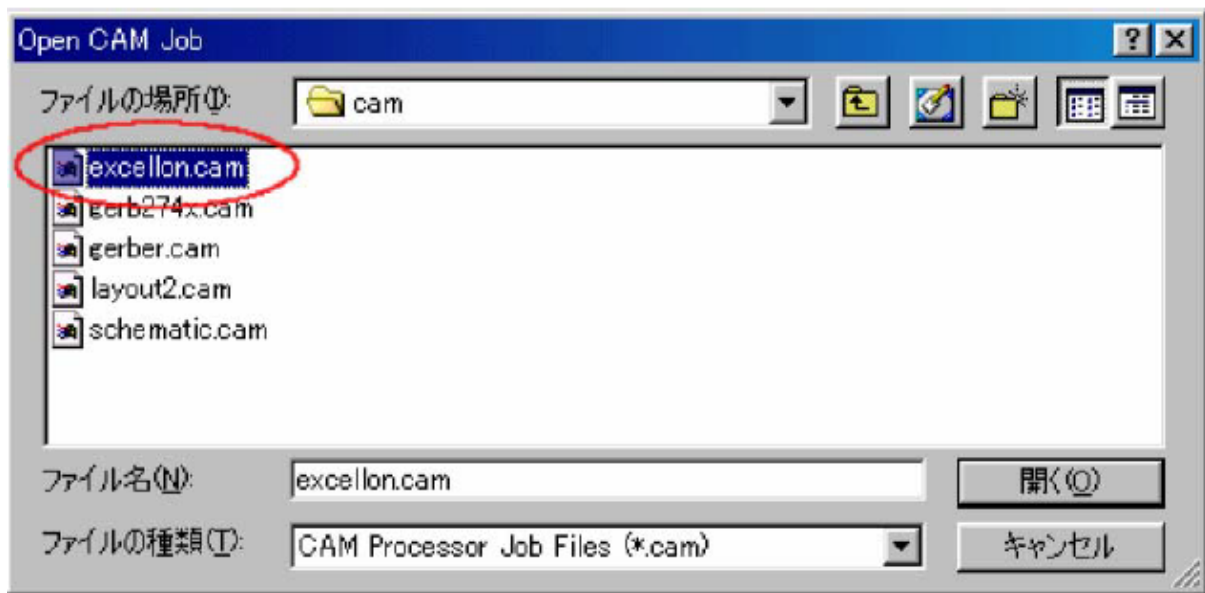
Create the Excellon drill file:



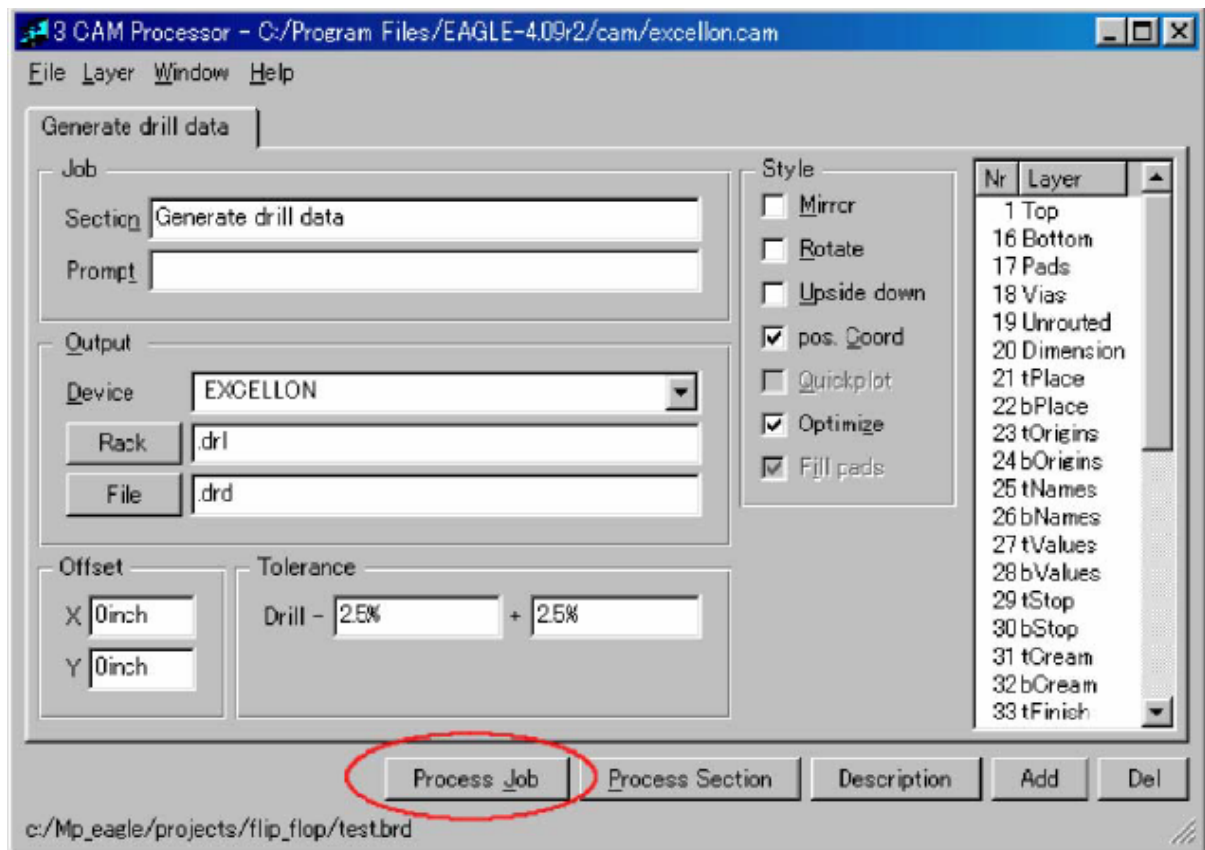
Choose “CAM” with an icon bar. Thereby, the following dialog of CAM Processor is displayed.



Choose "Open" by the file menu of a CAM Processor dialog, and choose "Job" further.



Choose “excellon.cam” from the list displayed, and push the open button.

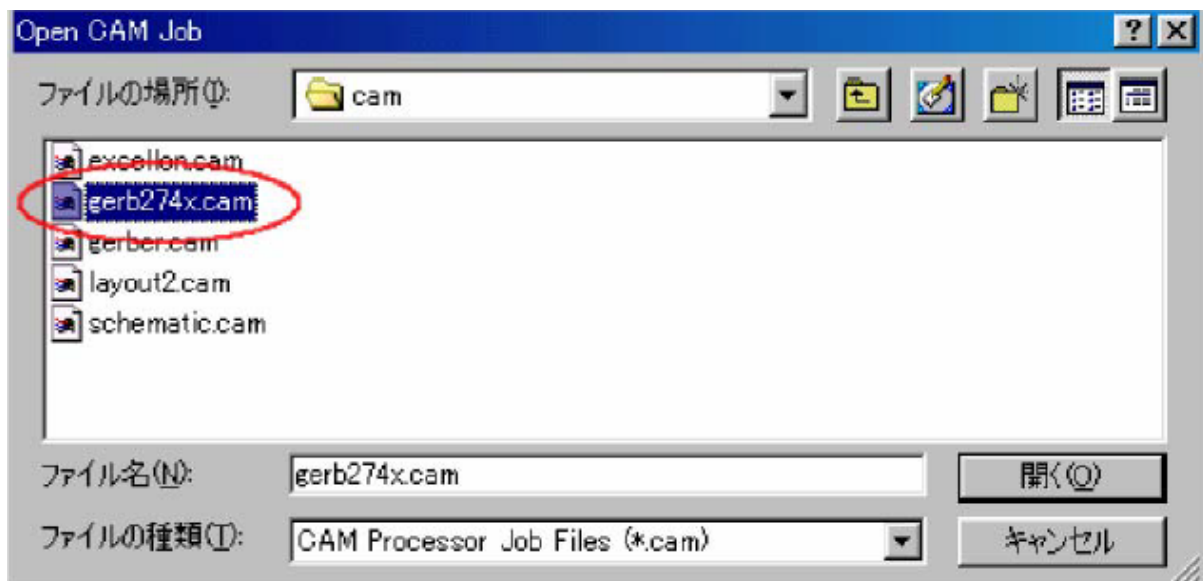


Check the items setup of Generate drill data, and push the “Process Job” button. The setting items are left default when details are not known. The Excellon drill files (*.drd, *.dri) are made by this processing.

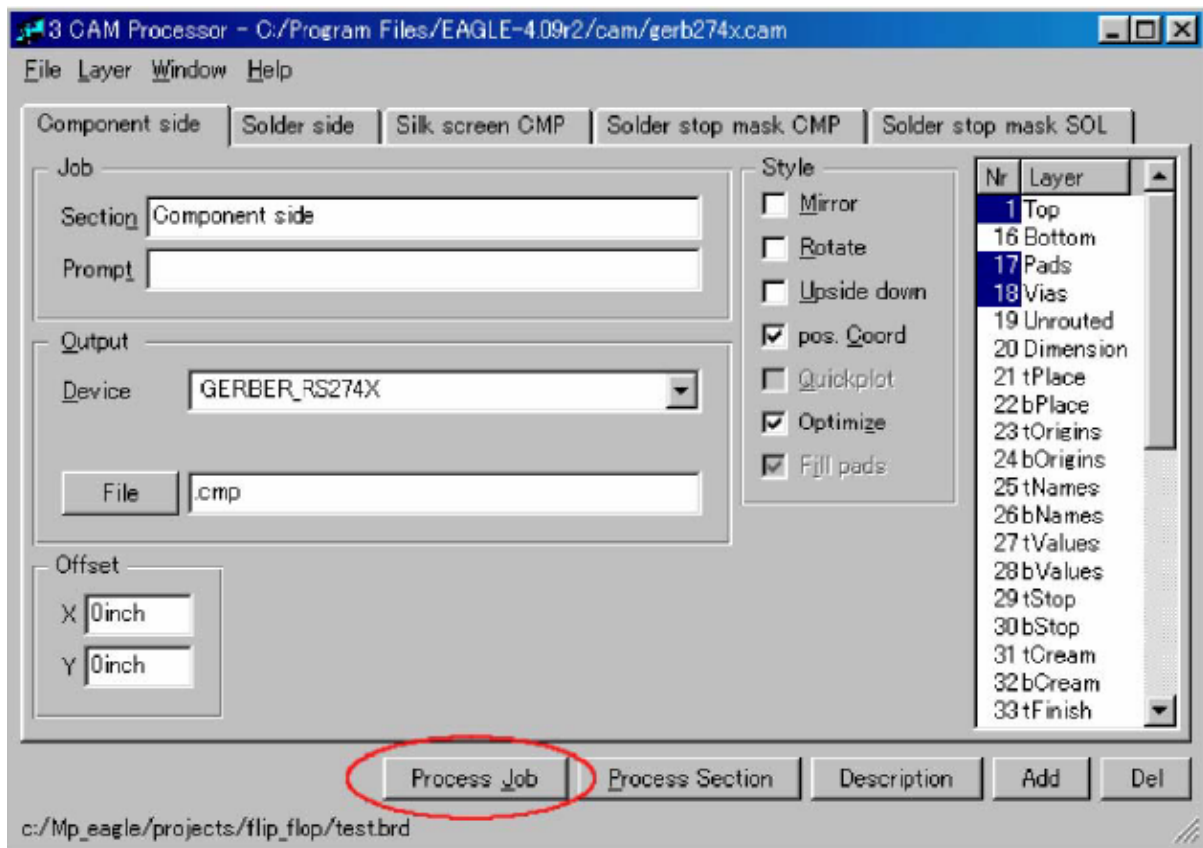
Create the Gerber files:



Choose File -> Open -> Job like creation of Excellon drill files.



Choose "gerb274x.cam" from the displayed list, and push the open button.



Leave the items setting as default when details are not known, and push the “Process Job” button.

The Gerber files (*.cmp, *.sol, *.plc, *.stc, *.sts, *.gpi) are made by this processing.

The following Gerber files can be made from the above processing.

- *.drl Drill rack data
- *.drd Excellon drill description
- *.dri Excellon drill tool description
- *.cmp Component side data
- *.sol Solder side data data
- *.plc Component side silk screen data
- *.stc Component side solder stop mask data
- *.sts Solder side solder stop mask data
- *.gpi Gerber photoplotter information