File Processing

A PCB design file can be processed based on the design specification it follows. File processing is done in multi steps. First we have to identify the file format and then drill down into each file to extract design information.

# File identification

It can be done using one of the following ways. Before we do the identification, we first have to find what format the design package is using. Sometime we get Gerber and ODB++ files together in the ZIP file from customer.

## Using file extension

Set of Gerber files can also be processed by looking at extensions. This method does not give complete information about the file but can be used to identify the file use. File extensions can be different based on the software used to create the files.

## Using standard or custom naming standard

We can also look at the file naming convention to figure out some details about the file’s purpose. This is only possible if all user’s follow one naming standard that we ask for or they give us their preferred format and always adhere to that naming. Again, this is not the most reliable way for file identification.

## Parsing the file and extract information

There are certain standards available for PCB design files. Each specification provides rules that we can refer and parse the files to extract important design information. Following are few common specifications -

* IPC 2581C
* Gerber
* ODB++

## Extracting design attributes from the files

Once we identify the format specification of the design package, we can parse the file’s content and look for important design attributes. System is designed to handle all possible specifications but currently implemented to process Gerber and ODB++ only.

System will parse the files and extract attributes by looking for information as per the design specification standard. All this information will be captured and stored in database for future reference.