**Design input validation Software**

Understanding of the requirements

* Once all the PCB design files are ready for manufacturing, they should be placed in the root directory of a folder and archived.
* Design to be exported to a format like Gerber or ODB++
* Customer need to provide your us with the following files -
  + Top Silkscreen
  + Top Soldermask
  + Top Copper
  + Bottom Copper
  + Bottom Soldermask
  + Bottom Silkscreen
  + NC Drill
  + Board Outline (sometimes embedded in another file)
  + Inner Layers (optional, depending on design)
  + Fab Drawing (optional, but recommended)
  + Readme (optional)
  + Any other that we missed?
* The file requirements will vary depending on the design.
  + What are different type of designs? -Single sided, with or without solder-masks, multi layered ?
  + PCB design software applications require the Drill file to be exported as a separate process from the rest of the design files, this is the most commonly forgotten file.
  + The Board Outline is another file that is often overlooked. This file can either be a stand-alone file, or be included within other layer files.
  + What are the common missing files you have experienced in past experiences?

**PCB Design File Extensions**

*ProtelAltium® EAGLE OrCAD™*

Top Silkscreen .gto .plc .sst

Top Soldermask .gts .stc .smt

Top Copper .gtl .cmp .top

Bottom Copper .gbl .sol .bot

Bottom Soldermask .gbs .sts .smb

Bottom Silkscreen .gbo .pls .ssb

NC Drill File .txt .drd thruhole.tap

Board Outline .gm1, .gko

Internal Layers .gp1, .g1

Fab Drawing .pdf .pdf .pdf

Readme .txt .txt .txt

**Common Tool Settings**

Basic settings in PCB design:

Units = Inches

Format = 2.3

Embedded Apertures = RS274X

For Fab we need:

1. Gerber

2. Drill layers (this can be on Gerber, Excellon ASCII or Excellon EIA). Prefer is Exellon ASCII

3. Fab Drawings (this can be on Gerber, DXF, DWG, PDF).

4. Layer Sequence ( this can be, readme, Fab Drawing, Gerber Header)

5. ODB++ (This could contain all the above)

For assembly we need:

1. All of the above

2. BOM

3. Assembly Drawings (this can be ODB++, gerber, PDF, DXF, DWG)

4. Paste Layers (this can be gerber or ODB++)

5. X&Y