



## ALLEGRO LAYOUT PCB CAD FILE NAMING CONVENTION (DELIVERABLES & README)

### NOTES:

Customer Delivery	Email delivery as attachment preferred, FTP delivery possible. Send data to the customer only unless they have specifically requested delivery to their assigned fab/asy house.
Manufacturer Delivery	Never send IP data (schematic, etc.) to the manufacturer. When delivering files, separate emails/FTP sites should go to the customer and to the manufacturer.
Archiving	After all processing has been performed in CAM, push all FILES data to history and copy the “*” files from CAM to FILES.

### CONVENTION: <PCB PN>.EEE

<PCB PN>.EXT	Use Bare PCB Board Part Number unless specified as in the assembly file data.
<PCB PN>.QQQ	Qualifier used for layer reference in gerber files (i.e. L1 – Top layer , ST - silk top)
<PCB PN>SS_PP.EXT	Optional qualifier used for TEMPORARY engineering iteration (i.e. S#[schematic iteration]_P#[PCB iteration])
<PCB PN>_REV-#.EXT	Drawing revision (shown as _REV-1.EEE or _REV-A.EEE)
<PCB PN>.EXT	File extension used to describe file type (i.e. .IPC for IPC-356 netlist)

### \*ARCHIVE DATABASE FILE: Given to Customer (units in mils unless stated) (Compressed = <PCB PN>\_ARC.ZIP)

*<PCB PN>_REV-#.Brd	Allegro binary database
*<PCA PN>_REV-#_ASY.PDF	Assembly Drawings in .PDF format
*<PCB PN>_REV-#_FAB.PDF	Fabrication Drill Drawing in PDF format

#### Optional or Requested Files

*<SCH PN>_S#.EXT	Schematic Drawing Must contain iteration #.
Stackup.EXT	PCB Layer Stack-up provided by fabricator.
<PCB PN>_REV-#_BOM.XLSX	Parts list generated from OrCAD or Allegro.

### FABRICATION FILE: Given to Fabricator (units are in mils unless stated) (Compressed = <PCB PN>\_FAB.ZIP) (Files in 274X format 2:5, RELATIVE, LEADING SUPPRESSED)

<PCB PN>_REV-#.IPC	CAD Generated IPC-356 netlist
<PCB PN>_REV-#_L#.ART	Metal Layer Gerber Data (# Indicates current metal layer)
<PCB PN>_REV-#_MB.ART	Soldermask-Bottom Gerber data
<PCB PN>_REV-#_MT.ART	Soldermask-Top Gerber data
<PCB PN>_REV-#_BD.ART	Board Outline Gerber data (Board Outline ONLY. No other references on this layer)
<PCB PN>_REV-#_ST.ART	Silkscreen-Top Gerber data
<PCB PN>_REV-#_SB.ART	Silkscreen-Bottom Gerber data
<PCB PN>_REV-#_ART_PARAM.TXT	Artwork Parameters report
<PCB PN>_REV-#.DRL	NC Drill size report
<PCB PN>_REV-#NCDRILL.LOG	NC Drill Log file
<PCB PN>_REV-#_FAB.PDF	Fabrication Drill Drawing in .PDF format

#### Optional or Requested Files

<PCB PN>_REV-#_ODB	CAD Generated ODB++
Stackup.EXT	PCB Layer Stack-up provided by fabricator.
<PCB PN>_REV-#_ICT.TXT	ICT file testpoints report (testprep)

### ASSEMBLY FILE: Given to Assembler (units are in mils unless stated) (Compressed = <PCA PN>\_ASY.ZIP) (Files in 274X format 2:5, RELATIVE, LEADING SUPPRESSED)

<PCB PN>_REV-#_PB.ART	Pastemask-Bottom Gerber data
<PCB PN>_REV-#_PT.ART	Pastemask-Top Gerber data
<PCB PN>_REV-#_XY<1 or mm>.XLSX	Pick & Place XY coordinate file (units in inches or millimeters)
<PCA PN>_REV-#_ASY.PDF	Assembly Drawings in .PDF format

#### Optional or Requested Files

<PCB PN>_REV-#_BOM.XLSX	Parts list generated from OrCAD or Allegro.
-------------------------	---

### MECHANICAL FILES (ONLY if req.): Given to Mechanical Engineer (units in inches) (Compressed = <PCB PN>\_MEC.ZIP)

<PCB PN>_REV-#_DXF	Database in AutoCAD (.DXF format)
--------------------	-----------------------------------