Customer Supplied Kit Requirements

Component Size/Type	Extra Qty	Extra %	Note
0402 & smaller	25	15%	Greater of Qty or %
0603 & larger	10	10%	Greater of Qty or %
SOT 6 legs or less	25	15%	Greater of Qty or %
All other parts	1		If more than 10 required, supply 2 extra

Printed Circuit Board Data:

Gerber Data in the following formats:

- 274X -- Preferred
- 274D – must include separate aperture list with all apertures defined.

Drill Data in the following formats (required if just gerbers supplied):

- Excellon 1
- Excellon 2

ODB++ format

- includes drill info (no separate drill files needed)
- generally .tgz extension

Printed Circuit Board Fabrication Instructions:

Fab drawing or separate instructions can be provided in the following formats:

- Adobe Acrobat (.pdf)
- Text file
- Gerber file
- HPGL
- AutoCAD (.dwg or .dxf)
- Graphic file (.jpg, .gif, .tif, etc.)
- Embedded in ODB++

The following information must be included either on the drawing or through separate instructions, if not the indicated defaults will be used:

- Quality Workmanship Standard
  - **Default:** IPC-6012 Class 2
- Material – RoHS unless otherwise noted.
  - **Default for RoHS:** FR4 Tg:170 Td:330
  - **Default for non-RoHS:** FR4 Tg:150 Td:310
- Starting Copper Weights (each layer)
  - **Default:** 1 oz. before processing
- Surface finish (plating) and thickness
  - **Defaults:** PWB Vendors choice of either:
    - ENIG (Immersion Gold) 3-7 micro inches thick. Preferred choice.
    - IAg (Immersion Silver) 6-21 micro inches thick.

Printed Circuit Board Fabrication Instructions Continued:

- Layer Count and Stack-up details
  - Per supplied data. If it cannot be determined, contact the buyer at Ninja Circuits
- Board outline dimensions and tolerances
  - **Default:** Outline per gerber file. Tolerances +/- 0.010”
- Hole or feature to board edge dimensions
  - **Default:** Location per gerber file. Tolerances +/- 0.005”
- Finished Board thickness with tolerance
  - **Default:** 0.062". Tolerance +/-10% of thickness.
- Hole sizes with tolerances
  - **Default:** Hole sizes per Excellon drill file. Tolerances +/-0.003”
- Silkscreen requirements
  - **Default:** includes white ink per gerber files. Vendor to clear as necessary.
- Soldermask requirements
  - **Default:** green semi-gloss or satin

- Special requirements must be provided (i.e. impedance, controlled dielectrics, plugged vias, special materials, etc.)

PCB FAB DATA REQUIREMENTS
TO BE SUPPLIED BY CUSTOMER

Printed Circuit Board Data:

Gerber Data in the following formats:

- 274X -- Preferred
- 274D – must include separate aperture list with all apertures defined.

  Drill Data in the following formats (required if just gerbers supplied):

  - Excellon 1
  - Excellon 2

ODB++ format

- includes drill info (no separate drill files needed)
- generally .tgz extension

Printed Circuit Board Fabrication Instructions:

Fab drawing or separate instructions can be provided in the following formats:

- Adobe Acrobat (.pdf)
- Text file
- Gerber file
- HPGL
- AutoCAD (.dwg or .dxf)
- Graphic file (.jpg, .gif, .tif, etc.)
- Embedded in ODB++

The following information must be included either on the drawing or through separate instructions, if not the indicated defaults will be used:

- Quality Workmanship Standard
    - **Default:** IPC-6012 Class 2
- Material – RoHS unless otherwise noted.
    - **Default for RoHS:** FR4 Tg:170 Td:330
    - **Default for non-RoHS:** FR4 Tg:150 Td:310
- Starting Copper Weights (each layer)
    - **Default:** 1 oz. before processing
- Surface finish (plating) and thickness
    - **Defaults:** PWB Vendors choice of either:
        - ENIG (Immersion Gold) 3-7 micro inches thick. Preferred choice.
        - IAg (Immersion Silver) 6-21 micro inches thick.
- Layer Count and Stack-up details
    - Per supplied data. If it cannot be determined, contact the buyer at Ninja Circuits
- Board outline dimensions and tolerances
    - **Default:** Outline per gerber file. Tolerances +/- 0.010”
- Hole or feature to board edge dimensions
    - **Default:** Location per gerber file. Tolerances +/- 0.005”
- Finished Board thickness with tolerance
    - **Default:** 0.062". Tolerance +/-10% of thickness.
- Hole sizes with tolerances
    - **Default:** Hole sizes per Excellon drill file. Tolerances +/-0.003”
- Silkscreen requirements
    - **Default:** includes white ink per gerber files. Vendor to clear as necessary.
- Soldermask requirements
    - **Default:** green semi-gloss or satin

- Special requirements must be provided (i.e. impedance, controlled dielectrics, plugged vias, special materials, etc.)

Technical Contact Information:

Customer must supply a technical contact (name, email, phone) should questions arise to prevent any delays in processing the order or build.