

MANUFACTURING INPUT

Files And Data Required

For Successful Printed Wiring Board Assembly

Overview

In order to assemble a printed wiring board, the people and machines at Custom Electronics Company (CEC) need the same information you or anyone else would need to sit down and put something together. There will need to be a list of parts and materials along with drawings and instructions about how those parts go together. These can include both human and machine readable data based on how your board is to be assembled, inspected, and in some cases tested.

CEC prides itself on providing intelligent manufacturing services. Upon your request, there are many default processes and steps we can take you through to derive some of the required information. We do not want to make assumptions about your design. Problems with input data usually result when a designer's own familiarity with a design clouds his or her judgment of what information is needed by the assembler, or when the designer is less familiar with the CAM portion of the CAD/CAM design software in use. At CEC we want you to have a consistent and dependable manufacturing experience. Regardless of our ability to provide you with engineering assistance going in, the manufacturing process must ultimately do exactly what you want it to do. Interfacing with a manufacturing floor can be considered similar to programming a machine. The more information and careful programming you provide, the faster and more accurate the results will be.

The items below are the most important elements of data which should be provided in order to complete a printed wiring board assembly. Clear and consistent information is critical in order to start your job. These items should be linked by a consistent version number and/or other unique identifying information consistent with your purchase order.

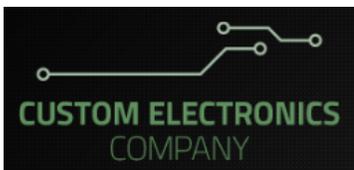
Bill Of Materials (BOM)

The BOM identifies everything someone should have available to them in order to build your assembly. This set of materials is what we will collect and verify as the "kit" for the job, either from materials supplied by you on a consignment basis or purchased by CEC for you on a turnkey basis. The kit will generally include the bare board, electronic components, and any hardware or other materials needed. Quantities of each item must be specified on the BOM along with location information, such as component reference designators that relate to an assembly drawing and/or markings on the board itself.

The items on the BOM should be specified by a part number and manufacturer or source information. However, this is not strictly required as long as there is enough information to uniquely identify each item in the kit. If we are purchasing any of the parts for you on a turnkey basis, then we will use the information you provide without substitution or further specificity. For example, if you say you want a 1K 0603 resistor, we will buy and/or check the kit for that item without regard to the unspecified tolerance, construction, voltage rating, etc. But if the BOM says you want a Panasonic ERJ-3GEYJ102V, then that is what we will buy and/or specifically look for in the kit. Delays may occur if we are unable to process the information that you provide.

You will not be required to uniquely specify or provide any common assembly materials such as solder or glue, unless you have unusual requirements. You will need to tell us if your assembly has any general materials requirements such as RoHS compliance, and we will ask you this specifically as part of your order placement process. It is generally a good idea to include notes on the BOM with this kind of information, along with any assumptions you are making, and confirming information such as reference designators you are not installing. This information might also appear on the assembly drawing.

The BOM drives all other processes and is the primary reference document if there is a conflict or inconsistency. The BOM is also the primary document used in accurately quoting a job. A printed document is acceptable, but a soft format such as an Excel spreadsheet is a good choice because we can read it, print it, make notes, and share results with you in that format if we need to help you prepare your job for production or discuss parts purchases when quoting and completing a turnkey job.



Assembly Drawing

The assembly drawing should include an outline of your board with part outlines and reference designators inside the part outlines. Be sure that the part outlines or other notes contain any necessary orientation information. In general, it is also helpful if the assembly drawing contains dimensional information, at least for the board outline and anywhere it may become important during assembly. Mechanical drawing techniques are usually needed to add hardware information or any other location specific materials or steps such as gluing or cutting. In some cases it may also be desirable to reference and include specific assembly instructions as an extension to an assembly drawing. Be sure that the assembly drawing and any attached documentation are complete enough that if you were to sit down with nothing other than your kit, BOM, and this drawing, you would be able to determine where the parts go and/or verify that they are placed correctly. As stated above, the assembly drawing might also be used to convey notes in addition to or instead of notes on the BOM, but to avoid delays all such information should be consistent.

Board Design Data

If we are purchasing bare boards for you, then we will need the Gerber files (generally RS-247-D format) and a fabrication drawing required for the board vendor to manufacture your boards. Not all board vendors can work with all file formats, but we can discuss other format options. These files are usually provided by an experienced designer familiar with the process of making printed circuit boards.

It is often helpful if we purchase the bare board, because that will give us an opportunity to help manage quality, cost, schedule, and perhaps also suggest or add useful features with your approval. For example, boards requiring surface mount parts to be mounted on automated equipment typically require certain visual fiducial features in the copper layers. Boards being assembled with any equipment will generally require certain edge to feature dimensional limits. Another common problem relates to surface finish and solder mask oversize in surface mount boards. There may be other considerations that directly affect the timeliness and quality of your completed PCB's, so we can review these items with you and/or try to support their inclusion when your boards are fabricated.

If your board will require surface mount assembly, at least beyond a few parts, then we will need to create tooling called a "solder paste stencil" for your board, which you will see on your quote. One stencil will usually be required for each board surface that contains surface mount components. The data required to make this stencil is the Gerber file for the "paste mask" or similarly defined layer of board design CAM output. In almost all cases we will need to make this stencil for you because our machines are very specific about stencil size and features required.

Pick And Place Location Information

If you have surface mount components on your board to be assembled using our automated pick and place equipment, then we will need numerical information identifying each part location. In general, this will also be a file generated by your board designer. We have become adept at translating varied formats of this information to what we actually use, but the more your format is simple and complete the faster it can be applied with less potential for problems. This information will be boiled down to a CSV file with the following basic form.

<designator>,<value or part number>,<X location>,<Y location>,<rotation>,<package type>

The X and Y locations must be a finely specified distance from a consistent origin location. These dimensions are usually given in mil units (1000th of an inch). The origin is typically the lower left corner of the board, but any consistent location is acceptable. The rotation is dependent on the machine we use, but consistency is important. For present machines 0 degrees is oriented toward the left and rotation is clockwise from there. For example, for a 14 pin SOIC, a 0 degree orientation would have pin 1 on the lower left and pin 7 on the lower right.

We will make sure to adjust this information after referencing your BOM and assembly drawing. There is no cost effective way to manually enter this information on most machines. Consistent information exported from your CAD/CAM software will usually be sufficient if the other documents are in good condition.