Accutronics Inc.

Data and Files Requested	Build		Quote	
	Turnkey	**Consignment**	**Turnkey**	**Consignment**
Bill of Material (BOM)	*Required*	*Required*	*Required*	*Required*
Gerber Files	*Required*	*Required*	*Required*	Important
ECAD Layout Design file(s) in ASCII format	*Required*	*Required*	Important	Important
PCB Fabrication Drawing	*Required*	Important	*Required*	Important
Assembly Drawing	Important	Important	Important	Important
Net List	Important	Optional	Optional	Optional
Solder Sample	Important	Important	Important	Important
Schematics	Optional	Optional	Optional	Optional
Programmable Devices Native Data files	Optional	Optional	Optional	Optional
Conformal Coating	Optional	Optional	Optional	Optional

Required	Accutronics can quote and build most assemblies without "Required" files but it requires significant manual programming and / or file manipulation. Without the required file(s) any estimate or quote can change significantly.
Important	Accutronics can quote and build without "Important" files. Without these file(s) the estimate or quote can change based on additional information identified.
Optional	Accutronics can quote and build without "Optional" files. "Optional" file(s) provide assembly specific requirements, Conformal Coating, Rework, Programmable Devices, Testing (Flying Probe), Debug … etc..

Bill of Material (BOM)	The BOM for an electronic assembly typically contains Assembly Part Number and Revision with a series of line items. Each line item is described using a Customer Part number, Description, Approved Vendor / Manufacturer List, Quantity, and Reference Designator(s). This information helps Accutronics procure material (Turnkey), inspect components, Pick and Place Programming, develop process documentation, and Testing (Flying Probe).
Gerber Files	Gerber files are representative images of each layer of the board to be fabricated. Standard Gerber format is RS-274X or extended. The Gerber files should include: • Paste Solder Mask Silk Screen (top & bottom) • All Layers Specialty layers Drill Information • Border Route & Tab V-Score
ECAD Layout Design file(s) in ASCII format	ECAD Layout Design file(s) in ASCII format are used for exchanging information to us about the PCA design. There is an IPC-2581 standard but generally each ECAD design package has the ability to output (export) the design files in ASCII format. We use these file(s) to develop our Pick and Place Software, Solder Paste Height inspection, AOI, and Flying Probe testing programs as well as creating our documentation. Accutronics can utilize a wide variety ASCII data formats from PADS, GENCAD, CADENCE, PROTEL, ODB++, as well as other standard outputs. See the Aegis "Cad Extract Import Guide" attached to this email.
PCB Fabrication Drawing	The Fabrication drawing contains information on board material, construction, and geometry. This information is used to help determine if SMT Fixtures are required, Process requirements, Inspection, and Turnkey procurement to of raw fabricated PCB's.
Assembly Drawing	The Assembly Drawing helps with the generation of process documentation, identifies the customers' requirements & standards, labeling, and other information to be communicated.
Net List	The Netlist file describes of the connectivity of the electronic circuitry. Standard format is the IPC-D-356 NET LIST. This is not required but is important especially when manufacturing or testing raw fabricated boards
Solder Sample	Solder Sample is used for thermal profiling and a physical representation of the PCB
Schematics	A graphical representation of an electrical circuit