

BOSCO Printed Circuits (Pty) Ltd

Gerber X2 Format

For us to supply you a quote, we require from you:

1. Your Gerber data files: **Gerber X2 preferred**



Gerber Format X2 FAQ

The Gerber Format Specification

The two Gerber Format PDF Files listed here above can be downloaded directly from Ucamco's Website "Downloads" page on this Link:

<https://www.ucamco.com/en/downloads>

Gerber X2 adds intelligence to the Gerber format by its use of attributes. When loading a new job, attributes allows the CAM software to put the layers directly in the correct position in the stack-up. Other attributes identify the via pads, the SMD pads and so on. **Gerber X2 files will make your data input faster and more secure.** It shows you are following the latest technology. Being first with X2 gives you an advantage over competitors whose CAM software does not yet support X2 – it is not often one can differentiate by the input format.

Furthermore, we all want to improve data transfer from design to fabrication in a safe and practical way.

2. Mechanical diagrams with special instructions where applicable
3. The PCB specifications and special instructions where applicable
4. Please complete and sign our Customer PCB Specification Sheet available on our Download web page.

Email your required documents as requested above to:

sales@bosco.co.za

