



Omni Circuit Boards PCB Design Checklist

All Gerber files present:		Design Checks:		Documentation:	
Top Copper Layer	<input type="checkbox"/>	Metal to Edges	<input type="checkbox"/>	Readme File	<input type="checkbox"/>
Bottom Copper Layer	<input type="checkbox"/>	Overlapping Drills	<input type="checkbox"/>	Part Number and Revision	<input type="checkbox"/>
Aperture Information*	<input type="checkbox"/>	Space and Cu Trace widths	<input type="checkbox"/>	Multi-Layers identified*	<input type="checkbox"/>
Top Overlay	<input type="checkbox"/>	Annular Ring Sizing*	<input type="checkbox"/>	Technical Contact Info	<input type="checkbox"/>
Bottom Overlay	<input type="checkbox"/>	Drill Sizes	<input type="checkbox"/>		
Top Soldermask	<input type="checkbox"/>	Ident on Pads	<input type="checkbox"/>		
Bottom Soldermask	<input type="checkbox"/>	Undersized Ident Text	<input type="checkbox"/>		
Mechanical (Outline)*	<input type="checkbox"/>				
NC Drill*	<input type="checkbox"/>				
Drill Tool List*	<input type="checkbox"/>				