# The Apple II, revision 0 CAD package

Do with these files as you please. As user of these files, you are entirely responsible for errors or problems that arise from usage of these files. This includes mistakes, omissions, and incompatibilities with manufacturing process. It is your responsibility to properly manage any trademark issues or copyright issues. In short, anything that may cause technical, manufacturing or legal problems is up to you to resolve.

This ZIP file contains the following files.

Nov 30 2013 appleiiboardV3a36.osm - OSMOND PCB CAD file

Jan 11 2014 applelogo-II-black-v2.3.ai – Adobe Illustrator

Apr 7 2015 SILK1.GBR – Gerber file –front silk screen

Apr 7 2015 MASK2.GBR – Gerber file – back solder mask

Apr 7 2015 MASK1.GBR – Gerber file – front solder mask

Apr 7 2015 LAYER2.GBR – Gerber file – back copper layer

Apr 7 2015 LAYER1.GBR – Gerber file – front copper layer

Apr 7 2015 DRILL.TXT – Gerber file – drill file

Apr 7 2015 AUX1.GBR – Gerber file – board outline

Sep 24 07:04 Read Me First.docx – This document

OSMOND PCB: This is the CAD program used to create the Apple II, revision zero gerber files. This program can be used to make changes to the design.

Gerber files: Zip these together and send them to the PCB manufacturer of your choice in order to get quotes and have your boards made. Be sure to include the drill.txt file, so the manufacturer knows where to put the holes into the board.

Adobe Illustrator files: Can be used to make Apple Computer logo decals, which is NOT part of Gerber silk screen file and also decals for memory jumper blocks.