

# Printed Circuit Board (PCB) Layout Application Note

## Summary

This application note covers some valuable layout concepts for doing printed circuit board designs using Microwave Office. It demonstrates how to use positive and negative layers to ease drawing ground flood regions. By utilizing some simple setup procedures it is possible to have all lines draw with a user-specified spacing to the ground flood automatically. This technique greatly simplifies drawing the ground flood when the initial layout for a circuit is complete. This application note also discusses various ways that Gerber files can be created in Microwave Office. One way is to merge the positive and negative layers to make one Gerber file. The other way is to keep the positive and negative layers separate if using a paint-scratch-paint processing technique.

## Downloads

[PCB\\_Layout\\_70.zip](#)

## Related Articles

[Printed Circuit Board \(PCB\) Layout Application Note 2 - Multi-Level Layout](#)