



DownStream Product Training – CAM350 & BluePrint-PCB

Whatever your reason for wanting to attend training courses, it is always a good investment. As software packages become more robust and powerful with each release, it is wise to brush up on your skills and learn the new features and functionalities. Of course if you are new to the software, training is a great way to familiarize yourself with the new product and company.

CAM350: April 20th, Marlborough MA

The CAM350 course is focused on Design Verification and PCB Post Processing. This is a great opportunity to learn about Release 10.0 and to get more out of your software. This course topics include: Getting familiar with the new Graphical User Interface; Output of data from various CAD systems - using the direct interface; Gerber identification and import; Streams Rule Checking; Netlist Comparison; Design Rule Checking; Design for Fabrication; Assembling the completed design package for Fabrication; and an Introduction to Universal Release Package Navigator.

Course Outline

Data Preparation

- Proper output of Gerber and NC files for fabrication
- RS-274d vs. RS-274x
- CAD system output hints

CAM350 CAM Editor

- Input options
- DXF to Gerber and Gerber to DXF
- Comparing layers for differences
- Adding Teardrops
- Clearing Silkscreen from lands

Netlist Compare for the Designer

- IPC-d-356 creation in the CAD system
- Using Streams RC for Netlist analysis
- Comparing an IPC-d-356 Netlist to a Gerber extracted Netlist in CAM350
- Verification of results in CAM350

DRC/DFM in CAM350

- Using Streams RC for Design Analysis
- Proper setup of DRC to find potential Fabrication problems
- Setup and save a generic DRC to run against all jobs
- Parameters for running a successful DFF Analysis

The complete data package for Fabrication

- What is in the complete data package, what not to send
- Format of Gerber files
- What to include on the fabrication print

- What to include in a “readme” file

Using the Universal Release Package Navigator

BluePrint-PCB: April 21st, Marlborough MA

This training course will focus on getting to learn the basics of BluePrint-PCB. The attendees will learn how to use the powerful features within the tool to generate documents and documents sets. Emphasis will be placed on using the Gallery and Document Templates to automate the process.

Course Topics and Outline

- User Interface
- Documentation Hierarchy
- Generating a release package
- Create Fabrication drawings
 - Defining Title Blocks
 - Placing Drill Charts and Patterns
 - Creating a layer Stackup detail
 - Creating and Linking Details
 - Auto Dimensioning
 - Adding Notes
- Create an Assembly Drawing
 - Creating top and bottom side views
 - Creating an exploded view
 - Import and modify DXF drawings
 - Hyperlinks
- Creating a parts list
- Creating Engineering Change Orders
- Other topics that may be covered (decided by attendees)
 - Variant Definition and Display
 - Process Steps Definition and display
 - Parts list management
 - 2d drafting tools
 - Interaction with CAM350
 - Array Panel Documentation

Registration and Details

These courses are scheduled from 9:00AM – 5:00PM each day. Course materials, refreshments, and lunch will be provided as part of the course and administration fees. For more details, and to register, please contact DownStream directly:

- Call 800-535-3226 x623
- Email customercare@downstreamtech.com