http://wiki.dxarts.washington.edu/groups/general/wiki/54cce/LPKF_Workflow.html

LPKF Workflow

1. Install the CAM processor for LPKF

copy this file to the cam directory of Eagle application folder. (ex: Applications/EAGLE-6.4.0/cam)

2. In EagleCAD

1. You should already have a complete schematic and board layout complete in Eagle.
2. Print your layout to check the layout and component fit.
    1. in the board view, select File->Print, and make sure you print at scale of 1
3. Export from Eagle CAM processor
    1. in your board view, click CAM icon or File->CAM Processor. CAM popup appears
    2. In the menu bar, select File->Open->Job:
        1. gerberForLPKF.cam. process job.
        2. excellon.cam. process job.
4. This results in a bunch of files. .CMP, .DRD, .STS, .GPI, .CRM, .VSC, .STC, .SOL, .PLC, .CMP, .BOR. These are your "Gerber Files". Some define images and some of them are drill hole information (excellon).
    1. If you want to view them, zip them to one .zip archive, and upload them here: http://circuitpeople.com/. There are other gerber viewers you can download or build on your own system, but I haven't found an easy solution on OS X. Let me know if you do.
    2. What is a gerber file: http://circuitpeople.com/Blog/WhatIsAGerberFile.aspx
5. Thumb drive to transfer files to E:/Projects/YourFolder on LPKF machine.

3. In CircuitCAM

1. Files->New->eagle.cat
2. File->Import:
    1. find your local folder and select the following four files to import.
    2. In "Rapid Import" popup, specify Layer/Template for each file. See laminated reference sheet.
        1. .BOR -> BOARD OUTLINE

        2. .CMP -> TOP LAYER
        3. .SOL -> BOTTOM LAYER
        4. .DRD -> DRILLS PLATED
    3. Import.
3. Save your file as a CircuitCAM project (.cam).
4. Edit->Insulate.
    1. Under "Job", select "Bottom (Solder Side)"
    2. Select Advanced tab. Delete the text in Rubout-1 and Rubout-2 fields.
    3. click Run.
    4. Repeat a-c for "Top (Component Side)"
5. Cut out board outline (with breakout tabs).
    1. click Edit->Contour Routing. Make sure you are cutting "Outside".
    2. select "Automatic" under Breakout. This will create breakout tabs to secure the board.
    3. Run.
    4. you should see a deep yellow line around the outside of the board contour. It should have a couple of gaps where the breakout tabs will be left.
6. Save your file again so you don't have to repeat this all.
7. Export for BoardMaster: File->Export->LPKF->LPKF Circuit Board Plotter.