

CAM350 6.0

2000/5/19

CHASSIS

Drawing , , , ,
Flash .

Aperture

Aperture Photo Plotter Draw Flash (,)
Photo Plotter Aperture Photo Plotter Data plotting
Aperture . Aperture D-code
D10, D11 ... D code 가 .

CAD



Photo Plotter
CAM350

D10 ROUND 55mil
D11 ROUND 8mil
D12 Square

Aperture



FLASH D10
FLASH D12
FLASH D13

Photo Plotter Data



Aperture
D-Code
Aperture

() . SPACE BAR
가

Data ,
() .ESC

Hotkeys()

View

+ (Zoom In)
- (Zoom Out)

PAGE UP

PAGE DOWN

HOME (All)

INSERT (Pan In)

C (Zoom Close)

Edit

F Fill
P
R Redraw
T Editing Hotkeys
W Window
X

Main

D D-Code
O (Any, Diag, Orth)
S Snap-To-Grid
V Grid On/Off
Y Layer Table
Z

Hot key

A Aperture Table
G
K Kill Layer
L
Q Query All
U Undo
CTRL-U Redo

Edit Edit Edit . Edit 가

. Edit

가 .

B Dcode

C Window
M Add/Part Mirror (Bottom)
T Add/Part
W

Menu Bar

Menu , ALT
File ALT-F

CAM350 가

Gerber CAM Editor , Aperture GAP Editor,
(foot print) Part Editor

CAM Editor

Title Bar

CAM350 Family

Menu Bar

CAM350 가

Tool Bar

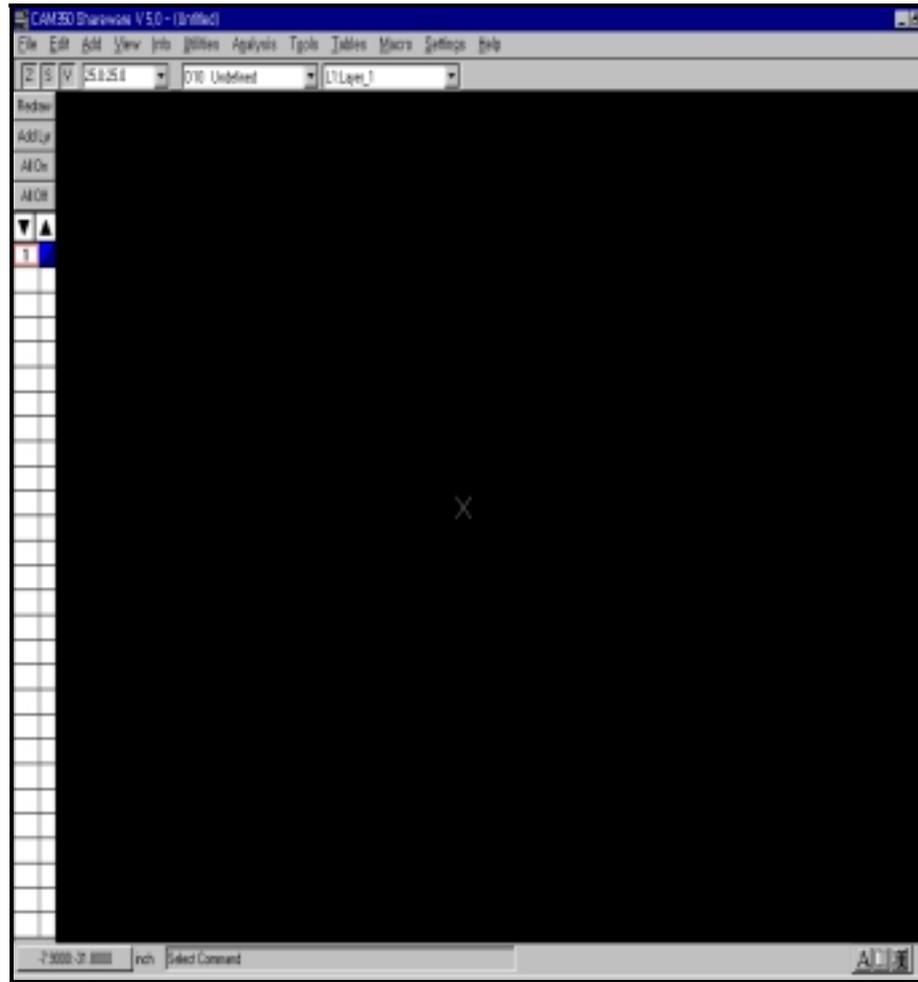
D-code, Layer

Edit

Tool

Bar가

Title Bar -----
 Menu Bar -----
 Tool Bar -----



Layer
 Control Bar

Status Bar ---

(CAM350)

Z

가 . Z .

S

On/Off

. S .

V

On/Off

. V .

GRID

25:25

Grid

ACTIVE

Dcode

D13 Round 20:20

Active Aperture

Layer

L1:DEMOSOLD.GBR

Active Layer

Status Bar

Command Prompt

가 (0, 0)

Layer Bar Layer Layer
Redraw, Layer

On/Off

All Off Active Layer 가

Redraw R Layer On/Off 가

가 Color Layer Color

Color 가 Layer Active Layer



Draw

Flash

(Workspace)

CAM350 가 ,

Origin Marker CAM350 (Center) X

가

Cursor Shape X

가

Cycle Picking 가 Copy, Move Cycle Pick

가

Edit

Edit

Move

Menu가

Draw Gerber Draw

Flash Gerber Flash

Filter Window

, Dcode

Dcode filter 70, 11:20, -15, 22

70

, 11

20

, 15

, 22

. Polygon

Text

Prev

SelectAll Dcode

CTRL-

W

CTRL-

. C

Window

가

가

ESC

File

New

File/New

가

가

Ok

Open

CAM350

(.PCB)

CAM350

File/Open

가

File name

가

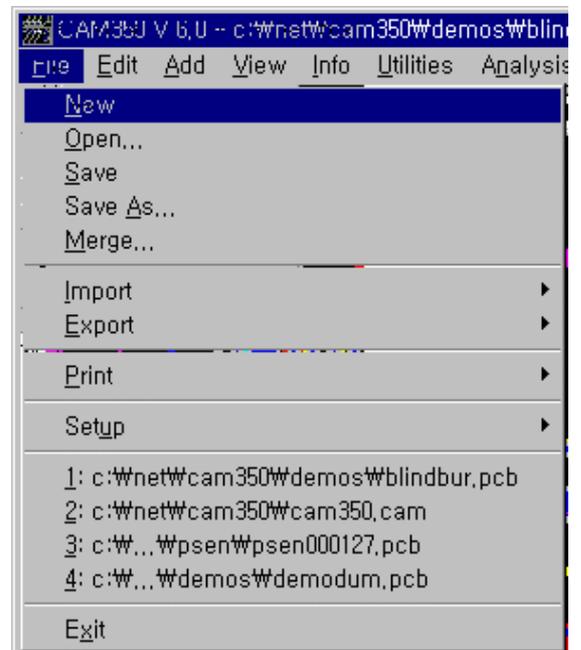
PCB

Ok

Example12350

File name

Ex



가 .PCB가

Files of type *.*

Save

CAM350

.PCB가

Gerber

Plot

PCB

Save As

File/Save as

가

Ok

Merge

.pcb

PCB

(CAM350)

Merge가 가

File/Merge

.pcb

PCB

가

.pcb

Import

가

Format

가

▪AutolImport Gerber

Import\Gerber

가 Gerber

가

Gerber

Plot

AutoImport

Aperture

Gerber

Import

▪Gerber Data

CAM350

Gerber

(Format)

Import

CAM350 가

RS-274 Laser Photoplotter

, RS-274X, Fire 9XXX(Symbolic Sciences)

Import/Gerber Data 가

Aperture file Aperture 가 Formats ECAM
Gap , Others
CAD Aperture

File

Aperture File Open Aperture file

Data Format 가

Gerber Gerber RS-274-X

Type Digits, Zero suppression 1page
Ok

CAD Data Format Inch
mm 가 가

Layer # 1 Open 가 Gerber
Layer Type ECAM

Layer
Component Side Signal Top Top
Layer Type

Gerber Import Layer Type Ok
Gerber 가

▪Drill Data NC/Drill

Utilities/Drill/Tables

Import|Drill Data Drill Data 가

Layer # 1 Drill

Filename

P Plated

Digits, Zero suppression, Type, Units

Layer

Legend Drill Report가

Legend

Layer 가 가 Yes

Layer , No 가

가

Drill Target

•Mill Data

Import\Mill Data Mill Data 가

.ROU

Layer # 1 Mill

Filename

Digits, Zero suppression, Type, Units

Mill 가

Layer 가 가 Yes

Layer ,No 가

가

•HPGL HPGL(Hewlett-Packard pen plotter) CAM350

가 HPGL pen #1 Dcode 10

Import\HPGL HPGL 가

Select Layer HPGL File Ok

Layer File name

Import

Gerber HPGL

가 Yes Layer HPGL

가

•IPC-D350 IPC-D350 Format CAM350 Import가 가

•ECAM dsn ECAM ECAM Design file Import

•Netlist CAD Format Netlist

X:Y Reference\Designator, Pin number

Netlist Import

Import\netlist Netlist Format 가

CAM350 Netlist Import Board Net(X, Y) Refdes, Pin

Number

CAD Netlist Import

•Aperture Aperture Import\Gerber Aperture가

Aperture 가 가 .

Import/Aperture . Aperture 가 .

Aperture .

Gerber Aperture 가

Gerber 가 . Gerber 가 Aperture

Aperture .

▪**Edit Aper Conv** CAM350 Aperture

Aperture , CAD

가 Aper Rule Files(.ARL) Aperture

▪**CAD Data** CAD PCB Import .

CAM350 Family CAM350 가 Option

Export

Import

▪**Gerber Data** Photo Plotter .

▪**Composites** Positive + Negative Tables/Composites

▪**Drill Data** Gerber Drill Data

NC Drill format Export .

▪**Mill Data** Mill 가 Utilities/Mill

▪**IPC-D350** IPC-D350 Format .

▪**Netlist** Netlist .

▪**Aperture Table** Aperture .

▪**CAD Data** Gerber CAD PCB .

Print

▪**Setup Printer** Windows

CAM350 Windows .

▪**Print Display** Setup Printer

1 View Window, All, In, Out .

▪Print

Print(Print Display) . Plot Scale 가 .
Plot Scale Ok .
Print(Print) . Print 가 . ,
Center Layer 가 . Plot
가 . Plot
All Layer가
Portrait Landscape .
Header
Setup Printer .
가
Port가
Layer Tile
Layer Tile
가
Layer Layer
Layer .
Rotate 90 .
Center 가 .
Layer Align 가 .
Fit to Page . Scale
가
Draws, Flashes .
Output All Data In Black
Block .
Separate Sheets Layer Plot
Filled Features Line, Polygon, Flash Fill ,
PenPlotter Check
Plot 가 .

Setup

▪ Preferences

▪ Paths

▪ Files Extensions

▪ Colors

가 Color

▪ FilmBox PhotoPlotter

Origin

▪ PhotoPlotter PhotoPlotter

▪ Router Router

▪ Drill Machine Drill

▪ Save Default Setup

▪ Access Code Serial key Access Code Product

Exit

Exit

Edit

Undo

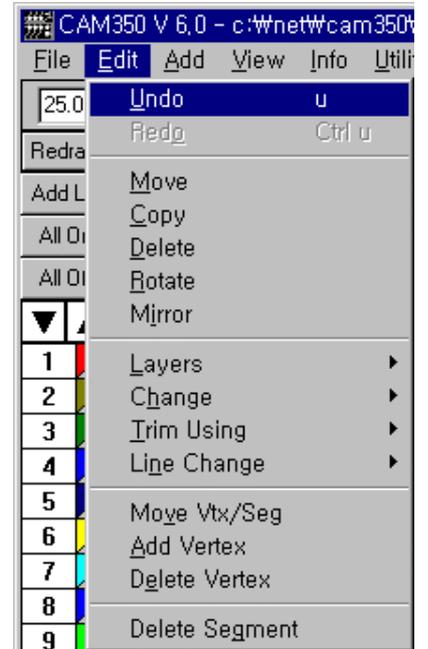
Redo

Move

Copy

Copy

Delete



Rotate

. 0.01 .

Mirror

Mirror Axis가 Tool Bar

가 , .

Layers

Layer

▪Add Layer Layer 가 .

▪Align . Gerber 가

▪Reorder Layer .

▪Scale (Target) Layer 가 Layer .

▪Remove Layer 가 가 가

Change

▪Refdes .

▪Device .

▪Netname Netname .

▪Padstack Padstack Padstack ID .

▪Dcode Dcode Dcode .

Edit Change/Dcode .

Flash, Trace Change Dcode 가

Dcode Ok .

Dcode 가 .

▪Text .

Change/Text .

Text .

Change Text 가 .

Ok .

Fit Text Mirror .

Text가

가

Fit Text Mirror

▪Explode

Aperture Part

Edit/Explode

Aperture

Group

Explode가

Draws/Flash

Aperture

Negative

Explode

Padstack

▪Sectorize

Photoplotter

Thru Circle

Circle

Arc

Thru Circle 가

가

Change/Sectorize

Circle, Arc

Sectorize

가

Angle

10

Ok

▪Origin

(0,0)

Space Origin

Layer (0, 0)

Grid Origin

point

(0, 0)

Move Vtx/Seg

Edit/Move Vtx/Seg

가

Add Vertex

가

Delete Vertex

Delete Segment

Delete
Segment

Delete

Add

Flash

Flash 가 . Tool Bar Active Dcode Gerber

Flash가 가 .

Line

가 . Tool Bar Active Dcode

Polygon

Polygon . CAM350 Pour가 가 Vector, Raster 가

. 가 Copper Pour

Fill . Polygon Copper . Copper

Layer Layer Trace, Pad Text Pour .

Copper Pour(Polygon) ()

. Copper Fill Area Fill

가 . Polygon Net

Polygon Void 가 . Polygon 가 Tool Bar

Netname Netname . Polygon Netname Netname

Clearance . Polygon Void, Add, Move,

Delete Vertex

Polygon Net name . Gerber

Import Utilities|Netlist Extract Trace Net name

Add\Polygon . Polygon Settings 가 .

가

가

Border Type

Draw Border 가 Polygon , Select

Border Polygon Line

Area Fill 가

가

Fill Mode

Outline Only draws Fill

Raster Fill Convert

Vector Fill Dcode

Vector Fill Dcode Solid, Hatch

Outline/Fill Dcode

Polygon Clearance

Active layer Polygon 가

Layer Target Layer

Hatched Dcode Fill Type Open .PAT

Import 가 Edit Pattern 가 가

Hatched Pattern 3가 Line

Open, New, Save, Save As Pattern

가

Polygon Ok

Text

Font Setting

Padstack

Padstack Tables

ID PadStack 가

Padstack ID가

Rectangle

가 . Dcode .

Wire

CAD

DRC

Wire

가

가

Netname

가가

Via

Flash

가

Layer

Signal Layer

Via

Part

Part Editor

Utilities/Buired Part

가

View

Window

All

Redraw

In

Out

Pan

Film Box

Setup/Preference Film Box
View/Film Box On/Off가 가
View/Film Box

Composite
Tables/Composite On/Off

Tool Bar
Tool Bar On/Off

Status Bar
Status Bar On/Off

Layer Bar
Layer Bar On/Off

Panoramic
Windows가
Window

Info

Query
All

Find
Tool Bar Dcode 가 Dcode
Dcode가 Zoom Center

Measure
가

Point - Point
가

Object-Object

Report

Status

Utilities

Gap Editor

CAM Editor

Aperture

Gap Editor

Gap Editor

Draws Flash

Line

Flash

Line

Round Line

Data가

Automatic Window

Draw type Pad

Flash type

Maximum Draw Size.

Maximum Size

Draw

Flash convert

가 height() width()

Draw

Selection Limits. Min Width

convert

Min Width

convert

가

1mil

Tolerance

Draw

2 가

Automatic Conversion

가 (round, oblong,

rectangle)

Draw가 가

Dcode Filter

Flash

User Check Rejected

가 Draw

convert

Draw가

size

User Check All

View All PCB

Zoom Out

Maintain View

Graphics

Exclude Existing Flashes Draw Flash convert

Ignore Single Stroked Lines

Draw

Minimum Width. Dcode Filter Dcode

Interactive Draw type Pad Dcode

Netlist Extract

Netlist Gerber HPGL

가 Export Netlist가 , CAM350

가 Gerber Netlist 가

Netlist Electrical DRC

Netlist Extract Layer type Top, Bottom, Internal, Plane
Tables\Layer Layer Type

Center Line [Fast] Netlist 가
가 , PAD Center point

True Shape [Slow] Netlist 가
True Shape

Part AutoCAD PCB 가

True Shape

Utilities/Extract

Center Line True Shape Ok

Pads -- Padstack

Flash Layer Flash Padstack

Gerber Import Pad

Padstack Padstack

Drill

Layer가 Top, Internal, Plane, Bottom Pad Stack

. SMT Pad Padstack .

Clear Silkscreen

Silkscreen

Pad

Clear Silkscreen

가

. Silkscreen

Solder Mask

Layer

. Layer Type

Dcode Filter

Clear

Dcode

Clear

Clear Silkscreen

Flash

Pad

가

Draw to Flash

Via

Clear

Dcode Filter

Dcode

Ok

Silkscreen

Clear

Pad Removal

Trace

Pad

. Layer

Pad

. Pad Removal

Layer

Pad

가

가

Pad To Padstack

Isolated Pads

. Window

Pad

Top Layer

Pad

가

. Ok,

No

No

Layer

Bottom

Bottom Layer

Pad

가

Ok

Redundant Pads

Dcode

Pad

Dcode

Ok

Teardrop

. Trace

Pad

Over/Under Size

		Layer	Layer
	. Solder Mask layer, Paste Mask layer,	Copper	Clearance
가	.		

Drill

NC/Drill

▪**Table** Tool Number Dcode Hole Size .

Dcode .

Custom aperture 가 .

Tool 가 가 가 .

Export Tool Number .

File\Import\Drill Drill Import .

Drill\Table 가 .

Tool # 가 .

Export Export Order

Load Drill .

Hole size Legend Dcode typing

labeled "..."

Dcode .

Tool Add Add Tool Drill

Tool 가 Drill Table

가 Tool Number, Hole Size, Legend Dcode

Drill 가

. Drill table Add Add Tool .

Tool Delete Tool .

Export Order Reorder Exported Drill Sequence 가

. Tool Number .

Drill Table Save .

Ok Tool .

▪**Create** NC Format Drill Export .

Drill\Create 가 Drill\Table

가 .

Padstack Tool . Padstack Top Bottom

Drill 가 가 .
 Active Tool Tool Table . Padstack Tool
 Padstack Hole size Tool
 Padstack Tool 2 가 .
 Plated Drill Layer Plated Layer Unplated
 Layer . Ok Layer가 가 Drill 가

▪Sort Drill Tool Path . Sort Panelization

▪Gerber To Drill Single Gerber Layer Drill .
 Drill\Table Hole size Legend Dcode .
 Drill\Gerber To Drill .
 Layer . Gerber Flash가
 Drill Layer .
 Drill Tool Legend Dcode .
 Plated Unplated hole .
 Ok Layer 가
 Layer Pad Dcode, Legend size가
 가 .

Mill

Net Check

DRC

DRC(Design Rule Check) . Trace
 Trace Trace 가 , 1
 Layer 가 가 가 .
 Round Flash 가 Drill Data Mask Data 가 ,
 Drill Point, Pad Pad, Drill Pad Rule . DRC
 가 Rule Check가 가 .

▪Clearance Area Layer
 Board Layer

Track-Track Trace

Track-Pad Trace Pad

Pad-Pad Pad Pad

Outline Board Outline 가

Min Trk Width Trace

Min Pad 가 Pad

Redundant Pad

▪**Layer To Test** Layer

Passes Area 가 가

Pass

Pass

▪**Annular Ring Area**

Pad-Mask Test Pad Mask

Drill-Mask

Drill-Pad

▪**Special Checks Area** 가

가

▪**Remove All Violations** DRC

DRC Histogram

DRC

DRC Histogram

DRC

Histogram

Utilities\DRC Histogram

가

Ok

가

Window

Panelization

Autofilm

Layer Gerber Layer

▪Autofilm Layer Gerber Layer

Panelization\Autofilm 가

Target Layer Film Layer

Layer가

X, Y

Border

Layer Center Image

Image Spacing X, Y

Space Between Image Spacing

가 Layer Offset

Ok

가 Panelization

가

Cancel Autofilm 가

File\Setup\Film Box View\Film Box 가

▪Panelize Panelize File\Setup\Preferences Origin

Origin Center가 Negative

Positive Plane

가 Layer Edit\Change\Explode\Panel

가 가 가 가

Utilities\ Panelize\Rotate Edit\Rotate

Gerber Layer Import

Layer Tag가 Tables\Layers Tag

Layer Edit\Layers\Reorder Layer

Number

Template Layer 가

Edit\Layers\Remove Panelize Layer

Board Layer

Border Layer가

Center Center Dcode Draw

Board Import 가 .
 Panelization\Panelize . Panelization 가 .
 Merge Panel . Template
 . Panel PCB .
 Merge Panel PCB Panelize
 . Generate Board/Vent Outline . PCB Board
 Layer Border Generate Board/Vent Outline
 . Dcode Layer .
 Border Layer Tag 가
 Layer Generate Board/Vent Outline .
 Border Image Step Off .
 Film Size
 File\Setup\Preferences Film .
 Ok .
 가 . Edit\Undo
 Panelization Unpanelization .
 Board 가 Unpanelization
 Venting .
 Venting
 Fill . Layer Vent
 Board
 Add\Polygon Raster, Vector Fill Venting 가 .
 Utilities\Venting 가 .
 Venting Fill . Add\Polygon Fill
 .
 Border Layer Border Layer Text Border
 Layer . Board Layer Board Layer Type 가
 .
 Vent Layer 1 Vent Layer Number
 .
 Vent Layer 2 Vent Layer Number
 .
 Galvanic Venting Border . Robber Bar
 . Galvanic Venting Fill Image가
 .
 Offset Data 가 Open 가 .

Polygon

Ok

Build Part

Gerber

Component

Part Library

Tables Menu

Aperture

Aperture Tables

Aperture

Tables\Apertures

가

Aper/Pen #

Cad

Aperture

Photo Plotter

Dcode 10

Go To Dcode

Dcode

*

Next Undefined

Dcode

Dcode

Flash

7가

Customer

가 Gap Editor

가

Size

Dcode가 Through-Hole

Smt Pad

Ok

Aperture

가

가

Padstacks

Gerber

Single Layer

Pad

Pad To Padstack

Through Hole

Padstack

가

Padstack

가

Tables\Padstacks

가

Padstack Id 1

1000

Padstack

가

가

Padstack

Padstack

Blind, Buried Via MCM

Layer가

Layer Name Layer Name . Aperture Id
 Aperture Padstack Aperture
 . Layer # . Aperture List 가 .
 Layer Aperture Ok .
 Add, Delete, Padstack Padstack List
 Remove가 가 . Padstack * .
 Ok .

Layers

Layer Draw, Flash Colors Active Layer
 .
 Tables\Layers 가 .
 # Layer Number , Name Layer Name
 . Layer Name 가 .
 Type Layer Type . Layer Type
 CAM350 가
 Layer Type . Draw
 Layer Draw Color
 .
 Flash Flash
 Layer . Flash Color .
 Status Layer 3가 . On
 Edit 가 , Off
 . Ref Edit
 .
 가 Act Active Layer . Active
 Layer On 가 .
 가 Top Layer
 Layer . All On/All Active Layer
 Layer On/Off .
 Add Layer Layer 가 .
 Layer Set Tables\Layer Sets
 On/Off Layer .
 Ok .
 Tables\Layer

