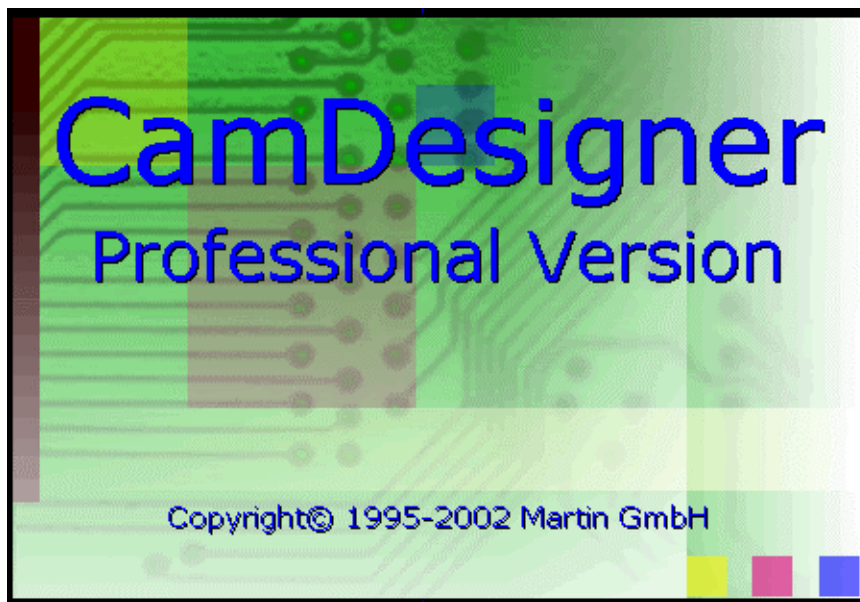


CamDesigner



Product Information

The main task of the MARTIN CamDesigner is the direct conversion of existing Gerber files into the Excellon type format needed for the Dot-Liner or Rework-Dispenser. The pad geometry is used to calculate the pad area from which the dispensing volume is generated. This process is largely automatic.

The volume dispensed in a single operation can be limited. This is important when the same size nozzle is used to dispense very small dots and larger volumes, too large for the nozzle. In such cases, the large volume can be automatically broken down into several smaller, manageable volumes for dispensing.

For long, narrow pads (eg QFP) a selection can be made to dispense one, two or three dots in a row. The overall volume per pad is naturally not affected by this choice.

Individual datum points can be inserted later into the completed Excellon or HPGL data to significantly simplify and accelerate Dot-Liner alignment.

Cam-Designer is a complete CAM program suitable for establishing complete board layouts.

Data, derived from different, widely used CAD programs, can be entered, changed and saved. Cam-Designer allows different production and document data to be displayed and checked. CamDesigner can also be used to convert several different data formats.

The output of Gerber and drill data in HPGL or DXF formats is very useful for preparing documentation. Additional support in this direction is provided by the opportunity to simultaneously open several files and to treat them as one layer. Within the same DIN Frame, one or more production files can be opened and placed.

Text and dimensioning functions can be added to the drawings, thus established, to obtain perfect documentation.

Supported Data Formats

- Gerber RS-274D
- Extended Gerber RS-274X
- Excellon
- Sieb & Meyer
- HPGL
- Pick & Place (data)

File Conversions

- Gerber to Gerber (eg mm to inch coordinates)
- Gerber data to Drill data (establishing of missing drill information)
- Drill data to Gerber data (establishing of solder masks)
- Excellon Drill data to Sieb & Meyer format and vice versa
- Output of Gerber data as HPGL or DXF data
- Output of Drill data as HPGL or DXF data

System Requirements

- Processor: Intel Pentium or higher
- RAM: 16 MB or more
- Graphic Card: 800 x 600 or higher
- Hard Disk: 20 MB available

Included is

- Program CD
- Hand Book
- Parallel Port Dongle (for Professional Version)