# Macaos Enterprise
Import Module
User's Guide

Introduction

The Import Module creates a production-ready PCB product from a set of Gerber and drill files as generated by an Electronic CAD system. The module parses and displays the contents of the Gerber and drill files, and guides the user through specifying additional data necessary for production.

Once a product has been completely specified, it may be published to the Macaos Enterprise product archive. The product is then available to be ordered through the Ordering module of Macaos Enterprise.

Module overview

The Import Module window consists of three main elements: a graphic display region (right), a layer overview (lower left) and a set of tabs for specifying/editing product data (upper left).
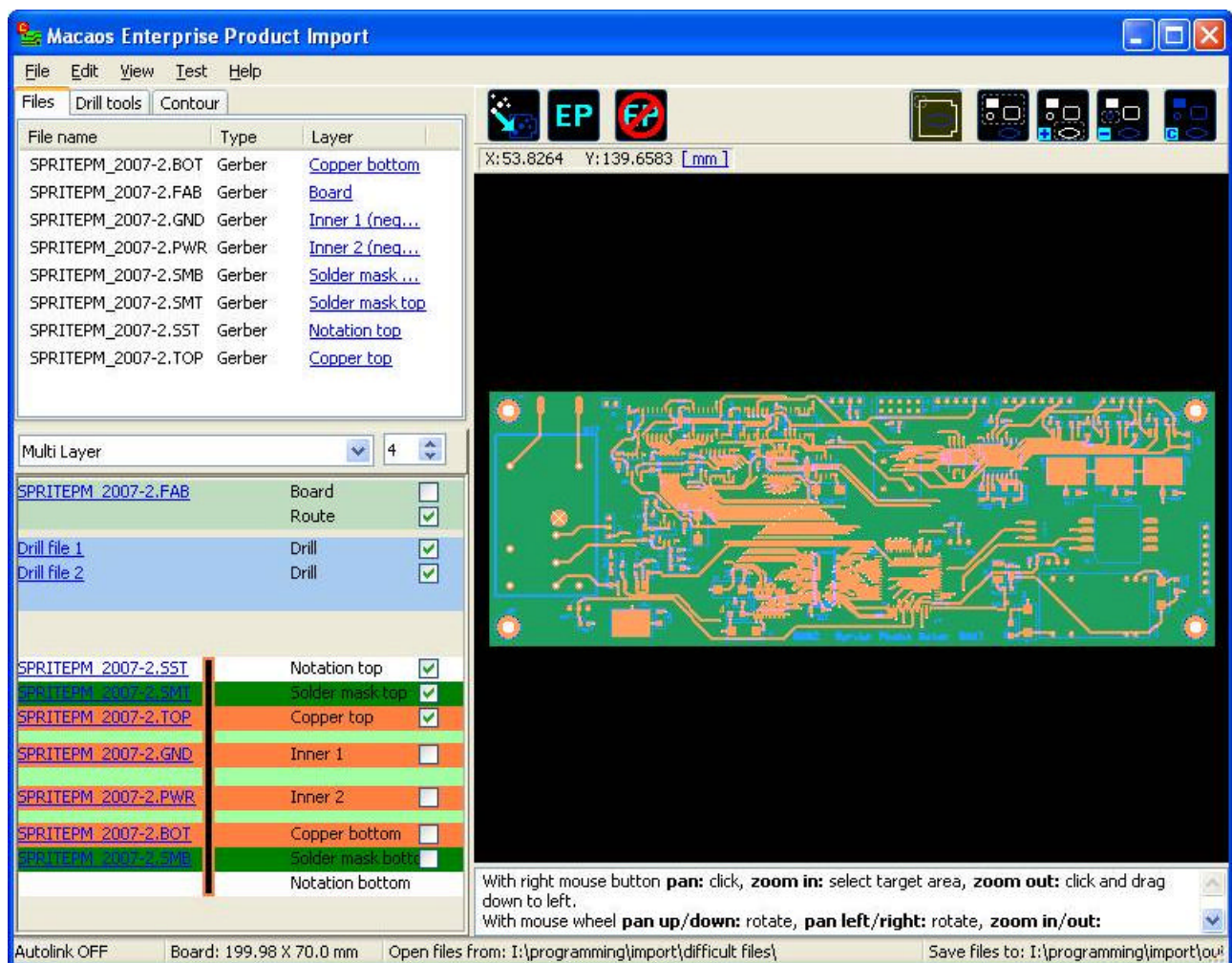
The *Files* tab lists the files in the project and specifies which layer in the board each file represents. The *Drill Tools* tab is used to inspect and modify the drill holes in the product. The *Contour* tab is used to specify the board's outer contour, as well as any cutouts or scoring lines.

Creating a product

To create a product, you need to do the following:

1. **Prepare for a new product.** Start the program, or choose *File/New product* if you have previously been working with a different product.

2. **Open the data files.** Choose *File/Open files* to open files and add them to the files listed in the *Files* tab. If a single zip file is added, then this will be unpacked and its contents added to the *Files* tab. You may select multiple files from the same folder. You can also choose *File/Open files* again to add additional files to the *Files* tab.

3. **Link files to their corresponding board layers.** If Autolinking is on, and the Import Module recognizes the filenames as matching one of the defined filters, then recognized files will be liked automatically to their corresponding layers. (See the autolinking section for more information.) Files which are not autolinked must be linked manually. This is done by clicking on the blue text in the *Layer* column of the *Files* tab and selecting the desired layer from the layer menu. The file image is displayed, in order to help select the correct layer. Right-click on a file to view the file as text or change the file type.

4. **Review drill holes.** In the *Layer* overview, set check marks by the drill layers and a copper layer, then go to the *Drill tools* tab. This tab lists the drill files and drill tools which have been linked. Ideally, the drill holes will be displayed centered on their corresponding copper pads. If not, see the drill tools section for information on how to make adjustments. Drill tool sizes are listed, rounded to the nearest actual tool size at the manufacturer. If tool sizes could not be extracted from the drill file, these must be entered manually by clicking in the *Manuf. size* column and typing in the correct size. Be sure to also check/correct plating for each tool, and drill depth for each file.

5. **Define the board contour.** Go to the *Contour* tab. The *Active layer* (by default, the Board layer) is shown in blue and other layers are shown in gray. Selecting a vertex of the outer contour, causes the selected segments to be highlighted. Using the + and – icon buttons, select the remainder of the contour (and no other segments). If the selected segments form a closed loop, then the *Outer* button in the *Contour* tab will be enabled. Click on the *Outer* button to define the outer contour. Select and define any additional contour objects (cutouts, routed tracks, scoring lines). See the contour section for information.

6. **Place the manufacturer's identifier.** The manufacturer adds a product/batch number to the board as part of the production process. You may select the position of the identifier by clicking on the EP icon button and placing the rectangle at the desired position on the board. The identifier will appear on the *Notation Top* layer. If you do not place the identifier, then the manufacturer will choose an appropriate location prior to production.

7. **Define board specifications.** Choose *Edit/Board specifications* to open the specifications dialog box. You must specify a product name and select a stackup. You must also specify minimum track width, clearance and annular ring for the product. In addition, you may choose non-standard surface finish, solder mask color and notation color. If the product is to have electrolytic (hard) gold applied to connector surfaces, then the gold area (in cm²) must also be specified.

8. **Publish the product.** Choose *Edit/Approval checklist* to open a list of items which must be approved before publishing to the Macaos Enterprise product archive. After having gone through each item, choose *File/Publish product* to complete product creation. After checking for missing information, the specifications dialog box is displayed for review and then the product is published. See the publishing section for more information.

Graphic display

The graphic region displays the Gerber and drill images either individually or together. Clicking on an image file in the Files tab causes only that image to be displayed; while the check boxes in the layer overview may be used to display several images simultaneously.

At the top there are eight icon buttons and a status bar which shows the current cursor position in Gerber coordinates. The left three buttons are described below. The remaining buttons are described in the Contours section.

Aligning drill files

In general, all Gerber and drill files should have the same coordinate origin in order to insure that all layers match. However, some CAD systems generate Gerber files separately from drill files, resulting in the two file types having different origins. This can be corrected as follows:

1. Click on the *Align drill with pattern* button to depress the button.

2. With the left mouse button, draw a selection rectangle around a drill hole. A rubber band segment stretches from the hole to the mouse cursor.

3. Draw a selection rectangle around the corresponding copper pad. All drill holes move to snap onto their pads.

Placing manufacturer ID marks (EP symbols)

The manufacturer places a part/batch number mark (EP symbol) at some position on the *notation top* layer of the board, in order to identify the boards during production. If you wish, you may specify the position of this mark, rather than leaving it to the manufacturer to place the mark. If you choose not to place a mark, the mark will be placed by the manufacturer (unless there is not enough "empty" space on the board for the mark). To place a mark:

1. Click on the *Place EP symbol* button to depress the button.

2. Place the rectangle at the desired location.

3. Press Escape to exit the placement mode.
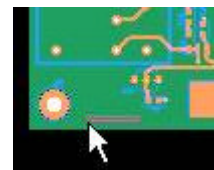
You should only place one mark on a board. If the "board" is a panel of boards, then a mark should be placed on each board in the panel. Use the *Clear EP symbols* button or the *Delete* key to remove all marks.

Pan and zoom

Panning and zooming can be done with the right mouse button, the mouse wheel or the keyboard.

|	Keyboard	Right mouse button	Mouse wheel
Pan		click	rotate or Shift+rotate
Zoom to fit	Home	click, drag upwards to left, release	
Zoom in	Page Up	click, drag upwards to right around area, release	Ctrl+rotate
Zoom out	Page Down	click, drag downwards to left, release	Ctrl+rotate
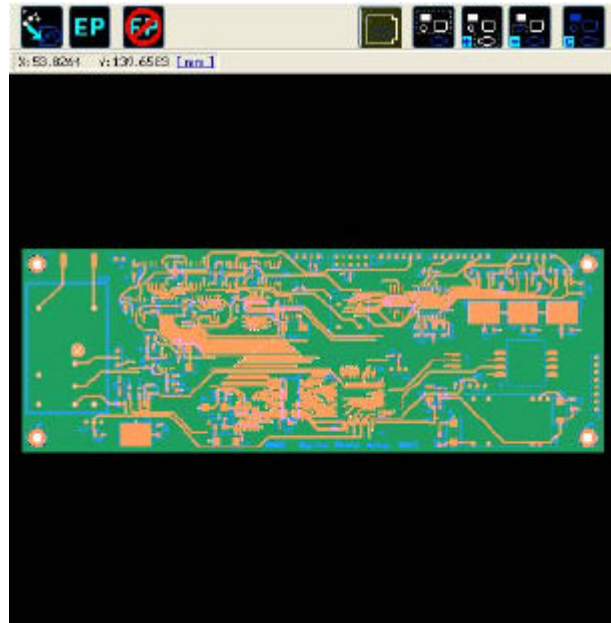
Layers

A printed circuit board is made up of several "layers". These include copper layers, mask and notation layers, drill files and more. Each layer is described by one (or sometimes two) image files, and normally a file describes only one layer. The *Layer* overview shows the filenames and layer names of the board. The check boxes for each layer may be used to view or hide the layer in the g*raphic display*.

The Route layer is automatically generated by adding contour objects (in the *Contour* tab).

The controls at the top of the *Layer Overview* may be used to specify the number of copper layers on the board. This is automatically increased if you link additional layers in the *Files* tab. However, if you want to reduce the number of copper layers, you must do so here.
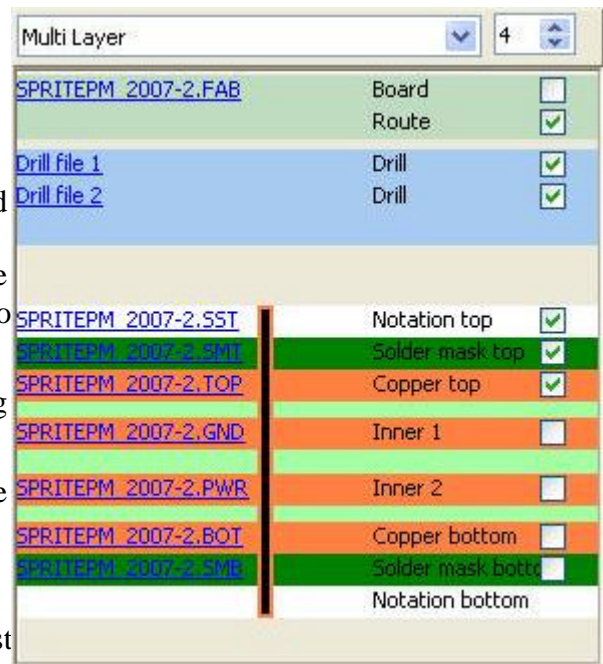
Files

The *Files* tab lists the files which have been added to the project using the *File/Open files* command. Each file is listed together with its file type and the layer to which the file has been linked.

When a file is added to the list, an attempt is made to determine the file type. If, for some reason, the file type is incorrect then it can be changed by right-clicking on the file type and choosing the correct file type from the *Change file type* submenu.

To link a file to a layer, click on the layer name (or *Not Selected*) and choose the desired layer from the menu. To unlink a file, choose *Not used*.

You can view the contents of a file in a text viewer by right-clicking on the file name and choosing *View file as text*. This can be useful if you wish to view a text file which lists the file names and their corresponding layers while you are linking the image files.
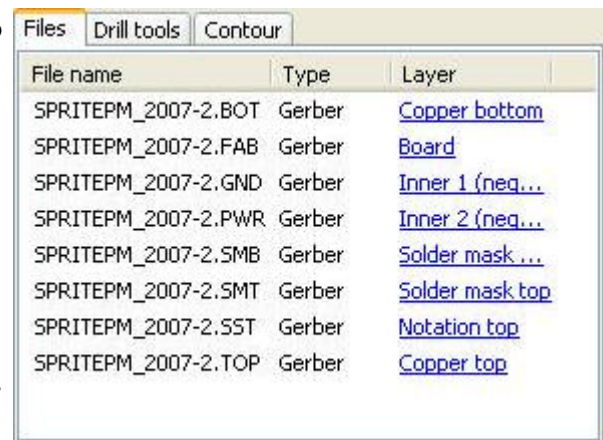
Sometimes, it may be necessary to use the same file for two different layers. For example, you may have only one solder mask file, which is to be used both for the top and bottom layers. To solve this problem, right-click on the file name and choose *Duplicate file*. This will add a copy of the file to the list, which can then be linked to another layer.

When an image file is imported into the system, warnings and errors (if any) are stored in a report which can be viewed by right-clicking on the file name and choosing *View file import report*.

Autolinking and filters

Drill tools

Contours

Board specifications

Publishing

Options

Shortcut keys

File	New product	Ctrl+N
File	Open files ...	Ctrl+O
File	Save product ...	Ctrl+S
File	Publish product ...	Ctrl+P
File	Exit	Ctrl+Alt+X
Edit	Board specifications ...	Ctrl+B
Edit	Approval checklist ...	Ctrl+A
Edit	Link filters ...	Ctrl+L
Edit	Create filter from current mapping	Ctrl+M
View	Hide linked files	Ctrl+Alt+L
View	Hide non-image files	Ctrl+Alt+N
View	Hide common part of file names	Ctrl+Alt+C
Zoom to fit	Home
Zoom in	PageUp
Zoom out	PageDown
Clear manufacturer's ID symbol	Delete
Escape from mode	Escape