

## MACAOS output generation

This program generates a MEI file that MACAOS can import directly for PCB production

More info about MACAOS can be found at: <https://www.macaos.com/>



## Benefits of using MACAOS output generation

- No manual import of Gerber files, drill, mill, pick and place data
- Eliminate error prone manual tasks
- MACAOS will automatically recognize stackup, blind/buried vias, cutouts etc.
- 1-click button to go from Cadence OrCAD/Allegro PCB Editor to MACAOS Enterprise
- Automatically setup default soldermask color, silkscreen color and surface finish.
- Automatically setup cutting tool for outline and inner cutouts.
- User can change default settings which will be stored in board file for reuse with next revision
- Better pricing when importing from OrCAD/Allegro
- Better pricing than ordering directly from PCB manufacturer

Watch the movie at [https://youtu.be/i5C-V\\_rh\\_yA](https://youtu.be/i5C-V_rh_yA) to see the program in action.

Zone	Stackup	Layers	Thickness	Stackup
1	None	PRIMARY	2	1.2000 mm 2005 1.2mm 1oz (35um)

## How to get MACAOS output generator

It is available free of charge.

All you need to do is signup at <https://www.nordcad.dk/macaos>

## Content

MACAOS output generation .....	1
Benefits of using MACAOS output generation .....	1
How to get MACAOS output generator .....	1
Requisites for using the MACAOS output generator .....	2
MACAOS output generator within the nsWare framework .....	2
MACAOS output generator manual installation .....	2
Program usage .....	3
Usage details .....	3
Defining cutouts in the board .....	3
Defining plated or non-plated milling/route/cutting paths .....	3
How to open the generated data in MACAOS Enterprise .....	5
Program user interface buttons and settings .....	7
General program usage .....	7
List of supported and not supported functionality .....	8
Supported features .....	8
Not supported features .....	8
Revision history: .....	8
Default data layers .....	9
Configuration file .....	9

## Requisites for using the MACAOS output generator

The program can be installed using either one of the following procedures.

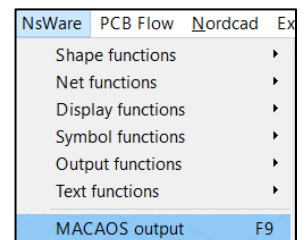
- Within the NsWare framework
- Manual installation

### MACAOS output generator within the nsWare framework

Download and install the NsWare framework using the NsWare installation guide

[http://nordcad.dk/download/nsware/nsware\\_install.pdf](http://nordcad.dk/download/nsware/nsware_install.pdf)

After the installation, place **ns\_macaos.il** file inside the NsWare folder. The next time PCB Editor is started it will automatically detect the MACAOS output generator and create the corresponding menu item.



### MACAOS output generator manual installation

1. Place ns\_macaos.il file inside a skill file directory
2. Add the following line to the allegro.ilinit file inside this directory
  - a. load("ns\_macaos.il" "nsWare")
3. The next time PCB Editor is started the MACAOS generator can be started running the command **"ns\_macaos"**

## Program usage

When a PCB Design is finished and needs to go to production this program will automate the process for use with MACAOS Enterprise.

Unless there are special requirements, the export procedure is as follows

1. The user run the program using NsWare → MACAOS or use the command 'ns\_macaos'
2. Select the desired Stackup from the Stackup dropdown. For complex stackups, this step may be omitted and the stackup defined in the Macaos import module.
3. Select "Create output" to generate .MEI output
4. Start MACAOS
5. Click "Import" and browse to the generated .MEI output
6. Make changes/choices in MACAOS if needed
7. Order PCB Fabrication

## Usage details

This section describes contain a small how-to on creating some of the special data like cutout and milling path definition. There is also a description of how to open the generated data in MACAOS.

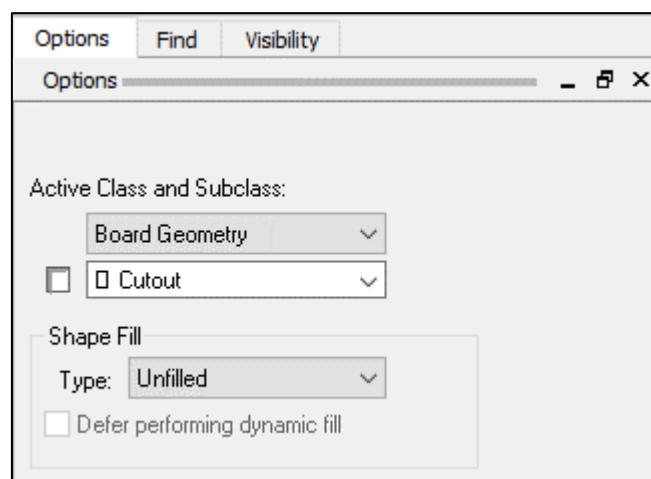
### Defining cutouts in the board

A cutout is a closed area that is cutout of the board with a cutting/milling tool that has a diameter. A number of cutting diameter sizes are available in MACAOS and can be defined directly in MACAOS output generator.

The cutting tool diameter determines the radius of the inner corners of the cutout. The minimum tool diameter is 0.6 mm. Default: Cutout toolsize = 1.6mm

To define a Cutout in OrCAD/Allegro PCB Editor do the following

1. Select Shape → Polygon/Rectangle/Circular
2. In Options panel select "Board Geometry/Cutout" and Shape Fill Type = Unfilled



3. Now draw or place the desired cutout area

### Defining plated or non-plated milling/route/cutting paths

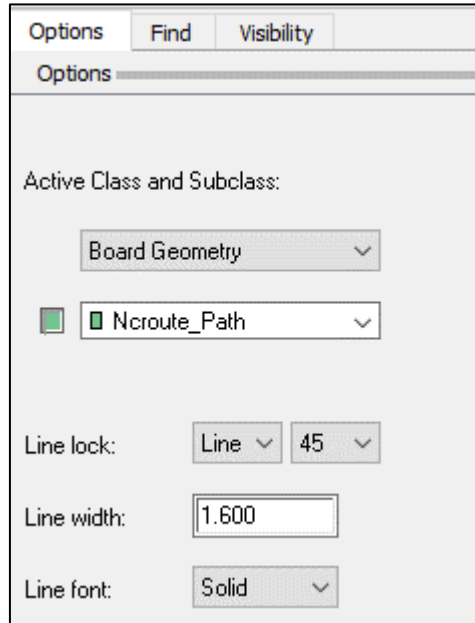
This is a path that is created by using a milling/cutting/route tool which will follow a path draw as lines inside the board.

The toolsize of this path is defined by the line width of the path. It is recommended to use standard tool sizes like for cutting the board contour.

Notice: The program will not run if tool sizes below 0.6mm are used. Tool sizes above 0.6 are automatically rounded to the nearest 0.1mm. Also, slots larger than 2.4mm will often be converted to cutouts/inner contours by manufacturer.

**To define a non-plated path do the following (see later in document for plated path)**

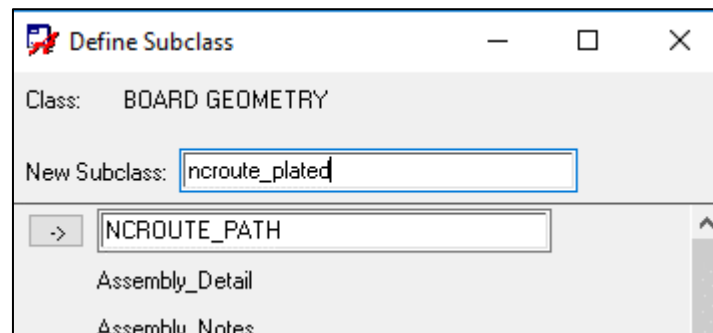
1. Select Add → Line
2. In Options panel select “Board Geometry/Ncroute\_Path” and set Line Width as desired
  - a. Preferably use one of the following tool sizes: 0.8mm, 1.0mm, 1.6mm, 2.0mm or 2.4mm



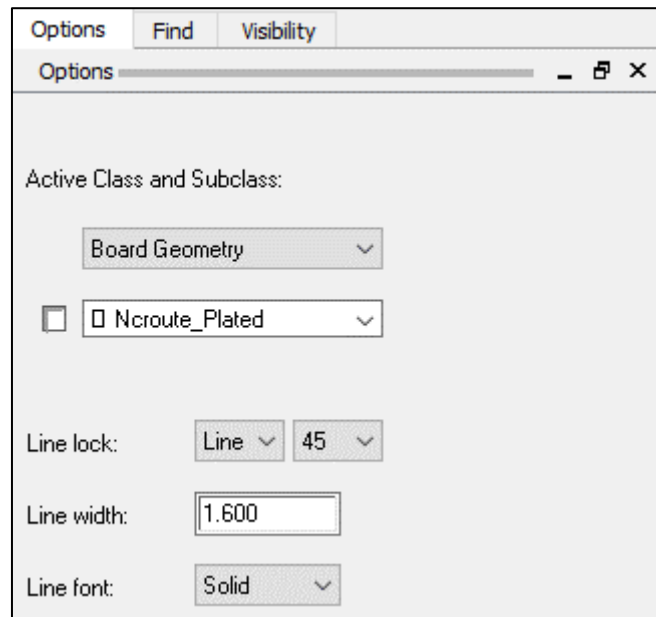
3. Now draw the desired non-plated path inside the board

**To define a plated path do the following**

1. Select Setup → More → Subclasses
2. Left click on “Board Geometry” in the dialog
3. Write “Ncroute\_plated” in the “New Subclass” field and press ‘TAB’ key



4. Close the “Define Subclass” dialog using the small cross in the upper right corner of the dialog
5. Close the other Define Subclass dialog with “OK”
6. Select Add → Line
7. In Options panel select “Board Geometry/Ncroute\_Plated” and set Line Width as desired
  - a. Preferably use one of the following tool sizes: 0.8mm, 1.0mm, 1.6mm, 2.0mm or 2.4mm)



- Now draw the desired non-plated path inside the board

### How to open the generated data in MACAOS Enterprise

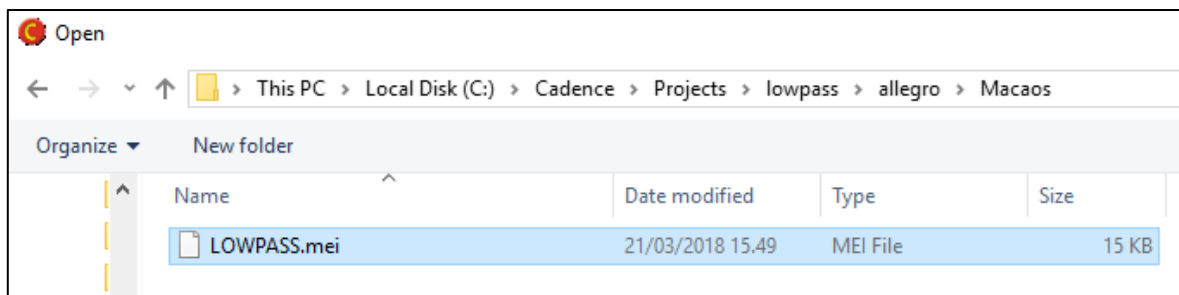
After the MACAOS output generation has run a subdirectory named MACAOS contain a .MEI file that can be directly imported in MACAOS.

To import the file do the following

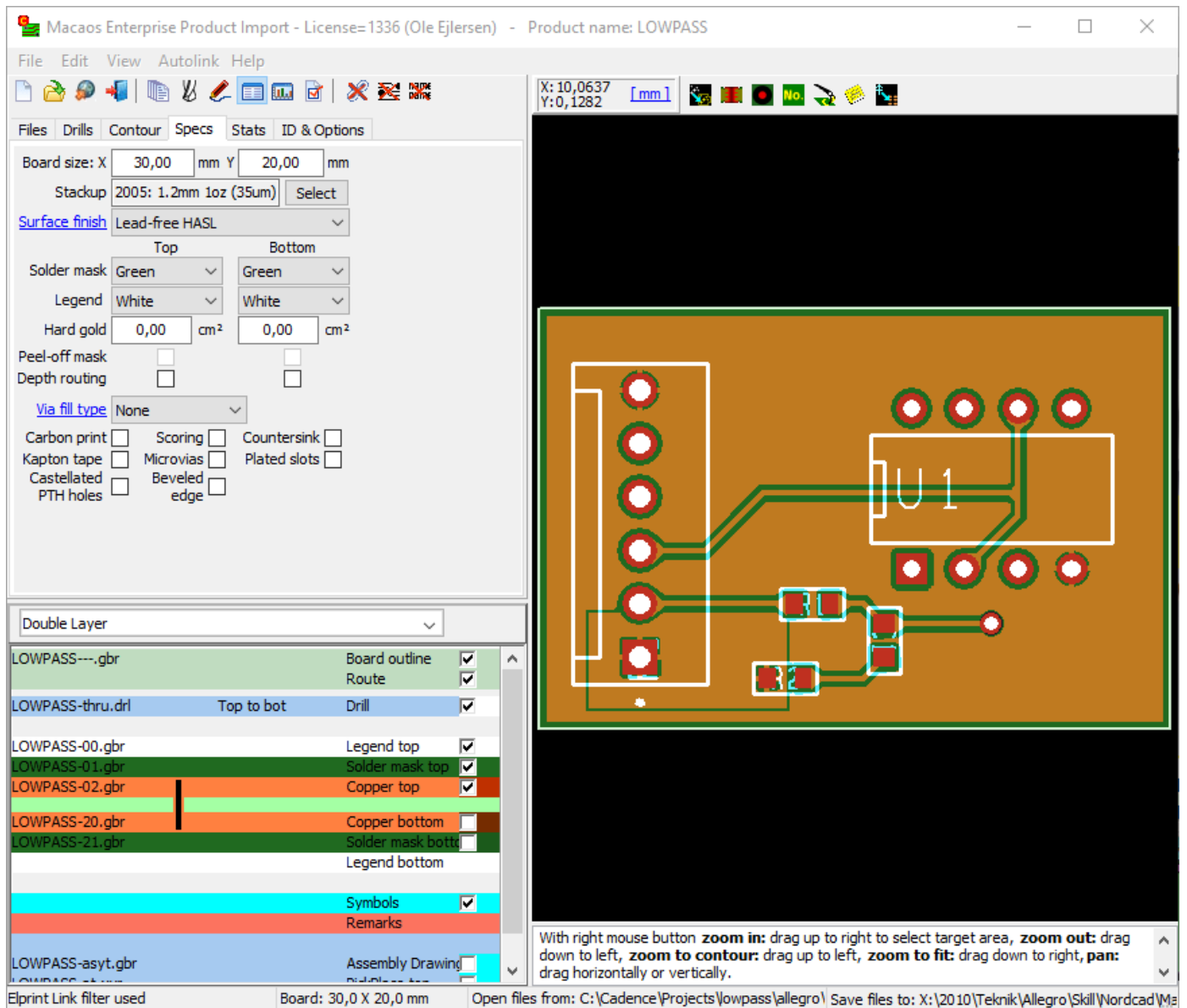
- Start MACAOS
- Left click "Import" button



- Browse board directory\MACAOS and the .MEI file that was just created



- Click "Open" to open the file
- Notice that MACAOS auto recognizes all data in the board and manual import and setup is needed



6. Typically, the only operation that will need to be performed in the Import module is to place the manufacturer's ID mark on the board. However, we do recommend that you check all parameters carefully.
7. Once you are satisfied with the import, click on the Publish button to publish the imported board. Once the board is published, you can step up to a panel, create a solder paste stencil, create a test fixture, get prices and place orders as desired

## Program user interface buttons and settings

Macaos output generation 1.04 Apr 24 09:30:48 2018

Productname: Lowpass

Article number: Art1234

Description: Lowpass filter

Board Type:  Rigid,  Flex,  Rigid-flex,  Metal-based

Silkscreen Color: 5 White

Soldermask Color: 6 Green

Outline toolsize: 1.6 Mm

Cutout toolsize: 1.6 Mm

Surface finish: 6 Lead-free HASL

Reset above to defaults

Zone	Stackup	Layers	Thickness	Stackup
1	None	PRIMARY	2	1.2000 mm 2005 1.2mm 1oz (35um)

Open MACAOS output dir: "C:/Cadence/Projects/lowpass/allegro/MACAOS"

Create MACAOS output

More info about MACAOS

Close

Help

Contact Nordcad at support@nordcad.dk / support@nordcad.no with feedback!

## General program usage

- **Productname:** Specify the name of data generated. Default is board file name
- **Article:** Here an article (part number) can be specified
- **Description:** Here a description can be specified
- **Board Type:** Select the board type
- **Silkscreen Color:** Select color for silkscreen data. Default is White
- **Soldermask Color:** Select color for soldermask data. Default is Green
- **Surface finish:** Select surface finish. Default is Lead-free HASL
- **Outline toolsize:** Cutter toolsize for cutting the outer contour of the board
- **Cutout toolsize:** Cutter toolsize for cutting cutouts inside the board
- **Stackup:** Here the desired MACAOS stackup can be selected.
- **Open MACAOS output dir:** This button opens the directory with the .MEI file
- **Create MACAOS output:** Clicking this button will generate the MACAOS.MEI output
- **More info about MACAOS:** Click to get more information about MACAOS
- **Close:** Close the dialog
- **Help:** Open the latest edition of this document which can be found at [https://www.nordcad.dk/download/nsware/nsware\\_macaos.pdf](https://www.nordcad.dk/download/nsware/nsware_macaos.pdf)

The above settings are saved within the board when MACAOS output are generated. The stackup selection will only be restored if the number of layers match with the number from the last run. Otherwise the stackup selection is cleared.

## Appendix

### List of supported and not supported functionality

It is important to notice the list of supported functionalities.

The program is developed and tested on OrCAD/Allegro version 17.2-2016s029 and newer.

#### Supported features

- Boards in millimeters
- 2 or more layers
- Design outline defined on "Board Geometry/Design\_Outline"
- Cutouts defined as closed objects on "Board Geometry/Cutout"
- Plated and non-plated circular drill holes
- Oval slots
- Non-Plated milling paths defined as lines with a width on "Board Geometry/NCRoute\_Path"
- Plated milling paths defined as lines with a width on "Board Geometry/NCRoute\_Plated"
- Silkscreen data (notation)
- Soldermask data
- Paste mask data
- Assembly drawing data

More functionality is in development.

#### Not supported features

At program startup a number of checks are run to check if the board has unsupported functionality. If this is the case data generation is prohibited and a list of issues is shown.

The MACAOS output generation ***program currently does not support*** the following

- Boards not in millimeters
- Single layer boards (complete testing has not finished yet, so check data carefully in this case.)
- Back-drilling
- Rectangular slots
- Rectangular drill
- Counter sink and counter bore
- Stiffener, coverlay and other flex specific data
- All the Non-Standard drill types (Laser, Plasma, Punch, Wet/dry etching, Photo imaging, Conductive ink formation, Dual, Other)

#### Revision history:

1.12: Layer name fixes for boards with more than 10 layers

1.11: Minor fixes

1.10: Minor fixes

1.09: Enhancement requests implemented

- Added functionality to customize output for silkscreen and assembly layers. Read more in the section named Configuration file on page 9



- Silkscreen data are generated for both top and bottom if such data exists. Previous behavior only top silkscreen data was generated if no bottom placed components existed.
- Maintain previously generated .mei files instead of deleting them.

File is renamed to “<original filename>\_backup-dd-mmm-yyyy-hh-mm.mei” where **dd** is the date, **mmm** is the month, **yyyy** the year, **hh** the hour and **mm** the minute of the file being renamed.

1.07: Fixed issue with board directory including spaces

1.05: First official release

## Default data layers

The MACAOS output is based on several data layers within OrCAD/Allegro PCB Editor.

The data structure is the following

- Copper layers (xxx represents layer names like Top, Bottom etc. –)
  - Etch/xxx
  - Pin/xx
  - Via Class/xxx
- Soldermask\_top/bottom
  - Pin/Soldermask\_top/bottom
  - Via Class/Soldermask\_top/bottom
  - Package Geometry/Soldermask\_top/bottom
  - Board Geometry/Soldermask\_top/bottom
- Pastemask\_top/bottom
  - Pin/Pastemask\_top/bottom
  - Via Class/Pastemask\_top/bottom
  - Package Geometry/Pastemask\_top/bottom
- Assembly\_top/bottom
  - Package geometry/assembly\_top/bottom
  - Ref des/assembly\_top/bottom
  - Board geometry/assembly\_top/bottom
- Silkscreen\_top/bottom
  - Package geometry/Silkscreen\_top/bottom
  - Ref des/Silkscreen\_top/bottom
  - Board geometry/Silkscreen\_top/bottom

Notice: If something else is wanted for silkscreen and assembly layers it is necessary to use a configuration file as described in the section named Configuration file on page 9

## Configuration file

Customizing output for Silkscreen and/or Assembly is possible with revision 1.09 or newer.

- This must be defined in configuration file (.txt) found in either one of the following directories
  - %cds\_site%\pcb\nsware\macaos
  - %allegro\_pcbenv%\nsware\macaos
  - %home%\nsware\macaos

- If one of the above directories are found and contain 1 or more .txt files this will expand the user interface at the bottom to contain the following



- Use config file if one of the configuration files should be used
- Configuration file can be selected in drop down
- The path for the configuration file is shown.
- Save will save the settings (including all other settings in the dialog). Settings are saved in the board and will be reused the next time if configuration file still exists.

Example: "%cds\_site%=C:\Cadence\Company" so file could be c:\Cadence\Company\pcb\nsware\macaos\my\_config.txt

Format is layer type and then semicolon separated class/subclass names as shown below. Do not embrace the class/subclass in quotes/double quotes.

```
1 assembly_top;package geometry/assembly_top;ref des/assembly_top
2 assembly_bottom;package geometry/assembly_bottom;ref des/assembly_bottom
3 silkscreen_top;package geometry/silkscreen_top;ref des/silkscreen_top
4 silkscreen_bottom;package geometry/silkscreen_bottom;ref des/silkscreen_bottom
```

Legal layer types are

- assembly\_top
- assembly\_bottom
- silkscreen\_top
- silkscreen\_bottom

**Notice:** If a layer is listed in a configuration file then this takes precedence over the default data structures used for these layers. So if assembly\_top is defined in configuration file then it will only contain the data listed here.