

Contents

Setup and Licensing

1. General

2. Project database

3. Program workspace and interaction principles

- 3.1. Immediate, modal and setup tools**
- 3.2. Interaction with graphical elements**
- 3.3. Main Menu**
- 3.4. Task and Control Tools**
- 3.5. Editing Tools and Options**
- 3.6. Control of Graphic Screen and cursor movement**
- 3.7. Pop-up dialogs**

4. Import

- 4.1. File and data formats for importing artworks, board outline and cutouts**
- 4.2. File and data formats for importing drilling data**
- 4.3. Import Parameters pop-up dialog**
- 4.4. Importing artworks from Gerber files**
- 4.5. Importing board outline and cutouts from Gerber files**
- 4.6. Usage of filter with import of artworks and outlines**
- 4.7. Importing drilling data from Excellon Format files**
- 4.8. Importing drilling data from Gerber files**
- 4.9. Plated and non-plated holes**

5. Preparation for generating toolpaths

- 5.1. Tool Database and tool categories**
- 5.2. Aligning layers**
- 5.3. Basic editing of PCB outline and cutouts**
- 5.4. Design Rules Check (DRC)**
- 5.5. Setting up contouring and fabrication parameters**
- 5.6. Introducing copper removal areas**
- 5.7. Optimizing toolpaths**

6. Contouring

7. Output of generated toolpaths

8. Editing function

- 8.1. Usage of snapping and contact points with editing**
- 8.2. Editing artwork images**
- 8.3. Editing board outline and cutouts**
- 8.4. Editing drilling data**
- 8.5. Editing contours**

9. Printing

Setup and Licensing

This software is supplied in form of executable self-extracting setup program ColiLiner.exe. It can be downloaded from the Colinbus website. The CD with ColiLiner.exe may be delivered as an option.

In both cases, the installation starts automatically by running ColiLiner.exe. The setup program installs all necessary software components. After installation, the user will find Colinbus-ColiLiner entries in Program Manager Menu.

The software has been tested to run properly with following operating systems:

Windows 2000, Windows XP and Windows Vista. Using this software in Windows 95, 98 or Millennium environments is not recommended.

Installed software occupies about 11 MB of hard disk space.

Following folders are created by setup program during installation process:

```
\ColiLinerV[revision number]
|----\SYS
|----\JOB
```

Folder **\ColiLiner[revision number]** contains all software components of the package. **This folder must not be renamed.**

Folder **\ColiLiner\SYS** contains important system files. **This folder must not be renamed. Users are warned not to change or delete any file in this folder because it can cause system malfunction.**

Folder **\ColiLiner\JOB** is intended for storing files containing user's Project Databases. It is also default folder for storing Gerber and Excellon format files for import. Nevertheless, import files may be opened from any location.

ColiLiner is licensed software. Upon installation, the program works in so-called evaluation mode. In this mode, the user may evaluate full functionality of the program. **Only output of machining programs and saving Project Databases is locked.**

The license has to be activated by entering License Activation Code obtained from manufacturer/supplier of this software. Licensing procedure is following:

- Run ColiLiner.
- Click on **Main Menu** entry **Activate License** for **License Activation** dialog. For every installation, the system generates unique License Identification Code that has to be sent to the vendor. It is most practical to click on **Send E-mail with ID code** button. In this case, the program will automatically prepare complete message in user's email program.

In order to activate you license, please do following:
1. Copy your unique License Identification Code
2. Send it to the vendor where you purchased this software (activation@colinbus.com)
3. You'll receive your License Activation Code from there
4. Enter this code and your name in the white boxes below and press Accept

Your License Identification Code:

Enter License Activation Code in the box below:

Enter user name in the box below:

- The same dialog is used to enter License Activation Code received from the vendor. The program requires also that user name is entered. Subsequent clicking on **Accept** button initiates activation.

In order to activate you license, please do following:
1. Copy your unique License Identification Code
2. Send it to the vendor where you purchased this software (activation@colinbus.com)
3. You'll receive your License Activation Code from there
4. Enter this code and your name in the white boxes below and press Accept

Your License Identification Code:

Enter License Activation Code in the box below:

Enter user name in the box below:

Successful license activation is confirmed by a message and disappearance of no longer needed **Activate License** entry from Main Menu.

Note:

The program will discard current Project Database after license activation.

1. General

The purpose of this program is to calculate and generate toolpaths for fabrication of printed circuit boards on milling machines. Using data generated by the program, the machine is able to perform following tasks:

- Mill along the contours of tracks, pads (footprints) and other conductive elements on a PCB layer in order to insulate connections from surrounding copper on the copper clad sheet of laminate.
- Optionally remove copper in those areas on the PCB where there are no connections
- Drill holes.
- Cut out the exact external outline of a PCB from the laminate as well as internal cutouts.

Source data for calculation and generation of toolpaths are images of PCB layer artworks and drilling data imported from files created by various software packages for PCB layout design (CAD EDA software).

Most PCB CAD software packages export PCB layer images in so-called Gerber format that is intended primarily for rendering artworks on photoplotters. These images in their raw form are not suitable for passing directly to milling machines. They require processing through application of a complex contouring algorithm in order to calculate correct toolpaths.

Drilling data are exported from PCB CAD packages in the format known as Excellon Program Format. In this case, the program needs only to convert drilling information to format acceptable by milling machine.

The procedure for generating of complete machining program for PCB fabrication consists of following steps:

- Import of artworks images (Gerber files), separately for each layer of the PCB
- Import of PCB outlines (Gerber files)
- Import of drilling data (Excellon files). There is also a provision to reconstruct drilling data from selectively imported artwork elements in Gerber files
- Editing of imported data - it is usually necessary to edit only PCB outline and / or internal cutouts. Other important editing step at this stage may be aligning imported layers to the common origin
- Setting up fabrication and contouring parameters (contouring strategies, tool sizes, required DRC checks etc)
- Defining areas where copper removal should take place (optional)

- Calculation of toolpaths (contouring)
- Output (to a disk file) of resulting machining program in format accepted by milling machine

The program provides also a set of advanced features that allow editing imported artwork images, drill template and – as already mentioned board outline and cutouts. Toolpaths - the contours generated by the program - may also be edited in cases where manual change (and eventual corrections) of automatically obtained results is required.

Editing tools enable to remove, relocate or transform graphical elements in artwork images, outlines and toolpaths. Drilled holes may be removed and relocated. There are also functions for changing geometrical properties of elements - lengths, sizes, diameters and shapes. Tools for defining shapes and sizes of copper removal areas are also incorporated into editing tools set.

Apart from operating on imported elements, the user may manually add elements to artwork images and create new holes. Quite often, it is necessary to create board outlines and cutouts without importing. Tools for this task are also provided.

2. Project Database

Project Database consists of imported artwork images, drilling data and board outlines. After contouring, the generated toolpaths become also part of the database. Content of the database is graphically presented on the computer screen within program's workspace.

The database base is logically divided into separate entities called layers. Initially, the program creates empty database consisting of following layers:

- **Artwork Top Layer** - intended for storing elements imported as artwork image of PCB top layer
- **Artwork Bottom Layer** - intended for storing elements imported as artwork image of PCB bottom layer
- **Board Outline/Cutouts** – intended for storing elements imported as PCB external outline and internal cutouts
- **Drill (Plated holes)** – intended for storing drilled holes
- **Drill (Non-plated holes)** – optional layer intended for storing non-plated drilled holes

Each of “artwork” layers has attached corresponding “contours” layer where results of toolpaths calculation for this layer are stored.

Note:

Professional version allows adding six extra artwork layers (Artwork Inner Layer (A) to Artwork Inner Layer (F) together with corresponding contours layers.

Images for each layer are composed of simple graphical elements, which are individually accessible for editing. Depending on layer type, certain elements are allowed and certain are not. For example, element “HOLE” is not allowed on artwork layers and Board Outline/cutouts layer.

Graphical elements of each layer are displayed with different default colors. User has an option to assign other colors from the palette of 32 colors. Other options allow selecting the sequence of layer display order and the visibility.

Recognized graphical elements are listed in the table below:

Element	Properties						Remarks
LINE	Style "Solid"	start (X,Y)	end (X,Y)	line width			Contoured unless 'Mill as line' property is assigned
RECTANGLE	Style "Solid"	A(X,Y)	length (a)	height (b)	rotation	line width	Only on artwork layers. Contoured unless 'Mill as line' property is assigned

RECTANGLE	Style "Filled"	A(X,Y)	length (a)	height (b)	rotation		Only on artwork layers
CIRCLE	Style "Solid"	Centre (X,Y)	radius	line width			Only on artwork layers. Contoured unless 'Mill as line' property is assigned'
CIRCLE	Style "Filled"	Centre (X,Y)	radius				Only on artwork layers
ARC	Style "Solid"	Centre (X,Y)	radius	start angle	arc angle	line width	Contoured unless 'Mill as line' property is assigned
POLYGON	Style "Outline"						Only on Board Outline/Cutouts and contours layers. Closed shape consisting of lines and arcs
POLYGON	Style "Filled"						Only on artwork layers. Closed shape consisting of lines and arcs, usually defining copper pour areas
POLY-LINE	Style "Solid"						Only on contours layers. Open shape consisting of lines and arcs
PAD POSITION	Small square	Position (X,Y)					Only on artwork layers, marks position of Gerber D03* command. Useful in recognizing which imported graphic elements constitute pads (foot prints)
HOLE		Position (X,Y)	diameter				
TEXT	Vector Font	position (X,Y)	letter X size	letter Y size	rotation	line width	Not imported - can be added. Fixed 'Mill as line text' property assigned
TEXT	True Type Font	position (X,Y)	Letter size	rotation			Not imported - can be added. Fixed 'Mill as line text' property assigned

At any stage of work, the Project Database may be saved in binary form to a disk file with extension .CBJ. This feature allows working with the same PCB project in several sessions.

Also at any stage, the current contents of database (in graphical form) may be printed out in desired scale. The current contents (or selected part) of the **Graphic Screen** may be copied as color bitmap for pasting into other documents.

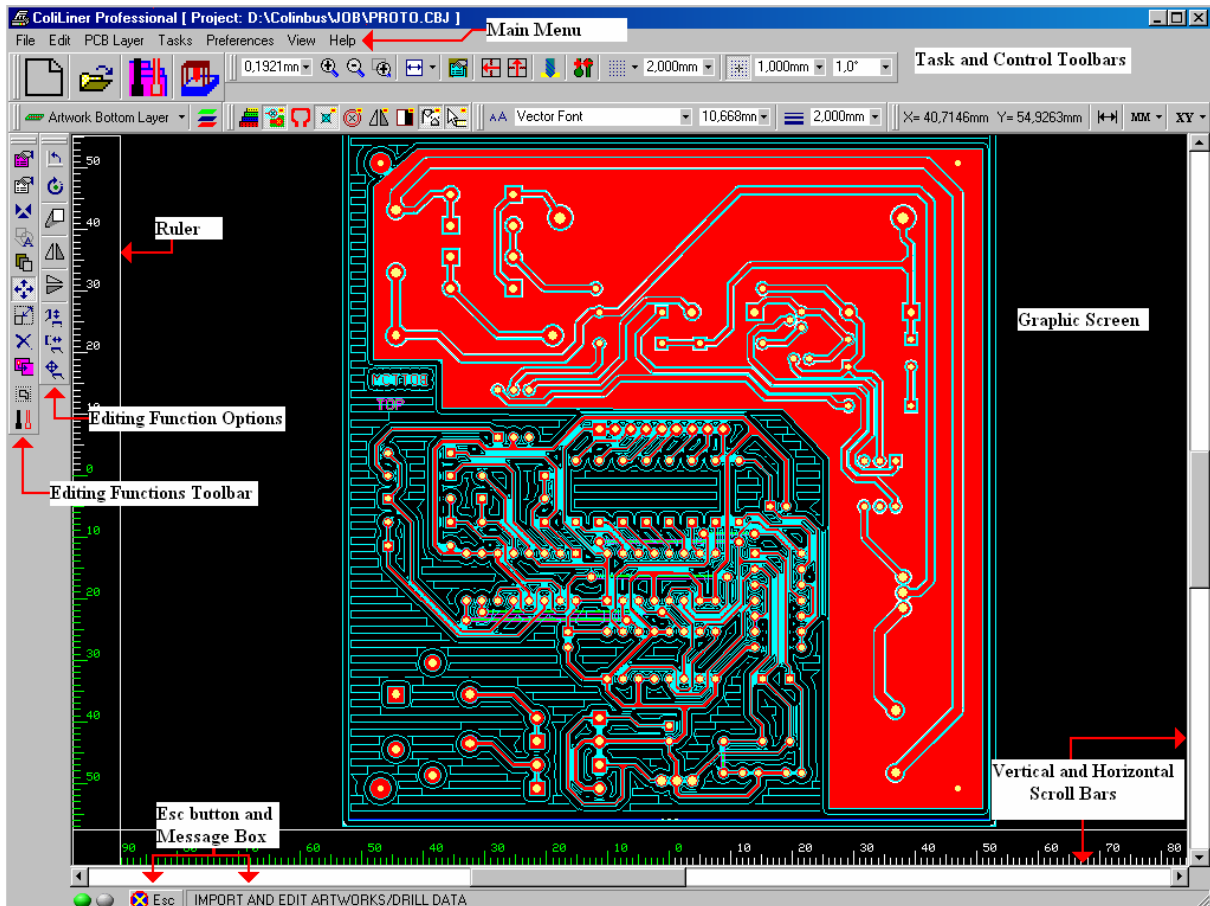
Internal precision (minimal unit) of the database is 1/100000 mm. Coordinates and sizes are stored with this precision. Angular values are stored with precision of 1/1000 degree. The program can process boards with maximal size of 4400 by 4400 millimeters.

In order to achieve maximal size, the X and Y coordinates are expressed in four quadrants of Cartesian coordinate system. It means that imported artwork images are always centered in the middle of coordinate system. If for example, the size of rectangle that frames imported image is 200 by 300 mm, then coordinates of its left lower corner will be $X = -100\text{mm}$ and $Y = -150\text{ mm}$.

Coordinates for toolpaths in the output file are always recalculated to first quadrant (only positive) in reference to the origin common for all layers.

3. Program workspace and interaction principles

Picture below shows the workspace of the program. The workspace may be roughly divided into following fields:



The operation of the program is controlled by clicking on tool buttons in the toolbars or selecting entries from the **Main Menu**. Tool buttons grouped in toolbars visible in the field **Task and Control Toolbars** are duplicated as entries in the **Main Menu**. Certain functions of the program can be invoked only from **Main Menu**.

Editing Functions Tools and Options are normally accessed through respective toolbars. Both these toolbars are duplicated in form of pop-up menus that appear when right mouse button is depressed while cursor hovers over the **Graphic Screen**.

By default, the toolbars are visible but visibility may be switched off in cases when increase of space available for the **Graphic Screen** is required. When a toolbar is invisible, then its functions are still accessible through **Main Menu** or pop-up menus.

The program displays so-called tool tips texts that shortly explain the function of each tool. This happens whenever the cursor hovers over tool button. Full explanation for each editing function and option is visible at the same time in the **Message Box** at the bottom of the screen.

3.1. Immediate, modal and setup tools

Depending on behavior, the tools may be divided into three general kinds; immediate, modal and setup.

Immediate tools invoke required program action directly upon selection. Example of such tool is **New Project**, which causes deleting of current database and creating a new, empty one.

Several of editing tools are **modal**. Such tool is activated upon selection but nothing immediate happens. The program waits for further user's action, which usually is selection of one or more objects visible on the **Graphic Screen**. Example of such function is **Align Layers**. When tool for this function is activated, the program expects that user selects first a location in one layer and subsequently a location in other layer with which the firstly selected location should be aligned. As long as modal tool remains active, the operation may be repeated. Modal tool remains active until other modal tool in the same toolbar is selected.

In all modal functions, the sequence of user's actions is the same. The actual execution begins when the object for interaction is selected by single left mouse button click when cursor points to the object. Progress of execution is in various forms directly visible on the **Graphic Screen** - presented image differs depending of function's nature. For example, for function **Relocate element** it would be highlighted graphic element glued to the cursor and following its movements.

Note:

Left mouse button should not be kept depressed while execution of a modal function (for example, dragging or relocating) is in progress and cursor moves over the graphic screen.

Operation is completed by clicking left mouse button when cursor reaches desired new location for relocated elements. Execution of modal functions can be at any time cancelled by pressing **Esc** key or clicking **Esc** button located on the bottom left corner of the screen.

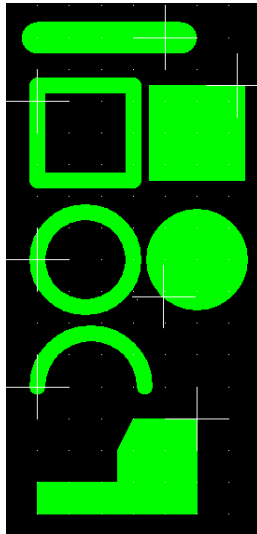
Setup tools are usually drop-down lists or text boxes, where user may either select one of possible settings or directly enter setting's value. Typical example is tool **Units**, a drop-down list, from where the current unit of measure (millimeters or inches) may be selected. Settings that are visible in setup tools remain valid until changed.

3.2. Interaction with graphical elements

Editing Tools operate on single graphic elements or blocks of graphic elements. Most of editing functions are modal. They are set active by clicking on the function's button in the toolbar. Afterwards the program awaits further user's action, which usually is selection of graphical elements.

Selection of single graphical elements

Single graphical element is selected by pointing with crosshair cursor and clicking left mouse button. Following rules apply to pointing different elements for selection (see picture below):

**LINES**

At the centerline. End of line that is closest to the cursor will be taken as the handle for relocation or stretching.

RECTANGLES

At the centerline of the side (not filled rectangle) or at the edge (filled rectangle). Corner that is closest to the cursor will be taken as the handle for relocation or stretching.

CIRCLES

At the centerline of the perimeter (not filled circle) or at the edge (filled circles). Centre of the circle is taken as handle for relocation. Circle is stretched by changing its radius calculated as distance between its centre and current cursor position.

ARCS

At the centerline of the perimeter. End of arc that is closest to the cursor will be taken as the handle for relocation or stretching.

POLYGON and POLY-LINE

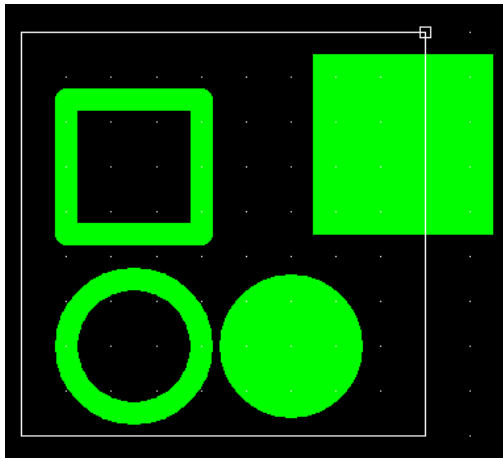
On the edge (filled polygons) or at the centerline of the segment. Vertex closest to the cursor will be taken as the handle for relocation or stretching.

Block selection of graphic elements

Block is selected by dragging bounding rectangle that completely encloses all elements that should be included in the selection. Elements that are only partially covered by bounding rectangle shall not be included in the block. Blocking consists of following steps:

- Placing cursor at location where bounding rectangle should start and clicking left mouse button. There must not be any elements placed in this location.
- Rectangle may be now dragged diagonally until it covers the entire block.
- Clicking left mouse button again confirms block selection.

See picture below. The filled right hand rectangle is not completely covered by the bounding rectangle and shall not be included in block selection:



Note:

Bounding rectangles that define:

- *screen area that should be copied to clipboard as a bitmap (Copy Block to Clipboard in File menu)*
- *block Zoom area*
- *rectangular Copper Removal Area*

are dragged in the same manner as bounding rectangles for block selection.

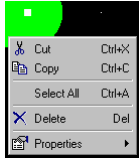
Bulleted

Bulleted is a selection method that allows to pickup one or more elements in cases when dragging bounding rectangle would cover even those that should not be included. Elements are bulleted by selecting them in the manner described above - by left mouse button click on the element - and simultaneously depressing **Ctrl** key on the keyboard. Bulleted elements are instantly marked by small highlighted rectangles. Apart from marking an element as selected, the bullets allow to initiate certain editing operations with this element. Type of operation is indicated by the form of cursor while it hovers over a bullet:



Clicking on the right hand bullet initiates relocation of the element. Clicking on the left hand bullet initiates stretching (transformation) of the element.

Other editing operations are also possible once one or more elements are bulleted. Clicking on the right mouse button pops-up copy of **Edit Menu** at the cursor position:



Any operation selected from this menu will be applied jointly to all currently bulleted elements.

Clicking on **Esc** button or pressing **Esc** key removes all bullets.

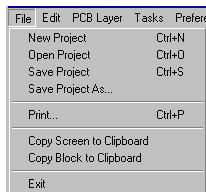
Note:

Irrespectively to selection method, only elements on currently edited layer may be selected. There are few exceptions to this rule (layer alignment, setting of contact points and transfer of outlines). Nevertheless, if an element does not respond to selection, it is an element from other visible layer.

3.3. Main Menu

Most of the functions accessible from **Main Menu** are duplicated as tool buttons in various toolbars. See explanation of toolbars below. Exceptions are **File**, **Edit**, **Preferences** and **View**. These menu entries open drop down sub-menus that generally follow Windows standard:

File Menu



New Project

Clears all layers and contours and creates empty Project Database.

Open Project

Loads previously saved Project Database from selected disk file (file extension .CBJ).

Save Project

Saves current Project Database to a disk file.

Save Project As ..

Saves current Project Database to disk file under a new name.

Print

Opens **Print Preview** dialog from where the current contents of the **Graphic Screen** may be printed in required scale. See chapter about printing below.

Copy Screen to Clipboard

Copies bitmap of entire **Graphic Screen** to clipboard for pasting to other documents.

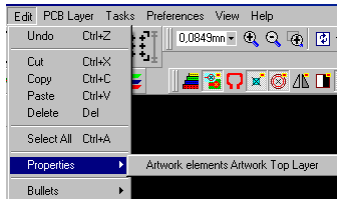
Copy Block to Clipboard

Copies bitmap of the portion of the **Graphic Screen** that is defined by bounding rectangle.

Exit

Closes the program.

Edit Menu



Undo

Restores contents of currently edited layer from before latest editing operation. Undo stack have four levels.

Cut, Copy, Paste and **Delete** functions accessible from **Edit Menu** have been provided for those that are familiar with standard Windows routines and should not be confused with editing functions that are invoked through tool buttons grouped in **Editing Functions and Options** toolbars.

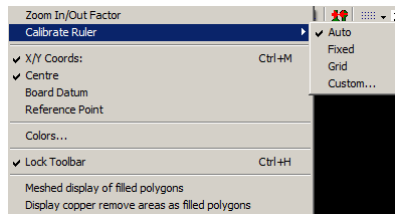
Graphical elements that may be Cut, Copied or Deleted have to be first selected. There are three methods for selecting elements for these functions:

- Depressing key **Ctrl** and clicking on an element allows selecting / deselecting single elements. Every selected element is marked by white rectangles called bullets. This selection method is called “bulleting”.
- Depressing key **Shift** allows dragging bounding rectangle inside boundaries of the **Graphic Screen**. Elements covered by bounding rectangle will be selected as **Block**.
- Clicking on menu entry **Select All** makes all elements selected
- There may be combination of above. Elements selected in the block or by **Select All** may be bulleted through clicking on menu entry **Bullets**. This in turn allows deselecting some elements through combination sequence of **Ctrl** key press and click on element.

Properties

Opens **Property** pop-up dialog allowing changing those geometrical and other properties that are common for all selected (bulleted) elements.

Preferences Menu



Zoom In/Out Factor

Allows adjusting factor by which scale of display is increased/decreases for each **Zoom In** and **Zoom Out** operation.

Calibrate Ruler

The program displays the ruler (Y coordinates on the left side of the graphic screen, X at the bottom). Visibility of the ruler may be switched ON and OFF – see **View Menu**. User may control the size of ruler intervals by selecting one of available options:

Auto - intervals are automatically adjusted to current display precision.

Fixed - 1 mm or 1/16" intervals depending on currently selected unit of measure.

Grid - intervals are equivalent to currently selected raster intervals.

Custom - user defined intervals.

X/Y Coords: / Distance

This menu entry is a toggle, selecting mode for presenting current cursor position in the **Sizes** toolbar. Cursor location is calculated as X/Y displacement or distance from one of following positions:

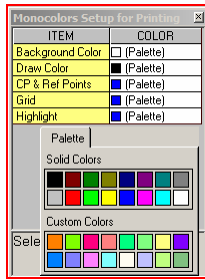
Centre - the centre of coordinate system $X=0$, $Y=0$

Board Datum - lower left corner of an imaginary rectangle that frames all imported elements.

Reference Point – user defined reference point. Pressing keys **Shift P** sets/resets reference point at current cursor position. Combination **Shift G** snaps reference point to grid coordinates closest to cursor position.

Colors

The image displayed on the screen may be turned to monochrome with selected background color. This feature enables printouts on printers that cannot handle colors. Clicking on this menu opens dialog where mono-colors for background and different object categories may be selected:



Lock Toolbars

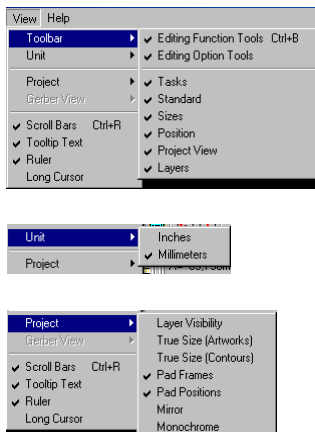
All toolbars may be relocated to desired location on the screen or docked within program workspace. To prevent unintentional relocation or un-docking, the toolbars may be locked in current positions.

Display copper remove areas as filled polygons

Switches mode of displaying polygons that define copper remove area from single yellow lines to filled yellow polygons. Filled polygons are displayed as background to other elements of currently edited or visible layer.

View Menu

Picture below presents **View** menu with its sub-menus. Most of the functions accessible from this menu can also be invoked by tool buttons in various toolbars.



Toolbar

Opens sub-menu for setting visibility of the toolbars.

Editing Function Tools toolbar groups all tools for editing graphic elements in currently selected layer.

Editing Option Tools toolbar changes depending on selected editing tool. For example, **Relocate Graphic Element** tool has an option to rotate element while relocating. Options for another tool, **Create Graphical Elements**, specify what kind of graphic element – line, rectangle or circle is to be created.

Tasks tools invoke major functions of the program – importing layers, contouring.

Standard toolbar contains tools for display control like **Zoom In** and **Out**, **Redraw**, raster and snap function settings as well as tools for secondary tasks like setting parameters for contouring; design rules check and access to machining tools database.

Sizes toolbar contains set up tools for selecting line widths for created graphical elements, text fonts and text sizes.

Position toolbar displays current cursor location in form selected in **Preferences** menu.

Project View toolbar contains tools controlling the way Project Database is presented on the **Graphic Screen**.

Layers toolbar is used to select on of layers for editing as well as for adding and removing layers from the project.

Unit

Opens sub-menu for selecting current unit of measure:

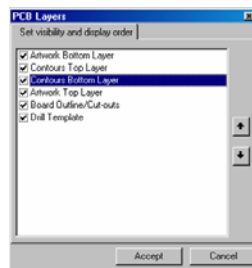
Inches - coordinates and sizes are expressed in inches. All values pertaining to dimensions or coordinates that are entered by the user are interpreted as in inches.

Millimeters - coordinates and sizes are expressed in metric units. All values pertaining to dimension or coordinates that are entered by the user are interpreted as in millimeters.

Project

Opens sub-menu that functionally duplicates toolbar **Project View**

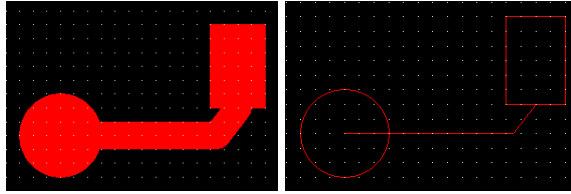
Layer Visibility opens a dialog that allows selection of visible layers as well as top to bottom sequence in which selected layers are displayed:



Checked layers are set as visible. Up and Down Arrow buttons allow moving highlighted layer one-step up or down in display sequence order. Visibility of the layer that is currently selected for editing cannot be switched off. Its place in display sequence order may be changed.

True size (Artworks)

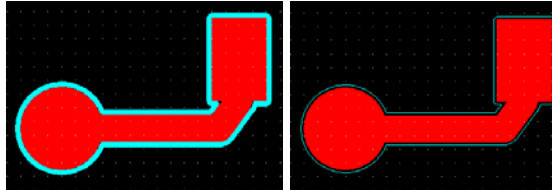
Toggles display of graphic elements on artwork layers between true size and centerline formats:



Filled circle, rectangle and lines displayed in true size and centerline formats.

True size (Contours)

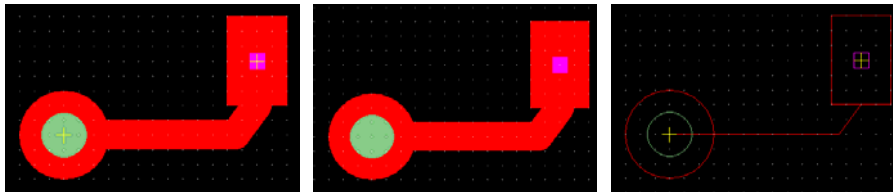
Same as above but for contours (calculated toolpaths):



Contours displayed in true size (line width equal to the diameter of corresponding milling tool) and in centerline formats (contour lines show exact trajectory of the tip of the milling tool).

Pad Position and Pad Frames

These two settings in combination with **True size (Artworks)** control display of pad positions on artwork layers and holes on Drill Template:



Left: Hole and pad position with True Size, Pad Position and Pad Frames ON

Middle: with True Size, Pad Position ON and Pad Frames OFF

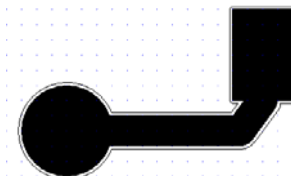
Right: with True Size OFF, Pad Position and Pad Frames ON

Mirror

Displays mirrored (around Y-axis) view of all visible layers. Must not be confused with functions **Mirror Layer (X)** and **(Y)** that recalculate coordinates of all elements in the layer selected for editing.

Monochrome

Sets view on the **Graphic Screen** to monochrome for printing on black & white printers:



Artwork layer and its contours in monochrome view.

Scroll Bars

Switches scroll bars for vertical and horizontal panning of the **Graphic Screen** ON and OFF.

Tooltip Text

Switches display of tooltip texts ON and OFF.

Ruler

Switches the ruler ON and OFF

Long Cursor

Long cursor is visible only where graphical elements are relocated or stretched. Its purpose is to help in aligning elements in reference to other elements or the ruler:



Long cursor, ruler and horizontal scroll bar.

3.4. Task and Control Tools

Task and control toolbars are by default docked at the top of program workspace. Tool buttons in these toolbars are duplicated in **Main Menu**. Visibility of these toolbar may be therefore switched off in cases when larger **Graphic Screen** is needed.

Tasks Toolbar

Tools from this toolbar are duplicated in **File** and **Tasks** menus.



New project

Discards current project, clears all layers and contours and creates empty Project Database.



Import artwork and drilling

Opens file selection dialog where import file (Gerber or Excellon format) may be selected. After selection, **Import Parameters** dialog pop-ups from where actual import is initiated. More details in chapter **Imports**.



Contour all layers

Orders generation of milling contour (toolpaths) for all artwork layers. More details in chapter **Contouring**.

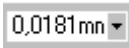


Fabricate PCB

Results of contouring, toolpaths for cutting board outlines and drilling data are stored in a disk file in one of the formats accepted by milling machines. In case of Colinbus machines, this format is binary, stored in files with name extension .BFF.

Standard Toolbar

Some of the tools in this toolbar are duplicated in **Tasks** menu



Precision

Precision means the size of one pixel on the graphic screen (minimal mouse movement) expressed in project database units. For example, precision value of 0.100 mm means that one pixel represents step of 0.1 mm. It also means, the cursor moves with this precision and graphic element may be relocated or stretched in steps of 0.1 millimeters. Required precision may be directly entered into the box or selected from drop-down list that pops up when this tool is clicked.



Zoom In and Zoom Out

Zoom In increases display precision by set factor (see **Preferences** menu) and **Zoom Out** decreases it.



Block Zoom

Adjust precision and scroll to display a portion of visible graphics defined by bounding rectangle.



- Current
- Center
- Project

Redraw mode

Current – refresh graphic screen with current precision and coordinates.

Center – redraw graphics with current precision, centered on X=0, Y=0 coordinates –center of coordinate system.

Project – redraw graphics with such precision and coordinates that the entire project is visible. This mode is set by default when the program starts.



Fabrication parameters setup

Opens pop-up dialog where parameters for contouring and other fabrication may be set. More details in chapter **Preparation for generating toolpaths**.



Mirror Layer (Y) and Mirror Layer (X)

Symmetrically rotates all elements in currently edited layer around respectively Y and X-axis. Assumed symmetry axis is in the centre of imaginary rectangle framing all elements in the layer.



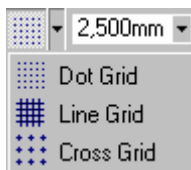
Tool Setup

Opens access to tools database where milling and cutting tools may be selected and assigned to tool categories. More details in chapter **Preparation for generating toolpaths**.



Design Rules Check – tool clearances

Invokes routine that checks whether insulation may be ensure with currently selected contouring tools. More details in chapter **Preparation for generating toolpaths**.



Raster ON/OFF and Raster Interval

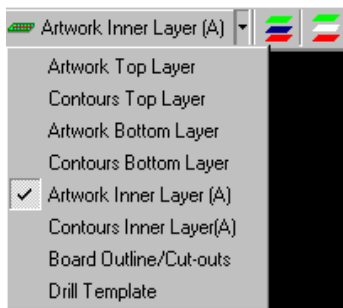
Combined tool for selecting type of visible raster, setting it ON and OFF and for setting raster interval. Value of required raster interval maybe directly entered in the box or select from drop-down menu.



Orthogonal snap and Angular snap

Combined tool for setting snap function ON and OFF and for defining snap step value. Value of required snap step maybe directly entered in the box or selected from drop-down menu. When an element is relocated or stretched, current cursor coordinates are rounded to closest multiplication of snap value. Same principle applies when dragging bounding rectangles. Angular snap takes effect when rotating elements or block of elements.

Layers Toolbar (with Add Inner Layer and Remove Inner Layer buttons)



Selects one of layers as active for editing. Only one layer may be edited at the time. Visibility of other layers may be switched ON and OFF but visibility of currently edited layer cannot be switched OFF. There are no restrictions regarding layer display order.

Professional version allows adding and removing up to six inner layers. In order to remove a layer, this layer has to be first selected as active. Top and bottom artwork layers cannot be removed.

This toolbar is duplicated as **PCB Layers** entry in **Main Menu**.

Project View Toolbar



Tools in this toolbar are duplication of functions found in various sub-menus of Main Menu entry **View**. Chapter **View Menu** contains detailed descriptions of each function.



Layer Visibility



True Size (Artworks)



True Size (Contours)



Pad Positions



Pad Frames



Mirror



Monochrome

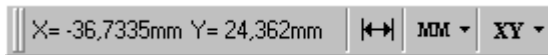


Ruler



Tool tip

Position Toolbar



Displays current coordinates of cursor position or distance from selected reference location.



Select mode for displaying cursor position

(For details, see **Preferences Menu**)



Select current unit of measure

(For details, see **View Menu**)

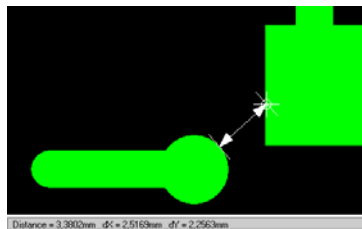


Measure distance (Shift M)

Switches function for measuring distance from selected graphical element. This function is especially useful for measuring gaps between edges of elements.

Procedure:

- Select this tool or key combination **Shift M**
- Click with left mouse button on graphic element to select
- Message box displays distance between cursor position and the edge of selected element:



- Measure function is cancelled by pressing **Esc** key or clicking **Esc** button

Sizes Toolbar



Setup tools used in connection with adding new graphical elements to artwork layers and contours.



Line with

Sets current line width for lines, rectangles (no filled), circles (not filled) and arcs. Line width value may be directly entered into the box or selected from drop-down menu.



Sets current font and size for text. Values are selected from drop-down menus.

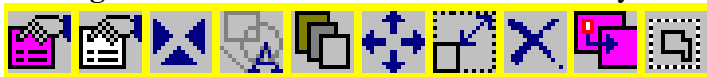
3.5. Editing Tools and Options

Set of **Editing Tools** varies depending on type of currently active layer:

Editing Tools for artwork layers



Editing Tools for Board Outline and Cutouts layer



Editing Tools for Drill Template layer



Editing Tools for contours layers



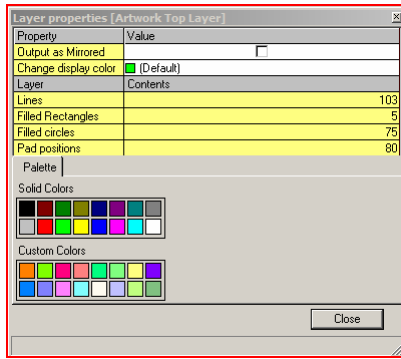
It applies to Options too. Options are separate for each editing tool and specify various modes and variants of editing operations. **Editing Options** as well as **Editing Tools** may be immediate or modal.

Options may be switched ON or OFF while editing function is in progress. Most practical way is to use keyboard functions keys (F1 to F9). F1 switches ON the first option counting from the top in the **Options** toolbar, F2 second from the top etc.

Layer Properties Tool



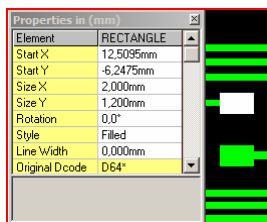
Immediate tool that displays pop-up dialog **Layer Properties** where user may assign display color to currently active layer and examine its contents.



Graphic Element Properties Tool



Modal tool that opens pop-up (as below) allowing examining and changing properties of selected graphic element or elements.

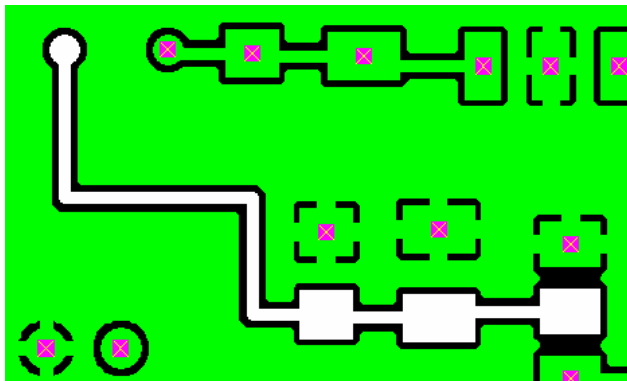


Option only for artwork layers:



Show net path (F1)

Highlights graphic elements that constitute uninterrupted path starting from selected element. See picture below:

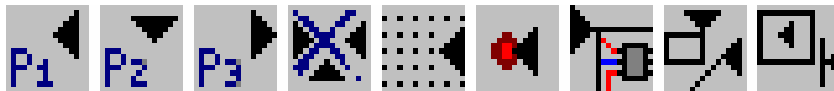


Set Contact Point Tool



Allows setting up to three (P1, P2 and P3) points on the **Graphic Screen**. Positions of contact points are marked with triangles, respectively in three different rotations. Options specify which of the points should be placed and in which location. If none of the options specifying the locations is set then the point will be placed at current position. How these points are utilized depends on other editing function. For example relocated graphic elements snaps to P1 position if this point is set. Contact points may be also used to specify predefined exact positions and sizes of created elements. For example, when P1 and P2 are set then line will be created starting at P1 and ending at P2. See detailed explanation in chapter **Editing function**.

Options available for this tool are the same for each type of layer:



Options reference:



Place point P1 (F1)



Place point P2 (F1)



Place point P3 (F3)



Clear all points (F4) – immediate option



Snap point to grid (F5)

Point will be placed at grid position nearest to current cursor position



Set on hole/pad position (F6)

Point will be placed at position of selected hole (Drill Template) or at pad position marker (artwork layers)



Set on board corner (F7)

Point will be placed at the corner of imaginary rectangle framing all layers that is closest to current cursor position



Set at element boundary (F8)

Point will be set at the boundary of selected graphical element (end or start of a line or arc, corner of a rectangle, perimeter of a circle)



Set at element centre (F9)

Point will be set at the centre of selected graphical element

Create Graphical Element Tool



Modal tool for creating graphical elements as specified by options. If none of contact point is selected then the procedure for creating elements is following:

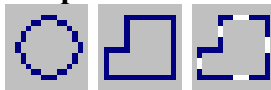
- Select this tool
- Select type of element to create by setting corresponding option
- Click with left mouse button on the graphic screen. Element is created and can be relocated. Option toolbar shows options for Relocate Element(s) Tool
- Click with left mouse button on the graphic screen when element is placed at desired location. Element may now be stretched (dragged) for desired size. Option toolbar shows options for Stretch Element Tool. Click with left mouse button to finish dragging

Options available for this tool vary depending of currently edited layer:

Graphic Element Create Options for artwork layers:



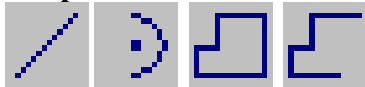
Graphic Element Create Options for Board Outline/Cutouts layer:



Graphic Element Create Options for Drill Template:



Graphic Element Create Options for contours layers



Options reference:**Create Line****Create Rectangle****Create Circle**

This option will produce circular outline (for board and cutouts) if applied when Board Outline/Cutouts layer is edited.

**Create Arc****Create polygon**

Polygons on artwork layers are filled, on Board Outline/Cutouts and Contours layers unfilled. Polygons are created by dragging from vertex to vertex. Minimal three vertexes must be inserted for a valid polygon.

**Create poly-line**

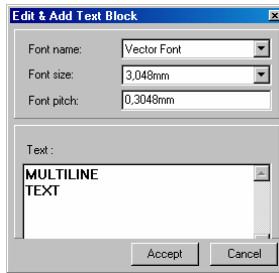
Poly-lines are created by dragging from vertex to vertex. Minimal two vertexes must be inserted for valid poly-line.

**Toggle to 'Breakout' Tab**

Option to insert/remove 'Break-out' Tab inside selected segment of outline or cutouts polygons. 'Break-out' tabbed segments will be skipped while cutting.

**Created drilled hole****Create text – immediate option**

Opens dialog for entering multi-line text blocks. User may specify type of font, size of letters and pitch between lines:



Filled elements

With this option set every created rectangle or circle will be filled.

Repeat or Transfer Elements Tool



Copies selected element or block of elements. Copied elements may be subsequently relocated to desired location. Options allow repeating selected elements specified number of steps.

Options available for this tool vary depending of currently edited layer:

Repeat Elements Options for artworks, contours and Drill Template layers:



Repeat or Transfer Elements Options for Board Outline/Cutouts layer:



Options reference:



Step repeats (F1)

Repeat element specified number of steps and X, Y displacement.



Repeat and rotate (F2)

Repeat and rotate element specified number of angular steps. Coordinate of contact P1 define rotation centre.



Transfer path of elements (lines) to Board Outline layer (F3)

Transfers path of lines starting from selected line (in any layer) and converts it to polygon on Board Outline/Cutouts layer. The path must form closed shape. More details in chapter **Editing Functions**.

Relocate Element(s) Tool



Relocates selected element or block of elements. Options that may be switched ON while element or block is relocated specify additional operations.

Options available for this tool take effect depending on what kind of element is relocated and whether relocation applies to a single element or a block.

Relocate Element(s) Options:



Options reference:



Rotate by 90 degrees (F1)

Each time this option is selected by click or F1 the relocated element or block is rotated by 90 degrees.



Free rotate (F2)

Allows rotating element or block by step specified in Angular Snap box.



Scale element in block (F3)

Elements in block will be scaled by entered scale factor.



Mirror block Y (F4)

Elements in block will be symmetrically rotated around vertical axis in the centre of bounding rectangle.



Mirror block X (F5)

Elements in block will be symmetrically rotated around horizontal axis in the centre of bounding rectangle.



Relocate segment vertically (F6)

Applies to polygons and poly-lines. Locks relocation of selected segment in vertical direction only.



Relocate segment horizontally (F7)

Applies to polygons and poly-lines. Locks relocation of selected segment in horizontal direction only.



Free segment relocation (F8)

Applies to polygons and poly-lines. Allows segment relocation in any direction.

Stretch Element Tool



This tool is used for various transformation of selected element. Options specify mode of transformation. Some options should be preset before selection of the element.

Stretch Element Options:



Options reference:



Stretch arc radius (F1)

Applies to arc elements. Arc is transformed by changing radius calculated as distance between cursor position and arc centre



Stretch arc by end-point (F2)

Applies to arc elements. Arc is transformed by dragging selected end-point.

Note:

If neither of above options is set, then the arc is transformed to pass through three points specified by fixed positions of both ends and changeable current cursor position.



Add vertex forwards (F3)

Applies to polygons and poly-lines. As long as set, allows inserting vertexes forwards from selected vertex. This mode is default when polygon or poly-line is created.



Add vertex backwards (F4)

Applies to polygons and poly-lines. As long as set, allows inserting vertexes backwards from selected vertex. This mode may be also set while polygon or poly-line is created.



Delete vertex (F5)

Applies to polygons and poly-lines. Removes currently inserted vertex. This mode may be also set while polygon or poly-line is created.



Finish insertion (F6)

Applies to polygons and poly-lines. Removes last inserted vertex and finishes transformation. This option completes creation of polygons or poly-lines.



Insert round miter (F7)

Applies to polygons and poly-lines. Inserts arc at selected vertex between two line segments. Miter radius may be entered in pop-up box or selected from combined drop-down list.



Remove miter (F8)

Applies to polygons and poly-lines. Replaces selected arc miter with vertex at intersection of adjoining line segments.



Remove arc (F9)

Applies to polygons and poly-lines. Replaces selected arc miter with straight segment.

Note:

Polygon transformation will not be allowed if it resulted in intersecting segments.

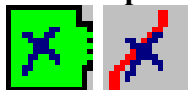
Delete Tool



Deletes selected element or block of elements. Before element or block is deleted, the user is asked for confirmation.

Options available for this tool vary depending of currently edited layer:

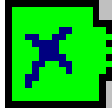
Delete Options for artwork layers:



Delete Options for Board Outline/Cutouts, Drill Template and contours layers:



Options reference:



Delete all elements (F1)

Clear currently edited layer.



Delete netpath (F2)

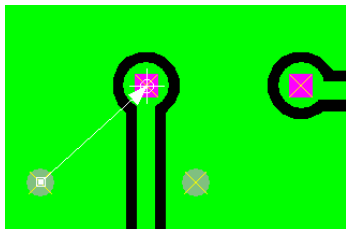
Deletes all elements that form uninterrupted path starting from selected element.

Align layers Tool



Aligns currently edited layer with other layer. This operation requires selection of alignment points (one on each layer) that after alignment should have the same coordinates. Options specify mode for selection of alignment point in other layer. Alignment procedure is following:

- Select this tool.
- Select element in currently editing layer. Position of this element is taken as first alignment point. Alignment vector is visible on the graphic screen and follows movement of the cursor.
- Select alignment point in other layer by clicking on element on this layer. Coordinate of this element (centre or end-point, depending on option) is taken as second alignment point
- In result of this function, all elements in aligned layer will be relocated by vector 'first alignment point -> second alignment point'. See picture below presenting alignment of Drill Template layer with artwork layer using pad position on artwork layer as the second alignment point:



Alignment Options:



Options reference:**Align to snap grid (F1)**

Second alignment point at snap grid coordinates nearest to current cursor position.

**Align to pad position or hole (F2)**

Second alignment point at selected hole (Drill Template) or at pad position marker (artwork layers)

**Align to end-point of graphic element (F3)**

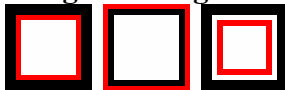
Second alignment point at selected at the boundary of selected graphic element (end or start of a line or arc, corner of a rectangle, perimeter of a circle).

**Align to centre point of graphic element (F4)**

Second alignment point at the centre of selected graphic element

Assign Milling/Cutting/Copper Remove Mode Tool

This tool groups several functions. Each function is specified by an option. Set of available options vary depending on currently edited layer. Not applicable to Drill Template and contours layers

Assign Milling/Copper Remove Mode Options for artwork layers:**Assign Cutting Mode Options for Board Outline/Cutouts layer:****Options reference:****'Mill as line' (F1)**

Clicking on an element when this option is set toggles its property from 'contoured' to 'milled as line'. Status 'milled as line' is signaled by this element being displayed in with dashed line. 'Milled as line' elements will not be

contoured but milled with tool assigned for Normal line milling. Toolpaths for 'Mill as line' elements are grouped in separate section of the machining program.



'Mill as line text' (F1)

Clicking on an element when this option is set toggles its property from 'contoured' to 'milled as line text'. Status "milled as line text" is signaled by this element being displayed in with dotted line. "Milled as line text" elements will not be contoured but milled with tool assigned for Normal line milling. Toolpaths for 'Mill as line text' elements are grouped in separate section of the machining program.



Set 'Remove all copper' mode (F3)

Immediate option that sets 'Remove all copper' mode for currently edited layer. During contouring, the program will calculate toolpaths for removing copper from all spaces on the layer that there is no copper defined in the artwork image.



Define rectangular area for copper removal (F4)

Allows dragging bounding rectangle that defines an area on the layer where copper removal should take place. During contouring, the program will calculate toolpaths for removing copper from all spaces within the rectangle that there is no copper defined in the artwork image.



Define circular area for copper removal (F5)

Allows dragging bounding circle that defines an area on the layer where copper removal should take place. During contouring, the program will calculate toolpaths for removing copper from all spaces within the circle that there is no copper defined in the artwork image.



Define polygonal area for copper removal (F6)

Allows dragging bounding polygon that defines an area on the layer where copper removal should take place. During contouring, the program will calculate toolpaths for removing copper from all spaces within the polygon that there is no copper defined in the artwork image.

Note:

All types of copper removal areas (circular, rectangular or polygonal) are polygons added to a layer and displayed with dashed line in yellow color. More that one area may be defined but they must not intersect. 'Remove all copper mode' automatically substitutes all previously defined areas with a single polygon that frames the whole board. All editing functions applicable to

polygons imported or added to artworks apply also to polygons defining copper removal areas.



Cancel copper removal area

Allows deleting selected copper removal area by clicking on one of the segments in bounding polygon.



Assign 'Cut on outline' mode

Assigns 'Cut on outline' mode to selected polygon on Board Outline/ Cutouts layer. The tip of cutting tool will move exactly in line with the segments of the polygon.



Assign 'Cut outside outline' mode

Assigns 'Cut outside outline' mode to selected polygon on Board Outline/ Cutouts layer. The tip of cutting tool will move on 1/2-diameter distance outside of the polygon.

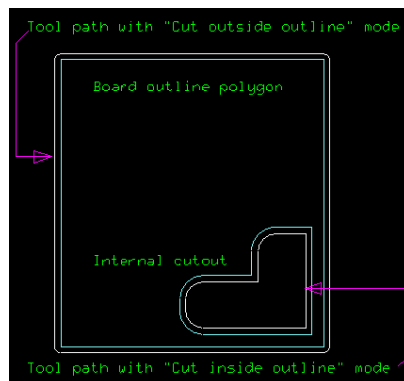


Assign 'Cut inside outline' mode

Assigns 'Cut inside outline' mode to selected polygon on Board Outline/ Cutouts layer. The tip of cutting tool will move on 1/2-diameter distance inside polygon.

Note:

Board outline and internal cutouts are defined as polygons that are directly converted to toolpath calculated according to one of above selected modes. See example below:



Contour Layer Tool



Immediate tool that starts contouring currently edited artwork layer. Should not be confused with **Contour all layers** task tool.

The purpose of this tool is to allow contouring of a single layer with individually assigned parameters.

3.6. Control of Graphic Screen and cursor movement

Numerous controls are provided for adjusting display on the **Graphic Screen**. Some of controls are grouped as tools in the **Standard** toolbar (**Precision**, **Zoom In**, **Zoom Out**, **Block Zoom** and **Redraw Mode**). See chapter **Standard Toolbar**. Same functions may be executed by pressing following key combinations on the keyboard:

+ or **Shift U** to **Zoom In**

- or **Shift D** to **Zoom Out**

Shift V to refresh graphic screen with current precision and coordinates

Shift B to **Block Zoom**

Additionally:

Home Centers display at coordinates of lower left corner of the imaginary rectangle that frames all graphical elements in the project.

Ctrl Home Redraws graphics with such precision and coordinates that the entire project is visible

Shift SPACE Panning – centers display at coordinates of current cursor position. Same function may be invoked by pressing middle mouse button

Mouse wheel Turning mouse wheel forward is equivalent to **Zoom Out**. Turning backward is equivalent to **Zoom In**.

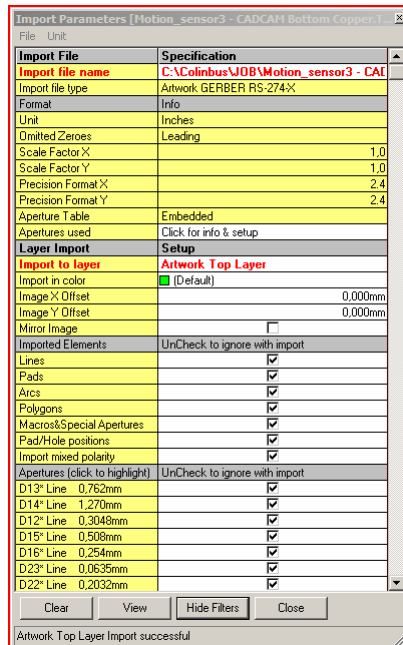
The contents of the **Graphic Screen** may be scrolled horizontally and vertically with help of scroll bars. Visibility of scroll bars may be switched ON and OFF. See **View Menu**.

Movement of the crosshair cursor over the **Graphic Screen** is normally controlled by the mouse. In cases when greater precision of cursor relocation is required, keyboard “arrow” keys (← ↑ → ↓) may be used. When any of these keys is pressed the cursor will be relocated in corresponding direction by single step specified in **Precision** box in the **Standard** toolbar

Pressing key combination **Shift Enter** while cursor hovers over the **Graphic Screen** is equivalent to clicking left mouse button.

3.7. Pop-up dialogs

Parameters for imports, contouring and machining parameters, layer and graphic properties are set through various pop-up dialogs. These dialogs have usually the form as presented below:



Parameters are listed in tabular form. Cells in the left hand column contain names of parameters. Current settings and values are displayed in the right hand column.

Cells displayed with yellow background present parameters or other additional information that cannot be altered. Grey background indicates rows that separate various categories of parameters.

Settings and values displayed with white background in right hand column cells may be altered. Clicking on these cells allows changing the current setting of the parameter. How actually the setting is changed depends on the type of cell which is immediately shown when the cell is clicked.

Text Box cell

Opens text box where value of parameter may be entered from the keyboard.

Drop-Down List cell

Opens drop list where one of possible settings may be selected.

Combined Text Box / Drop-Down List cell

Parameter values may be either entered from the keyboard or selected from the drop-down list.

Check Box cell

Clicking on this cell checks and un-checks the box which is equivalent to setting corresponding parameter ON and OFF

Button Cell

Clicking on this cell reveals small button, which for example may be clicked to open another pop-up dialog.

Most of the pop-up dialogs are resizable and the program remembers their sizes and positions after closing. Tables are scrollable up and down with the help of scroll bars and mouse wheel.


4. Imports

The program starts its operation from displaying file selection dialog focused on the folder from where the last import took place (\JOB is default folder in the first run of the program).

Type of the file (Gerber, Excellon or saved Project Database) is automatically recognized upon selection. Since there are no uniform naming conventions regarding Gerber and Excellon files, the program will open files with any name extension.

If Project Database (.CBJ name extension) file is selected then no import takes place and the project is loaded and displayed.

If opened file does not match one of accepted import formats, the program displays its contents in the Notepad along with error message.

During program run, the import may be initiated by clicking on tool button  **Import artwork and drilling.**

4.1. File and data formats for importing artworks, board outline and cutouts

Gerber data is a simple, generic means of transferring printed circuit board information to a wide variety of devices that convert the electronic PCB data to artwork produced by a photoplotter. Virtually every PCB CAD system generates Gerber data because all photoplotters read it. It is a software structure consisting of X, Y coordinates supplemented by commands that define where the PCB image starts, what shape it will take, and where it ends. In addition to the coordinates, Gerber data contains aperture information, which defines the shapes and sizes of lines, holes, and other features.

Gerber Format, which is the format in which Gerber data is expressed, actually is a family of data formats that are subsets of EIA Standard RS-274D. *Extended Gerber Format*, which is also called *RS-274X*, provides enhancements that handle polygon fill codes, positive/negative image compositing, custom apertures and other features. RS-274X also encapsulates the aperture list in the header of the Gerber data file and therefore allows files to pass from one system to another without the need to re-input the aperture table. RS-274X produces a variety of Gerber data called *X data*.

RS-274X is a *superset* of the EIA Standard RS-274D format. RS-274X supports some of the parameter data codes (G codes) and aperture codes (D codes) contained in RS-274D, as well as codes referred to as *mass parameters*. Mass parameters are plot parameters that define characteristics that can affect an entire plot, or only specific parts of the plot, called *layers*. Mass parameters extend the capabilities of Gerber Format. Their presence makes the Gerber data *X data*.

The program automatically recognizes whether selected Gerber file is in Standard RS-274D or in Extended RS274X format. Files in both formats may be imported.

Formatting parameters for Gerber files

Following information needs to be provided for proper interpretation of Gerber files:

Aperture Table

Assigns sizes (line widths, pad diameters and shapes) to aperture codes (D-Codes) in the file.

Unit

Specifies units (metric or imperial) in which coordinates are expressed in the file.

Precision format

Specifies how many integer and how many decimal places to expect in the coordinate data. For example, precision 2.3 specifies two integer and three decimal places. A maximum of four integer and four decimal places may be specified (9999.9999). Different precision format may be defined for the X- and Y-axes.

Zero omission

Zero omission compresses data by omitting either leading or trailing zeroes from coordinate values. Any given string of digits may be interpreted very differently depending on the zero omission specification. Precision format also affects how zero omission is interpreted.

Leading zero omission eliminates all zeroes that precede non-zero digits but retains following zeroes. For example, with 2.3 precision format, 15 would be interpreted as 0.015.

Trailing zero omission eliminates all zeroes following non-zero digits but retains preceding zeroes. For example, with 2.3 precision format, 15 would be interpreted as 15.000.

Scale factor

Specifies coefficient by which coordinates in the file should be multiplied. Different scale factors may be defined for the X- and Y-axes. Default X and Y scale factor is 1.0.

All this information is embedded in Extended RS-274X files and artworks in this format may be imported directly. Import Standard RS-274D files is more complicated because all formatting information must manually be entered to the program.

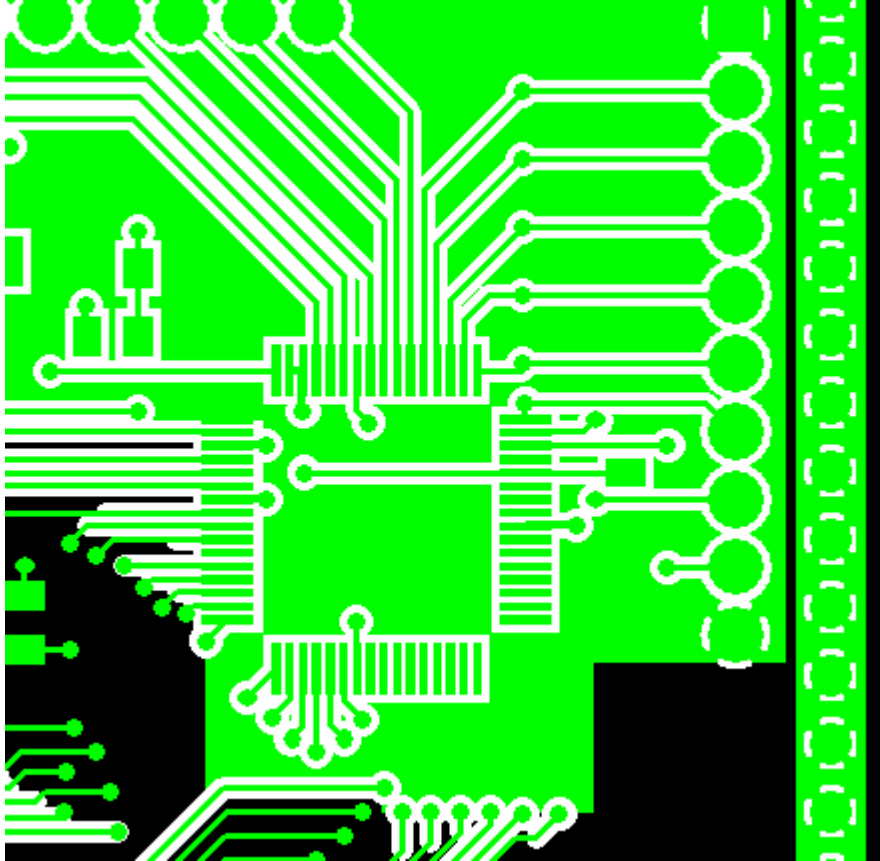
Mixed polarity artworks

PCB CAD systems use various methods for creating copper pour areas on the artworks. By far most dominant (and time consuming with contouring) method is so called stroke fill, where areas are filled by large amount of thin lines. Some systems take advantage of capabilities of Extended RS-274X format that allows generating mixed polarity plots. Single PCB layer artwork is the result of plotting series of superimposed images in positive and negative polarity.

Filled polygons defining copper pour areas are plotted first with positive polarity. Next superimposed image is plotted with negative polarity. Pads and tracks with sizes increased by air gap that should insulate them from surrounding copper are plotted in this stage as well as other shapes that will relieve copper on the board. In the third step, the polarity is reversed again to positive and plot of pads and tracks in normal sizes is superimposed on previous

images. Photographically, the resulting image contains black copper pour polygons in the background and black pads and tracks within transparent air gaps.

This program can process artwork images generated according to this method. See example of imported mixed polarity image below:



Elements drawn in white color represent air gaps plotted with negative polarity. Copper pour area is represented in green as well as tracks and pads within air gaps.

4.2. File and data formats for importing drilling data

Most of PCB CAD packages outputs drilling data in Excellon format. Basic drilling program contains commands for tool (drill bit) change and coordinates of the holes.

Formatting parameters for Excellon files

Following information needs to be provided for proper interpretation of Excellon format files

Tool Size Table

Assigns diameters to tool code (T01, T02 ... T0n)

Unit

Specifies units (metric or imperial) in which coordinates are expressed in the file.

Precision format

As in Gerber files. According to specification of Excellon format following precisions are used:

- For imperial units: 2.4
- For metric units: 3.2, 4.2 and 3.3

Zero omission

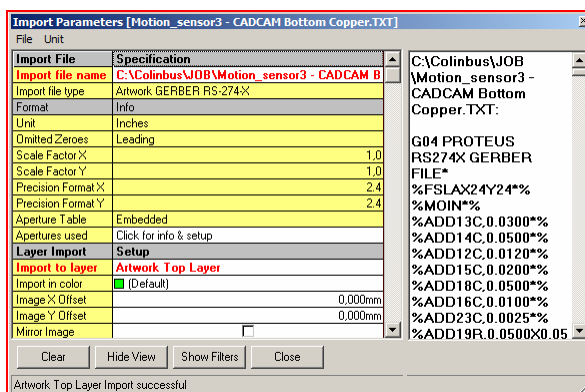
As in Gerber files but Excellon format follows opposite convention and specifies instead which zeros are **not** omitted. Trailing Zeros (TZ) parameter means that coordinates contain *trailing zeros* and that *leading zeros* are omitted. Parameter Leading zeros (LZ) means that coordinates contain *leading zeros* and that *trailing zeros* are omitted.

Above information is absent in basic drill programs that are known as Excellon Format One and have to be manually entered. This is not needed when importing drill programs in more advanced Excellon Format Two where formatting information is embedded in the file in similar fashion as in Gerber RS-274X. Therefore, Format Two files may be imported directly.

Experience shows that PCB CAD systems not always follow specification and output data in inches with other precision format than 2.4. Sometimes also, the information necessary to accept a file as Format Two is partially missing or invalid. In all such cases, the file is trade as Format One and missing parameters entered manually.

4.3. Import Parameters pop-up dialog

After selecting file from **Select File for Import** dialog, the program opens **Import Parameters** pop-up dialog where further import parameters are set. Import procedure and number of parameters to set differs depending on detected file type and format.



Picture above shows **Import Parameter** dialog with opened viewing field presenting contents of Gerber file imported to Artwork Top Layer.

Several controls on this dialog are common for all imports. This are:

File Menu

Opens sub-menu with two entries:

Open for Import

Usually more than one file is imported. This menu allows selecting another file in the same import session.

Open for View

Opens any file for viewing only. This feature is useful for viewing so called report files that are often generated along with Gerber and Excellon files. Report files usually contain information that may be required for setting of formatting parameters.

Unit Menu

Opens sub menu that allows selecting unit of measure that is most suitable for pasting parameters values (for example, drill tools diameters) from ASCII report files. This setting is valid only for this dialog.

Inches

Values of parameters will be displayed and may be entered in imperial units.

Millimeters

Values of parameters will be displayed and may be entered in metric units.

Buttons:**Clear**

Clear contents of recently imported layer.

View and **Hide View**

A toggle - opens and closes viewing field for presenting contents of the file selected for viewing.

Show Filters and **Hide Filters**

A toggle – opens and closes section of the parameters table that contains import filters. This button is visible after import layer has been selected.

Close

Closes current import session and returns control to **Graphic Screen**. Last imported layer is selected as active and on top of display sequence order.

4.4. Importing artworks from Gerber files

Apart from opening Gerber file (**Select File for Import** dialog) it is necessary to select the layers where the artwork from this file should be imported. If user wants to import artworks to other layers than those that were created by default (Top and Bottom) then an inner layer or layers has to be added to the project prior to importing session (see chapter **Layer Toolbar**) The import of opened file is executed directly upon selection of a layer from drop-down list in the right hand cell for **Import to layer** parameter. Immediately after import (and re-import caused by change of parameters), the layer is displayed on the **Graphic Screen**. **Graphic Screen** is at all time accessible and display may be zoomed in and out, scrolled and panned.

Note:

General principle valid for all types of imports. Whenever any parameter in Import Parameters dialog is changed, the opened file is automatically re-imported applying all currently set parameters values.

Importing artworks from files in Extended RS-274X format

Since all formatting information is embedded in RS-274X format files the only parameter to set is **Import to layer**. Usage of other parameters is optional.

Import Parameters for Gerber Extended RS-274X format

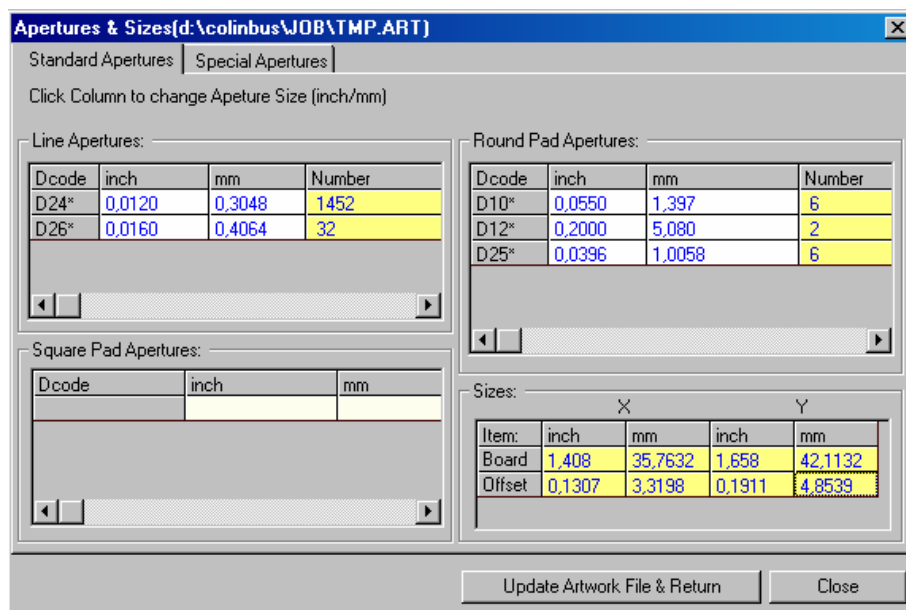
Parameter name	Type	Settings and values
Import File Name	Button	Path to currently opened import file. Click to open next file.
Import File Type	Fixed	Displays detected format.
Unit	Fixed	Unit of measure specified in the import file.
Omitted Zeros	Fixed	Type of zero omission specified in the import file.
Scale Factor X	Fixed	Scale factor for X-axis specified in the import file.
Scale Factor Y	Fixed	Scale factor for Y-axis specified in the import file.
Precision Format X	Fixed	Precision format for X-axis coordinates specified in the import file.
Precision Format Y	Fixed	Precision format for Y-axis coordinates specified in the import file.
Aperture Table	Fixed	Embedded - meaning aperture table is embedded in the import file.
Apertures Used	Button	Opens Apertures & Sizes dialog presenting all apertures used in the import file. Apertures & Sizes dialog allows changing of aperture sizes and modifiers.
Import to layer	Drop-Down List	For selecting import layer
Reconstruct polygons for Outline/Cutouts	Check Box	Parameter is visible only when importing to Board Outline/Cutouts layer. If checked causes automatic reconstruction of polygons from lines found in imported image.
Import in color	Drop-Down List	Assigns display color for imported layer. Offers two options: Selected from palette or Use default color

Image X Offset	Text Box	Imported image will be shifted horizontally by offset entered in this box.
Image Y Offset	Text Box	Imported image will be shifted vertically by offset entered in this box.
Mirror Image	Check Box	If box is checked then imported image will be symmetrically rotated around Y-axis.

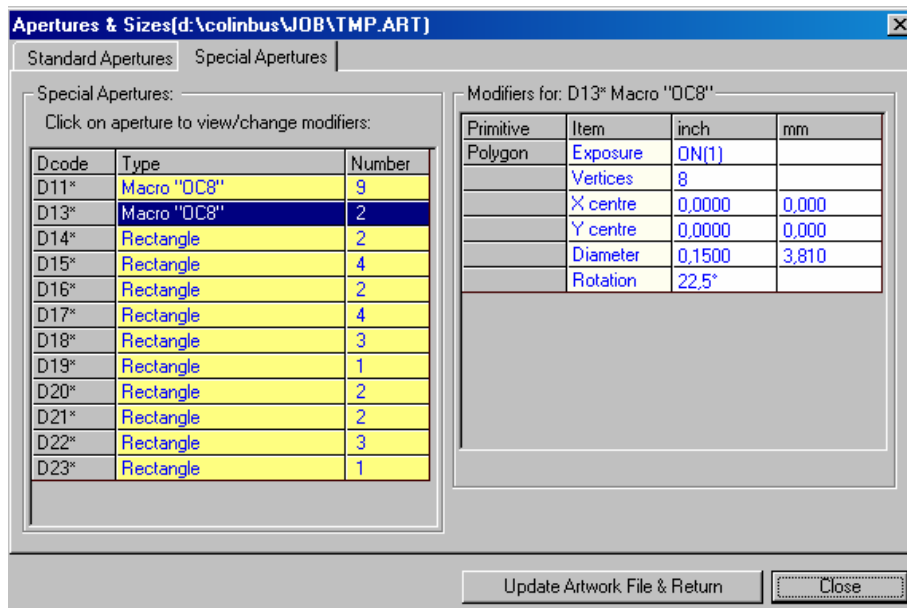
The type of import file and formatting parameters are in this case only listed as additional information and cannot be altered.

Aperture table is embedded in the import file so normally it is not necessary to do any changes. Nevertheless, aperture sizes for standard apertures as well as modifiers for special apertures and macros may be changed in the dialog **Apertures & Sizes**. The same dialog may be called in order to examine aperture details without entering any changes.

Aperture & Sizes dialog is called by pressing button for parameter *Apertures used*



Aperture & Sizes dialog for setting standard apertures. Aperture sizes may be entered either to “inch” columns in inches or to “mm” columns in millimeters – whatever more convenient.



Aperture & Sizes dialog for changing modifiers for special apertures and aperture macros. Clicking on aperture in the left hand table opens table that lists current modifiers values. New values may be entered either to “inch” column in inches or to “mm” column in millimeters – whatever more convenient. Changing modifiers requires advanced familiarity with Gerber Extended RS-274X format.

Clicking on button **Update Artwork File & Return** will effect in re-import of the artwork image using updated aperture sizes. This function **does not** change aperture table embedded in the original import file.

Note:

When Gerber file is opened for import, the program preprocesses the image and stores it in intermediary file TMP.ART (artwork file) in order to facilitate faster re-import with new parameters. All parameter changes affect only preprocessed image in the artwork file.

Importing artworks from files in Standard RS-274D format

Formatting parameters and aperture table are obligatory information that needs to be provided for successful import of files in RS-274D format. Initially, the program uses parameters that are remembered after previous import session. Setting of formatting parameters should be done prior to assigning aperture sizes since it always causes repeated preprocessing of original import file.

Import Parameters for Gerber Standard RS-274D format

Parameter name	Type	Settings and values
Import File Name	Button	Path to currently opened import file. Click to open next file.
Import File Type	Fixed	Displays detected format.

Unit	Drop-Down List	Select Inches or Metric - unit of measure for the import file.
Omitted Zeros	Drop-Down List	Select type of zero omission for the import file
Scale Factor X	Text Box	Enter X-axis scale factor for the import file.
Scale Factor Y	Text Box	Enter Y-axis scale factor for in the import file.
Precision Format X	Drop-Down List	Select X-axis precision format for the import file.
Precision Format Y	Drop-Down List	Select Y-axis precision format for the import file.
Aperture Table	Button	Opens Aperture Table dialog where current aperture table may be selected and edited. This aperture table is used when Standard RS-274D format file is preprocessed for import.
Apertures Used	Button	Opens Apertures & Sizes dialog presenting all apertures used in the import file. Apertures & Sizes dialog allows changing of aperture sizes and modifiers. Setting made in this dialog override currently selected aperture table.
Import to layer	Drop-Down List	For selecting import layer
Reconstruct polygons for Outline/Cutouts	Check Box	Parameter is visible only when importing to Board Outline/Cutouts layer. If checked causes automatic reconstruction of polygons from lines found in imported image.
Import in color	Drop-Down List	Assigns display color for imported layer. Offers two options: Selected from palette or Use default color
Image X Offset	Text Box	Imported image will be shifted horizontally by offset entered in this box.
Image Y Offset	Text Box	Imported image will be shifted vertically by offset entered in this box.
Mirror Image	Check Box	If box is checked then imported image will be symmetrically rotated around Y-axis.

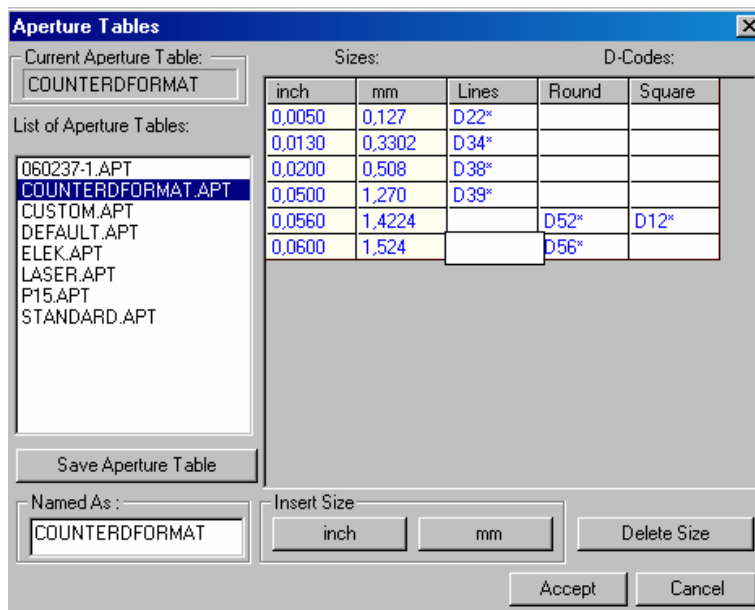
There are three ways of assigning proper sizes to apertures found in the import file:

- Selecting and eventually updating already existing external Aperture Table.

- Creating new Aperture Table (in this case, some existing Aperture Table may provide suitable template).
- Temporary assigning aperture sizes using dialog **Aperture & Sizes**.

Aperture Table, which is used in preprocessing, must contain sizes for all apertures in the import file. The program is supplied with default Aperture Table (Default.APT) that is installed in \SYS folder. It is very unlikely that apertures defined in default Aperture Table will match aperture in every import file. Same aperture codes (D-Codes) may mean different sizes in different files or necessary aperture codes may be missing in the Aperture Table.

Aperture Table may be accessed by clicking button for parameter *Aperture Table*



Current Aperture Table may be selected from the list on the left hand side of the dialog. It simultaneously opens the table for editing. Aperture Table is arranged by sizes listed in inches and millimeter in ascending order and for each size three different apertures (for lines, round pads and square pads) may be assigned.

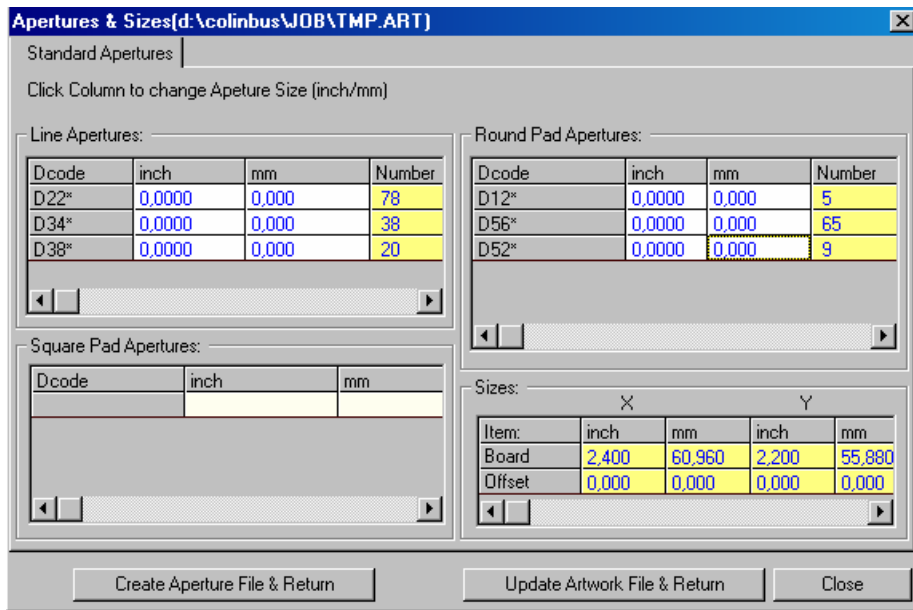
New sizes in selected unit may be inserted by clicking on **inch** and **mm** buttons. Aperture codes (D-Codes) are entered directly into cells in the table.

When new Aperture Table is to be created it is best to select one of existing that is the closest match. Set of necessary sizes may be updated by inserting new and deleting old unneeded. Next step would be assigning missing aperture codes.

Updated Aperture Table is saved under name entered in the text box **Named As:**

Creating own Aperture Tables may be worth the trouble in cases when RS-274D files from same source are repeatedly imported. It most likely that set of aperture codes and their sizes are identical in all these files.

Simplest method for assigning sizes to apertures in RS-274D format files is through **Apertures & Sizes** dialog, opened by clicking on button for parameter *Aperture used*.



Aperture & Sizes dialog for setting standard apertures. Aperture sizes may be entered either to “inch” columns in inches or to “mm” columns in millimeters – whatever more convenient. Most practical is to open ASCII report file (if available) with aperture D-codes and corresponding sizes used in the import file and paste the sizes directly into cells. Report file may be for this purpose displayed in viewing field of **Import Parameters** dialog.

Clicking on button **Update Artwork File & Return** will effect in re-import of the artwork image using assigned aperture sizes. This function overrides but **does not** change the Aperture Table that has been selected.

Clicking on button **Update Aperture Table & Return** allows creating external Aperture Table from data entered in the dialog. This is practical solution in cases when other RS-274D files from the same source will be imported in this session. It is most likely that similar apertures will be used in those files.

4.5. Importing board outline and cutouts from Gerber files

Outline of the board and eventually outline of internal cutouts may be imported from Gerber files in both formats. Parameters setup for import to Board Outline/Cutouts layer is identical as when importing to artwork layers. One exception is parameter ***Reconstruct polygons for Outline/Cutouts***.

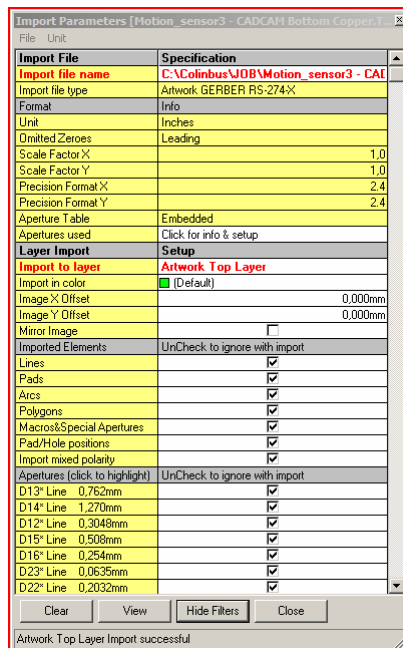
Some PCB CAD systems generate separate Gerber files containing only external and internal outlines of the board. In such cases the import is simplified. It is nevertheless necessary to ensure that the outlines are build of closed shape polygons, as only for those the program will generate toolpaths. Any other shape imported to this layer will be ignored in generation.

Board outline and cutouts may be reconstructed after import with help of editing functions. Imported elements could be used as template and eventually removed from the layer when reconstruction is completed.

Other solution is to order automatic reconstruction of polygons during import by checking box for parameter **Reconstruct polygons for Outline/Cutouts**. The program will in this case scan all imported elements to find out those that form closed no-intersecting shapes and convert them to polygon segments. No other elements will be imported.

4.6. Usage of filters with import of artworks and outlines

Clicking on **Show Filters** button opens section of the parameters table that contains import filters. This button is visible after import layer has been selected.



Although elements imported to layer may be removed with the help of editing functions, it may be practical to omit them already at import stage.

By default all types of elements are imported. Import of selected category may be suppressed by un-checking corresponding check box in the section **Imported Elements**.

Section **Apertures** lists all apertures detected in the import file. Un-checking of checks box in this section cause re-import of the image where all elements defined with corresponding aperture will be ignored.

Clicking on aperture description in the left hand column of this section highlights on the **Graphic Screen** all elements defined with this aperture. This feature is especially useful before deciding which elements should be ignored.

Some artwork images include elements (lines) that constitute board outlines. These elements should not be contoured and have to be removed from the layer. Quite often, such lines are defined by certain aperture that is not used for anything else. In such cases, suppressing import of elements with this aperture solves the problem in advance.

Board outlines incorporated in artwork images may be imported to Board Outline/Cutouts layer. Here the image contains too many unneeded elements. The automatic polygon reconstruction will remove most but it may be time consuming, especially for artworks with stroke filled copper pour areas. Images of this kind contain thousands of elements that program has to scan. It is therefore advisable to import board outlines initially without reconstructing polygons and try to reduce as many as possible of unneeded elements with help of filters.

4.7. Importing drilling data from Excellon Format files

When opened import file (**Select File for Import** dialog) is detected to be in Excellon Format One or Two, it automatically implies import to Drill Template layer. In this case selection of import layer is not needed and the file is imported directly.

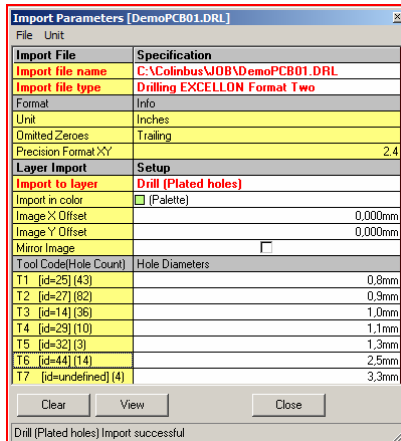
Importing drilling data from Excellon Format Two files

Since tool diameters and all formatting information are embedded in Format Two files there is normally no need to set any parameters. Usage of settable parameters is in this case optional.

Import Parameters for Excellon Format Two		
Parameter name	Type	Settings and values
Import File Name	Button	Path to currently opened import file. Click to open next file.
Import File Type	Drop-Down List	Displays detected format. Allows switching to Format One in cases when user wishes to override or correct formatting parameters specified in the import file.
Unit	Fixed	Unit of measure specified in the import file.
Omitted Zeros	Fixed	Type of zero omission specified in the import file
Precision Format	Fixed	Precision format for X and Y-axis coordinates specified in the import file.
Import to layer	Fixed	Drill Template
Import in color	Drop-Down List	Assigns display color for Drill Template. Offers two options: Selected from palette or Use default color
Image X Offset	Text Box	Imported holes will be shifted horizontally by offset entered in this box.

Image Y Offset	Text Box	Imported holes will be shifted vertically by offset entered in this box.
Mirror Image	Text Box	If this box is checked then imported holes will be symmetrically rotated around Y-axis.

Import parameters dialog contains section listing all tool codes and corresponding diameters found in the import file:



Clicking on a tool code displayed in the left hand column in this section causes highlighting on the **Graphic Screen** of all holes with corresponding diameter.

Only holes with non-zero diameter are imported so entering zero into hole diameter box acts like a filter. No holes for corresponding tool code will be imported.

Importing drilling data from Excellon Format One files

Import in this format requires setup of formatting parameters and assigning diameters to tool codes detected in the import file. All this information may be pasted from a report file if available.

In absence of report file and when formatting information are not readily available it is practical to assign tool diameters first. Initially, the diameters for all tool codes are set to zero and no import results may be visible on the screen. As stated before, the program imports only holes with non-zero diameters. With diameters assigned, the formatting parameters may be adjusted by trial and error. In such case, it is useful to import artwork layers before drills. Drill template display superimposed over artwork layers may provide clues for adjustment of formatting parameters.

Import Parameters for Excellon Format One

Parameter name	Type	Settings and values
Import File Name	Button	Path to currently opened import file. Click to open next file.
Import File Type	Fixed	Displays detected format.
Unit	Drop-Down List	Select Inches or Metric - unit of measure for the import file.

Omitted Zeros	Drop-Down List	Select type of zero omission for the import file
Precision Format	Drop-Down List	Select X and Y-axis precision format for the import file.
Import to layer	Fixed	Drill Template
Import in color	Drop-Down List	Assigns display color for Drill Template. Offers two options: Selected from palette or Use default color
Image X Offset	Text Box	Imported holes will be shifted horizontally by offset entered in this box.
Image Y Offset	Text Box	Imported holes will be shifted vertically by offset entered in this box.
Mirror Image	Text Box	If this box is checked then imported holes will be symmetrically rotated around Y-axis.

4.8. Importing drilling data from Gerber files

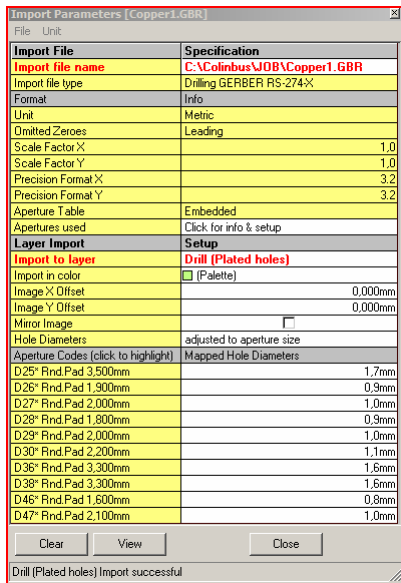
Drilling data may be also imported from Gerber files in both RS-274X and RS-274D formats. This method may be used when Excellon Format file is unavailable. It may be sometimes even easier to import drills from Gerber than from Format One, especially when formatting parameters for the latter are missing.

Selection of Drill Template as import layer when Gerber file is opened implies automatically import of drilling data. The file is immediately imported and results are shown on the Graphics screen.

Import of drills from Gerber files assumes that coordinates of holes correspond to coordinates of pads defined by flash command D03*. Diameters are calculated from aperture sizes used for plotting pads.

There are two ways of calculating hole diameters. They may be adjusted to either pad sizes (0.5 of aperture diameter) or assume explicit aperture size. For pads plotted with special apertures or with aperture macros where sizes may be ambiguous, the assumed diameter is always 1 millimeter. Section **Layer Import Setup** of **Import Parameter** dialog contains parameter **Hole Diameter** where one of above methods may be selected from drop-down list.

Import Parameters dialog contains section **Aperture Codes** listing all pad apertures detected in file with corresponding calculated holes diameters:

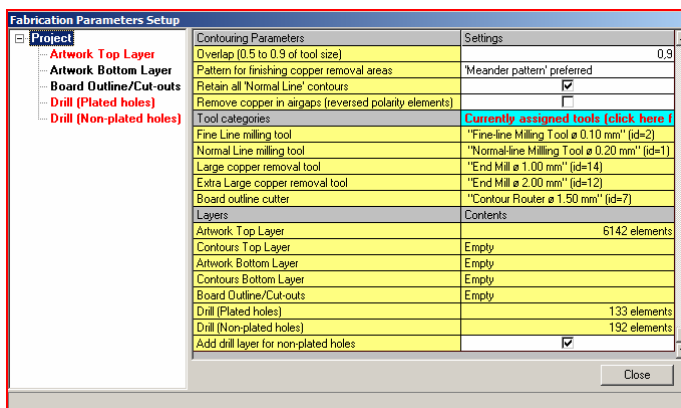


Holes diameters calculated by the program can be changed. In many case artworks image contains pads for Surface Mounted Devices where no hole is required. In such cases, setting diameter to zero will suppress import of holes for pad defined with corresponding aperture.

4.9. Plated and non-plated holes

Drilling of plated and non-plated holes is usually done in two separate machining operations. Therefore, the machining program for drill has to be divided into two sections.

It is assumed by default that all imported holes are plated. Therefore, the program automatically selects layer “Drill (Plated holes)” to store drill data imported from Excellon files. All holes from this layer will be grouped in one section in the generated machining program. In those cases when a separated section in the machining program is needed for non-plated holes, an additional layer – “Drill (Non-plated holes)” may be added to the layer stack. This is accomplished by checking appropriate check box in the **Fabrication Parameters Setup** pop-up dialog:




When layer for non-plated holes has been added, the program will ask to which of the drill layers the holes should be imported, each time an Excellon file is selected.

5. Preparations for generating toolpaths

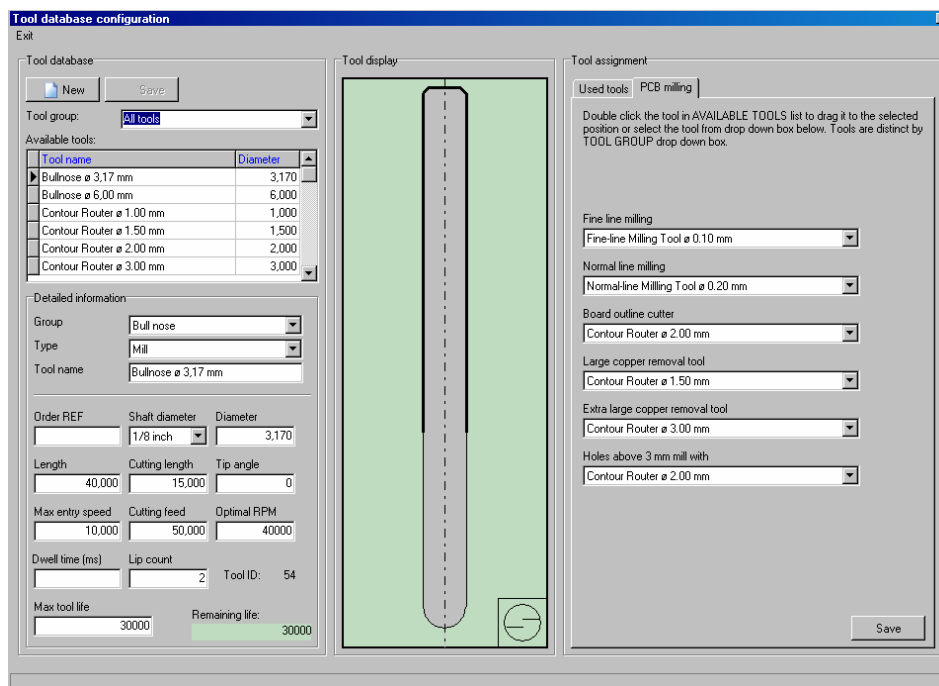
There is usually necessary to execute several tasks before contouring of artwork images and generation of final fabrication data may take place. Amount of work at this stage depends on what has been imported.

5.1. Tool Database and tool categories

Tool Database containing all tools needed for fabrication of PCBs is installed along with the program. Tool Database may be accessed from the program ( **Tool Setup** button) and user may add new tools, remove tools unneeded or unused and change tools' machining parameters.

The information that the program retrieves from Tool Database is tool diameters and tool identification number. Tool diameters are needed for calculating toolpaths and tool identification numbers are passed in machining program to the software that controls milling machine.

Clicking on  **Tool Setup** button opens dialog accessing **Tool Database configuration**:



The program recognizes five tool categories. Using this dialog, the user may select suitable tools from available in the database and assign them to each of the categories. Physically the same tool may be assigned to more than one category.

These five categories are:

Normal Line milling tool

Used for milling contours and for finishing copper removal areas at places where larger tools cannot pass.

Fine Line milling tool

Used for milling contours in places that Normal Line tool cannot pass. On very dense boards where insulation without removing needed copper cannot be assured by Normal Line tool, all contour milling may be done will Fine Line tool.

Extra large copper removal tool

Used in the first stage of copper removal milling.

Large copper removal drilling

Used in the second stage of copper removal milling.

Board outline cutter

Used for cutting external and internal outlines of the board.

Tool Database may contain drill bits. The program checks whether required drill bits are present in the database and informs (during import) if some is missing.

Note:

Colinbus software that controls milling machines (ColiDrive) uses the same database as this program. In cases that ColiDrive is installed on other computer system, user must ensure that changes to Tool Database settings made in one system are repeated in the other.

5.2. Aligning layers

Quite often images imported to artwork layers and drill data in imported to Drill Template are misaligned.



There is a special tool provided (see **Editing Tools and Option, Align Layers Tool**) for relocating currently edited layer by a vector between two points. First of these points is selected on edited layer and the second on any of other visible layers. In result of this displacement, these two points will have the same coordinates (by shifting currently edited layer). This is the principle for layers' alignment.

Since most of the boards have at least few of SMD pads or via holes, it means that on the top layer there will be imported pad positions that have corresponding pad positions on the bottom layer (and eventually on inner layers). Moreover, most likely there will be a corresponding hole in the Drill Template. Therefore, pad positions and hole positions are best



suitable for selection alignment points. In such case, option **Align to pad position or hole (F2)** should be used.

There is also an option for aligning edited layer to snap grid coordinates nearest to current



cursor position: **Align to snap grid (F1)**. In this case, the second point is not selected in other layer. The alignment vector is calculated as distance between first alignment point (end-point of any selected element in edited layer, pad position or hole) and snap grid coordinates closest to current cursor position. This alignment mode may be useful when some editing including adding, relocating or transforming of elements is necessary for this layer. Most of the PCBs are designed with certain placement grid but after import, this grid alignment may be lost. Snapping elements to grid makes accurate relocation and transformation easier. This option allows restoring grid alignment.

5.3. Basic editing of PCB outline and cutouts

Assuming that polygon that defines outline of the board and eventually polygons defining outlines of internal cutouts are correctly imported and reconstructed, there are two operations that should be done before generating toolpaths for cutting.


Assigning cutting mode

In most cases, outer outline of the board is cut with cutting tool moving outside the polygon by distance of half tool's diameter. Internal cutouts are cut with tool moving inside of the polygon, also by a distance of half tool's diameter. Since the program cannot automatically recognize which cutting mode assign to polygons on this layer, the required cutting mode has to be assigned manually to each of them. After import, all polygons are assigned 'Cut on outline' mode meaning that tool's tip will move exactly in line with segments of the polygons.



Options of the **Assign Cutting Mode** tool allow assigning cutting modes 'Cut outside



outline' or 'Cut inside outline'  to selected polygon. Procedure for assigning cutting mode is simple; select the tool and option for require mode. After this click on the polygon. Assignment is confirmed by cutting toolpath visualized outside or inside the polygon depending on selected option.

Inserting 'break-out' tabs

'Break-out' tabs are short sections in outline polygon segments that are not cut. This prevents material inside of the outline from shifting while cutting is in progress. When cutting is completed, finished board may be easily broken out of the surrounding material. Inserting of 'break-out' tabs is second of editing operations that is usually needed to generate correct tools

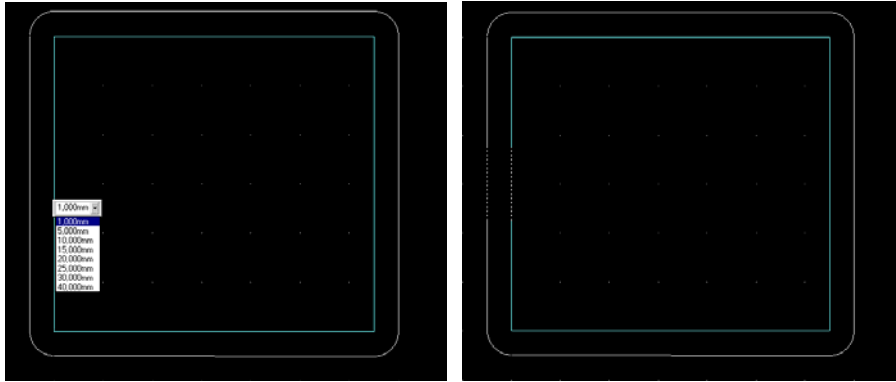


paths for cutting external and internal board outlines. For this purpose, option **Toggle**



to **'Breakout' Tab** for tool **Create Graphic Element** is provided.

'Break-out' tabs are inserted by selecting segment of polygon and entering tab length into small text box that pops out:



Inserting is confirmed by a section drawn with dotted line in the middle of selected segment. Picture above presents board outline polygon ('Cut outside outline' mode) before and after inserting an 'break out tab'.

If entered length of the tab exceeds length of the segment, the program will insert tab with 80% length of the segment.


'Break-out' tab is formed by inserting two extra vertexes collinear with selected segment. This means that after the operation is completed the selected segment will be divided into three parts. Middle part is the new tab segment. This feature allows free relocation of vertexes at the beginning and the end of the tab in order to adjust its position and size.

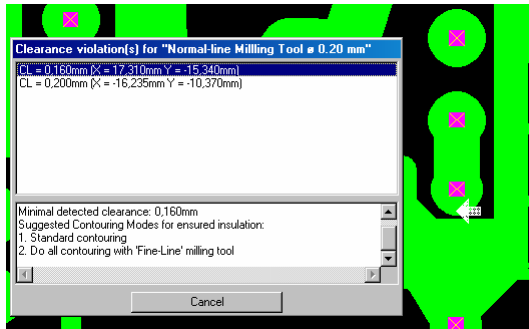
Function for inserting 'break-out' tabs works like toggle, which means that previously inserted tab is removed if it is selected instead of a segment without a tab

5.4. Design Rules Check (DRC)

Insulation (without shaving copper from contoured element) may be ensured only if the distance between two contoured elements is equal to or greater than diameter of applied tool. Contouring routine performs design rules check in order to establish whether elements' sizes and locations allow for necessary tool clearances. Depending on selected contouring mode, the program may apply various measures in order to ensure insulation in any case.

Nevertheless, it is advisable to perform design rules check prior to contouring in order to detect possible clearance violation. Knowledge about where and what kind of violation has been detected helps in selecting suitable contouring mode. In certain situation it possible to fix the problem by editing artwork image. In some cases, clearance violation is of such character that the artwork cannot be contoured with selected set of tools.

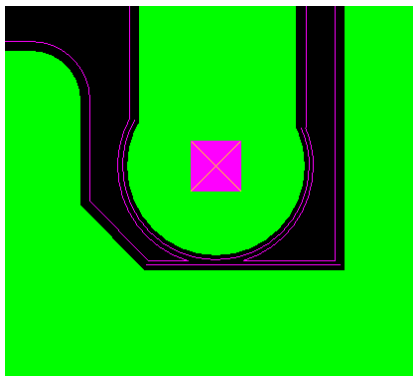
Check for tool clearance violation is invoked by clicking on  **Design Rules Check – tool clearances** button. The user is then asked for which of milling tools (Normal Line or Fine Line) the DRC should be performed. It is also possible to enter custom size. If the DRC detects problem following message dialog is displayed:




Locations on the board where clearance violations were detected are marked with white arrows. Upper box of the dialog contains list of all detected errors, informing of measured clearances and coordinates of faulty element. Clicking on error in the list focuses display on the **Graphic Screen** to its location.

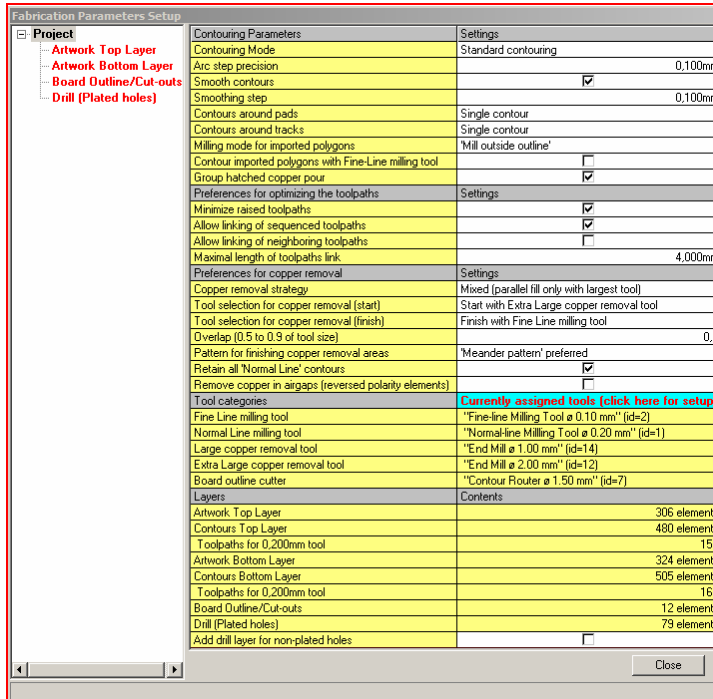
Lower box of the dialog presents suggestion about most suitable contouring mode to overcome the problem.

In above case, with standard contouring mode applied, the insulation would be ensured by contouring around element with Fine Line tool in the place where Normal Line tool cannot pass:



5.5. Setting up contouring and fabrication parameters

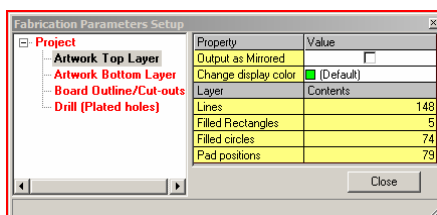
Contouring and fabrication parameters are set in pop-up dialog opened by tool  **Fabrication parameters setup** from **Standard** toolbar:




This dialog, apart from sections dedicated for entering parameters, contains also section that displays current tool category assignments and section with statistics regarding number of graphical elements on each layer. These information-only sections are displayed in rows with yellow background color.

Note:

Explorer window on left hand side provides shortcut to Layer Property dialog for every layer in the project without necessity of selecting layer as active:



All parameters in **Fabrication parameters setup** dialog are instantly stored by the program whenever changed. Settings may be therefore reused in following sessions and apply to all

layers (when tool  **Contour all layers** is used). In some cases, it may be necessary to alter some parameters in order to adjust contouring strategies specifically for one of the

artwork layers (contouring may be executed layer by layer ( **Contour layer** tool), every

time with different parameters). Users should be warned that settings in this case are not temporary. Return to values from before those set individually for a single layer have to be done manually.

Fabrication Parameters

Parameter name	Type	Settings and values
Contouring Mode	Drop-Down List	Select contouring mode. Options: Standard contouring (default) Forced insulation with Fine Line milling tool Forced insulation with Normal Line milling tool Do all contouring with Normal Line milling tool Do all contouring with Fine Line milling tool
Arc step precision	Text Box	Set interpolation step for arc toolpaths. Toolpaths for arcs are interpolated by linear steps. Entered value specifies length of interpolating line.
Smooth contours	Check Box	Set contours smoothing function ON or OFF
Smoothing step	Text Box	This row is visible if smoothing functions is ON. Toolpaths will be smoothed using entered value. Smoothing removes vertexes from toolpath poly-lines that differ in colinearity with neighboring vertexes by less then value of smoothing step.
Contours around pads	Drop-Down List	Number of contours around pads. Only elements that were defined in the import file with flash command D03* are considered as forming pads. Multiple contouring will be applied only to such elements. Options: Single contour (default) Double contours Triple contours Quadruple contours

Contours around tracks	Drop-Down List	Number of contours around elements that were defined in the import file with draw commands D02* and D01* and are considered as forming tracks. Multiple contouring will be applied only to such elements. Options: Single contour (default) Double contours Triple contours Quadruple contours
Overlap (0.5 to 0.9 of tool size)	Text Box	This row is visible if multiple pad contouring is selected. Enter tool overlap coefficient for multiple pad contouring. Value is expressed as fraction of tool diameter selected for contouring.
Milling mode for imported polygons	Drop-Down List	Select mode for milling of imported copper pour polygons. Options: 'Mill outside outline' (default) 'Mill on outline'
Contour imported polygons with Fine Line milling tool	Check Box	If this box is checked then copper pour polygons will be contoured with Fine Line milling tool.
Group hatched copper pour	Check Box	If this box is checked then the program will attempt to reconstruct copper pour polygons from vertical and horizontal stroke fill lines. It may significantly reduce contouring time.
Minimize raised toolpaths	Check Box	If this box is checked then sequence of toolpaths will be re-arranged in order to reduce movements with raised tool. This option shortens milling time.
Allow linking of sequenced toolpaths	Check Box	If this box is checked then program will attempt to link two or more sequenced toolpaths into one in order to avoid raising and lowering of the tool head. Length of toolpaths link is defined by a parameter below. Linking applies only to toolpaths generated when removing copper.
Allow linking neighboring toolpaths	Check Box	If this box is checked, the program will attempt to link all toolpaths located within defined linking distance. This operation takes place before toolpaths sequencing and may be time consuming with boards that are more complex. This routine

		applies only to toolpaths generated when removing copper.
Maximal length of toolpaths link	Text Box	Enter value of maximal allowed toolpaths link in range 0-50 mm
Copper removal strategy	Drop-Down List	Select method for generating toolpaths for copper removal. Options: Parallel fill Expand contours Mixed (parallel fill only with largest tool)
Tool selection for copper removal (start)	Drop-Down List	Select sequence of tools used for copper removal. Options: Start with Extra Large copper removal tool Start with Large copper removal tool Use only Normal Line milling tool
Tool selection for copper removal (finish)	Drop-Down List	Select sequence of tools used for finishing of copper removal. Options: Finish with Normal Line milling tool Finish with Fine Line milling tool
Overlap (0.5 to 0.9 of tool size)	Text Box	Enter tool overlap coefficient for copper removal. Value is expressed as fraction of tool diameter used in all three stages of copper removal using parallel fill method.
Pattern for finishing copper removal areas	Drop-Down List	Select toolpath pattern for finishing copper removal. Applies to all selected tools. Options: Plain hatch 'Meander pattern' preferred (default) 'Zigzag pattern' preferred

Retain all 'Normal Line' Contours	Check Box	If this box is checked then all toolpaths resulting from contouring with Normal Line tool diameter will be retained. Otherwise, Normal Line contours will be retained only in places where Extra Large and Large copper removal toolpaths cannot pass.
Remove copper in air gaps (reversed polarity elements)	Check Box	Applies to mixed polarity artworks only. If this box is checked (default) copper will be removed from air gaps. Otherwise, air gaps will be only contoured.
Tool categories	Button	Opens dialog Tool database configuration where physical tools from database can be assigned to below listed tool categories.
Add drill layer for non-plated holes	Check Box	If this box is checked, an additional layer for importing non-plated holes is permanently added to the layer stack.

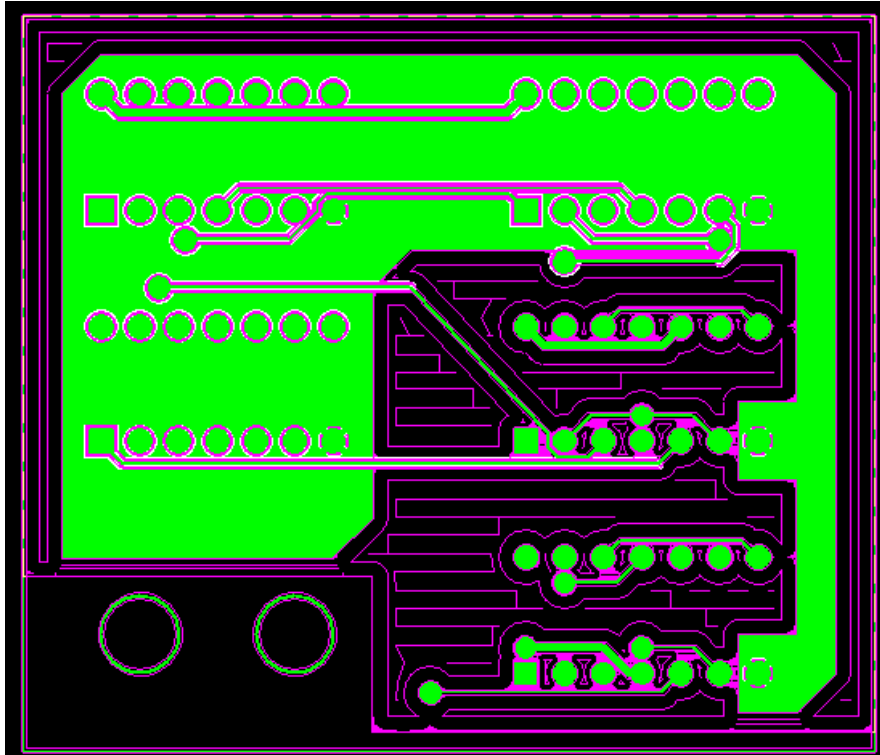
5.6. Introducing copper removal areas

The program can optionally generate toolpaths for removing copper in spaces on the artwork where there are no elements. Areas where copper removal should take place are specified by bounding polygons. There are several modes for introducing these polygons (see options for



tool **Assign Milling/Copper Remove Mode**). In all cases, the procedure is always the same and follows rules specified in **Fabrication Parameters** dialog. Assuming for example, that parameter *Tool selection for copper removal (start)* is set to *Start with Extra Large copper removal tool* and *Tool selection for copper removal (finish)* is set to *Finish with Normal Line milling tool*; this procedure using strategy *Parallel fill* consists of three stages:

- Copper is first removed by contouring elements within copper removal area with Extra Large tool. When this is done, selected finishing pattern is applied in spaces where there is still clearance for this tool.
- Contouring of elements within copper removal area is repeated with Large tool. This will farther remove copper in places where Extra Large tool could not pass. When this is done, selected finishing pattern is applied in spaces where there is still clearance for this tool.
- The board is contoured in selected contouring mode. After this, the remaining copper is removed with Normal Line tool, applying selected finishing pattern.



Picture above shows contoured layer where copper removal area has been defined by bounding polygon. Observe that copper has not been removed from circular cutouts at the bottom of the layer because they are located outside bounding polygon.

Irrespectively to the way in which they were created (circular, rectangular or polygonal) all copper removal areas are in fact polygons added to a layer. They are presented on the **Graphic Screen** with lines in yellow color. More that one area may be defined for one layer but they must not intersect. 'Remove all copper' mode automatically substitutes all previously defined areas with a single polygon that frames the whole board. All editing functions applicable to polygons imported or added to artworks apply also to polygons defining copper removal areas.

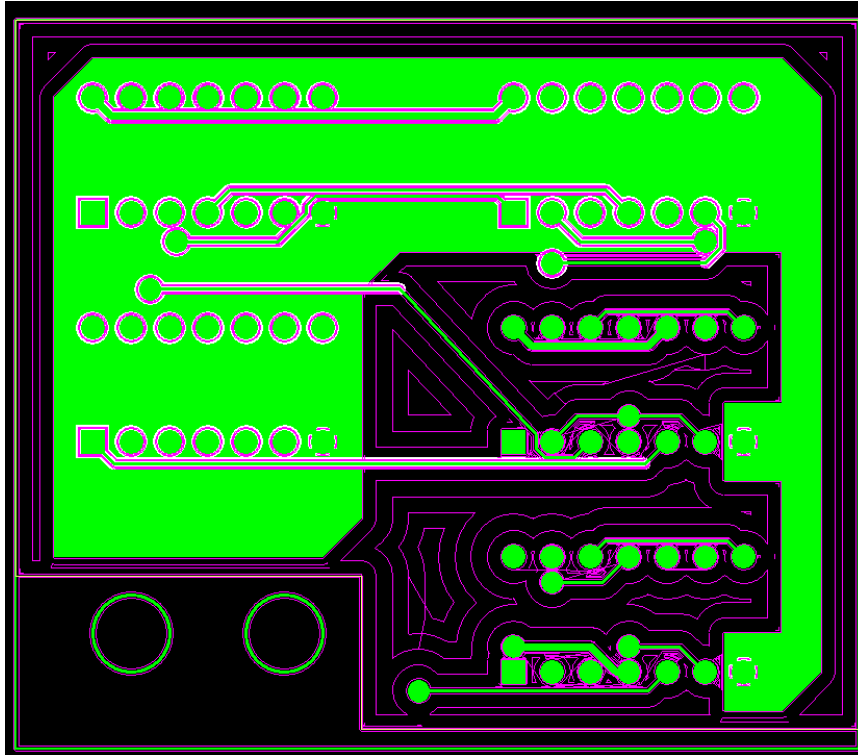
Note:

Copper removal areas must be defined prior to contouring.

To allow for tolerances in tool diameters and ensure that copper is thoroughly removed, the toolpaths must slightly overlap. Tool overlap is controlled by value of parameter ***Overlap (0.5 to 0.9 of tool size)*** that specifies spacing between parallel toolpaths

'Meander pattern' is recommended for finishing copper removal because it is most efficient in terms in reducing raised toolpaths.

If ***Expand contours*** method had been selected instead off Parallel Fill, the copper removal procedure would also consist of three stages. In this case, the artwork would be contoured repeatedly with each tools (in steps of half tool diameter) until there is no clearance left for tool to pass. This method takes more time to complete but may produce better results in terms of less and shorter toolpaths. A board with copper removed by expanding contours is presented on the picture below:



In some cases, it may be more efficient to remove copper over large empty areas using mixture of both above described methods - strategy *Mixed (parallel fill only with largest tool)*.

5.7. Optimizing toolpaths

Generated toolpaths may be optimized in several ways in order to shorten milling time and reduce wear and tear of the machine. The basic optimizing is done by toolpaths sequencing individually for each tool used. The program arranges milling sequence by finding first the toolpath that starts closest to the lower left corner of the board. Subsequently, it searches for next toolpaths that starts at location closest to the end of previous. This is continued until all toolpaths are sequenced. Sequencing ensures that length of movements with raised toolhead between separate toolpaths will be minimized.

Another method (in complement to toolpath sequencing) may be used for boards where copper has been removed. It may be in some case more time effective to move from toolpath to toolpath with tool down instead of raising tool, moving to new location and then lowering it again. Copper removal produces large number of closely located toolpaths that may be linked in this way providing that the linking path does not cut through the copper that must be left. With this condition and if the distance between the end and start of two consecutive toolpaths is less or equal than defined link length, the program will join these two toolpaths (parameter *Allow linking sequenced toolpaths*).

Some toolpaths may be easier linked before sequencing. This function may be ordered by setting parameter *Allow linking of neighboring toolpaths* on.

6. Contouring



Contouring routine is invoked with help of task tool button **Contour all layers**. All layers starting from Top to Bottom are contoured. Progress can be observed on the **Graphic Screen**. Contouring may be cancelled at all stages except design rules check by clicking on **Esc** button or pressing **Esc** key.



Single (currently active) layer may be contoured by pressing **Contour Layer** tool in **Editing Functions** toolbox. This feature is provided for cases when layers must be contoured with individually adjusted parameters.

Contouring algorithm follow rules specified in **Fabrication Parameters** dialog. Most important parameters for calculating contours are diameters of tools assigned to categories Normal Line and Fine milling tool. Successful contouring depends on whether there is enough clearance for toolpaths between elements belonging to different nets.

Contouring Modes

In accordance to selected contouring modes, the program applies various measures in solving clearance violations.

Standard contouring mode

In this mode, the contouring algorithm calculates toolpaths using diameter of Normal Line milling tool. Before contouring, the program performs design rules check in order to detect possible clearance violation. Elements causing violation will be contoured using diameter of Fine Line milling tool. In this case, the insulation will be ensured if minimal clearance is bigger than diameter of smaller tool.

Forced insulation with Fine Line milling tool

In this mode, the contouring algorithm calculates toolpaths using diameter of Normal Line milling tool. Before contouring, the program performs design rules check in order to detect possible clearance violation. Elements causing violation will be contoured using diameter of Fine Line milling tool after being forcefully shrunk to ensure necessary clearance. Amount of copper shaved from those elements will be equal to half size of Fine Line milling tool.

Forced insulation with Normal Line milling tool

In this mode, the contouring algorithm calculates toolpaths using diameter of Normal Line milling tool. Before contouring, the program performs design rules check is in order to detect possible clearance violation. Elements causing violation will be contoured using diameter of Normal Line milling tool after being forcefully shrunk to ensure necessary clearance. Amount of copper shaved from those elements will be equal to half size of Normal Line milling tool.

Do all contouring with Normal Line milling tool

No design rules check is performed in this mode and toolpaths are calculated using diameter of Normal Line milling tool. Insulation will be ensured if minimal clearance is bigger then this diameter.

Do all contouring with Fine Line milling tool

No design rules check is performed in this mode and toolpaths are calculated using diameter of Fine Line milling tool. Insulation will be ensured if minimal clearance is bigger than this diameter.

Multiple contours around pads

There is an option for ordering multiple contours around the pads. Which elements constitute pads is not always clear from the data in imported files. In most Gerber files, the pads are defined by flash command D03* but there are exceptions from this rule. Nevertheless, only this command allows the program to recognize elements of the pads.

Note:

Recognized pads elements are indicated on the Graphic Screen by pad position markers (small reddish rectangles)

To allow for tolerances in tool diameters and ensure that copper is not left between multiple contours, the toolpaths must slightly overlap. Tool overlap is controlled by value of parameter *Overlap (0.5 to 0.9 of tool size)* that specifies spacing between parallel toolpaths

Multiple contours around tracks

Multiple contours may also be ordered for tracks (traces) that are defined in artwork file as lines with help of D01* and D02* commands. Same tool overlap is applied as for multiple pad contours.

Special considerations for copper pour polygons

With some boards, the program may have problems in ensuring insulation of copper pour polygons when gap between some segments and neighboring elements is less than diameter of tool assigned for Normal Line milling. In such case user may use parameters *Milling mode for imported polygons* and *Contour imported polygons with Fine Line milling tool* in combination. By default, copper pour polygons are contoured outside their outlines. Other selectable mode is **'Mill on outline'**

Automatic reconstructions of copper pour polygons from stroke fill lines

Some PCB CAD systems define copper pour by hatching areas with thin horizontal and /or vertical lines instead of using more efficient mixed polarity artwork images.

Stroke fill creates problems for contouring algorithm by sheer amount of data, which results in longer processing time. In extreme cases, the program may run out of available memory for storing intermediate data during contouring.

This problem may be overcome by setting *Group hatched copper pour* parameter ON. The program will in this case attempt to convert as much as possible of hatch-defined area to polygons.

7. Output of generated toolpaths

Complete PCB machining program consisting of toolpaths for contours, copper removal, board outlines and drills is stored to a single disk file. This action is initiated by pressing task

tool button  **Fabricate PCB.**

Note:

Sequence of toolpaths for each tool can be re-arranged in order to reduce movement with raised tool. This option considerably shortens machining time and decreases tear and wear of the machine. Optimization function may be switched ON (before contouring) by checking box for parameter Minimize raised toolpaths in Fabrication Parameters dialog.

The structure of machining program is presented in the table below:

The structure of machining program

Layer Sequence	Toolpaths Sequence
Blocks for artwork layers from Top to Bottom Layer	Toolpaths for copper removal with Extra Large tool Toolpaths for copper removal with Large tool Contour toolpaths for Normal Line milling tool Contour toolpaths for Fine Line milling tool 'Mill as line' mode toolpaths for Normal Line milling tool 'Mill as line text' mode toolpaths for Normal Line milling tool
Board Outline/Cutouts layer	Toolpaths for Board outline cutter
Drill (Plated holes)	Coordinates of holes sorted by their tool diameters in ascending order. Drill bit diameter is specified for each hole.
Drill (Non-plated holes)	Coordinates of holes sorted by their tool diameters in ascending order. Drill bit diameter is specified for each hole.

Actual order of executing this program depends on the control software for particular milling machine.

Machining programs passed to ColiDrive – dedicated software that controls Colinbus milling machines are stored in disk files with name extension .BFF.

If ColiDrive is installed on the same computer system then it is started immediately when generation of machining program is completed.

8. Editing

Apart from selecting cutting mode for board outlines and inserting 'break-out' tabs in outlines' segments, no editing of source data for contouring is required in most cases. Introducing of copper removal areas is optional.

Other editing functions have been provided for those rare occasions where corrections as well as removing/adding of redundant/missing elements may be required. It is in fact possible to create simple PCB layouts, complete with layer artworks, outlines and drilled holes with help of editing functions of this program.

8.1. Usage of snapping and contact points with editing

Editing includes often relocation or stretching (transforming) of elements. During this process, the highlighted element follows cursor movement on the **Graphics Screen** or its form changes following this movement. The cursor moves over the screen in steps of one pixel. Depending on currently set display precision, this step covers different distances measured in database coordinates. For example if precision is set to 0.010 mm, then elements may be moved or stretched in steps of 0.10 mm.

It is often difficult to adjust display precision to required precision for cursor movement. This precision may be forced by setting orthogonal snap grid step (see **Standard** toolbar) and switching the snap function ON. Relocated or stretched elements will 'snap' to grid locations nearest to current cursor position, irrespectively to display precision. Orientation with movement will be enhanced by setting ON display of visible raster (see **Standard** toolbar) with its interval adjusted to multiplication of grid step. Snap function takes also effect with dragging and relocating bounding rectangles.

Angular snap may be applied when single element or block of elements are rotated. All settings for raster and snap functions may be changed while elements are being moved or stretched.

Sometimes an element has to be aligned with other element that is placed outside of the grid.



In such cases, it is convenient to set contact point the "off-grid" element (see **Set contact point** tool and its options. Elements selected for relocation or stretching will 'snap' to first contact point.

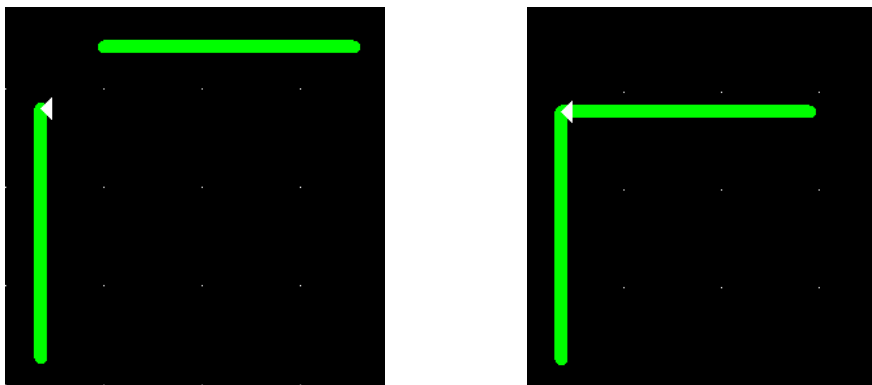
Combination of **Shift** key and one of numeric keys can be used as shortcuts for setting contact points without switching to **Set contact point** tool from other editing tools

Options:

- Press **Shift 1** and then click on the **Graphic Screen** to set contact at snap grid coordinates closest to the cursor position.
- Press **Shift 2** and then click on pad position or hole to set contact point at coordinates of pad position or center of hole.

- Press **Shift 3** and then click near the corner of board framing rectangle to set contact point exactly on this corner,
- Press **Shift 4** and then click on the element to set contact point at the end-point of this element closest to cursor location.
- Press **Shift 5** and then click on the element to set contact point at the centre of this element.


Picture below illustrates usage of contact point in exact end-point-to-end-point alignment of two off grid elements. (**Relocate graphic element(s)** function).



8.2. Editing artwork images

Imported artworks may contain elements that are redundant or that not need to be machined on prototyped board. Not always, such elements may be filtered out in import because they are defined with the same apertures as elements that must be imported.

Examples of such elements are texts, logos and other graphics that may be a part of artwork image but do not constitute the actual circuit. There is no need to mill these if board is



prototyped and may be removed with help of  **Delete** tool.

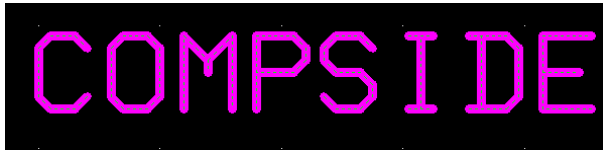
'Mill as line' and 'Mill as text line' modes


The program has no possibility to recognize textual information automatically. Texts in artworks images are composed of lines that normally will be contoured. This may be undesirable since it would considerably prolong machining time. In some cases, machined text image may be difficult to read:



Picture above presents contours of elements composing text. Machining time may be in this case reduced and readability enhanced by assigning these element 'Mill as line text' mode

(see  option of  **Assign milling mode** tool). After this operation and repeated contouring, the toolpaths for milling text elements will be as on the picture below:



There is another option for setting  **'Mill as line'** mode. It will produce identical results. Then only difference is that toolpaths for elements milled in 'Mill as line' and 'Mill as line text' modes are grouped in two separate sections of layer block in the machining program. These sections may be skipped during machining.

Correcting clearances

Another example that editing functions may help in obtaining correct machining results is correction of design rules violation discovered in imported artworks. Picture below shows evident shortcut between two nets. This shortcut could not be detected by program's DRC that considers all overlapping element as members of the same net path:



By slightly lifting up (0.2 mm) the line above shortcut pads, enough clearance has been provided for the toolpath to pass and ensure insulation.

8.3. Editing board outline and cutouts

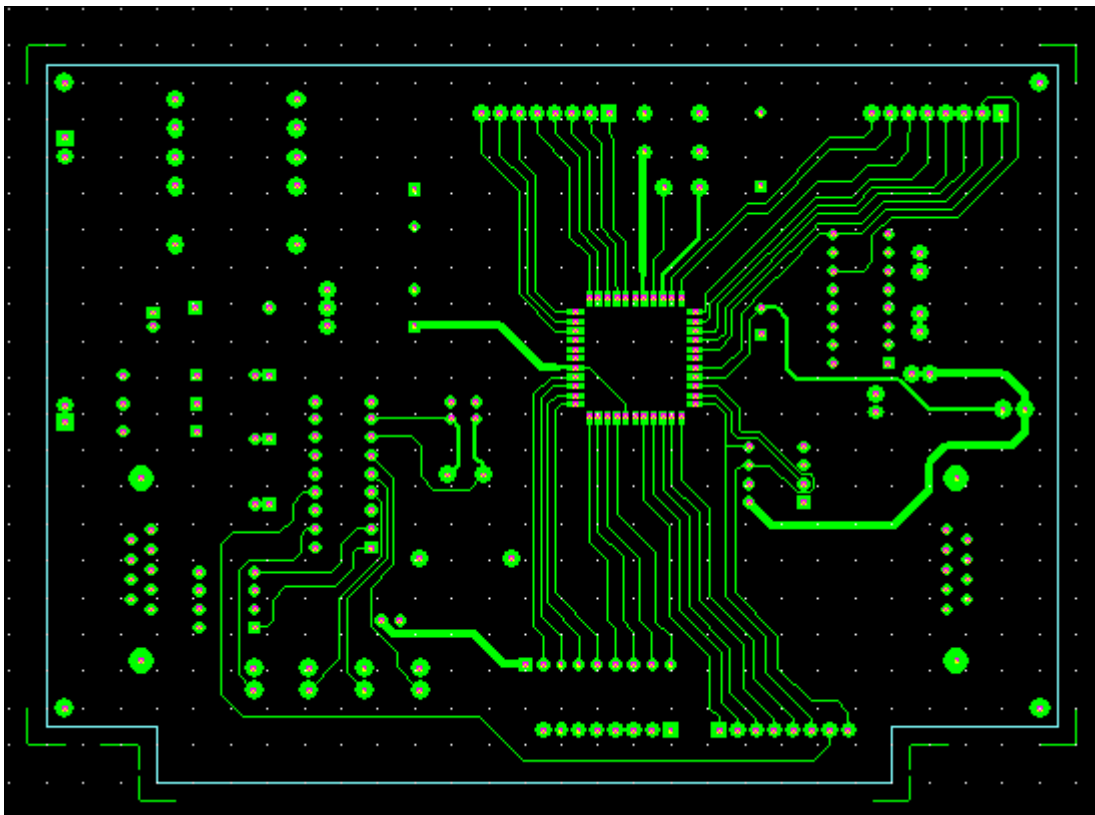
Not always board outlines can be imported directly to Board Outline/Cutouts layer. In such case, it is possible either to create necessary outlines from the scratch or to reconstruct those using elements on artwork layers as template.

Re-creating missing board outlines and cutouts


Board outline and cutouts may be simply created as polygons ( **Create polygon** and  **Create Circle** options of  **Create Graphic Element** tool.

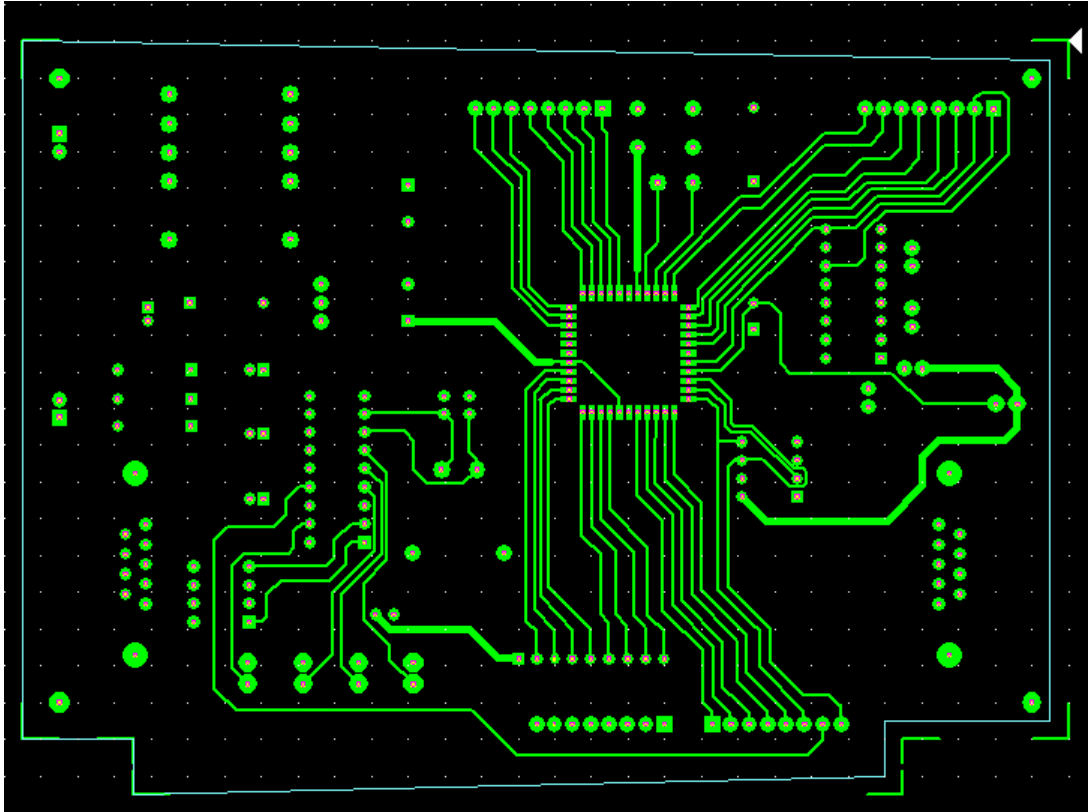
It can be sometimes difficult or laborious to transform newly created outline polygons to required dimensions and forms. This problem may be solved by using elements of artwork layers as template. Artwork images often include elements that mark boundaries of the board and cutouts. Vertexes of the polygons may be dragged to these markers to obtain proper coordinates. Contact points are very useful for this task

Picture below shows artwork image with markers indicating corners of the board outline. Re-construction starts with creating a polygon (shown in white color on the picture) that roughly approximates the shape of the board. Note that this polygon has the same number of vertexes as corners marked on the image.



Next picture show reconstruction process half way through. Four vertexes have been already

dragged to locations of respective corners with the help of  **Stretch Element** tool. Contact point is set at upper right corner in preparation for snapping nearest vertex to its exact position

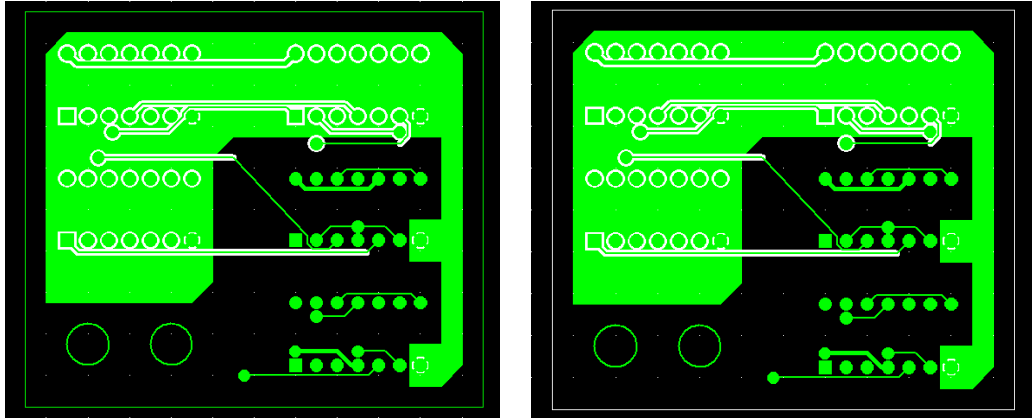


Transferring outlines from artwork layer to Board Outline/Cutouts layer

Artwork images often contain board outline markings that form continuous path. They are usually not needed there and should be removed otherwise they will be unnecessarily contoured with the rest of elements in this image. Instead removing them, they may be transferred to Board Outline/ Cutouts layer and converted to polygon. Example of such case is illustrated below. Left hand picture shows the artwork image as imported with full board outline markings. Picture to the right show artwork layer and Board Outline/Cutouts layer



after the transfer with the help of option  **Transfer path of elements (lines) to Board**

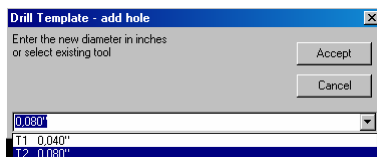
Outline layer of  **Repeat or Transfer Elements** tool.




8.4. Editing drilling data

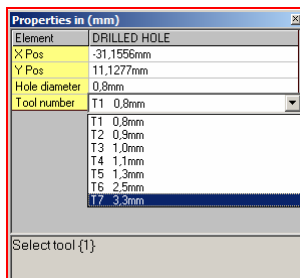
Necessity for editing drills may arise when holes must be added to or removed from the Drill Template layer.

Holes may be added with help of  **Created drilled hole** option for tool  **Create Graphic Element**. The program will ask to enter diameter for created hole:




Diameters for selected holes may be changed through **Properties** dialog (opened by clicking

on hole when  **Graphic Element Properties** tool is active) by entering new diameter or assigning other tool code from drop-drop down list.








There is also a possibility to assign new diameter to a tool code, which will result in change of diameter for all holes imported with this tool code. This function may be executed by entering

new diameter for tool codes displayed in dialog opened by clicking button for  **Layers Properties** tool:

Property	Value
Output as Mirrored	<input type="checkbox"/>
Change display color	<input type="checkbox"/> (Palette)
Layer	Contents
Drilled holes	192
Tool List(Hole Count)	Diameters
T1 [d=25] (43)	0.8mm
T2 [d=27] (82)	0.9mm
T3 [d=14] (36)	1.0mm
T4 [d=29] (10)	1.1mm
T5 [d=32] (2)	1.3mm
T6 [d=44] (14)	2.5mm
T7 [d=undefined] (5)	3.3mm

8.5. Editing contours

In case when contouring fails due to clearance problems, it is possible to correct such errors by adding filling segments to the toolpaths. Toolpaths that result from contouring are either closed shape polygons or poly-lines. Polygons and poly-lines as well as single lines and arcs

may be added manually after contouring (options  **Create Line**,  **Create Arc**,  **Create polygon**,  **Create poly-line for tool**  **Create Graphic Element**).

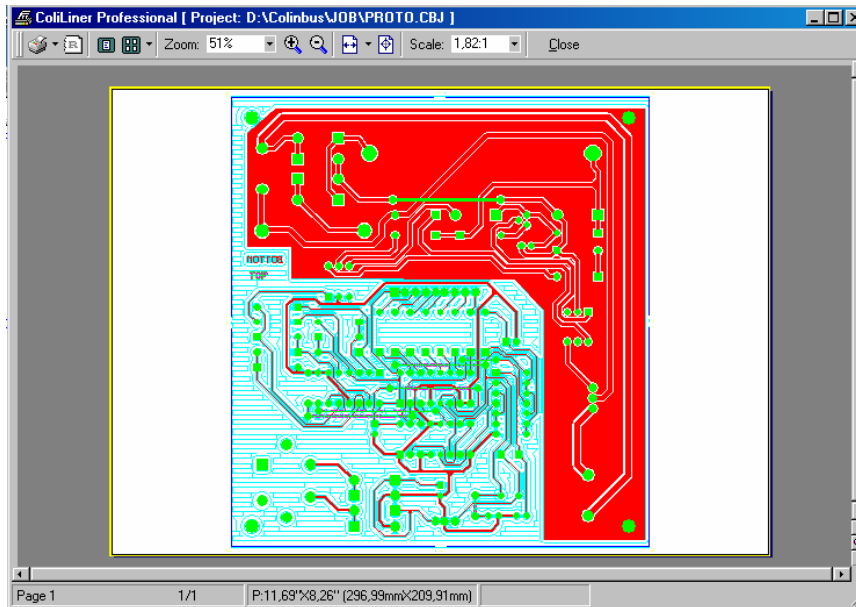
All added element are considered as toolpaths when final machining data is generated. Sizes (line widths) for these elements may be selected from drop-down list in **Sizes** toolbar. For contours layers, this list contains diameters of tools currently assigned to all categories except Board outline cutter.

Picture below shows how shortcuts due to clearance problem may be corrected but adding two arcs with line size equal to diameter of tool assigned to Fine Line milling category. Some copper would be shaved from the shortcut pads and neighboring line but the loss is negligible in this case:



9. Printing

At any stage, the image of the project may be printed in exactly same form as currently displayed on the **Graphic screen**. Entry **Print ..** in **File** menu opens print preview dialog:



Controls in this dialog allow printer selection, setting print format and positioning image on the page. The image may be scaled to fit the page or printed in any selected scale. If the size of image in given scale exceeds available page size then there is an option to print the image in several pages:

