CamDRAW™

User manual

© 2025 Camdraw

www.camdraw.com

Table of contents

12.03.2025

1 Definitions and abbreviations

CNC	Computerized Numerical Control
CAM	Computer-Aided-Manufacturing
DXF	Drawing Interchange File Format
HPGL	Hewlett-Packard Graphics Language
G-Code	CNC specific language code to control the movements and I/O of e.g. a milling machine
GUI/UI	Graphical user interface / program interface

2 Technical data

- Windows 11, 10, 8.1 or 7 with the latest updates and service packs
- Compatible with CorelDraw Graphics Suite X7, X8, 2017, 2018, 2019, 2020, 2021, 2022, 2023, 2024, 2025
- Compatible with CorelDraw Technical Suite X7, 2017, 2018, 2019, 2020, 2021, 2022
- Compatible with CorelDraw Standard 2020, 2021, 2024
- Compatible with CorelDraw Special Editions (OEM)
- Compatible with CorelDraw Home & Student X7, X8, 2018, 2019
- Compatible with CorelDraw Essentials 2021
- Compatible with CorelDraw 32/64 bit
- At least 4 GB RAM
- At least Microsoft .NET Framework 4.6
- An Internet connection is required once for software activation

3   Introduction

CamDRAW is a CAM plugin (Addon) for CorelDraw. It allows you to create G-code (CNC programs) for your CNC machine directly in CorelDraw. The software masters a large number of machining methods. Among others milling, engraving, drilling, laser cutting, plasma cutting and waterjet cutting. It creates 2.5D toolpaths from CorelDraw drawings. At the same time, the application is very flexible and supports all CNC controls. Some of them are for example:

- EdingCNC
- Mach 3/4
- Planet CNC
- Isel
- Estlcam
- CNCGraf
- BZT
- NC-Easy
- WinPC
- GRBL
- 3D Carbide GRBL
- Industrial CNC Router
- Langmuir CrossFire Systeme
- CAMaster Router

You can customize CamDRAW to fit your application. These can be among others:

- Advertising technology (signs)
- Graphics industry
- Furniture making
- Trade fair and exhibition construction
- Jewelry making
- Model making

With CorelDraw GCode plugin you can improve your workflow and reduce process time at the same time. CamDRAW in combination with CorelDraw combines all necessary functions in a single program interface. The task of drawing can be performed in the innovative design interface of CorelDraw. There are no limits to what can be done. The task of CAM software (generation of G-code) is done for you by CamDRAW.

Function list of CamDRAW:

Feature	CamDRAW
Create G-code directly in CorelDraw	✓
Workpiece data input and document creation	✓
Parallel working on documents (document tabs)	✓
Save and open CamDRAW projects (saves all settings and toolpaths made in CorelDraw document)	✓
Postprocessors (Profiles/Customization/Import/Export)	✓
Automatic node algorithm (interpolation precision for curves)	✓
Integrated tool list with technological data of the tools	✓
3D representation of the created G-code	✓
Contour machining (center, left, right, inside, outside)	✓
Pocketing	✓
Drilling machining	✓
Contour cutting (laser/plasma/water)	✓
Engraving with depth regulator	✓
Measure surface (measuring grid)	✓
Rotary axis support (incl. mapping/unwinding)	✓
Tool path optimization	✓
Lead-In & Lead-Out options for contour milling and contour cutting (Laser-, Plasma-, Waterjet machining)	✓
Tab function for laser, plasma and waterjet machining. (This function leaves small material tabs to fix the components during cutting).	✓
Tangential axis/tangential knife for contour machining (This function calculates the tangent at any point of contours and outputs it as an angular coordinate)	✓

4   Installation

After downloading the application, you can install CamDRAW. The setup (the installer) is an EXE file. If you have downloaded the software as a zip archive, please unzip the file accordingly. The setup must be run as administrator.

(1) Start the setup by double-clicking.
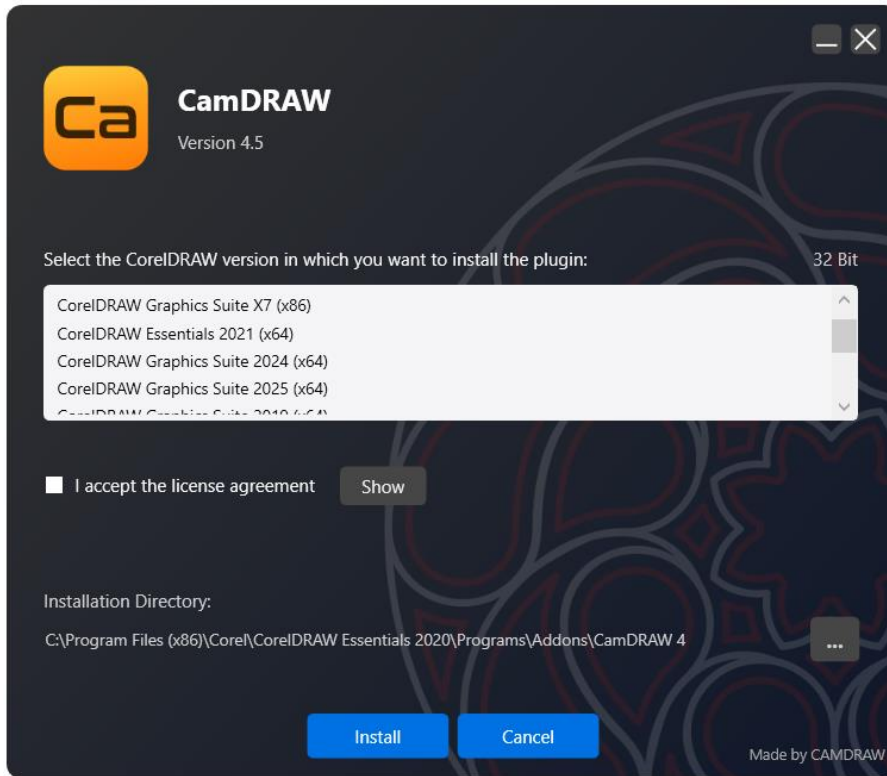
(2) The installer automatically detects all installed CorelDraw versions (32/64 bit). Select the CorelDraw product in which you want to install the plugin/addon. The 32/64 bit versions as well as the installation directory are set automatically when selecting the CorelDraw product.

(3) Accept the license terms.

(4) Click on "Install".

(5) After installing, you can close the setup program.

(6) Open the CorelDraw version in which you installed the plugin.

(7) Go to the CorelDraw main menu "Tools" and click on CamDRAW at the end of the list.
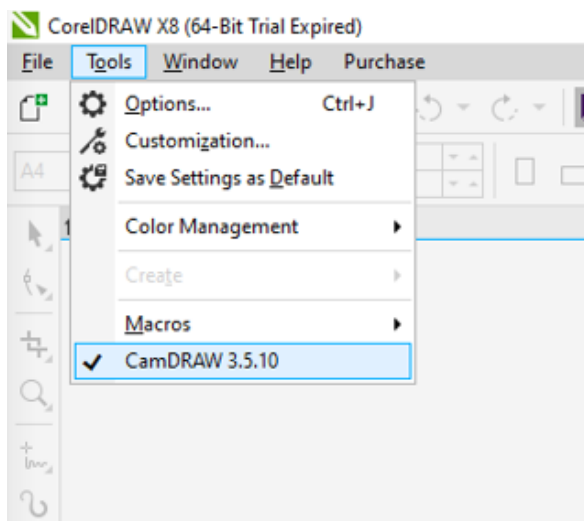
(8) The plugin is integrated into the CorelDraw interface on the right (Addons). If you install CamDRAW for the first time, the trial version will be activated automatically at startup.

(9) Adjust the docker width (by dragging with the mouse).

(10) Ready!

Important information:

If you do not see CamDRAW listed under the "Tools" menu item. Then quit CorelDraw. Restart CorelDraw and press/hold the F8 key at the same time. A message appears to reset the workspace. Confirm with OK. Now you will see CamDRAW under "Tools".

5   First Start

When you start the application for the first time, the window for selecting the default settings for your CNC machine opens. Here you can select the CNC machine you want to use. If you cannot find your machine in the list, please select "G-Code (Standard)" or "G-Code (Standard) - ATC". Both default settings are the standard configuration for G-code output, which is compatible with most CNC machines. The window appears in the same language as you have set your CorelDRAW.

Standard-Voreinstellungen:

G-Code (Standard):   Industriestandardformat ohne Werkzeugwechsel (Bei Verwendung von verschiedenen Bearbeitungswerkzeugen erfolgt die Ausgabe in separate G-Code Dateien. Sinnvoll für Werkzeugmaschinen, welche nur ein Werkzeug pro Datei unterstützen)

G-Code (Standard) – ATC: Industriestandardformat mit Werkzeugwechsel (bei Verwendung von verschiedenen Bearbeitungswerkzeugen erfolgt die Ausgabe in eine einzige G-Code Datei)
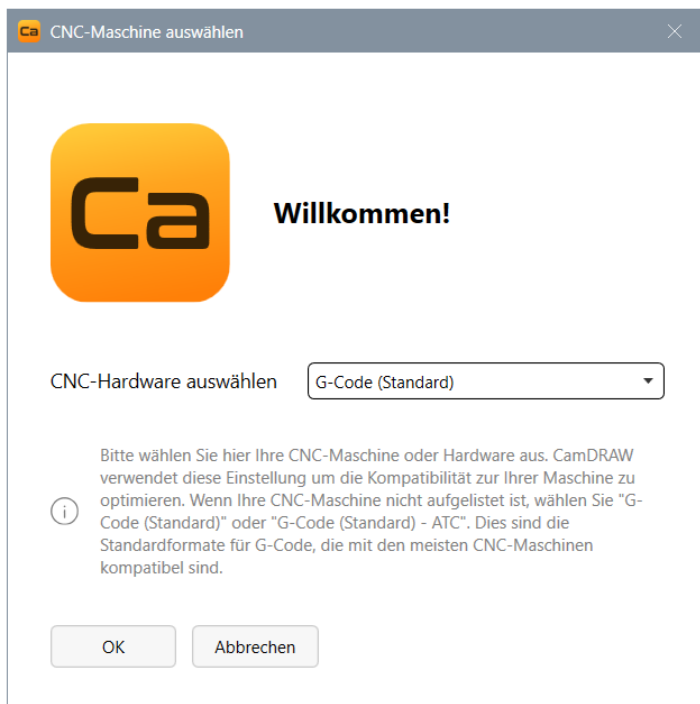
6   User interface

This chapter explains the basic structure of CamDRAW. The program interface is divided into different sections/expander, which can be displayed or hidden individually. The workflow for the production of G-Code takes place thereby always from above downward.
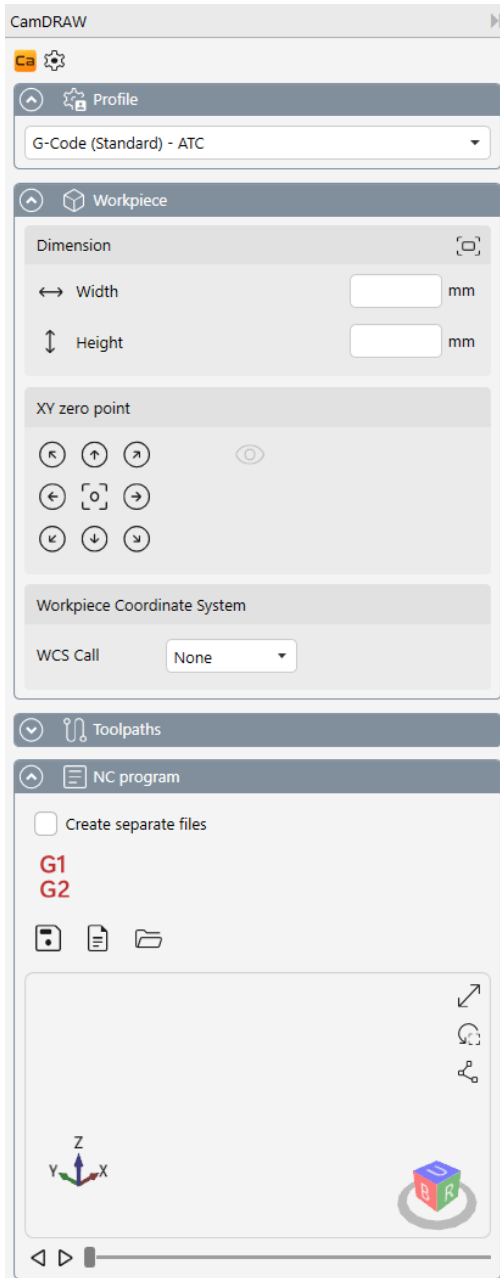
CLICK ON GEAR ICON: Open settings

PROFILE: selection of the profile / postprocessor

WORKPIECE: Input and settings of the current workpiece information.

TOOLPATHS: Creation of toolpaths (contour, pockets, holes)

NC PROGRAM: Creation / export / display / 3D representation of the CNC program (G-Code)

CLICK ON GEAR ICON: Open settings

Click on the logo at the very top of the interface to open the settings of CamDRAW.

PROFILE: selection of the profile and the postprocessor

Here the profile (the postprocessor settings) is selected, with which CamDRAW generates the CNC program. One must always be selected. Different postprocessor settings can be stored in the profiles. For example, if you want to perform a flat machining operation, select a postprocessor with the X, Y and Z machine axes set. If you want to execute a rotary machining instead, select a corresponding postprocessor with the A-axis set, for example. This allows a quick changeover from different machining operations on the same CNC machine.

CamDRAW comes with some postprocessors. In addition, all postprocessor settings can be changed in the application settings and adapted to your application.

WORKPIECE: Workpiece data

All workpiece settings are entered in this menu. These include the workpiece dimensions, XY zero point, Z zero point for rotary machining and the selection of whether rotary internal or external machining is to be performed.

Depending on which postprocessor you have selected, the workpiece menu is set automatically. This is necessary because different data are required for different machining operations. Here are the masks of the workpiece menu for a flat machining or a rotary machining.
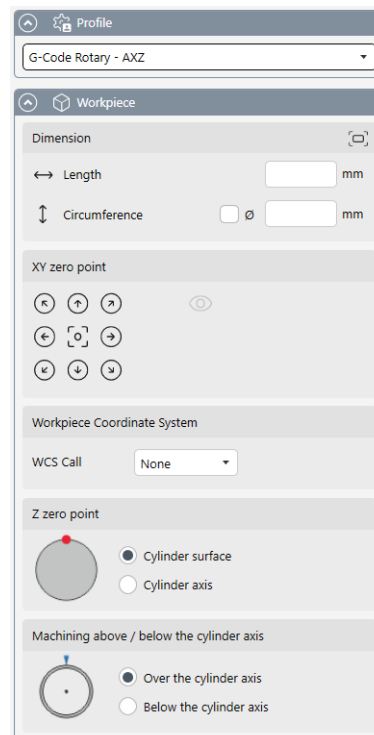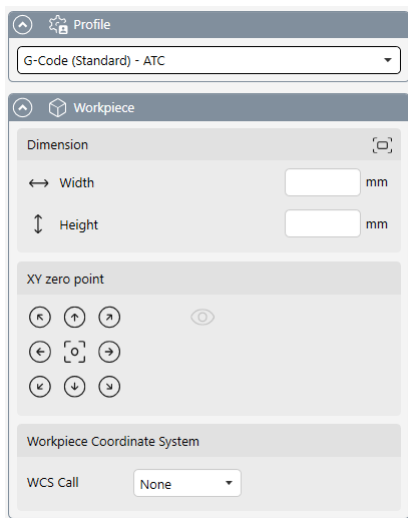
TOOLPATHS: Create toolpaths

In this menu you can create your toolpaths and call up the tool list. In addition, there is an overview/listing of the programmed toolpaths and their data. By right-clicking on an operation, you can edit, calculate or delete it. Clicking on the respective tool operation icon opens a new mask for adding the operation.

For more information on the individual functions, see the corresponding chapter.
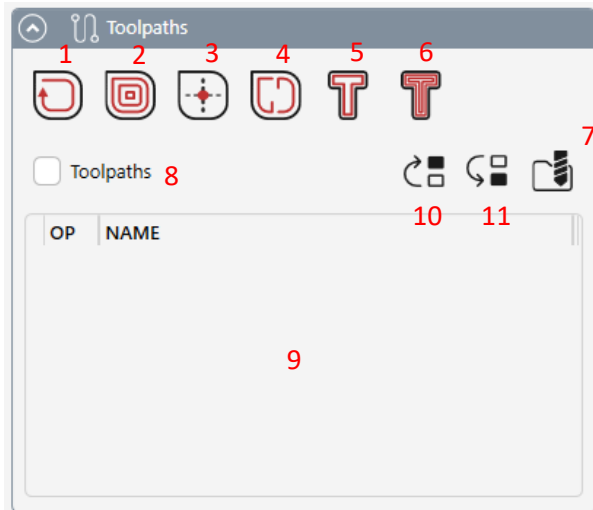
1	Contour milling
2	Pocket milling
3	Drilling machining
4	Contour engraving
5	Pocket engraving
6	Laser / Water / Plasma Cutting
7	Opens the tool list
8	Activates / deactivates all toolpaths
9	Listing of all created tool operations
10	Move selected tool operation one up
11	Move selected tool operation one down

Right-clicking on a created operation opens a menu that allows you to edit, calculate or delete the selected operation. If you click (select) an operation, then CamDRAW displays the associated toolpath in red. Additionally you can show or hide single or all toolpaths.
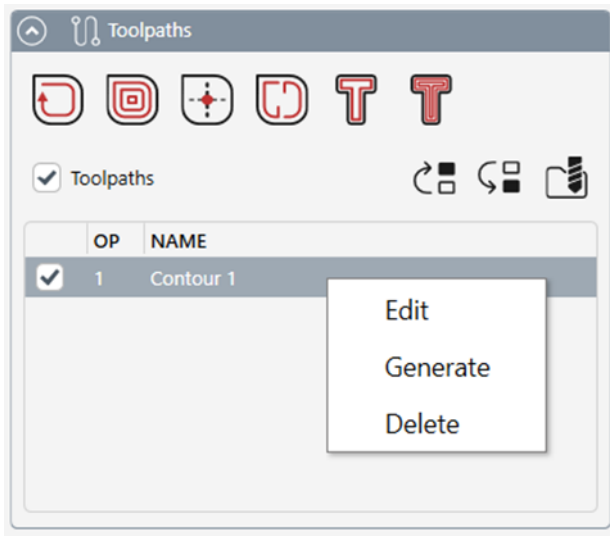
When you hover the mouse over an operation, the settings and information of the operation are displayed (cutting data, tool, etc.).

12.03.2025
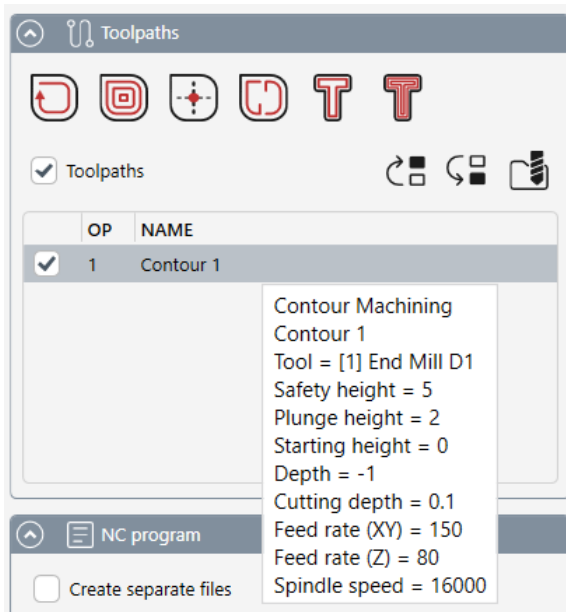
NC PROGRAM: Creation / export / display of the NC program

Here you will find all the functions for creating and viewing the CNC program. Furthermore, you can export and save the created CNC program. To control the G-Code, the G-Code is rendered (visualized) three-dimensionally after each creation. You can see the tool paths in blue and the rapid movements (approach and departure movements) in red. Double-click on the 3D rendering to open the 3D rendering in a separate window.
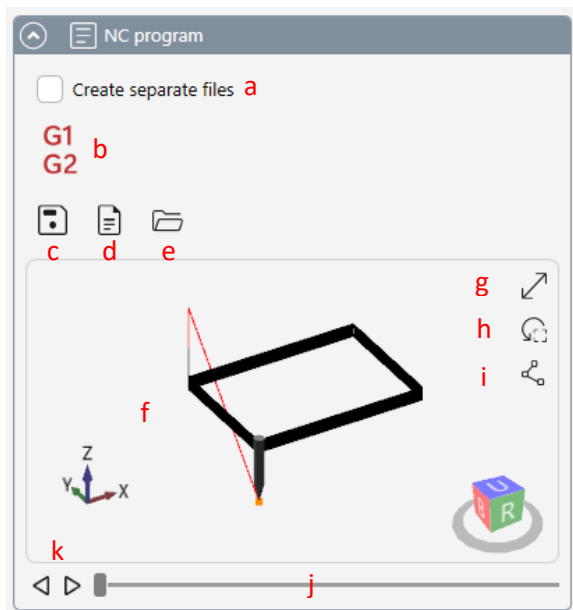
a	Create separate files for the tools used
b	Creating the NC program of the operations listed under Toolpaths
c	Export / save the created NC program
d	Open the created NC program with the text editor
e	Open the output directory of the G-Code files
f	3D view of the created NC program
g	Open the 3D view in a separate and larger window
h	Reset camera view
i	Display/hide all node positions of the NC program
j	Slider to simulate the tool paths
k	Single step forward and backward to simulate the tool paths

7 Functionality of the application (Docker)

7.1 General

CamDRAW is an addon that integrates with the CorelDraw program interface. With it you can create and save G-code for your CNC machine.

The application has a very flexible structure. In principle, it works like the "Object Manager" in CorelDraw. All settings, input values, created tool paths, etc. stored in CamDRAW are always stored in the currently open document. If you change/open/close a document, CamDRAW automatically adapts to the new document. This also works, for example, when changing the document via the "Document tabs" in CorelDraw.

Example (2 open documents):

12.03.2025
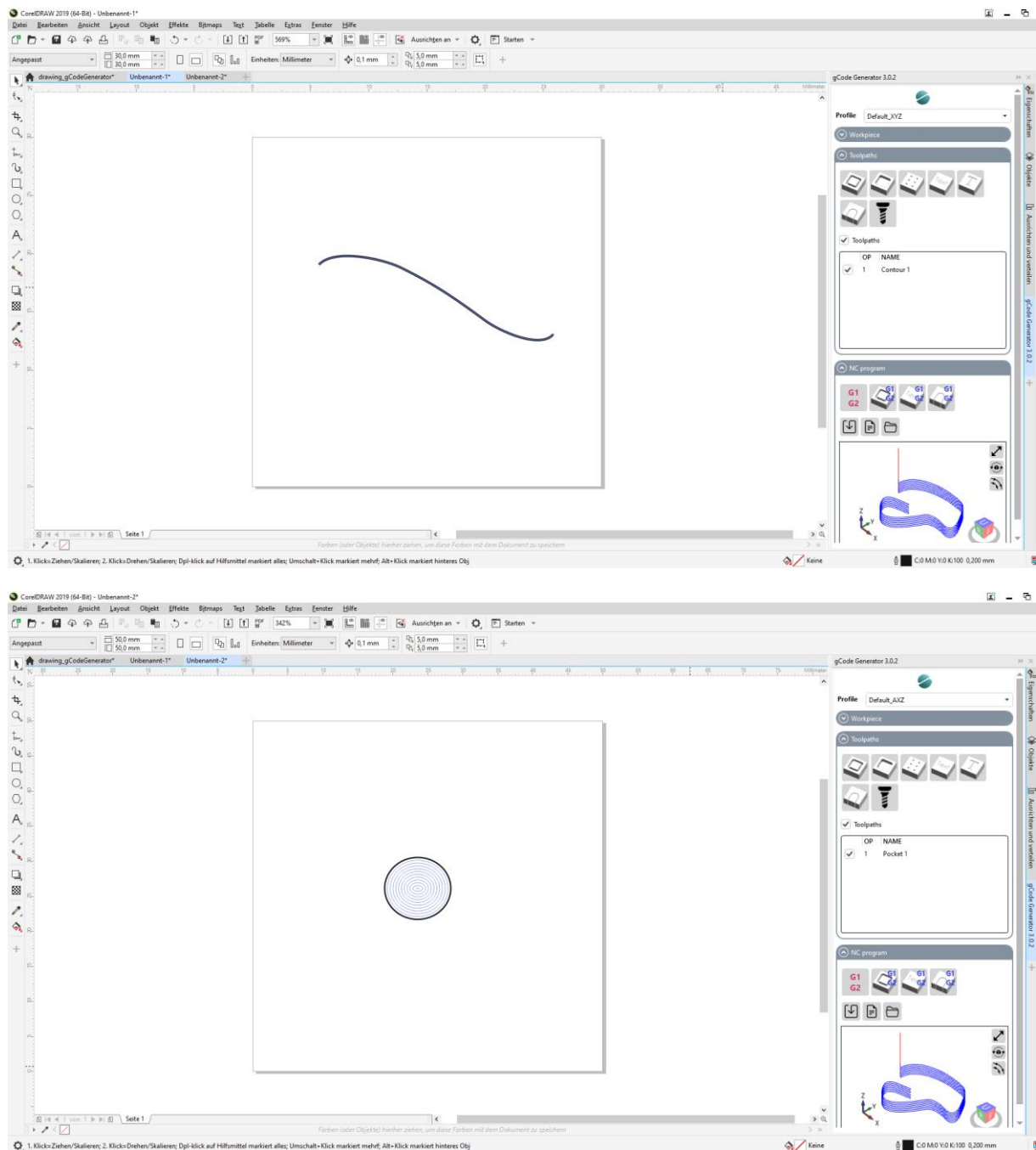
7.2   Layers / Layers

For CamDRAW to work properly, the application needs defined layers. On them are e.g. the tool paths.

If no document is currently open in CorelDraw and you enter workpiece dimensions, the application will automatically create a new document with all the necessary layers. These are:

- "ToolPaths" (created tool paths)
- "ToolPathPreview" (preview the tool paths while before creating them)
- "ToolPathWorkspace" (workspace for creating tool paths)
- "DesignArea" (user layer, this is where you draw)
- "HelpObjects
- "ZeroPoint" (X-Y zero point object)
- "Measurement" (measuring grid/measuring points for surface measurement)

Important: You don't necessarily have to create a document via CamDRAW for the layers to be added! You can also start with any or saved document. All layers will be created automatically as soon as the application needs them for the respective function!

7.3   Save / Open

Now that you have a whole set of toolpaths and settings stored in the current document, you can save all your work. To do this, simply save your document using the CorelDraw "Save..." or "Save as..." function. The document and all stored settings (selected postprocessor, workpiece dimensions, created toolpaths, etc.) are saved with the document. You can close the document.

If you now open the same document again, CamDRAW reads all settings from the document and applies them.

8   Default settings of the application

If you start the application for the first time, some settings are predefined.

8.1   Language

The default language after installation is the language that is set and displayed in CorelDRAW. You can also change it to another language. To do this, simply go to the settings of the application and change the language. Click on "Save" and restart CorelDraw. Available languages are:

- English
- German
- Spanish
- French
- Portuguese
- Chinese
- Japanese

12.03.2025
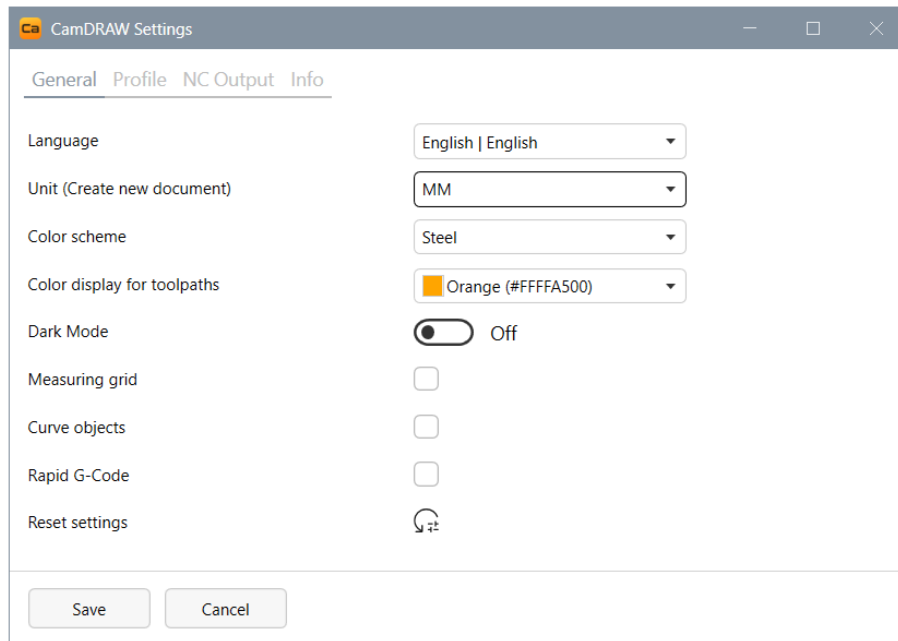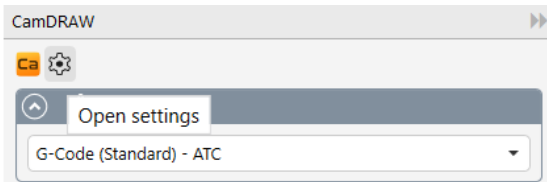
8.2 Units

In the settings you can set the unit of length (inches or millimeters) with which CamDRAW will create a new document. The set length unit of the document influences:

- Creating a new document in CorelDraw
- Text abbreviations behind the input boxes
- Input values of the individual functions
- Creation/generation of the G-code (CNC program)
- Choice of the tool list

Important information:

CamDRAW always works with the current CorelDraw document unit. Even if the unit stored in the settings differs. Thus all CorelDraw length units are supported.

You can change the length units of the CorelDraw sheet as follows:
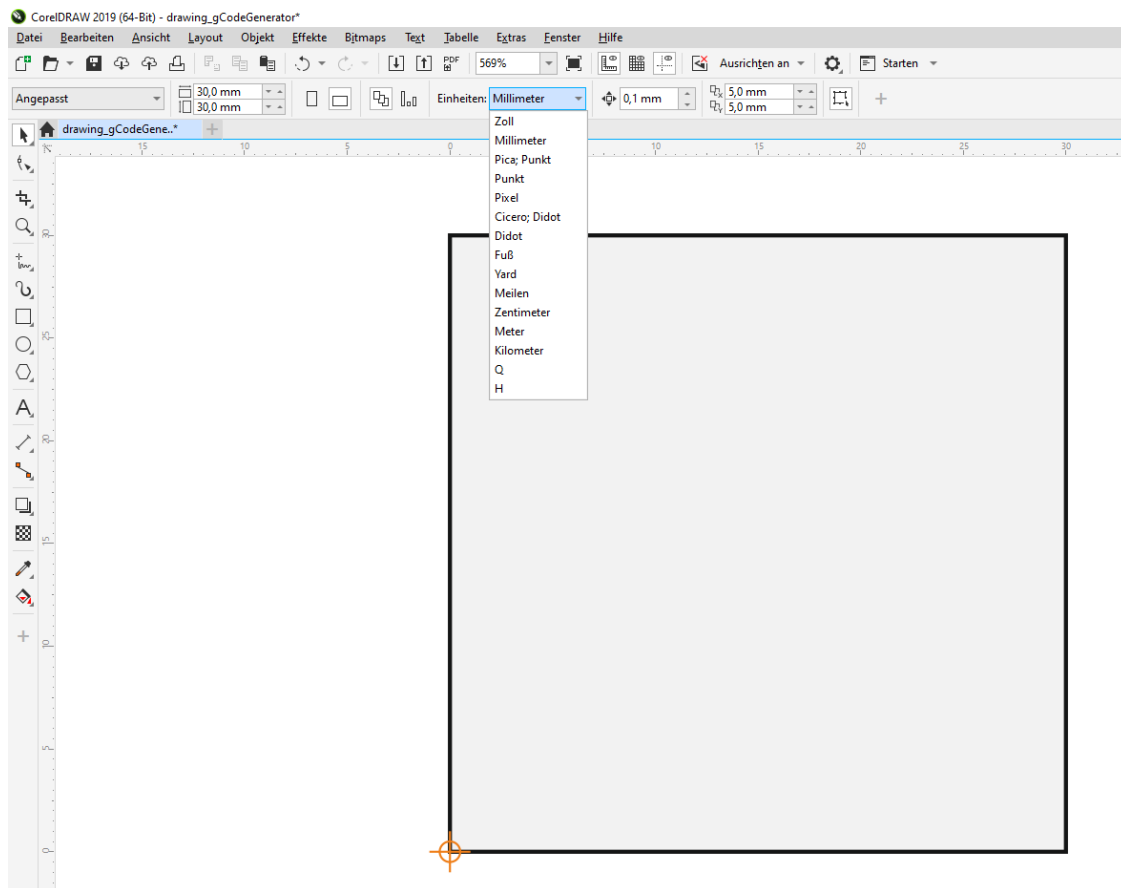
8.3 Presetting / Postprocessor

When you first start the application, a new window will automatically open where you can select a suitable preset (postprocessor) for your CNC machine.

In addition, all other preset files will be installed. You can easily import these into a profile via the CamDRAW settings.

The current list of available presets / postprocessors can be viewed on our website.

8.4 Tool list

Two different tool lists are installed with. An inch tool list, in which tools with parameters in inches are stored. Additionally a millimeter tool list, in which tools with parameters in millimeters are stored. Depending on which document unit you are working with, CamDRAW selects the appropriate list. As a rule, you always work with one unit (document unit = machine unit = tool unit).

9 Application directories

9.1 Application settings

All application settings of CamDRAW are located in:

"C:\Users\AppData\Roaming\CamDRAW 4"

These are:

- CamDRAW.ini (application settings)
- Tools_INCH.tdb (Tool List Inch)
- Tools_MM.tdb (Tool List Millimeter)
- Log.txt (log messages of the software)
- Backup" folder (application data is backed up each time the application is launched)
- Folder "postprocessors" (Here are the presets files stored in the profiles - postprocessor settings. If new postprocessor files are imported in the application, the file is copied to this folder).

9.2 Installation directory

CamDRAW is installed in:

32Bit: "C:\Program Files (x86)\Corel\CORELDRAW VERSION\Programs\Addons\CamDRAW 4"

64Bit: "C:\Program Files\Corel\CORELDRAW VERSION\Programs64\Addons\CamDRAW 4"

In the folder "postprocessors" there are the standard postprocessors, which are installed. In the folder "ToolList" are the standard tool lists, which are installed.

10 Basic functions

This chapter describes the functions of CamDRAW.

10.1 Workpiece data

The menu (mask) "Workpiece" is intended for the workpiece parameters. Various data are defined there depending on the machining type (postprocessor). The workpiece data input is not mandatory. It is only intended to help define the workpiece. You can also define the workpiece using CorelDraw's own functions (document options). For example, you can resize your CorelDraw page. Also you can define the zero point (rulers) via CorelDraw (double click on the rulers).

The first input field for the workpiece dimension is always the CorelDraw X axis (X direction). The second input field is always the CorelDraw Y axis (Y direction).

Flat machining

For flat machining operations, the dimensions and the X-Y zero point can be set here. The width of the workpiece, referred to here as "Width", is entered at the very top. This defines the sheet size in the X direction. The height of the workpiece, here called "Height", defines the sheet size in Y-direction. The names of the workpiece dimensions, here called "Width" and "Height" can be changed in the postprocessor settings. For example, you could designate the X dimension as "Width" and the Y dimension as "Length". Depending on which workpiece is present, it makes sense to adjust the designations.

Once the dimensions are set by values in the two input fields, CamDRAW creates or modifies the CorelDraw page dimensions.

The X-Y zero point is set by clicking on the corresponding button. It is indicated by an black crosshair on the CorelDraw page. In addition, the zero point can be shown or hidden.
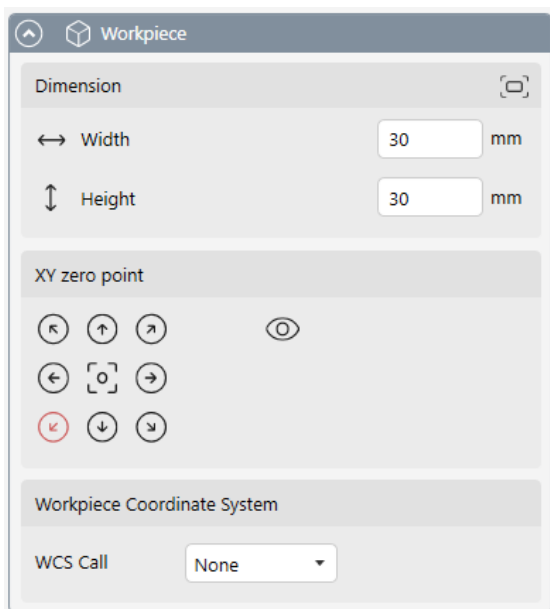
Rotational machining

For rotation editing, some additional inputs are supported. Here, too, the designations (X and Y direction), here designated as "Length" and "Circumference", can be adapted. "Length" designates here the length of the cylindrical workpiece. "Circumference" designates the circumference of the cylindrical workpiece. The workpiece dimensions are always displayed on the CorelDraw page with the unrolled circumference. I.e. you see the entire circumference/shell area of the cylindrical workpiece.

In addition to entering a circumference, you can also enter a workpiece diameter. To do this, activate the checkbox in front of the input field.

For the most flexible adaptation to your workpieces or machine, the circumference input/diameter input (checkbox) can be swapped in the postprocessor settings. This is necessary, for example, if your rotary axis has a different orientation on the machine (X or Y machine axis). If you have any questions, please contact Camdraw. We can gladly make adjustments to the postprocessor to suit your application.

Under the item "Z zero point" you can set whether the Z zero point is located on the cylinder surface or the cylinder axis.

Under the item "Machining above/below cylinder axis" you can set whether there is machining on the outer circumference or on the inner circumference of the workpiece. This is necessary, for example, if you want to perform internal machining (internal engraving) of jewelry rings.
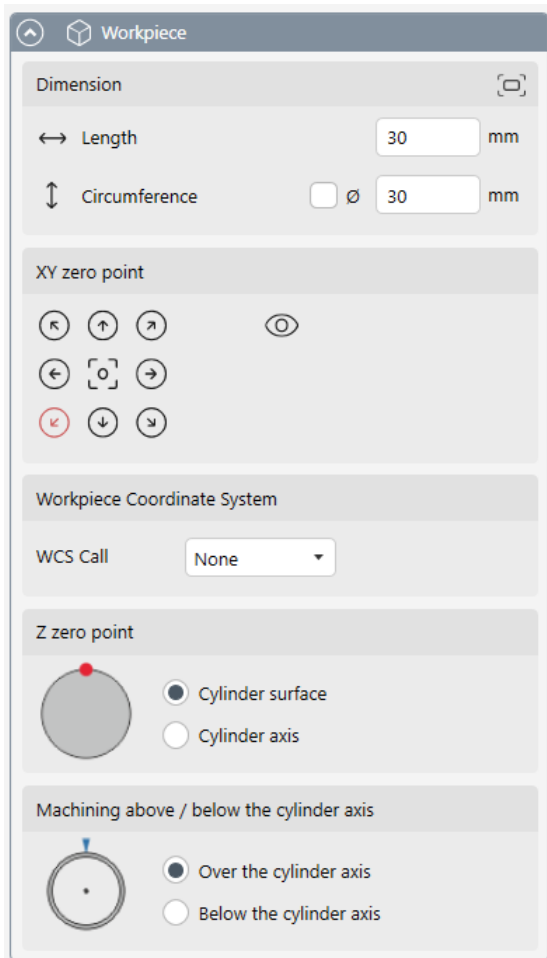
10.2 Create toolpaths

Under the menu (expander) "Toolpaths" you can create your machining operation and toolpaths.

Please select the corresponding curve and drawing objects in CorelDraw before each creation of an operation. Adding an operation will then apply the selected drawing objects for the creation. If you want to create G-code for your whole CorelDRAW document, simply select all drawing objects and add the corresponding operation. **The methods and tool operations described in this chapter are always recommended for creating the NC program.**

10.2.1 Contour machining / contour milling

Open and closed curve objects are supported. Open e.g. a freehand line, closed e.g. a rectangle.

Clicking on "contour machining" opens the window (input mask) for contour machining. Once the window is open, you will see a preview of the current operation with its settings on the CorelDraw page. Each change causes an adjustment of the preview.
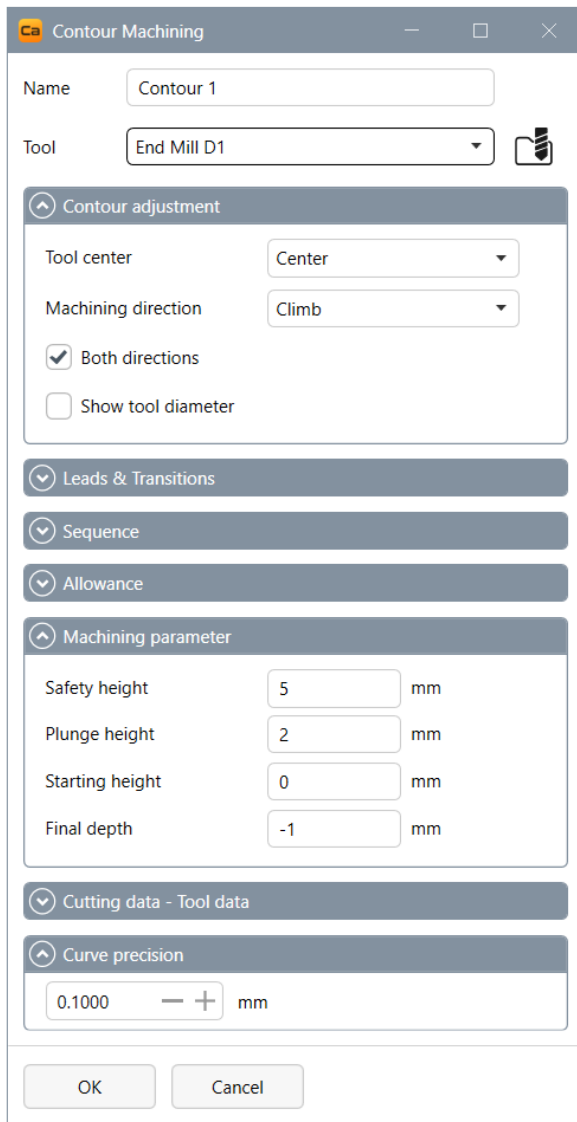
NAME: Name of the machining operation

TOOL: Select tool / open the tool list

COUNTOUR ADJUSTMENT
Center / left or inside / right or outside
Climb or conventional milling
Both directions (climb milling and conventional milling)
Show tool diameter

LEADS & TRANSITIONS
Here you can define leads and transitions for contour machining

SEQUENCE
Automatically optimize or keep the order of objects on layer

ALLOWANCE
Setting an allowance to the contour

MACHINING PARAMETERS
Safety height
Immersion height
Start height
Depth

CUTTING DATA – TOOL DATA: Tool parameters

CURVE PRECISION
Specifies the accuracy (length unit) of the interpolation for curves

12.03.2025

NAME

Sets the name of the machining operation.

TOOL

Select the tool for the operation here. All tools from the tool list are automatically listed here. In addition, you can open the tool list.

COUNTOUR ADJUSTMENT

Here you can make settings for the contour path. The contour path can be centered / left or inside / right or outside on your selected curve. In addition, you can set climb milling or up-cut milling. The "Both directions" option activates the use of climb milling and up-cut milling. You also have the option to display the exact tool diameter. For this, CamDRAW draws the contour path with a path thickness of the tool diameter.

LEADS & TRANSITIONS:

With this option you have the possibility to set approaches and departures for contour machining. The contour path is then extended accordingly by line segments or arc segments. You can set approach and departure separately from each other.

Leads & Transitions	
Position	15 — + %
Hold tool down	☐
Lead-In	☑
Tangential	☑
Lead-In radius	0.6 mm
Lead-In Sweeping angle	90 Degree
Lead-In distance	2 mm
Turn over Lead-In	☐
Lead-Out	☑
Like Lead-In	☑

Position	Defines the position of the approaches and exits for closed contours as a percentage. The percentage refers to the contour curve. 0 % contour start, 100 % contour end
Hold Tool down	Keeps the tool down when approaching the next cutting plane. No safety height is approached. Counts only for the same contour/drawing object. Caution: When machining open contours, the start and end points are approached directly without a safety height.
Lead-In	Activates the Lead-In for the selected contour(s)
Tangential	Activates a tangential Lead-In. The Lead-In then consists of a circular arc and a linear segment.
Lead-In radius	Lead-In radius

Lead-In Sweeping Angle	Angle at which the linear Lead-In segment is at the contour
Lead-In distance	The length of the linear Lead-In segment
Turn over Lead-In	Turns the Lead-In to the other side (rotated 180 degrees)
Lead-Out	Activates the Lead-Out
Like Lead-In	Sets the Lead-Out in the same way as the Lead-In. If disabled, then all settings of the Lead-Out can be set separately/independently to the Lead-In.

SEQUENCE

Here you can define the order of processing. With "Automatic", CamDRAW optimizes the toolpath and determines the order in which the individual contours are processed. With "As on layer", exactly the order of the drawing objects on the CorelDraw layer is followed. Thus you have direct influence on the sequence.

ALLOWANCE

With a contour allowance you can adjust the distance to your drawing object. This is intended for roughing/finishing.

MACHINING PARAMETERS

The cutting data of the operation can be entered here. These are:

Safety height: Z safety level for approach, retract and rapid traverse above the workpiece.

Plunge height: Z plunge plane. From here on, the plunging feed rate of the tool is used. Material plunging usually takes place at a lower feed rate.

Start height: Z-level, from when machining starts. Mostly zero, when the Z zero point is on the workpiece surface.

Depth: Z-plane of the last cut plane (depth of machining).

CURVE PRECISION

Here you can set the curve precision. All curves of the toolpaths are approximated by straight segments. The length of these segments is the curve precision. With a high value, the individual segments are longer and the precision is lower. At a low value, the individual segments are shorter and the precision is higher. With a high precision, the creation of the NC program can take longer, among other things.

Example contour path centric
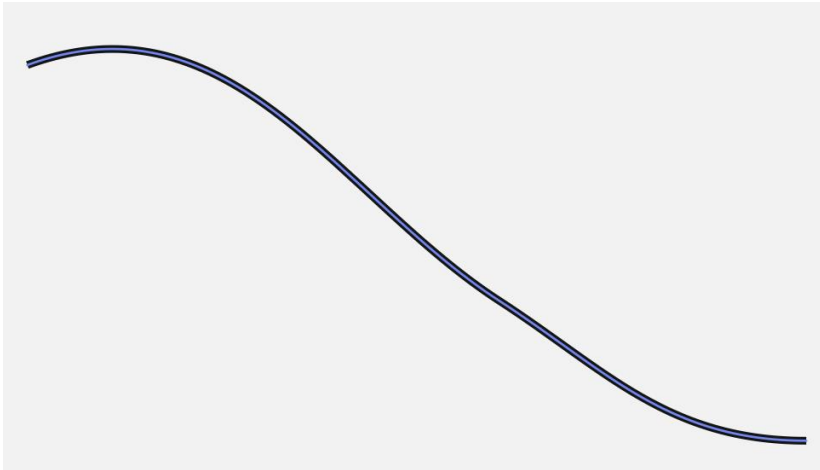
Example contour path left
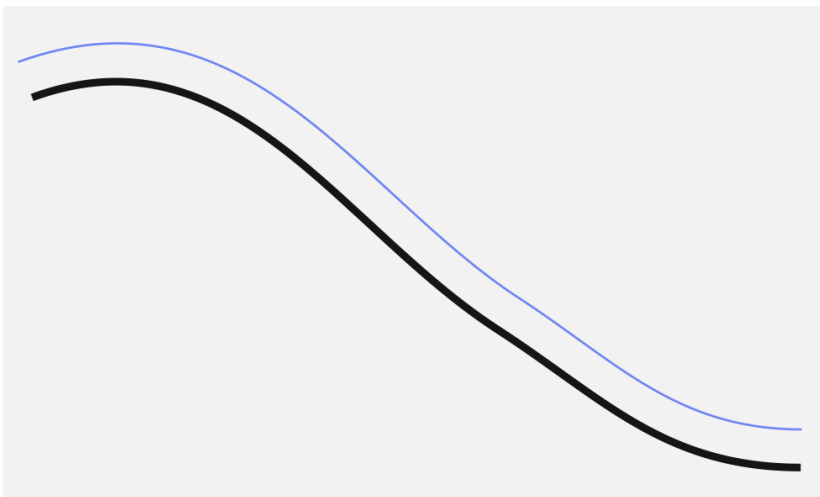
Example contour path right

12.03.2025
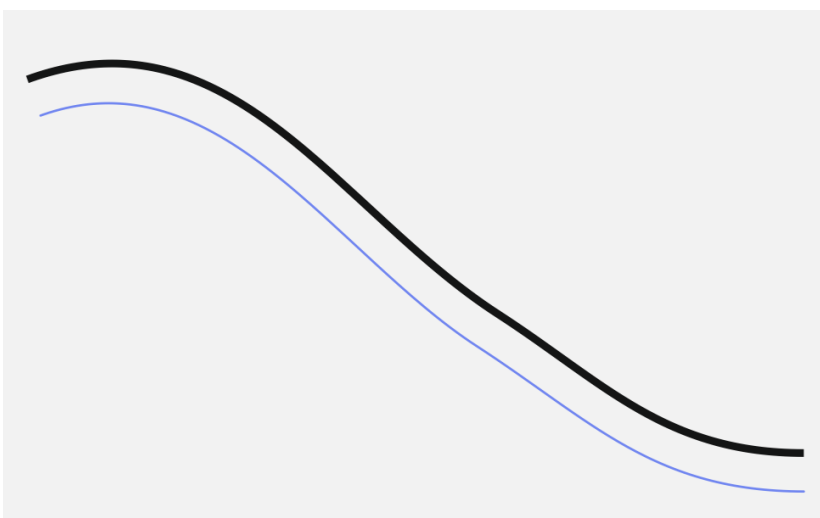
Example contour path inside (without/with display of tool diameter)
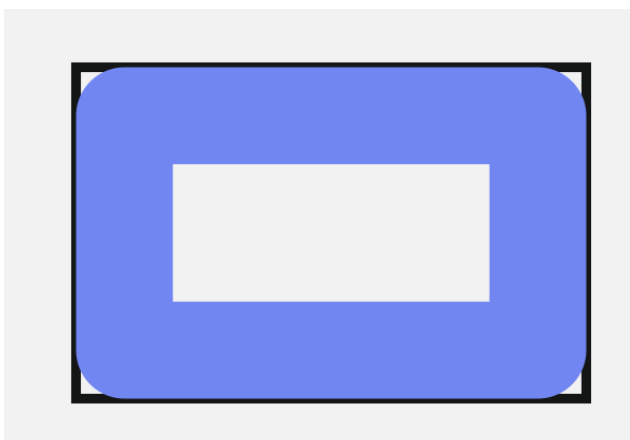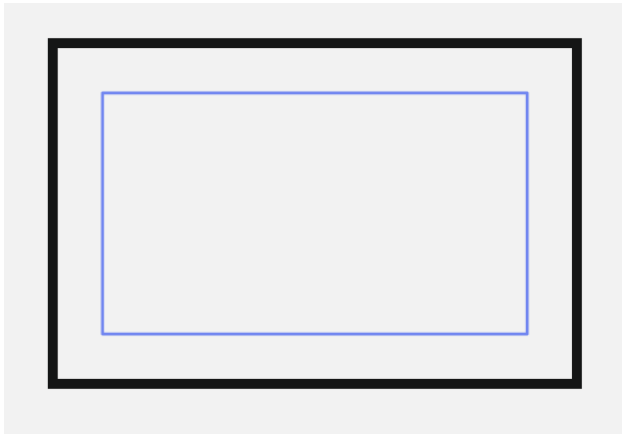
Example contour machining of combined drawing objects

With CamDRAW you have the possibility to create contour operations with combined curve objects. Once combined curves are present, the application automatically creates contours according to the intersections of the combined curves.

Here in the example a contour machining from the inside. Without and with display of the tool diameter.
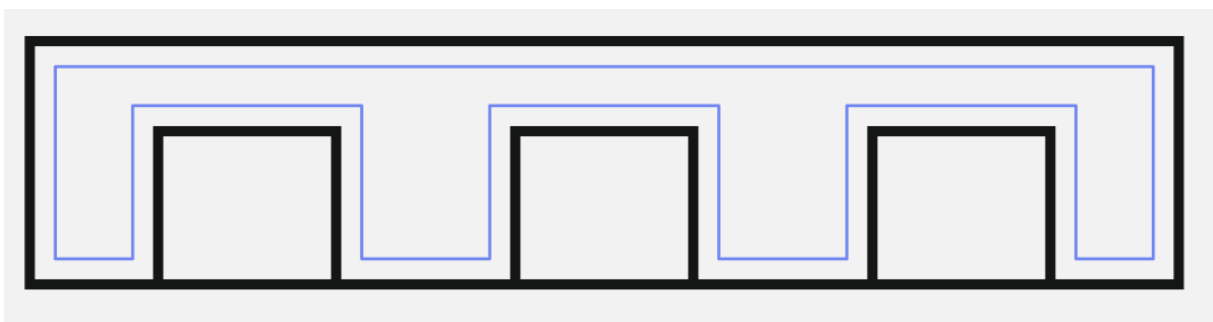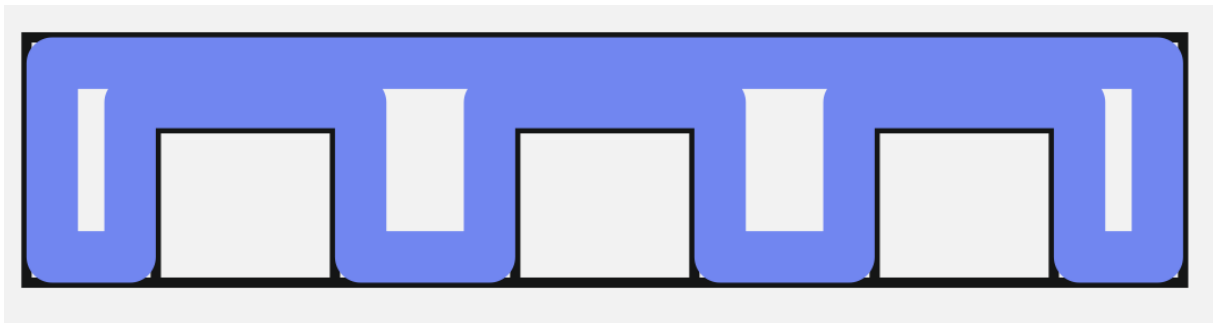
Example contour editing of text objects

You can use CamDRAW to create contour edits of any text drawing objects. Supported are, as with other curves, contours with center/inside/outside.

Example Lead-In & Lead-Out with 45 degree sweeping angle (inside)
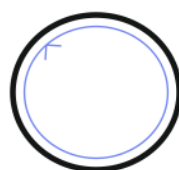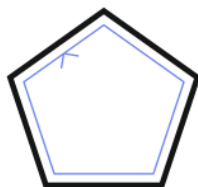
Example Lead-In & Lead-Out with 45 degree sweeping angle (outside)
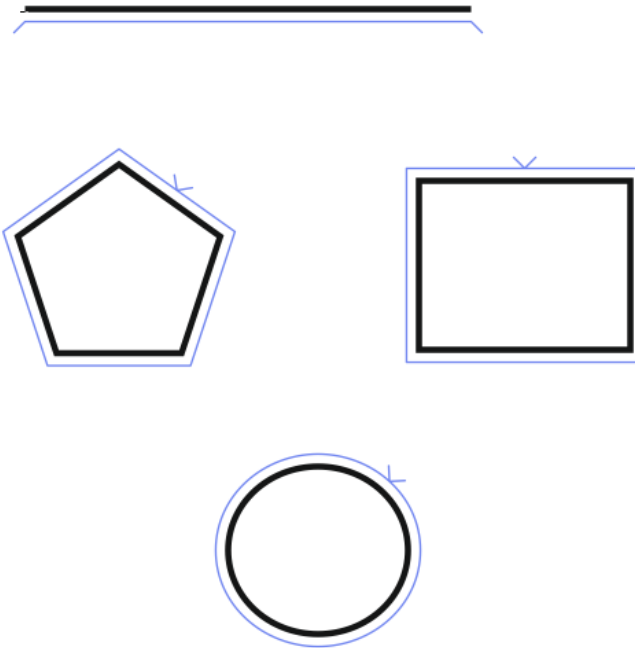
Example of tangential entry and exit with 90 degree angle
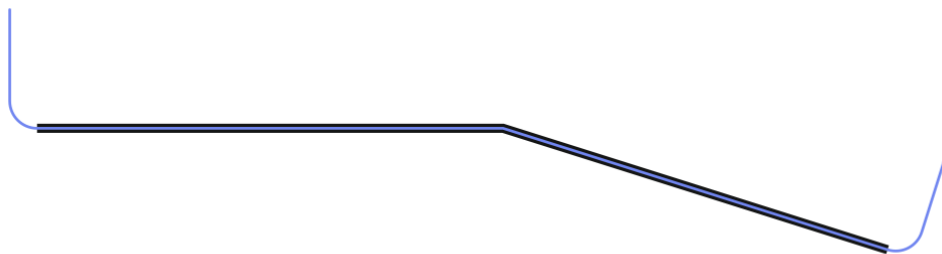
10.2.2 Pocket machining / pocket milling / clearing

Only closed curves/contours are supported when creating pocket operations. I.e. drawing objects where the X-Y position of the start node corresponds to the X-Y position of the end node. This is the case, for example, with a circle or rectangle.

First select your drawing object and then click on "Pocket machining". Again, a pocket machining input screen will open in a new window. You will see a preview of the current toolpath in the CorelDraw document.
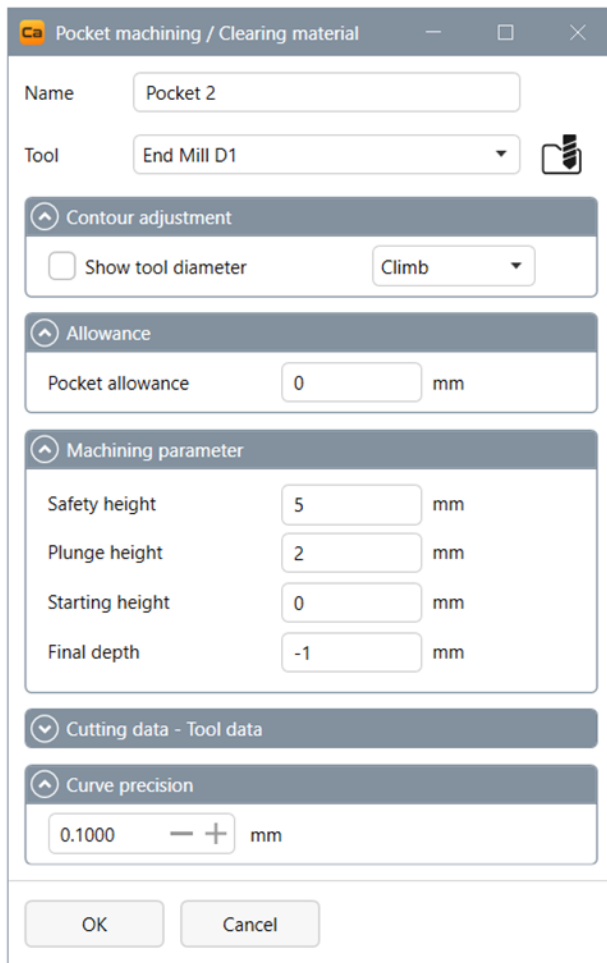
Pocket machining / Clearing material	**NAME:** Name of the machining operation
Name: Pocket 2	
Tool: End Mill D1	**TOOL:** Select tool / Open tool list
Contour adjustment	
☐ Show tool diameter — Climb	**CONTOUR ADJUSTMENT**
	Show tool diameter
	Climb or conventional milling
Allowance	
Pocket allowance — 0 mm	**ALLOWANCE**
	Contour allowance (roughing/finishing)
Machining parameter	
Safety height — 5 mm	**MACHINING PARAMETERS**
Plunge height — 2 mm	Safety height
Starting height — 0 mm	Immersion height
Final depth — -1 mm	Start height
	Depth
Cutting data - Tool data	**CUTTING DATA – TOOL DATA:** Tool parameters
Curve precision	
0.1000 — + mm	**CURVE PRECISION**
	Specifies the accuracy (length unit) of the interpolation for curves
OK Cancel	

30

12.03.2025

NAME

Sets the name of the machining operation.

TOOL

Select the tool for the operation here. All tools from the tool list are automatically listed here. In addition, you can open the tool list to make adjustments to the tool parameters.

CONTOUR ADJUSTMENT

Here you can make settings for the pocket contour. You have the possibility to display the exact tool diameter. For this, CamDRAW draws the contour path with a path thickness of the tool diameter. In addition, you can set climb milling or up-cut milling.

ALLOWANCE

With a contour allowance you can adjust the distance to your drawing object. This is intended for roughing/finishing.

MACHINING PARAMETERS

The cutting data of the operation can be entered here. These are:

Safety height:	Z safety level for approach, retract and rapid traverse above the workpiece.
Plunge height:	Z Plunge plane. From here on, the plunging feed rate of the tool is used. Material plunging usually takes place at a lower feed rate.
Start height:	Z-plane, from when machining starts. Mostly zero, when the Z zero point is on the workpiece surface.
Depth:	Z-plane of the last cut plane (depth of machining).

CURVE PRECISION

Here you can set the curve precision. All curves of the toolpaths are approximated by straight segments. The length of these segments is the curve precision. With a high value, the individual segments are longer and the precision is lower. At a low value, the individual segments are shorter and the precision is higher. With a high precision, the creation of the NC program can take longer, among other things.

Example pocket machining (without/with display tool diameter) - rectangle
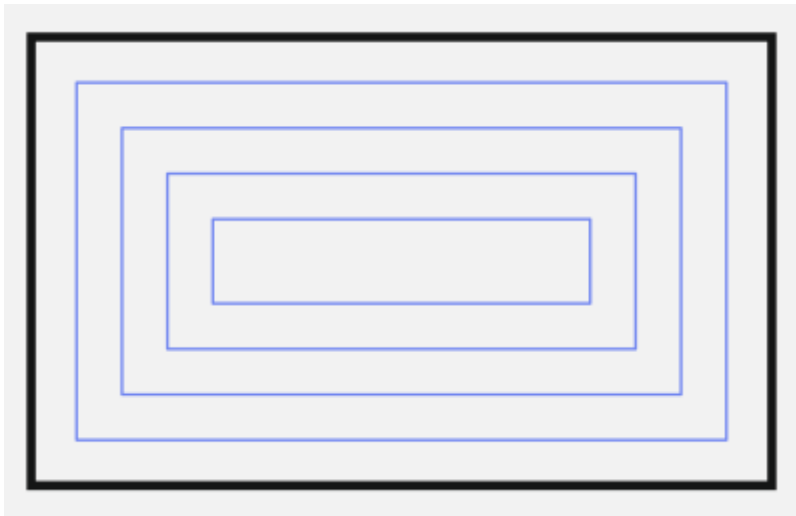
Example pocket machining - ellipse
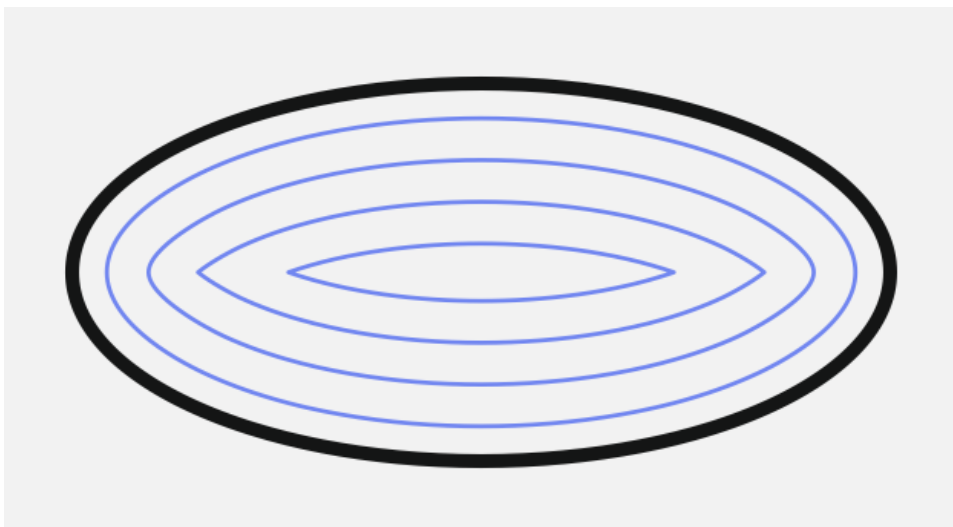
Example pocket machining of combined objects (islands)

Pocket edit paths can be created from any drawing objects. They only have to be closed! E.g. you can combine several drawing objects (curves) in CorelDraw and then create a pocket machining.

Here two rectangles are combined. One rectangle lies completely inside the other.
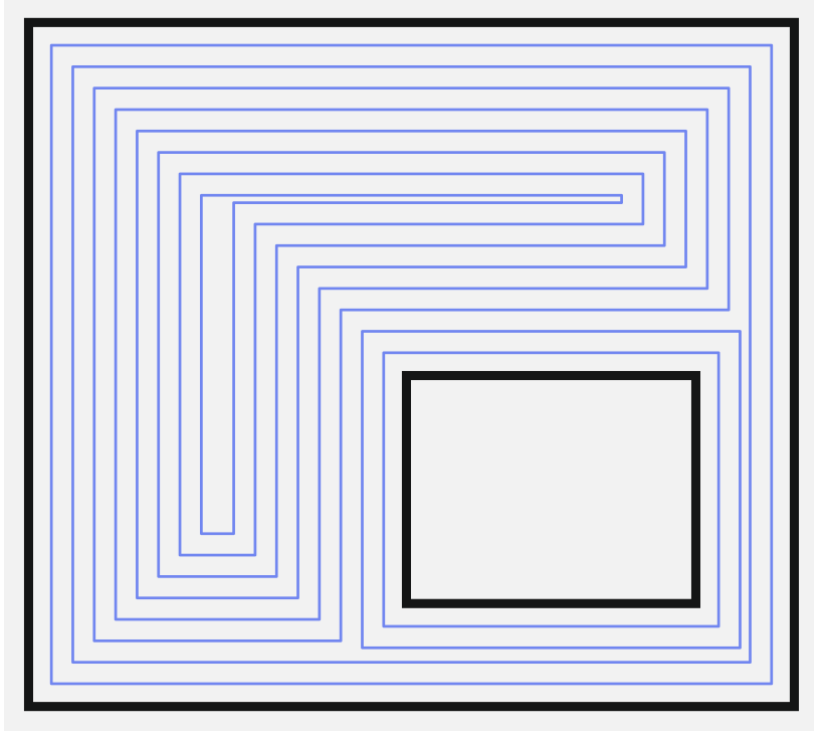
Here the one rectangle is a bit outside.

12.03.2025
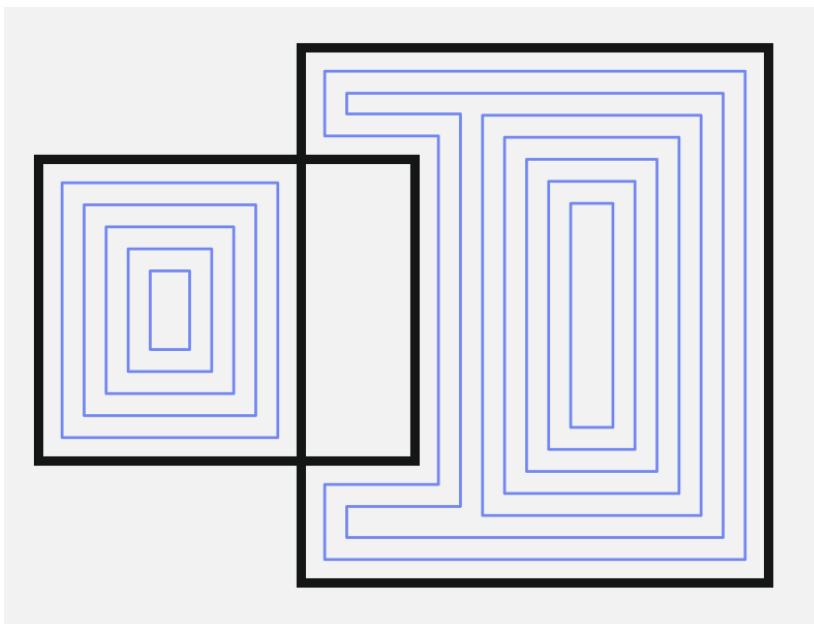
Here more than two rectangles are combined. CamDRAW creates the appropriate pocket path so that your CNC machine can get the material out. Example without and with tool diameter display. The tool diameter display is a good way to check where and how much material is taken out. In addition, you can see if there is any material left over somewhere. Here, for example, in the corners (radius of the milling tool).
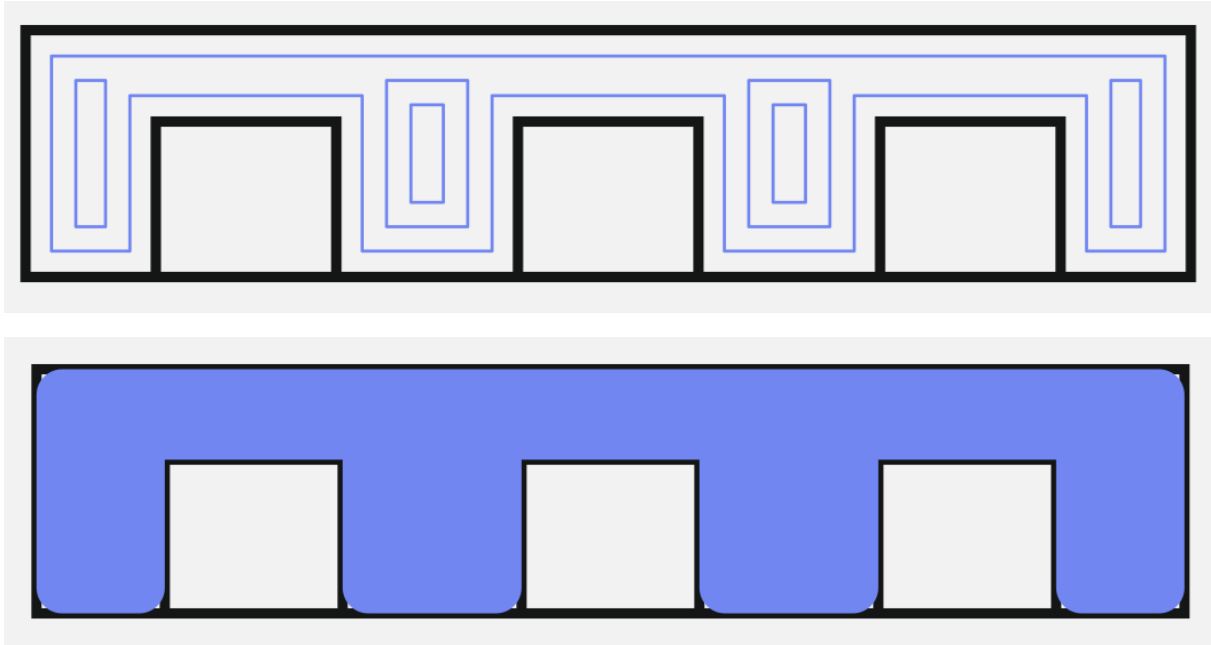
Example pocket machining of text objects (without/with display tool diameter)
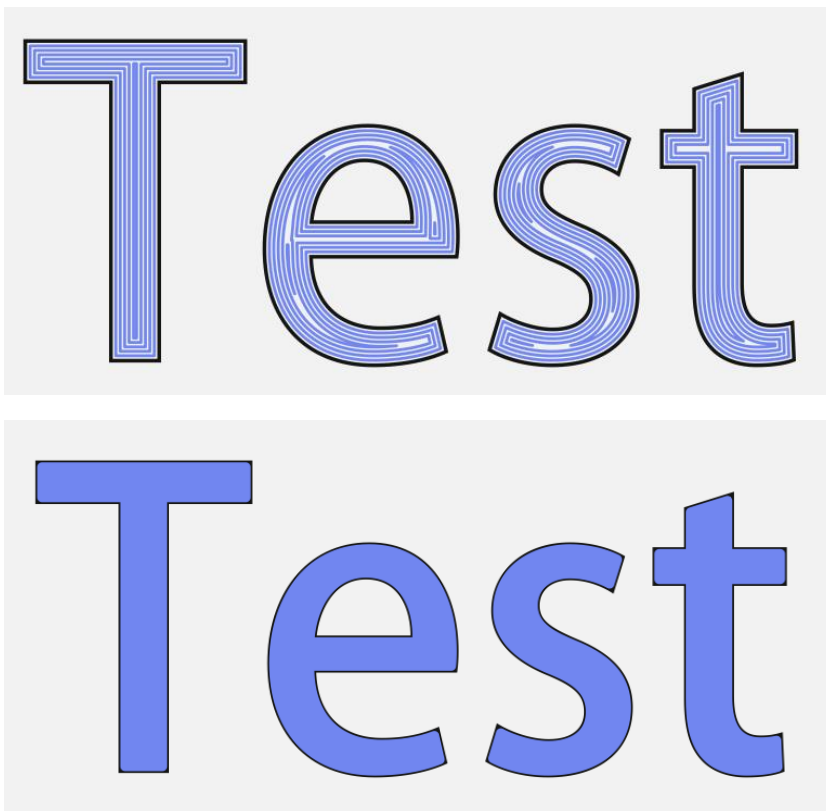
10.2.3 Drilling machining

As an additional feature, CamDRAW also supports a drilling operation. With this you can quickly and easily drill holes on your workpiece.

To create a drilling operation, select the desired drawing objects. The application always creates a hole in the center of each drawing object (outer border). All types of drawing objects are supported (open/closed/combined curves/circles/rectangles).

Info: In hole machining, combined curves behave differently from contour and pocket machining. Here, drill points are also created in the center of each individual drawing object for combined curves! Thus you have the possibility to combine the curves in case of very many bores. This makes the handling with these drawing objects much easier.

First select your desired drawing objects and then click on Create drilling operation. The input mask for the drilling operation opens. Here, too, you will directly see a preview in the CorelDraw document. The positions of drill holes are marked with a small blue dot.
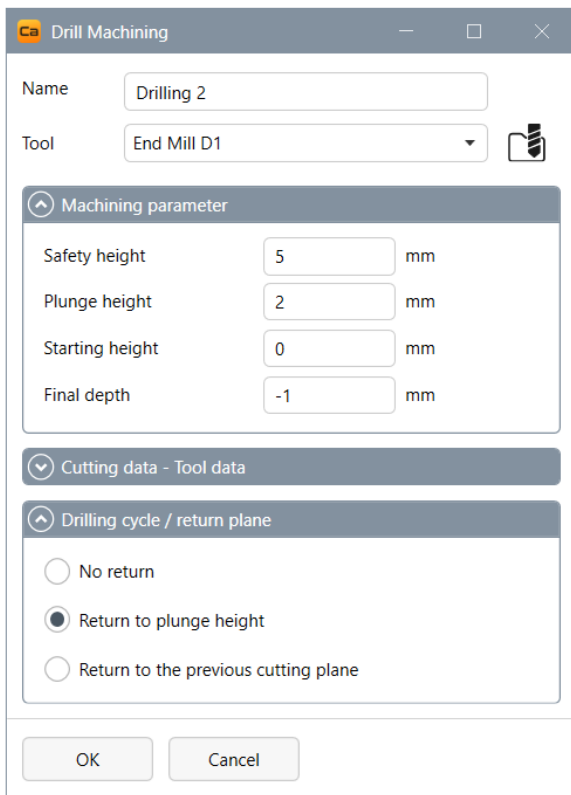
NAME: Name of the machining operation

TOOL: Select tool / Open tool list

MACHINING PARAMETERS
Safety height
Immersion height
Start height
Depth

DRILLING CYCLE
Retreat type setting
Without retreat
Retreat to immersion height
Retract to previous cut level

NAME

Sets the name of the machining operation.

TOOL

Select the tool for the operation here. All tools from the tool list are automatically listed here. Opens the tool list in a new window. If you want to add a new tool or make adjustments to the tool parameters.

MACHINING PARAMETERS

The cutting data of the operation can be entered here. These are:

Safety height: Z safety level for approach, retract and rapid traverse above the workpiece.

Plunge height: Z Plunge plane. From here on, the plunging feed rate of the tool is used. Material plunging usually takes place at a lower feed rate.

Start height: Z-plane, from when machining begins. Mostly zero, when the Z zero point is on the workpiece surface.

Depth: Z-plane of the last cut plane (depth of machining).

DRILLING CYCLE: retraction settings

The drilling cycle can be selected here. Three different cycles are supported. One is a drilling cycle without retraction. This means that the hole is drilled in one pass without a retraction plane. The second drilling cycle is performed with a pullback to the plunge height (here 0.5 millimeters). This means that the plunge height (retraction) is approached each time the new cutting plane is reached. During the third drilling cycle, a retraction to the last (previous) cutting plane takes place. By default, the drilling cycle with retraction to the plunge height is always selected.

Example of drilling machining with circular objects
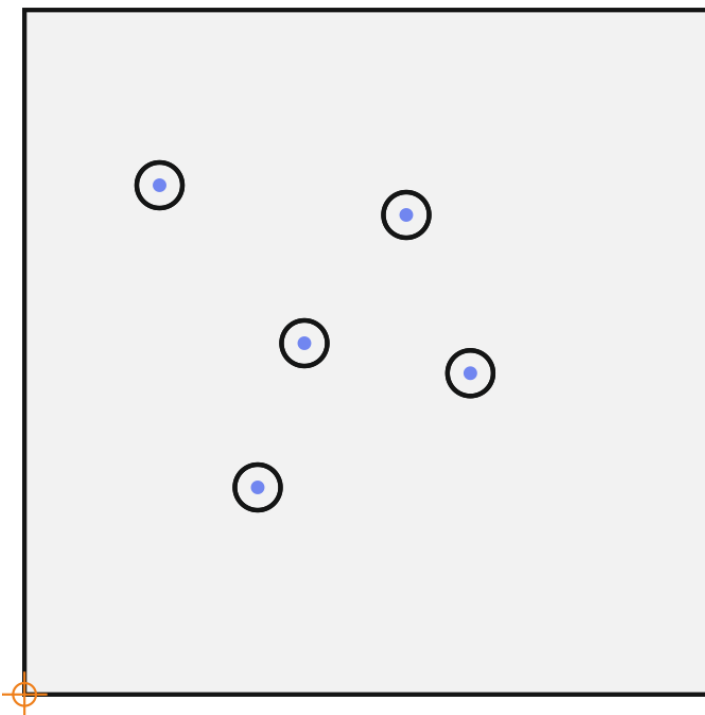
10.2.4 Contour engraving

You can engrave any contours with this machining strategy. This strategy is intended for contour machining, e.g. using an engraving depth controller. In contrast to contour milling, no depth or cutting depth is taken into account here. The engraving depth controller of your machine can be preloaded by a constant engraving height (mostly negative Z direction).
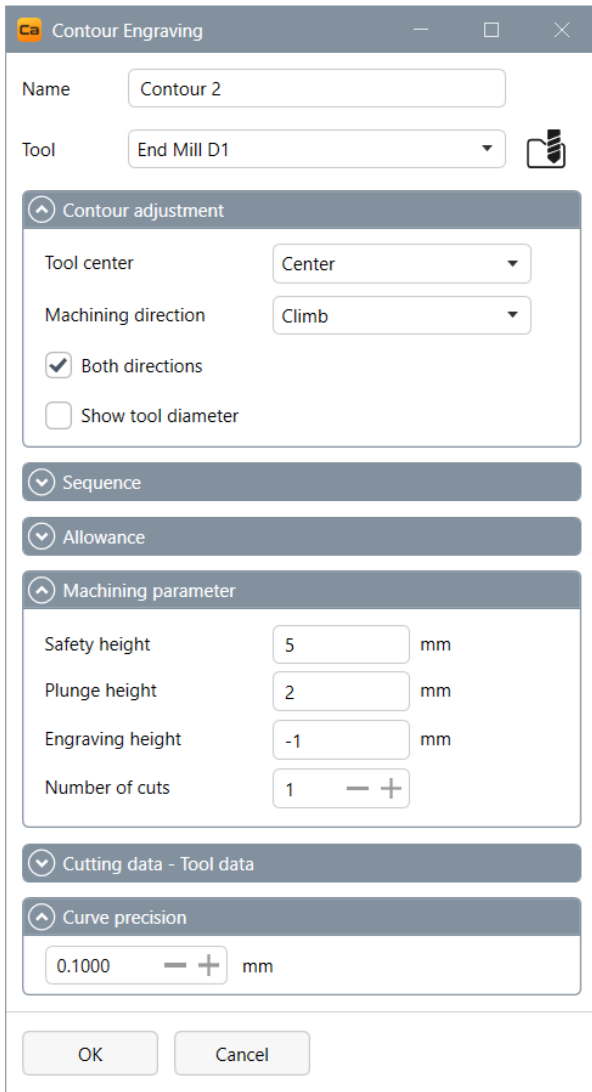
NAME: Name of the machining operation

TOOL: Select tool / Open tool list

COUNTOUR ADJUSTMENT
Centric / left or inside / right or outside
Climb or conventional milling
Both directions (Climb and conventional milling)
Show tool diameter

SEQUENCE
Automatically optimize or keep the order of objects on layer

ALLOWANCE
Contour allowance

MACHINING PARAMETERS
Safety height
Immersion height
Engraving height
Number of passes

CURVE PRECISION
Specifies the accuracy (length unit) of the interpolation for curves

NAME

Sets the name of the machining operation.

TOOL

Select the tool for the operation here. All tools from the tool list are automatically listed here. Opens the tool list in a new window. If you want to add a new tool or make adjustments to the tool parameters.

COUNTOUR ADJUSTMENT

Here you can make settings for the contour path. The contour path can be centered / left or inside / right or outside on your selected curve. In addition, you can set climb milling or up-cut milling. The "Both directions" option activates the use of climb milling and up-cut milling. You also have the option to display the exact tool diameter. For this, CamDRAW draws the contour path with a path thickness of the tool diameter.

SEQUENCE

Here you can define the order of processing. With "Automatic", CamDRAW optimizes the toolpath and determines the order in which the individual contours are processed. With "As on layer", exactly the order of the drawing objects on the CorelDraw layer is followed. Thus you have direct influence on the sequence.

ALLOWANCE

With a contour allowance you can adjust the distance to your drawing object. This is intended for roughing/finishing.

MACHINING PARAMETERS

The cutting data of the operation can be entered here. These are:

Safety height: Z safety level for approach, retract and rapid traverse above the workpiece.

Plunge height: Z Plunge plane. From here on, the plunging feed rate of the tool is used. Material plunging usually takes place at a lower feed rate.

Engraving height: Z plane from when machining starts. Mostly zero, if the Z zero point is on the workpiece surface. For engraving depth controllers, a negative value is set here.

Number of cuts: Number of passes

CURVE PRECISION

Here you can set the curve precision. All curves of the toolpaths are approximated by straight segments. The length of these segments is the curve precision. With a high value, the individual segments are longer and the precision is lower. At a low value, the individual segments are shorter and the precision is higher. With a high precision, the creation of the NC program can take longer, among other things.

10.2.5 Area Engraving

This machining strategy is intended for engraving closed drawing objects over a wide area. An engraving depth control is also supported here, which can be set with a constant negative engraving height. To do this, simply select your desired drawing objects and click on "Area Engraving" in the "Tool paths" menu. As with all other machining operations, an input mask opens here.

Only closed drawing objects are supported. These can be single or combined. Combined objects are used to create islands.

NAME: Name of the machining operation

TOOL: Select tool / Open tool list

CONTOUR ADJUSTMENT
Show tool diameter
Climb or conventional milling

ALLOWANCE

MACHINING PARAMETERS
Safety height
Immersion height
Engraving height
Number of passes

CURVE PRECISION
Specifies the accuracy (length unit) of the interpolation for curves

NAME

Sets the name of the machining operation.

TOOL

Select the tool for the operation here. All tools from the tool list are automatically listed here. Opens the tool list in a new window. If you want to add a new tool or make adjustments to the tool parameters.

CONTOUR ADJUSTMENT

Here you can make settings for the pocket contour. You have the possibility to display the exact tool diameter. For this, CamDRAW draws the contour path with a path thickness of the tool diameter. In addition, you can set climb milling or up-cut milling.

ALLOWANCE

With a contour allowance you can adjust the distance to your drawing object. This is intended for roughing/finishing.

MACHINING PARAMETERS

The cutting data of the operation can be entered here. These are:

Safety height:	Z safety level for approach, retract and rapid traverse above the workpiece.
Plunge height:	Z Plunge plane. From here on, the plunging feed rate of the tool is used. Material plunging usually takes place at a lower feed rate.
Engraving height:	Z plane from when machining starts. Mostly zero, if the Z zero point is on the workpiece surface. For engraving depth controllers, a negative value is set here.
Number of cuts:	Number of passes

CURVE PRECISION

Here you can set the curve precision. All curves of the toolpaths are approximated by straight segments. The length of these segments is the curve precision. With a high value, the individual segments are longer and the precision is lower. At a low value, the individual segments are shorter and the precision is higher. With a high precision, the creation of the NC program can take longer, among other things.

10.2.6 Contour cutting (laser/water/plasma)

The "Contour cutting" machining strategy is specially intended for laser/ plasma or waterjet cutting. For this purpose, specific lines can be set in the postprocessor for switching the laser or waterjet on or off. These commands/lines are always taken into account at the start and end of a contour and are also output in the G-Code file. Thus, for example, you can switch on the laser at the start of the contour and switch it off again at the end of the contour in order to move on to the next contour cut. To do this, select your desired objects in the drawing area and click on "Contour cutting". An input mask opens.
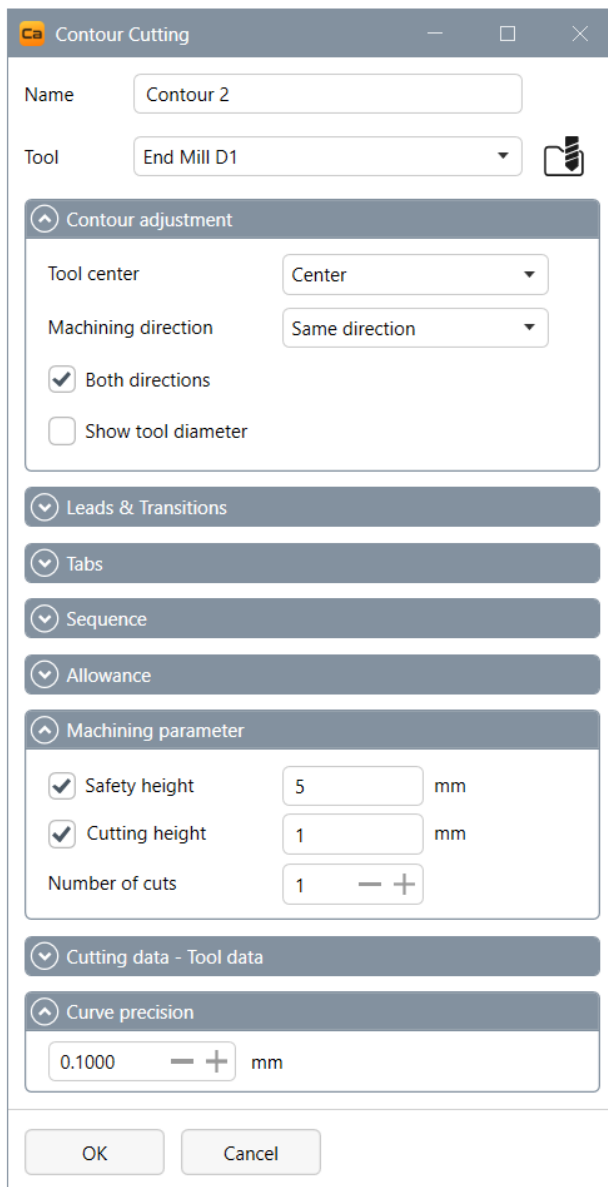
NAME: Name of the machining operation

TOOL: Select tool / Open tool list

COUNTOUR ADJUSTMENT
Centric / left or inside / right or outside
Machining direction same or opposite
Both directions (same/opposite)
Show tool diameter

LEADS & TRANSITIONS
Here you can define leads and transitions for contour cutting

TABS: Material bridges
Material bridges can be added here

SEQUENCE
Automatically optimize or keep the order of objects on layer

ALLOWANCE

MACHINING PARAMETERS
Safety height
Cutting height
Number of passes

CURVE PRECISION
Specifies the accuracy (length unit) of the interpolation for curves

12.03.2025

NAME

Sets the name of the machining operation.

TOOL

Select the tool for the operation here. All tools from the tool list are automatically listed here. Opens the tool list in a new window. If you want to add a new tool or make adjustments to the tool parameters.

CONTOUR ADJUSTMENT

Here you can make settings for the contour path. The contour path can be centered / left or inside / right or outside on your selected curve. In addition, you can change the direction of the contour path.

LEADS & TRANSITIONS:

With this option you have the possibility to set approaches and departures for contour machining. The contour path is then extended accordingly by line segments or arc segments. You can set approach and departure separately from each other.
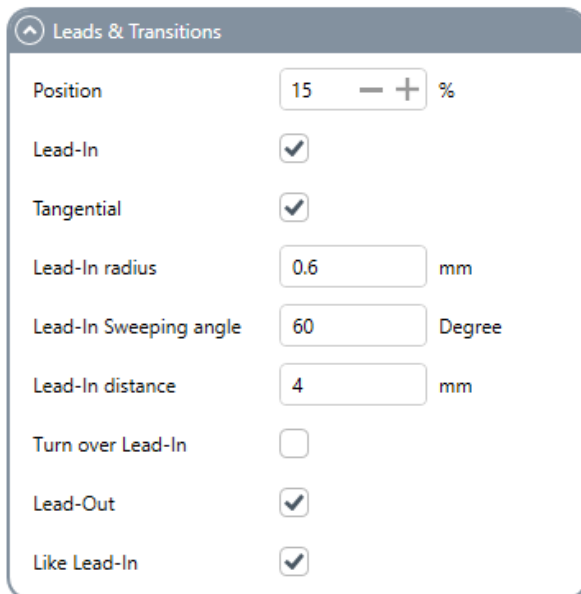
Position	Defines the position of the approaches and exits for closed contours as a percentage. The percentage refers to the contour curve. 0 % contour start, 100 % contour end
Lead-In	Activates the Lead-In for the selected contour(s)
Tangential	Activates a tangential Lead-In. The Lead-In then consists of a circular arc and a linear segment.
Lead-In radius	Lead-In radius
Lead-In Sweeping Angle	Angle at which the linear Lead-In segment is at the contour
Lead-In distance	The length of the linear Lead-In segment
Turn over Lead-In	Turns the Lead-In to the other side (rotated 180 degrees)
Lead-Out	Activates the Lead-Out

12.03.2025

Like Lead-In	Sets the Lead-Out in the same way as the Lead-In. If disabled, then all settings of the Lead-Out can be set separately/independently to the Lead-In.

TABS: Materialstege

This option allows you to add material bars to the current contour cutting operation. The cutting contours are then interrupted. This is useful if the material to be cut is to remain fixed. Once activated, the bars are displayed as red dots in the CorelDRAW document. You can move these to any position with the mouse if required. This is how you determine the tab position. You can set the following input parameters:

Width: Width of the tap

Numbers: Number of tabs

Activate/Deactivate automatic tab position

Scaling of the red dots in the CorelDRAW document (visual setting only)
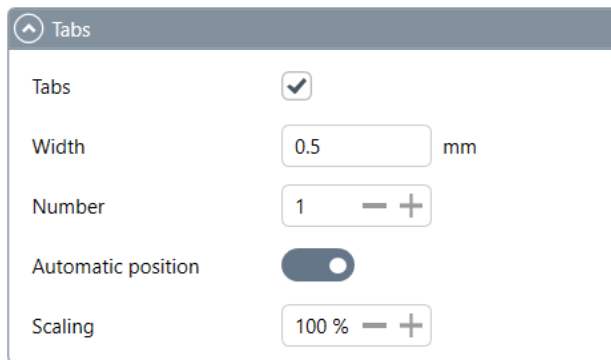
SEQUENCE

Here you can define the order of processing. With "Automatic", CamDRAW optimizes the toolpath and determines the order in which the individual contours are processed. With "As on layer", exactly the order of the drawing objects on the CorelDraw layer is followed. Thus you have direct influence on the sequence.

ALLOWANCE

You can use a contour allowance to adjust the distance to the drawing objects.

MACHINING PARAMETERS

The cutting data of the operation can be entered here. These are:

Safety height:	Z safety level for approach, retract and rapid traverse above the workpiece. Can also be deactivated if required.
Cutting height:	Z level, from when machining starts. Can also be deactivated if required.
Number of cuts:	Number of passes

CURVE PRECISION

Here you can set the curve precision. All curves of the toolpaths are approximated by straight segments. The length of these segments is the curve precision. With a high value, the individual segments are longer and the precision is lower. At a low value, the individual segments are shorter and the precision is higher. With a high precision, the creation of the NC program can take longer, among other things.

12.03.2025

Example Lead-In & Lead-Out for contour cutting
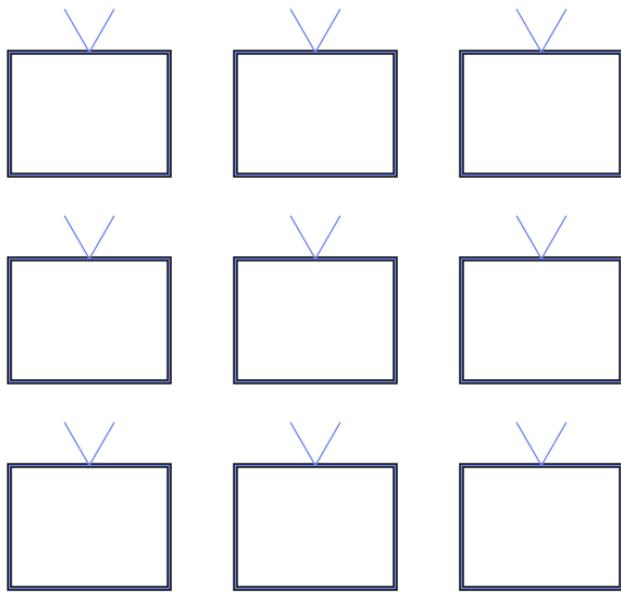
Created NC program
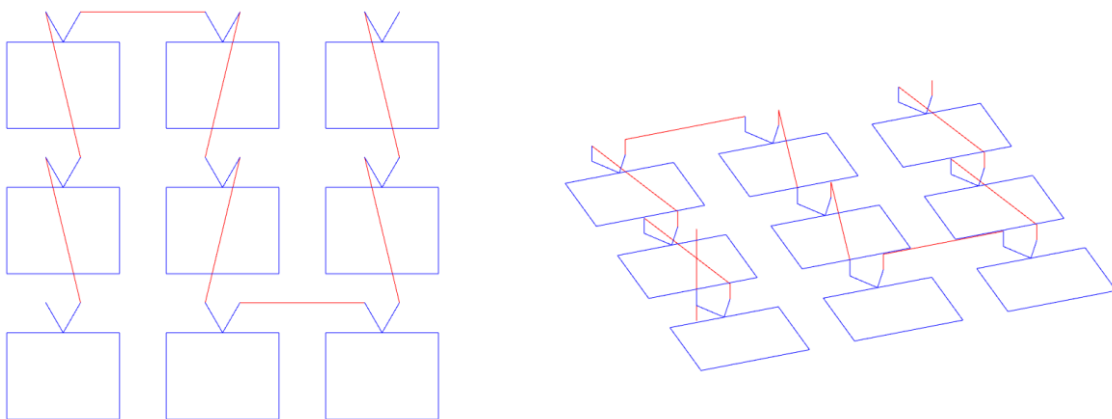
Tangential Lead-In and Lead-Out in a circular contour

12.03.2025
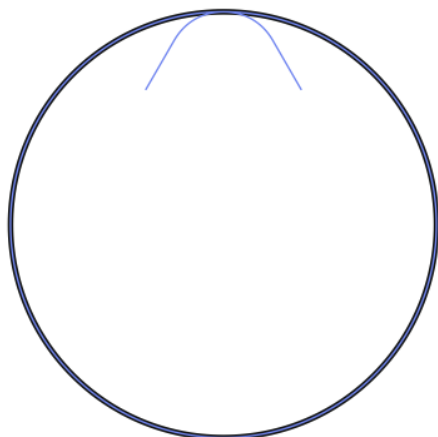
10.3 Tool list

A tool list/database is integrated in CamDRAW. In this you can create and save your machining/milling tools. Laser, plasma and water jet tools can also be defined here. After installation and the first start of the application, two tool lists are automatically created in the application settings ("C:\Users\USER\AppData\Roaming\CamDRAW 4"). These each contain 7 standard tools with default parameters.

You can open the tool list by clicking on the tool icon (under "Toolpaths"). Depending on which document unit is set, the corresponding tool list opens (inch → inch list, millimeter → millimeter list, other unit → inch list).

Once you have adjusted your tool parameters, you can save all changes by clicking on "Save". If you click on "Cancel" or close the window, no changes will be applied.

All settings in the tool list are assigned to the tool. When selecting the tool in a machining operation, these settings are automatically adopted for the operation.

The following window will be displayed:

10.3.1 Tool list tab "General settings"
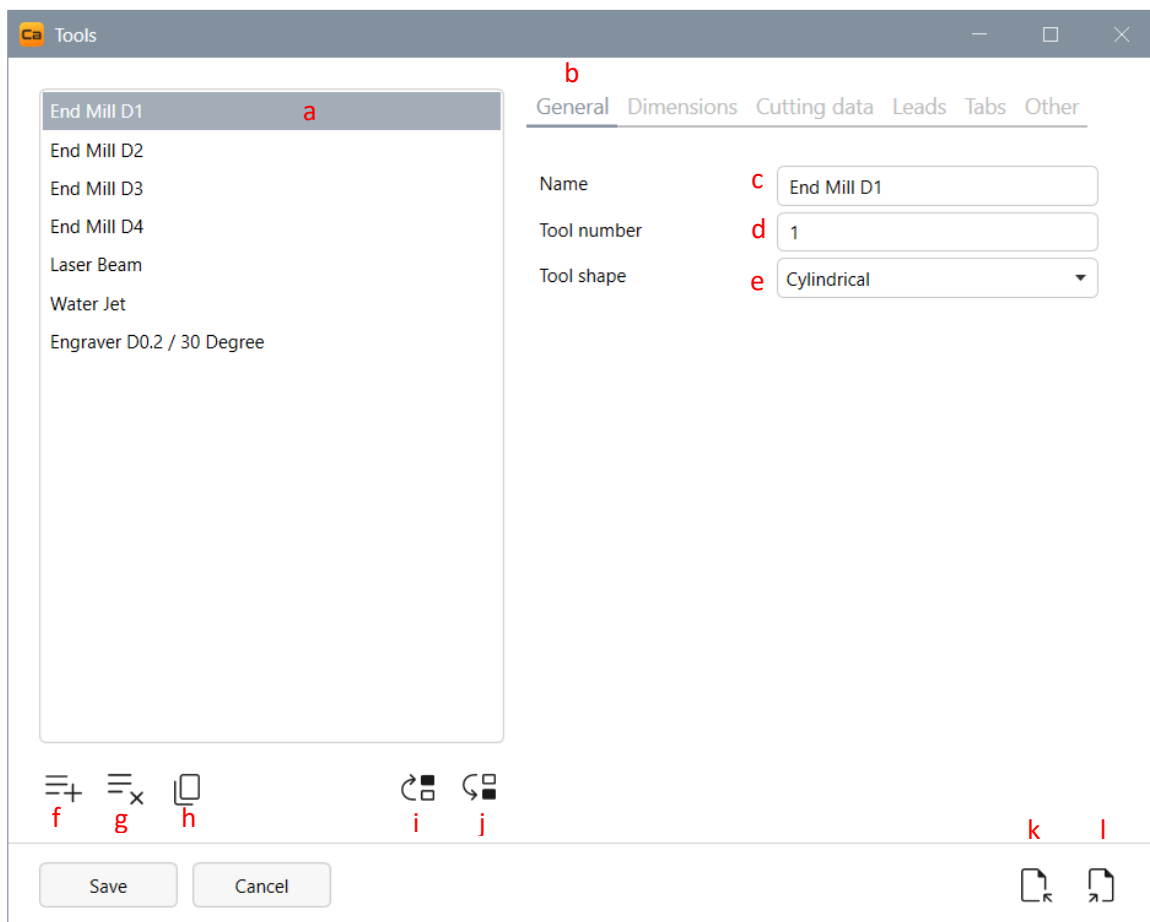
a	Currently selected tool from the tool list
b	Currently selected tab in the tool list: Various parameters can be set in the respective tab
c	Name of the tool
d	Tool number

e	Tool shape: Cylindrical or conical
f	Add new tool
g	
h	Duplicate currently selected tool
i	Move the currently selected tool one up
j	Move the currently selected tool down one
k	Importing a tool list from a file
l	Export tool list to a file

10.3.2 Tool list tab "Dimensions"
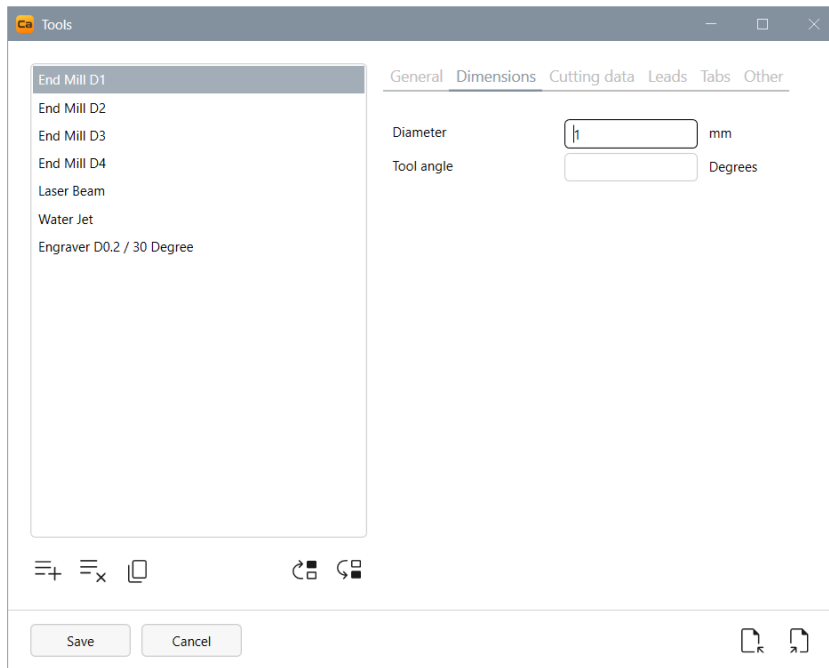
You can define the general dimensions of the tool in the "Dimensions" tab.

Diameter: Tool diameter (beam diameter for laser/plasma/water)

Tool angle: Angle of the tool if the tool shape is conical

12.03.2025

10.3.3 Tool list tab "Cutting data"
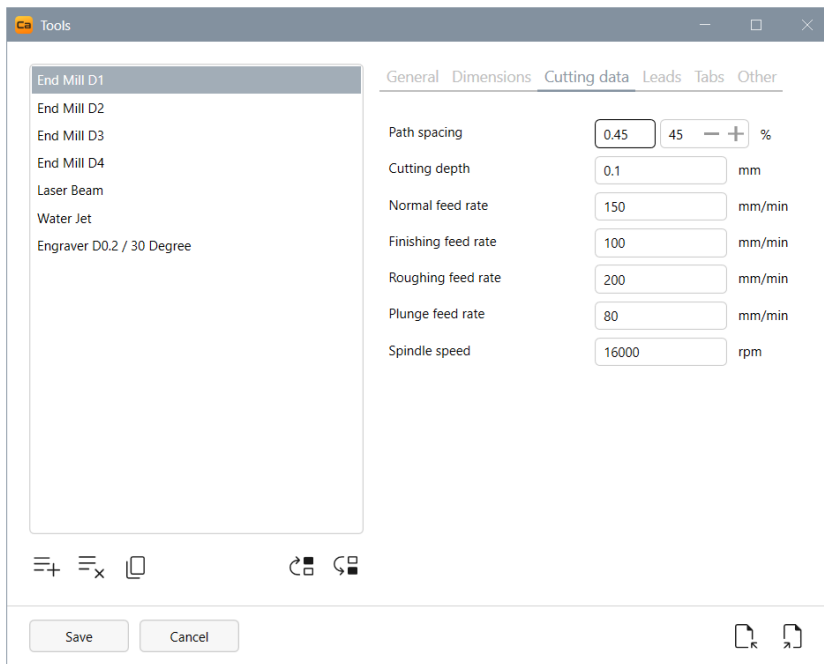
You can define the cutting data for the tool in the "Cutting data" tab.

Path spacing:	Distance of the tool paths (cross infeed) in percent for pocket machining (broaching operations). Should not be more than 50%. The percentage refers to the tool diameter.
Cutting depth:	Depth of cut / increment per pass. The total depth of the machining operation is achieved in individual steps with the depth of cut.
Normal feed rate:	Standard feed of the tool
Finishing feed rate:	Feed rate for finishing
Roughing feed rate:	Feed rate for roughing
Plunge feed rate:	Speed for plunging into the material (Z-direction)
Spindle speed:	Rotational speed of the spindle if the tool is a rotating tool

10.3.4 Tool list tab "Leads"
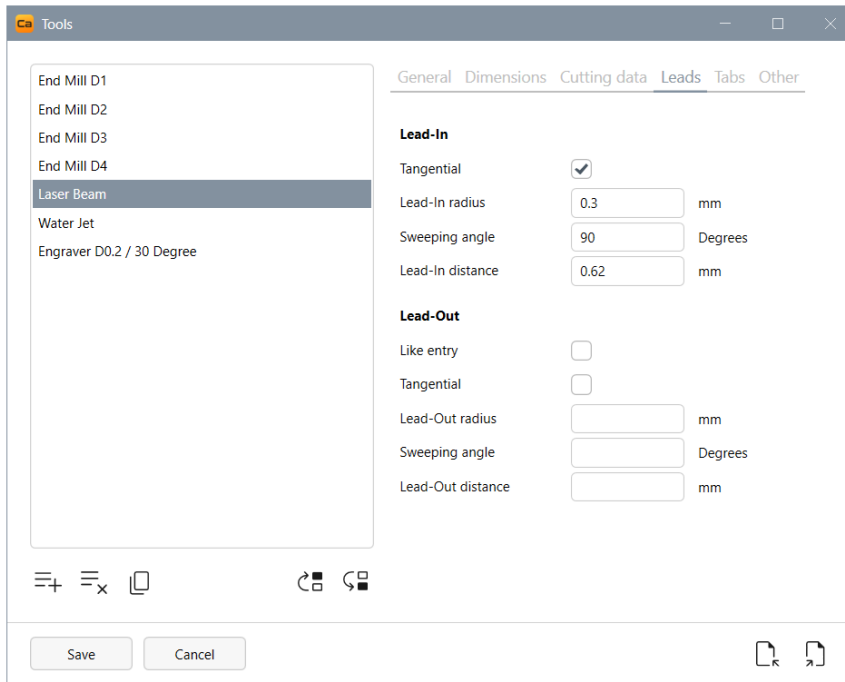
In the "Leads" tab, you can define the settings for the tool's entrances and exits.

Tangential:	Determines whether the entry or exit should be tangential (if the function is deactivated, the entry or exit is linear)
Radius:	Radius for tangential entry or exit
Sweeping angle:	Angle at which the entry and exit should take place
Distance:	Distance / length at which the entry and exit should take place

10.3.5 Tool list tab "Tabs"
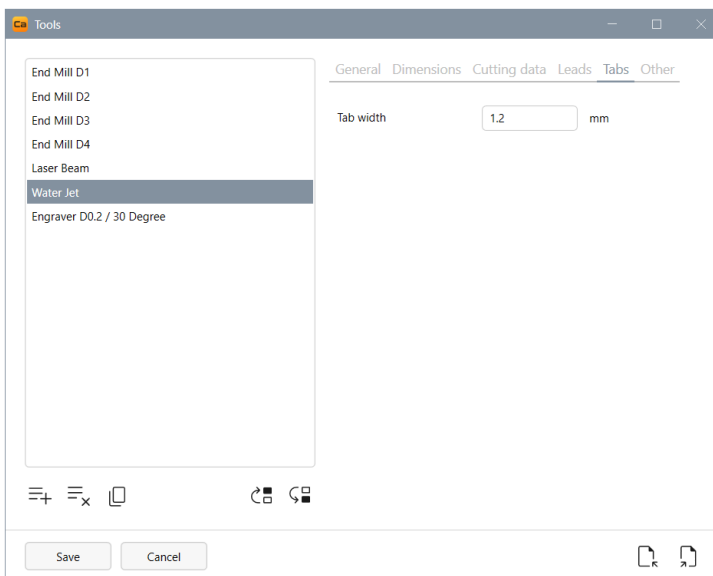
In the "Tabs" tab, you can define the settings for the material bridges of the tool.

12.03.2025

Tab width: Defines the width of the material bridge. Distance at which the individual contours are interrupted.

10.3.6 Tool list tab "Other"
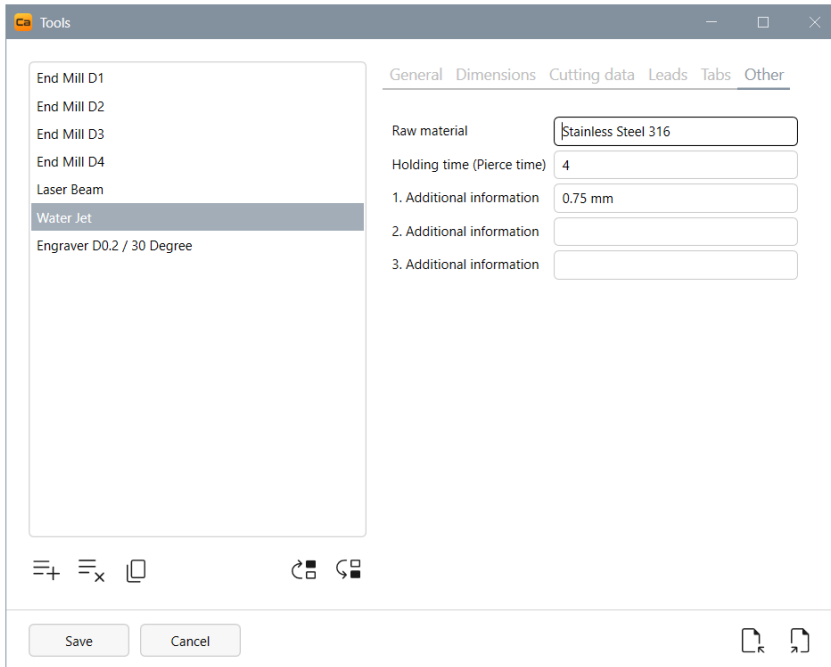
You can store further specific settings for the tool in the "Other" tab. These parameters and information are also available via placeholders (<...>) in the G-code output and can be output at a suitable location using the placeholders.

Raw material: Information about the raw material. Available via the placeholder <material>

Pierce time: Here you can define the pierce time for laser, plasma and waterjet processing. The time specifies how long the beam needs to penetrate the material at the start. Available via the placeholder <pierce_time>

Additional information 1: First additional tool information. Available through the placeholder <info1>

Additional information 2: Second additional tool information. Available via the placeholder <info2>

Additional information 3: Third additional tool information. Available via the placeholder <info3>

10.4 Display of toolpaths (overview)

After creating machining operations (click on "OK" or Enter key), they are saved and entered in the machining operation overview list. There you will find an overview of all created machining operations.
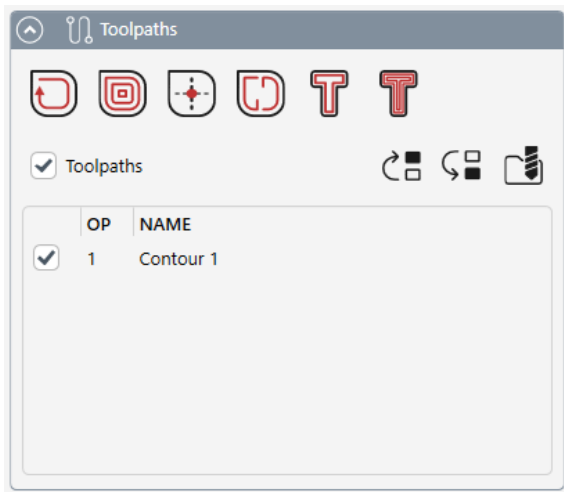
Each machining operation can be activated/deactivated by a checkbox. In addition, you can activate or deactivate all machining operations by using the "Toolpaths/Toolpaths" checkbox.

Left-click on a created operation to select it and it will be displayed in red on the CorelDraw page.
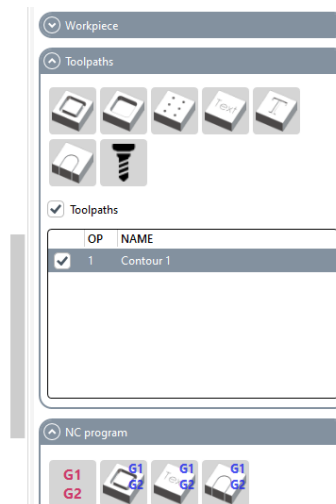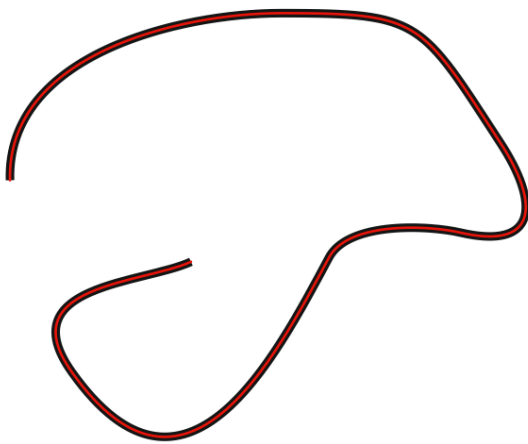
Right-clicking on an operation opens a menu with the options:

Edit operation   Reopens the window of the operation to edit it afterwards.

Generate operation   Recalculates the operation. This can be done, for example, after one or more basic character objects have been moved.

Delete operation   Deletes the selected operation.
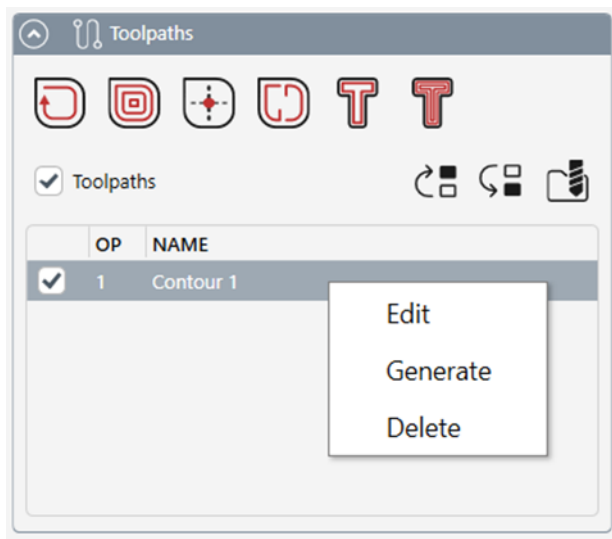
10.5 NC program

After you have created and set your machining operation, you can use CamDRAW to create the NC program (G-Code). The following menu/expander is for this purpose.
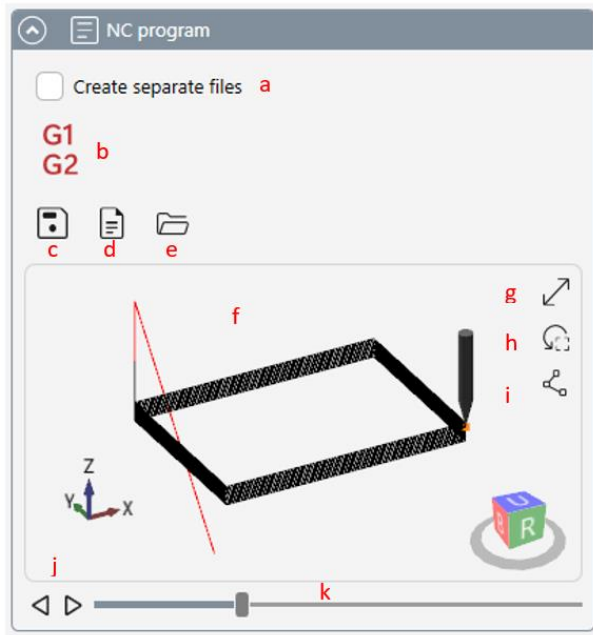
a   Create a separate file for each tool used. Useful if your CNC machine only supports one tool per file.
b   Create NC program (G-code) of the created operations
c   Exporting / saving the created NC program
d   Open / display the created NC program
e   Open the NC output directory
f   3D view of the created G-code
g   Open 3D view in a separate window (also double-click)
h   Reset 3D camera view
i   Show or hide node / support points
j   Tool simulation: single step forwards and backwards
k   Tool simulation: Slider for quick passage of the NC program

12.03.2025

10.6 Create G-Code of all drawing objects (Rapid G-Code Function)

In addition to manual programming of individual toolpaths, you can also use CamDRAW to create NC programs of all drawing objects in the current CorelDraw drawing. This is only possible for contours, i.e. the created toolpaths are located in the center of the curve objects. Three different strategies are available for this, which you can call up in the "NC program" menu/expander. If you do not see the options, you must first activate the "Rapid G-Code" option in the CamDRAW settings window. The methods and tool operations described in chapter 10.2 are preferred and recommended over the methods described in this chapter.

10.6.1 Contour machining / milling

Click on the "Contour milling" button to open the following window. All necessary parameters can be entered there. Clicking on "OK" starts the generation of the G-code.

SEQUENCE
Setting the order of editing. Automatically optimized order or numbered order.

LAYER FEEDRATE
Set the feed rate for each layer in the CorelDraw sheet.

MACHINING PARAMETERS
Set cutting data and parameters
Safety height
Plunge height
Start height
Depth
Depth of cut / increment
Plunge feed rate
Spindle speed
Tool number

CURVE PRECISION
Specifies the accuracy (length unit) of the interpolation for curves

SEQUENCE

Set the desired sequence for the individual curve objects here. The automatic tool path optimization optimizes the sequence of the partial machining according to the shortest path. With the numbered sequence CamDRAW follows the numbers of the text fields. You simply insert normal text fields into the CorelDraw workspace and enter numbers. The formatting of the numbers can be e.g. 1, 2, 3, 4, ... However, 1.1, 1.2, 1.3, 1.4, ... are also supported. For open curves, you can define the start of the contour by the position of the text field (start or end node). At the start of the contour, machining or plunging starts.

Requirement for a numbered sequence:

- Formatting of the text fields with 1, 2, 3 or 1.1, 2.1, 3.1
- Text field must be on the curve (touch)
- Text field must be on the same layer/level as the curve
- Text field and curve must have identical colors

Example
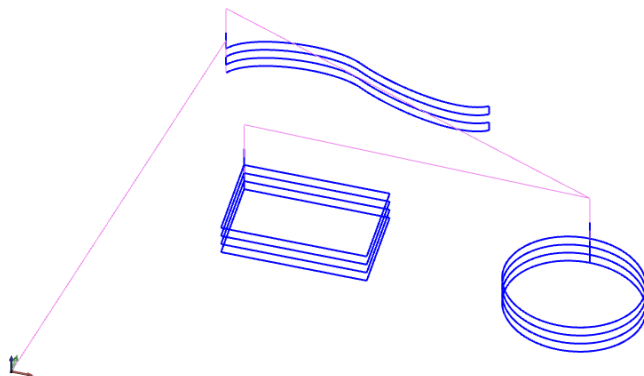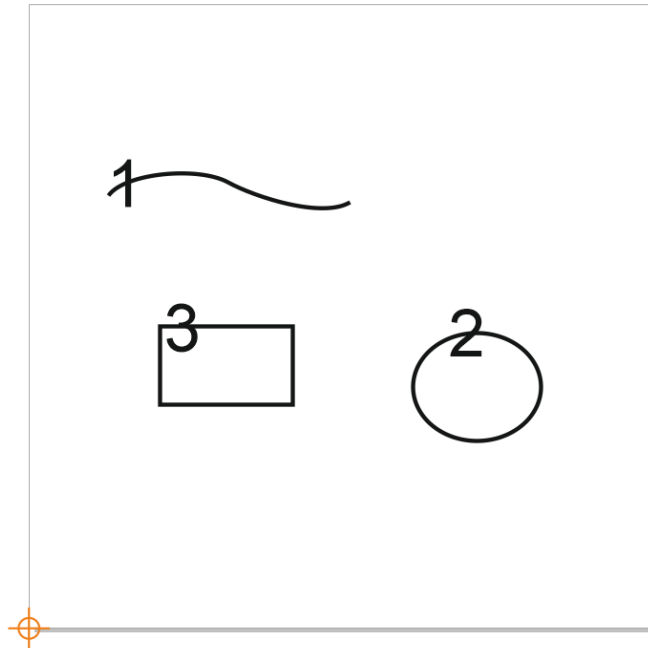
LAYER FEEDRATE

All available layers of the drawing are listed here. Different feed rates can be set for each layer. If you do not want to edit certain curves or layers, then move the curves to a layer and hide it. This layer will then no longer be taken into account when creating the G-code.

MACHINING PARAMETERS

All necessary parameters of the milling operation are set here:

Safety height	Z safety plane for approach, retract and rapid traverse above the workpiece.
Plunge height	Z plunge plane. From here on, the plunging feed rate is used. Material plunging usually takes place at a lower feed rate.
Start height	Z plane, from when machining starts. Mostly zero, when the Z zero point is on the workpiece surface.
Depth	Z-plane of the last cut plane (depth of machining).
Depth of cut / increment	Depth of cut (depth per pass)
Plunge feed	Feed rate when plunging into the material
Spindle speed	Spindle speed
Tool number	Tool number for tool change

CURVE PRECISION

Here you can set the curve precision. All curves of the toolpaths are approximated by straight segments. The length of these segments is the curve precision. With a high value, the individual segments are longer and the precision is lower. At a low value, the individual segments are shorter and the precision is higher. With a high precision, the creation of the NC program can take longer, among other things.

12.03.2025

10.6.2 Contour engraving

This strategy is specifically for engraving contours. Engraving depth controllers are also supported. Clicking on the button again opens a window in which all parameters can be set. Clicking on "OK" starts the generation.
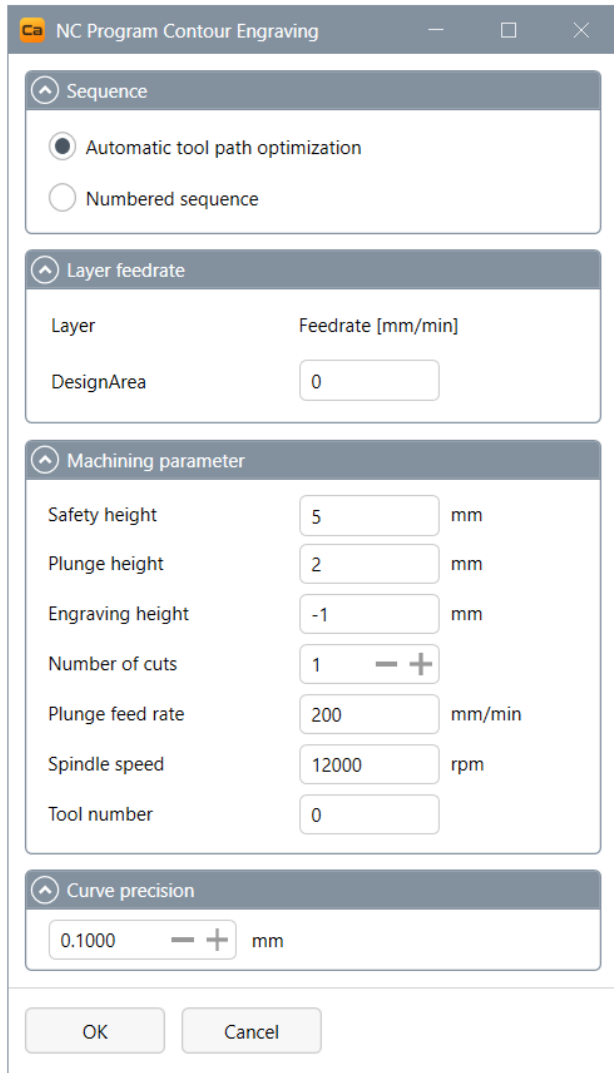
SEQUENCE

Setting the order of editing. Automatically optimized order or numbered order.

LAYER FEEDRATE

Set the feed rate for each layer in the CorelDraw sheet.

MACHINING PARAMETERS

Set cutting data and parameters.
Safety height
Plunge height
Engraving height
Number of cuts
Plunge feed rate
Spindle speed
Tool number

CURVE PRECISION

Specifies the accuracy (length unit) of the interpolation for curves

SEQUENCE

Identical to contour milling. See Contour milling (page 54)

LAYER FEEDRATE

All available layers of the drawing are listed here. Different feed rates can be set for each layer. If you do not want to edit certain curves or layers, then move the curves to a layer and hide it. This layer will then no longer be taken into account when creating the G-code.

MACHINING PARAMETERS

All necessary parameters of engraving machining are set here:

Safety height	Z safety plane for approach, retract and rapid traverse above the workpiece.
Plunge height	Z plunge plane. From here on, the plunging feed rate is used. Material plunging usually takes place at a lower feed rate.
Engraving height	Z plane, from when machining starts. Mostly zero, if the Z zero point is on the workpiece surface. For engraving depth controllers, a negative value is set here (preload).
Number of cuts	Number of passes. How often the individual contours are to be traversed.
Plunge feed	Feed rate when plunging into the material
Spindle speed	Spindle speed
Tool number	Tool number for tool change

CURVE PRECISION

Here you can set the curve precision. All curves of the toolpaths are approximated by straight segments. The length of these segments is the curve precision. With a high value, the individual segments are longer and the precision is lower. At a low value, the individual segments are shorter and the precision is higher. With a high precision, the creation of the NC program can take longer, among other things.

12.03.2025

10.6.3 Contour cutting

This strategy is intended for cutting contours using laser, water or plasma. Clicking on the button opens a window in which all necessary parameters can be entered. You can select whether a safety height or a cutting height is to be approached. Simply activate/deactivate the corresponding checkbox. Clicking on "OK" starts the creation of the G-code.
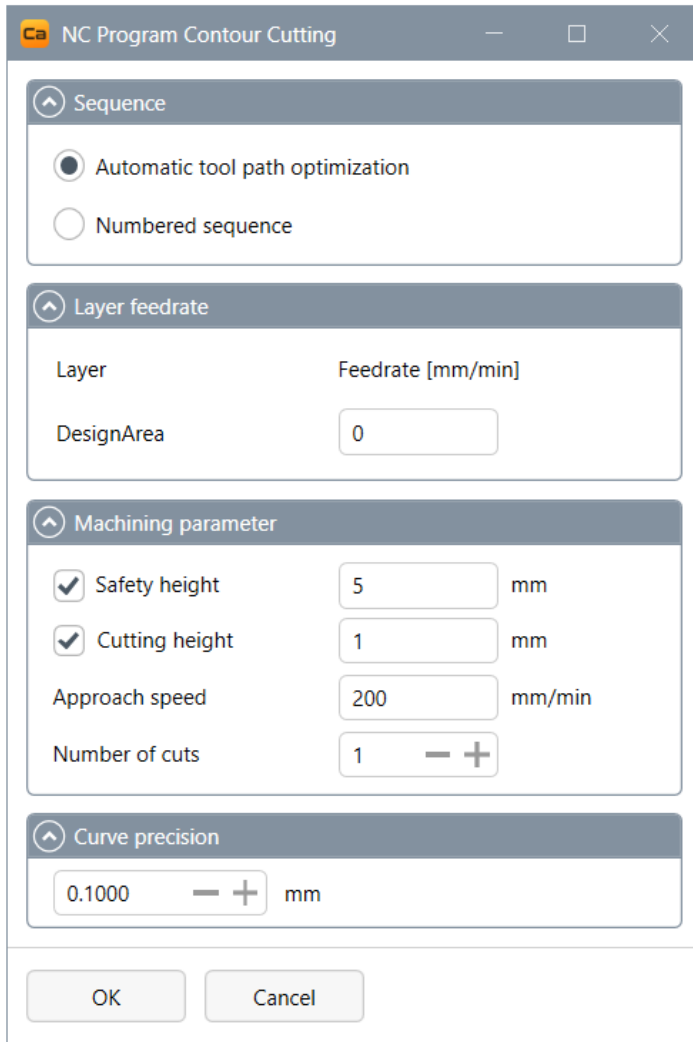
SEQUENCE
Setting the order of editing. Automatically optimized order or numbered order.

LAYER FEEDRATE
Set feed rate for each layer in the CorelDraw sheet.

MACHINING PARAMETERS
Set cutting data and parameters.
Safety height
Cutting height
Approach speed
Number of cuts

CURVE PRECISION
Specifies the accuracy (length unit) of the interpolation for curves

SEQUENCE

Identical to contour milling. See Contour milling (page 54)

LAYER FEEDRATE

All available layers of the drawing are listed here. Different feed rates can be set for each layer. If you do not want to edit certain curves or layers, then move the curves to a layer and hide it. This layer will then no longer be taken into account when creating the G-code.

MACHINING PARAMETERS: Cutting data

All necessary parameters of the cutting process are set here:

Safety height	Z safety plane for approach, retract and rapid traverse above the workpiece.
Cutting height	Z cutting plane. Z height of the cutting plane.
Approach speed	Feed rate between safety height and cutting height. Feed rate for approaching the Z cutting plane.
Number of cuts	Number of passes. How often the individual contours are to be traversed.

CURVE PRECISION

Here you can set the curve precision. All curves of the toolpaths are approximated by straight segments. The length of these segments is the curve precision. With a high value, the individual segments are longer and the precision is lower. At a low value, the individual segments are shorter and the precision is higher. With a high precision, the creation of the NC program can take longer, among other things.

10.7 3D view

The 3D view supports various gestures and functions. These can be controlled with the mouse.

Gesture	Functional description
Left mouse button and move mouse	Moves the 3D view according to the mouse movement
Hold down right mouse button and move mouse	Rotates the 3D view
Move mouse scroll wheel	Zooms in/out of the 3D view (magnification)
Left mouse button double click	Opens the 3D view in a separate and large window
Mouse click on cube pages	This cube always shows the current perspective of the 3D view. Clicking on the corresponding side of the cube activates the side views: U → Top view R → Front view B → View from left F → View from right L → View from behind D → View from below

10.8 Tool simulation

You have the option of simulating the whole NC program and the tool paths. A slider is provided for this purpose. You can use this to access each individual G-code line but also to simulate it quickly. The current position is indicated by an orange dot and a tool dummy. The coordinates of the current position are also displayed in the separate window of the 3D view. The simulation functions are located below the 3D view (slider and two buttons for single-step simulation). Single-step means that the previous or next position in the NC program is read in and simulated. The tool simulation is helpful for recognizing errors or the machining sequence of the individual tool operations.
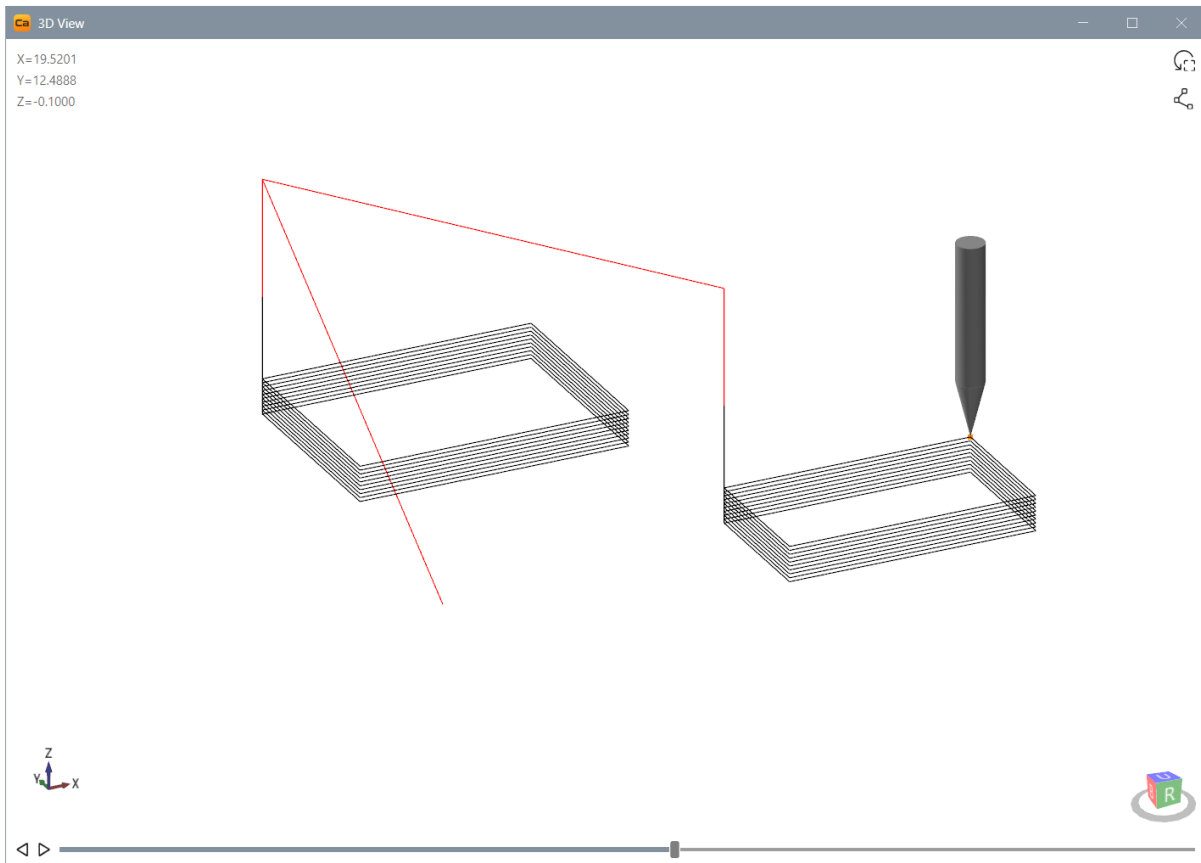
11 Additional functions

The application has additional functions that are hidden by default. These include the creation of a measurement grid, the function for searching for duplicate curves and the numbering and sorting of drawing objects. The functions can be activated via the CamDRAW settings window. They are then displayed in the user interface.

11.1 Search curve duplicates

This function supports you when searching for curve duplicates. This is particularly useful for drawings with a large number of curve objects. The function automatically searches for curve duplicates and lists duplicates. You can then decide whether individual duplicates or all duplicates should be removed.
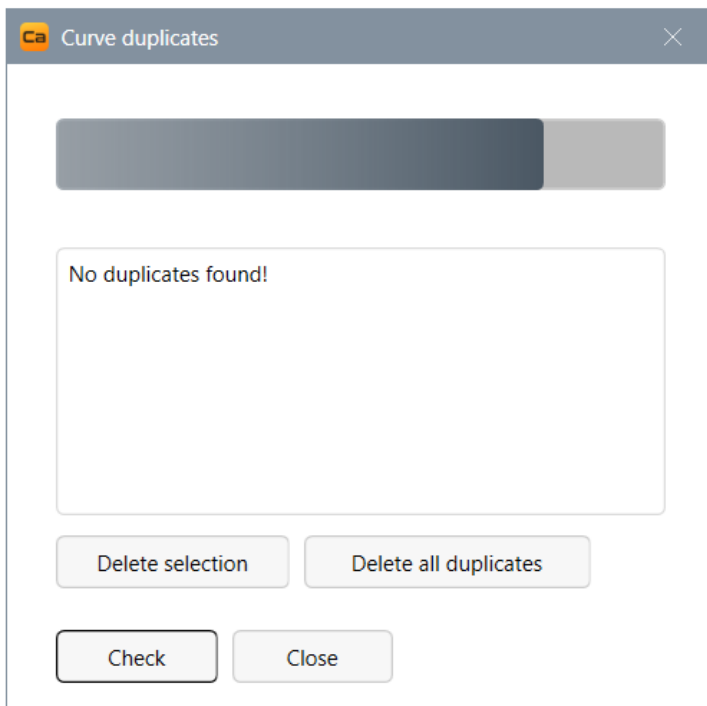
11.2 Create measurement grid

CamDRAW supports the creation of a measurement grid for your drawing objects. This is needed for a surface measurement of the workpiece, e.g. to perform a Z-height correction for uneven or curved workpiece surfaces. For this you need a probe (Probe) at your CNC machine.

The measuring grid can be created for any type of machining (flat or rotary machining).

The input mask of the surface measurement looks as follows. You first select your desired drawing objects, enter the measurement data and create a measurement grid. The measurement grid is then automatically placed over the selected drawing objects. The measurement points are represented by dots. The edge of the measurement grid is represented by a rectangle. You can also select no drawing object at all. Then CamDRAW automatically creates a measurement grid over the entire CorelDraw drawing sheet. This is useful if you want to scan the entire workpiece surface.

The red measuring point represents an arbitrary and adjustable point for e.g. setting the Z zero point on your CNC machine (before the measurement).

The X-Y coordinate of this point, as well as all other measuring parameters, is available through postprocessor parameters in the G-code file. A transfer of the measuring parameters to your CNC machine is done via the NC program or via the machine setup program.

You can define all measurement data parameters in the G-Code and then, for example, call a subprogram (subroutine) of your CNC control. In this subroutine, the actual measurement of the workpiece surface is then performed using the measurement parameters of CamDRAW (see chapter Postprocessor settings).

Measuring grid

Round workpiece	a	☐
Number of points X	b	10
Number of points Y	c	10
Border distance X	d	0.400 mm
Border distance Y	e	0.400 mm
Z safety height (WCS)	f	4.000 mm
MinZ	g	-4.000 mm
Z increment (X)	h	2.000 mm
Z increment (Y)	i	2.000 mm
Probing feed rate	j	100.000 mm/min

k l m n o Z-Auto

		p
X-distance	1.8967	mm
Y-distance	1.0300	mm
Measuring points	100	

a	Input to activate a round workpiece
b	Number of measuring points in the X direction of the CorelDraw sheet
c	Number of measuring points in Y-direction of the CorelDraw sheet
d	Margin distance to the selected drawing object in X direction
e	Margin distance to the selected drawing object in Y-direction
f	Safety height between measuring points or first measuring point
g	Minimum Z-height (plane): Lowest point
h	Z increment for measuring series in X direction (extended measuring cycle)
i	Z increment for measuring series in Y direction (extended measuring cycle)
j	Feed rate for measuring
k	Create measuring range
l	Update measuring range
m	Delete measuring range
n	Show or hide measuring range
o	Selection of the first measuring point
p	Info display of the created measuring range

12 Settings

The application has its own settings window. There you can customize CamDRAW to your needs and applications. Click on the gear icon to open the settings window.
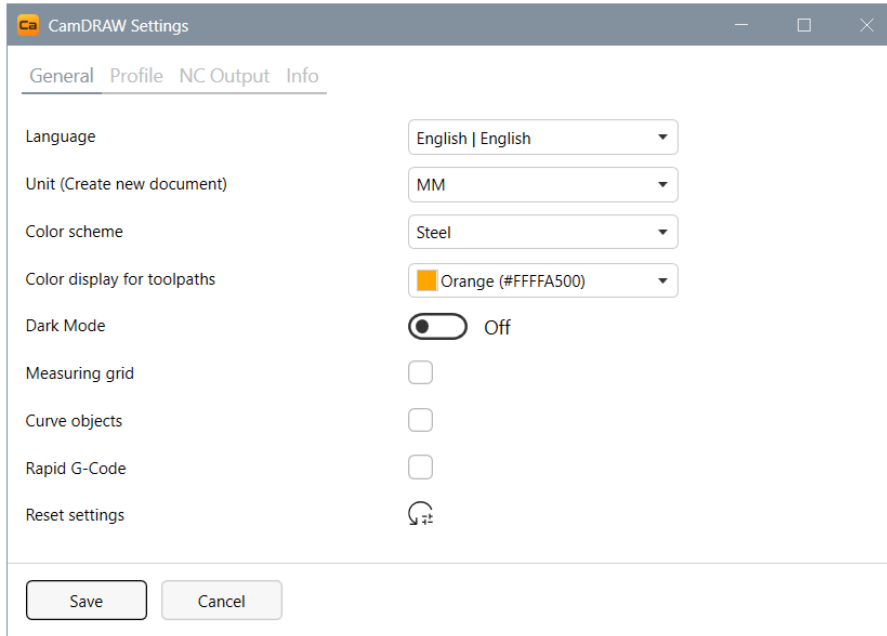
12.1 General settings

On the first page in the settings you will find general settings. Here you can set the language of the user interface or the units for creating a new document. After changing the language, CorelDraw (CamDRAW) must be restarted. The units are only for creating a new document in CorelDraw (default unit). You also have the possibility to use other units. You can change them directly in the document settings of CorelDraw.

You have the possibility to customize the user interface of CamDRAW to your taste. For this you can set the theme color or enable or disable the dark mode.

In addition, you can activate or deactivate the additional functions (Search curve duplicates / Create measuring range) here. When deactivated, the respective function is also hidden from the plugin interface of CamDRAW.

Other settings include

Color of the toolpaths:	Here you can select the color in which the toolpaths of the operations should be displayed
Rapid G-Code:	Show/hide the "Rapid G-Code" functions
Reset:	Here you can reset the entire plugin settings. The factory condition is restored. All settings such as profiles, post processors, tool lists are reset. This causes a re-display to select your CNC machine.

12.2 Profile / Presettings

Under the "Profiles" tab, you can set the profiles and stored preset files (postprocessors).

The profiles represent the individually selected preset files. You can use the profiles to store and save various presets. This is very helpful if, for example, you want to perform several types of machining (flat and/or rotary machining) on the same machine. Then simply save the different presetting files (settings) in two different profiles. You can then select your desired profile on the main page of CamDRAW. The correct settings are loaded automatically and the software is immediately ready for use. You can also assign a name to each profile.

When you start CamDRAW for the first time, a window appears in which you can select your CNC machine. After confirmation, this preset file is automatically saved in profile 1.
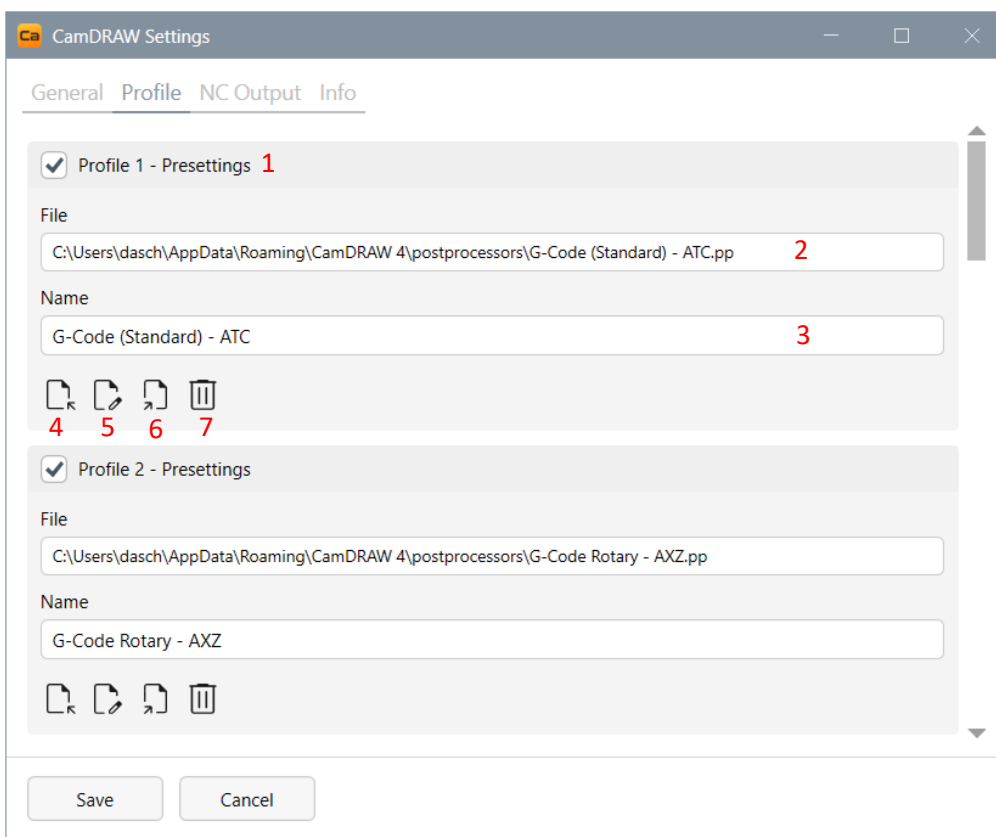
1	**Activate / deactivate the profile**
	By activating the checkbox, you activate the respective profile. It becomes active. Deactivating the checkbox deactivates the profile, but does not delete it.
2	**Preset file**
	Display of the preset file (postprocessor)
3	**Name of the profile**
	Any name for the profile can be entered here
4	**Import**
	A preset file can be imported into the profile here. After clicking, the installation directory of all preset files (postprocessors) opens automatically. Each imported file is copied to the application settings directory (AppData).
5	**Edit**

The profile's settings can be called up and edited here. A new window opens with all the options for the presets (post-processors).

6   **Export**

Export the current profile settings to a file.

7   **Removing / emptying the profile**

Click on "Delete" to remove the existing preset file from the respective profile and it will be deactivated. However, the file is not deleted. It can be found in the directory.

12.2.1 Presettings (Postprocessor)

Click on "Edit" to open the presetting file window. All settings can be made here. The settings are divided into corresponding areas. All settings can be individually adapted to your CNC machine (CNC control) or your application.
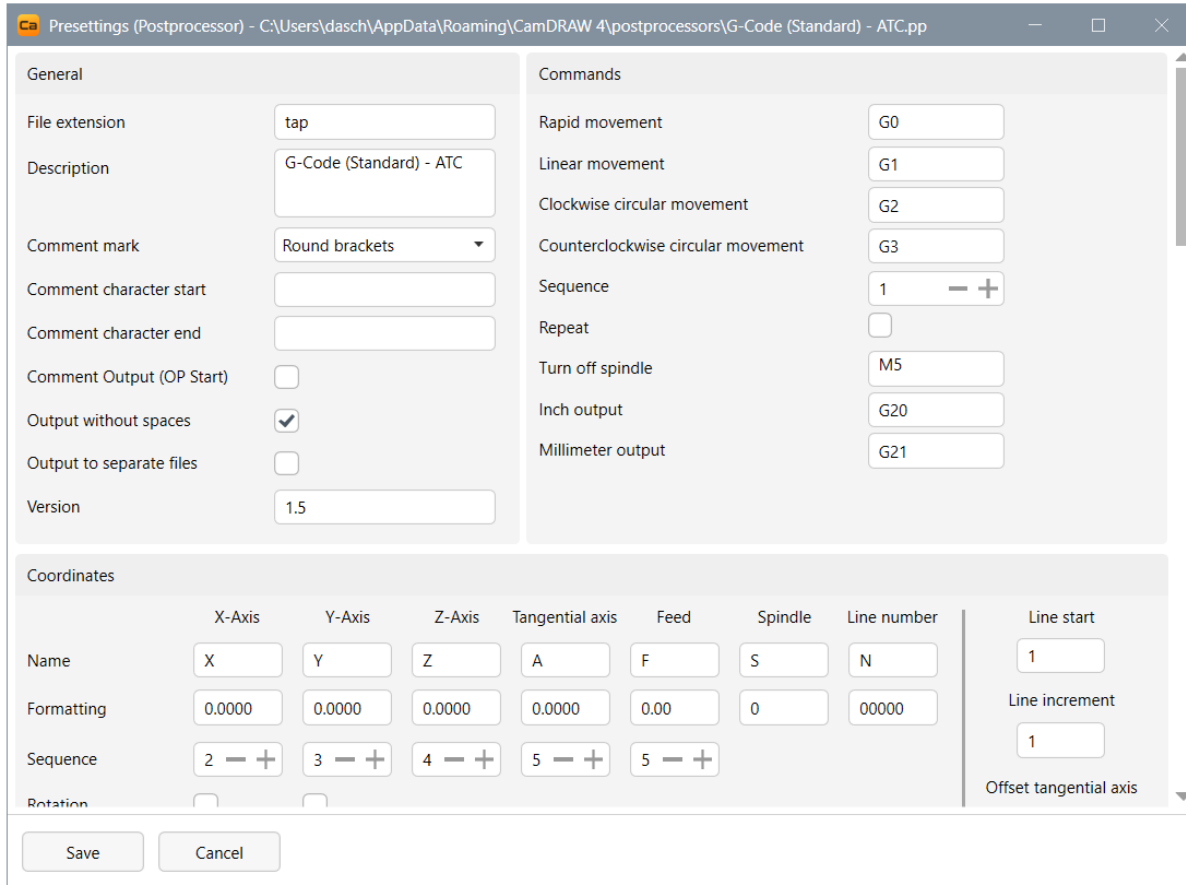
The settings and your explanations are as follows:

Settings in the "General" area	Description
File extension	File extension of the created G-code file (CNC program)
Description	Post processor description
Comment mark	The character for comment lines in the G-code output
Comment character start	Defines an individual character at the beginning of the comment. Empty by default.
Comment character end	Defines an individual character at the end of the comment. Empty by default.
Comment output (OP Start)	Comment output at the beginning of a machining operation
Output without spaces	Outputs the individual G-code lines without spaces
Output to separate files	If active, tool operations with different tools are output to separate files. If deactivated, the output is in a single file.
Version	Version number of the postprocessor

Settings in the "Commands" area	Description
Rapid movement	Command for rapid traverse in G-code
Linear movement	Command for linear movement in G-code

Clockwise circular movement	Command for circular movement in G-code
Counterclockwise circular movement	Command for circular movement in G-code
Sequence	Sequence at which the movement command is output in the G-code
Repeat	Should the motion commands be issued repeatedly? (Yes/No)
Switch off spindle	Command to switch off the spindle (e.g. M5), which is issued during tool change at the end of an operation.
Inch Output	Command for output in inches (mostly G20). If the CorelDraw document unit is inches, then the contents of the inch output.
Millimeter Output	Command for output in millimeters (usually G21). If the CorelDraw document unit is millimeters, then the contents of the millimeter output.

Settings in the "Coordinates" area	Description
Name	Designation/name for the respective axis, feed rate, spindle and line number X: Corresponds to the CorelDraw X axis Y: Corresponds to the CorelDraw Y axis Z: Corresponds to the height axis of your machine (mostly Z)
Formatting	Formatting of the numbers/values in the G-code file behind the respective designation
Sequence	Sequence at which the respective axis designations and values are output in the G code
Rotation	Activates a rotation axis for the respective axis E.g. for "X": Means that the CorelDraw X axis is a rotation axis. Length dimensions (X) are converted to angle dimensions based on the workpiece radius
Mapping	Activates a mapping for the respective axis E.g. for "Y": Means that the CorelDraw Y axis is a rotation axis. However, length dimensions (Y) are not converted to angle dimensions. If your CNC controller supports this: Y →A Mapping
Factor	Factor for the respective axis or the line number E.g. the direction of rotation of a rotary axis can be reversed by -1.
Repeat	Here you can set whether or not the XYZ coordinates should be written repeatedly for each line in the G-code.
Output	Here you can determine whether the corresponding coordinate or designation is to be written into the G-code or not. For example, the output of individual axes can be deactivated.
Line start value	Start value of the line numbering
Line increment	Increment of the line numbering
Offset tangential axis	Positive or negative constant offset for the tangential axis
Z-Up angle for tangential axis	Defines a constant angle at which the machine must lift off to orient the tangential axis. For example, in the case of a rectangle, the machine must lift off in the corners and orient the tangential axis.

12.03.2025

Settings in the "Flat machining" area	Description
Workpiece shape	If there are postprocessor settings for a flat machining, the workpiece shape (round/rectangular) can be set here. This is only active for the function to create a measuring range.

Settings in the "Rotation machining" area	Description
Swap diameter input	If postprocessor settings are available for a rotary machining, the diameter input can be exchanged here. I.e. in the workpiece parameters, either the X or Y axis is then provided with a diameter input.
Z zero point	Setting the Z zero point: Either on the cylinder surface or cylinder axis
Machining cylinder axis	Over cylinder axis →rotary external machining Under cylinder axis → rotary internal machining (e.g. internal engraving of jewelry rings)
Settings in the "Custom lines" area	**Description**
Program start	Lines/sections at the beginning of the program in the G-code file (CNC program): E.g. G90 / machine equipment / etc.
End of program	Lines/sections at the end of the program in the G-code file (CNC program): E.g. M5 / M30 / free movements / approach park position
Operation start	Lines/sections at the start of a machining operation in the G-code file (CNC program): E.g. switch on coolant
Operation end	Lines/sections for output at the end of a machining operation
Tool change	Lines/sections for the tool change in the G-code file (CNC program): E.g. M5 M6 T1 M3
Skip on first tool operation (checkbox)	Skips the output of the tool change text during the first tool operation
Start of cut (laser/water/plasma)	Lines/sections when the cutting contour starts. E.g. Switch on laser
End of cut (laser/water/plasma)	Lines/sections when the cutting contour ends. E.g. Switch off laser
Machine Setup (Setup)	This section is used to set up the CNC machine. During the creation of the G-code job, an extra file is created with the contents of the machine setup. By default in the AppData directory. You can use the file to measure workpiece surfaces or set workpiece datums, for example. The machine setup must be performed before the job file is executed. All placeholder/postprocessor parameters are also available in this file.

Settings in the "Workpiece data input" area	Description
Text workpiece input (X)	In this area the texts for the workpiece data input can be defined.

	Workpiece input for the CorelDraw X axis E.g. "Width" or "Circumference"
Text workpiece input (Y)	Workpiece input for the CorelDraw Y axis E.g. "Length" or "Circumference
Text diameter input	Workpiece input for the diameter. E.G. E.g. "Outer diameter" or "Inner diameter".

Settings in the "Default values" area	Description
Position first measuring point	Defines the position of the first measuring point for the measuring grid. This position is required during measurement to approach the first measuring point. For flat machining usually "LL". LL → Bottom/left UL → Top/Left UR → Top/right LR → Bottom/right
XY zero point orientation	Default XY origin as soon as a new document is created by CamDRAW.
Tool number	Default tool number for creating a machining operation. This tool number is selected when creating an operation for the first time (opening the operation window).
Safety height for new operation	These default values are taken over when a new machining operation is created. If empty, a suitable value is determined by CamDRAW. All default values can be overwritten in the respective operations.
Plunge height for new operation	
Starting height for new operation	
Depth for new operation	
Engraving height for new operation	
Cutting height for new operation	
Number of cuts for new operation	
Plunge feed rate	
Cutting depth	
Spindle speed	
Curve precision	
Allowance / offset	Default value for the allowance or offset when adding a new tool operation.
Safety height during contour cutting (laser/water/plasma)	Here you can determine whether the safety height should be approached during contour cutting or not.
Cutting height during contour cutting (laser/water/plasma)	Here you can determine whether the cutting height should be approached during contour cutting or not.

Settings in the "Rendering" area	Description
Direction factor axis of rotation	Depending on the direction of rotation of your rotary axis, either -1 or 1 can be set here (default = -1)
X-axis designation	Designation/name of the X axis (default = X)
Y-axis designation	Designation/name of the Y-axis (default = Y)
Z-axis designation	Designation/name of the Z axis (default = Z)
A-axis designation	A-axis designation/name (default = A)
B-axis designation	Designation/name of the B axis (default = B)

12.03.2025

Settings in the "Workpiece zero points" area	Description
Zero point preselection (default)	Defines the standard call of the workpiece zero point (WCS call) when selecting a profile. Number "0" means no call. Numbers "1-9" mean a call of the workpiece zero point according to the number. If no call is made, there will be no output in the NC program.
Zero point 1	Output text for the G-code for workpiece zero point 1 (usually "G54")
Zero point 2	Output text for the G-code for workpiece zero point 2
Zero point 3	Output text for the G-code for workpiece zero point 3
Zero point 4	Output text for the G-code for workpiece zero point 4
Zero point 5	Output text for the G-code for workpiece zero point 5
Zero point 6	Output text for the G-code for workpiece zero point 6
Zero point 7	Output text for the G-code for workpiece zero point 7
Zero point 8	Output text for the G-code for workpiece zero point 8
Zero point 9	Output text for the G-code for workpiece zero point 9

12.2.2 Machine setup file (automation)

Each time you create G-Code (job file), CamDRAW automatically creates an extra file in parallel, which is suitable for a setup of your CNC machine. There you can e.g. scan workpiece surfaces or set workpiece zero points. The content of the file is defined in the presettings. Under "Machine Setup" in you can set user-defined lines, which are then written to the machine setup file during G-Code generation. In this file all placeholders and postprocessor parameters are also available. E.g. measurement parameters which are defined in CamDRAW. You can also use this file for a parameter transfer to your CNC control. Or call macros from there.

Always run the machine setup file before running the actual job file.

12.2.3 Placeholder / Postprocessor parameters

In the sections of the user defined lines (program start, program end, operation start, operation end, tool change, cut start, cut end and machine setup) placeholders are supported. With these placeholders you can transfer parameters, which are defined in CamDRAW, into the G-Code file (CNC program). The placeholders are always marked by a "<...>" and the respective parameter inside. These placeholders are then replaced by the corresponding value during the creation of the G-code.

The following placeholders are supported:

Placeholder	Description
<project>	Project/document name in CorelDraw
<version>	Version number of CamDRAW
<name>	Name of the current machining operation
<number>	Number of the current machining operation
<units>	Current unit of the CorelDraw document (G20/G21)
<s>	Spindle speed for the current machining operation
<t>	Tool number for the current machining operation
<f>	Feed rate for the current machining operation
<tool_name>	Tool name of the current tool
<tools>	Listing of all tools needed for the job
<workpiece_radius>	Workpiece radius (currently in CamDRAW)
<workpiece_size_x>	Workpiece size in X direction (currently in CamDRAW)
<workpiece_size_y>	Workpiece size in Y-direction (currently in CamDRAW)
<workpiece_left_x>	X-position of the left edge of the workpiece (currently in CamDRAW).
<workpiece_bottom_y>	Y-position of the bottom edge of the workpiece (currently in CamDRAW).
<workpiece_center_x>	X-position of the workpiece center (currently in CamDRAW)
<workpiece_center_y>	Y-position of the workpiece center (currently in CamDRAW)
<toolpaths_left_x>	X-position of the left edge of all toolpaths (bounding box)
<toolpaths_right_x>	X-position of the right edge of all toolpaths (bounding box)
<toolpaths_top_y>	Y-position of the upper edge of all toolpaths (bounding box)
<toolpaths_bottom_y>	Y-position of the lower edge of all toolpaths (bounding box)
<IsRotationMachining>	Is there a rotational machining? 1 → yes 0 → no
<surface_measurement_activated>	Is a measurement grid created and activated? 1 → yes

	0 → no
<x_position_first_node_shape>	X-position of the first node of the first shape
<y_position_first_node_shape>	Y-position of the first node of the first shape
<x_position_zAuto_point>	X position of the Z-Auto point
<y_position_zAuto_point>	Y-position of the Z-Auto point
<nx>	Measuring grid: Number of measuring points in X direction
<ny>	Measuring grid: number of measuring points in Y-direction
<gridsize_x>	Measuring grid: spacing of the measuring points in the X direction
<gridsize_y>	Measuring grid: distance of the measuring points in Y-direction
<corner_measure_grid_x>	Measuring grid: X-position of the corner/first measuring point
<corner_measure_grid_y>	Measuring grid: Y-position of the corner/first measuring point
<minZ>	Measuring grid: MinZ
<zSafetyHeight>	Measuring grid: Safety height for approaching or freeing between the measuring points
<z_increment_x_direction>	Measuring grid: additional parameters for adjusted measuring cycle
<z_increment_y_direction>	Measuring grid: additional parameters for adjusted measuring cycle
<probeFeedRate>	Measuring grid: scanning speed
<round_workpiece_activated>	Is there a round workpiece shape? 1 → yes 0 → no = rectangular
<default_tool_number>	Standard tool number, which is stored in the default settings (under Standard values)
<default_safe_height>	Default safety height, which is stored in the default settings (under Default values)
<default_plunge_height>	Default plunge height, which is stored in the preferences (under Default values)
<default_start_height>	Default starting height, which is stored in the preferences (under Default values)
<default_depth>	Standard depth, which is stored in the preferences
<default_cutting_depth>	Standard cutting depth, which is stored in the presets (under Standard values)
<default_engraving_height>	Standard engraving height, which is stored in the presets (under Default values)
<default_cutting_height>	Default cutting height, which is stored in the presets (under Default values)
<default_number_of_cuts>	Default number of cuts, which is stored in the preferences (under Default values)
<default_plunge_feed>	Default plunge feed, which is stored in the preferences (under Default values)
<default_spindle_speed>	Standard spindle speed, which is stored in the presets (under Standard values)
<default_interpolation_precision>	Standard curve precision, which is stored in the preferences (under Default values)

12.3 NC output

The output directory for the created NC programs and the machine setup file is set here. In addition, the file name can be set. The default name for the NC program is "Job". The default directory is "C:\Users\USER\AppData\Roaming\CamDRAW 4\". You can select a new directory by clicking on the folder icon.
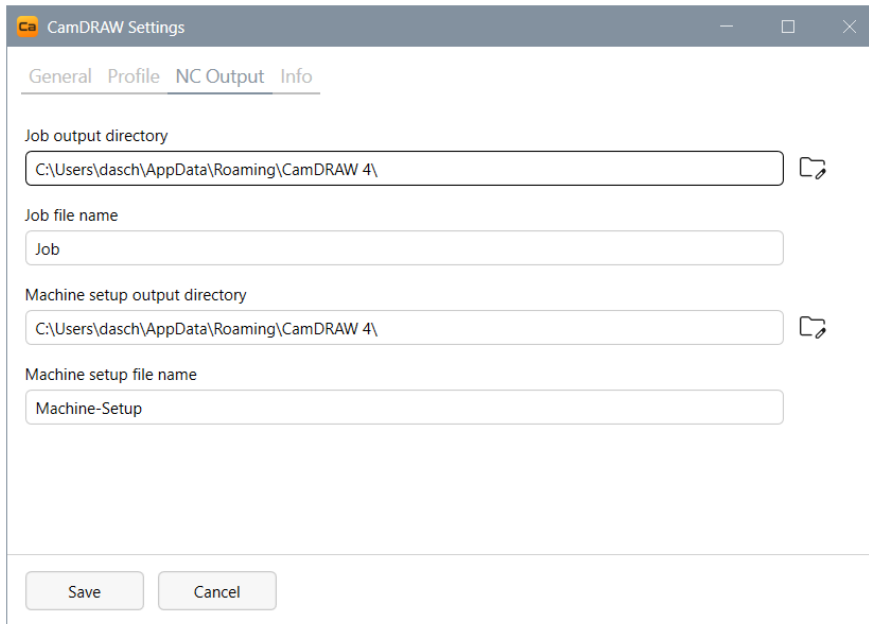
13 Trial version

CamDRAW is automatically activated as a trial version at the first start. The software is immediately ready for use. After the trial period, the demo version expires and the software is locked. For the first automatic activation of the trial version you need an internet connection. With the trial version you can test the workflow and all features in the complete range of functions.

14 Activate full version with license key

After purchasing a license, you will receive a license key with which you can unlock the software. You will receive this license key by e-mail.

The software can be activated at any time (also during the trial period) by a valid license key. For the software activation you need an internet connection once. After activation, you can use the software without an Internet connection. You can use the software only on one computer (hardware bound)! To use it on another computer, it must first be deactivated and then reactivated. An Internet connection is required once for each activation and deactivation.

To activate the software, open the settings and navigate to "Info". There you click on "Activate license". The following window opens:
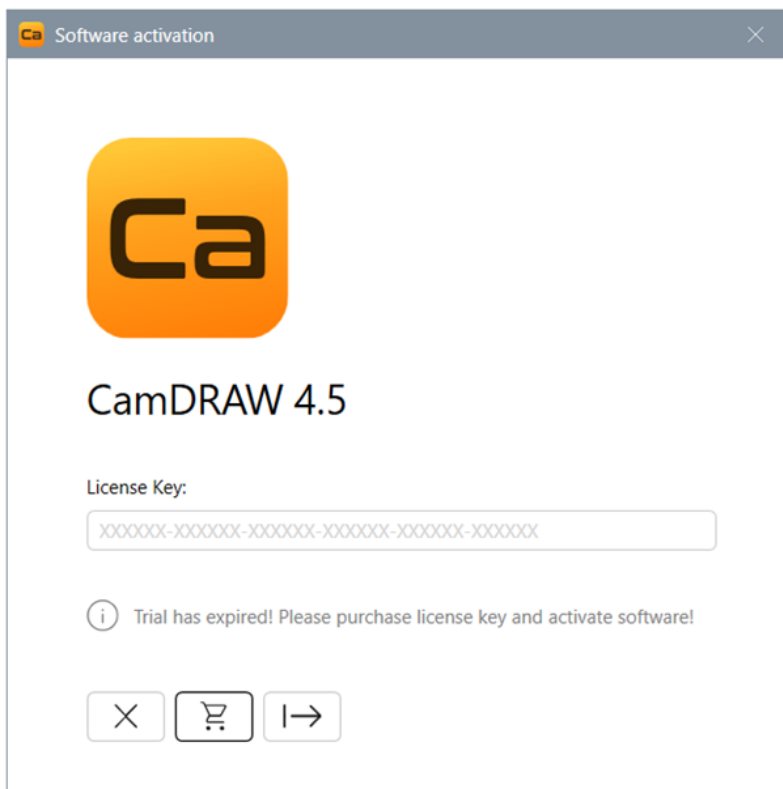
Now you can insert or enter the license key. Then click on the arrow-icon. The software will be activated and unlocked! You will get the status of the activation at the very bottom of the window. After successful activation you can close the window.

77   12.03.2025

15 License information / Deactivate license on computer

You can deactivate your license, which is activated on the current computer, at any time. This is necessary, for example, if you have a new computer.

To deactivate, open the settings of CamDRAW and navigate to "Info". There you click on the button "License information". Here you can view all license information:

- Product
- Your license key
- License status
- Remaining days (trial version)
- Licensee information

You can then click "Deactivate license" to deactivate it. The license will be deactivated on the current computer and will thus be unlocked for reactivation.

16 Install updates

Product updates are released for CamDRAW on a regular basis. You can view them at https://www.camdraw.com/en

Updates of the same major version (CamDRAW 3 → Major 3) are always installed into the same installation directory. Thus, the application is updated. Download the installer (setup) of the latest version, run the installer and install into the directory where the older version is already installed. I.e. you must select the CorelDraw product where you installed the previous version. See chapter 4.

17 Uninstall

Uninstalling CamDRAW can be done by reopening the setup (installer) or via the "Windows Control Panel" and "Uninstall a program". Only the addon directory in CorelDraw will be deleted. All application data (postprocessors, individual settings of the plugin, etc.) will be preserved.