
gsDXF2gCode

User guide

Illustrated guide, menus, machining and post-processors

English | v.1.0.0

<https://store.gardos-software.com/en/#cnc>

Contents

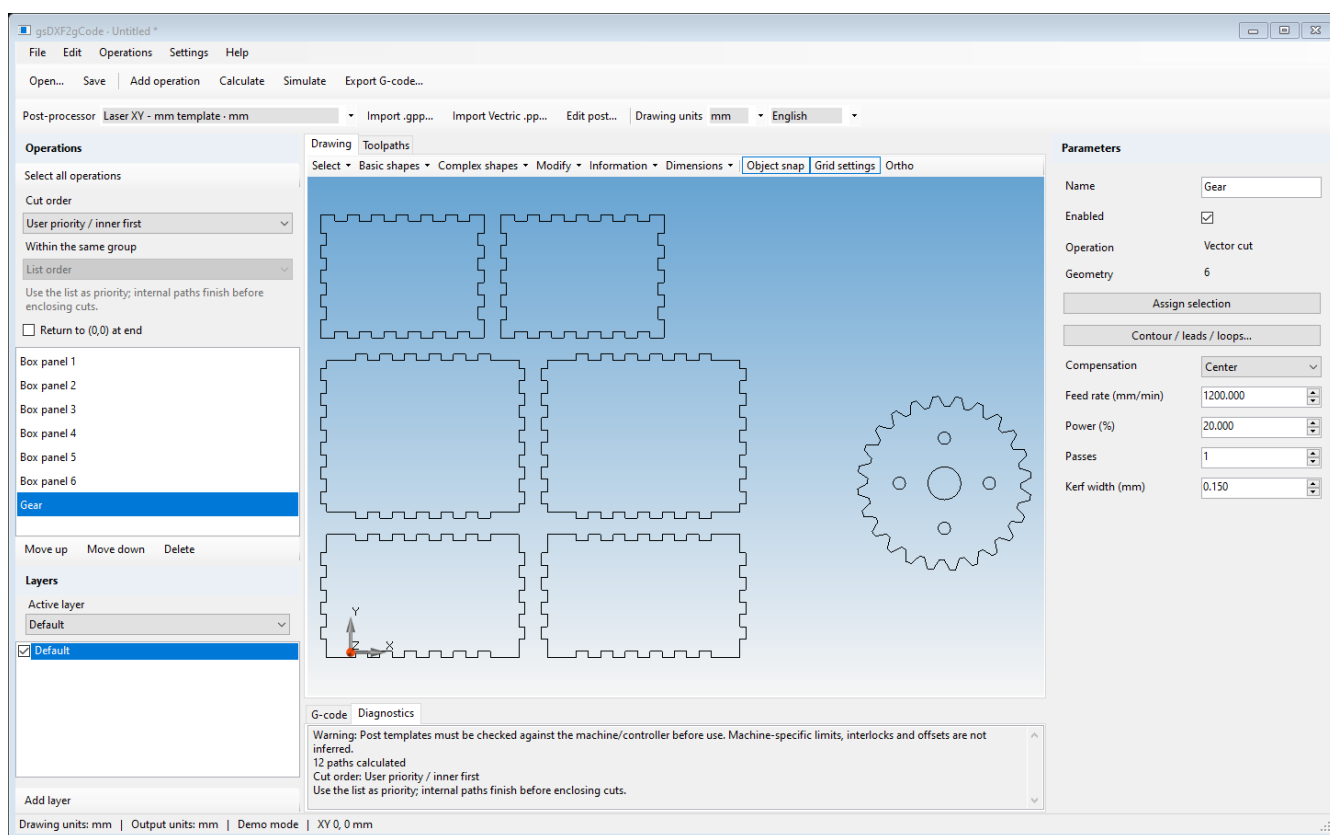
01 Getting to know the program	3
02 Drawing and machining functions	5
03 Post-processor: setup and commands	8
04 Your first job	13
05 Drawing, selecting and editing	14
06 Organizing the drawing with layers	15
07 Importing and creating ready-made shapes	16
08 Preparing machining operations	17
09 Pipe developments and 3D preview	19
10 Checking the toolpath and simulation	21
11 Saving and exporting a finished job	22
12 Troubleshooting, language and licensing	23

01 Getting to know the program

Start with the workspace and menus. The task-based questions later in this guide explain individual jobs in more detail.

gsDXF2gCode brings drawing, machining preparation and preview into one project. You can open a DXF/DWG, draw parts, generate shapes and arrange them on a sheet. Choose a suitable post-processor, define the operations, calculate, inspect and export. Save the .gscam project to keep an editable job.

Screenshots show the English interface. The same controls are available in the selected application language; some shape parameters retain their English names. The examples illustrate the workflow, not material-specific cutting settings.



The workspace: operations and layers on the left, drawing in the middle, selected operation settings on the right.

Areas of the window

Drawing	Create, select, edit and measure drawing objects.
Toolpaths	Inspect calculated machining paths and run the simulation.
Operations / Parameters	Select an operation on the left to edit its process settings on the right. Move up/down changes its priority or manual order.
Layers	Organize objects. Checkboxes control visibility; Active layer receives newly drawn objects. Add layer creates another layer.
G-code / Diagnostics	Generated machine text and calculation messages. G-code text and exports require activation. Read Diagnostics after every calculation.

The main menus

File	New, Open, Save, Save as, Export DXF and Exit. Save preserves the complete job; Export DXF exchanges drawing geometry.
Edit	Undo, Redo and Select all. Ctrl+Z undoes supported edits; Ctrl+Y redoes them.
Operations	Add operation, select operations, assign selected geometry, calculate, export G-code and edit contour/leads/loops.
Settings	Set/reset work origin, choose curve output, edit the post, configure snaps/grid and the default operation for DXF import.
Help	Open this PDF guide (also F1), visit the Gardos Software site, activate/deactivate the license and see About/version.

The upper toolbars

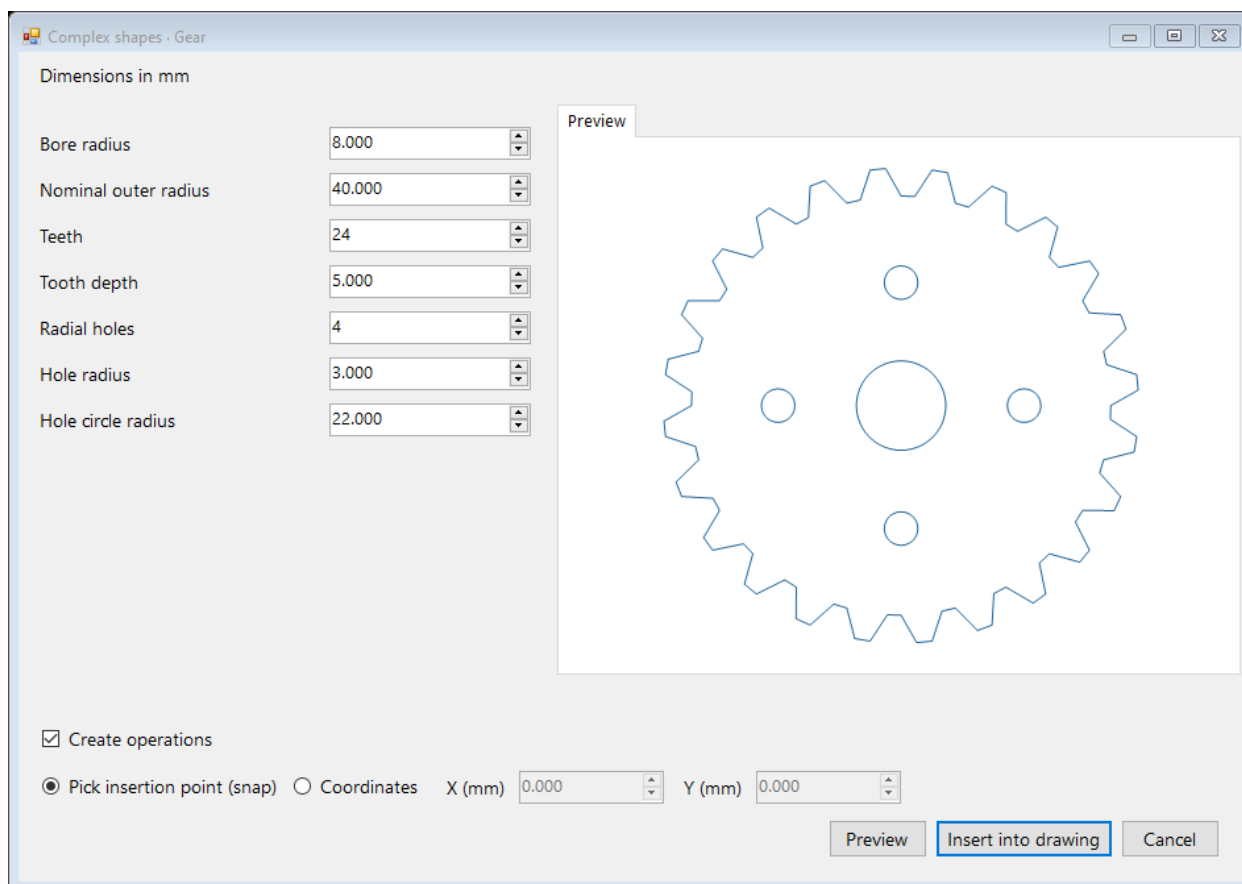
Open and Save manage files. Add operation prepares machining; Calculate updates the paths; Simulate previews the job; Export G-code writes the machine file. The next row selects the post-processor and offers Import .gpp, Import Vectric .pp and Edit post. Drawing units and language are on the same row. Zoom to fit is on the drawing/toolpath control itself.

02 Drawing and machining functions

Geometry describes the part. An operation specifies what the machine will do to that geometry.

Tools under Drawing

Select / Basic shapes	Choose a picking mode or draw lines, polylines, rectangles, circles, arcs, ellipses, splines and text. Esc cancels the current command.
Complex shapes	Import file at an insertion point; create parametric parts, finger joint boxes, gears, polygons, text and triangles. Includes nesting and Pipes.
Modify	Delete, move, copy, rotate, scale, mirror, offset, trim, extend, fillet, chamfer and explode. Change layer moves selected objects to another layer.
Information / Dimensions	Inspect length/details, area/perimeter and vertex coordinates. Add linear, aligned, radial, diameter, angular, ordinate or leader dimensions; use temporary dimensions for checks.
Object snap / Grid / Ortho	Use snaps for exact geometric points, configure grid appearance and spacing, and use Ortho for horizontal/vertical drawing.



Ready-made shapes: enter dimensions, preview, then insert using a snap point or coordinates.

Types of operation

VectorCut / VectorEngrave	Follow a contour for cutting or line engraving. Check feed, power, passes and compensation.
HatchEngrave	Fill a closed area with parallel engraving paths. Hatch spacing and angle set the pattern.
Profile / Pocket	Router operations for contour machining or clearing an area. Set tool diameter, final depth, step-down, feed and safe Z.
Punch	Apply the configured punch command at generated positions. The post and command must suit the machine.

Available operations depend on the selected post. Automatic operations can already exist after import or shape insertion: inspect the list before adding another Cut. Cut order controls inner-first priorities, grouping, manual order or alternating distant cuts. Return to (0,0) adds a final return to the job origin, not a machine homing cycle.

Nesting and Pipes

Complex shapes - Rectangular nesting

Dimensions in mm

Parts: width x height x quantity

100x50x25
50x50x10

Sheet width: 535.000

Sheet height: 330.000

Minimum spacing: 3.000

Allow 90° rotation: ☒

Preview

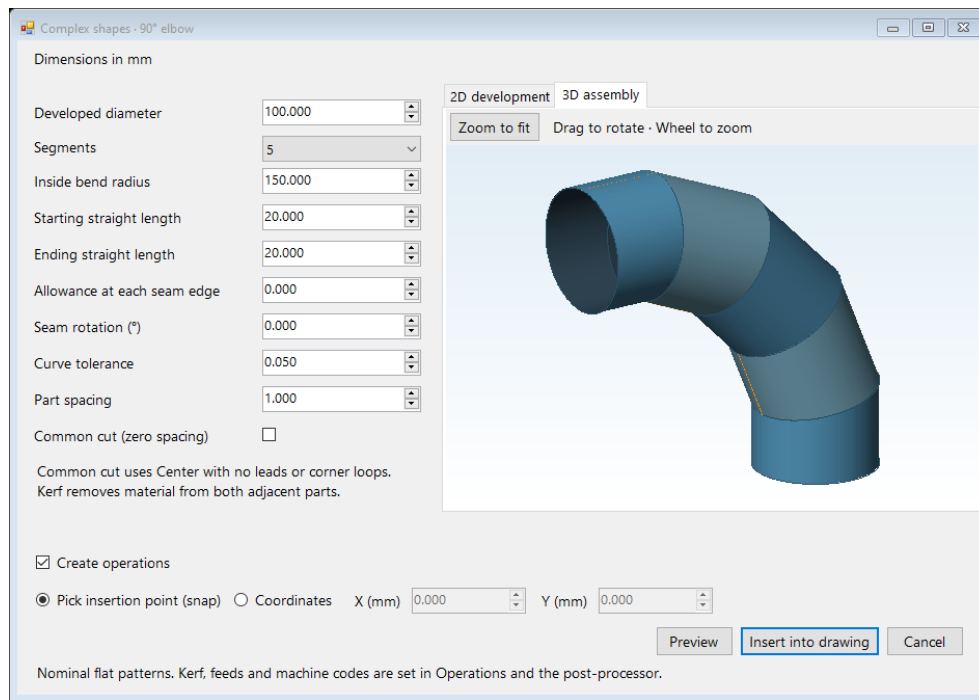
☒ Create operations

☒ Pick insertion point (snap) ☐ Coordinates X (mm) 0.000 Y (mm) 0.000

Sheet 1 / 2 Preview Insert into drawing Cancel

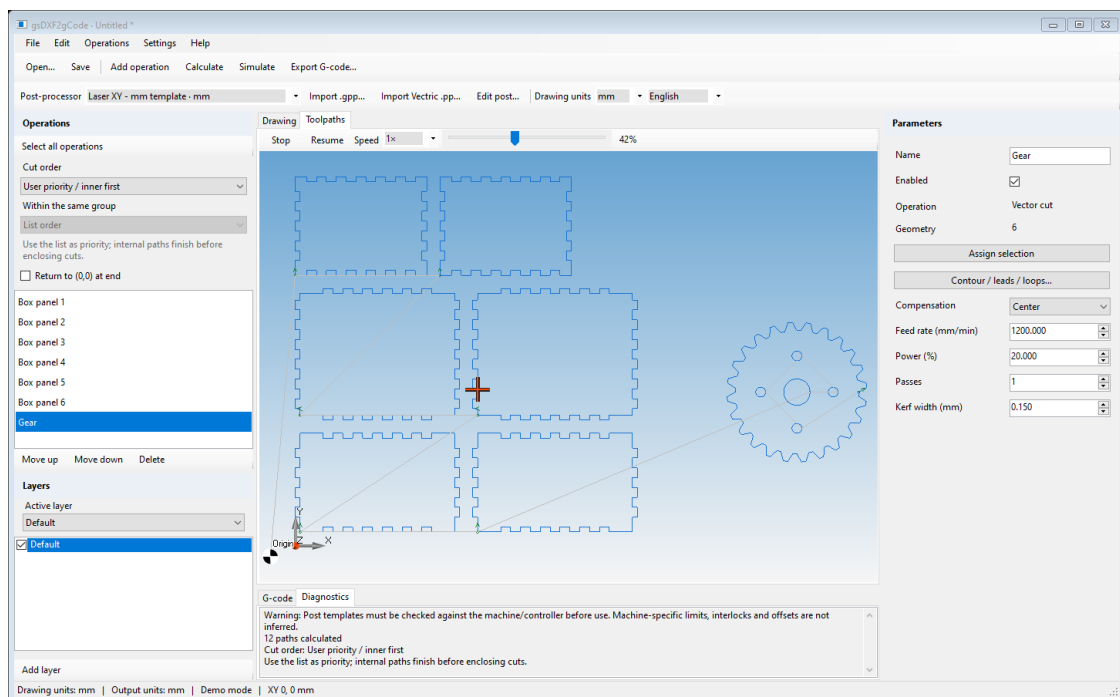
Rectangular nesting: enter part sizes and quantities, sheet size and spacing. Preview each resulting sheet.

DXF nesting arranges imported drawing parts. Pipes creates flat developments for cylinders, elbows, S/U bends and intersections. Set the dimensions and preview both the 2D development and the assembled 3D shape. Inserting a pipe adds its flat contours to the drawing. Common cut is available for supported bend layouts; inspect shared cuts and compensation before calculation.



Pipes: the 3D assembly helps you understand the final shape before inserting its flat development.

Calculate and simulate



Toolpaths: inspect the approach, the cutting sequence and the final return.

Calculate again after changing geometry, operations or the post. Use Restart/Stop and Pause/Resume, adjust Speed and drag the progress slider to inspect any part of the job. The crosshair marks the current position. Simulation shows the calculated paths; it is not a machine-controller emulator for arbitrary custom commands. Save the project before exporting.

03 Post-processor: setup and commands

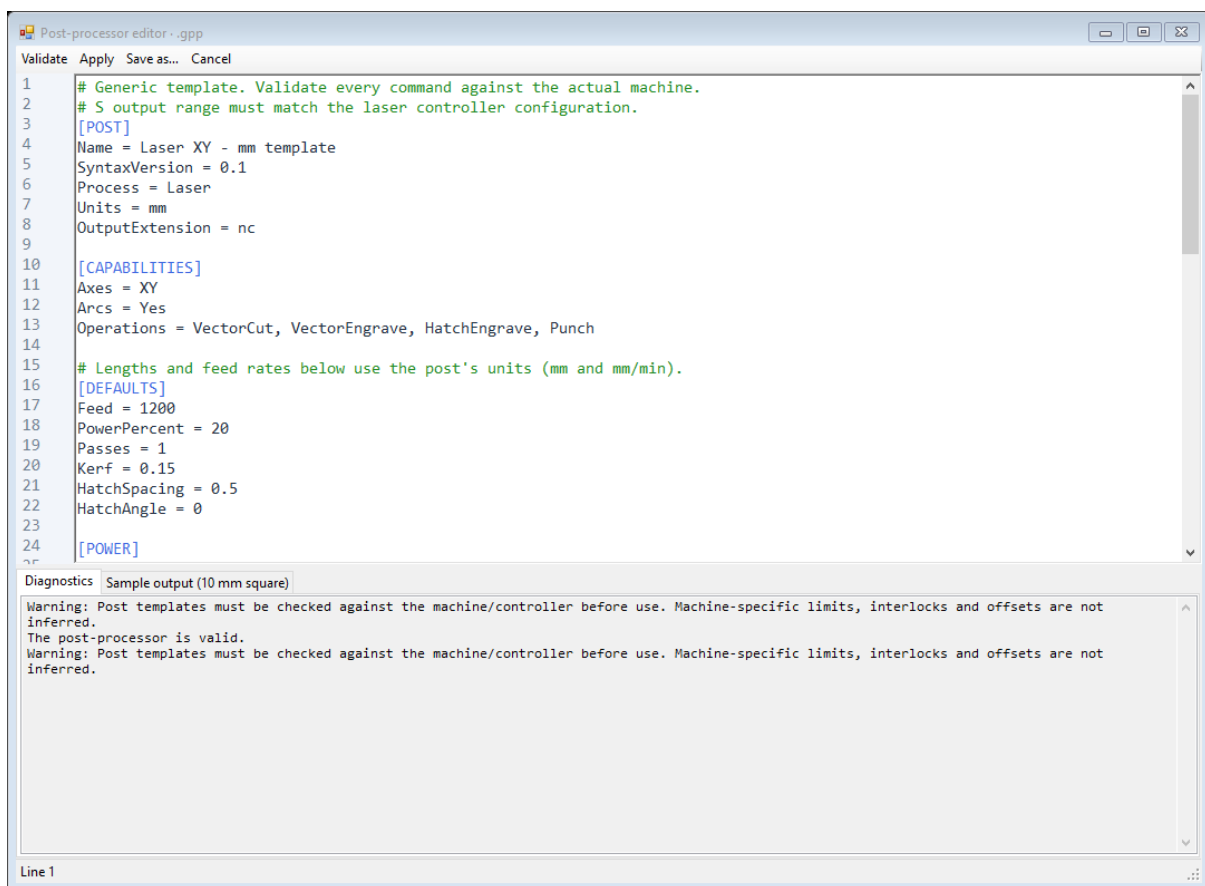
The post-processor translates calculated paths and operation settings into the commands expected by the machine controller.

Choose and edit a post

Choose the process before creating operations. The supplied Laser XY, Waterjet XY, Plasma XYZ and Router XYZ templates are starting points; Router also has an inch template. They are not certified profiles for every machine. Match units, axes, power/spindle commands and clearance heights to your controller.

Import .gpp loads a Gardos post. Import Vectric .pp converts supported definitions; review all diagnostics and the resulting .gpp, because not every Vectric feature is supported. Edit post opens the editor. Validate checks the syntax and produces a 10 mm square sample. Apply uses the valid post in the job. Save as writes a separate .gpp copy; export and generated sample text require activation. Save the project to retain its post settings.

After any post change, inspect existing operation values and calculate again. A successful Validate checks the post syntax, not the suitability of every command for your machine. Plasma templates do not configure probing or torch height control; the router template assumes a fixed tool and does not implement automatic tool changes.



Post editor: sections and variables above, validation messages below. Edit a copy when adapting a template.

How a .gpp is organized

Sections use [BRACKETS]. Settings use Key = Value. A line beginning with # is a comment. Command sections contain the text to output; values in {BRACES} are replaced automatically. Keep section names, keys and variable names exactly as shown, regardless of interface language.

[POST]	Name: display name. SyntaxVersion = 0.1. Process = Laser, Plasma, Waterjet or Router. Units = mm or inch. OutputExtension = nc, for example, without a dot.
[CAPABILITIES]	Axes = XY or XYZ; Router requires XYZ. Arcs = Yes or No. Operations is a comma-separated list of supported operation names. Profile/Pocket require Router.
[DEFAULTS]	Initial operation values. Lengths and feed rates use the post units and units/min. Review actual operation settings after importing a different post.
[POWER]	OutputMin/OutputMax map PowerPercent to POWER_S. With 0..1000, 20% produces S200. Match this range to the controller.
[FORMAT]	CoordinateDecimals, FeedDecimals, PowerDecimals: decimal places (0..8). DecimalSeparator = . only. LineEnding = CRLF or LF. LineNumberStart/Increment/Maximum control {LINE_NUMBER}; add N{LINE_NUMBER} where needed.

Operation defaults

Feed / PlungeFeed	Cutting feed / Z plunge feed.
PowerPercent / SpindleSpeed	Process power percentage / spindle speed. They drive different S variables.
SafeZ / CutZ	Clearance height / final cutting Z. SafeZ must be above CutZ. The router template uses stock-surface Z0 and negative cutting depths.
PierceZ / PierceDelay	Plasma piercing height / delay. The template writes G4 P for the delay; verify the controller's P time unit.
Kerf / ToolDiameter	Cut width for compensation / cutter diameter for router calculations.
HatchSpacing / HatchAngle	Distance between hatch lines / hatch direction in degrees.
Passes / StepDown	Repeated passes for XY/Punch / maximum depth increment for milling.
ToolNumber	Tool identifier available as {TOOL_NUMBER}. Setting it alone does not add a tool-change command.

When command sections run

[PROGRAM_START] / [PROGRAM_END]	Once at program start/end: units, positioning mode and shutdown.
[OPERATION_START] / [OPERATION_END]	Optional hooks around each generated path/pass, not just once per item in the operation list.
[RAPID_MOVE]	Position at the next start with X/Y. Also used for the optional final return.
[PATH_START] / [PATH_END]	Enable/disable the process around a contour, for example M3/M5. Plasma may include piercing here.
[PLUNGE_MOVE] / [RETRACT_MOVE]	Required for XYZ: descend at plunge feed and retract to SafeZ before travel and after a contour.
[FIRST_LINEAR_MOVE] / [LINEAR_MOVE]	Optional special first straight move, then ordinary straight moves. Without FIRST_LINEAR_MOVE, LINEAR_MOVE handles all.
[ARC_CW] / [ARC_CCW]	Clockwise G2 / counterclockwise G3. Required when Arcs = Yes; each section contains one arc command.
[PUNCH]	Required when Punch is offered. Replaces PATH_START, plunge, cutting moves and PATH_END for a punch point. Usually outputs {PUNCH_COMMAND}.

The six base command sections are PROGRAM_START, RAPID_MOVE, PATH_START, LINEAR_MOVE, PATH_END and PROGRAM_END. Add the XYZ, arc and Punch sections required by your capabilities. Typical XY sequence: start program -> rapid -> process on -> cutting moves -> process off -> next path -> end program.

Variables inserted in commands

{X} {Y} {Z}	Absolute destination coordinates in the selected output units.
{I} {J}	Arc-center offsets from the arc start, along X/Y. Only in arc sections; these are not absolute center coordinates.
{FEED} {PLUNGE_FEED}	Cutting/plunge feed, converted to output units per minute.
{POWER_S} {POWER_PERCENT}	Mapped S value / original percentage. Prefix POWER_S with S where required.
{SPINDLE_SPEED} {TOOL_NUMBER}	Spindle speed / tool number from the operation settings.
{SAFE_Z} {CUT_Z}	Safe height / current path Z, including the current depth pass. CUT_Z follows that path, not always the final depth.
{PIERCE_Z} {PIERCE_DELAY}	Piercing height / delay value for the plasma sequence.
{PATH_INDEX} {LINE_NUMBER}	Path/pass counter starting at 1 / configurable output line number. Numbering needs an N prefix in the template.
{UNITS_CODE}	G21 for millimeters or G20 for inches. Prefer this variable to a fixed units command; check the job units before export.
{PUNCH_COMMAND}	The punch command configured in the operation. It is not a universal command for every punching machine.

G-code commands used by the templates

G0 / G1	Rapid positioning / straight movement at feed F. X/Y/Z specify the destination.
G2 / G3	Clockwise / counterclockwise circular motion in XY; I/J locate the center relative to the start.
G17 / G90 / G91.1	XY plane / absolute X/Y/Z / incremental I/J arc centers. G91.1 does not make X/Y incremental. Ordinary G91 is unsupported.
G20 / G21 / G94	Inches / millimeters / feed in units per minute. F1200 under G21/G94 means 1200 mm/min.
M3 / M5 / S	Templates use M3 to enable spindle/process and M5 to disable it. S sets spindle speed or mapped process power. Exact laser/torch behavior depends on the controller.
G4 P / M2	Dwell with a P value / program end. Confirm P time units and supported end codes in the controller manual.

Example: from template to movement

```
[LINEAR_MOVE]
G1 X{X} Y{Y} F{FEED}

G1 X48.000 Y40.000 F1200.0
```

The first two lines define a post section. The last line illustrates its output for a destination of (48,40) and feed 1200 in a millimeter job. The braces disappear; the actual coordinates and feed are inserted. This excerpt is not a complete machine program.

Arcs and full circles

With Arcs = Yes, provide ARC_CW and ARC_CCW with X/Y/I/J/FEED and also Z for XYZ. PROGRAM_START must contain G17 and G91.1. Only XY arcs with incremental I/J are supported; radius R format, other planes and absolute arc centers are not supported. Settings > Curve output chooses arcs or G1 segments. Full circles use two semicircles. Arcs = No causes linear fallback.

For a circle of radius 10 centered at (0,0), starting at (10,0), the two example G3 moves below return to the start. Each I/J pair is measured from the start of that particular semicircle. Check that your controller supports the declared arc-center mode; if it does not, use line segments and a suitable line-only post.

```
G21
G90
G17
G91.1
G94
G0 X10.000 Y0.000
G3 X-10.000 Y0.000 I-10.000 J0.000 F1200.0
G3 X10.000 Y0.000 I10.000 J0.000 F1200.0
```

This is a geometry example, not a production-ready program: process start/stop and machine setup are intentionally outside the excerpt. The controller manual is the authority for supported commands. Reference definitions: LinuxCNC G-codes and M-codes, <https://linuxcnc.org/docs/stable/html/gcode/g-code.html> and <https://linuxcnc.org/docs/stable/html/gcode/m-code.html> . Using these references does not make a template a verified LinuxCNC machine profile.

04 Your first job

Work from the drawing to operations, then calculate, inspect and export. Keep the editable project as well as the final machine file.

What do the main areas of the window show?

Drawing is where you select and edit geometry. Toolpaths shows calculated movements and simulation controls. The left side lists operations and layers. Selecting an operation displays its settings on the right. G-code and Diagnostics appear below the view.

The upper bars contain file commands, the post-processor, drawing units and language. Drawing tools appear below the Drawing tab. Zoom to fit is available on the viewport itself. When starting a new task, first check the units and the selected post-processor.

How do I turn my first DXF into G-code?

Use the following sequence for a simple cutting job. Export requires activation; in Demo you can prepare, simulate and save the editable project. Automatic operations may already be present after opening the DXF, depending on your import preference. Inspect them before adding more, so the same geometry is not assigned twice.

The post-processor must match the intended process and machine. Simulation helps you inspect the movements, but does not demonstrate that every setting suits your actual machine or material.

1. Choose the appropriate post-processor, then Open your DXF.
2. Confirm Drawing units is mm and measure a known dimension.
3. Inspect the operations on the left. Add Cut or Profile only where needed.
4. Set feed, power or depth, compensation and any required passes.
5. Press Calculate, read Diagnostics, then inspect Toolpaths and simulation.
6. Save the project, then Export G-code with an activated license. Check the exported file before running it.

What should I check about millimeters and inches?

A DXF opened as a drawing starts in mm. DWG follows its declared drawing units. Measure a known feature instead of judging size from the screen. A line that should be 100 mm long should measure 100 in an mm drawing.

Changing Drawing units reinterprets the drawing numbers; it is not a physical-size-preserving conversion. A 100-unit line becomes 100 inches if you switch that drawing to inches. If the imported numeric geometry itself needs conversion, use Scale with the appropriate factor, for example 25.4 from inch numbers to mm numbers. Then verify a known dimension again. The selected drawing unit also controls the generated output units; verify it again after choosing or importing a post.

How do I save my work and continue later?

Use File > Save or Ctrl+S to save an editable .gscam project. Save As creates another file, useful before trying a different machining strategy. Reopen the project with Open; supported older .gdx projects can also be opened.

The project retains the drawing, operations and their settings, work origin, cutting order and the selected post settings. Export G-code produces the machine file, not a substitute for the editable project. Export DXF is useful for exchanging drawing geometry, but is not the way to preserve the complete machining job.

How do I start over or undo a mistake?

File > New starts a new drawing; save the current project first if you want to keep it. Undo (Ctrl+Z) reverses supported editing actions, while Redo (Ctrl+Y) restores an undone action. A shape insertion and the operations created with it can be undone together.

Esc cancels the current drawing or point-picking command. Cancel in a shape dialog discards that pending insertion. After a geometry edit, calculate again before trusting the previous toolpath or exported file.

05 Drawing, selecting and editing

Use the Drawing tab to work on the actual geometry. Selection determines which objects an editing or machining command will use.

How do I draw a line, circle or another basic shape?

Open Drawing > Basic shapes and choose Line, Polyline, Rectangle, Circle, Arc, Ellipse, Spline or Text. Follow the prompt for the points and dimensions required by that tool. Use object snaps to reuse exact points from existing geometry.

Choose the active layer before drawing. Return to Select when you want to select existing objects. If a command is still waiting for another point, press Esc before starting a different task.

How do I place something precisely with snaps or coordinates?

Enable the object snaps you need in Object snap, such as endpoints or centers. Move the pointer close to the desired feature and accept the indicated point. A grid is a visual aid; seeing a grid crossing is not proof that an object endpoint was selected.

Where a point command offers dynamic input, enter the required X/Y values instead of estimating a position. Shape dialogs also offer Coordinates with insertion X and Y. Check the displayed unit, the reference point being requested and the position in the preview before accepting.

Which selection mode should I use?

Select opens the available picking modes. By pick is useful for individual objects. Crossing box or polygon includes objects touched by the boundary as well as those inside it. Enclosed box or polygon requires the objects to be completely inside.

Drag to make a box. For a polygon, click successive corners and close it by clicking the first corner again; Esc cancels. Dynamic visible picking helps when working with visible objects. Changing the selection mode preserves the selection already made. Use the mode suited to the shape of the area you want to select.

How do I select everything, clear selection or select the opposite set?

Use Select all, Clear selection or Invert selection in the Select menu. Select all and Invert selection work on visible, selectable objects; hidden layers are not silently included.

Clearing the drawing selection also clears the related selection on the operations list. If the right-hand parameter area becomes empty, select the operation you intend to edit again. Do not confuse selecting geometry with enabling an operation: these serve different purposes.

How do I move, copy, rotate, resize or delete objects?

Select the objects, then choose the appropriate command from Modify. Move and Copy request placement points; Rotate and Scale use a reference and the requested value. Read the active prompt rather than clicking an approximate point. Mirror, Offset, Trim, Extend, Fillet and Chamfer are also in Modify.

Delete removes the selected drawing objects and updates their operation references. Deleting an operation from the operations list removes that machining assignment rather than deleting the drawing. After editing, inspect the operation list and calculate again. Use Undo if the wrong objects were changed.

How can I measure geometry and add dimensions?

Information contains Length / Details, Area / Perimeter and Vertex coordinates. Use them to check a known length, a closed shape or a precise point before creating operations. Dimensions contains linear, aligned, radial, diametric, angular and ordinate dimensions, plus leaders.

Temporary dimensions help while inspecting or editing; use the corresponding toggle and clear command when needed. Dimensions and annotations are for documentation, not automatically cutting contours. For machining text as outlines, use Linear text or Circular text from Complex shapes.

06 Organizing the drawing with layers

Layers organize geometry. Operations describe how that geometry will be machined; a layer name alone does not create a cutting instruction.

How do I create a layer and make it active?

Click Add layer below the Layers list, type a meaningful name and confirm. Choose it in Active layer, or double-click the layer in the list. New drawing objects and inserted generated shapes use the active layer.

For example, create separate layers for outlines, lettering and reference geometry. Existing objects do not move merely because you change the active layer. Imported files retain their own layer organization, and dimensions use their dimension layer.

How do I move existing objects to another layer?

Select the objects and choose Modify > Change layer. Choose the destination and confirm. Their geometry and machining assignments are retained. The command does not change your active layer.

If the destination layer is hidden, the moved objects will disappear from the view with that layer. Show it again to inspect them. Cancel leaves everything as it was, and Undo can reverse an accepted transfer.

Does hiding a layer stop its objects from being machined?

Use layer visibility to simplify the drawing view. Do not use hiding as a substitute for disabling a machining operation. An existing operation can still reference geometry on a hidden layer.

To exclude a cut from output, disable or remove the corresponding operation and calculate again. Hidden objects are excluded from bulk drawing selections and from initial automatic assignment when opening a DXF, which is a different stage of the workflow.

07 Importing and creating ready-made shapes

Complex shapes adds geometry to your drawing. Preview first, choose where it belongs and decide whether to create operations at the same time.

What is the difference between Open and Import file?

Open loads a drawing or saved project as the current document. To add another drawing to the document you are already editing, use Complex shapes > Import file. Choose a DXF/DWG, then place its base point in the existing drawing using snap or typed coordinates.

The imported base point is often (0,0), but can be defined differently in the source file. Imported layers and blocks are preserved; conflicting names are distinguished automatically. DXF is interpreted in mm, DWG uses its declared units, and appended geometry is converted into the current drawing units. Esc or right-click cancels the pending placement.

How do I insert a parametric shape, gear, polygon or finger-joint box?

Choose the corresponding command in Complex shapes. Parametric shapes offers dimensioned templates; the other commands have their own parameters. Fill in the dimensions in the unit shown by the dialog and press Preview. Check holes, widths, quantities and any reference image before insertion.

Use Pick insertion point (snap) to place it on the drawing after accepting, or choose Coordinates and enter X/Y. Create operations adds machining assignments using the current preference when available. If it is off, the geometry is inserted without operations and you can assign them later.

How do I create lettering in a straight line or around a circle?

Use Linear text or Circular text in Complex shapes. Enter the text, choose a font and set the requested size or circular placement parameters. Preview the resulting outlines before inserting them. A different font changes both the appearance and the openings inside letters.

The created outlines can be cut or engraved according to your chosen operation. Inspect the operations already created before adding a second one. For cut-out lettering, the inner-first order helps machine the openings before the letter outline. Letters with enclosed islands may need bridges in your design if those islands must remain attached to a surrounding sheet.

How do I arrange many rectangular parts on sheets?

Open Rectangular nesting, specify the sheet dimensions and the part sizes and quantities. Set spacing and whether rotation is allowed, then preview the arrangement. The total quantity limit is 100,000 parts, but a large job can require more time and several sheets.

Review each sheet using the sheet selector. Insertion uses the selected sheet, so repeat for other sheets when required. Allow enough spacing for the intended cut width, compensation and entry paths. A packing result that looks compact on screen is not by itself a suitable machining clearance.

How do I nest parts from DXF drawings?

Choose DXF nesting. Add the drawing files and quantities, enter the sheet dimensions, spacing and available rotation settings, then run the nesting calculation. Review the result, including holes and orientation, before inserting a selected sheet.

Changing quantities or dimensions invalidates the previous result: run the calculation again. Use clean, closed part outlines and check the imported dimensions. Nesting places parts; the operation settings and selected post still determine how those parts are machined.

08 Preparing machining operations

Geometry describes the part. An operation specifies how to machine it. Inspect every operation before calculation and export.

How do I create an operation for selected objects?

Select the required contours in Drawing, then choose an available operation from Operations. Select the new operation in the left list to edit its parameters. Name operations meaningfully, for example Holes, Outside cut or Marking.

The available kinds depend on the selected post-processor. Cut is used for vector cutting, Engrave for marking along contours, Hatch for filling an area with engraving lines, Profile for milling along a contour and Pocket for removing material from an area. Select the process first, then set its appropriate values; changing the name alone does not change the machining kind.

How can a DXF arrive with Cut already assigned?

In Settings choose the default DXF operation, for example Cut when supported by your post. Opening the next DXF can then create operations immediately. Choose None when you want to assign every operation yourself. This preference is for new imports; it does not rewrite the machining decisions in an existing saved project.

Generated shapes also offer Create operations. Check that option to insert the geometry together with the selected default operation. Clear it when you only need drawing objects or intend to assign different processes to separate contours.

Why can the same text or contour be machined twice?

The same geometry can belong to two operations, for example an automatically created Engrave followed by a manually added Cut. Simulation then shows both, and their ordering rules may differ. This is different from intentional depth passes within one operation.

When adding an operation to already assigned objects, leave Replace existing operations enabled if the new process should replace the old one. Clear it only when you intentionally want both processes. Review the left list, remove an unwanted operation and calculate again. Keep the drawing objects when you only want to remove their machining assignment.

How do Center, Inside and Outside affect the result?

Center follows the drawn contour. Inside and Outside move the toolpath to the chosen side using the cutting width or tool diameter. For a finished outside edge you usually place the tool outside; for a hole you usually place it inside. Check the displayed path rather than relying only on the label.

Offset compensation needs a valid closed contour. Tiny gaps, overlaps and self-intersections can prevent a valid result. Do not compensate the same width twice, once in the geometry and again in the operation. Common cut pipe layouts use Center because a shared boundary cannot independently offset for both neighboring parts.

Which values should I set for cutting or engraving?

Check the feed, power where supported, cutting width, number of passes and any process-specific delay. Use settings established for your machine and material. The application cannot infer a suitable power or speed merely from a DXF outline.

For milling, also check tool diameter, safe height, final depth and stepdown. Final depth below the work surface is negative; stepdown is a positive amount per pass. For Pocket, use a positive stepover no greater than the tool diameter. Recalculate after changing these values and inspect the pass count and clearance moves in simulation.

Can I edit several operations together?

Use the operation selection controls on the left, including Select all operations when appropriate. Inspect which operations are selected. In Common parameters check only the fields you want to change, enter their values and press Apply to selected operations; editing a value alone does not apply it. If the operations use different processes, some fields may not apply to every selected item.

For a change that should affect only one contour, select its own operation and verify the highlighted geometry. After a bulk change, inspect representative operations and calculate again. Saving a copy of the project before a major strategy change makes it easier to compare the previous result.

How do I choose where a contour starts and its direction?

Select the relevant operation or drawing contour and open its path settings. Choose the start point and cutting direction, then inspect the calculated path. When more than one contour is involved, identify the intended contour before applying its settings.

Entry and exit paths can move the start of machining away from the finished edge. Leave sufficient space for them around nearby parts and sheet borders. A lead that fits one contour may collide with an adjacent part after nesting. Common cut layouts restrict offset, entry/exit leads and loop features; start and direction choices remain available.

What is the difference between User priority / inner first and Manual operation order?

User priority / inner first uses your operation priorities while keeping inner cutting contours before the enclosing outside contour. It is useful when you want influence over the order without losing the hole-before-outline rule.

Manual operation order follows the operation order you set and does not automatically impose that nesting rule. Use it when you need a specific sequence and have checked the consequences yourself. Use Up and Down to order operations, calculate and watch the sequence. This orders operations, not every individual

contour within a multi-contour operation. A part cut free too early may move before its remaining features are machined.

When should I choose By nesting level or One family at a time?

By nesting level groups work by how deeply contours sit inside other contours. It is useful for completing a similar depth of internal features across a job before proceeding outward. One family at a time keeps a part and its contained features together before moving to another family.

These choices organize the machining order; they do not change the drawing geometry or automatically solve every production constraint. Inspect a representative part with holes and a neighboring part in simulation. For intentional exact sequencing between unlike operations, use Manual operation order and verify the resulting order explicitly.

Can I alternate cuts in distant areas to spread heat?

First choose By nesting level or One family at a time, then choose distant-cut alternation in the spatial ordering settings. It is unavailable in User priority / inner first and Manual operation order. It favors work away from the recently cut area while respecting the applicable cutting-order rules. This can help distribute cutting activity across the sheet.

It is an ordering aid, not a thermal calculation or a guarantee against distortion. Heat also depends on material, thickness, power, speed and how much material remains connected. Review the longer rapid moves and the inner/outer sequence. Exact Manual operation order takes precedence over automatic spatial reordering.

Where is Punch and when can I use it?

Punch is a machining operation, not a separate ready-made shape. It is available when the selected post supports that process. Select the relevant geometry, choose Punch from Operations and set the fields presented for the operation.

If Punch does not appear, check the selected post instead of looking for it in Complex shapes. A laser or router post may not support the same commands as a punching machine. Inspect the calculated result and ensure the selected post describes the machine process you actually intend to use.

09 Pipe developments and 3D preview

Create flat cutting patterns and inspect the nominal assembled surface. Dimensions describe the chosen geometry; manufacturing allowances must be considered separately.

Which tool should I choose from Complex shapes > Pipes?

Use Cylinder for a straight rolled cylinder, Elbow for a segmented bend, S bend for an offset connection and U bend for a return connection. Pipe intersection creates the joining outlines for a branch meeting a trunk pipe.

Enter the dimensions, press Preview and inspect both the flat pattern and 3D tabs before inserting. The result placed in the drawing is the 2D geometry. You can then choose its insertion point with snaps or coordinates and create machining operations just as for other generated shapes.

How do diameter, bend radius and segment count affect a bend?

Diameter defines the cylinder size. The bend radius field describes the inside turning radius; do not confuse it with an outside or centerline radius from another drawing. The first and last straight lengths extend the end sections. More segments make a segmented bend approximate the intended turn more closely, but add seams.

Elbow accepts 2 to 100 segments. S and U bends use an odd count from 3 to 99; follow the angle conditions shown for the selected form. After changing a count, angle or length, refresh Preview and check that the end directions and spacing match the intended assembly.

How should I set seam margins and curve tolerance?

A seam or margin allowance adds material to the flat layout where provided by the form. For a cylinder, the developed width is the circumference plus the margins. Curve tolerance controls the approximation of curved outlines: a smaller value gives finer detail but more segments.

The tools show nominal geometry; they do not automatically calculate neutral-axis corrections, springback, weld preparation or wall-thickness allowances. Set dimensions and allowances according to the fabrication method. The 3D view represents the nominal assembled surface, so a seam margin in the flat layout is not a visible thick overlap in that preview.

Can I place bend segments closer or use Common cut?

The flat bend layout packs matching segment boundaries closely, with a default gap of 1 mm. Adjust the gap to suit cutting width, entry paths and handling. Common cut is available for Elbow, S and U bends and shares matching boundaries so those lines are cut only once.

With Create operations enabled, the shared dividers are cut before the outer boundary in one centered operation. Common cut removes material from both adjacent pieces and does not independently compensate each finished edge. Use separate closed parts with adequate spacing when both edges require their own dimensional compensation. Do not add a second cut over the shared lines.

How do I inspect the assembled result in 3D?

Select the 3D preview tab in the pipe form. Drag with the left mouse button to rotate, use the mouse wheel to zoom and use its fit control to bring the complete assembly back into view. Rotate far enough to inspect both the joining side and the rear side.

After editing parameters, press Preview again; both previews update while the chosen tab remains selected. The view helps you understand orientation, bend shape and intersections. It is not a thick-walled solid or a manufacturing simulation, and the geometry inserted into the drawing remains the flat development.

What should I check for a pipe intersection?

Enter the trunk diameter and length, the branch diameter and length, the intersection angle and the seam position. The branch diameter must not exceed the trunk diameter. The branch length is measured from the intersection of the axes; allow for that when comparing it with an exposed length on a workshop drawing.

Choose whether to create both developments, only the trunk or only the branch. Rotate the seam to place the opening sensibly on the flat pattern. If an opening crosses the seam, it can appear as matching edge cutouts rather than one central hole. Check both 2D and 3D before cutting.

10 Checking the toolpath and simulation

Calculate after changes, read diagnostics, then inspect the complete movement sequence before exporting.

How do I calculate and inspect a toolpath?

Press Calculate after defining the operations and their parameters. Open Toolpaths to see the resulting movements and read Diagnostics for warnings or errors. Use the viewport fit icon to display the whole result, then zoom into a hole, corner or lead that needs closer inspection.

Changing geometry, operation parameters, cutting order or the work origin requires a fresh calculation. Do not assume a previously displayed toolpath still represents the changed project. If calculation reports a problem, fix the stated contour or value before exporting rather than judging only the portions that remain visible.

How do Restart, Pause and Stop work?

Restart starts the simulation from the beginning. During playback the controls allow you to pause or stop; Resume continues from the paused point. The crosshair marks the current tool position and stays a readable size as you zoom.

For a detailed check, pause close to a transition between operations and inspect the entry move, contour direction and exit. The G-code pane highlights the current line when code display is available. In Demo you can inspect the simulated movements, while the G-code text and file exports require activation.

Can I change speed and jump to a particular point?

Choose a playback speed from 0.25x through 16x. Slow playback is useful around entries, small holes or changes of direction; faster playback helps review a long job. The setting changes the visualization speed, not the programmed feed sent to the machine.

Drag the progress slider to move to another part of the calculated path. It works while playing or paused, so you can stop at a difficult area, move backward and inspect it again. The percentage indicates progress along the simulated path, not a guaranteed real machining time including every controller delay.

Does simulation start at (0,0), and can the job return there?

The initial rapid move from the work origin (0,0) to the first programmed XY position is shown. This lets you see the approach instead of starting the display at the first contour. Set or reset the work origin from Settings when the job zero needs to change.

Enable Return to origin to add a final move back to work XY (0,0). For an XYZ process the return follows the safe-height handling. Work origin is the job coordinate origin; it is not a command to home the machine or locate its physical limit switches. Inspect the approach and return before exporting.

What does a successful simulation confirm?

It helps you check the calculated path, contour order, travel moves, directions, entries and repeated passes. Confirm that holes are completed before a part is released where that is required, and that no contour appears twice unintentionally.

It does not model the full machine, clamps, stock movement, thermal distortion or every controller-specific action. Check clearances and process values against the actual setup. Use the machine manufacturer's normal verification and trial procedures before production. Zoom to fit is built into both Drawing and Toolpaths views, so you can review either without a separate toolbar button.

11 Saving and exporting a finished job

Save the editable project separately from the machine file. Every export requires an activated license; Demo can prepare, calculate and simulate a job.

How do I select the right post-processor?

Choose the post matching your process and axes, and confirm the selected Drawing units for the output. The supplied choices include Laser XY mm, Plasma XYZ mm, Waterjet XY mm, Router XYZ mm and Router XYZ inch. A post determines the available machining kinds and how the machine file is written.

Do not select a post solely because its name sounds similar to your machine. Confirm the units, axis handling and supported commands with your machine requirements. Changing the post can change available operation fields; inspect existing operations and calculate again before using the new output.

Can I import or adjust a post for my machine?

Use the post controls to import a supported .gpp or .pp file and inspect it in the post editor. Review the displayed units, file extension, axis options and process commands before applying it. A saved copy is useful when adapting an existing post for another machine.

Use a post supplied or verified for the intended controller. Importing a file successfully does not prove that its commands are suitable. After changing or applying a post, calculate a small representative job and review its output. Exporting a post file with Save as is also unavailable in Demo.

How do I export G-code or exchange the drawing as DXF?

With an activated license, calculate the job and resolve diagnostics, then choose Export G-code. Choose the destination and file name and inspect the resulting machine file. Keep a separate .gscam project so you can change geometry or operations later.

For exchanging drawing geometry, choose Export DXF from File. This is a drawing exchange file, not the complete machining job and not the machine program. All export commands are blocked in Demo, including G-code, DXF and post-file export. Saving an editable project remains available, so you can keep work prepared before activation.

What should I review before sending the file to the machine?

Check the post, output units, work origin, overall dimensions, cutting order, feed, power or depth, safe height, compensation and pass count. Read Diagnostics and inspect the complete simulation, including the first approach and any return to origin.

Compare the file with the controller's accepted program format and the actual machine setup. A project save and an export are separate actions; after a change, recalculate and export again rather than using an old file with the same name. A clear job name or revision makes it easier to identify the approved file at the machine.

How can I output circles as arcs or control the line-segment density?

Open Settings > Curve output. Circular arcs (G2 / G3) is the default. Recognized circular contours use G2 or G3 according to their direction, and a full circle is written as two semicircles. Simulation follows the circular path.

Choose Line segments (G1) to approximate curves with straight segments. Set Maximum segment length in the displayed drawing unit: for example, 0.5 mm gives denser segments than 2 mm. Segments can be shorter where needed to follow a curve accurately. Press OK, then Calculate again. The choice and length are saved in the project.

The post must support circular arcs. If the selected post only supports lines, the program uses segments and reports this in Diagnostics. For an older saved project, select a current arc-capable post and check its machining settings. Noncircular curves and paths already converted to segments remain linear; a polygon is not automatically converted into a circle.

12 Troubleshooting, language and licensing

Start with the visible drawing, the operation list and Diagnostics. The questions below help identify common causes without discarding the project.

Why is an imported drawing the wrong size?

Measure a dimension whose real value you know. A DXF opens in mm, so coordinates originally written as inch numbers need the appropriate geometry scaling if they were not converted by the drawing author. A 25.4 ratio is a useful clue, not a reason to scale without measuring.

Do not try to correct the physical size simply by changing the units label. Confirm the source dimensions, apply the intended Scale factor and measure again. When importing into an existing drawing, also check the current drawing units and the insertion location.

Why do I see geometry but no machining operations?

The default import operation may be None, or Create operations may have been cleared in the shape dialog. Geometry alone is not a machining instruction. Select the intended contours, add the appropriate operation and enter its parameters, then calculate.

Also check whether you have opened Drawing instead of Toolpaths or changed the post to one that does not support a previous process. Inspect the operation list rather than repeatedly importing the same geometry; a second import can create overlapping duplicates that later lead to repeated cuts.

What should I do when a contour must be closed?

Zoom into the reported contour and inspect its endpoints, especially at corners and joins. A shape that looks closed at a distant zoom may still contain a small gap, duplicate segment or crossing. Use endpoint snaps when correcting it and recheck the outline before recalculation.

Inside/outside offsets and area operations need suitable closed boundaries. An open line may be appropriate for centered cutting or engraving, but changing the process merely to suppress an error can change the finished part. Decide whether the intended feature is an open path or a closed area, then correct its geometry and operation accordingly.

Why did an object disappear, or why is it still selected in the tree?

Check layer visibility first, then use the view fit control and confirm the active Drawing or Toolpaths tab. A hidden layer is different from deleted geometry. Undo can restore a recent supported deletion. Deleting drawing objects also updates their operation references, so the left list should not retain a stale machining assignment.

Use Clear selection to clear selection consistently in the view and corresponding tree. If several objects overlap, check whether there are actual duplicates before deleting. Save a copy of an unfamiliar project before extensive cleanup, then calculate again after changes.

How do I change language and open the instructions?

Choose a language from the main language selector: English, Greek, German, French, Spanish or Italian. The first item in Help opens the PDF guide for the selected language; F1 opens the same guide. The manuals are installed locally and can be read without an Internet connection.

A PDF reader must be associated with PDF files on the computer. If the guide is missing or cannot be opened, check that the application was installed completely and that a PDF reader is available. Some ready-made shape names and parameter captions remain in English in languages other than Greek; the guide explains those names in context.

What can I do in Demo, and how do I activate or deactivate?

Demo lets you draw, import, configure operations, calculate, simulate and save the editable project. It does not allow any export: G-code, DXF and post-file exports require activation. Generated G-code text is not displayed in Demo.

Choose Activate in Help and complete the activation form with your license information. A success message confirms activation. When active, Help offers Deactivate instead. It opens the same form, displays the serial number (SN) and provides the Deactivate button. Successful deactivation is confirmed and returns the application to Demo export restrictions. Keep your license details for future activation.

Where can I find Gardos Software CNC products and support information?

The Gardos Software website item in Help opens <https://store.gardos-software.com/en/#cnc> in your browser. An Internet connection is needed for the website, while the installed PDF guide remains available offline. About shows the application version and publisher.

When describing a problem, include the application version, selected post, drawing units, the action that caused the problem and the relevant Diagnostics message. A small example project that reproduces the behavior is more useful than a large unrelated drawing. Remove any confidential design content before sharing a project.