



TopSolid
2025



| What's New
TopSolid 6.26



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Version 6.26

Rev. 01

Note: If you are experiencing problems using this document, please feel free to send your feedback and comments to edition@topsolid.com.

Tip:

Any questions about TopSolid? Find the answers you need in our comprehensive [TopSolid'Faq](#)! It's your go-to resource for solutions to the most common queries.

Elevate your skills with TopSolid'Learning. Ready to take your skills to the next level? Whether you're a beginner or looking to sharpen your expertise, our online platform, [TopSolid'Learning](#) is designed to help you grow. Learn at your own pace and unlock the full potential of our software, making it an essential part of your projects.

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Welcome to TopSolid 2025!

Discover the innovative brand-new features and significant enhancements of **TopSolid 6.26**, designed to elevate your user experience and boost your daily productivity.

For detailed insights on these updates, contact your local branch for expert guidance tailored to your needs.

What's New in TopSolid'Wood 6.26

Explore the latest features and enhancements in **TopSolid'Wood**, the leading CAD/CAM software for woodworking professionals!

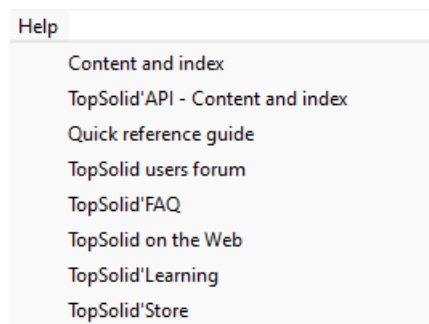
User interface

Simplified access to TopSolid'Learning and TopSolid'Faq

[TopSolid'Learning](#) is now directly accessible from the home page.



Additionally, links to [TopSolid'Faq](#) and [TopSolid'Learning](#) have been added to the **Help** menu, simplifying navigation to these platforms.

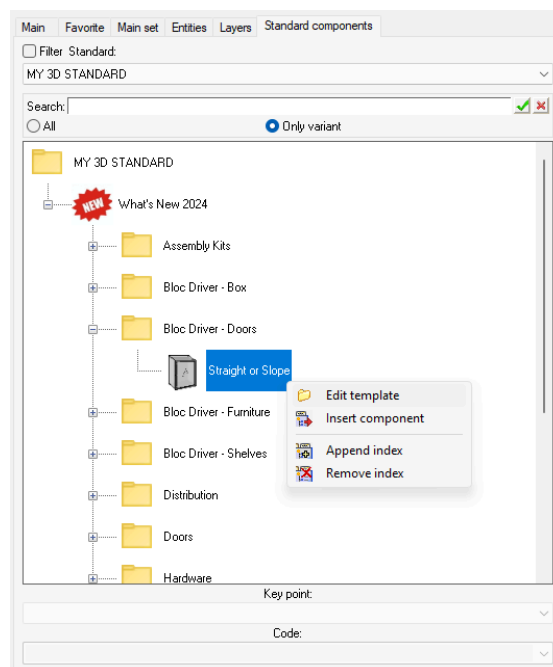


Direct access to Group and Config folders

You can now directly access the **Config** and **Group** folders in the File Explorer via the **Help** > **Configuration** menu.

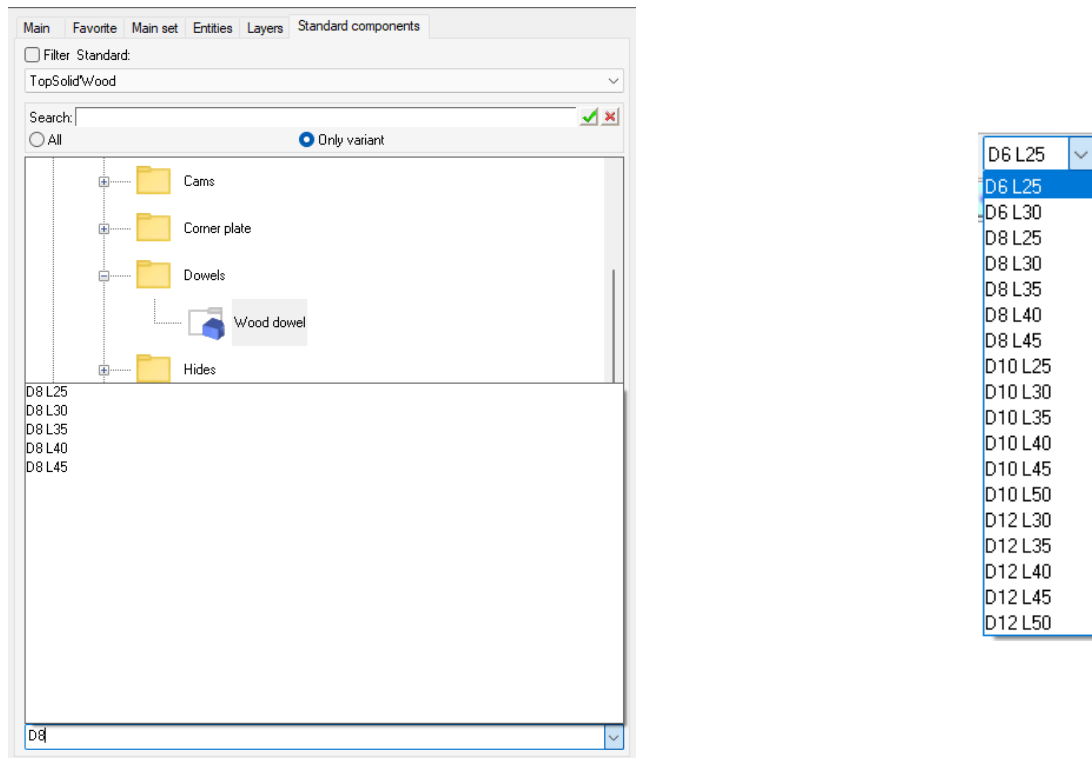
Opening a component from the tree

You can now edit a component model by right-clicking on the corresponding node in the standard component tree.



Code entry

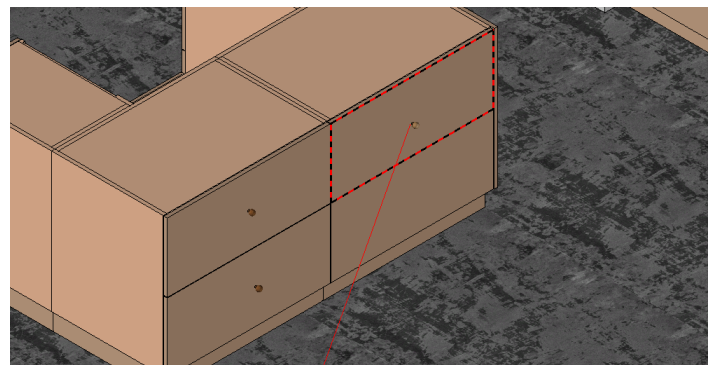
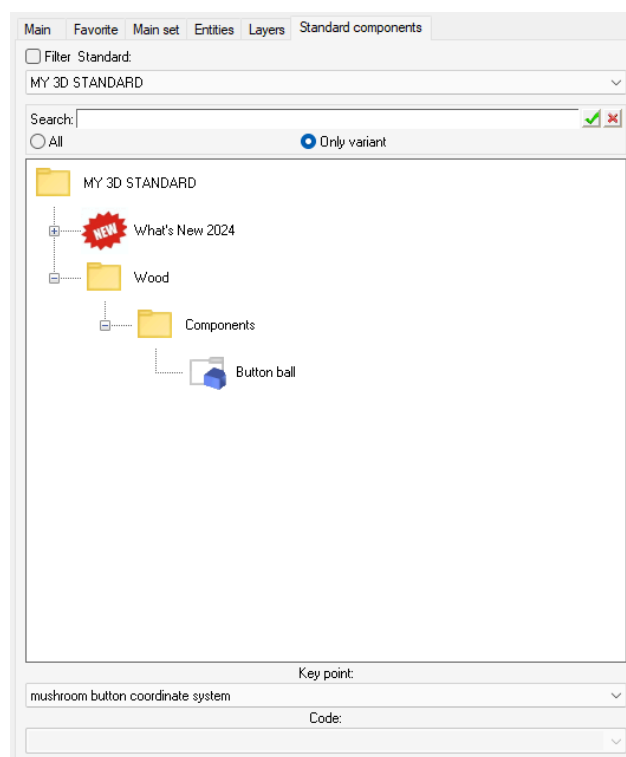
When selecting a standard component, you can now enter a code directly in the dedicated field.



Example of a dowel with a list of various codes.

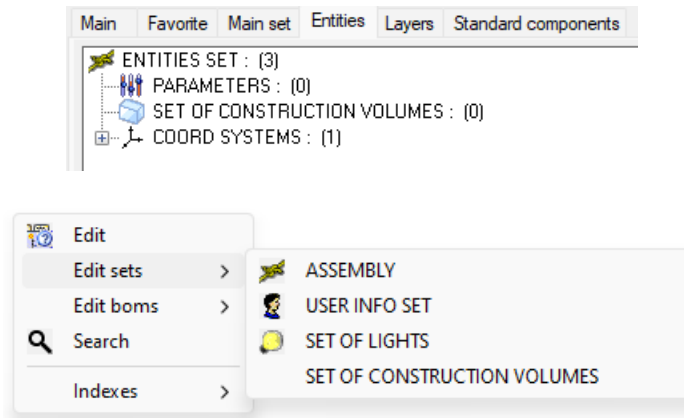
Inclusion of the same component

When a component with at least one driver is included via the tree, the **Same component** button becomes available. This feature allows for quick and easy re-inclusion of the same component.



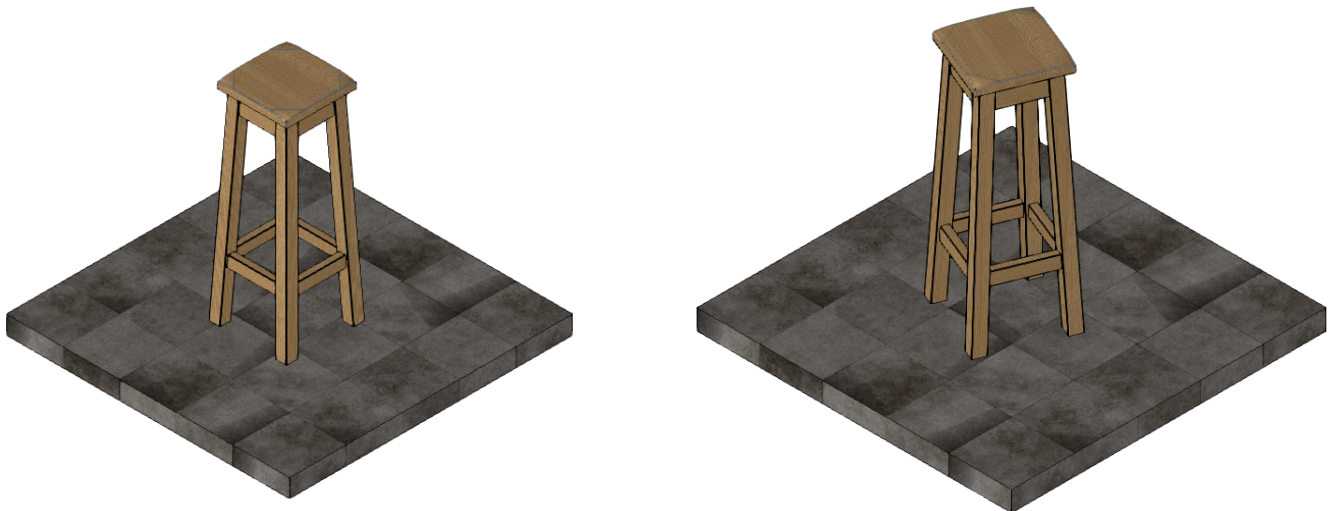
New "Construction volumes" section in Entities tree

The **Set of construction volumes** section is now visible in the Entities tree when a new document is created.



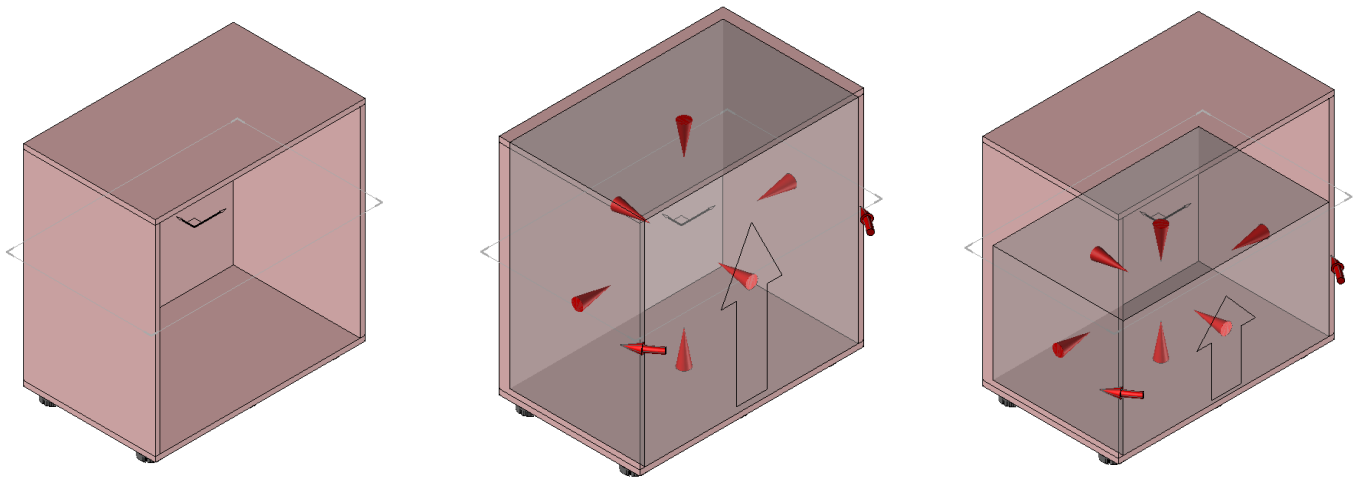
Rotate command in neutral mode

In neutral mode, the rotate command is now available after clicking on a component.



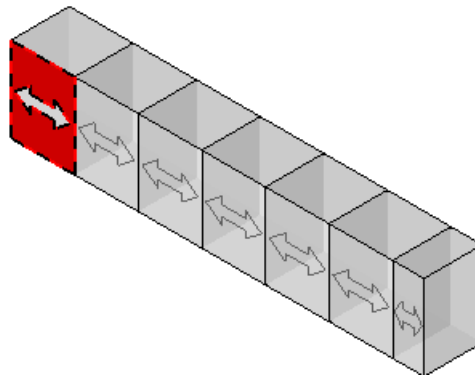
Reconnecting driver block arrows to coordinate systems

You can now connect or reconnect driver block arrows to coordinate systems, including those from a component.



Modification of a distributed component in neutral mode

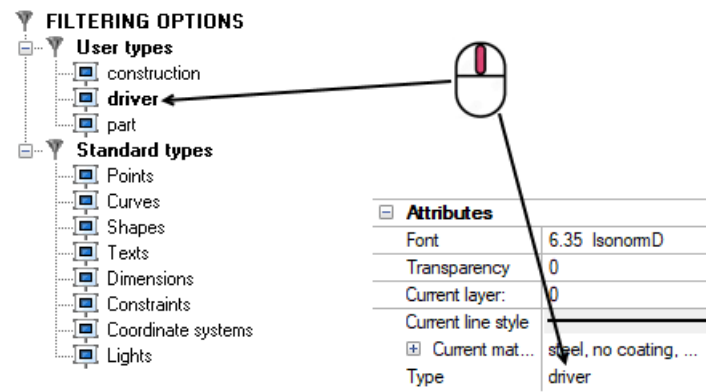
In neutral mode, a left-click on an element from a distribution now allows direct modification using the wrench icon.



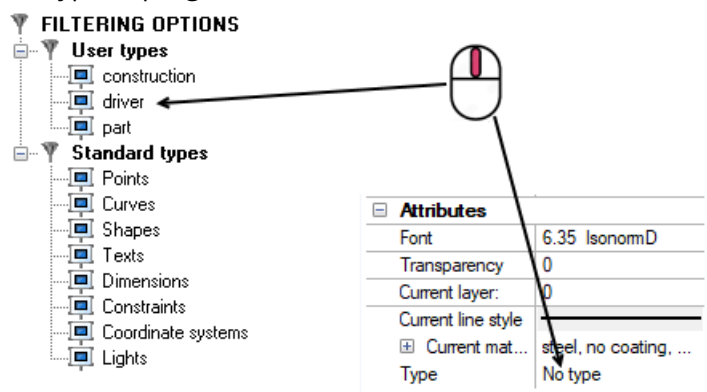
Purging the current part type

New middle-click actions in the **Filtering options** tab offer improved management of the current part type:

- Middle-click to activate the selected part type.



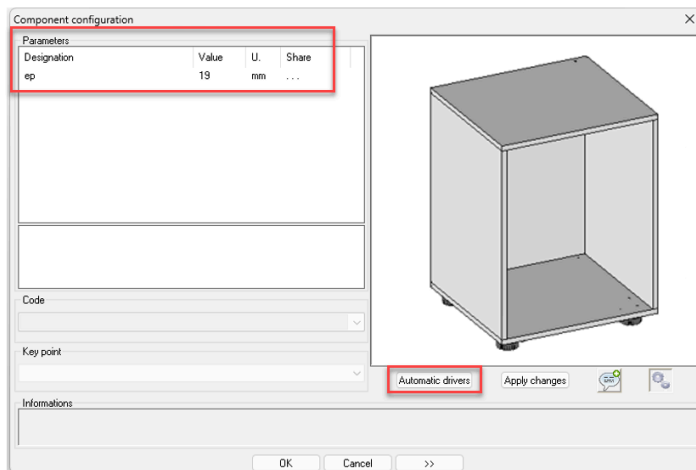
- Middle-click on an active part type to purge it.



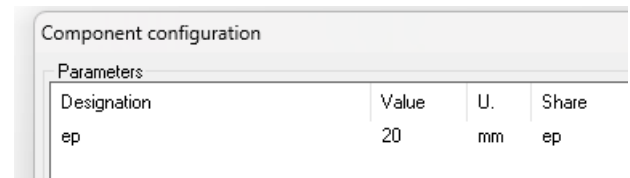
Driver merging

When including a component with drivers, the **Automatic drivers** button now merges the component parameters with those already present in the inclusion document.

For example, when including a cabinet with a driver parameter named identically to an existing parameter in the document, using the **Automatic drivers** button automatically connects the cabinet driver to the inclusion document's driver.

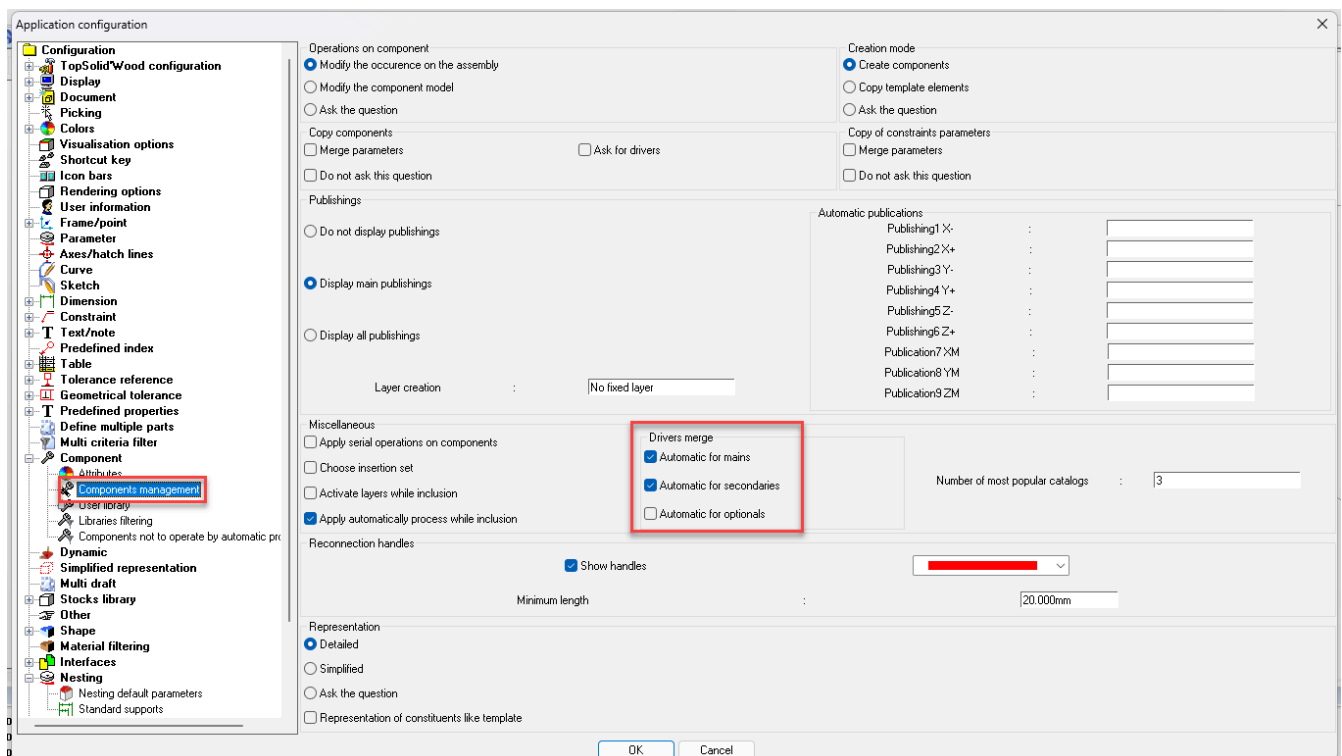


Example with the **ep** parameter.



After using the **Automatic drivers** button, the **ep** parameter is now shared, with its updated value displayed.

In addition, new options in the component management parameters allow configuration of behaviors for various types of drivers.



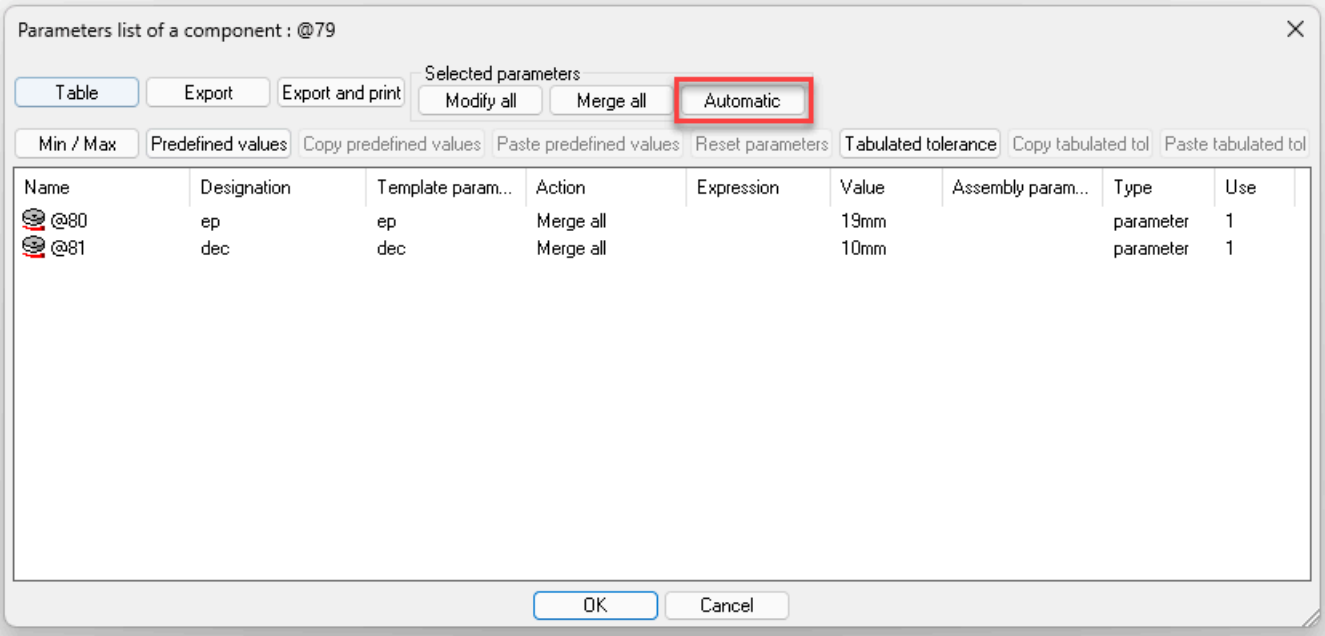
The associated configuration words in the top.cfg file are:

- D_COMPO_DRIVER_AUTOMATIC_FOR_MAINS 1
- D_COMPO_DRIVER_AUTOMATIC_FOR_SECONDARIES 1
- D_COMPO_DRIVER_AUTOMATIC_FOR_OPTIONALS 0

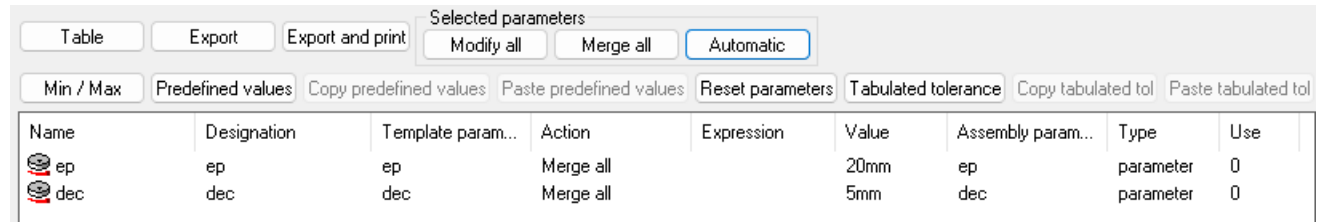
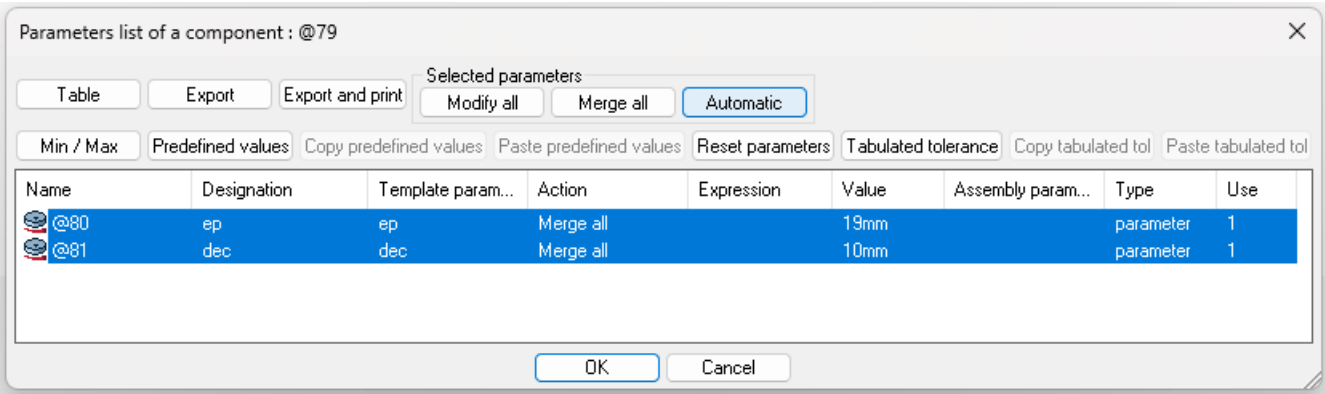
Important: This new feature does not apply to Boolean drivers.

Driver merging after inclusion

A new **Automatic** button is available in the multiple parameter modification window.



It connects the assembly drivers to the component drivers with the same name, for the selected lines.

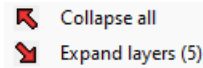


Result after driver merging.

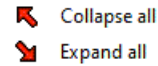
Changing the number of expanded nodes

The **D_TREE_EXPAND_NUMBER** configuration word has been added to the top.cfg file, with a default value of 5.

The value set for this configuration word determines the number of expanded nodes in the construction tree when the **Expand layers** command is used on a line.

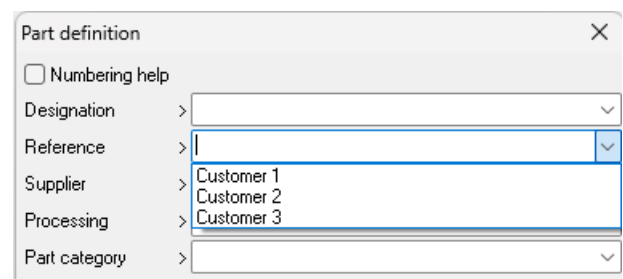
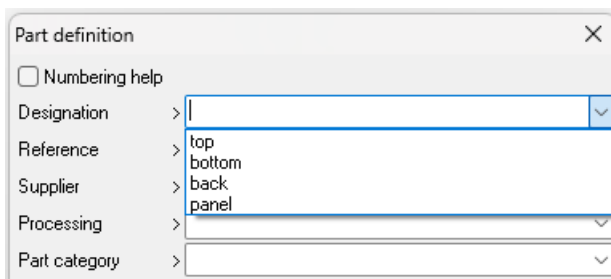


A value of 0 restores the behavior of the **Expand all** command.



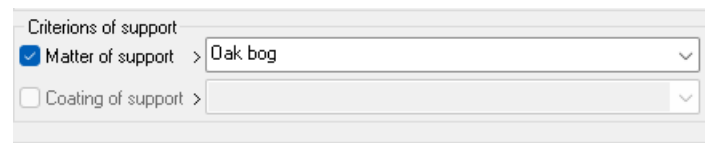
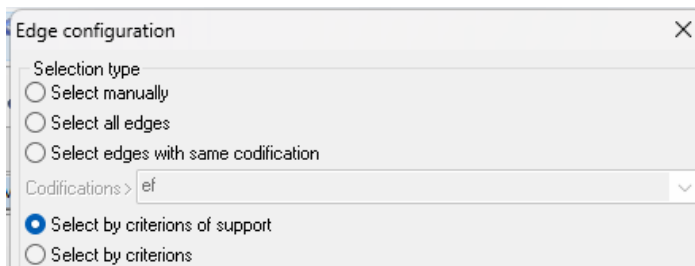
Using predefined designations and references

It is now possible to access predefined designations and references set in **Tools > Options**.



Edge selection by support criteria

In the edge configurator, you can now select edges based on the **Matter of support** and **Coating of support** criteria.



Customizing value increments

Parameter values can now be modified using the keyboard's arrow keys.



New configuration words are available to customize parameter increments:

- **D_PARAM_LENGTH_DELTA**: defines the default pitch for a length parameter (default 10mm)
- **D_PARAM_NOUNIT_DELTA**: defines the default pitch for a unitless parameter (default 1)
- **D_PARAM_ANGLE_DELTA**: defines the default pitch for an angle parameter (default 5)

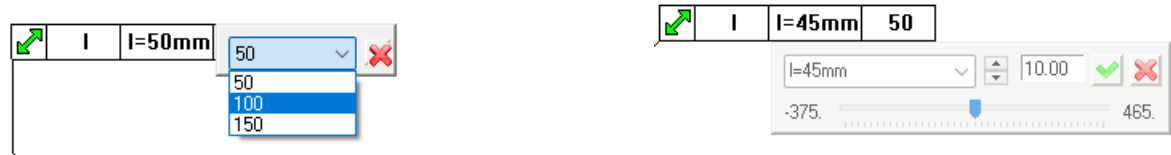


If mouse scrolling is used and the parameter has predefined minimum and maximum values, the pitch will be calculated as: $(\text{MaxValue} - \text{MinValue}) / 40$.

Access to predefined parameter values from the label

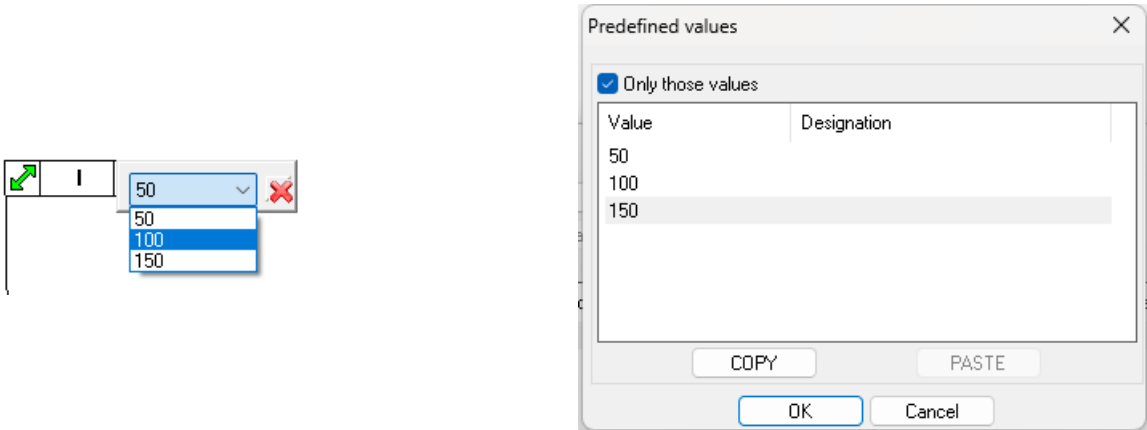
You can now directly access predefined values for a parameter from its label.

Mode with predefined values



This mode allows you to select a value from the predefined options directly from the label. You can still modify the value of parameter I without using the predefined values.

Mode with strict values



*In this mode, the parameter has predefined values and the **Only those values** option is checked, limiting choices to the specified values.*

Design

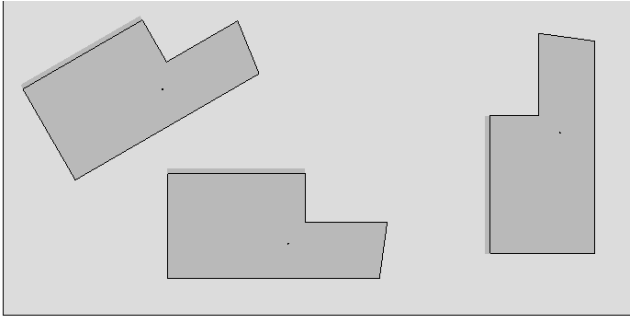
Orientation angle between the part's sawing-up coordinate system and the nesting support

A new BOM property, **NESTING_LABEL_ANGLE_2**, indicates the value of the angle between the nesting support and the part's sawing-up coordinate system.

Creation or modification of bom file

Title	Definition
ELEMENT IDENTIFI...	ELEMENT_IDENTIFIER
DESIGNATION	DESIGNATION
LABEL ANGLE	NESTING_LABEL_ANGLE
LABEL ANGLE 2	NESTING_LABEL_ANGLE_2
Double click to enter a value	

90	90
0	0
30	30
0	0
LABEL ANGLE	LABEL_ANGLE_2



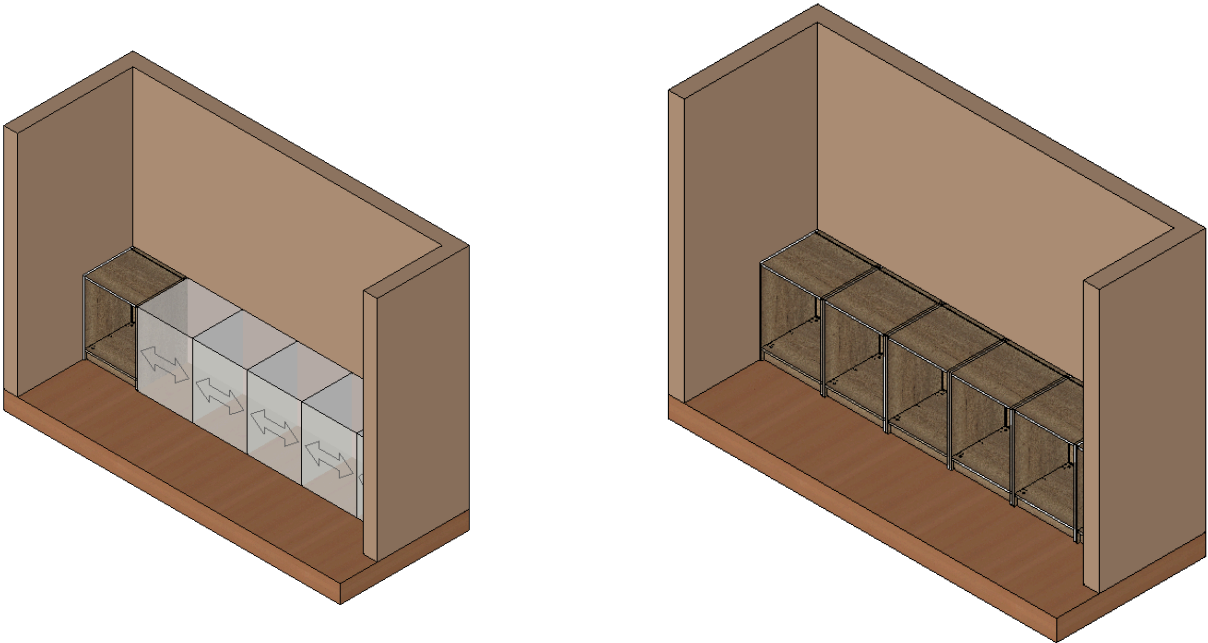
In the example above, the machining coordinate system is set identically to the sawing-up coordinate system.

The **NESTING_LABEL_ANGLE** property indicates the angle between the nesting support's X axis and the part's machining coordinate system, while the new **NESTING_LABEL_ANGLE_2** property gives the angle between the nesting support's X axis and the part's sawing-up coordinate system.

New button for serial copy

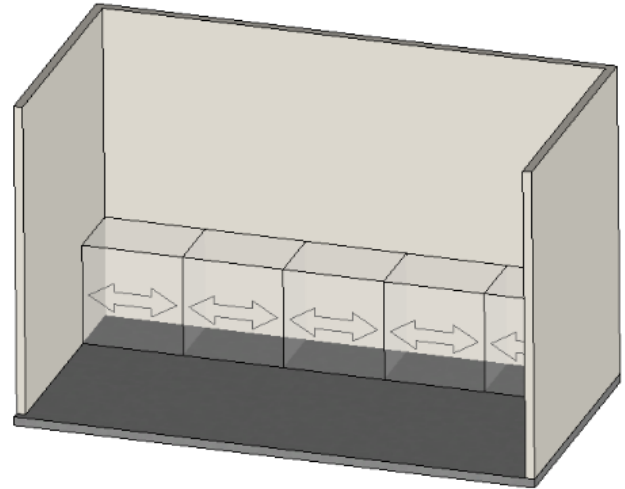
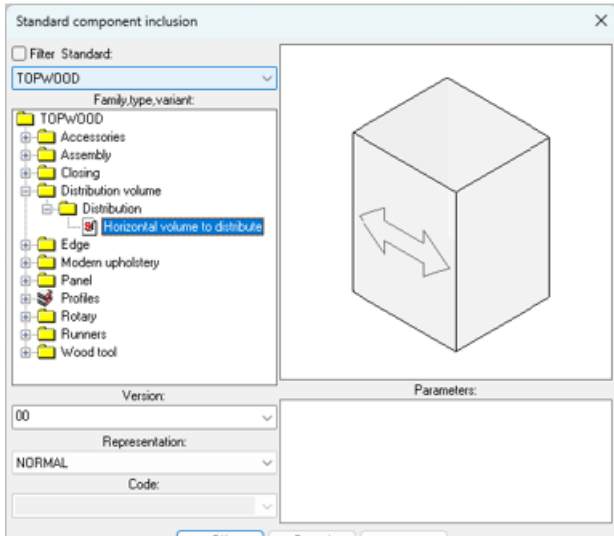
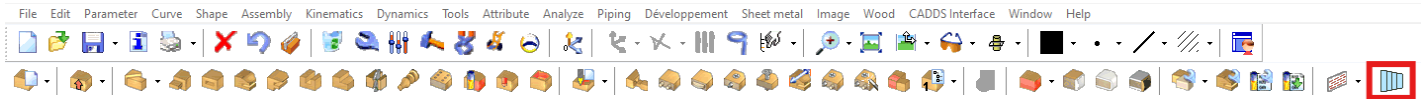
When a driver block component is included in the result of a distribution, the **Serial copy** button is now automatically proposed.

Housing mode= Hide block= **SERIAL COPY** block:

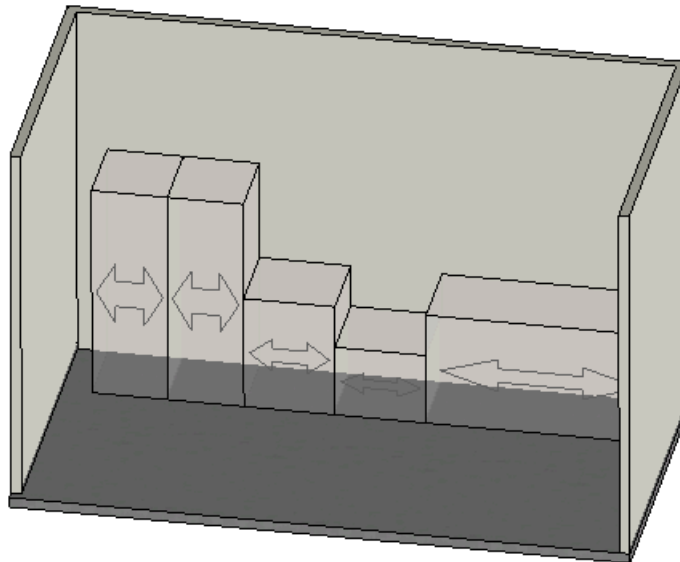


Horizontal distribution

This new command allows you to include a new standard component and immediately launch horizontal distribution.



Result of horizontal distribution.

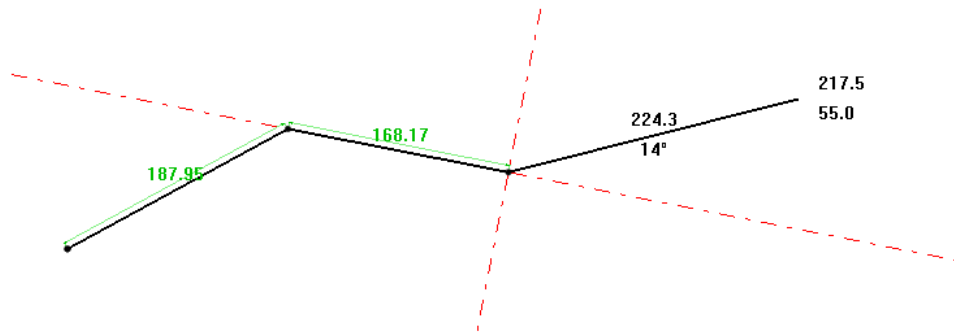


Result of horizontal distribution after modifying the parameters of the distributed component.

New auto-dimensioning mode in sketches

Sketches can now be automatically dimensioned. A new **Auto dimension = YES / NO** option is available when creating a contour.

STOP Link= INTERSECTION Segment length= Auto dimension YES >> Next curve, near to the fork:



QR code in bills of materials

A new feature allows you to convert a BOM property into a QR code for a designated column.

To activate it, check the **QR code** option when editing the relevant column.

Modification

Title: MACHINING FILE NAME

Modules

Defined modules > TopSolid'Wood - Advanced properties

Function

Defined functions > MACHINING FILE NAME

Other parameters >

Parameters

Type > CHARACTER STRING

Alignment > LEFT

Title alignment > LEFT

Width : 15.00mm

Format :

Bar code > NONE

☒ QR code

☒ Visible

Column number : 4

Use of column

☒ Sort by this column

☐ Make sum on this column

☐ Make average on this column

OK Cancel

7144	Kit excentriques et tourillons	-
6988	Kit excentriques et tourillons	-
6070	Top	
6042	Base	
5982	Right Side	
5952	Left Side	
827	Back	
583	Plinth	
286	Plinth	

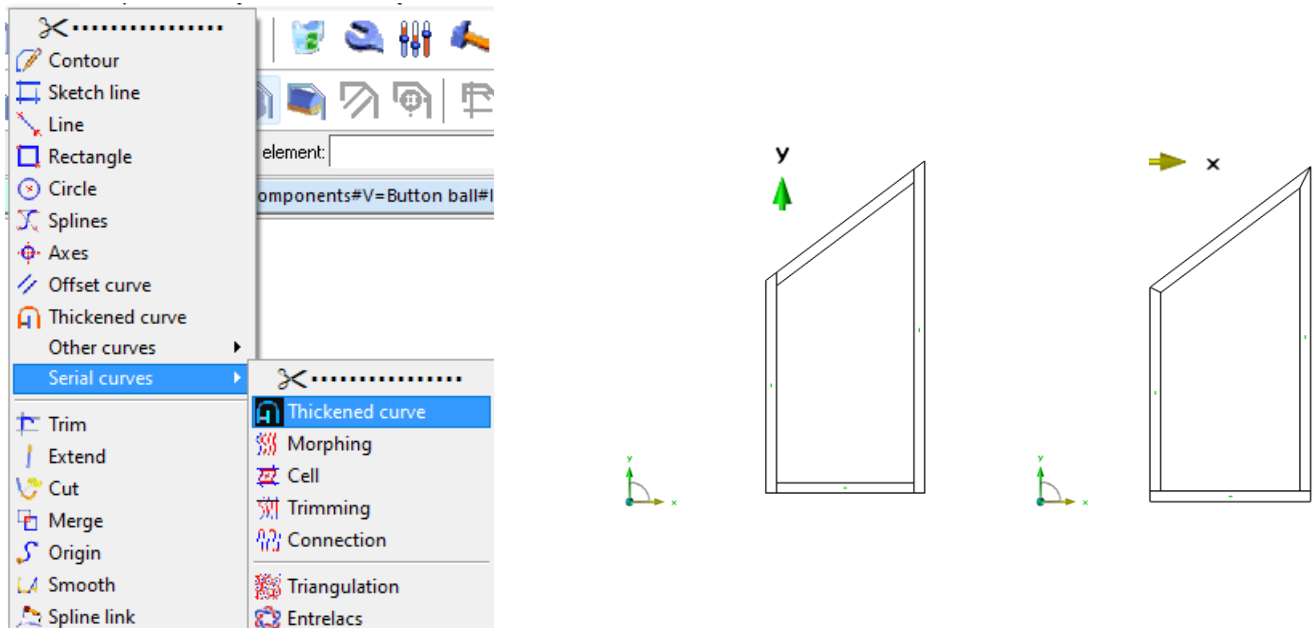


Note: To maintain performance, it is recommended to filter the bill of materials to avoid generating QR codes for parts that do not require them.

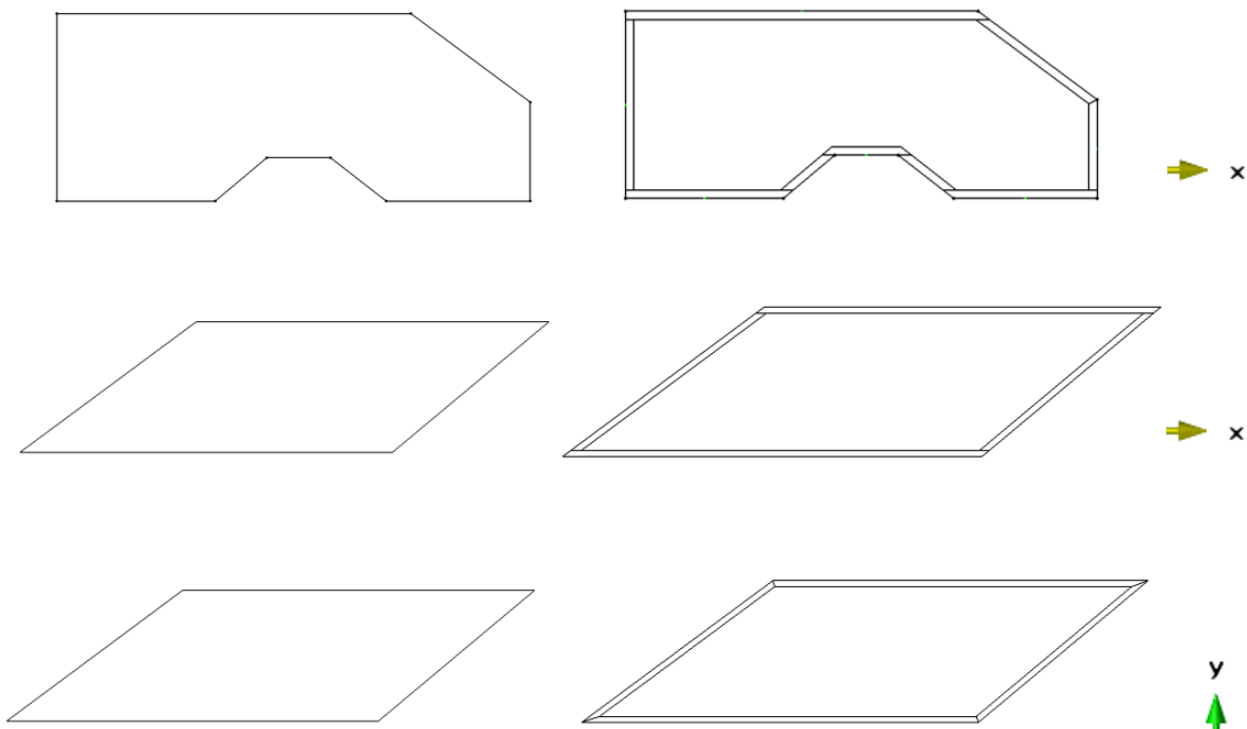
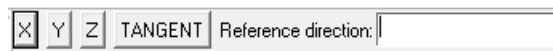
New cutting mode for thickened serial curves

A new cutting mode is available when using the **Thickened curve** command in serial mode.

Perpendicular cuts are generated between two perpendicular curves, along the given direction. If this is not the case, miter cuts are generated.



After selecting the **Direction** mode, you can define the reference direction.

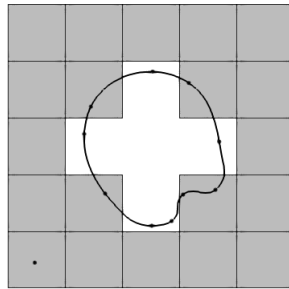


Closed curve exclusion

When excluding instances from a propagation by a closed curve, two exclusion modes, accurate and strict, are now available.

Accurate exclusion mode = NO

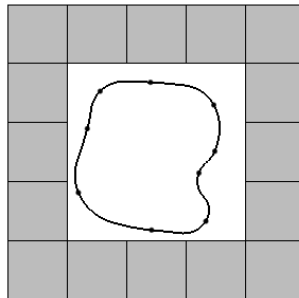
OK Accurate exclusion mode= NO ↕



This mode was already available in previous versions.

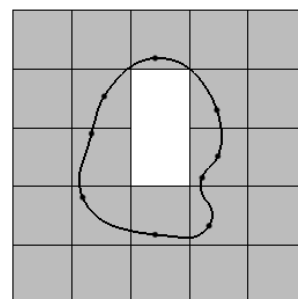
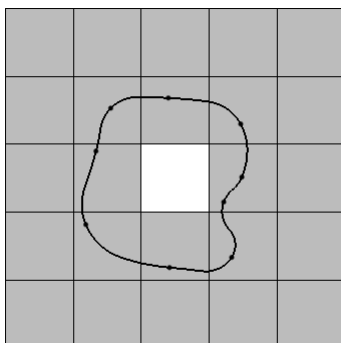
Accurate exclusion mode = YES + Strict exclusion mode = NO

OK Accurate exclusion mode= YES ↕ Strict exclusion mode= NO ↕



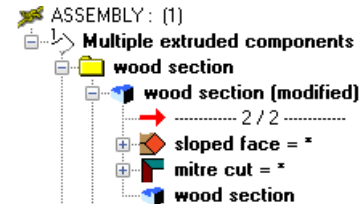
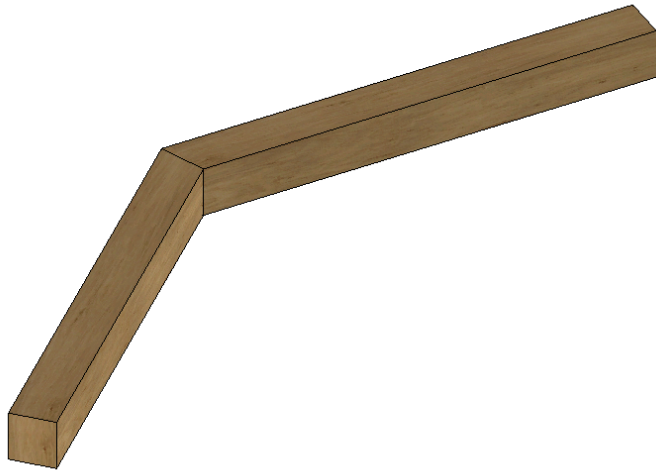
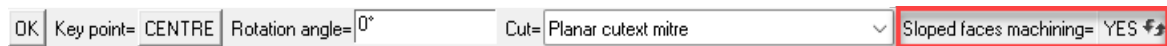
Accurate exclusion mode = YES + Strict exclusion mode = YES

OK Accurate exclusion mode= YES ↕ Strict exclusion mode= YES ↕



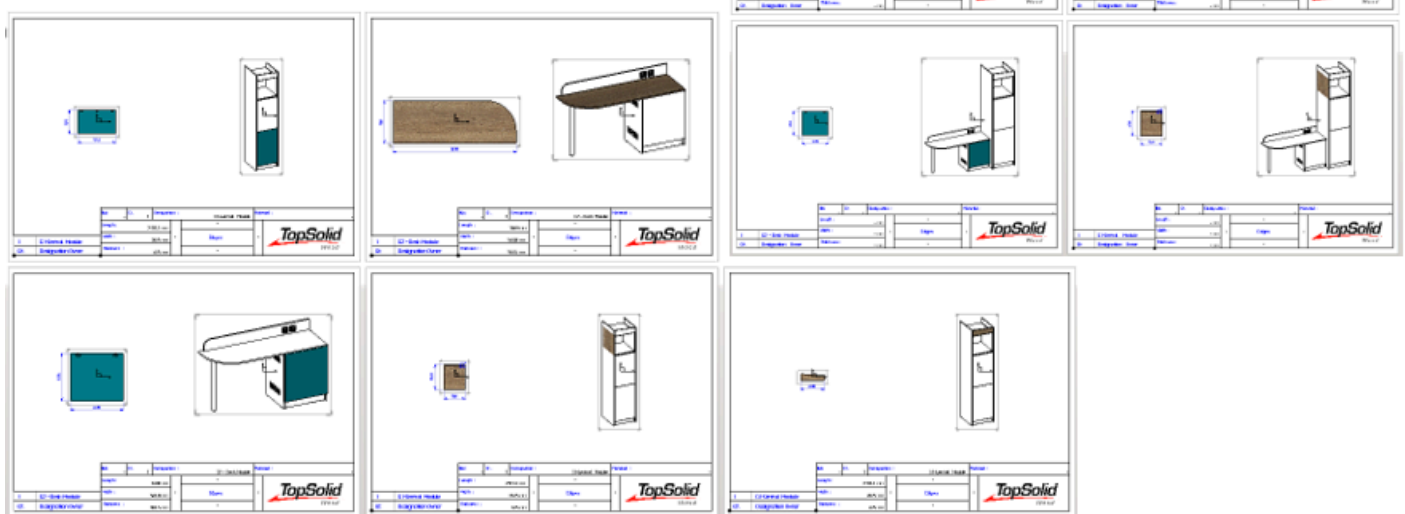
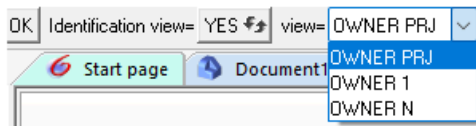
Creation of user machining operations for extruded component cutting

When cutting extruded components, the new **Sloped faces machining** option automatically generates the **Sloped face** user machining operation.



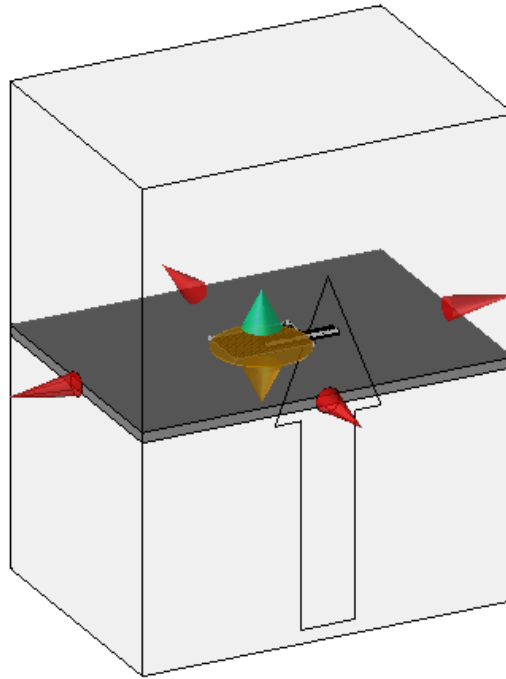
Highlighting a part in multi-drawings

When creating a multi-drawing template, new view creation options are available. For example, they can be used to highlight a specific part within its subassembly, making it easier to visualize.



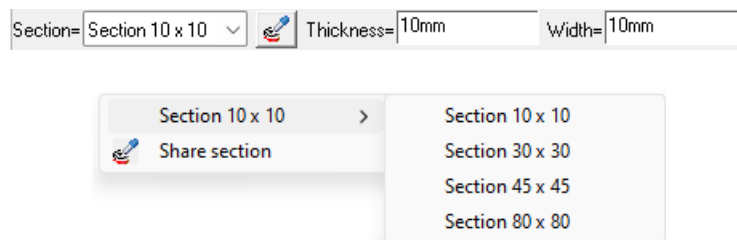
Editing arrows added after creating a constrained block or section

It is now possible to use edit arrows after creating a constrained section or block in manual mode.



Saving the last used constrained section

Previously, the last used constrained section was not saved. Now, it is automatically selected, eliminating the need to search for it in the drop-down list.



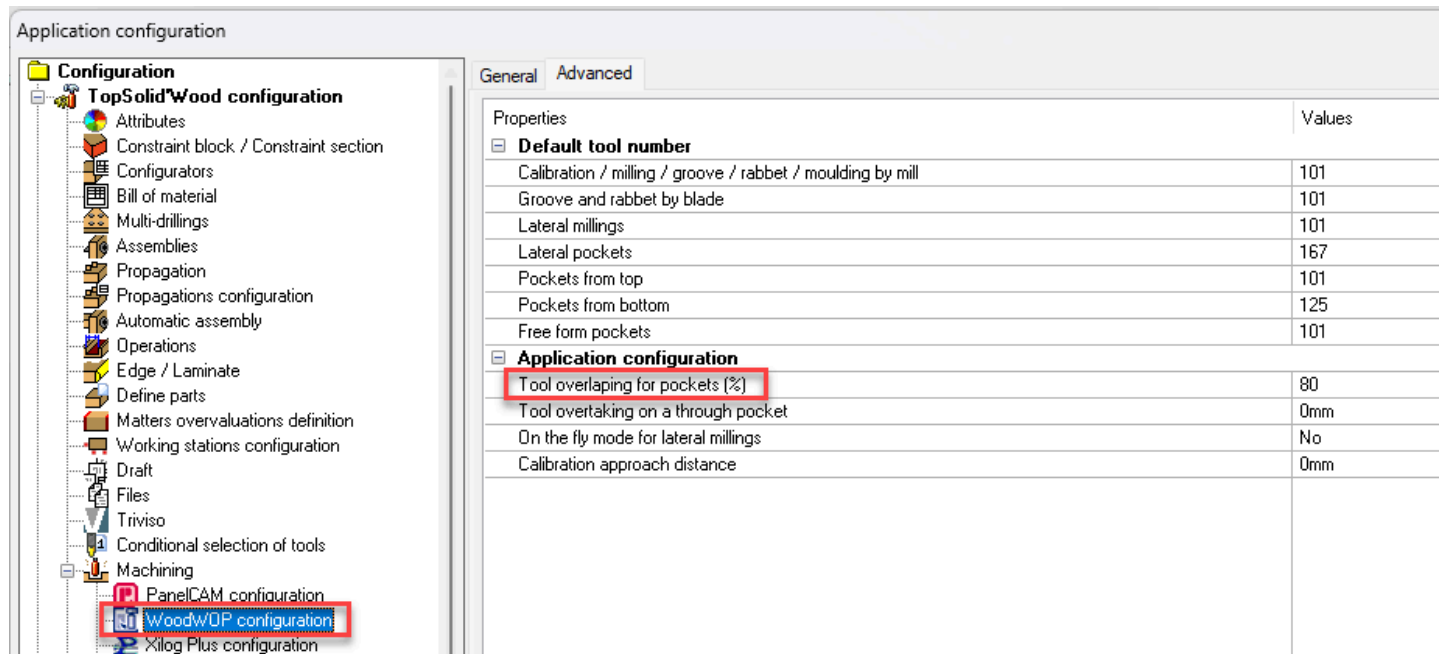
WoodWOP interface

Tool overlap for pockets

The tool overlap property now applies to both through pockets and blind pockets.

The associated configuration word is:

ZX_ZWOO_CAM_MPR_THROUGH_POC_ADVANCE (default value: 80%)



Miscellaneous improvements

Nesting enhancements for panel entities

Recent updates to panel entity nesting include:

- Edge-free panel nesting management.
- **Parent** properties management for panel entities.
- The ability to transfer material to a panel entity.

Behavior of edges disabled by operations

A new enhancement optimizes management of edges disabled by operations.

An additional configuration word has been introduced: **ZWOO_V623_EDGE_OVERDIM_COMPUTATION**

When enabled, this configuration word preserves the overdimension values for edges disabled by processes.

What's New in TopSolid'Sheetmetal 6.26

Explore the latest features and enhancements of **TopSolid'Sheetmetal**, the integrated CAD/CAM solution for sheet metalworking!

Import

Import of current v7 unfolded document across multiple machines

You can now import the current v7 unfolded document via multi-document import, enabling access to automatic machining and oriented part creation features tailored to the entire machine fleet.



During this process, a destination folder for the PCH file must be specified.

However, PCH files are automatically named according to the v7 part name and, if applicable, organized into subfolders corresponding to the machine name.

Import from a v7 part family

When importing from a v7 part family, if the command is executed from an already-machined PCH punching document, the system attempts to retain existing machining operations for new documents created from the family instances.

However, if the thickness and/or material are controlled within the family document, some tools used in the template PCH document may become incompatible with the new material or thickness.

Now, when automatic machining is activated for each part, the system checks tool compatibility if the material or thickness of the instance has changed.

If a tool is incompatible, the system searches for a replacement tool with the same dimensional characteristics. If a compatible tool is found, it replaces the previous one on the same turret station and updates all associated machining operations.

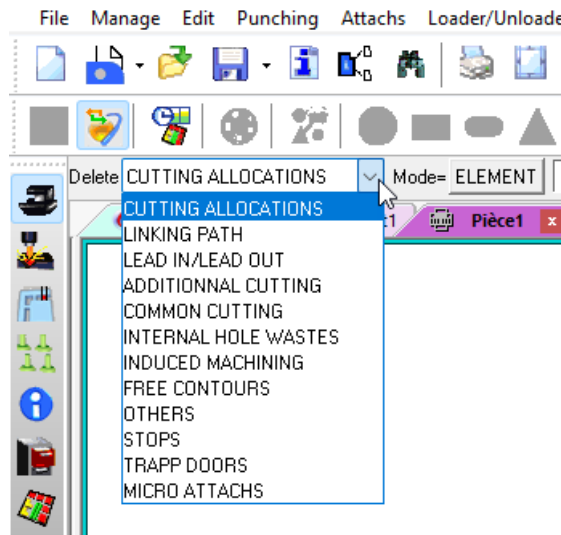
If no compatible tool is available, the tool is removed from the turret, and all its assignments, including those related to attachment tools, are deleted.

Finally, during the subsequent automatic machining phase, the system selects the most suitable tools to complete the remaining machining operations.

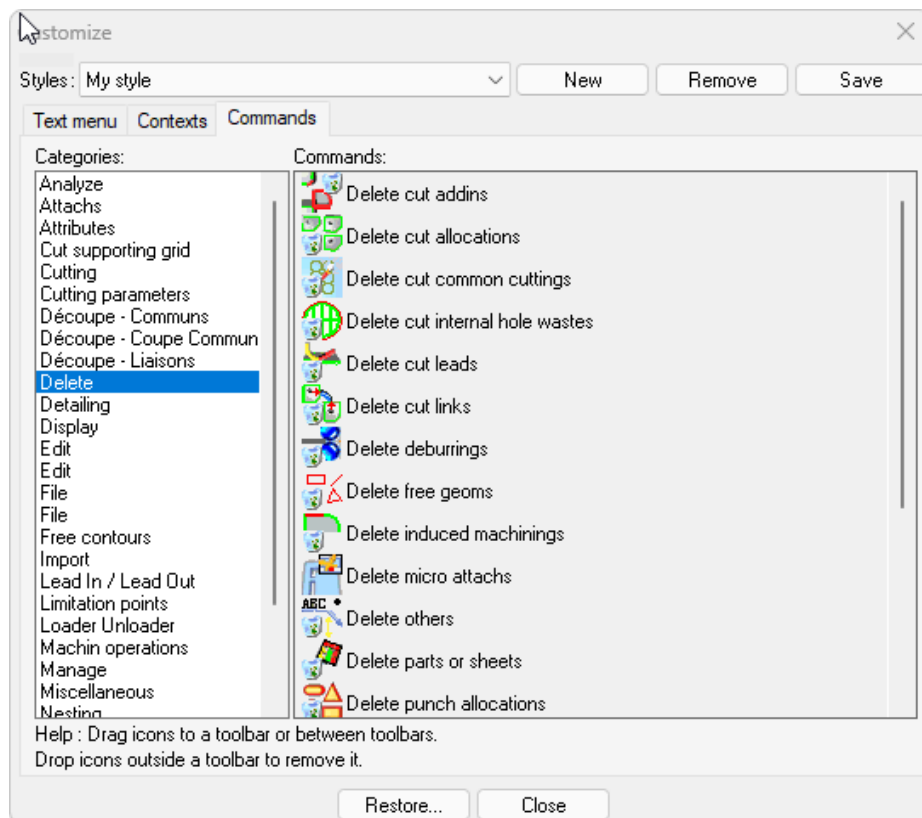
Cutting and Punching

Improved delete command

Previously, the **Delete** command displayed the different types of entities to delete in a drop-down list.

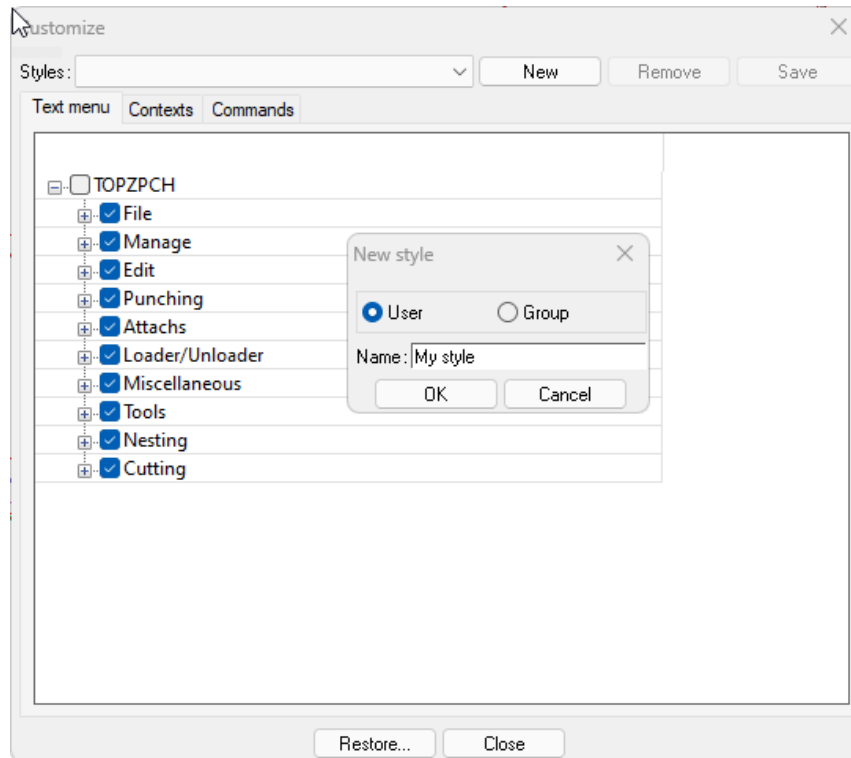


Each entity type now has its own dedicated delete command. This enhancement lets you configure a customized menu containing one or more specific delete commands, all grouped together under the **Delete** category.

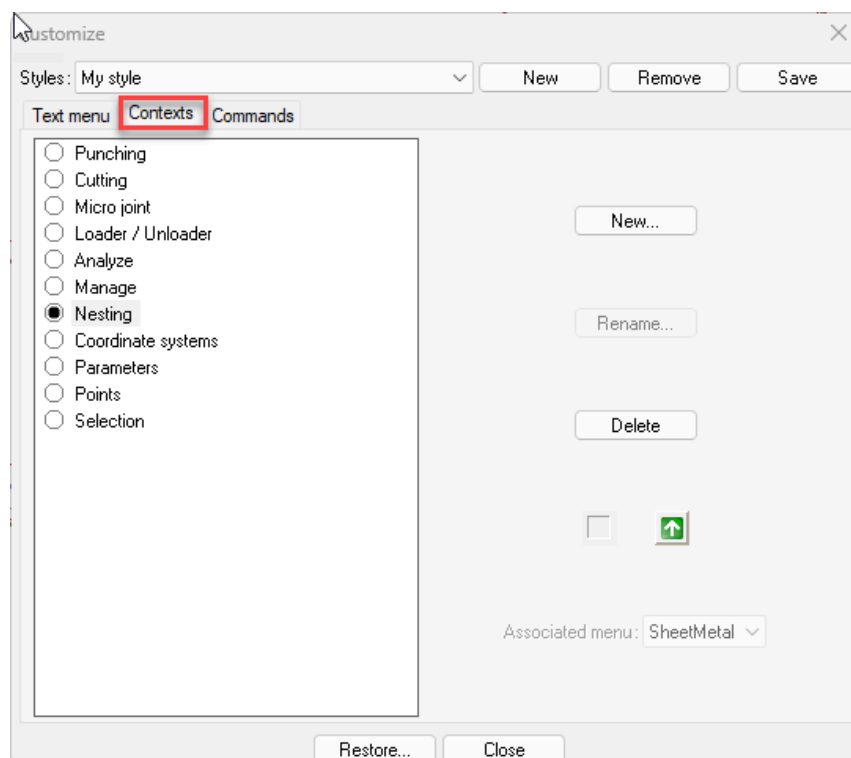


How it works

To access customization, open the **Tools > Customize** menu and create a new style.

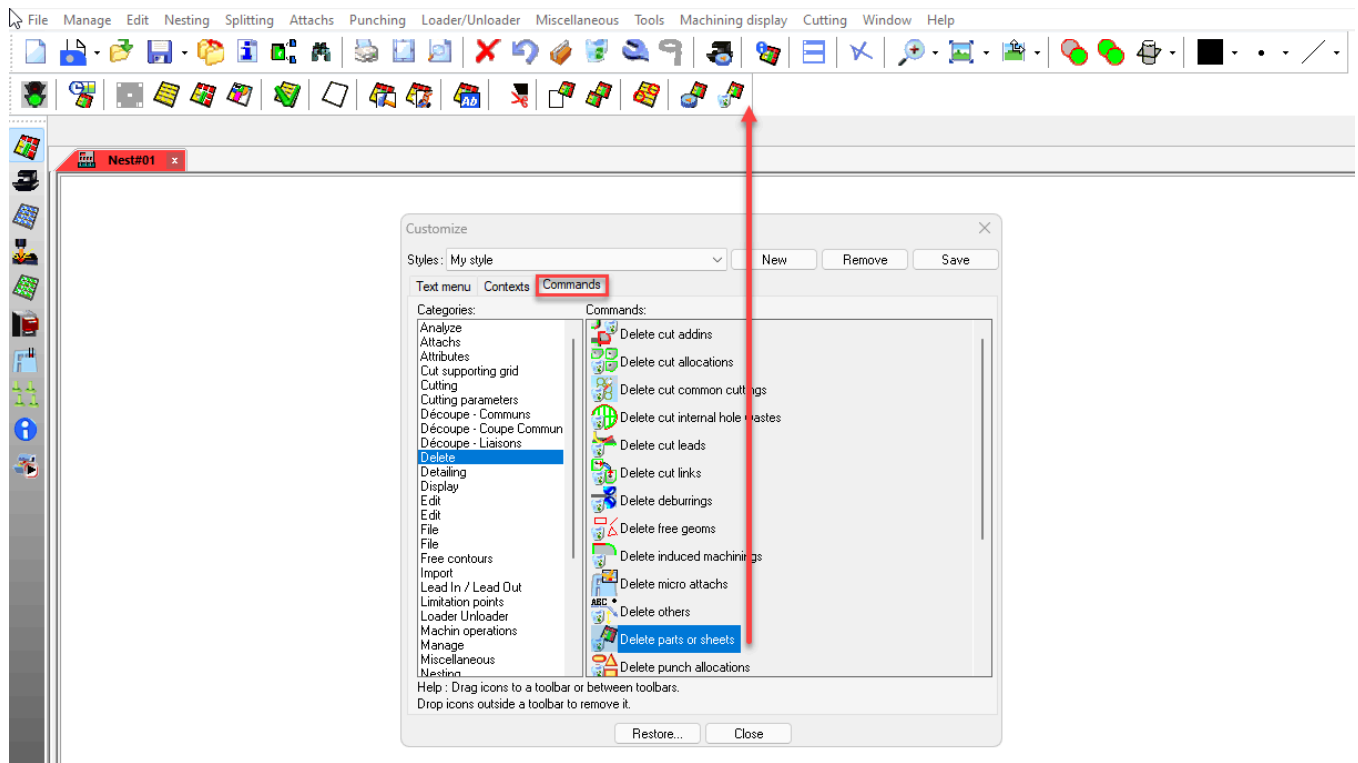


In the **Contexts** tab, select the icon bar where you wish to add the commands.

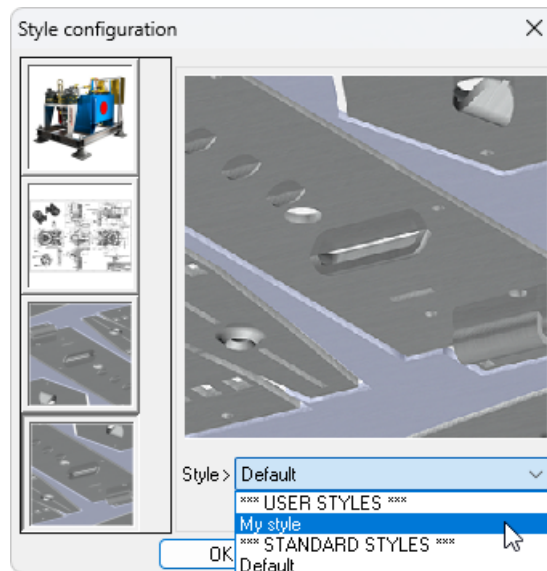


In the example above, the commands will be added to the nesting context.

In the **Commands** tab, select the **Delete** category. Drag the desired command to the desired icon bar or context bar.



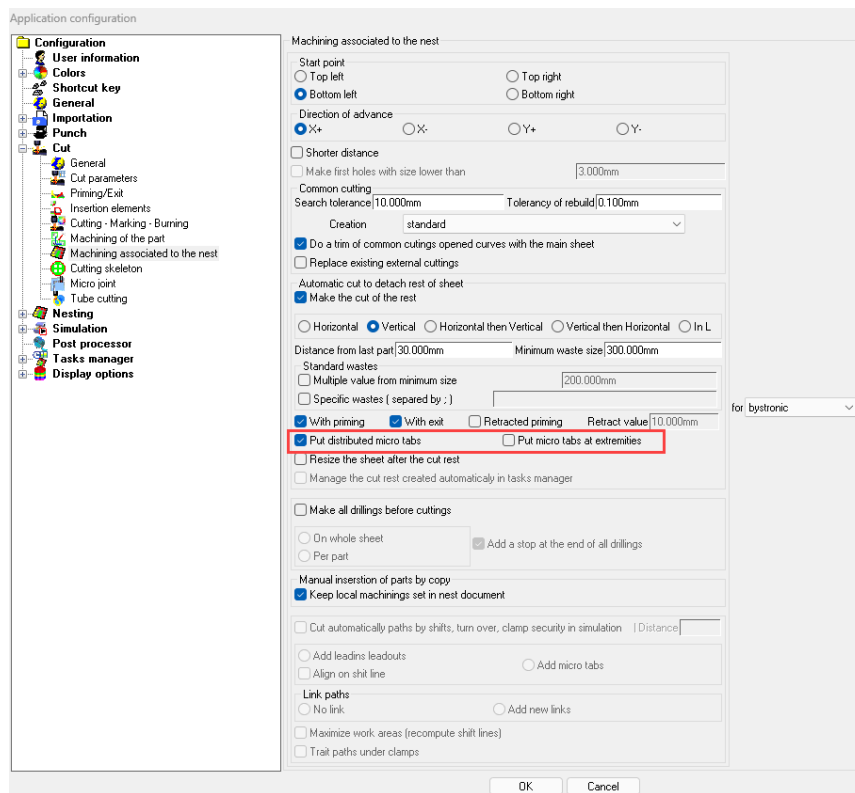
To activate the custom style, go to **Tools > Configure styles** and select the newly created style.



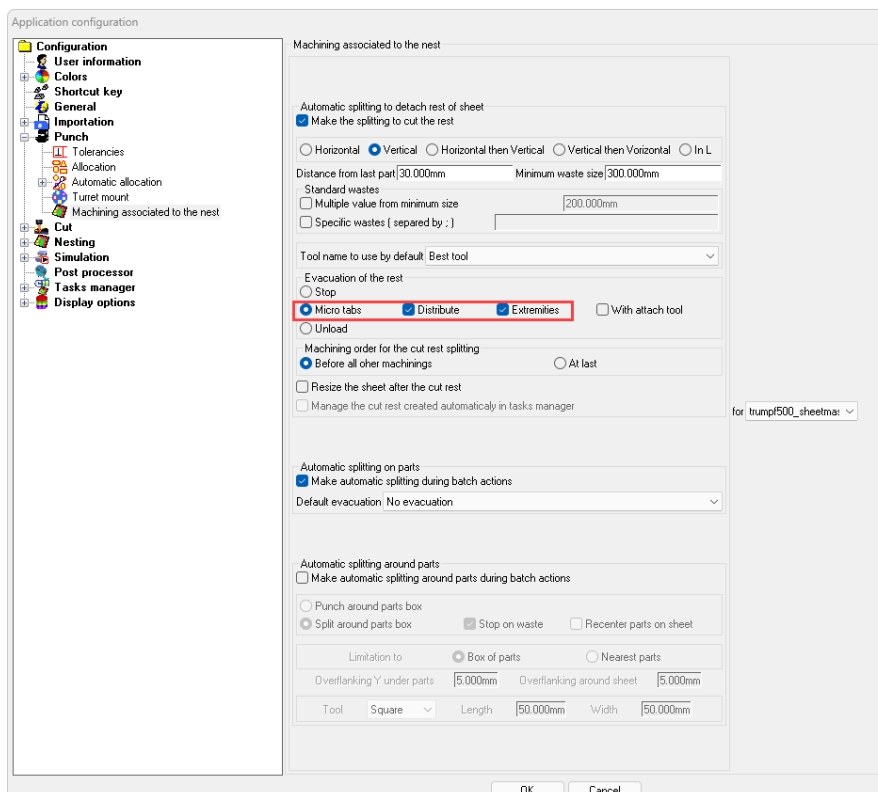
Micro-tabs at the ends of waste cuts

In previous versions, micro-tabs for waste cuts were only available in distributed mode.

Micro-tabs can now be added to the ends of waste cuts. This feature is available for both punching and cutting.

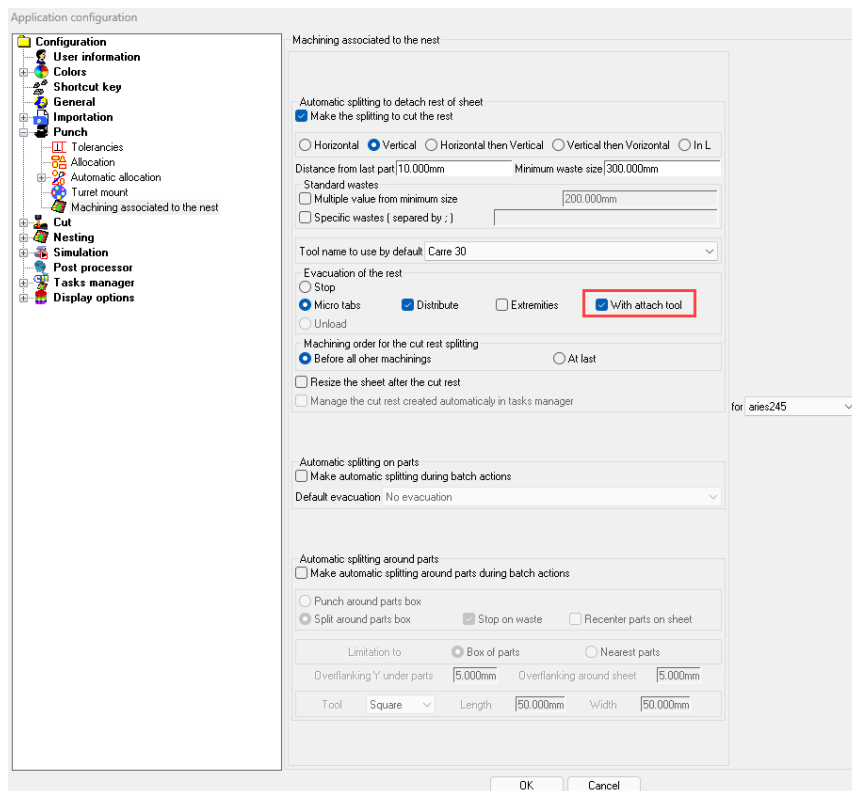


Punching access.



Cutting access.

In punching, default attachment tools can also be placed.



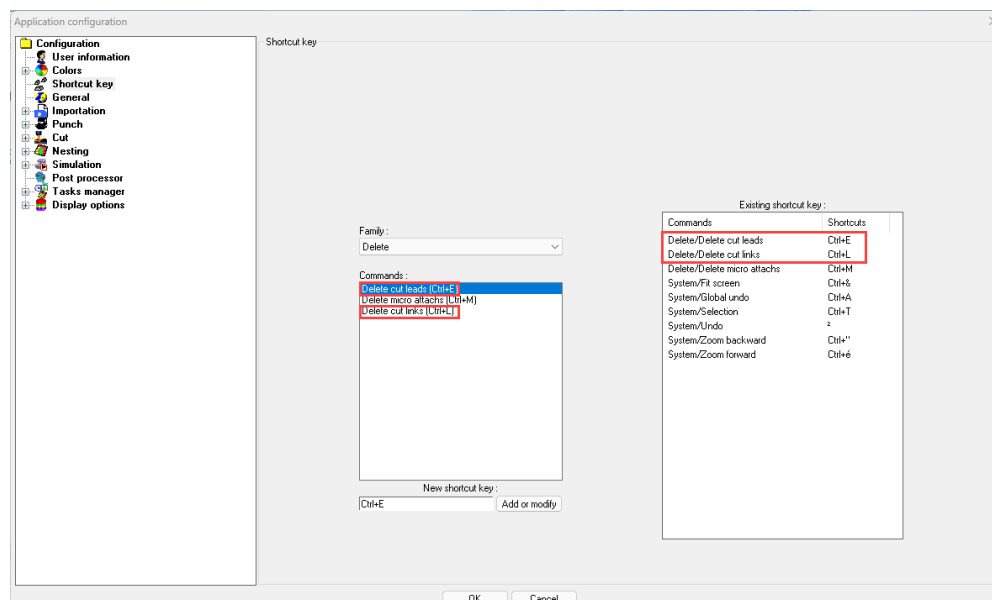
New keyboard shortcuts for delete commands

You can now assign keyboard shortcuts to delete commands by entity type, grouped under the **Delete** category, when customizing icon menus. This feature provides quick, optimized access to these commands.



Example of menu customization with the addition of the **Delete cut links** and **Delete cut leads** commands.

Keyboard shortcuts for these commands can be configured via **Tools > Options > Shortcut key**.



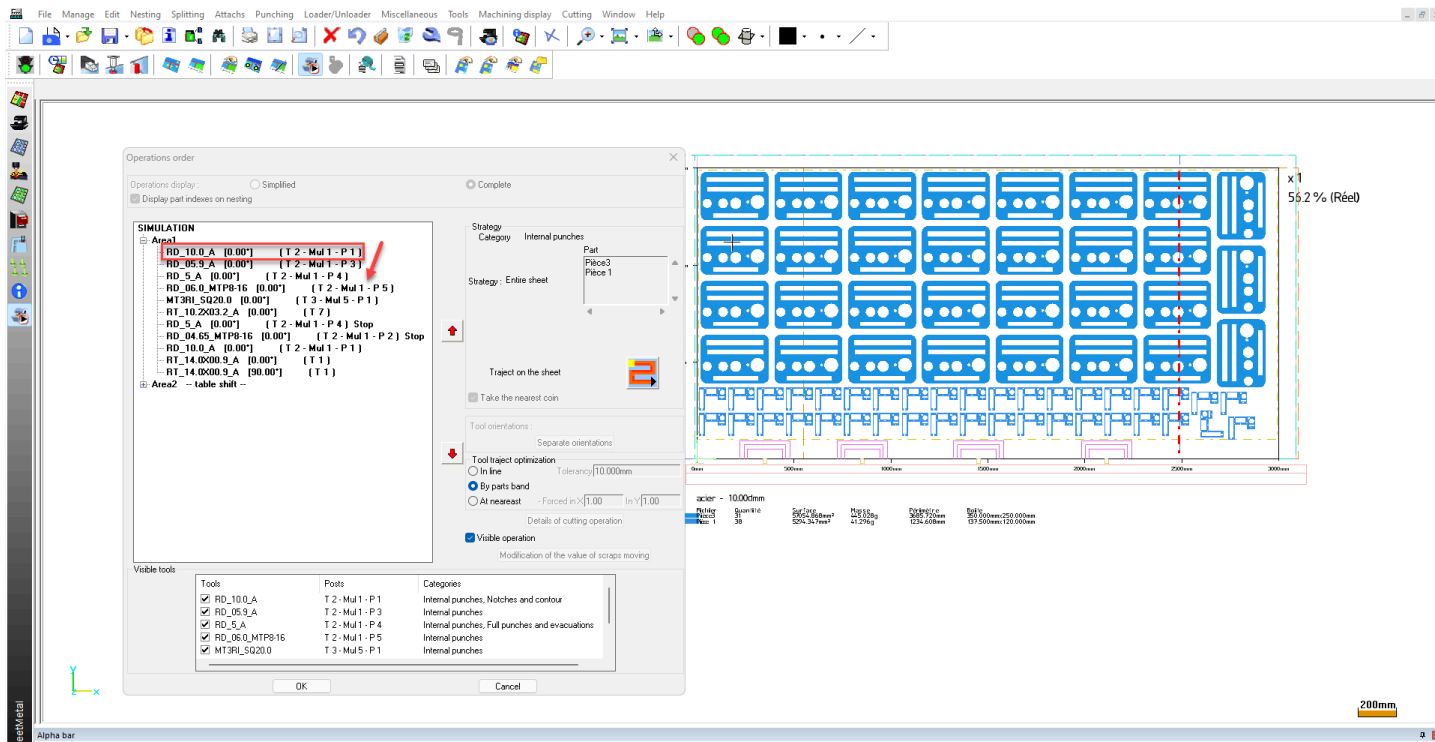
In the example above, the added commands have a dedicated keyboard shortcut and are grouped together in the **Delete** family.

Punching

Improved tool reordering in operation sequence

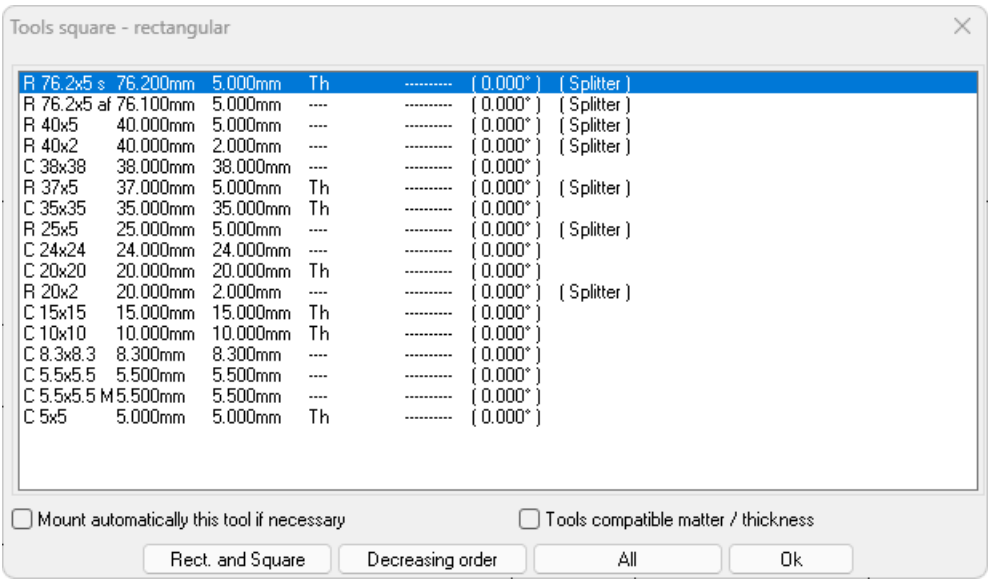
The management of tools in the operation order dialog box has been simplified with the ability to reorder tools using drag-and-drop.

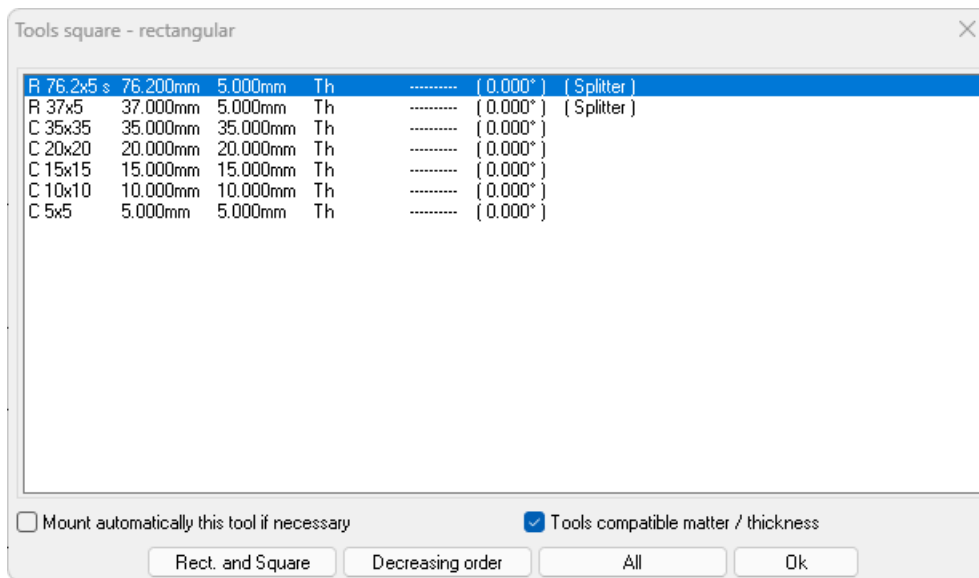
Only one line can be moved at a time.



Display of available tools by material and thickness

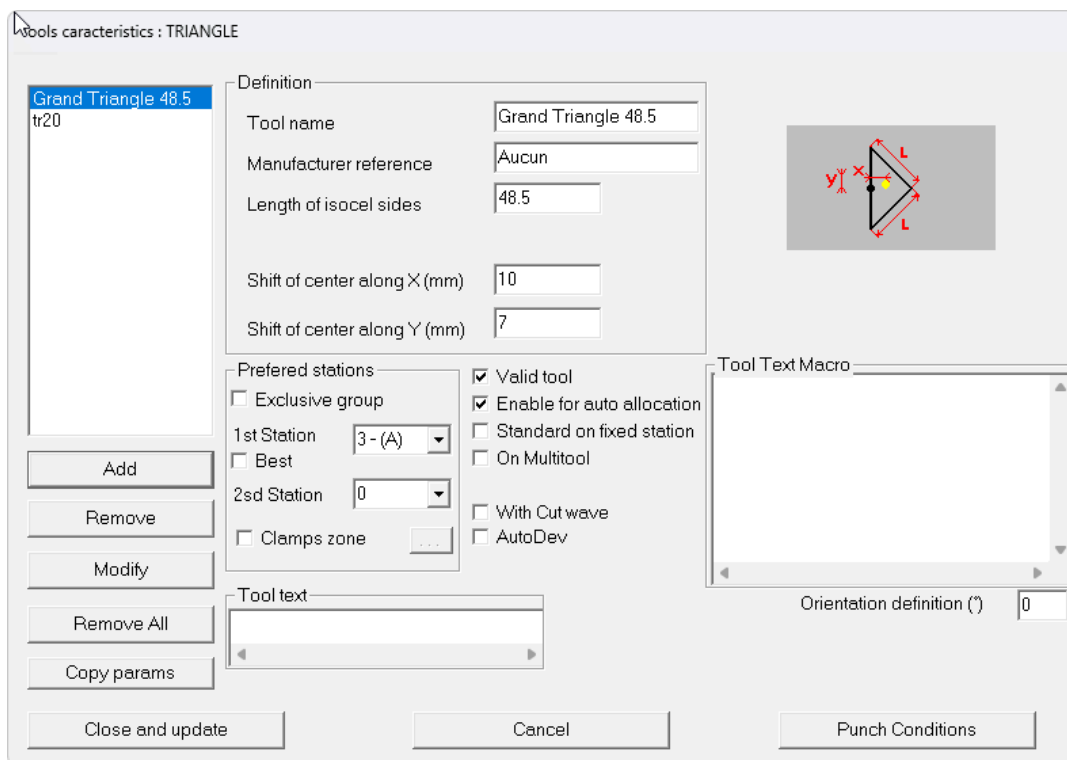
A new feature now limits the display to tools compatible with the current document's material and thickness.



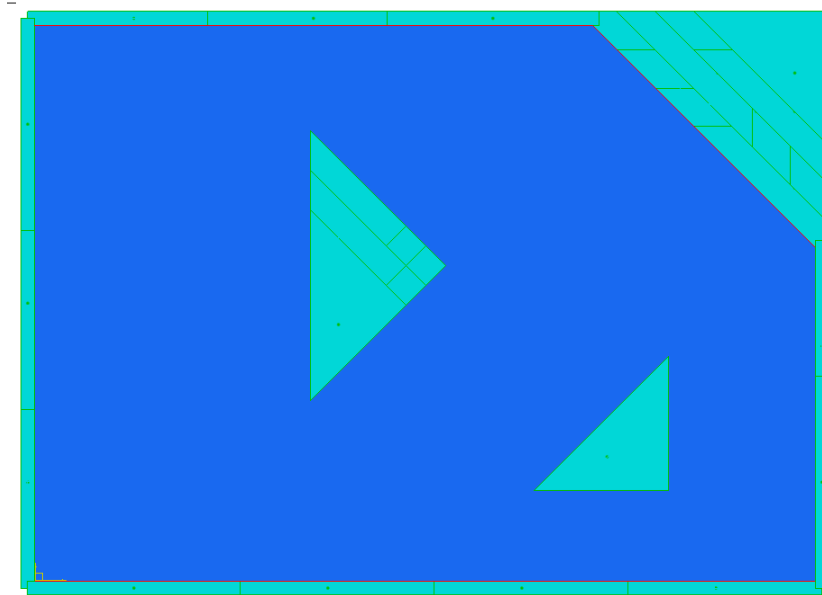


Punch point management for triangle tools

The management of right-angled isosceles triangle tools has been optimized with a new feature that lets you define the exact punch point. You can now position the center point by specifying offsets on the X and Y axes from the point on the vertical line.



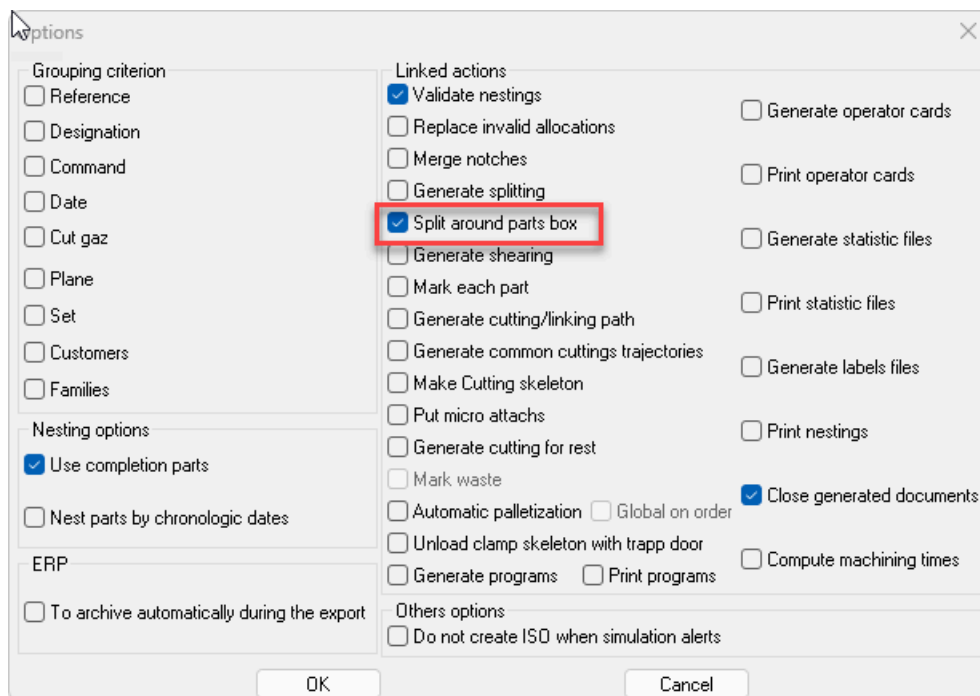
This enhancement simplifies the use of triangle tools and eliminates the need to create special tools.



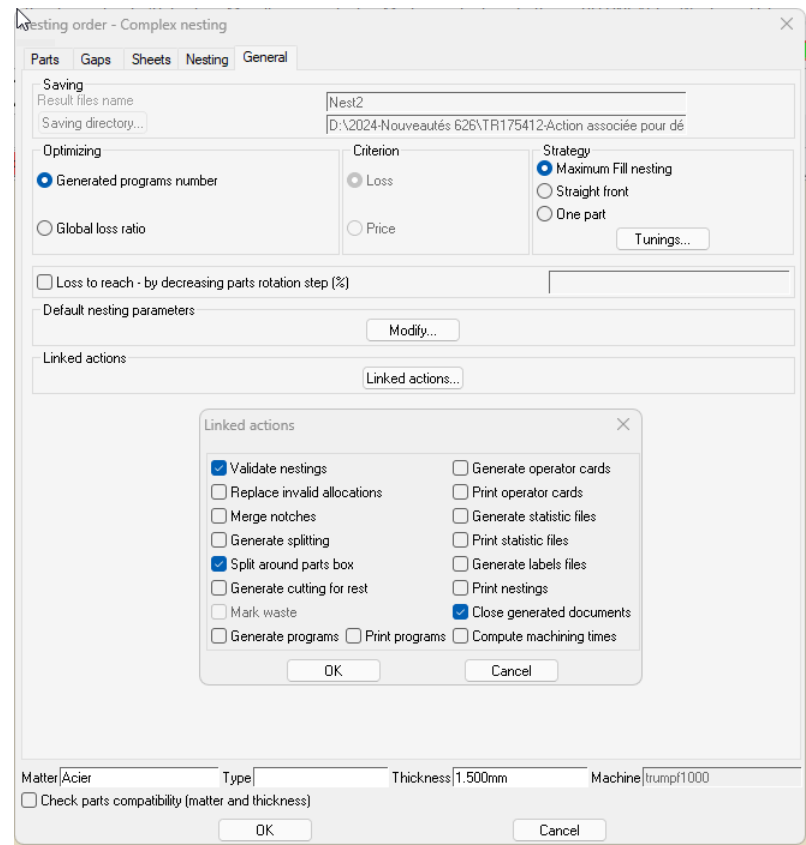
Example of using the **Triangular profile** and **Notch with triangular tool** commands.

Associated action for splitting around parts

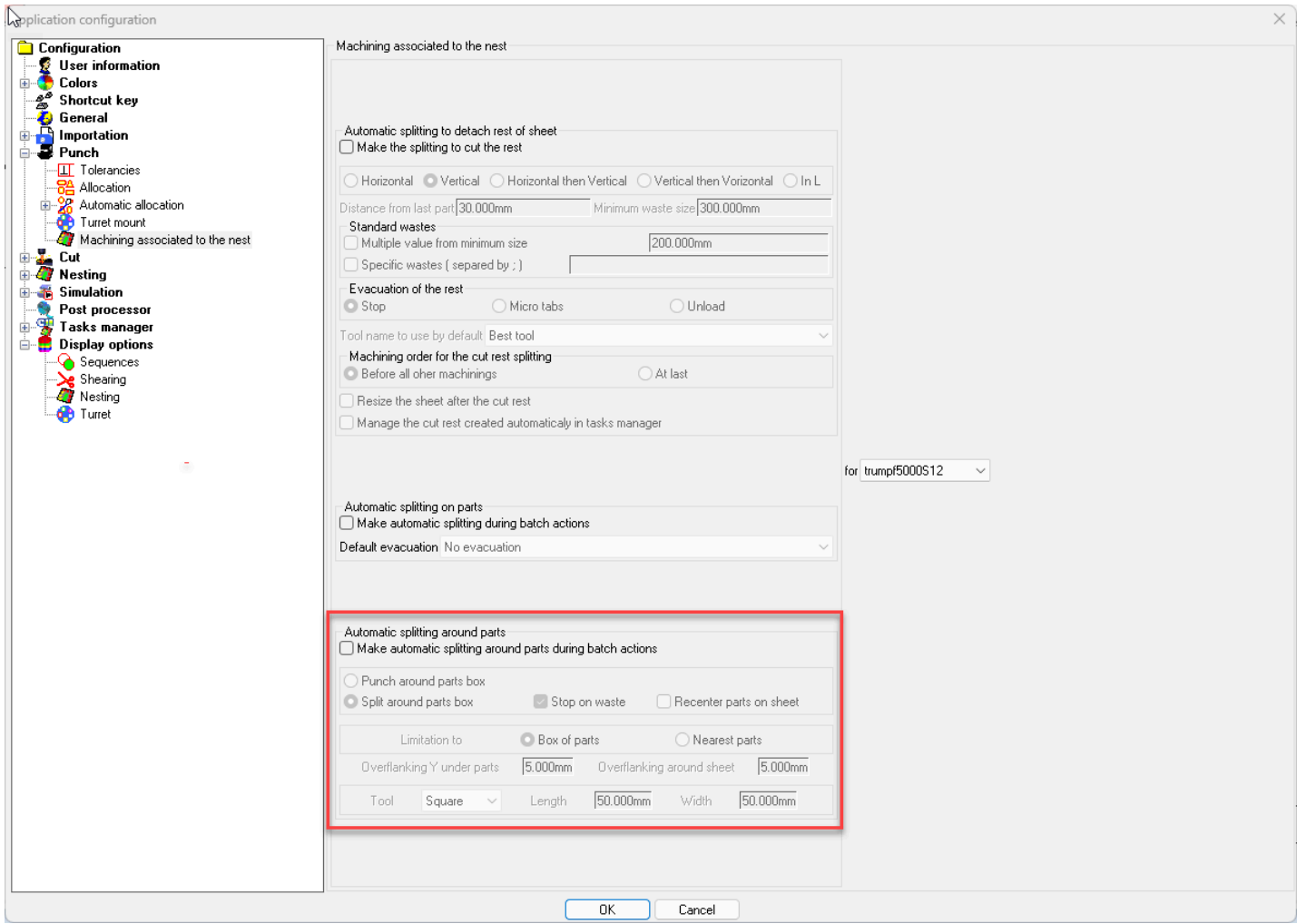
A new associated action, accessible in the Task Manager options, automates the **Border line** command with the **Split around parts box** or **Punch around parts box** option, creating a splitting (or punching) operation around the parts on the sheet.



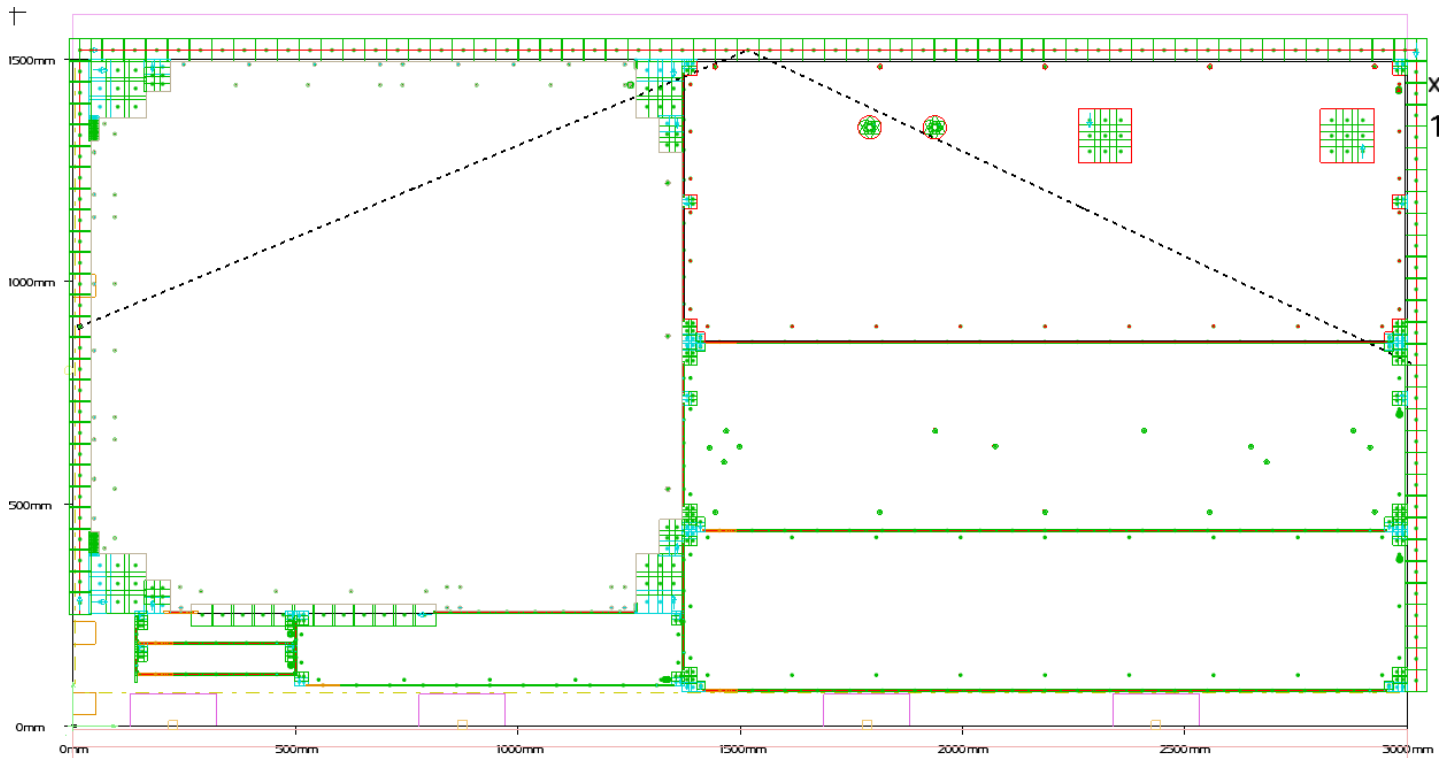
This feature is also available in the nesting order.



Machine settings are configurable via **Tools > Options**.



The selected tool will be the one whose dimensions are less than or equal to those specified. If the tool is not present on the turret, it will be mounted automatically. Scheduling is systematically added when splitting operations are created.

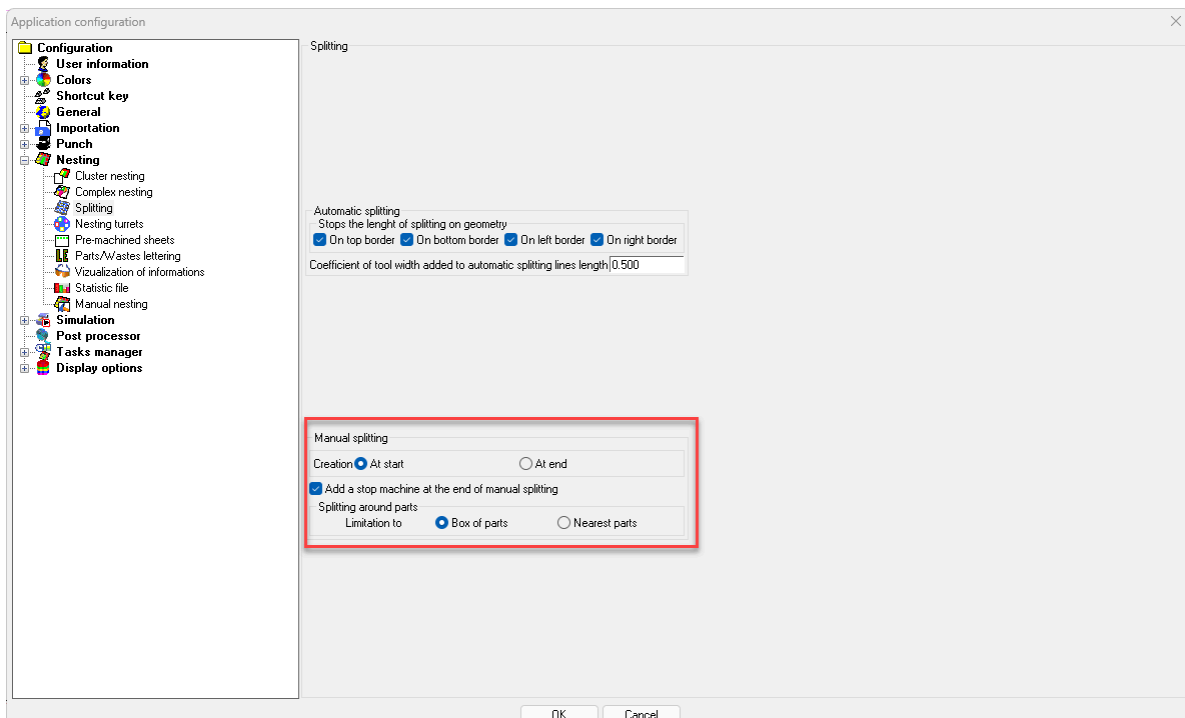


Machine stops at the end of splitting

It is now possible to specify whether a machine stop should be added at the end of a sheet splitting operation. This option can be adjusted directly from the question bar.

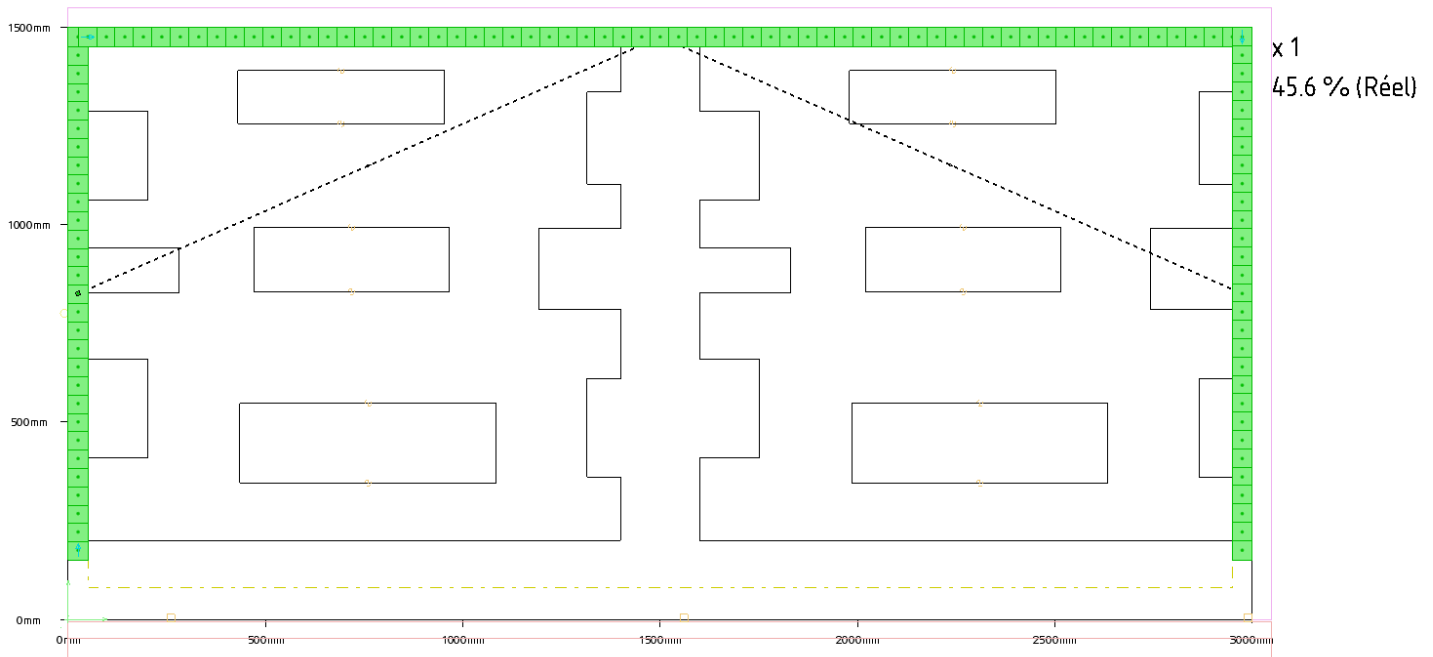


Default behavior can be configured via **Tools > Options > Splitting**, in the section dedicated to manual splitting.



Improved stop position for splitting around parts

The **Border line** command, with the **Split around parts** box option, has been optimized. Enhancements include automatic deletion of unnecessary lines and the ability to add one or more stops as needed.

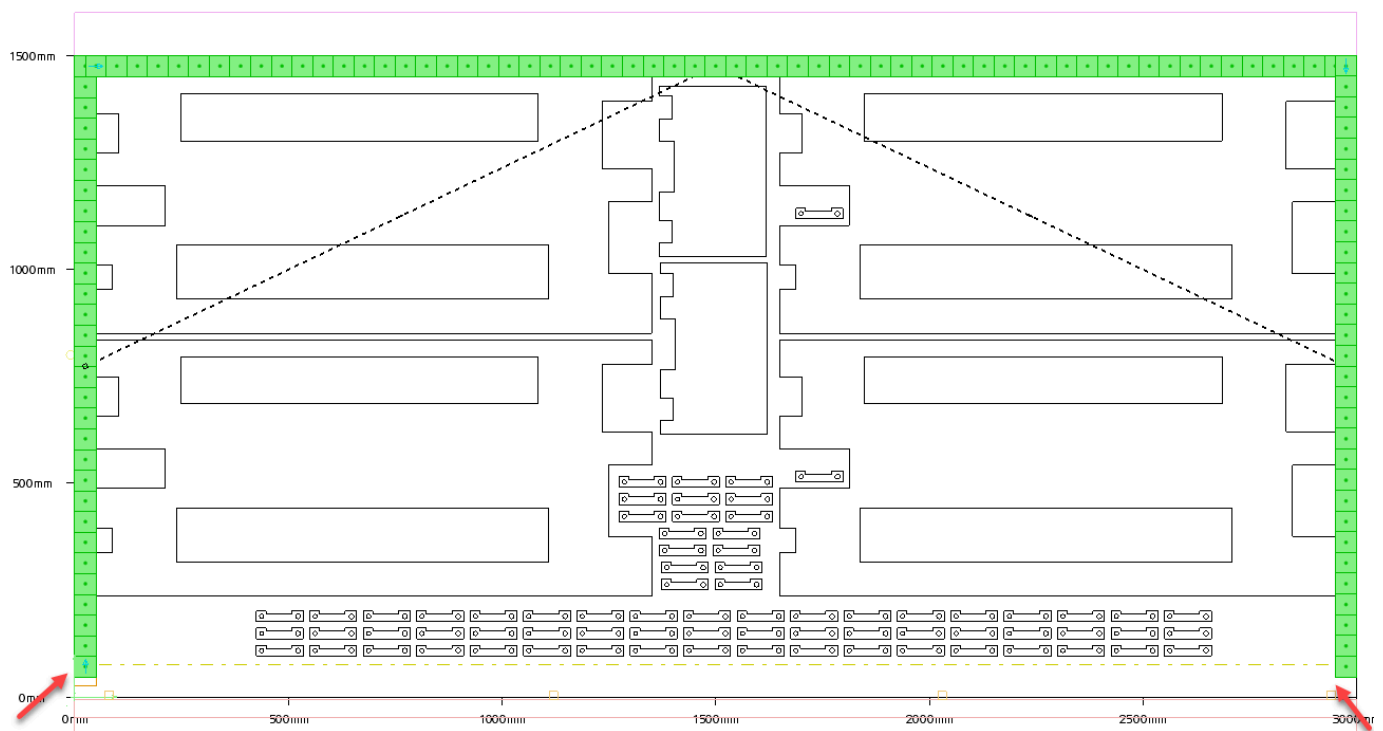


In the optimized example above, the stop on the right is no longer displayed.

Optimized splitting around parts command

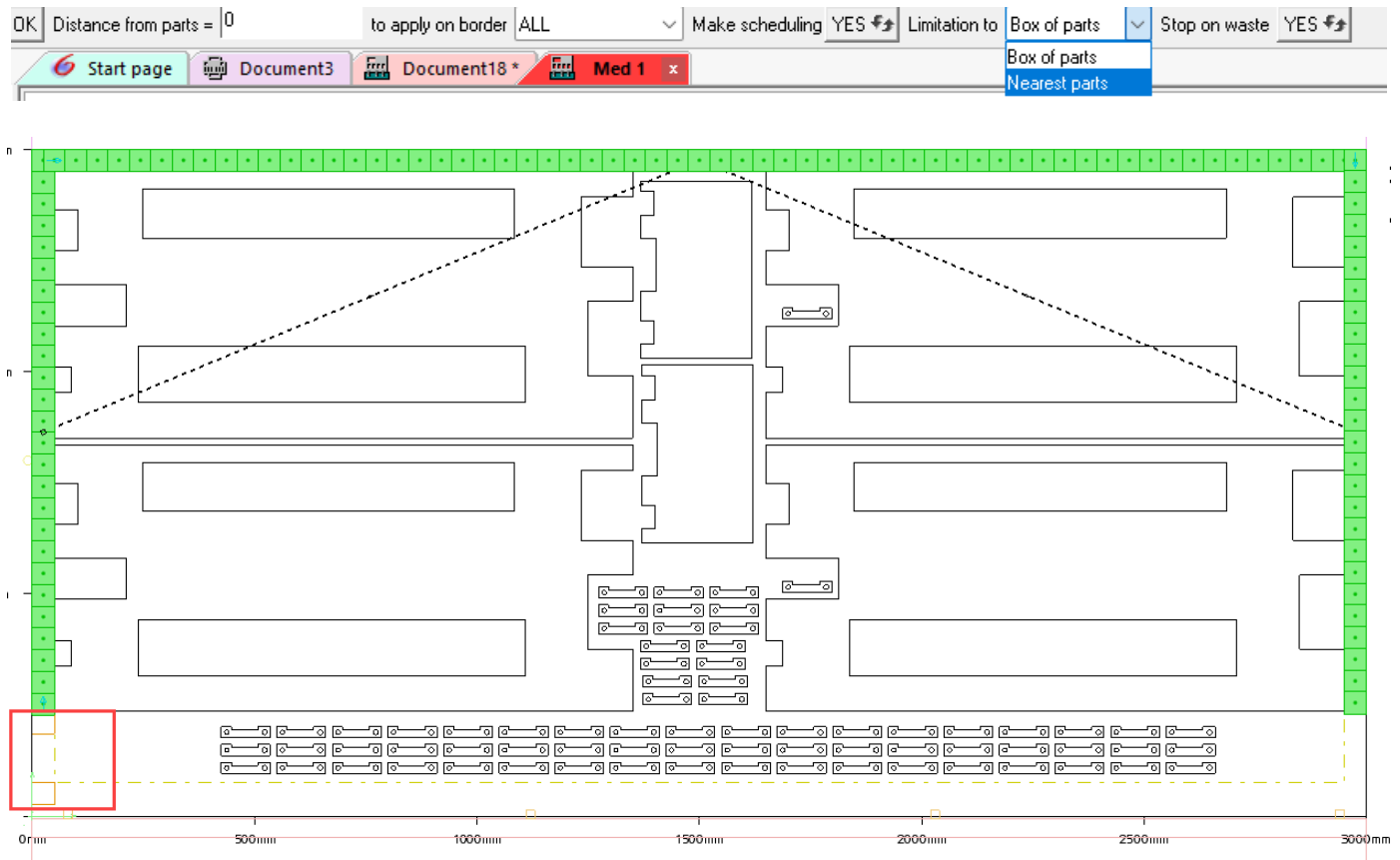
A new feature has been added to the **Border line** command, when the **Split around parts** box option is selected.

Previously, splitting was only calculated relative to the bounding box of the parts, with automatic overflanking based on the tool dimensions.



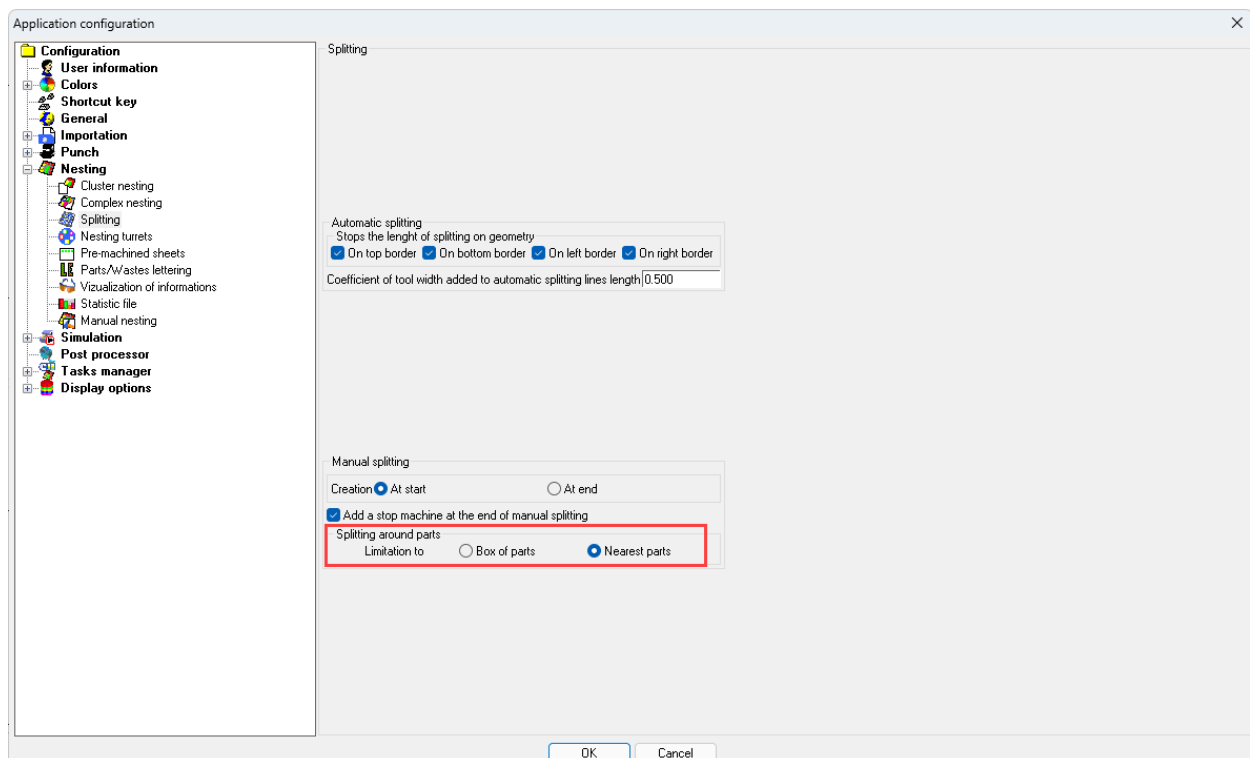
On some machines, small clamps are used to evacuate the skeleton in clamps, leaving an unpunched area. Previously, it was necessary to adjust the splitting line to allow this operation.

You can now choose whether splitting operations are calculated based on the parts' bounding box or in relation to the parts close to these splitting operations (for left and right edges).



In addition, the length of the left splitting has been optimized. This improvement also applies to the **Punch around parts box** option.

Default behavior can be configured via **Tools > Options > Splitting**, in the section dedicated to manual splitting.



Optimized sheet splitting based on waste size

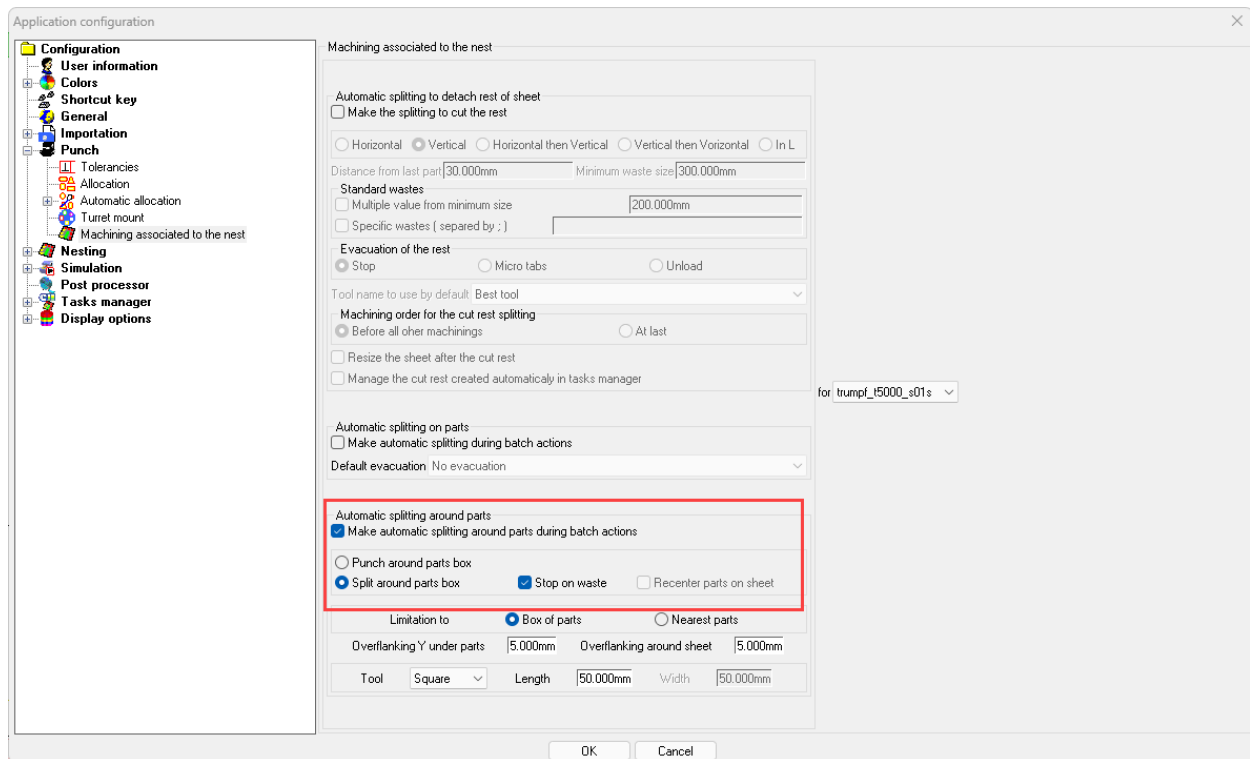
When using the **Border line** command with the **Split around parts box** option, a machine stop could be generated if the left margin was too large in relation to the width of the tool used for splitting.

Stops can now be avoided by splitting only the top horizontal and right vertical sections.

An additional option allows parts to be recentered on the sheet when the left and right margins can be split in a single pass, optimizing the process.

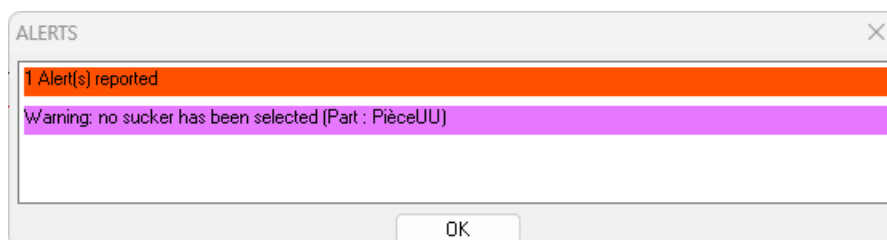
OK Distance from parts = 10 to apply on border ALL Make scheduling YES Limitation to Box of parts Stop on waste NO Recenter parts on sheet YES

For automatic splitting around parts during associated nesting actions, you can configure the **Stop on waste** and **Recenter parts on sheet** options in **Tools > Options > Machining associated to the nest**.



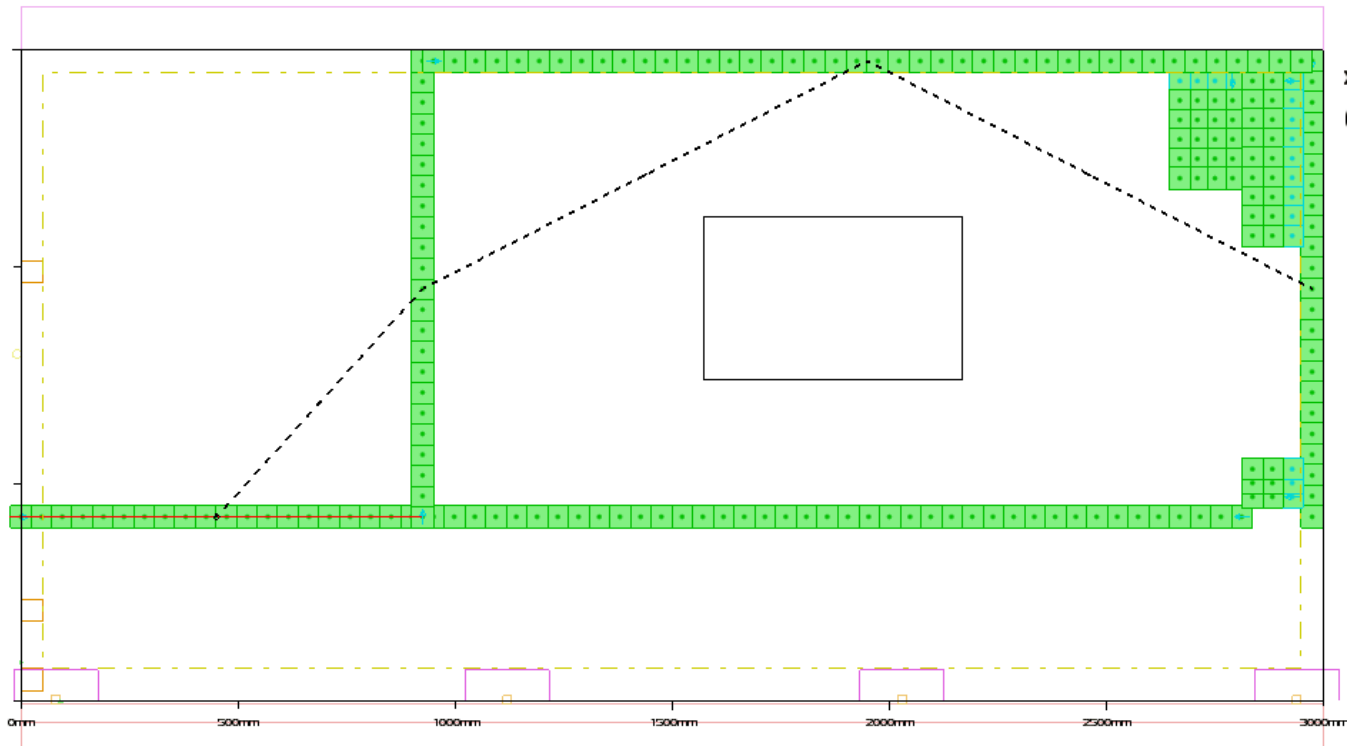
Sucker check alert

A new alert has been added to the simulation phase. It triggers if unloading is programmed for a part but no sucker has been assigned, and the part has not been placed on a pallet.



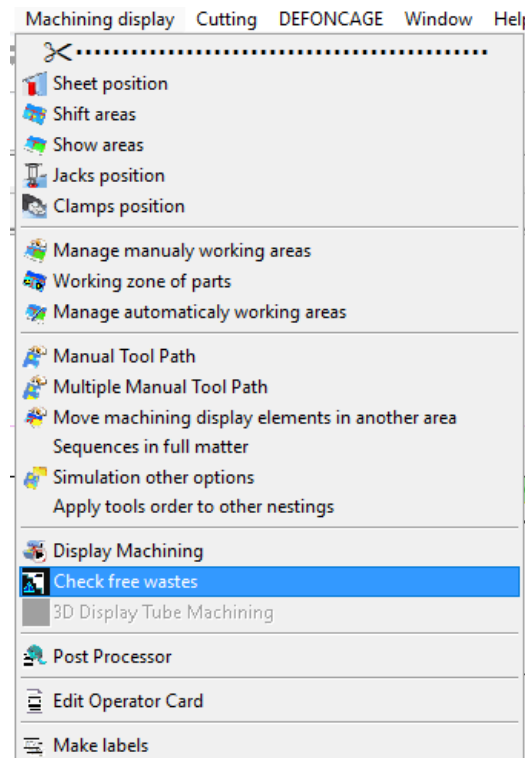
Free waste detection with simulation alert

A new feature detects free waste and non-evacuating parts, and issues an alert during simulation.



In the above example, S-shaped notching generates a free waste if splitting around the sheet occurs before notching.

In the **Simulation** menu, select the **Check free wastes** command and click on the **Start** button to launch detection.



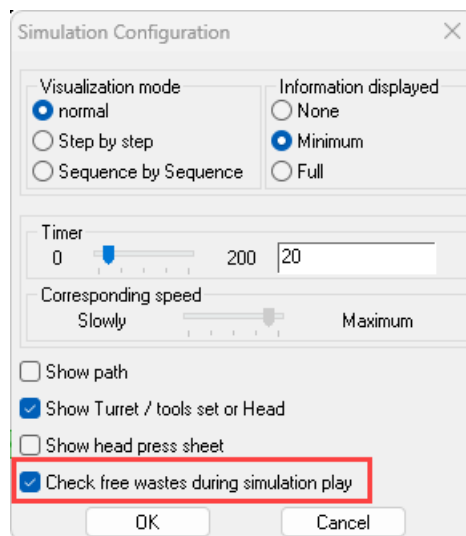
If no strategy has been defined, the system calculates default operations. Otherwise, it follows the order of operations defined by the strategies for detecting free waste.

When free waste is detected, the system displays the sheet in shaded mode, zooming in on the anomaly in the punching progress. The punch responsible for the free waste is highlighted in yellow. A scroll wheel zoom is available for greater precision.

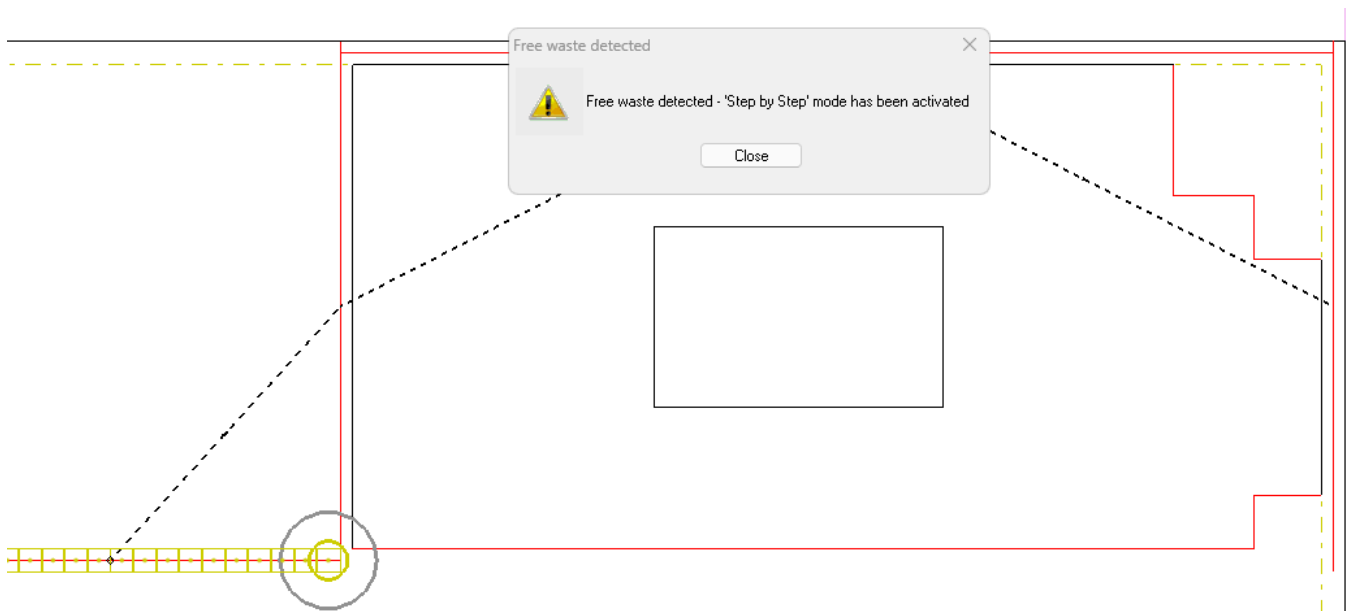


It is possible to continue free waste detection or abandon the search.

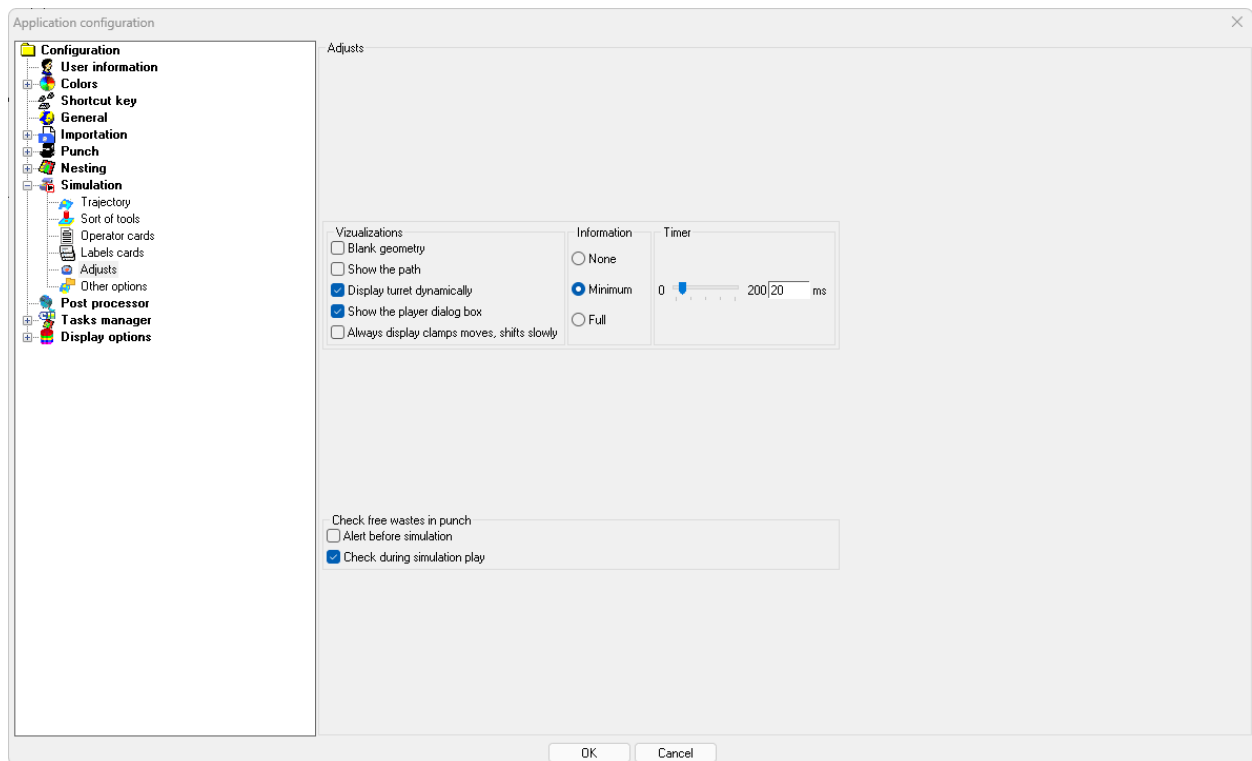
Detection can also be carried out during simulation by checking the **Check free wastes during simulation play** option in the simulation configuration.



If an anomaly is identified, the simulation pauses and switches to step-by-step mode at the start of the relevant sequence for more precise control. Any problematic punches are displayed in red.

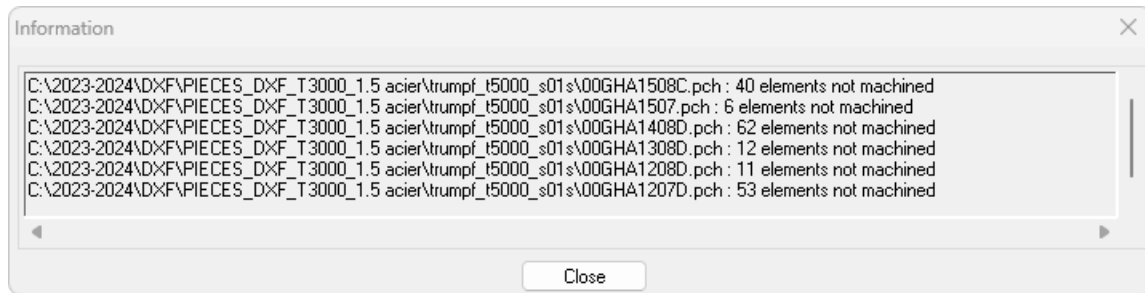


An alert can be added to notify the presence of free waste when calculating operations. This alert is set up in **Tools > Options > Adjusts**.

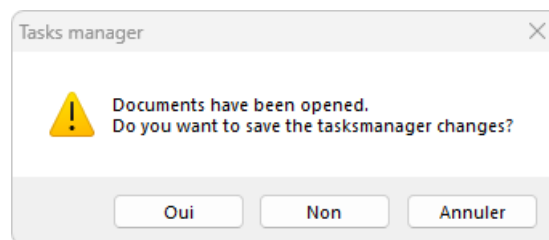


Search for unmachined profiles

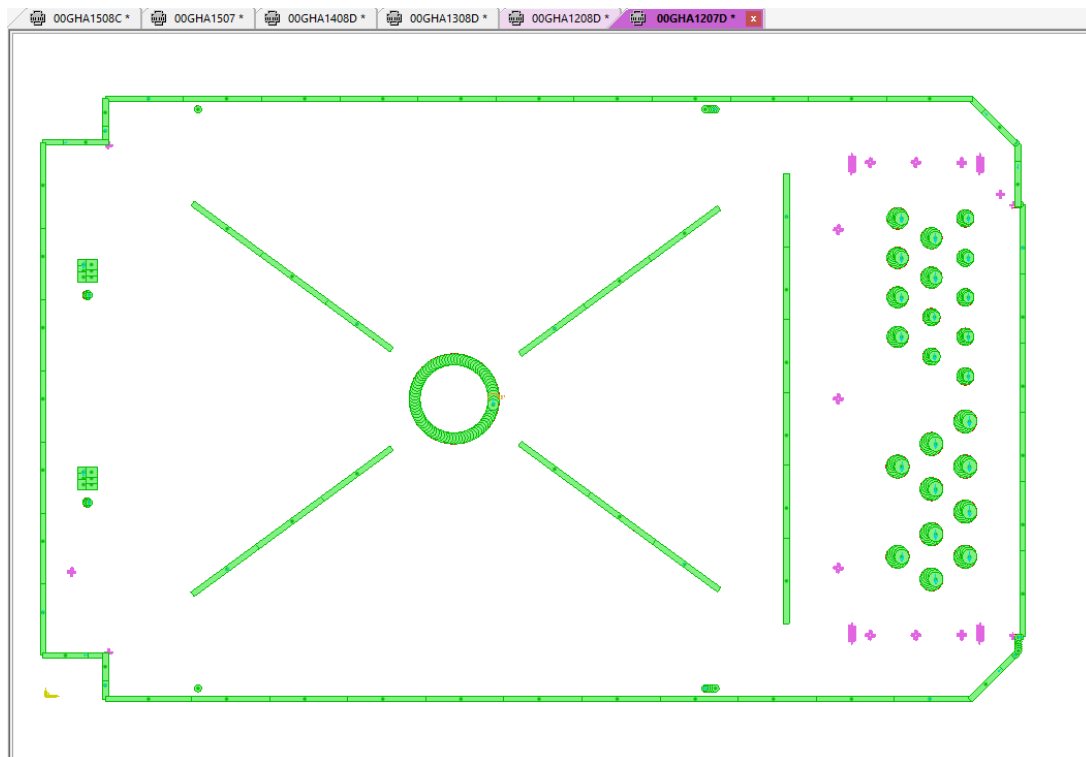
A new feature has been added to the Task Manager, under the **Parts to machine** tab, allowing the detection of unmachined elements in the PCH files of selected parts. When enabled, the system analyzes the documents and provides a summary listing the documents containing unmachined elements, along with the count of such elements per document.



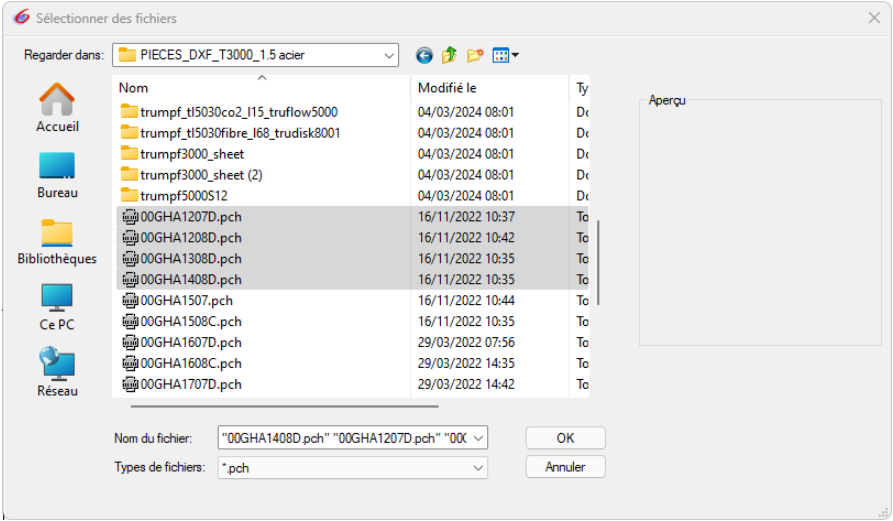
Once processed, the relevant documents are opened and organized into tabs, and a message prompts you to close the Task Manager.



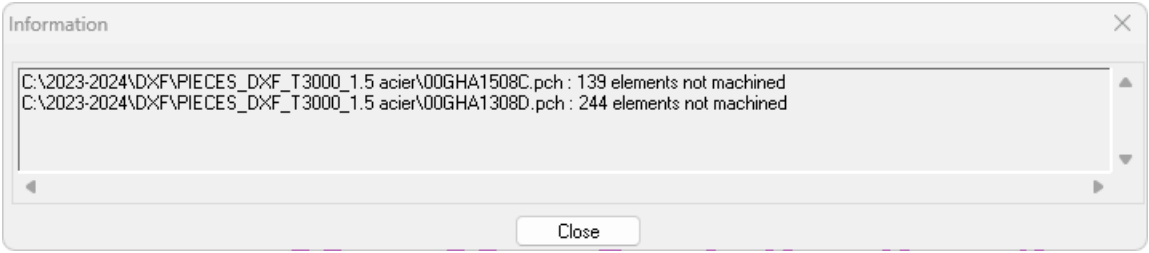
Unmachined geometries are highlighted in purple.



This feature can also be used outside the Task Manager. In this case, select the **Edit > Display not allocated profiles** command. Then click on the **Browse** button to open the file explorer and select the documents to be analyzed.



At the end of the process, a summary dialog box displays the documents containing unmachined elements along with their count. The relevant documents are then opened in tabs, and unmachined geometries are colored purple.



Dynamic adjustment of machining lengths

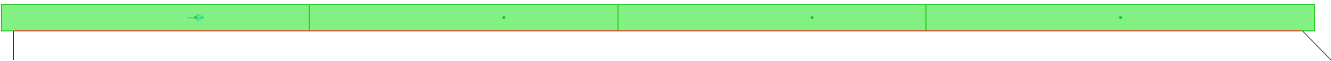
A new feature has been added to the **Punching > Modify overflanking** command, allowing the machining length to be modified dynamically.

How it works

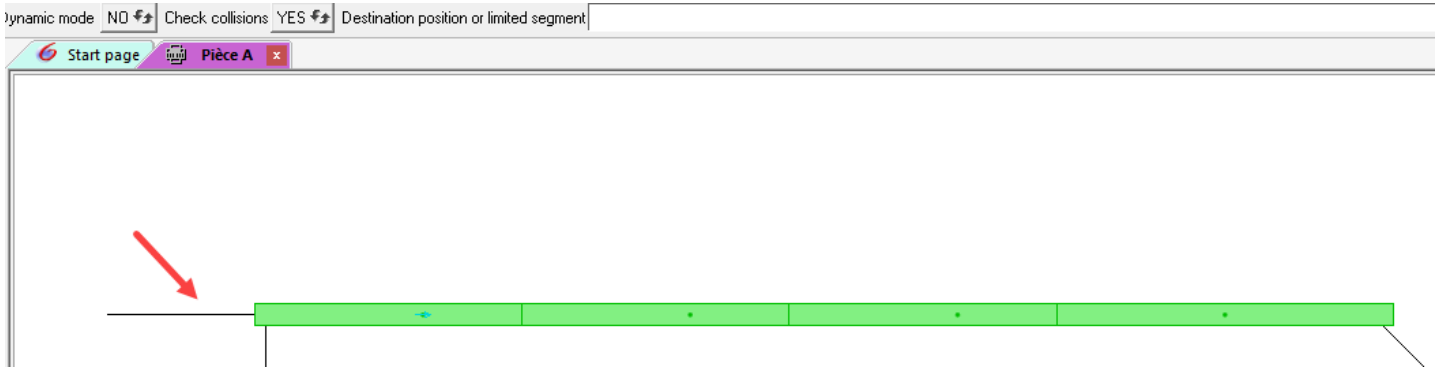
In the question bar, click on the segment near the end where you wish to modify the overflanking value.



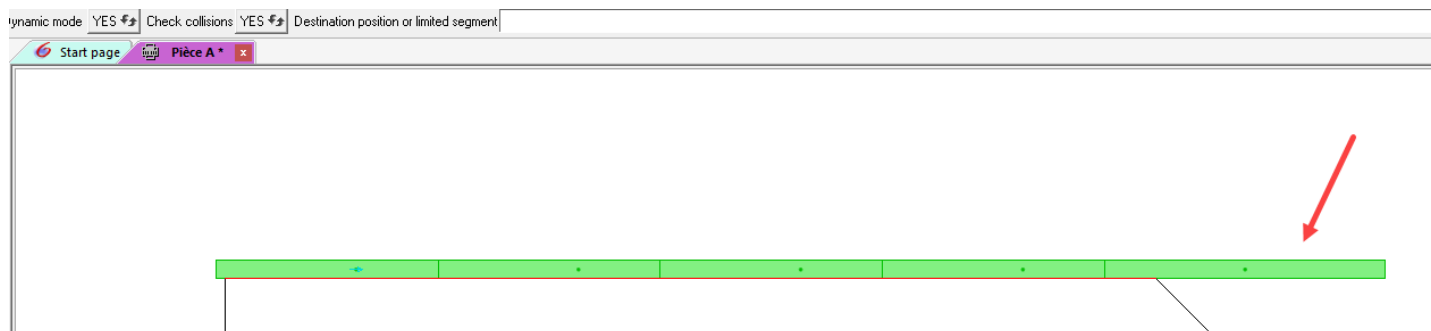
Then, click on the red-colored geometry.



In **Dynamic mode = NO**, a line follows the mouse cursor.

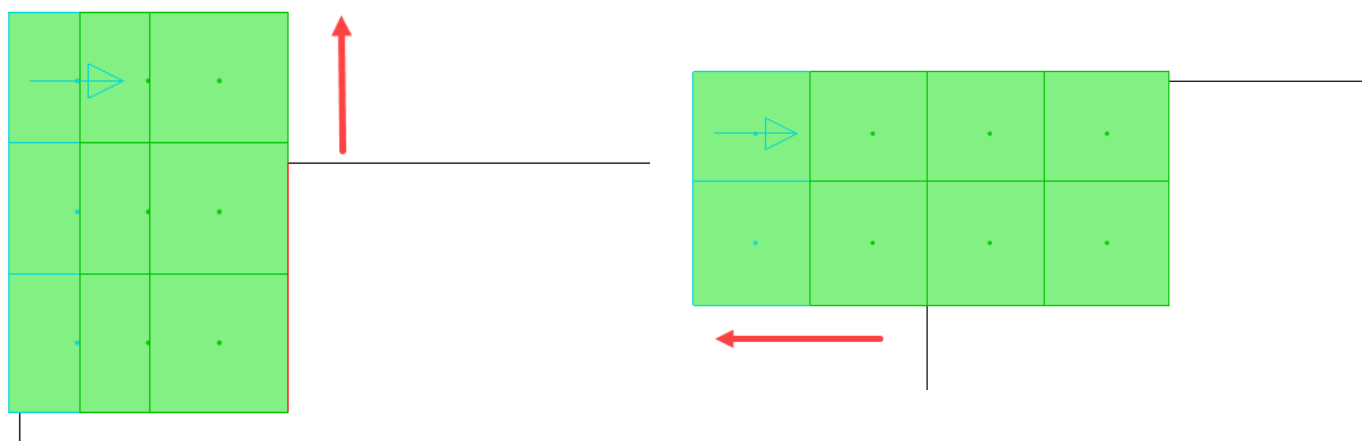


In **Dynamic mode = YES**, punches are recalculated in real time as the mouse moves. Click to define the final position or click on a geometry segment, and the overflanking will be calculated according to the projected point.

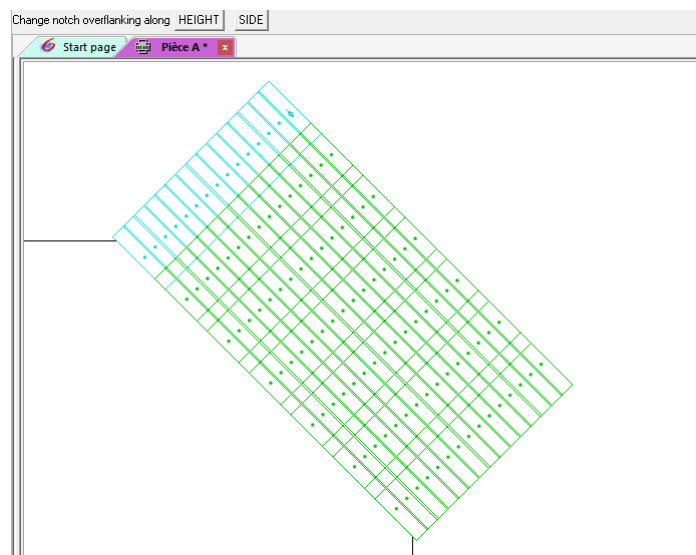


For automatic splitting lines, the principle remains the same: the selected geometry will be at the center of the splitting, without collision management. Overflanking modification is also possible for notching, without collision management.

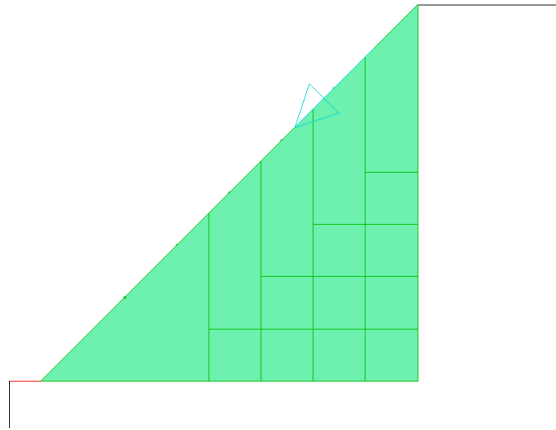
For two-segment notching, click on the corresponding segment to modify the overflanking according to side or height.



For single-segment notching machined with a square or rectangular tool, an additional question allows you to choose whether to modify the overflanking based on height or side.



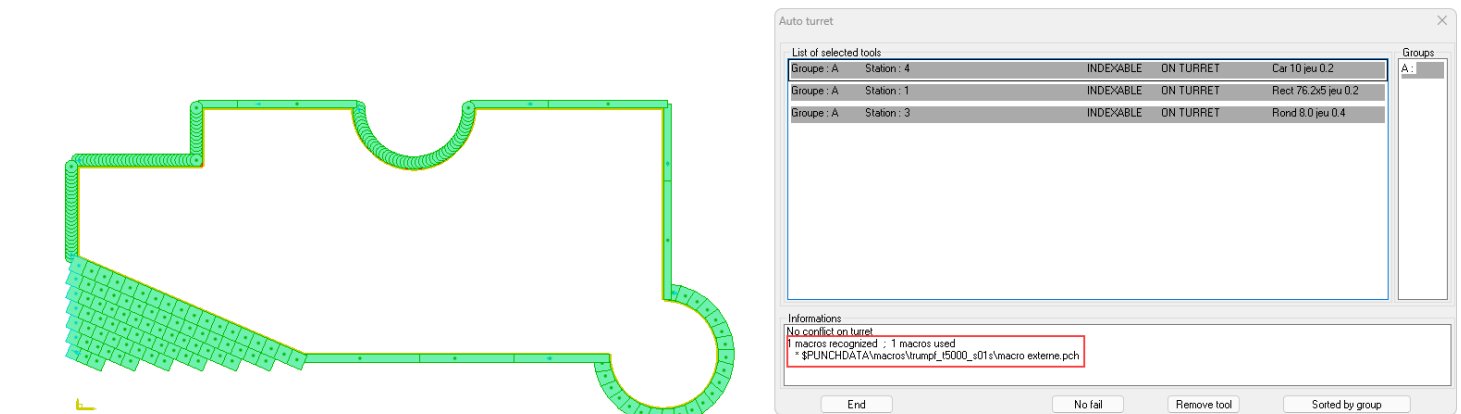
Overflanking modification is also available for notching performed with a triangle tool.



Creation of macro on an external contour

It is now possible to create a macro on the external contour of a part. To do so, two options are available when creating the macro:

- Designate the first and last contour segments, taking into account the direction of the arrow.
- Designate the same segment twice.



Macro creation on a complete external contour is not supported on cutting machines.

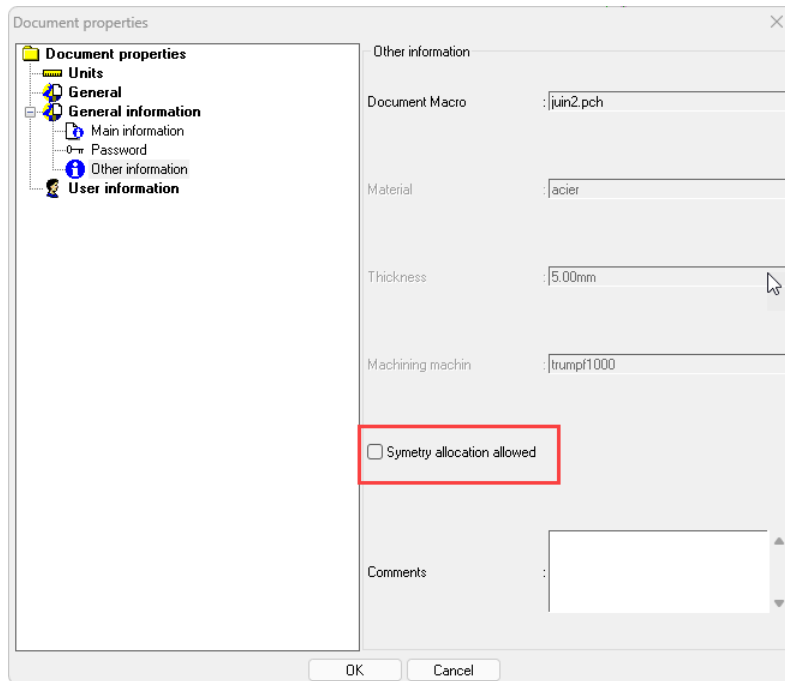
Symmetry macro allocation

Previously, external macros were recognized in their original position, whatever their orientation, including vertical symmetry, which could cause issues depending on the associated tooling.

You can now specify whether an external macro can be used symmetrically when it is created.

Keep PCH macro doc opened= YES Keep Punch points= Only those in box Symetry allocation allowed= YES Subdirectory = EXPLORER Name of the Macro

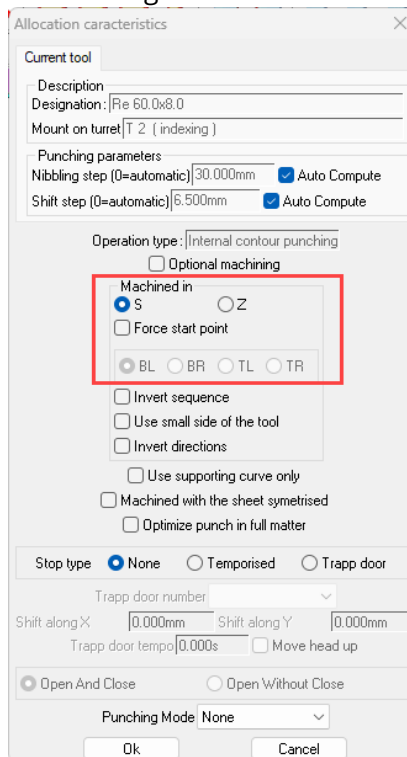
For macros created in previous versions, which are by default recognized in symmetry, you can adjust this behavior without recreating them. To do so, go to **Manage > Macros > Edit**, select the macro in the explorer, change the checkbox setting in the document properties, then save the macro document.



Improved starting points for rectangle machining

New options are available for machining rectangles in S mode. It is now possible to:

- reverse directions;
- choose between vertical and horizontal machining.



The simulation ensures strict adherence to the corner, preventing any reversal of the rectangle's path to optimize the overall trajectory on the sheet. In S mode, the path maintains a "S" movement, unlike Z mode, which follows a "Z" path.

Definition of a default attachment tool compatible with material and thickness

Previously, a single attachment tool could be defined in **Tools > Options** for a given machine, which limited flexibility, especially when the tool could not punch all thicknesses, or when different tools were used for different materials and thicknesses.

The **Attach tool** feature can now be defined directly in tool management, enabling this property to be assigned to multiple tools.

The 'Punch conditions' dialog box is shown with the following settings:

- Punch thickness (mm):** Material: Acier, Thickness: 4. Buttons: 'Ok for thickness', 'Active Material' (checked), 'Activate All Materials' (unchecked).
- Tool particularities:**
 - ☒ Normal
 - ☐ Marking
 - ☐ Noching
 - ☐ Whisper tool
 - ☐ RollerBall
 - ☐ Rolling Shear
 - ☐ Bending
 - ☐ Multi shear
 - ☐ Deburring
 - ☒ **Attach tool** (highlighted with a red box)
- Overlapping:**
 - Length: 0 mm
 - Width: 0 mm
 - Button: 'Overlappings Per Matter / Thickness...'
- Work direction:** ☐ Only within length
- Speed Code:** 0
- Punch cadence Code:** (empty)
- Speeds Per Matter / Thickness...** (button)
- Overflanking (mm):** -1 (-1 = defined in tools/options)
- Nibbling (mm):**
 - ☒ Can nibble
 - Minimum step: 0.5, Maximum: 1
 - Coef of length for min. step: 0
- Minimum tool feed:** Thickness (x): 1
- Lubrication:** 0 Punches, -1 = None
- Dead Points:** Top: 0, Bottom: 0
- Per Matter/Thickness** (button)
- Die allowance Per Matter / Thickness.....** (button)

Buttons at the bottom: 'Ok', 'Cancel'.

This feature is available only for the following tool shapes:

- Calked end
- Trapezoid
- Rectangle
- Square

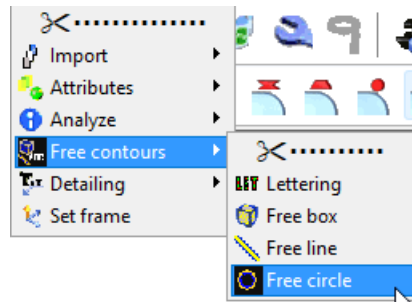
During automatic machining, **TopSolid** automatically selects the first tool with the **Attach tool** feature that is compatible with the material and thickness of the current part.

Note:

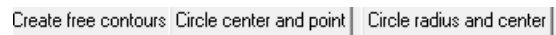
- The **Auto adapted covering** option, which was previously configurable in **Tools > Options**, has been removed.
- By default, in automatic mode, the overlap of the attachment tool on the main tool is now automatically adjusted to prevent unfinished machining.
- In manual machining, it is still possible to impose an overflanking value.

New free geometry

A new feature is available in the punching menus to create free full-circle geometries.



A circle can be created in two ways.



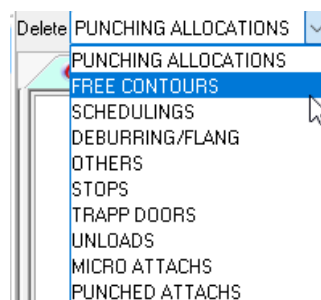
Once the circle is created, it can be punched.



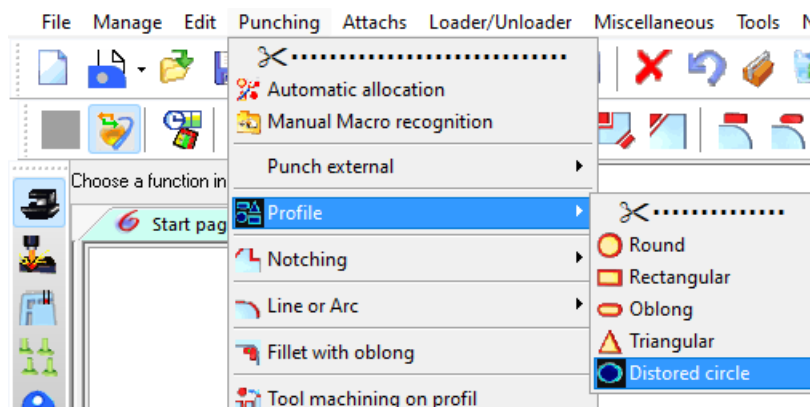
If **YES** is selected, various choices are available for punching the circle.



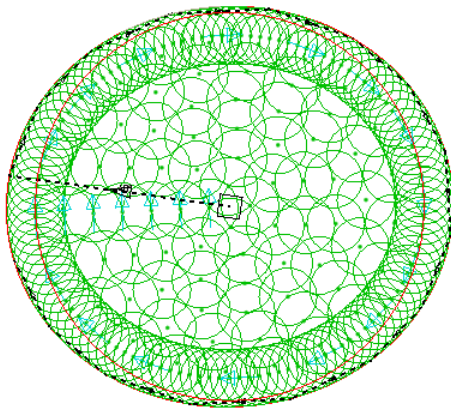
The created circles can be deleted using the **Delete** command, by selecting **Free contours**.



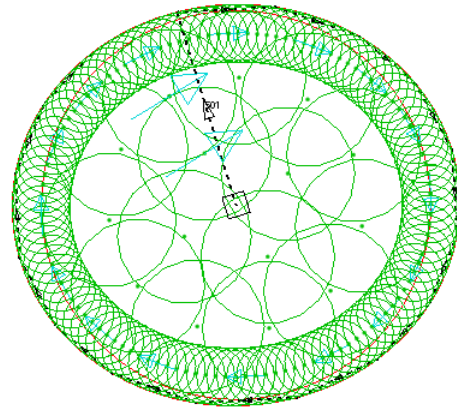
In addition, a new command lets you punch a contour resembling a deformed circle, using a round tool.



All contour segments are punched with this tool. A circle is then created at the center point of the contour and punched with either another round tool or the tool used for the contour. The machining starting point is defined by the clicked segment. A scheduling links the inner circle and all the contour segments.



Example of machining with the same tool.



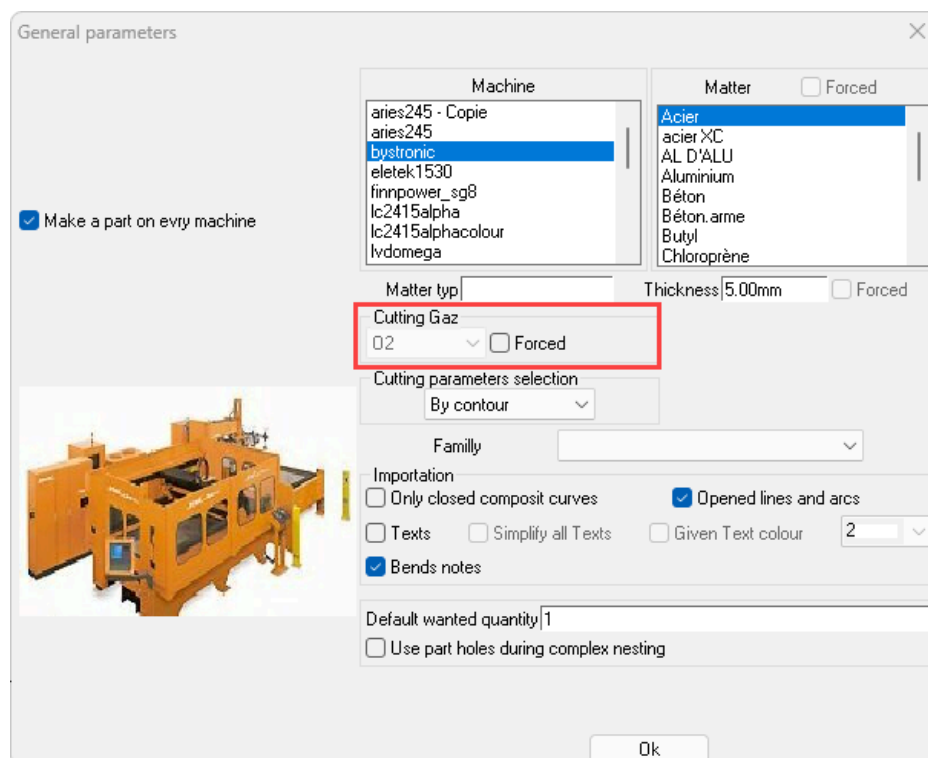
Example of machining with two separate tools.

Cutting

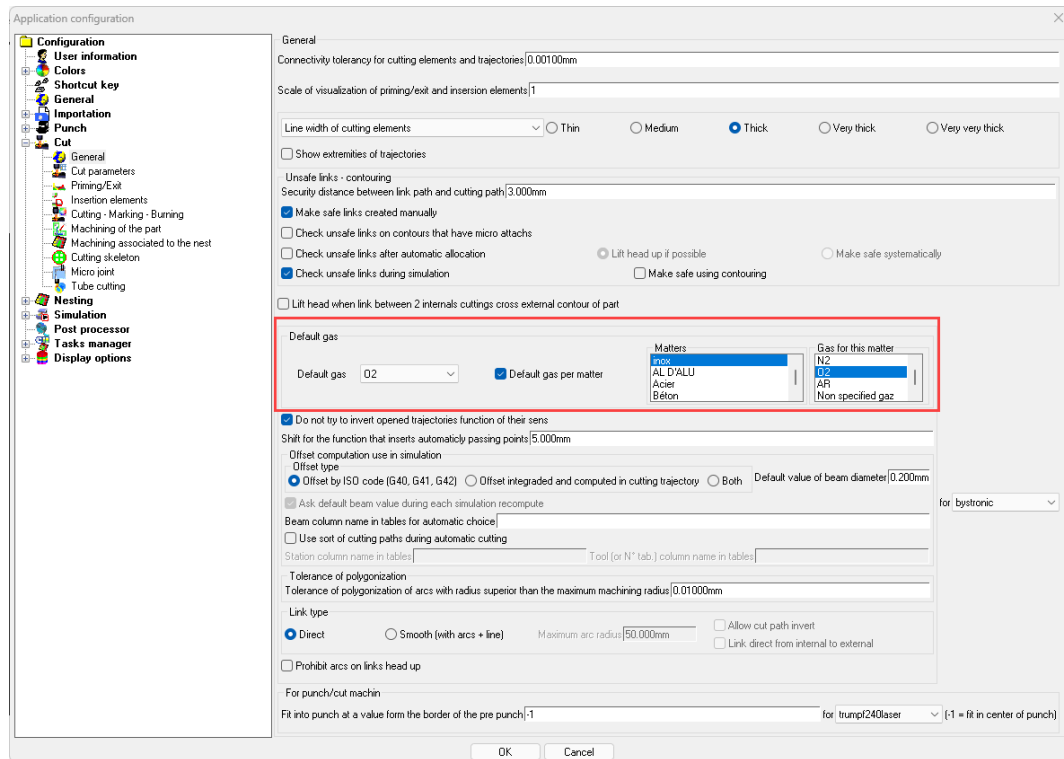
Force the cutting gas during a multi-document import

During a multi-document import, PCH files can now be generated for all machines.

The **Forced** option, accessible in the general parameters, allows the use of a specific cutting gas for laser machines. When this option is enabled, the selected gas in the drop-down list is applied to all machines.



A new option now allows you to not impose a particular gas. In this case, the system automatically selects the appropriate gas for the machine and part material, according to the settings defined in **Tools > Options**.



Optimization of part marking positioning

Part marking positioning has been improved in the following situations:

- During multi-document import, a marking is created for each individual part.
- During the associated nesting action relating to the marking of each part in the nesting.
- In the manual function for re-marking parts after nesting changes.

For example, if “centered” positioning is requested, the system checks for openings in the part. If openings are detected, the marking is automatically moved to the nearest valid position.

