NC Express e^3

Software version release: 23.2

12 December 2023

New features in NC Express e³ 23.2

Unfolding

Supported 3D formats and versions

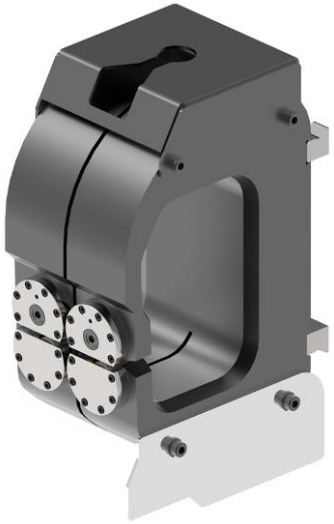
- Autodesk Inventor (*.ipt, *.iam), up to 2024
- SolidWorks (*.sldprt, *.sldasm), up to 2023
- Solid Edge (*.par, *.psm, *.asm), up to 2023
- Siemens JT (*.jt), up to 10.7
- Siemens NX (*.prt), up to 2306
- PTC Creo (*.prt.x, *.asm.x), up to 10.0
- Catia (*.catproduct, *.catpart), V4 (4.15 to 4.26) and V5-3DX (R10 to R32, R2023x), V6
- IGES (*.igs, *.iges), up to 5.3
- STEP (*.stp, *.step), AP203 (E1, E2), AP214 (up to E3), AP242 (E1, E2, E3, BO XML), AP209
- Spatial ACIS (*.sat), up to 2023 1.0
- Parasolid (*.x_t, *.x_b), up to 36.0

The BEND machines

ACP dialogue enhancements

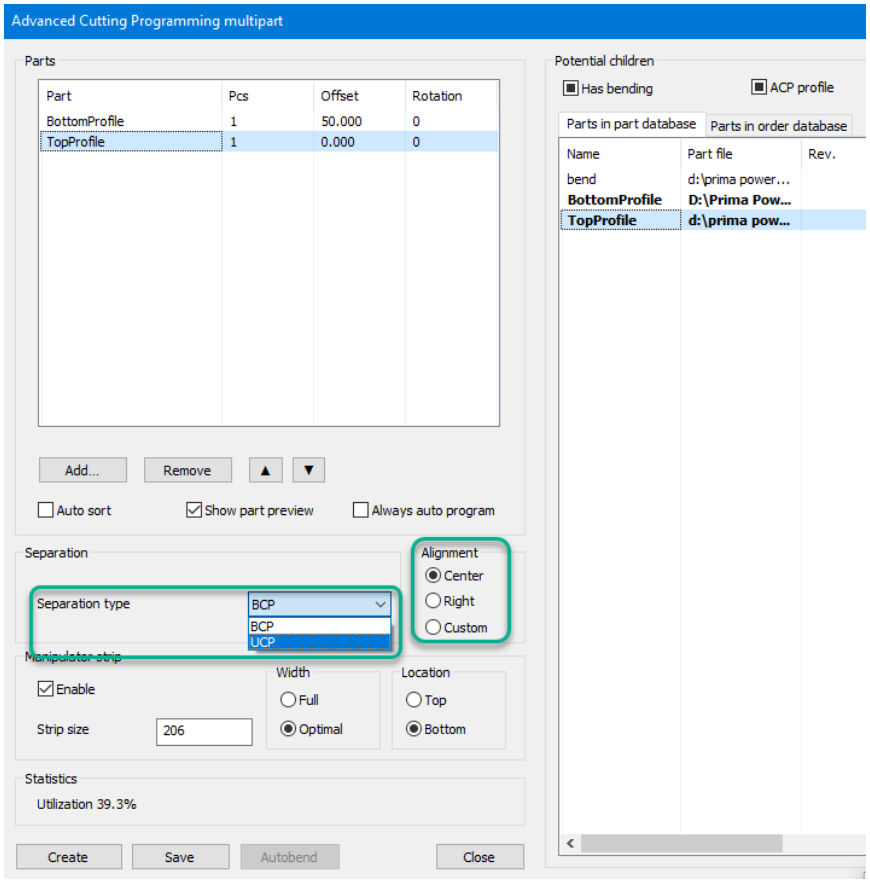
The ACP (Advanced Cutting Programming) dialogue has new options for BCP/UCP and alignment:

- BCP — Bend Cutting Profile means a 0-degree bend line between parts. Part separation is achieved with a Panel Bender with a wheel tool (option)

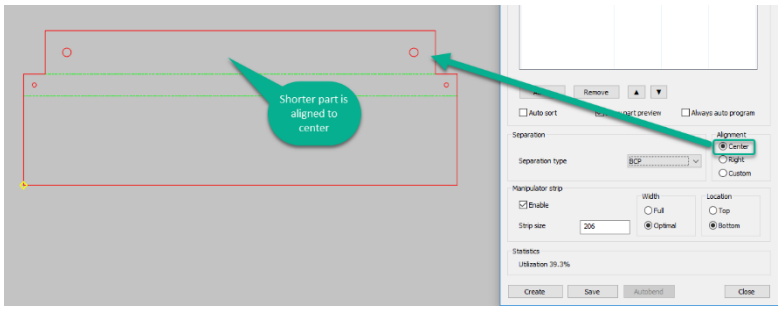


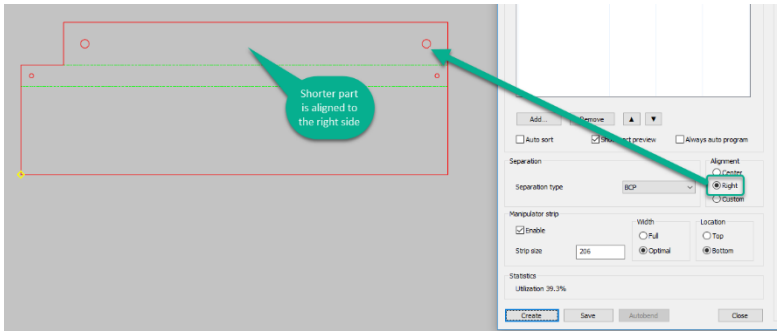
- UCP — Unloading Cut Profile means a 0-degree bend line between parts. We make a groove with a pincher wheel on the main machine (for example, SG), and actual part separation is achieved with a Panel Bender with a bending tool

The separation type can be now selected.



The alignment affects how the ACP mother parts are prepared.





Scripting command to create bending super-part

The PartNCX interface now supports the preparation of bending super-parts (ACP parts). The new methods and properties are the following:

```
boolean ACPAddChild(BSTR lpFilename, short amount,
                    double offset, double rotation)
```
Add Advanced Cutting Programming child parts with amount, offset and rotation.

```
boolean ACPCreate
```
Create the Advanced Cutting Programming multipart.

```
boolean ACPAddManipulatorStrip(double size, short type,
                               short location)
```
Add Advanced Cutting Programming manipulator strip.

```
short ACPAlignmentType
```
Type of alignment of the Advanced Cutting Programming parts:
0=Custom, 1=Centre, 2=Right. Default is 0.

```
short ACPSeparationType
```
Type of separation of the Advanced Cutting Programming parts:
1=BCP, 2=UCP. Default is 1.

```
long ACPProfile
```
Default 0, 1 if the part is to be manufactured as a child part of an Advanced Cutting Programming multipart.

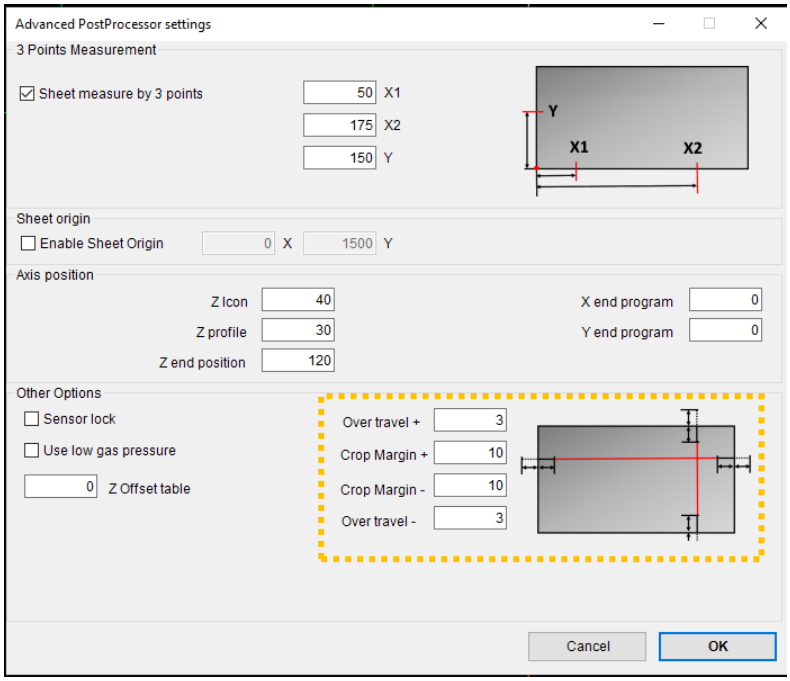
A short sample script:

```
Sub CreatePartA()
  Part.ACPAddChild "2804556-02", 5, 0, 0
  Part.ACPAddManipulatorStrip 210, 0, 1
  Part.ACPAlignmentType = 2
  Part.ACPSeparationType = 2
  Part.ACPCreate
End Sub
```

The LASER machines

Crop margin and over travel enhancements

Crop margins and over travels are the parameters of post-processing configured in the ***Advanced Post-Processing*** dialogue within *NC* settings to cut lines that start/end at the edges of a sheet (scrap/crop lines).



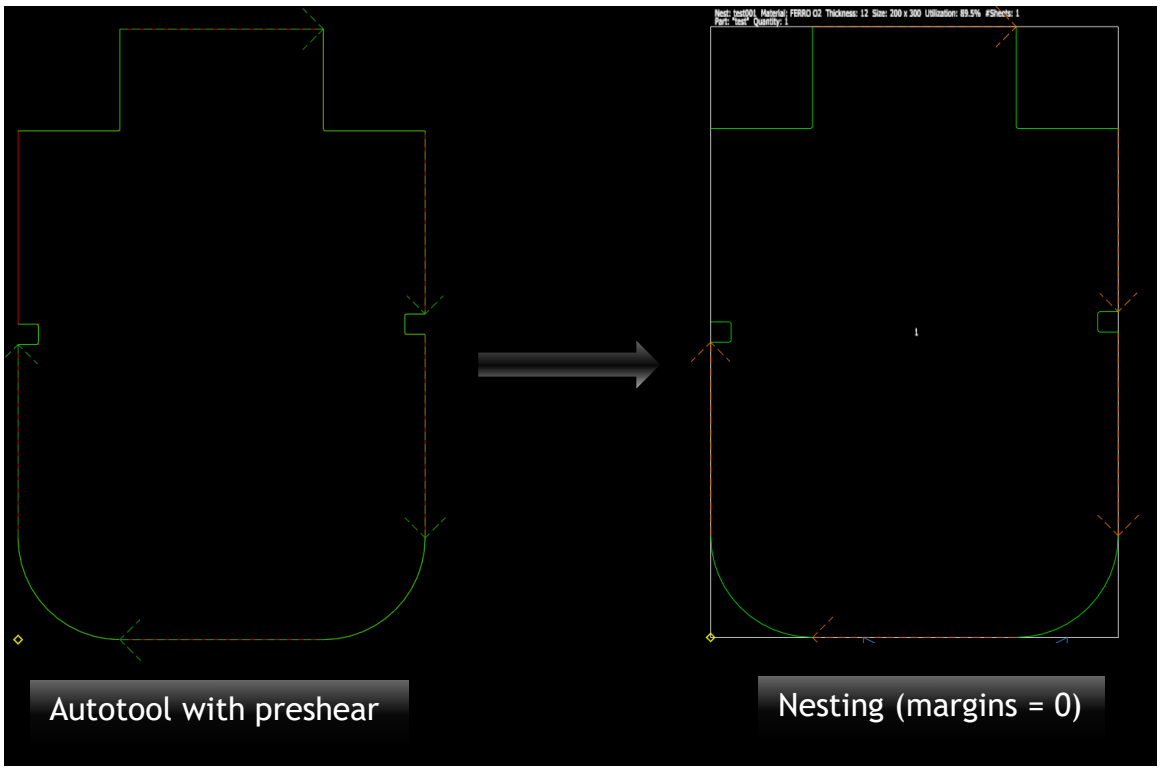
The current version introduces some enhancements related to these parameters for Tulus Laser 2D machines:

1) Circular movements are now supported. This allows the execution of the ***Preshear*** option in ***Autotool*** on parts with an external profile containing lines and arcs.
2) System tolerance is introduced to force the presence of cutting sections with a frozen head in the NC program starting/ending cutting from/to the edge of a sheet.
3) Automatic resize of these parameters can be executed during post-processing. When this happens, the system alerts the user in the log. A crop margin or over travel can be decreased for the following reasons:
    a. Geometry constraints for crop margins. This happens when the first or last segment of the scrap/crop line is shorter than the configured margin. This can occur when there are little notches on a pre-sheared sheet or when the distance among parts inside a nesting is shorter than the margins.
    b. Working area violation for over travels. The working area is defined by the travels of axes configured in Machine Settings. High configured values in the over travel and/or sheet probing parameters can cause the evaluation of points to be out of the working area of the machine.

Circular movements supported

Tulus Laser 2D machines support circular movements executing a scrap/crop line starting/ending from/to the edge of a sheet. This helps when using pre-sheared sheets.

The user can enable the ***Preshear*** option in the ***Autotool*** dialogue to cut the subsections of the external profile containing linear and arc segments on a rectangular sheet, for example as in the following image.



Tolerance management

Three sub-phases define a cut that starts and ends at the edges of a sheet:

1) Initial section with frozen head

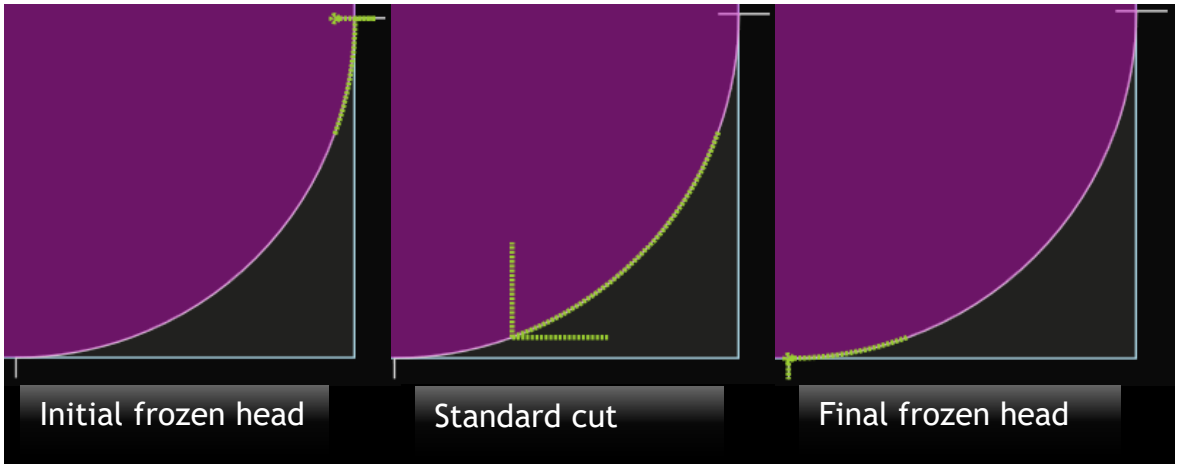
2) Standard cutting section

3) Final section with frozen head

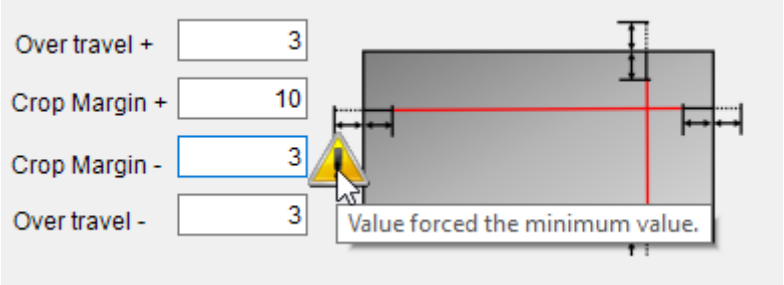
When the initial and/or final section are not defined in the NC program, collisions can happen on the real machine.

NC Express forces a minimum value for both crop margins and over travels to execute a motion on the machine inside and outside the sheet with the tracking sensor disabled (we call it “frozen head”) to cut the initial/last section of the scrap line. This value is 3 mm (0.11 inches) for both crop margins and over travels.

The following images show a subsection of an NC program that executes a scrap on a circular segment.



When the user tries to force a value smaller than the tolerance in the *Advanced Post-Processing* dialogue, the system automatically forces the tolerance value.



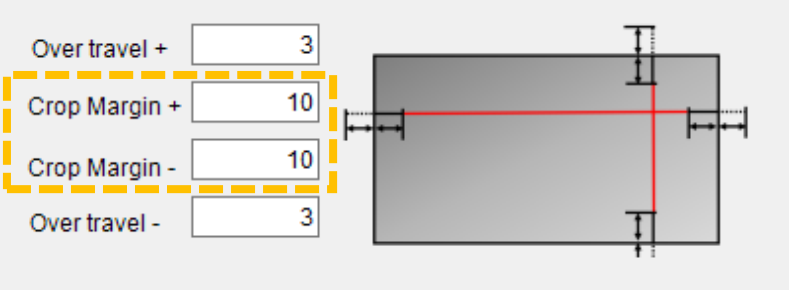
Auto-resize

Crop margins

The post-processing phase resizes crop margins automatically when the configured crop margin is bigger than the geometric length of the segment (initial or final) of the scrap profile, because the configured margin overrides the geometry of the nested part if maintained.

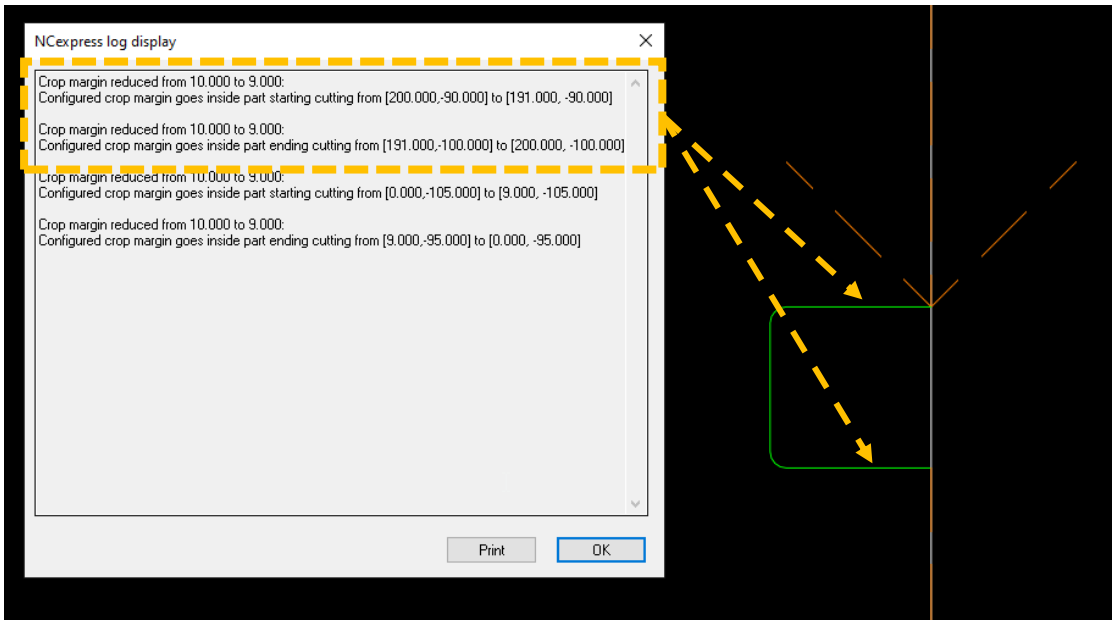
Every resizing action is added to the log produced at the end of post-processing to advise the user.

In the following example, we consider the part of the previous pre-shearing example. The crop margins are 10 mm in the *Advanced Post-Processing* dialogue.



The part has the same rectangular notch on two sides. The length of the initial and final segment is 9 mm, which is smaller than the configured crop margin.

The post-processing log contains the resize warning about the four resizes (two for each side) when approaching and departing the sheet.

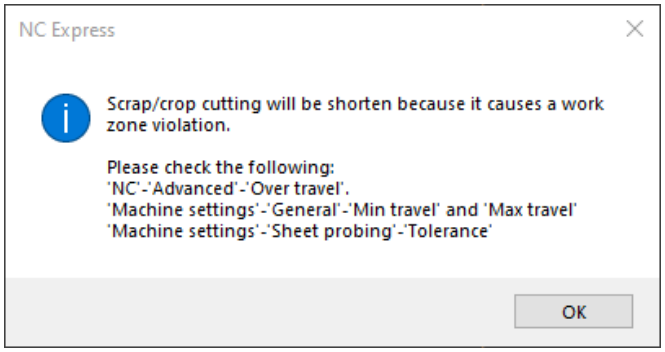


Over travels

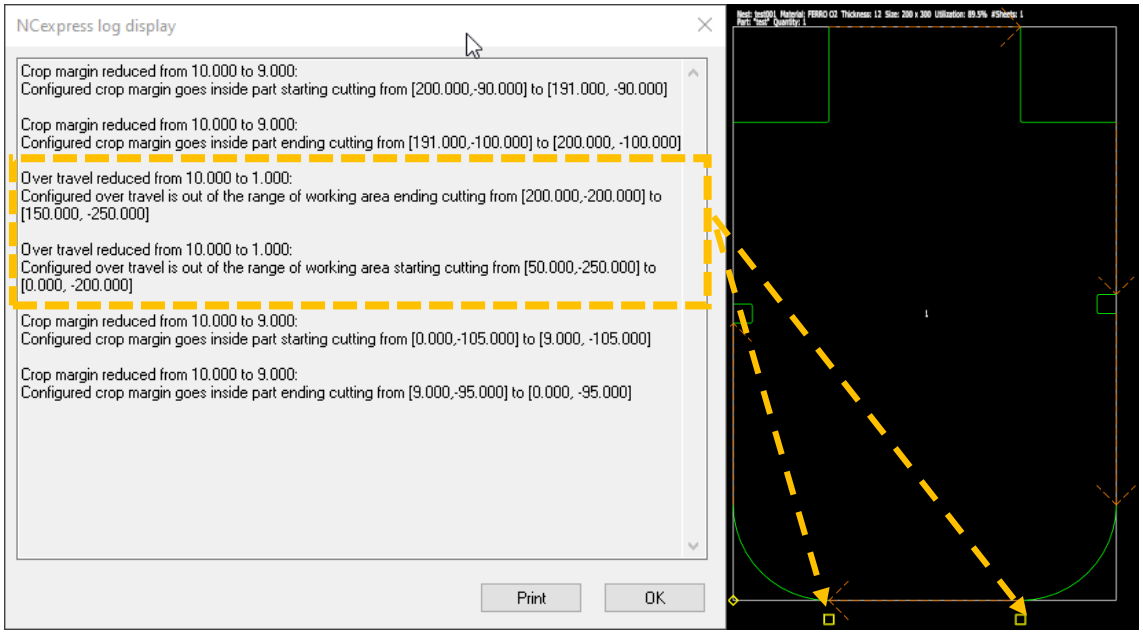
The post-processing phase resizes an over travel automatically when it is out of the working area.

This automatic resizing of over travels generates additional warning messages regarding the resize of crop margins because it depends on the working area of the machine.

A message is displayed before executing post-processing, advising the user to check the current parameters that can impact it.

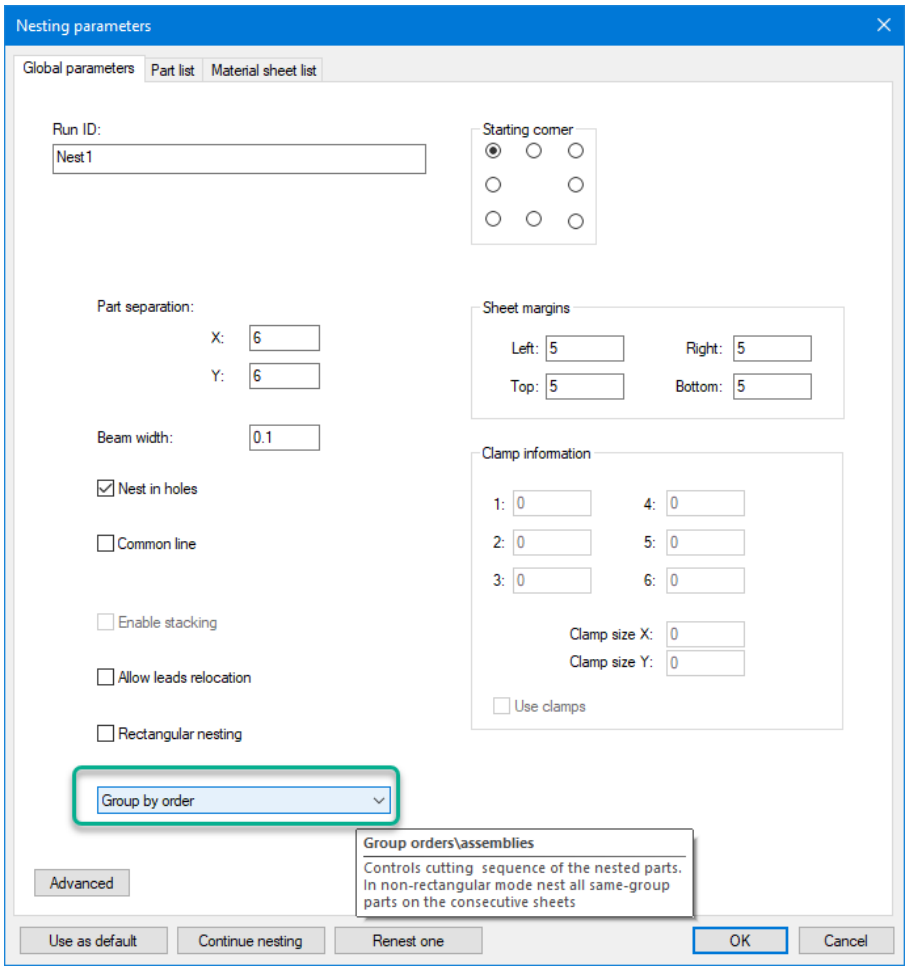


Every resizing action is added to the log produced at the end of post-processing to advise the user.

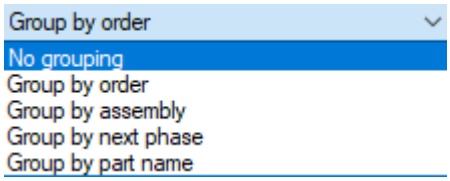


Support grouping in free-form nesting

With the grouping functionality, you can implement the part nesting order to consecutive sheets. All parts with same group are nested on consecutive sheets.

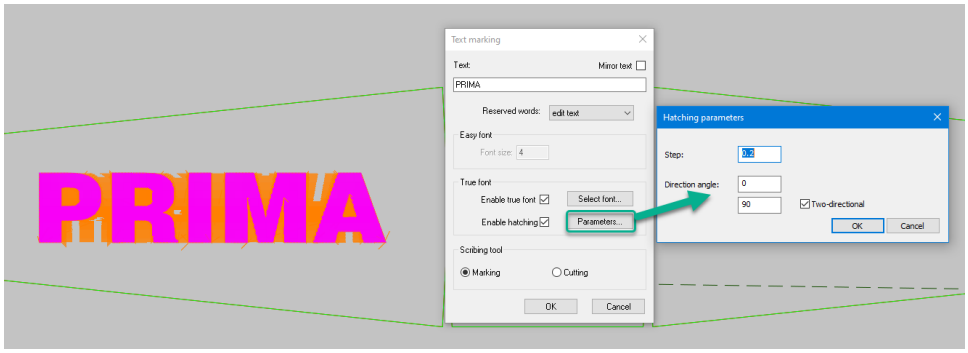


The grouping method can be one of the following:



Hatching of true type fonts

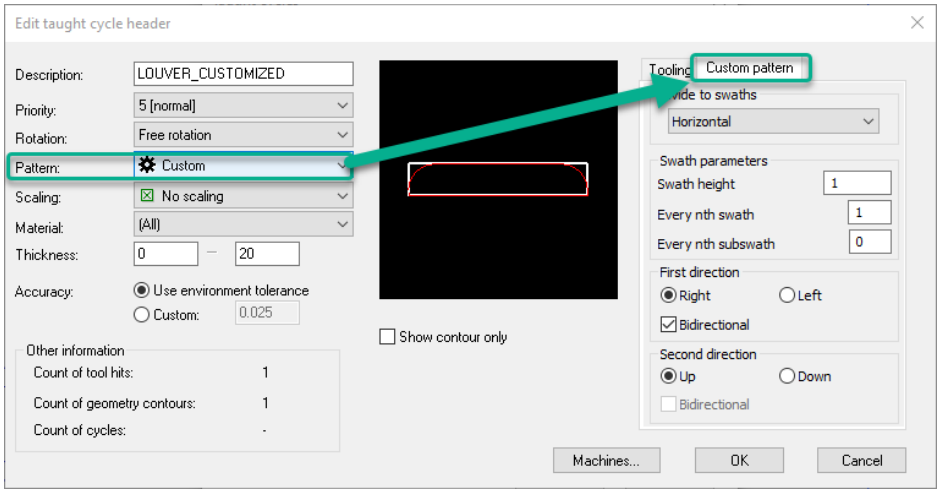
True type font hatching can be activated from the Text marking dialogue, by selecting the ***Enable hatching*** tick box. Parameters can be given for the step, hatching angle and "***two-directional***" hatching. Two-directional hatching offers the possibility of better visibility for marking but takes longer to complete.



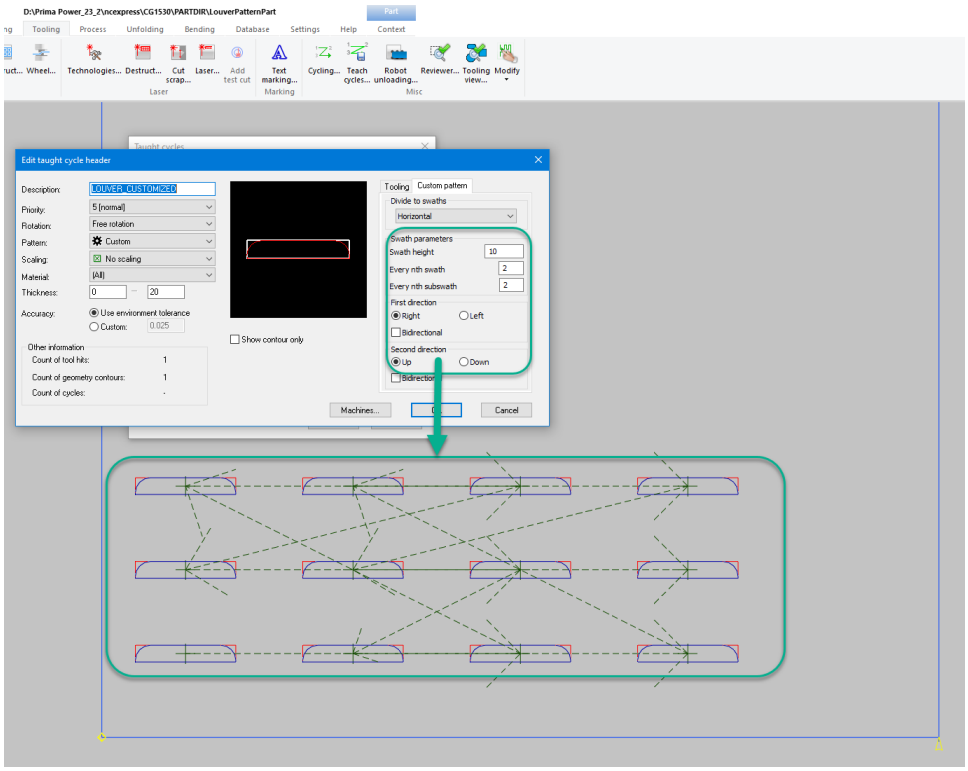
The COMBI, Punch-Shear machines

Custom pattern in Teach cycles

There is a new option to be found in ***Teach cycles*** -> ***Pattern*** -> ***Custom***. With this you can set a more complex pattern that is automatically applied to these tool hits when ***Teach cycles*** are set in ***Autotool***. The swath parameters are the same as those available in the ***Cycling*** -> ***Sort selected*** dialogue. This can be used, for example, when a forming tool is distorting the sheet flatness while machining. A custom pattern may help to keep sheet flat during machining.



Here is a sample of a Teach cycle with custom pattern.

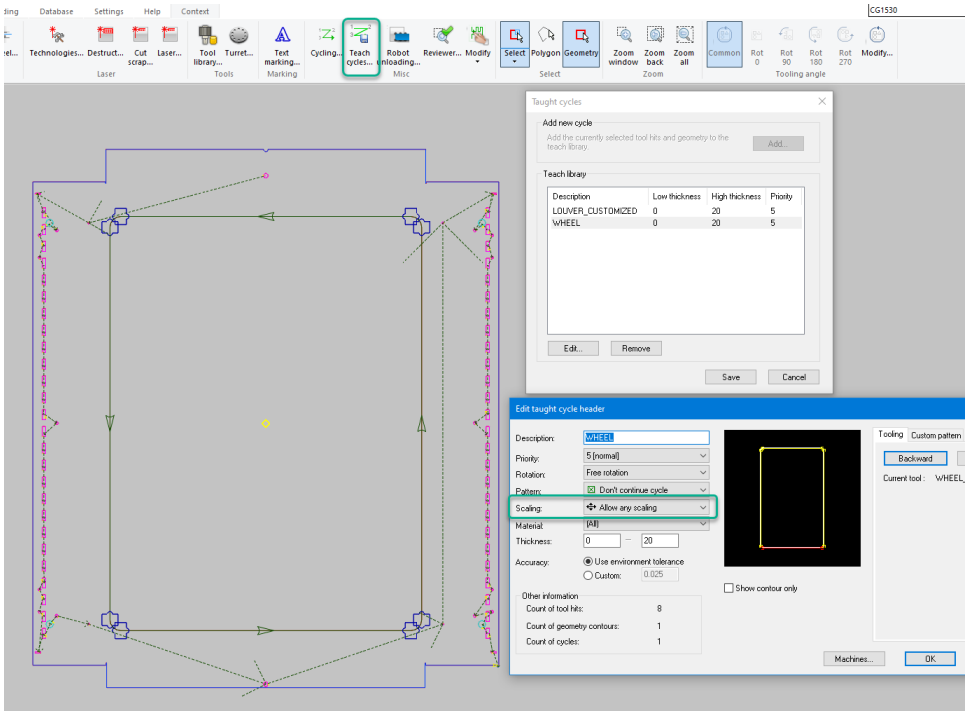


Allow scaling to change aspect ratio in Teach cycle

There is a new option, *"**Allow any scaling**"* in ***Teach cycle -> Scaling***, which allows free scaling (without keeping the original aspect ratio). This is useful in cases where parametric parts are produced with different dimensions. In this instance, you need to teach only one basic shape and ***Autotool*** can scale the cycle for different part sizes.

Please note the option *"**Allow scaling in X and Y**"* will otherwise perform the same, but it requires the shape to keep the same aspect ratio.

In this example, the inside obround shape is taught to the WHEEL tool that makes the stiffening rib for a panel. That same panel is produced with different X/Y sizes.



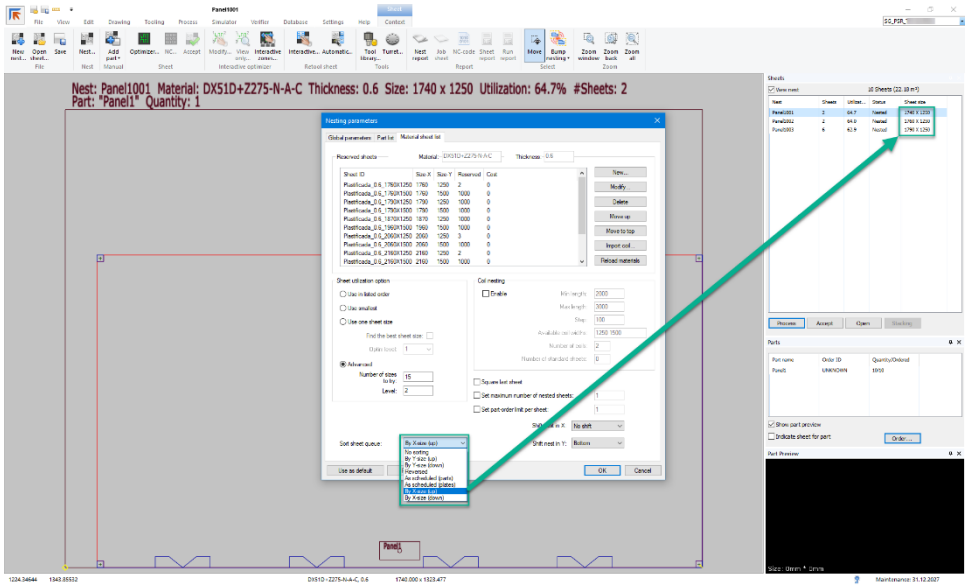
Sort sheet queue in nesting

The following new options sort sheets primarily by X size in ***Sort sheet queue***:

- ***By X size (up)***
- ***By X size (down)***

Meanwhile, the following options primarily sort by Y size (same coil width), and then the same Y size sorting is made by X size:

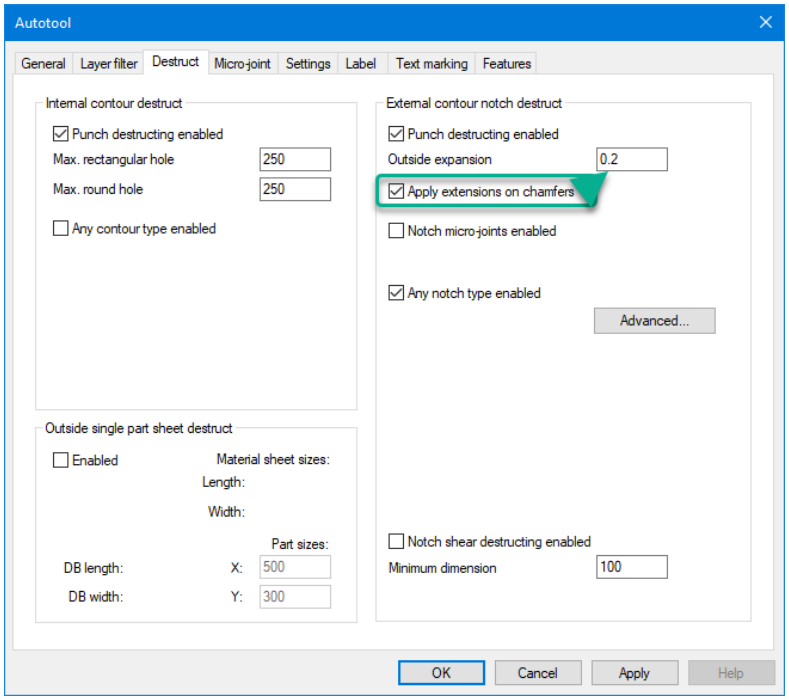
- ***By Y size (up)***
- ***By Y size (down)***

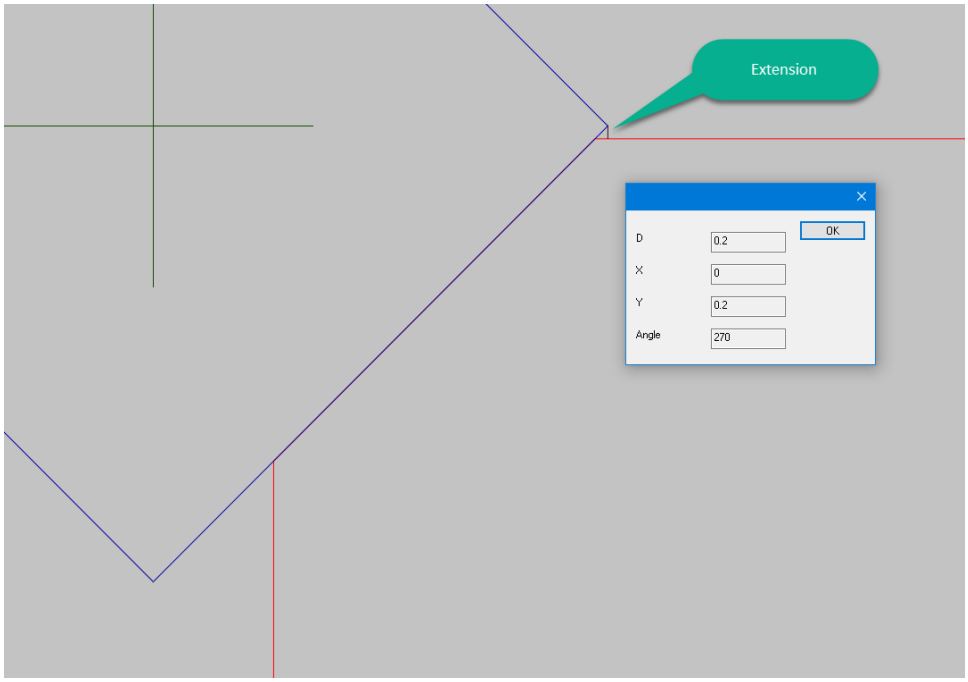


These new options can help to keep clamp positions the same between nested sheets, making production faster as less clamp moves will be required between sheets. The optimiser setting ***Prefer clamp positions of previous sheet*** is useful in this case.

Extend tool hits on a chamfer

Autotool -> Destruct -> Apply extensions on chamfers is a new function that prevents spikes in part chamfers (on a non-right-angular notch). When it is active, it prevents the common line of the part from nesting, but can produce better quality parts. Tooling is extended outside the part to the value of the “Outside expansion” parameter.





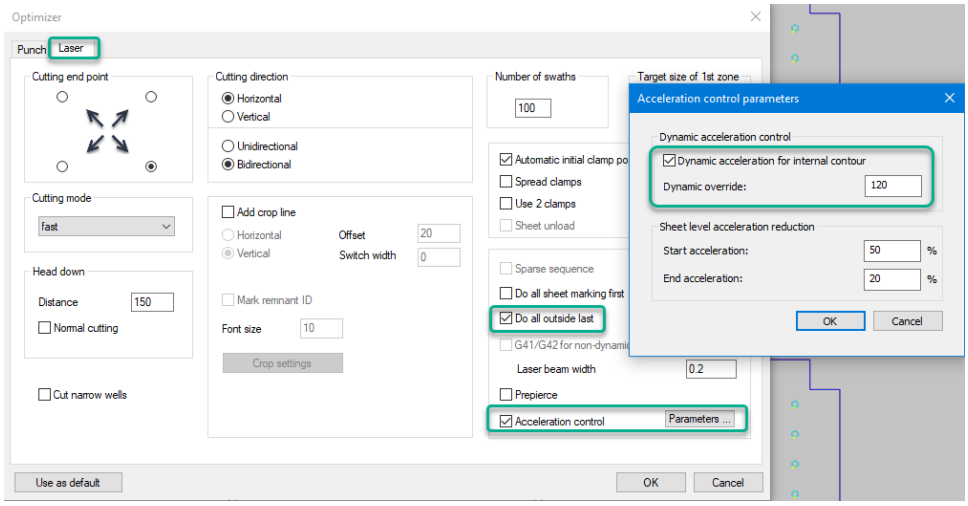
The COMBI, Laser-Punch machines

Dynamic acceleration control

Dynamic acceleration control is a new feature for the COMBI laser machines optimiser that allows the user to activate the higher (or lower) axis dynamic for laser cutting of internal holes on parts. It raises the acceleration (m/s^2) and jerk (m/s^3) with a given dynamic override percentage of 120%. This means that the dynamics of the X- and Y-axes are increased by 20% compared to normal laser database cutting values.

This is the same functionality that can be activated with the real machine control panel's Dynamic button.

NOTE: It is highly recommended to activate ***"Do all outside last"*** when dynamic acceleration is activated. This means more holes are cut with higher dynamic values, which equates to faster production.



The command RAMP_MODE is written to the NC-code to activate the functionality:

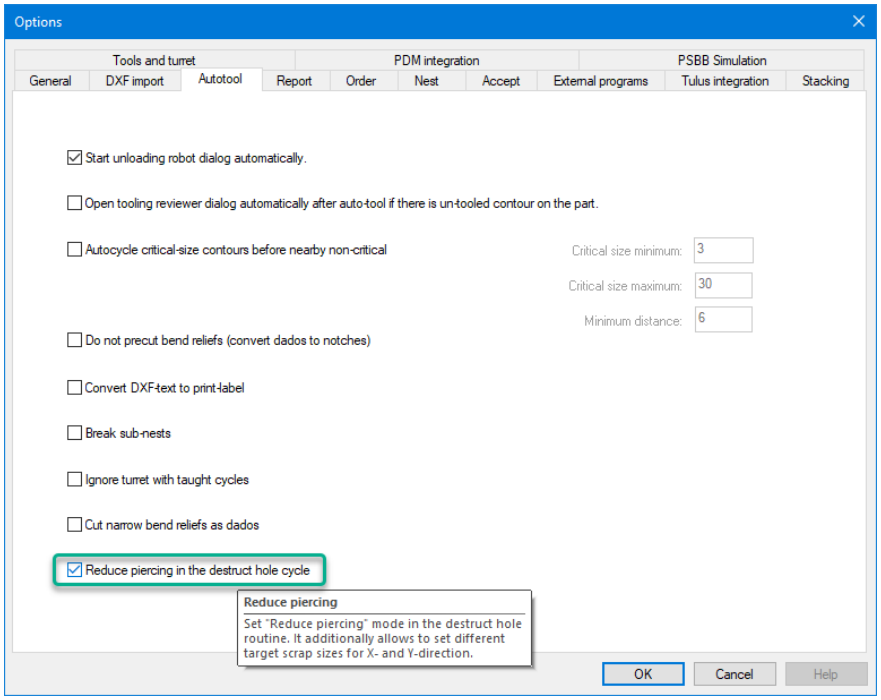
```
<SUB_PART SubName="CGPart1_0_0">
        <SUB_START/>
        <RAMP_MODE Mode="1" DynamicOverride="120"/>
```

And the same command with Mode="0" cancels the dynamic acceleration function. This is written to the NC-code when the outside contour cut starts.

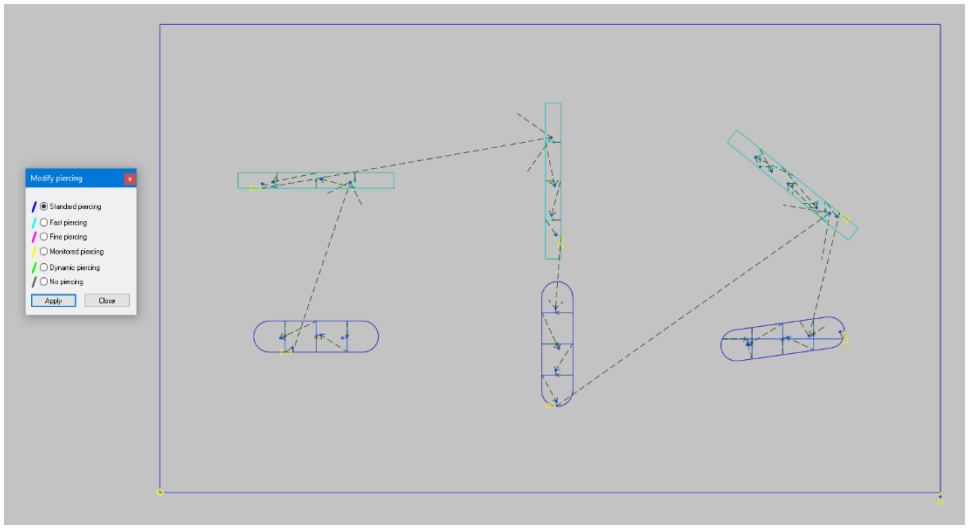
```
<SUB_PART SubName="CGPart1_0_2">
        <SUB_START/>
        <RAMP_MODE Mode="0" DynamicOverride="120"/>
```

Reduce number of piercings in the laser destruct

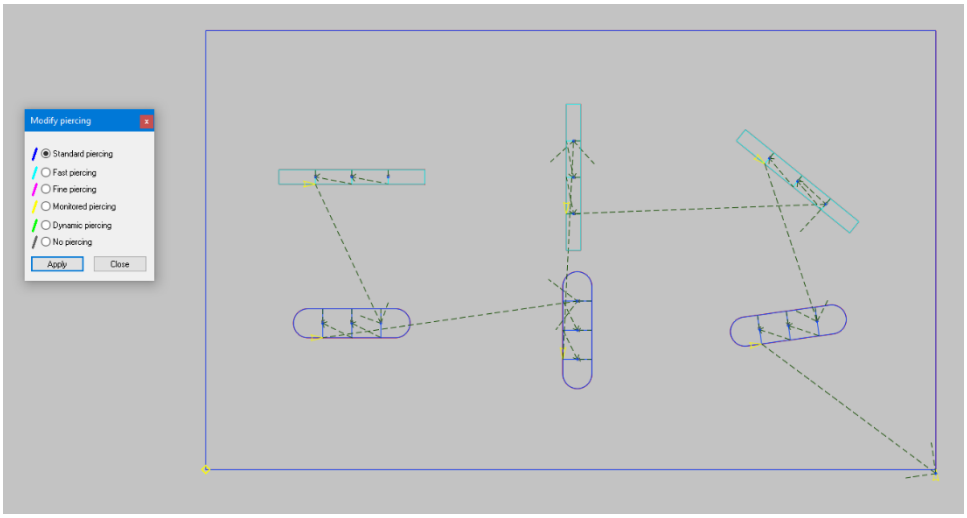
Internal hole laser destructing becomes faster when you activate this functionality.



The Autotool result before activation is shown below. This produces four piercings for each internal hole destruction.



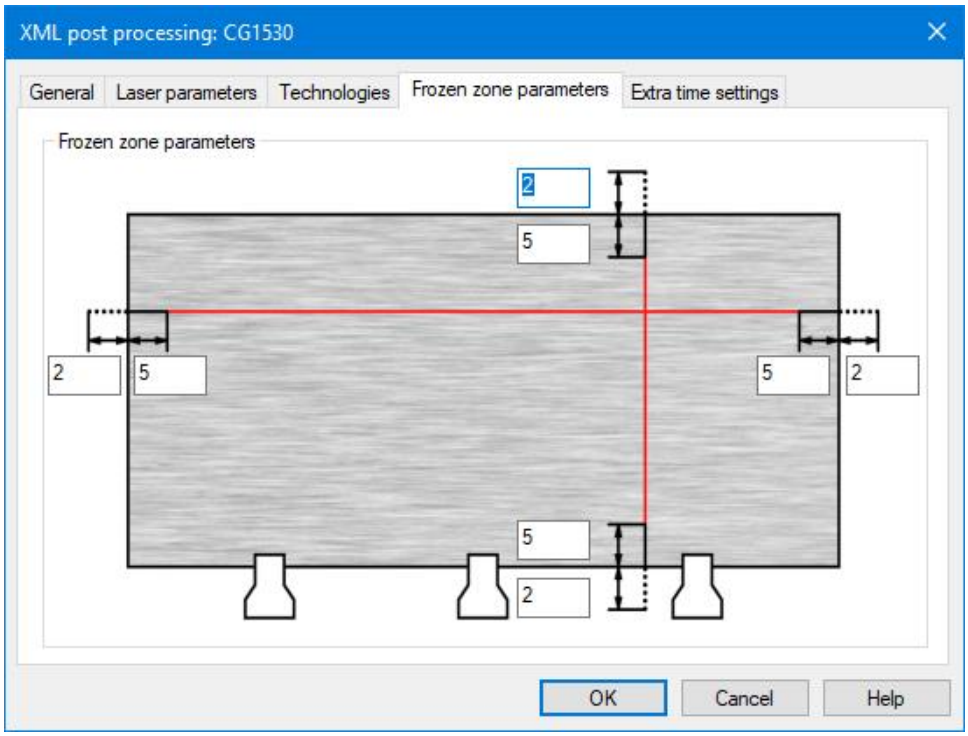
Below is the Autotool result after activation. Now three piercings are produced for each internal hole destruct. In this case, that equates to six piercings fewer for each part, thus saving production time. This is especially beneficial when cutting thicker materials for which piercing time is longer.



Over travel and crop margin for all sheet sides separately

There is a new tab for ***Frozen zone parameters*** in the post-processing dialogue. Now it is possible to set the crop margin and over travel separately for each side of the sheet.

This helps to set smaller over travel to the sheet sides, which are accurate in the specific machine. Typically, the loading pin side and clamp side are “accurate”, meaning smaller values can be used than on inaccurate sides (sheet size tolerance).



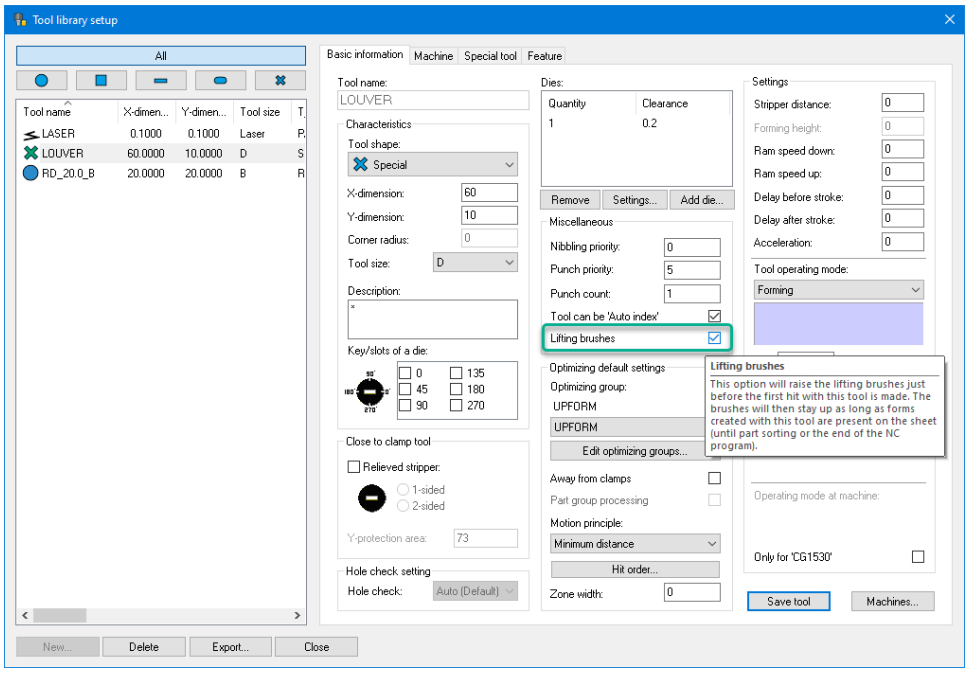
The PUNCH

Lifting brushes activation for specific tool in tool library

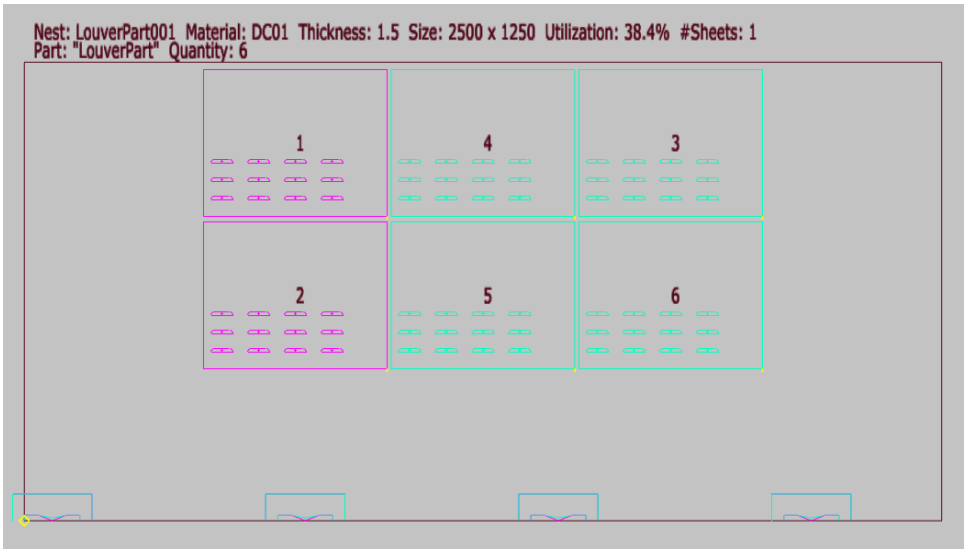
Scratch-free production with sheet-floating system SFS

A new Genius Series option is to have vertically moving brushes also in front and inside of the turret, which effectively prevents scratching of sensitive materials. The movement is activated by the program when needed.

The optimiser reads the lifting brushes state from the tool library and sets the tool control mechanism internally. Brushes are lifted just before the first hit or forming with this tool. The brushes will then stay up if forms created with this tool are present on the sheet until part sorting or the end of the NC program.



Nest and NC-code sample:



LIFTING_BRUSHES_UP command is given before the first hit with the LOUVER tool.

```
<Program PPVersion="23.2" PPDate="Dec 1 2023" Programmer="NC Man">
  <Main>
    <SETUP/>
    <SHEET_LIFT/>
    <TOOL_START Number="10" X="538.5" Y="979.8" ToolName="LOUVER"/>
    <Forming_height UpValue="10" UpIncr="0" DownValue="0"
                    DownIncr="0"/>
    <LIFTING_BRUSHES_UP/>
    <PUNCH_ON/>
    <MOVE FlowID="1" X="538.5" Y="979.8" C="0"/>
```

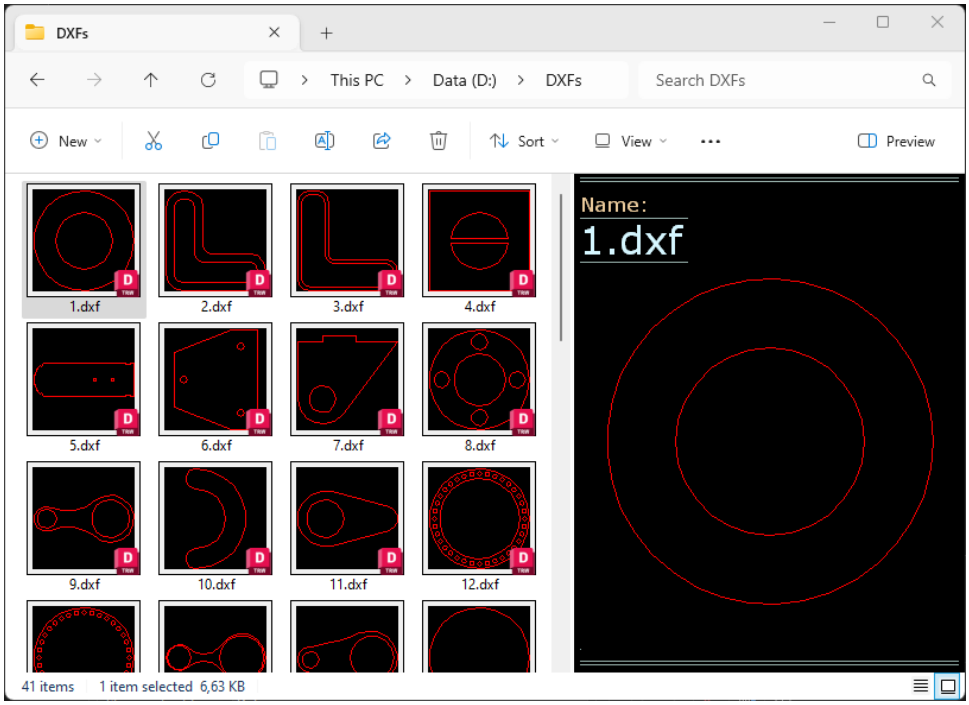
LIFTING_BRUSHES_DOWN command is given after all the parts that have hits made with the LOUVER tool are sorted or unloaded.

```
  <STOP_DPM_PROCESS NextMainPointProcessTechnology="1"
                    NextMainTrajectoryProcessTechnology="1"
                    NextPrePointProcessTechnology="2"
                    NextPreTrajectoryProcessTechnology="2"
                    ExtraHeight="1"/>
  <INCREMENTAL_PROGRAMMING/>
  <SORTING FlowID="0" LastPart="1"/>
  <LIFTING_BRUSHES_DOWN/>
  <SUB_END/>
  <SUB_PROGRAM_END/>
</SUB_PART>
```

Other usability enhancements

DXF/DWG file preview

The preview handler in NC Express e³ has been extended to also preview .dxf and .dwg file formats in the Windows operating system File Explorer. Thumbnail creation for the file formats is also available. In the image below, an illustration has been given in which .dxf file thumbnails are presented on the left-hand side of the File Explorer. One file is selected, namely 1.dxf, and it is previewed on the right-hand side.

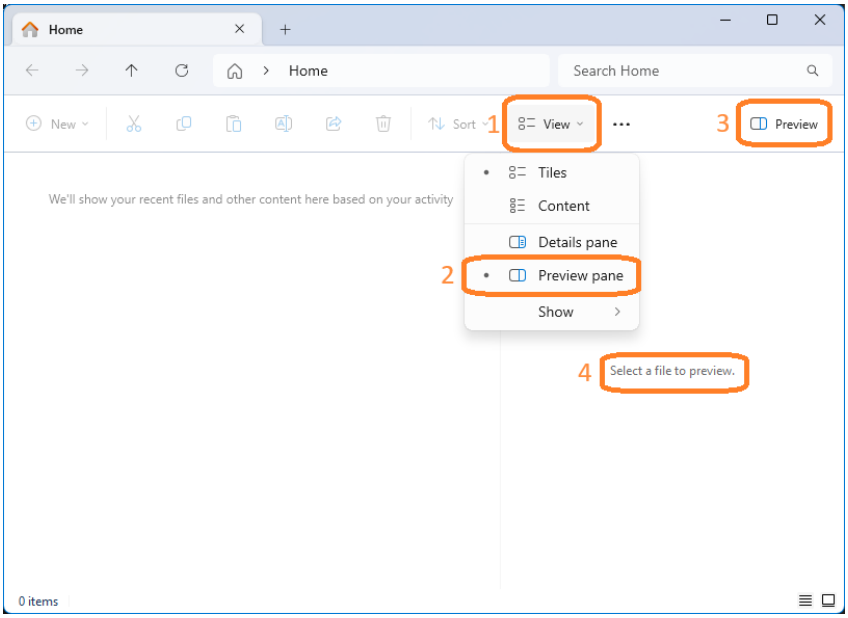


An illustration of .dxf thumbnails and preview.

To activate the new preview and thumbnail Windows functionality for these file formats, NC Express e³ 23.2 must be installed. The following simple steps must then be taken to make File Explorer show previews and create thumbnails:

Preview (Windows 11 and File Explorer):

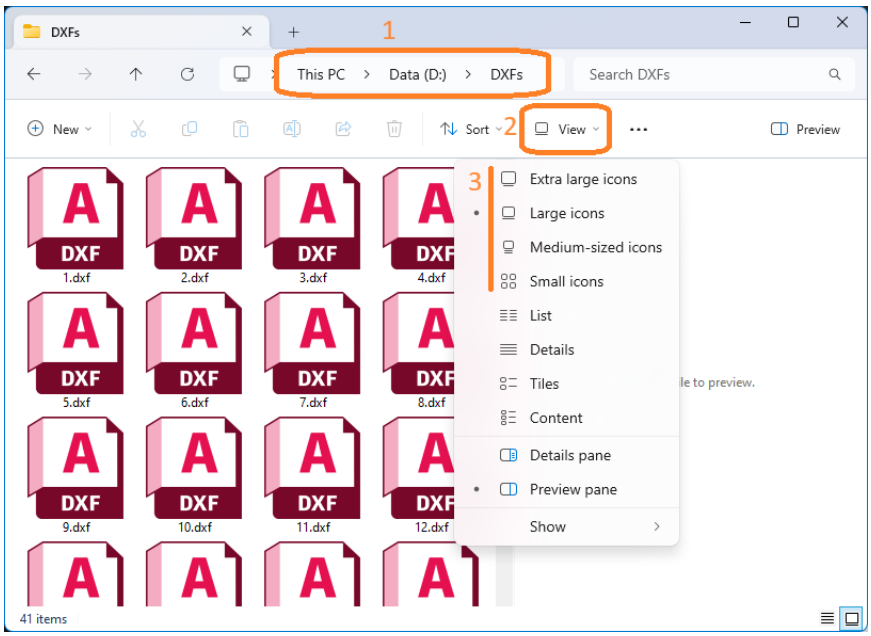
1. Click the “View” drop-down menu button in File Explorer.
2. In the drop-down list, choose the “Preview pane” option.
3. Activate the actual preview in File Explorer.
4. The preview pane shows if it was not visible before (press the button again if the preview pane has disappeared). After selecting a file from the file listing on the left-hand side of File Explorer, a preview will show in the pane.



How to activate preview pane in Windows.

Thumbnails (Windows 11 and File Explorer):

1. Navigate to a folder containing .dxf or .dwg files.
2. Click the “View” drop-down menu button in File Explorer.
3. Choose from the available display options for icons, e.g. “Large icons”.
4. Thumbnail images are created for files for which thumbnails are missing.



How to activate thumbnails.

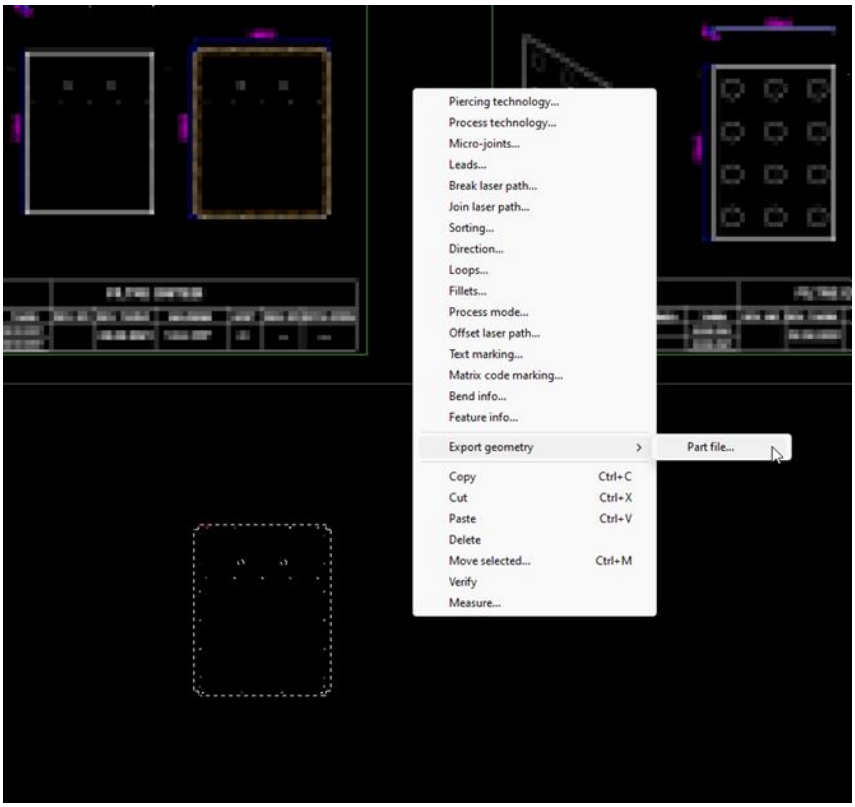
In case the user is using the 32-bit version of NC Express e³ 23.2 with a 64-bit Windows operating system, File Explorer must be started in 32-bit mode. Otherwise, the thumbnails will not be created automatically. To open File Explorer

correctly in this case, File Explorer should be opened for example via NC Express and the "***Open part***" dialogue. Note, already created thumbnails are shown automatically.

Export part from multipart drawing

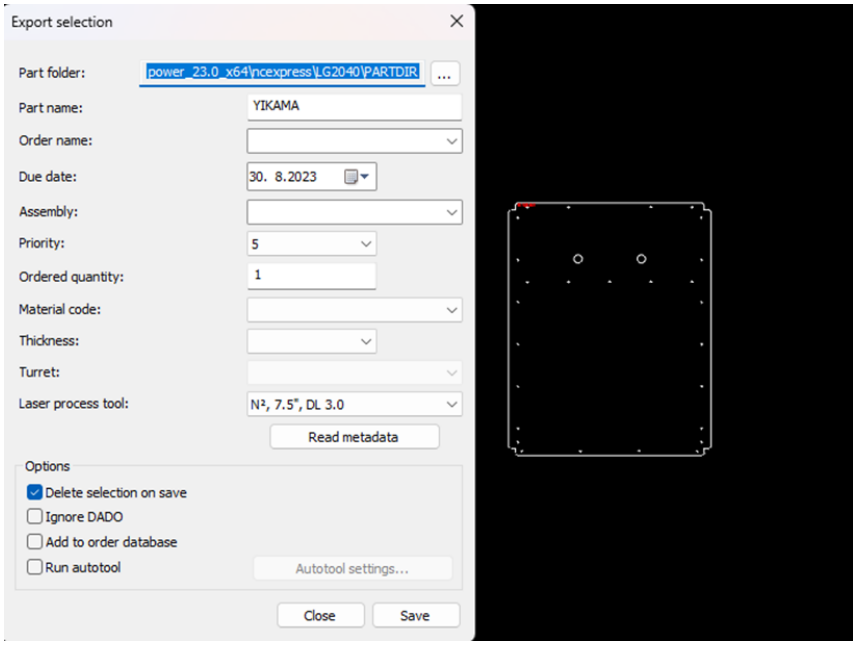
It is now possible to import multipart DXF/DWG files, select parts individually, export geometries and perform set functions with each part, for example add it to an order database or Autotool it.

- A new dialogue "***Export part***" can be opened by right-clicking the sub-menu ***Export geometry -> Part file***.
- After clicking ***Part file***, select the geometry to be exported.

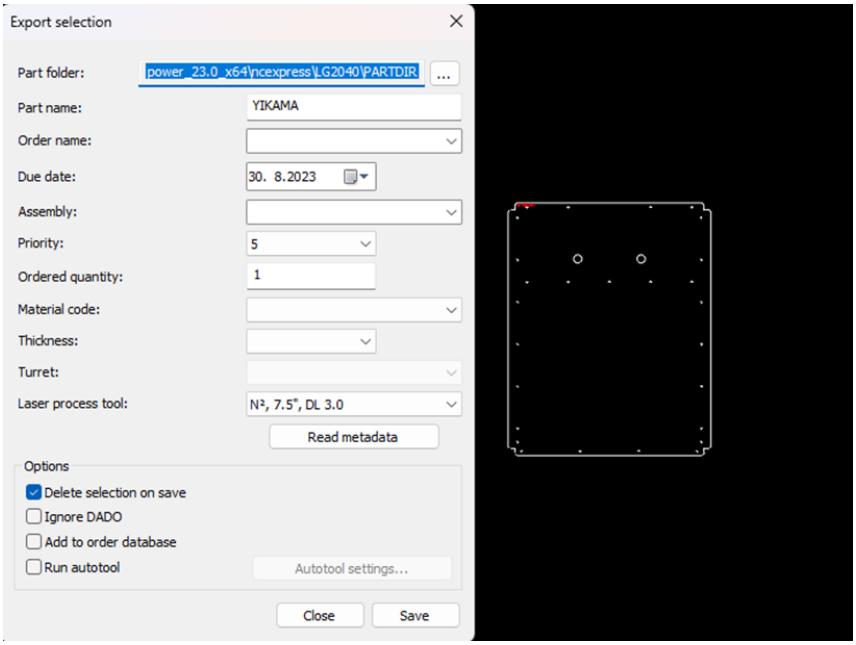


Clicking ***Part file*** will open an ***Export selection*** window from which it is possible to set the desired properties before exporting the geometry.

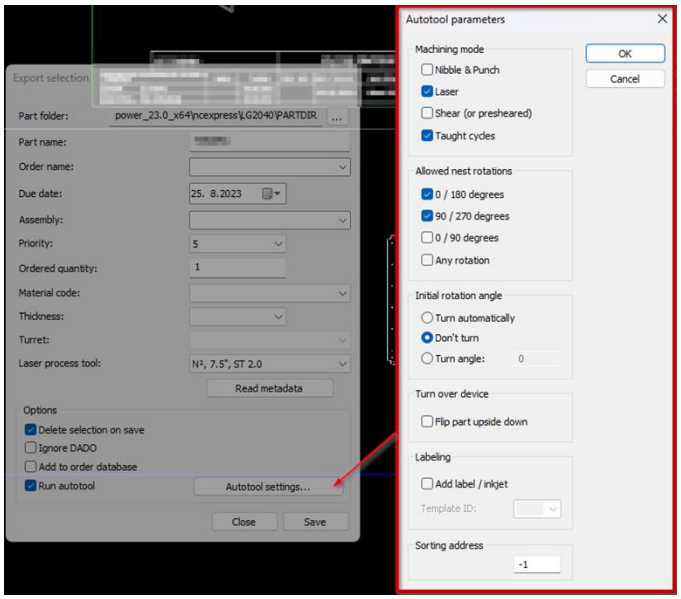
- The part name stays the same except automatic suffix stepping (user can modify the part name)
- Other field values stay the same unless changed by the user
- User can set the sub-folder to which the part is saved



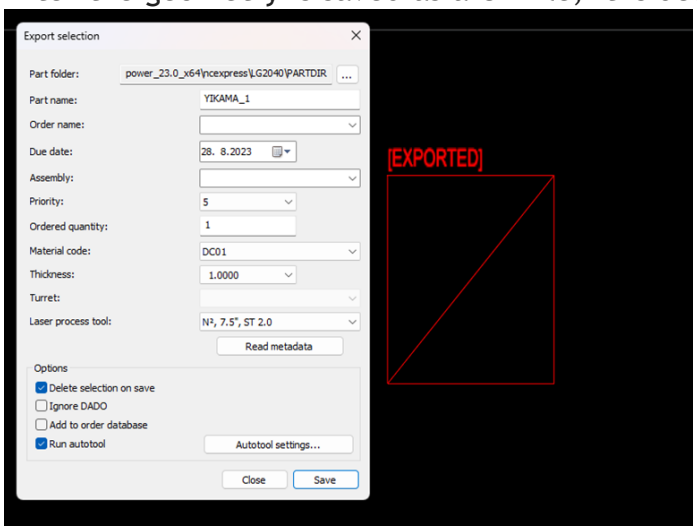
* Note that there are steps to follow before reading metadata will work. (Instructions on this topic can be found from Prima Power Connect -> News -> Reading part metadata from a DXF drawing.)



It is possible to set basic Autotool parameters from ***Autotool settings***. Here the most-used functions can be found, for example the machine mode and allowed nest rotations.

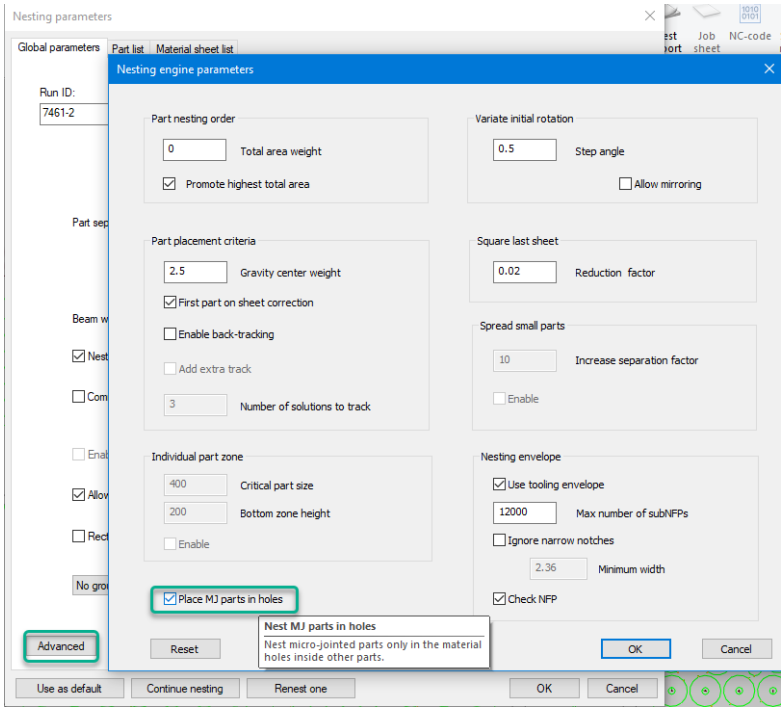


After the geometry is saved as a CP file, it is deleted from the original drawing.

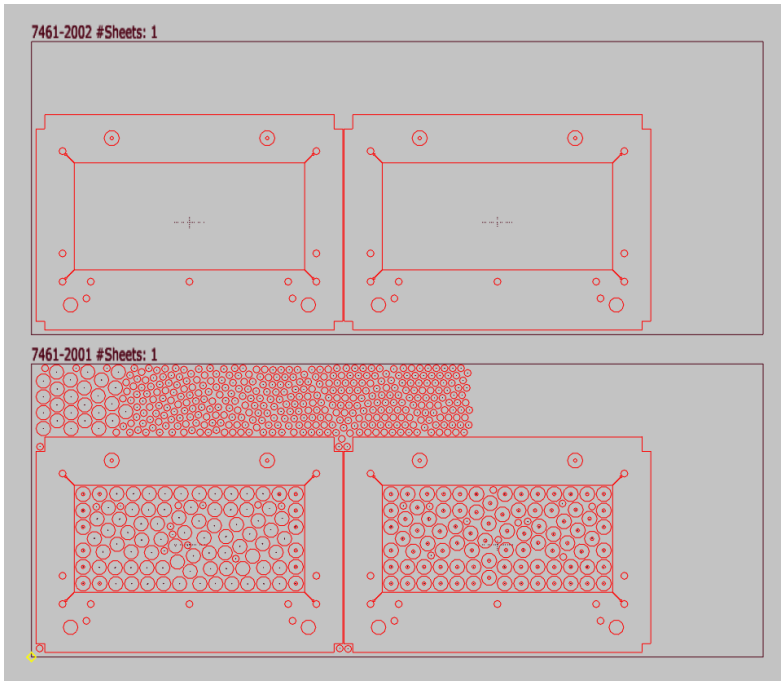


Prefer nest micro-jointed parts in holes

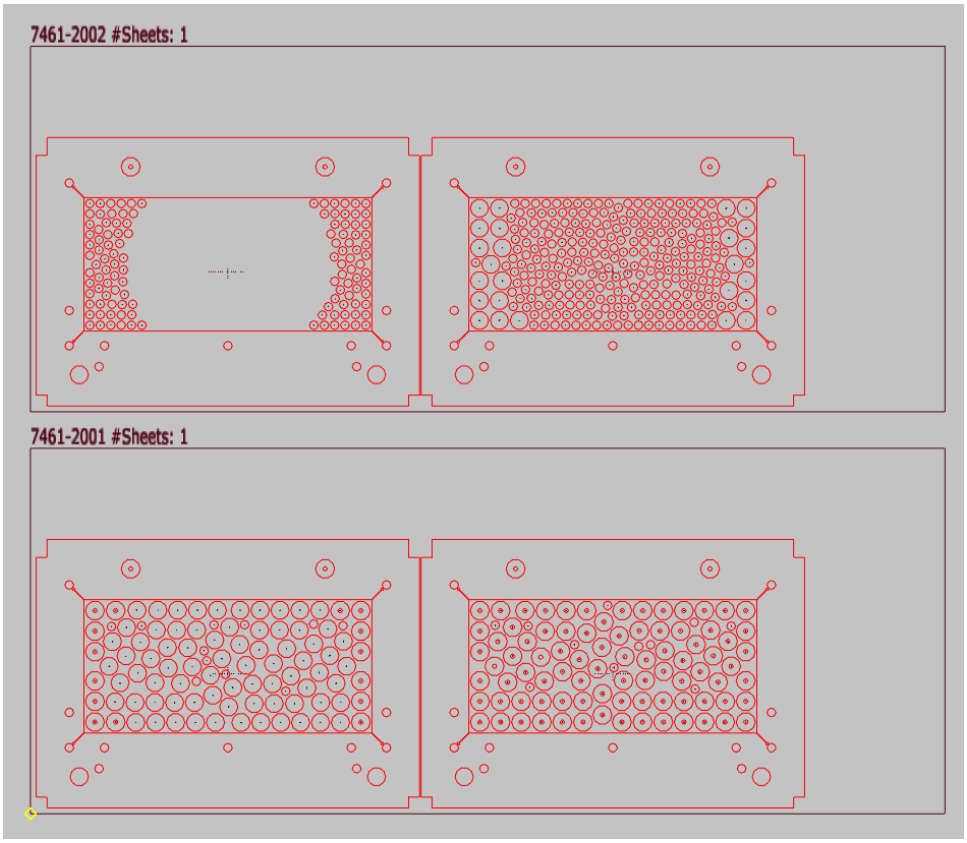
Activation of the "***Place MJ parts in holes***" functionality is done in ***Advanced*** nesting parameters of the High Performance free-form nester. To enter Advanced nesting parameters, you must set the administrator user to the environment settings. When "***Place MJ parts in holes***" is active, the micro-jointed parts are nested inside the holes of other parts.



Nest run when "***Place MJ parts in holes***" is OFF:



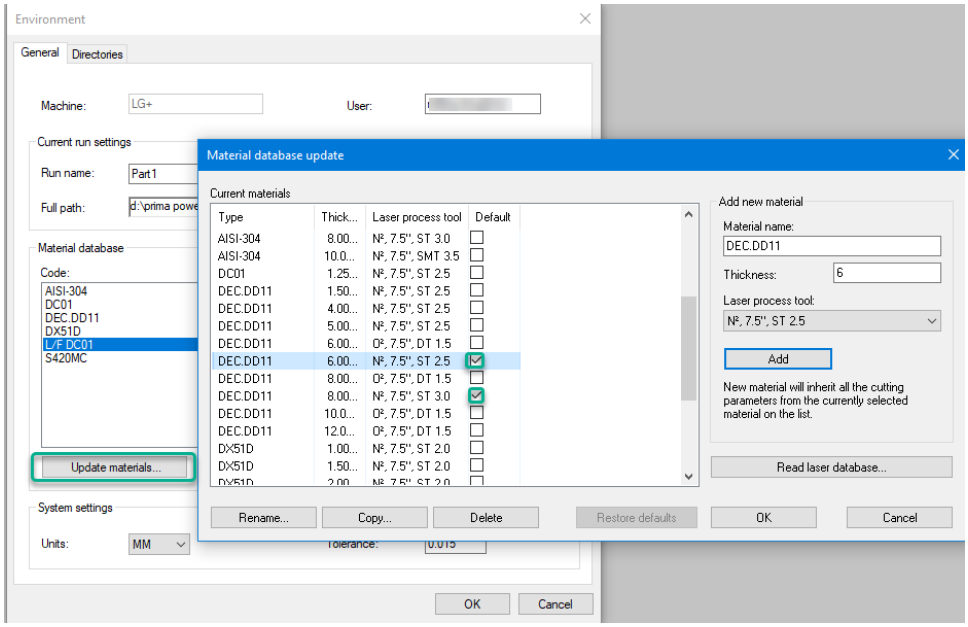
Nest run when “***Place MJ parts in holes***” is ON. All small micro-jointed parts are nested inside holes. This can help to create more of the same shaped remnants, which are easier to reuse in production.



Default laser tool selection

Users can now choose a default laser tool for each material and thickness, in the ***Update materials*** dialogue.

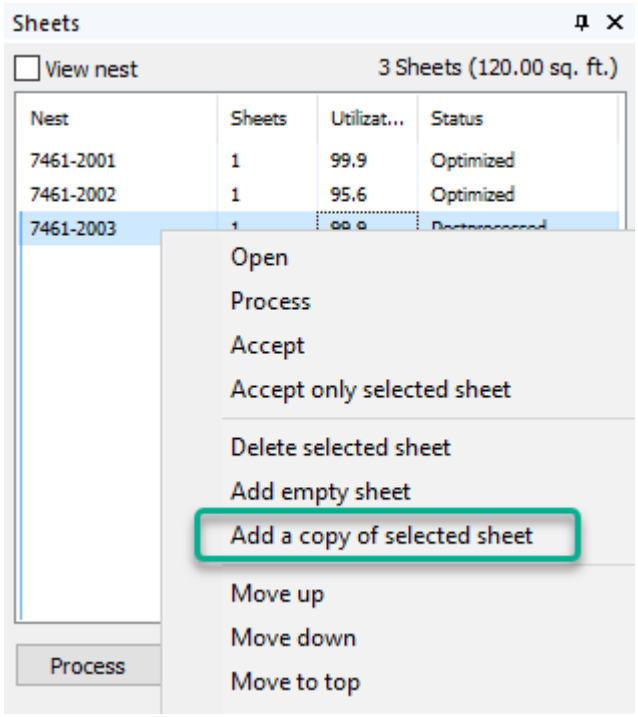
This is especially useful when there are many laser tools (“triplets”) available on a single material and thickness, and ***Database -> Orders*** is used. The order file specifies material and thickness, but does not specify a laser tool. The default laser tool is now automatically the same for all users. This makes automatic order processing possible.



Copy-optimised sheet

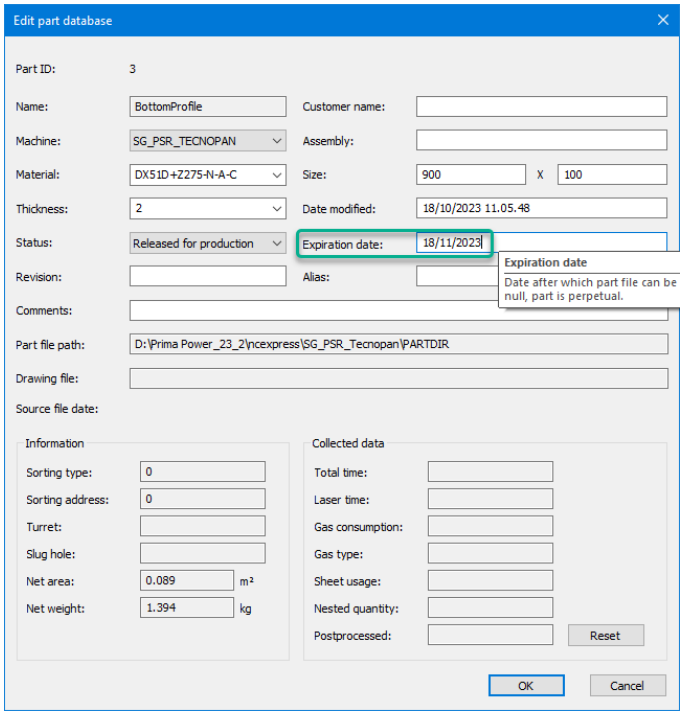
We have changed the behaviour of Copy selected sheets so that it also preserves the optimised sheet (.opm). Previously, only the nested sheet (.nst) was copied and the copied sheet had to be re-optimised.

If the selected sheet is in “postprocessed” or “accepted”, it will be rolled back to “optimised”. The sheet will not be post-processed again with the new name.



Expiration date

Based on the "***Expiration date***" it is possible to automatically delete old parts. This is especially needed when automatic production is done with Tulus Office or customised automatic production run with the ParaNCX interface.

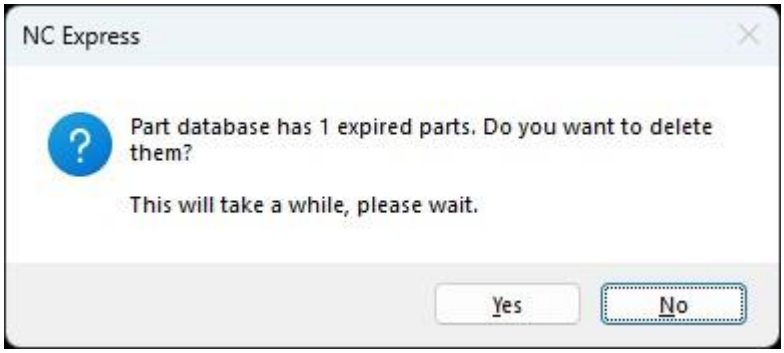


The only way to set this is through the PartNCX property:

`DATE ExpirationDate`
Date after which part file can be deleted. When null, part is perpetual.

The default value of ExpirationDate is null, meaning the part is perpetual (normal, not to be deleted).

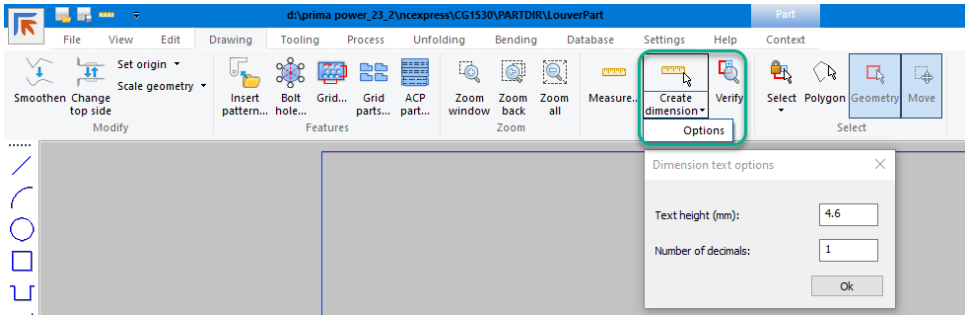
The house-cleaning routine is run when NC Express is started. The following message is shown when expired parts are found in the database:



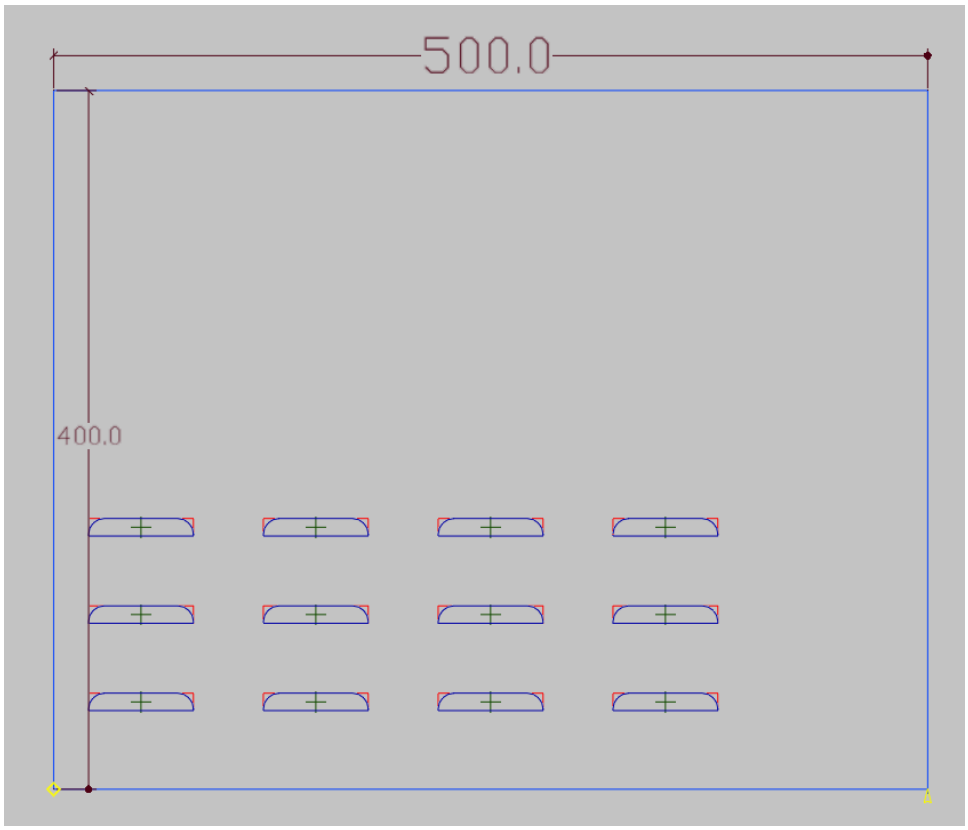
Deleted parts are moved to the recycle bin if the user selects ***Yes***.

Improved dimension line visibility

With ***Dimension text options*** you can set the font size and number of decimals for dimension creation. Font size is also preserved when you print the PDF from the part.



In this sample part, the height is dimensioned with a 10 mm font and the width with a 20 mm font:



Windows support

NC Express e³ 23.2 supports Windows 7 and Server 2012 to 2022 up to the latest Windows 10 and Windows 11 versions.

This version is also available as a 64-bit build. New installations are recommended to be made in 64-bit, whereas updates remain in 32-bit.

If you update an existing installation to 64-bit and it uses customised report templates, please be prepared to redo those report templates for new reporting.