

NC Express

Software version release: **26.1**

25th June 2026

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UNFOLDING

Supported 3D formats and versions

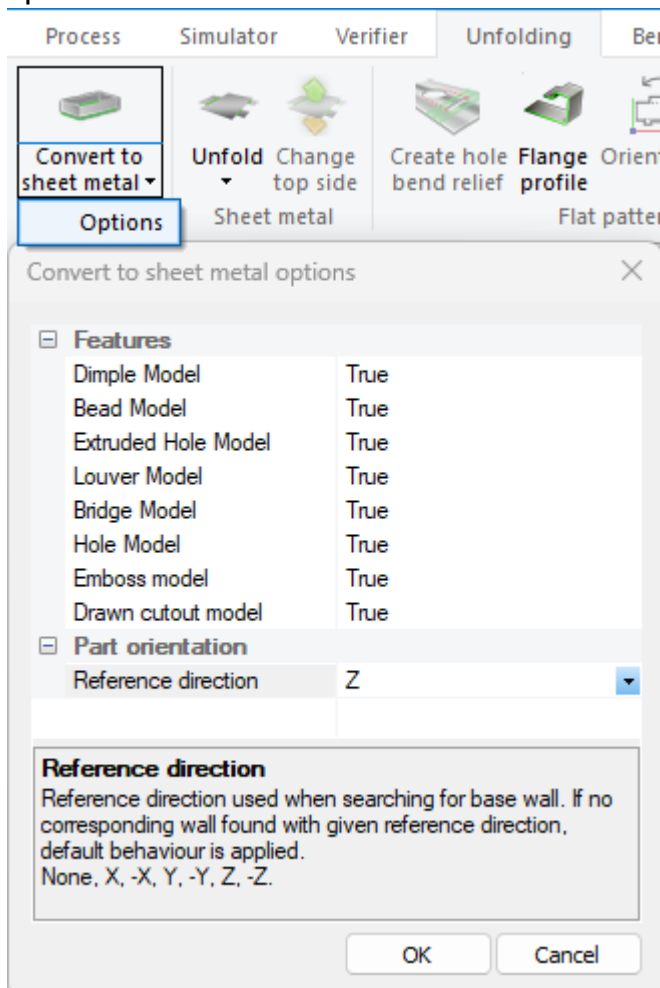


- Autodesk Inventor (*.ipt, *.iam), up to 2026
- SolidWorks (*.sldprt, *.sldasm), up to 2026
- Solid Edge (*.par, *.psm, *.asm), up to 2026
- Siemens JT (*.jt), up to 10.10
- Siemens NX (*.prt), up to 2512.1700
- PTC Creo (*.prt.x, *.asm.x), up to 12.0
- Catia (*.catproduct, *.catpart), V4 (4.15 to 4.26) and V5-3DX (R10 to R35, R2023 to R2025), V6 (R2026)
- IGES (*.igs, *.iges), up to 5.3
- STEP (*.stp, *.step), AP203 (E1, E2), AP214 (up to E3), AP242 (E1, E2, E3, E4, BO XML), AP209
- Spatial ACIS (*.sat), up to 2023 1.0
- Parasolid (*.x_t, *.x_b), up to 38.0

Option for defining desired reference direction

It is now possible to set a desired reference direction in Convert to sheet metal options. This can be useful in cases where the standard behaviour will give inconsistent results for different variations/dimensions of the same product. For example, if the ratio of different walls is changing, making the NC Express standard behaviour to change the reference selection, or if there are some other special criteria different than the standard behaviour.

This option can be set to apply globally for all parts in Convert to sheet metal options:

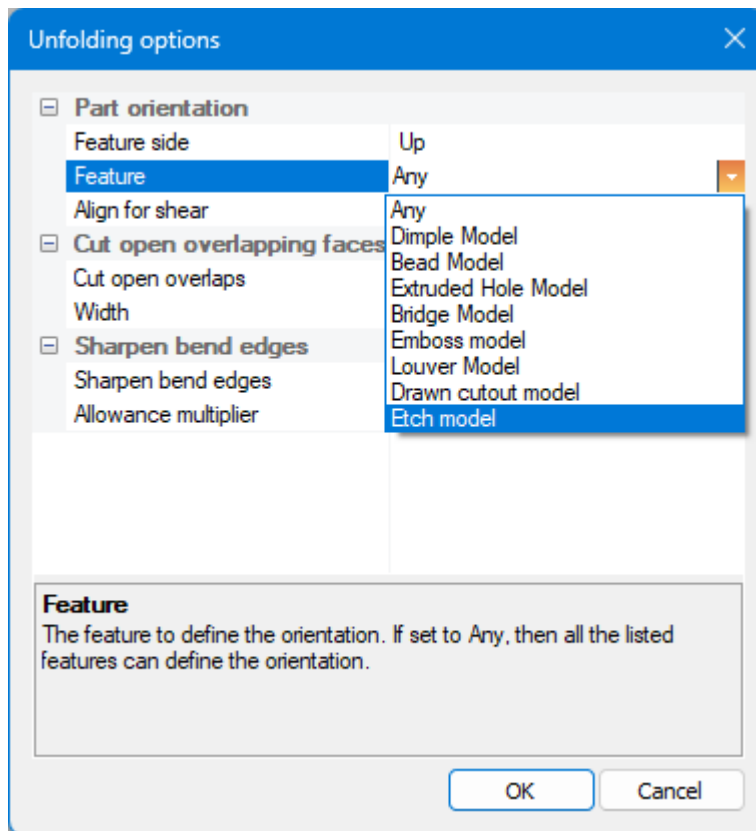


Default: None, having the standard behaviour.

In addition, it can be overridden with Property mapping.

Option for defining blanking side by Etch feature

A new value ***Etch model*** has been added to the ***Feature*** selection of ***Part orientation***:



With this, it is possible to automatically change the top side of the part to align Etch on top or bottom, based on the ***Feature side*** selection.

PANEL BENDER

BeltPhase: Possibility for setting default BufferingMode

BufferingMode is now available under both Default program parameters and Program parameters.

Default program parameters

ASP simulacrum

Preferred edge safety thickness mult1.2

☒ Flange parametric adjust trajectory

Validation version1

Maximum length without blade [mm]5

Center frequency0

☒ Default use pins

Center reliable CMove [mm]0

Center reliable XMove [mm]0

Default negative LBN mode

☒ Mode 0☐ Mode 1☐ Mode 2☐ Mode 3

☐ Angle measure move blade ref. used

☐ Default automatic angle correction

API ROI Width Theta [deg]12

☐ Clamp small part center☐ Helper cycle enabled

Helper cycle repetition factor [mm]400

Bending Length To Add Center [mm]0

Additional centeringNone

Shift calculate strategy

☒ Bend length☐ Press length

BufferingNot buffering

Max Lenght Force SSPL Standard [mm]Not buffering

Autobend preferences

Preferred cycle

☒ Short side☐ Long side

Slot_EBe2720_LeftLoading_v3

manipulator acceleration [%]100

DNP speed [%]100

DNP acceleration [%]100

☐ Angle measure move blade ref. used

☐ Default automatic angle correction

API ROI Width Theta [deg]12

ASP simulacrum

Preferred edge safety thickness mul1.2

☒ Flange parametric adjust trajectory

Validation version1

Maximum length without blade [mm]5

Center frequency0

☐ Clamp small part center

☐ Helper cycle enabled

Helper cycle repetition factor [mm]400

Bending Length To Add Center [mm]0

Additional centeringNone

Shift calculate strategy

☒ Bend length☐ Press length

BufferingNot buffering

Max Lenght Force SSPL Standard [mm]Not buffering

Autobend preferences

Preferred cycle

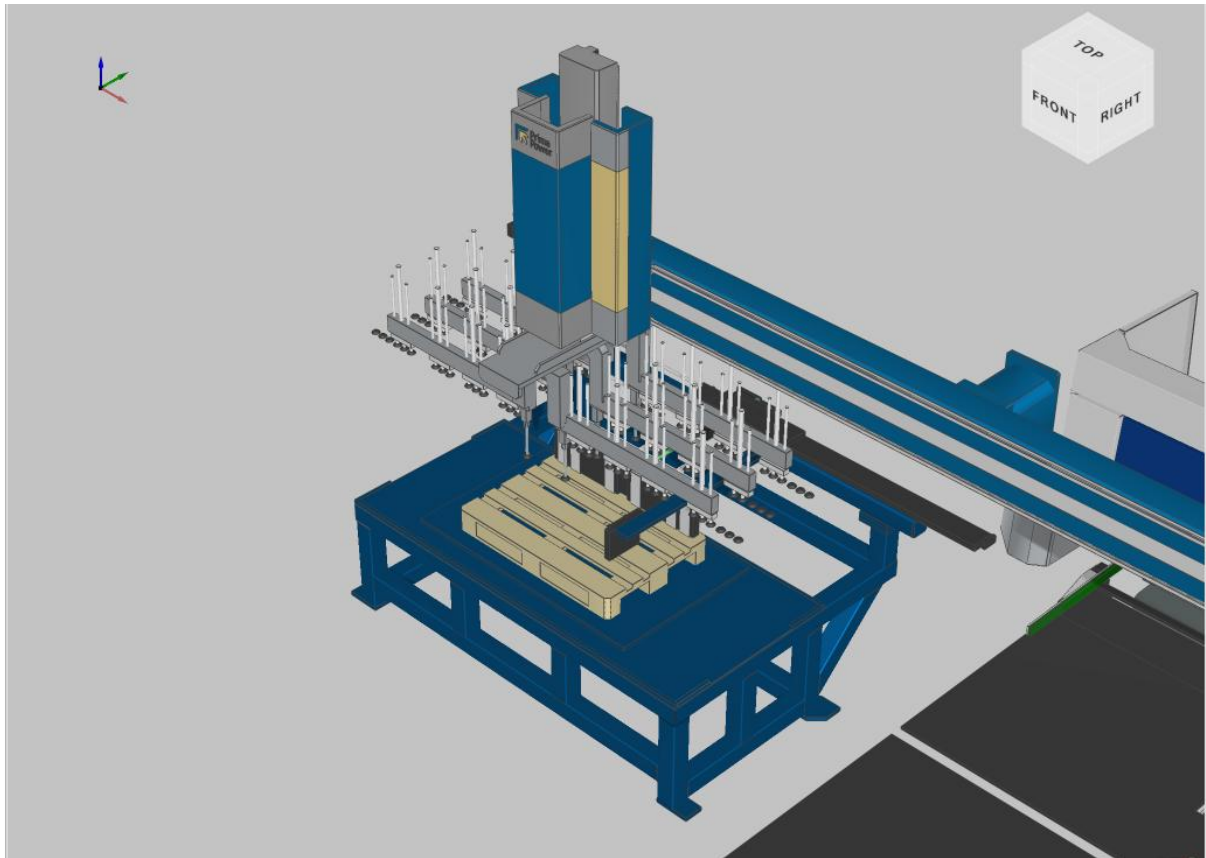
☒ Short side☐ Long side

Barcode

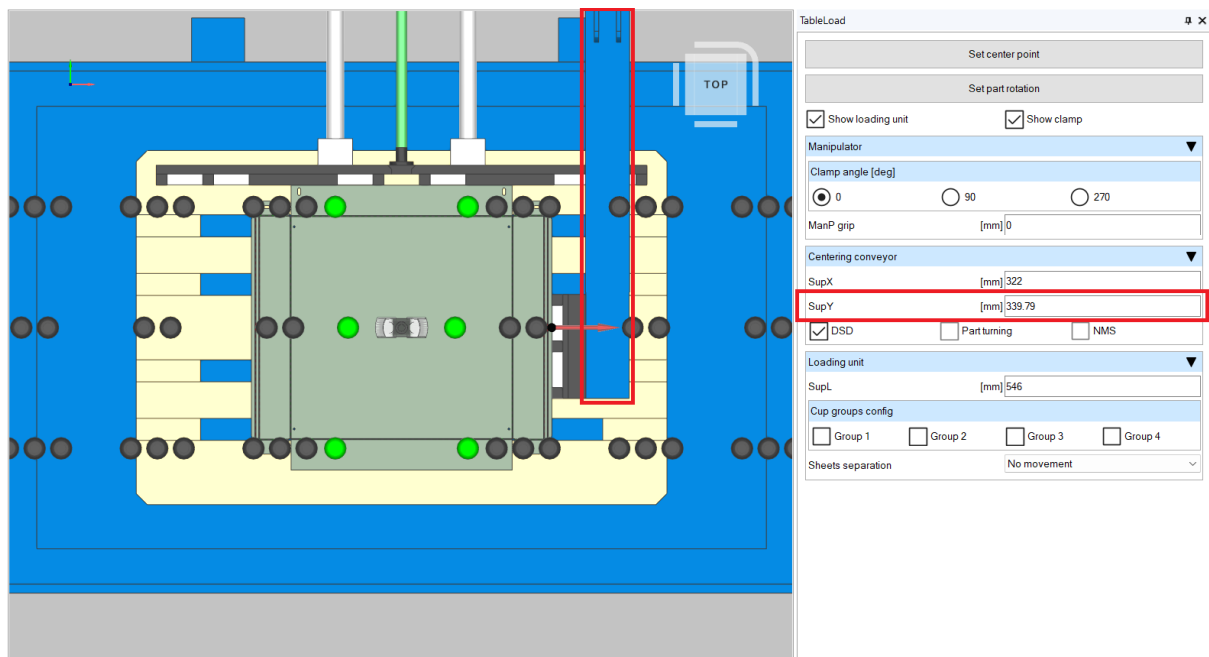
PropertiesSlot_EBe2720_LeftLoading_v3

Support for LoadingTable device

LoadingTable device is now supported.



In LoadingTable device, changing the SupY setting results in moving the side bar against the part (instead of Loading Unit).



Enhancements on USS unloading

It is now possible to set Default unloading to be activated for the exported program. Previously, the operator needed to change the selection to USS at the machine HMI, but now this can be defined already in NC Express.

Slot_EBe2720_LeftLoading_v1

ASP simulacrum	
Preferred edge safety thickness multiplier	1.2
☒ Flange parametric adjust trajectory	
Validation version	1
Maximum length without blade [mm]	5
Center frequency	0
☐ Clamp small part center	☐ Helper cycle enabled
Helper cycle repetition factor [mm]	400
Bending Length To Add Center [mm]	0
Additional centering	None
Shift calculate strategy	
☒ Bend length	☐ Press length
Default unloading	
☒ BeltUnload	☐ USSPicking
Max Lenght Force SSPL Standard [mm]	0
Autobend preferences ▼	
Preffered cycle	
☒ Short side	☐ Long side
Barcode	

Properties Slot_EBe2720_LeftLoading_v1

This can be also defined in Property mapping.

Support for DNPPosCorrection

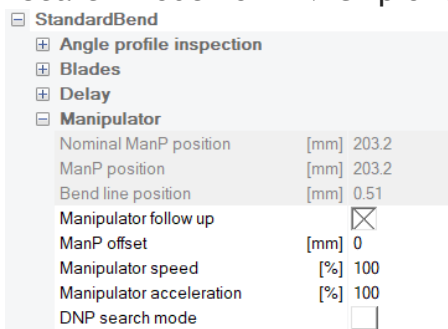
In some specific conditions, a part is not reached by the suction cups during the DNPgrip phase, which is caused by real-time offset, corrections or bend status.

```
<Phase Type="DNPgrip" FlowID="8">
  <Parameters>
    <Parameter Name="UpperToolIn" Type="Length" Value="9" NominalValue="9" />
    <Parameter Name="UpperToolOut" Type="Length" Value="3" NominalValue="3" />
    <Parameter Name="Force" Type="Force" Value="20" NominalValue="20" />
    <Parameter Name="ManpPos" Type="Length" Value="252.91" NominalValue="252.91" />
    <Parameter Name="ManpRepos" Type="Length" Value="0" NominalValue="0" />
    <Parameter Name="DNPgripMode" Type="Number" Value="0" NominalValue="0" />
    <Parameter Name="DNPHeightMode" Type="Number" Value="1" NominalValue="1" />
    <Parameter Name="ManpClamp" Type="Length" Value="9" NominalValue="9" />
    <Parameter Name="DNPPos" Type="Length" Value="301" NominalValue="301" />
    <Parameter Name="DNPPosOffsetR" Type="Length" Value="0" NominalValue="0" />
    <Parameter Name="KeepWedgeDown" Type="Boolean" Value="false" NominalValue="false" />
    <Parameter Name="MoveManp" Type="Boolean" Value="true" NominalValue="true" />
    <Parameter Name="SearchMode" Type="Boolean" Value="false" NominalValue="false" />
    <Parameter Name="SearchTorque" Type="Number" Value="35" NominalValue="35" />
    <Parameter Name="SearchDelta" Type="Length" Value="5" NominalValue="5" />
    <Parameter Name="SyncWithPrevCenter" Type="Boolean" Value="false" NominalValue="false" />
    <Parameter Name="DNPPosCorrection" Type="Length" Value="0" NominalValue="0" />
    <Parameter Name="HMI_TMDB_Interaction" Type="Boolean" Value="true" NominalValue="true" />
    <Parameter Name="Phase ID" Type="String" Value="5" NominalValue="5" />
  </Parameters>
</Phase>
```

With this new parameter introduced in the exported pfp, the user can edit this offset from the Tulus interface upon measuring the empirical distance between the cups and the flange, allowing DNP to grip the part.

Support for DNPSearchMode on StandardBend & DNPgrip

With DNP on machine series version ≥ 4.33 , the operator can enable the option “Search mode” on DNPgrip or StandardBend phases (if DNP is holding the part).



StandardBend		
+ Angle profile inspection		
+ Blades		
+ Delay		
- Manipulator		
Nominal ManP position	[mm]	203.2
ManP position	[mm]	203.2
Bend line position	[mm]	0.51
Manipulator follow up		☒
ManP offset	[mm]	0
Manipulator speed	[%]	100
Manipulator acceleration	[%]	100
DNP search mode		☐

In addition, two new parameters, “SearchTorque” and “SearchDelta”, will be present on the exported pfps in the mentioned phases.

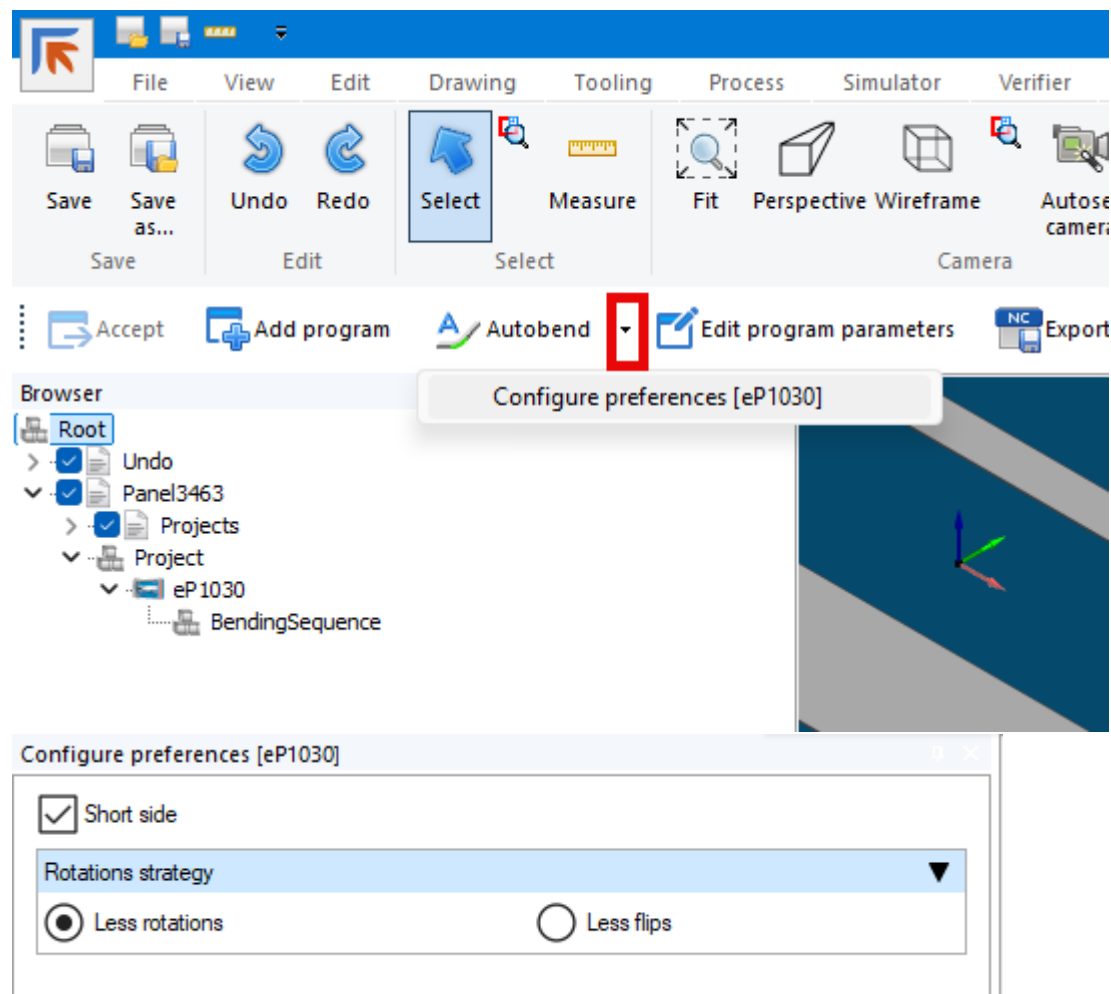
Upon activation, the searching mode for DNP will make it “search” the flange surface to make a firm grip on it. The search will end when the desired torque is reached, meaning the DPN cups are touching the metal sheet.

PRESS BRAKE

Enhanced Autobend

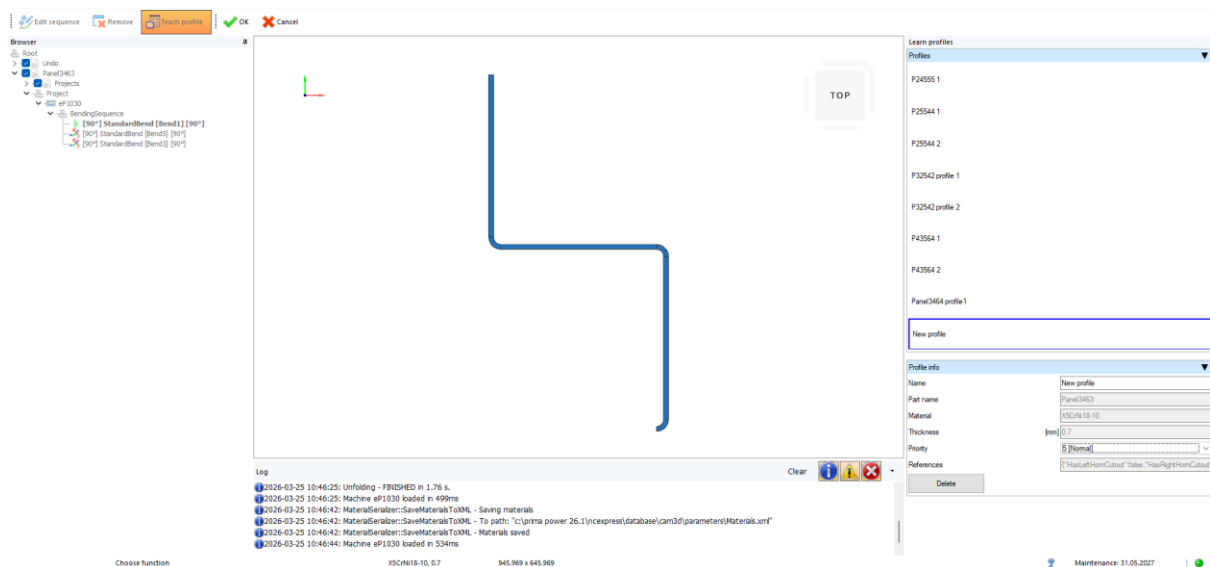
Autobend algorithms have numerous improvements. A few are listed below:

- Prioritizing more stable finger positioning when computing the sequence.
- Avoiding sequences that lead to inevitable collision with negative bends and the die tool.
- Possibility to set preferences. Current options are to prefer short sides first cycle and to prefer avoiding rotations or flips during the sequence.

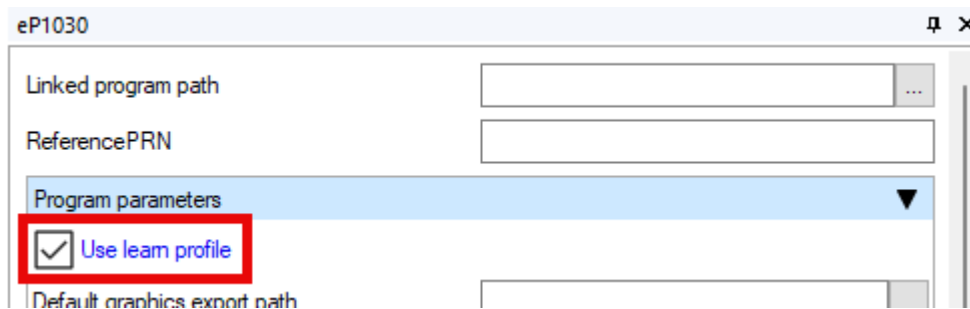


Support for Learn profile

Learn profile usage is now supported with Press Brake programming.

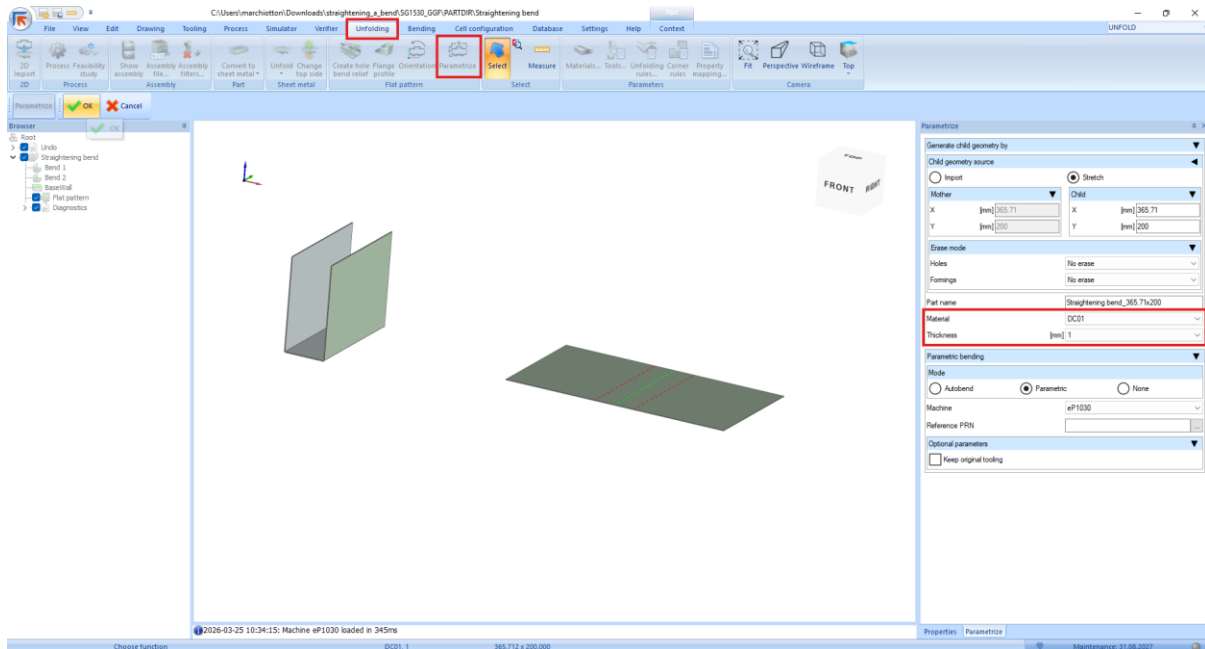


With Learn profile, it is possible to teach a manually programmed profile of the part, including the sequence, process parameters and tools. When Program parameter “Use learn profile” is enabled, stored profiles are applied on bends automatically on the Autobend and Edit sequence (while holding alt on selection) for similar profiles on different parts.



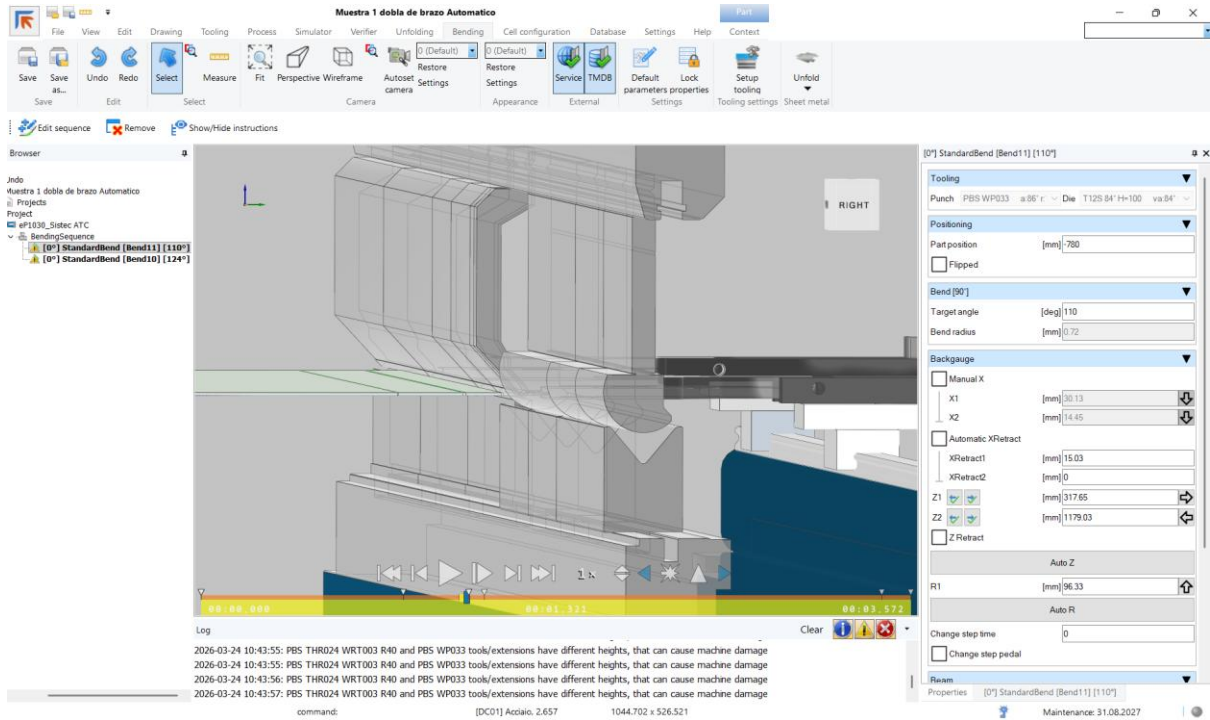
Material & Thickness porting

Users can now perform Material & Thickness porting on Brake programs. Starting from a mother program, this feature allows the user to change the material and thickness properties of the part and generate a child program in which all the programming settings of the mother program will be preserved.



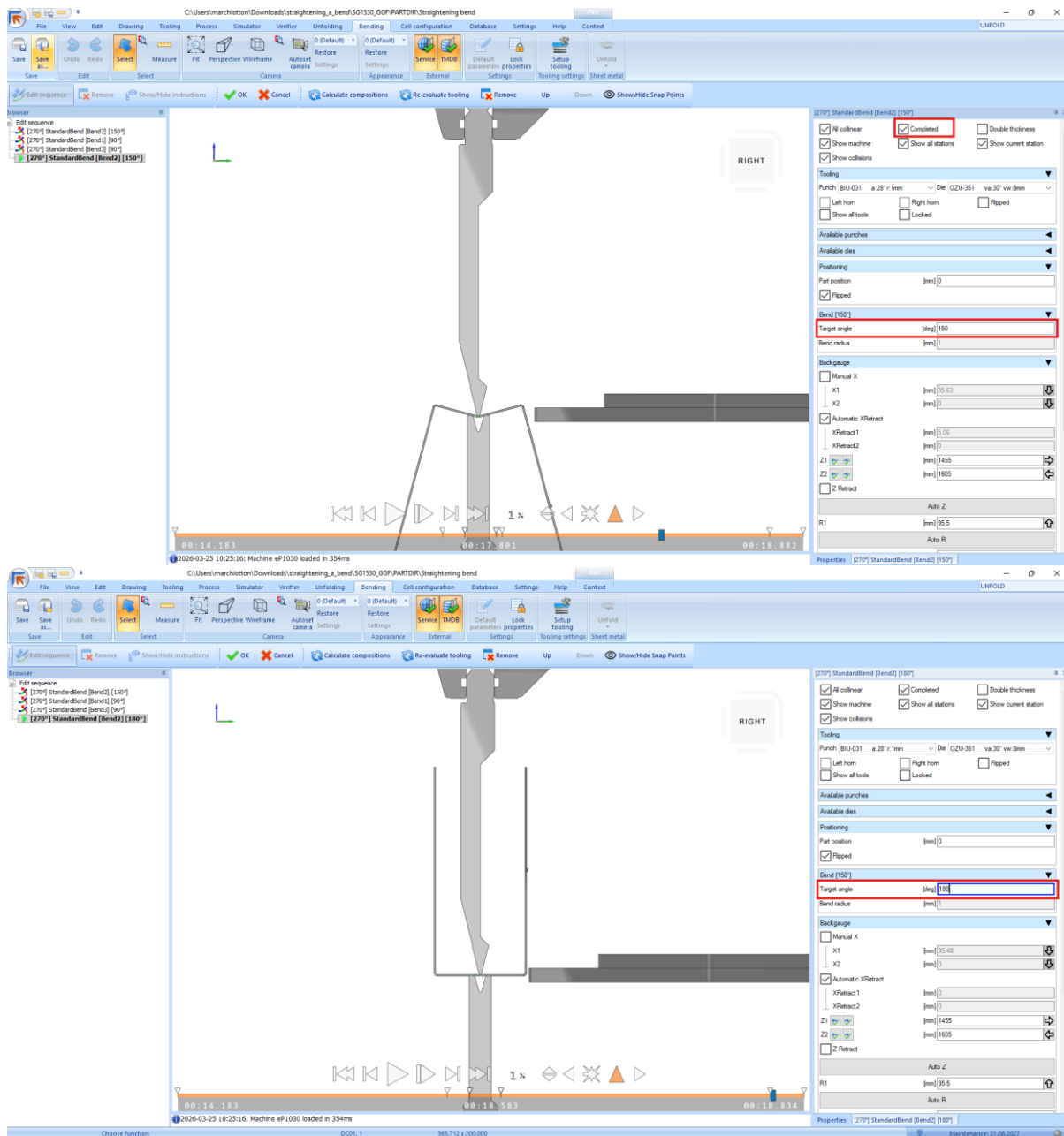
Allowing tools with different height

It is now possible to accept programs that use tools of different heights. NC Express will notify the user with a warning message when tools with different heights are used.



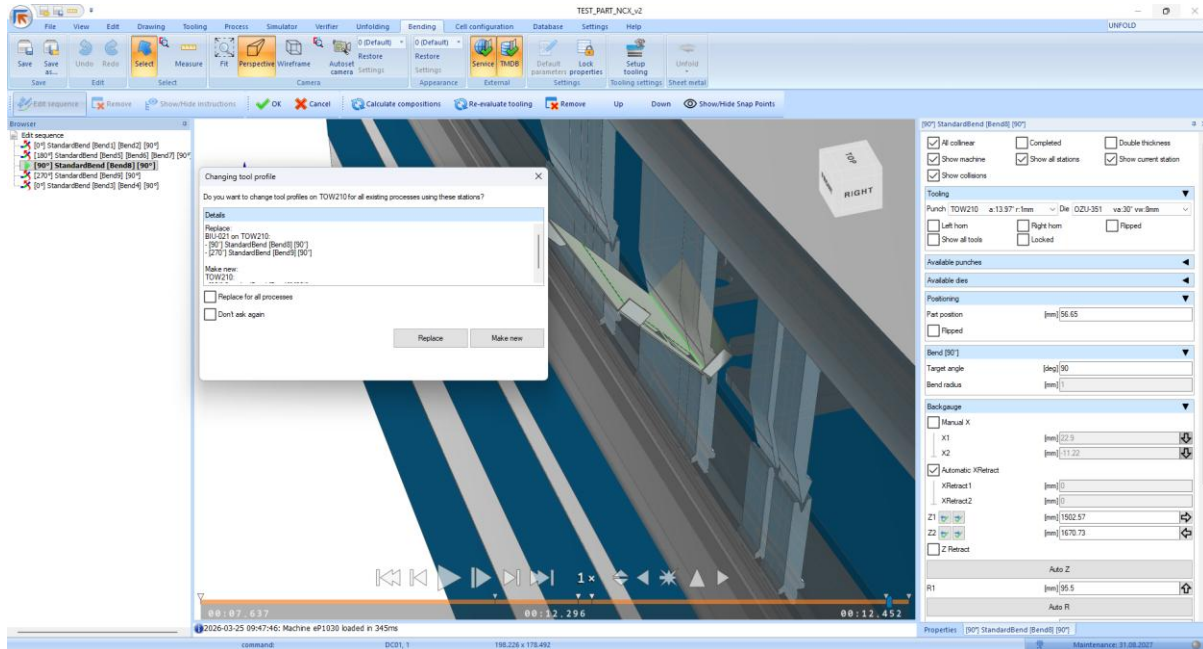
Straightening a bend

Users can now perform an additional bend process on an already completed bend to straighten it. To create the straightening process, the *Completed* option must be enabled, bend to straighten must be selected and the bend angle must be set to the desired end angle.



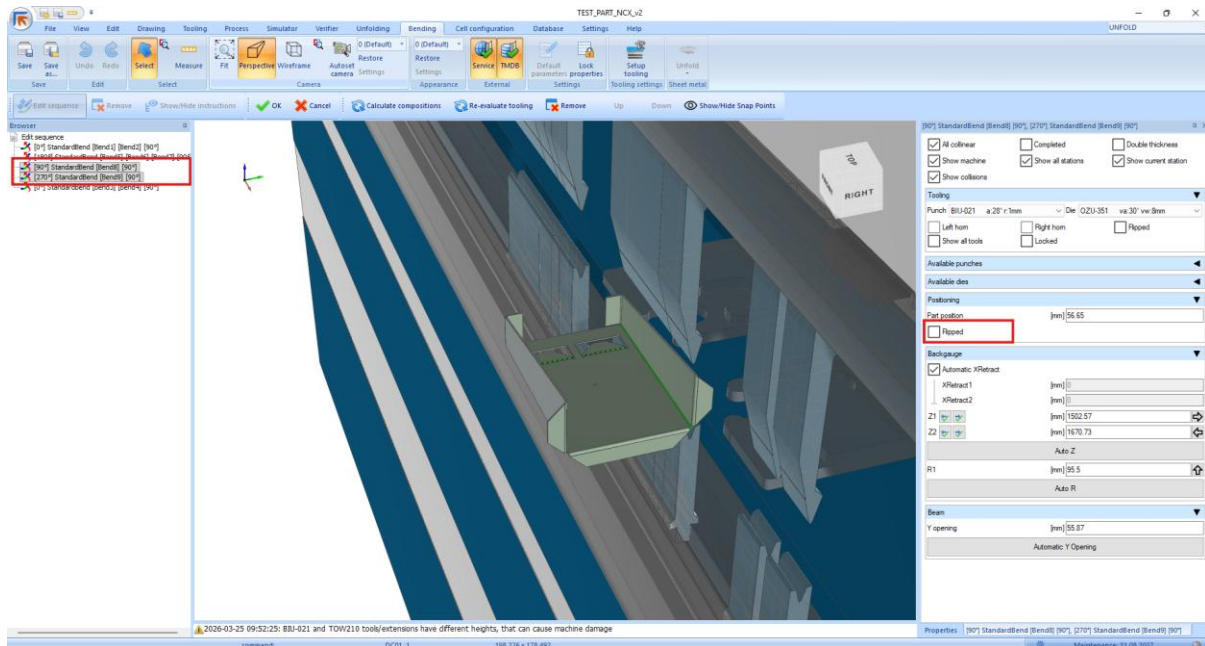
Possibility to change tool profile for all stations at once

Upon selecting a new tool to be used for a process, if the station is used also by other processes, NC Express will ask if a new station is to be created with the new set of tools or if all the processes are to use the new tools.



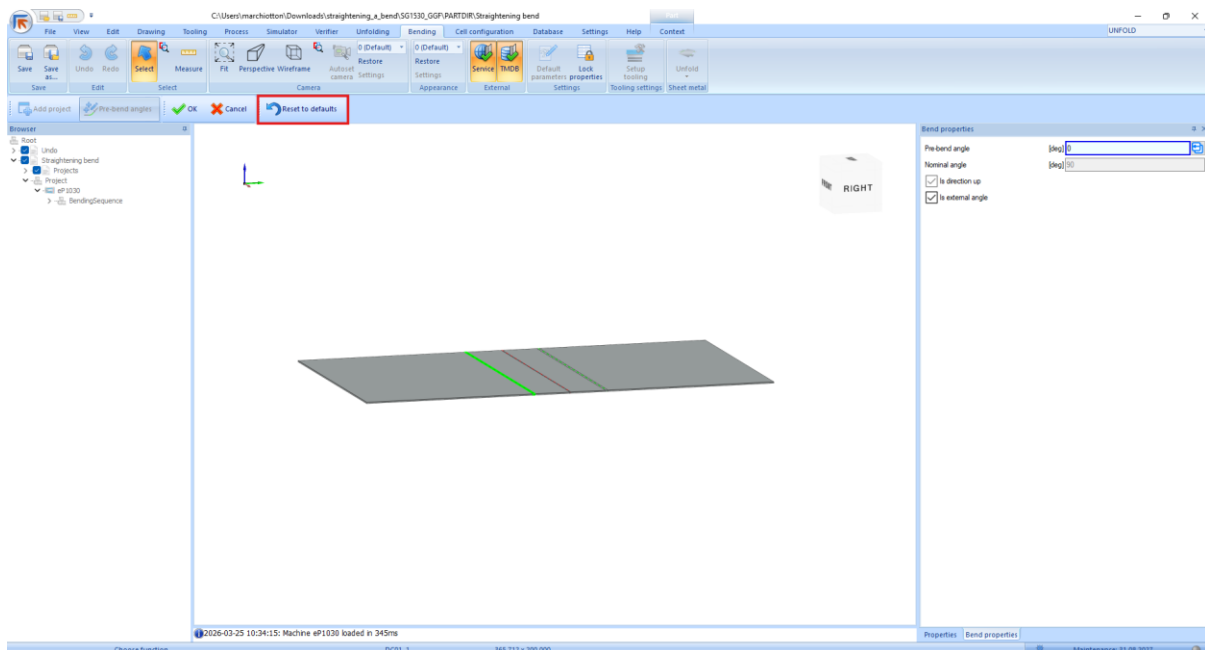
Allowing user to change part flip when multiselecting processes

It is now possible to change the Flip property while multiselecting processes.



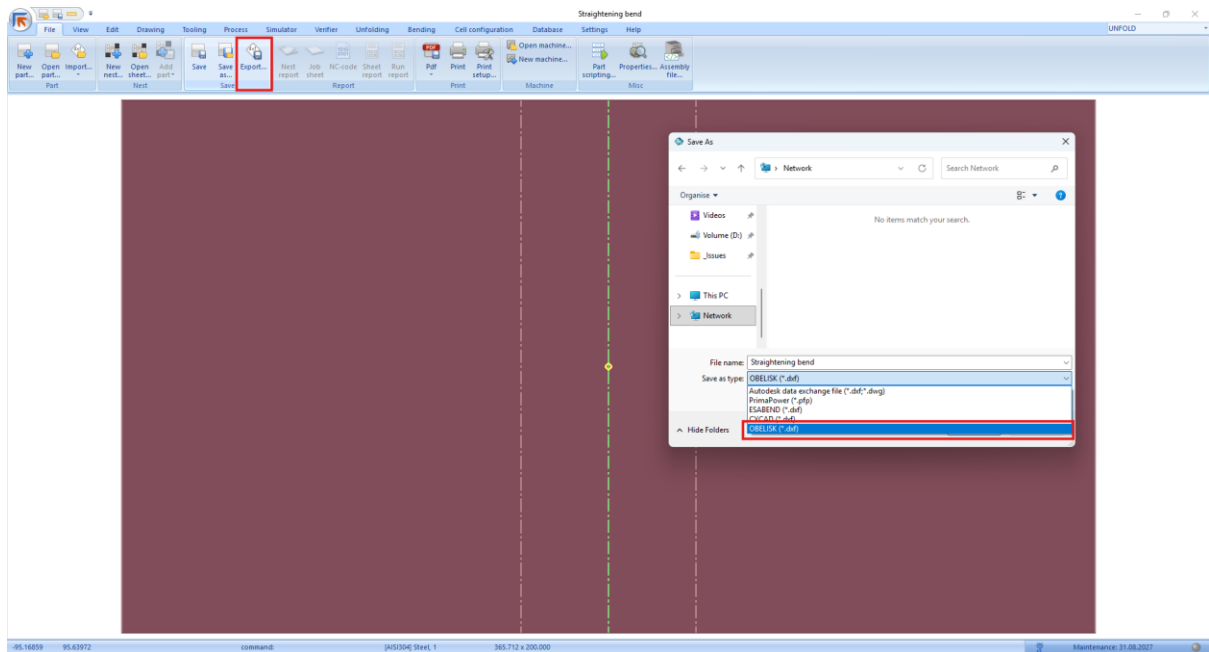
Added possibility to remove all pre-bend angles in one click

Introduced a command that allows you to remove all pre-bend angles in one simple click.



Export for Obelisk

Introduced the possibility to export programs compatible with Obelisk. Exporting will generate .dxf and .polp files.



LASER

LogicNest

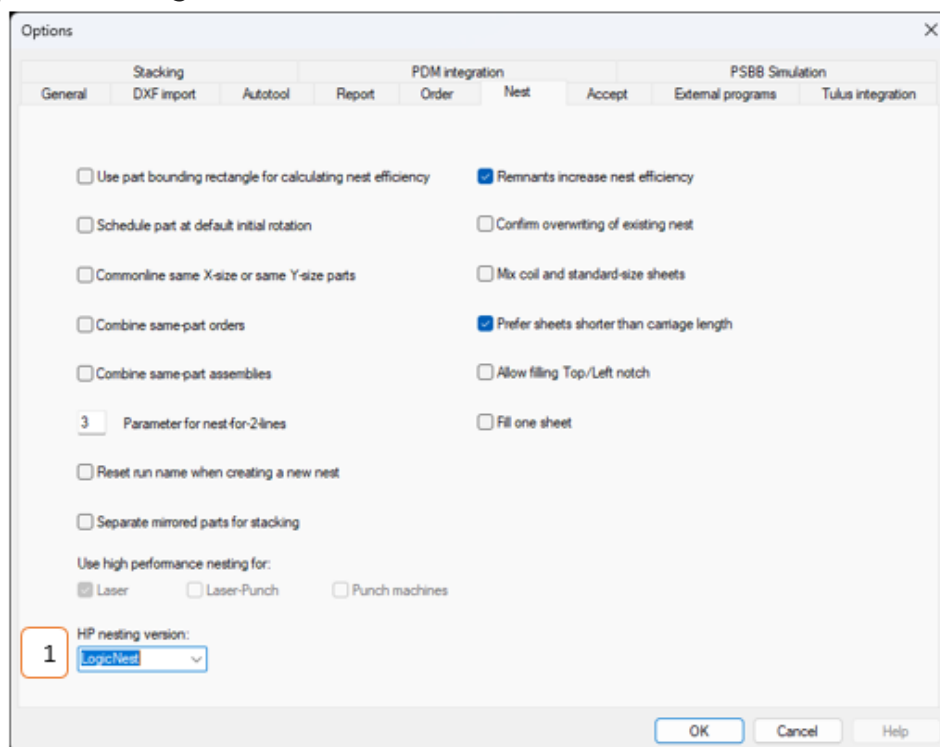
This section covers basic instructions for a new nesting module called *LogicNest* for 2D laser machines ^[1]. LogicNest offers various nesting features, including the following:

- Adjustable computation time to control how long a computer can search for nesting results.
- Improved computer processor utilization to run the computation in parallel.
- A possibility to study intermediate results found during the nesting computation.
- A new improved control for guiding LogicNest to search solutions in which the number of different layouts is minimized alongside the goal of having good sheet utilizations.

On activation

To start working with LogicNest, it must first be set as an active nesting module as follows.

1. Select **LogicNest** ^[2] as the high performance nesting version from the drop-down menu located at the bottom left corner on the **Nest** tab of the **Options** dialog.



^[1] Support for other machine types will be added later.

^[2] If no proper license including LogicNest is found, LogicNest is not listed in the drop-down menu.

On creating a nesting

The use of LogicNest is similar to the existing nesting modules already available in NC Express. There is, however, a small number of new features that are briefly explained in this subsection.

1. LogicNest is active when the title of the first page of the dialog below is **LogicNest**.
2. To control the minimization of the number of different layouts, select the desired importance of the goal from the drop-down menu ^[3].
3. Adjust the allowed computation time. The time is given in seconds ^[4] ^[5].
4. Input other desired options, parts and sheets to be used in the nesting.
5. Finally, click the OK button to start the nesting computation.

The screenshot shows the 'Nesting parameters' dialog box with the 'LogicNest' tab selected. The dialog is divided into several sections. Callout 1 points to the 'LogicNest' tab. Callout 2 points to the 'Minimization of the number of layouts' dropdown menu, which is set to 'The number is not minimized.'. Callout 3 points to the 'Maximum nesting time (sec):' input field, which is set to '60'. Callout 4 points to the 'Part list' and 'Material sheet list' tabs. Callout 5 points to the 'OK' button at the bottom right of the dialog. Other visible settings include 'Part separation: 5', 'Beam width: 0.1', 'Nest in holes' (checked), 'Common line' (unchecked), 'Enable stacking' (unchecked), 'Allow leads relocation' (unchecked), 'Rectangular nesting' (unchecked), 'Use tooling envelope' (checked), 'Sheet margins' (Left: 10, Right: 10, Top: 10, Bottom: 10), 'Clamp information' (1: 0, 2: 0, 3: 0, 4: 0, 5: 0, 6: 0, Clamp size X: 0, Clamp size Y: 0), and 'Use clamps' (unchecked).

^[3] Note that a smaller number of different layouts may reduce sheet utilization or increase the number of sheets required.

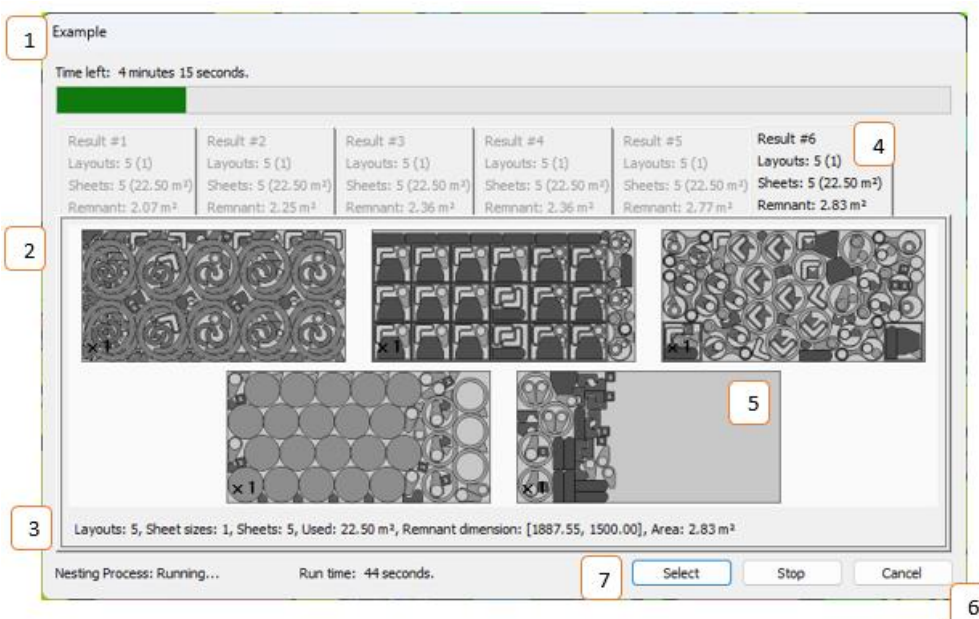
^[4] The more computation time is given, the more time the computer has to find better results.

^[5] Due to the multithread computation and stochasticity of such, LogicNest may not be able to find the same nesting results between the following runs. Also, the hardware setup and possible background tasks the computer is running may affect the computation power the computer can allocate for the nesting computation.

On nesting computation

LogicNest introduces a new progress dialog for the nesting computation. The dialog is introduced below.

1. The name of the nesting is shown in the title of the dialog.
2. A nesting result found during the computation is previewed in the dialog.
3. Generic information of the result is presented at the bottom of the preview:
 - number of different layouts
 - different sheet sizes used
 - the total number of sheets required
 - how much material is used
 - remnant size on the last sheet
 - area of the remnant.
4. Different results found are listed, and they can be viewed by clicking the tab of the result.
 - The title of a tab shows similar information as explained in step 3.
5. When the mouse pointer is moved on top of a layout, detailed information about the layout is presented.
6. The dialog is resizable, and thus the layout previews can be made larger.
7. The nesting computation can be controlled from the dialog ^[6] ^[7].
 - **Select:** Select the previewed result and continue with it.
 - **Stop:** Stop the nesting computation without closing the dialog.
 - **Cancel:** Cancel the nesting computation.



If the computation time runs out, and the NC Express operator has not clicked anything on the dialog:

1. The dialog will close automatically.
2. The latest result found will be selected and continued in NC Express.

^[6] If a desired result is found during the computation before reaching the given end time, the nesting computation can be stopped with the result selected, and thus there is no need to wait until the very end.

^[7] Cancelling or stopping the nesting may take some extra time to stop all the processes the module may have started.

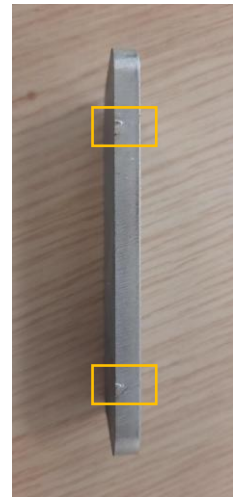
Clean-joints

Tulus laser machines (Laser Genius+ and Platino Linear) support the **clean-joint** in NC programs starting from the following versions:

- PLC/CNC Application: *Open2DLaserApp-8-5*
- Machine HMI: *Tulus version 5.17.1*
- TOB: *Version 2.17*

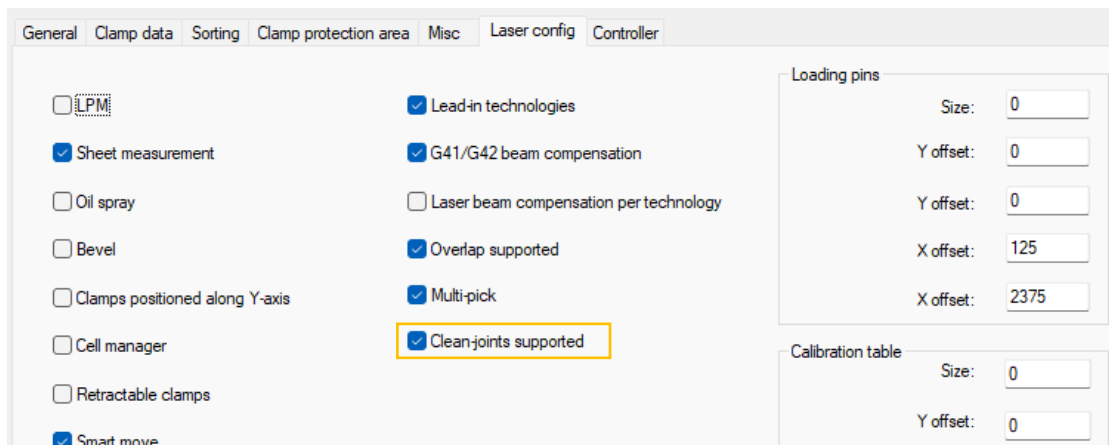
A clean-joint is a minimal micro-joint without leads, subject to the following constraints:

- It cannot be associated with lead-in or lead-out.
- Its length is fixed and not configurable.
- Each geometric profile can contain only one joint type; mixing clean-joints and micro-joints within the same profile is not permitted.
- It cannot be placed on profile vertices.



In NC Express, clean-joint programming extends micro-joint functionality (both automatic and manual).

Activation requires Administrator credentials in **Machine Settings-Laser config**.



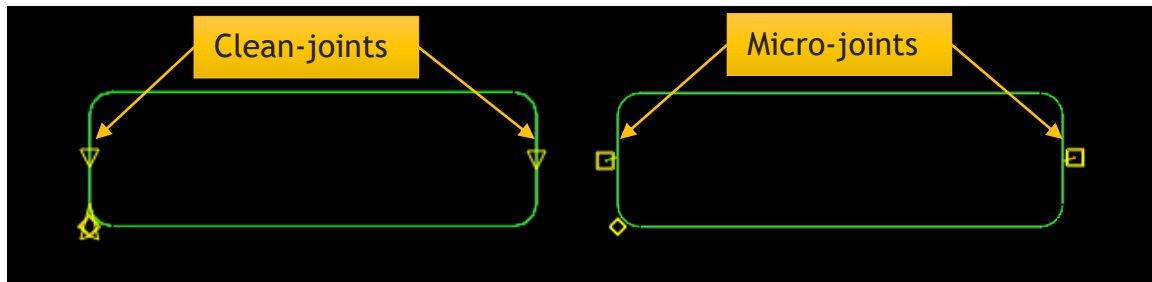
Visualization

Micro-joint visibility is controlled via **View-Micro-joints** in the ribbon.

Joint symbols:

- Micro-joint: square
- Clean-joint: triangle

Symbols are not customizable. An example of joint symbols is shown below.



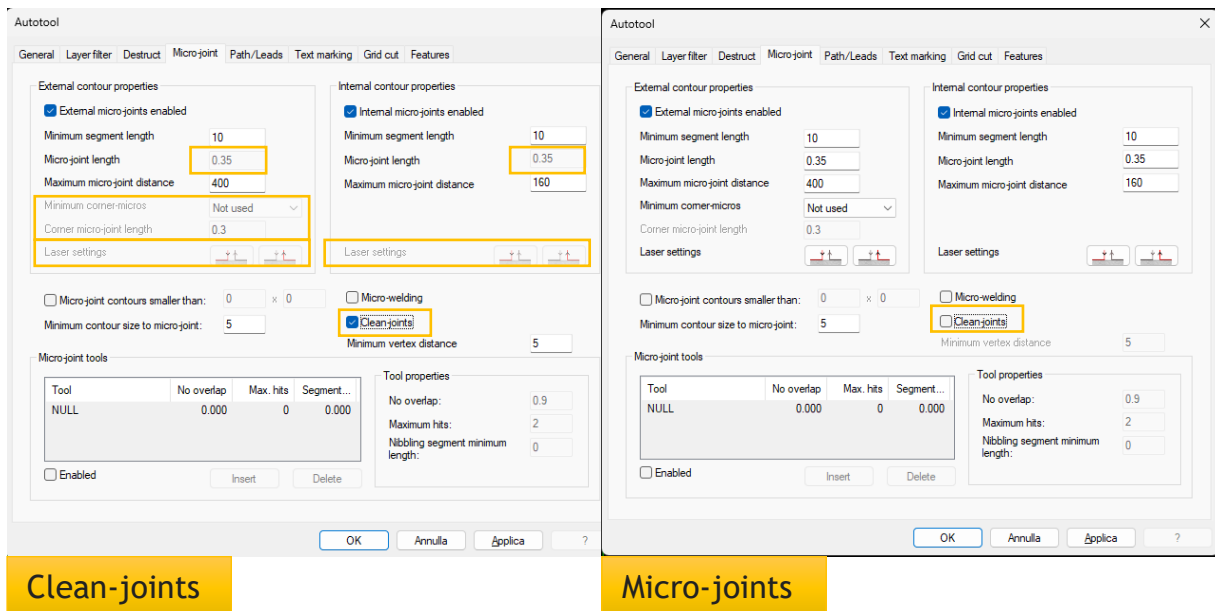
Automatic tooling

The **Clean-joints** option in **Autotool - Micro-joint** tab allows selecting the joint type (clean-joint or micro-joint) used during automatic tooling.

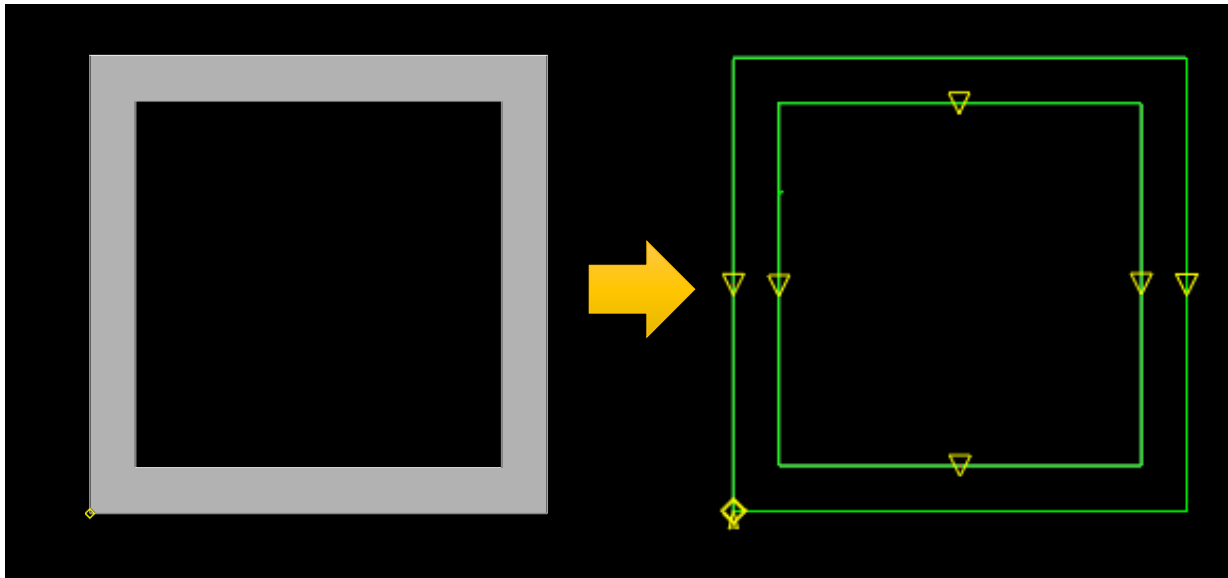
When clean-joints are enabled, the following features are unavailable:

- Joint length configuration
- Corner evaluation algorithm
- Lead parameter customization

These options remain visible but are disabled, as shown in the image.

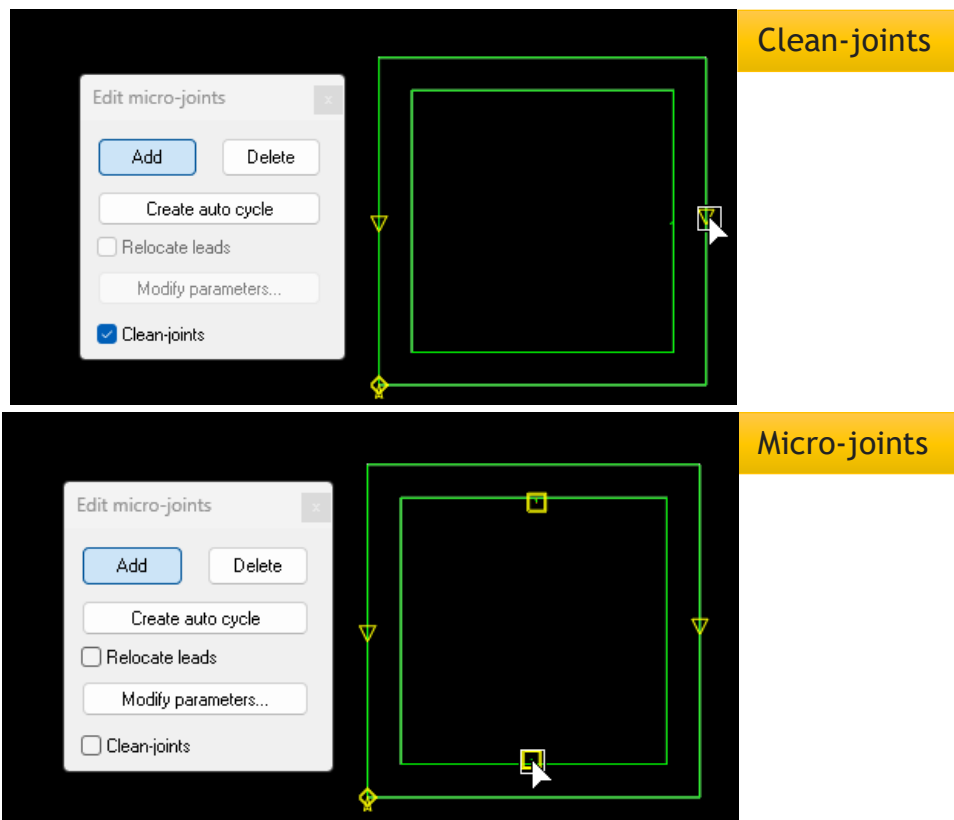


The following example shows the output with clean-joints enabled on a 200 mm × 200 mm frame-shaped part.



Manual tooling

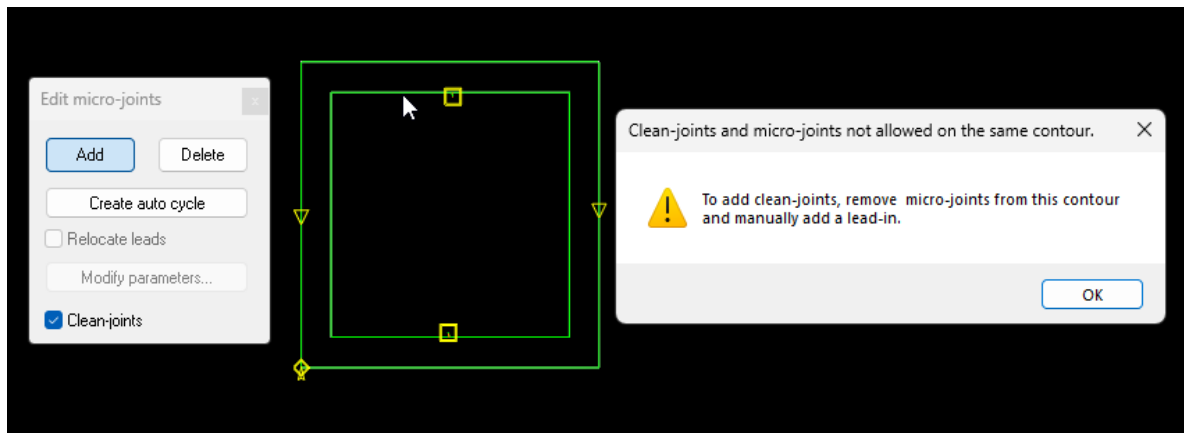
Clean-joints option of *Edit micro-joints* dialog defines the joint type for insertion or deletion. An example is shown in the image.



System constraints enforced during manual operations:

- Only one joint type per geometric profile is allowed.
- After removing all micro-joints from a profile, a lead-in must be manually added before inserting a clean-joint.

In the following example, the system indicates that a clean-joint cannot be added to a contour that already contains micro-joints.



Grid parts

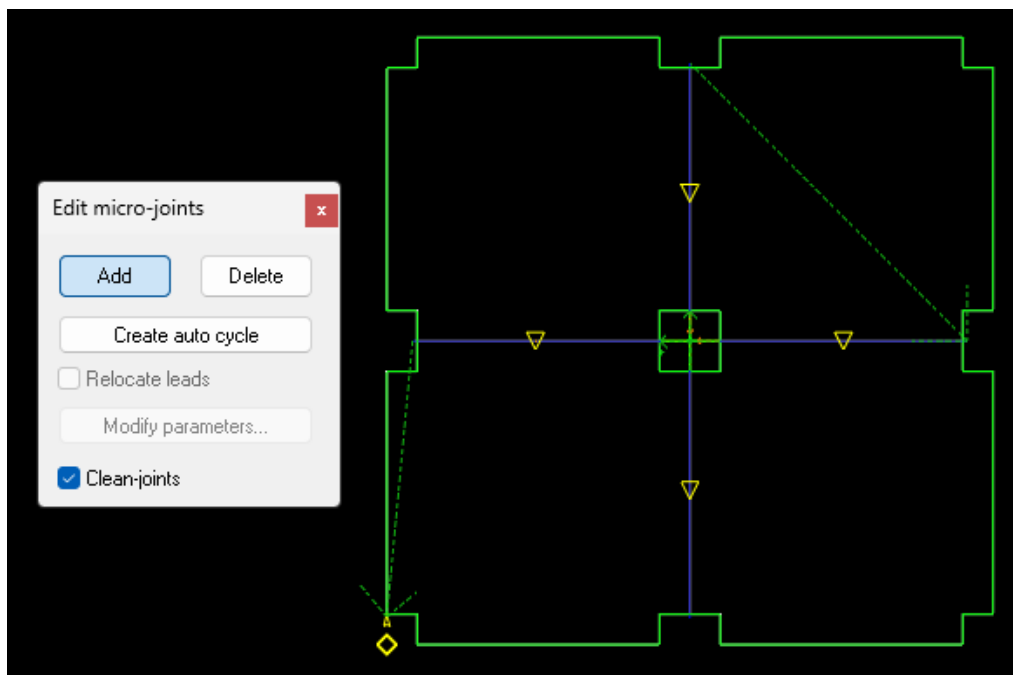
A *Grid part* is a cluster of identical parts created by manually defining a common-line pattern using ***Drawing-Grid Parts***.

In NC Express, joints on common lines are supported exclusively for Grid parts. As a result, clean-joints on common lines are available only when using Grid parts.

The use of Grid parts is recommended when an unloading robot is involved. In such setups:

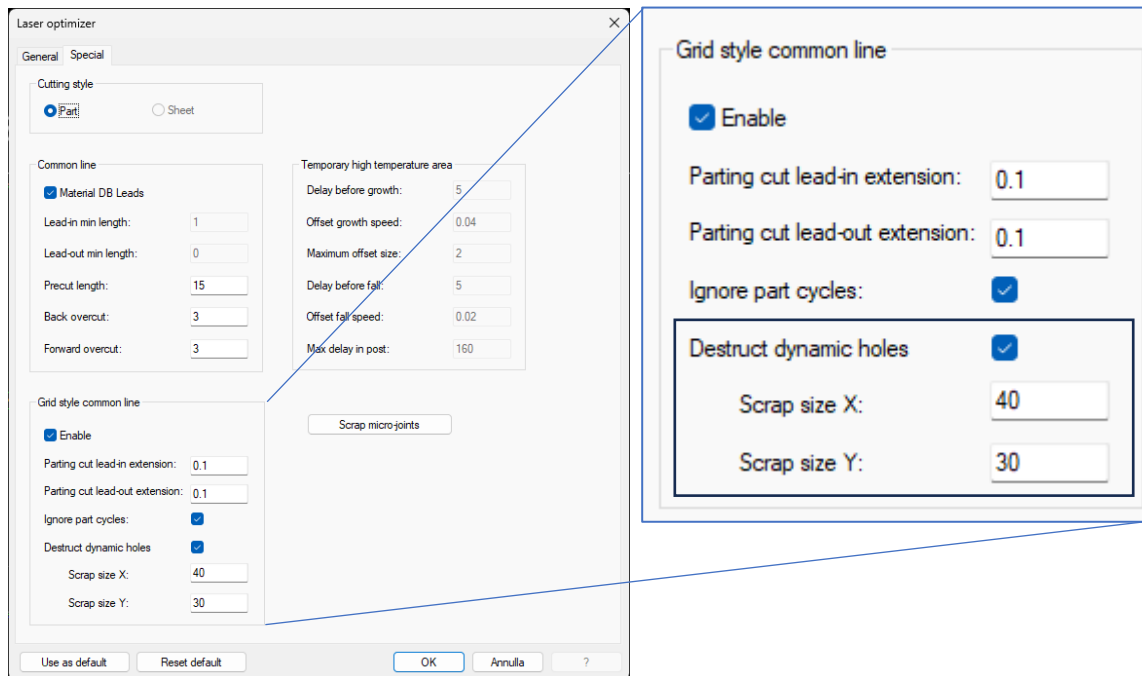
- The user can define the gripping position on the part.
- Clean-joints can be manually placed on common lines to keep parts connected.
- This approach helps reduce marks on the part edge compared to standard micro-joints.

An example is shown in the image:

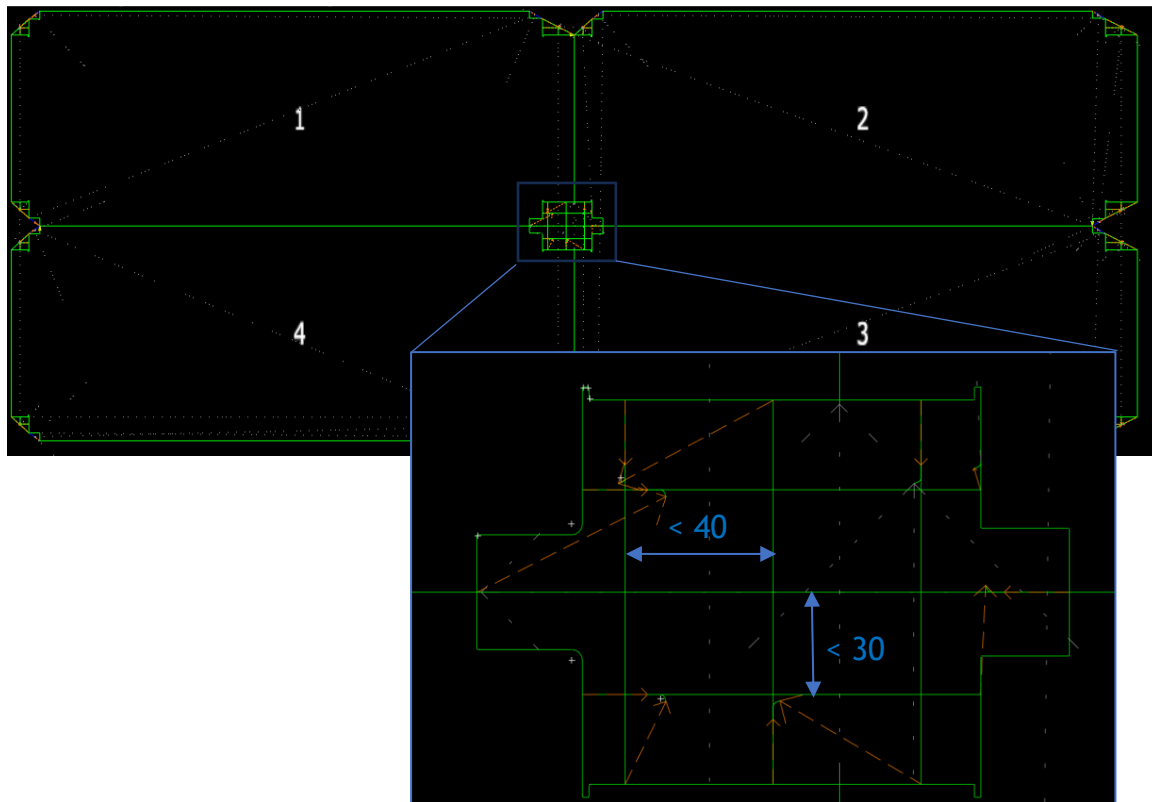


Customization of X and Y scrap sizes during Grid Style common line

It is now possible to configure explicitly different scrap sizes along X and Y axes in the *Laser optimizer* dialog under the *Special* tab's *Grid style common line* option.



These values are the maximum allowable values of the scrap size on the axis applied during the destruction of the common area defined by at least 4 parts in a 2 x 2 matrix grid.



Laser Next Core machine support

Laser Next Core is the new entry of the 3D machine series with Siemens Sinumerik control.

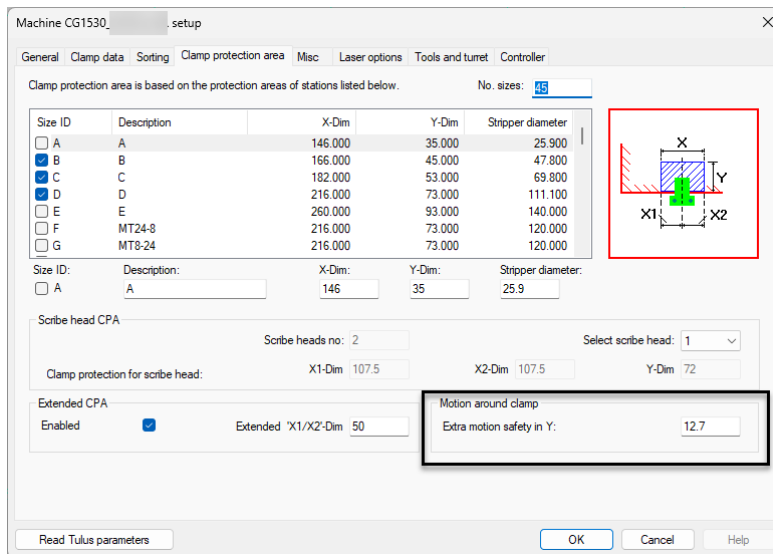
NC Express is the 2D cutting CAM for this type of machine. The post processor integrates the macros of the machine's 2D cutting module.



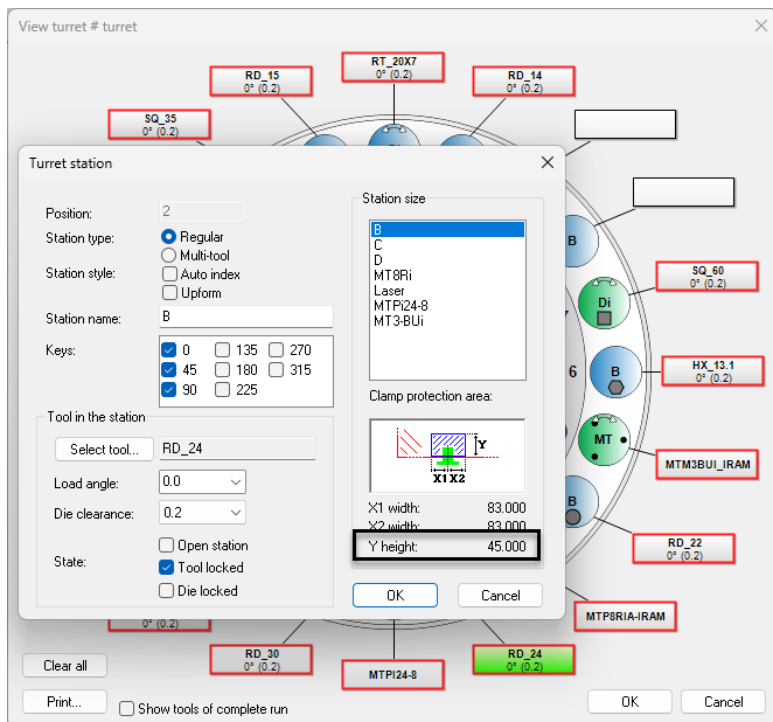
PUNCH

Adjustable Y extra safety around clamps

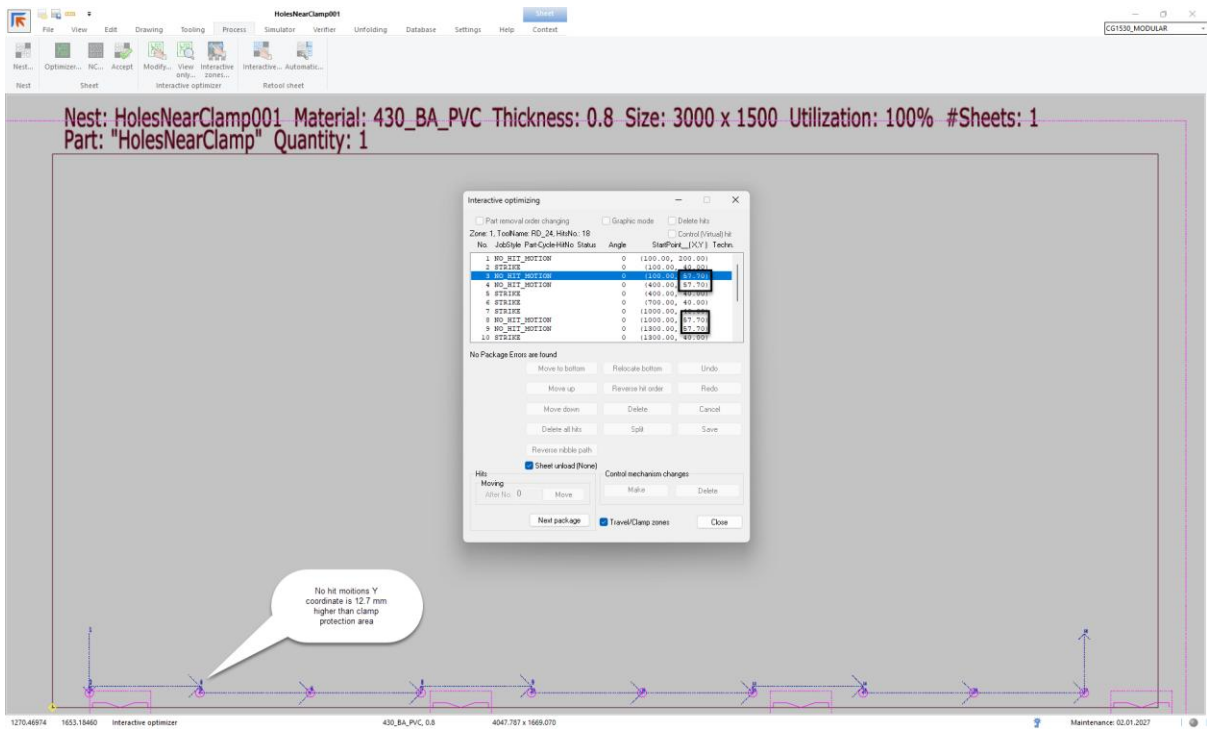
A new *Extra motion safety in Y* parameter has been added to **Machine settings - Clamp protection area**. The default value is 12.7 mm or 0.5 inches. This gives more space for all clamp avoidance movements in optimizing. In many cases, it helps to avoid the clamp's small touches to die and to prevent collision noise and physical wearing of the clamp and die holder.



An example with a B station that has a normal clamp protection Y area of 45 mm.

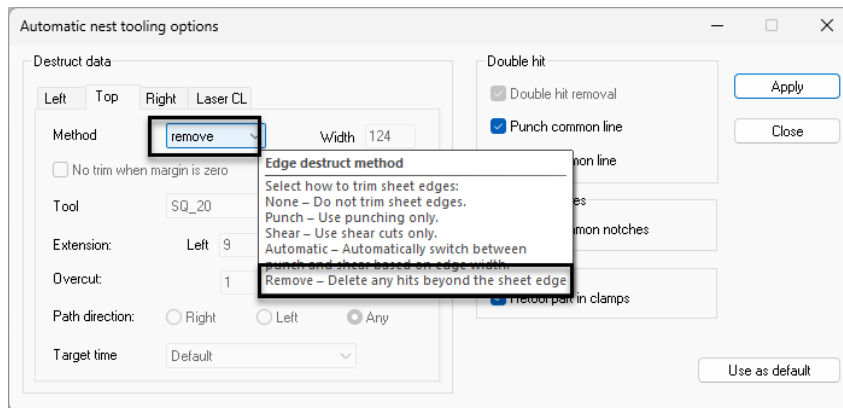


The optimizer results show that all clamp avoidance movements are done 12.7 mm over the normal clamp protection value: $45+12.7=57.7\text{mm}$.

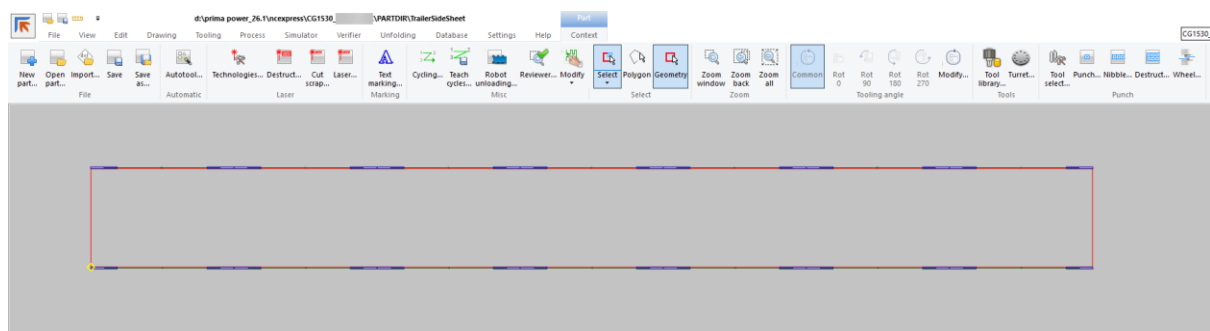


Option to Remove hits outside of sheet boundary

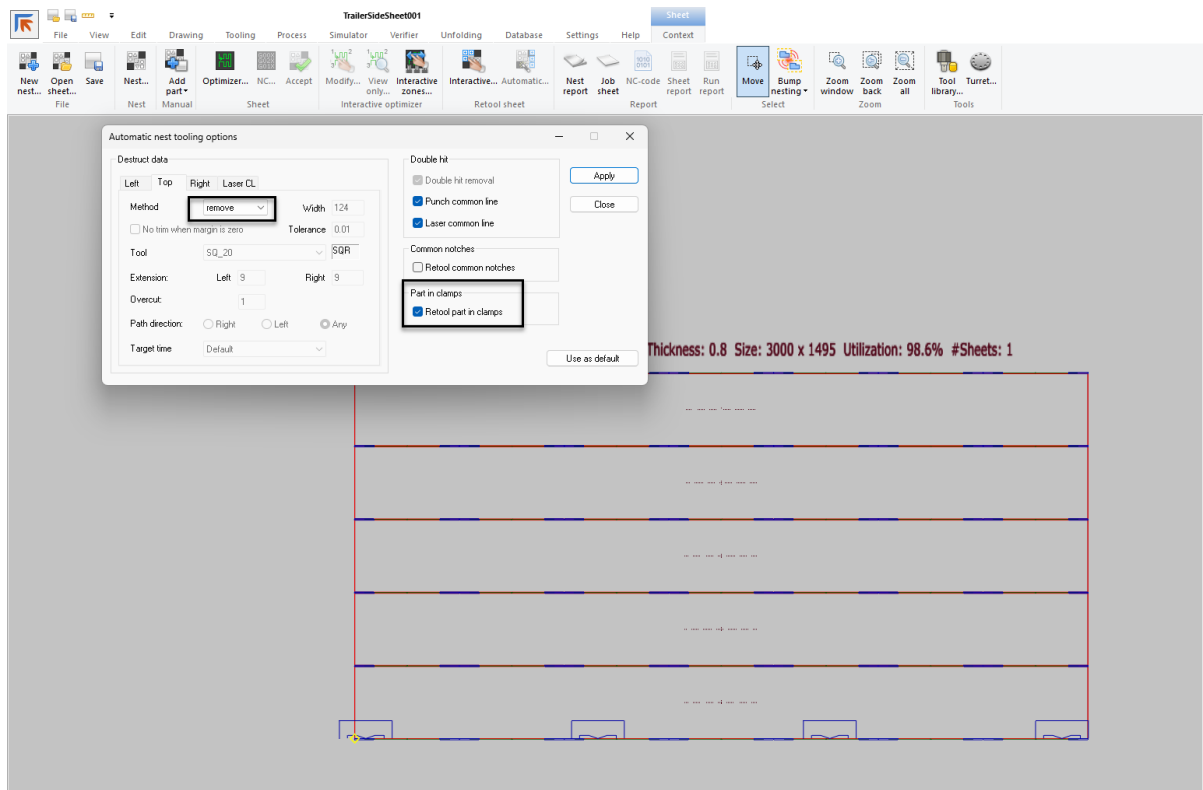
Automatic nest tooling has a new option to remove tooling hits that are outside of the sheet. This is useful in cases where the product only just fits on the sheet, and you can nest the same part that has the tooling on both sides.



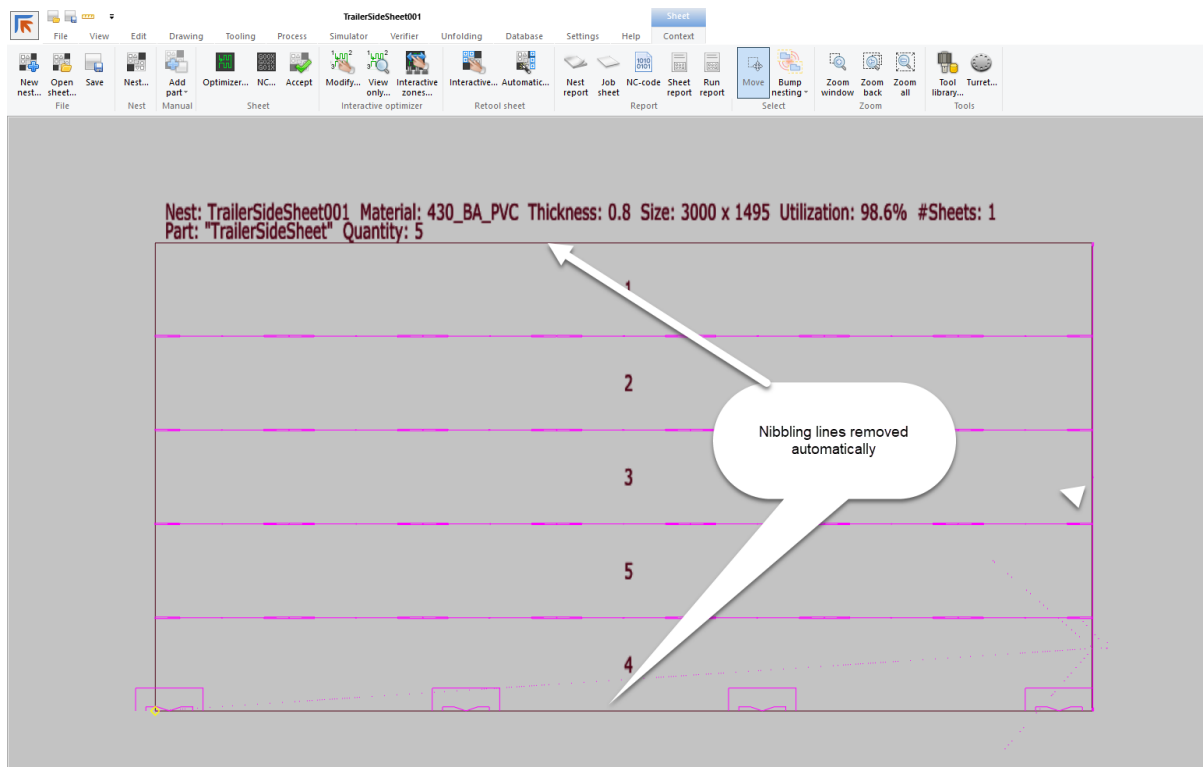
An example of a part that has the nibbling line on the top and bottom side:



You can nest the part so that the tooling is outside of the top side and bottom side of the sheet. Automatic nest tooling set to 'remove' then automatically deletes those nibbling lines in optimizing.



You can see from the optimizing result that top and bottom nibbles were removed automatically. There is no need to make copy parts with different tooling for top and bottom parts in the nest anymore.



PUNCH-SHEAR

Adjustable Y pull-out movement between SHEAR_CUT's

With the 'Minimum Y pull-out' parameter for shear, you can set how much the sheet is released in the Y direction from the X blade between shear cuts when processing parts that are longer than the shear X blade. This parameter helps in cutting the thick sheets with a shear, as it prevents sheets from getting stuck on the X blade during movements to the next shear cut coordinate. This parameter is in use only with machines that use the SHEAR_CUT NC syntax.

The screenshot shows the 'Machine SG1530_DEMO setup' dialog box with the 'Shear and related' tab selected. The 'Min. Y pull-out' parameter is highlighted with a red box, and a tooltip is displayed over it.

Machine SG1530_DEMO setup

General | Clamp data | Sorting | Clamp protection area | Misc | **Shear and related** | Tools and turret | Controller

Blade length

X: 1000
Y: 1500

Max. part size

X: 3000
Y: 1500

Max. scrap size

X: 3000
Y: 1500

Separation factor: 4

Last part in clps min. size coeff.: 2

Don't nest at the bottom parts longer than: 3000

Punch trim strip max. length: 3000

Automatic clearance setting. ☒

Shear pre-start move ☐

Clamp detector turn off mode ☒

Part group processing ☒

Material growth compensation ☐

Fast cut ☒

Release buffer ☐

Shear protection area

X left: 90
X right: 1090
Y: 39.75

Miscellaneous

Bottom overcut: 1
Min. cut width: 10

RS table open & close

☒ Open ☒ Close

Max X: 700
Trim clamp strip max Y: 100

SHEAR_CUT NC syntax

☒ Use SHEAR_CUT NC syntax

X-blade overlap: 10

Min. Y pull-out: 0.5

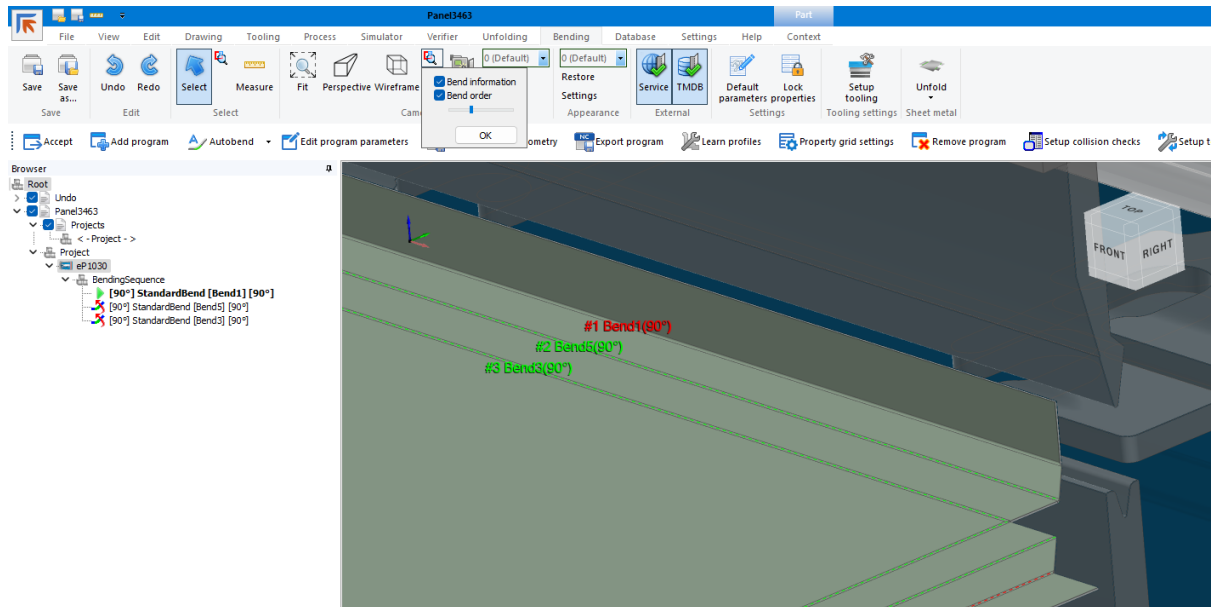
Min. Y pull-out
Minimum pull-out movement between SHEAR_CUT commands on long cuts. This setting controls how far the shear tool retracts along the Y-axis before the next cut.

Read Tulus parameters | OK | Cancel | Help

Other usability enhancements

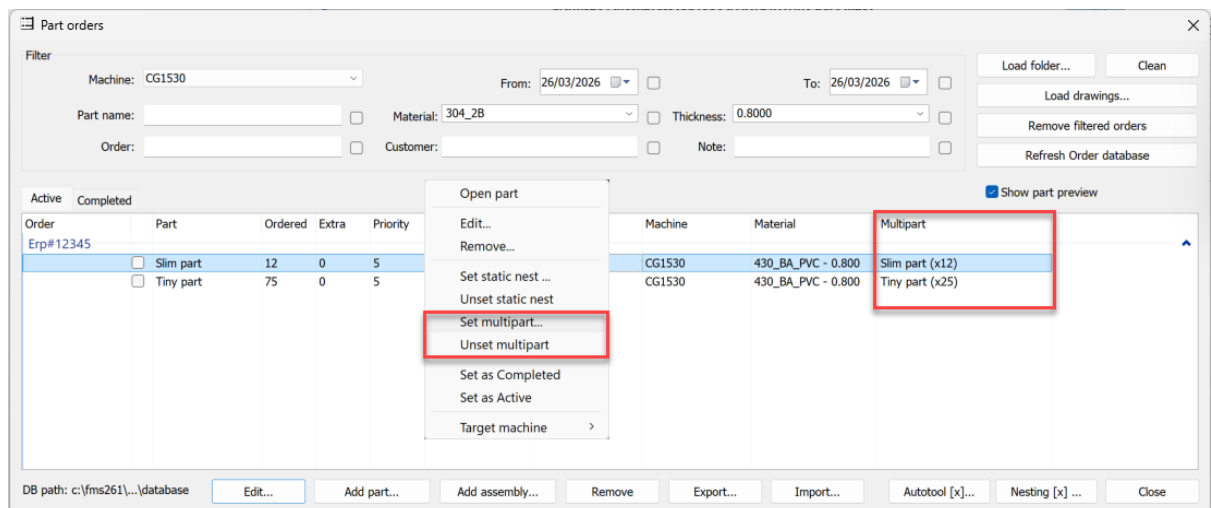
Overlay current bending sequence in 3D view

It is now possible to have a 3D overlay of the bend information containing also the sequence of each bend marked with the “#” prefix. This can be enabled from the Camera section on the ribbon:

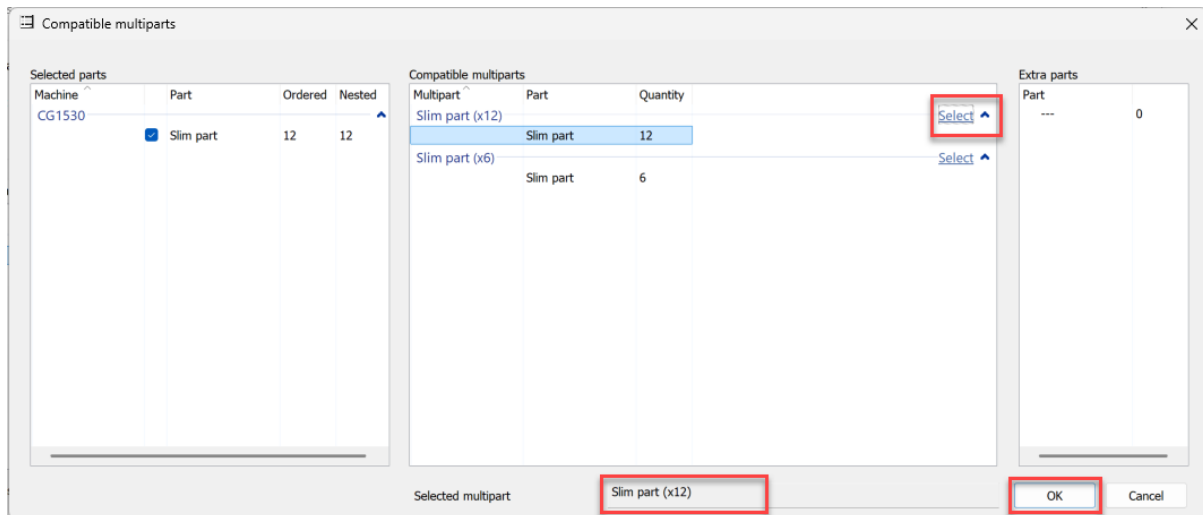


Multiparts in Order database

You can assign a *multipart* in *Database - Order* by using the right-click menu option *Set multipart*.



This opens a small dialog showing all available multipart stored in the part database. Select the desired multipart using the *Select* button and confirm with *OK*.



The assigned multipart name will then appear in the **Multipart** column.

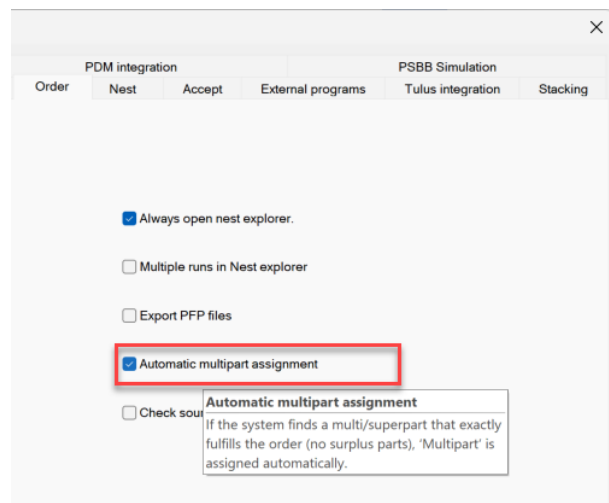
This functionality works for both:

- ACP multipart (created via *Drawing* → *ACP part*)
- Regular super-parts (created via *Drawing* → *Grid parts*)

There is also an automatic assignment option. You can enable this under **Settings - Options - Order - Automatic multipart assignment**.

Note:

The automatic method assigns a multipart *only when a perfect match exists* – it will *not* create any additional parts automatically.

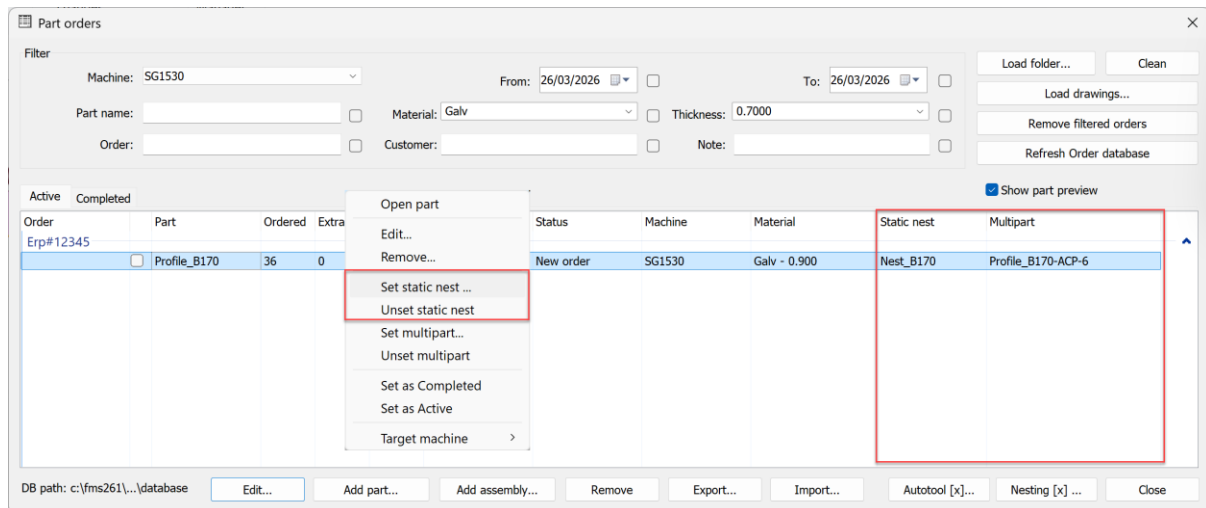


Static nests of ACP parts in Order database

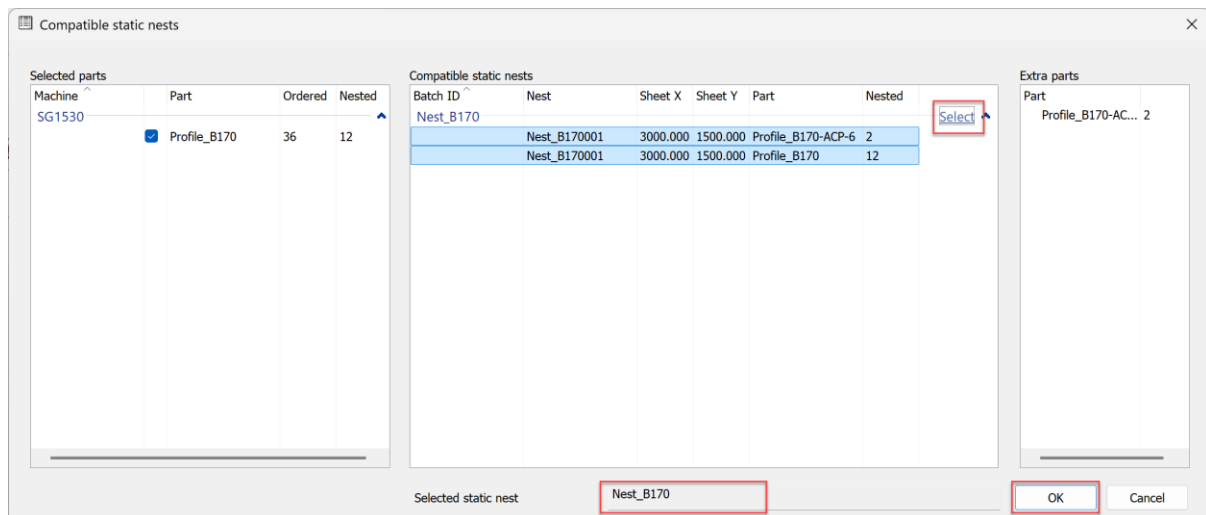
You can now allocate **static nests** also for sheets that contain a multipart (ACP parts), and the Work Order ID allocation will be handled correctly.

The workflow is the same as for regular static nests:

1. Select the desired work orders.
2. Right-click and select **Set static nest**.



A dialog will open showing all available static nests. Select the best matching static nest, press **Select** and confirm with **OK**.



After allocation, make sure that both the **Static nest** and **Multipart** columns are populated correctly.

Support for displaying 'Formings' in Database-Orders

You can check whether a part contains any forming type- tooling directly in **Database--Orders**. To enable this, right-click- on the column header area and activate the **Formings** column as shown below.

Note:

Formings are calculated automatically based on the tooling used in the part. A part is considered to contain *formings* if it includes any of the following:

- Forming wheel
- Forming
- Deep draw
- Bottom stroke

This information is stored in the part database when you press the **Save** button. The order database then retrieves the forming status from the part database and displays it in the **Formings** column.

The screenshot shows the 'Part orders' window with a filter section at the top and a main table area. The filter section includes fields for Machine (CG1530), From (31/03/2026), To (31/03/2026), Part name, Material (304_2B), Thickness (0.8000), Order, Customer, and Note. The main table area has a 'Show part preview' checkbox and a table with columns: Order, Part, Ordered, Extra, Priority, Machine, Material, Static nest, Multipart, and Formings. A context menu is open over the column headers, with 'Formings' selected. The table data shows two rows for Order Erp#12345: one for 'Slim part' (Ordered: 12, Extra: 0, Priority: 5) and one for 'Dimple part' (Ordered: 100, Extra: 0, Priority: 5). The 'Formings' column shows values 0 for 'Slim part' and 4 for 'Dimple part'.

Order	Part	Ordered	Extra	Priority	Machine	Material	Static nest	Multipart	Formings
Erp#12345	☐ Slim part	12	0	5	CG1530	430_BA_PVC - 0.800		Slim part (x12)	0
	☐ Dimple part	100	0	5	CG1530	430_BA_PVC - 0.800		Tiny part (x25)	4

Windows support

NC Express 26.1 supports operating systems from Windows 7 and Windows Server 2012 up to the latest versions of Windows 10 and Windows 11. The software is available exclusively as a 64-bit version.

Note 1: If you are upgrading an existing 32-bit installation to 64-bit and your system uses customized report templates, these templates must be recreated for the current reporting system.

Note 2: If you have software integrated with NC Express via the ParaNCX interface, contact Prima Power before updating to the 64-bit build. This applies to applications such as Tulus Office Premium, Tulus Office Power Processing, Batch Processing, PowerPlan and others. Any integrated software must also be compiled for 64-bit.

Note 3: NC Express 25.2 is the last release available in both 32-bit and 64-bit versions.