NC Express

Software version release: 24.1

31 May 2024

New features in NC Express 24.1

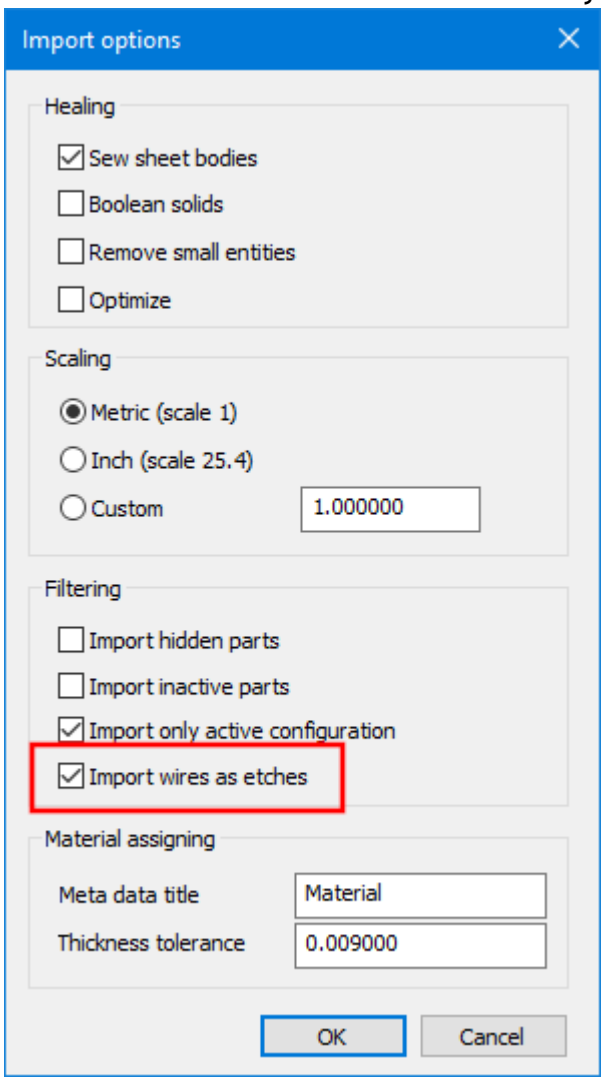
Unfolding

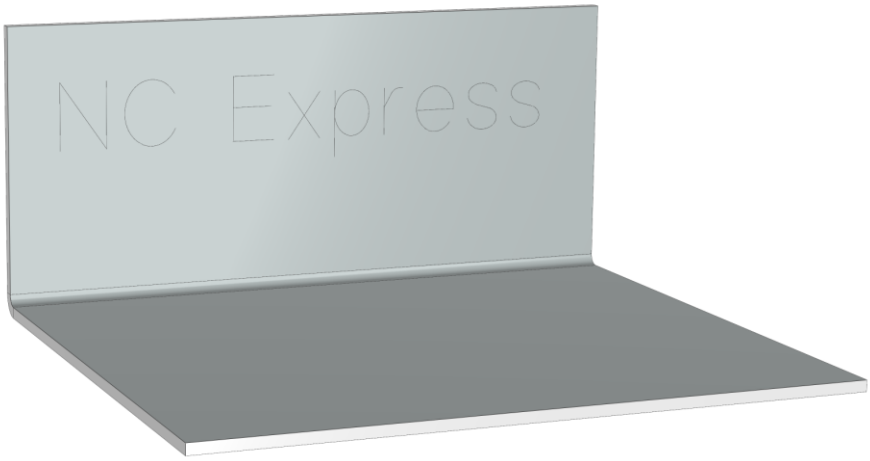
Supported 3D formats and versions

- Autodesk Inventor (*.ipt, *.iam), up to 2024
- SolidWorks (*.sldprt, *.sldasm), up to 2024
- Solid Edge (*.par, *.psm, *.asm), up to 2024
- Siemens JT (*.jt), up to 10.7
- Siemens NX (*.prt), up to 2312.4000
- 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

Reading a marking from a 3D model

The new import option ***Import wires as etches*** allows you to import a 3D model's wires. This results in etch features that you can use for marking:





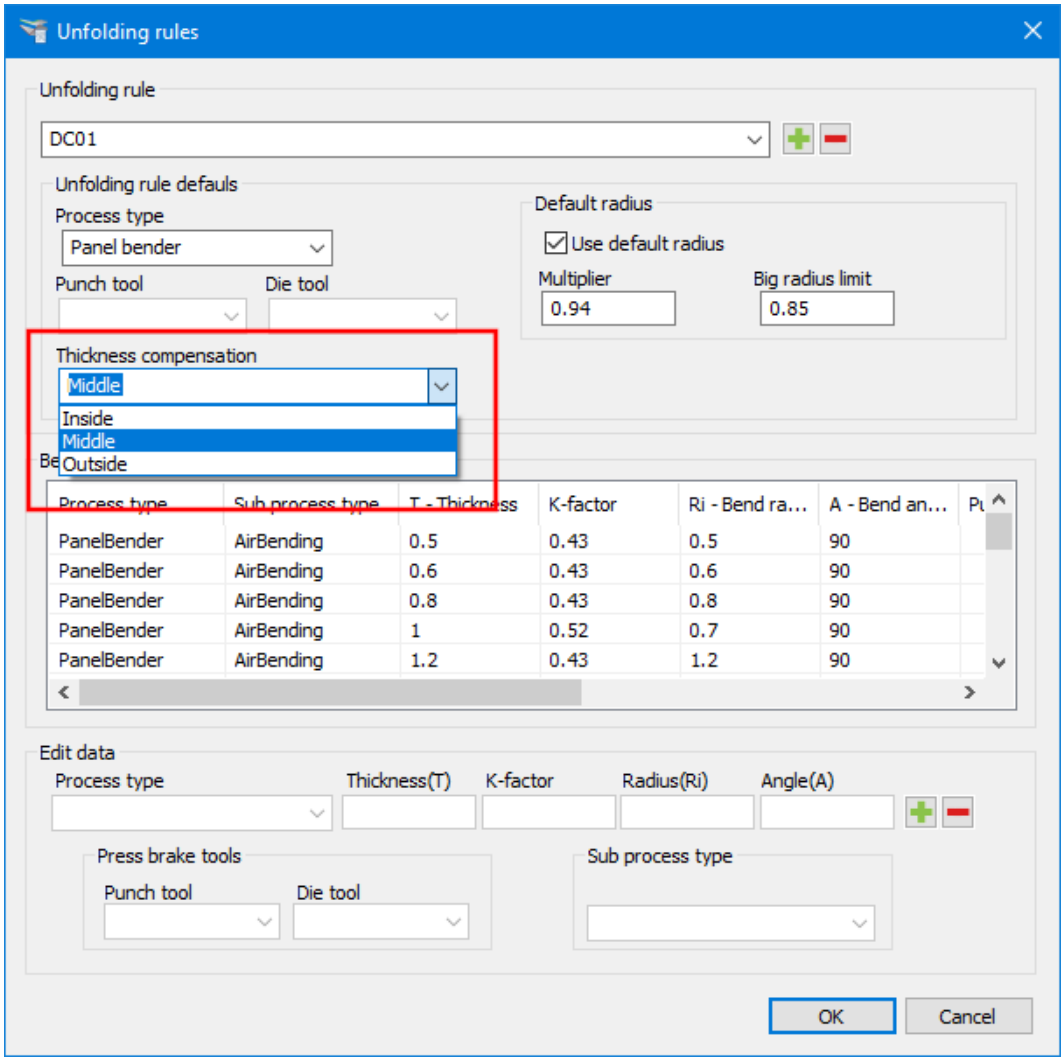
Thickness difference compensation

You can now select the compensation method for unfolding in case the design and manufacturing thicknesses differ.

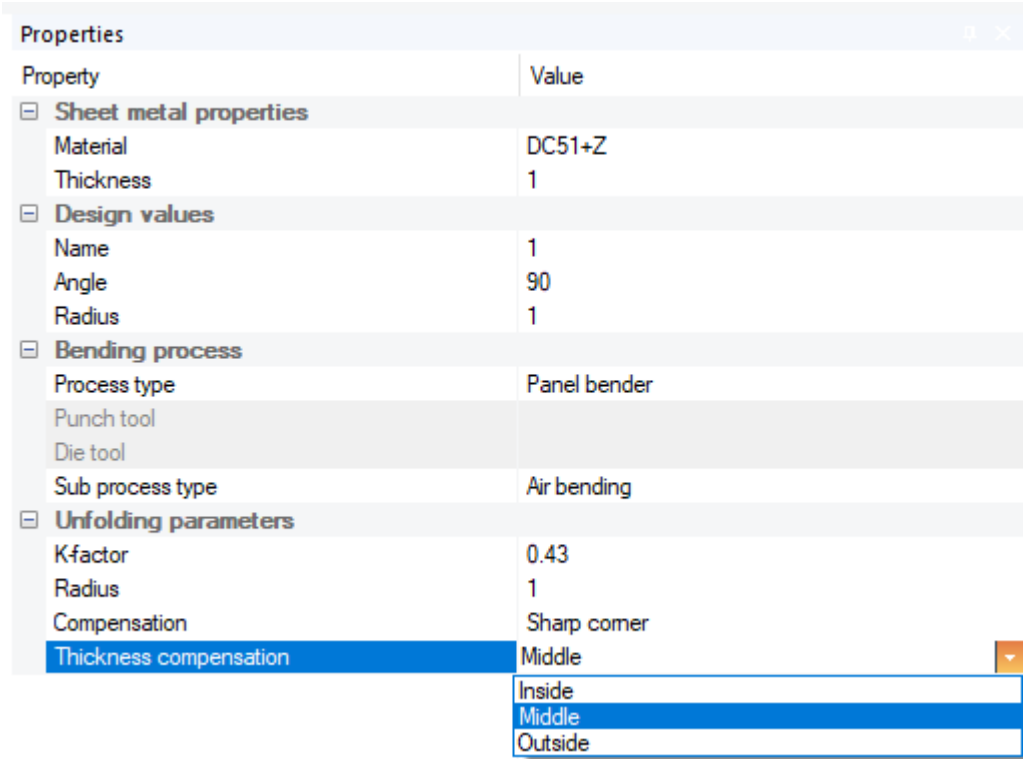
Possible values are:
0 = Inside
1 = Middle (default)
2 = Outside

Inside means that the material difference is considered to be at the inner side of the bends while calculating unfolded length.
Middle means that the material difference is set on both sides equally while calculating unfolded length.
Outside means that the material difference is considered to be at the outer side of the bends while calculating unfolded length.

You can set a preferred compensation for each unfolding rule:



Or you can modify it separately for each bend:



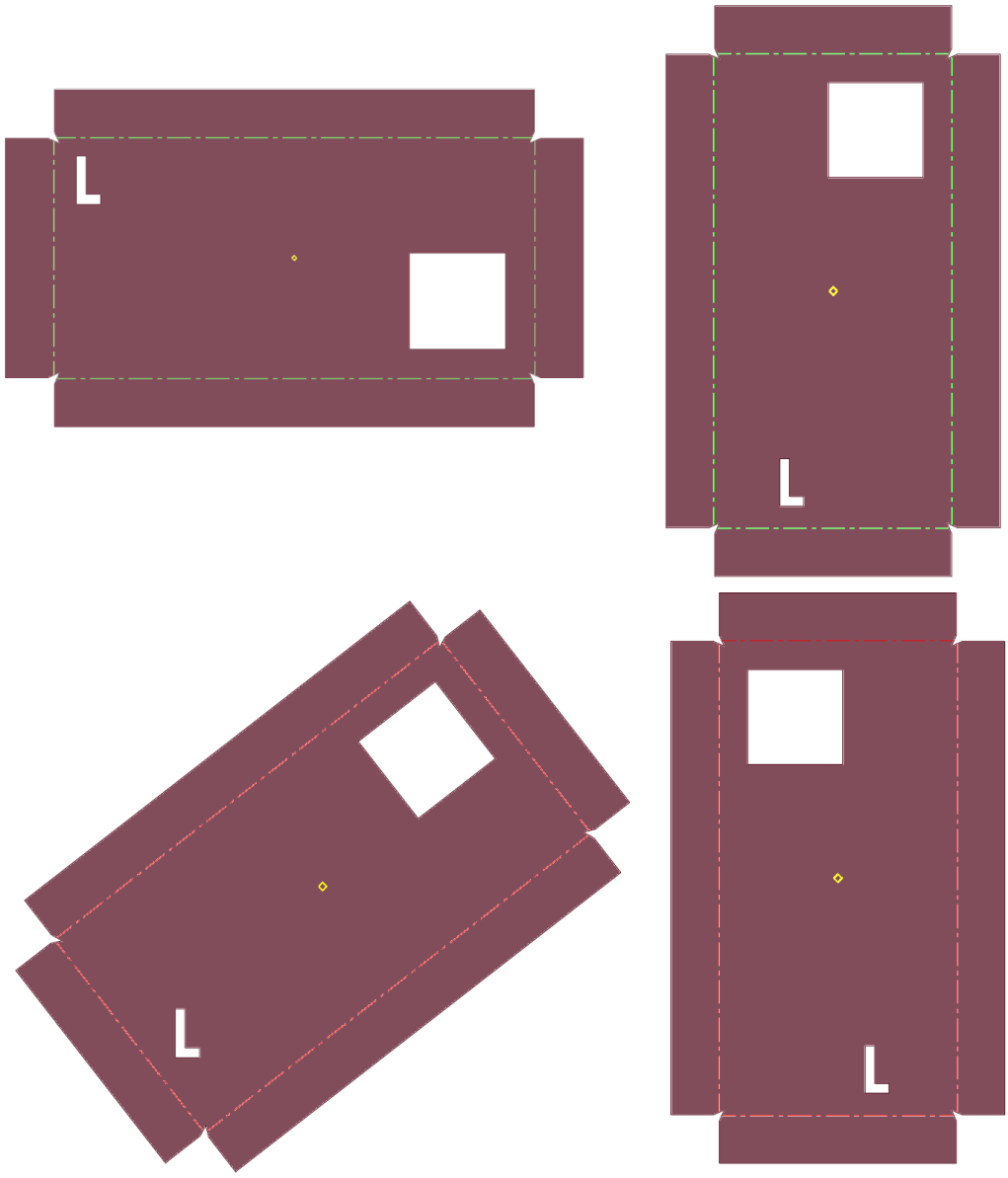
Forcing blanking orientation

You can now force blanking orientation by defining a reference geometry that is used to orient the part for blanking. You can do this by placing “orientation.cp” inside the machine setup’s configuration folder. This orientation.cp should contain a geometry to be searched from the 3D model that is used to align the part geometry for the blanking.

Note that the geometry to be used should not be symmetric around any axis. Otherwise, it can lead into radiomic results, since it can be oriented in multiple ways to match the reference geometry.

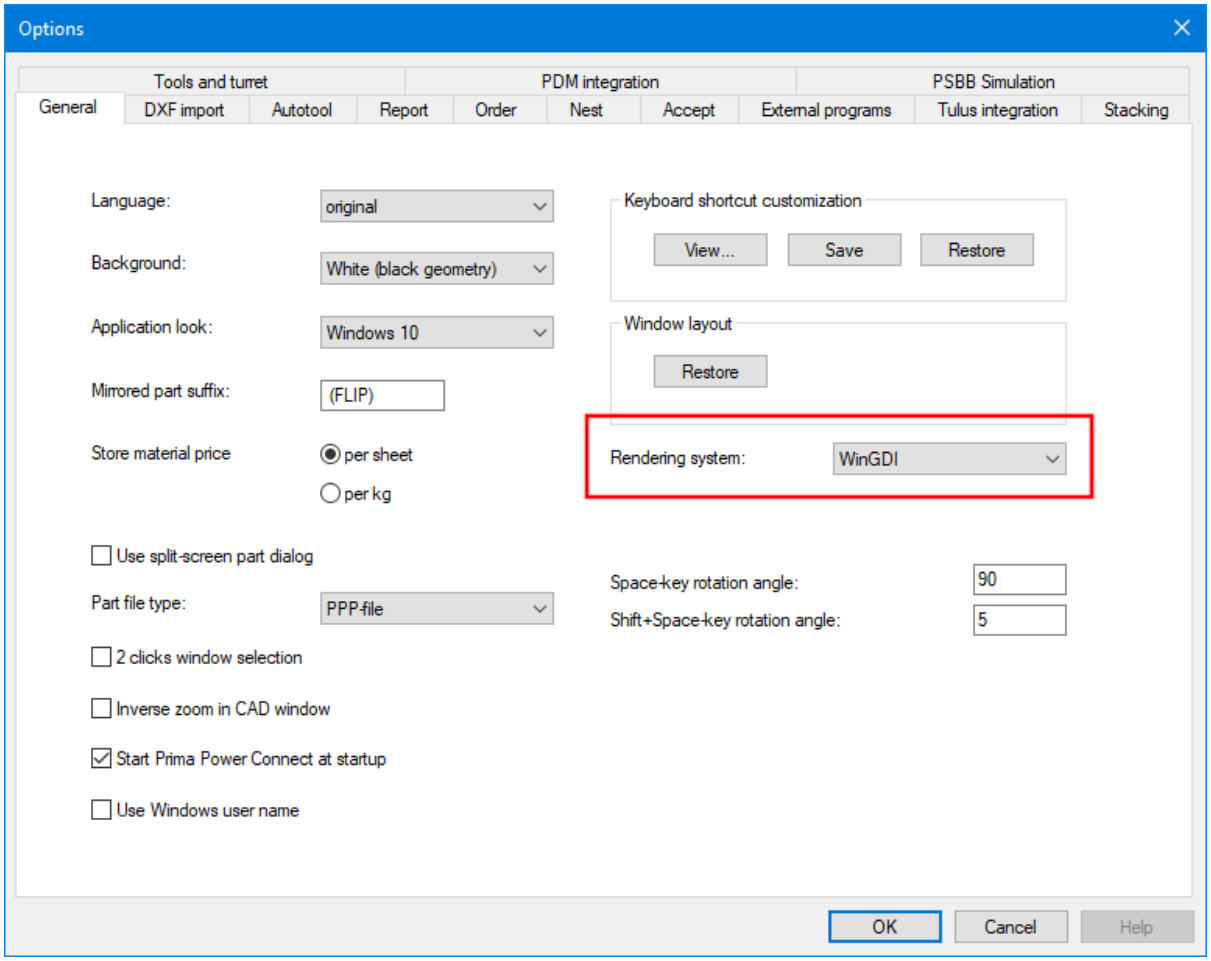
The example below shows the result you can get with such an orientation.cp:

The same part rotated and flipped based on the reference geometry location and position:



Software rendering on 3D canvas

Enabling the WinGDI rendering mode now also enables software rendering on 3D canvas:



Earlier software rendering on 3D canvas was made only as a fallback in case no sufficient GPU could be found.

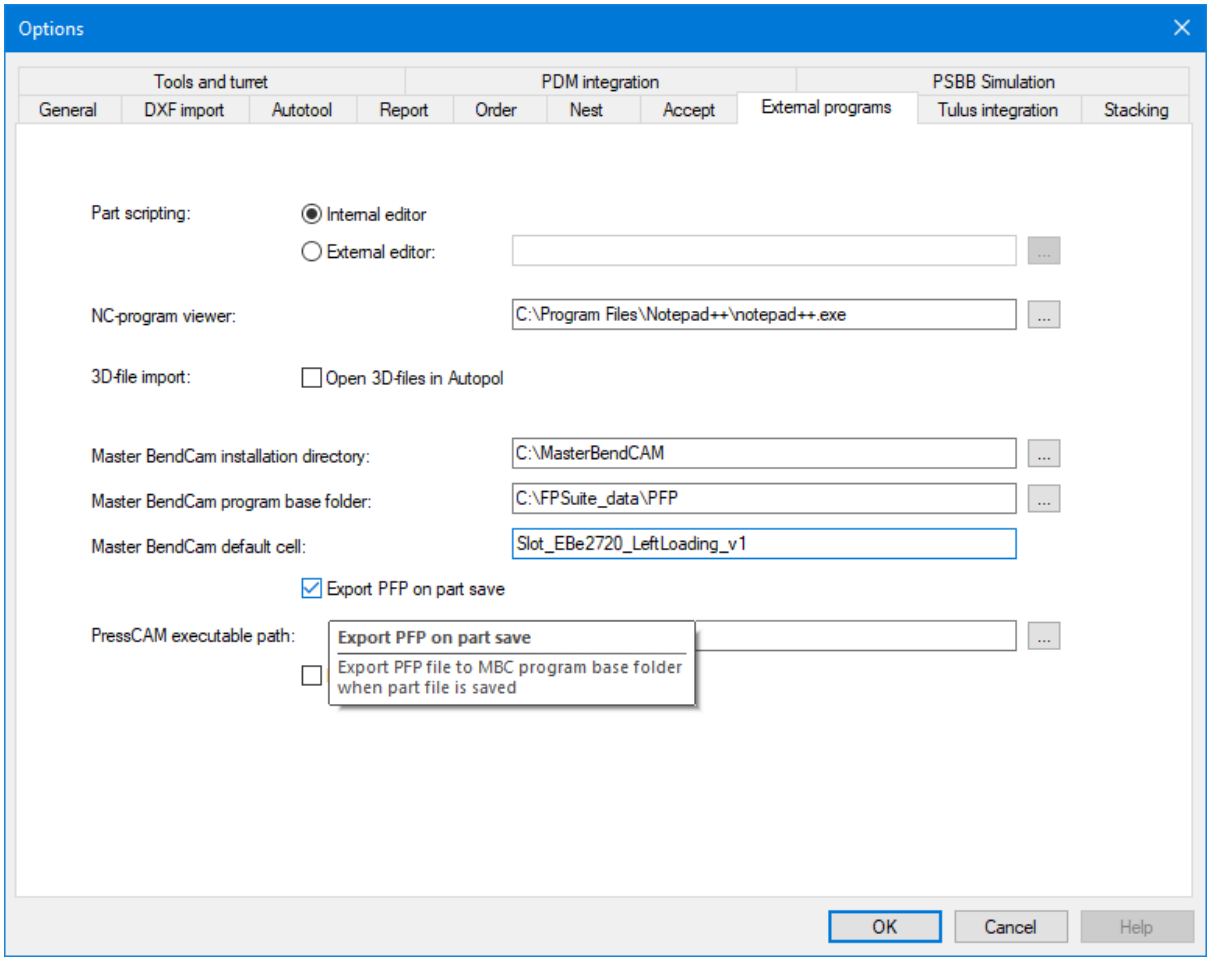
The BEND machines

Panel bender programming

More information on programming EBe and BCe machines will be available later.

Exporting a geometry PFP file on save

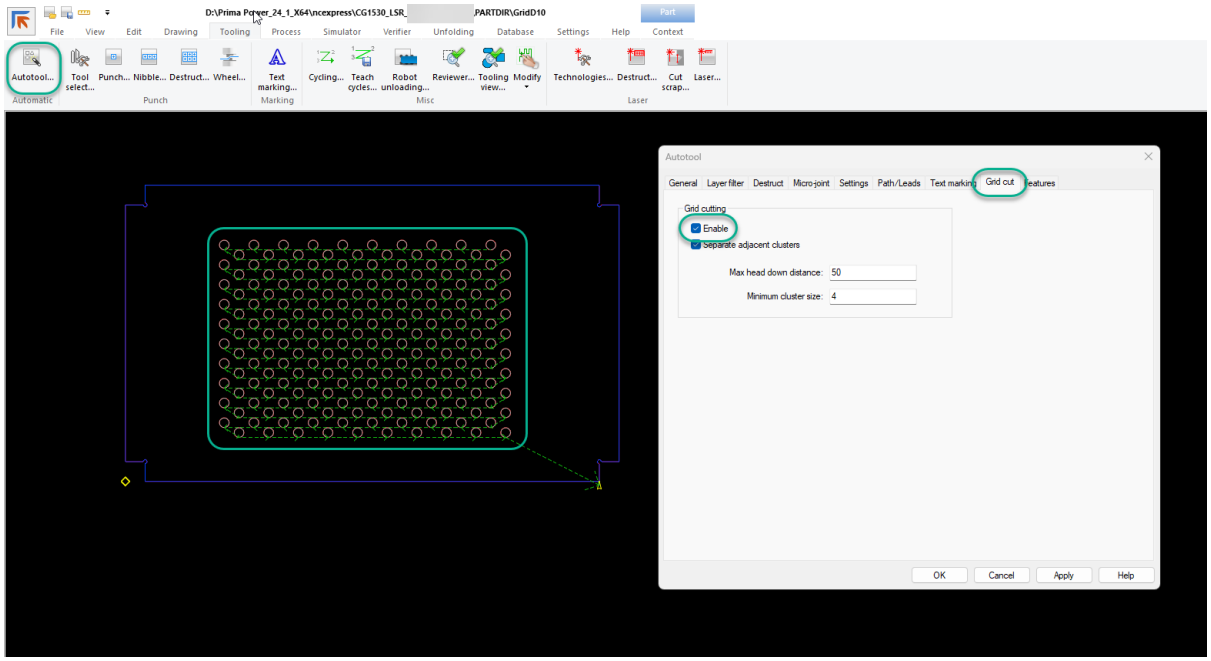
You can now export a geometry PFP file for Master BendCAM automatically on part save:



The LASER machines

Grid cutting for round hole patterns with Autotool

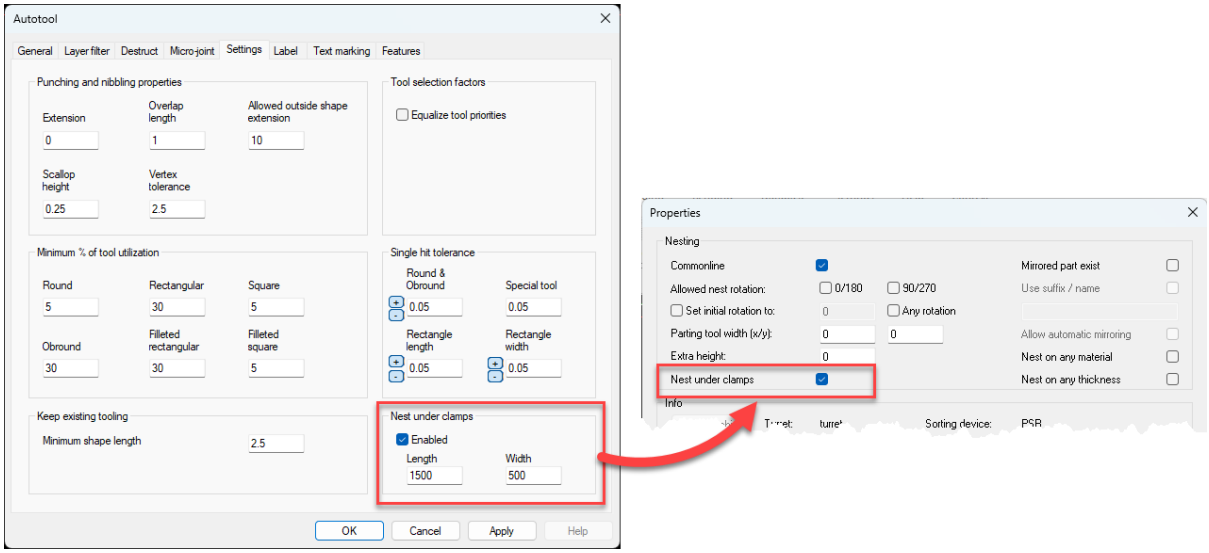
Autotool has the ***Grid cut*** tab where you can enable grid cutting with a grid cutting style (tangential motion, head down, no piercing). Enabling it results in much faster hole pattern cutting than traditional cutting with piercing and lead-in.



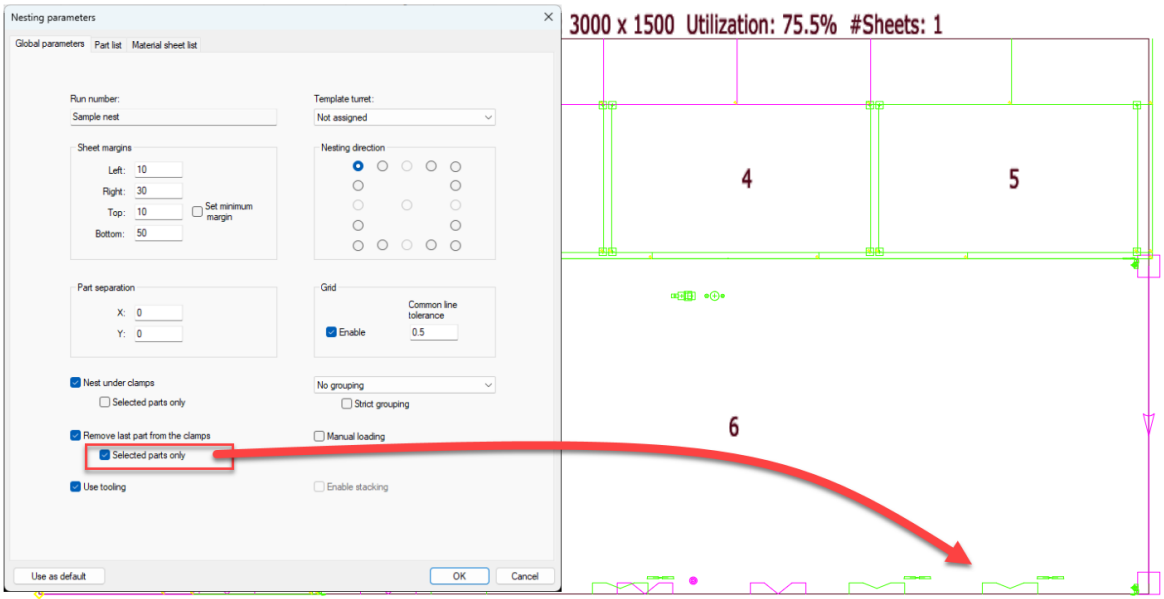
The COMBI, punch-shear machines

Activating Autotool's "Nest under clamps" setting

You can activate the ***Nest under clamps*** setting under ***Autotool - Settings*** to only place large parts on the bottom right corner of a sheet.



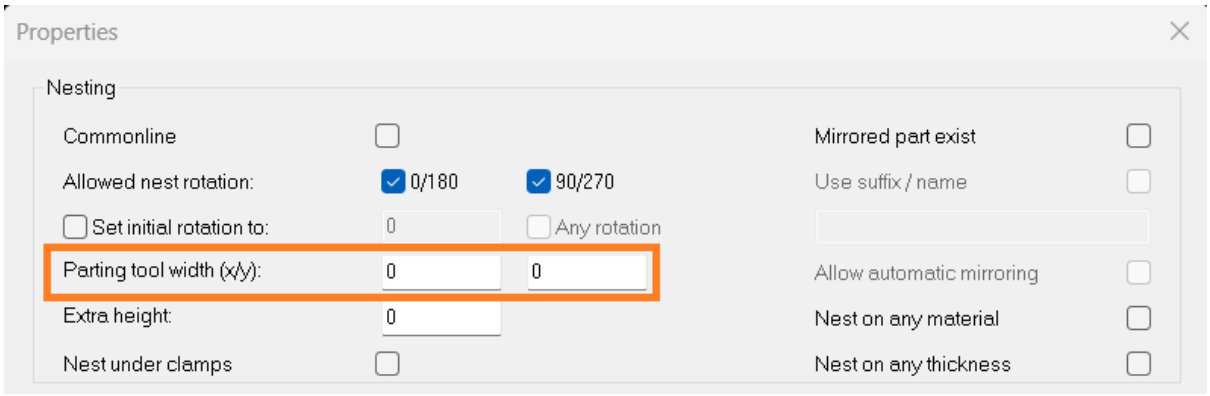
This setting has an effect only when you also activate ***Nest - Global parameters - Remove last part from clamps - Selected parts only:***



Parting tool width in nesting

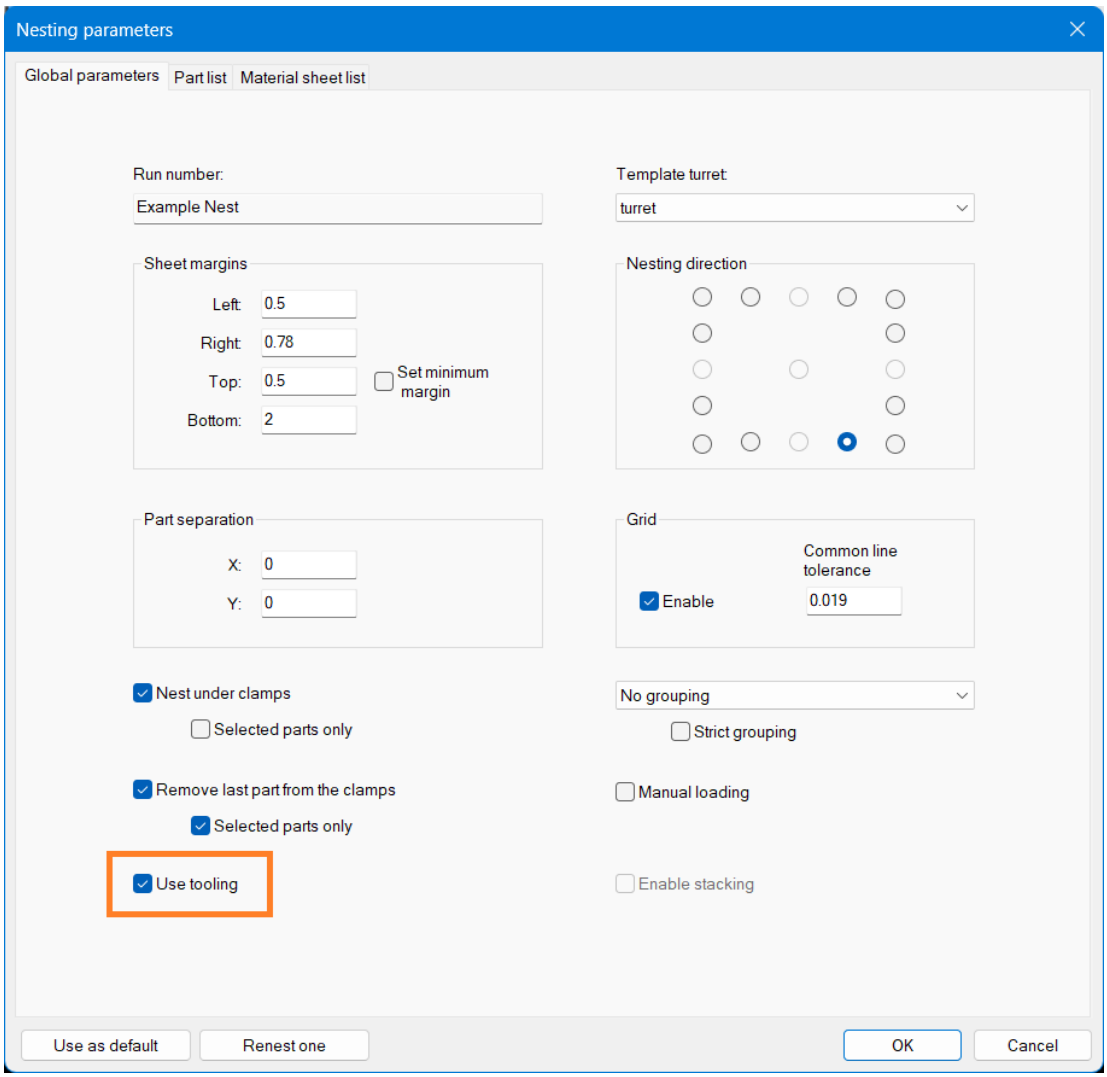
The rectangular nester for punch-shear machines has been improved in NC Express 24.1. See the below example:

You have a part that is not going to be common lined. The part, however, requires some extra space between itself and the adjacent parts on the sheet. This might be done to make the sheet's skeleton around the part stronger, for instance. To make this happen automatically in the nesting step, you can use the parting tool widths for x and y directions to set desired extra separation values respectively for the part. The parting tool widths can be set in the Part Properties dialog presented below. The input fields are highlighted with an orange box.



Using tooling envelope off/on in nesting

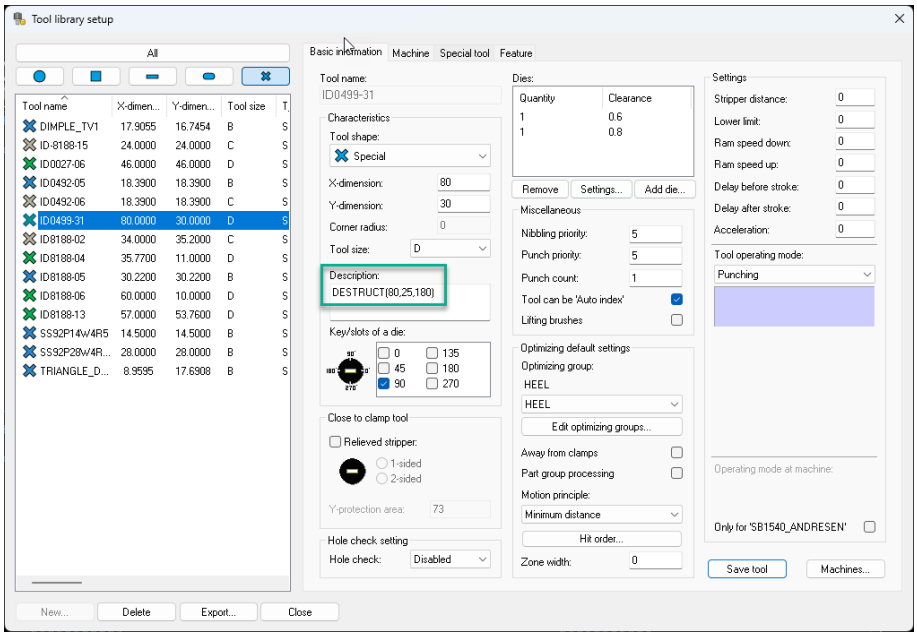
The rectangular nester for punch-shear machines uses tooling envelopes by default when it is positioning parts onto a sheet that already contains parts. In this way, the nester ensures that possible punch hits do not violate the geometries of any adjacent parts. In NC Express 24.1, you can choose whether tooling envelopes or part geometries are used when parts are positioned next to other parts. In other words, you can obtain better nesting results by using only part geometries if all parts to be nested are, for example, pure rectangle shaped parts, the punch hits of which are only on the inside geometries of the part and the outer geometry of which has only a shear releasing cut. When part geometries are used, it is easier to obtain parts next to each other if, for example, no part separation is given. This is, however, a very special case, and using tooling envelopes is usually recommended. The following ***Use tooling*** setting has been selected in the below nesting dialog. Otherwise, possible punch tool hits may violate the geometry of an adjacent part.



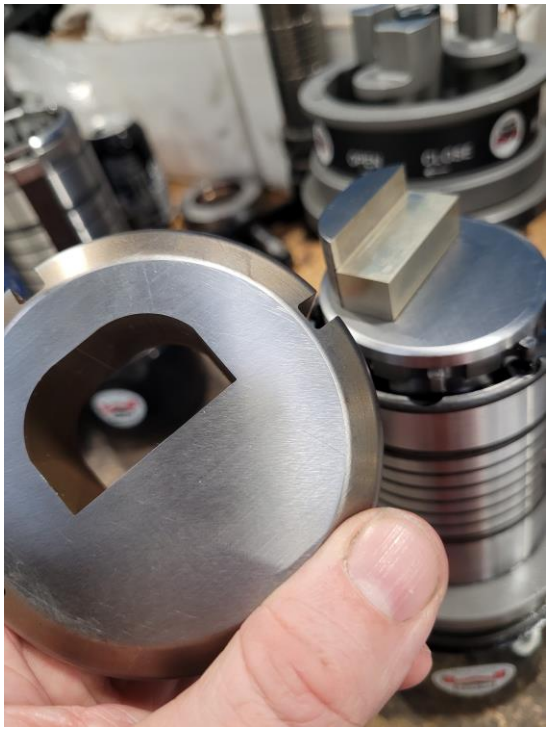
Destructing sheet edges with a special tool

You can configure automatic sheet edge destructing to also use a special tool.

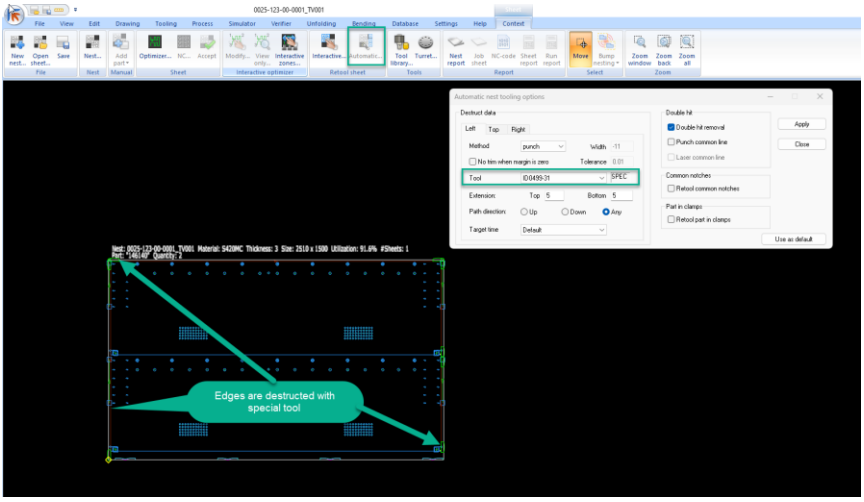
To activate the special tool for this purpose, you need to add a line to the tool library's ***Description*** field. The command line is `DESTRUCT(max_destruct_X_size, max_destruct_Y_size, tool_angle)`. For example, `DESTRUCT(80,25,180)`.



The tool is specially made for edge destructing, and it has the side support for punching narrow sheet edges for rather thick material. A normal rectangular tool will not last for long in this kind of use.



You can set automatic nest tooling to use this special tool when it has the description set to activate its usage as an edge destructing tool.



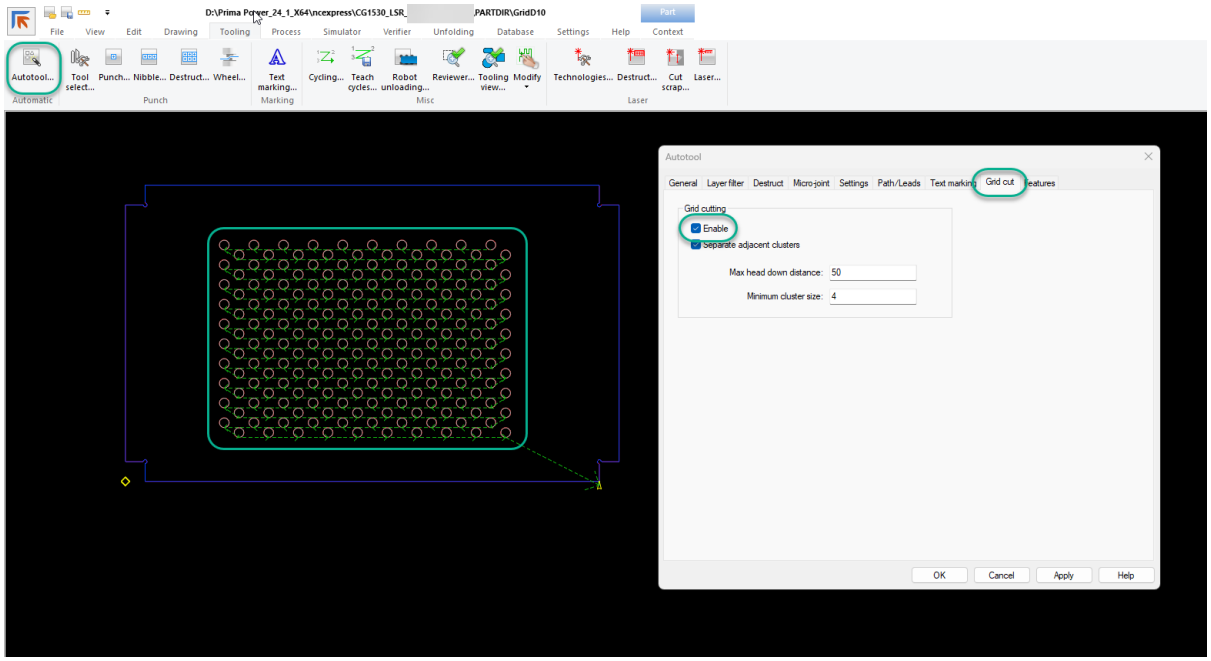
3000 mm sheets with trim strip chute on SB1530/1540 16-series

The optimizer has been improved so that it can automatically use the XCarrPos parameter to automatically run 3000 mm sheet to Trim Strip Chute (TSC option) without reposition. This saves time by approximately 3 seconds per sheet.

The COMBI, laser-punch machines

Grid cutting for round hole patterns with Autotool

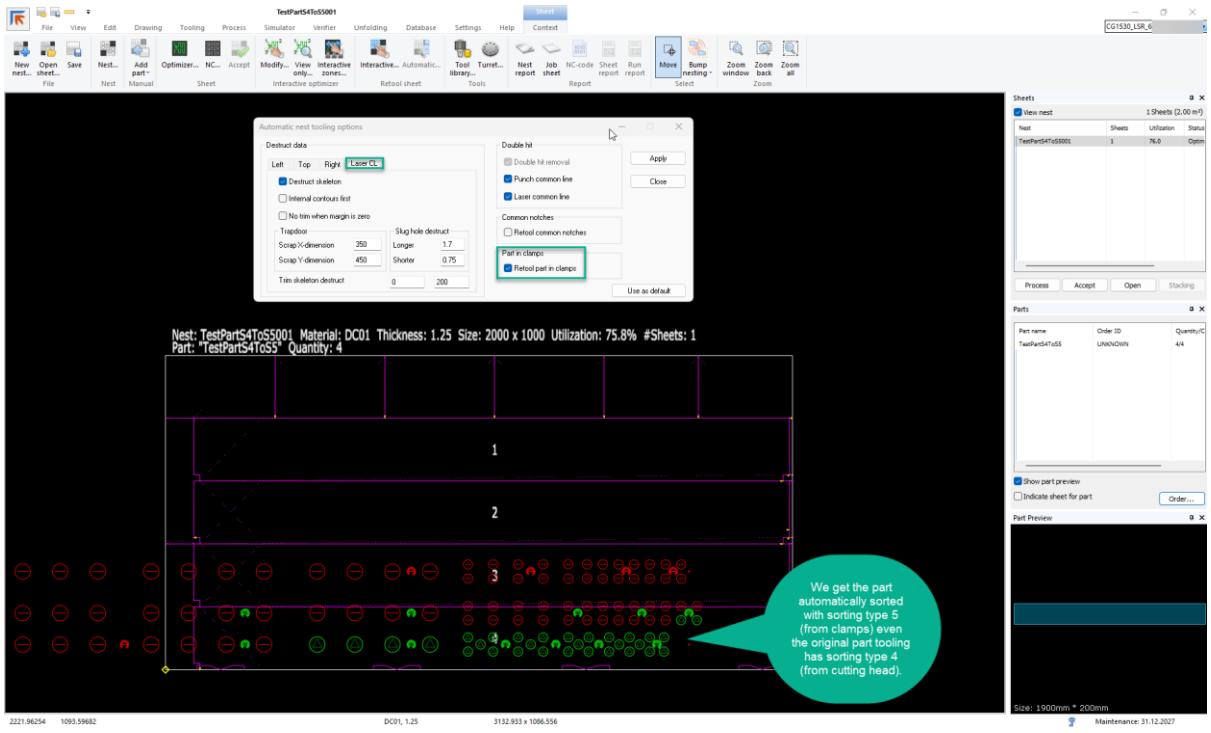
Autotool has the ***Grid cut*** tab where you can enable grid cutting with the DPM+ style (tangential motion, head down, no piercing). Enabling it results in much faster hole pattern cutting than traditional cutting with piercing and lead-in.



Automatically changing sorting type 4 to 5 when a part is in clamps

The optimizer has been improved so that it can automatically change the sorting type in the optimizing phase. It must be activated with the parameter SORTTYPE_4_LIKE_SORTTYPE_5 1 in the robot configuration file (.rob in LIB-folder). This is convenient when a common-line, skeleton-free nest is used. Previously, you had to do the part nested to clamps manually with suitable tooling and sorting type 5 (a robot picks the part from clamps).

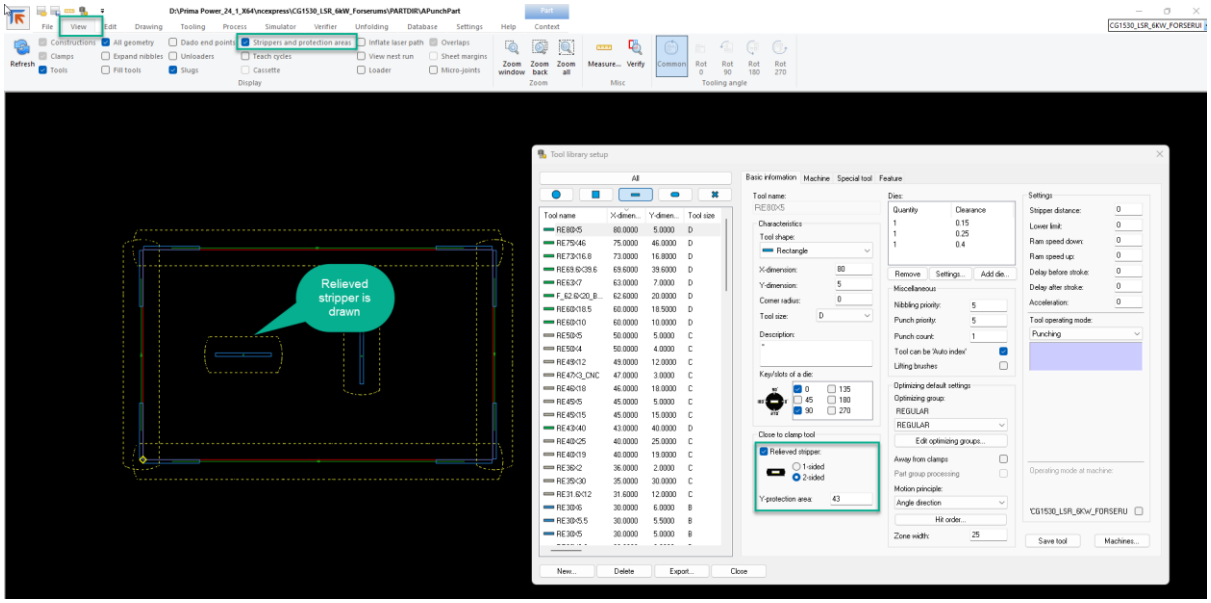
See the below example of a common-line nest with the automatic nest tooling option **Part in clamps - Retool part in clamps** enabled. You can see the sorting type 5 automatic robot placement after optimizing in the same manner as with the last scrap automatic robot placement on shearing machines. You cannot edit the automatic robot placement.



The PUNCH

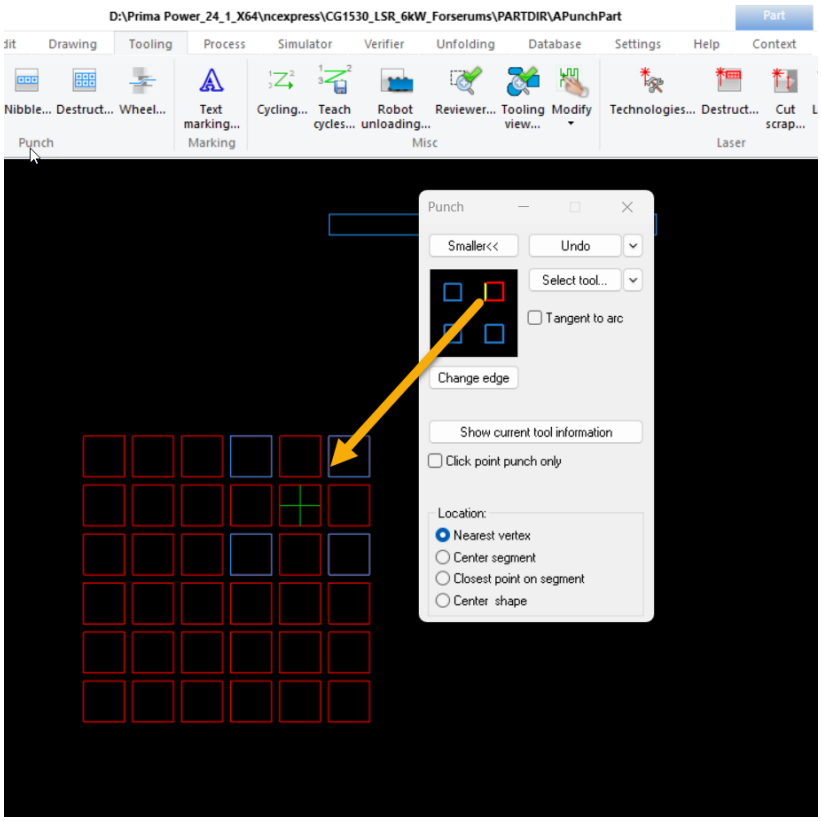
Viewing a close to clamp tool stripper

A close to clamp tool with a relieved stripper is now viewable with true shape. Nibbling lines are drawn as a combined stripper area, and single hits are drawn with true shape. This works in both the part and nest level.



Improvements on cluster tool placement

The “active” contour of a special tool with multiple contours is now drawn differently (red while the secondary contours are blue) in the tool preview of the interactive tooling punch dialog. This helps the manual tooling because you can see which is the active contour and edge in the cluster tool.

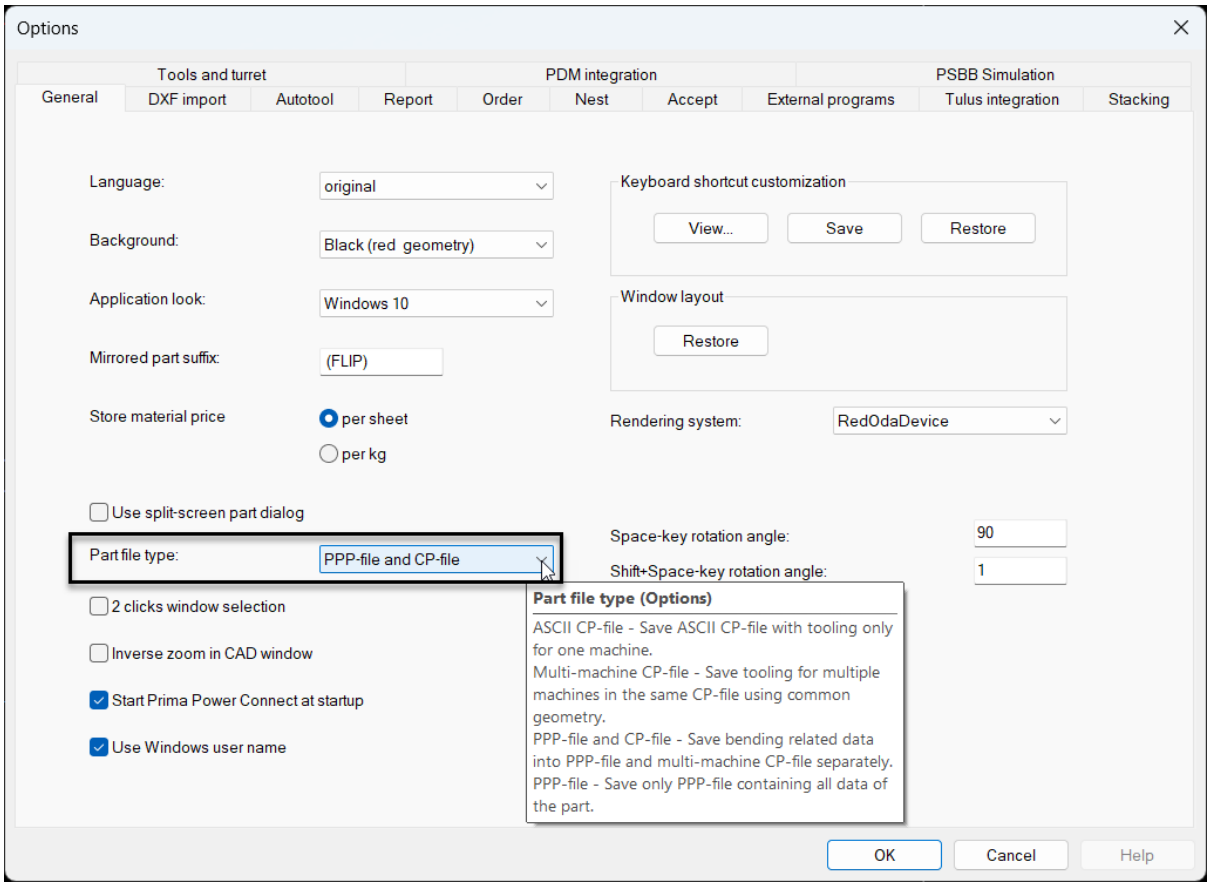


Other usability enhancements

PPP part file syntax

You can choose the part file syntax according to your needs. When NC Express bending is programmed with NC Express, we need a PPP file to keep the bending geometry and the tooling. The recommended setting is the PPP file and the multi-machine CP file. You can also preview PPP files.

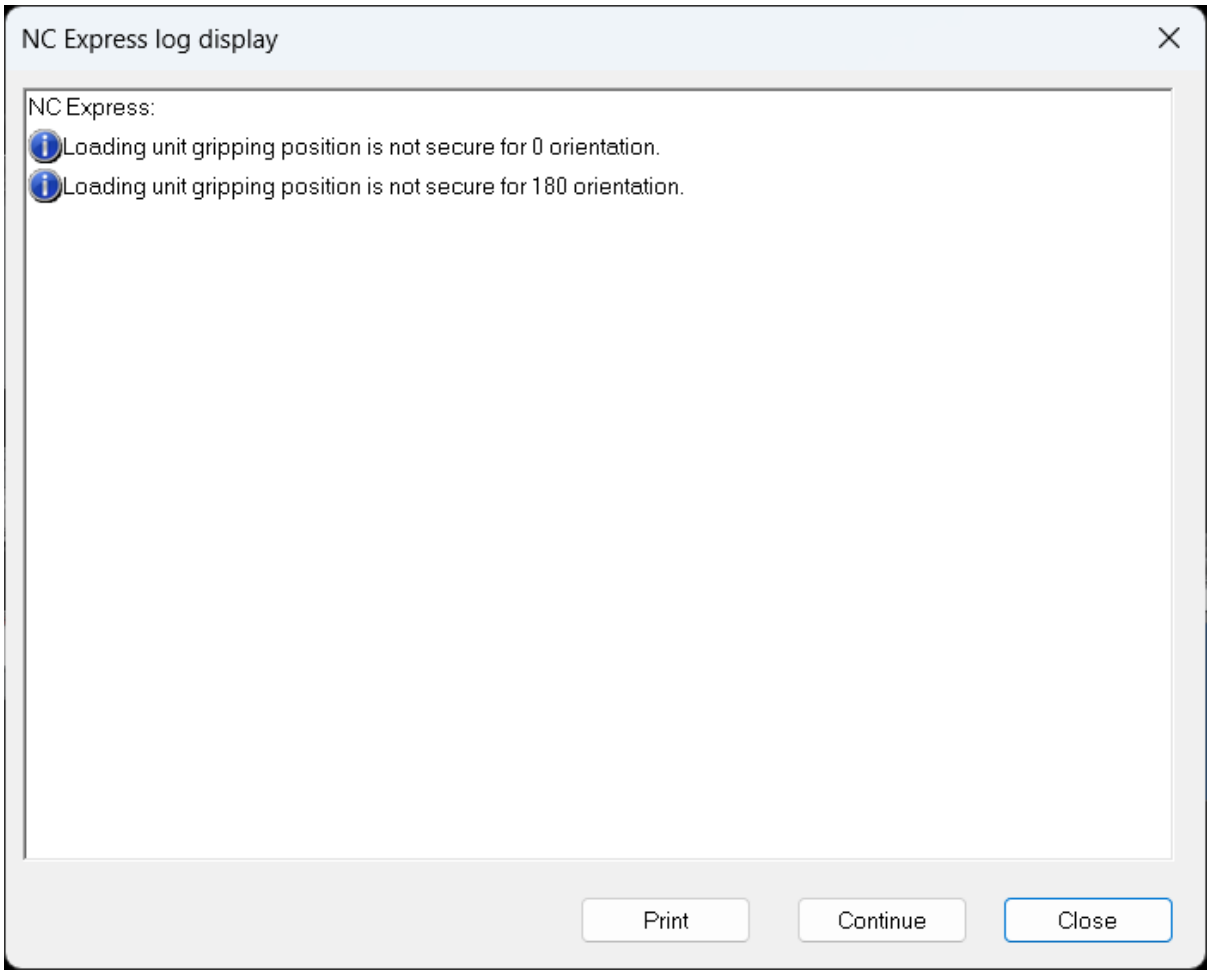
PPP = **P**rima **P**ower **P**art



Improved log display dialog

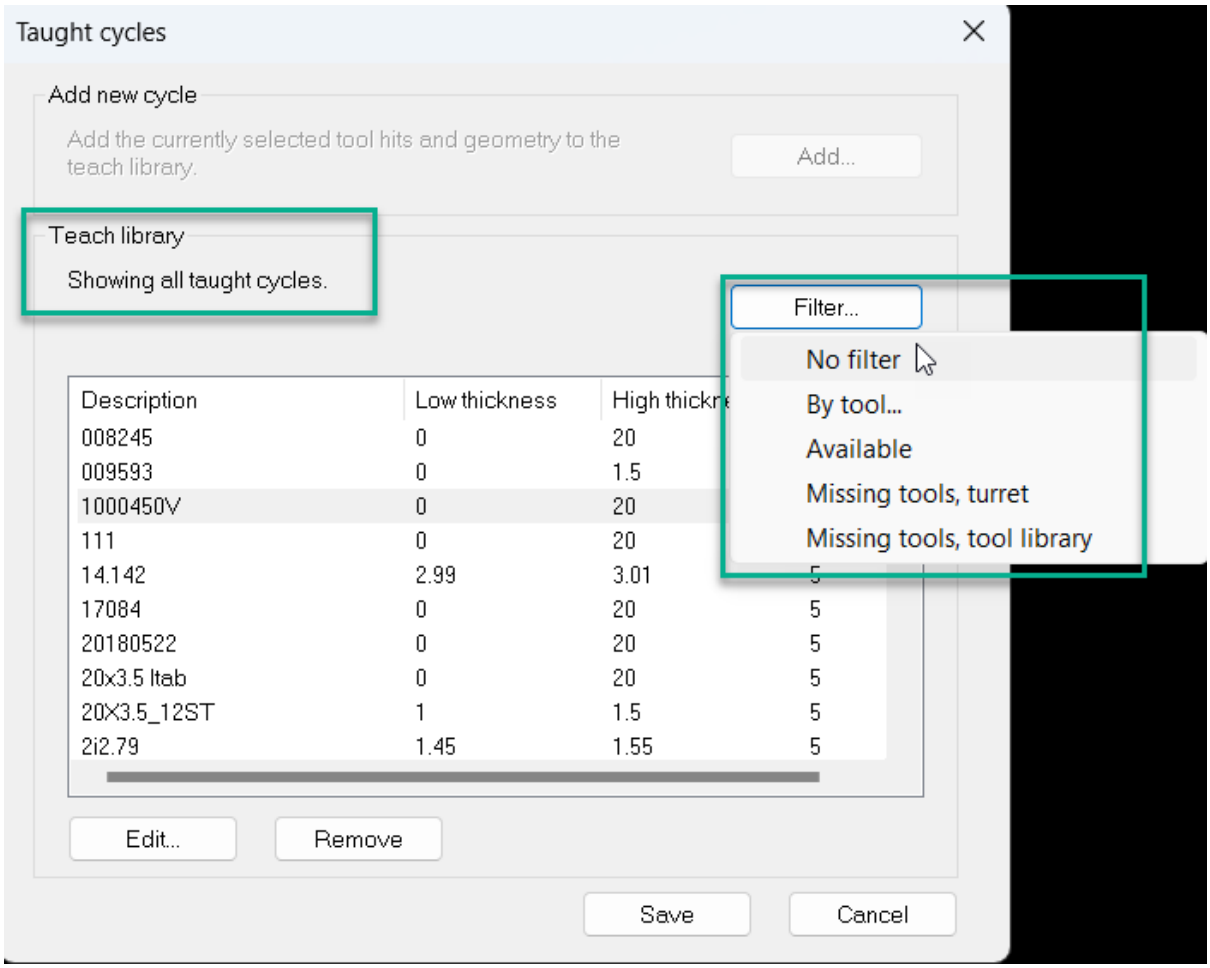
The displaying of multiple messages in the ***Log display*** dialog has been improved:

- Shows the severity of each message (info, warning, etc.) and the icons from the log panel.
- Consecutive messages with the same title are grouped to avoid duplicating the title.
- *The* ***Continue*** button unblocks the program without closing the dialog, and the ***Close*** button closes the dialog.



Filtering teach cycles

Long term request for filtering ***teach cycles*** is now implemented!



You can filter teach cycles by different criteria:

No filter: Shows all teach cycles.

By tool: Opens the tool selection dialog. After selection, you see only the teach cycles that use the selected tool.

Available: Shows teach cycles that have available tools in the active turret with the die clearance allowed for the actual material.

Missing tools, turret: Shows teach cycles that do not have available tools in the active turret or the die clearance of which is out of range for the actual material.

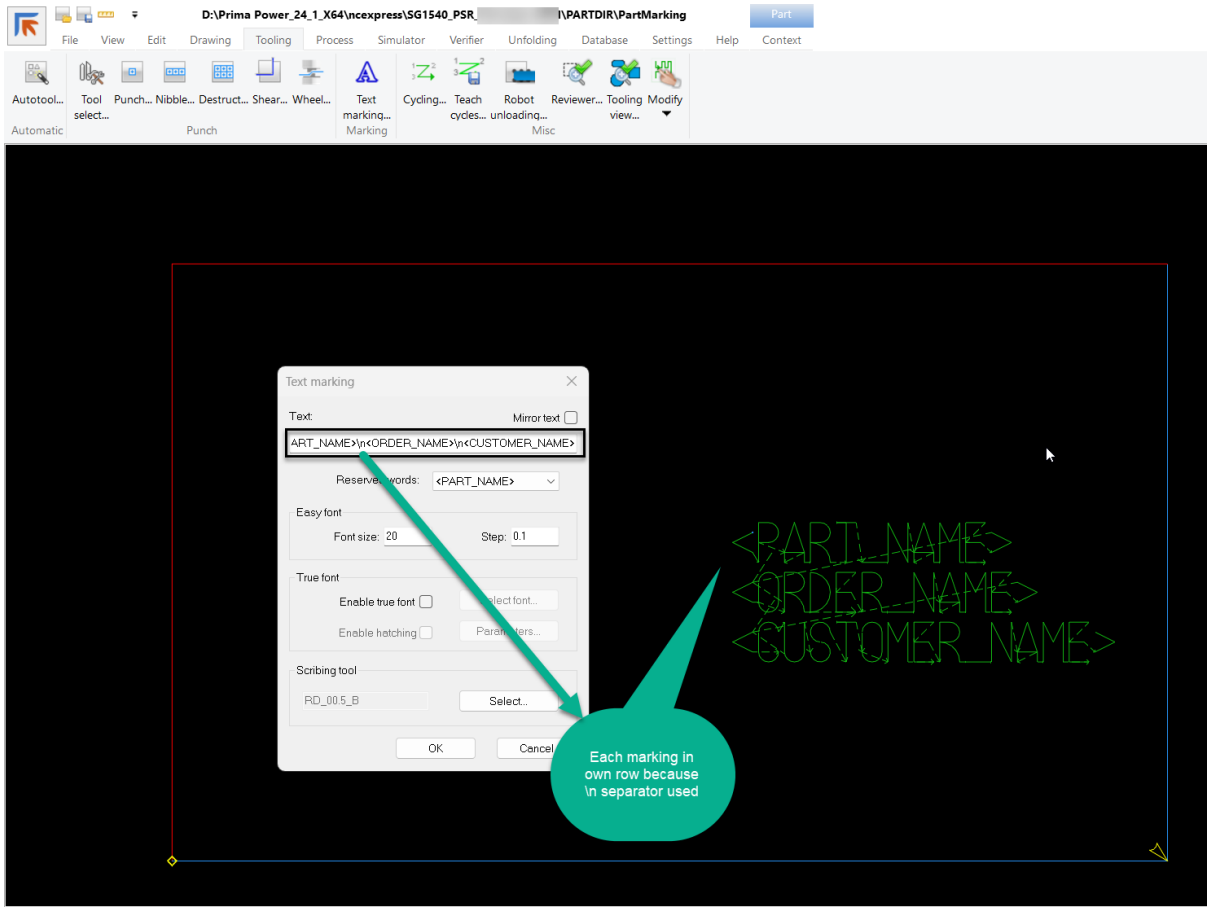
Missing tools, tool library: Shows teach cycles whose tools are not in the tool library anymore.

Text marking substitute for many reserved words

Text marking can now substitute many reserved words in the ***Text marking*** and ***Autotool - Text marking*** dialogs:

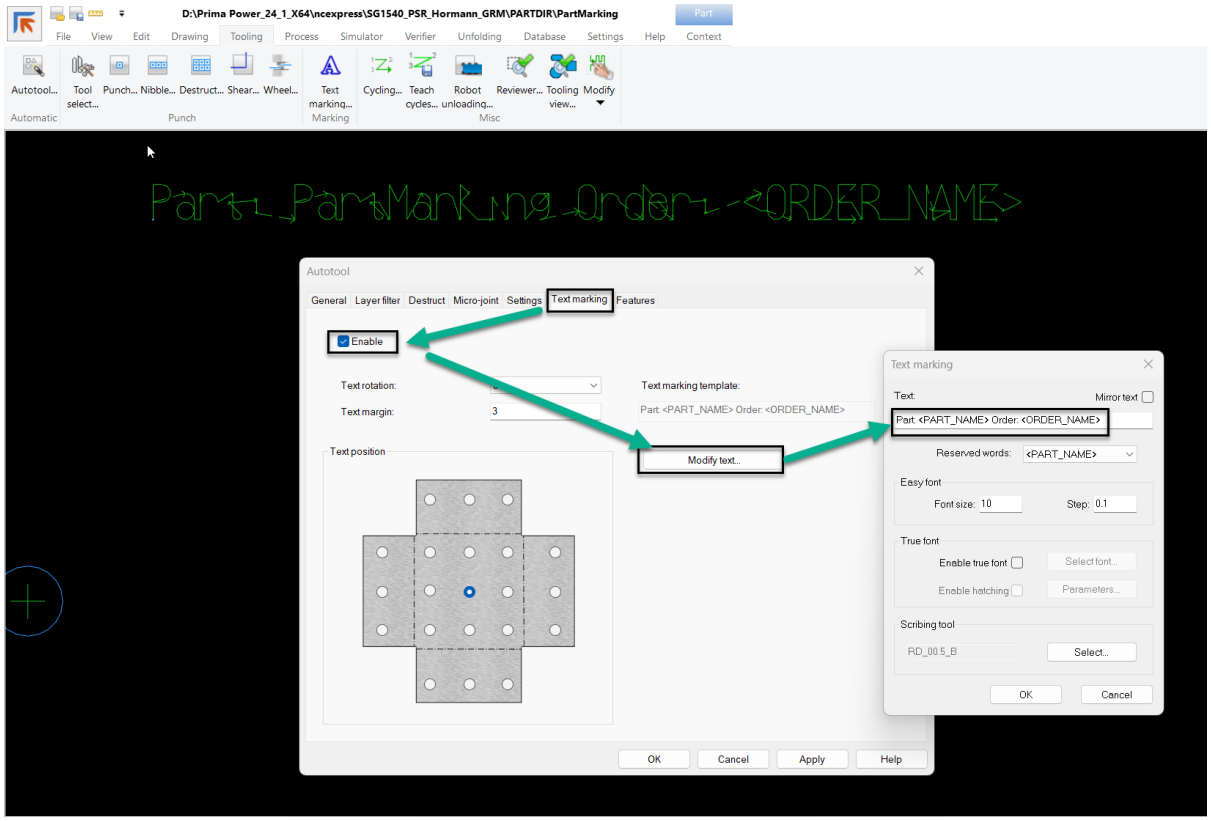
A new line is defined by adding the `\n` separator between reserved words.

Below is an example of three reserved words each in their own lines while using the separator \n.

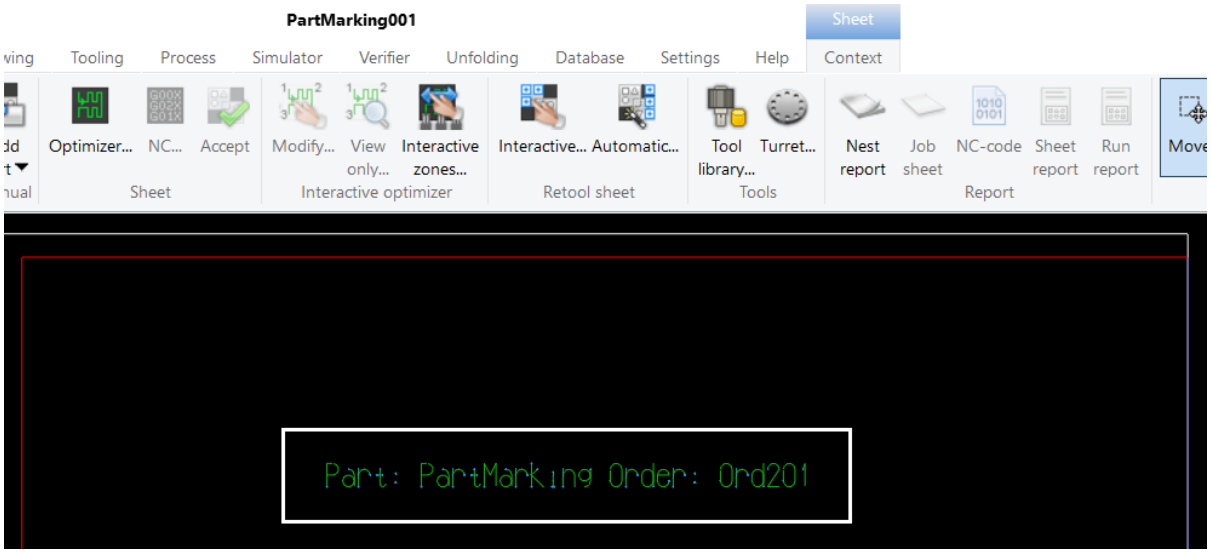


You can also add free text between reserved words if necessary.

Below is an example made with ***Autotool - Text marking***:



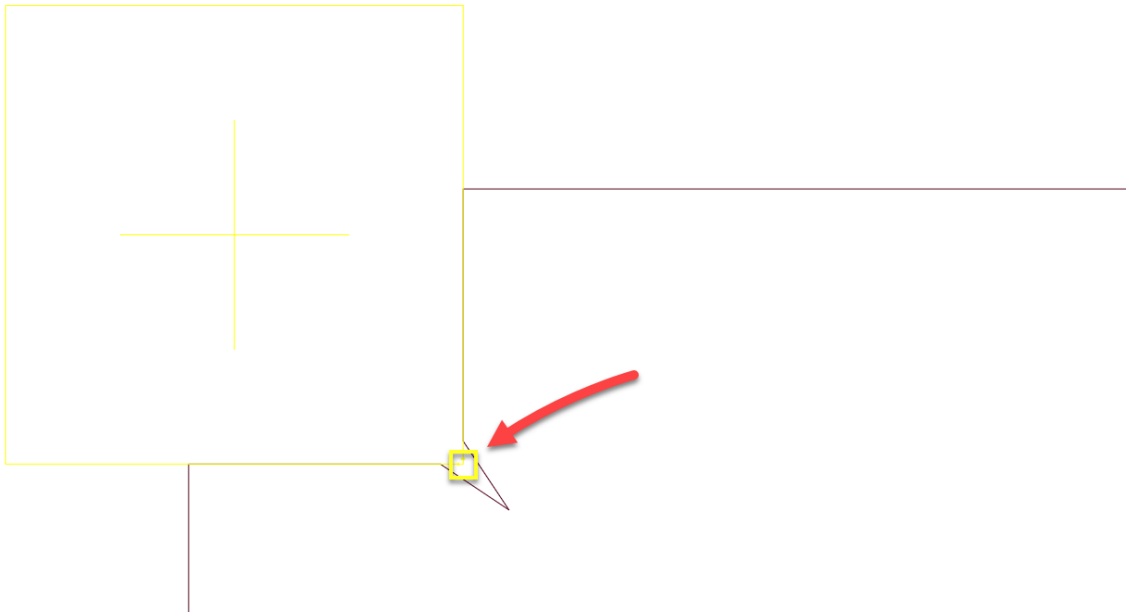
The result after nesting when reserved words are substituted with final data:



Snap to extension of lines

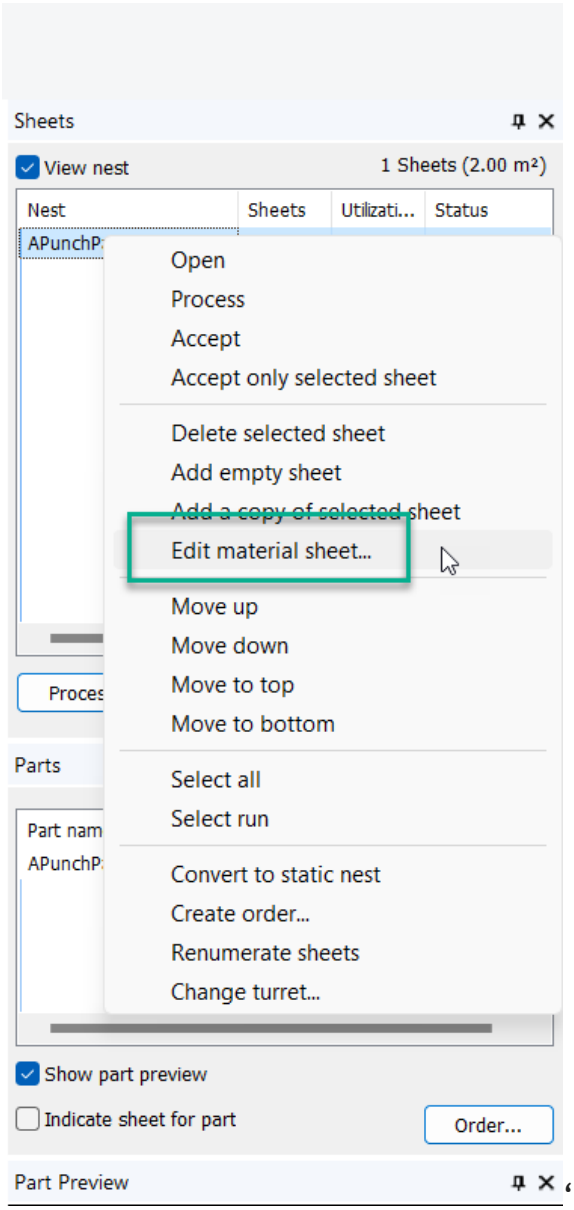
When drag-and-drop tools' snapping activates on "apparent intersections", note the following:

- If a snapping point does not appear, zoom outwards. This will include more line segments in the calculation window.
- Drop the tool inside the yellow square viewfinder.

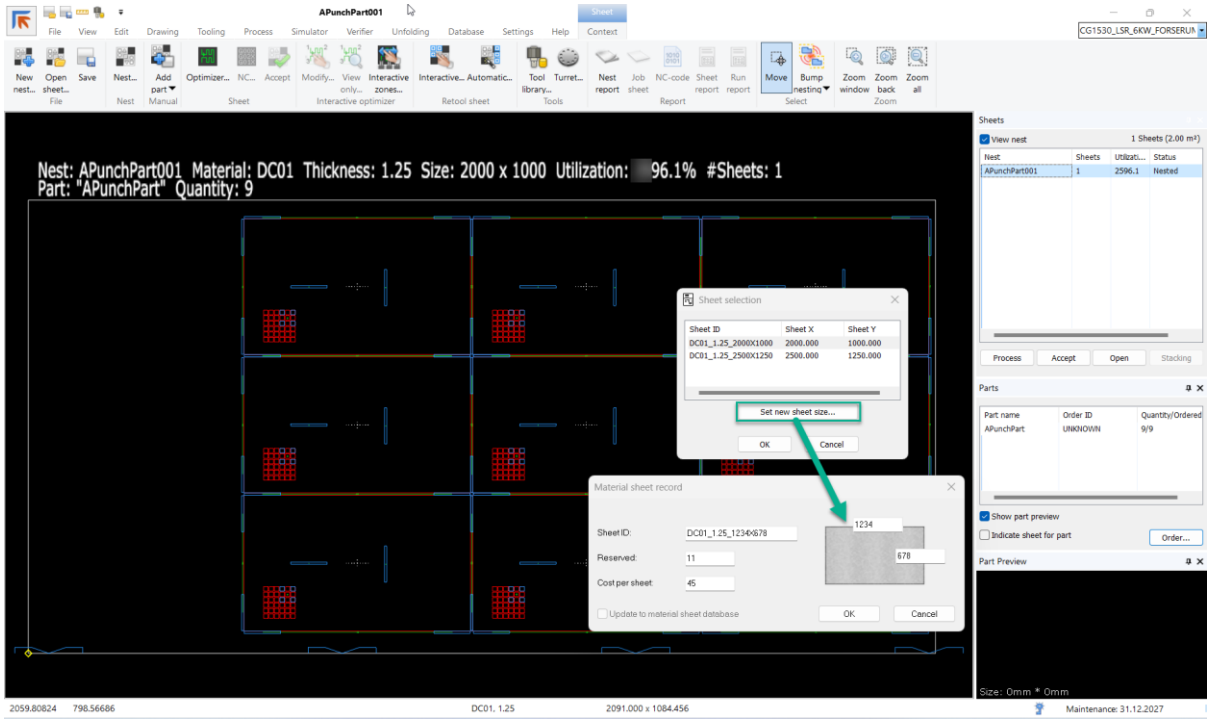


Changing the material sheet ID of an existing sheet/nest

The nest explorer's **Sheets** context menu has a new **Edit material sheet** functionality.

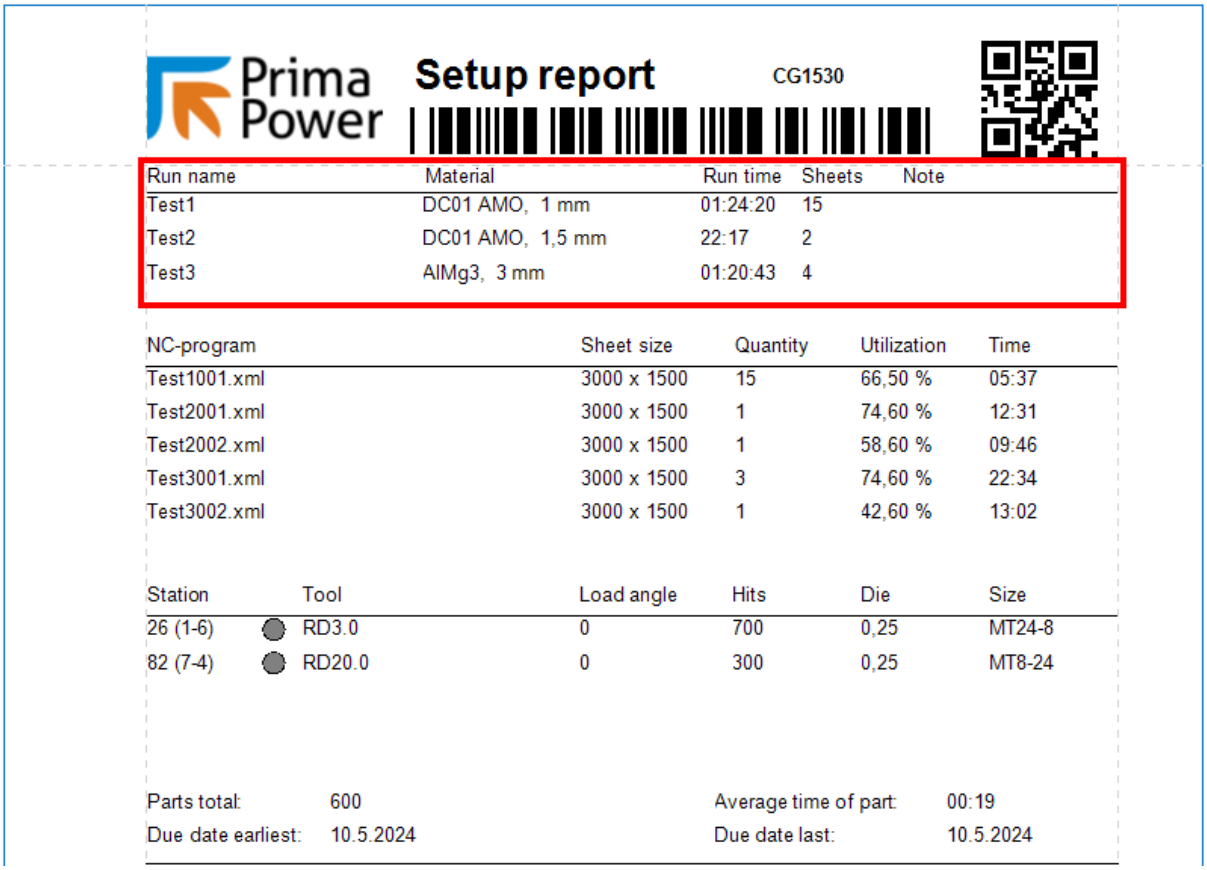


When you click the functionality, you open the **Sheet selection** dialog where you can select one of the existing sheet sizes. You can also create a new sheet size with the **Set new sheet size** button and enter the sheet size with the **Material sheet record** dialog. The Sheet ID is automatically filled in, but you can modify it freely according to your needs. That might be necessary when you have a sheet storage where material sheets are named according to your company policy.


APunchPart001
Nest: APunchPart001 Material: DC01 Thickness: 1.25 Size: 2000 x 1000 Utilization: 96.1% #Sheets: 1
Part: "APunchPart" Quantity: 9
Sheet selection
Sheet ID
Sheet X
Sheet Y
DC01_1.25_2000X1000
2000.000
1000.000
DC01_1.25_2500X1250
2500.000
1250.000
Set new sheet size...
OK
Cancel
Material sheet record
Sheet ID:
DC01_1.25_1234X678
Reserved:
11
Cost per sheet:
45
Update to material sheet database
1234
678
Sheets
View nest
1 Sheets (2.00 m²)
Nest
Sheets
Utilizati...
Status
APunchPart001
1
2596.1
Nested
Process
Accept
Open
Stacking
Parts
Part name
Order ID
Quantity/Ordered
APunchPart
UNKNOWN
9/9
Show part preview
Indicate sheet for part
Order...
Part Preview
Size: 0mm * 0mm
2059.80824
798.56686
DC01, 1.25
2091.000 x 1084.456
Maintenance: 31.12.2027

Report from multiple runs

In run reports, NC Express now has a new report template called “Multi run report”. Multiple runs that are in the same project are now shown in the report.



Prima Power

Setup report CG1530

Run name	Material	Run time	Sheets	Note
Test1	DC01 AMO, 1 mm	01:24:20	15	
Test2	DC01 AMO, 1,5 mm	22:17	2	
Test3	AlMg3, 3 mm	01:20:43	4	

NC-program	Sheet size	Quantity	Utilization	Time
Test1001.xml	3000 x 1500	15	66,50 %	05:37
Test2001.xml	3000 x 1500	1	74,60 %	12:31
Test2002.xml	3000 x 1500	1	58,60 %	09:46
Test3001.xml	3000 x 1500	3	74,60 %	22:34
Test3002.xml	3000 x 1500	1	42,60 %	13:02

Station	Tool	Load angle	Hits	Die	Size
26 (1-6)	RD3.0	0	700	0,25	MT24-8
82 (7-4)	RD20.0	0	300	0,25	MT8-24

Parts total: 600 Average time of part: 00:19

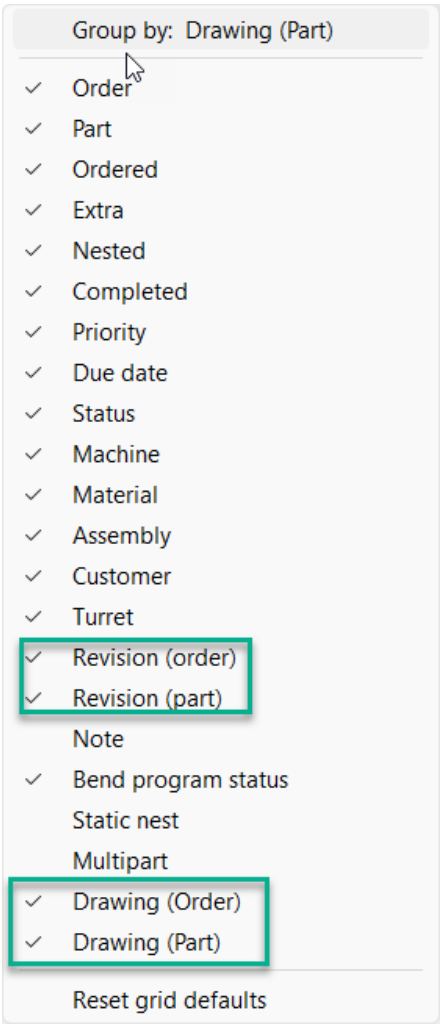
Due date earliest: 10.5.2024 Due date last: 10.5.2024

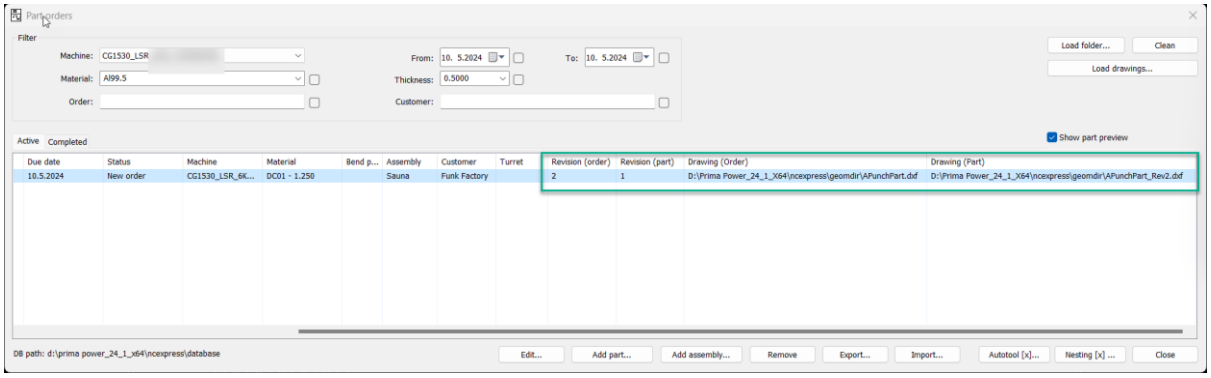
Full path to the drawing and actual revision in the Orders dialog

You can now see the full path to the part drawing and ordered part drawing.

You can also see the ordered part revision and actual revision of the part.

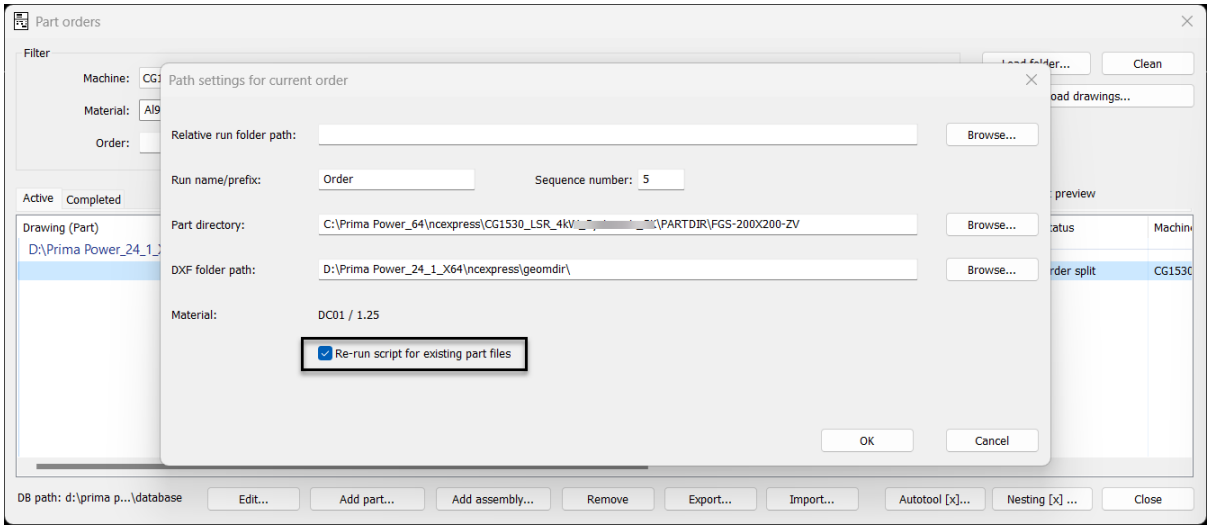
They are not active by default, but you can open filtering by clicking the right mouse button on the top order dialog header and set them on when needed.





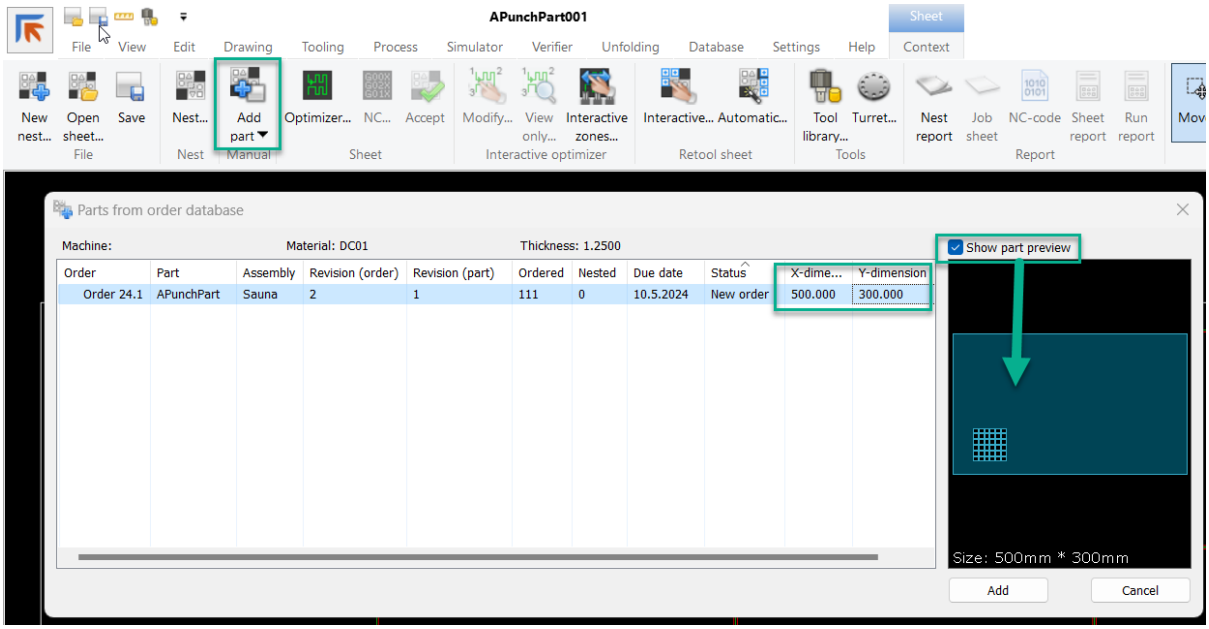
Option to re-run a script in the Orders dialog

You can now re-run a script for an existing part file to ensure that you always have a new and updated part file for orders.


Part orders
Filter
Machine:
Material:
Order:
Clean
Active
Completed
Drawing (Part)
Path settings for current order
Relative run folder path:
Browse...
Run name/prefix:
Order
Sequence number:
5
Part directory:
C:\Prima Power_64\ncexpress\CG1530_LSR_4kV
\PARTDIR\FGS-200X200-ZV
Browse...
DXF folder path:
D:\Prima Power_24_1_X64\ncexpress\geomdir\
Browse...
Material:
DC01 / 1.25
Re-run script for existing part files
OK
Cancel
DB path: d:\prima p...\database
Edit...
Add part...
Add assembly...
Remove
Export...
Import...
Autotool [x]...
Nesting [x] ...
Close

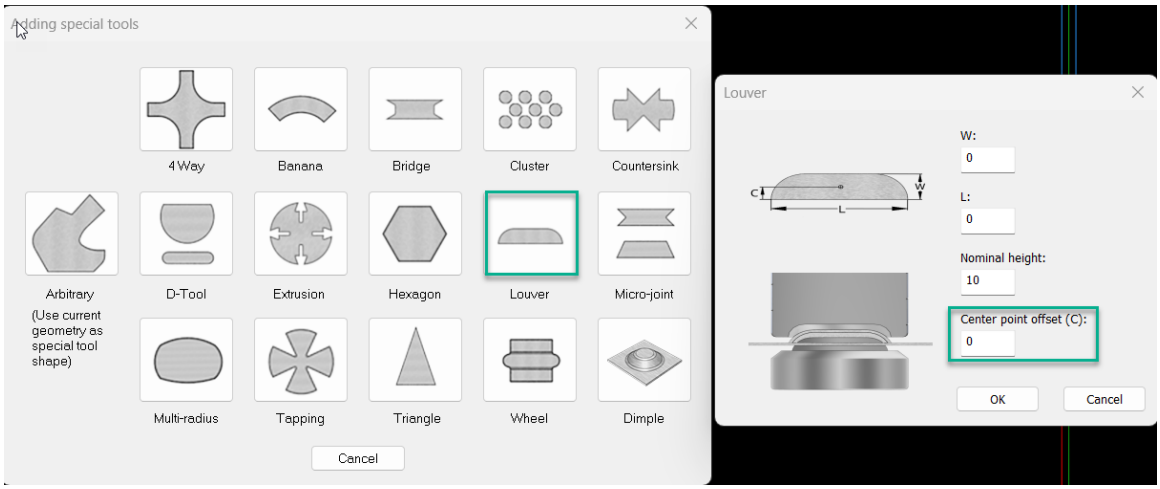
Improvements in “Add parts from order database”

When new parts from the order database are added to the nesting, you can now also see the part’s X and Y dimensions and a preview. This can help when selecting suitable parts to be added to available space on the sheet.

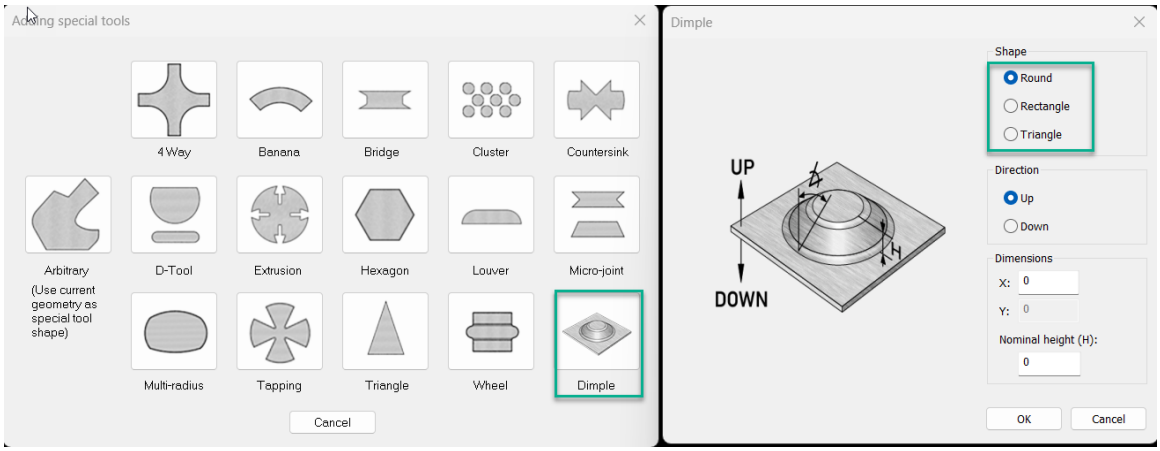


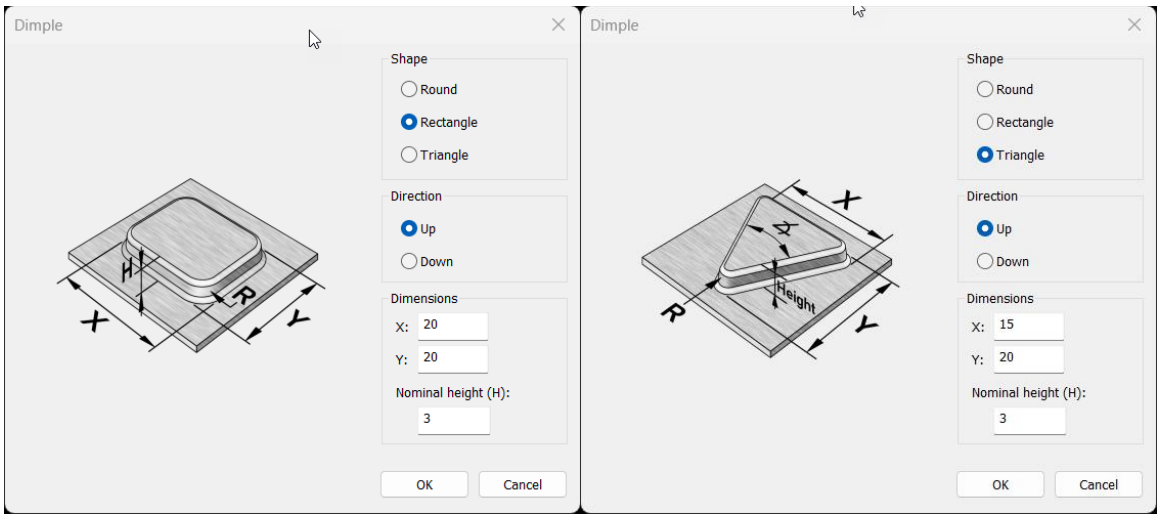
Enhancements to tool wizards

The Louver tool wizard has the new parameter ***Center point offset (C)***, which you can set to move the tool center point to correct position in a convenient way.



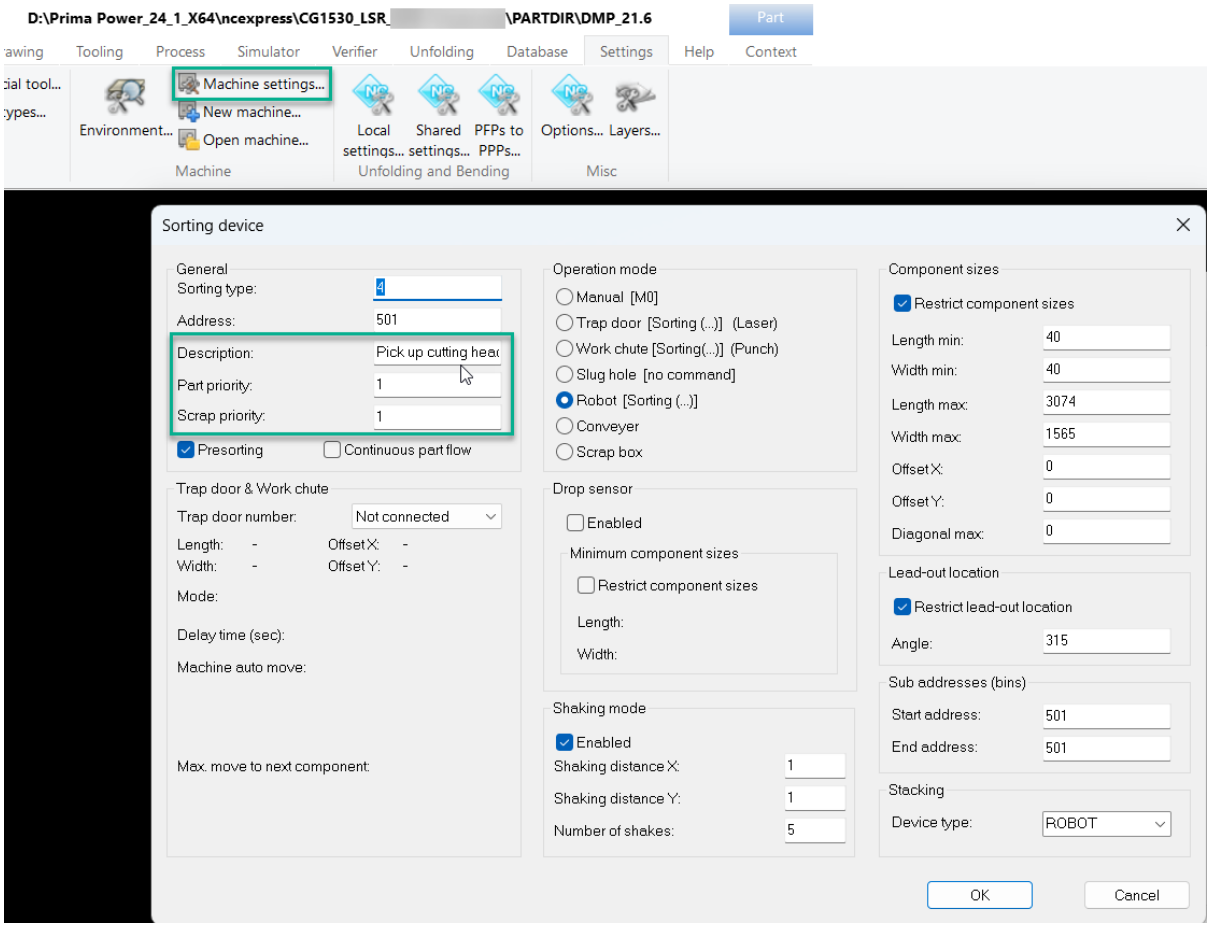
NC Express also has the Dimple shape special tool wizard, which can create three basic shapes of dimple tools: round, rectangle and triangle. The wizard also adds the values to the tool library according to the given values.





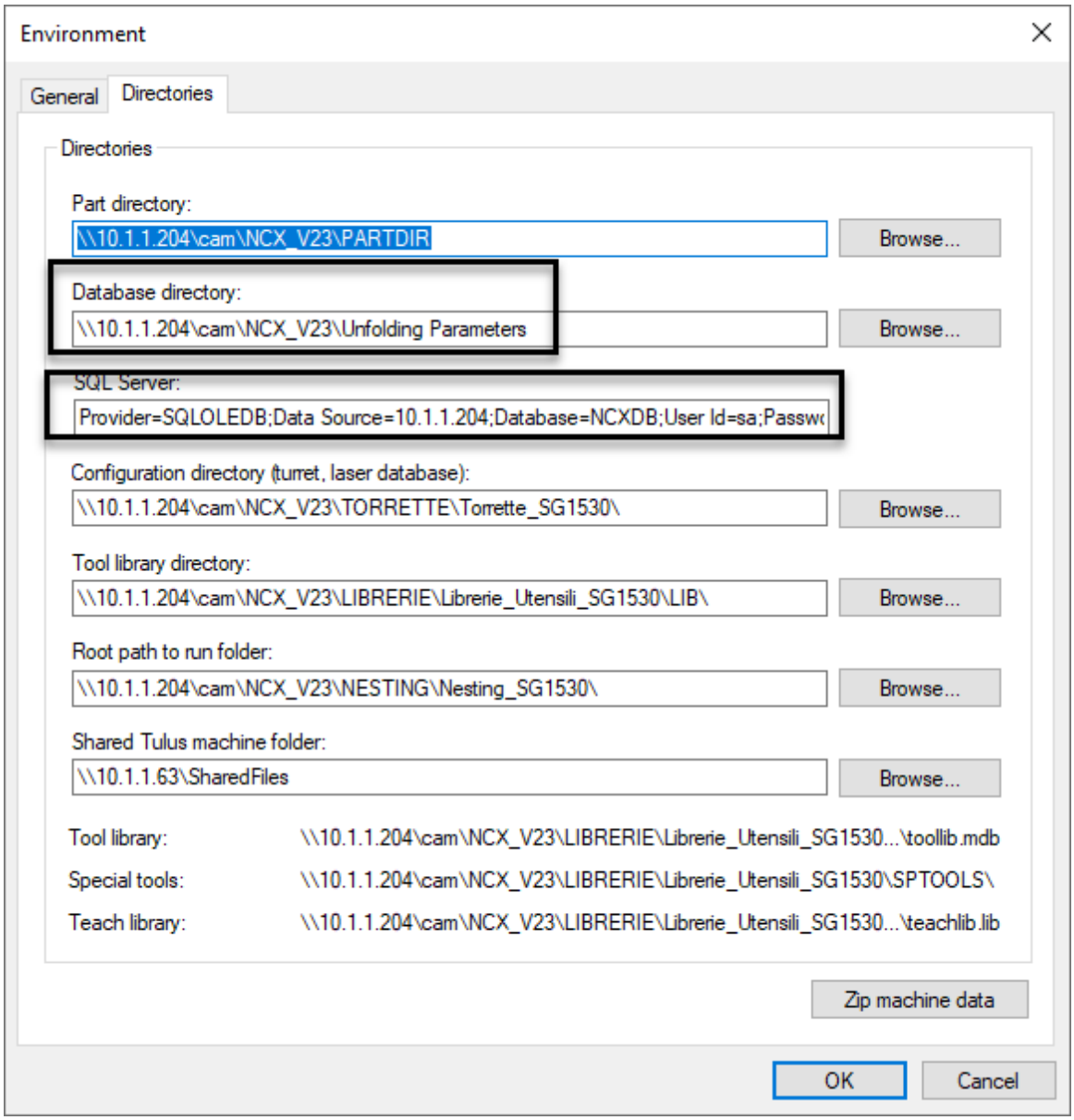
User-configurable sorting device description and priority

Users can now change the sorting device description and priority without giving the NC Express administrator password. This is convenient when you want to translate the sorting device description to your own language or to change the priority of the sorting device.



SQL server connection string

Environment settings for the SQL server connection string are separated from unfolding parameters. You can now set the unfolding parameters to “Database directory” and use a separate SQL server definition for PartDB, OrderDB, MaterialDB and NestDB.



New commands in PartNCX API

Method to add mapped properties

New method added PartNCX::SetMappedProperty(string propertyName, string propertyValue), which can be used in script as shown below:

```
Part.SetMappedProperty "Tested", "true"
```

Method to compare geometries

Get an object representing the similarity of the part to the given drawing:

```
PartSimilarity PartNCX.Similarity(lpFilename, bIncludeFeatures)
```

Symmetry of the part with the given error margin. Returns PART_SYMMETRY bitmask:

```
PART_SYMMETRY PartNCX.Symmetry(bIncludeFeatures, dMaxError)
```

Bend lines collection

New bend lines collection added:

`PartNCX.BendLines` method
`BendLineCollection` class
`BendLine` class

You can find more information in “NC Express PartNCX description.pdf”.

New commands in NestNCX API

PlateCollection.Accept method added with two parameters

Writes queue file (same as WriteQueue) and additionally based on iLevel:

```
long Accept([optional]VARIANT lpFilename, [optional]VARIANT iLevel)
```

+1 = Run accept.exe (default)
+2 = Write part estimates to part database (default)
+4 = Write to Nest DB, write order records completed, reduce raw material count

SheetProbing property set for plate

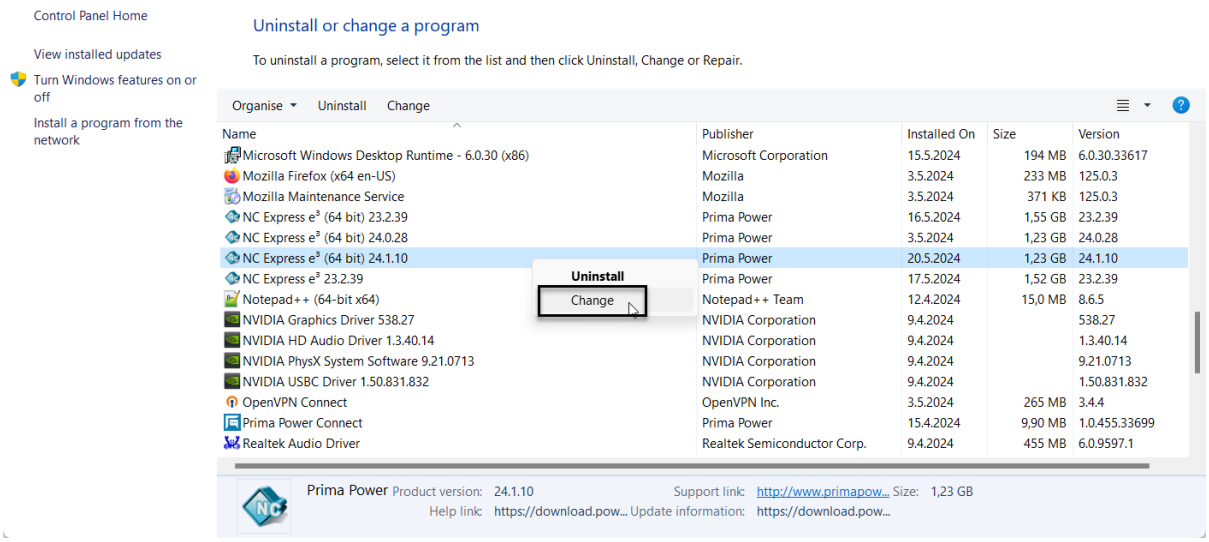
Optimizer reserves space around clamps and adjusts axes’ limits by the amount of sheet probing tolerance:

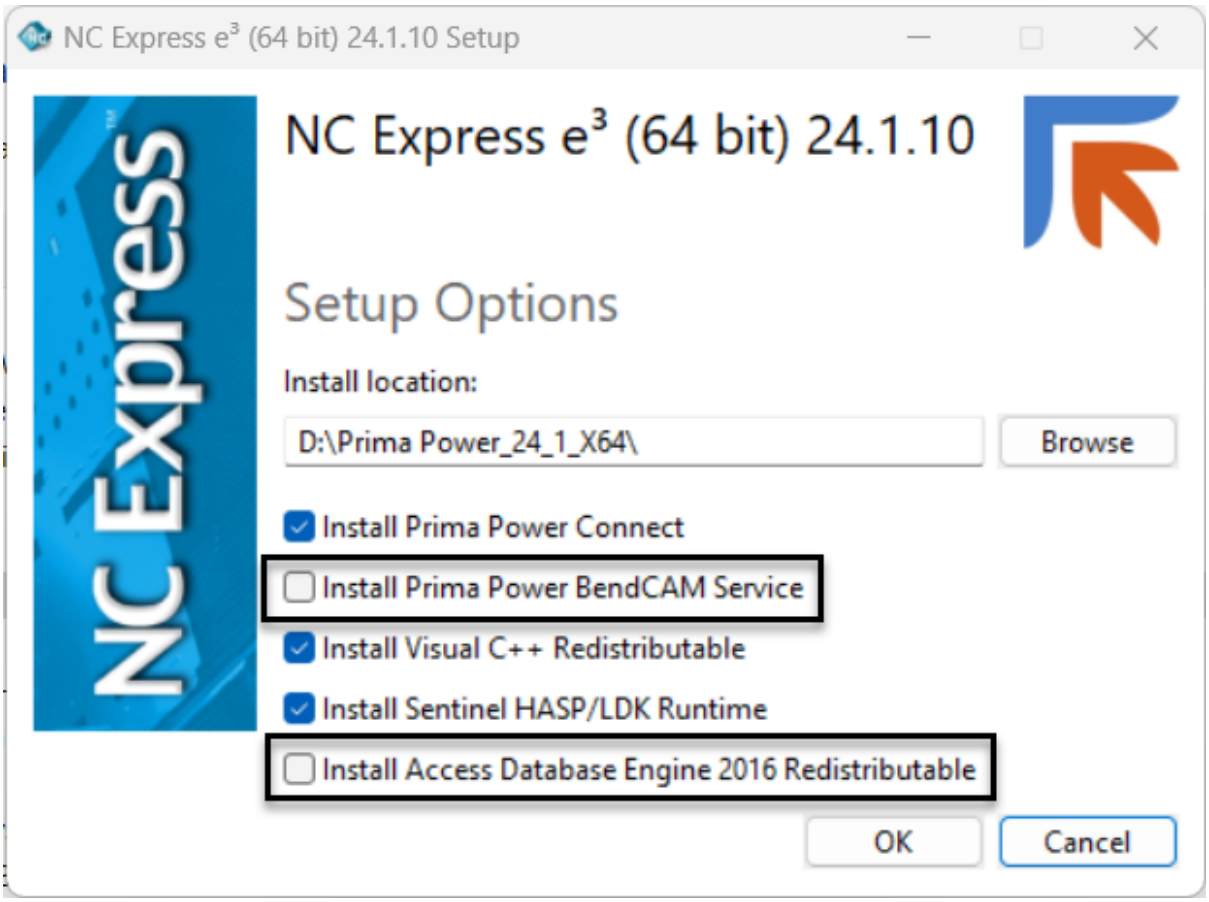
```
Plate.Punch.SheetProbing=True
```

You can find more information in “NC Express NestNCX description.doc”

Modify options in setup

Allow users to choose options when running the NC Express setup program again. This allows users to install Prima Power BendCAM Service and Microsoft Access Database Engine 2016.





Windows support

NC Express e³ 24.1 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 customized report templates, be prepared to redo those report templates for future reporting.