



# New mCAM features (version 2.0)

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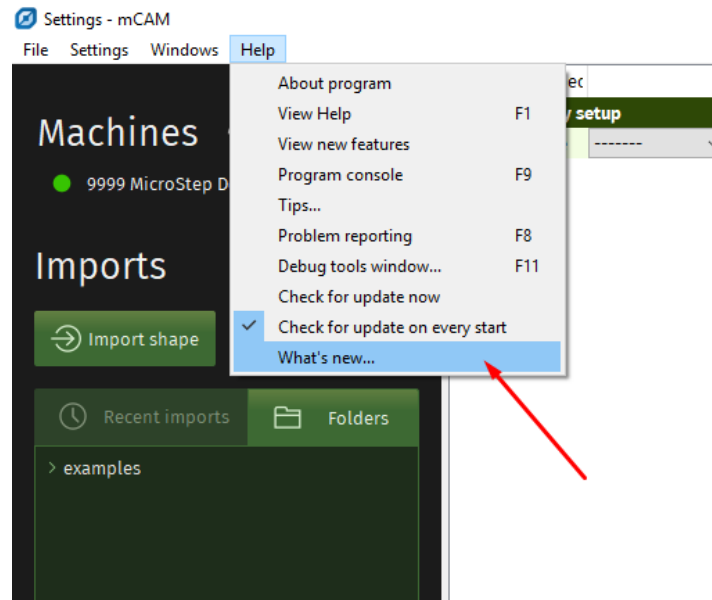
# Overview

New mCAM 2.0 brings many new features that are described in this document. Among these are:

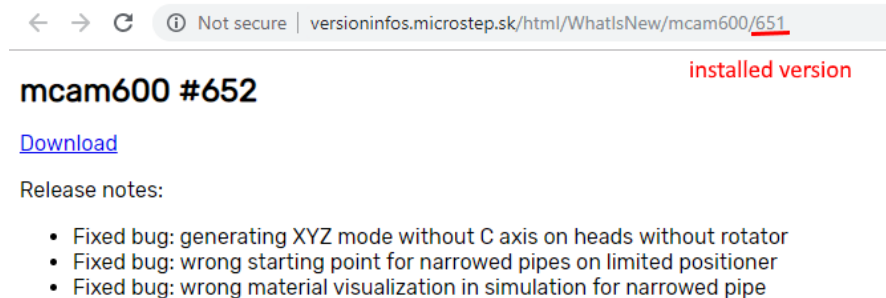
- more intuitive interface with new sets of controls (see [New interface](#), [Visualization screen controls](#), [Right control panel](#) and [Simulation screen](#))
- changes in initial position of beams on the machine (see [Initial beam position changes](#))
- manual nesting for positioning of plates and circular pipes (see [Manual nesting](#))
- possibility to adjust cutting speed for individual cut paths (see [Properties area](#))
- possibility to add welding preparations for existing cut paths (see [Welding preparations](#))
- recognition of several types of H-beam welding macros (see [H-beam welding connection macros](#))
- refined generation of Lead-ins/outs and minor changes in Measuring modes (see [Settings changes](#))

# “What’s new” feature

It is possible to check what is new in newer versions of mCAM (if they exist) directly from mCAM. This new feature is located under menu Help -> What’s new. This function will open a webpage in browser with information about new features in newer version of mCAM. If the most recent version of mCAM is installed, the page will contain text: “You have the latest version”.



**Fig. 1** What's new feature



**Fig. 2** Web page with information about changes in newer version of mCAM

# New supported shapes

Two new supported shapes were added in mCAM 2.0 – Bent U and L profiles.

## Technical explanation

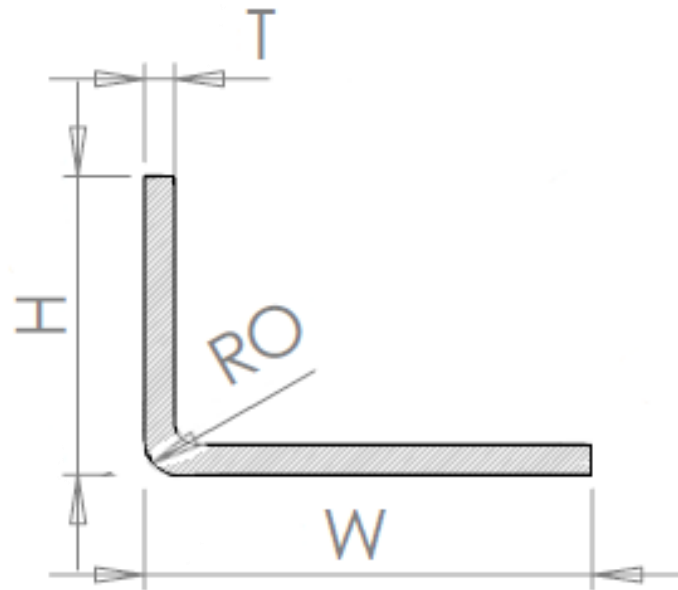
**W** – width

**H** – height

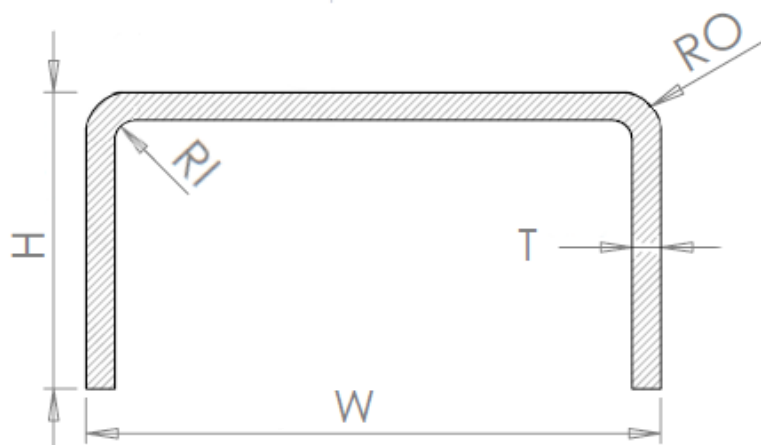
**T** – thickness

**RO** – outer radius

**RI** – inner radius



**Fig. 3 Bent L profile**

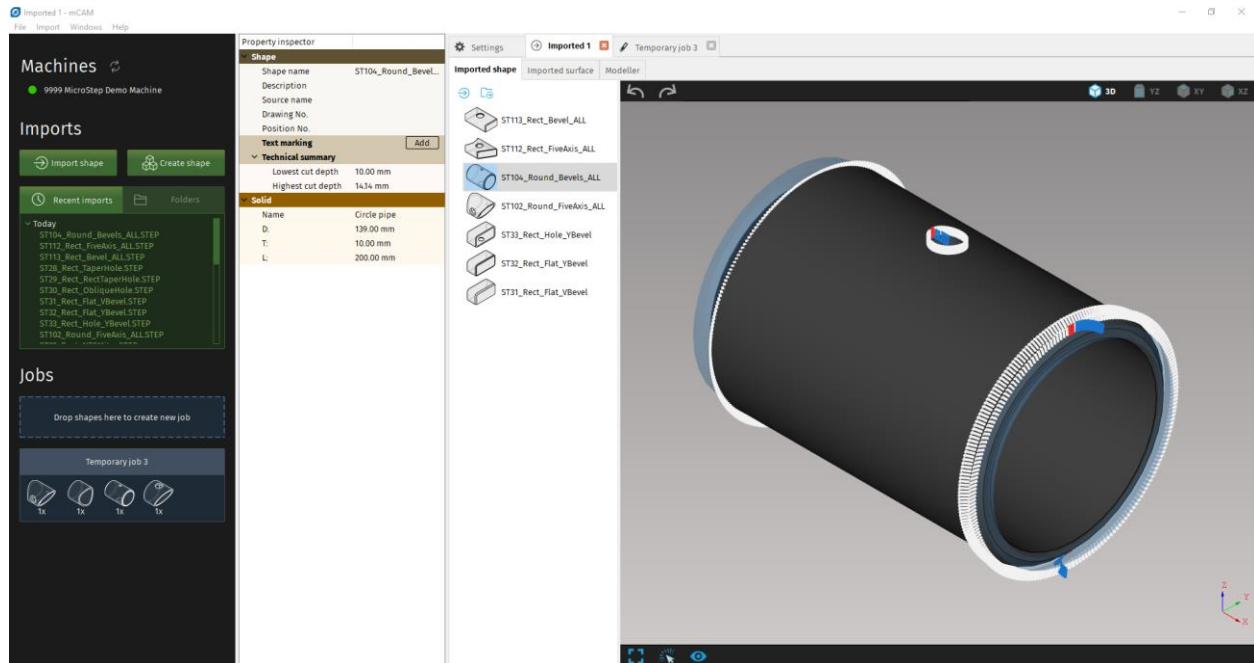


**Fig. 4 Bent U profile**

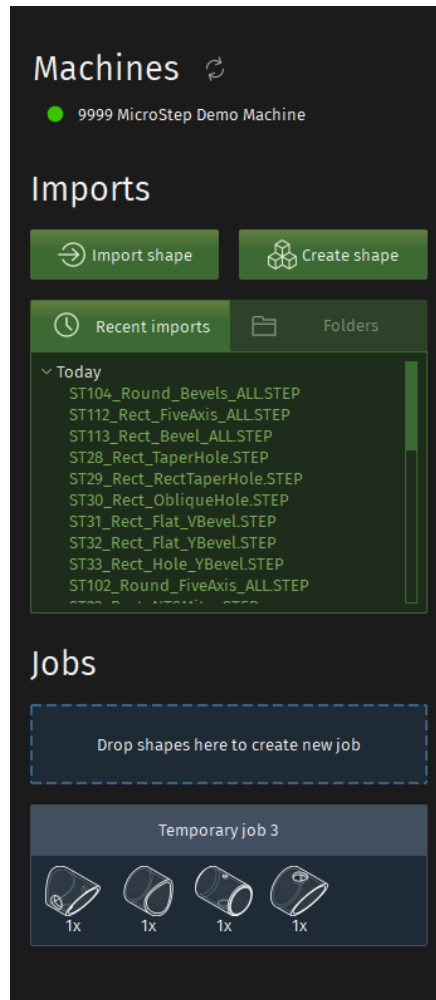
# New interface

## Master panel

The Master panel in mCAM 2.0 was redesigned to be more intuitive to use. The new interface provides user with more modern look and new set of controls.







**Fig. 5** New interface



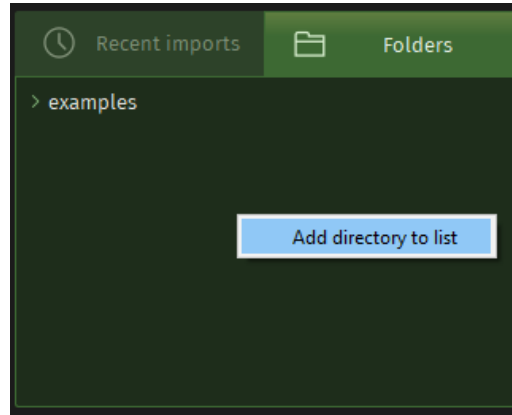
**Fig. 6 New master panel**

At top of the master panel is a list of connected machines and the actual status of connection.

Icon	Description
	Connection is active
	Connection is not active
	mCAM is connecting to the machine
	Refresh connection

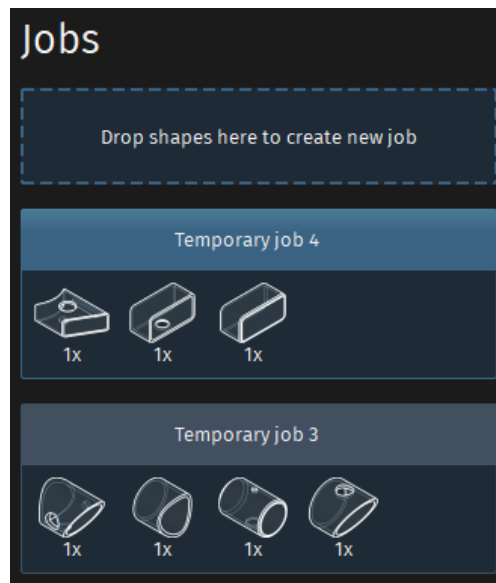
Underneath are two buttons, one for importing a shape and one for creating a shape in modeller.

Next is a list of recent imports and user's most used folders. A folder is added to this list by clicking the right mouse button in the list and selecting a folder.



**Fig. 7 Adding a directory to list of frequently used folders.**

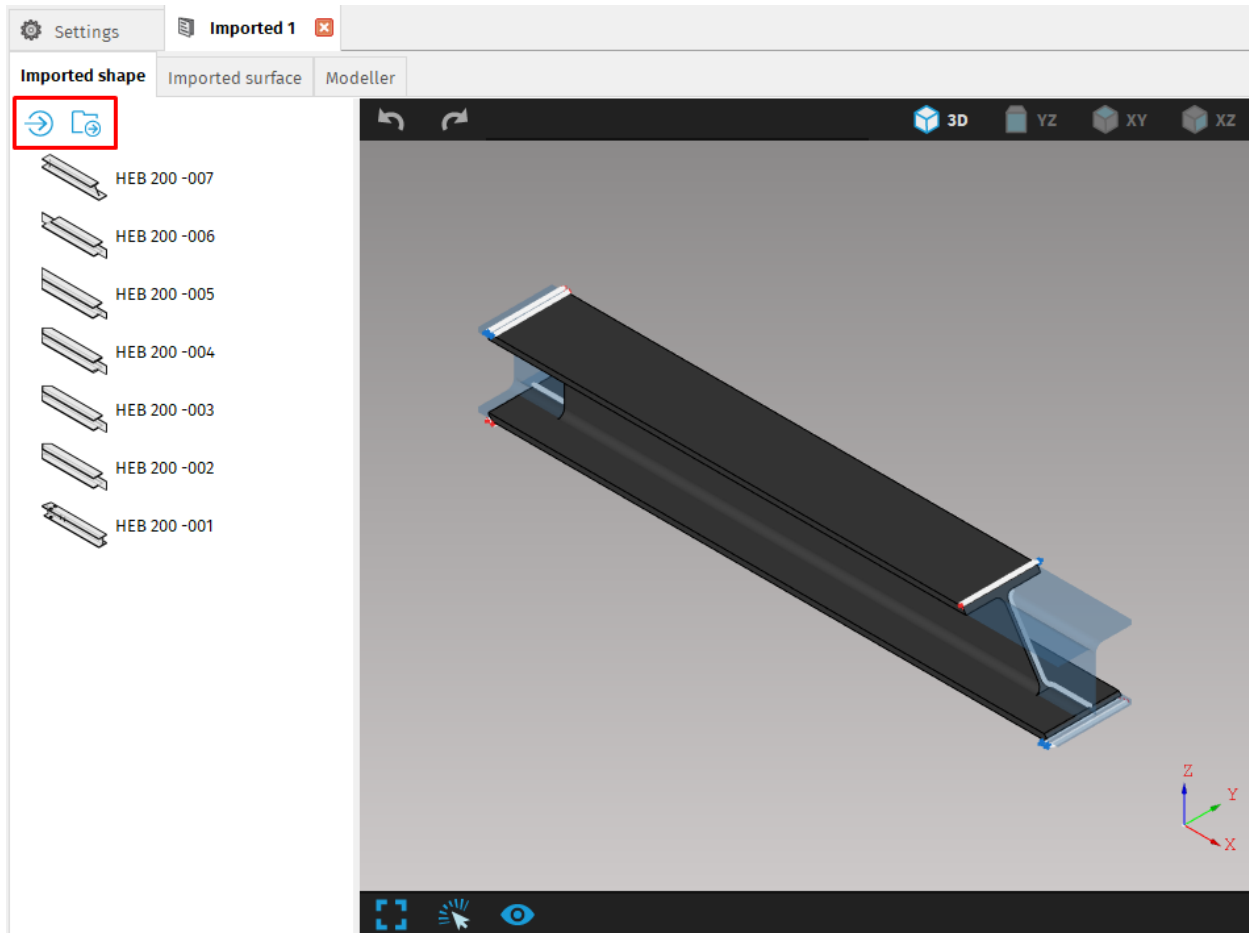
Creating a job works the same as in previous version of mCAM, but now there is a rectangle area where the parts have to be dropped. Dragging and dropping parts here will create a job, that will also appear underneath and will display all shapes that are included in respective job. Switch between jobs is possible by clicking on job header as well as in tabs from mail application window. Selected job is highlighted in brighter blue colour.





**Fig. 8 List of jobs under the area for creating jobs**

# Working area – Import tab

Import tab was also reorganized. The Import all button, that was located at the bottom of the Working area of Import tab, was removed. New buttons were added on the top of the Working area of Import tab for importing a shape and importing a whole folder of shapes.



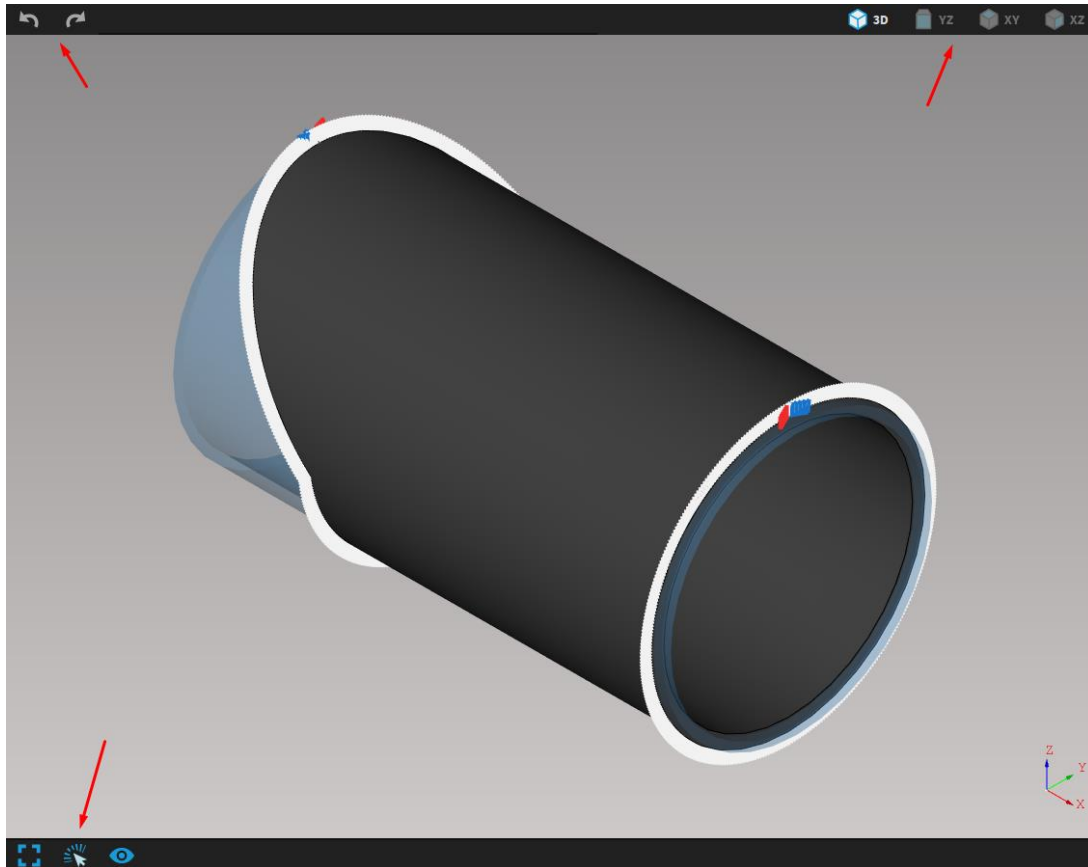
**Fig. 9** New buttons in working area of import tab

Icon	Description
	Import shape to active import tab
	Import whole folder to active import tab












# Visualization screen controls

The view in visualization screen is now controlled with new controls at the top and bottom of the visualization screen.



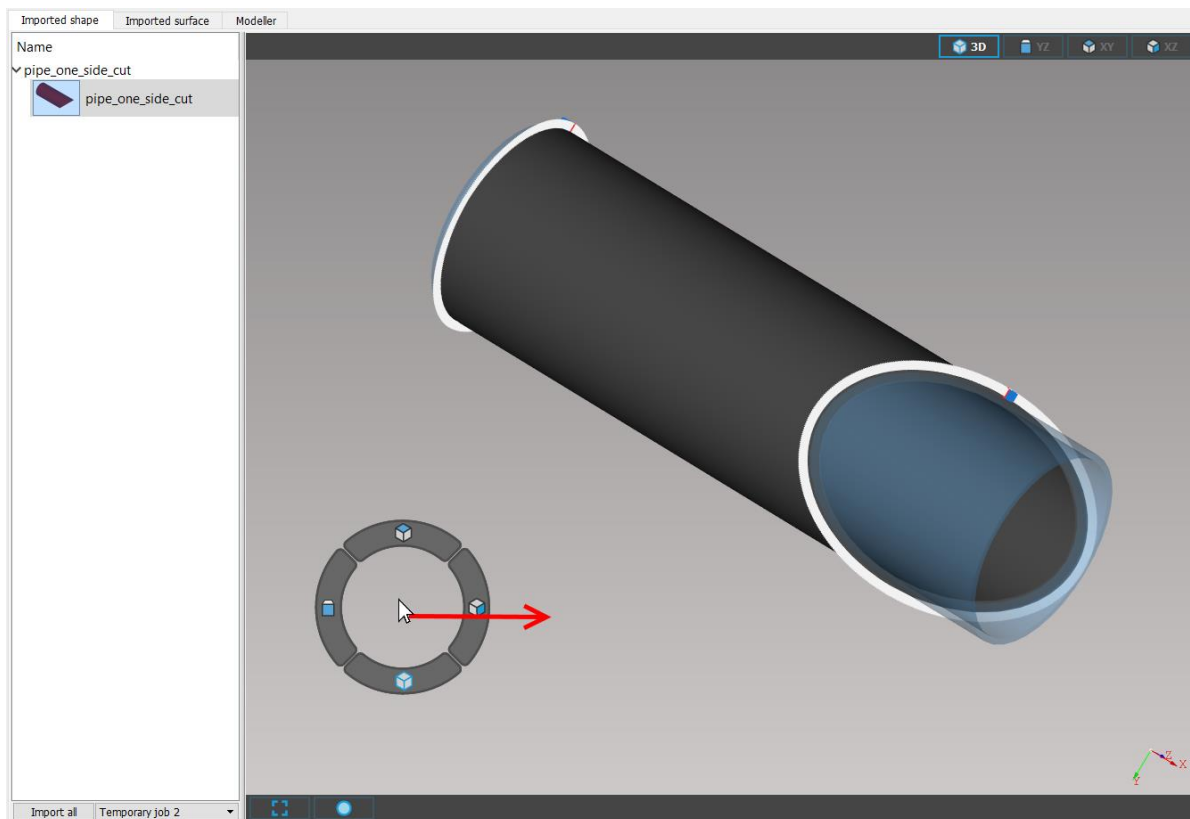
**Fig. 10** Controls of visualization screen with active 3D view. Model is displayed as shaded view.

Icon	Description
	Fit to window
	Opens Select cut
	Displays menu with settings for visualization screen
	Undo action
	Redo action

 3D	3D view
 YZ	YZ plane view (locked)
 XY	XY plane view (locked)
 XZ	XZ view (locked)


Rotation of model is disabled with YZ, XY and XZ view selected. Only panning and zooming in/out of the model is active.







User can also switch between individual views with the radial view widget, that will pop out by right clicking in visualization screen. Without releasing the mouse button, it has to be dragged over and through the desired view to change it.

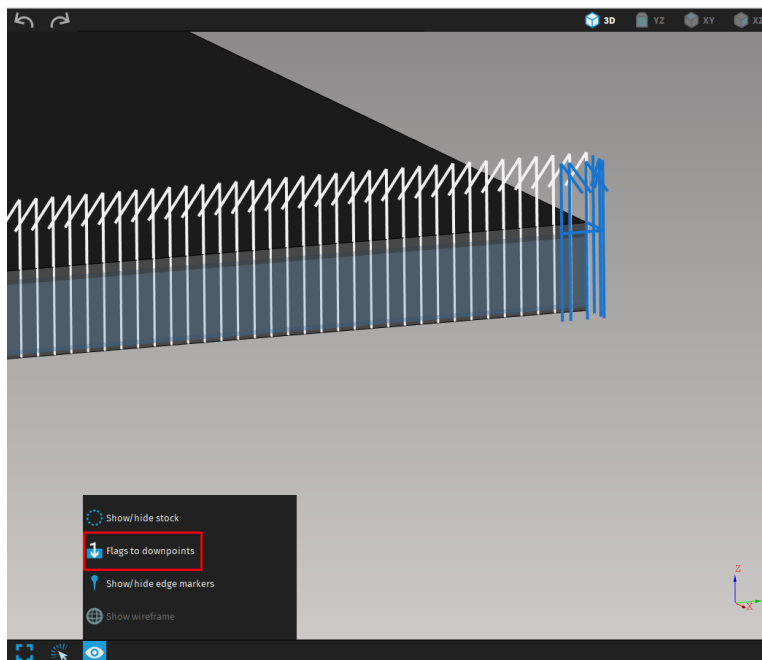


**Fig. 11 Without releasing the right mouse button it has to be dragged over the view user wants to select (in this case, XZ view would be selected)**

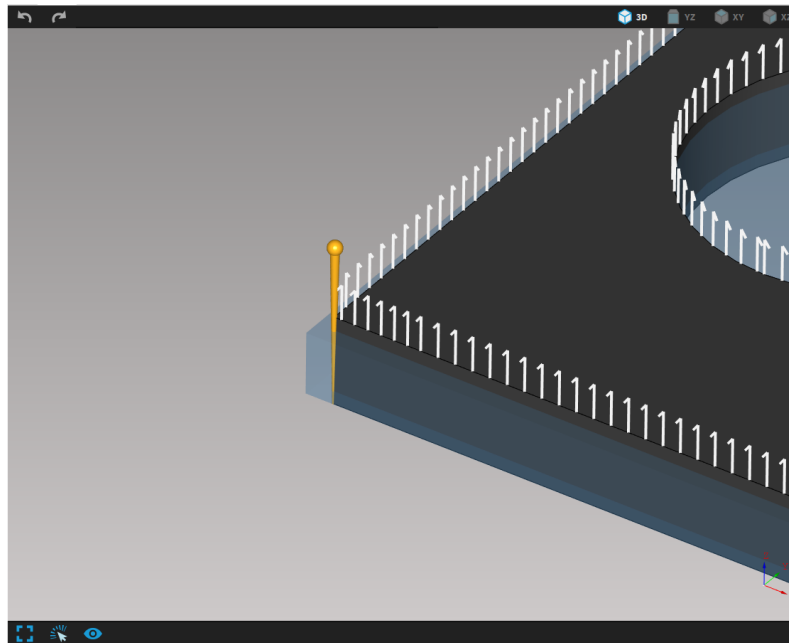
Individual settings for what and how is displayed in the visualization screen are now

located in the new combo box with eye icon  in the bottom panel of visualization screen instead of in the Properties area. After clicking on the icon with left mouse button a list of settings appears. Active settings are displayed in bright blue color, disabled settings are displayed in grey-blue color.

Icon	Description
 Show wireframe	When activated, displays the model (including the tool in simulation) in wireframe rendering
 Show/hide stock	Displays the stock (in light blue color)
 Show/hide edge markers	Show/hide edge markers
 Show/hide decorations	Displays decorations – e.g. measuring points ( <i>only in simulation</i> )
 Show/hide tool	Displays the tool ( <i>only in simulation</i> )
 Flags to downpoints	Displays cutting points along the entire thickness of the material

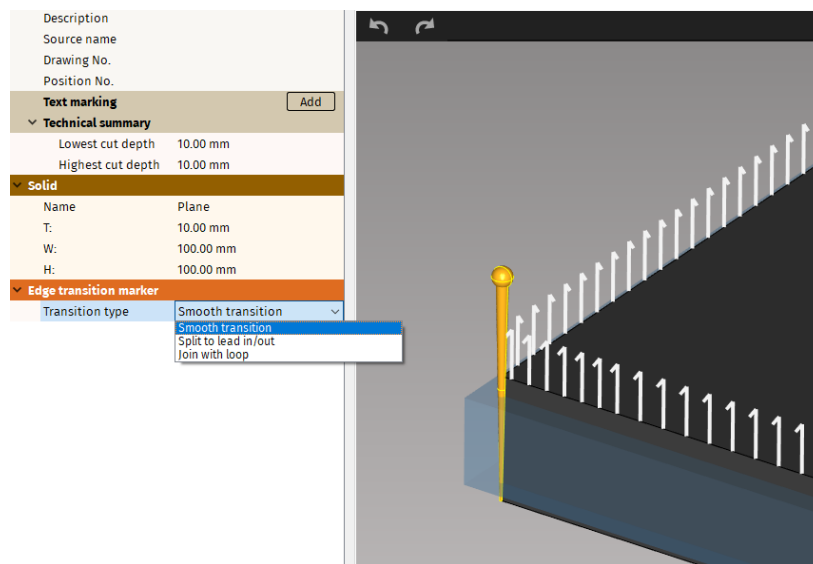


**Fig. 12 Cutting points are displayed along entire thickness of the material**



**Fig. 13 Edge marker on corner of the shape**

Edge markers serve for selecting suitable transition type on edges. After selecting a marker, its settings appear in Properties area.



**Fig. 14 Selected edge marker with its settings**

There are three transition types for edges:


1. **Smooth transition**
2. **Split to lead-in/-out** – path will be split on selected edge
3. **Join with loop** – loop will be added on corner. It is possible, that instead of loop, the lead-in/-out will be moved to the selected edge.

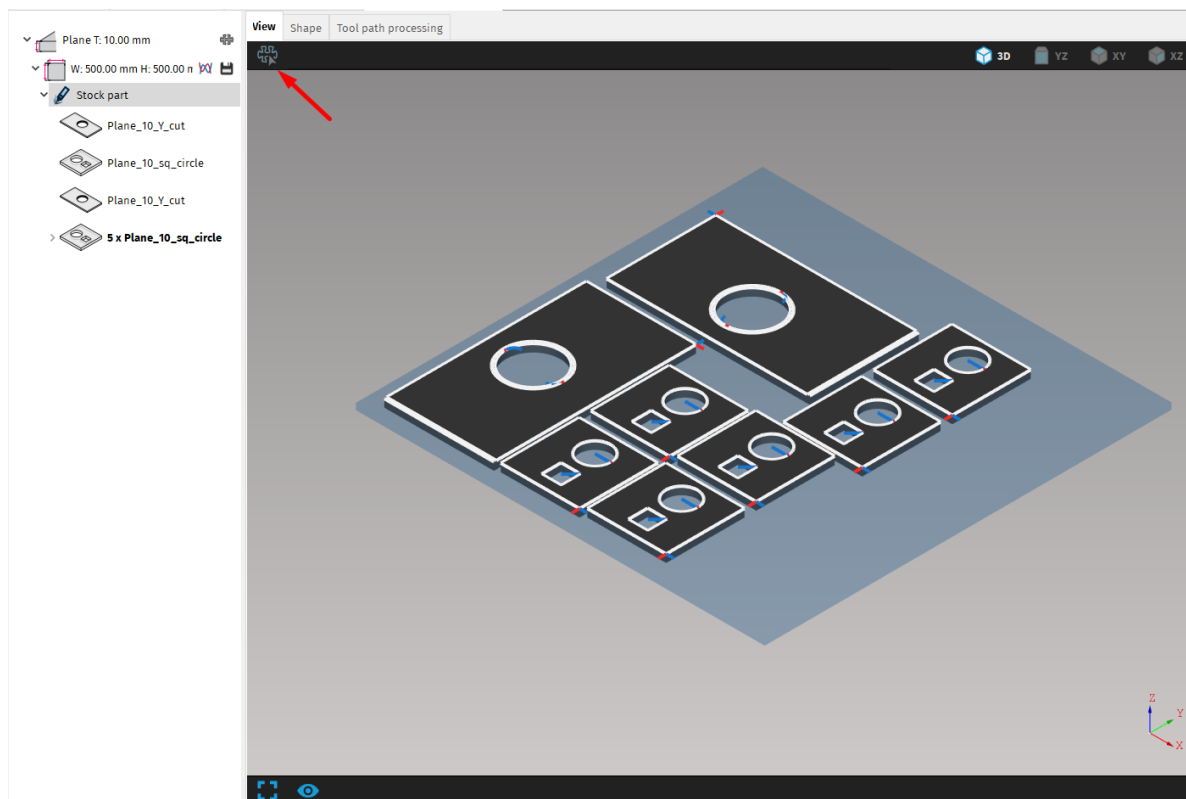
# Manual nesting

Manual nesting is a feature designed for positioning of plates and circular pipes. It helps user to more efficiently move individual parts on stock to achieve their best placement, if he wasn't satisfied with automatic nesting.

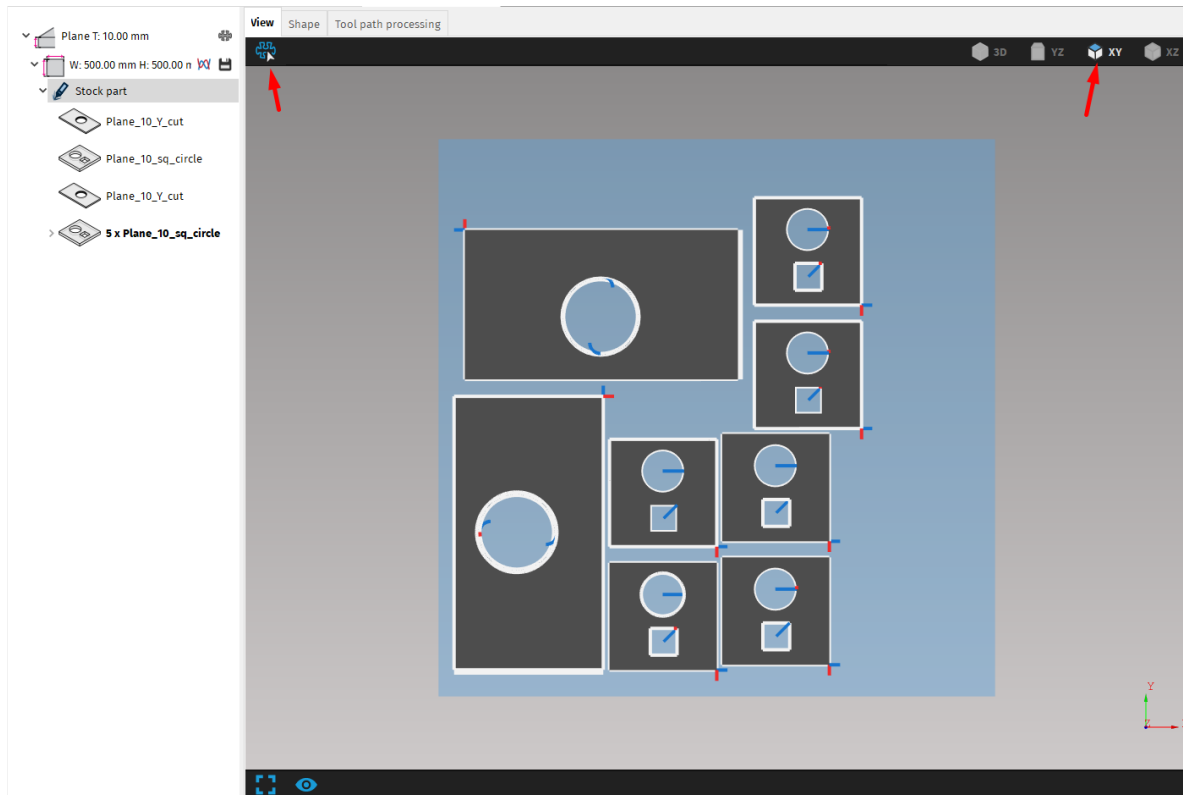
Manual nesting is activated in job tab. It's important to adjust the parameters of stock and move all parts to stock first. This is done either by automatically nesting all parts by clicking on puzzle icon or move them manually to the stock (with drag and drop), right clicking on the stock and selecting "Move rest parts to stock". The same manual procedure applies if not all parts were moved to stock at automatic nesting.

All parts that need to be nested has to be on the stock before manual nesting. If another part is added later, the nesting will reset.

Manual nesting is activated by clicking on a manual nesting button  at the top left corner of the visualization screen.



**Fig. 15 Manual nesting button – manual nesting is not activated**



**Fig. 16 Manual nesting is activated and the view is locked in 2D top view**



After activation of manual nesting, the view is automatically changed into locked 2D view. When nesting 2D parts, the view is locked to XY plane. For pipes, the view can be changed between XY plane and XZ plane.

Selected part is displayed with yellow outline and can be:

1. **Moved** – parts are moved by simply clicking and dragging the selected part into its desired position
2. **Rotated** – selected part can be rotated with left and right arrow keys by  $15^\circ$ . Holding the ALT key while rotating the parts will rotate them by  $5^\circ$ .
3. **Flipped** – only for pipes, flips the part around Z-axis (by  $180^\circ$ ) with up and down arrow keys.

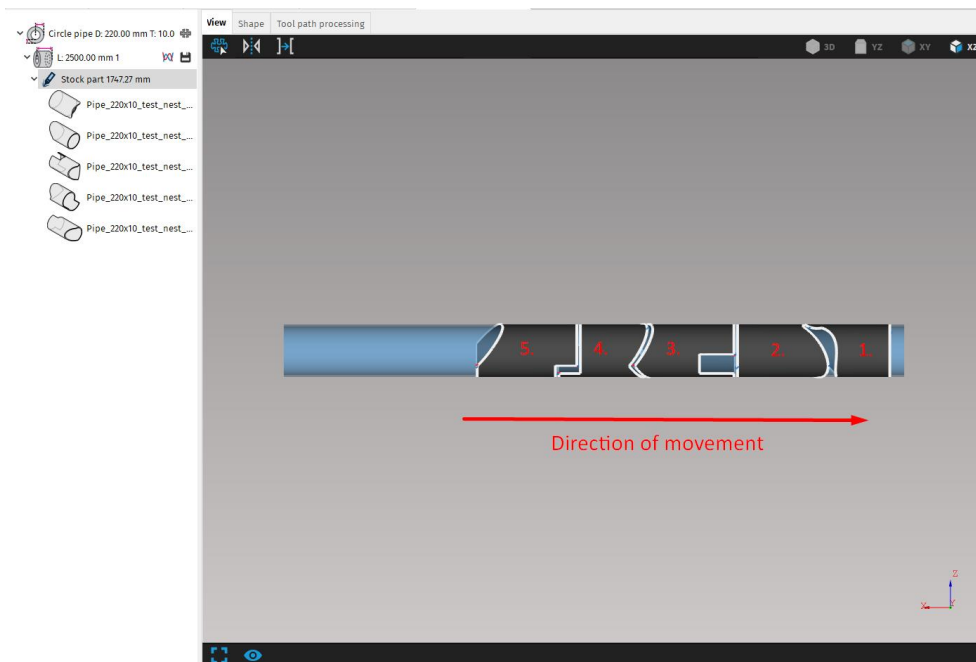
Note: The best and highly recommended way to find an optimal ordering, orientation and placement of all shapes is to use automatic nesting. Overlapping cutting contours and other related issues are not controlled/checked by mCAM (when using manual nesting), therefore user takes full responsibility for possible subsequent cutting problems on machine.

Next to the manual nesting button at the top left corner of the visualization screen are also two buttons with specific functions intended for more efficient manual nesting of circular pipes. These controls are active only in manual nesting mode for circular pipes.

Icon	Description
	<b>Flip</b> – flips the selected part (also with up and down arrow keys)
	<b>Move</b> – moves selected part closest to the previous part on stock (applies common cut if active)

## Move function

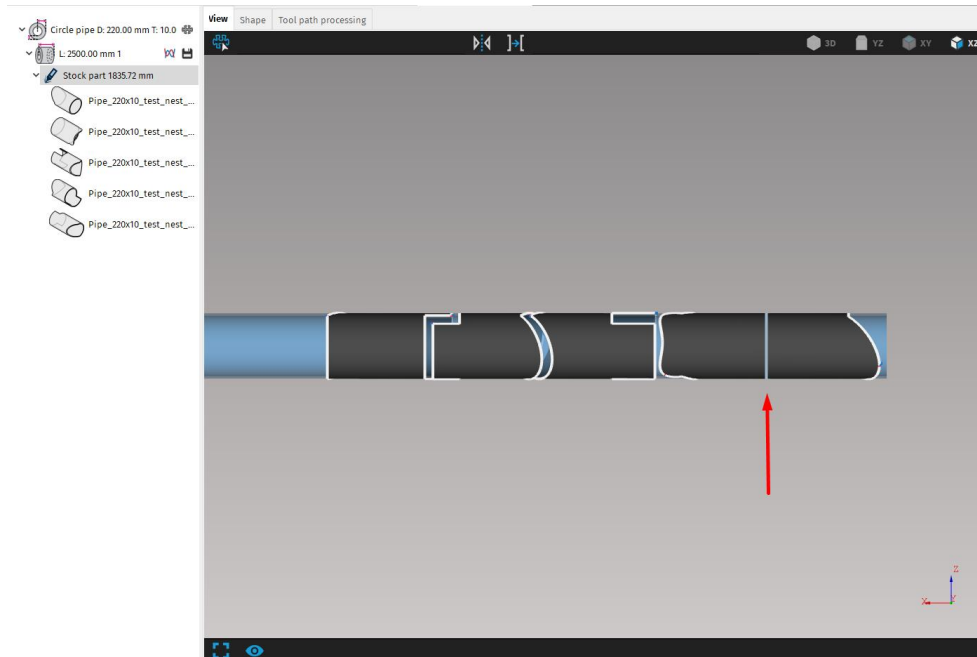
Move function is used to move selected part closest to the previous part on stock. The distance is set according to the value in Nesting gap in Properties area. The order of parts is set from the beginning of the stock material to the chuck location. So, in locked view in manual nesting, the parts are moved in X- direction.



**Fig. 17 Direction of movement of parts in active manual nesting mode in XZ view**

The Move function will also automatically rotate selected part to achieve optimal placement with maximum utilisation of the stock material according to the previous part position.

If common cut is activated for the whole cutting plan (by checking the checkbox in Nesting settings in Properties area) using the Move function will also apply common cut if possible (on straight trim cuts). If common cut was applied, the cutting path will change color to light blue-grey.



**Fig. 18** Parts nested with applied common cut

**Note:** Selecting parts in manual nesting in visualization screen won't affect the selection in Working screen. Because the information in Properties area are based on selected part in Working tree, these could not correspond with selected part in visualization screen. We recommend not making any changes in Properties area while the manual nesting is active or to double check if correct part is selected.

## Kerf compensation on common cut

If common cut will be used, it is necessary to select the specific tool, that will be used for generating CNC, so that respected kerf length can be used to maintain the most precise length of individual parts. Due to kerf compensation, the common cut is not displayed in visualization screen as one cut, but the parts are spaced apart by a length of kerf. Even though, they are displayed as two cuts in job, they will be cut by single cut.

The size of kerf is always the size of kerf for straight cut for respective thickness, even if the actual cut is bevel cut.



# Initial beam position changes

Initial beam position was changed in April 2017 for some shapes, to make it clearer how the beam should be loaded correctly on the machine. The original placement was changed for H, I and U profiles. Due to these changes, the new instruction M122 was added, that defines the position used for generating CNC code in mCAM.

M122 parameter is intended only for ROTPOL\_MODE2, because only in this mode it is possible to cut all profiles. ROTPOL\_MODE1 is used for cutting pipes, where the M122 is not necessary.





The setting for which initial position should be used is located in Settings -> Internal -> Position type and is automatically set according to parameter from flashbin. It should only be changed by Microstep staff.

The parameter in flashbin has four versions:

- p\_M122\_VERSION=0 – represents the most recent version (now it's the version 3, if in the future there will be version 4, it will represent version 4 etc.)
- p\_M122\_VERSION=1 – M122 is ignored
- p\_M122\_VERSION=2 – original placement of profiles as was before the change in 04/2017
- p\_M122\_VERSION=3 – new placement of profiles after the change in 04/2017

Setting for Position type in mCAM contains two options:

- **Version 1 without M122** – p\_M122\_VERSION=2
- **Version 1 with M122** – M122 is generated into the CNC code, but the positions of beams are as before the change in 04/2017
- **Version 2 with M122** – p\_M122\_VERSION=3

Position	Version 1 (without M122)	Version 2 (with M122 v3)
I/H beam		
U beam		

There are two scenarios:

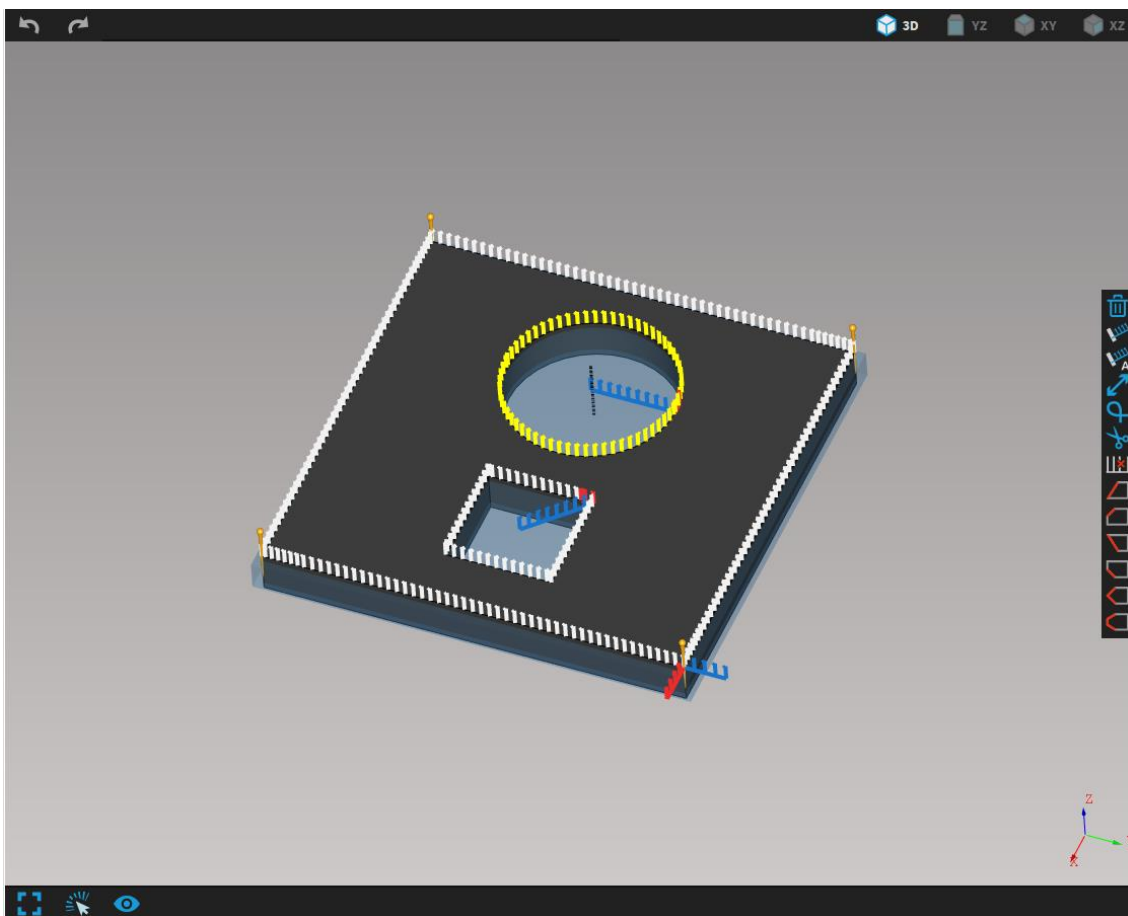
1. Older machines with old RTPC SW don't recognize the M122 instruction – In this case, at installation of mCAM, the Position type is automatically set to Version 1. The CNC code will be generated with original initial position, without using the M122 instruction. If Version 2 is selected and the machine does not recognize the M122 instruction, it will display error message.
2. New machines with new RTPC SW recognize the M122 instruction – the setting for Position type in mCAM is automatically set to Version 2 and the CNC code is generated with new initial positions of beams.

# Cut path editing








## Right control panel

New control panel was added in mCAM version 2.0. The panel pops out on the right side of the screen after selecting any cut path. It contains several buttons with various functions for changing some attributes of cut paths.

After selecting a cut path, a new panel pops out on the right side of the screen with several buttons intended for editing some attributes of the cut path.



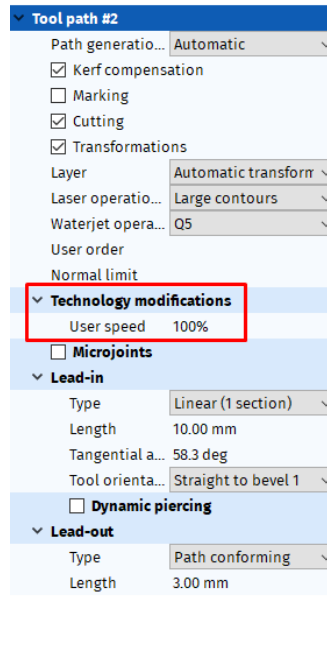
**Fig. 19** Cut path editing panel on the right side of the visualization screen

Icon	Description
	Set lead-in/out to custom position
	Set lead-in/out to automatic position as generated by mCAM
	Revert the cut path
	Add loop on selected cut point
	Split path on selected point
	Erase selected cut point
	Delete cut path

There are also welding attributes, that are described in section Welding

## Properties area

- **User speed** – It is now possible to change speed of cutting for each cutting path individually. It is set as percentage of speed that is in expert tables.



**Fig. 20** Setting the user speed for selected cut

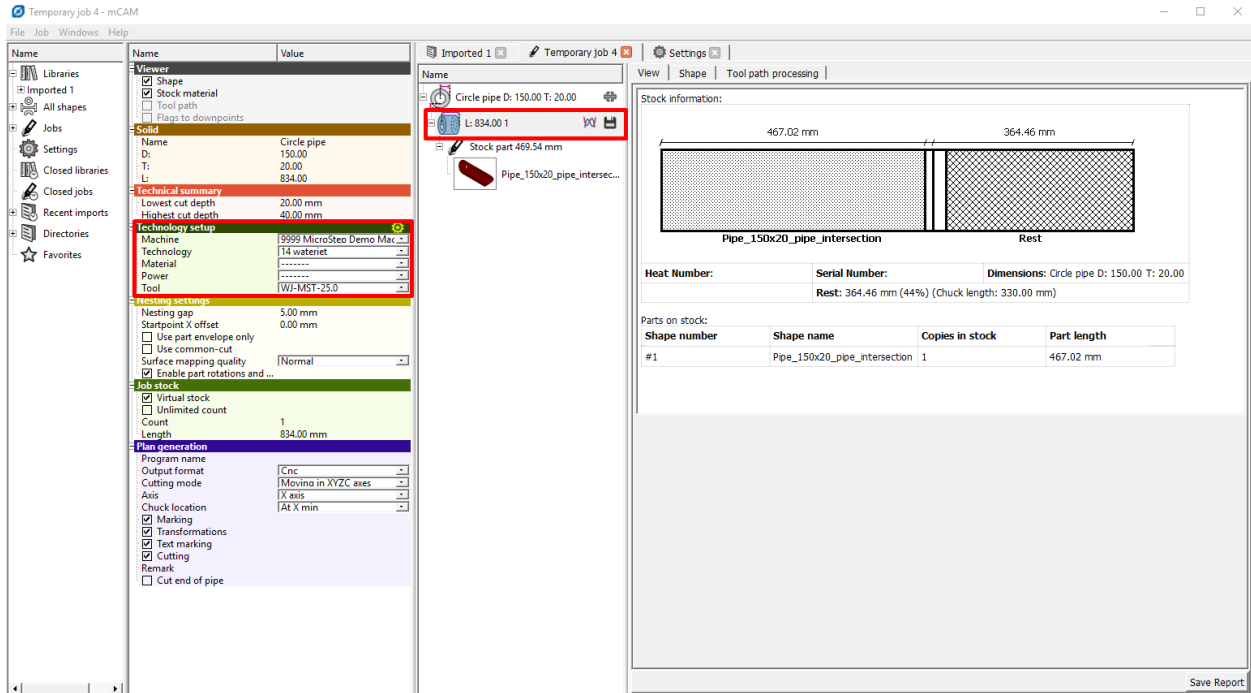
- **Stop (M0) before cutting** – can be checked on or off for each cut path individually. If checked, the instruction M0 is generated into CNC code before respective cut path.
- **Tool selection for marking** – it is possible to assign different technology and tool to marking and text marking.

The shape has to be already loaded in the job. The option to choose separate technology and tool for marking is not available in Import tab.

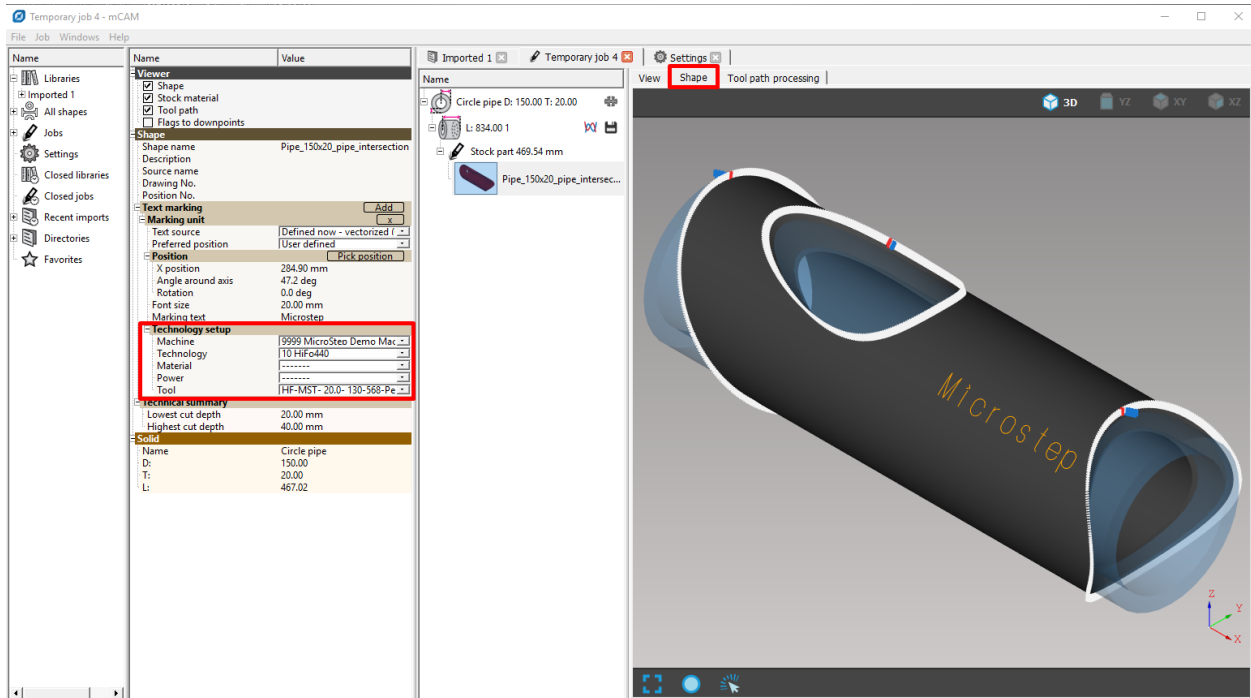
Technology and tool for text marking is assigned in Shape tab in section Text marking (see Fig. 22). Technology and tool for marking of cut path is assigned at the bottom of the properties area after selecting respective cut path (see Fig. 23).

Technology selected for marking has to be different from technology used for cutting the whole plan, otherwise a complain will pop up “Some cut paths have set invalid tool”. This is because it is not possible to change tools in iMSNC during cutting, so one technology can have only one tool assigned in one CNC plan.

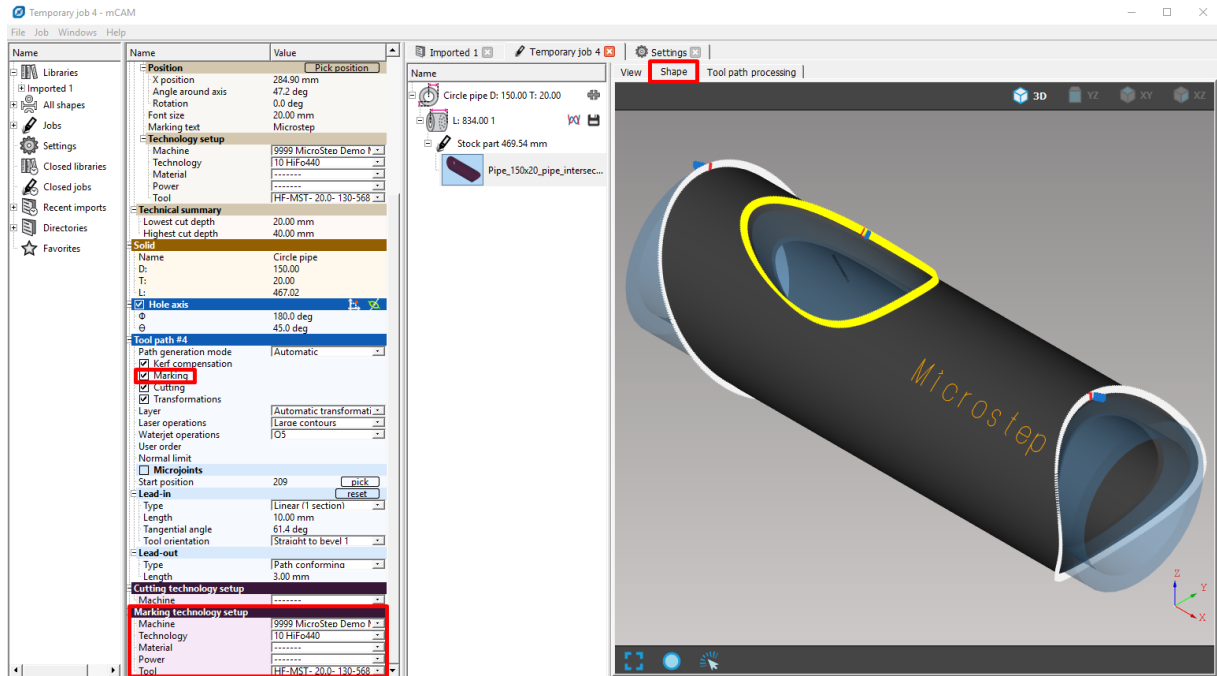
This option is especially useful for machines with inkjet markers.



**Fig. 21 Waterjet technology with respective tool is selected for cutting**



**Fig. 22 Plasma with respective tool is selected for text marking in Shape tab**



**Fig. 23** Marking is enabled for selected cut path and plasma technology with respective tool is selected in Properties area

```

(PART 'Pipe_150x20_pipe_intersection' 1 2954400)
(SIZE 1; 20 150; 467.019)
(SIZE: CirclePipe Thickness=20 Diameter=150; Length=467.019)
(mcam.expert.tool Operation 4. thickness= 0.00 speed= 3400.00 kerf= 3.60
(mcam.key.tool machine:9999 technology:10 tool:HF-MST- 20.0- 130-568-PerCut 4
(mcam.text= src=userTextVectorized pref=toUserCoordinates rot=0.0 height=20.
M6 T410 (TEXT TEXT 4)
G0 C-41.525
M94 D5
G94 X-140.473 Y-2.428 Z-0.039 (upAngle=1.9)
G94 X-182.473 Y-7.416 Z-0.368 (upAngle=5.7)
M94
M38 L-5.087 L14.292 (Corr A from B 30.0. Add=1.0)
G0 X-182.473 Y-7.416 Z-0.368 A-5.675 B0.000 C-41.525 (setB=0.000) (upAngle=
M20 D2
M3 (letter_M)
M90 D1.000
G1 X-182.473 Y-6.421 Z-0.275 A-4.911 B0.000 C-41.525 (upAngle=4.9)
M90 D1.000
G1 X-182.473 Y-5.424 Z-0.196 A-4.147 B0.000 C-41.525 (upAngle=4.1)
M90 D1.000
G1 X-182.473 Y-4.426 Z-0.131 A-3.383 B0.000 C-41.525 (upAngle=3.4)
M90 D1.000
G1 X-182.473 Y-3.427 Z-0.078 A-2.619 B0.000 C-41.525 (upAngle=2.6)
M90 D1.000
G1 X-182.473 Y-2.428 Z-0.039 A-1.855 B0.000 C-41.525 (upAngle=1.9)
M90 D1.000
G1 X-182.473 Y-1.428 Z-0.014 A-1.091 B0.000 C-41.525 (upAngle=1.1)
M90 D1.000
G1 X-182.473 Y-0.429 Z-0.001 A-0.327 B0.000 C-41.525 (upAngle=0.3)
M90 D1.000
G1 X-182.473 Y0.571 Z-0.002 A0.437 B0.000 C-41.525 (upAngle=0.4)
M90 D1.000
G1 X-182.473 Y1.571 Z-0.016 A1.201 B0.000 C-41.525 (upAngle=1.2)
M90 D1.000
G1 X-182.473 Y2.571 Z-0.044 A1.964 B0.000 C-41.525 (upAngle=2.0)
M90 D1.000
G1 X-182.473 Y3.570 Z-0.085 A2.728 B0.000 C-41.525 (upAngle=2.7)
M90 D1.000
G1 X-182.473 Y4.569 Z-0.139 A3.492 B0.000 C-41.525 (upAngle=3.5)

```

Blocks				
Id	M94 D	M20	M6	Cut off
L1				
3	5:2	2	410	marking with plasma
11	---		410	
7	7:1	2	114	waste
4	7:1	2	114	item
L3173				cutting with waterjet

Line: 68      search


**Fig. 24 Example of CNC code**

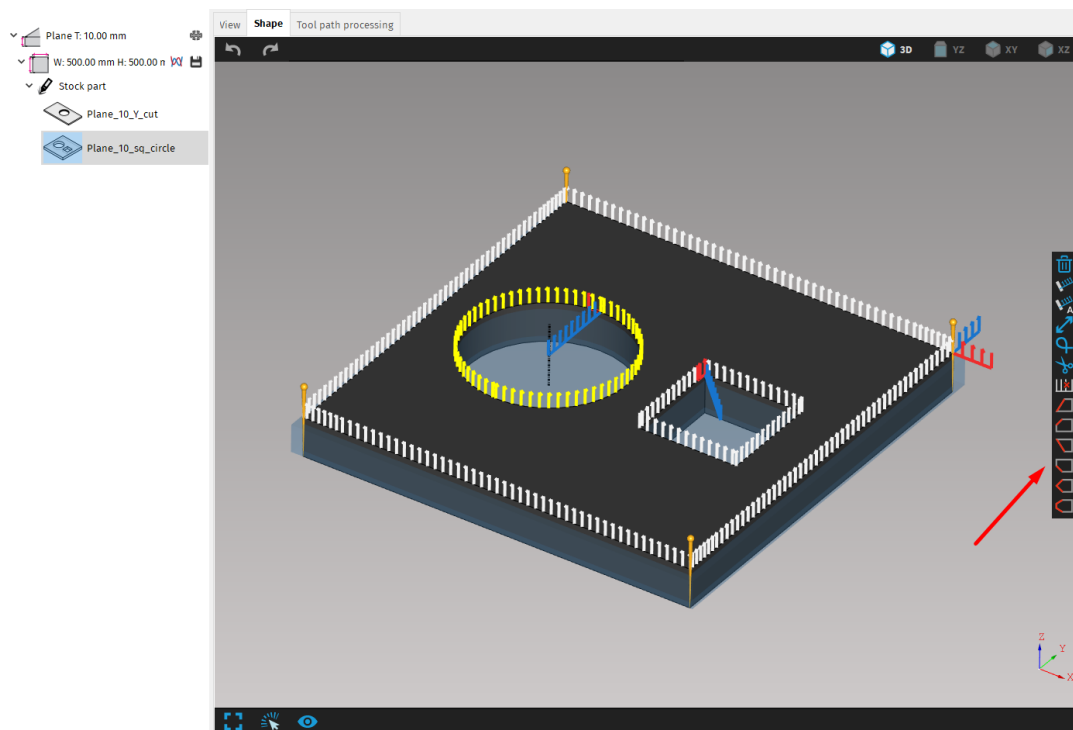


# Welding preparations

mCAM now allows to add welding preparations to existing cut paths. Adjusting a cutting contour for welding is done simply by selecting a cut path and clicking on an icon of a desired shape of welding preparation in the panel on the right side of the visualization screen. mCAM will then generate all necessary cutting paths accordingly.







Cut paths that are modified for welding preparation are displayed in **orange** color

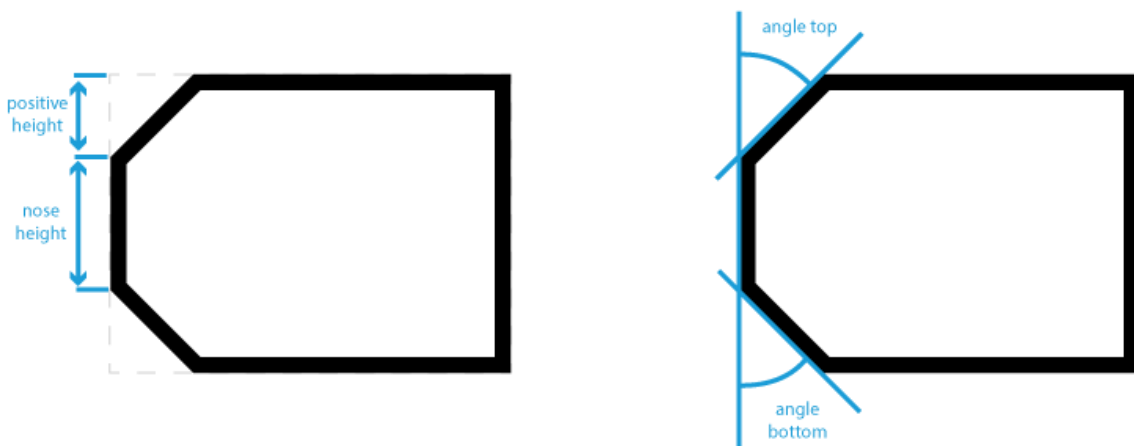
Each type of welding preparation can be further adjusted in Properties area – Welding section. Welding preparation is deleted by selecting a welding preparation cut path and clicking on an icon  in the cut path widget.



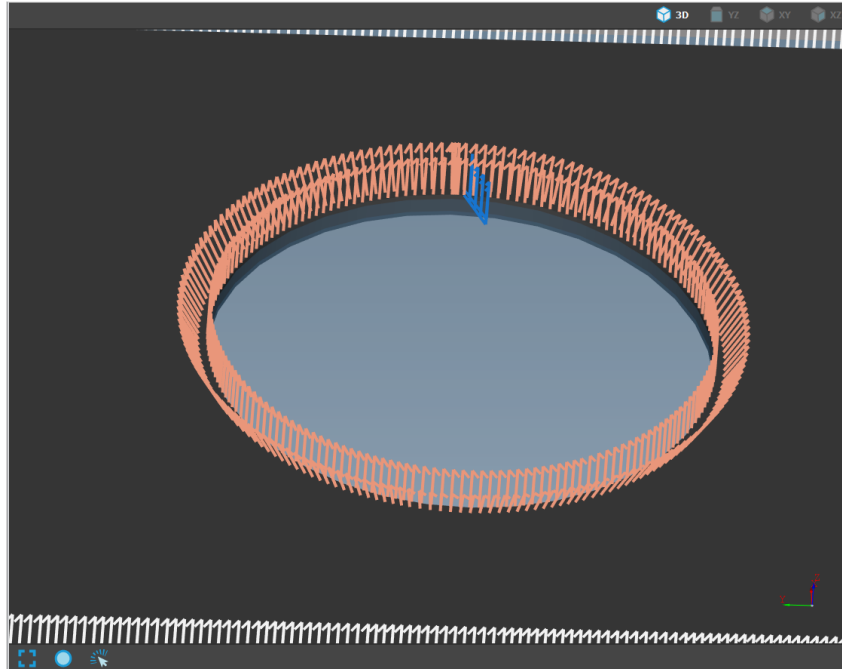
**Fig. 25 Panel with various options including welding preparations shows up on the right side of the screen when the cut path is selected**

Parameters that can be adjusted depend on the type of the welding:

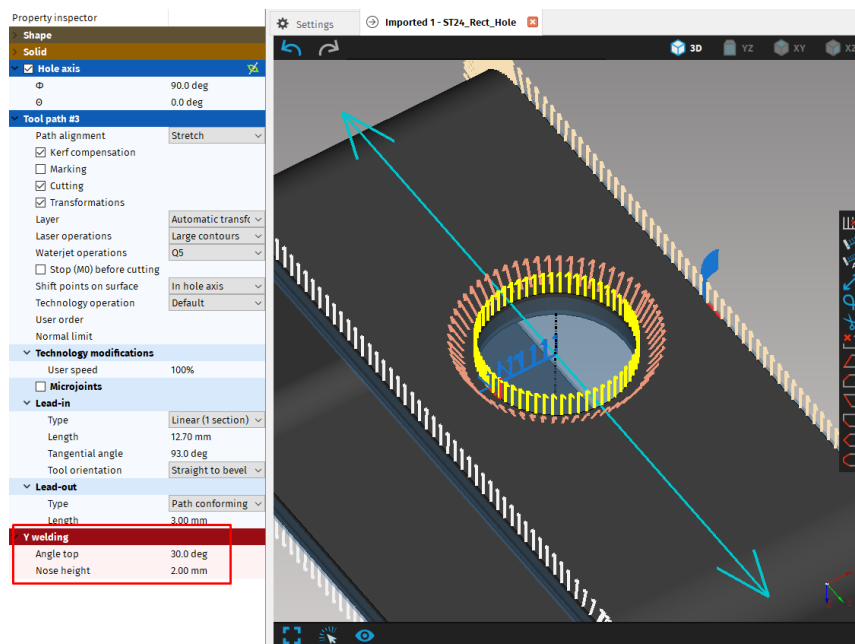
Name	Parameters
 <p>V welding</p>	angle top
 <p>Y welding</p>	angle top, nose height
 <p>A welding</p>	angle bottom
 <p>L welding</p>	angle bottom, nose height
 <p>X welding</p>	angle top, angle bottom, positive height
 <p>K welding</p>	angle top, angle bottom, positive height, nose height



**Fig. 26** Welding preparation parameters



**Fig. 27 Hole with Y welding preparation**



**Fig. 28 Welding settings in Properties area**

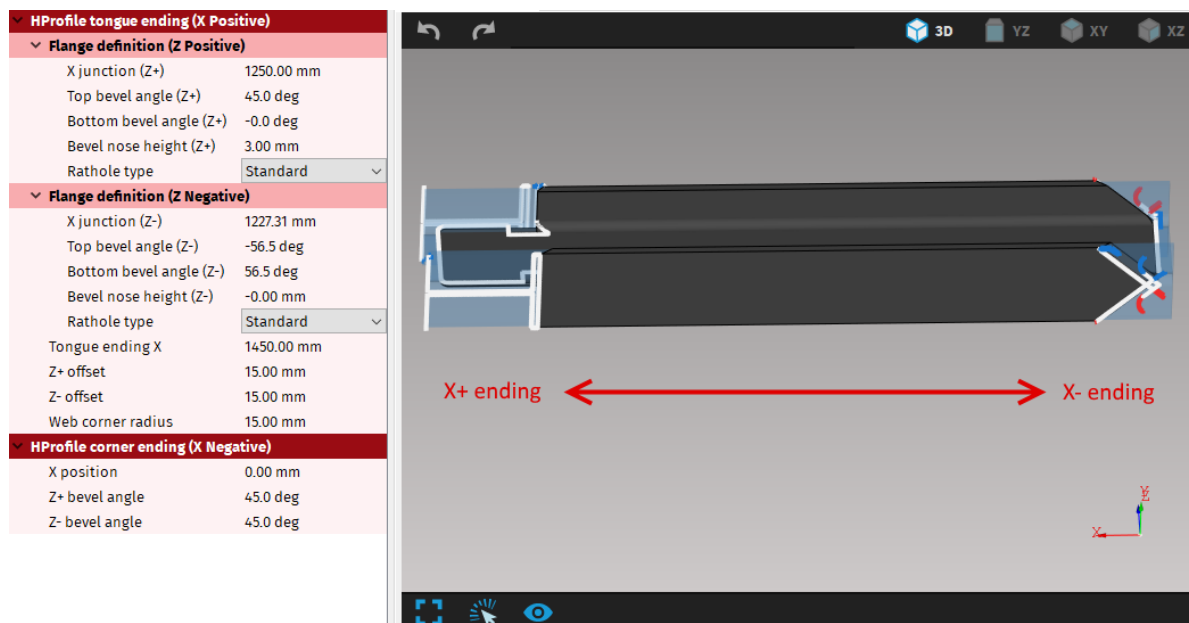
# H-beam welding connection macros

mCAM supports several types of welding connections for H-profiles, that are automatically recognized at import.

1. Tongue ending
2. Inverse tongue ending
3. Corner ending

Each type of welding connection has its set of parameters that are automatically set according to the imported shape. They are of informative character and do not need to be adjusted.

Each macro for welding connection ending can be oriented in X+ or X- direction, depending on which end of the part it is located. Each part contains maximum of two macros – one for each end of the part.



**Fig. 29** Each part can contain maximum of two macros – One in X+ and one in X- direction

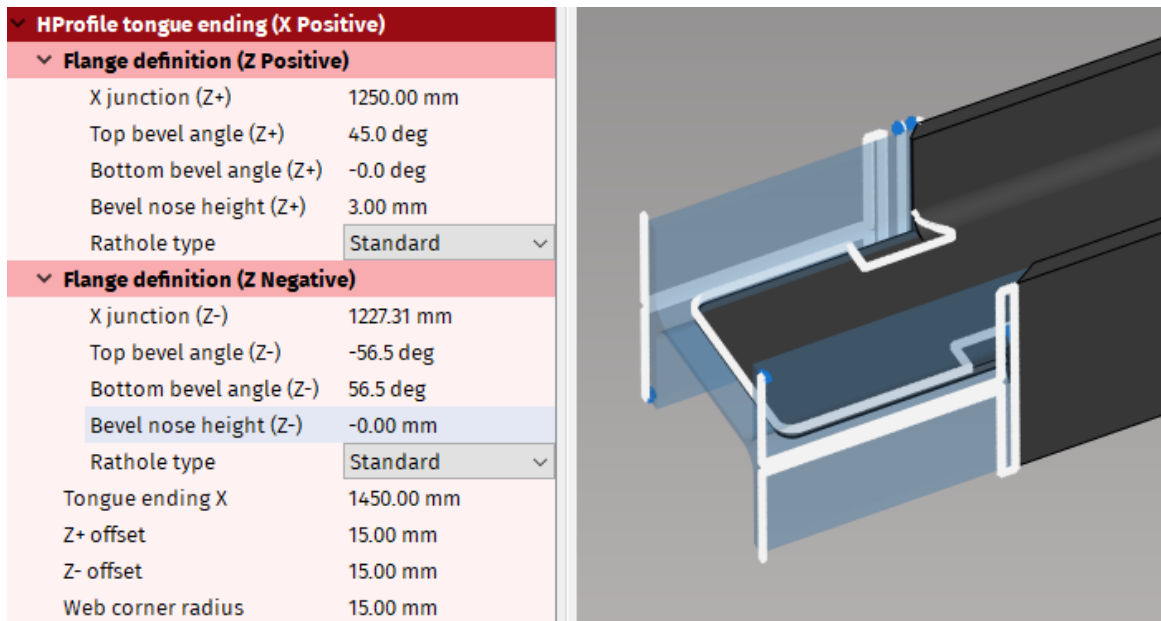
## 1. Tongue ending parameters

→ Flange definition Z+/Flange definition Z- – defines the length and shape of respective flanges. Contains several parameters:

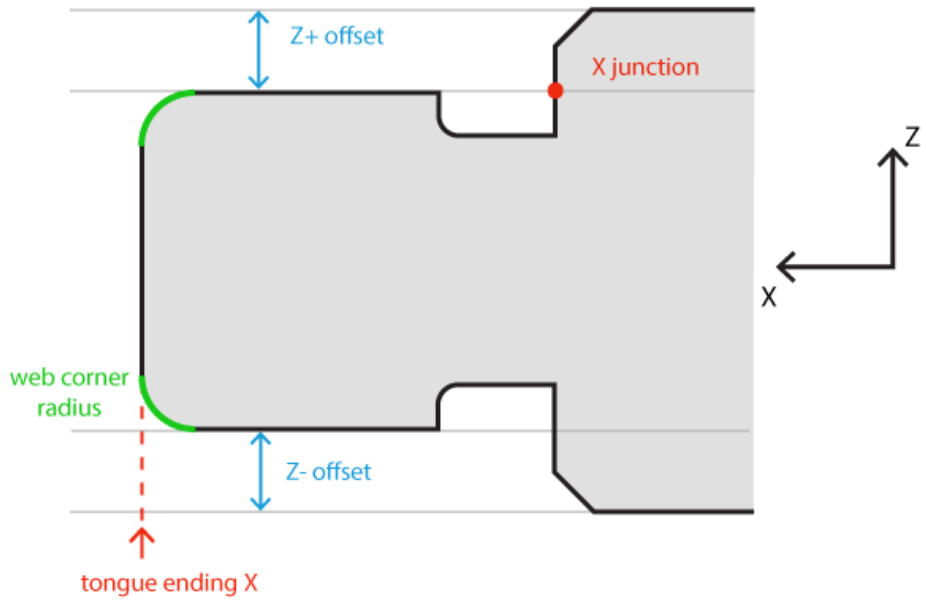
- *X junction* – Point on an inner edge of the Z+/Z- flange, which defines the length of the respective flanges. (see Fig. 31)
- *Top bevel angle* – defines shape of the respective flange (see Fig. 32)
- *Bottom bevel angle* – defines shape of the respective flange (see Fig. 32)
- *Bevel nose height* – defines shape of the respective flange (see Fig. 32)
- *Rathole type* – None, none with grinding and standard (see Fig. 33, Fig. 34, Fig. 35, Fig. 36)

- *Tongue ending X* – defines the length of the web. Unlike the inverse tongue ending, this type can have only straight ending with Z+ and Z- value the same (hence only one parameter instead of two as is in inverse tongue ending)
- *Z+ offset* – offset of web from outer edge of Z+ flange (see Fig. 31)
- *Z- offset* – offset of web from outer edge of Z- flange (see Fig. 31)
- *Web corner radius* – rounding of the corners on tongue ending (see Fig. 31)

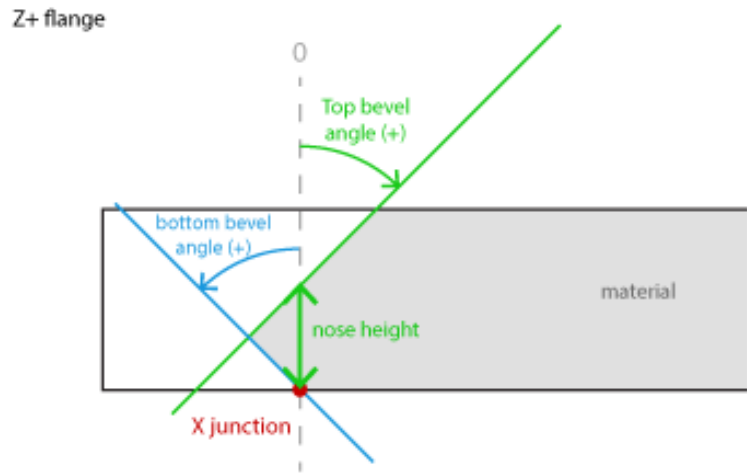
If Z offset is bigger than the cutting head diameter, flanges don't need to be cut, because the cutting head will fit between the flange and the cutting contour. The cutting paths are automatically generated for each flange separately with regard to this information.



**Fig. 30 Tongue ending and its parameters in Properties area**

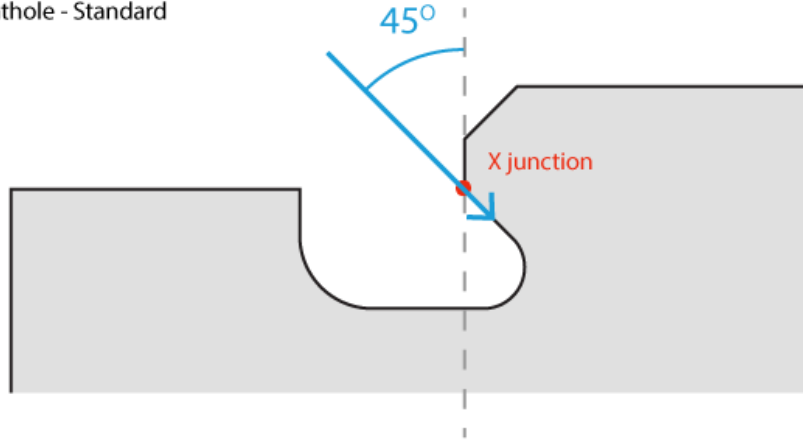


**Fig. 31 Parameters for Tongue ending welding connection**



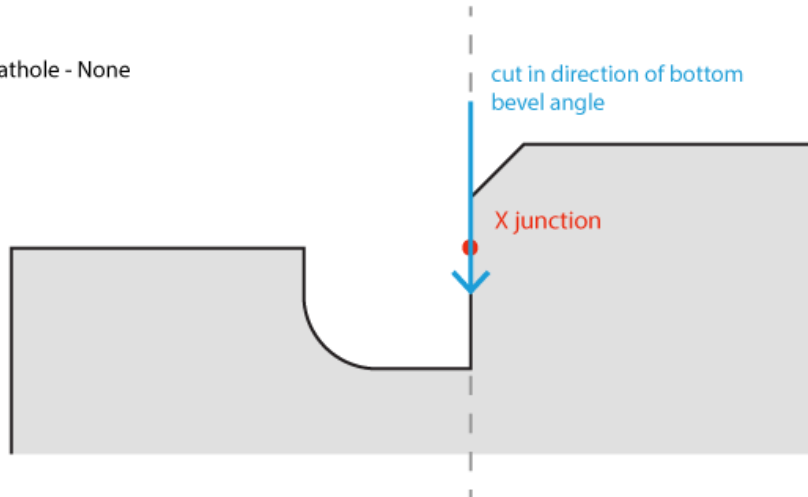
**Fig. 32 Z+ flange with bevel parameters**

Rathole - Standard

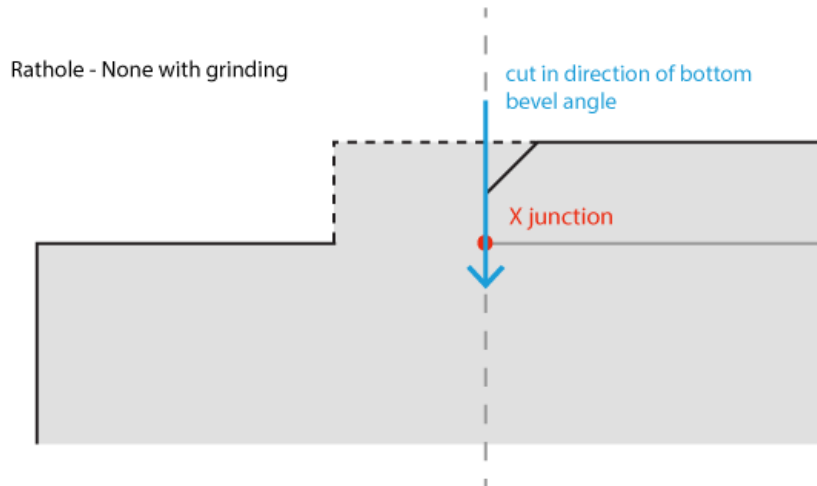


**Fig. 33 Rathole Standard is cut in 45 degrees angle**

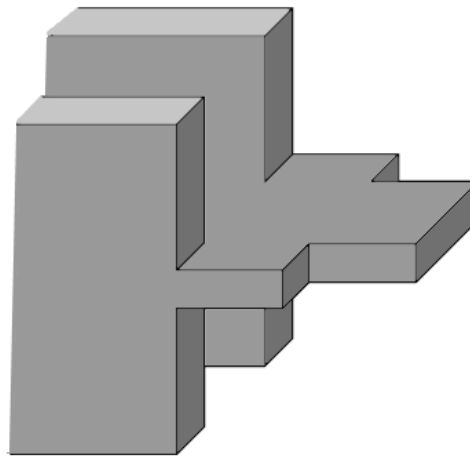
Rathole - None



**Fig. 34 Rathole None is cut in direction of bottom bevel angle**



**Fig. 35 Rathole - None with grinding**



**Fig. 36 Rathole - None with grinding**

## 2. Inverse tongue ending parameters

→ Flange definition Z+/Flange definition Z- – defines the length and shape of respective flanges. Contains several parameters:

- *X junction* – Point on an inner edge of the Z+/Z- flange, which defines the length of the respective flanges.
- *Top bevel angle*



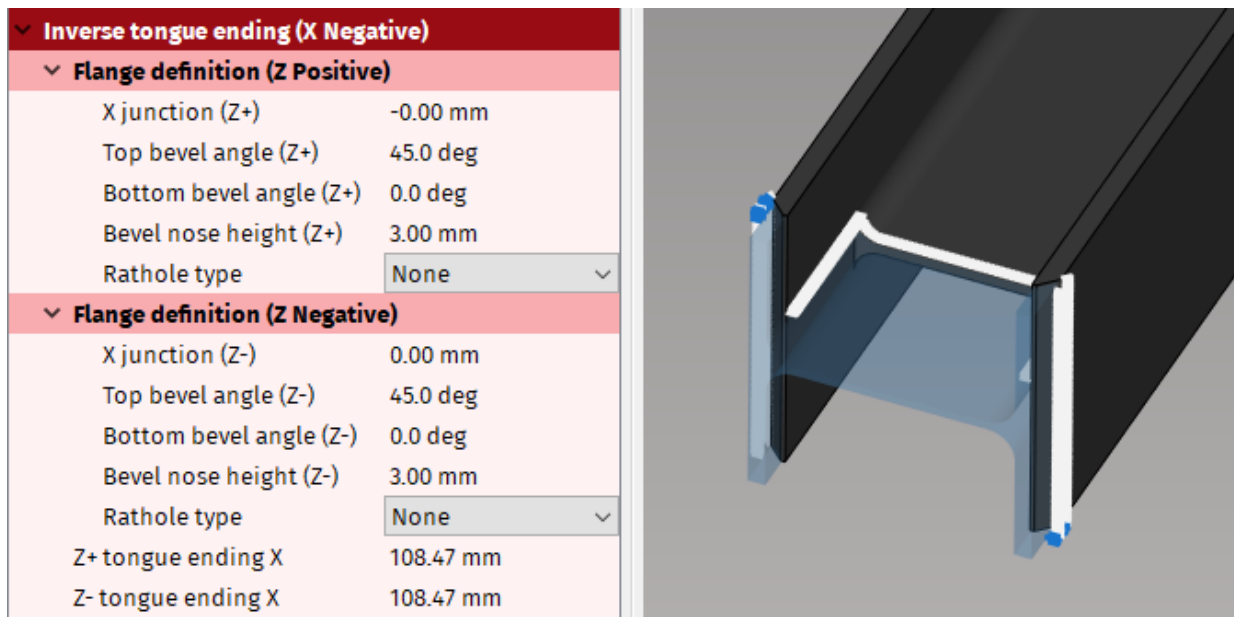
- *Bottom bevel angle*
- *Bevel nose height*

Bevel angle parameters (Top and bottom bevel angle, bevel nose height) are the same as for Tongue ending welding connection (see Fig. 32)

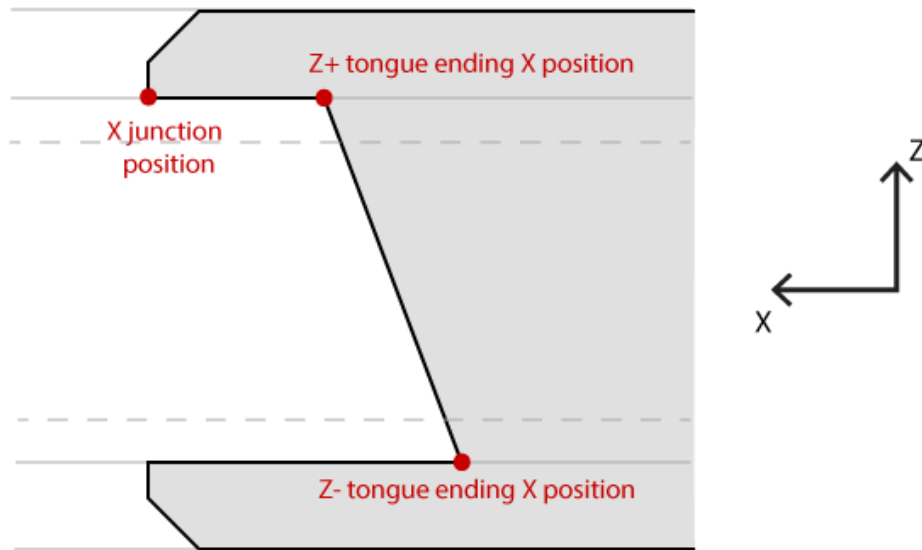
- *Rathole type* – None and standard (see Fig. 33 and Fig. 34)

- Z+ tongue ending X – defines the length of the Z+ side of the web
- Z- tongue ending X – defines the length of the Z- side of the web.

Z+ and Z- tongue ending X parameters can be different values each. Because it is inverse tongue ending welding connection type, the position of Tongue ending X must be located in direction of material away from position of Junction X.



**Fig. 37 Inverse tongue ending and its parameters in Properties area**

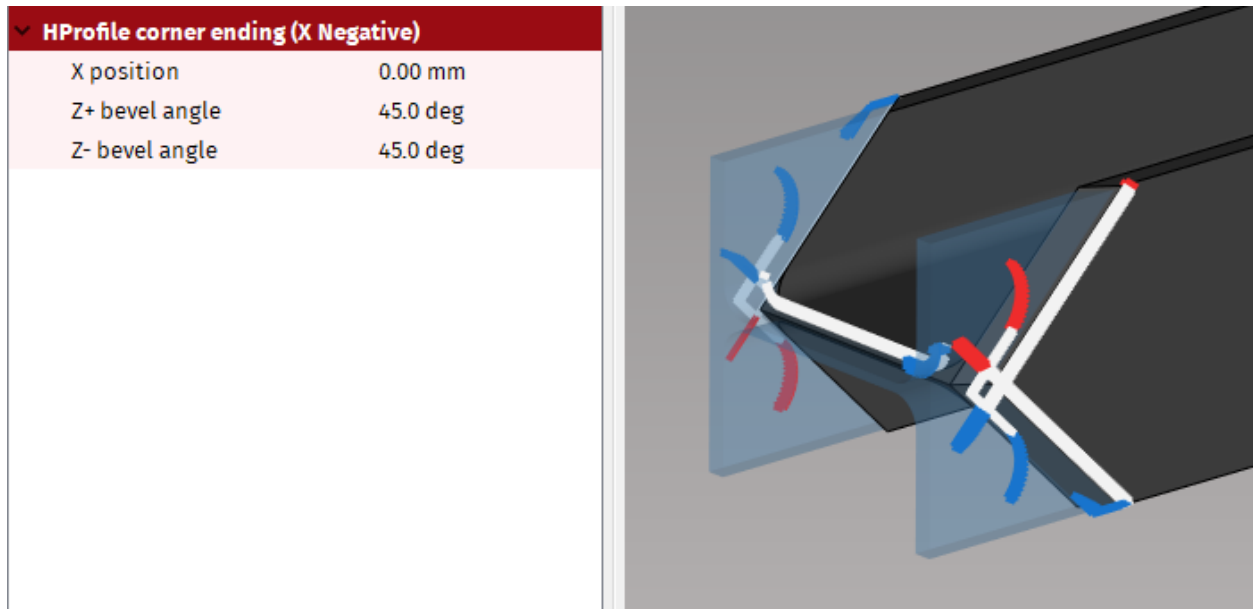


**Fig. 38 Inverse tongue ending with parameters**

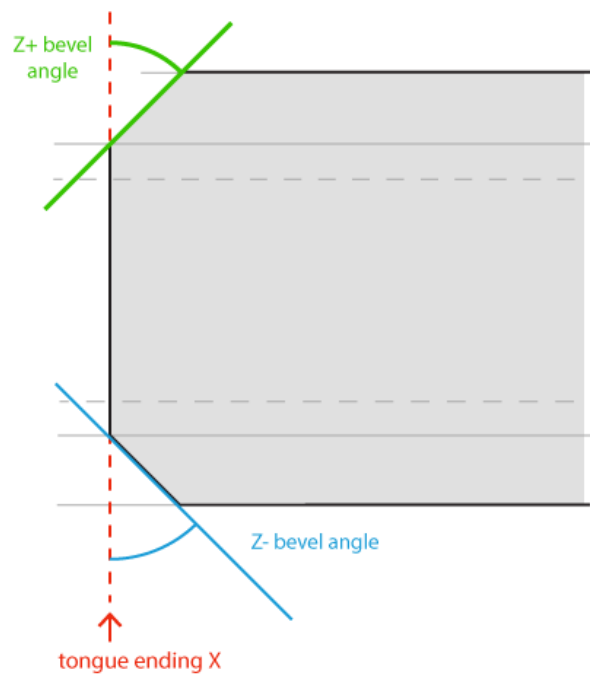
All bevels on web for both tongue ending and inverse tongue ending are being ignored, even if they are designed in 3D model of the part. mCAM will ignore them, and generate basic straight cut on the web, regardless of the drawing.

### 3. Corner ending parameters

- X position – defines the length of the web
- Z+ bevel angle – defines the bevel angle of Z+ flange
- Z- bevel angle – defines the bevel angle of Z- flange

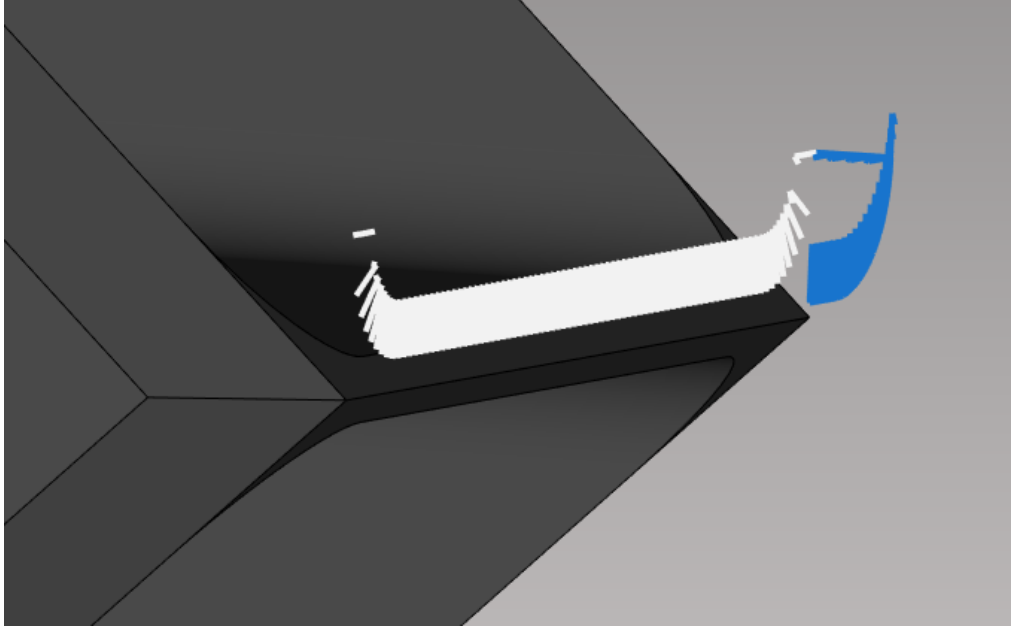


**Fig. 39 Corner ending and its parameters in Properties area**



**Fig. 40 Top view of corner ending welding connection with its parameters**

Corner ending has to be designed with X-cut on the web for mCAM to identify correctly the welding connection, but it will be ignored and the final product will have only straight cut on the web.

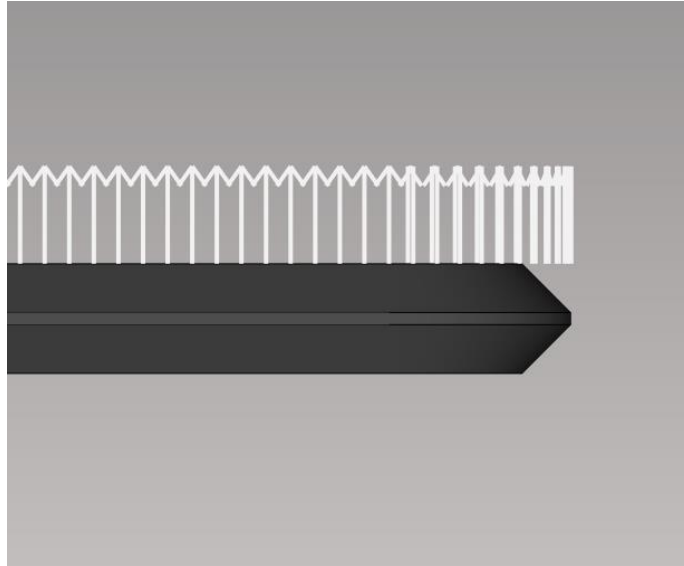


*Fig. 41 Step file is designed with X cut on the web, but the final cut is straight*

## Limitations on H-beam welding connections macros

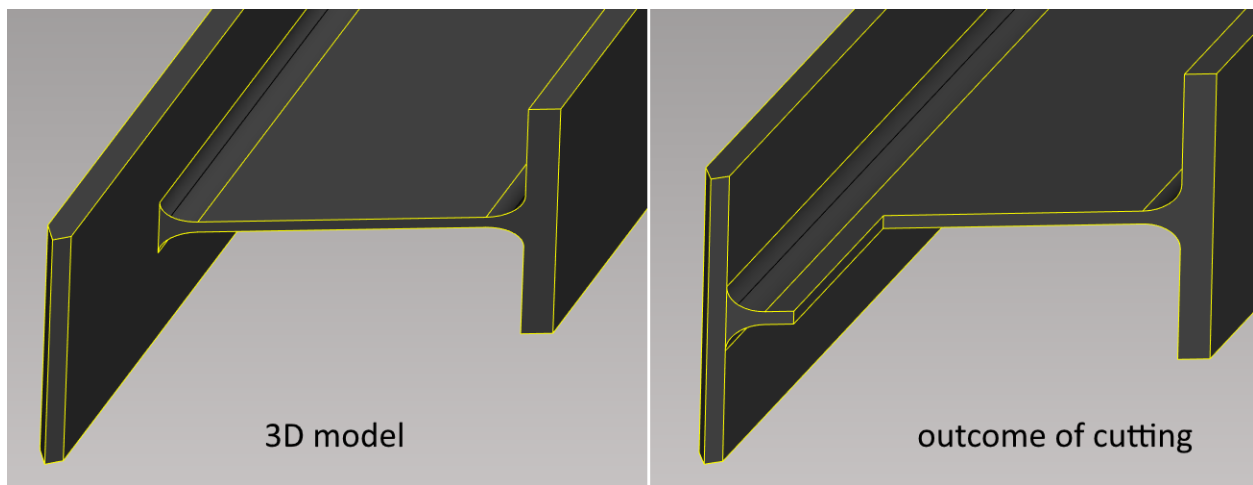
There are some limitations on what type of cuts are supported on H-beam welding connections.

1. Y/X/K cuts on web – Because of the dimensions of the cutting head it is not possible to cut the bevel cuts on web. If they are designed in step, they will be ignored and replaced with straight cut. It doesn't matter if they are designed in step file. The exception is Corner ending macro, where, for the correct recognition of the welding connection, X-cut must be on the web.



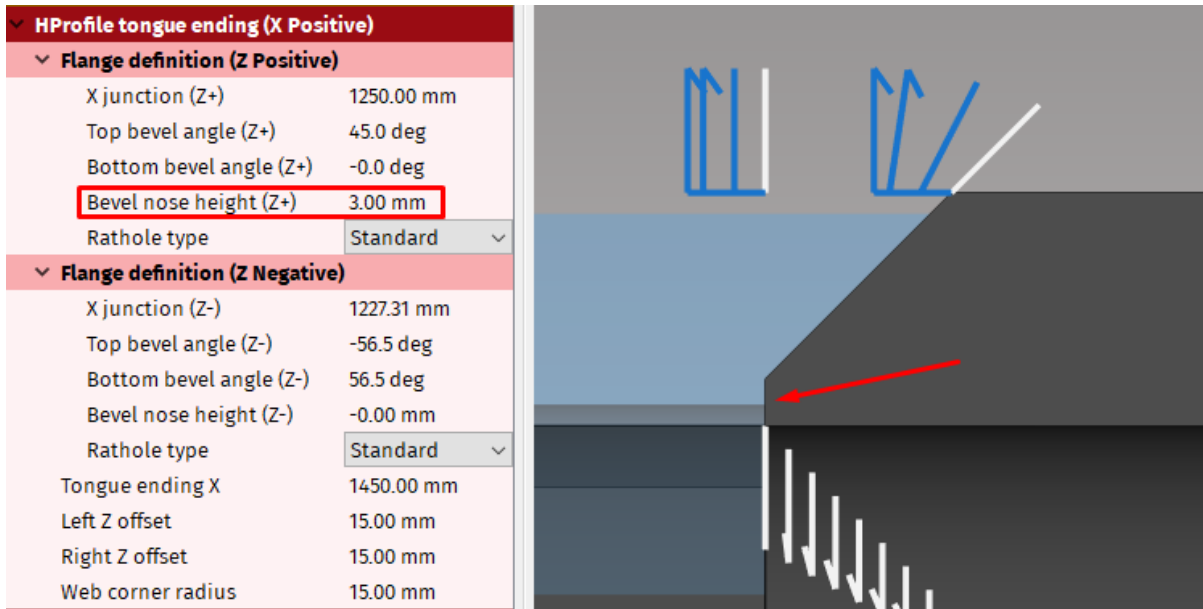
**Fig. 42 Detail of the web on tongue ending with K-cut replaced with straight cut**

2. In case of inverse tongue ending macro, due to technology limits, it is not possible to cut the web completely off the flange. There will be a section of the web that will remain attached to the flange. It is not possible to cut exactly near the flange due to the dimensions of the cutting head.



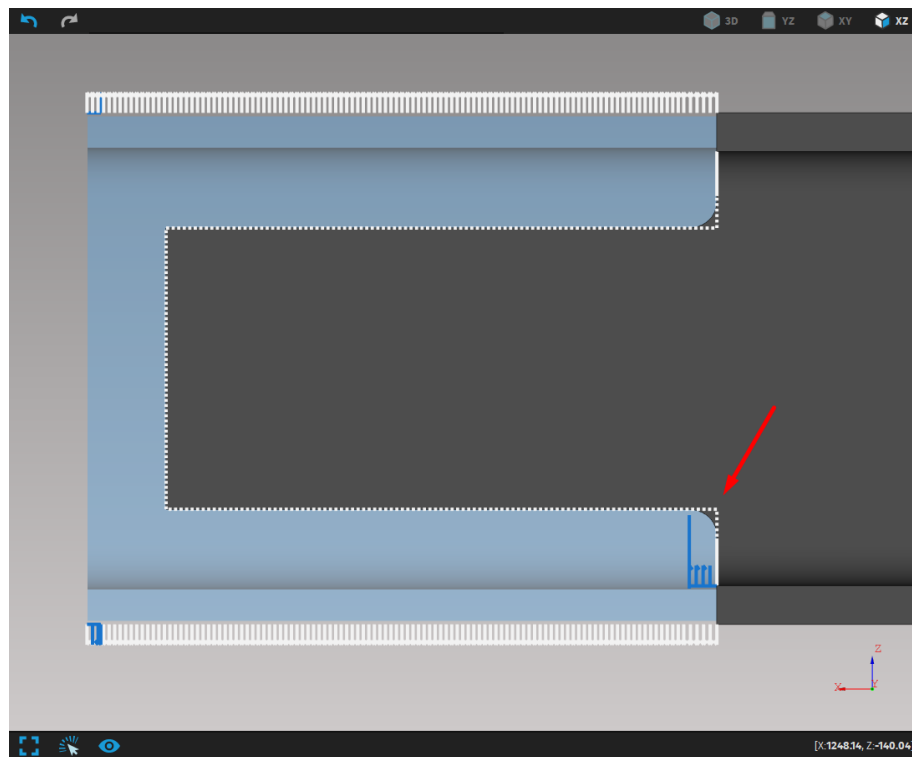
**Fig. 43 Difference between the 3D model and the real outcome of cutting**

3. The height of the nose on the flanges should be consulted with technology department to determine what size of the nose is the best for given thickness.



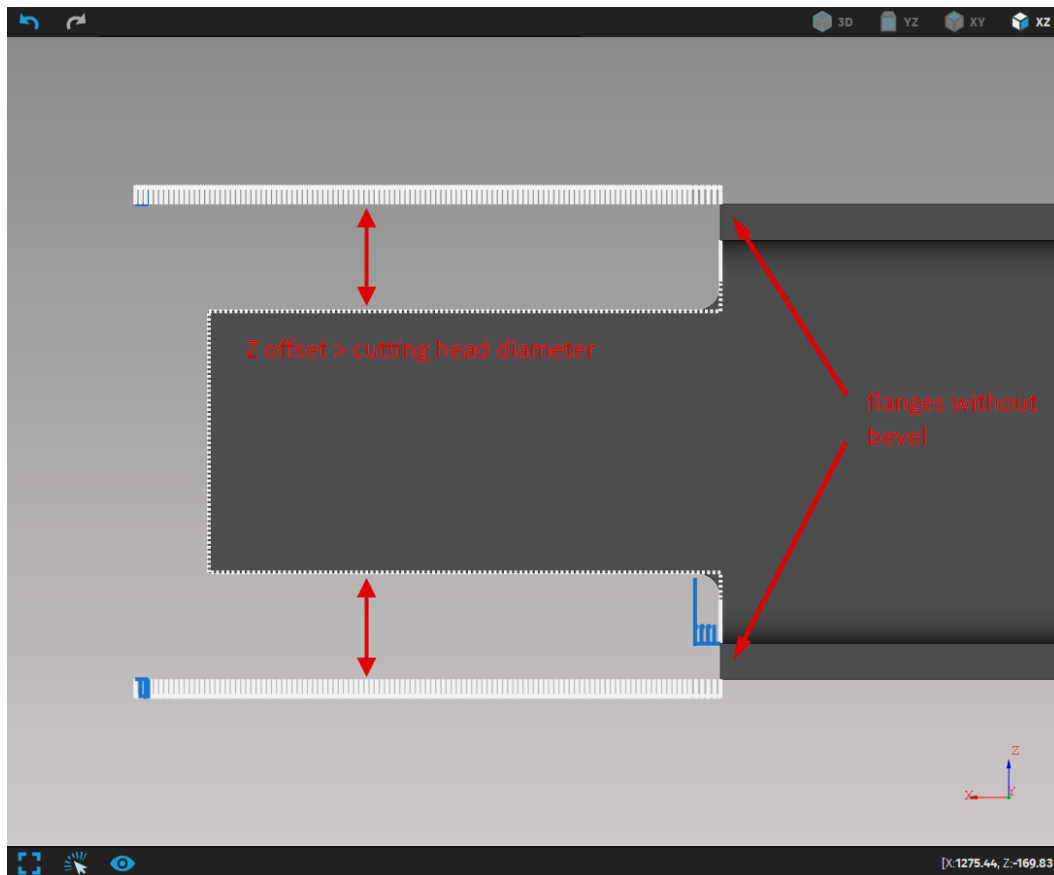
**Fig. 44** It is recommended to consult the height of the nose with technology department

- It is recommended not to draw radiuses on the inner side of the web, where the rathole is located.



**Fig. 45** The model should be drawn without the radiuses

5. If the Z offset (+ and -) is bigger than the diameter of the cutting head, the tongue ending could be cut without using the macros. If this is the case, the inner radiuses (as mentioned in previous point) should be drawn in step file (if they need to be cut). Also, if the tongue ending is cut using standardly generated cutting path, the flanges have to be without bevels.



**Fig. 46** Cutting the tongue without using the macros

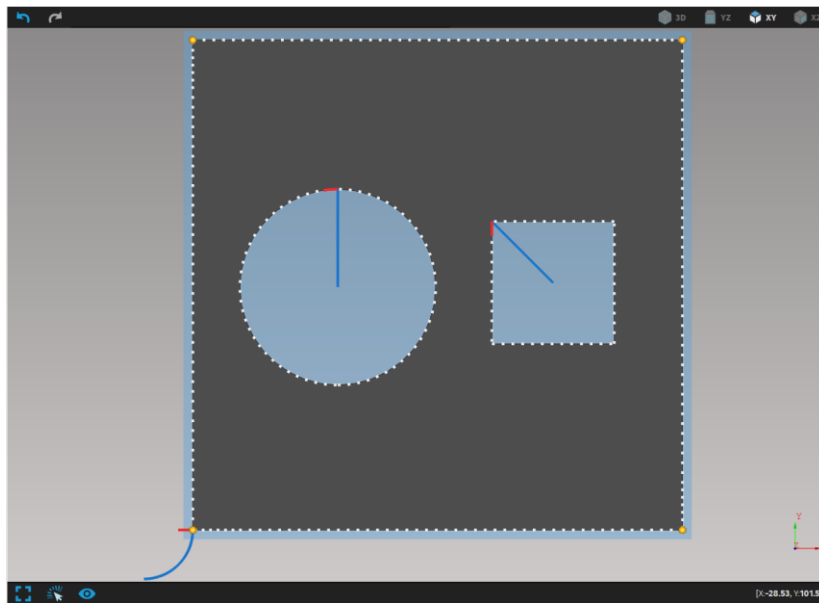
# Settings changes

## *Lead in/out*

In section Lead in/out, two types of leadins/outs for Hole and Trim cut were removed and two new types of leadins/outs were added – Convex Corner and Contour. They set the leadins/outs based on the location of the leadin/out and not based on the type of the contour (hole or trim cut). Also, separate section was added for setting the leadin/out for welding cut paths.

The final length of lead-ins/out is calculated as base length plus (thickness multiplier times thickness of the material.) Thickness of the material is base thickness not effective cutting thickness.

For small holes or cases when the lead in/out does not fit inside of the hole, the lead in will begin in the middle of the hole, regardless of what is set as default in settings.



**Fig. 47** Lead-ins on small holes begins in the center of the hole










Application	Cut path	Machine limits	Tool operations	Nesting / supports	Lead in/out	Measure	Generate	Intensity	Transformations	Expert table	Internal
<b>Convex corner</b>											
		<b>Lead-in</b>				<b>Lead-out</b>					
		Type	Linear (1 section)		Type	Linear (1 section)					
		Length	0.00 mm		Length	0.00 mm					
		Length multiplier	1.000		Length multiplier	1.000					
		Tangential angle	0.0 deg		Tangential angle	0.0 deg					
		Check collision	<input type="checkbox"/>		Check collision	<input type="checkbox"/>					
		Section angle			Section angle						
		Tool orientation	Straight to bevel 2		Tool orientation	Bevel					
<b>Contour</b>											
		<b>Lead-in</b>				<b>Lead-out</b>					
		Type	Arc (90 deg)		Type	Path conforming					
		Length	0.00 mm		Length	4.00 mm					
		Length multiplier	1.000		Length multiplier	0.000					
		Tangential angle			Tangential angle						
		Check collision	<input type="checkbox"/>		Check collision	<input type="checkbox"/>					
		Section angle			Section angle						
		Tool orientation	Straight to bevel 2		Tool orientation						
<b>General contour</b>											
		<b>Lead-in</b>				<b>Lead-out</b>					
		Type	None		Type	Linear (1 section)					
		Length			Length	3.00 mm					
		Length multiplier			Length multiplier	0.000					
		Tangential angle			Tangential angle	0.0 deg					
		Check collision	<input type="checkbox"/>		Check collision	<input type="checkbox"/>					
		Section angle			Section angle						
		Tool orientation			Tool orientation	Bevel					
<b>Slot</b>											
		<b>Lead-in</b>				<b>Lead-out</b>					
		Type	None		Type	None					
		Length			Length						
		Length multiplier			Length multiplier						
		Tangential angle			Tangential angle						
		Check collision	<input type="checkbox"/>		Check collision	<input type="checkbox"/>					
		Section angle			Section angle						
		Tool orientation			Tool orientation						
<b>Flange cut</b>											
		<b>Lead-in</b>				<b>Lead-out</b>					
		Type	Slide edge		Type	None					
		Length	0.00 mm		Length						
		Length multiplier	1.000		Length multiplier						
		Tangential angle			Tangential angle						
		Check collision	<input type="checkbox"/>		Check collision	<input type="checkbox"/>					
		Section angle	90.0 deg		Section angle						
		Tool orientation	Straight to bevel 2		Tool orientation						
<b>Web cut</b>											
		<b>Lead-in</b>				<b>Lead-out</b>					
		Type	Web lead		Type	None					
		Length	10.00 mm		Length						
		Length multiplier	0.000		Length multiplier						
		Tangential angle			Tangential angle						
		Check collision	<input type="checkbox"/>		Check collision	<input type="checkbox"/>					
		Section angle			Section angle						
		Tool orientation			Tool orientation						
<b>Welding</b>											
		<b>Lead-in</b>				<b>Lead-out</b>					
		Type	Linear (1 section)		Type	Linear (1 section)					
		Length			Length	5.00 mm					
		Length multiplier			Length multiplier	0.000					
		Tangential angle	0.0 deg		Tangential angle	0.0 deg					
		Check collision	<input type="checkbox"/>		Check collision	<input type="checkbox"/>					
		Section angle			Section angle						
		Tool orientation			Tool orientation	Bevel					

Fig. 48 New section for lead ins/-outs

## ***Generate***

New section C axis movements was added to Generate settings, and respective settings that are related to C axis movements were moved here. This section contains settings: Default mode for cutting circle pipe, Default mode for cutting profiles, Linearize C-axis movements on profile corners without rotator and Prefer to use the same C-axis constant value for cuts on one hole.

New settings were added:

- **Prefer to use the same C-axis constant value for cuts on one hole** – if checked, the C axis value will be the same for all cuts in complex Y, K and X cuts (if it is possible)
- **Collision limitations on dome** – if it's necessary to cut near the bottom edge of the dome, the collision limitations can be turned off by unchecking the checkbox beside this setting. It has to be used with great caution and turned off only for specific cases because if the dome was placed on the ground, without the limitations of the cut path, the whole cutting head would collide with the ground and break. The dome has to be placed on some kind of supports and there has to be enough space around/below it that the cutting head will not collide with the ground.

## ***Measure***

There are several changes in settings for measuring modes for profiles and pipes. All settings in this tab, that were previously associated with laser scanner detection for profiles are now applicable also to surface detection for pipes.

The section Laser detection was renamed to Profile, beam and pipe shape detection. Regardless if Scanner surface detection or Default surface detection mode is selected, the settings in this section apply. The following settings apply for these instructions:

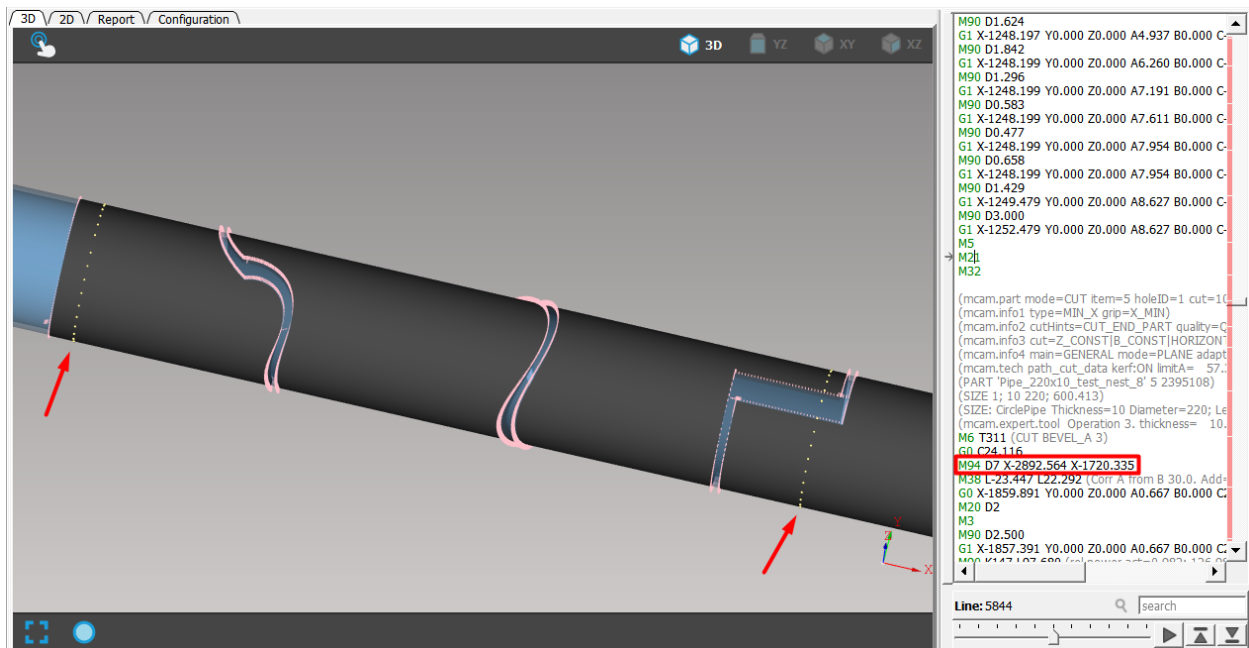
- M94 D7 – measuring pipes – only new machines support measuring on two X-positions
- M94 D12/D13 – measuring beams – laser scanner measuring always supports measuring on two X-positions

## **Profile, beam and pipe shape detection settings**

- **Measure mode**
  - Use one measurement for more cuts – one measurement is made for cut paths that are grouped according to set length in Maximum length for one/two X-position measurements
  - Measure every cut separately - for each cut separate measurement is made
- **Can use two measure positions** – for machines, that can execute the surface detection on two X positions for one group of cuts.

- New machines (only for circular pipes) – mCAM gets the information if it is possible to measure pipes on two X-positions from machine via webservice and automatically checks/unchecks this setting based on this information.
- Old machines (for profiles) – it is necessary to manually check/uncheck this setting, if the machine is capable to perform measurement for profiles on two X-position or not or if the 2 position measurement should/shouldn't be used
- **Maximum length for one X-position measurement** – one measurement is sufficient for group of holes and cuts that don't exceed set width in X-axis
- **Maximum length for two X-position measurements** – for group of holes and cuts that exceeds the width in X-axis for one measurement, two measurements are necessary
  - When the length of cut path exceed the set length in X-axis for two X position measurements, it is recommended to split the cut path
- **Length of cut off material when new measurement is necessary** - when the length of cut-off material exceeds the entered value, new measurement is necessary


The measurement cannot be done on X positions where some holes were already cut. If it is the case, it uses the closest possible position for measurement.






**Fig. 49 Measuring on two X positions for group of cuts on pipe**

# Simulation screen

## Touch controls










It is now possible to use touch controls in visualization screen in simulation. Touch controls are activated by clicking on the icon  in the top left corner of the visualization screen. After activation, new icons appear, that serve for different kinds of movement.

Icon	Description
	Move the view
	Rotate the view
	Zooming in/out of the view

Active icons are displayed in blue colour, non-active icons are grey. Zooming in is done by swiping with finger across the area which should be zoomed in. Zooming out is done by tapping anywhere in the visualization screen. Multiple taps may be needed to zoom out completely.

## 2D visualization

New controls are available for 2D visualization of CNC code:

Icon	Description
	Movie the view
	Zoom the view
	Select line from plan
	Fit all to screen
	Display uncompensated cutpaths if they are generated by machine control system
	Display tool movement
	Display shape type
	Display supports
	Track tool during cutting

**Edit CNC** – serves for editing the CNC code

**Abort edit** – Cancels all changes made to CNC code

**Save** – saves all changes made to CNC code

Two new plots are also available in simulation.

1. **Plot XYZ axes (s)** – shows changes in X, Y and Z axes in time during cutting
2. **Plot A/B/C axes (s)** – shows changes in A, B and C axes in time during cutting

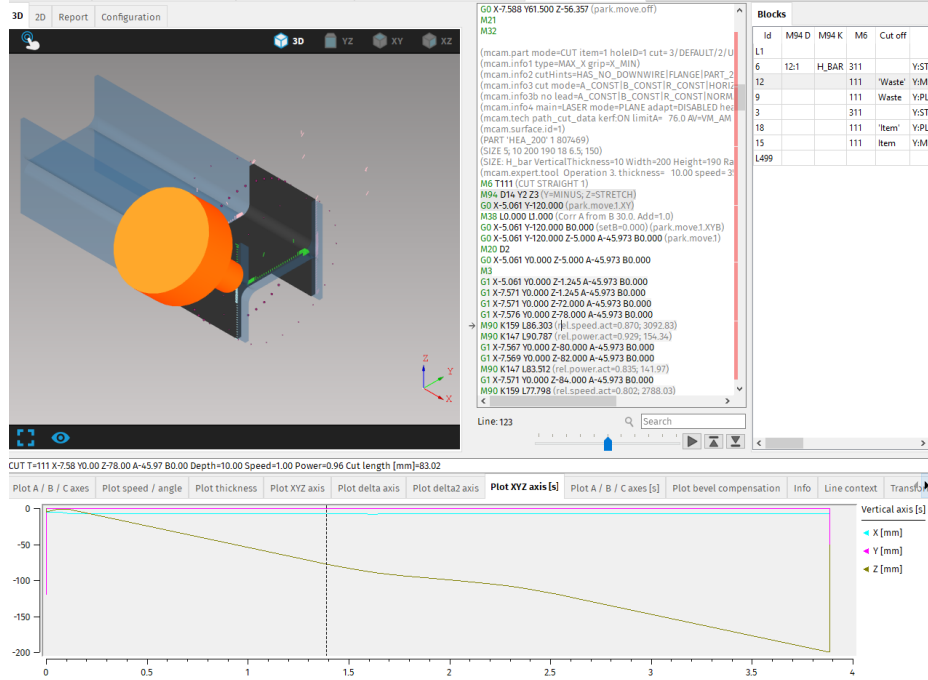


Fig. 50 Plot XYZ axes (s)

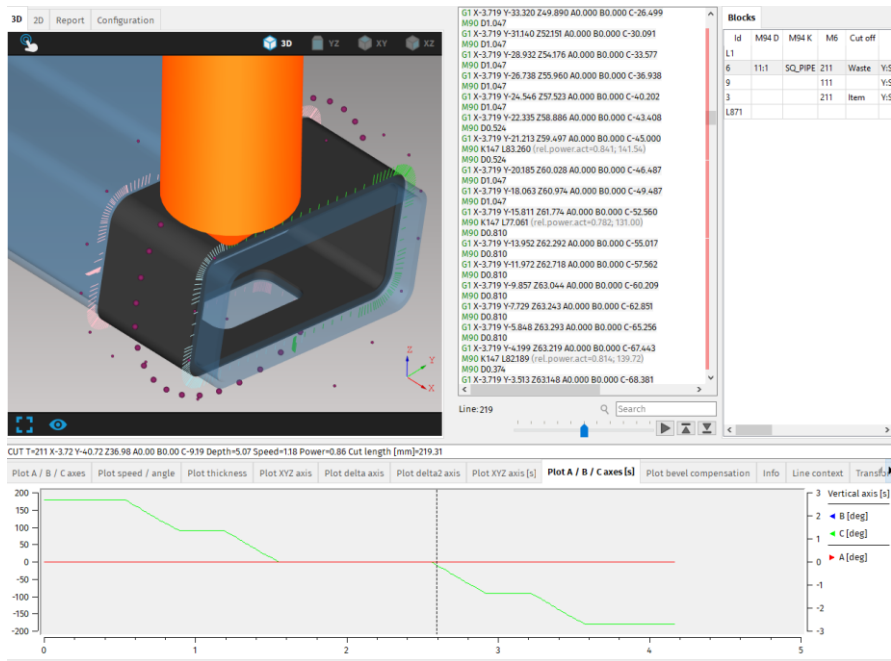


Fig. 51 Plot A/B/C axes (s)