mCAM

&

mCAM Lite

&

mCAM Pipe+

USER MANUAL

November 2023

Document UAW-02-V2.7-23EN

Table of contents

Using the manual

The manual is organized into several parts that individually deal with particular section or topic of the program. The first part deals with introductory information about the basic program principles and possibilities.

The second section *Installation* specifies basic system requirements, describes installation procedure and the verification system.

The next part *File formats* specify available input and output formats, all supported shapes, as well as the databases that can be involved in input formats.

The section *User interface* contains fundamental information about the program, program appearance and model rendering possibilities.

The section *Application workflow* introduces a basic procedure of working process summarised in few steps starting from loading of input files until generation of CNC program.

The next section *Cutting path modifications* describes how cut paths can be modified.

Section *Cutting plan preparation* that deals with program task (*Import, Job task, Library*), the function Drag and drop, machine and WebService setup, expert tables setup – kerf width compensation, nesting or cutting plan generation.

The section *Settings* describes all properties and configurations of *machine, technology, tool codes, expert tables of cutting parameters, transformations* and other presets.

The section *Troubleshooting* contains the program information, access to program documentation, describes *shape geometry data* inspection and *Reporting* system.

The last section *Working with MPM* describes standard working procedure in *mCAM* integrated in *MPM*.

> Note: The mCAM is in the state of permanent development, so this user guide describes version 2.0. Different versions might contain some new functions, some functions can be removed or modified. The version number and release date can be checked in the menu: "Help → About...".

Introduction to mCAM Lite

mCAM Lite is an easy-to-use CAM software tool combining the powerful mCAM engine with a user-friendly interface for easy and professional preparation of cutting plans for straight cutting of any supported part – by means of cutting with straight cutting head only (not bevelling heads). By using *mCAM Lite* it is possible to process any workpiece shapes standardly supported by *mCAM* –pipes, tubes, profiles, domes, but they are limited only to straight cutting with straight cutting torch (not possible to process beams as H-beams/I-beams, U-channels or L-bars as the bevelling is needed).

Note: In case the 3D model contains welding preparations (bevels) mCAM will limit these contours and generate them in straight cutting path only. Contours which are supposed to be bevelled or chamfered after straight cutting (by manual grinding or manual chamfering) can be marked (marking of ideal bevelled contour) in order to visualize the area for subsequent grinding (possible to use only for top side bevels = V-type and Y-type bevel).

Supported Automation level – manual loading only

mCAM Lite is designed for advanced straight cutting, using energy-beam technologies (e.g. plasma, laser, oxy-fuel, water jet) and supports only machines with manual loading of workpiece into the cutting area (tubes, profiles, beams, etc.). Machines with extended level of automation of workpieces are not supported (tube loaders, beam loaders, etc.).

Supported formats – STEP and IGES formats only

The software can process any supported 3D model geometry generated by commonly used CAD softwares (such as SolidWorks, 3D Inventor, Autodesk 3D, CATIA, Parasolid, PTC Creo or any other CAD SW) with possibility to export STEP or IGES formats (STEP format - *.step, *.stp; IGES format - *.iges, *igs).

Additional *"mCAM friendly formats"* that are standardly supported (exported) by structural BIM Softwares (such as TEKLA, Aveva BoCAD, Revit, Rhinoceros or many other), such as format DSTV (*.nc, *.nc1) or parametrized tube connections (*.xlm), are available only in higher versions of *mCAM* (not supported by *mCAM Lite*).

Simple shapes and standardized connections (notches) – *mCAM macros*

mCAM Lite integrates also the functionality to prepare standardized connections and notches by using *mCAM macros*. Integrated library contains standard and most-commonly used types of connections for pipes, tubes and profiles (similar to SolidSel or PipeSel). In case of using macros, the cutting path generated after will still be limited to straight cutting only. Most common types of macros are already included in mCAM and mCAM Lite, and new shapes and types will be added gradually to the mCAM library with following updates.

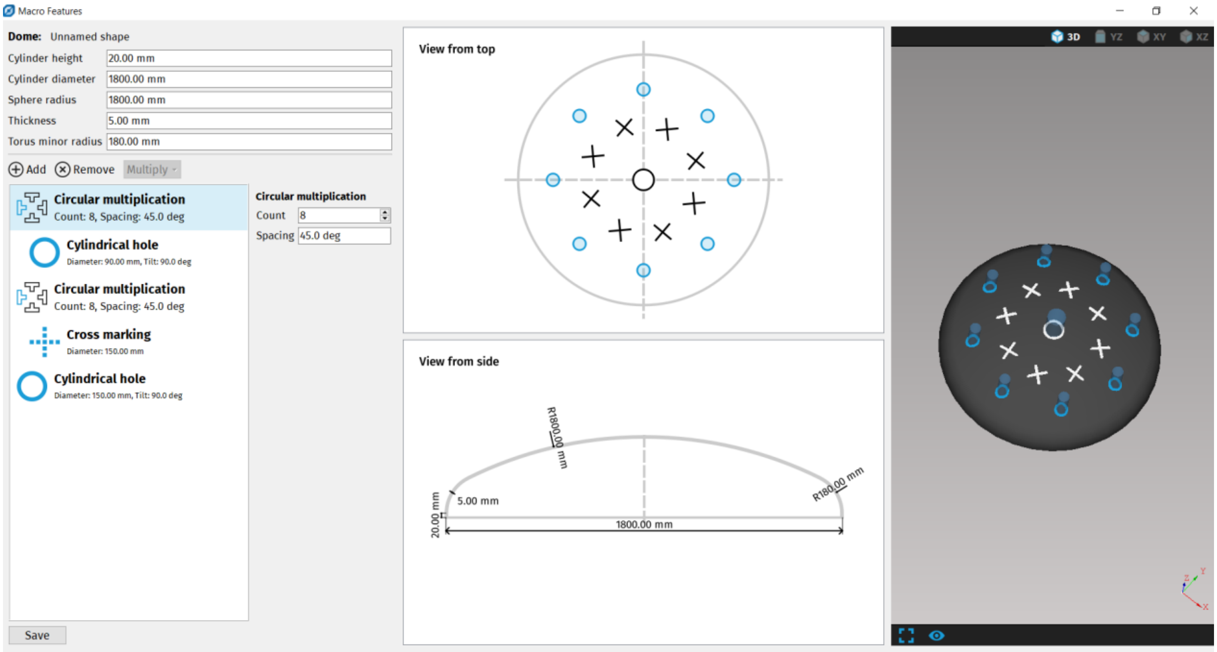


Fig. 1: An example of macro – Cylindrical hole

mCAM Lite functionality – standard mCAM functionality included

Rest of *mCAM* functionality including the marking process (Ink-jet, Laser marking, etc.), additional operations such as Drilling or Punching, 2D/3D visualization of parts, 3D cutting simulation, WebServices (connection to machine) is available and fully supported by *mCAM Lite* version.

Introduction to mCAM Pipe+

mCAM Pipe+ is a CAM (Computer Aided Manufacturing) software that is capable of preparing cutting plans only for cutting of circular pipe, circular pipe segments and sheets in a full 3D mode.

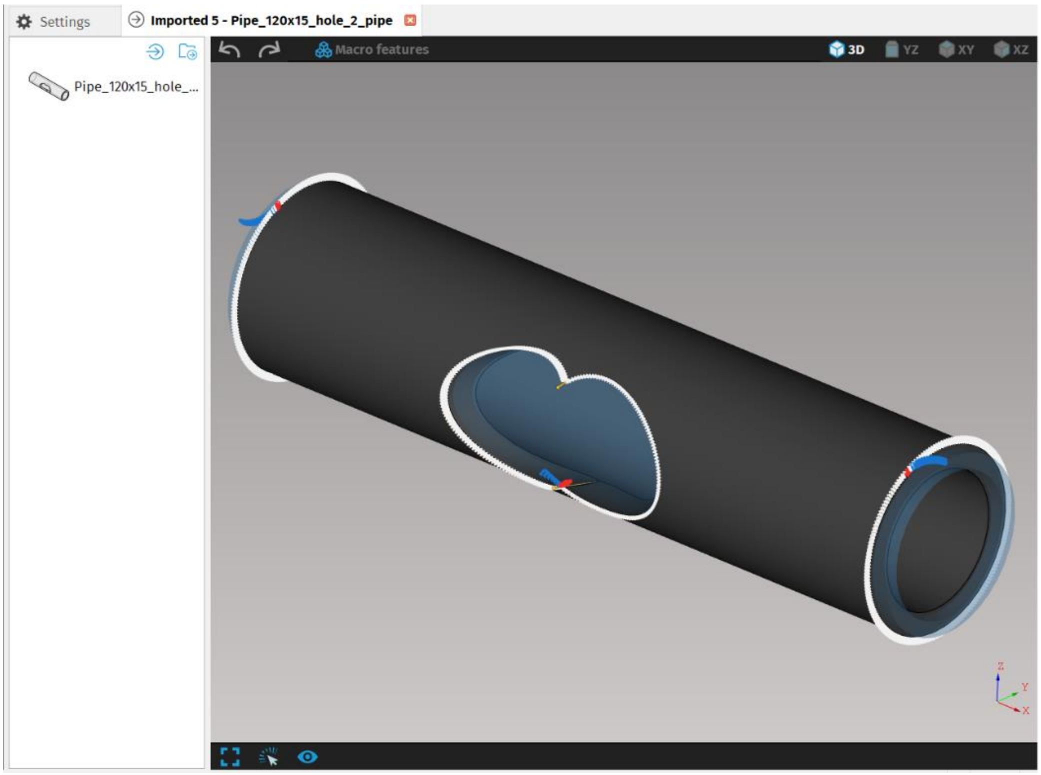
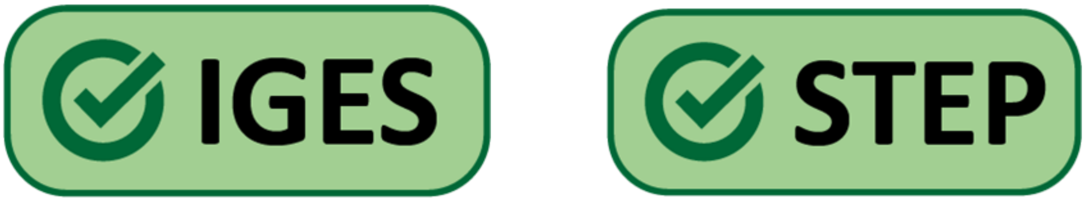


Fig. 2: Imported circular pipe from STEP file

mCAM Pipe+ can process 3D geometry, has support for bevel cutting and for preparations for welding; also supports cutting sheets at an angle and supports Additional Bevel Processing (ABP) for sheets.

Supported are only STEP or IGES formats (STEP format - *.step, *.stp; IGES format - *.iges, *igs).



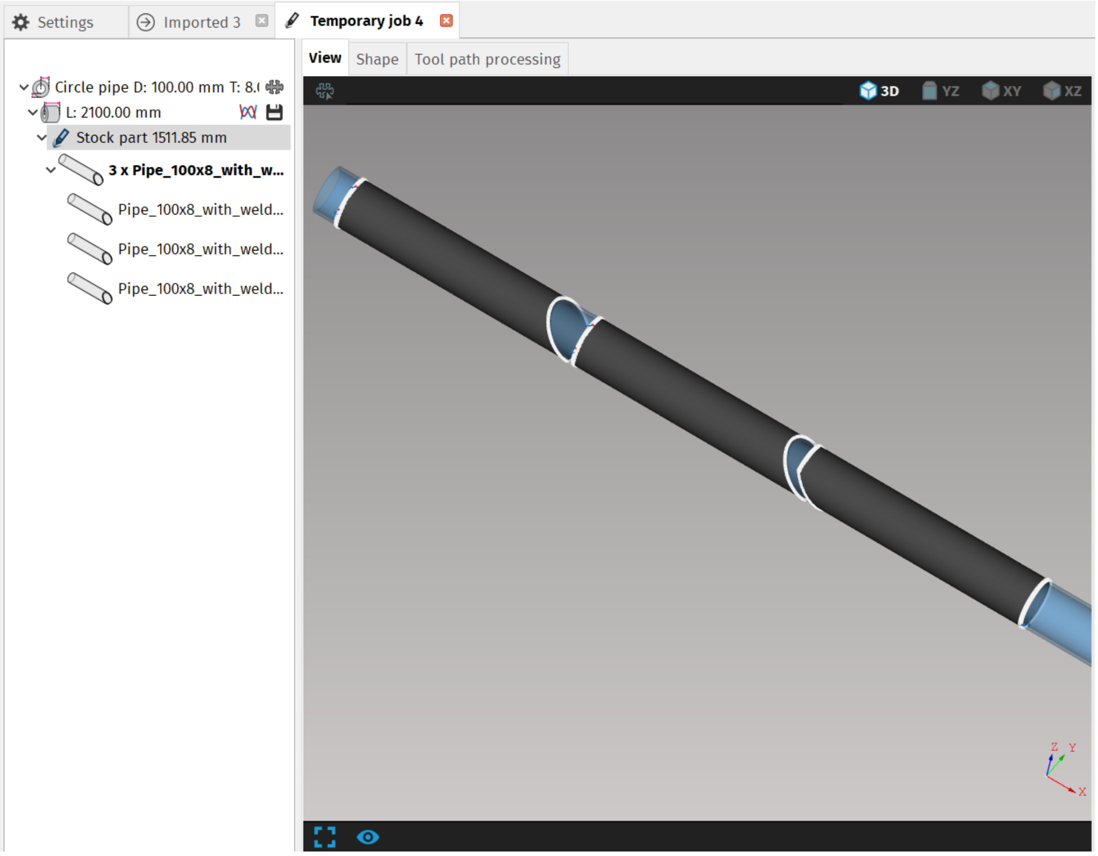


Fig. 3: Nested circular pipes in a Temporary job

This software also contains a library for macros. All macros contained in this library are only for circular pipe.

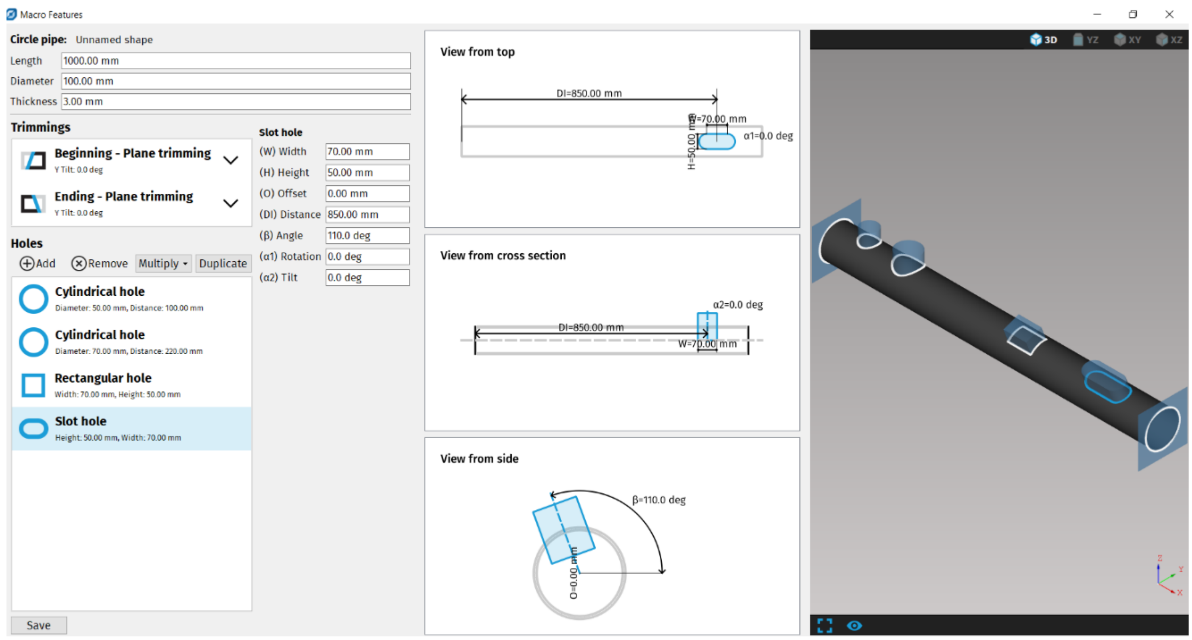


Fig. 4 Examples of macros for Circular pipe in Macro feature

Introduction

mCAM is a CAM (Computer Aided Manufacturing) software for professionals preparing cutting plans for 3D cutting. *mCAM* is designed for advanced automated 3D machining using energy-beam technologies (e.g. plasma, oxy-fuel, water jet, laser, etc.).

The software processes any supported 3D model generated by commonly used CAD software (Computer Aided Design) or any model–based software for steel constructions (e.g. Tekla) that can export steel structures in DSTV or xml. formats but generates cutting paths only for those shapes that are supported by mCAM.

The software creates libraries of frequently used parts, offers advanced visualization and helpful previews. Well-arranged user interface with variety of settings allows simple, highly efficient, and precise manipulation and setup of cutting paths and its features. The software includes also a built-in CNC simulator that simulates generated CNC code so user can check cutting paths.

Program features

The *mCAM* uses boundary representation (B–rep) for processing true realistic 3D shapes. *mCAM* supports several options for import of shape data:

- 3D CAD models
- Model–based steel structures (DSTV, xml)
- 2D surfaces geometrically projected on 3D parametrized shapes
- libraries of standard shapes

The *mCAM* analyzes the shape of 3D models and automatically generates all cut paths. The program provides useful and powerful functions such as tool kerf compensation, placement of microjoints, management of lead–in s/–outs, simulation of machining, as well as interconnection of *mCAM* with *MPM* (*MicroStep Production Management*). The functions in combination with used framework and user – friendly interface make the program efficient, simple, and well organized.

Boundary representation

Boundary representation is a method for representing shapes using the bounds. A SOLID is represented as a collection of connected surface elements, the boundary between solid and non–solid. The 3D space inside SOLID is bounded by FACES. Every single FACE is defined analytically as bounded surface, e.g. cylindrical, spherical or BSPLINE surface. These faces put together form encapsulated solid. A face is bounded by one WIRE that consists at least of one EDGE. Every EDGE is defined analytically (lines, arcs, splines) and has two ending points (VERTEX).

The *mCAM* approach is based on a tree structure that describes the details of a 3D shape. SHELL contains several FACE-s, every FACE contains several WIRE-s. FACE could possibly contain some holes that are also defined by WIREs. Every WIRE contains at least of two EDGEs and an EDGE has two VERTEX-es (end points).

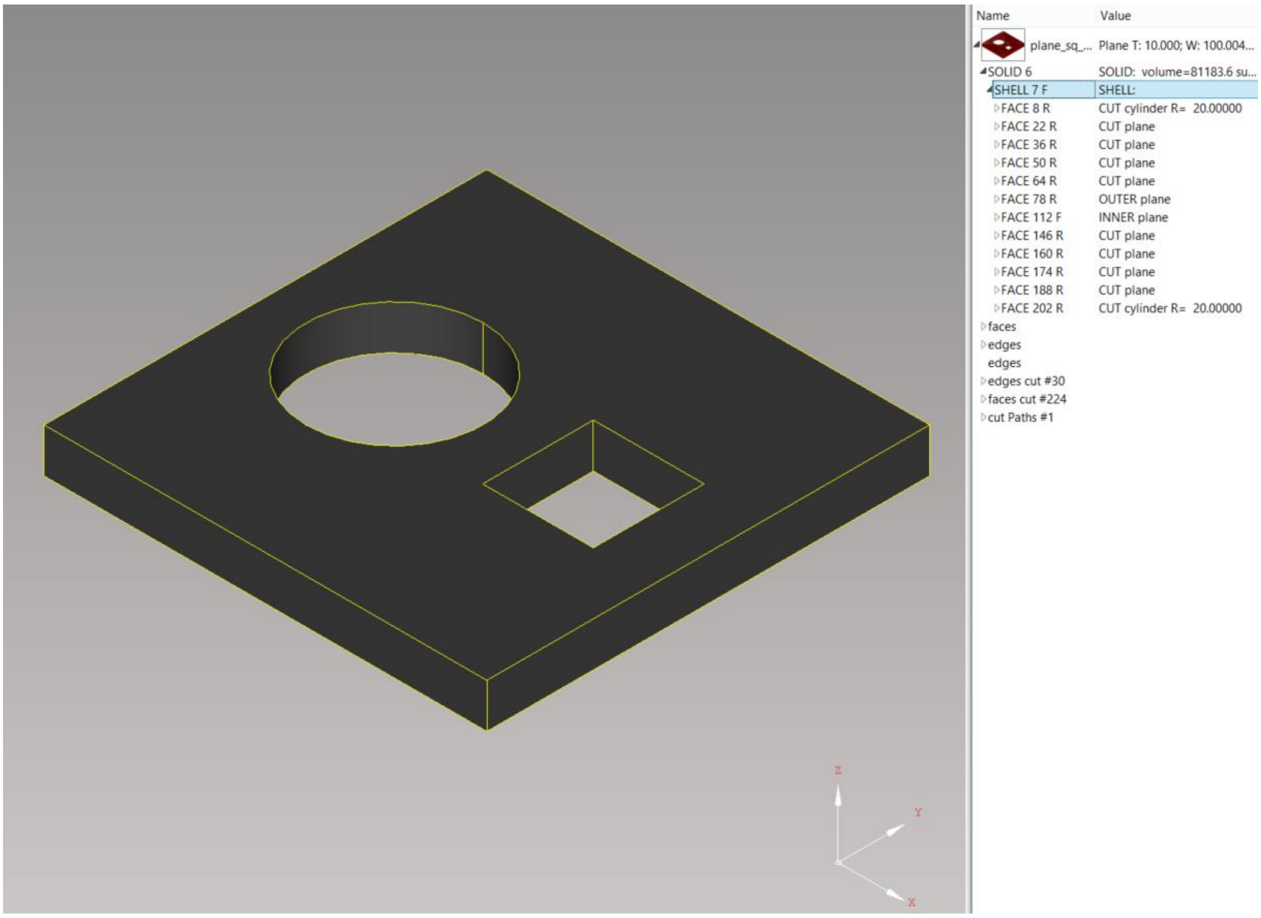


Fig. 5: B–rep model using 12 faces

In–depth shape analysis

By analysing raw geometrical data of 3D model, mCAM is able to recognize shape type and process cutting faces automatically. Due to shape type analysis and deep processing of all faces in solid geometry (that defines B–rep structure), *mCAM* is able to process almost any cutting surfaces and shapes including complex variable bevel cuts, Y, K or X–cuts.

After all cutting faces are processed, user is able to prepare a cutting plan in few steps. All cutting features, modifiers and tools are easily accessible in interface and it’s possible to modify them according to user requirements.

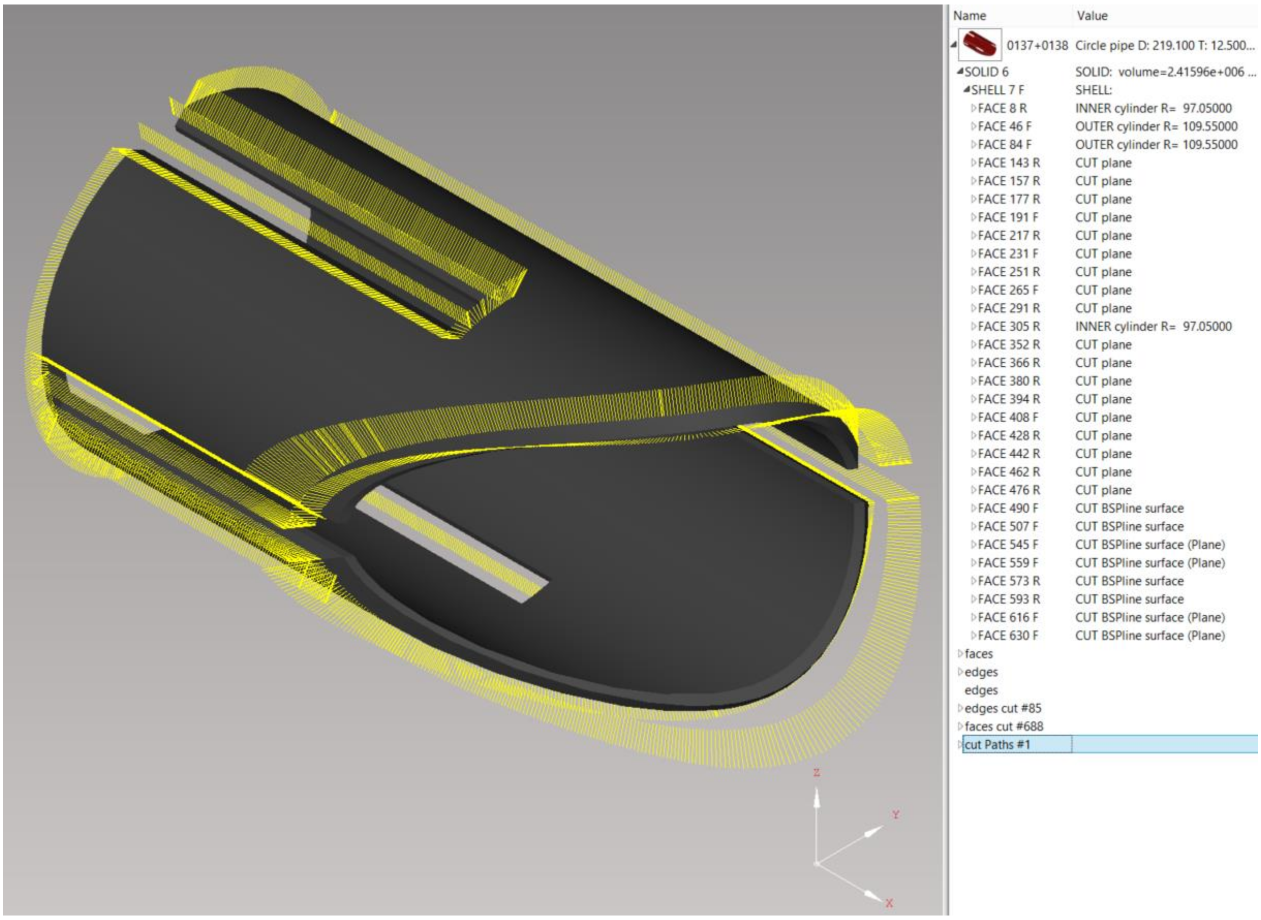


Fig. 6: In–depth shape analysis

Assembly importing

In mCAM it's possible to import a whole assembly of 3D models (STEP/IGES) and process every single part individually. It is necessary to save the assembly as separate solids when exporting from SolidWorks or another 3D CAD software. All imported parts are sorted according to type and dimensions, making the nesting, reports and generation of cutting plans very effective.

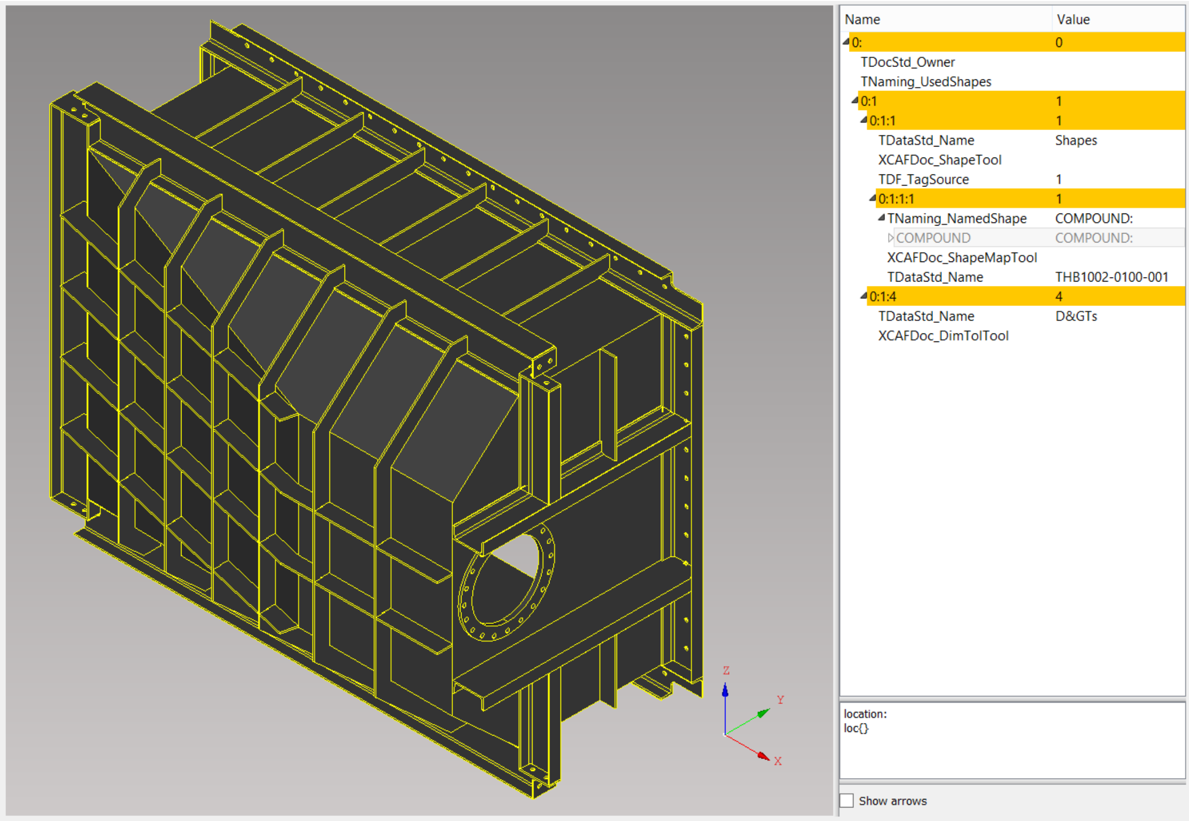


Fig. 7: Assembly import

WebService connectivity

WebService is a mandatory network application tool that connects a machine, its technology and tools database with *mCAM* via company network. With WebService user can take advantage of expert system of cutting parameters, download tool database from iMSNC automatically and fully integrate technological setup for particular machine and technology within company.

WebService is very useful when working with several machines, several technologies or advanced cutting because it increases effectivity when preparing CNC programs.

Installation

mCAM can be installed only under certain conditions. All necessary system requirements are specified in the next sections including a brief description of the installation procedure.

System requirements

mCAM program can be used on any commonly used personal computer under 64 bit version of operating system Windows 10 (version 1607 and above) and Windows 11. A necessary condition is a presence of a graphic card that supports OpenGL interface in version 2.0 or higher. Microsoft .NET framework 4.5 has to be installed. A storage disk drive has to have at least 500 MB free space.

Note: There are no minimum hardware requirements and *mCAM* can be installed on any computer with previously mentioned software requirements but due to high advanced graphic visualizations, complex nesting procedures and sophisticated limitation algorithms it is highly recommended to install the application on computers with sufficient performance as for standardly used 3D CAD software.

Installation procedure

Installation file can be downloaded from **https://customer.microstep.sk/**. It is possible to choose stable or beta version of the program in “Type” field on the website. Name of installation file looks like this: mcam_YYYY_MM_DD_8XX.release.x_64_setup.exe. YYYY is year, MM month and DD day when the built was created, 8XX.release.x refers to mCAM version and number of the release.

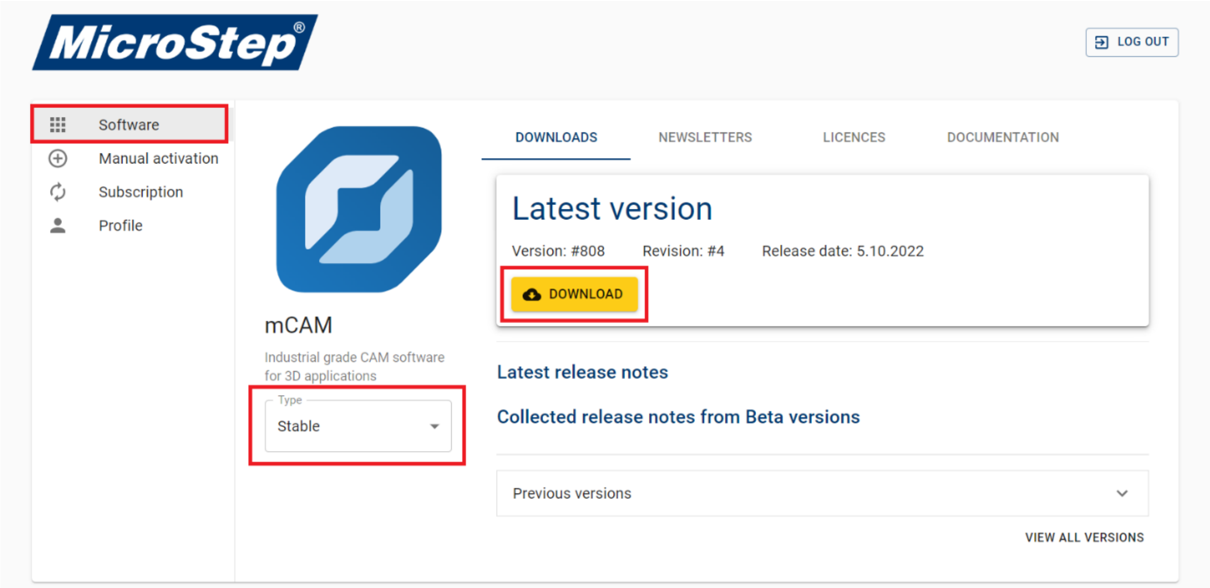


Fig. 8: **MicroStep Customer portal**

To install the *mCAM*, run the installation file from downloaded files folder in computer. In this procedure user selects which components should be installed, e.g. *MPM* and destination folder for application. The default installation folder is *C:\msnc\mcam*. The option *Clean installation* removes already existing files from previous installation.

Note: The installation target directory should not contain international characters.

Types of licenses

License is issued forever (except for service license). License is registered for specific computer. Therefore, user can have installed multiple versions of the program in different directories and use the same license for them. To release a license from a PC is possible in the mCAM program or by using the customer portal.

For proper functionality it is necessary to allow access via https protocol, to the addresses - "license.microstep.sk" and "auth.microstep.sk".

- ## Online license

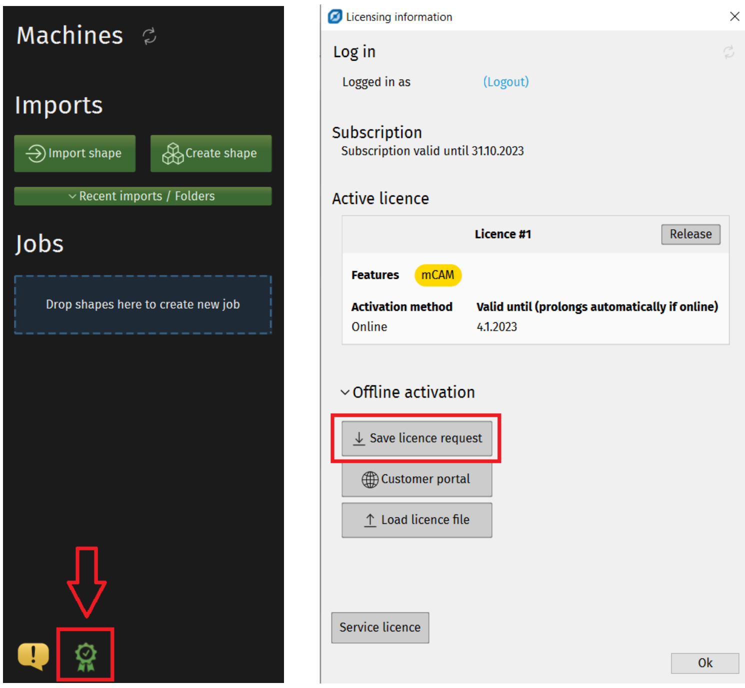
Online license is registered on servers and transferred to PC. It requires internet connection for proper functioning. Without Internet access the online license stored in the license client program directory is valid for 30 days.

- ## Offline license

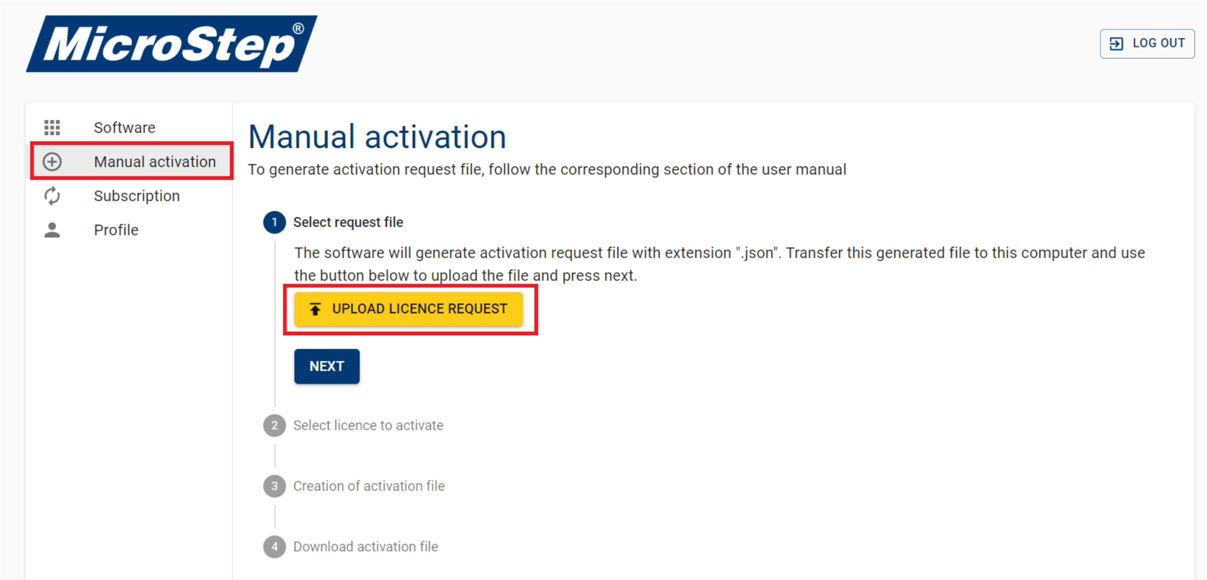
Offline license is generated in the mCAM/Asper program and stored as a file in license client program directory. The file can be transferred to an internet connected device and uploaded to MicroStep customer portal from where user will get the response in form of a file that is used for registration. Validity of this license is one year. Offline activation process must be done at the customers environment (it is possible to assist by using TeamViewer).

<u>Procedure for creating an offline license:</u>

1. In the mCAM program, by clicking on the license client icon (lower left), open the licensing information and choose **"Offline activation"** - **"Save license request"**



2. Select the folder, where the file will be saved
3. Log in to the Customer portal using your login data and select Manual activation



4. Click the "Upload license request" button to upload the file with the extension **".json"** that you have downloaded in the mCAM program
5. You will receive the activation file, select the folder for its location
6. In the mCAM program, click on the license client icon (bottom left), choose **"Offline activation"** - "Load license file" and upload the activation file that has been downloaded in the previous step
7. Offline activation is complete

- ## Floating license

Floating licenses are similar to online licenses with the difference that they are registered for a very short time - a few minutes, so they are easy to share between several computers. If the program is correctly terminated, it is automatically logged out of the floating license, which is immediately accessible to another computer.

- ## Service license

This license is available only for MicroStep employees. It is only valid for a few days. This license is the only one that does not transfer automatically to another installation on the same PC.

- ## Releasable license

A license that binds to a given PC upon the first registration and cannot be transferred to another PC. Serves as a machine license.

Supported shapes

By using in depth analysis of imported shapes, mCAM processes all cutting faces and generate cutting paths automatically. Shape type recognition is based on a principle of geometry detection of imported 3D shapes, so the geometry of any imported 3D shape has to meet geometrical parameters determined by each shape type. Current version of the mCAM supports detection of these 3D shapes:

- plane shape
- pipe (circle pipe) + circle pipe segments
- rectangular profile pipe
- torispherical domes – this includes all types of domes with various heights and radiuses ratios e.g. elliptical domes, elliptical domes (1,9:1 and 2:1) torispherical domes (DIN 28011), semi-elliptical domes (DIN 28013), tank domes for single and double skin tanks (DIN 6608/6616, DIN 6608/2, 6616/2) etc.
- EHA/EHB Elliptical domes – ISO9001:2008; GB/T25198-2010 (these domes have to be designed in a special way, for more information contact support team)
- conical dome
- dished discs
- dished bottom domes
- flat bottom domes
- inverted head domes
- H–beam (HE, HEA, HEB, HEM, HEAA, HD, HL)
- I–beam (IPN, IPE, IPEA, IPEAA, IPEAO)
- U–beam (UE, UPE, UPN, UAP)
- bent U-beam
- L–beam (equal/unequal angles sizes - EN 10056, EN 10162)
- Bent L-beam

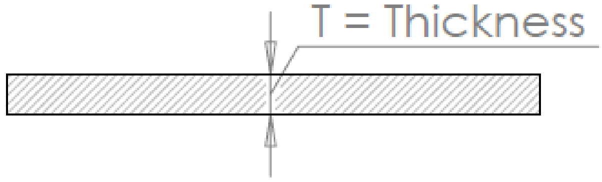


Fig. 9: Plane shape

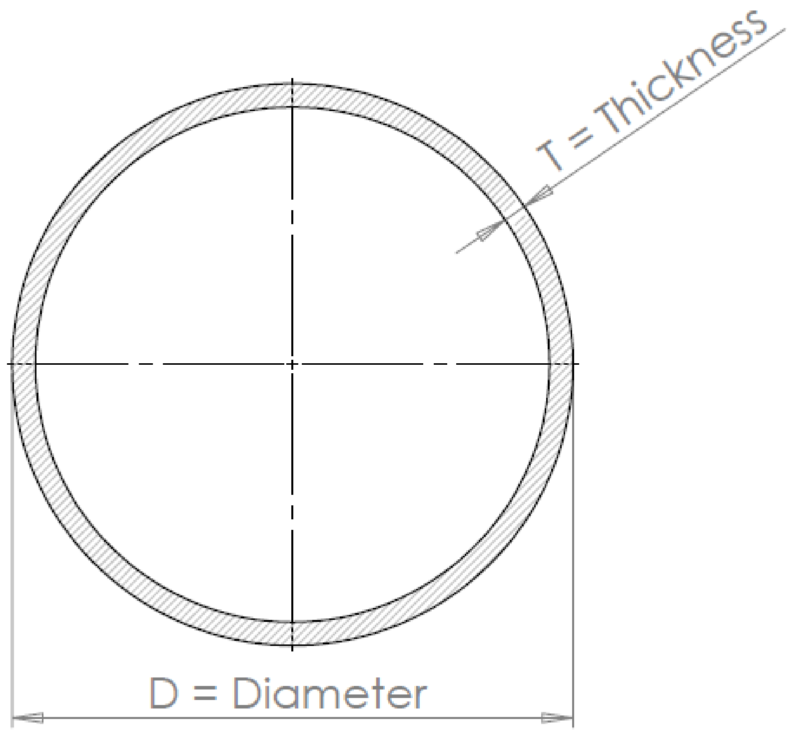


Fig. 10: Circle pipe

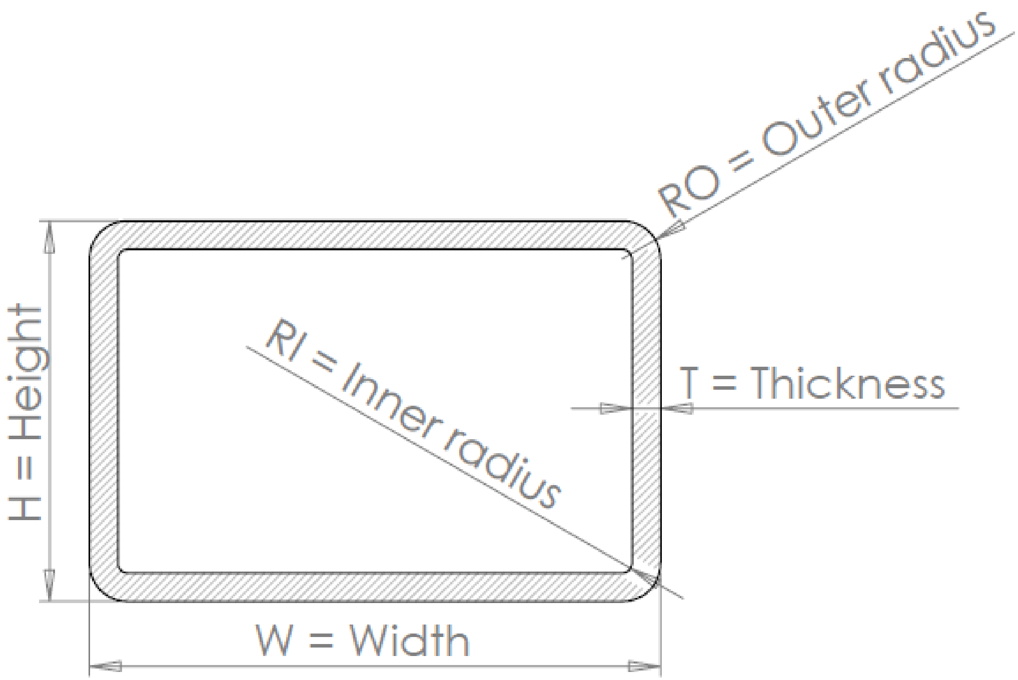


Fig. 11: Rectangular profile pipe

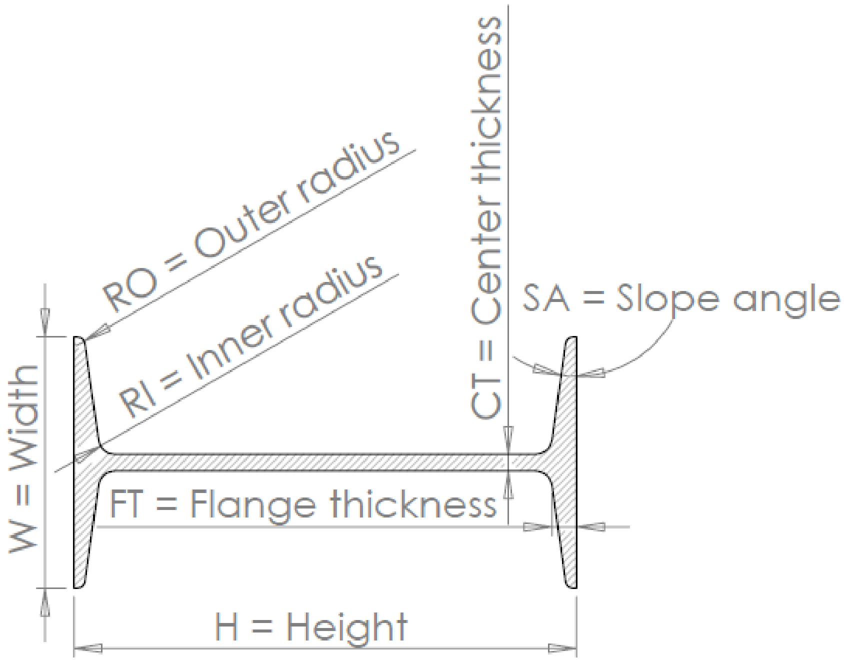


Fig. 12: I profile - IPN beam (non – parallel flanges)

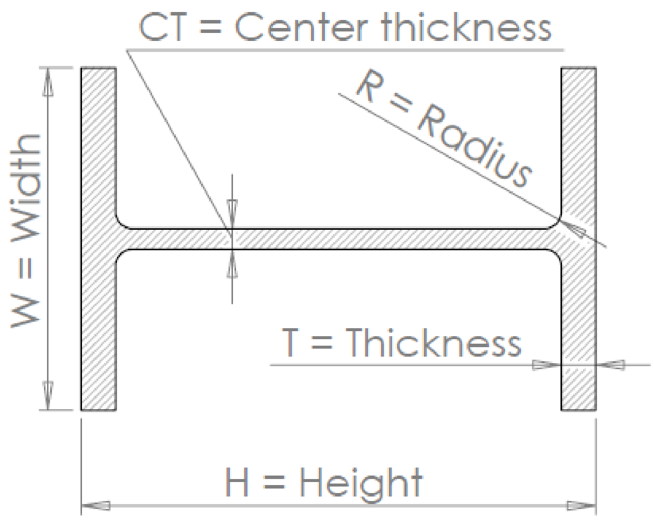


Fig. 13: H Beam - HEA, HEB, HEM and IPE beams (parallel flanges)

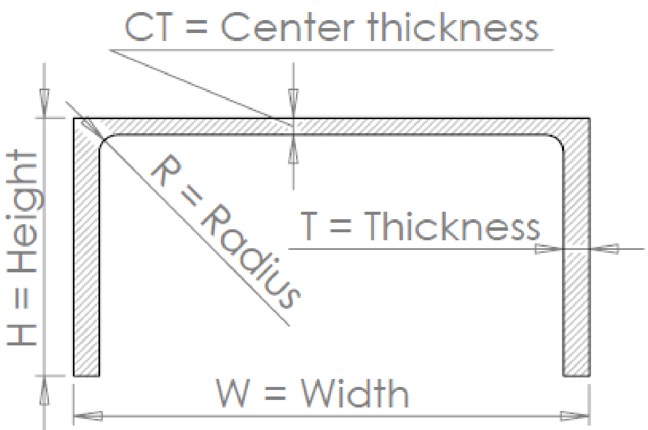


Fig. 14: UPE, UAP beams (parallel flanges)

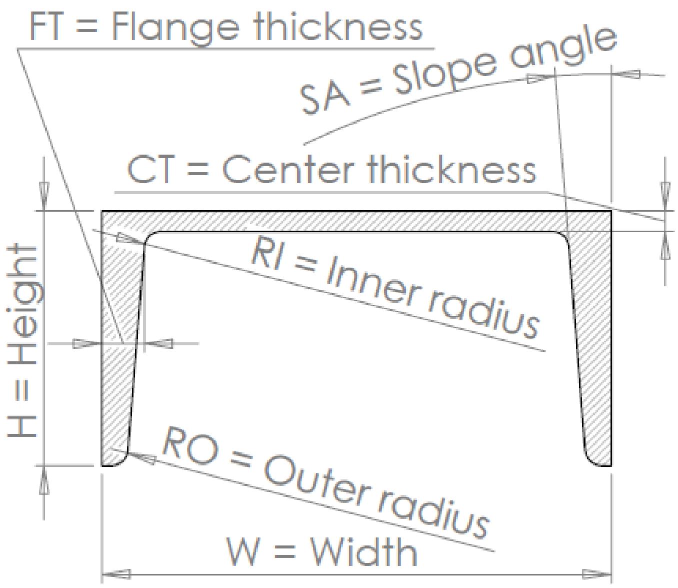


Fig. 15: UPN beams (non – parallel flanges)

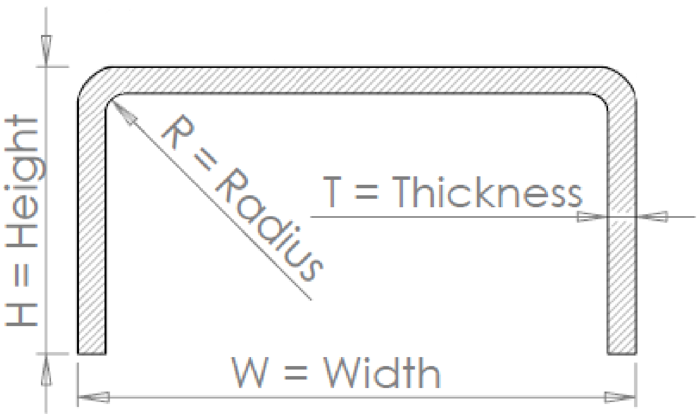


Fig. 16: Bent U-beam

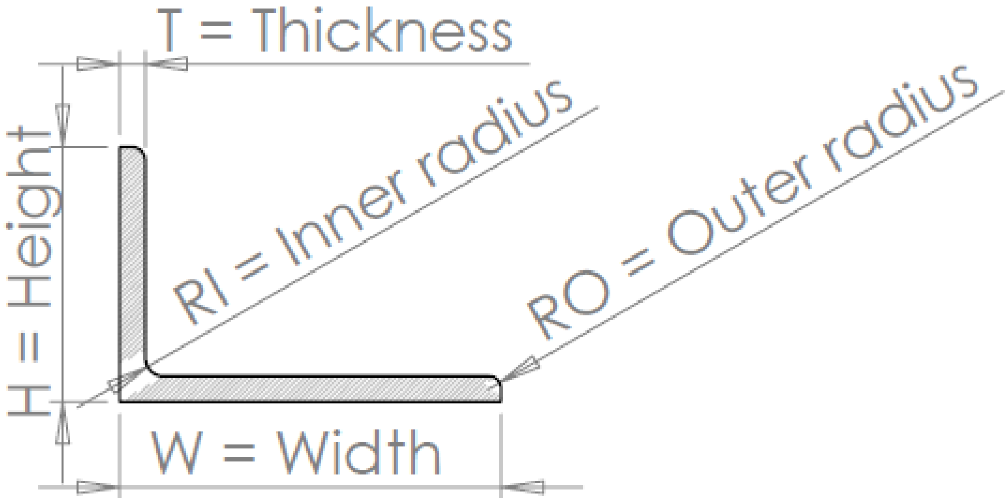


Fig. 17: L beams (equal / unequal leg angles)

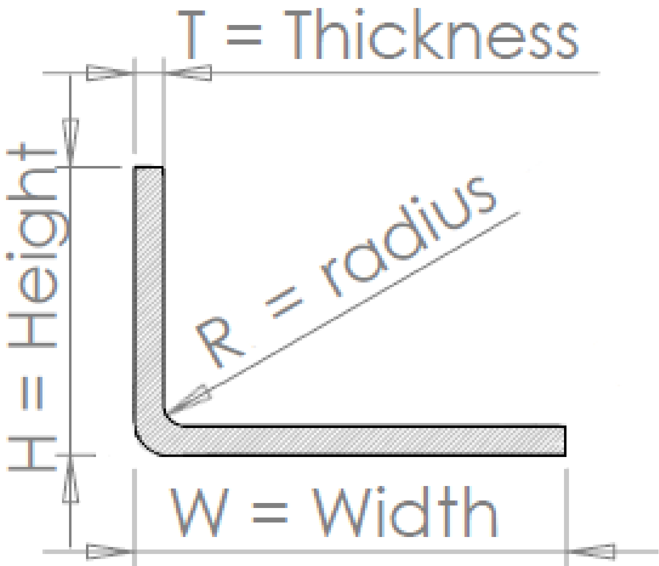


Fig. 18: Bent L-beam

Explanations of technical parameters

od – outer diameter
cr – crown radius
kr – knuckle radius
sf – straight flange
t – wall thickness

mw – margin width (dishedbottomed dome only)

oa – opening angle (cones only)

ahr – apex hole radius (cones only)

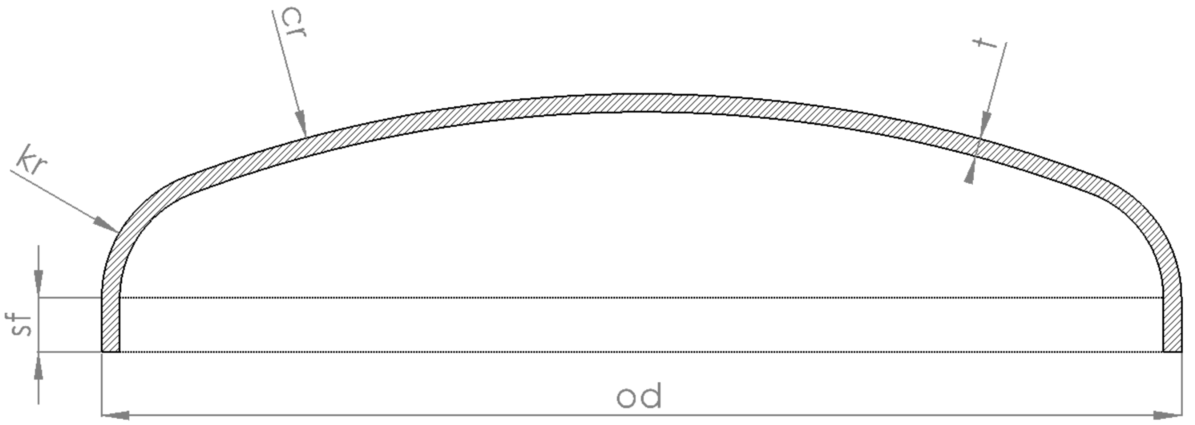


Fig. 19: Torospherical dome

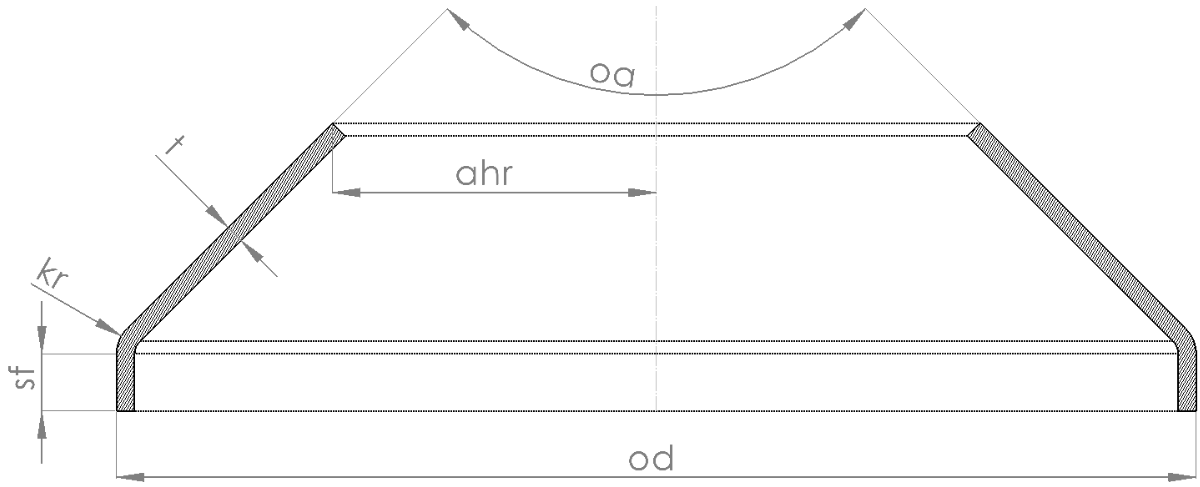


Fig. 20: Conical dome

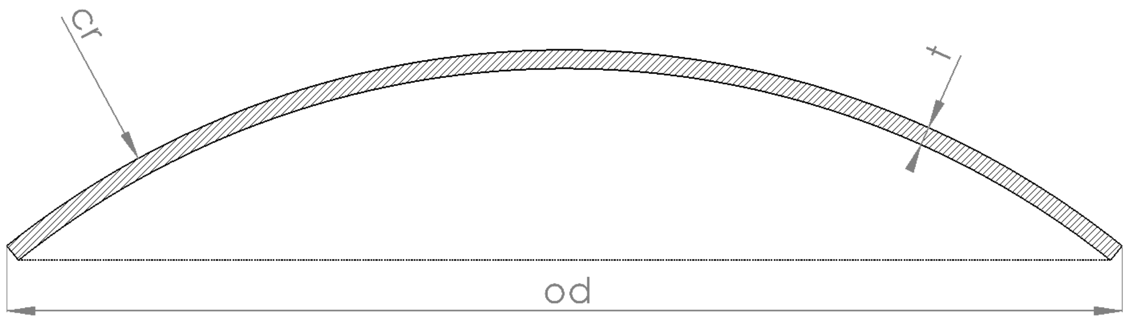


Fig. 21: Dished disc

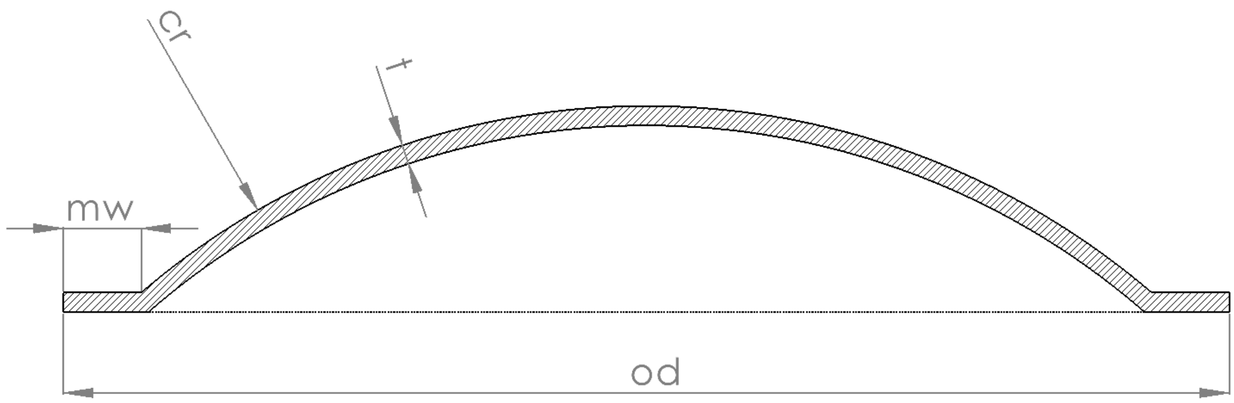


Fig. 22: Dished bottom dome

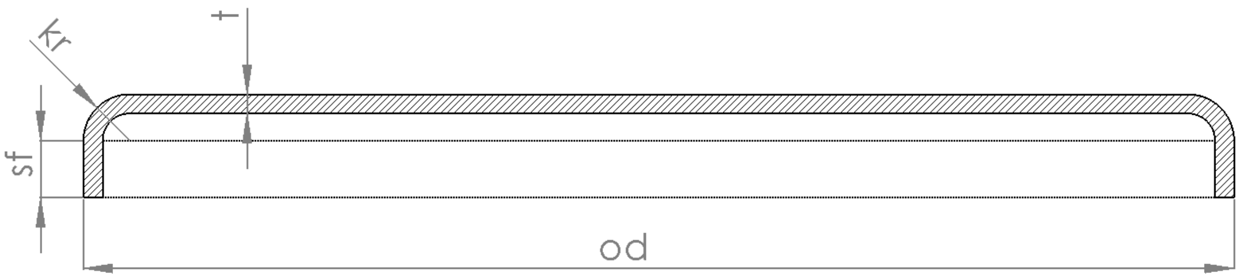


Fig. 23 Flat bottom dome

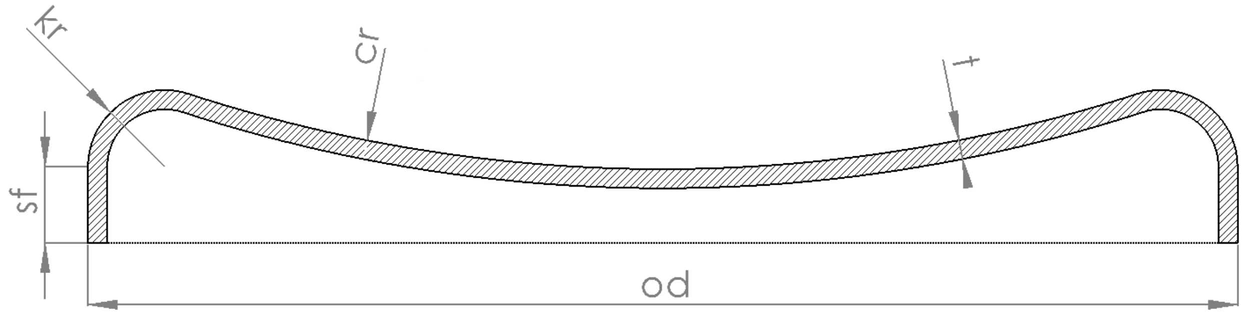


Fig. 24 Inverted head dome

Special recognition of shapes

When importing cones, it is necessary for correct recognition of shape type and all cut paths to define parameters of all types of cone semiproducts that are being used. This is done by manually editing a *shape_type.ini document located in “C:/msnc/mcam/config/”* folder.

Example:

```
[domecone]

domecone_thickness:3.00_cylinderRadius:1154.0_torusMinorRadius:100.00_apexHole
Radius:20.00_openingAngle:130.0=DOMECONE 1154

domecone_thickness:3.00_cylinderRadius:1154.0_torusMinorRadius:50.00_apexHole
Radius:20.00_openingAngle:140.0=DOMECONE 1154
-------------------- domecone
```

Each row represents one type of cone and its parameters – thickness, cylinder radius, torus minor radius, apex hole radius and opening angle. All dimensions are in [mm] except angle [°].

File formats

This section defines all input and output formats supported by the *mCAM*. Input formats are divided in groups according to their type.

Input formats

All supported input formats can be divided into four groups:

- CAD models (*.step, *stp, *.iges, *.igs)
- Model–based steel structures (*.nc1, *.nc, *. xml, *.ifc)
- Surfaces (*.cnc, *.dxf)

CAD models

To define a geometry of 3D shapes the *mCAM* supports commonly used STEP and IGES formats. STEP and IGES formats are generally included as export formats of all commonly used CAD software. Format names, file extension, and its descriptions are summarized in the table below:

Format name	**File extension**	**Description**
STEP	*.step, *.stp	Standard of Exchange of Product Data Model
IGES	*.iges, *.igs	Initial Graphics Exchange Specification neutral data format

Tab. 1: CAD models – Input formats

Model–based steel structures

Model–based steel structure data represents some commonly used formats that are possible to export from BIM software (Building Information Modelling) such as Tekla Structures or Tekla BIM sight. Exported model–based structure data contains geometry of each part, position in structure (in assembly) and information about used intersection type with another part in assembly nodal point.

Format name	File extension	Description
DSTV	*.nc, *.nc1	Industrial standard defined by the German Steel Construction Association (Deutsche Stahlbau–Verband)
XML	*.xml	Standardly used NC files for creation tubular hollow sections, tube–to–tube and tube–to–plate connections
IFC	*.ifc	IFC (Industry Foundation Classes) data format is intended to described building and construction industry data

Tab. 2: Model–based steel structures – Input formats

Note: Processing of these formats in mCAM is limited by their capabilities and limitations, therefore we recommend to use STEP format as main input format.

Surfaces – CAD drawings and SolidSel library

Final 3D model can be also defined by 2D drawing that is projected or wrapped around a solid shape. The *mCAM* supports projection or wrapping of 2D drawing on round pipes, rectangular pipes, and domes.

Creation of 3D model by using 2D surfaces originated from the SolidSel library and is similar to the case of 2D surfaces from DXF file. A surface shape generated by SolidSel in CNC code can be created by parametrized models defined by group of particular parameters as intersections of round pipe or other 3D shapes included in the SolidSel library. There are limited possibilities of bevelled cuts in case of DXF format, and limited feature set for CNC format.

Format name	File extension	Description
DXF	*.dxf	AutoCAD open format available in most CAD programs
CNC	*.cnc	G–codes and M–codes

Tab. 3: 2D surfaces – Input formats

Output formats

The *mCAM* offers the output in form of a CNC code that is sent or used directly on a machine.

Format name	**File extension**	**Description**
CNC	*.cnc	G–codes and M–codes

Tab. 4: Output formats

User interface

It is possible to access and edit simultaneously several working tasks due to MDI structure of the program (MDI – Multiple Document Interface). The main menu bar of the program is displayed on the top of the screen. This menu contains basic functions of mCAM. The content of the menu bar is variable and it changes depending on the active task bar (import/job…)

The layout of the program screen can be edited to make it convenient for a user by several window appearance options and by modifying the size of each area (*master panel*, properties, *working tree* or visualization screen).

Program areas

User interface is divided into four main areas in order to provide simple manipulation with all items.

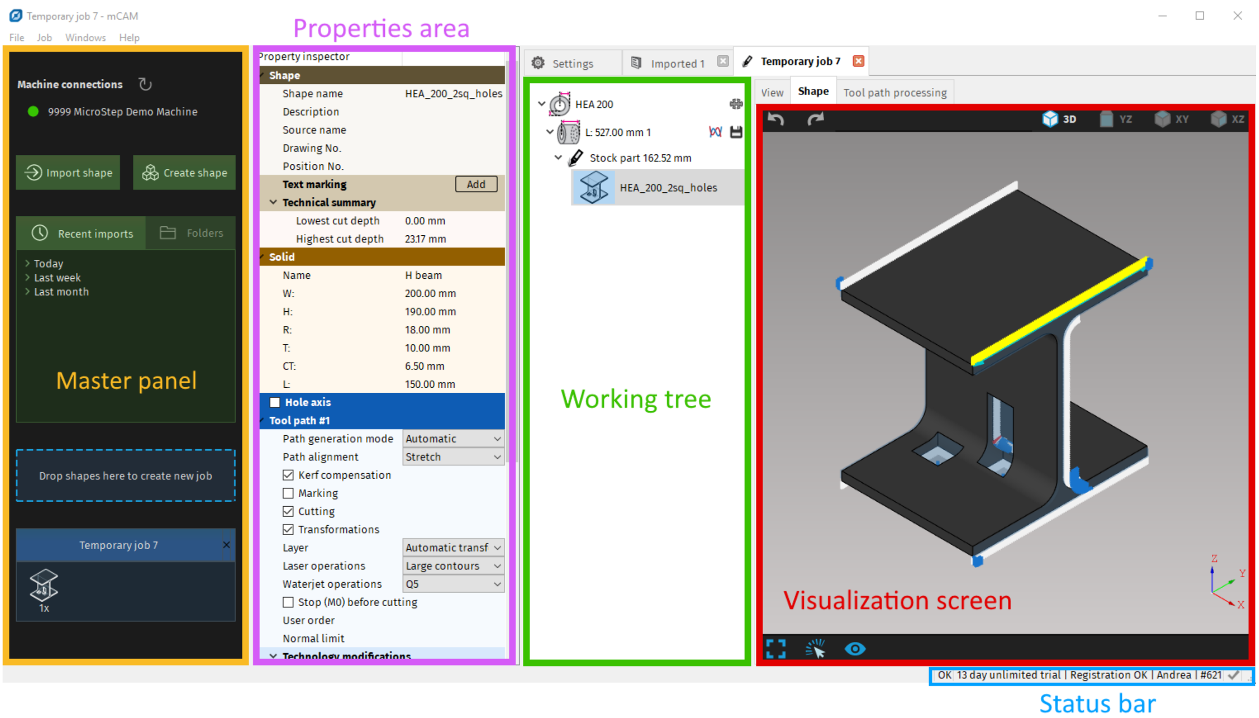


Fig. 25: Appearance of mCAM

Basic controls

This section just describes commonly used control elements.

- **Context menu** – when user performs right click with mouse on the element in the working tree in import or job tab (e.g. on template, stock or part) a context menu appears with various actions, depending on the clicked element.

The Mirror function in Context menu is capable of creating a mirror image of an imported 3D shape.

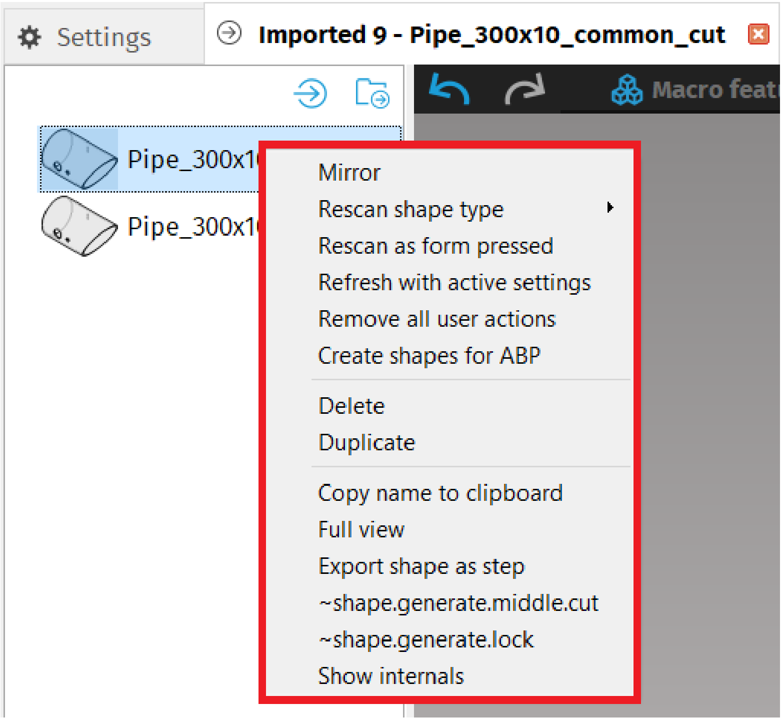


Fig. 26: Context menu with actions for part in Import tab

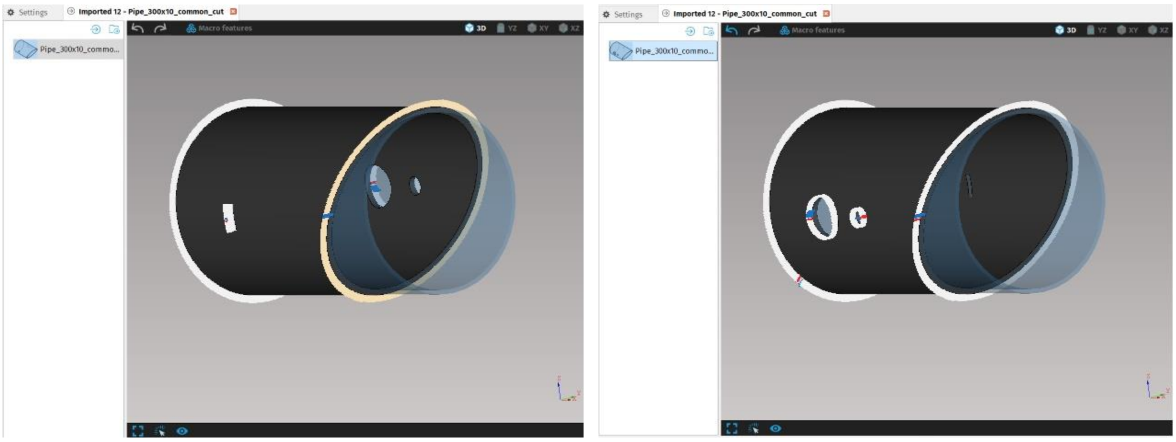


Fig. 27: The original imported 3D shape and its mirror image

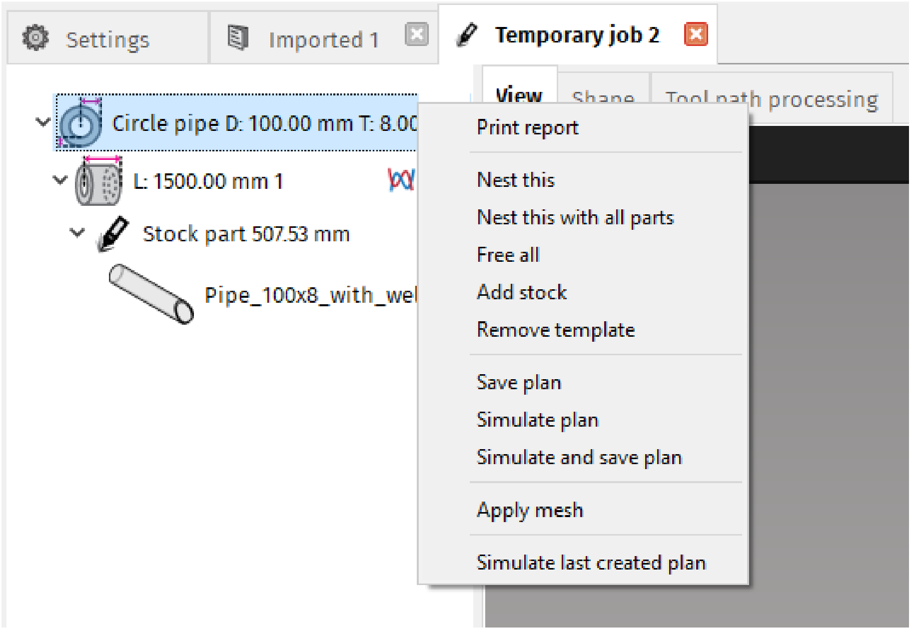


Fig. 28: Context menu with actions for template in job tab

- **Shortcuts –** Shortcuts provide an efficient way to work without continually moving the pointer to the main menus or the toolbar buttons:

→ **Alt + F** – Opens File menu

→ **Alt + W** – Opens Windows menu

→ **Alt + H** – Opens Help menu

→ **Ctrl + I** – Opens Import dialog window

→ **Ctrl + S** – Opens Raw view

→ **Alt + X** – Exits *mCAM*

→ **F1** – Views Help (user guide)

→ **F9** – Opens Console

→ **F11** – Opens debug tools window

Master panel

Master panel is on the left side of the screen. It displays an overview of all connected machines, recently imported shapes, favourite folders and opened jobs. *Drag and drop* function is described in more details in section Drag and drop on page 192.

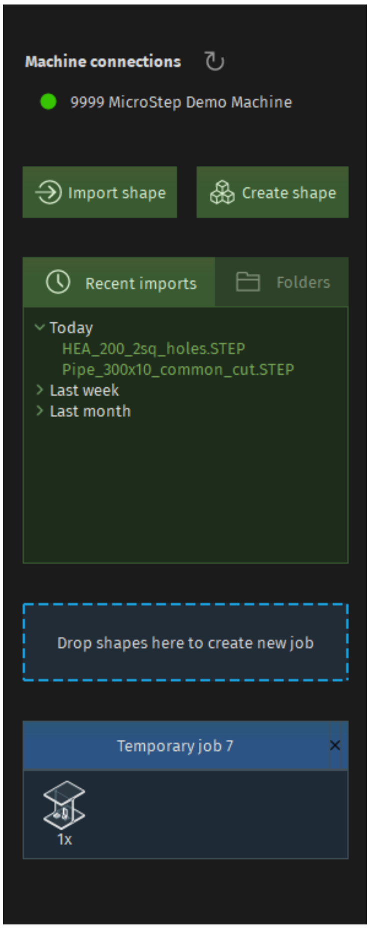


Fig. 29: Master panel

Properties

The Properties area displays details and all important properties of currently selected task tab (import, job, full view, library, etc.), template, stock, stock part, or cutting path.

> **Note:** The editable features of the properties menu are changing according to currently selected part, stock, template, cutting path or any feature that is possible to modify.

One of the main purposes of properties bar is to adjust different features of cutting paths as lead–ins/-outs, microjoints, transformations, text marking, contour marking or define used operation (for laser and water jet cutting). All cutting path editing functions are displayed in Properties area when cutting path is selected in Import task tab, Library or in Shape tab in Job task tab.

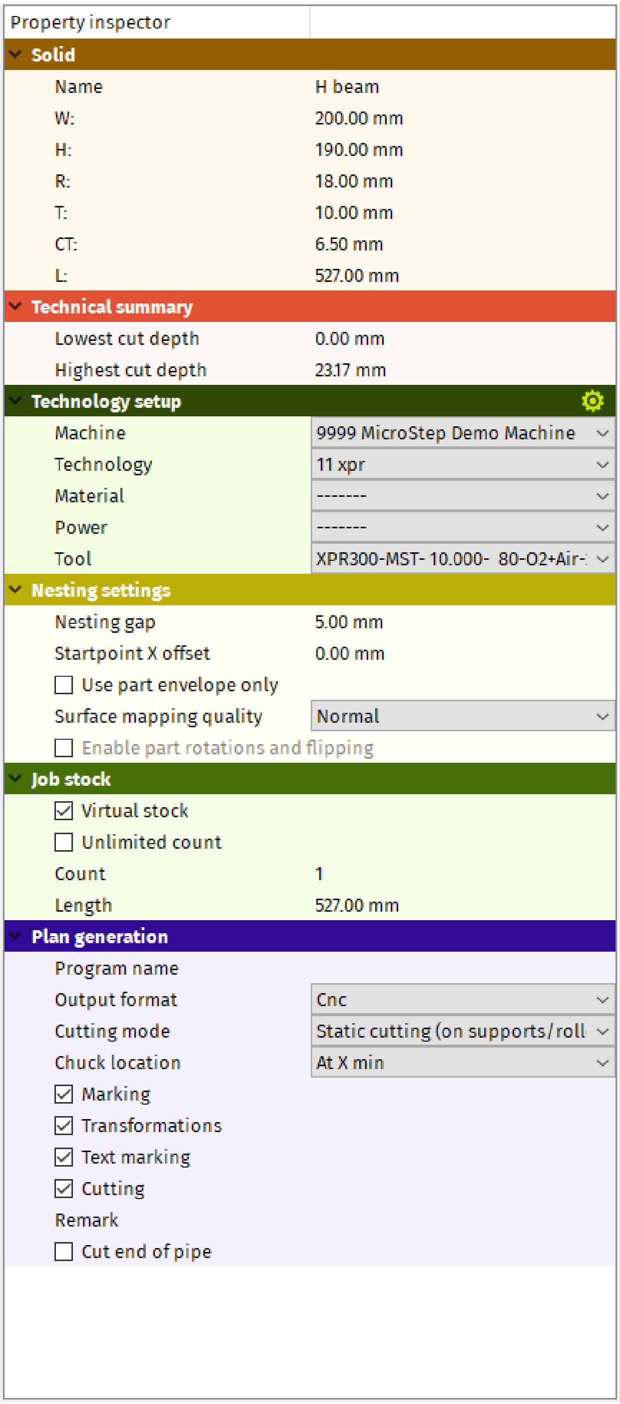


Fig. 30: Stock properties menu

Working tree

The *Working tree* is next to Properties and it displays all items that you are working with including parts, shapes, stocks, libraries and jobs. Selected item is displayed in the *Visualization screen* and its properties in the *Properties area*.

User can drag and drop parts from the *Working tree* into the job in *Master panel* (see section Job task creation) or drop parts on the stock.

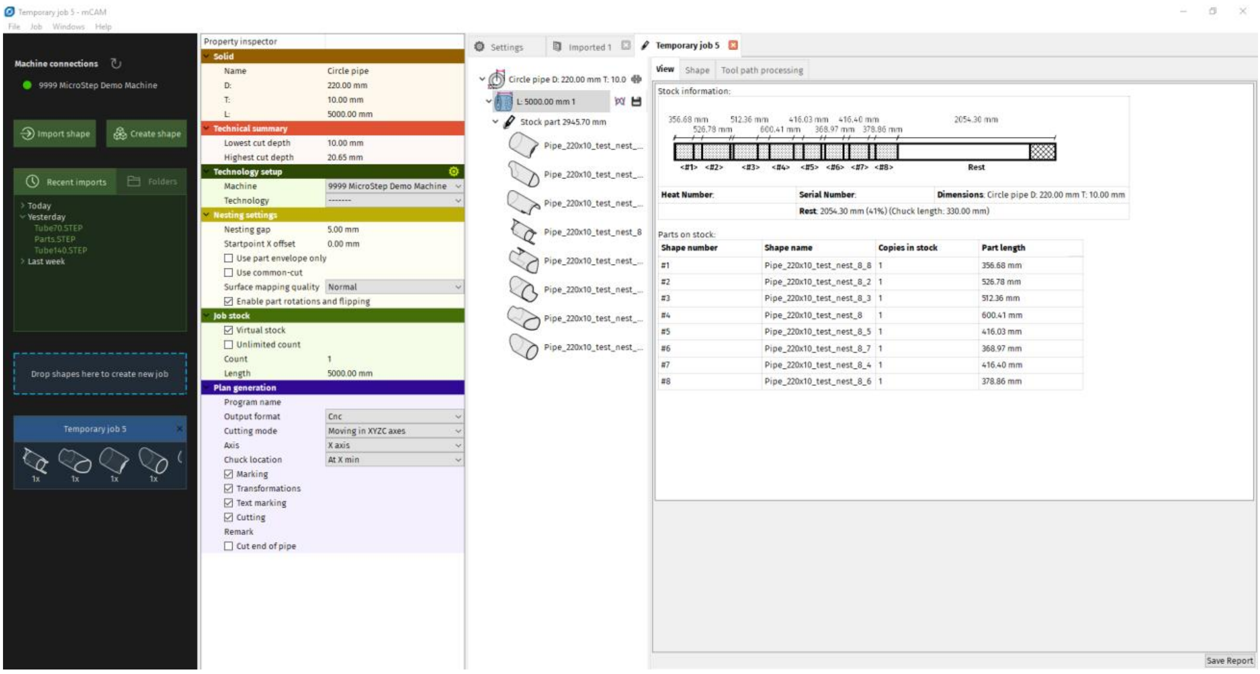


Fig. 31: Selected stock item in working tree – properties editing

Visualization screen

Visualization screen displays 3D model of the active item, part, stock, etc. It is possible to zoom–in and zoom–out the model by turning a mouse–wheel, when the mouse–wheel is pressed and held, it is possible to move the model.

Coordinate system of the active view is displayed in the bottom right corner.

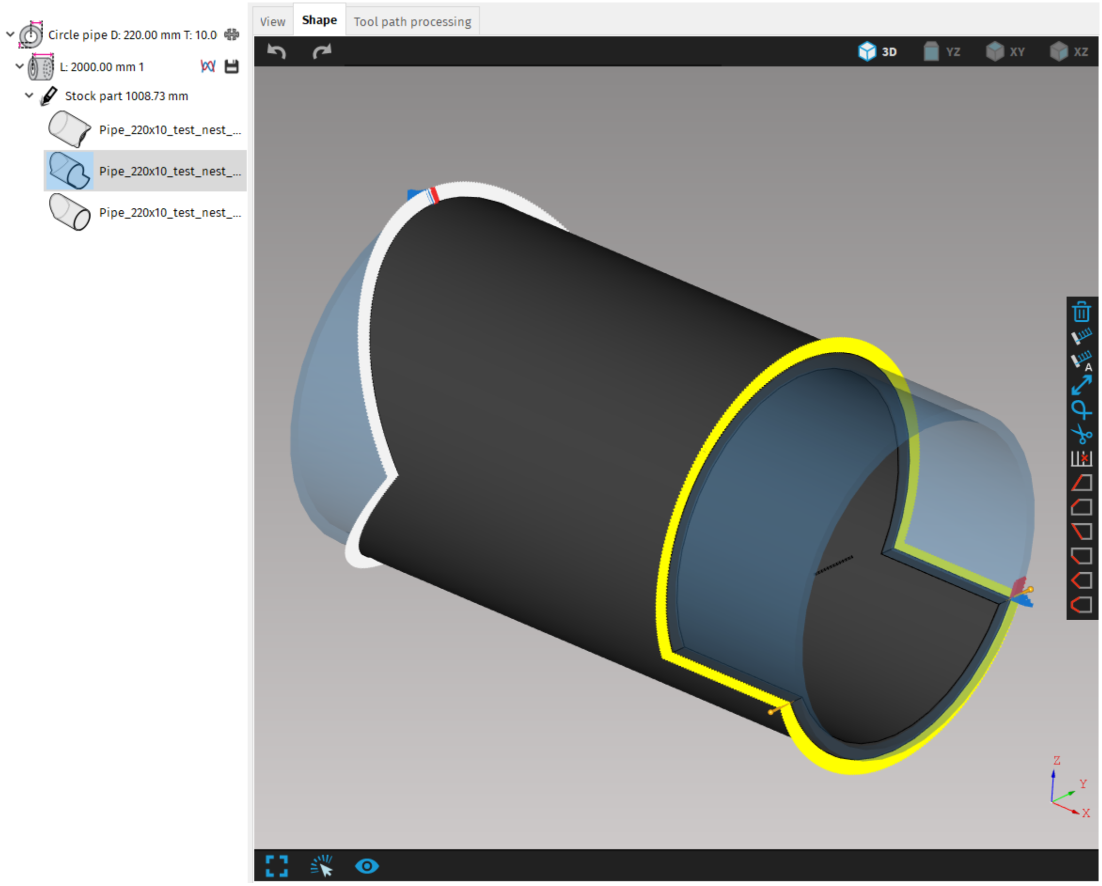


Fig. 32: Cutting path selection in visualization screen

After selecting a cutting path by clicking on it, you can choose another one by holding **Shift** and clicking on it, like this you can select any number of cutting paths you want.

Visualization screen controls

The view in visualization screen is controlled with standard mouse controls and special control elements at the top and bottom of the visualization screen.

Mouse controls

- **Left mouse button** – rotates the view of a part
- **Shift + Left mouse button** – zooms in selected area (fast zoom)

- **Ctrl + Left mouse button** – moves the item in visualization screen
- **Middle mouse button (scroll)** – Zooms the view of part or group of nested parts on stock material
- **Shift + Middle mouse button** – Smooth zooming

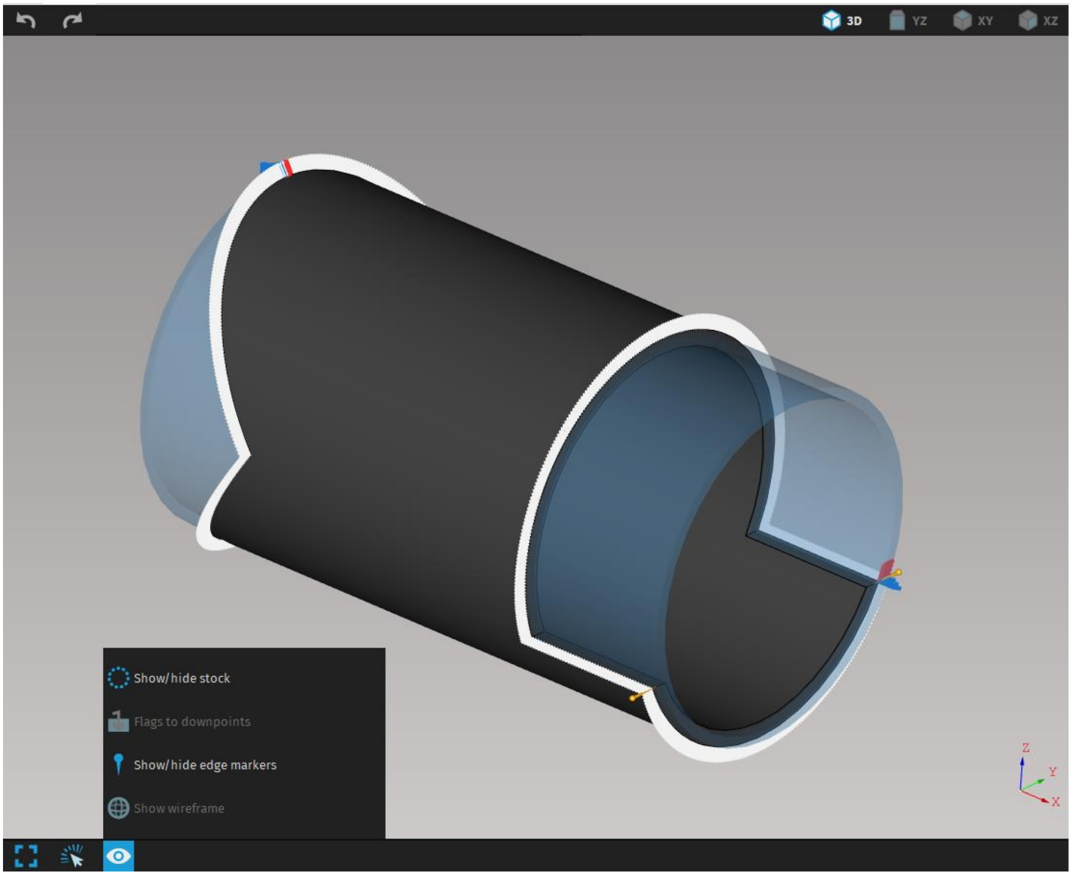


Fig. 33: Controls of visualization screen with active 3D view.

Icon	Description
	Fit to window
	Opens Select cut dialog
	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.

When 2D view is selected, it is possible to access measuring tool by clicking and holding ALT key and clicking and dragging mouse pointer across the screen (e.g. to measure a hole diameter, thickness etc.). The measurement is displayed in the bottom right corner of the screen.

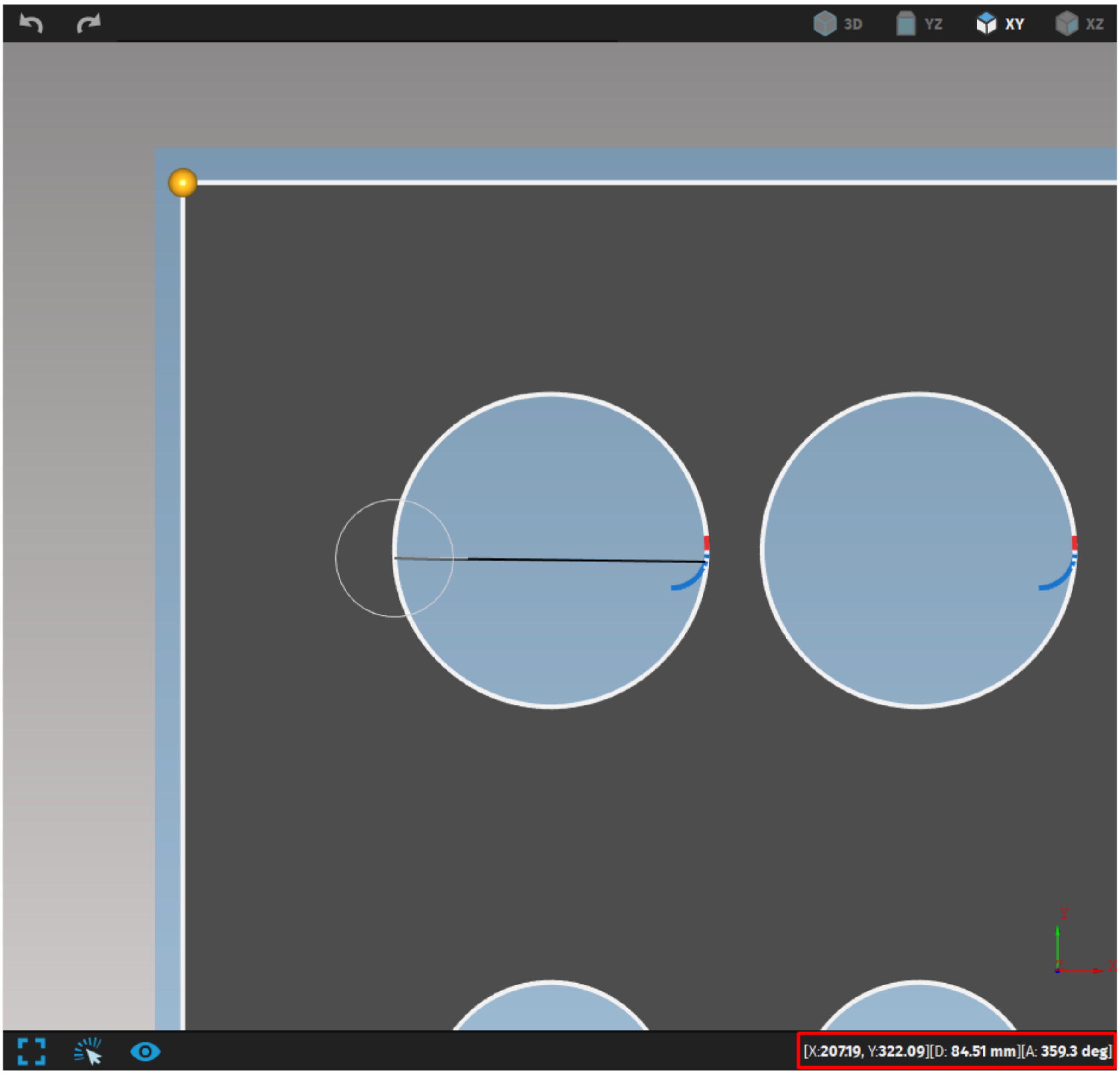


Fig. 34: Measuring the diameter of the hole

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.

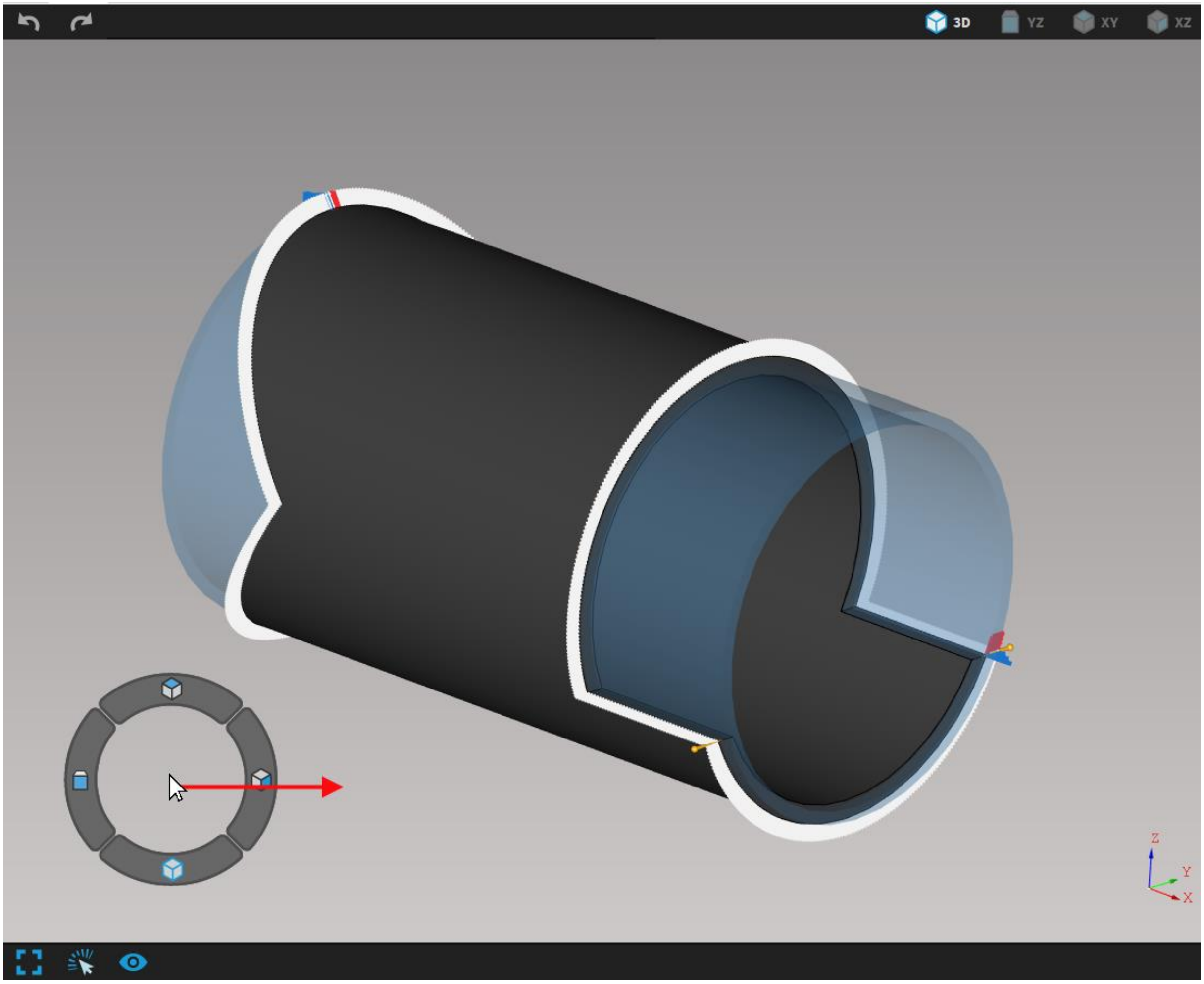


Fig. 35: Without releasing the right mouse button it has to be dragged over the view user wants to select (in this case, XZ view would be selected)

Individual settings for what is displayed in the visualization screen are located in the combo box with eye icon in the bottom panel of visualization screen. After clicking on the icon with left mouse button a list of settings appears. Active settings are displayed in bright blue color with white text, disabled settings are displayed in grey-blue color with grey text.

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 (edge markers are described in more detail in section Edge markers on page 133)
Show/hide decorations	Displays decorations – e.g. measuring points (*only in simulation*)
Show/hide tool	Displays the tool (*only in simulation*)
Flags to downpoints	Displays cutting points along the entire thickness of the material

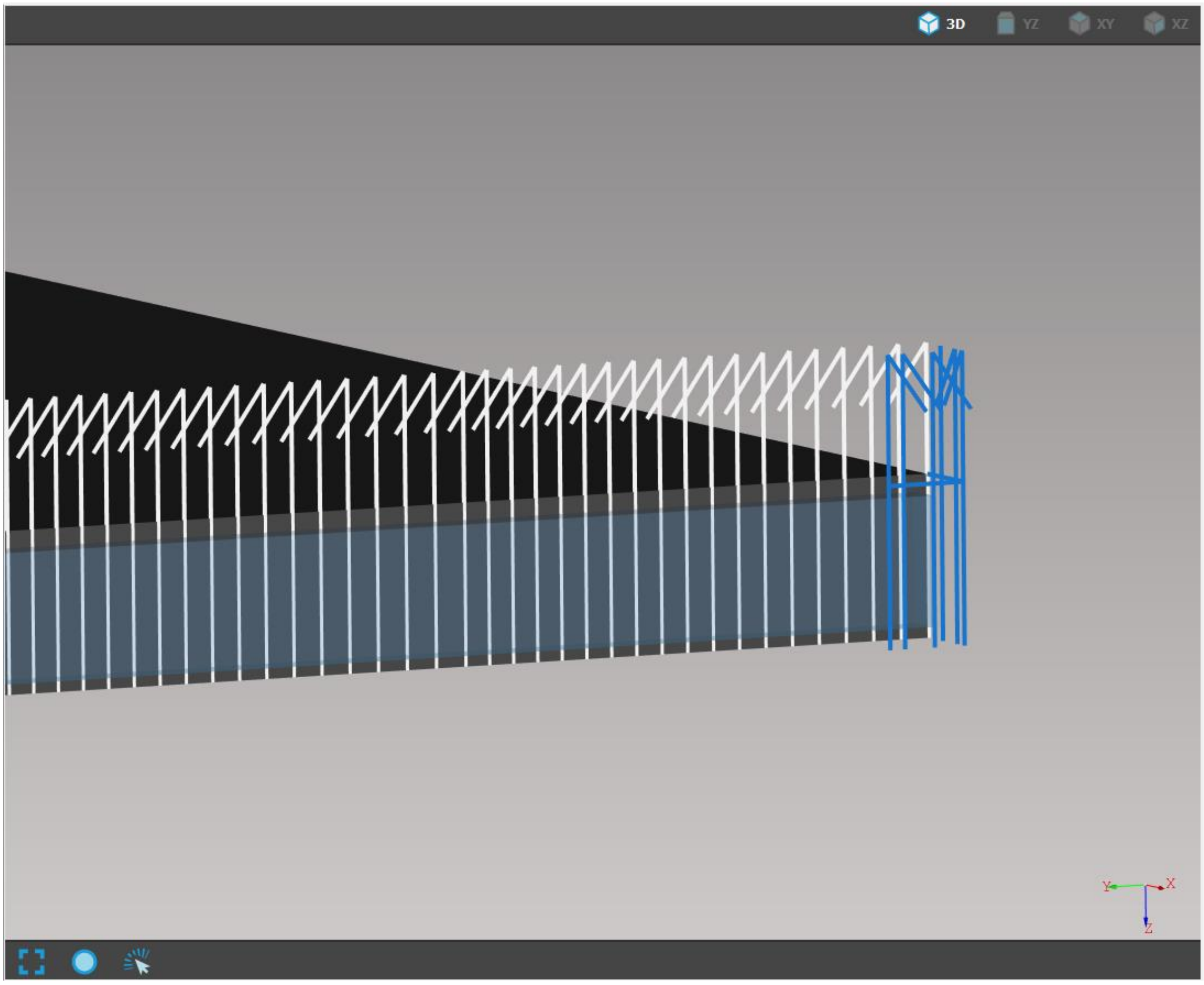


Fig. 36: Cutting points are displayed along entire thickness of the material

Selection of cutting paths using Lasso

This feature is best used when you are dealing with a lot of cutting paths on one shape. To select a certain amount of cutting paths you can press Shift, clicking the right mouse button and by dragging you can choose which cutting paths you wish to edit.

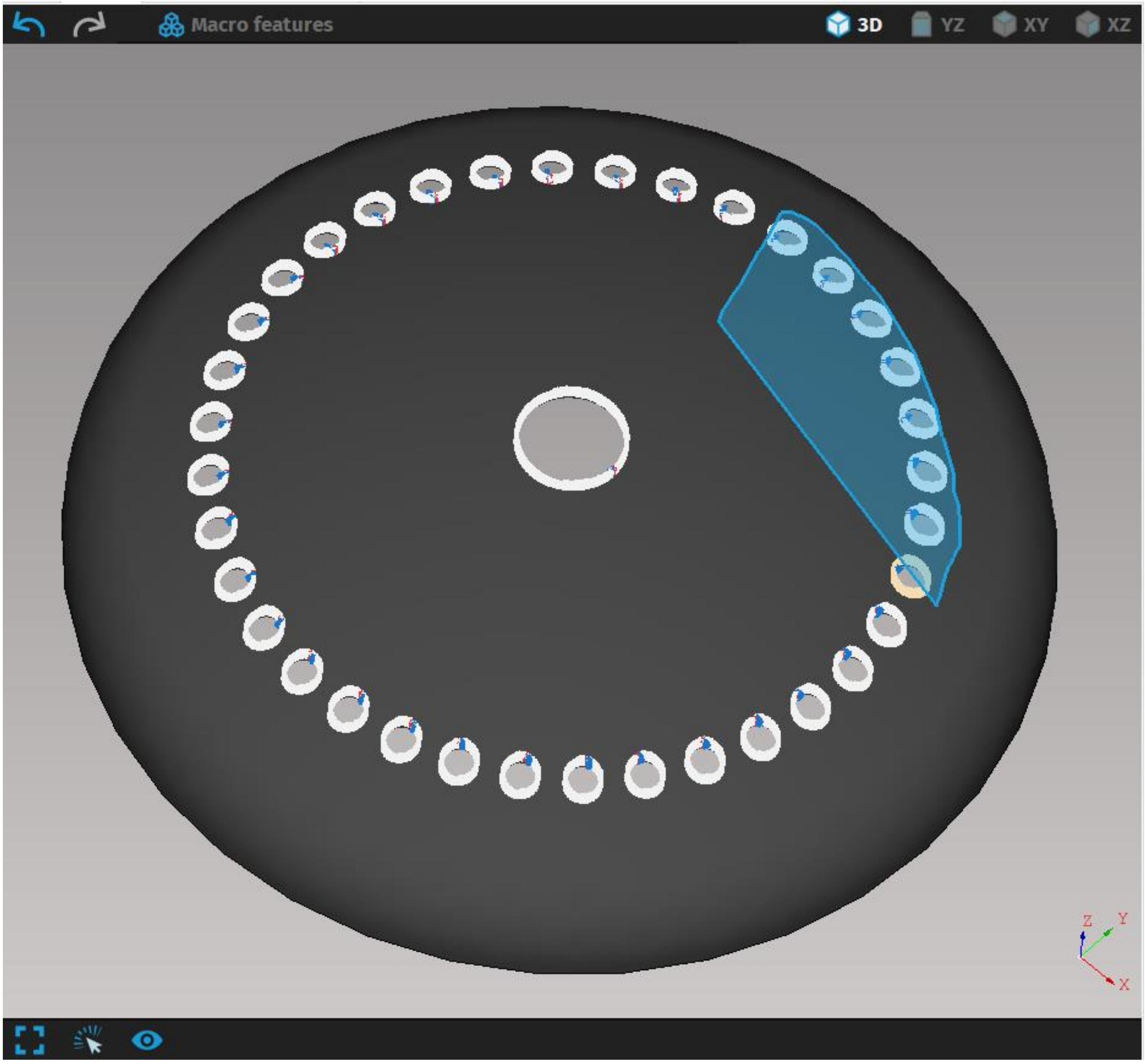


Fig. 37: **Selecting the cutting paths**

These selected paths will become highlighted in yellow color.

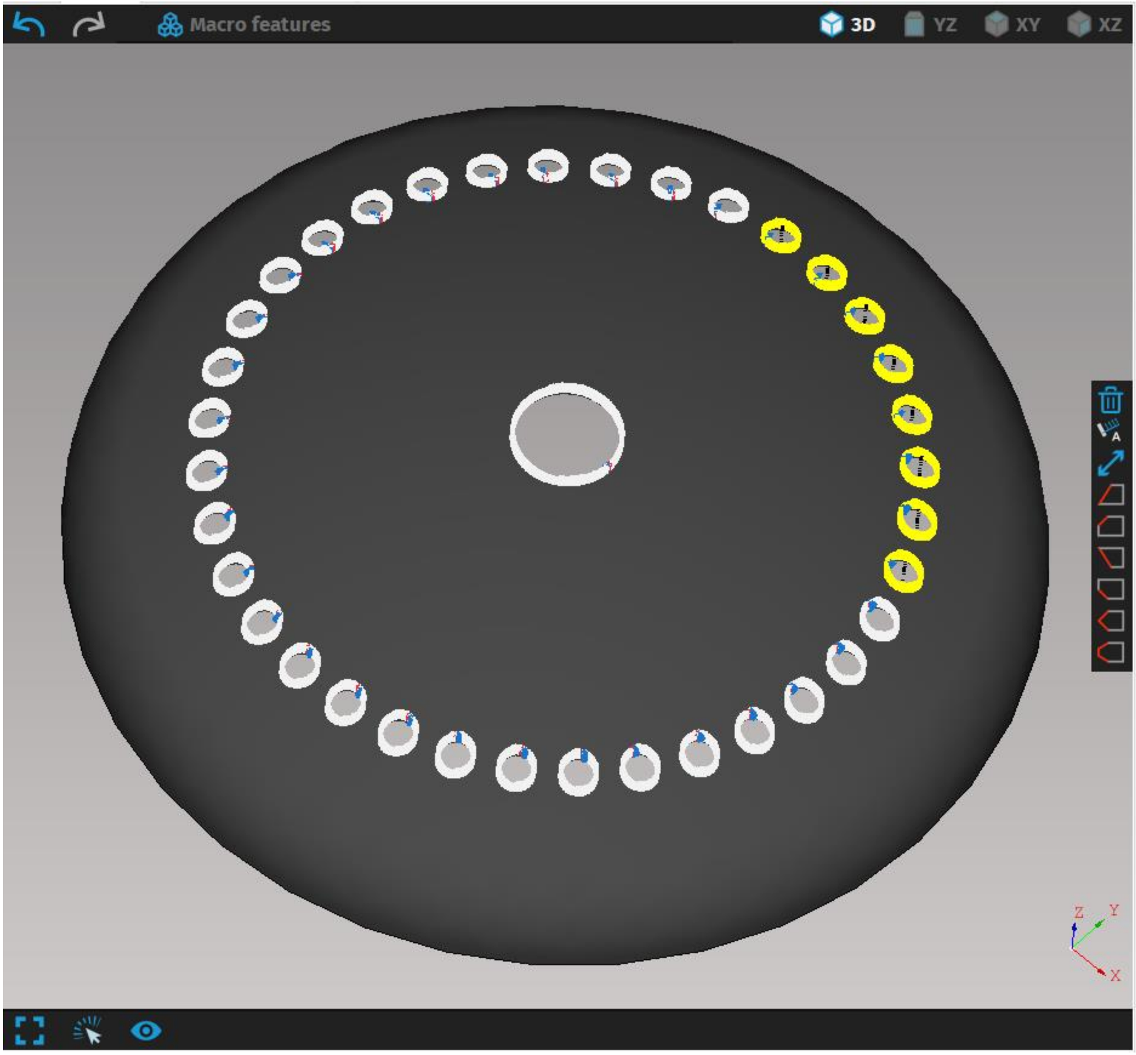


Fig. 38: **Highlighted cutting paths**

By pressing Alt alongside Shift, right mouse button and dragging you can unselect previously selected cutting paths, choosing only a few particular ones you need.

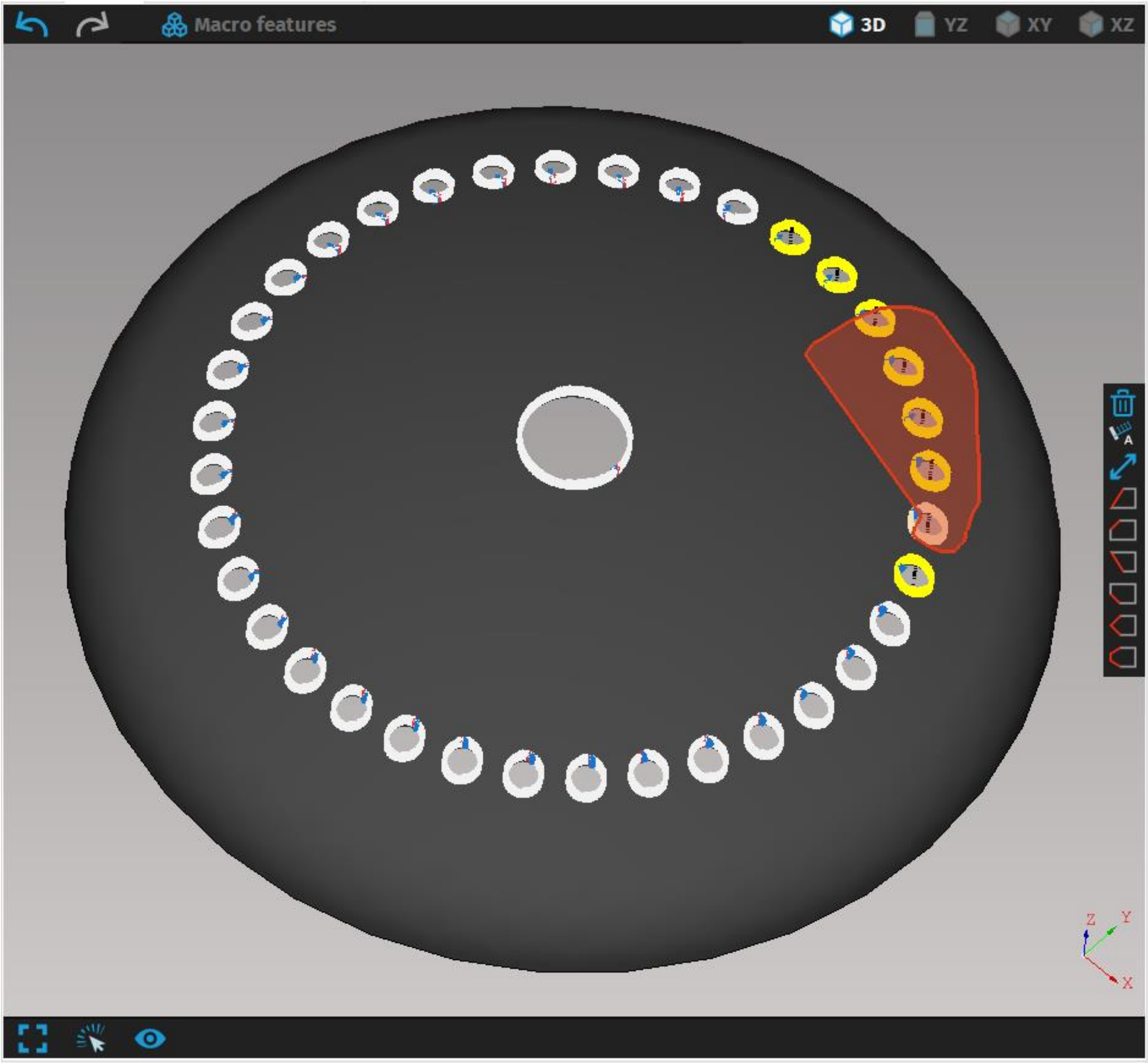


Fig. 39: **Unselecting particular cutting paths**

Rendering

There are two rendering modes that allows user to choose proper and more clear displaying mode – *Shaded view* or *Wireframe view. Shaded rendering* is considered as standard rendering option where a model is displayed without highlighting edges, corners or other sharp shapes.

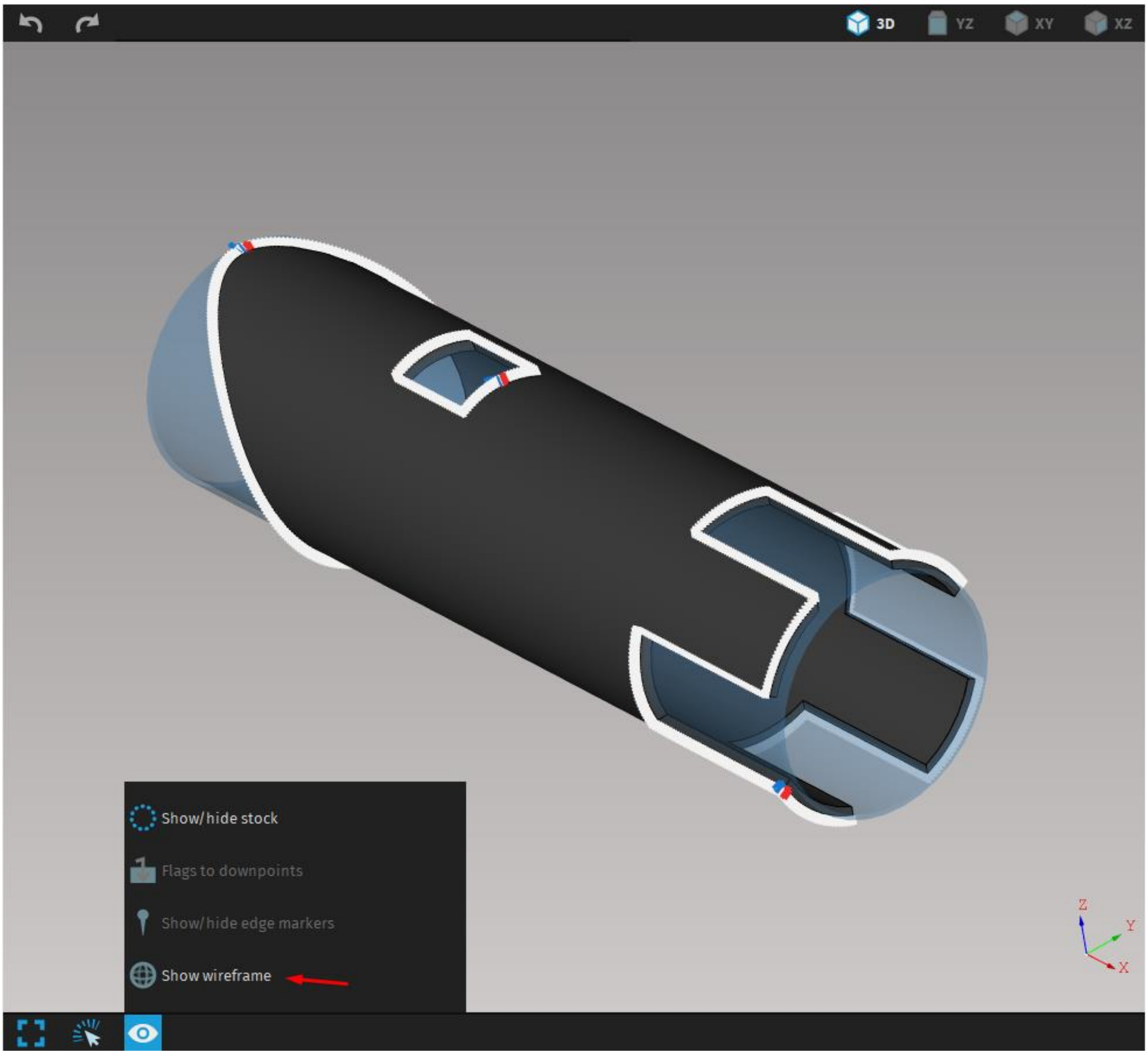


Fig. 40: The option to display the part in wireframe rendering is located in viewer menu under the eye icon

Wireframe rendering is useful if any detail associated with contours needs to be displayed and shaded mode does not make the situation sufficiently clear. The object is displayed as a wire model that consists of contours, edges, auxiliary edges, and significant contours, so-called wires. Compared to a shaded model that displays complete material, a wireframe model is transparent. The option to display the part in wireframe rendering is located under the eye icon that contains the various settings for visualization screen.

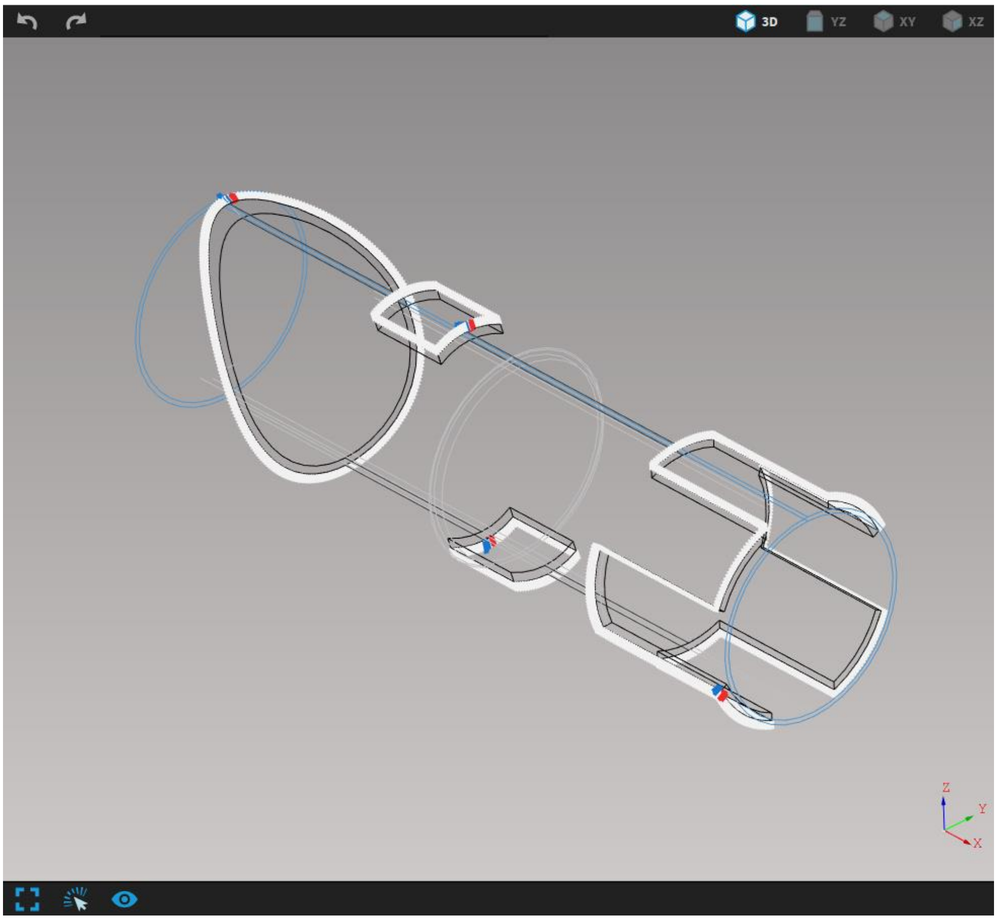


Fig. 41: Wireframe rendering

Status bar

The status bar at the very bottom of the screen displays information about validity of program registration, build version of the program, customer name and icon for complaints.

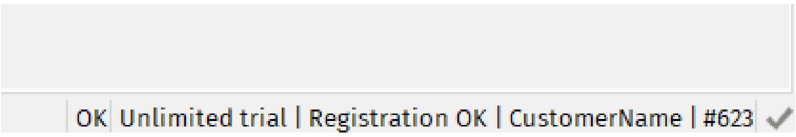


Fig. 42: Status bar

- No complaints/No new unread complaints
- Information
- Moderate warnings
- Fatal

Application workflow

This section describes typical sequence of activities to illustrate proper usage of the program. Every partial step is described in more details in particular section. The basic sequence consists of these operations:

1. importing new parts
2. cutting path editing
3. job task creation
4. job stock and part editing
5. nesting of parts
6. machine and tool selection
7. expert tables setup
8. simulation and saving of CNC program and related reports

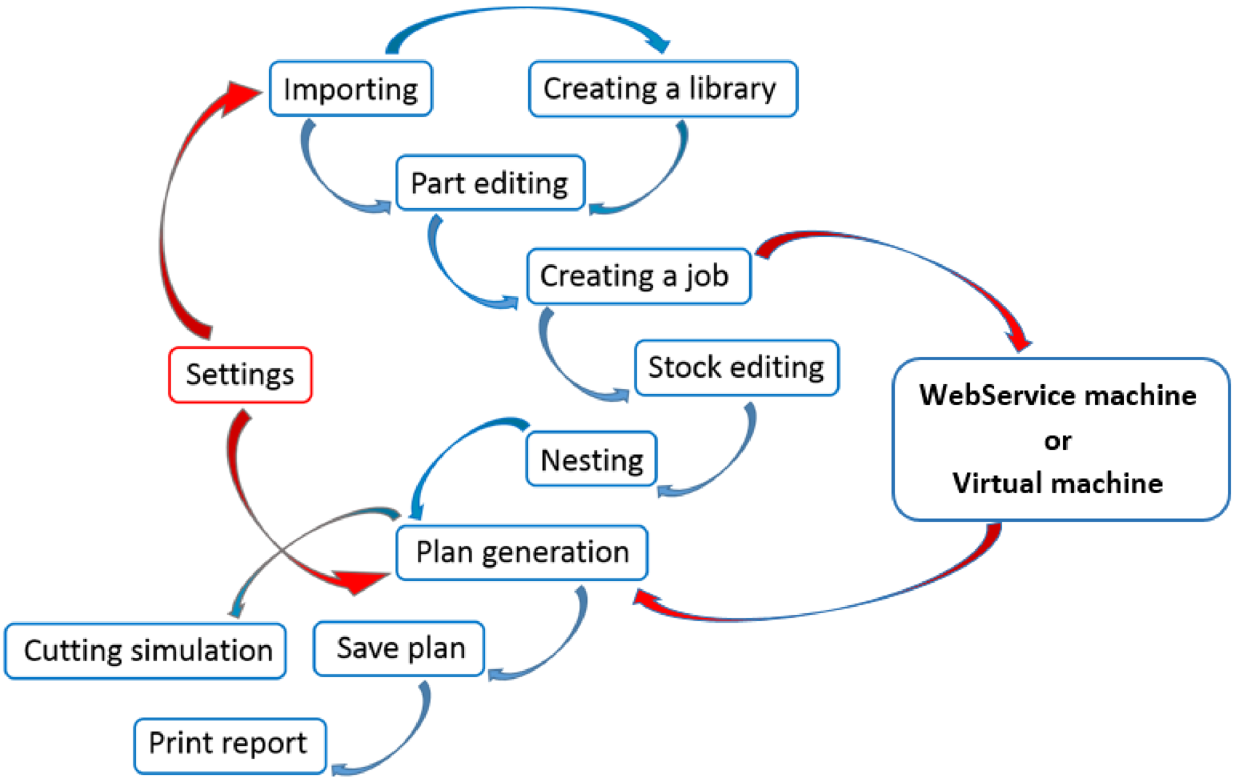


Fig. 43: Standard working procedure schema

Importing

The first step usually begins with importing data. There are several ways of loading data. *mCAM* enables to read parts is all formats specified in section Input formats on page 31).

List of supported formats:

- STEP (*.step, *.stp) – used for all supported shapes
- IGES (*.iges, *.igs) – used for all supported shapes
- DXF (*.dxf) – used only for straight pipe cutting (circular or square pipes)
- CNC (*.cnc) – used for pipe cutting (including bevel cutting) by SolidSel NC code
- DSTV (*.nc, *.nc1) – used for profile cutting (HEA, HEB, HEM, IPE, IPN, UPE, UPN, and square pipes)
- IFC (*.ifc) – used for all supported shapes (quality of processing is given by possibilities and limitations of ifcOpenShell open source library)

Import 3D shape

Direct import (*File – Import*) (key shortcut: Ctrl+I) of any file in supported format from a selected directory enables to load files in new Import task (temporary library).

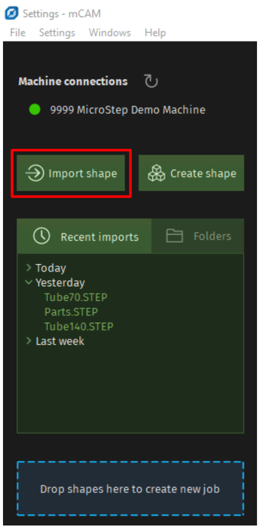


Fig. 44: Import new shapes

Import 2D surface

Default dimensions of the generated shapes match the dimensions of the 2D surface. Shape dimensions can be manually modified in the *Shape type menu* in the *properties menu*, in subsection of selected shape type.

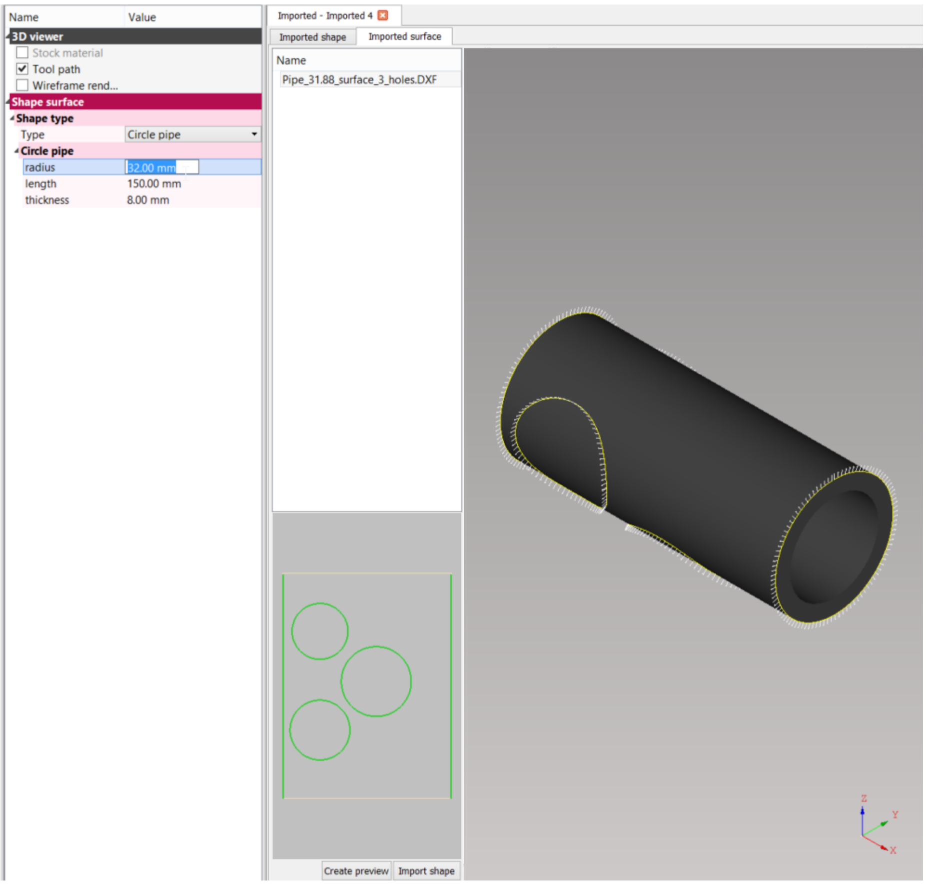


Fig. 45: Parametrized 3D shape (import 2D surface – *.dxf)

Imported 2D surface is used for projection onto or wrapping around a solid shape and so it brings few differences.

At first, the *mCAM* imports selected 2D surfaces into a database. Assignment of the surfaces to particular available shapes (round pipe, rectangular pipe, or sphere) should be executed manually. After selecting a shape type, it is possible to change pre–set dimensions of the assigned stock material manually, even if it was previously set automatically by *mCAM*. Dimensions of automatically selected stock material fit the smallest possible dimensions of the 2D surface.

It is possible to create a preview of 3D model defined by specified shape type with projected or wrapped surface by pressing button Create preview under the *working tree*. The final import of particular shape with projected or wrapped surface is executed by the button *Import shape*.

Coordinate systems during import surface

The *mCAM* uses two different coordinate systems according to the shape type that is used at import. The coordinate system for a spherical shape is located in the centre of the sphere when viewed from the top. It is important to set the coordinate system correctly and to match this attribute to the DXF file during preparation of the model created in a CAD software. In the example, the coordinate system of the imported geometry (DXF file with the ellipse) has been set to the centre of the ellipse.

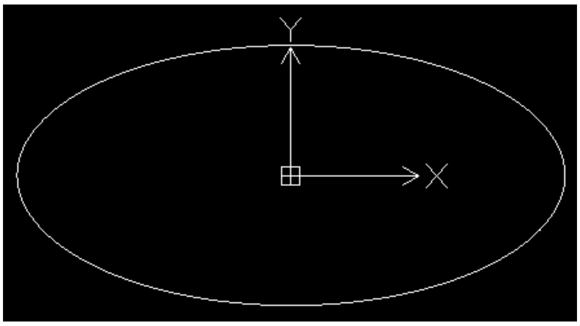


Fig. 46: Coordinate system of imported surface (projection on sphere–cap)

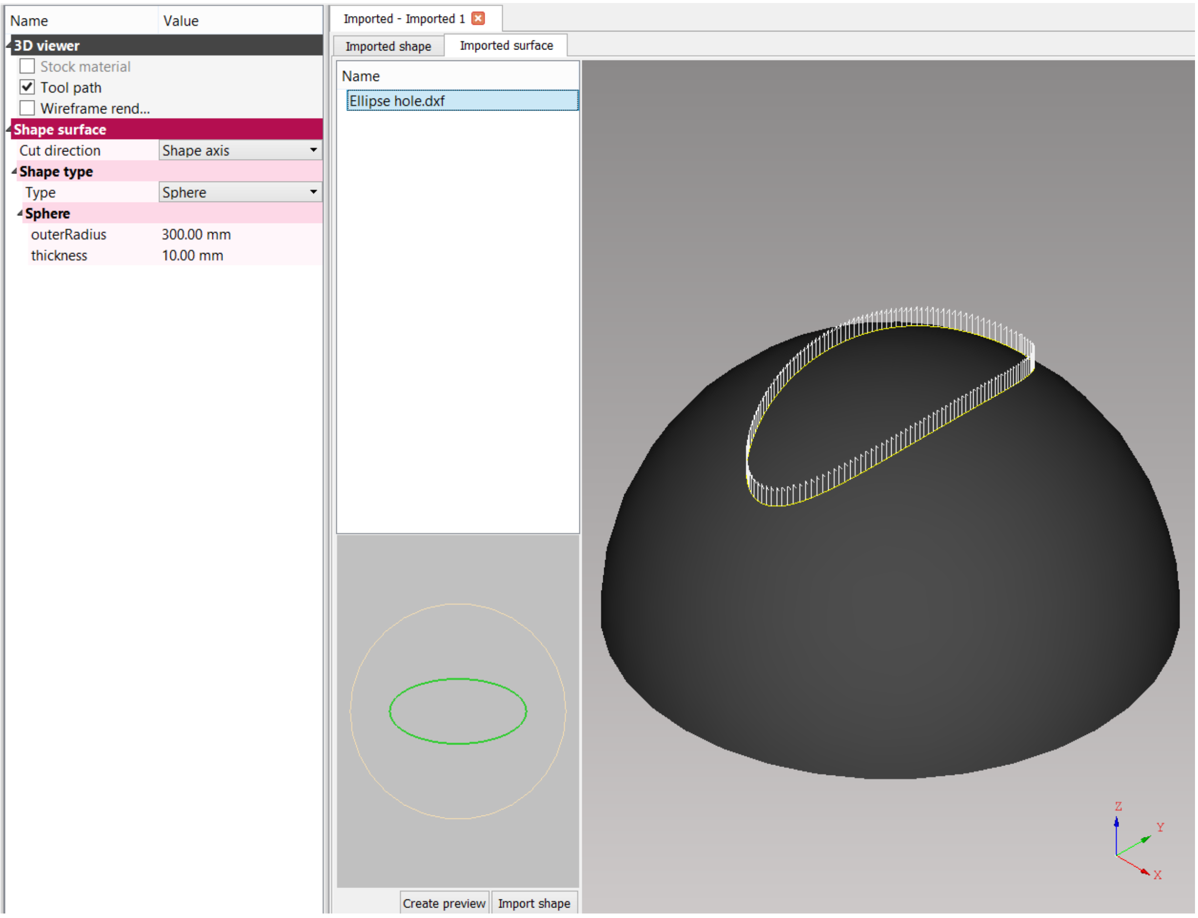


Fig. 47: Importing of projected 2D surface on sphere–cap

Another example illustrates a situation where imported surface is wrapped around a round pipe or a rectangular pipe. The *mCAM* uses the coordinate system as displayed in Fig. 48. Round or rectangular pipes are unwrapped to the planar shape (rectangular sheet) and the imported surface is projected on the shape according to coordinate system of DXF. The coordinate system of the DXF file with the ellipse is not set in the centre of the ellipse but in the area displayed in the bottom left corner.

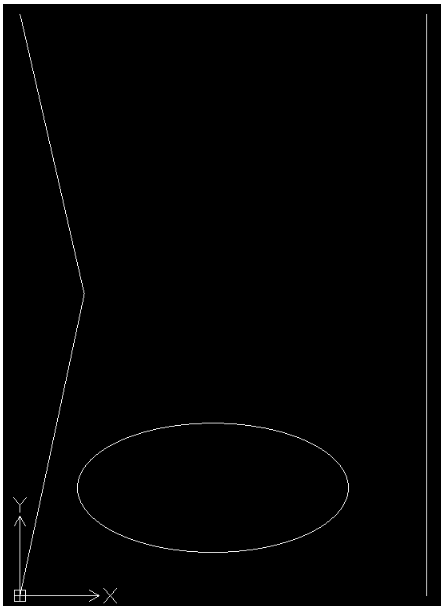


Fig. 48: Coordinate system of imported surface (wrap around a circle pipe)

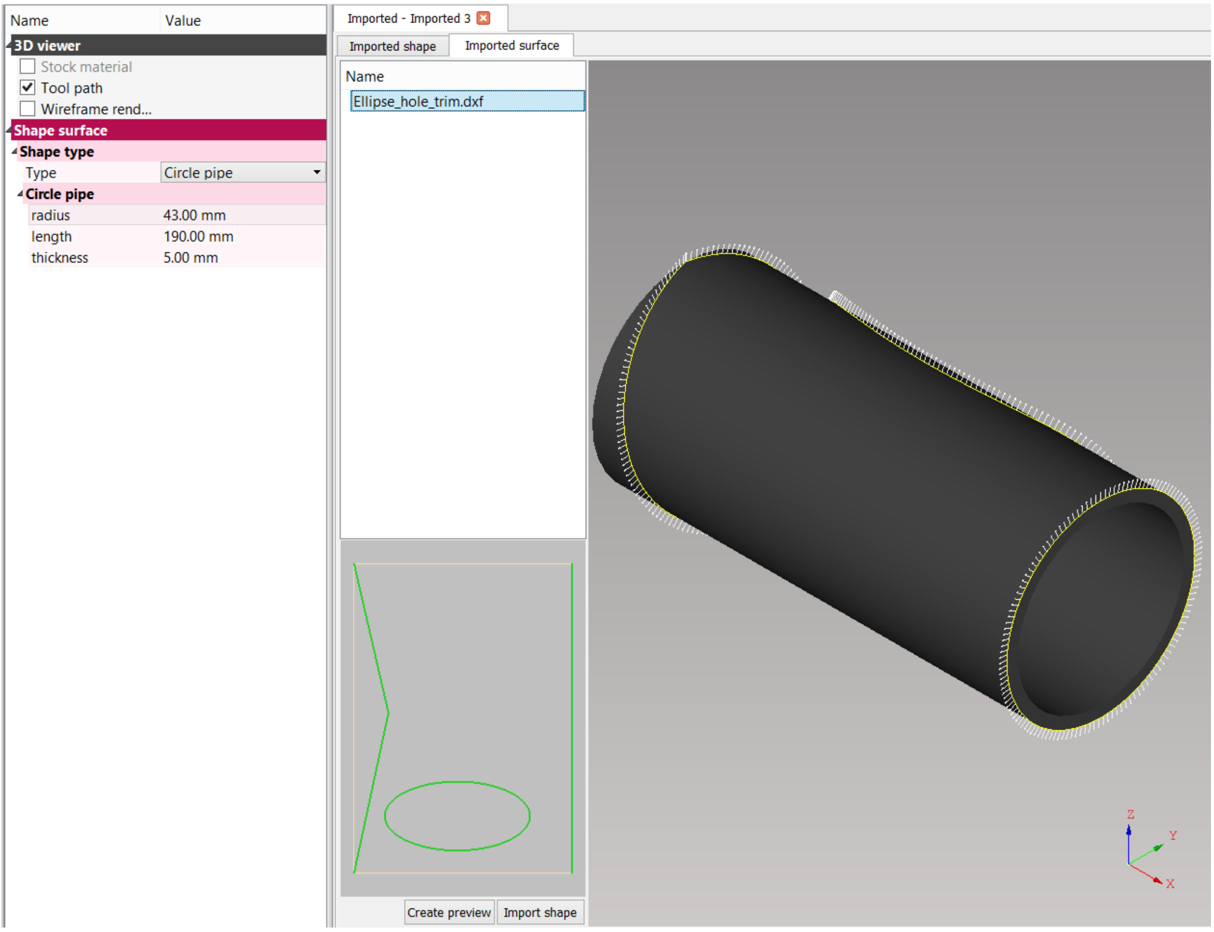


Fig. 49: Importing of wrapped 2D surface around circle pipe

Note: Coordinate systems for imported ellipses and the used shape types are identical. It means that DXF coordinate systems of both ellipses are set according to the. In the first case, it is located in the centre of the ellipse and in the second case it is placed approximately at coordinates: x= 60; y= 30.

When importing 2D surface (*.cnc or *.dxf) intended for wrapping around or projection on a part, Surface importing formats needs to be selected.

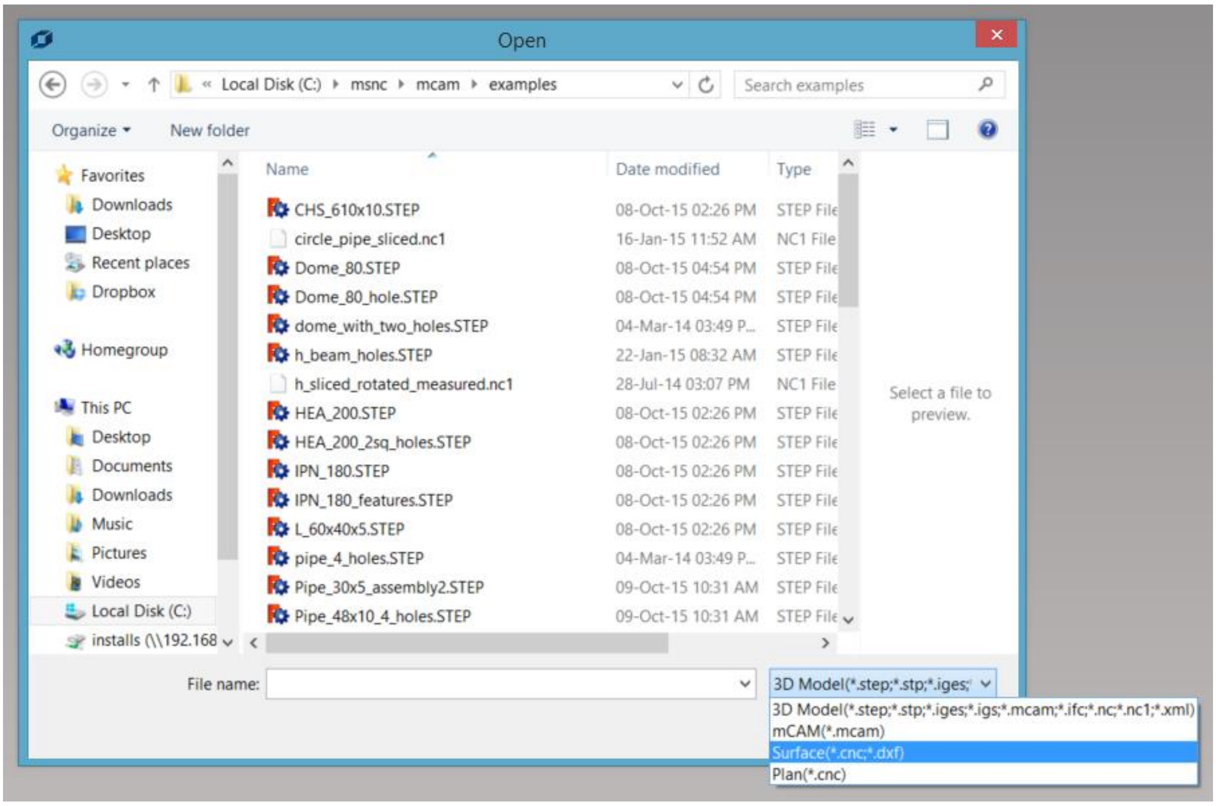


Fig. 50: Import file format selection

Cutting path editing

Cutting path editing includes these operations:

- splitting paths and loops creation
- lead-ins/outs modifying
- start point modifying (on cyclical paths)
- microjoints creation
- text marking
- transformations assignment or disabling (drilling/ punching/ quality hole cutting),
- marking of contours
- user ordering (priority cutting)
- assignment of cutting operation for each cutting path (Q1/Q2/Q3/Q4/Q5; Large/Medium/Small contours).

> Note: It is possible to edit part and cut-paths also in shape tab in *Job task* the same way as in *Import tab*.

It is possible to edit each feature by selecting respective cutting path or multiple cutting paths at once. Multiselecting more cutting paths at once is performed (as in all commonly used Windows applications) by ctrl/shift + left mouse click. Cutting paths are  possible to multiselect also by using *Select cut* wizard (opened by pressing the H-key or via button at the bottom bar in the visualization screen) where cutting paths are grouped by set filter.

Select cut dialog

In Select cut dialog it is possible to select multiple paths at once according to various parameters:

- **None –** displays list of all cuts without division according to parameters
- **Hole –** displays hole as one item even if they are composed of multiple cut paths (e.e. complex cuts Y/K/X)
- **Type (Hole/slice/gap) –** groups hole according to type of contour
- **Bounding box size –** groups hole according to the size of the area of the rectangle (box) that is circumscribed about individual cuts.
- **Path length –** groups cuts according to the path length.
- **Web/Flange –** groups cuts according to their position on the part
- **Polar angle –**
- **Cut type –** groups cuts according to cut type – straight/bevel

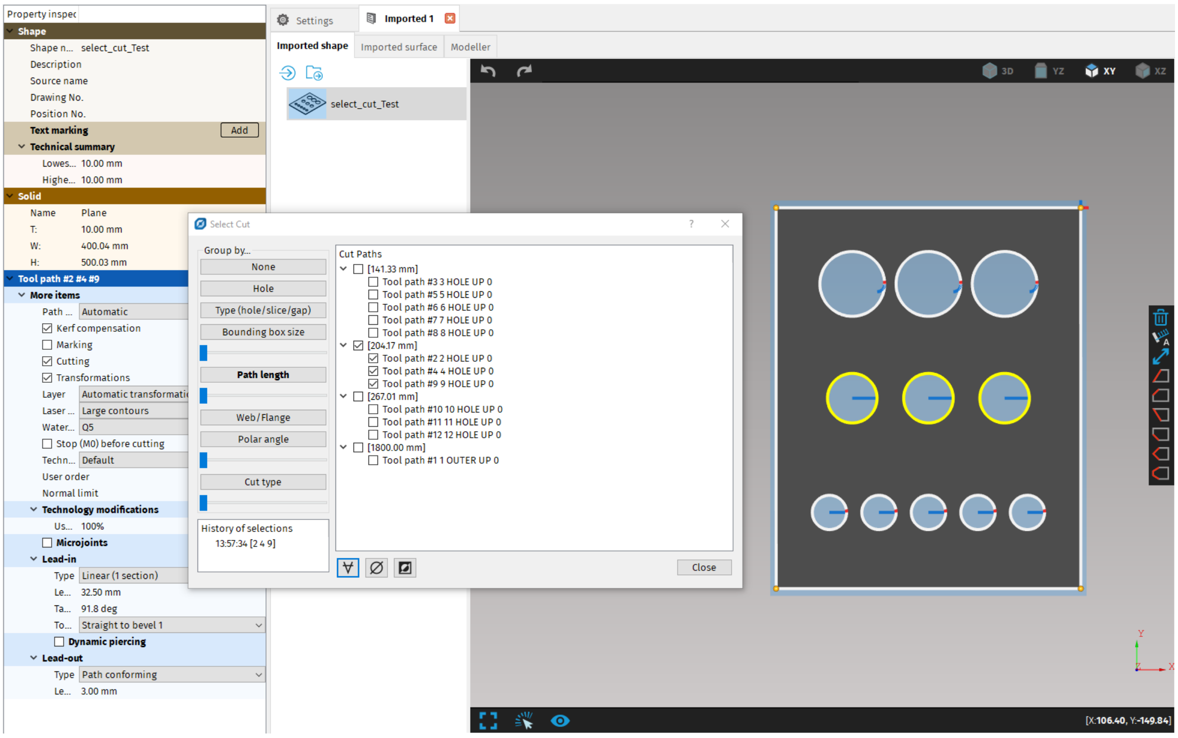


Fig. 51: Select cut wizard

Right control panel

After selecting the cutting path, the panel pops out on the right side of the screen. This cut path editing widget contains several buttons with various functions for changing some attributes of cut paths.

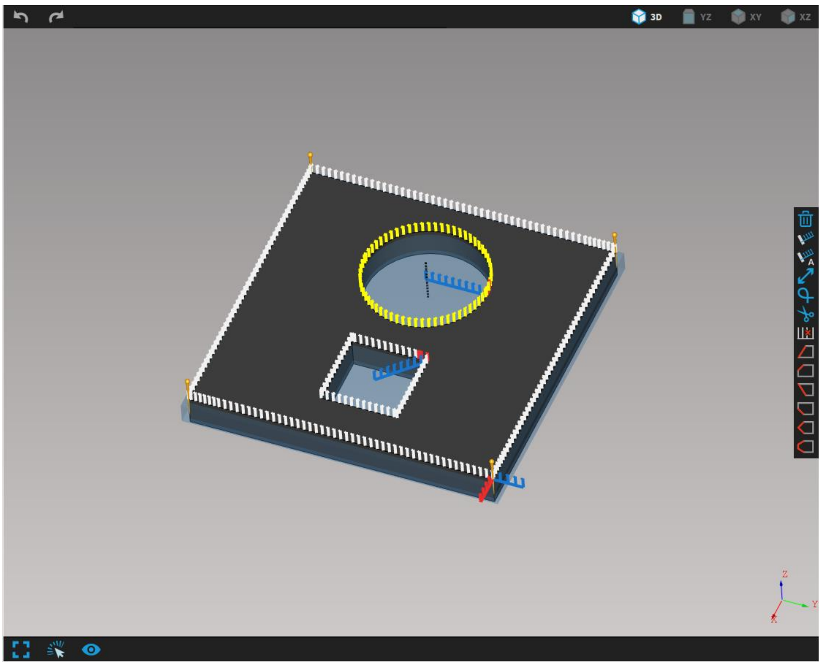
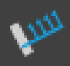
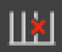


Fig. 52: Cut path editing panel on the right side of the visualization screen

Icon	Description
	Set lead-in/out to custom position (described in section Lead–in s/–outs)
	Set lead-in/out to automatic position as generated by mCAM
	Revert the cut path (described in section Reversing a cut path)
	Add loop on selected cut point (described in section Loops)
	Split path on selected point (described in section Splitting paths and creating loops)
	Join non-circular path points (described in section Splitting paths and creating loops)
	Erase selected cut point
	Delete cut path

There are also welding preparations, that are described in section *Welding preparations.*

Properties and features that modify selected cutting path/s are displayed in *Properties menu*. All selected cutting paths are highlighted by yellow colour.

Editing of all cutting path features is described in more details in section *Cutting path modifications.*

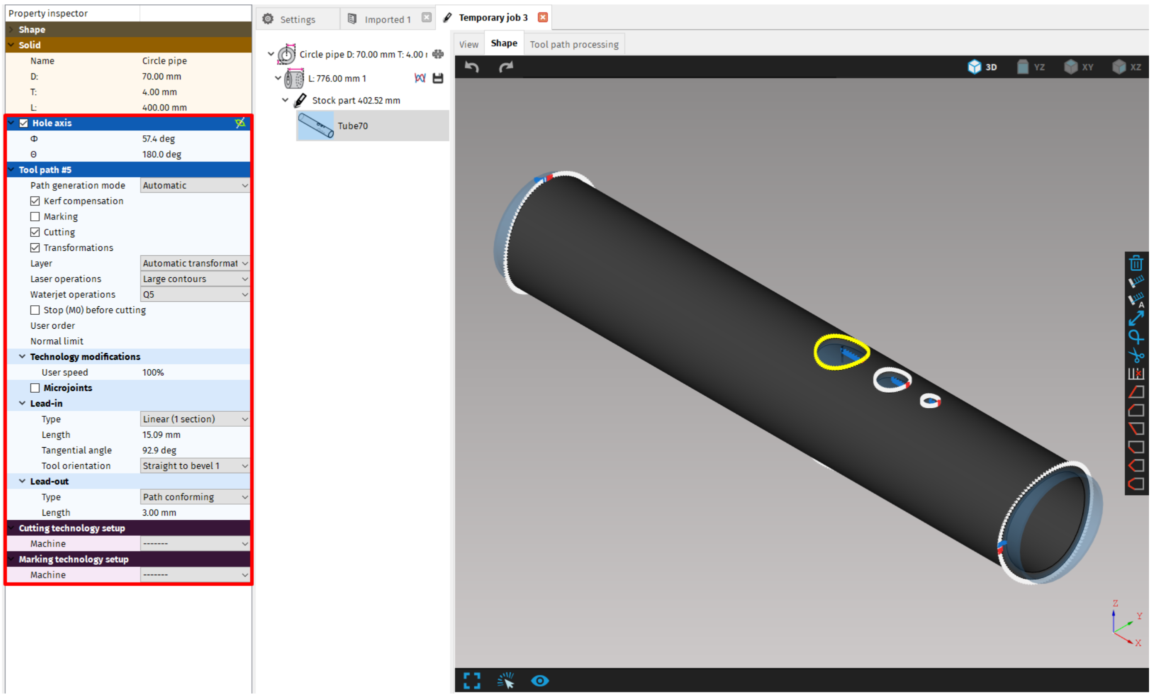


Fig. 53: Cutting path editing

Job task creation

The next step is to create a *Job task*. Job can be created in two ways:

1. **Using drag&drop function** – this is a preferred method. Required shapes are dragged from Working tree in Import to designated area in Master panel. This method creates Temporary job task. *Temporary job task* is the same type of database as standard *Job task*, but the *.job file with all settings, parts, nestings, and generated CNC programs is possible to save to hard drive only when user closes particular *Temporary Job task*.
2. **In function Menu – File – Created/Load job** - This type of *Job task* is saved as a database, so all applied settings and modified features including cutting paths, lead-ins/outs and all other data are saved automatically. When creating a job task, its name has to be defined.

After creation of job, mCAM automatically sets a template (e.g. Circle pipe D219.100 mm T 16.000 mm) and stock with default length (e.g. L:3003.000 mm). More details about parts, stocks and templates are in section *Job task*).

More parts can be easily added to particular *Job* by drag and drop selected parts from *Working tree of Import tab* to Master panel to respective job.

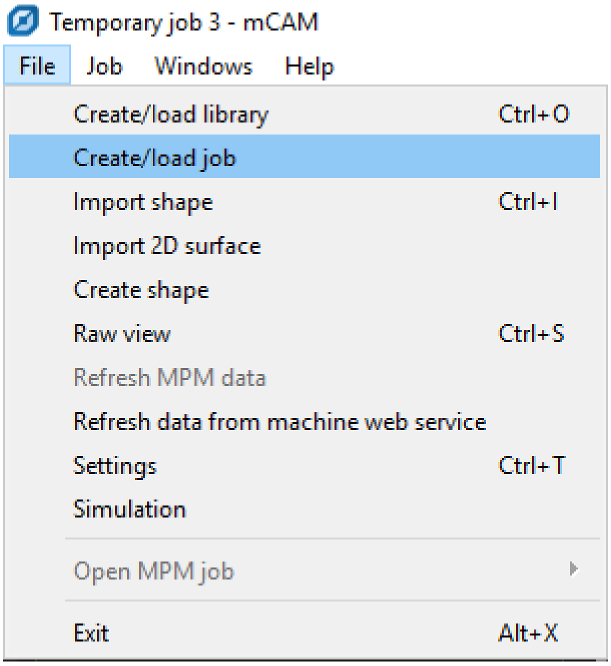


Fig. 54: Creating a new job or loading existing job

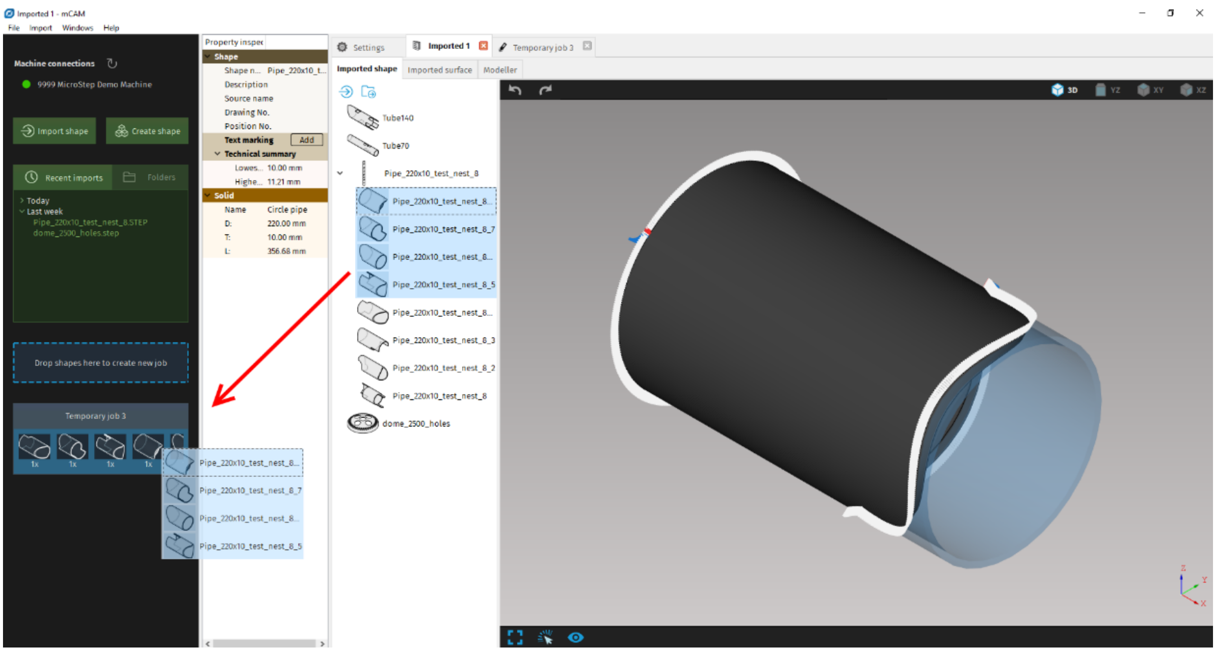


Fig. 55: Adding parts to Job task

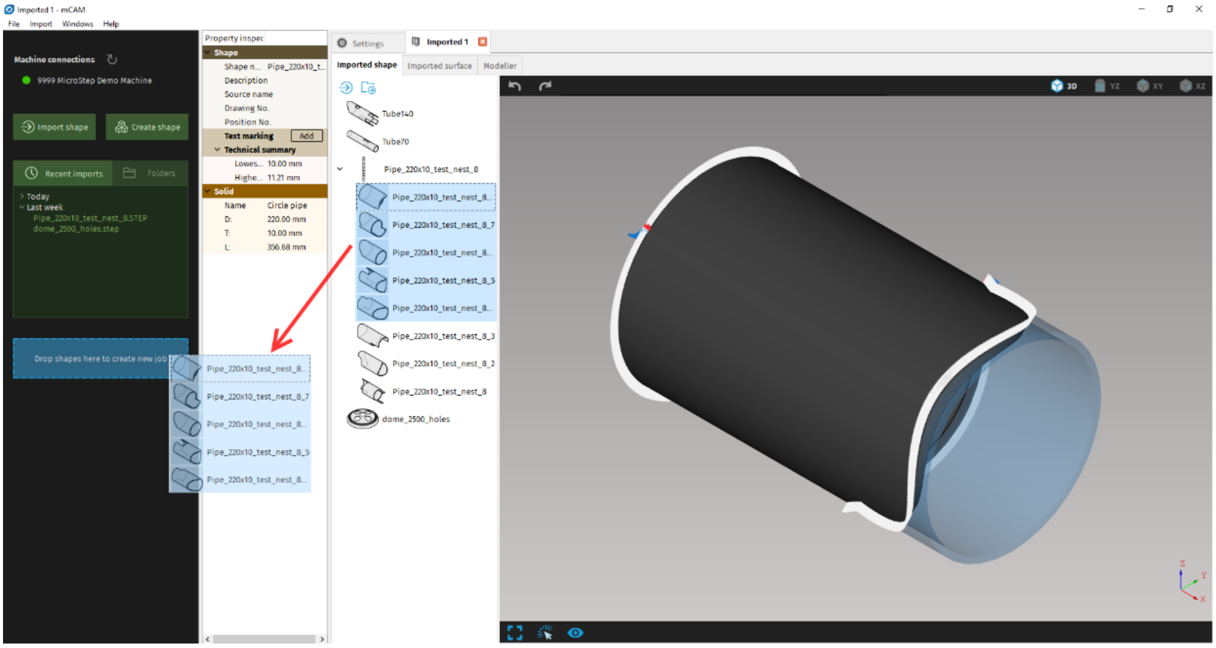


Fig. 56: Temporary job task creation

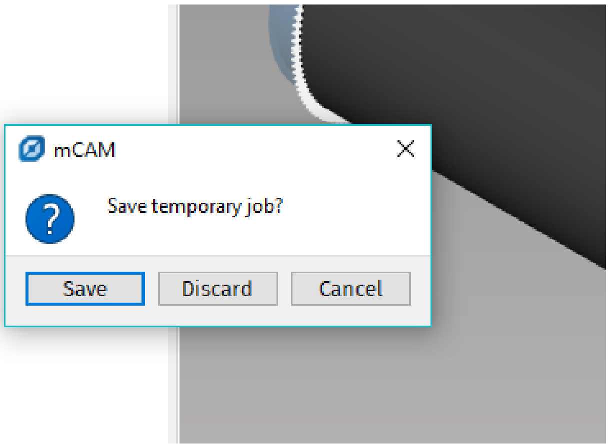


Fig. 57: Saving temporary job task

Job stock and part editing

After a job is created or loaded, it is possible to:

- add new stocks (with different length than existing stocks within one template – material and dimensions)
- change a count of stocks
- modify length of stock material
- change quantity of shapes

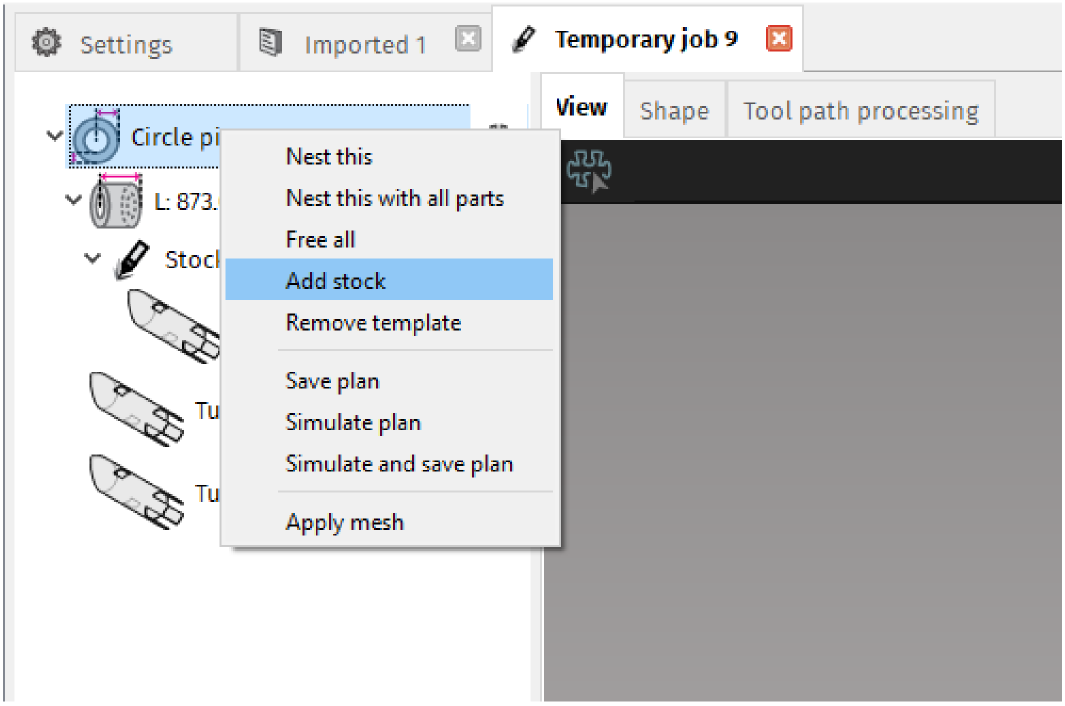


Fig. 58: Adding a new stock

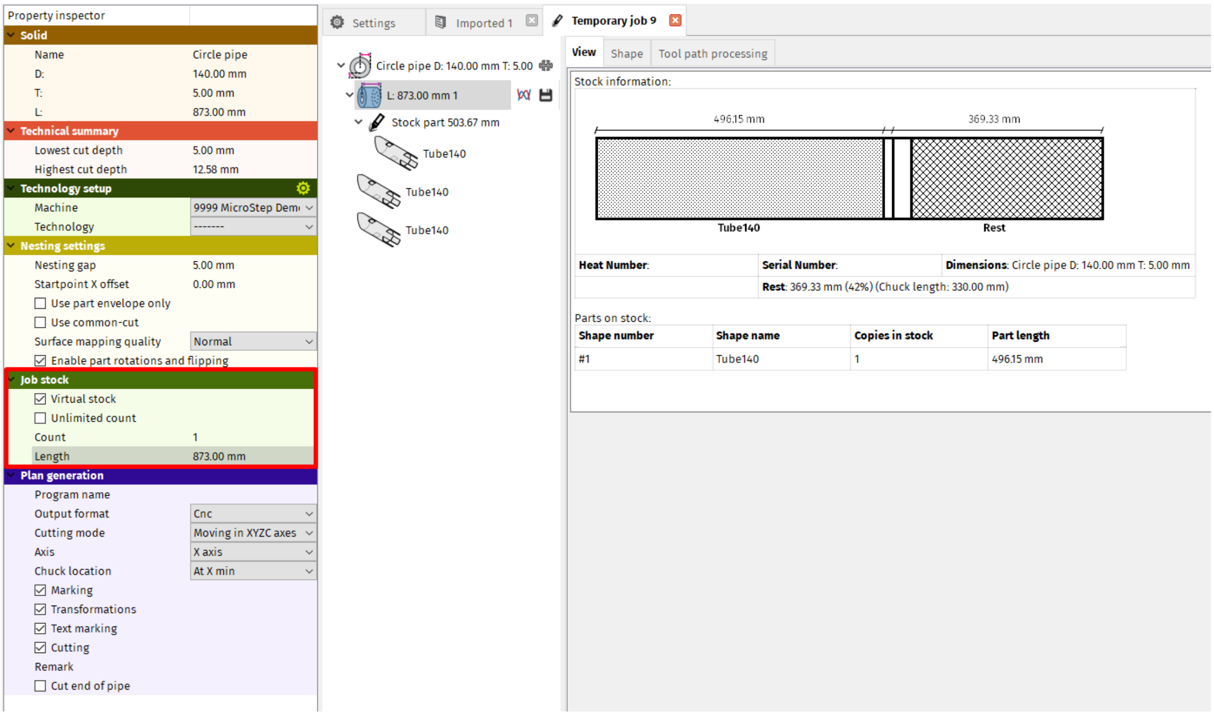


Fig. 59: Stock editing (count and length)

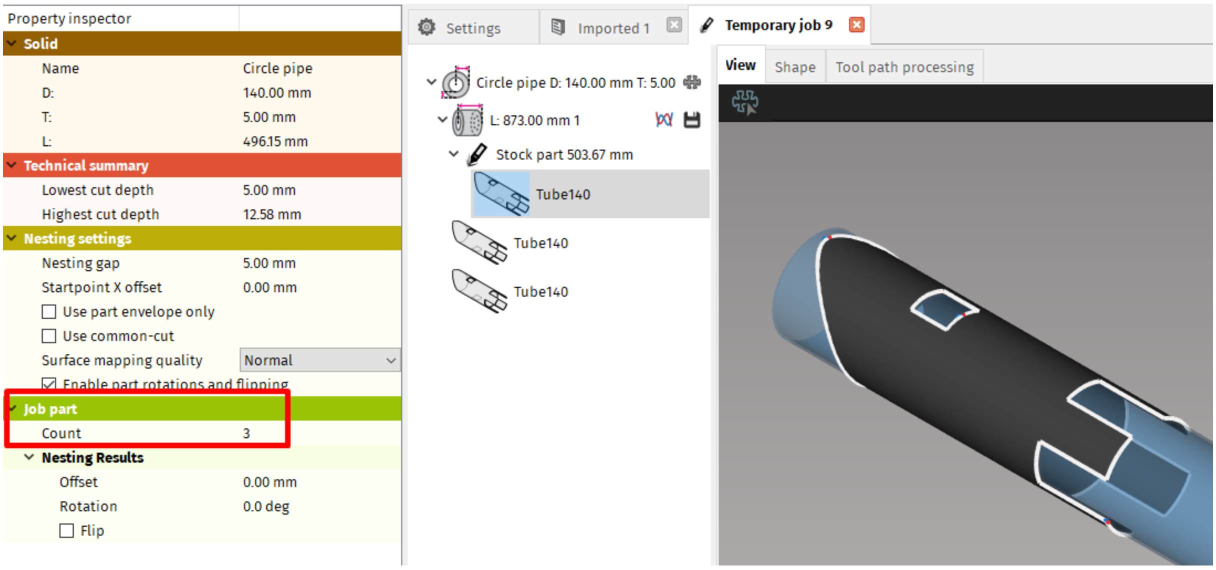


Fig. 60: Defining count of selected part

> Note: Other Job stock operations are described in more detail in section *Stock (stock material)*.

Nesting of parts

Nesting is a function used to create optimal placement of parts on the stock material. The function employs a sophisticated algorithm to place parts so that maximum utilisation of the stock material is achieved. The algorithm rotates, mirrors, shifts, and orders the parts to use the minimum amount of the stock material for particular job. Minimum gap between neighbouring parts as well as other nesting settings are defined *Profile nesting* settings in particular *Job task*. Other settings that define type of the machine, supports and general nesting parameters as *Chuck length* or *Chuck location* are set in *Nesting* (*File – Settings – Nesting/supports*) in *Application* configuration level with no machine selected.

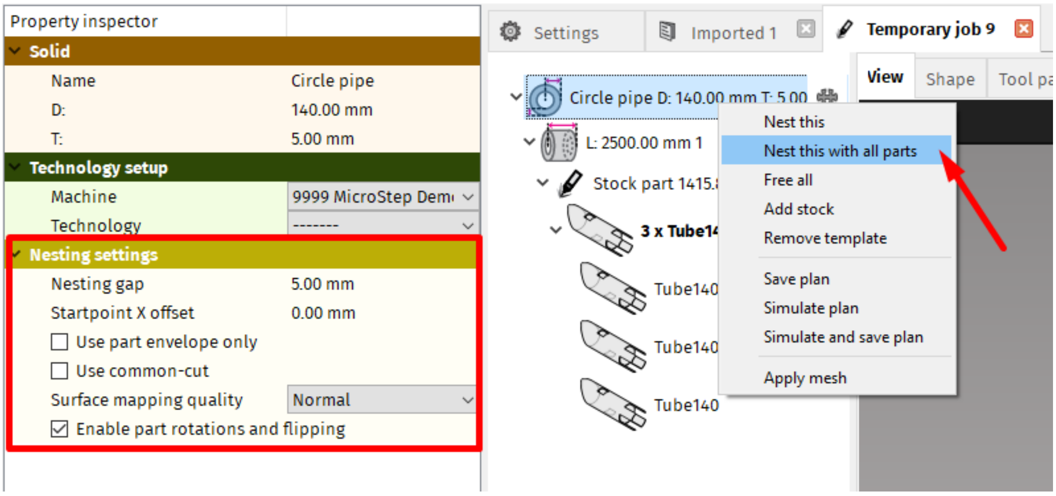


Fig. 61: Nesting settings and nesting execution

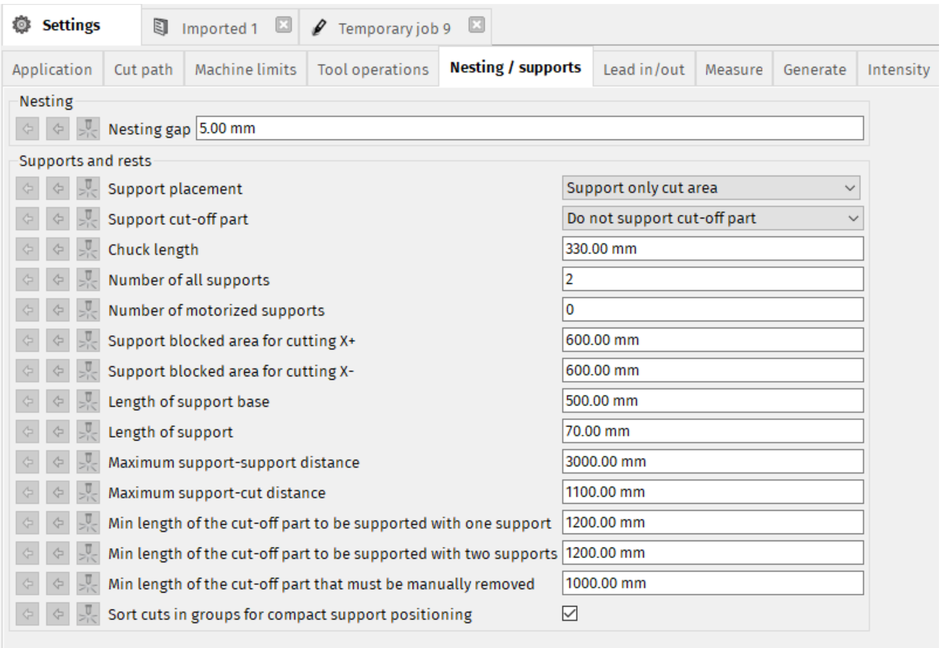


Fig. 62: General nesting and supports parameters

Expert tables setup

Expert table of each tool operation:

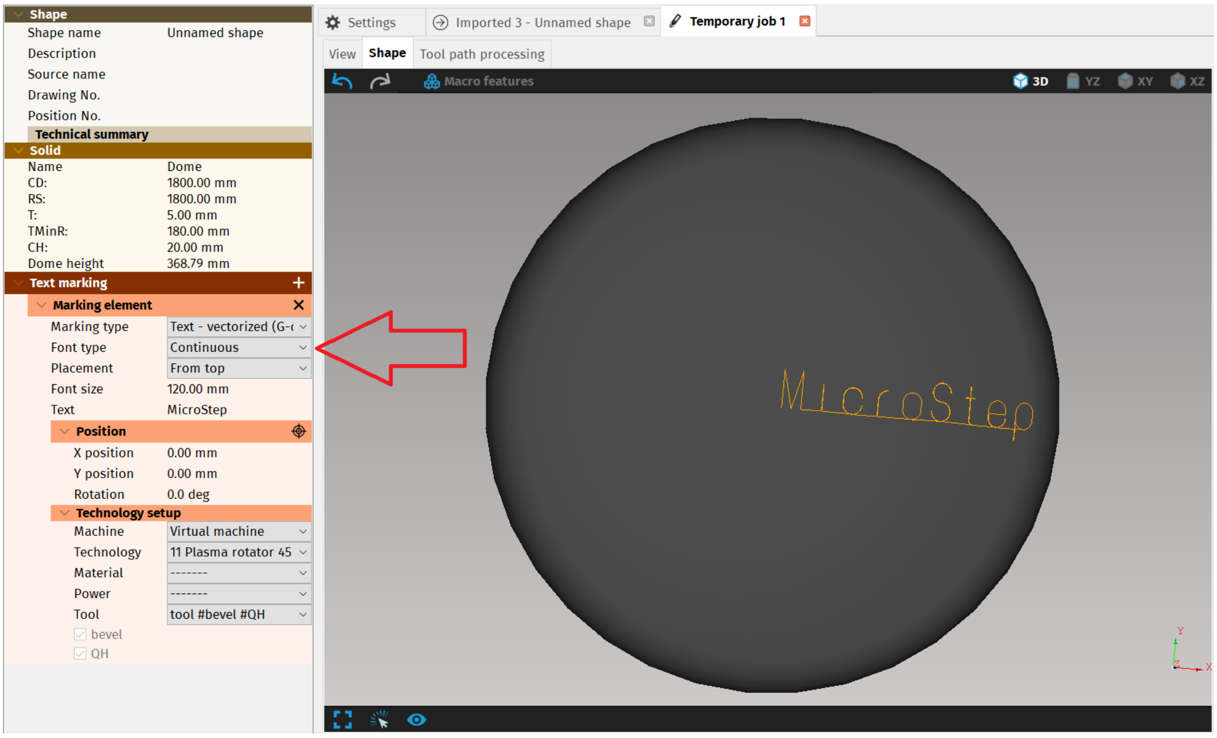


Fig. 194: Font type in Text marking section of Properties menu

Tool operations directly affects the processing of cutting paths, cutting speed control and kerf width compensation in each cut-point of cutting contour. mCAM controls cutting speed and compensates cutting path according to set Expert tables of particular tool which define exact cutting parameters for each technological group of parameters = operations. Cutting tool can be assigned to stock or whole template.

Expert table initial setup is described in more details in section Expert tables.

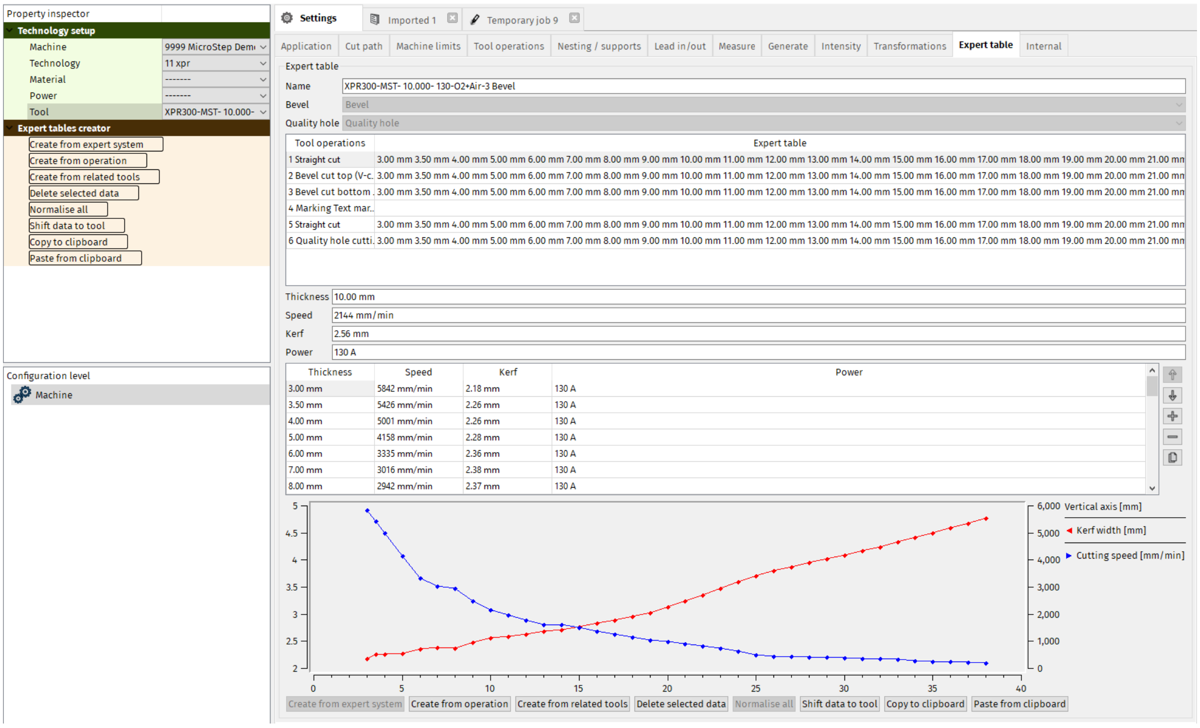


Fig. 63: Expert cutting tables – cutting parameters (kerf width/cutting speed)

Machine and tool selection

Machine and tool have to be selected every time when preparing cutting plan in *Job task*.

Machine limitations (according to the type of the cutting head and range of rotator axes) and technology limitations (given by maximum angle that is possible to reach between torch and material normal while cutting) for each cutting path are based on a *machine – technology* (with defined ranges and limitations) that is assigned to particular *stock* or whole *template*.

Machine and machine limitations are described in more details in section WebService or Virtual machine on page 194 and section Machine limits.

Selection of tool directly affects the kerf width compensation (according to *expert tables* of particular tool). Principles of *kerf width compensation* and initial setup of *expert tables* are described in section Expert tables.

Note: Machine and tool can be set for each stock individually or for whole template at once (all stocks within template shares template´s machine settings, technology and tool). Cutting parameters of chosen tool (with applied expert tables) while generating CNC program are defined/possible to edit in Settings – Expert table.

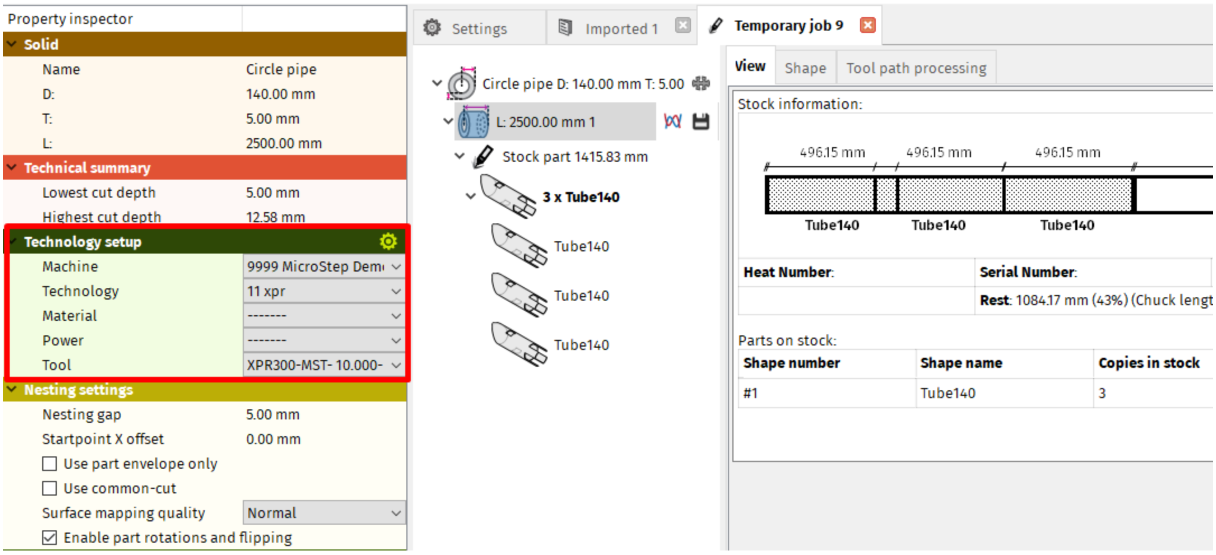


Fig. 64: Machine and tool selection

Displaying final cutting paths (Tool path processing) – because all cutting paths (kerf width compensation, technological and machine limitations) are processed during CNC generation, final cutting paths can be displayed only in Tool path processing tab in Job task.

If necessary, machine and tool can be selected for each cut individually by selecting the cut path and assigning a desired machine and tool to in "Technology setup (cut path)" in Properties area.

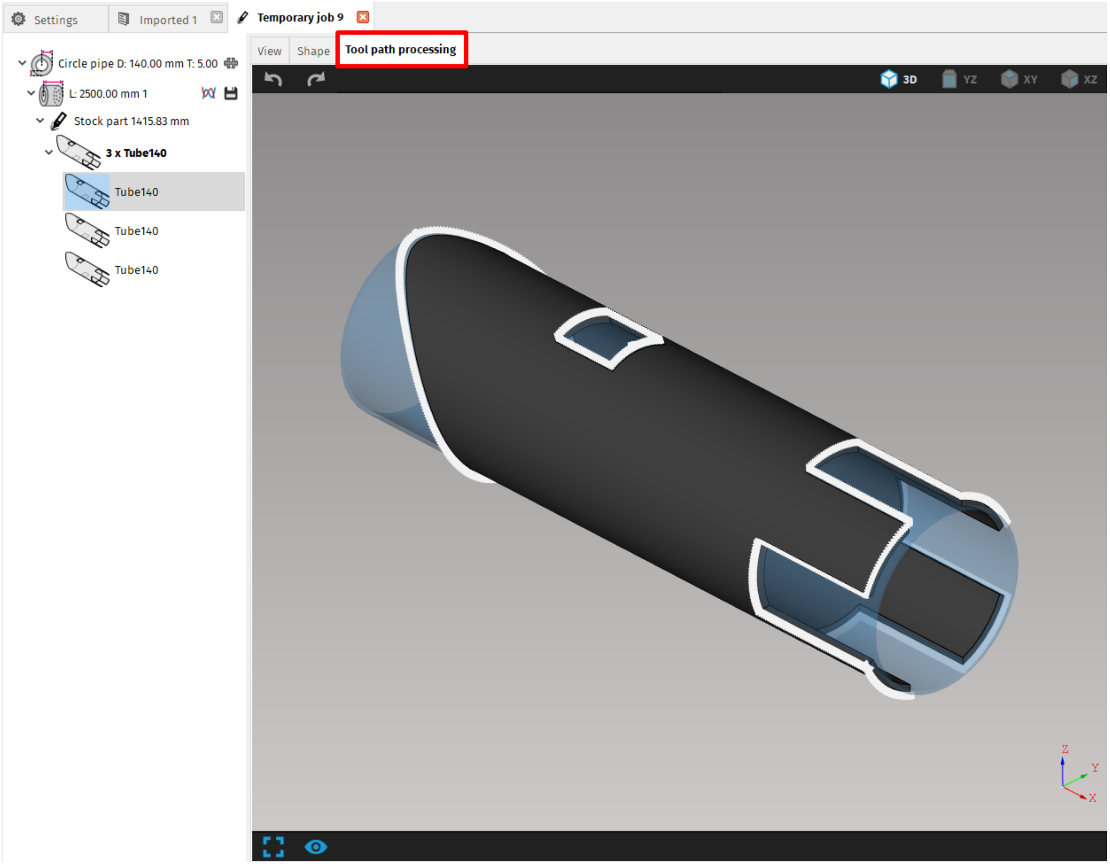


Fig. 65: Tool path processing – limited and compensated according to set machine and tool

Simulation of CNC program and graphical visualization

The *mCAM* contains a machining simulator for verification of the CNC code. The simulator displays a stock material defined in the program, outline of a cutting tool, all cutting paths and a graph that helps to detect undesirable and inappropriate motion of tool or paths. Functions of the simulator and graphs are described in more details in section Cutting simulation of CNC program.

CNC program is generated by clicking on one of two buttons located next to the stock piece in *working tree* of particular Job. Left button generates and simulates CNC program and right button only generates and saves CNC program without running cutting simulation.

 generates and simulates CNC program

 generates and saves CNC program without running cutting simulation

Nesting report can be saved via button located in the bottom right corner in the View tab when respective stock is selected.

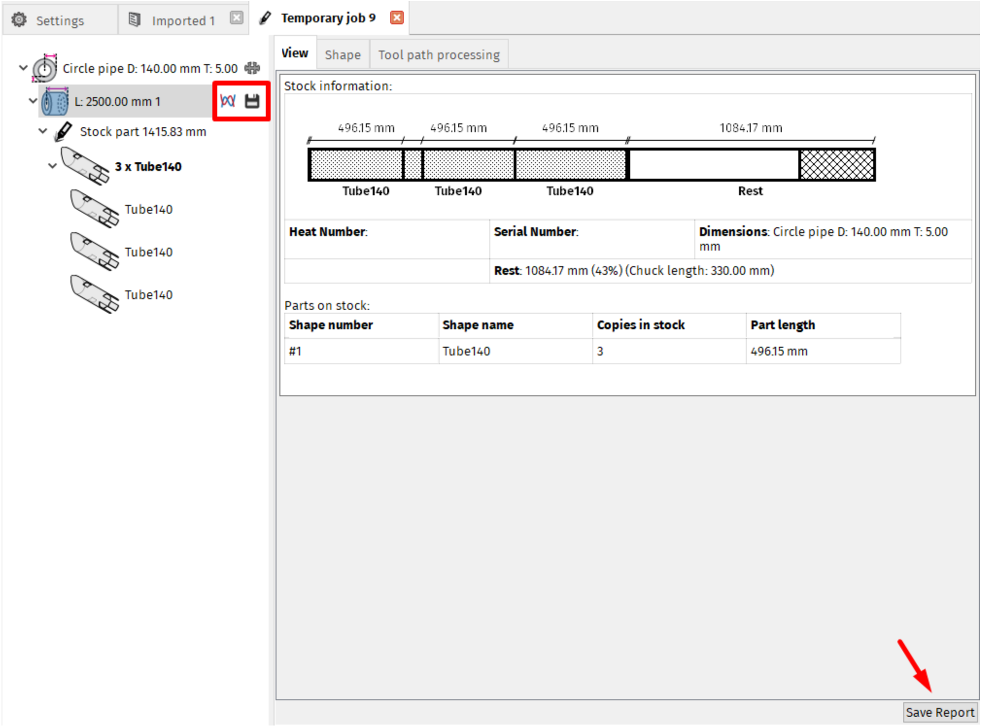


Fig. 66: Cutting simulation, saving CNC plan to the right of selected stock, saving nesting report in pdf. file in the bottom right corner

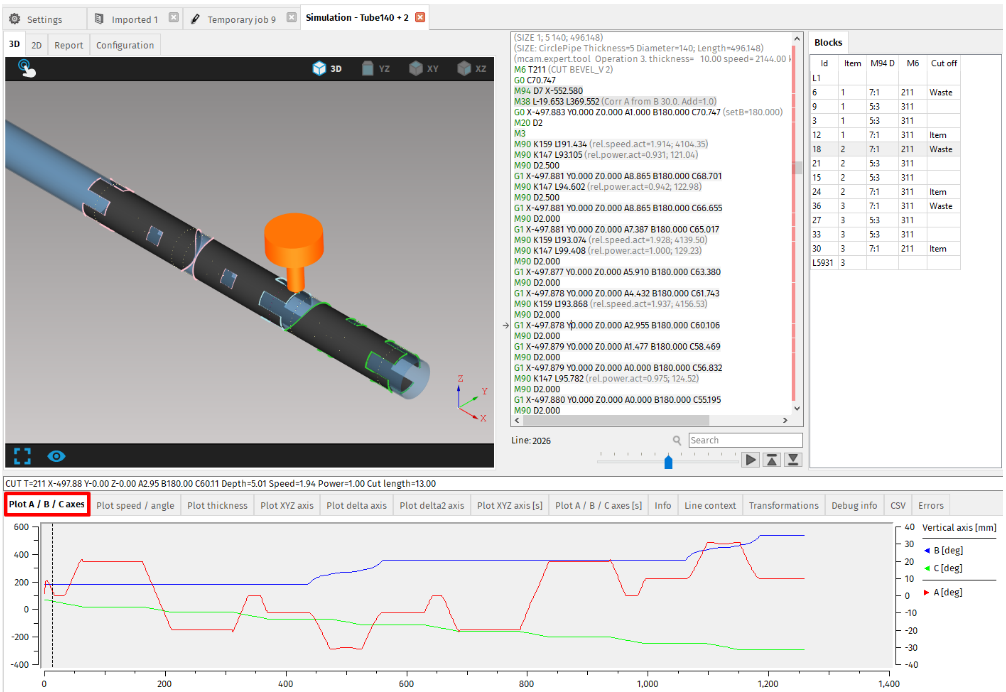


Fig. 67: Cutting simulation and plotting A, B and C axes movements

Related cutting reports are automatically saved in *.html file format into directory of saved CNC program, so it is possible to open it in any Web Browser.

mCAM report	
Created:	08.01.2019 10:20:29
Name:	Tube140 + 2
File:	C:/Users/AB/Desktop\Tube140 + 2.cnc
Stock:	Circle pipe D: 140.00 mm T: 5.00 mm
Stock:	L: 2500.00 mm
Material:	
Chuck location:	At X min

	Starts	Length
Cutting	12	7 m 587.404 mm
Detection	12	
Total	24	7 m 587.404 mm

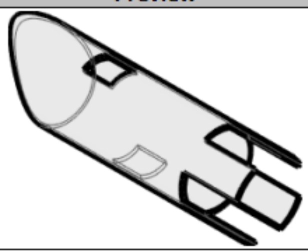
<table>
<tr><th>Preview</th><th>Info</th><th>Part name</th></tr>
<tr><td></td><td>
<table>
<tr><th></th><th>Starts</th><th>Length</th></tr>
<tr><td>Cutting</td><td>4</td><td>2 m 195.801 mm</td></tr>
<tr><td>Detection</td><td>4</td><td></td></tr>
<tr><td>Total</td><td>8</td><td>2 m 195.801 mm</td></tr>
</table>
<table>
<tr><th></th><th>Value</th></tr>
<tr><td>Count</td><td>3</td></tr>
<tr><td>Size</td><td>L: 496.15 mm</td></tr>
</table>
</td><td>Tube140</td></tr>
</table>

Fig. 68: Cutting report

Report customization in mCAM

Reports in mCAM

mCAM currently supports two kinds of reports.

The job report:

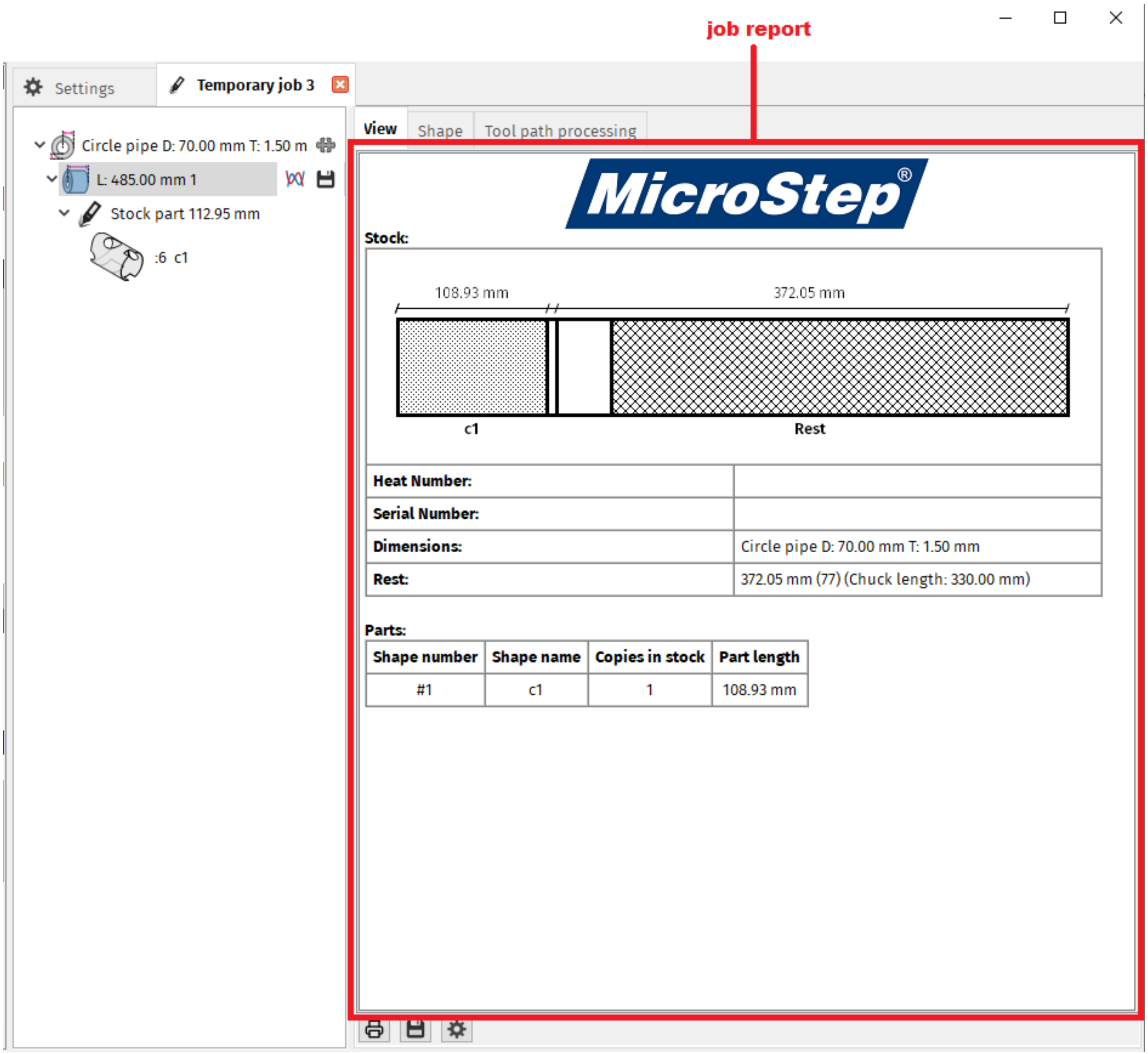


Fig. 69: The job report located in the View section

And the Shape cut report:

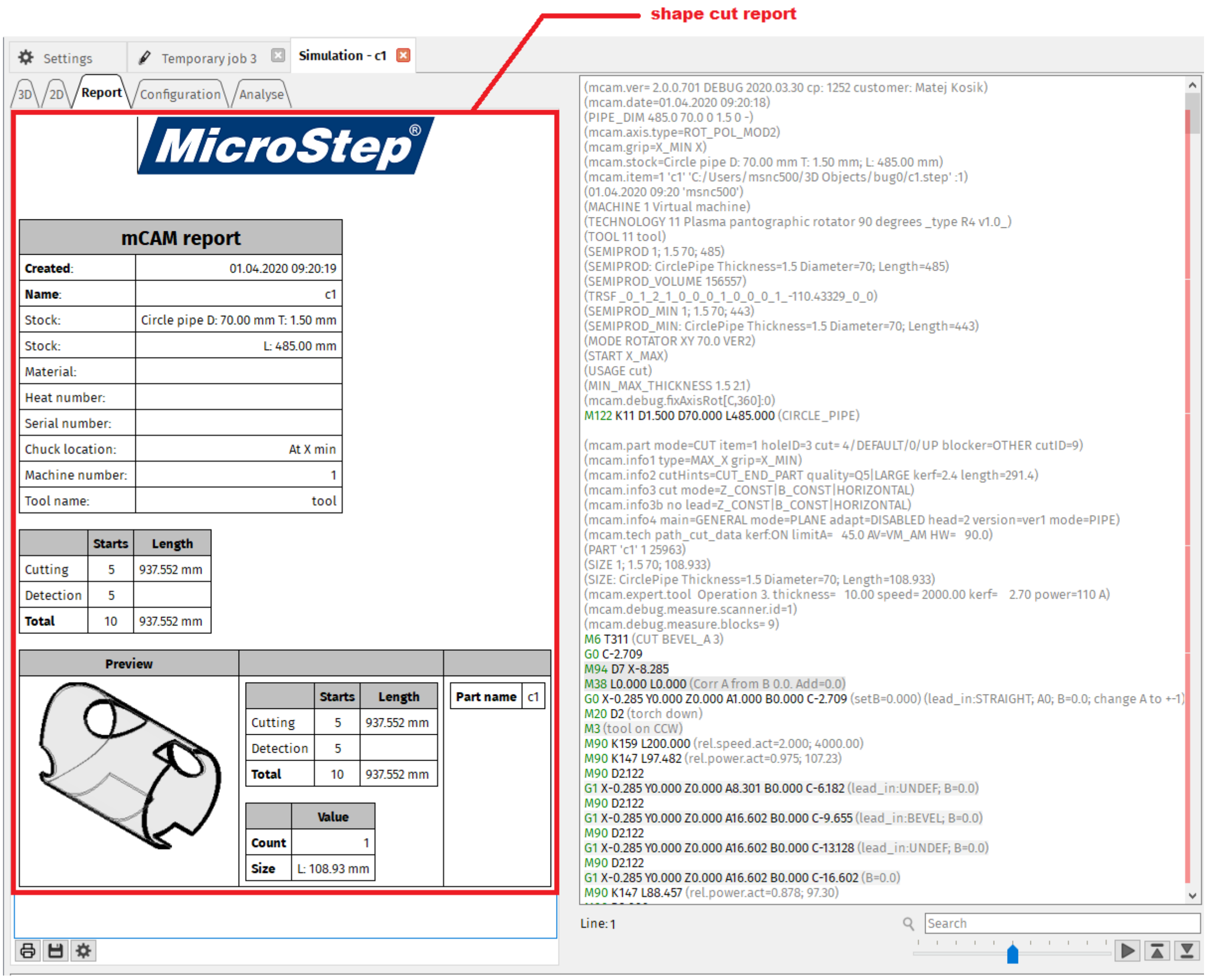
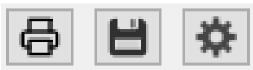


Fig. 70: The shape cut report in Simulation located in Report section

In the bottom left corner of each report is located a tool bar:



Through this tool bar it is possible to print, save or customize the report.

The Print button

For the moment, mCAM only has indirect support for printing. By clicking on the Print button, the report is opened in the user´s default browser. The browser then can be used for printing the report in the best available quality.

The Save button

The Save button  enables saving the current report to the disk as a single HTML file.

The Settings button

The Settings button makes it possible to customize the report. Pressing this button, a new window will appear:

Fig. 71: Check box for confirmation for report customization

After checking the box and by it confirming that the user wishes to customize the report and has at least some previous experience with the technologies mCAM uses for generating reports (HTML, CSS, XSLT, XPath, XQuery), the window will change.

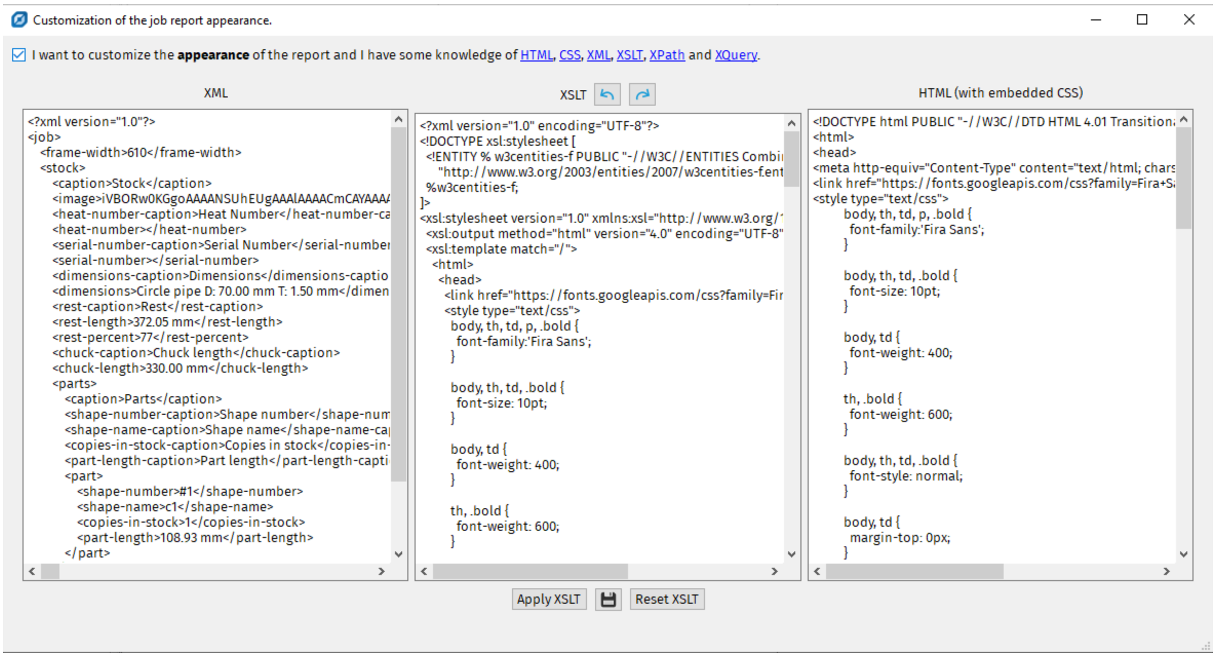


Fig. 72: The XML, XSLT and HTML text areas

The text area on the left shows an XML document that contains the data, that are supposed to appear in the report.

The text area in the middle contains XSLT transformation that defines how to create HTML document on the right from the XML data on the left.

The text area on the right shows HTML encoding of the actual report that will appear on the screen.

Trivial changes of the XSLT document

Trivial changes of the XSLT document can be done as follow:

- by directly modifying the XSLT code shown in the central text area

- and then pressing the "Apply XSLT" button

These modifications will be saved and will not be affected by mCAM restart.

Nontrivial changes of the XSLT document

Nontrivial changes of the XSLT document can be done as follow:

- by exporting the relevant files out of mCAM by pressing the Save button

- editing the XSLT code in the development environment of the user´s choice

- pasting the final XSLT code back to the central window

- pressing "Apply XSLT" button.

To revert the changes in XSLT instructions, simply press the "Reset XSLT" button.

Cutting in practice

Several conditions have to be met for the part to be cut correctly:

Profiles have to be placed correctly on the machine. There are different initial positions for beams depending on whether the beams are cut using rotary positioner or stationary beam holder. This position is defined by mCAM and has to be preserved.

Initial beam position in rotary positioner (if C=0) and *Initial beam position in stationary beam holder*).

The starting positions needs to be fine-tuned by operator before the start of cutting:

→ X-coordinate of cutting head has to be aligned with the end of beam. Some machines are equipped with optical sensor, which can fine-tune X-position of head on operators command.

→ R-axis should also be fine positioned, so that beam exactly conforms to required initial position. Mostly this means, that top surface of beam is in level

→ Technological parameters on the machine has to be set exactly in the same way as parameters that were used in mCAM while generating CNC program

M122 – initial beam position

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

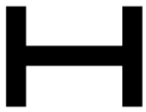
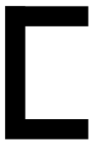
- p_M122_VERSION=2 – original placement of profiles as was before the change in 04/2017
- p_M122_VERSION=3 – new placement of profiles after the change in 04/2017

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 fikus 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 fikus 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.

Initial beam position in rotary positioner (if C=0)

Listed below are default starting positions of different types on beams in rotary positioner on machines as generated by mCAM, if C is 0.

Due to the change in default position in april 2017, mCAM will generate M122 instruction with information about what is the default position in generated CNC. The machine will have defined which M122 are compatible and will reject all that aren't. If this happens, it is necessary to check, if mCAM is set correctly. Otherwise, it is possible that mCAM or fikus should be updated – in this case, please, contact MicroStep support team.

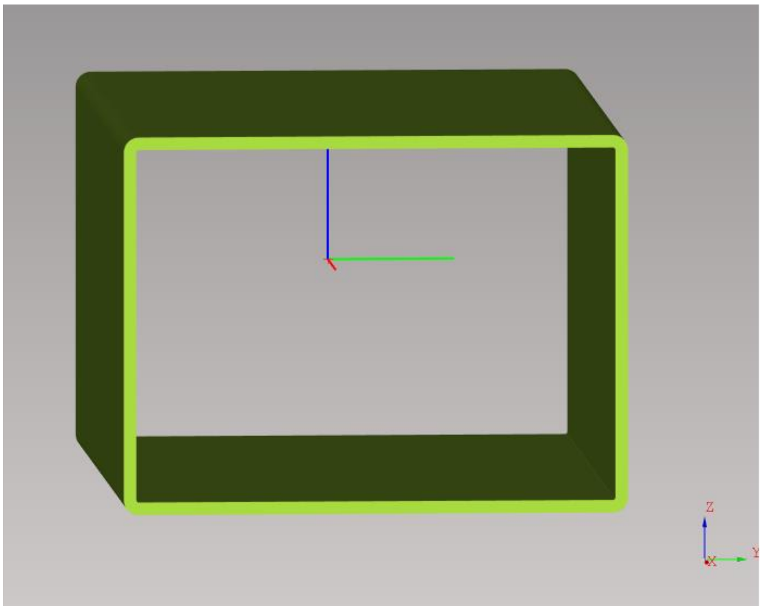


Fig. 73: Square pipe – longer side is in Y-axis

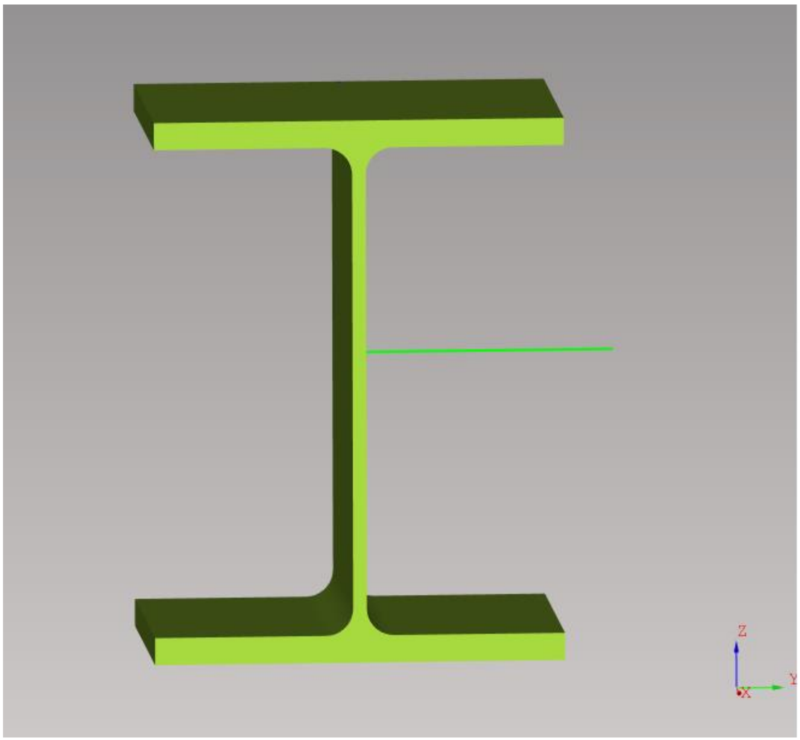


Fig. 74: H-bar – (older position) top edge of cutting contour, where the cutting starts, needs to be in Y+ direction, the flanges are in Y-axis

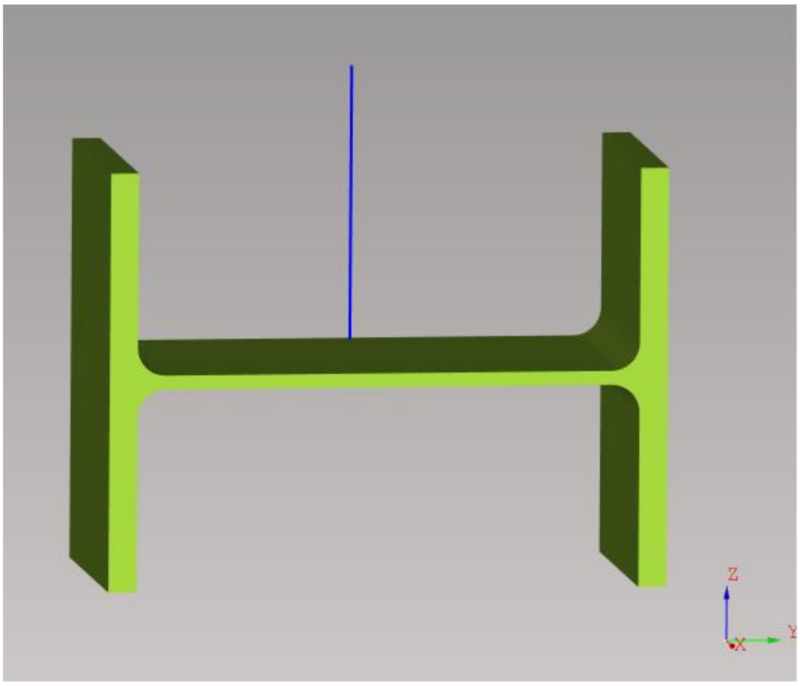


Fig. 75: Flanges are in Z-axis direction (new position)

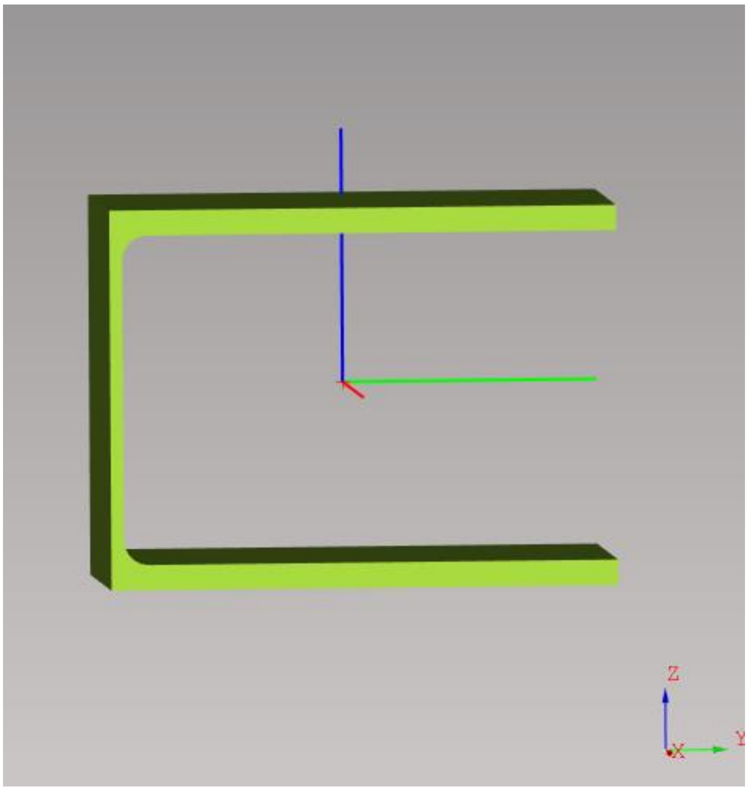


Fig. 76: U-bar (old position) Flanges are in Y axis with web facing in Y- direction

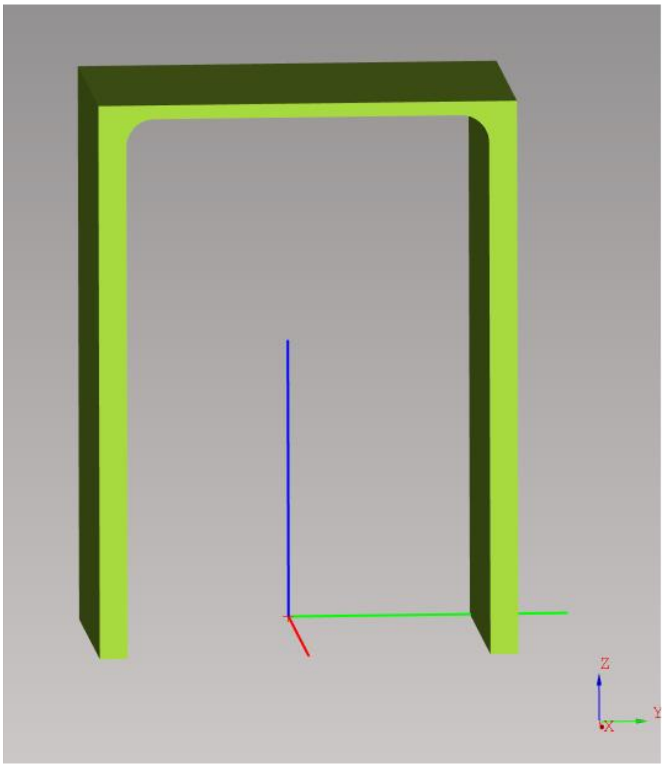


Fig. 77: U-bar – Flanges are in Z-axis, with web facing Z+ direction (new position)

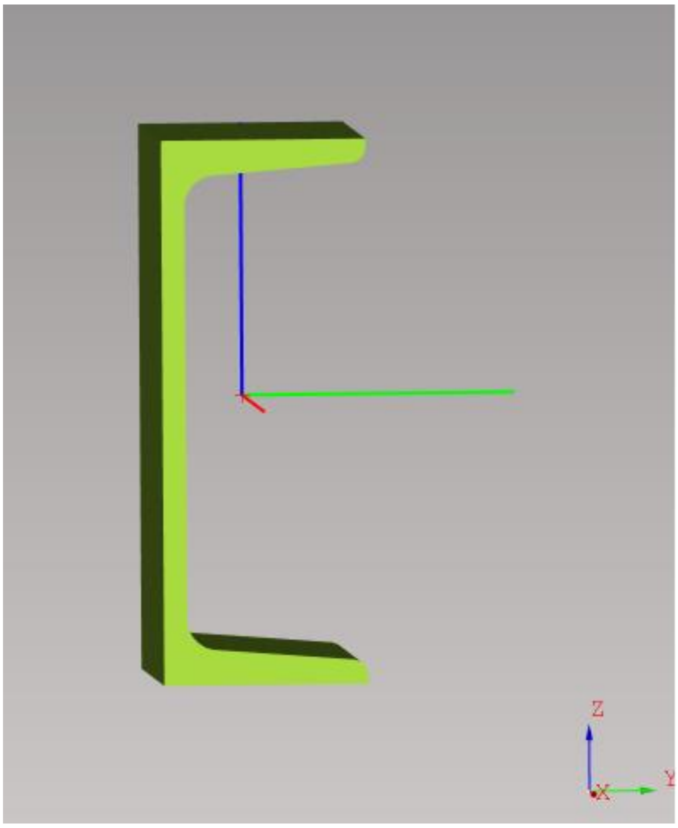


Fig. 78: UPN bar (old position) Flanges are in Y axis with web facing in Y- direction

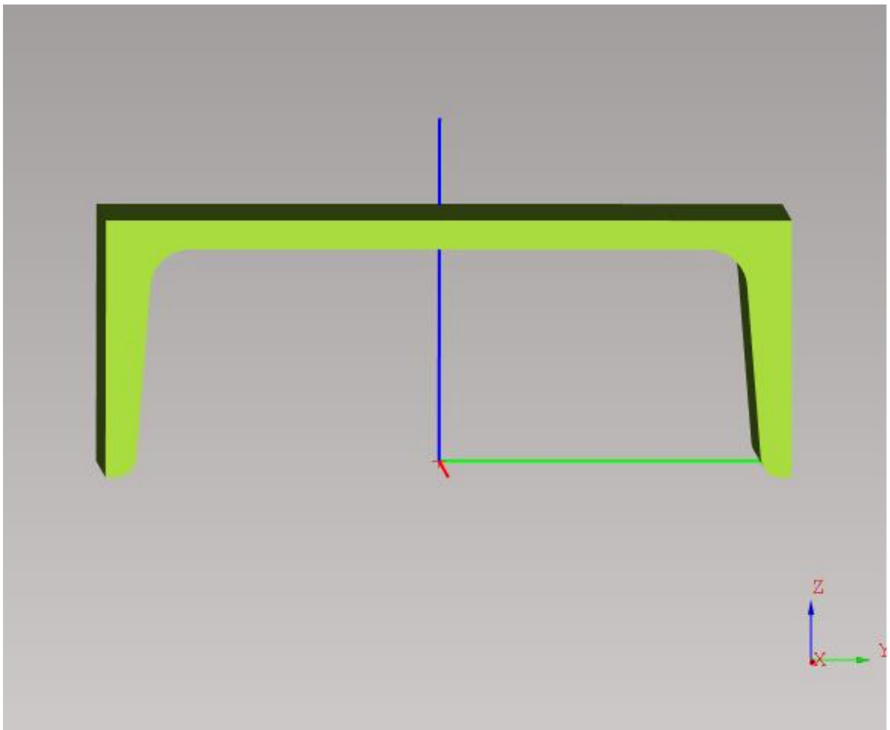


Fig. 79: UPN-bar – Flanges are in Z-axis, with web facing Z+ direction (new position)

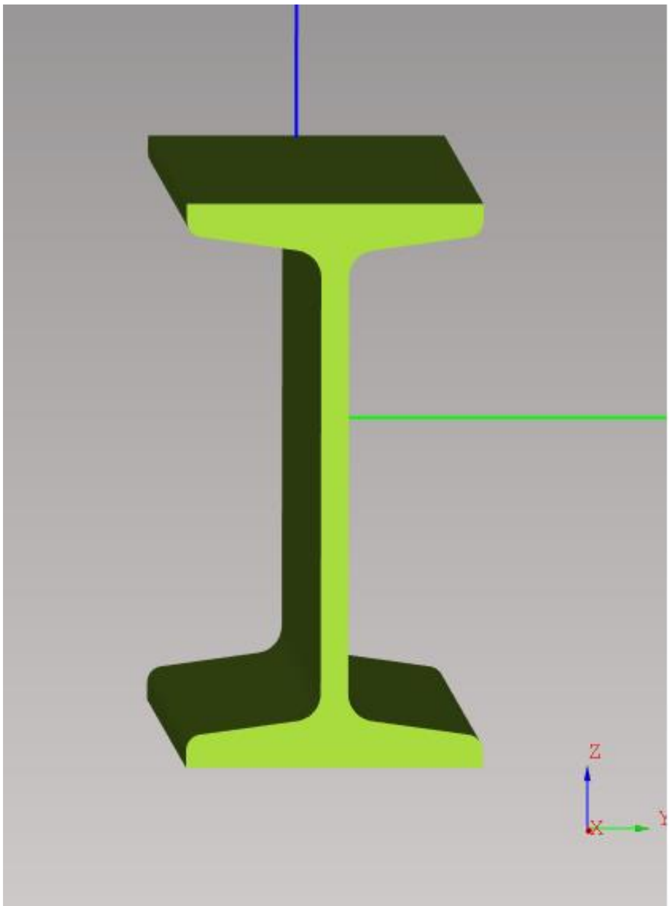


Fig. 80: I-bar – (older position) top edge of cutting contour, where the cutting starts, needs to be in Y+ direction

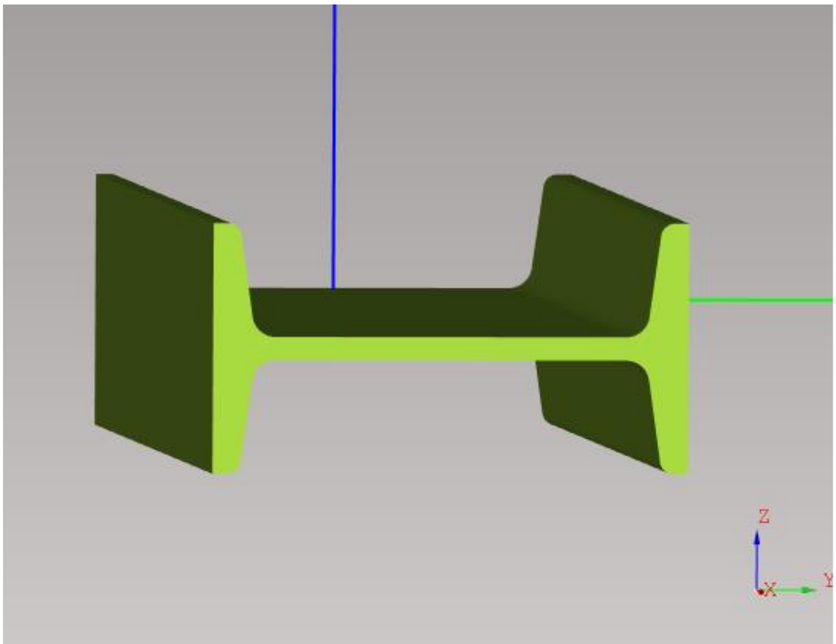


Fig. 81: Flanges are in Z-axis direction (new position)

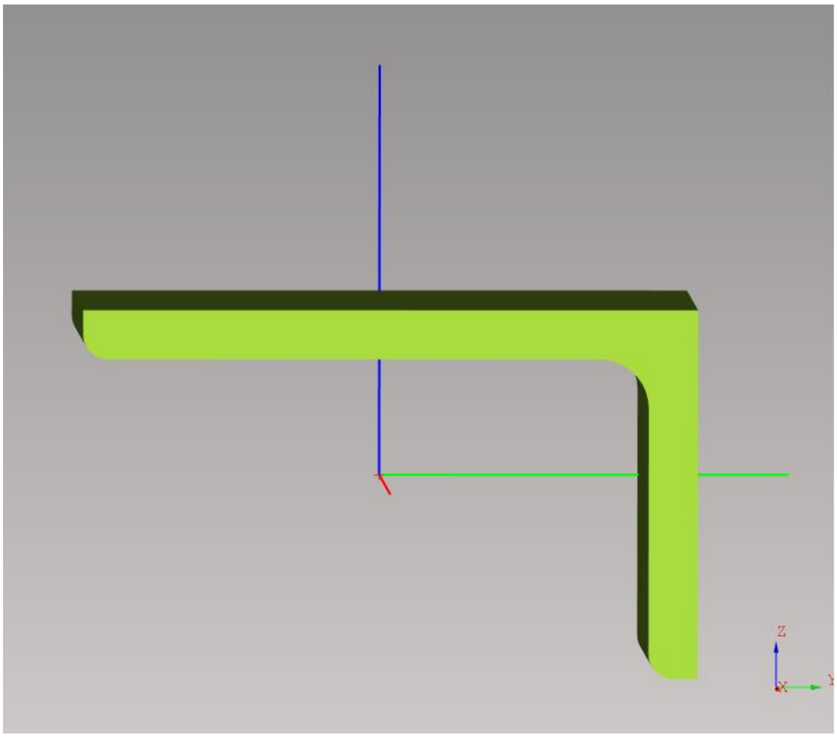


Fig. 82: L-bar – shorter side has to be on the Y+ side, longer side is facing upward in Z+ direction

Initial beam position in stationary beam holder

Listed below are default starting positions of different types on beams in plane cutting.

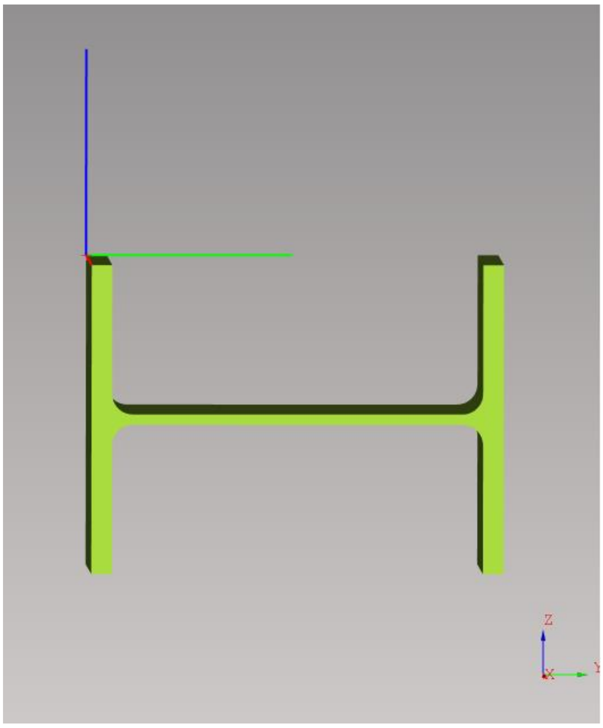


Fig. 83: H-bar – Flanges are in Z-axis direction

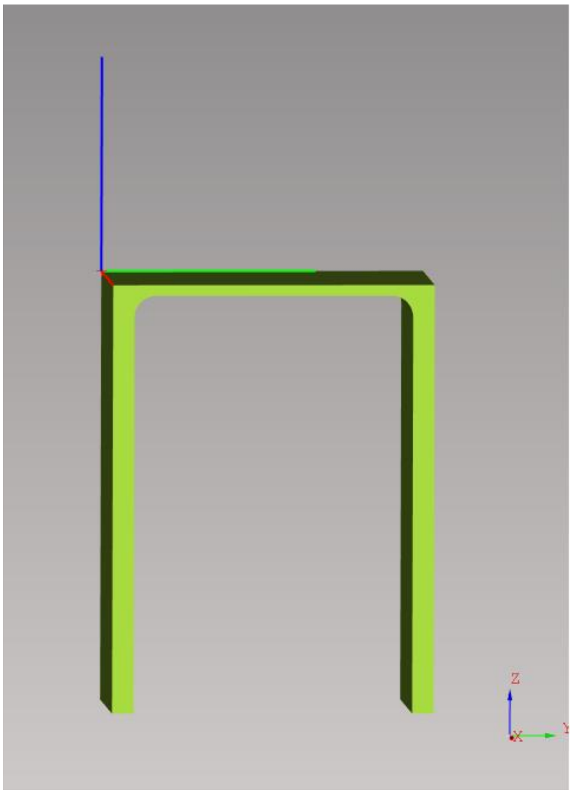


Fig. 84: U-bar – Flanges are in Z-axis, with web facing Z+ direction

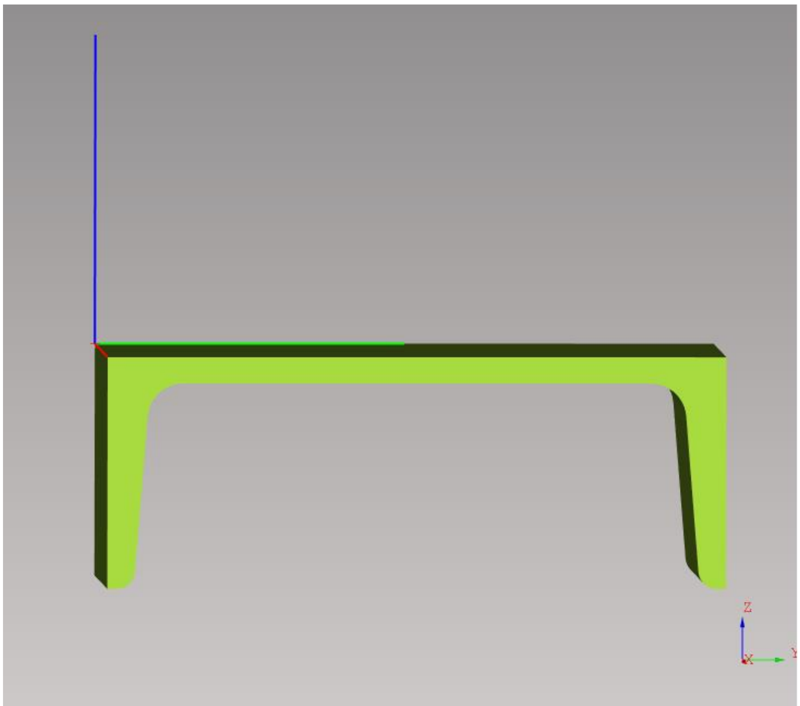


Fig. 85: UPN-bar – Flanges are in Z-axis, with web facing Z+ direction

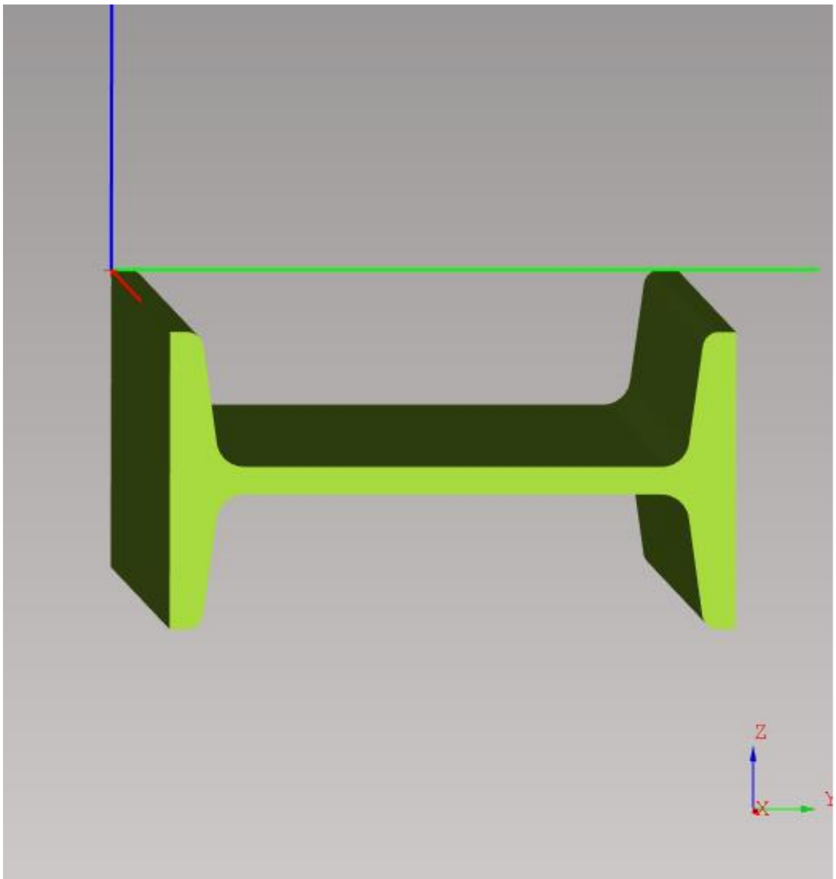


Fig. 86: I-bar – flanges are in Z-axis

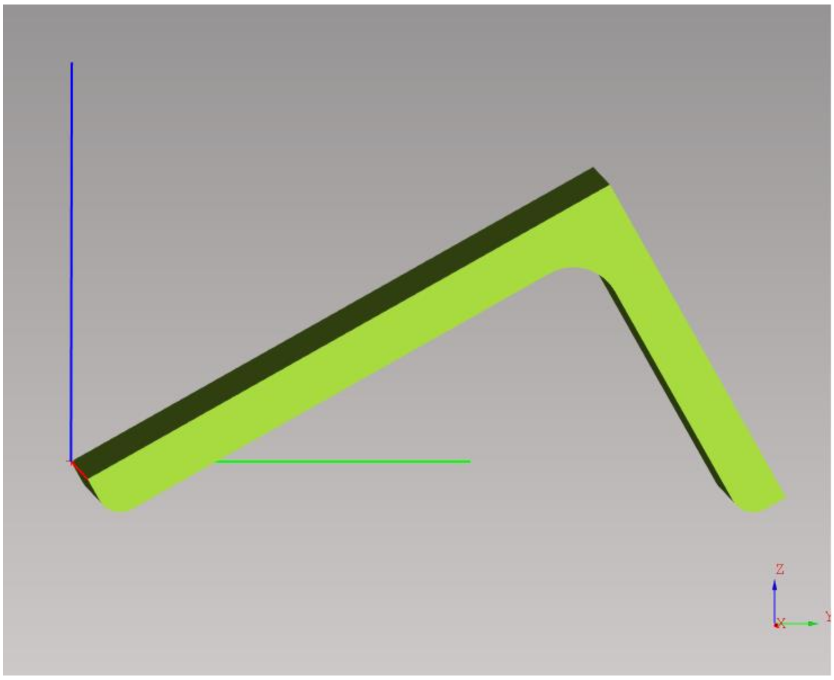


Fig. 87: L-bar – shorter side has to be in Y+ direction

Modeller

Modeller is a simple cad design feature, where you can design simple parts as subtraction of one shape from another. It is located in menu File -> Modeller or in Import tab as third sub-tab. Modeller is not replacement of any complex CAD programs as it only allows to create shapes based on supported shapes with straight cut-outs.

Add a new base shape by clicking on the right mouse button in the work area and selecting “Add”. It will add a default shape - circular pipe. This part acts as a parent (result, base) part. Adding a second shape (by clicking with right mouse button in the work area) will create a helper shape that is automatically subtracted from a parent part. Position of helper shape can be adjusted by applying individual steps of movement or rotation in one of the three axes X, Y and Z in Transformation part in Properties area to create a desired part.

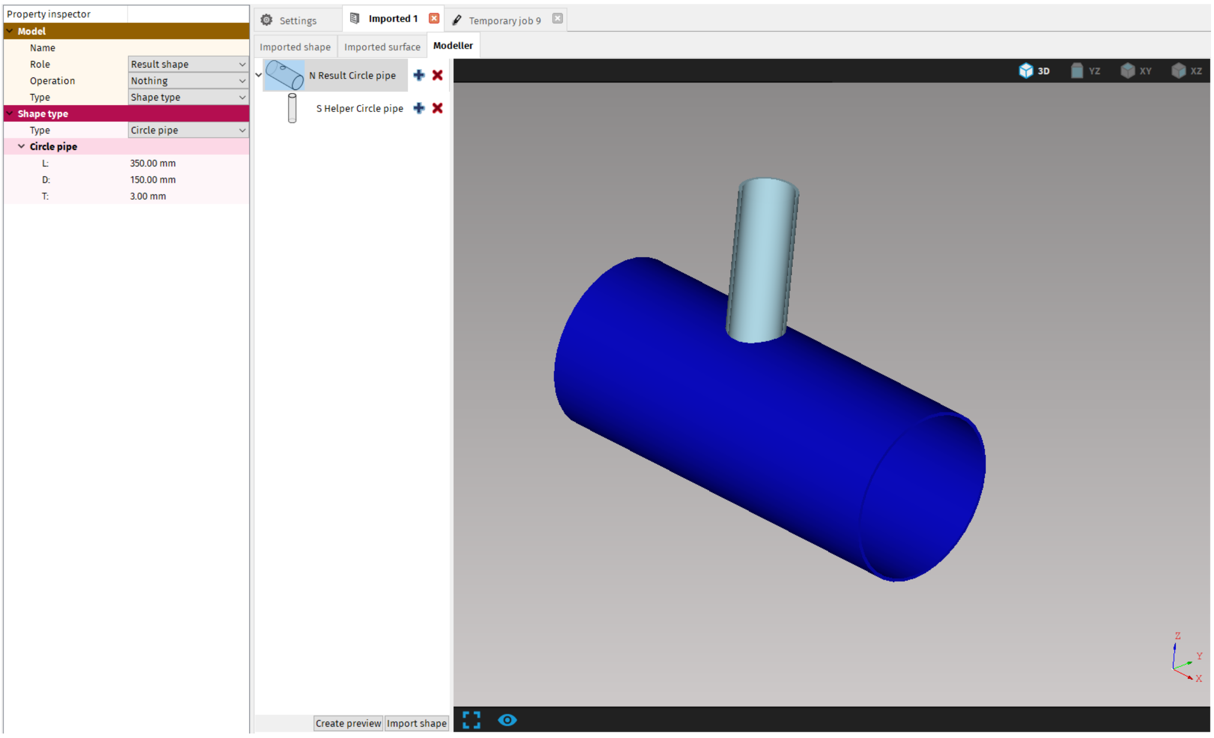


Fig. 88: Modeller

Helper shape is displayed in light blue color, result shape is dark blue. Options for specifying the setting for each shape are displayed in a properties area.

1. **Model** – you can specify Name, Role, Operation and Type

    - **Role –** specifies if the selected shape is result or helper shape. It is set automatically.
    - **Operation –** two operations are available – Nothing and Subtract. Nothing is intended only for result shape, that acts as a base shape for designing a new

part. Subtract is intended for helper shapes, that are subtracted from result shape to create the desired part.

- **Type –** specifies the form of the added shape. A new shape can be added as a Shape type, Solid, Macro or a Multiplier
  - → **Shape type –** added shape is a general shape that you can further specify
  - → **Solid –** adds a shape from imported shapes in active import tab
  - → **Macro –** added shape is a premade shape with special characteristics. Right now, only one macro is available – Circle pipe cut to demonstrate the use of Macros. If needed, more shapes can be added.
  - → **Multiplier –** described in more details on the next page

2. **Shape type** – you can select various shapes from the dropdown menu and adjust the dimensions of the respective type.
3. **Transformation** – visible only when helper shape is selected. Transformation commands allows you to move or rotate the shape to the desired position.

Multiplier

Multiplier automatically multiplies respective helper shape/shapes according to the given instructions.

1. To add a multiplier right click on a result shape and select „Add“.
2. In a Model section in Properties area change the Type to Multiplier
3. Add a helper shape by right clicking on Multiplier shape in Work area to assign it to Multiplier. You can add more shapes than one.
4. Adjust the dimensions of a helper shape and its position on a result shape, from which you want to start multiplying it across the result shape
5. Select Multiplier and change how many times you want to multiply the helper shape.
6. In transformation add steps that define the movement and/or rotation of multiplied shapes. The helper shapes will automatically multiply and distribute evenly according to this instruction. If you have more helper shapes assigned to one Multiplier, all will follow these set of instructions. If you want different quantity or shapes spaced out differently, you have to create separate multipliers for them.

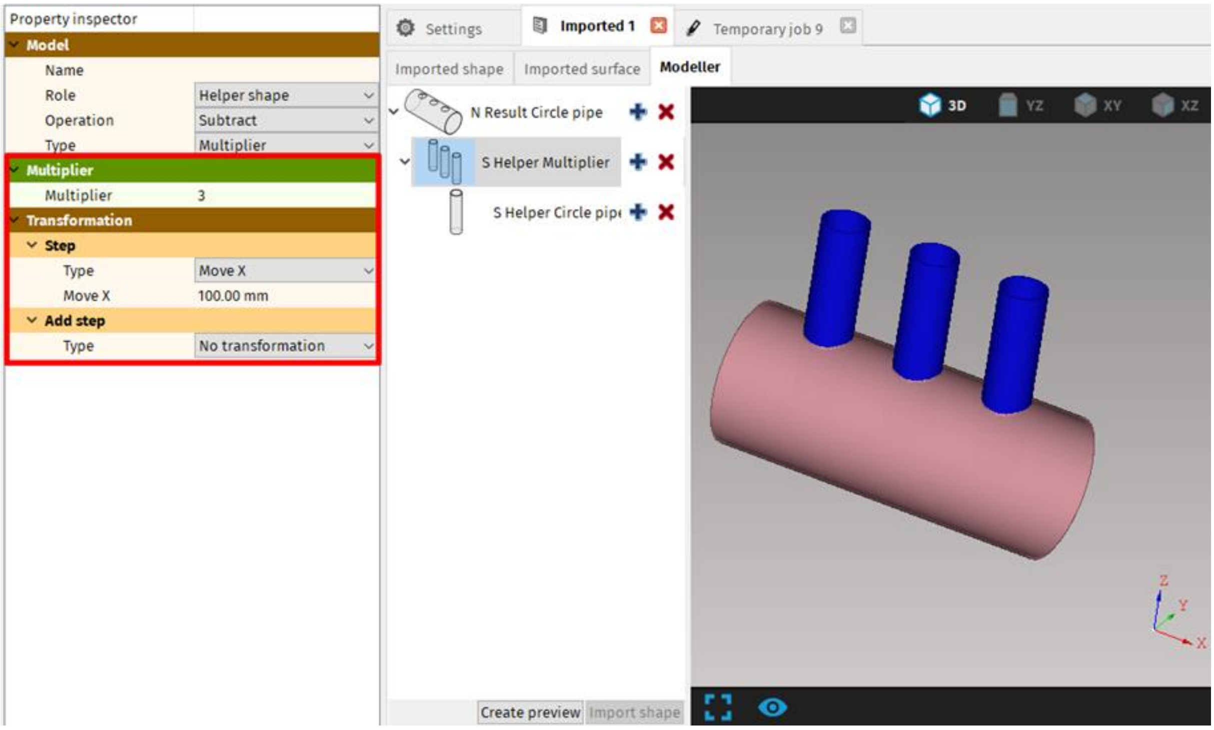


Fig. 89: Three circle pipes spaced out across the result shape by 120mm

Macro features

Macro features is a simple and time saving feature where you can design basic shapes. New shapes and macros are being added gradually.

To create a new shape, go to the Import tab, click **Create shape→ Macro features**. To edit a previously imported shape, go to the visualization screen and click the **Macro features** button at the top. Like this you can edit shapes made in Create shape, but also imported shapes, such as the ones from Step files.

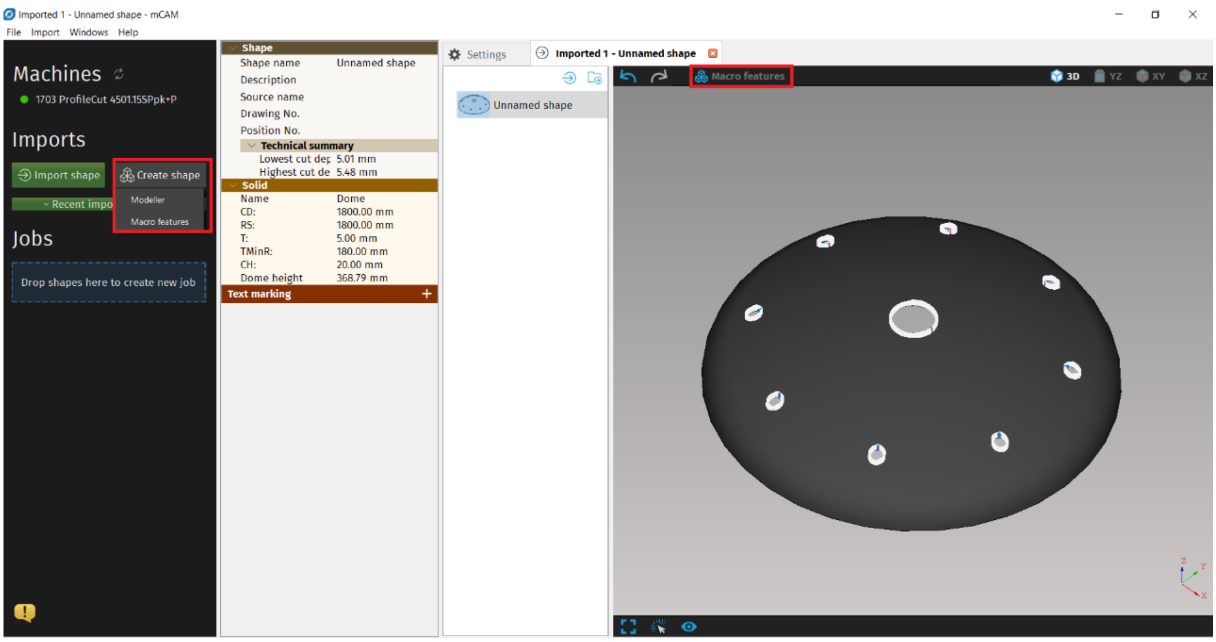


Fig. 90: Macro features

Create shape

Click Create shape and then Macro features, a table shows up and you can choose from the currently supported shapes categorized in “Domes” and “Profiles” groups. By clicking on one of them you will get a default shape.

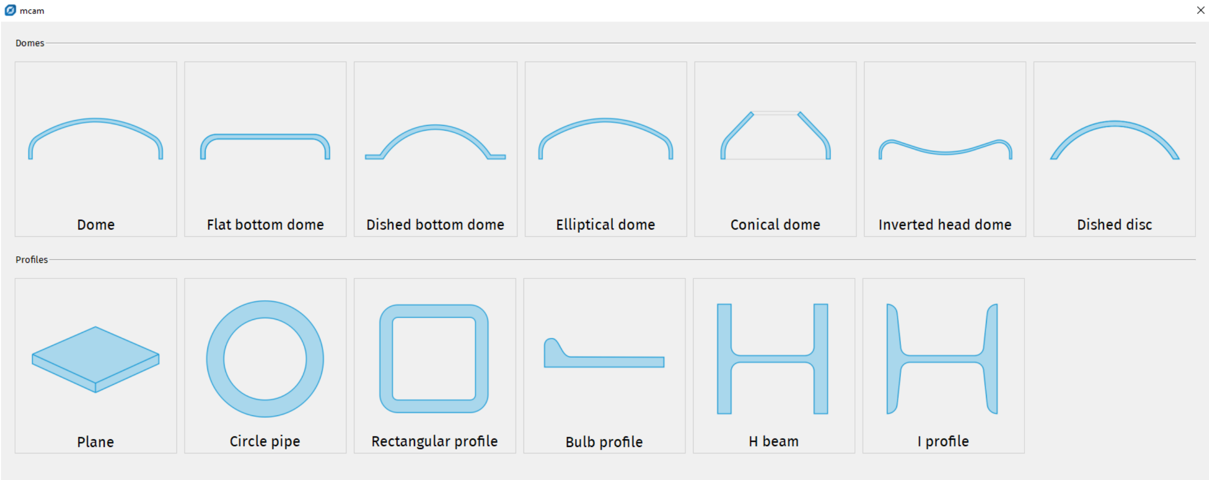


Fig. 91: Basic shapes table in Macro features

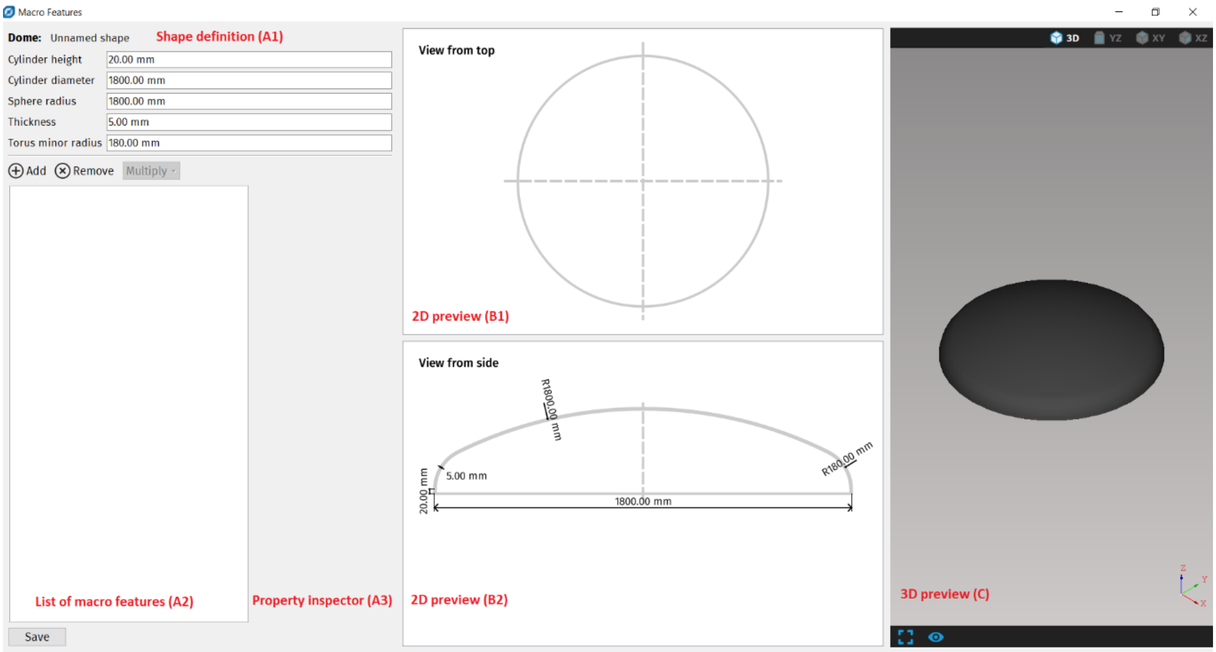


Fig. 92: Default shape - dome

The selected shape is then displayed on a visualization screen (C), the 3D view with visualization behaves the same as the other views in mCAM. The middle screen (B) shows the shape from two different angles, from top (B1) and from the side/cross section (B2).

Shape definition

If you are editing an already existing shape (previously imported or saved), shape dimensions cannot be changed.

When creating a new shape, its dimensions can be changed. You can do it like this:

- You can change the parameters of the shape by clicking on a desired dimension or change it in the Shape definition on the left screen (A1).

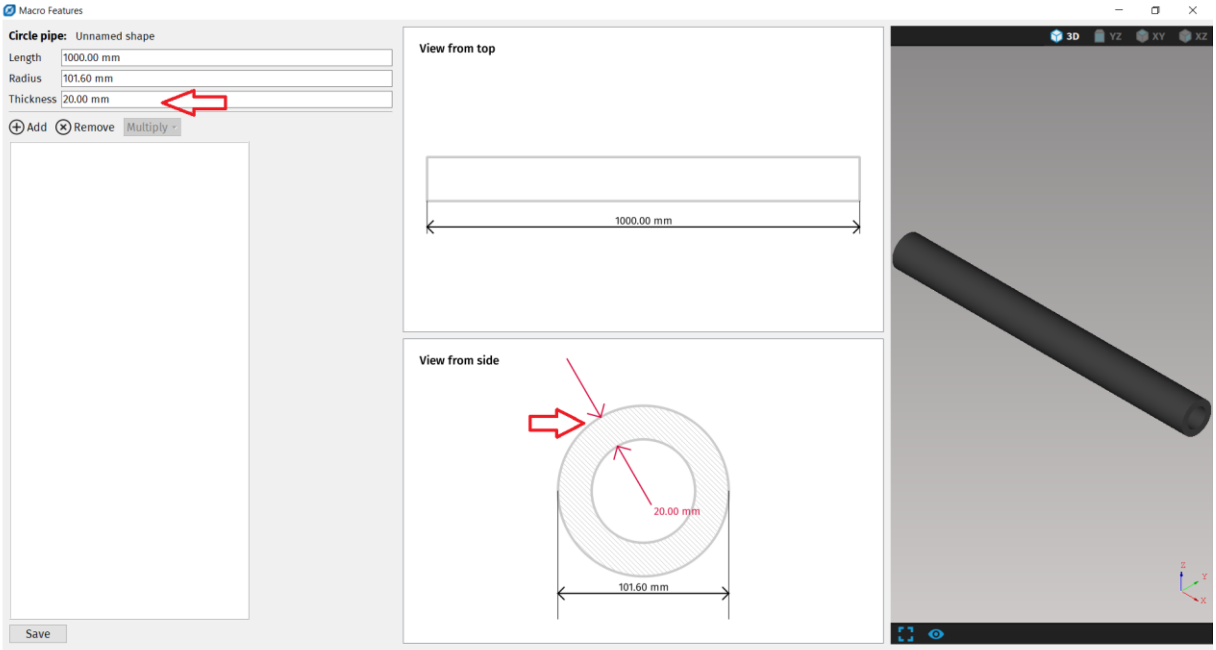


Fig. 93: How to change shape´s dimensions

Adding Macros

The **add button** on the left-most screen (A2) gives you a choice to add one of the supported macro features. Each shape of the stock enables you to add a different set of macros.

Macros available for dome

→ **Cylindrical hole** – is a macro of a circular shape, which properties such as diameter, distance, angle and tilt can be changed.

→ **Cross marking -** is a cross shaped type of marking, which properties can be changed. These are diameter, distance, angle and rotation.

→ **N – gon hole** – is a macro of polygon character where the number of sides is possible to adjust as needed. Other adjustable properties are diameter, distance, angle, rotation, tilt and an option to place the hole perpendicular to the surface. By checking the option for perpendicular hole, the hole is cut at right angle (90°) to the surface of the dome.

→ **Splitting in half** – is a macro that allows for the dome to be cut in half. The parameters are side offset, center offset, rotation. If the side offset and center offset are zero the dome is cut completely in half. Side offset determines how many millimeters from the edge should be left uncut. Center offset determines how many millimeters from the center should be left uncut.

→ **Side cylindrical hole -** is a macro of cylindrical shape, with properties such as diameter, offset, angle, rotation, tilt and an option for the hole to be placed perpendicular to the dome.

→ **Side cross marking -** is a cross shaped type of marking, with properties such as diameter, offset, angle and rotation. This type of marking is placed on the side of the shape.

→ **Side line marking –** is a line marked on the side of the shape, with properties such as length, offset, angle and rotation.

→ **Cut around –** is a macro that allows the shape to be cut around its circumference. The parameter for this macro that can be changed is height.

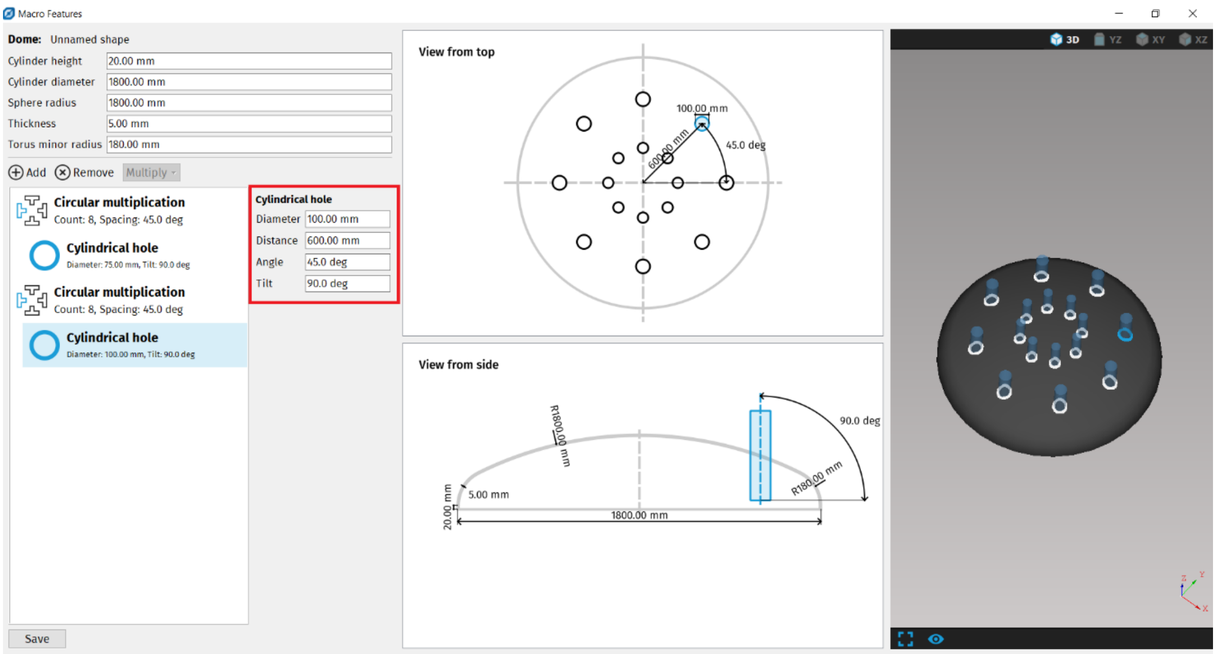


Fig. 94: The properties of Cylindrical hole

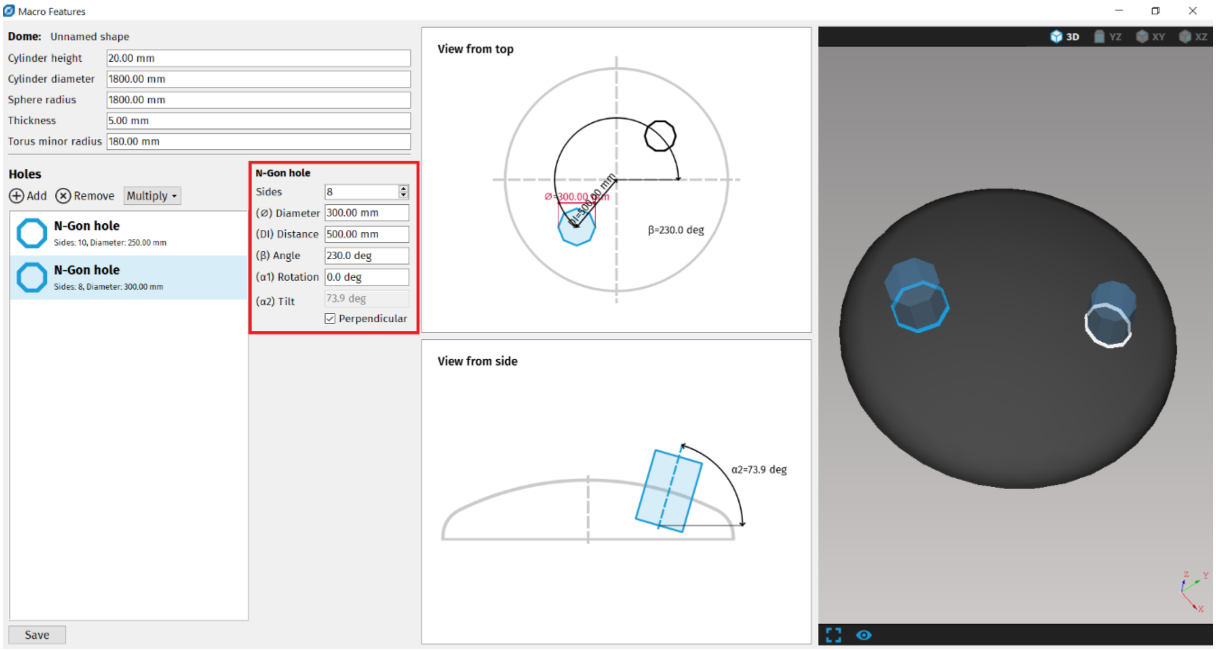


Fig. 95: The properties of N – gon hole

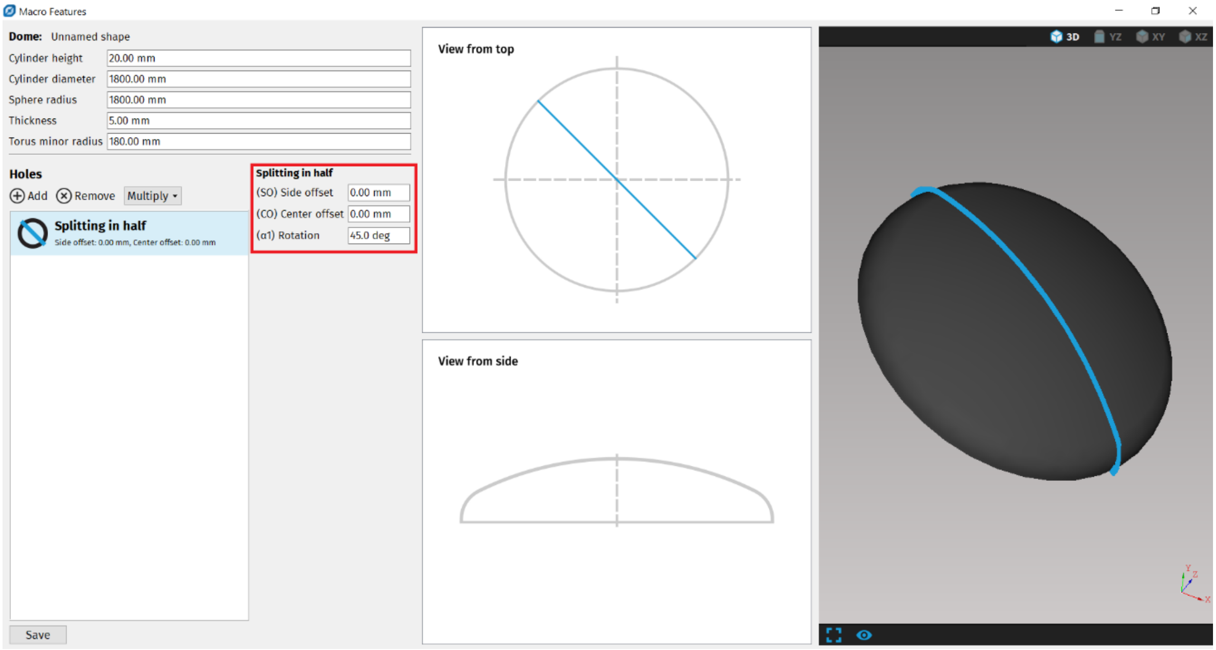


Fig. 96: Splitting in half for Dome in Macro features

Macros available for circle pipe

→ **Cylindrical hole** – the same as a cylindrical hole on a dome**.**

→ **Rectangular hole** – is a macro of a rectangular shape, which properties (width, height, distance, angle, offset, rotation) can be edited.

→ **Slot hole** – is a macro in a shape of an oval hole with properties such as width, height, offset, distance, angle, rotation and tilt.

→ **Laying pipe from top** – is a macro that allows to cut a hole in the shape of a pipe where it is possible then later in production to attach a desired pipe. The properties that can be changed are diameter, distance, angle, offset, rotation.

→ **Cross pipe** – is a macro that cuts two holes through the top and bottom side of the circle pipe. Both holes are circular in shape and their properties can be changed (diameter, distance, offset, angle, rotation, tilt).

Trimmings

Both ends of the circle pipe have the option for trimmings or no trimmings at all.

- No trimming
- Plane trimming – the cut off is straight and perpendicular (90°) to the cross – section of the circle pipe
- Cylinder trimming – the cut off is in the shape of a pipe applied perpendicular (90°) to the cross – section of the circle pipe
- Two-cylinder trimming – the cut off is in shape of two pipes which are perpendicular to each other and by default in 45° angle to the shape. The angle can be adjusted.

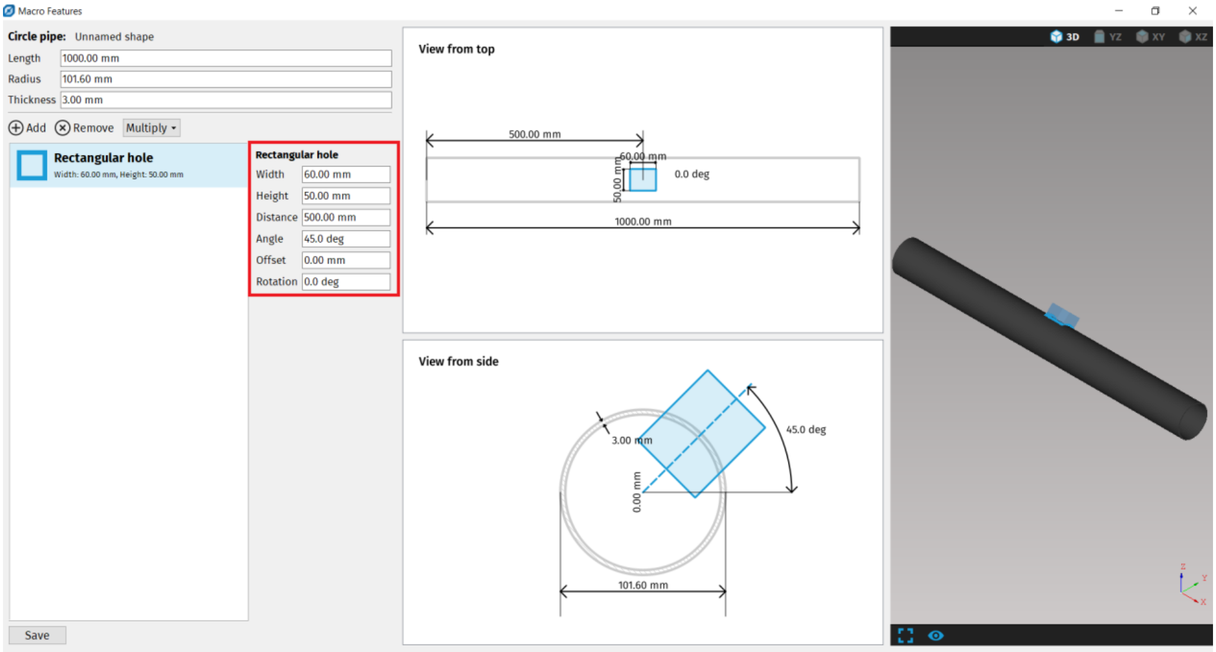


Fig. 97: Properties of a Rectangular hole

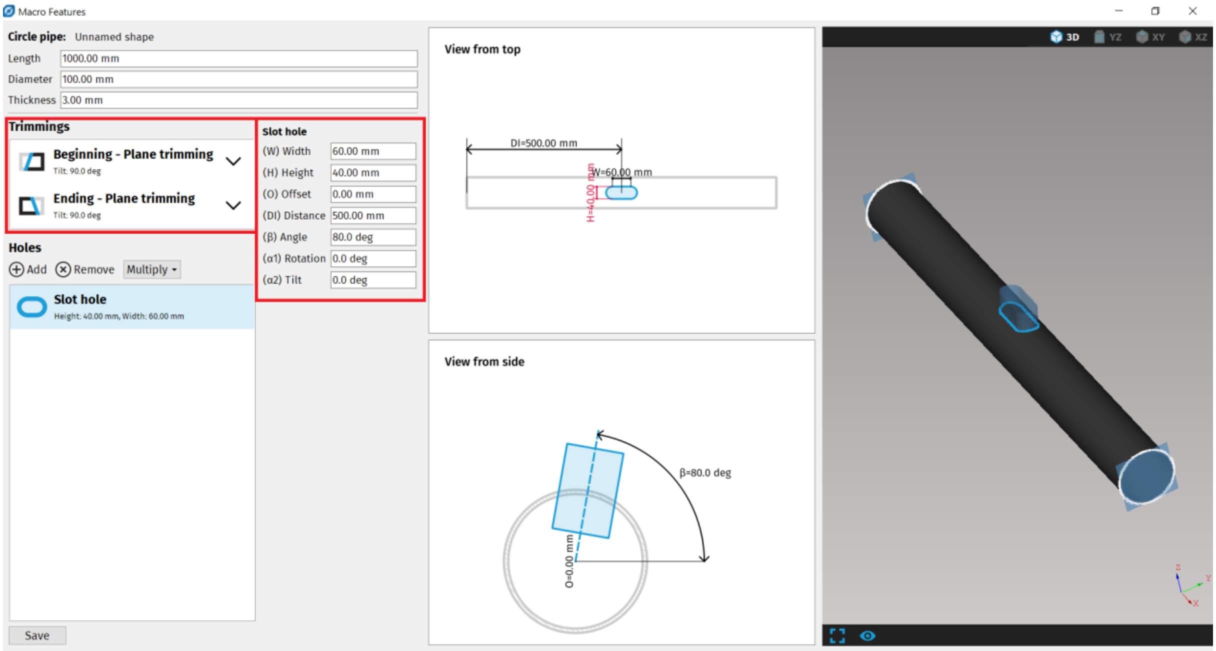


Fig. 98: Trimmings and properties of Slot hole for circle pipe

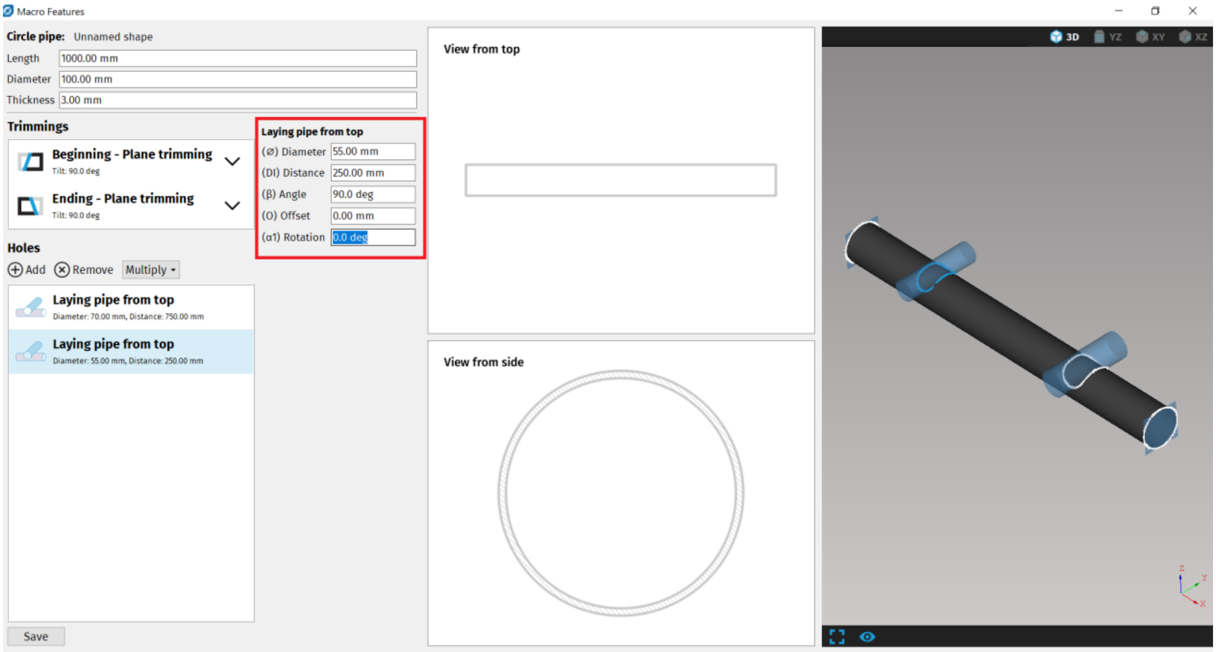


Fig. 99: Properties for Laying pipe from top in Macro features

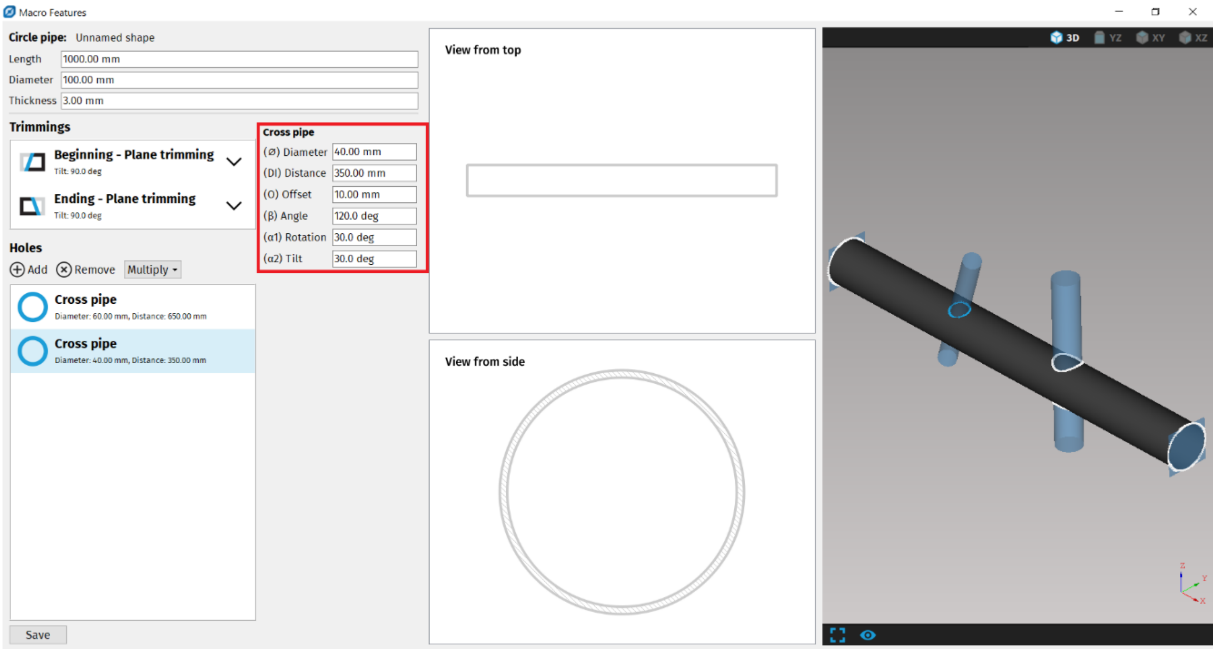


Fig. 100: Properties for Cross pipe in Macro features

Macros available for rectangular profile

→ **Cylindrical hole** – is a macro of a circular shape, which properties such as diameter, distance, angle and tilt can be changed.

→ **Rectangular hole** – is a macro of a rectangular shape, which properties (width, height, distance, angle, offset, rotation) can be edited.

→ **Slot hole** – is a macro in a shape of an oval hole with properties such as width, height, offset, distance, angle, rotation and tilt.

Trimmings

Both ends of the rectangular profile have the option for trimmings or no trimmings at all.

- No trimming
- Plane trimming – the cut off is straight and perpendicular (90°) to the cross – section of the rectangular profile.

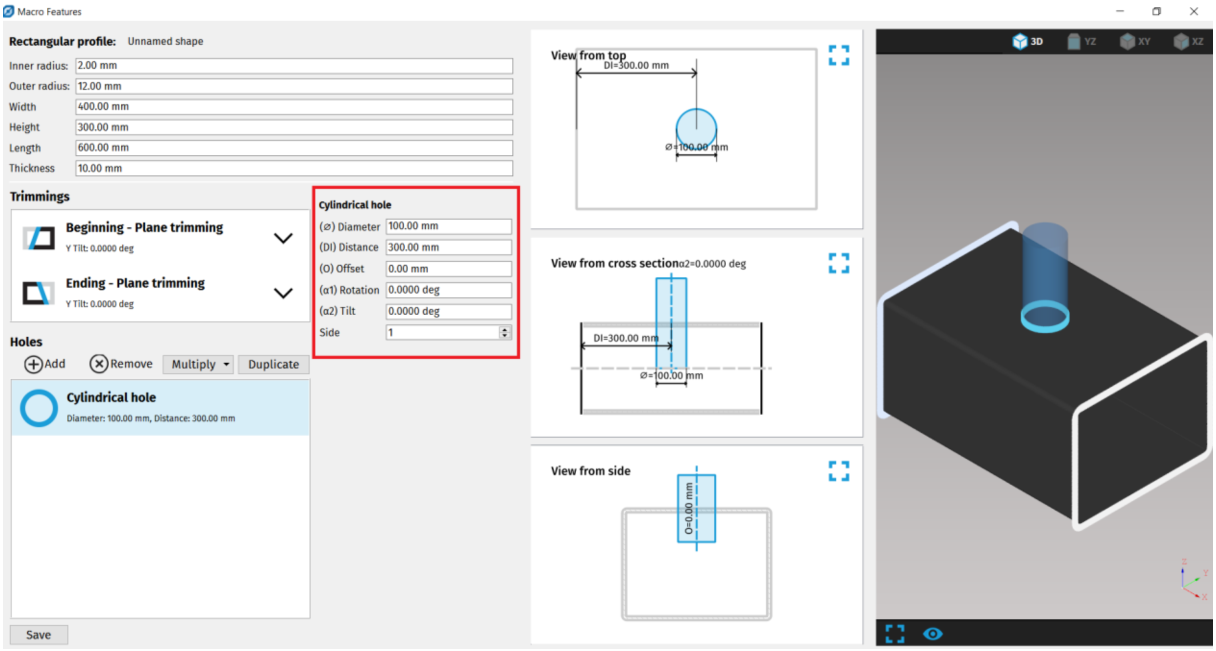


Fig. 101:Example of cylindrical hole on rectangular profile

Macros available for bulb profile

→ **Circular hole -** is a macro of a circular shape, which properties such as diameter, distance (X) and height (Y), rotation and tilt can be edited.

→ **Half circle** – is a macro that cuts a half circle to the straight (non – rounded) side of the bulb profile. Parameters that can be edited are distance (X) and the radius of the half circle.

→ **N1** – is a macro that can cut a rectangle shaped cut – out to the straight (non – rounded) side of the bulb profile. Parameters that can be changed are distance (X), width (A), length (B) and radius (R1).

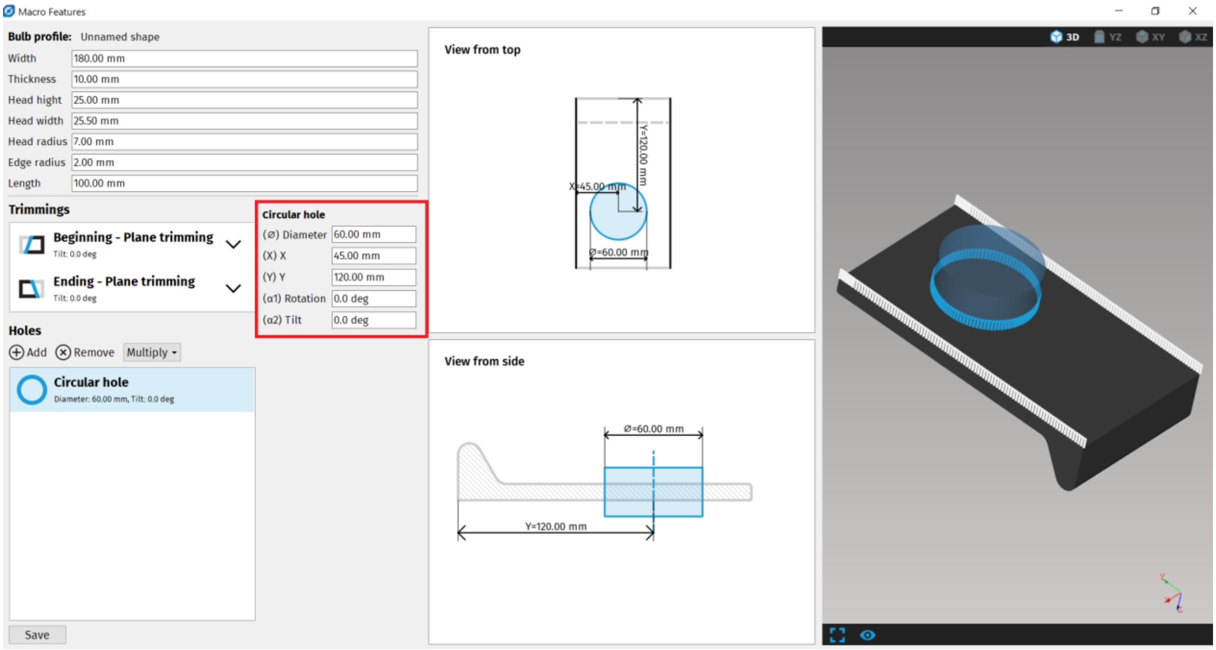


Fig. 102: Properties of Circular hole for Bulb profile

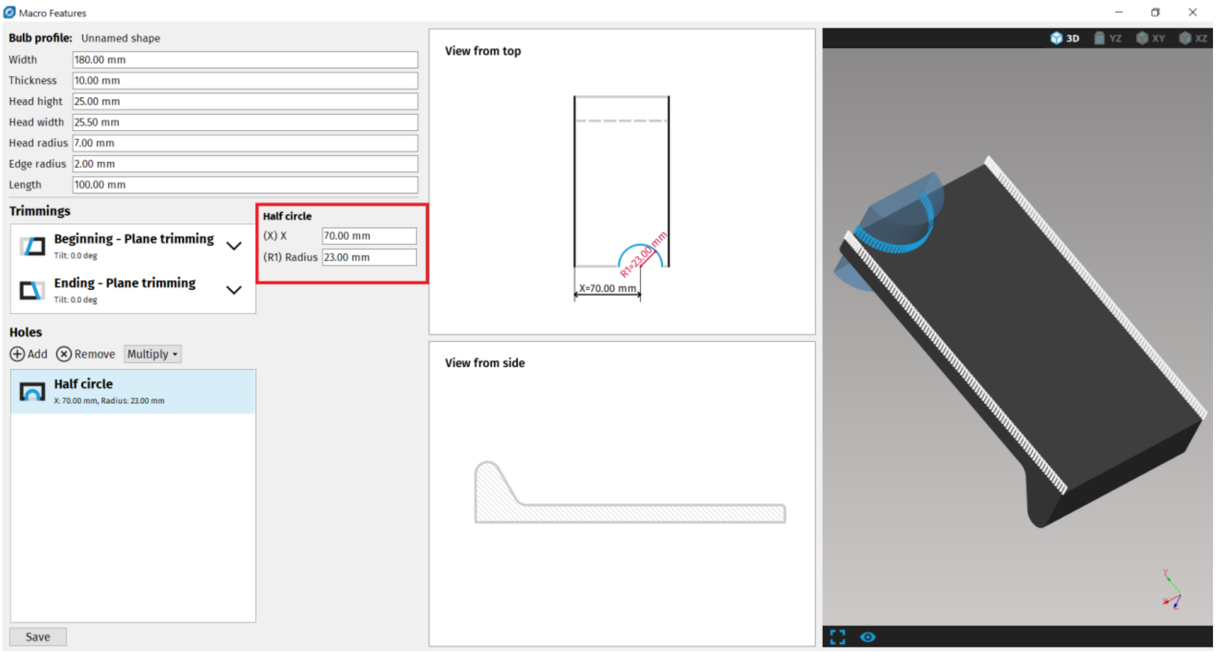


Fig. 103: Half circle macro for Bulb profile in Macro features

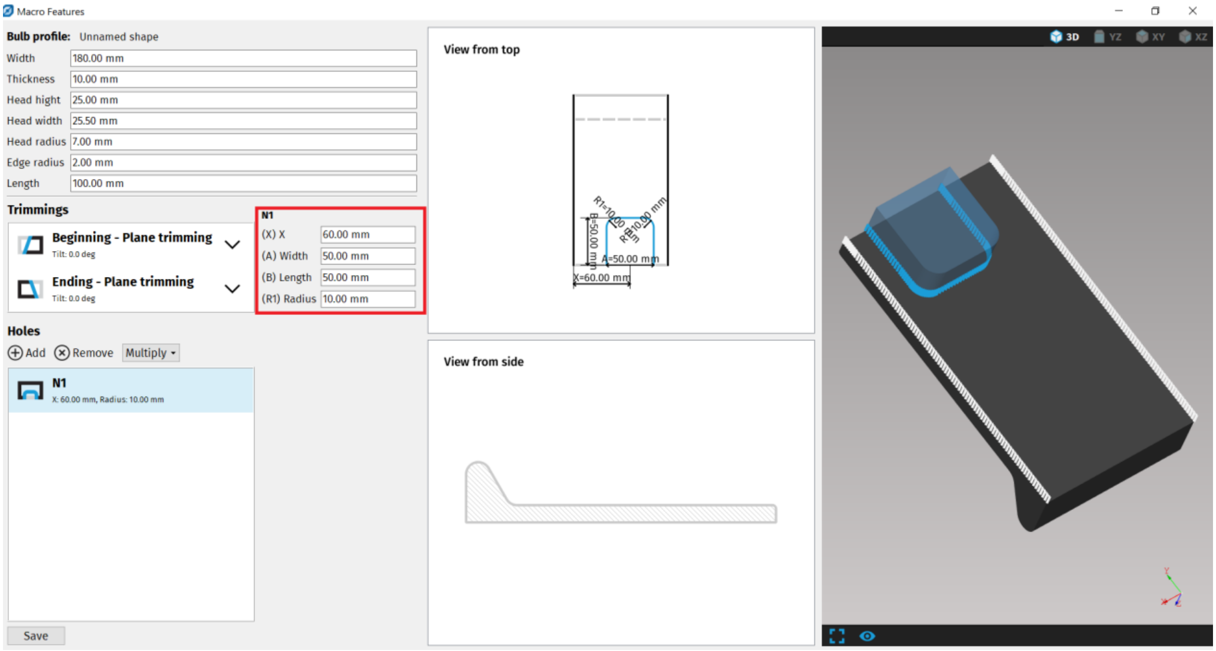


Fig. 104: N1 macro for Bulb profile in Macro features

Macros available for H beam

- **Web circular hole –** is a macro of a circular shape, with properties such as diameter, distance and offset that can be changed.
- **Flange circular hole –** is a macro of a circular shape, with properties such as diameter, distance, offset and side (Positive Z/Negative Z) where the hole can be added

Trimmings

- **Beginning/Ending - Corner cut –** this type of macro cuts off selected corners of the shape. Parameters for this macro are Z+ bevel and Z- bevel.
- **Beginning/Ending – Tongue –** this type of macro trims away a part of the shape so it can be fitted into another and welded together. Parameters that can be changed are X Offset (Z+), Top bevel (Z+), Nose height (Z+), Bottom bevel (Z+), Rathole type (Z+), X Offset (Z-), Top bevel (Z-), Nose height (Z-), Bottom bevel (Z-), Rathole type (Z-), Tongue length, Corner radius, Z Offset (Z-), Z Offset (Z+), Web top bevel.

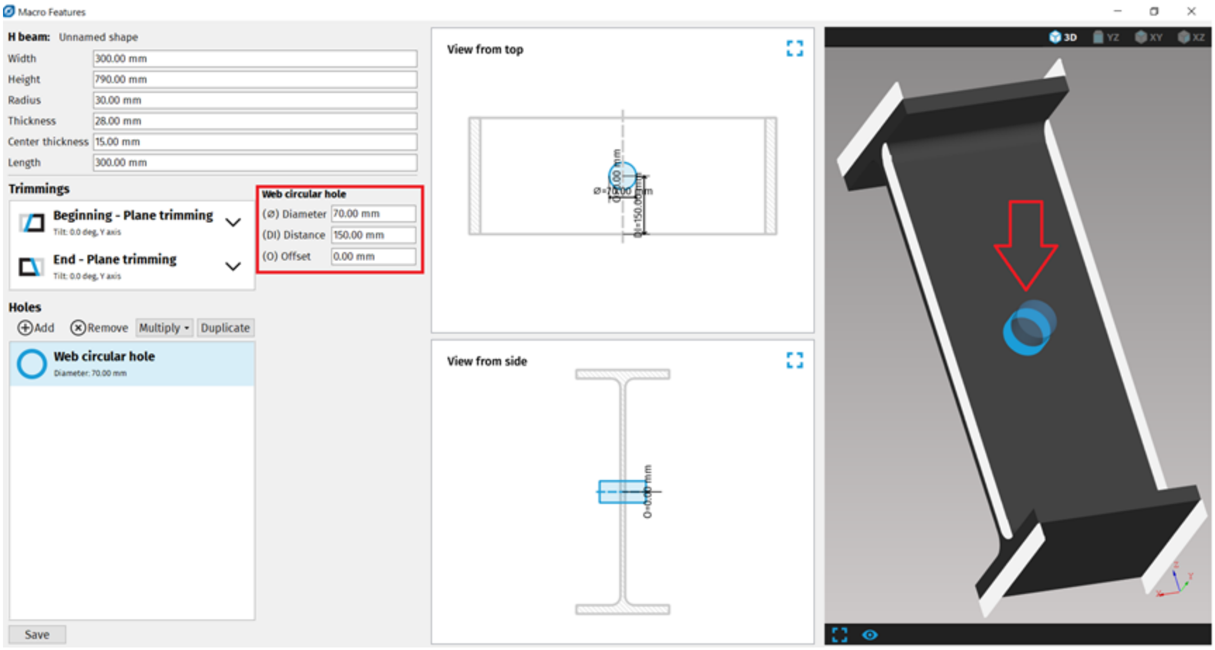


Fig. 105: Example of Web circular hole macro placed on H beam

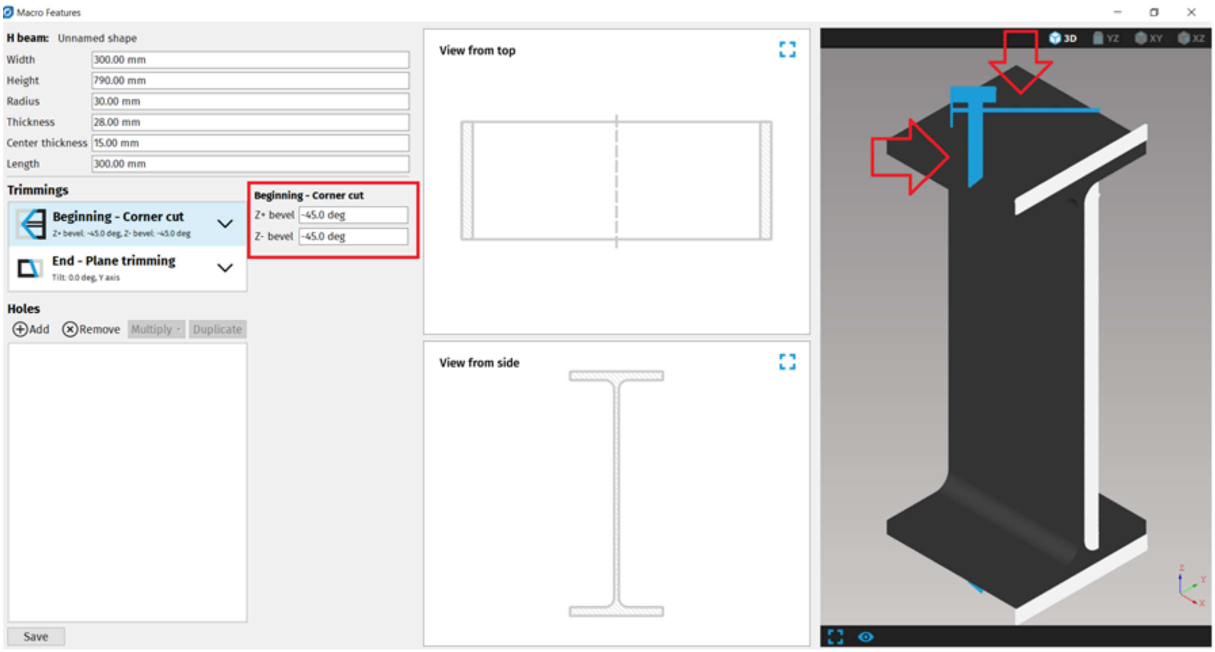


Fig. 106: Example of Beginning – Corner cut in Trimmings for H beam

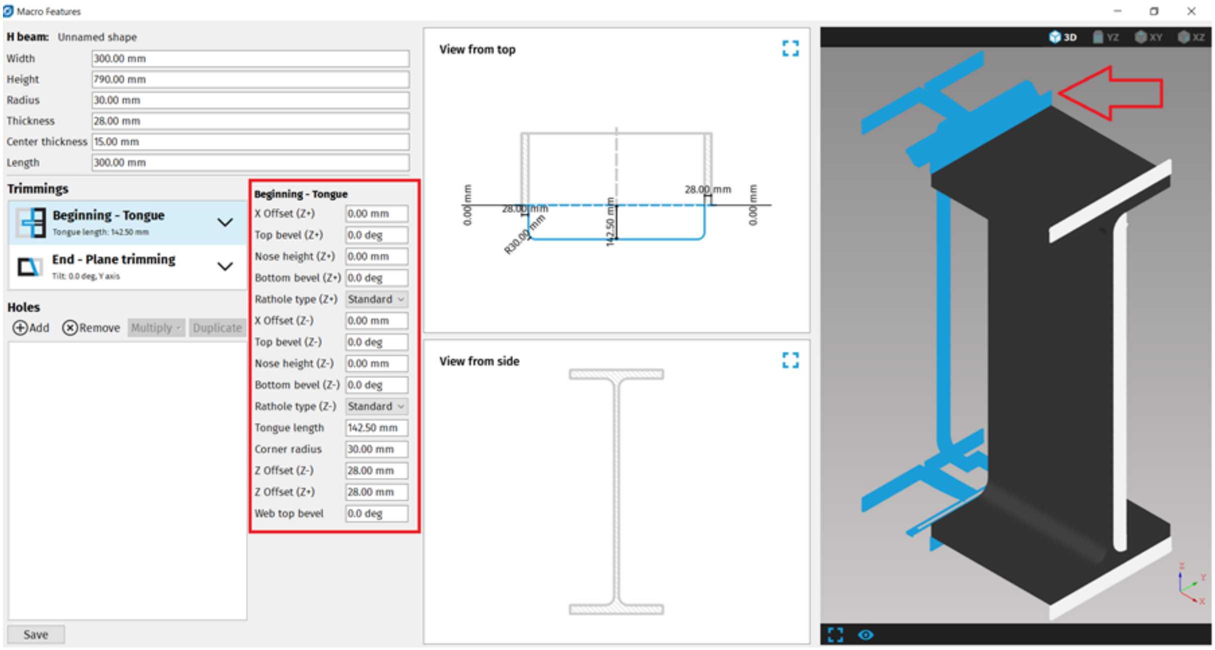


Fig. 107: Example of a Tongue in Trimmings for H beam

Macros available for I profile

- **Web circular hole –** is a macro of a circular shape, with properties such as diameter, distance and offset that can be changed.
- **Flange circular hole –** is a macro of a circular shape, with properties such as diameter, distance, offset and side (Positive Z/Negative Z) where the hole can be added.

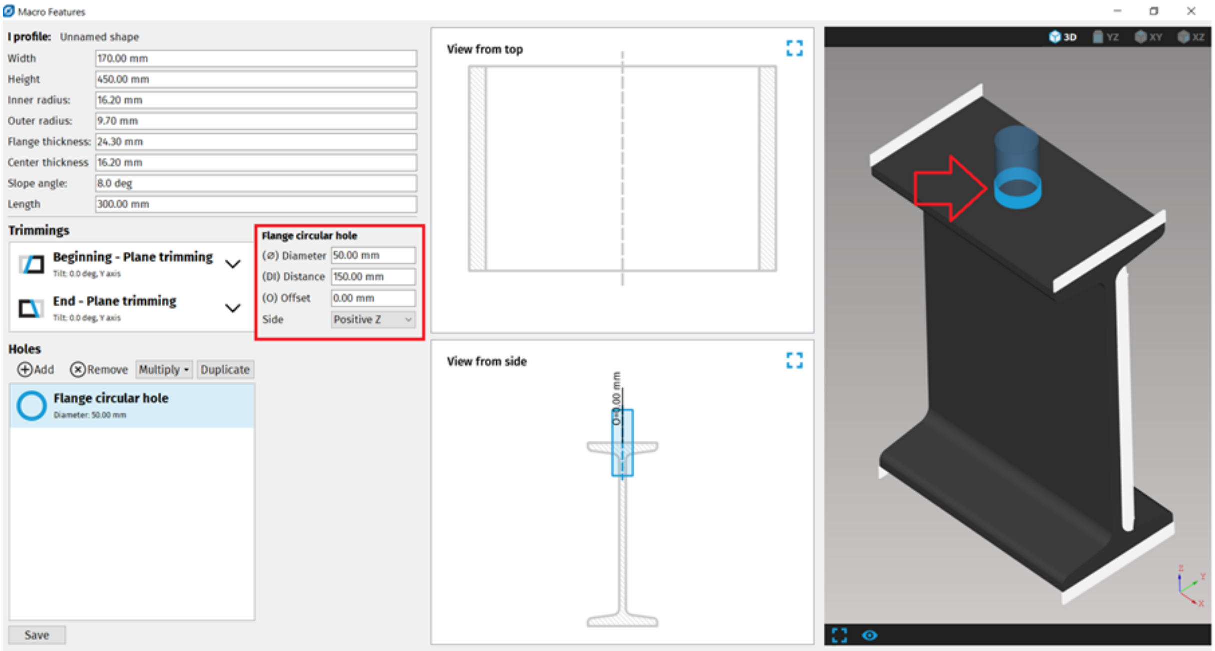


Fig. 108: Example of Flange circular hole macro placed on I beam

Macros available for Flat bottom dome

→ **Cylindrical hole** – is a macro of a circular shape, which properties such as diameter, distance, angle and tilt can be changed.

→ **Cross marking -** is a cross shaped type of marking, which properties can be changed. These are diameter, distance, angle and rotation.

→ **N – gon hole** – is a macro of polygon character where the number of sides is possible to adjust as needed. Other adjustable properties are diameter, distance, angle, rotation, tilt and an option to place the hole perpendicular to the surface. By checking the option for perpendicular hole, the hole is cut at right angle (90°) to the surface of the dome.

→ **Splitting in half** – is a macro that allows for the dome to be cut in half. The parameters are side offset, center offset, rotation. If the side offset and center offset are zero the dome is cut completely in half. Side offset determines how many millimeters from the edge should be left uncut. Center offset determines how many millimeters from the center should be left uncut.

- **Side cylindrical hole -** is a macro of cylindrical shape, with properties such as diameter, offset, angle, rotation, tilt and an option for the hole to be placed perpendicular to the dome.

- **Side cross marking -** is a cross shaped type of marking, with properties such as diameter, offset, angle and rotation. This type of marking is placed on the side of the shape.

- **Side line marking –** is a line marked on the side of the shape, with properties such as length, offset, angle and rotation.

- **Cut around –** is a macro that allows the shape to be cut around its circumference. The parameter for this macro that can be changed is height.

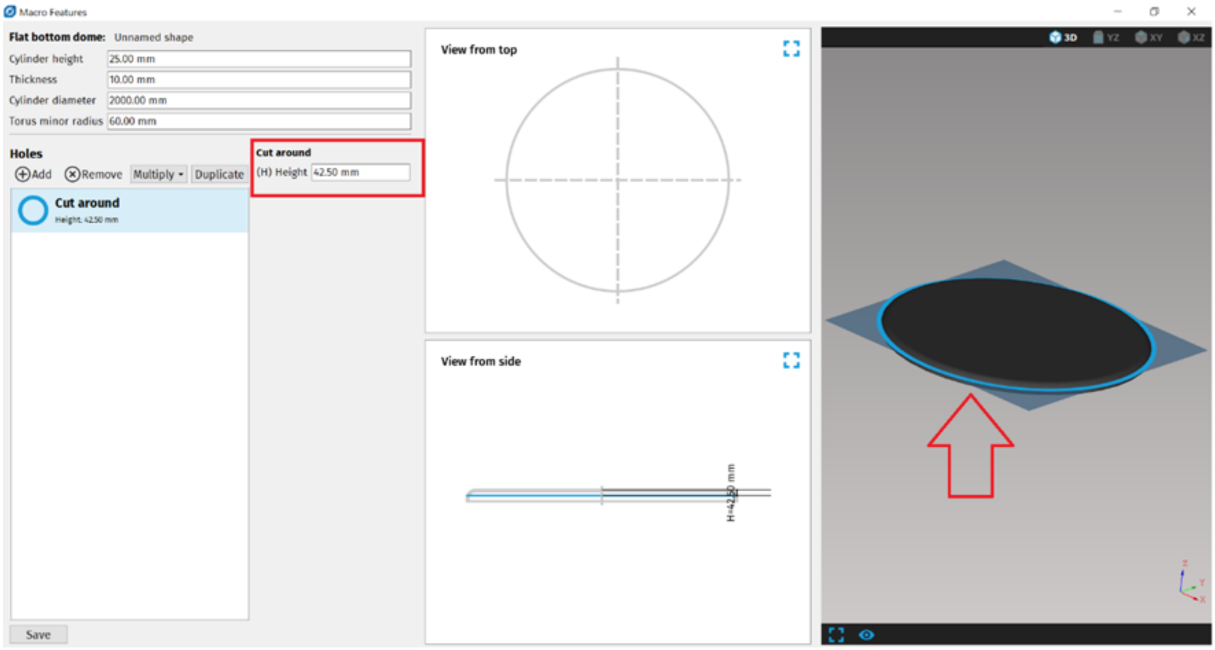


Fig. 109: Example of Cut around macro placed on Flat bottom dome

Macros available for Dished bottom dome

→ **Cylindrical hole** – is a macro of a circular shape, which properties such as diameter, distance, angle and tilt can be changed.

→ **Cross marking -** is a cross shaped type of marking, which properties can be changed. These are diameter, distance, angle and rotation.

→ **N – gon hole** – is a macro of polygon character where the number of sides is possible to adjust as needed. Other adjustable properties are diameter, distance, angle, rotation, tilt and an option to place the hole perpendicular to the surface. By checking the option for perpendicular hole, the hole is cut at right angle (90°) to the surface of the dome.

→ **Splitting in half** – is a macro that allows for the dome to be cut in half. The parameters are side offset, center offset, rotation. If the side offset and center offset are zero the dome is cut completely in half. Side offset determines how many millimeters from the edge should be left uncut. Center offset determines how many millimeters from the center should be left uncut.

- **Side cylindrical hole -** is a macro of cylindrical shape, with properties such as diameter, offset, angle, rotation, tilt and an option for the hole to be placed perpendicular to the dome.

- **Side cross marking -** is a cross shaped type of marking, with properties such as diameter, offset, angle and rotation. This type of marking is placed on the side of the shape.

- **Side line marking –** is a line marked on the side of the shape, with properties such as length, offset, angle and rotation.

- **Cut around –** is a macro that allows the shape to be cut around its circumference. The parameter for this macro that can be changed is height.

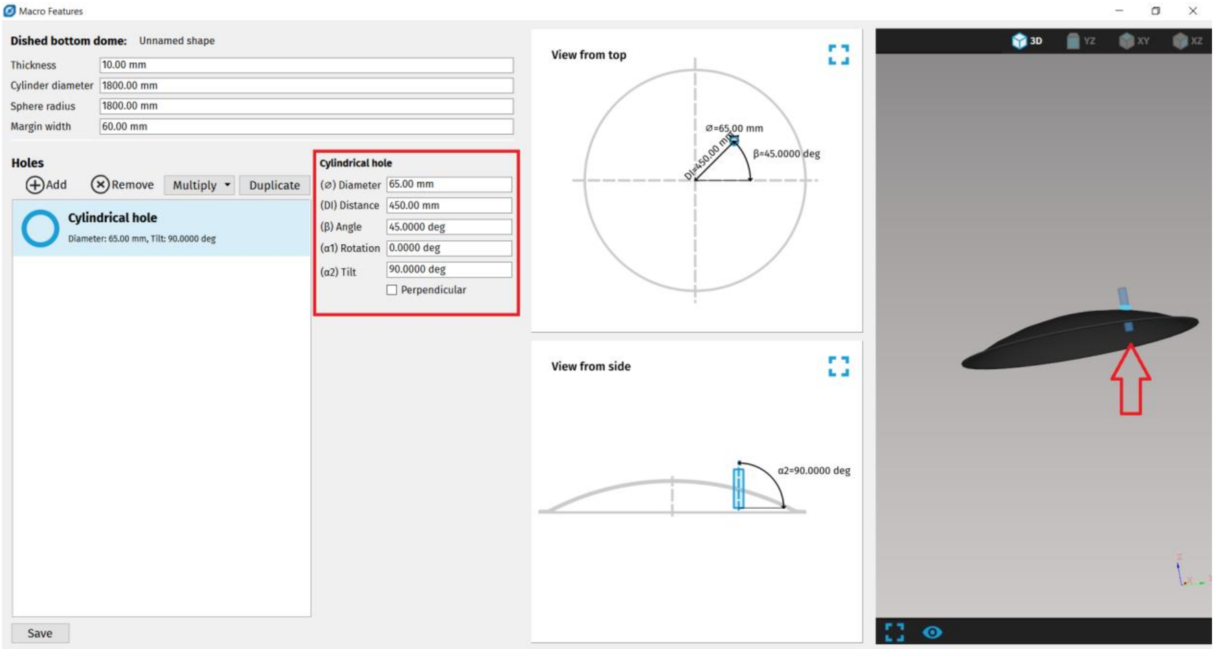


Fig. 110: Example of Cylindrical hole macro on Dished bottom dome

Macros available for Elliptical dome

→ **Cylindrical hole** – is a macro of a circular shape, which properties such as diameter, distance, angle and tilt can be changed.

→ **Cross marking -** is a cross shaped type of marking, which properties can be changed. These are diameter, distance, angle and rotation.

→ **N – gon hole** – is a macro of polygon character where the number of sides is possible to adjust as needed. Other adjustable properties are diameter, distance, angle, rotation, tilt and an option to place the hole perpendicular to the surface. By checking the option for perpendicular hole, the hole is cut at right angle (90°) to the surface of the dome.

→ **Splitting in half** – is a macro that allows for the dome to be cut in half. The parameters are side offset, center offset, rotation. If the side offset and center offset are zero the dome is cut completely in half. Side offset determines how many millimeters from the edge should be left uncut. Center offset determines how many millimeters from the center should be left uncut.

- **Side cylindrical hole -** is a macro of cylindrical shape, with properties such as diameter, offset, angle, rotation, tilt and an option for the hole to be placed perpendicular to the dome.

- **Side cross marking -** is a cross shaped type of marking, with properties such as diameter, offset, angle and rotation. This type of marking is placed on the side of the shape.

- **Side line marking –** is a line marked on the side of the shape, with properties such as length, offset, angle and rotation.

- **Cut around –** is a macro that allows the shape to be cut around its circumference. The parameter for this macro that can be changed is height.

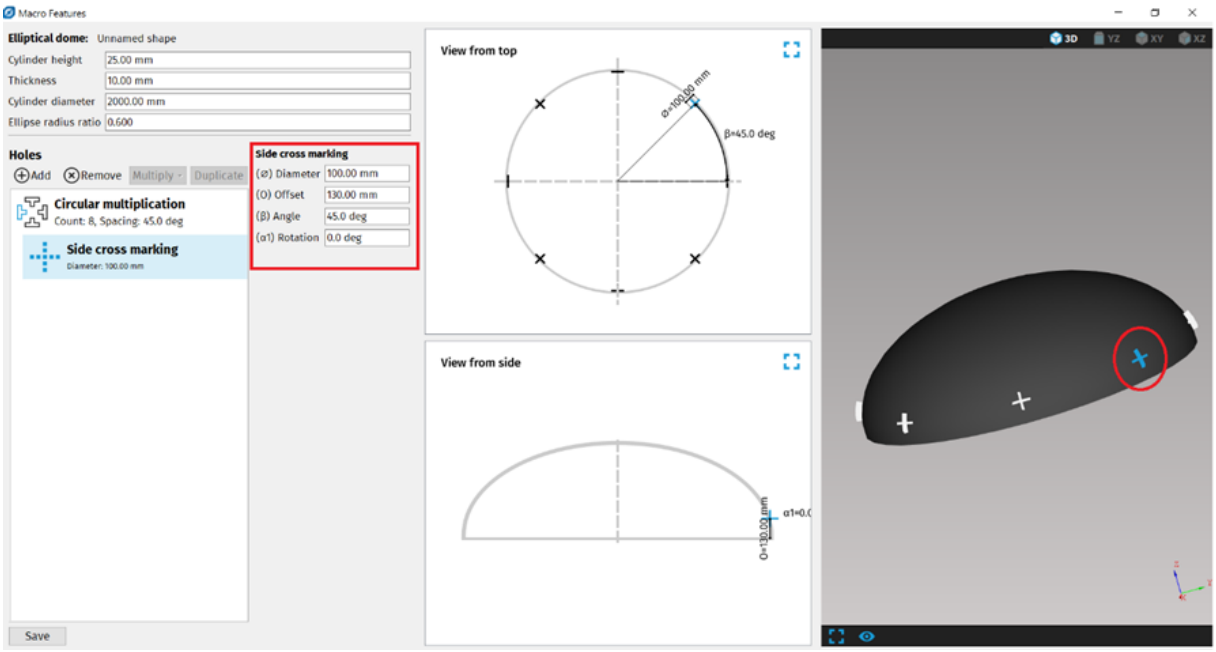


Fig. 111: Example of Side cross marking macro placed on Elliptical dome

Macros available for Conical dome

- **Cylindrical hole** – is a macro of a circular shape, which properties such as diameter, distance, angle and tilt can be changed.

- **Cross marking -** is a cross shaped type of marking, which properties can be changed. These are diameter, distance, angle and rotation.

- **N – gon hole** – is a macro of polygon character where the number of sides is possible to adjust as needed. Other adjustable properties are diameter, distance, angle, rotation, tilt and an option to place the hole perpendicular to the surface. By checking the option for perpendicular hole, the hole is cut at right angle (90°) to the surface of the dome.

Side cylindrical hole - is a macro of cylindrical shape, with properties such as diameter, offset, angle, rotation, tilt and an option for the hole to be placed perpendicular to the dome.

- **Side cross marking -** is a cross shaped type of marking, with properties such as diameter, offset, angle and rotation. This type of marking is placed on the side of the shape.
- **Side line marking –** is a line marked on the side of the shape, with properties such as length, offset, angle and rotation.
- **Cut around –** is a macro that allows the shape to be cut around its circumference. The parameter for this macro that can be changed is height.

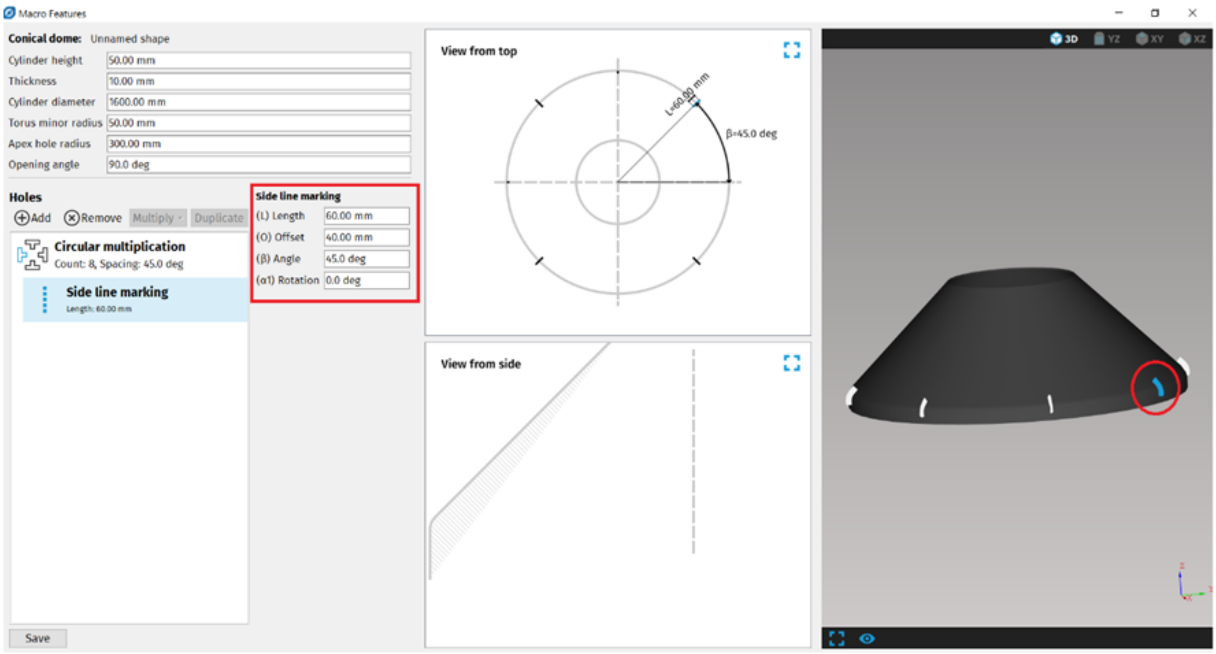


Fig. 112: Example of Side line marking macro placed on Conical dome

Macros available for Inverted head dome

→ **Cylindrical hole** – is a macro of a circular shape, which properties such as diameter, distance, angle and tilt can be changed.

→ **Cross marking -** is a cross shaped type of marking, which properties can be changed. These are diameter, distance, angle and rotation.

→ **N – gon hole** – is a macro of polygon character where the number of sides is possible to adjust as needed. Other adjustable properties are diameter, distance, angle, rotation, tilt and an option to place the hole perpendicular to the surface. By checking the option for perpendicular hole, the hole is cut at right angle (90°) to the surface of the dome.

→ **Splitting in half** – is a macro that allows for the dome to be cut in half. The parameters are side offset, center offset, rotation. If the side offset and center offset are zero the dome is cut completely in half. Side offset determines how many millimeters from the edge should be left uncut. Center offset determines how many millimeters from the center should be left uncut.

- **Side cylindrical hole -** is a macro of cylindrical shape, with properties such as diameter, offset, angle, rotation, tilt and an option for the hole to be placed perpendicular to the dome.

- **Side cross marking -** is a cross shaped type of marking, with properties such as diameter, offset, angle and rotation. This type of marking is placed on the side of the shape.

- **Side line marking –** is a line marked on the side of the shape, with properties such as length, offset, angle and rotation.

- **Cut around –** is a macro that allows the shape to be cut around its circumference. The parameter for this macro that can be changed is height.

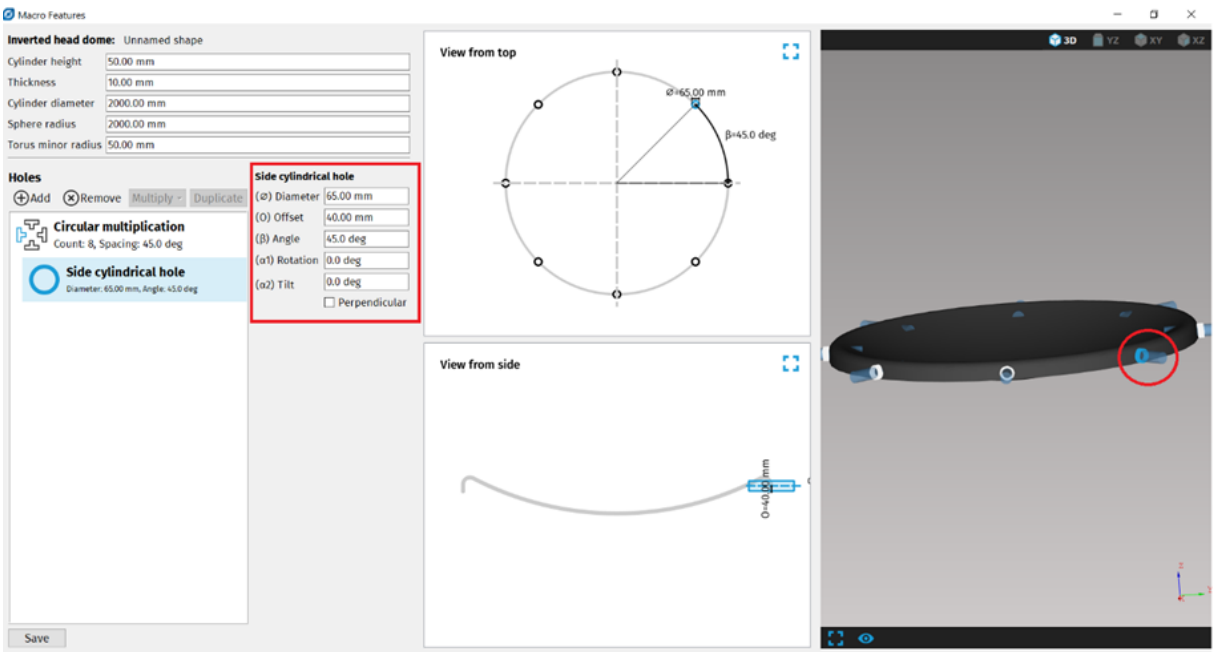


Fig. 113: Example of Side cylindrical hole macro placed on Inverted head dome

Macros available for Plane shape

- **Cylindrical hole –** macro of a circular shape with specific parameters. Diameter, angle, offset position and tilt can be modified.

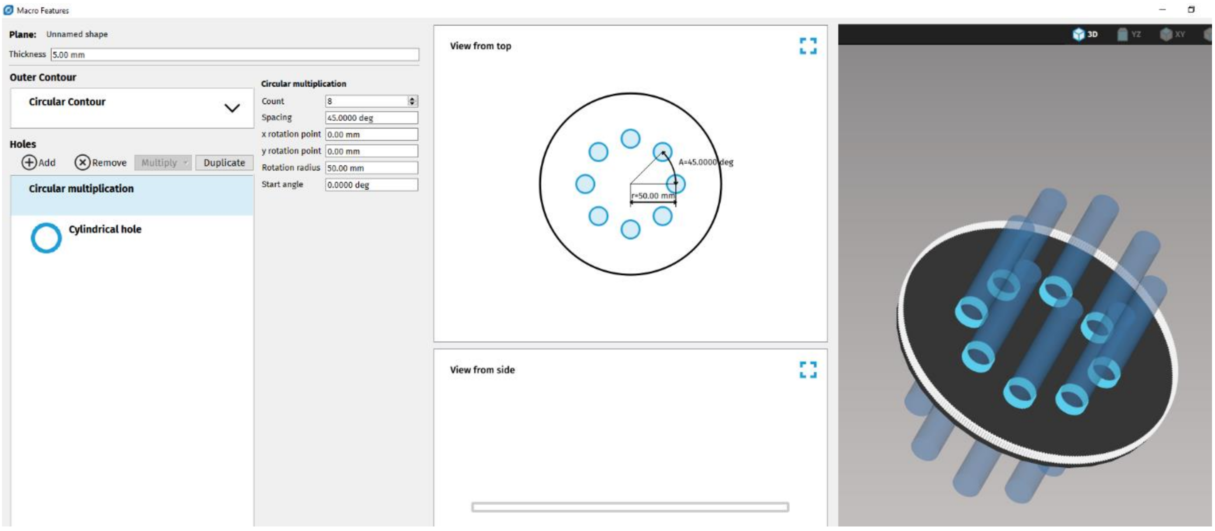


Fig. 114: Example of cylindrical hole macro placed on circular plane profile

Macros available for Dished disc

- **Cylindrical hole –** is a macro of a circular shape, which properties such as diameter, distance, angle and tilt can be changed.
- **Cross marking -** is a cross shaped type of marking, which properties can be changed. These are diameter, distance, angle and rotation.
- **N - Gon hole -** is a macro of polygon character where the number of sides is possible to adjust as needed. Other adjustable properties are diameter, distance, angle, rotation, tilt and an option to place the hole perpendicular to the surface. By checking the option for perpendicular hole, the hole is cut at right angle (90°) to the surface.
- **Splitting in half -** is a macro that allows cutting in half. The parameters are side offset, center offset, rotation. If the side offset and center offset are zero the disc will be cut in half. Side offset determines how many millimeters from the edge should be left uncut. Center offset determines how many millimeters from the center should be left uncut.
- **Side cylindrical hole -** is a macro of cylindrical shape, with properties such as diameter, offset, angle, rotation, tilt and an option for the hole to be placed perpendicular to the disc.

- **Side cross marking -** is a cross shaped type of marking, with properties such as diameter, offset, angle and rotation. This type of marking is placed on the side of the disc.
- **Side line marking -** is a line marked on the side of the shape, with properties such as length, offset, angle and rotation.
- **Cut around -** is a macro that allows the shape to be cut around its circumference. The parameter for this macro that can be changed is height.

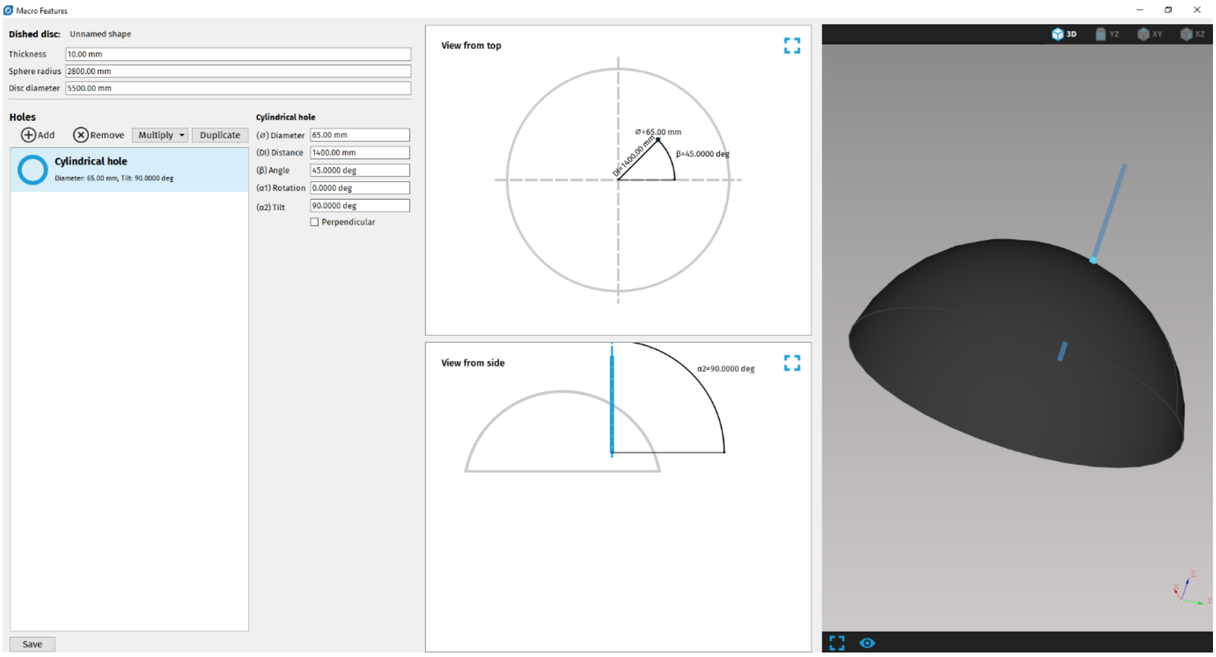


Fig. 115: Example of cylindrical hole macro placed on dished disc dome

To change properties of a macro, first you have to select the desired macro. You can do so by selecting it in the List of macro features (A2), by clicking on the desired macro on the 2D preview screen (B1) or by clicking on it on the 3D preview screen (C). After selecting the macro, you can choose the property you want to change by clicking on it in the Property inspector (A3) or by clicking on the property on the 2D preview screen (B1).

If you see this message "Error: Invalid macro, please change the parameters" under the macro´s properties, it means the dimensions you put in for the macro are incorrect (e.g.: the macro would protrude outside the shape´s edges). In this case you have to change the macro´s parameters.

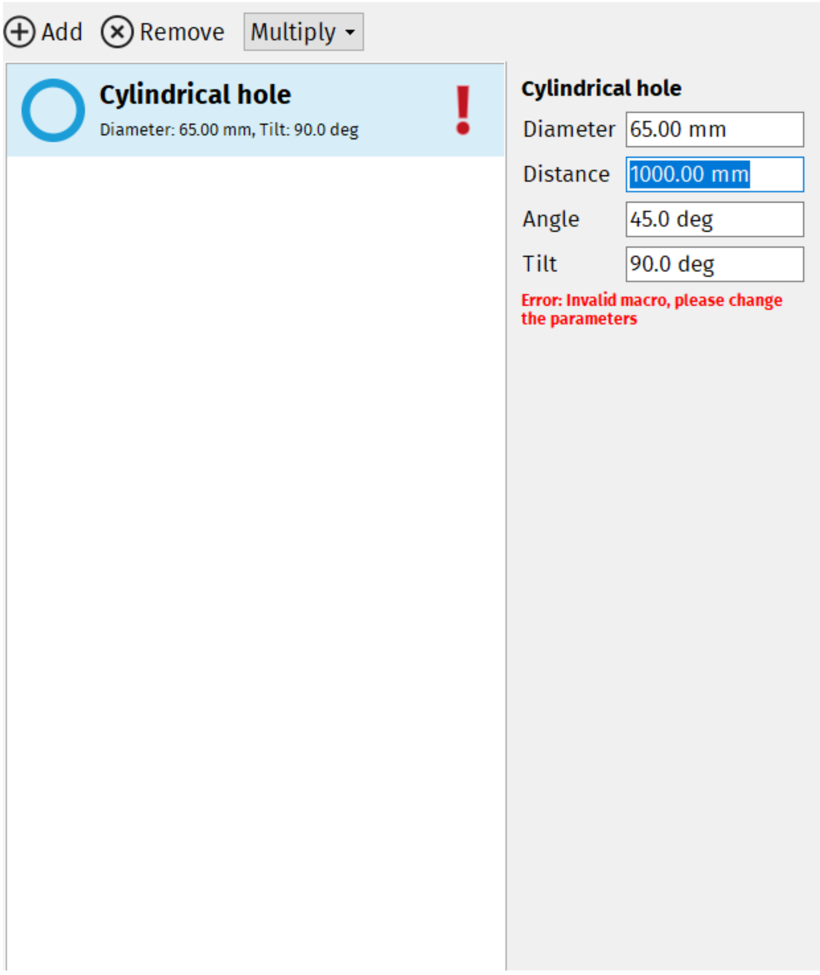


Fig. 116: Invalid macro warning

Adding multiplications

Multiplication is used when you need to add the same macro several times. To use multiplication option first choose a macro you wish to multiply by clicking on it on one of the screens or in the List of Macro features (A2). After choosing a macro, click on the Multiply button on the left screen (A2) and select desired multiplication.

→ **Circular multiplication** - the number of multiplied macros can be changed and the spacing between two macros can be chosen too, in the Property Inspector after clicking on a circular multiplication or by clicking on a macro in the 2D preview screen (B1).

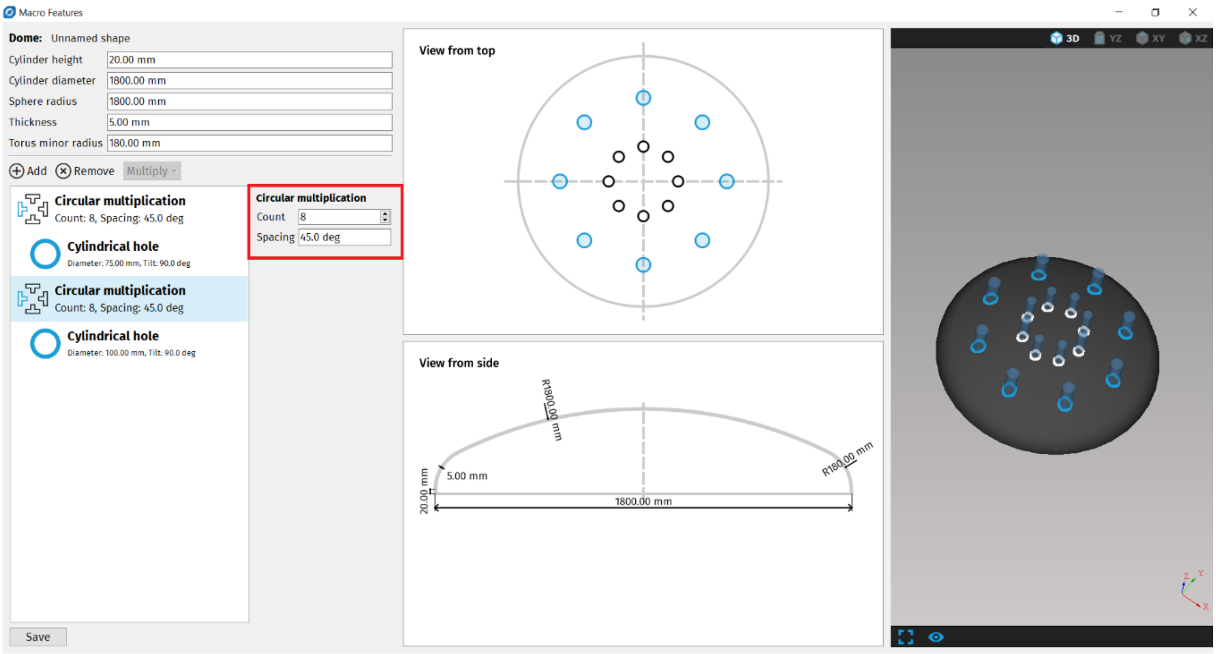


Fig.117: Circular multiplication

Removing features

Remove button will remove a previously selected macro or a multiplication by either choosing it on 2D preview screen (B1) or in the List of Macro features (A2). To remove a desired macro, you can also click a chosen macro/multiplication on the 2D preview (B1)/ in the List of Macro features (A2) and use a delete key on your keyboard.

Multiplications can be removed either by clicking on a macro on the 2D preview screen (B1) or on the Circular multiplication in the List of Macro features (A2) and either clicking the remove button or pressing the delete key.

Saving shapes

The created shape can be Saved by clicking the button on the bottom of the List of Macro features screen (A2). The mCAM also automatically saves the shape when you press X to leave. This shape, containing the cutting paths, can be further edited afterwards if you wish so.

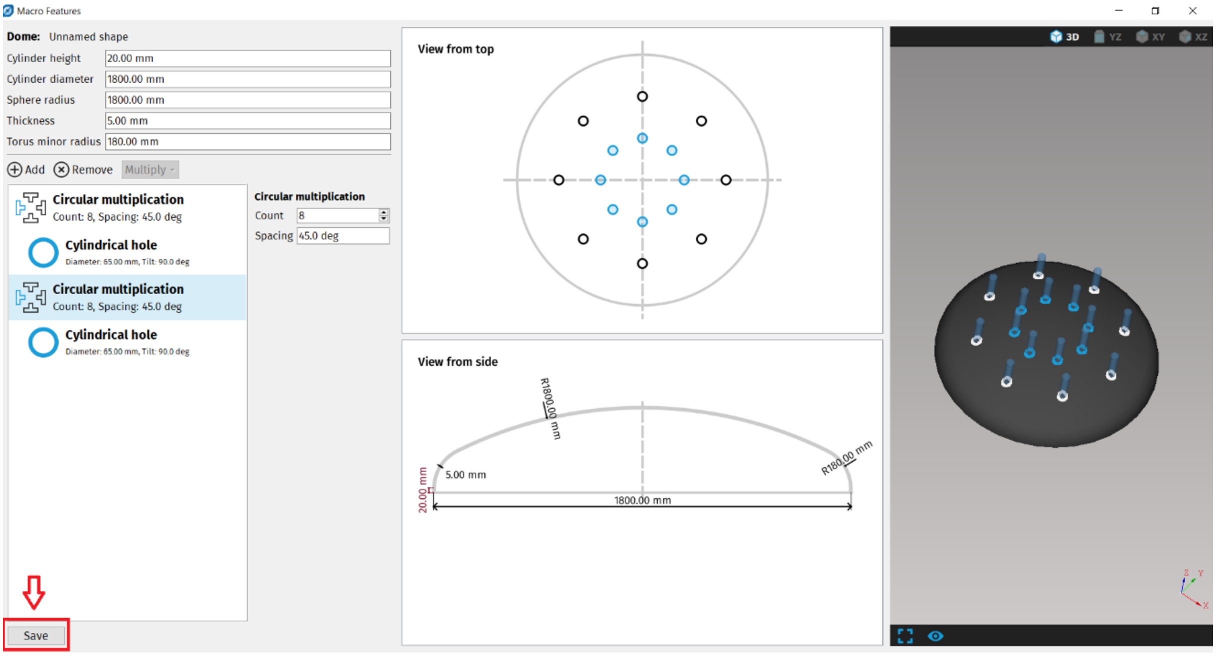


Fig. 118: Save button and the final shape

The final shape is imported into mCAM as a default shape and can be used and edited like the other imported shapes. Its cutting paths can be edited, lead ins/outs can be changed, text markings can be added, the preparation for welding can be added and so on.

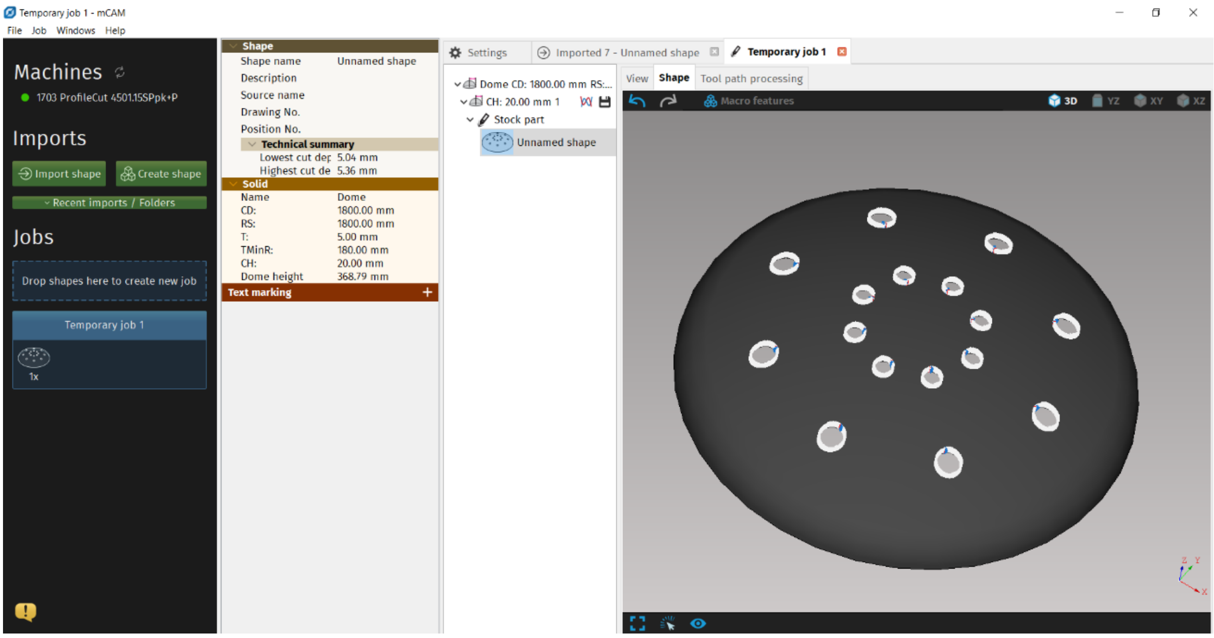


Fig. 119: Final shape used in a job

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.

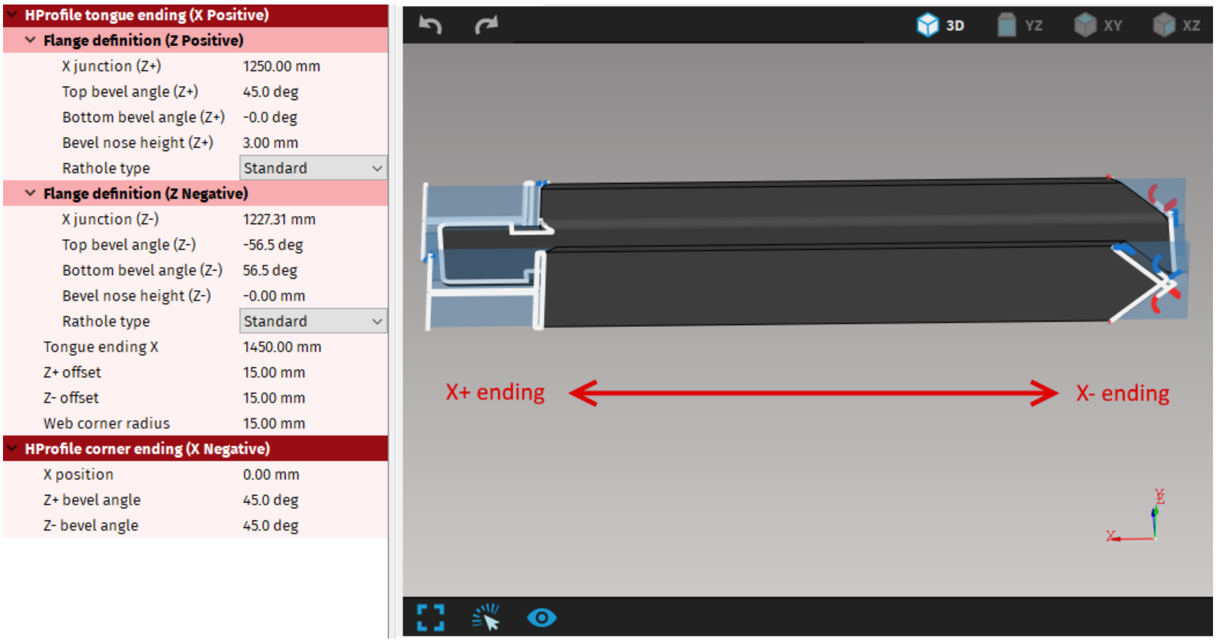


Fig. 120: Each part can contain maximum of two macros – One in X+ and one in X- direction

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.
- *Top bevel angle* – defines shape of the respective flange
- *Bottom bevel angle* – defines shape of the respective flange
- *Bevel nose height* – defines shape of the respective flange
- *Rathole type* – None, standard, None with grinding

→ *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

→ *Z- offset* – offset of web from outer edge of Z- flange

→ *Web corner radius* – rounding of the corners on tongue ending

If Z offset is bigger than the cutting tool 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.

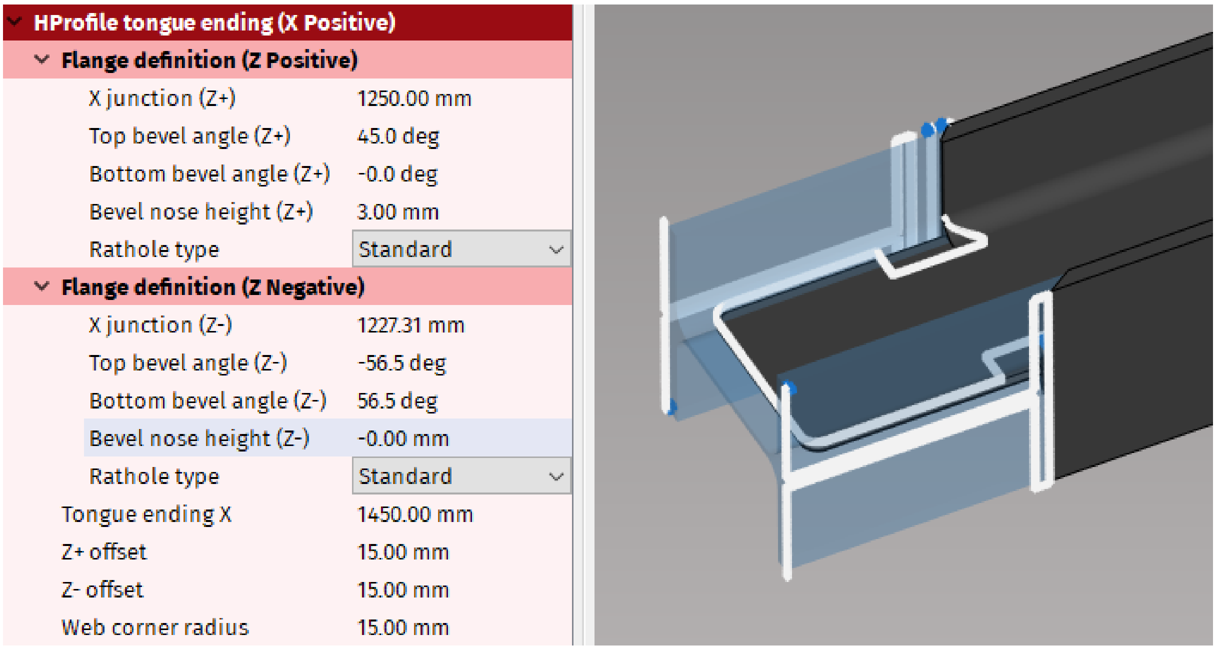


Fig. 121: Tongue ending and its parameters in Properties area

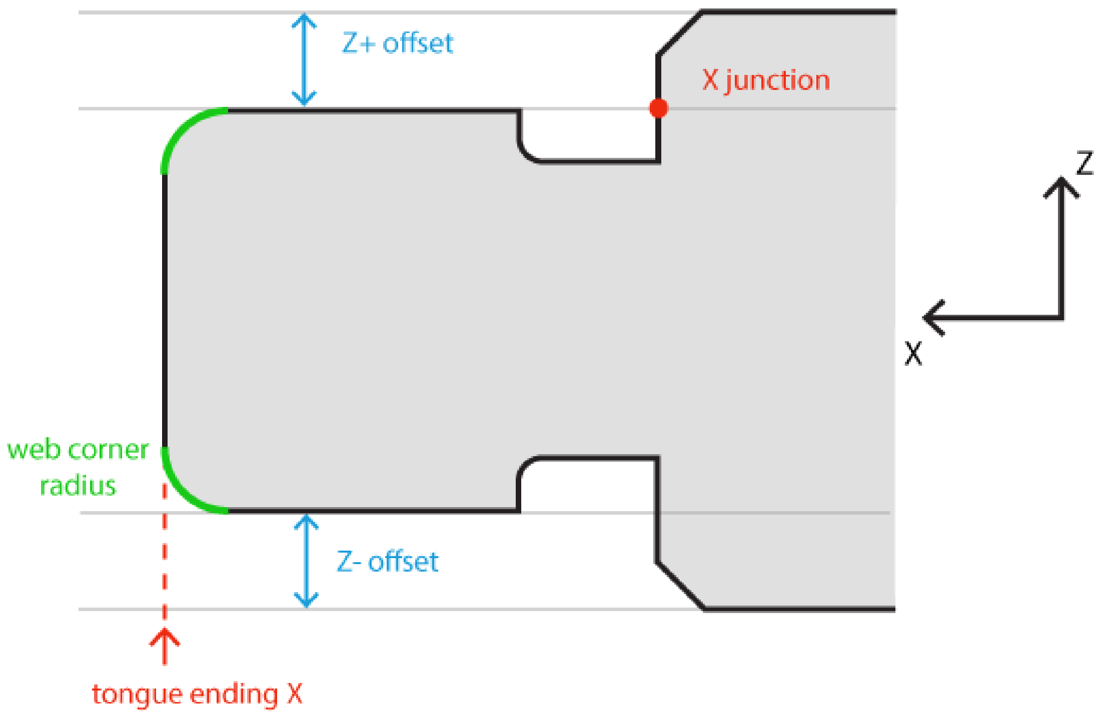


Fig. 122: Parameters for Tongue ending welding connection

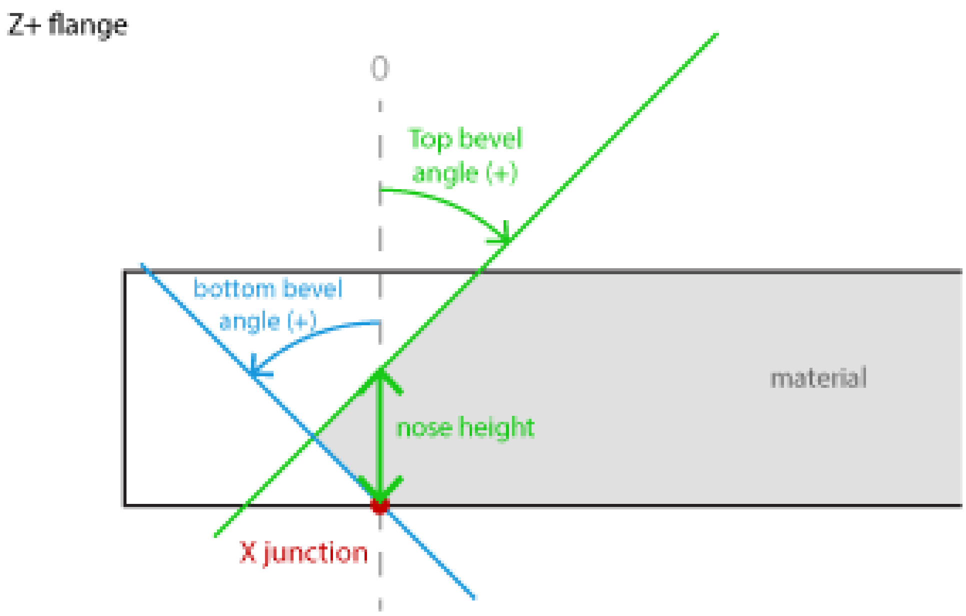


Fig. 123: Z+ flange with bevel parameters

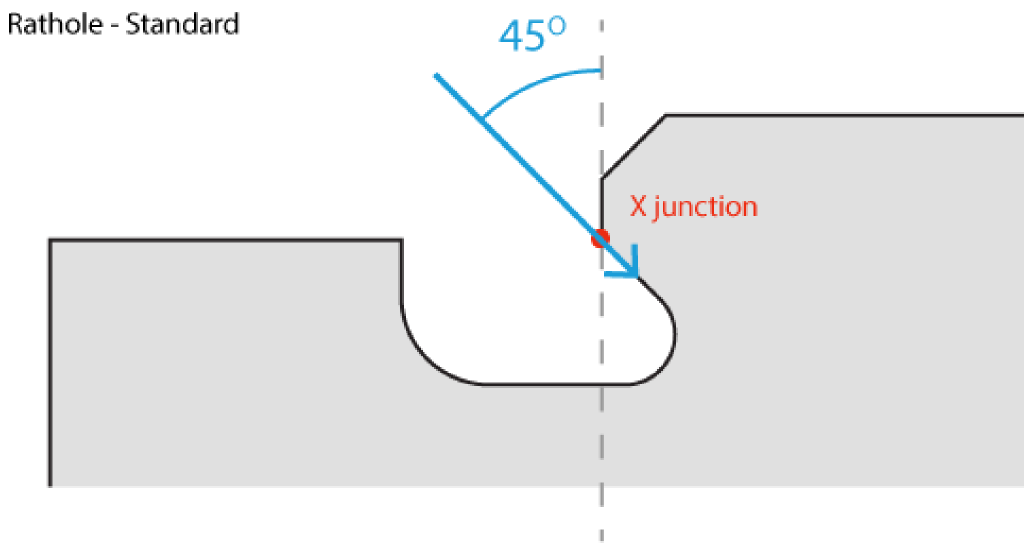


Fig. 124: Rathole Standard is cut in 45 degrees angle

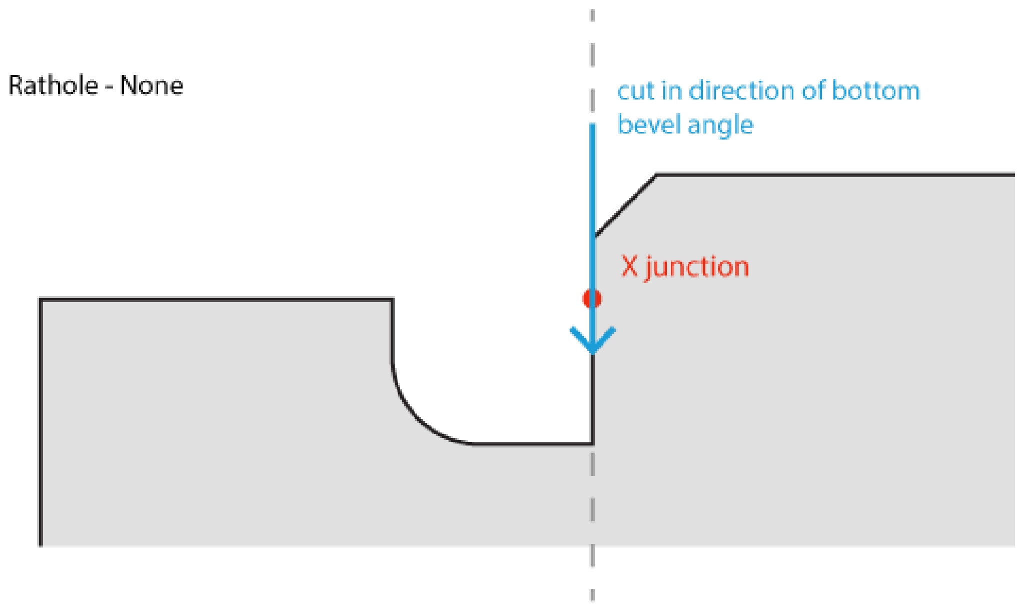


Fig. 125: Rathole None is cut in direction of bottom bevel angle

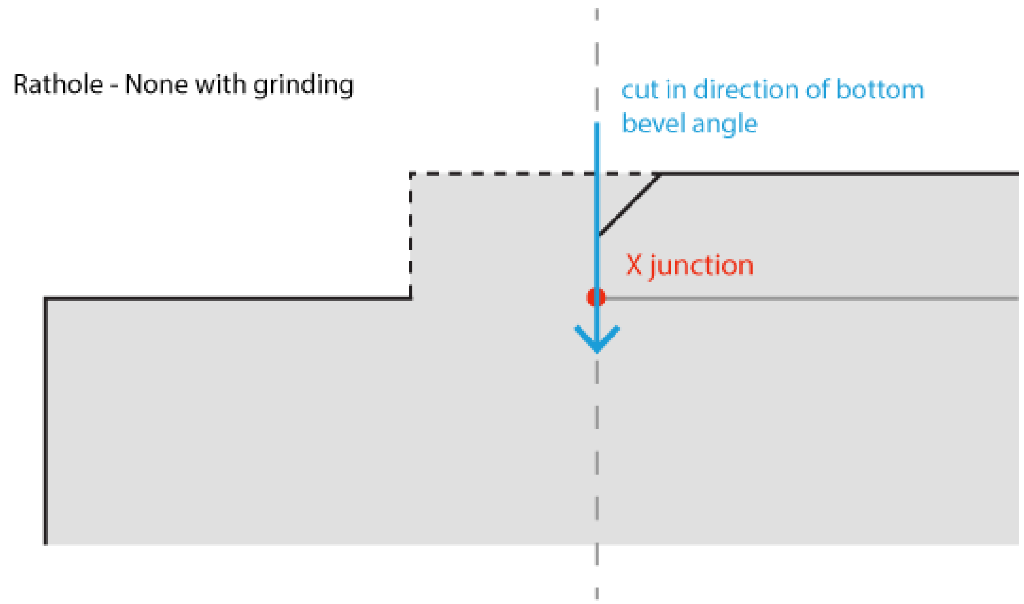


Fig. 126: Rathole - None with grinding

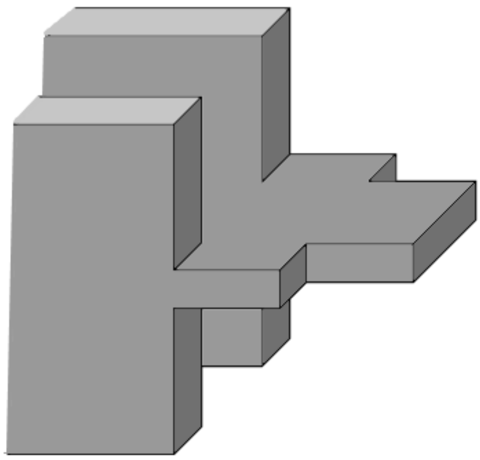


Fig. 127: Rathole - None with grinding

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

- *Rathole type* – None and standard

→ *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.

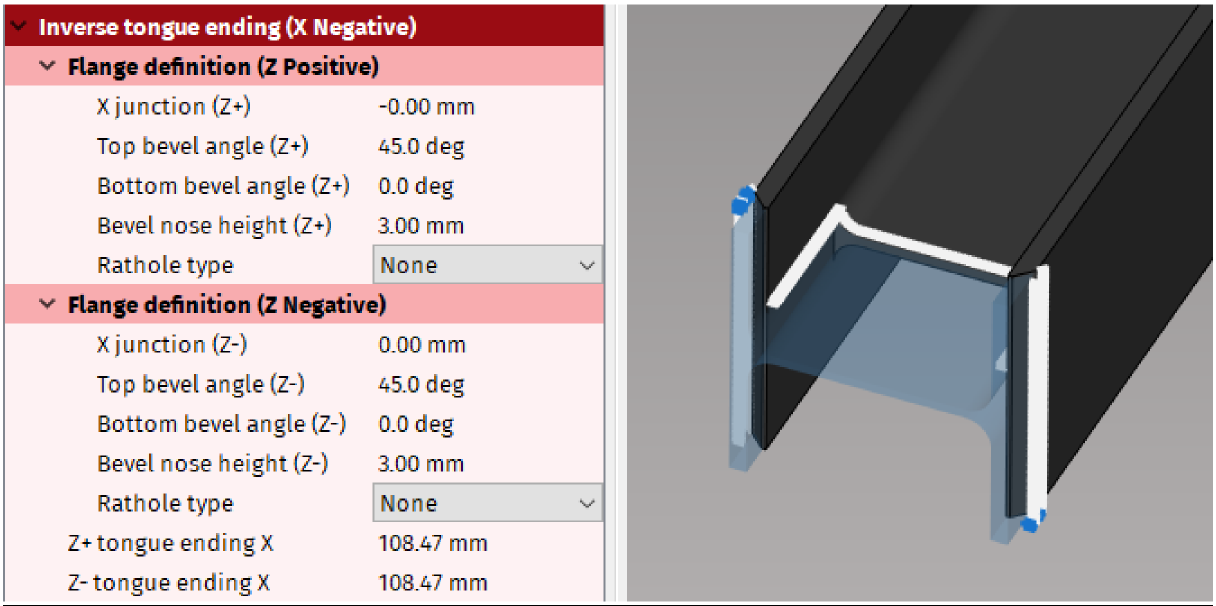


Fig. 128: Inverse tongue ending and its parameters in Properties area

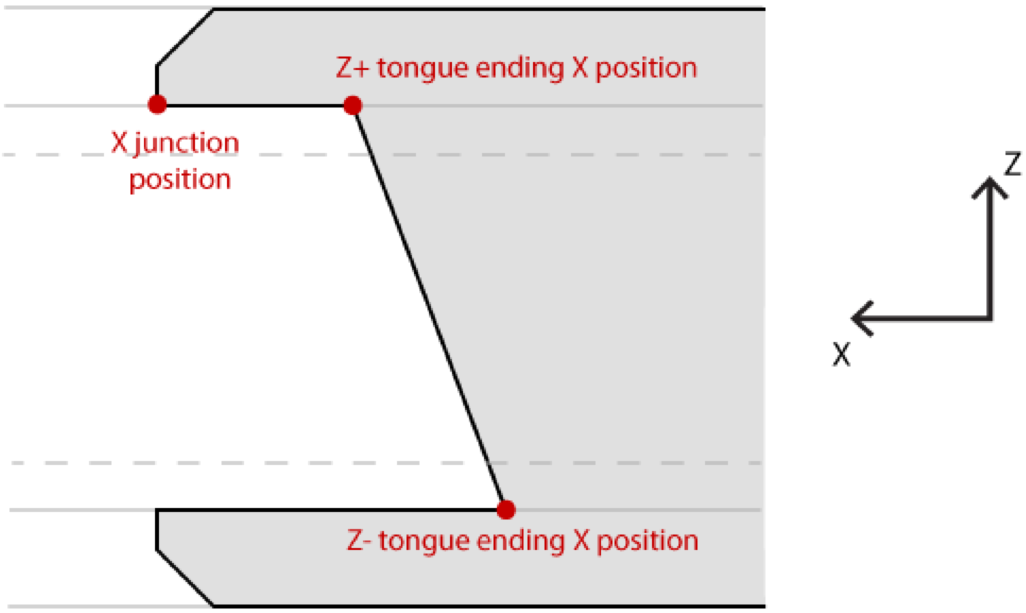


Fig. 129: Inverse tongue ending with parameters

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

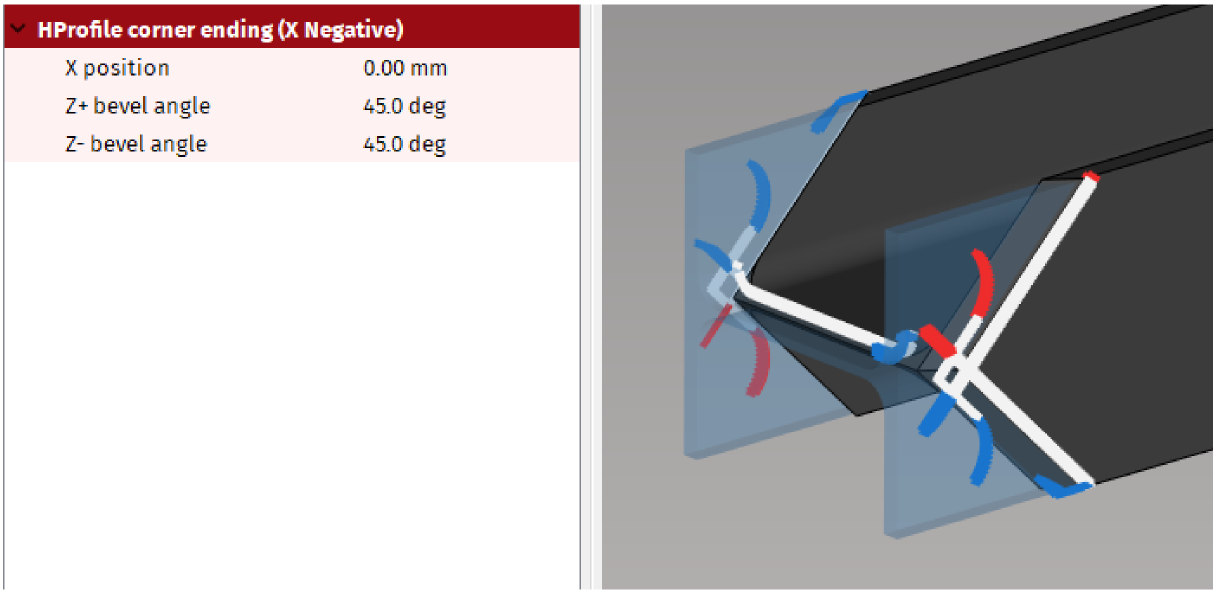


Fig. 130: Corner ending and its parameters in Properties area

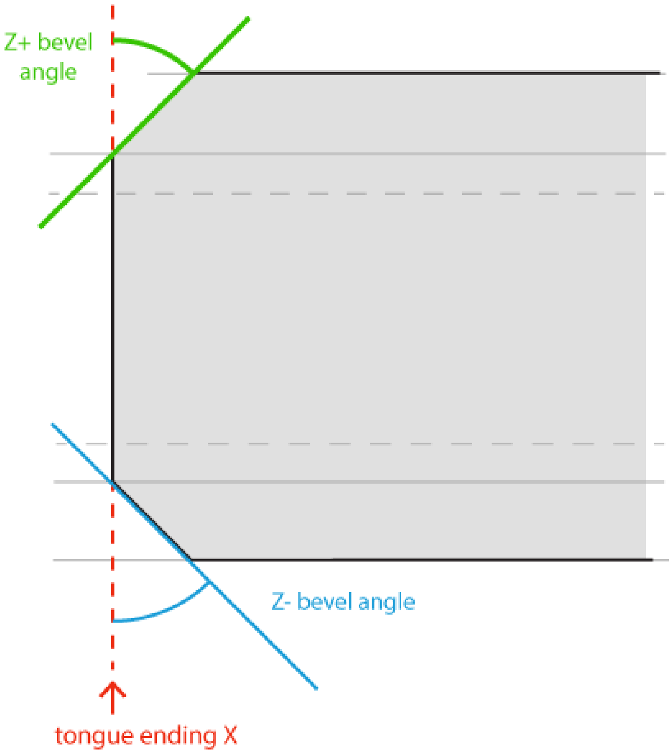


Fig. 131: Top view of corner ending welding connection with its parameters

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.

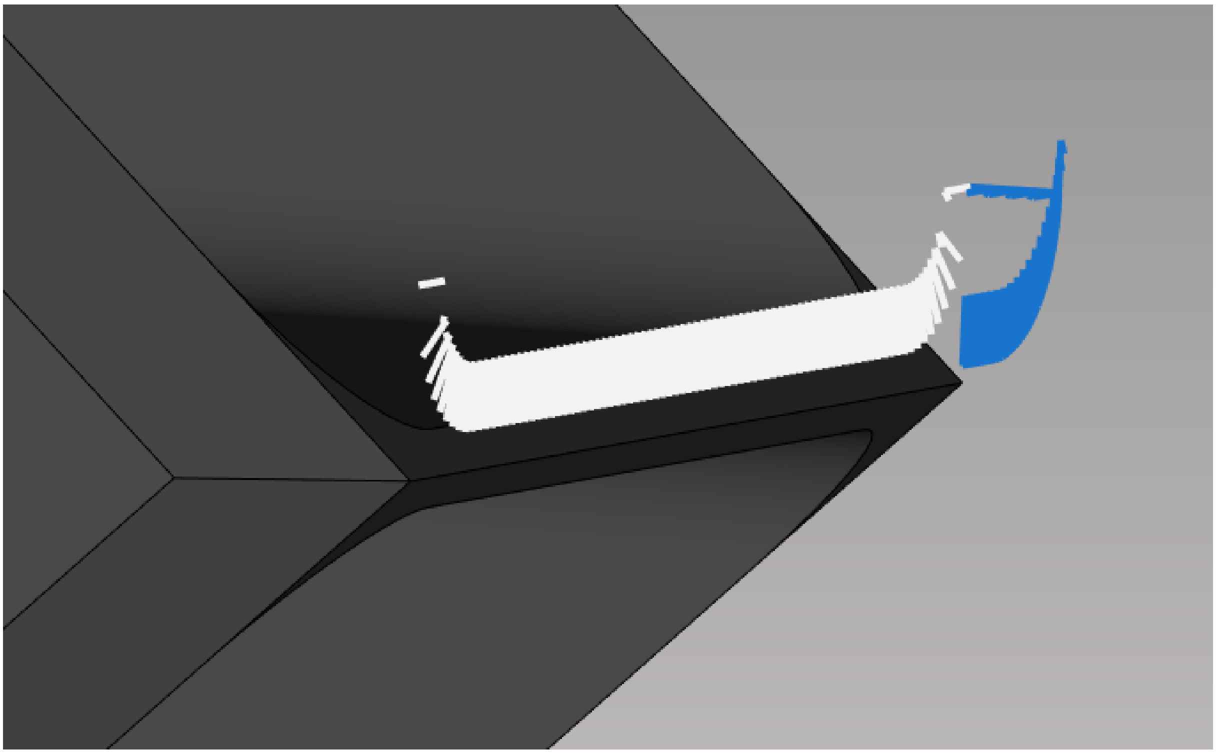


Fig. 132: Step file is designed with X cut on the web, but the final cut is straight

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.

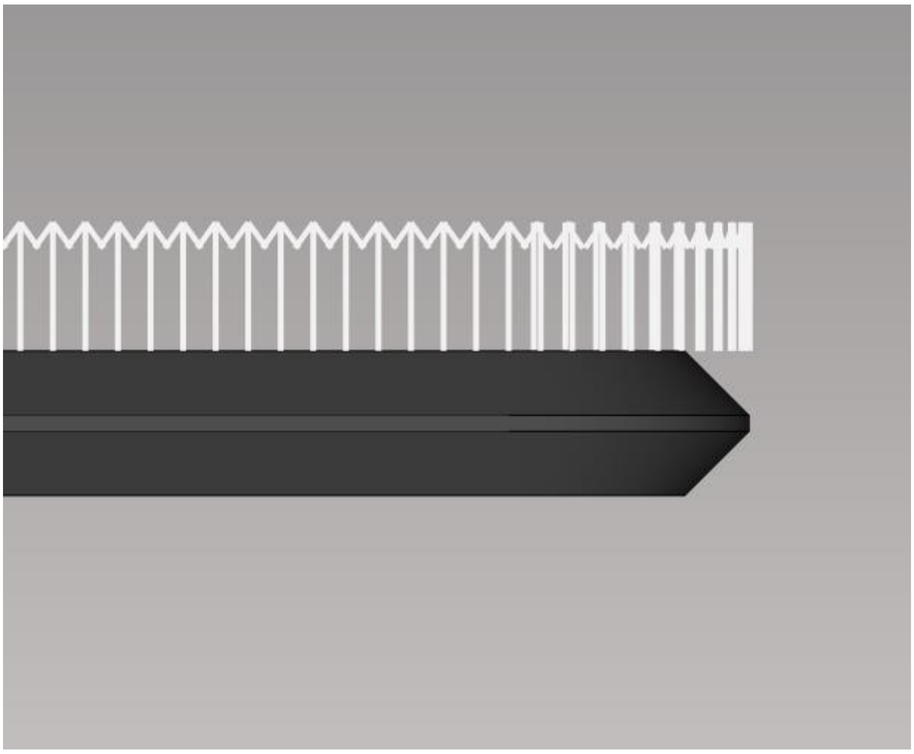


Fig. 133: Detail of the web on tongue ending with K-cut replaced with straight cut

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.

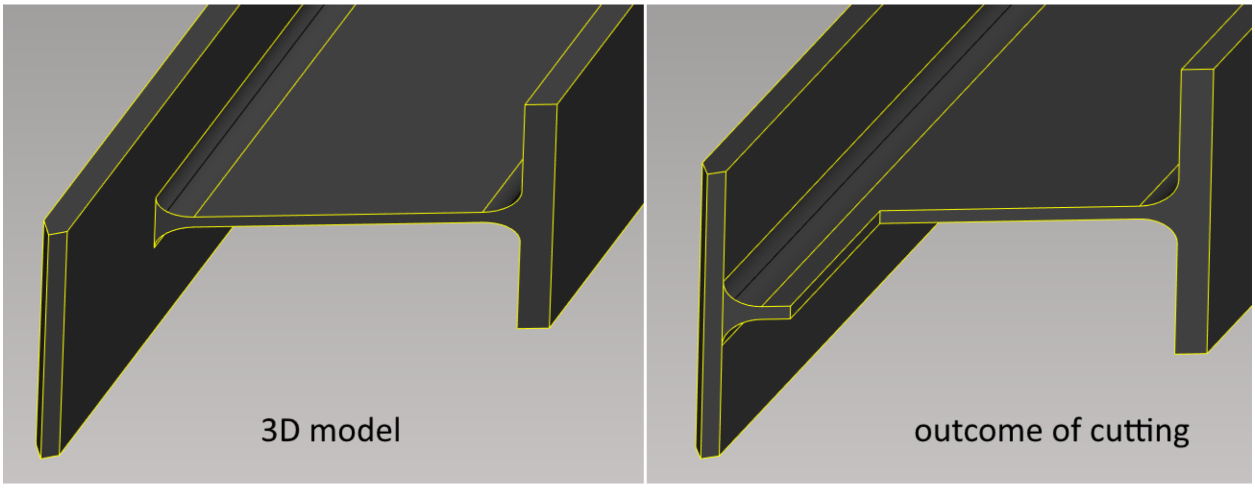


Fig. 134: Difference between the 3D model and the real outcome of cutting

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.

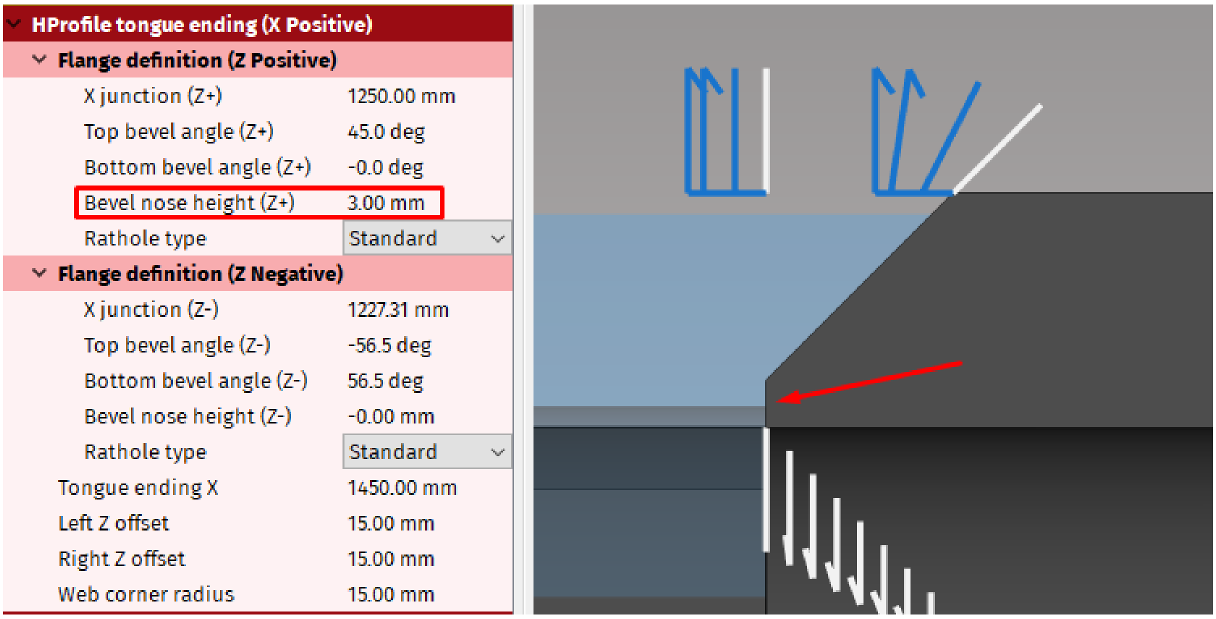


Fig. 135: It is recommended to consult the height of the nose with technology department

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

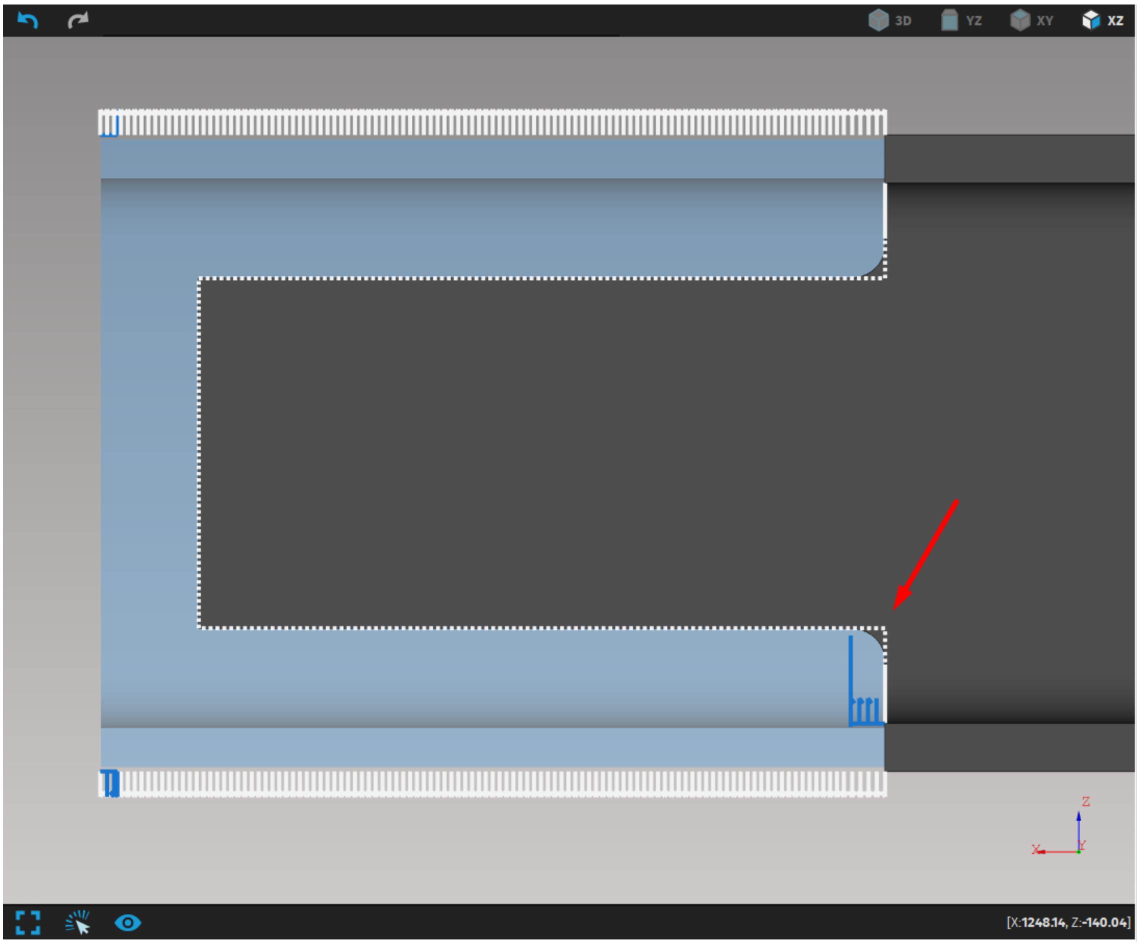


Fig. 136: The model should be drawn without the radiuses

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.

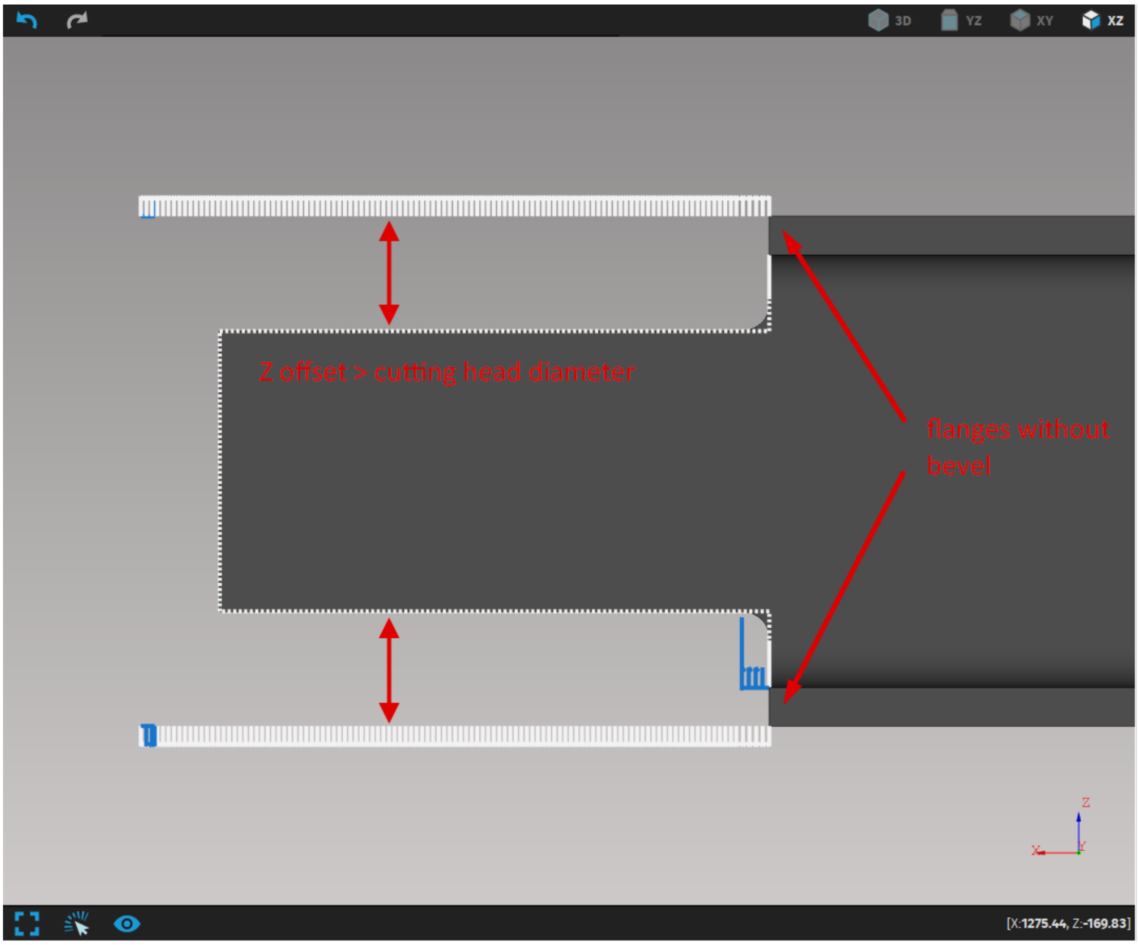


Fig. 137: Cutting the tongue without using the macros

Cutting path modifications

Cutting paths are displayed as a set of lines that represent the path of the tool during machining process. Every single line represents mutual position of the cutting head and the work piece. The top of each mark is oriented forward with respect to the direction of motion.

Lead–ins are displayed in blue colour and lead–outs in red colour. Selected cutting path is displayed in yellow colour. When selected, user can edit its features that are displayed in Properties area. Some features can be changed using cut path editing widget.

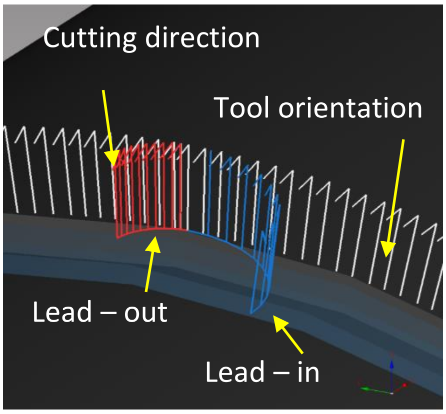


Fig. 138: Cutting path and its sections

Cut path editing includes these operations:

- splitting paths and loops creation
- lead-ins/outs modifying
- start point modifying (on cyclical paths)
- microjoints creation
- text marking
- enabling or disabling transformations (drilling/ punching/ quality hole cutting)
- marking of contours
- user ordering (priority cutting)
- assignment of cutting operation for each cutting path (Q1/Q2/Q3/Q4/Q5; Large/Medium/Small contours)
- alignments of cuts
- applying common–cut (during *nesting procedure*)

Right control panel

After selecting the cutting path, the panel pops out on the right side of the screen. This cut path editing widget contains several buttons with various functions for changing some attributes of cut paths.

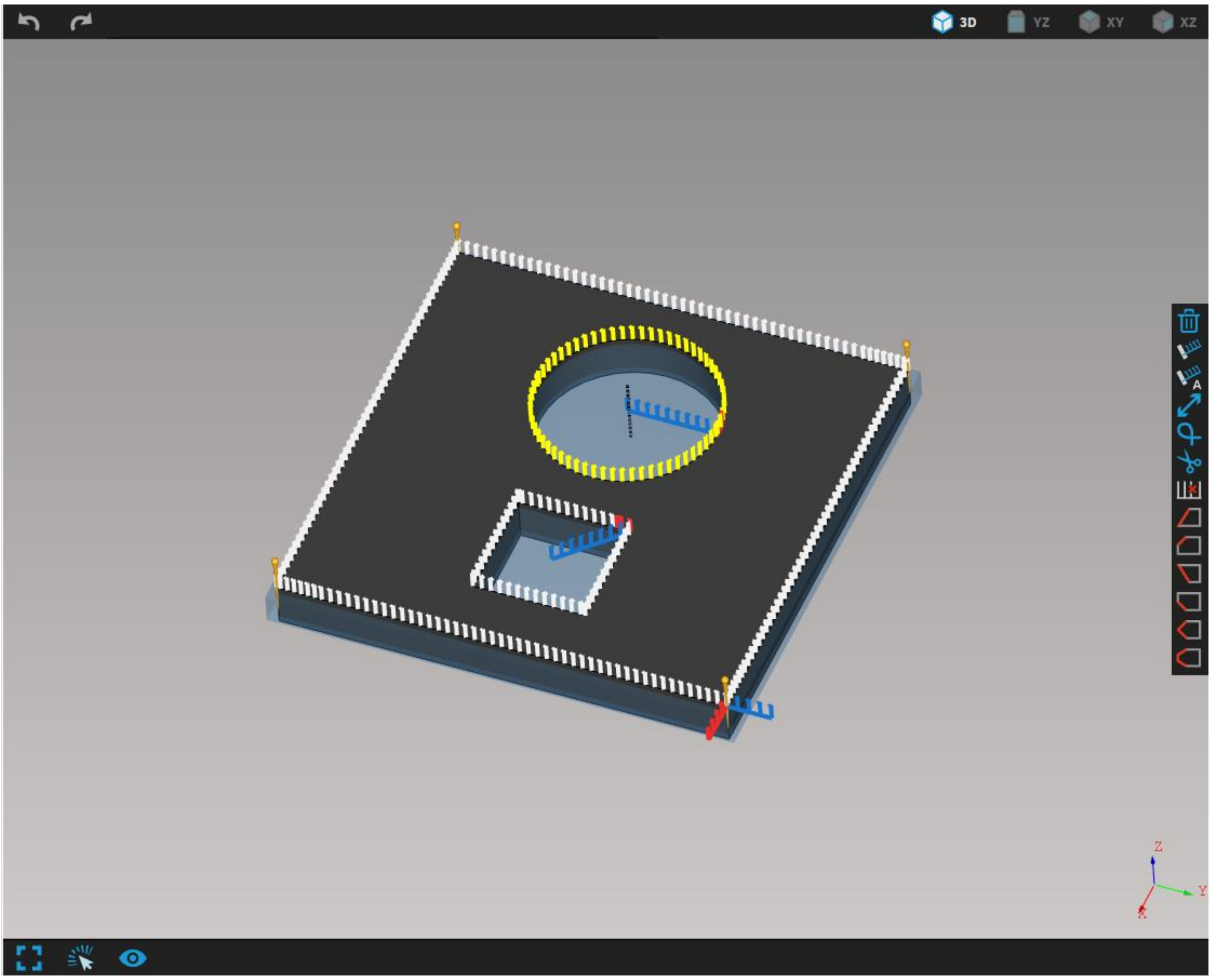
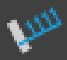
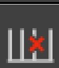


Fig. 139: Cut path editing panel on the right side of the visualization screen

Icon	Description
	Set lead-in/out to custom position (described in section Lead–in s/–outs)
	Set lead-in/out to automatic position as generated by mCAM
	Revert the cut path (described in section Reversing a cut path)
	Add loop on selected cut point (described in section Loops)
	Split path on selected point (described in section Splitting paths and creating loops)
	Join path points (described in section Splitting paths and creating loops)
	Erase selected cut point
	Delete cut path

There are also welding attributes, that are described in section *Welding preparations.*

Cutting path editing shortcuts:

- **E –** erase cutting path point on selected cutting path
- **L –** add loop on selected cut–point
- **Shift + L** – add loop between two points on cutting path
- **R** – remove selected loop
- **T –** delete selected cutting path
- **Y –** disable all technologies on selected cutting path
- **H –** open select cut dialog
- **B –** split path on selected cutting point

Alignments on beams

Alignments are used when machine uses laser scanner for beam measurement/ (H–beams, U–beams, L–beams, Square pipes). Laser scanner measures the exact size of beams (width, height and Web offset) and adapts the size or position of *cutting contours* to measured beams. Laser scanning of beams partially solves the problem of unprecise dimensions of beams by the stretching or positioning the cuts along the whole width of flanges or web.

Alignments function allows to *align* cuts to positive or negative edge of flange/web, *align* cuts to middle of flange/web (centering holes) or *stretch* them along the whole width (trim cuts).

Note: *Stretch* function should be used on all trim cuts to make sure that parts will be completely cut–off. It is not recommended to use *stretch* function on closed holes and closed features with special geometry due to possible deformation of required shape – when beam size differs from 3D model too much.

Alignments functions:

- ***stretch*** – stretches cuts along whole width – affects the length of cut
- ***align positive*** – aligns cuts to positive edge of side (FACE) – affects only position of the cut (not dimensions)
- ***align middle*** – aligns cuts to middle of side – affects only position of the cut (not dimensions)
- ***align negative*** – aligns cuts to negative edge of side – affects only position of cut (not dimensions).
- ***align to top point*** – used when cut is passing through two or three sides of a part (square pipe U-profile, L-profile) and align cuts to top point of the edge through which the cut is passing. Affects only position of cut (not dimensions).

Trim cuts on beams are set to *stretch* by default and closed holes are set to *align to middle* by default.

Align functions (*to positive/ middle/ negative*) are used to keep the same distance of cut from the edges/from middle position as modelled in 3D model and defined in drawing of part.

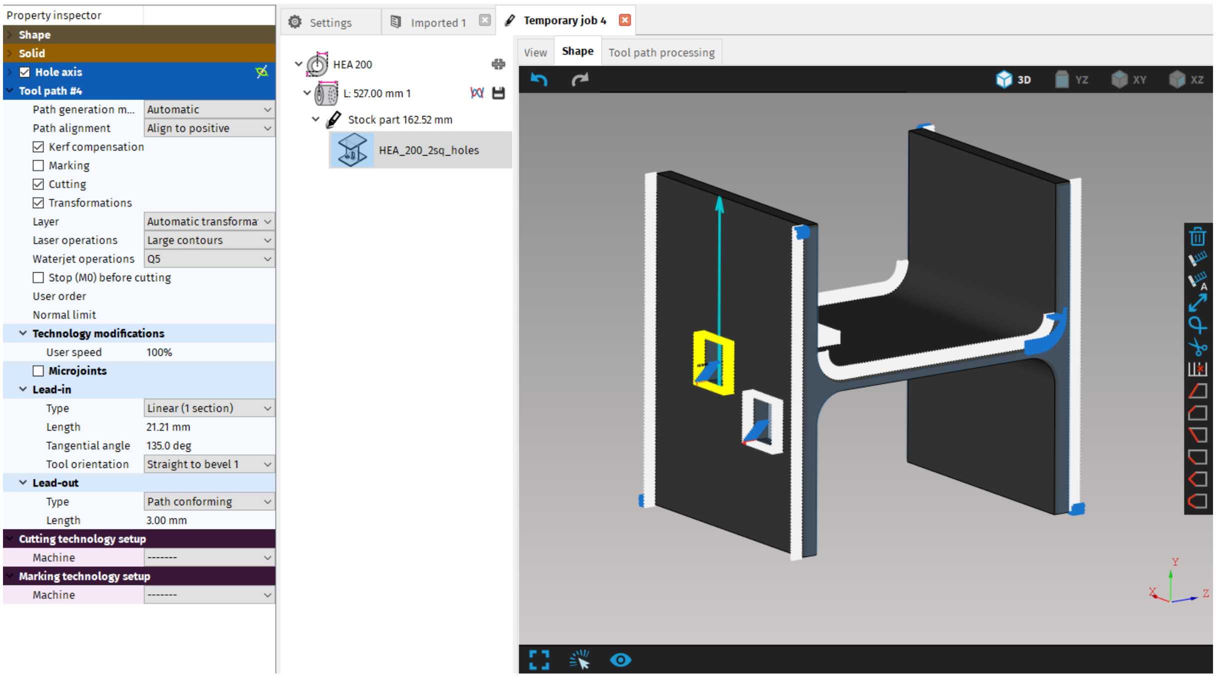


Fig. 140: Align to positive – hole will maintain the distance from the top edge

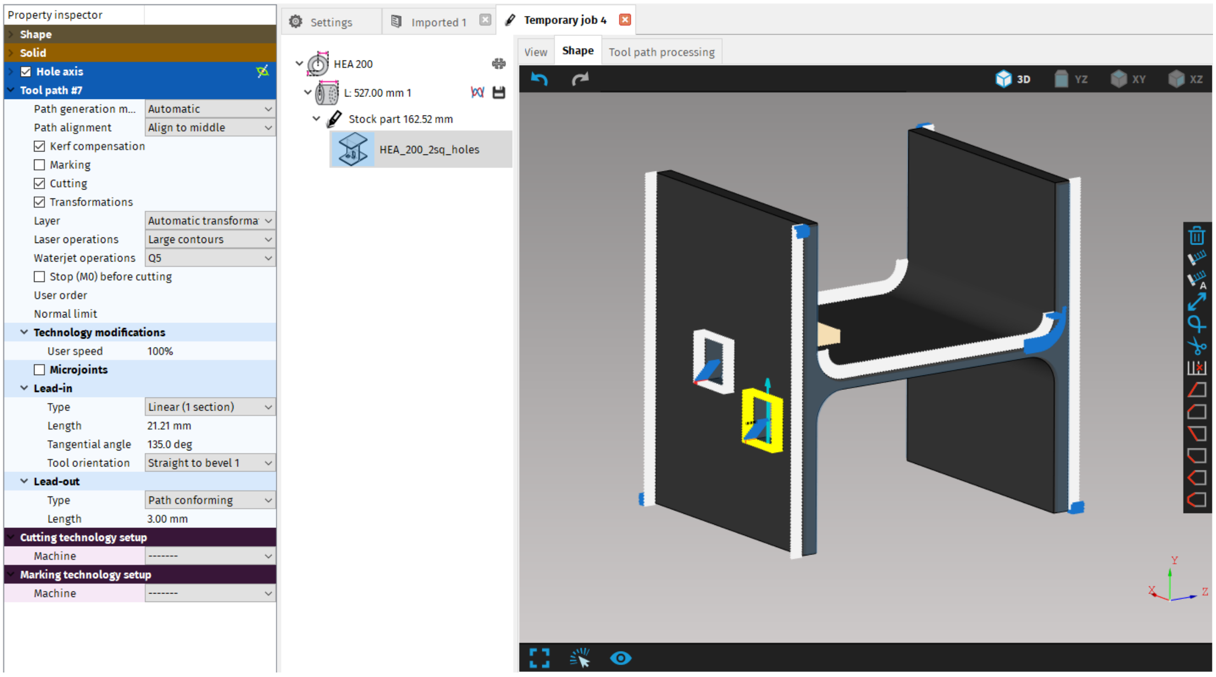


Fig. 141: Align to middle – hole will maintain the distance from the middle of flange

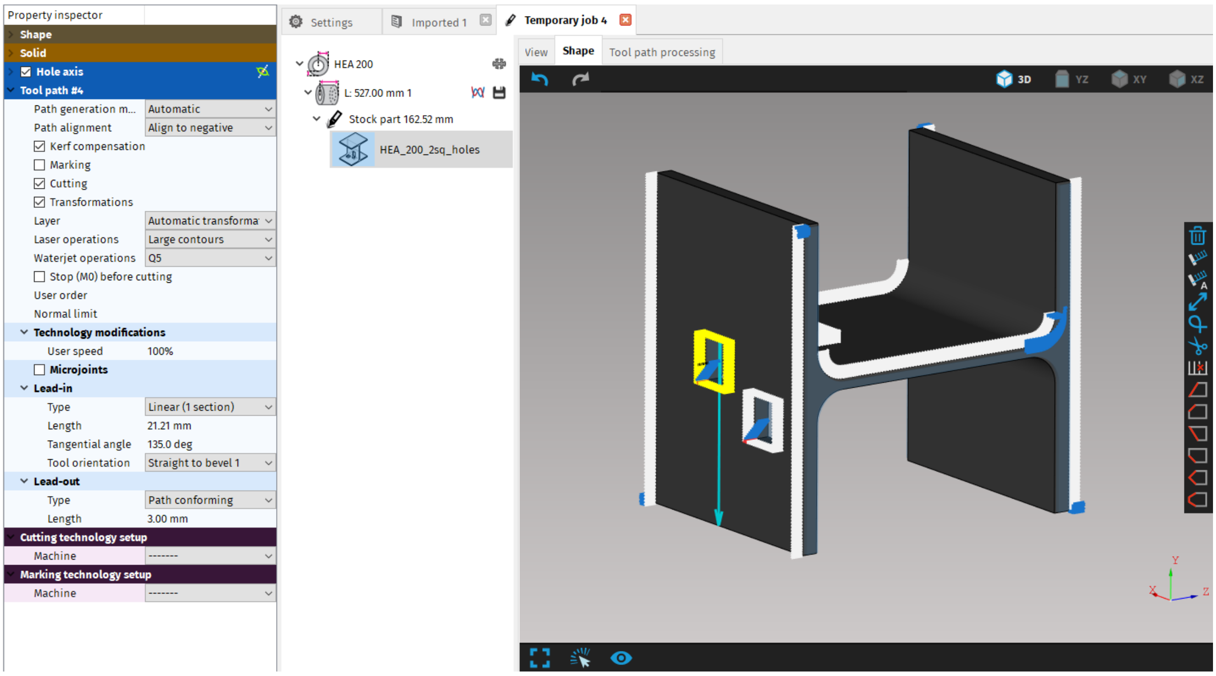


Fig. 142: Align to negative – hole will maintain the distance from the bottom edge

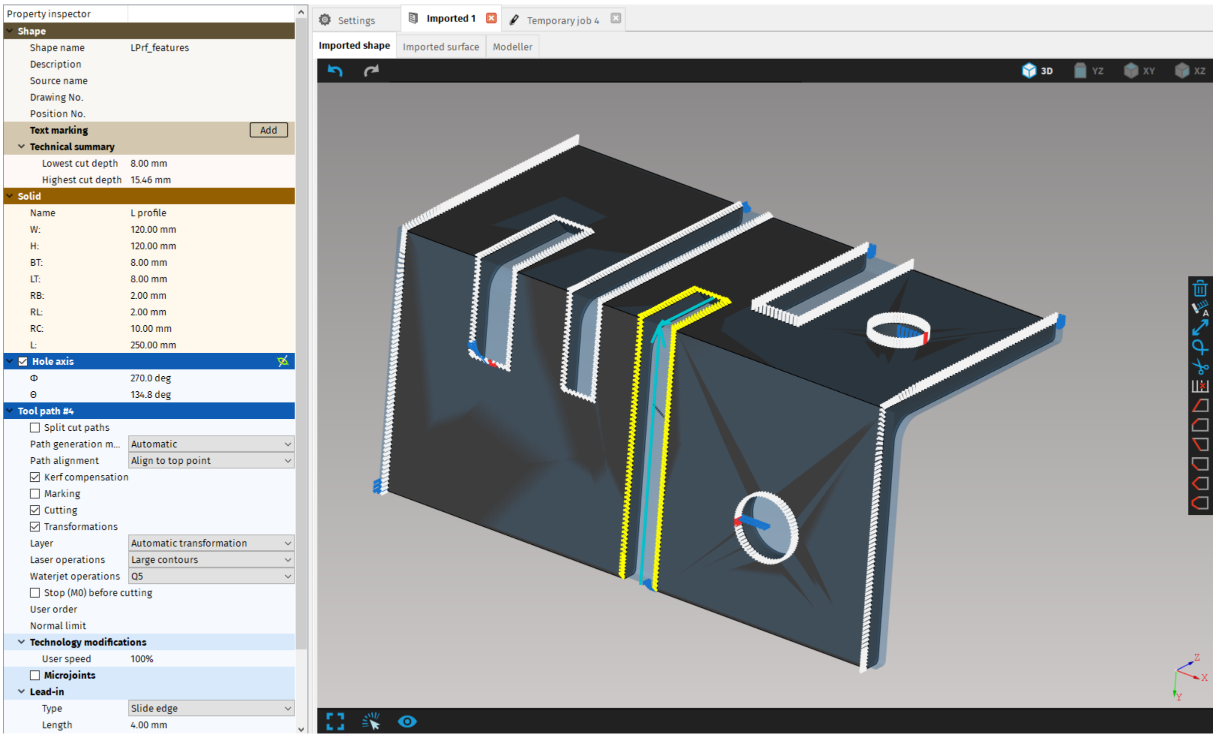


Fig. 143: Align to top point – hole will maintain the distance from the edge that it is passing through

Edge markers

Edge markers serve for selecting suitable transition type on edges. To display edge markers, they need to be enabled in the visualization screen settings menu located in the bottom panel under the eye icon.

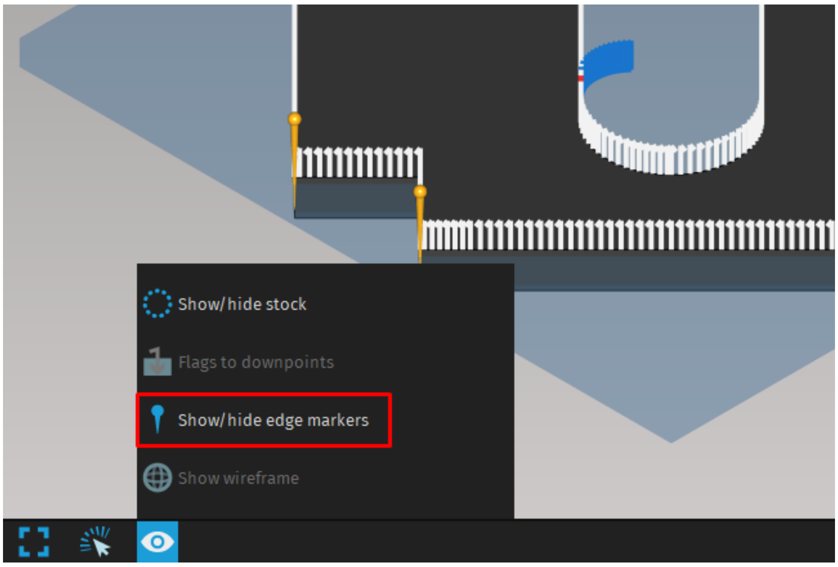


Fig. 144: Show /hide edge markers. The edge markers are enabled.

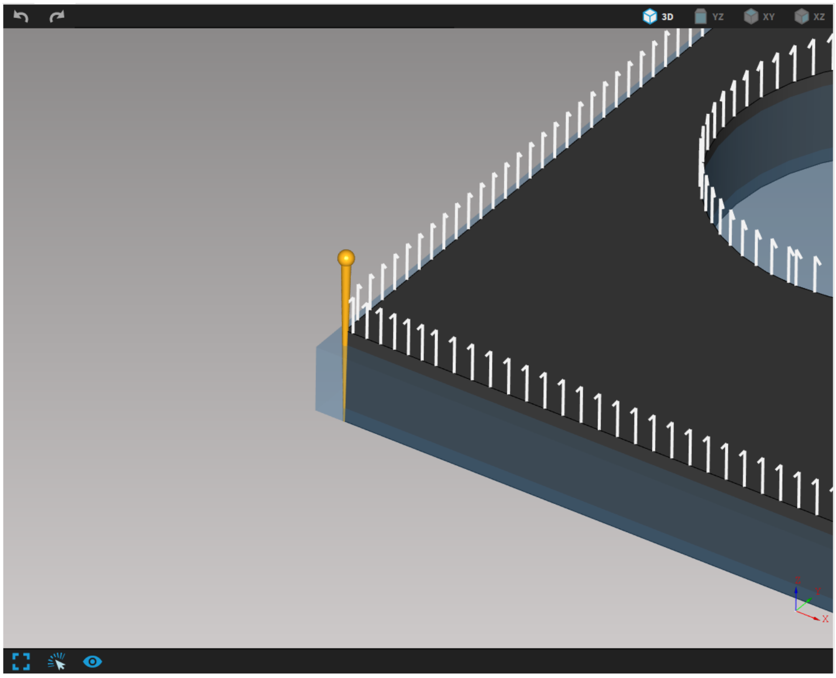


Fig. 145: Edge marker on corner of the shape

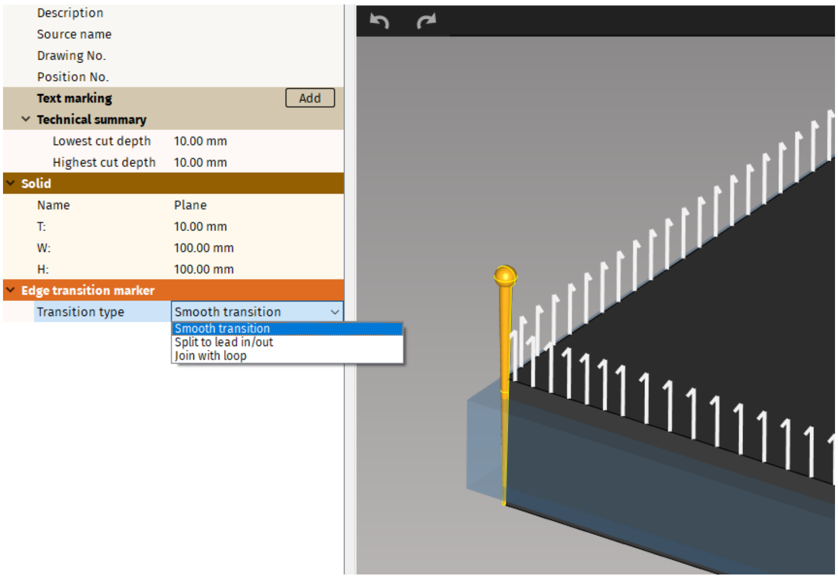


Fig. 146: Selected edge marker with its settings

After selecting a marker, its settings appear in Properties area. 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. Loops are described in more detail in section *Splitting paths and creating loops*.

Fitting cut path to a real shape

mCam now cooperates with mScan, another software developed by MicroStep, that serves for mapping a real shape and creating digital representation of a manufactured part by using data from a scanner. By applying this digital representation onto an ideal shape in mCAM, it adjusts all cutting paths and generates modified CNC instructions according to the new data.

The direction in which the cutting paths are shifted is determined via hole axis.

Hole axis

Hole axis determines the direction in which a cutting path is shifted from 3D model to a real surface. If mCAM detects a hole and is able to generate the hole axis automatically, the cutting path will shift according to the generated hole axis as pictured below, unless the hole axis is changed manually.

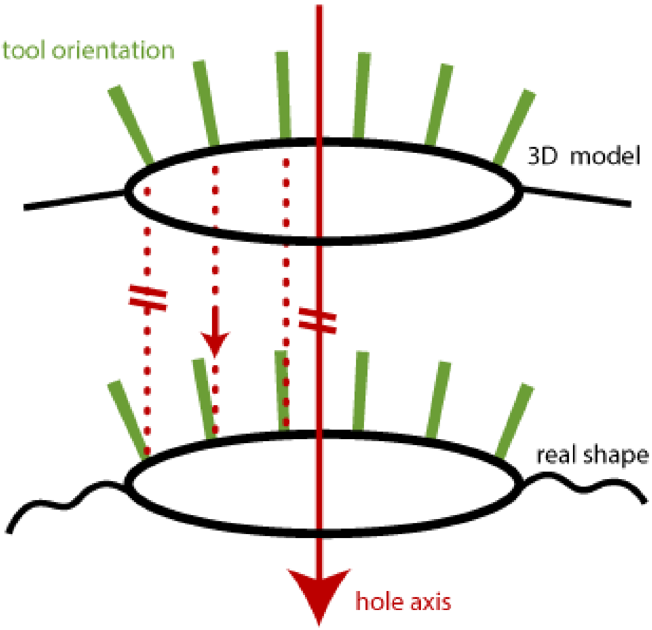


Fig. 147: Direction of shifting the cutting path with generated hole axis

The direction of hole axis can be changed manually by changing the corresponding values in X, Y and Z in cartesian coordinate system or ϕ and Θ in polar coordinate system in Properties area.

There are cases when holes either don't have hole axis (e.g. gaps) or mCAM can't generate them automatically. If a hole doesn't have a hole axis generated, the cutting path will shift in the tool direction as pictured below.

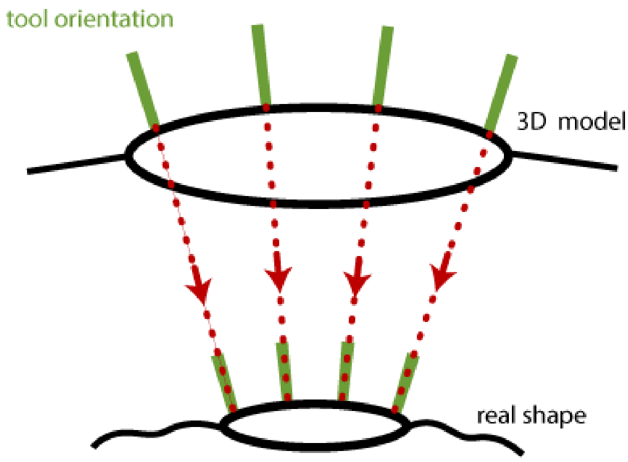


Fig. 148: Direction of shifting the cutting path without generated hole axis

After the import of the 3D shape to the mCAM, the program automatically detects holes and generates hole axes if it's possible. User can manually check if the axes are generated and if it's done correctly by selecting a cutting path. The hole axis settings show up in Properties area and hole axis of the hole appears on a visualization screen if present. If the box beside Hole axis is unchecked, the hole axis was not generated.

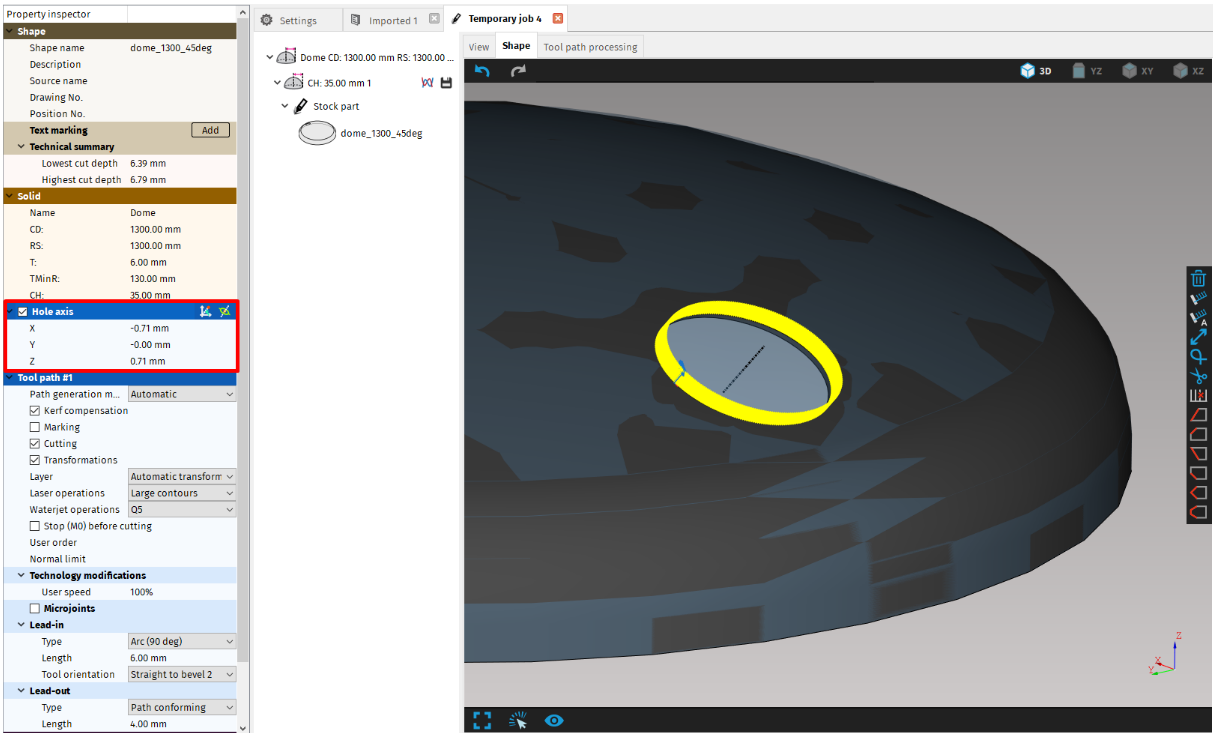


Fig. 149: Hole axis properties

In case of complex cuts, such as Y and K cuts, hole axis will be generated only for one part of cut – most likely on straight cut. The remaining cuts will then assume the same axis and the cutting path will be shifted accordingly.

If some holes don't have hole axis (e.g. holes with welding preparation) you can manually generate them by pressing J. The direction of the hole axis is then a normal of a best plane fitted to the selected cut path. In this instance pressing J is the same as clicking on a picture of the hole axis in the top right corner of Hole axis settings. In other cases, when the hole axis is generated automatically at import, the Reset hole axis button uses default algorithm and reverts the hole axis direction to its original placement.

If there are any holes without hole axis and you'll generate CNC code, a complaint will pop up with announcement that „Missing hole axis for some cut path(s).". If that was not your intention, you can recheck the holes, generate missing hole axes and then re-generate CNC code to ensure the correct and desired result.

You can change the type of the coordinate system in which the axis direction is displayed from cartesian to polar by clicking on a picture of the coordinate system.

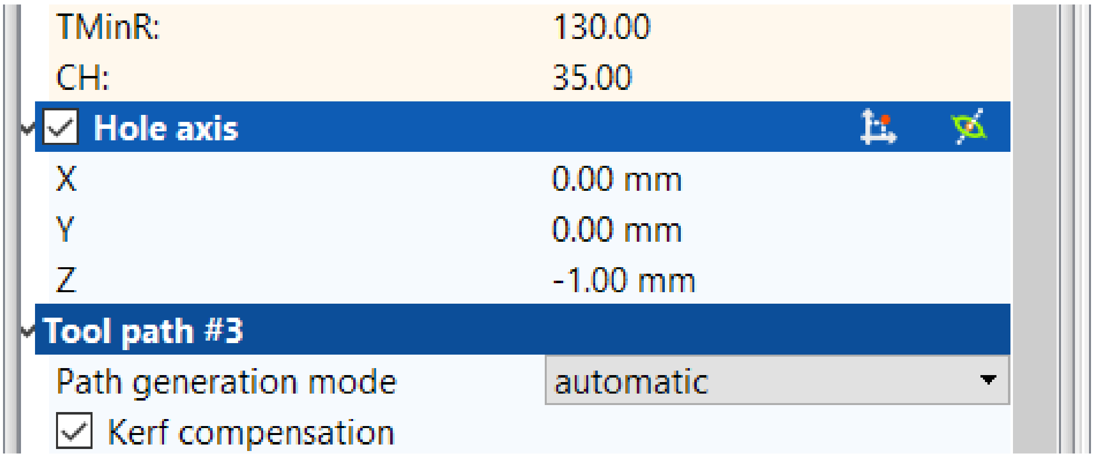


Fig. 150: Hole axis properties

Resets hole axis

Displays axis direction in polar coordinate system

Displays axis direction in cartesian coordinate system

Useful shortcuts

These keyboard shortcuts provide quick and easy way to set a hole axis in the two most occuring ways.

N – sets the direction of the hole axis according to the Z-axis.

J – the direction of the hole axis is based on a normal of a plane, that is fitted between points belonging to the cut path.

Lead–in s/–outs

In the process of piercing, most energy-beam machining technologies damage a small area around the piercing point. To eliminate the damage on the final work–piece, the piercing process is started on the outer part of the contour (in the waste area). Starting point can be manually placed by selecting a cut path, clicking on a set a lead-in/out on custom position button located in the cut path editing widget on the left side of the screen. The second button with letter A returns a lead-in/out to the automatic default position as set by mCAM with default settings as set by user in Settings.

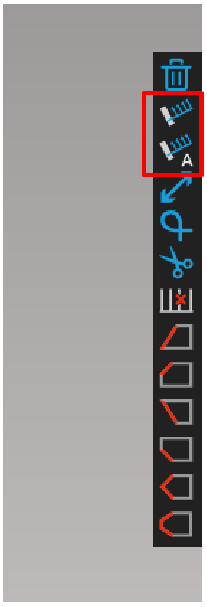


Fig. 151: Buttons for setting the lead-ins/outs to custom and default position

mCAM displays individual features of a cutting path in several colours. Lead-ins are displayed in blue colour, lead-outs in red colour and cutting path is in white colour.

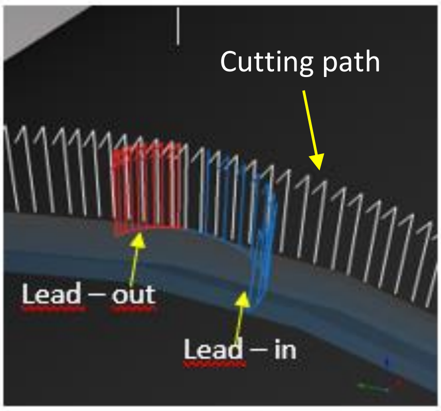


Fig. 152: Lead-ins/outs are displayed in individual colors

Default lead-in/-out parameters as type, length, tangential angle, tool orientation and user speed are defined in Settings -> Lead-in/-out section or can be changed individually for each cut path in Properties menu of selected cutting path. All parameters of lead-ins/outs are described in more detail in section *Lead in/out.*

There is also possibility to use **Dynamic piercing** on plates, where the tool moves from side to side until the material is pierced through, and only then it continues cutting respective cutpath. This technique is used when cutting plates that are too thick with water-jet. Dynamic piercing has these parameters:

- **Length –** specifies the length of piercing area

- **Repeat –** how many times does the tool move along the piercing area
- **Direction –** in which direction the tool is moving – X or Y axis

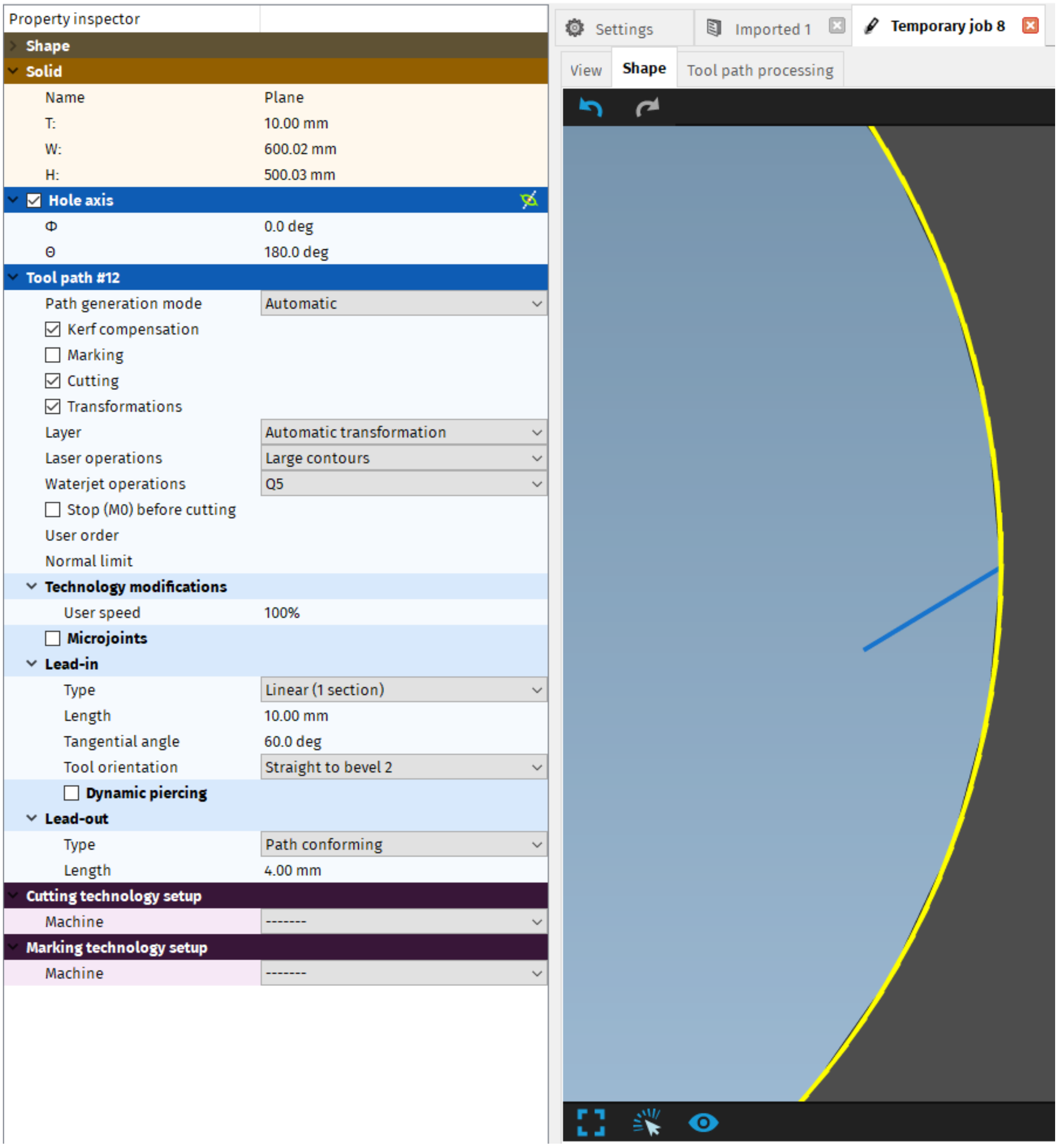


Fig. 153: Lead-in with disabled dynamic piercing

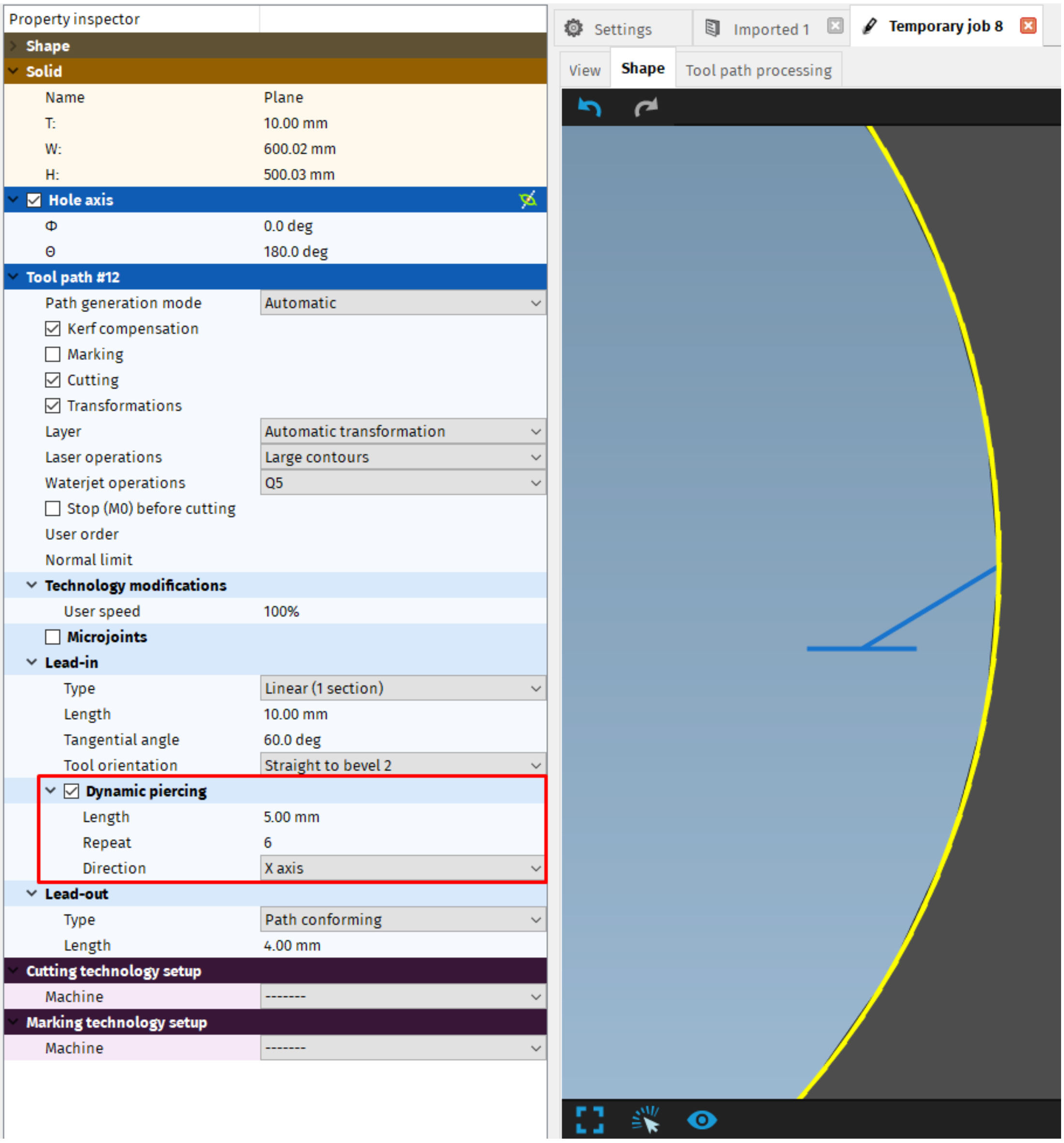


Fig. 154: Lead-in with enabled dynamic piercing

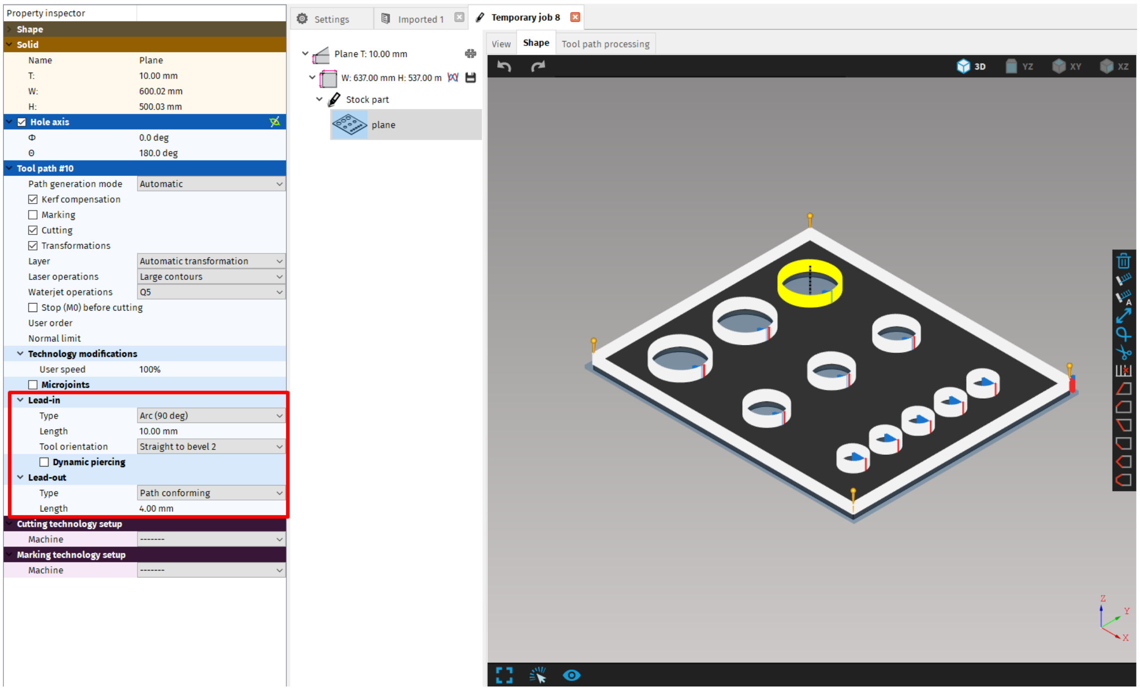


Fig. 155: Setting the lead-in/out for selected cutting path in Properties area

M0 – Stop before cutting

This setting 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.

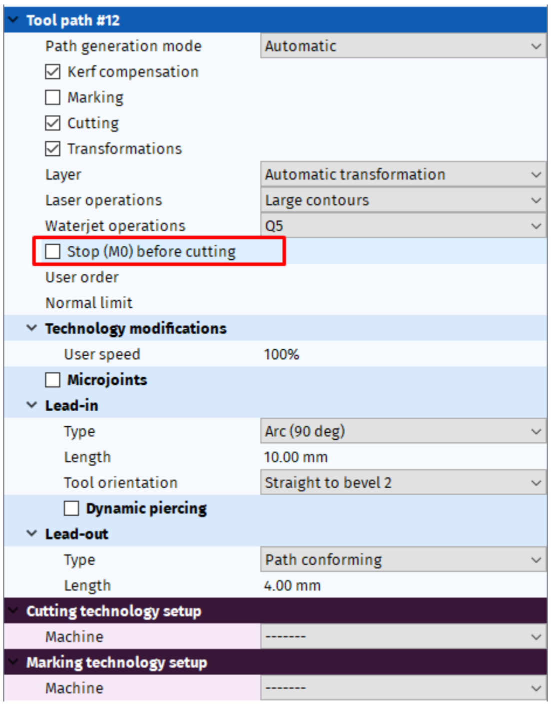


Fig. 156: Stop (M0) before cutting

Marking (contour marking)

Marking is a technological operation used in energy beam cutting technologies where reduced power, pressure, current or different type of gases needs to be used. Marking operation affects only few millimetres (or less) of material and is mainly used to mark cutting contours or cutting features that are intended to be machined by other machining method (e.g. drilling or milling holes, etc.).

There are several cases when marking is useful. It is frequently used in laser cutting when semiproducts (pipes, sheets or beams) are too rusty for cutting and all contours (intender for cutting) need to be marked first.

Marking is also used for marking the cutting contours that have been limited due to maximum angle of torch to material normal that is possible to reach. Standardly used torch tip in plasma bevel cutting is able to reach 45 degrees (or 50 degrees for special torch tips). Laser cutting head standardly uses nozzles that are able to reach 20 degrees to material normal. Maximum angle between torch and material normal while cutting is defined in *Settings – Cut path – Technology limits*.

Marking operation is enabled or disabled by using the *Marking* checkmark option in *properties* of selected cutting path. Marking is disabled for all contours by the default.

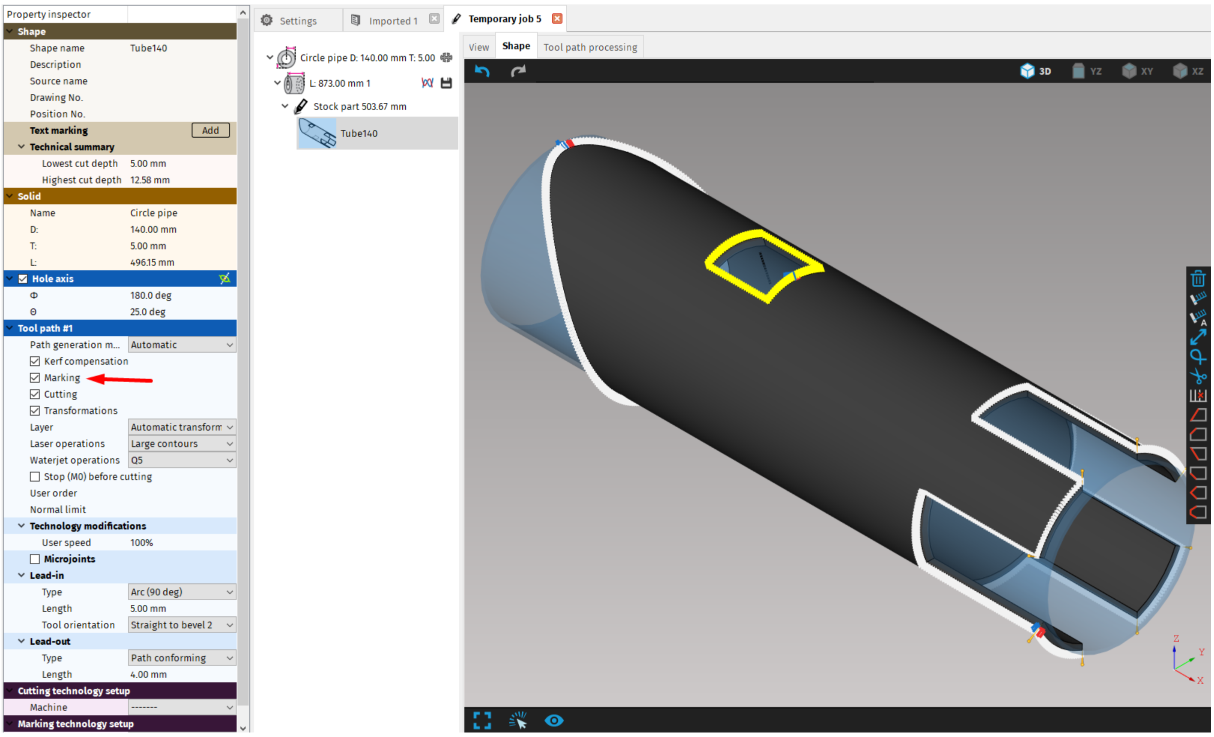


Fig. 157: Marking of cutting contour

Application of marking instruction (all contours that have been set to mark) into CNC program has to be confirmed in the *stock properties* in *Job task.* Generation of all required marking instructions into CNC program is enabled by the default.

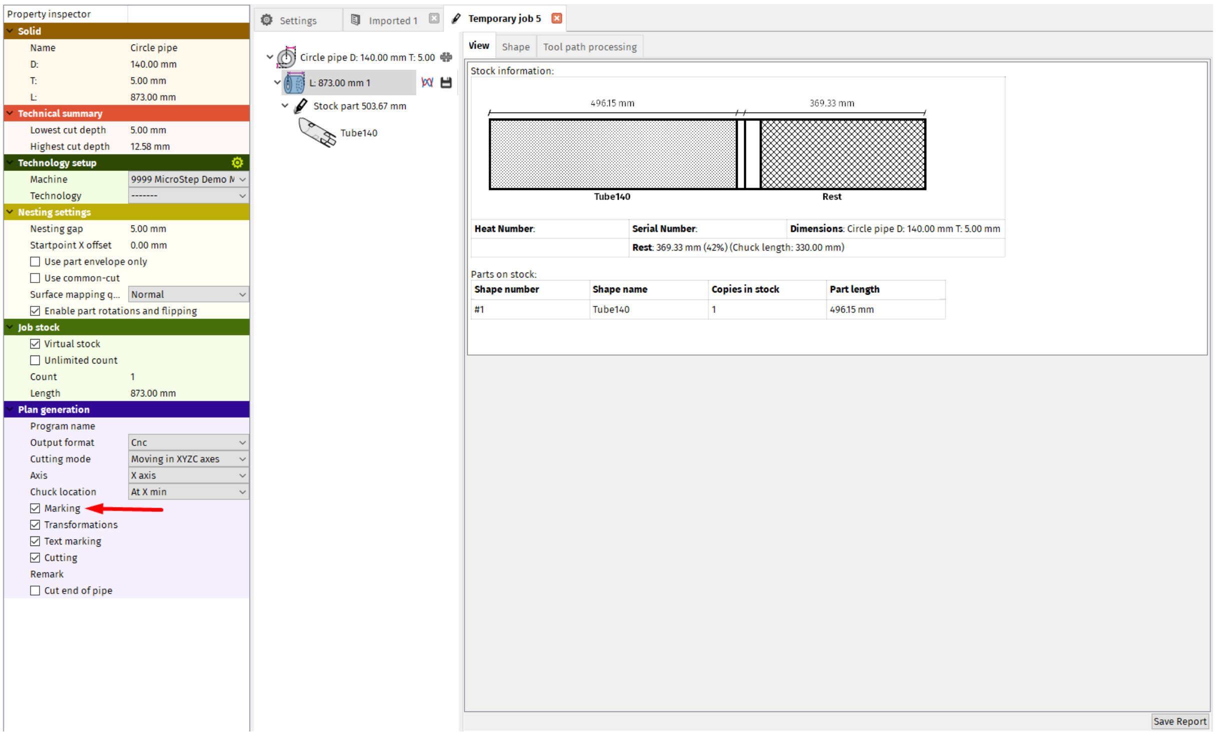


Fig. 158: Marking instructions generation – enabled

Microjoints

Microjoints are used when a cutting path needs to be divided into smaller sections separated by uncut segments to preserve integrity of a work-piece and rest of the stock material and to prevent distortions or damage of parts during cutting. Microjoints are defined by their length and quantity. Microjoints are placed regularly on the whole length of selected cutting path. Starting point of the first microjoint is set by *starting point* position. Lead–ins and lead–outs types of all microjoints on particular cutting contour are defined by *Lead–in* and *Lead–out properties* of the selected cutting path.

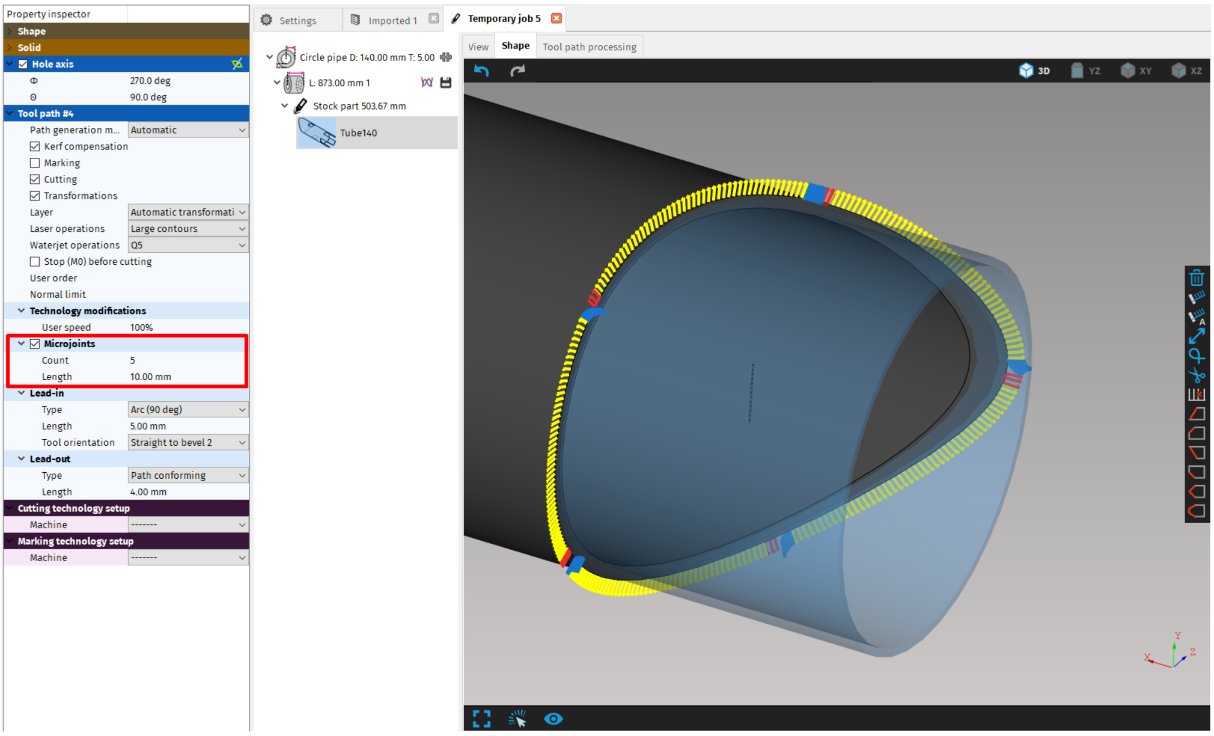


Fig. 159: Microjoints application

Path generation mode

Path generation mode determines how the cutting points (tool path) are generated. There are seven modes that can be used in different cases and generates different cutting paths with specific characteristics. Default mode used (for all newly imported parts) is set in *File –> Settings –> Cut path –> Path generation mode* with no *machine/technology* selected.

Path generation mode can be changed for each selected path individually according to the contour properties and cutting edge shape type in *Properties menu* in section *Path generation mode*.

There are seven path generation modes:

- **automatic**
- **smooth cut**
- **tangential**
- **minimal distance**
- **linear mapping**
- **surface normal**
- **face tangential**

Automatic (recommended to use)

This mode is the combination of several modes: minimal distance, surface normal, linear mapping, and face tangential. It combines algorithms according to actual shape of the cutting face. Due to the combination of the four modes, it generates the best possible cutting path in most cases.

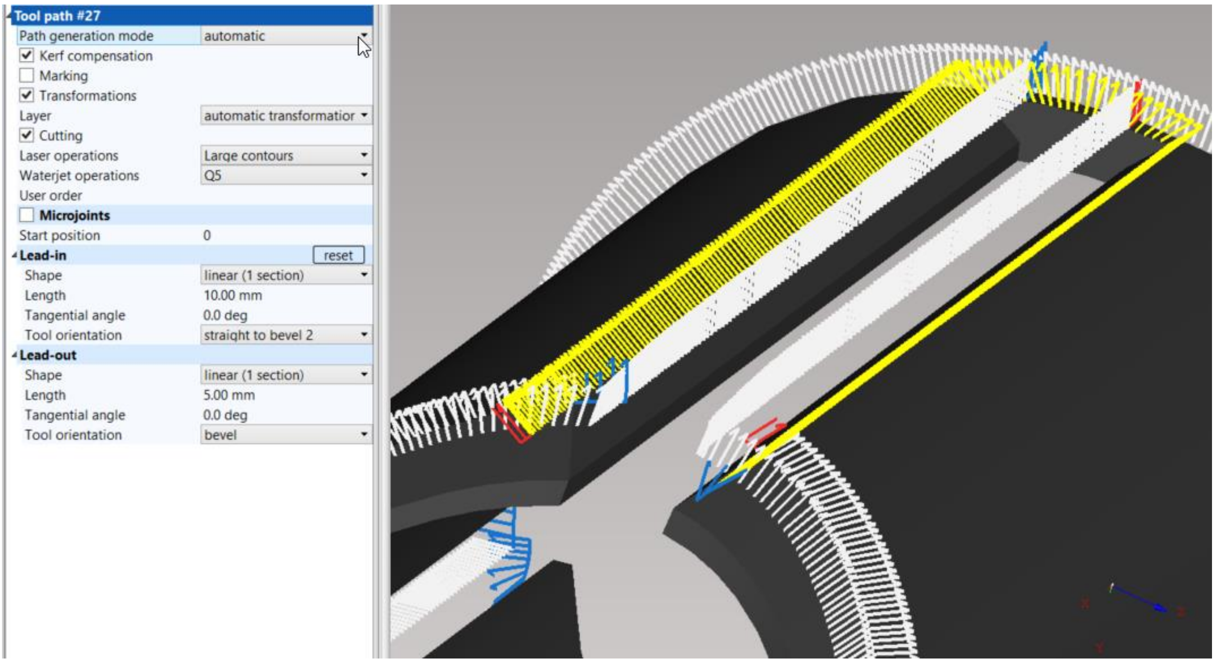


Fig. 160: Complex surfaces and weld edge preparation – automatic mode

Smooth cut (2D planar cutting)

Smooth cut mode is a very precise method for contours that are perfectly trackable by one beam from top to bottom edge of the shape. It is not recommended to use on convex or concave surfaces (e.g.: intersection of pipes and other rounded shapes) or for shapes with Y and K cuts.

This method is recommended to use especially in water-jet cutting of complex shapes where cutting faces are completely traceable by one straight beam in all sections of contour. Another important characteristic of this method is its speed.

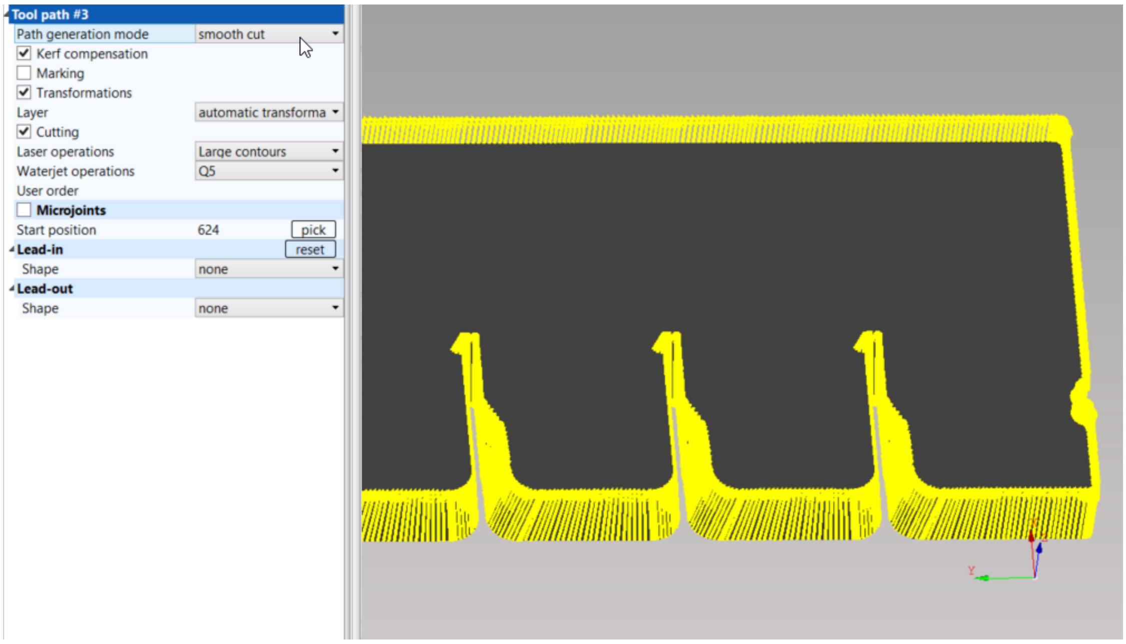


Fig. 161: Complex cutting surfaces – planar shapes – smooth cut mode

Tangential

Tangential mode generates cutting path that connects two points. The first point is defined as an intersection of the top contour and the plane given by the axis of the angle between two cut-path points belonging to the top contour. The second point is defined as an intersection of that particular plane with the bottom contour.

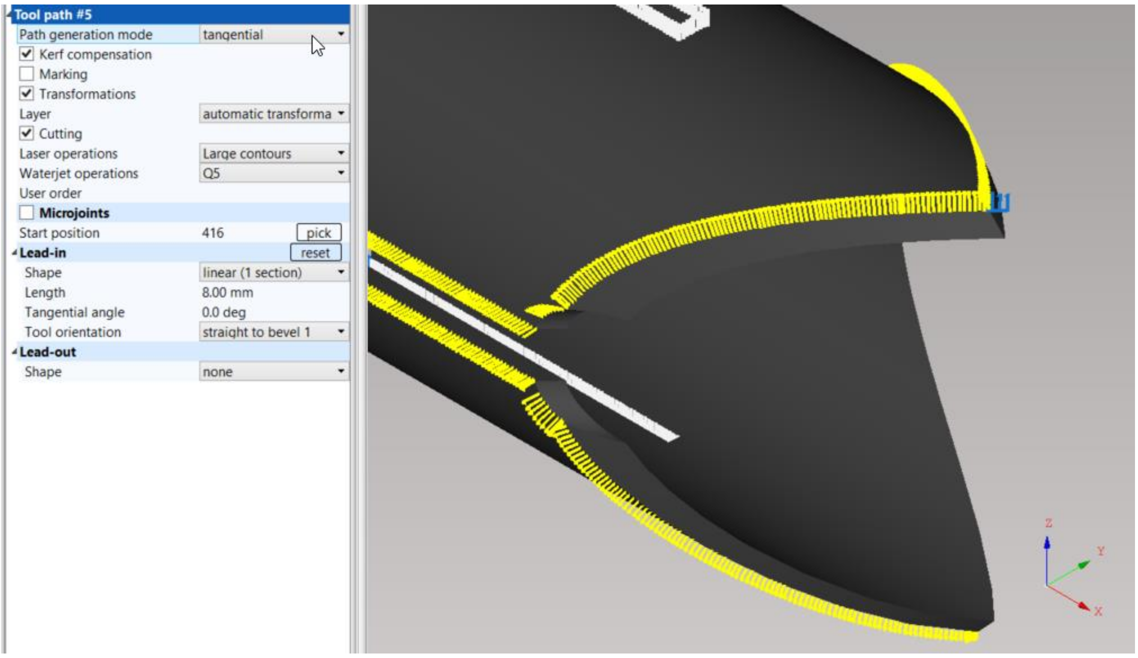


Fig. 162: Tangential tracking of cutting path – tangential mode

Minimal distance

Minimal distance creates a cutting path as the shortest path from a top cutting point to the bottom cutting edge.

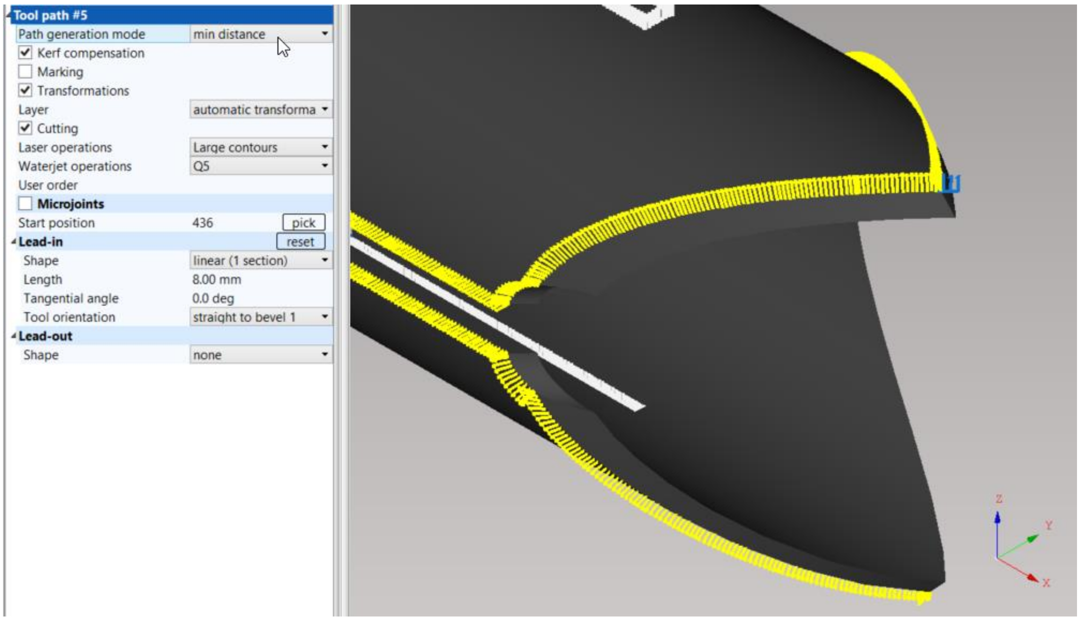


Fig. 163: Minimal distance mode

Linear mapping

Linear mapping mode creates the cut–path as a linear movement with linear speed at the top and the bottom cutting edges from the starting cutting point.

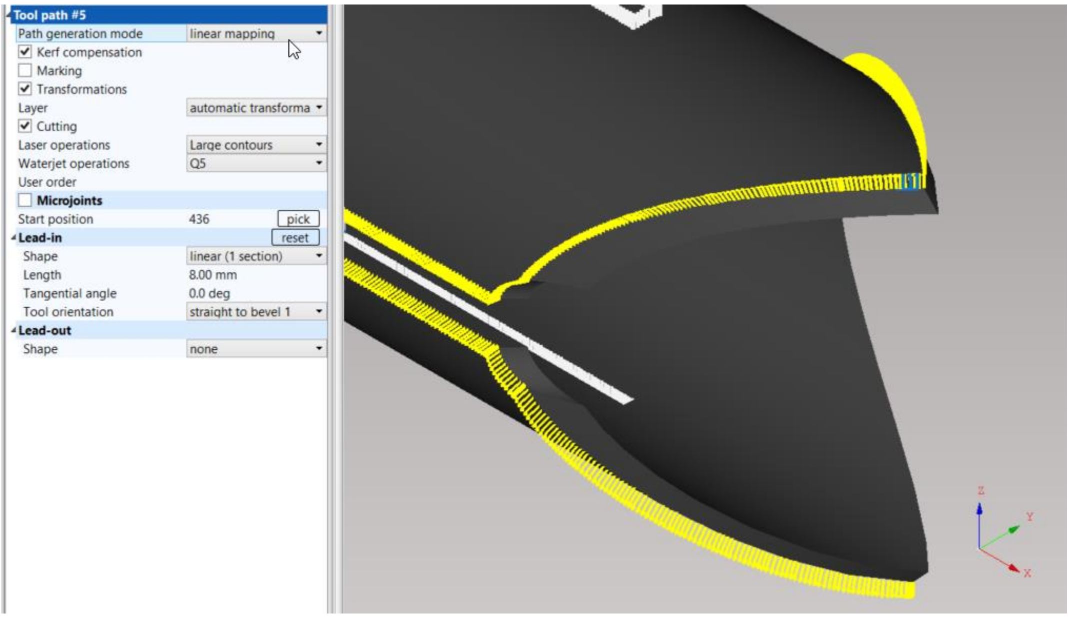


Fig. 164: Linear mapping mode

Surface normal (straight cutting)

Surface normal mode creates a cut–path as the normal to the surface on top edge of cutting contour. This mode is recommended to use only with straight cutting and straight contours.

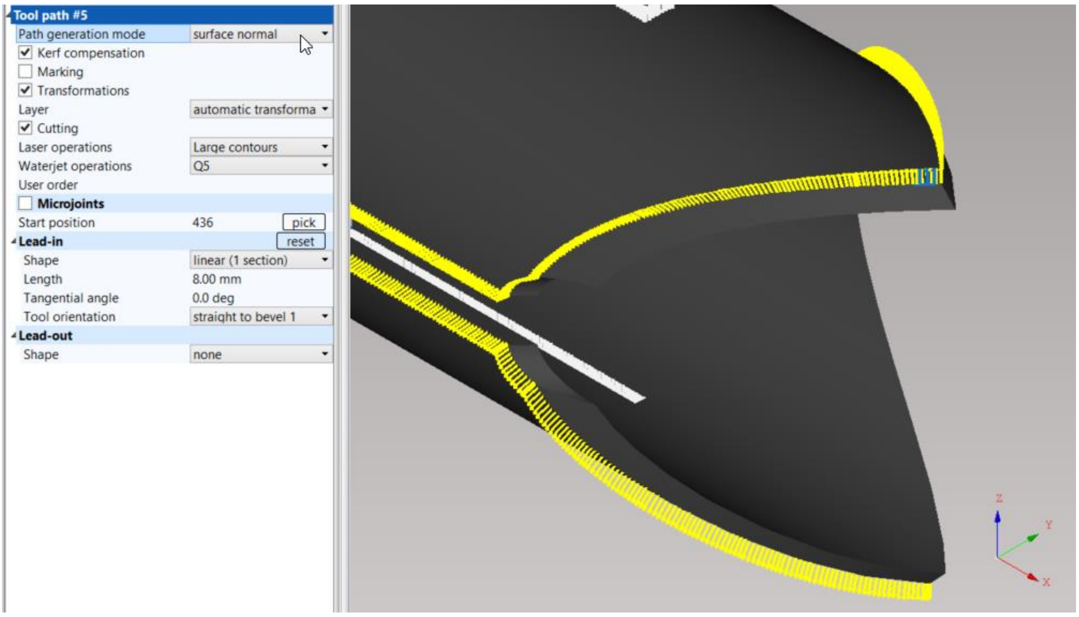


Fig. 165: Surface normal mode

Face tangential

Face tangential mode is similar as the tangential but it uses a different algorithm.

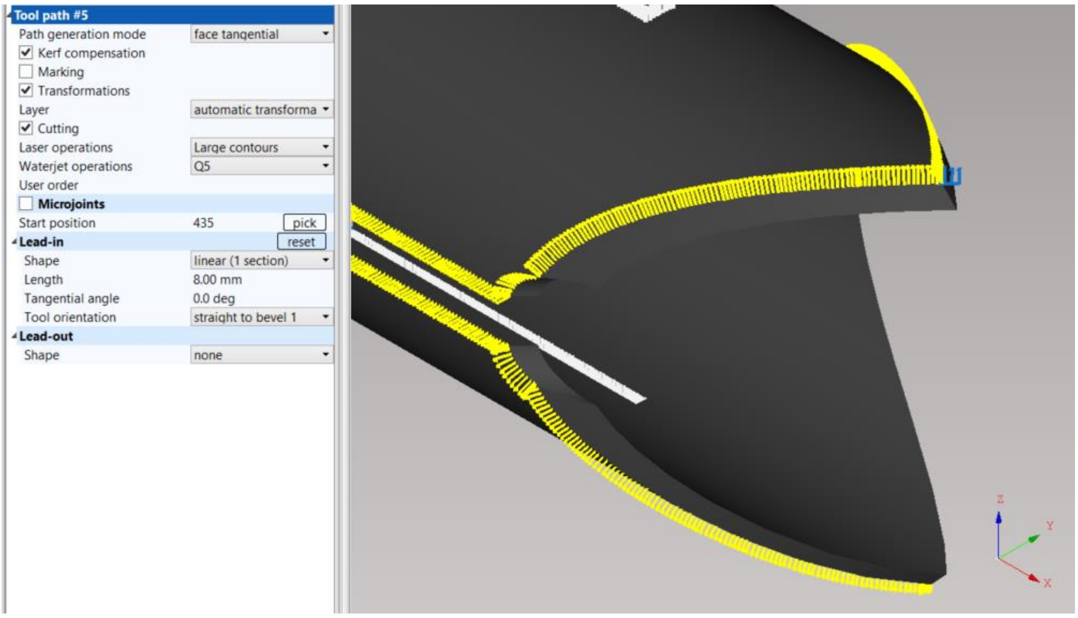


Fig. 166: Face tangential mode

Keep in mind that using other path generation modes (except Automatic path generation mode) may cause non–smooth cut especially on sharp cutting edges, too complex shapes or weld edge preparations.

Erasing cut path points

When cutting path crosses another cutting path in the wrong way, there is an option to erase some cutting points manually by selecting respective cutting path and using function *Erase* cutting path points (key shortcut: “E”) or  icon in cut path editing widget.

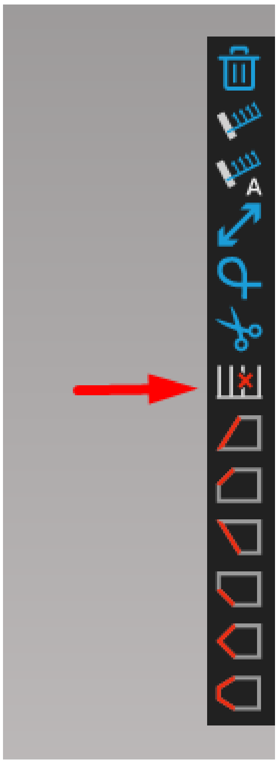


Fig. 167: Erase cutpoint icon in cut path editing widget

Erasing cutting points will cause that cut path will be generated by linear interpolation between previous and next cutting point surrounding the erased cutting point, therefore it can cause problems especially in variable bevel cutting.

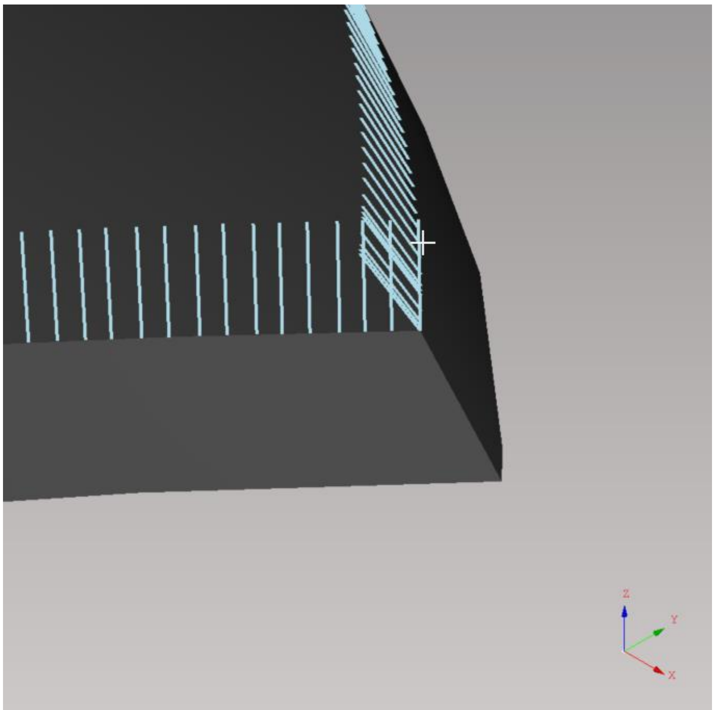


Fig. 168: Selection of cutting path points to be erased

Punching

Punching is a special technological operation when a cutting tool, typically a plasma torch, is used as a marking tool to create mark points on the surface. These are used for preparation of subsequent operation (*drilling, boring*, etc.). The centre point of a future hole is marked on the work–piece surface by a short punch of a machining tool. Identification of holes for punching as well as quality hole cutting or drilling is managed in *Transformations* and defined by *transformation tasks* (*punching, drilling, quality hole cutting,* etc.*). Transformations* are described in more details in section Transformations.

Reversing a cut path

A cut path can be reversed if the inner part of the hole is a final product. To reverse a cut path, it has to be selected either in Import or in “Shape” tab in Job. More holes can to be selected at once with “Select cut”, by pressing an “H” key. When all respective cut paths are selected, they can be reversed by pressing the “U” key or corresponding button for reversing a cut path in cut path editing widget.

This will cause all lead-ins to be generated on the outer part of the contour. Also, the kerf width is compensated outwardly from the hole so the dimensions of the inner part of the hole will be preserved.

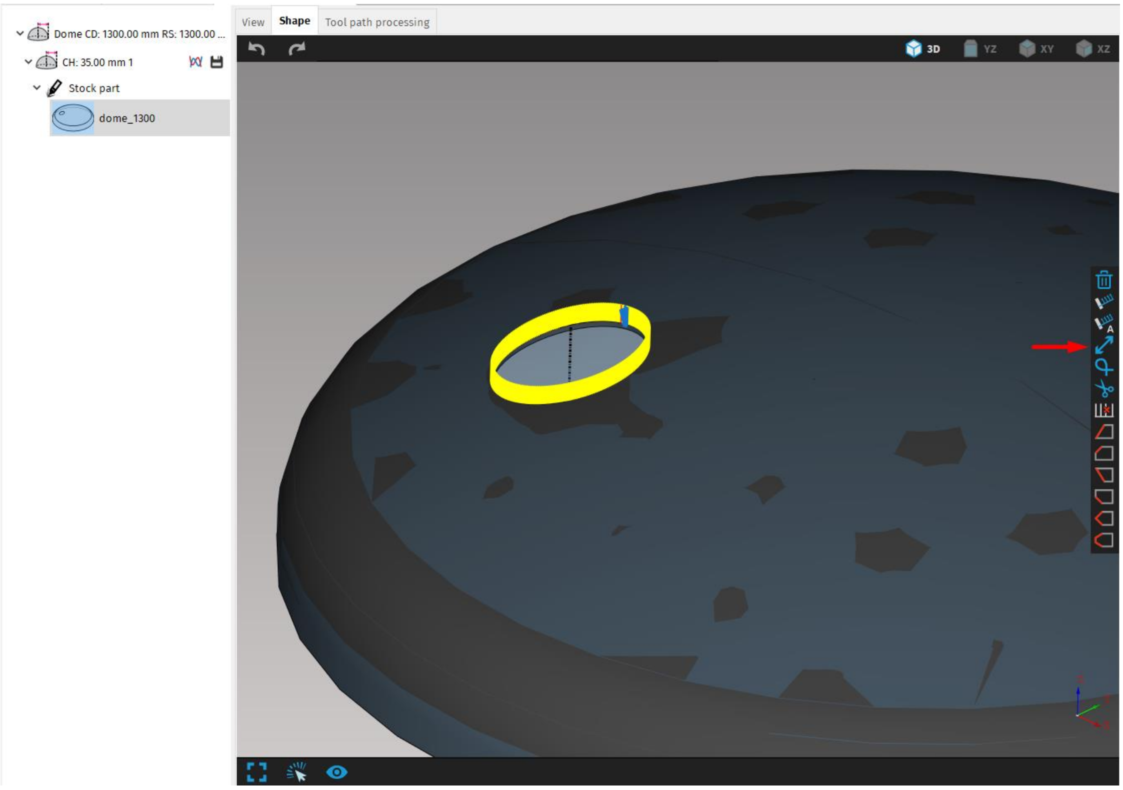


Fig. 169: Reversing cut path button in the cut path editing widget

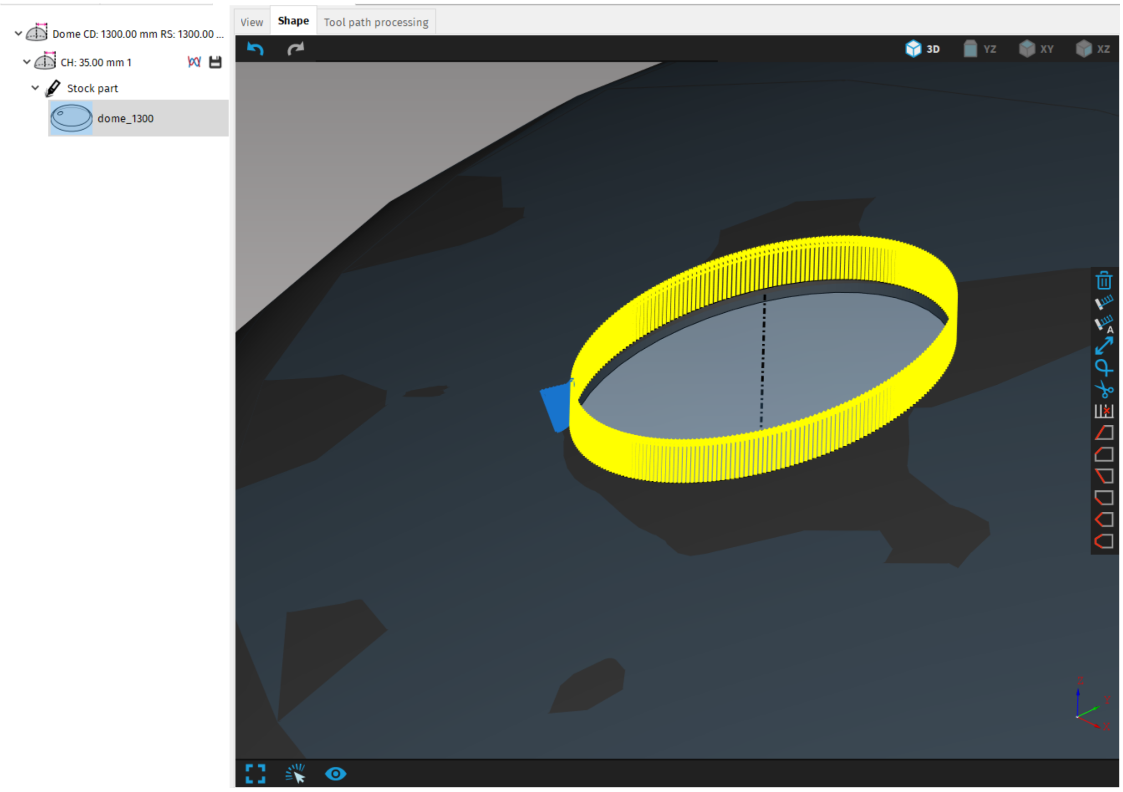


Fig. 170: Reversed cut path with lead-in on the outer contour

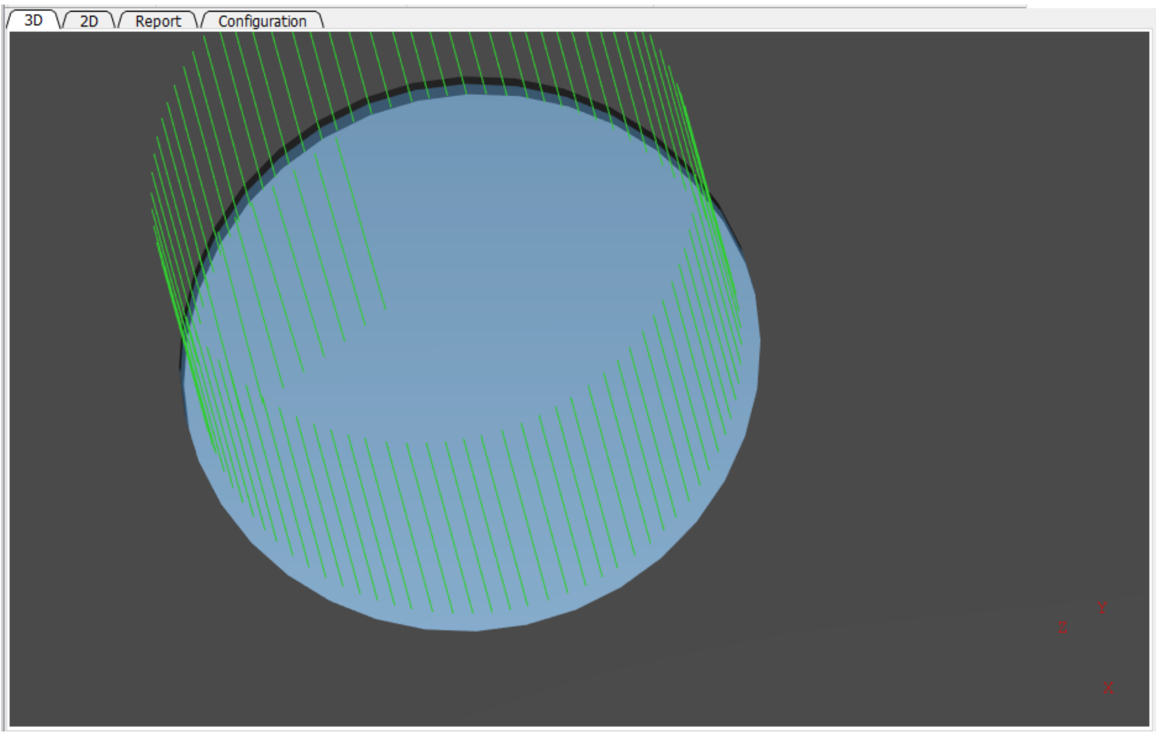


Fig. 171: Kerf width compensation on the inside of the hole while the cut path is not reversed.

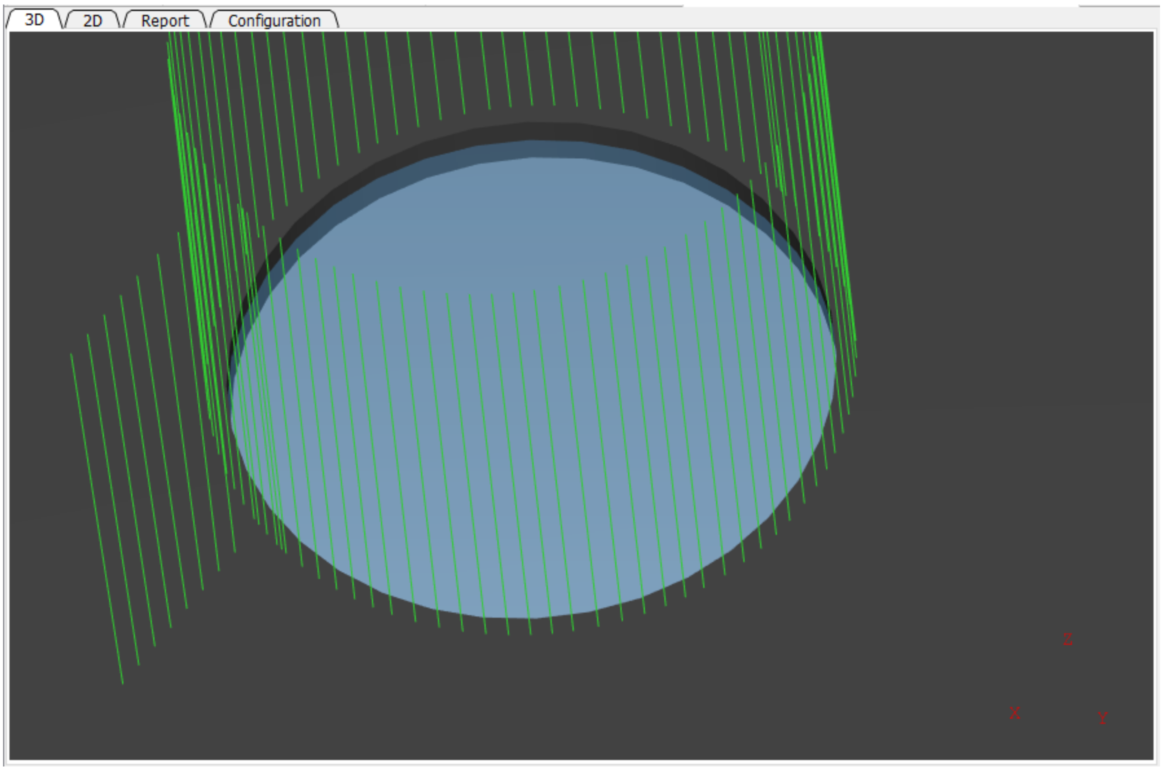


Fig. 172: Kerf width compensation on the reversed cut path is on the outer side of the hole thus preserving the desired dimension of the final product.

Splitting paths and creating loops

Cutting path can be split by selecting a cutting path, clicking on an icon of scissors  (or pressing the key shortcut B) and selecting a cutting point where the path has to be split. It is possible to split cutting path on any *cutting point* regardless if there is or isn't an edge.

Division of cutting paths (*Split paths*) is recommended to use especially when cutting special technological features separately (slots, straight cut sections, etc.) where we want to use more precise surface detection and more precise cutting.

Join paths button can be seen only with non-circular paths and can be used to join previously split or otherwise not-connected paths.

Connection of the cutting paths (*Join with loops*) is recommended to use when cutting sharp edges where by using *Spline loops* can user avoid overburn of the material, increase the quality of the edge, reduce number of ignitions or extends lifetime of consumables.

Cutting of joined cuts (with or without *loops joining*) is based on one surface detection procedure (one measurement). Therefore, if the cut path is long with wide X-axis range, it can cause tool collisions due to the inaccuracy of the stock material.

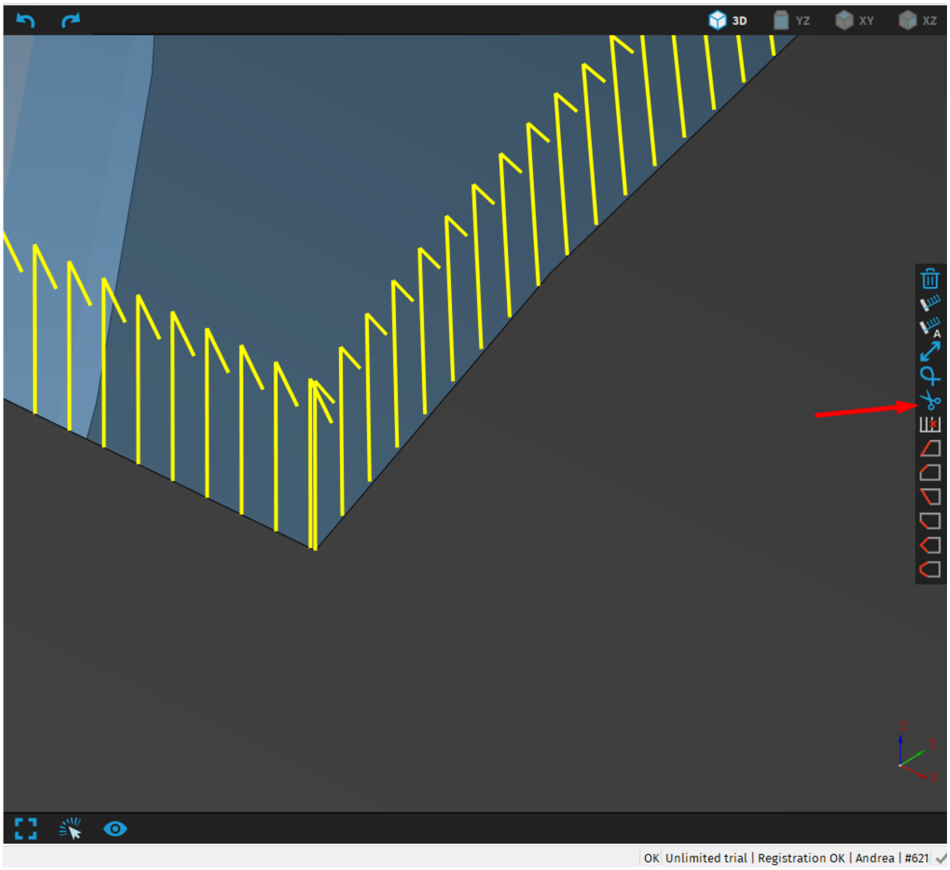


Fig. 173: Splitting path using scissors in cut path editing widget

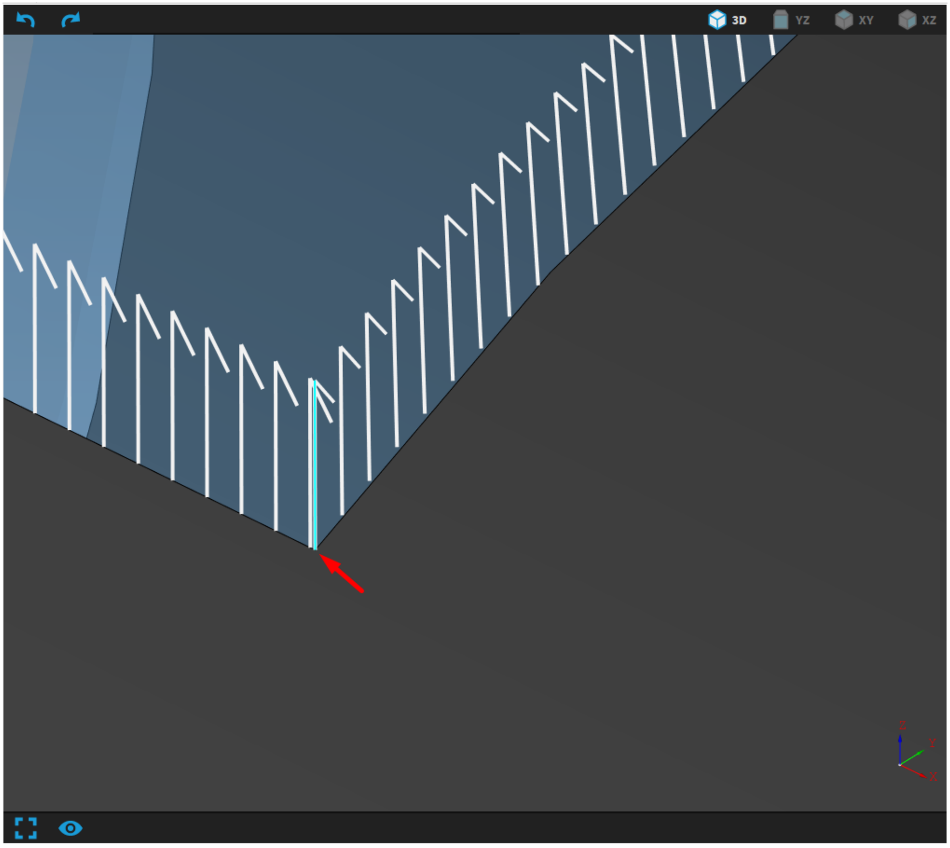


Fig. 174: Selected cut point is displayed in blue color

Loops

Loops are usually used in the corners, where the direction of contour is suddenly changed (i.e. a place of intersection of two linear entities). Loops are located in waste material and they significantly improve corner cutting quality. Besides significant improvement of the accuracy of the cut, loops can be used to reduce cumulative rotation in B-axis to meet its limitation. More information about the limitation in B-axis are specified in the parameter B-axis speed limit definition in section Generate.

Loops can be created in two ways:

- via Edge markers (described in section *Edge markers*)
- manually, using cut path editing widget (described in section Right control panel on page 88) When start position (lead–in /out position) is moved into the place of a loop, the loop is automatically removed.

When loop is selected, its settings are displayed in Properties area. There are parameters that define the type and size of the selected loop and option to reposition B axis.

There are three types of loops:

- **corner**,
- **triangle**,
- **arc**
- **spline**

The selection of loops in the visualization screen is available only when the Selection style is switched to displaying Loops. The Selection style option can be found by right clicking in the visualization screen.

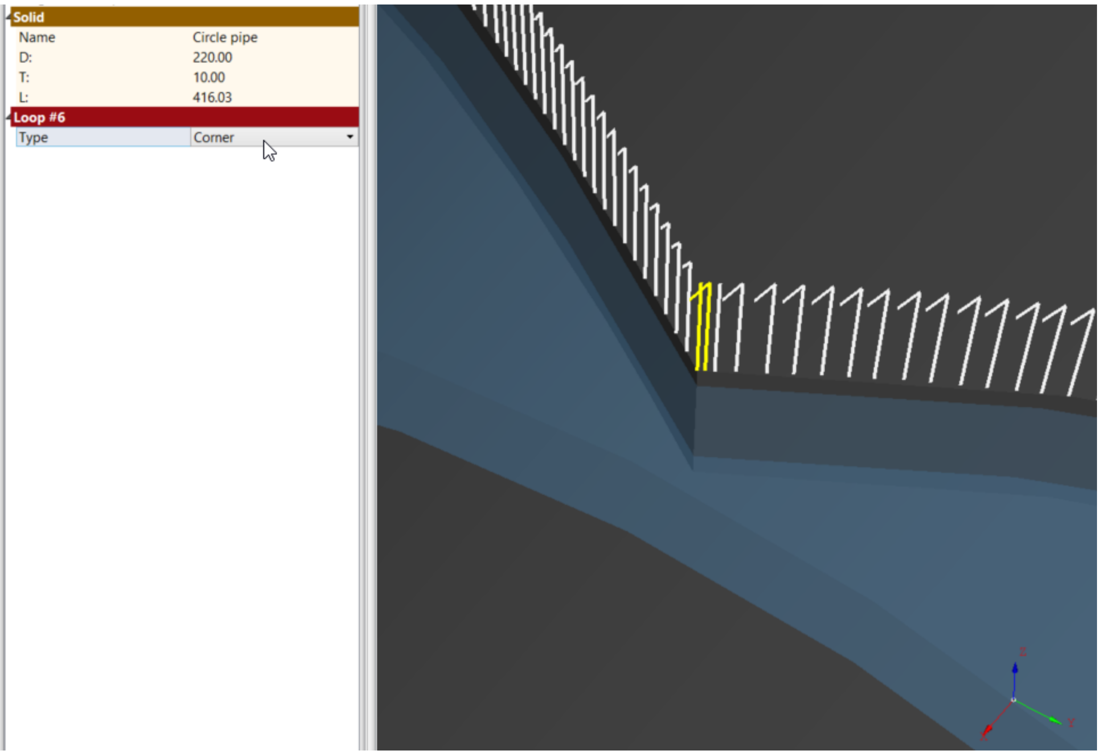


Fig. 175: Loop type – corner

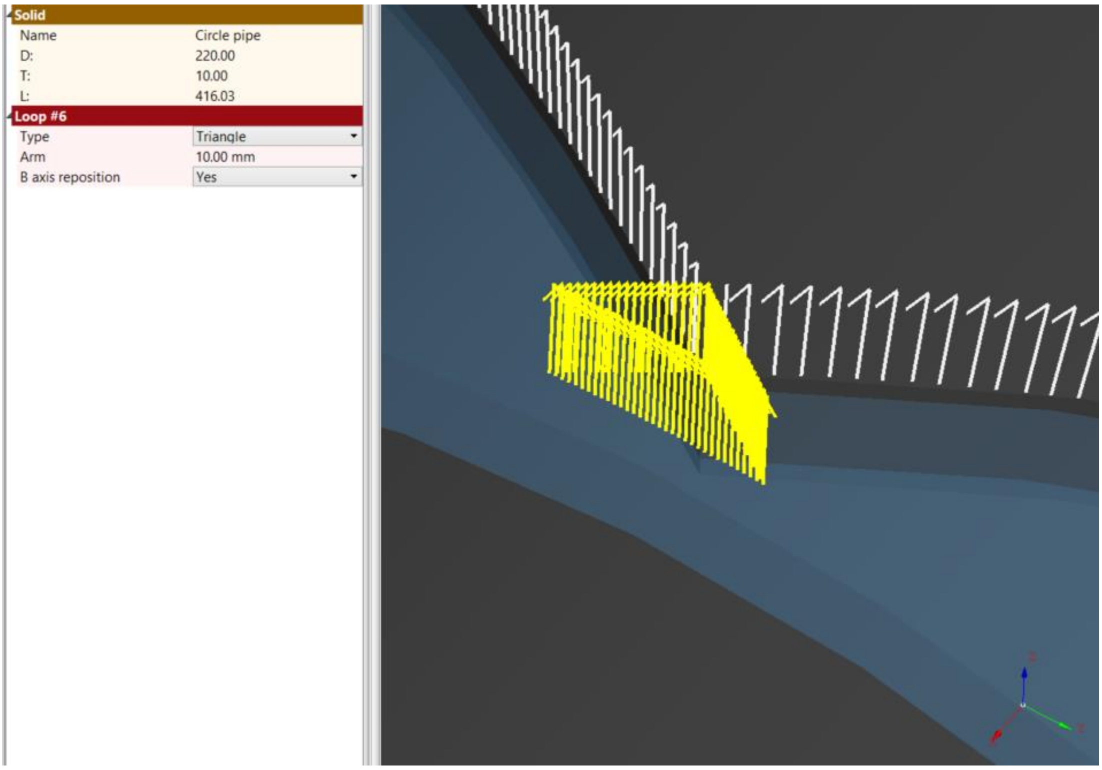


Fig. 176: Loop type – triangle

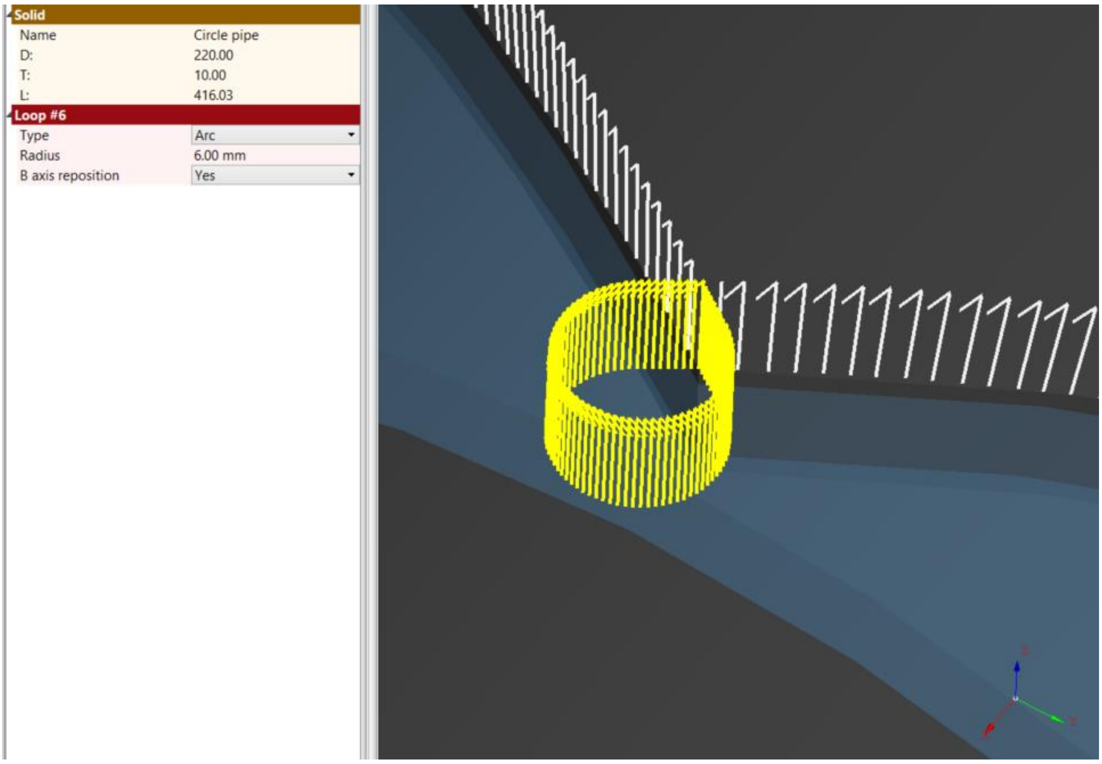


Fig. 177: Loop type – arc

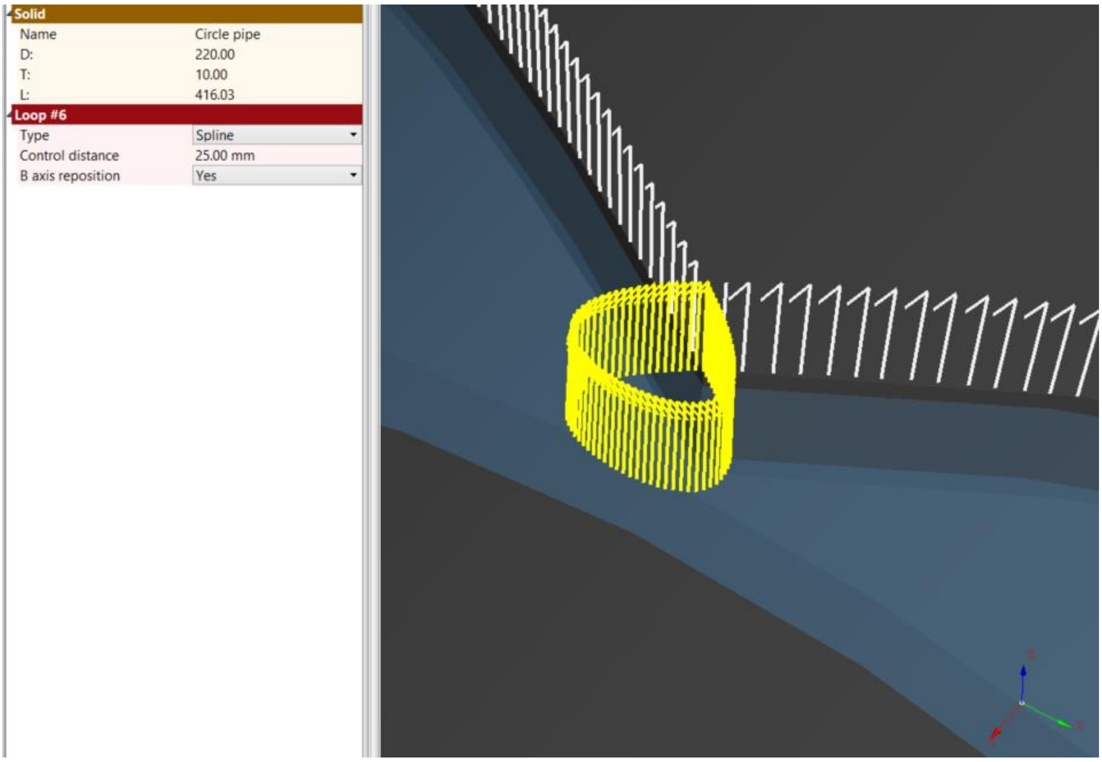


Fig. 178: Loop type – Spline

B-axis reposition

Loops can be used for backward rotation of rotator in B-axis. The function B-axis reposition is useful when cumulative value of B-axis on cutting path reaches the limit value (which is approximately 400°– depends on type of the CNC machine). If B-axis value exceeds the limit, *mCAM* displays a warning that signals the exceeded range of B-axis. Number and the places of loops to apply can be identified from a cutting simulation graph where the process of B-axis values is displayed. Backward rotation of B-axis is represented as M38 instruction in CNC code.

Shortcuts

There are several key–shortcuts and function Cut division wizard that can be used to add or remove loops, split cuts and join cuts. Key – shortcuts used for manual adding loops (on selected cutting path) within visualization screen are:

- add loop on cut–point (on selected path): key shortcut: <L>
- add loop between two cut–points (on selected path): key shortcut: <Shift+L>
- remove loop: key shortcut: <R>

Start position (cutting contour)

Start position marks the location where cutting head begins cutting into the material and defines the location of lead-in/-out on the cutting path. Starting point can be manually changed by placing the lead-in to the custom position on selected cutting path, clicking on a set a lead-in/out on custom position button located in the cut path editing widget on the left side of the screen. The second button with letter A returns a lead-in/out to the automatic default position as set by mCAM.

> Start position can be changed only on cyclical contour (closed holes or circular trim cuts)

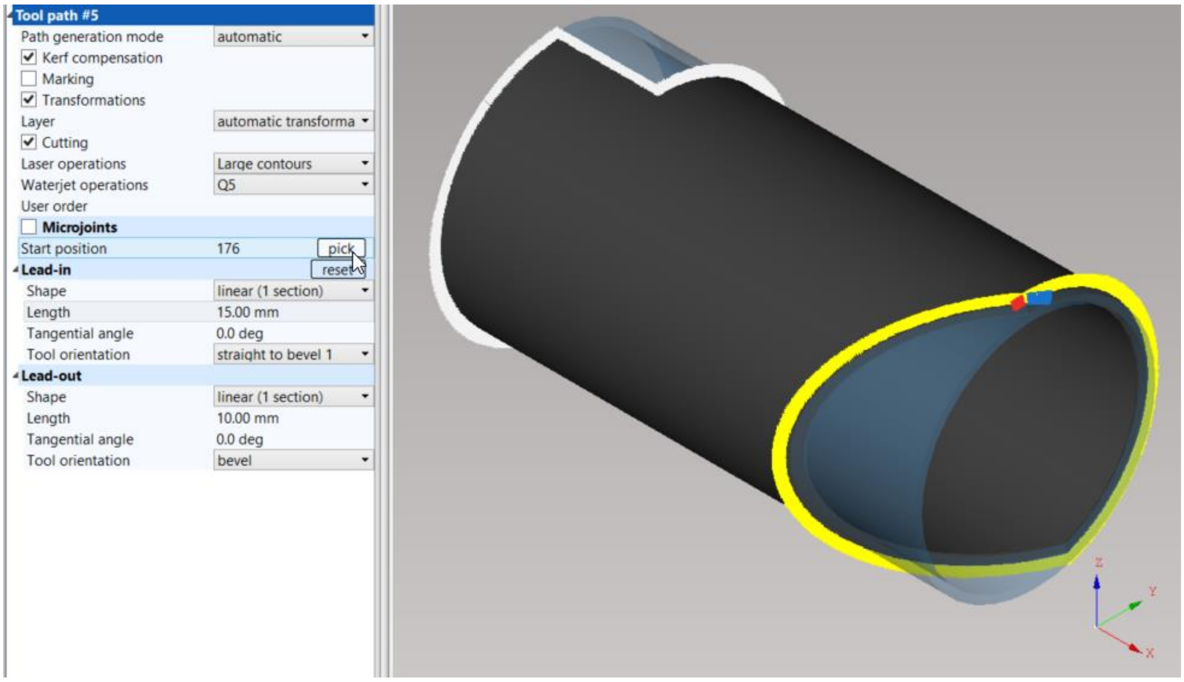


Fig. 179: Function Pick start position

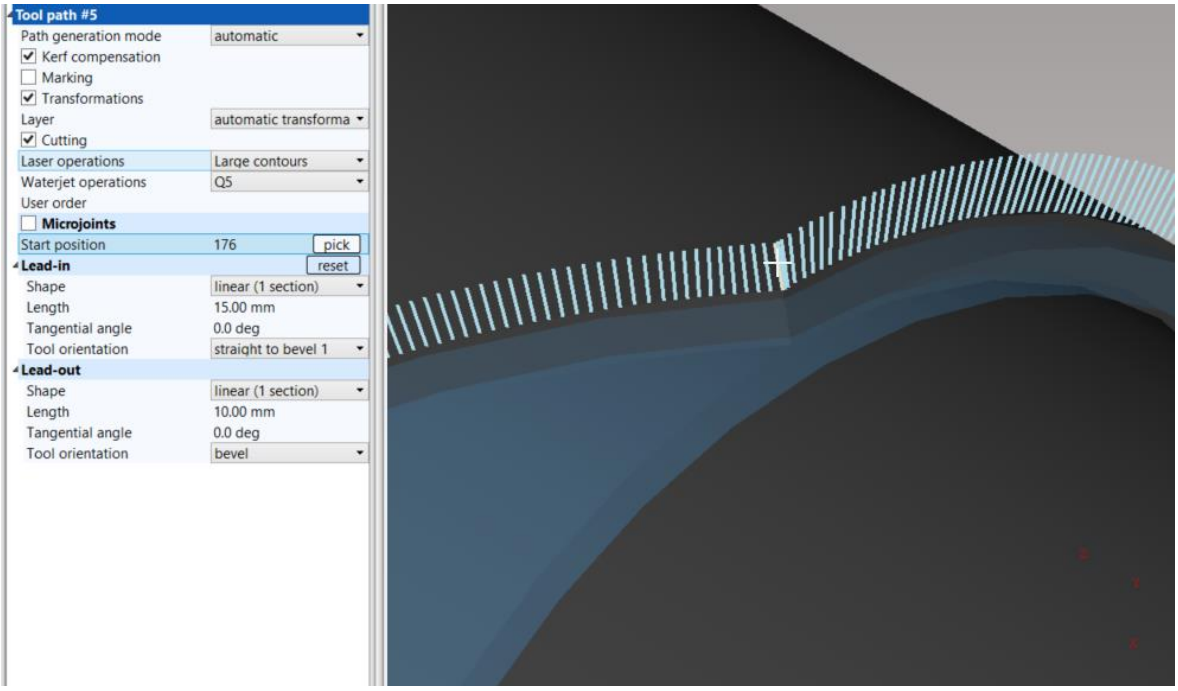


Fig. 180: Selection of cutting point – start position

Start point (CNC program)

CNC program start point is defined by the *shape type* (*circle pipe, H–beam, U–beam,* etc.) and used *Chuck location*. *CNC program* starting point is displayed in simulation of particular *CNC program* when mouse cursor is placed in the first *NC code line* (not commentaries in brackets).

In special cases when machine is equipped with optical sensor that detects the edge of the pipe (start point), it is possible to use *Startpoint X offset* if the detection was not entirely accurate. The function shifts the start of *CNC code* a bit towards the *Chuck location. The* parameters are defined for whole *Job task* in *Profile nesting properties*.

Default CNC program starting point in plate cutting is defined in *Settings – Generate – Ordering – Start point of plate CNC program*.

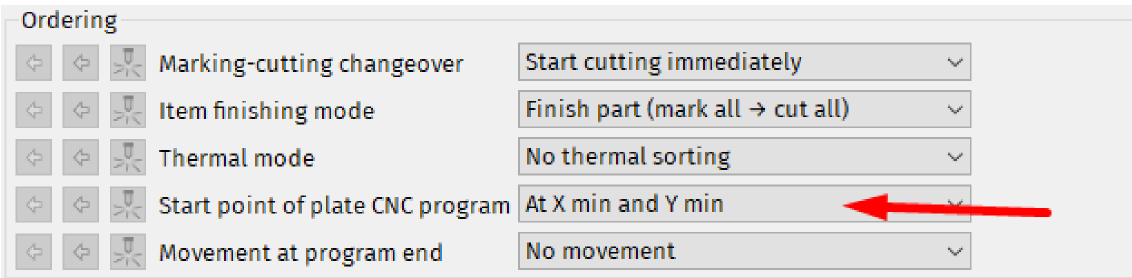


Fig. 181: Start point of plate CNC program

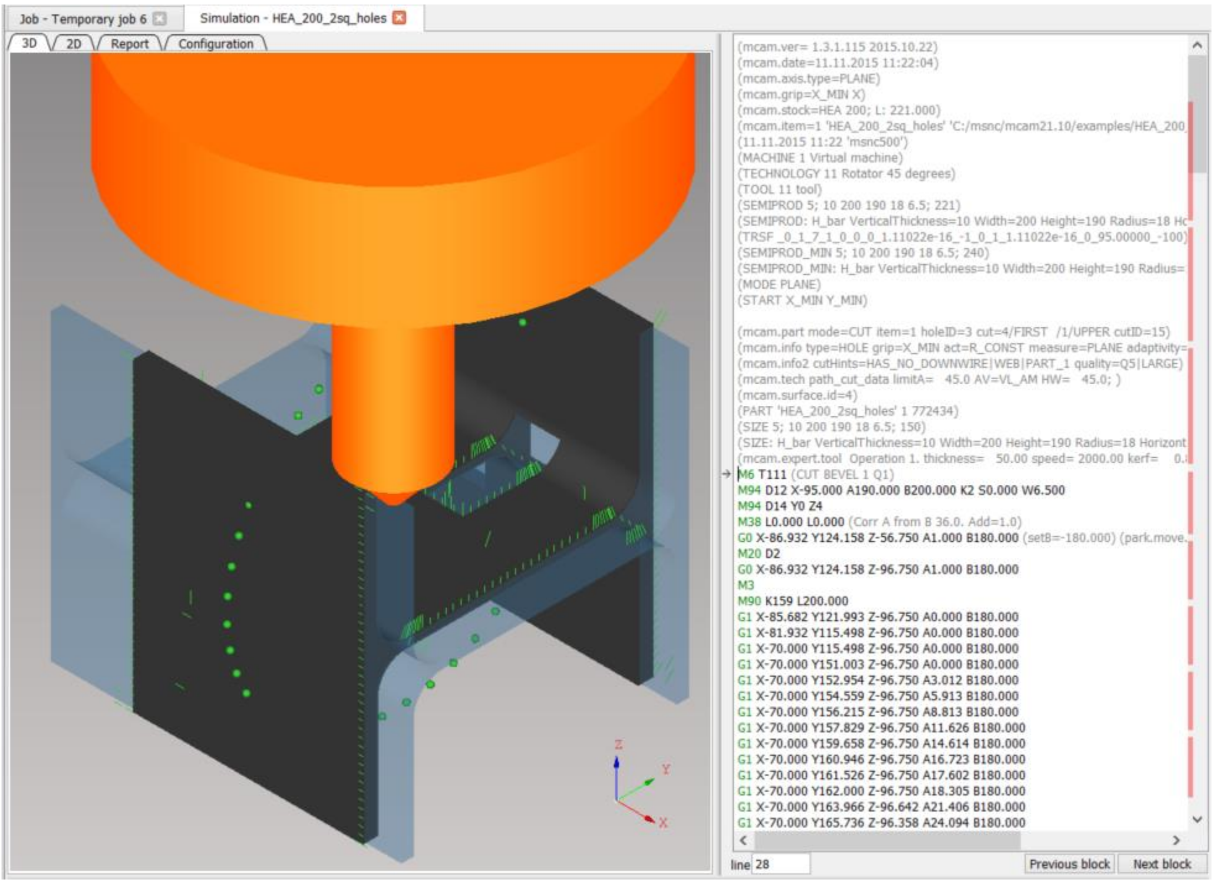


Fig. 182: CNC program starting point

Technology modifications

This section in Properties area for selected cut path contains the setting for adjusting the user speed for each cut path individually. It is set as percentage of speed that is in expert tables.

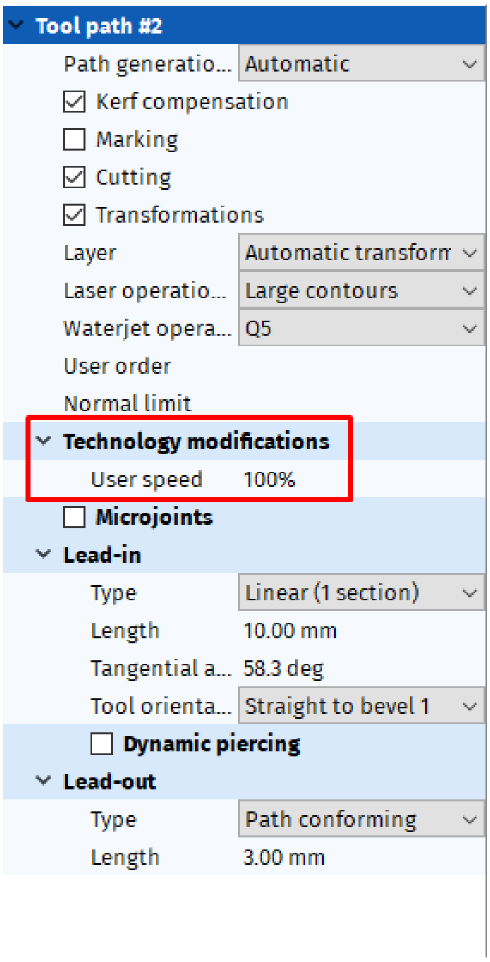


Fig. 183: Setting the user speed for selected cut

Technology setup (for individual cut-paths)

After selecting the cut path, it is possible to assign different technology and tool for cutting and/or marking or text marking. The shape has to be already loaded in the job. The option to choose separate technology and tool for marking or cutting is not available in Import tab.

1. **Technology setup for text marking –** Technology and tool for text marking is assigned in Shape tab in job in section Text marking.

2. **Technology setup for cutting –** Different technology and tool for cutting of cut path is assigned at the bottom of the properties area after selecting respective cut path.

3. **Technology setup for marking -** Different technology and tool for marking of cut path is assigned at the bottom of the properties area after selecting respective cut path.

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.

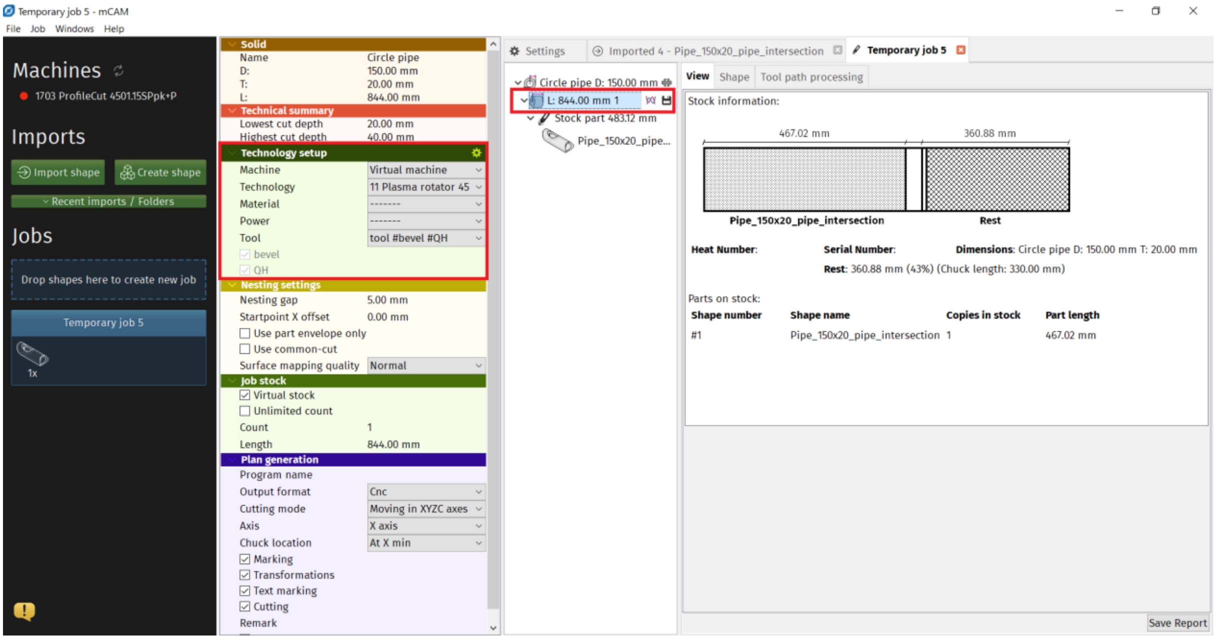


Fig. 184: Waterjet technology with respective tool is selected for cutting

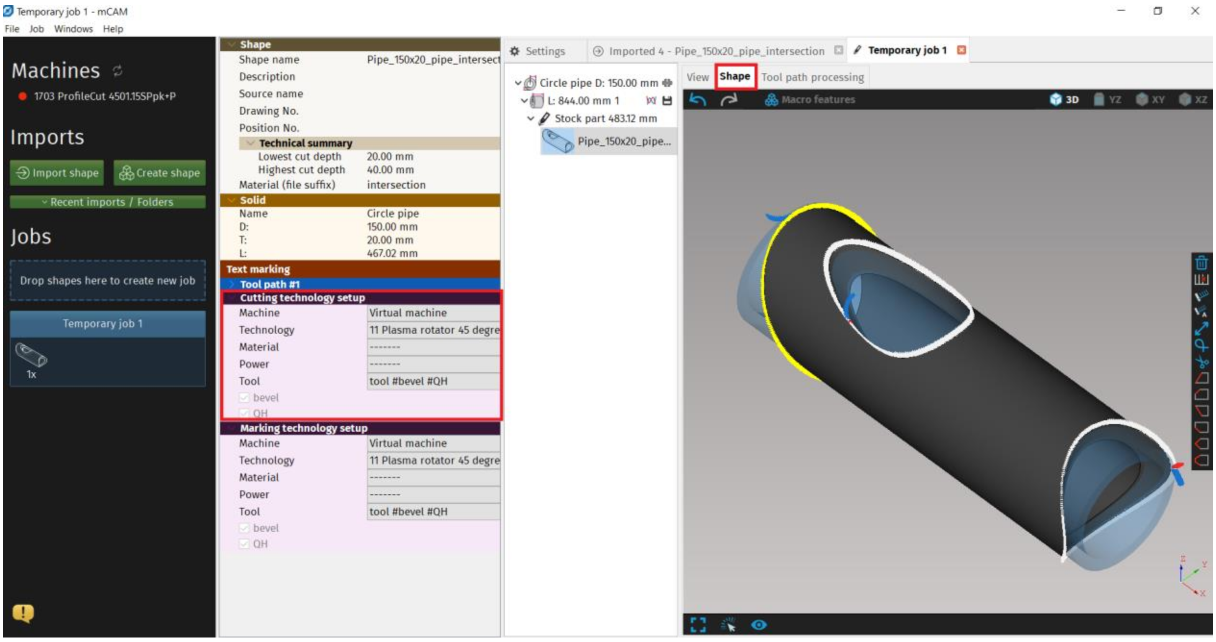


Fig. 185: Plasma with respective tool is selected for text marking in Shape tab

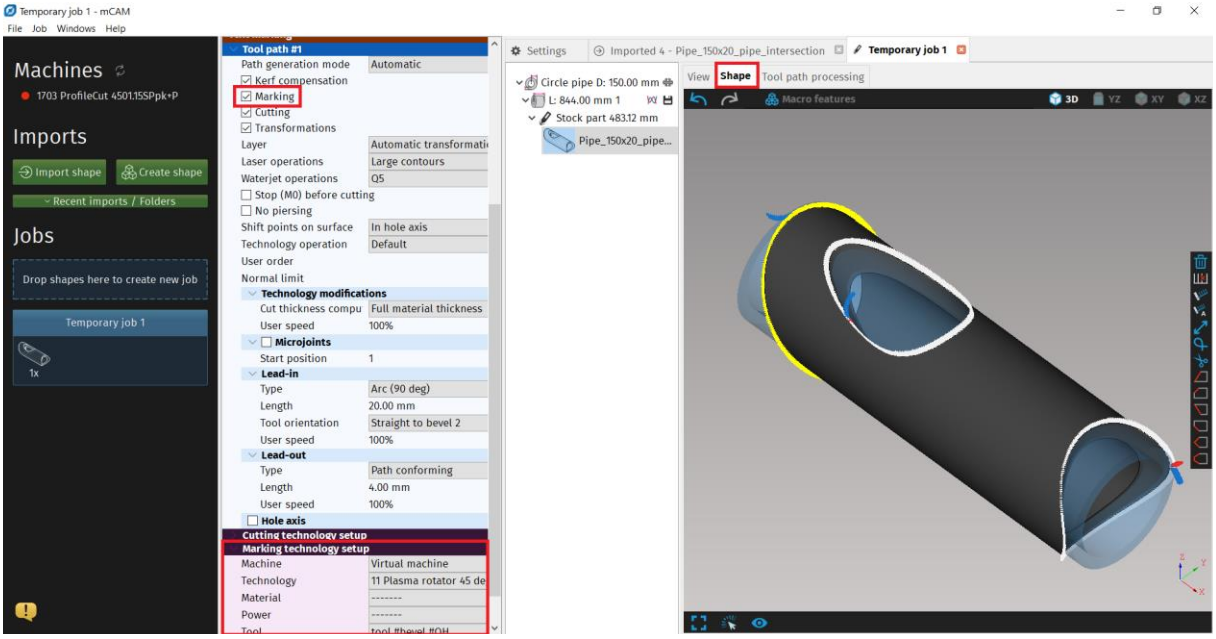


Fig. 186: Marking is enabled for selected cut path and plasma technology with respective tool is selected in Properties area

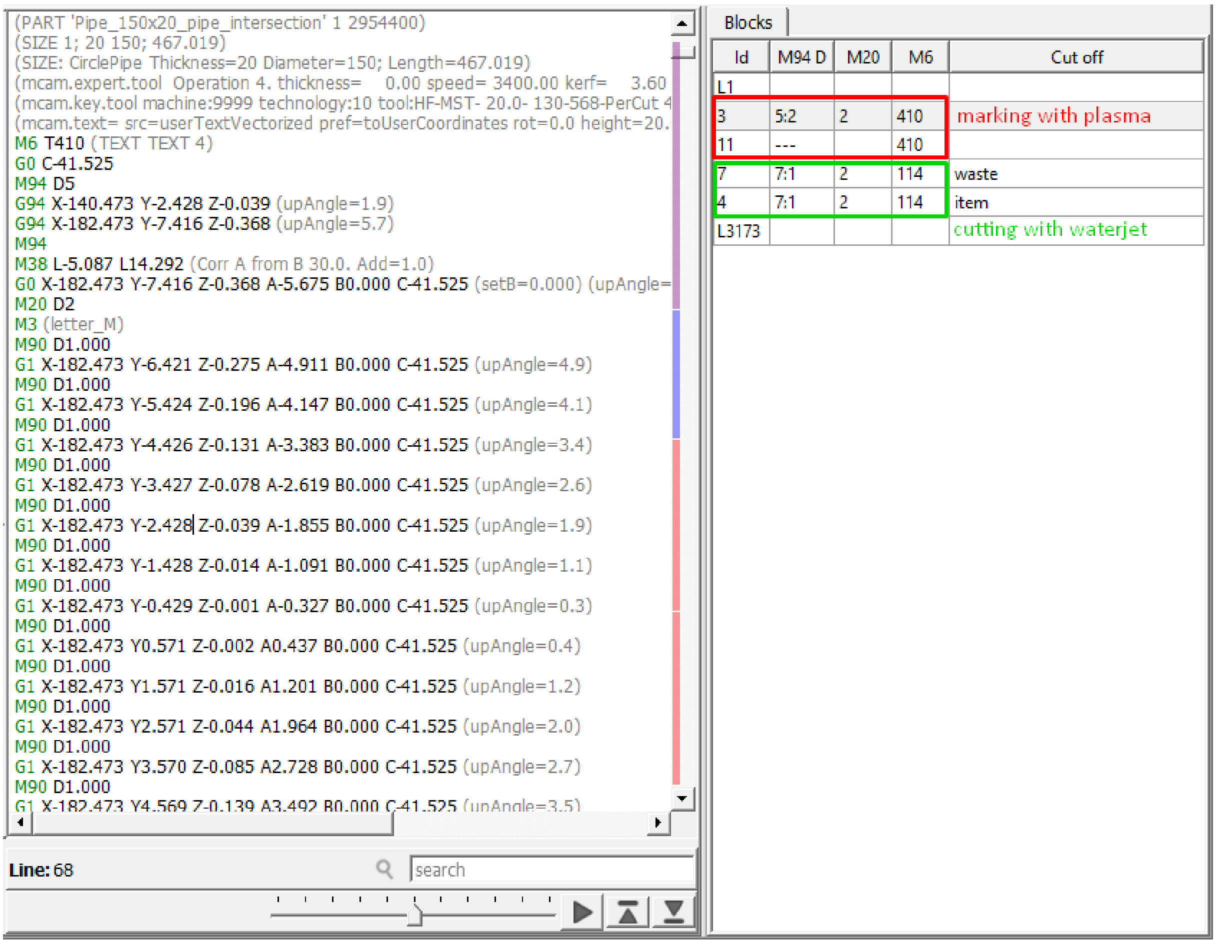


Fig. 187: Example of CNC code

Adaptive Bevel Compensation

Bevel Angle Correction in iMSNC:

The parameter adjusts the distance between the tool and the intersection point of tool axes during tool rotation. The system uses this value to calculate compensation of contour dimensions depending on the tool angle. Then the cut contour is not influenced by the cut angle. Parameter value is dependent on a mechanical structure and adjustment of the rotator. Usually, it is approximately 7-8 mm. When during bevel cutting a deviation from the desired part dimensions is observed it is possible to make corrections of dimensions by modification of this parameter.

The parameter value is obtained experimentally:

- If the part dimension during positive bevel (i.e. the torch is inclined to the left in relation to the direction of movement) is bigger than the required dimension, the correction value must be increased.
- If the part dimension during negative bevel is bigger than required, the correction value must be decreased.
- If the part dimension during negative bevel is smaller than required, the correction value must be increased.

Settings in iMSNC:

mCAM uses only parts of MSNC data for adaptive bevel compensation because it can´t differentiate between A-V, X and K cuts. For this reason, it views everything as either top bevel (V) or bottom bevel (A).

Activation of ABC correction:

In mCAM in Technology setup click Tool and choose the one that has the #ABC setting.

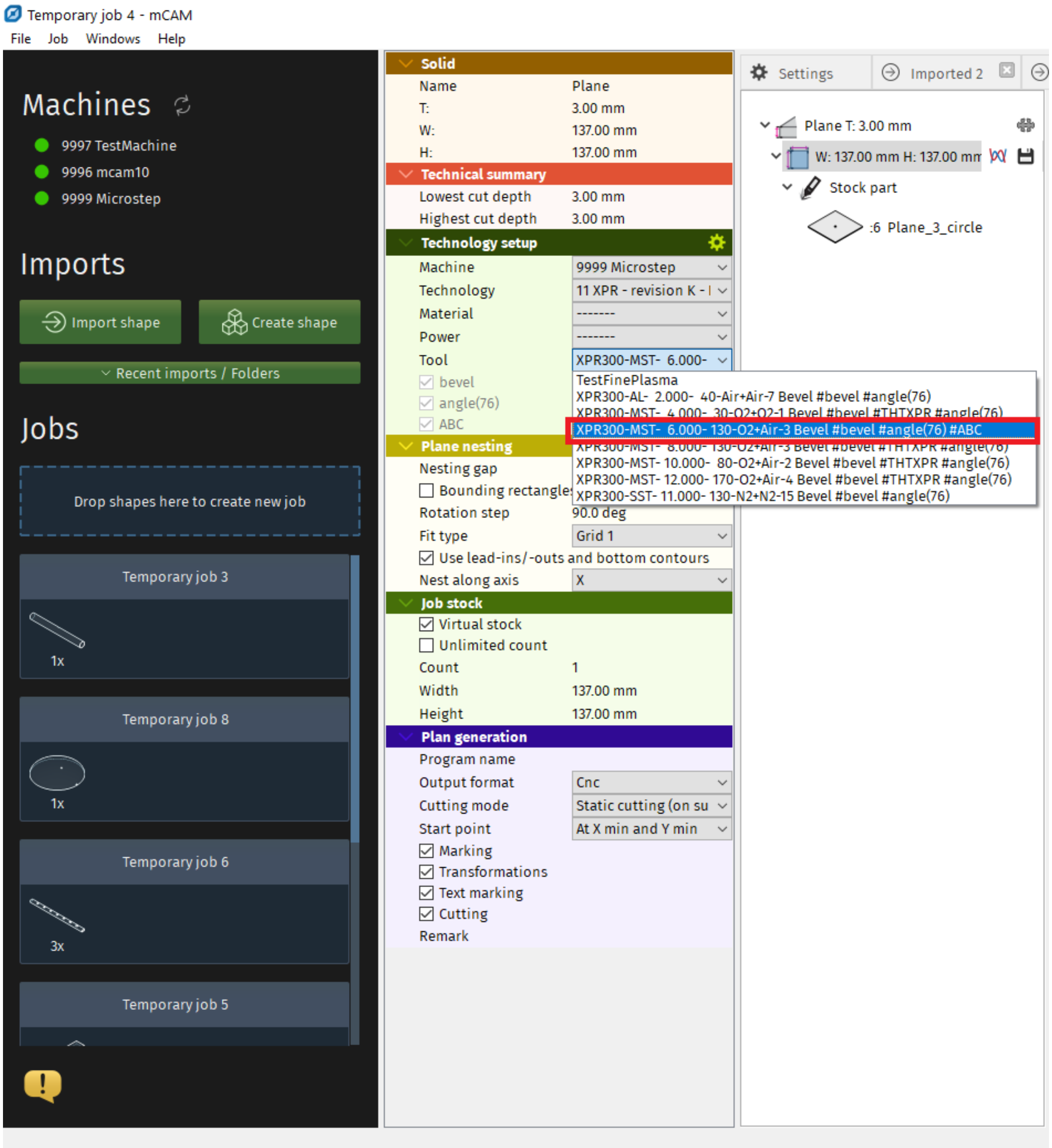


Fig. 188: Tool selection – click on the one that has the #ABC setting

Then in Settings click on Generate, find Bevel correction and choose Automatic bevel correction v1.0 (ABC).

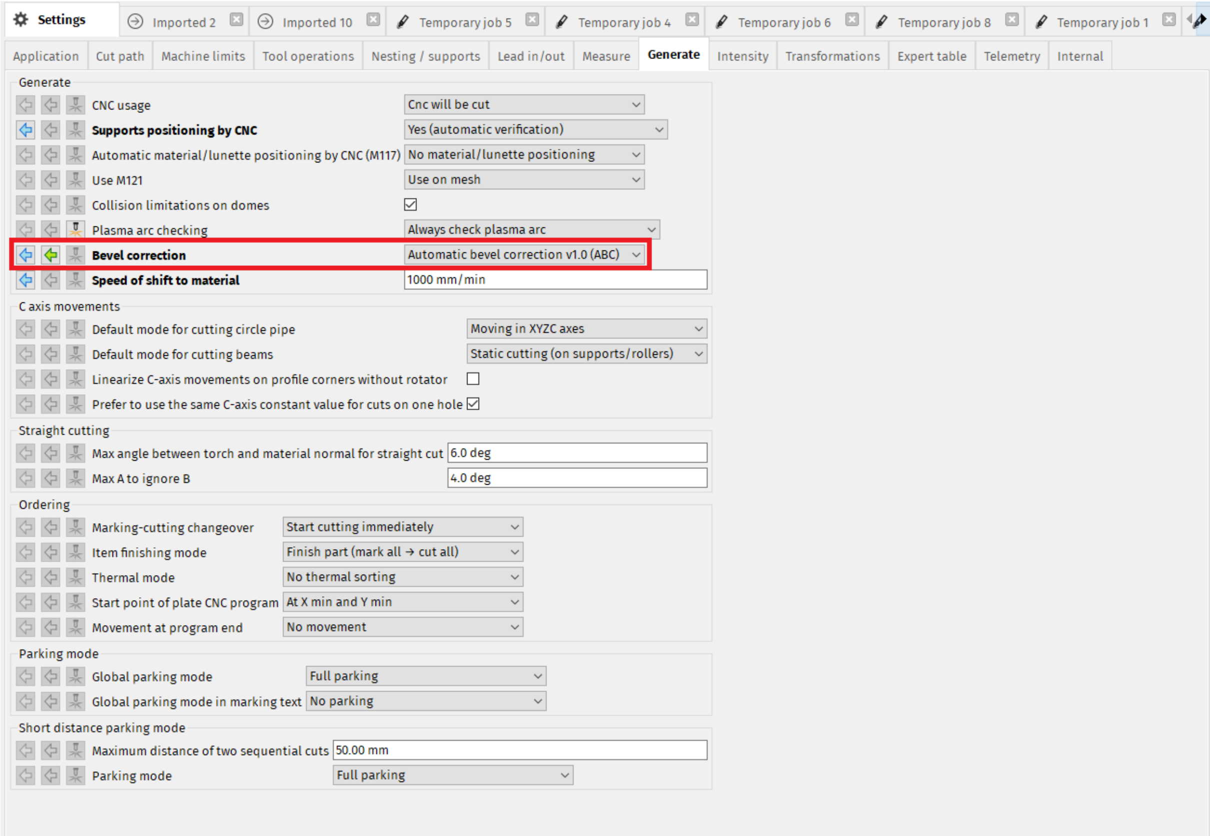


Fig. 189: Bevel correction – Automatic bevel correction v1.0 (ABC)

mCAM results review:

The results of adaptive bevel correction can be reviewed by enabling comments. In Settings, Internal section, find Generate plan comments and choose Full comments. The CNC will now include comments, such as:

-bevel_size_corr

-bevel_angle_corr

G1 X54.936 Y64.215 Z0.000 A20.539 B241.324 (A=20.5 B=-118.7; I<mT:0.53:s,cA:1.39:s,eS:1.20:s:d,cS:1.41:s:d>; B=-118.7; length=0.15; deep=3.19; speed=136.9%; kerf=3.0; **bevel_size_corr**=0.8; **bevel_angle_corr**=0.6; kerf-C=LINE_TOUCH; dir=FaceAxis; CP=10; operT=BEVEL_A)

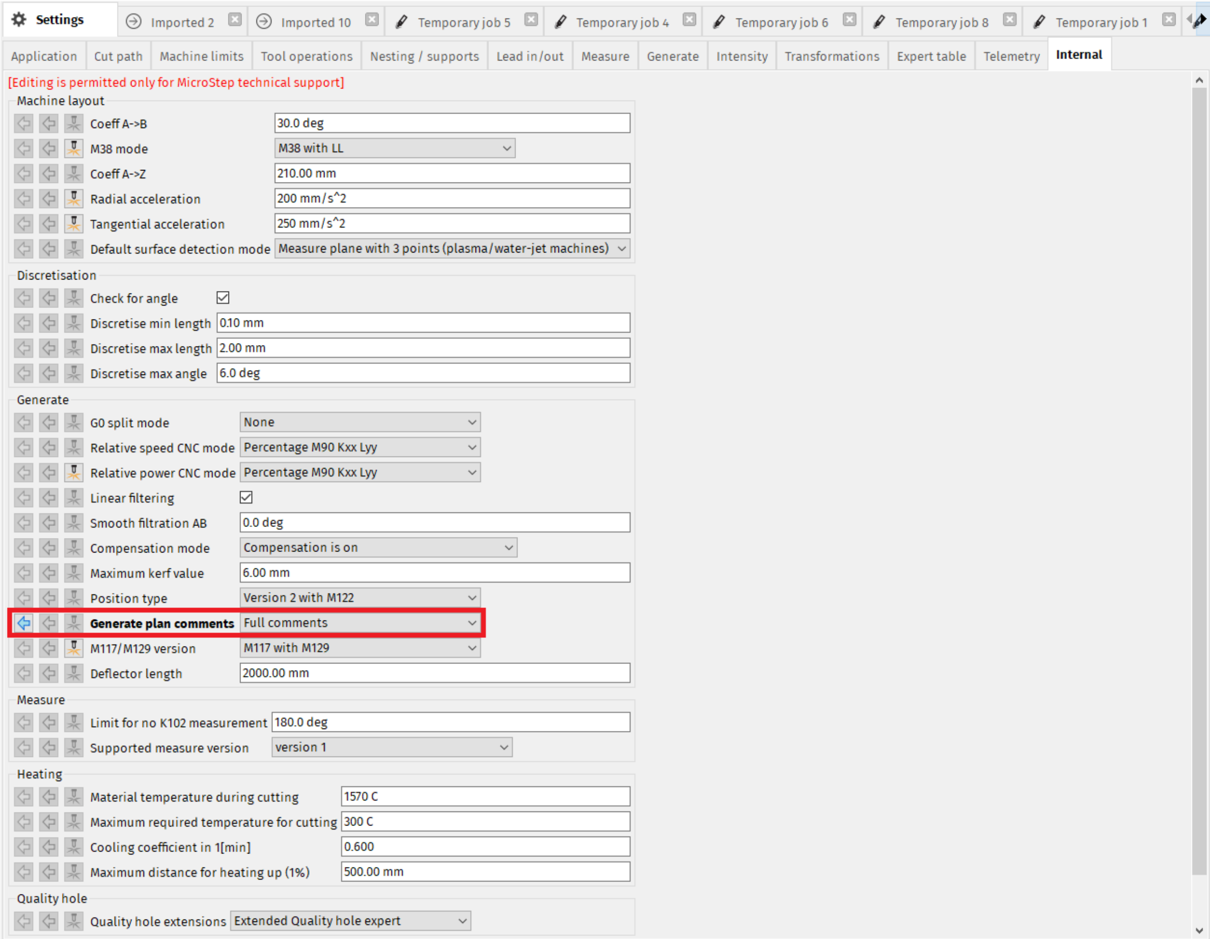


Fig. 190: Generate plan settings in Internal section of mCAM Settings

Method of computing ABC:

- The beveling head is rotated around the top point even for the bottom parts of the X and K cuts.
- The positive angle always means increase of bevel (even for top and bottom bevel).

To 15 degrees – the correction is not performed

For 25 degrees – data <15,35> from iMSNC are being used

For 45 degrees – data <35,55> from iMSNC are being used

Above 45 degrees – extrapolation is used

(This is how Licris does it)

mCAM data can be checked in mCAM application console in View. The angles in mCAM and iMSNC hold the same value. In mCAM size corrections are recalculated into the change of tool diameter.

Text marking

Text marking is a technological operation used for permanent inscription of arbitrary text, e.g. production batch ID number or outlines of contours onto material surface. For marking, energy beam cutting technologies utilise reduced power to affect only the material surface. The marking operation is enabled by using the *Text marking* option. Marking function can be enabled in the same section as *Chuck location, Transformation* or contour *Marking*.

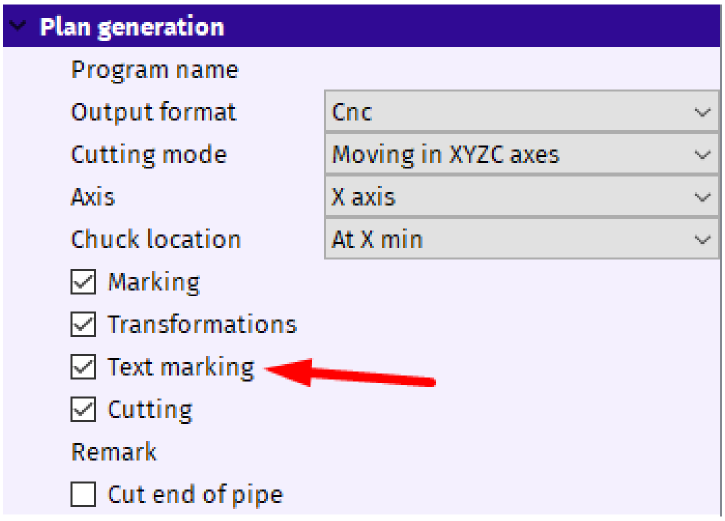


Fig. 191: Text marking generation – enabled

Adding a text, changing its position, size, and other parameters is really simple. All individual options are in *Properties menu* of selected part in *Import task*. To add the text, click on the Add button next to Text marking in Properties section. To delete the text, click X button in the upper right corner of Marking element section.

Each marking element can have several parameters defined:

- **Marking Type** – can be defined by three methods:
  - → *Text - vectorized (G code):* this method splits marking text into small sections of vectorized paths and marking is executed by the partial G-code instructions (G0 and G1 – as standard cutting instructions).
  - → *Text - ASCII writer:* when using this method, the section of marking instructions (G0, G38, G39) is generated in the CNC code according to marking text. This method is suitable only for machine equipped with special ASCII marking unit.
  - → *QR code, Data matrix, Pixel matrix:* the section of marking instructions (G38, G39) is generated in the CNC code according to marking text. This method is suitable only for machines equipped with special marking unit.
- **Font type -** this feature in Text marking allows you to choose between two font types:

→ *Disjoint* – also available in previous versions of mCAM. The text is written one letter at a time, with the tool stopping and traversing between the letters.

→ *Continuous* – the text is written in one continuous movement, without the tool having to stop and move to the next letter.

- **Text will be entered on machine***:* the marking text intended for marking is defined after loading CNC code to a machine by an operator. To choose this option click on the checkbox *Text will be entered on machine* under the Marking type after choosing the preferred source (ASCII writer, QR code, Data matrix, Pixel matrix).
- **Face** – defines on which face the text should be placed (individual options depend on actual shape type)
- **Placement (only for domes)**

→ *From top* – places text from top view

→ *From side* – places text from side view and rotates it around axis

- **Position** - position can be picked manually by clicking on a 'Pick position' button and clicking on a place on a part where the text should be placed or by selecting concrete face and set X and Y coordinates. User can also adjust rotation of text.
- **Font size [mm] –** defines size of a font.

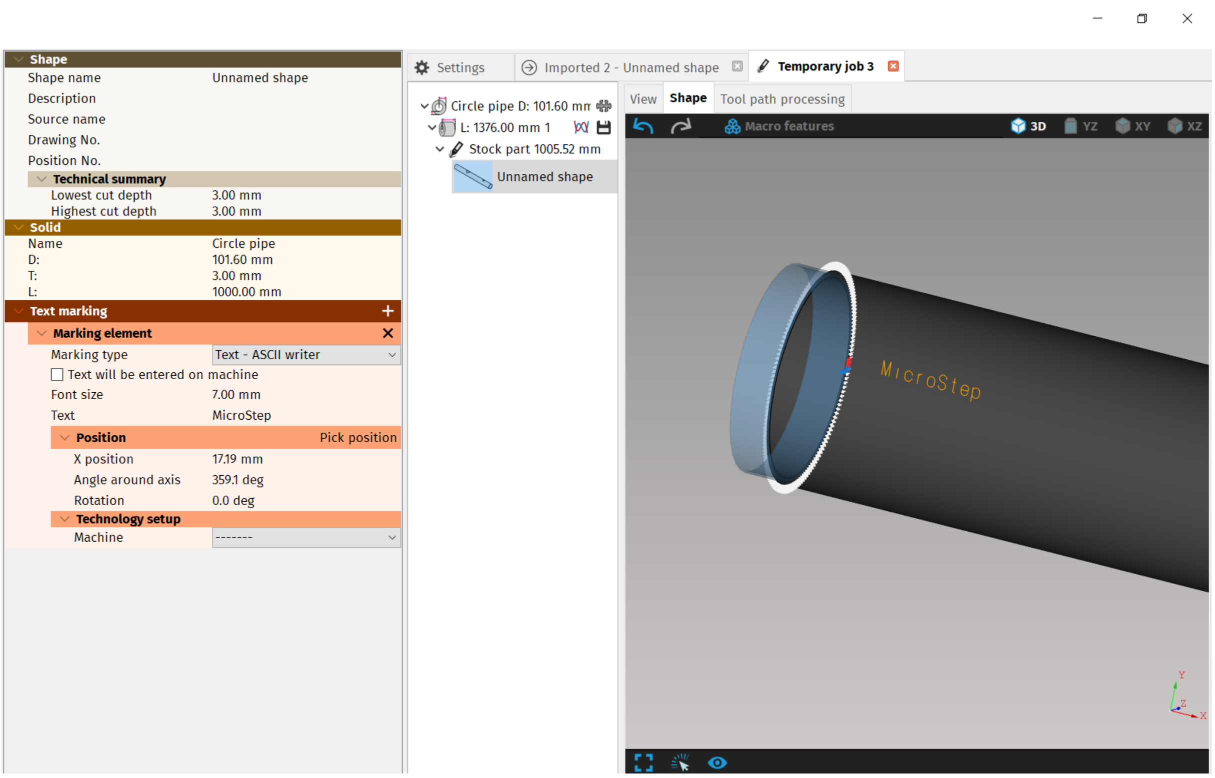


Fig. 192: Text marking element

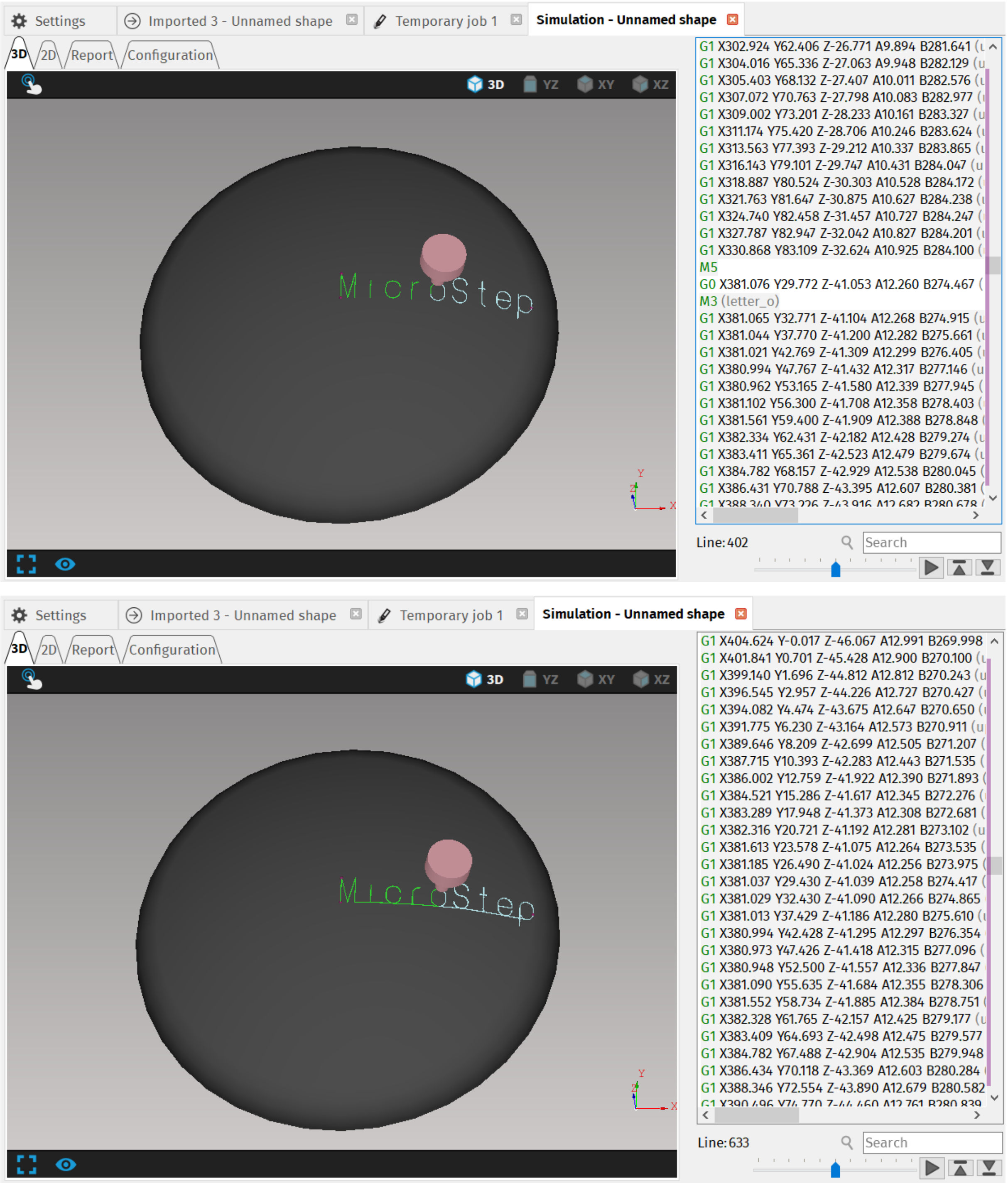


Fig. 193: Disjointed and Continuous Font type used in Text marking on dome

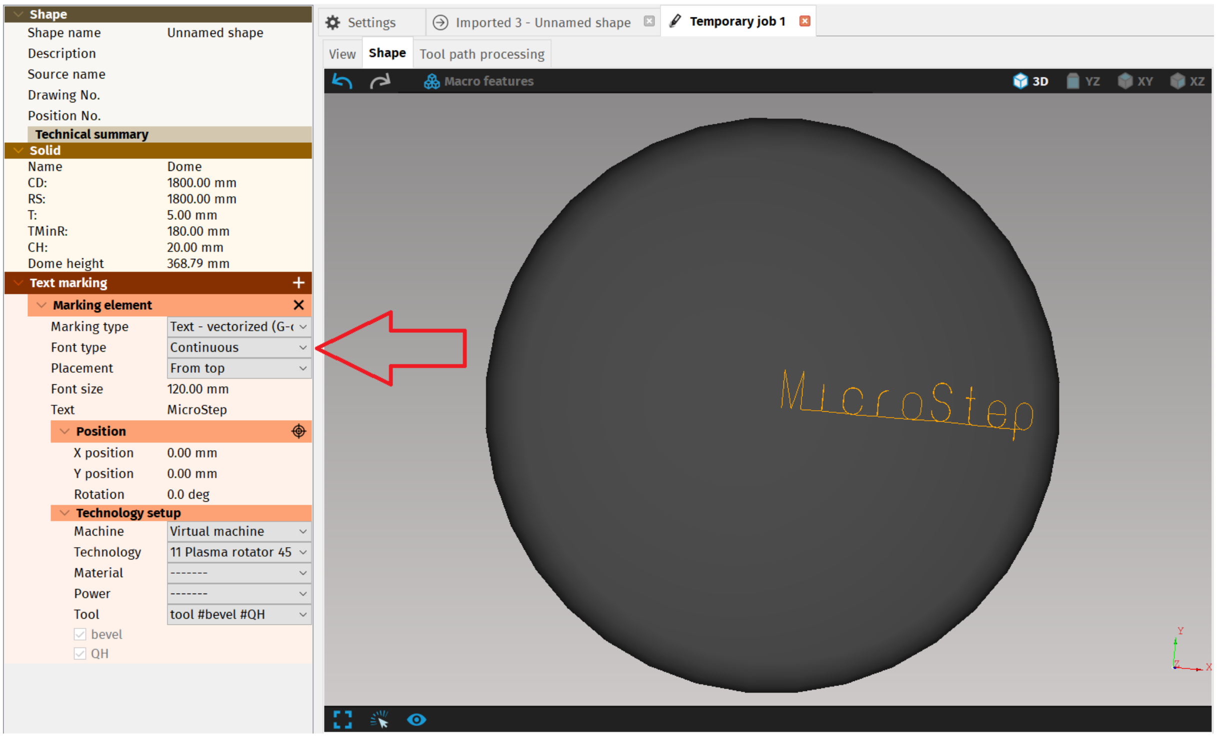


Fig. 194: Font type in Text marking section of Properties menu

Tool operations

Tool operations (also known as technological groups of cutting parameters) refer to columns of tool parameters in iMSNC. In plasma cutting machines operation 1 type/operation 1 quality are related to straight cut parameters. In water – jet machines first row is related to Q1 (first column) cutting parameters. In laser machines first operation type row is related to Large contours.

Keep in mind to choose particular machine and technology during the setup *Tool operations*.

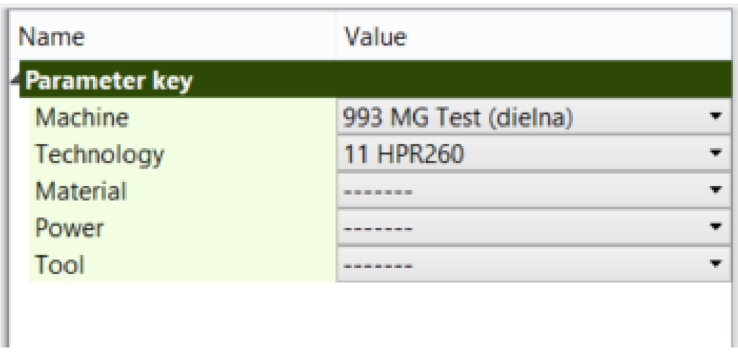


Fig. 195: Machine and technology selection

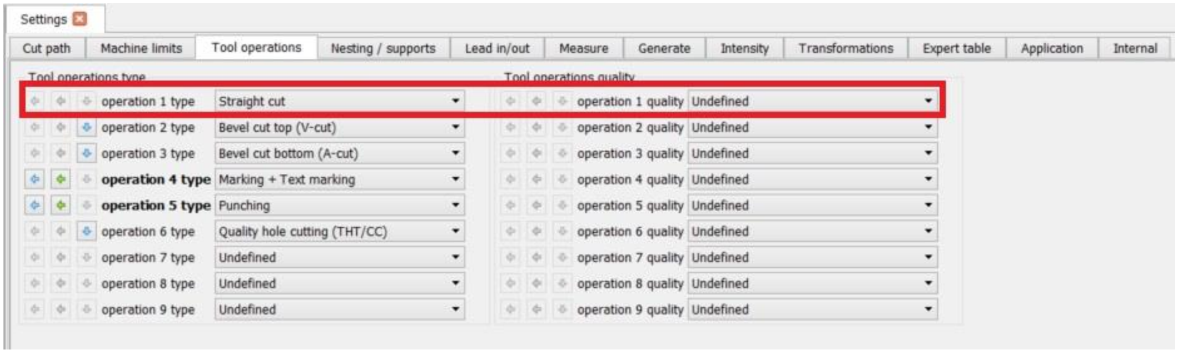


Fig. 196: Standard plasma tool operations setup

Every tool number (e.g. for plasma cutting T111) that is used in every CNC code is created from two numbers: operation (technological group) number and technology number. First number of the tool number (T111) defines operation (technological group of parameters) that should be used for particular cut, in this case cutting parameters from straight cutting. The other two numbers (T111) define the technology code that defines used technology. Plasma cutting technology on MicroStep machines standardly uses technology number 11 (laser 17, water jet 14, oxy fuel 12, driller 15).

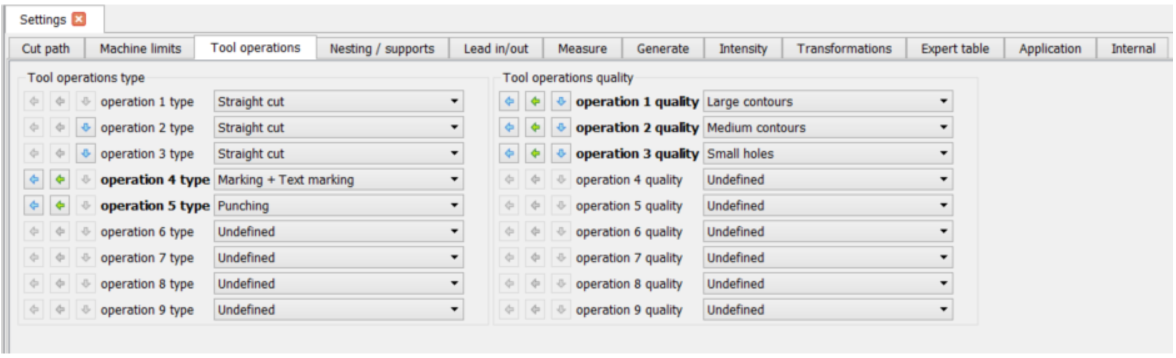


Fig. 197: Standard laser tool operations setup

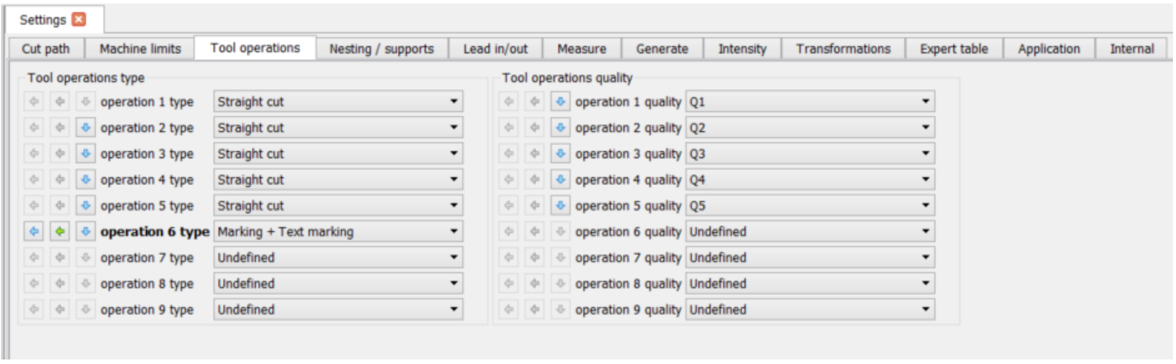


Fig. 198: Standard water – jet tool operations setup

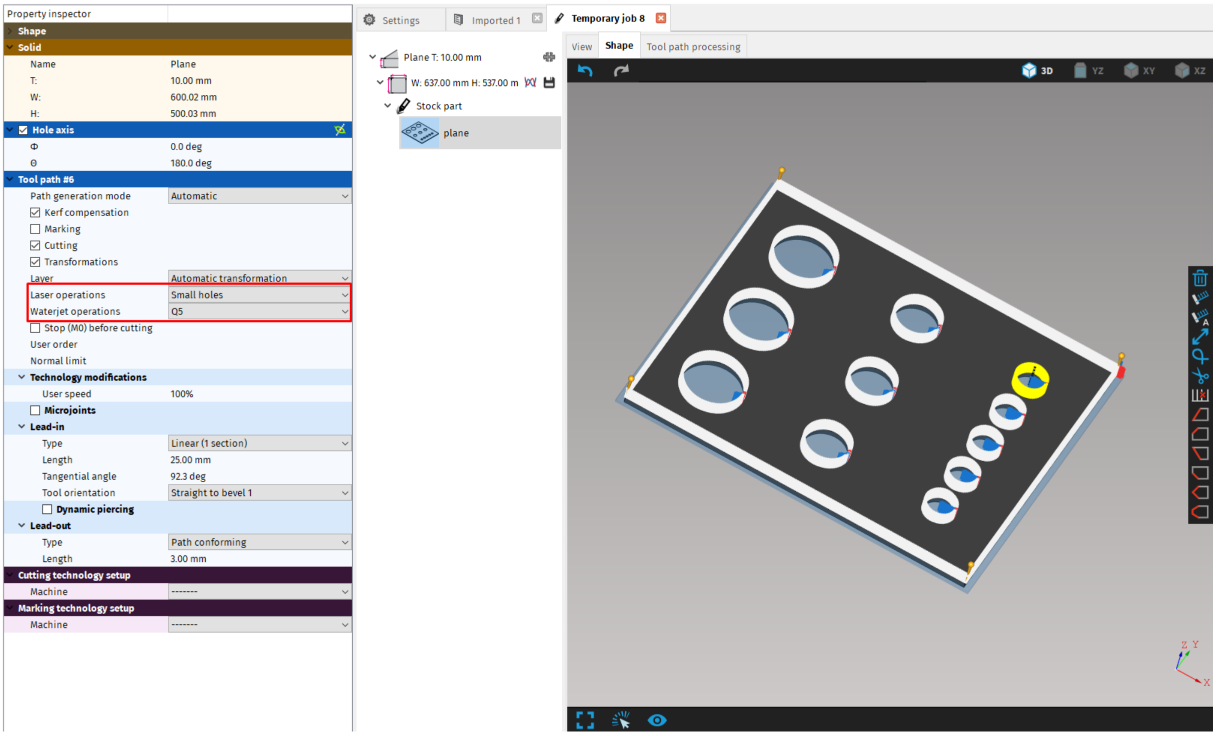


Fig. 199: Tool operation settings (for laser or water jet cutting)

Transformations

Transformations are used when cutting path (especially circle holes) is supposed to be processed by some special feature or technological operation such as drilling, tapping, punching, marking or quality hole cutting. Transformation parameters define a class of holes to be transformed and the way of processing classified holes.

Circular holes are very common shapes found in cutting programs and unlike other types of holes they can be processed in several different ways:

- Drilling the holes provided that the machine is equipped with a drilling head using drilling cycles
- Punching of small holes (for subsequent operation – manual drilling)
- Quality hole cutting (True Hole Technology and Contour Cut)

Setup transformations

Transformations are defined in Settings – Transformations. Setting up a transformation rules are described in more detail in section *Setup transformations*.

Transformations are set for whole application (not for particular machine or technology), therefore it is not necessary to select any machine, technology or tool. All transformations can be edited in program configuration level.

Transformations are processed during generation of CNC program, therefore user is able to see whether transformations were applied as expected only in cutting simulation.

Transformation application

Transformations are automatically applied on cutting contours that match with defined transformation rule, unless transformation for particular cut is completely disabled, some other transformation is forced on it or transformation rule that matches the cutting contour is completely disabled in Settings – Transformations. All transformations on cut are turned on by the default.

Note: When applying quality hole transformation, selected tool must be intended for THT and for the exact same thickness, as is the thickness of the material.

Force transformation without condition matching

Any transformation can be manually forced to be performed on any cutting path, even if this feature does not match any transformation condition in particular transformation rule (when rule is enabled in global).

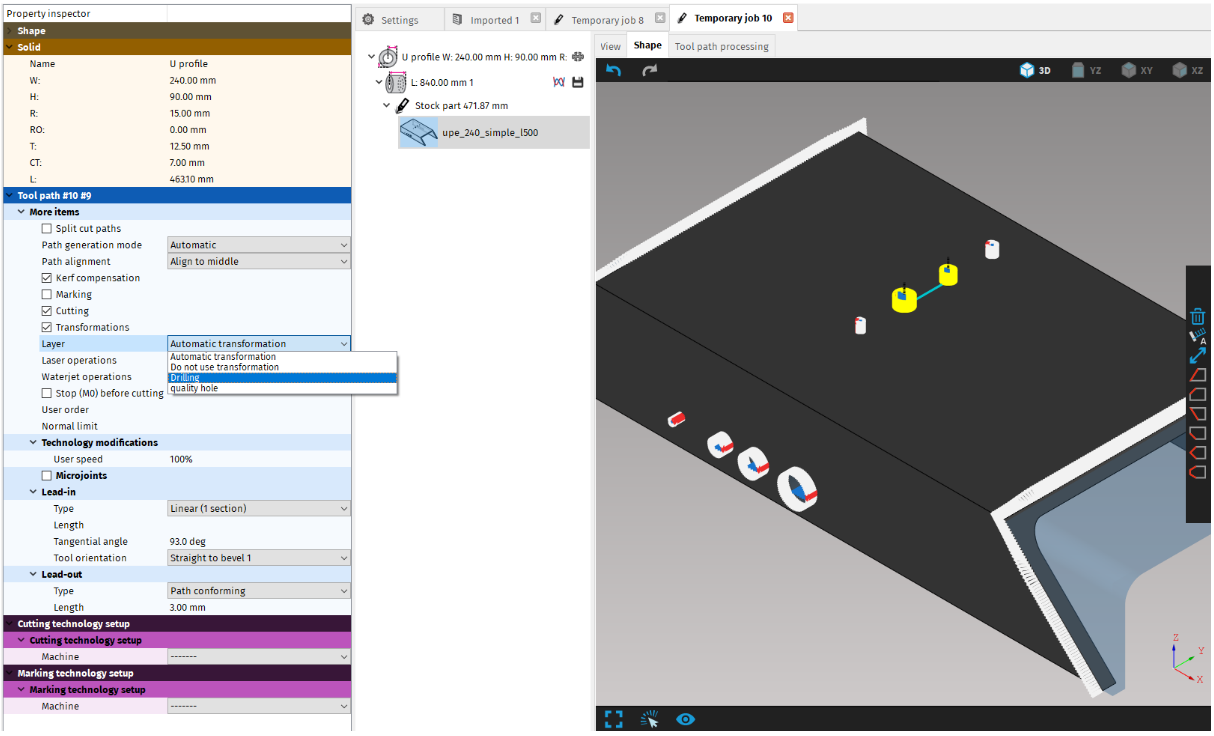


Fig. 200: Forcing transformation without condition matching

Disabling transformation for matching cut path

If some feature matches the transformation condition but user does not want to perform related transformation tasks on this particular cut path, all transformations for that particular cut path can be completely disabled by unchecking Transformation's checkbox in properties of selected cut path.

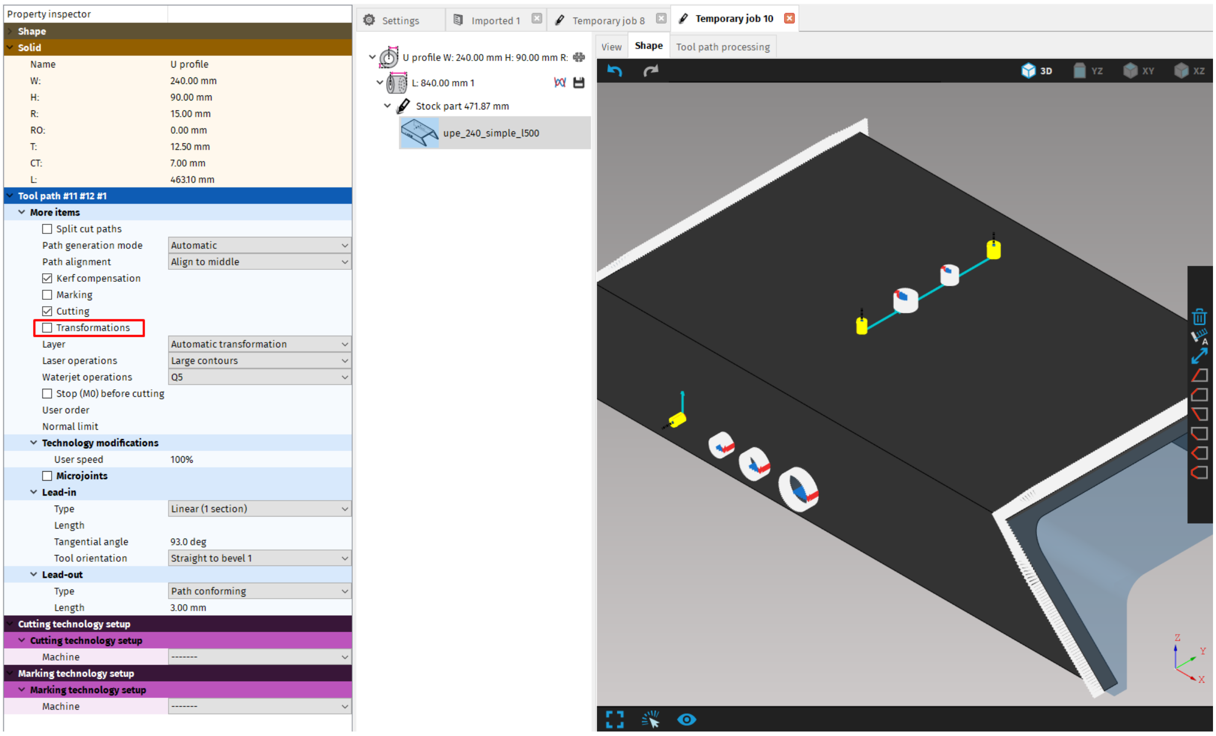


Fig. 201: Disable transformation for matching cut path

Welding preparations

mCAM 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.

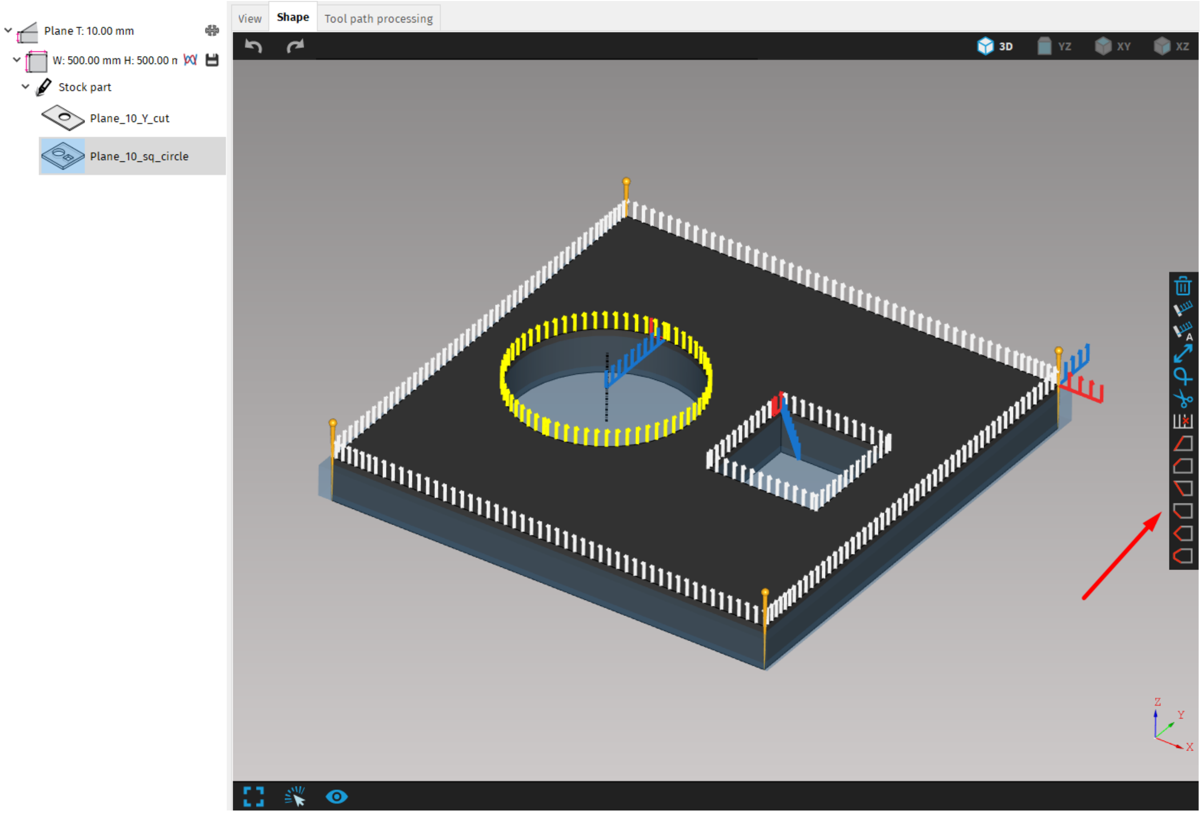
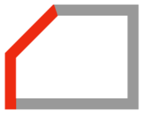
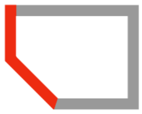
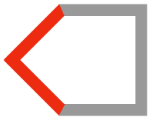
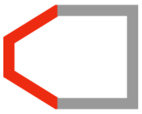
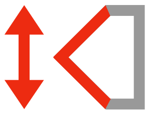


Fig. 202: Panel with various options including welding preparations shows up on the right side of the screen when the cut path is selected

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

Name		Parameters
	V welding	angle top
	Y welding	angle top, nose height
	A welding	angle bottom
	L welding	angle bottom, nose height
	X welding	angle top, angle bottom, positive height
	K welding	angle top, angle bottom, positive height, nose height
	X - variable nose height welding	angle, offset

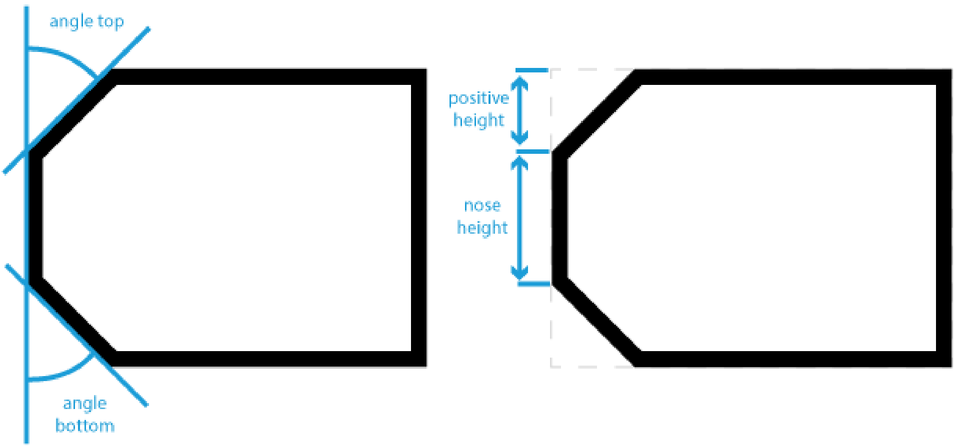


Fig. 203: Weld preparation parameters

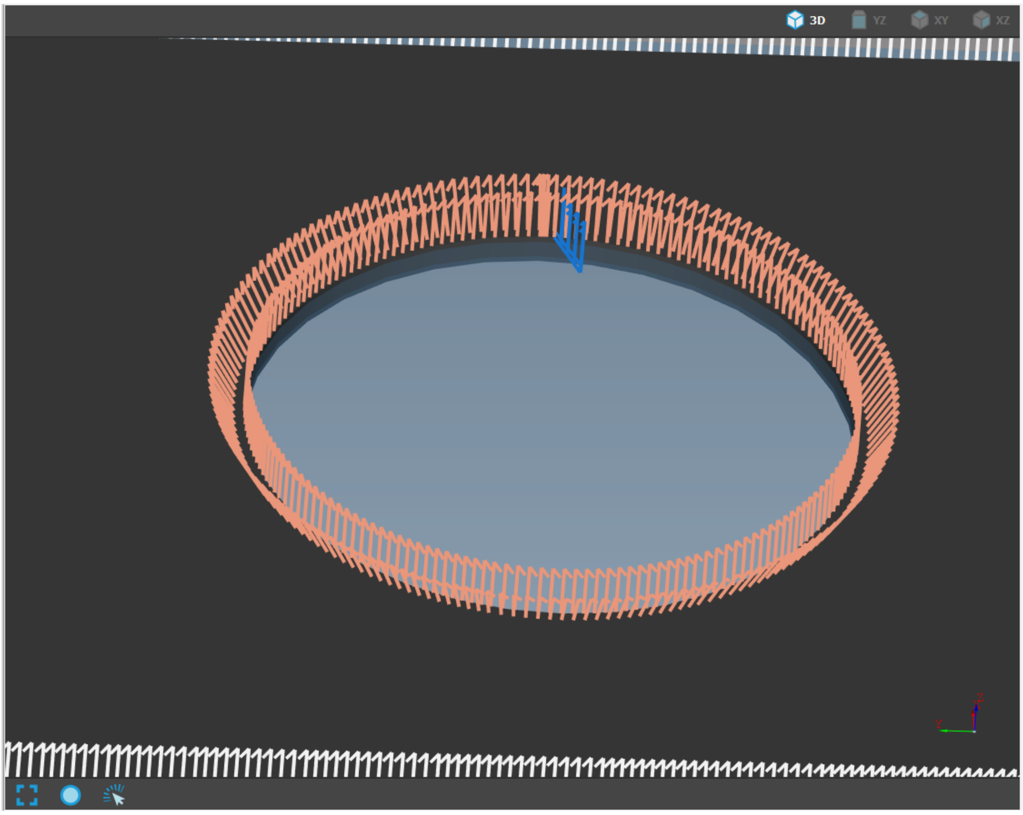


Fig. 204: Hole with Y welding preparation

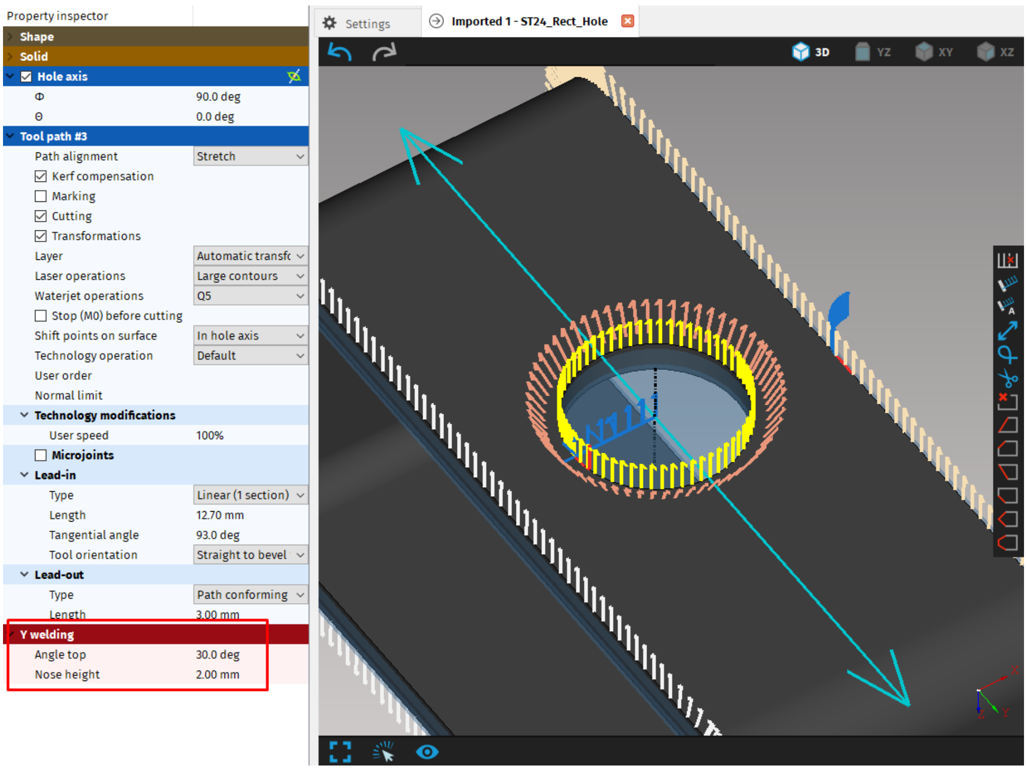


Fig. 205: Welding settings in Properties area

Automatic Welding preparation

This function enables imported shapes to have automatic welding preparation. It can be found in *Settings→ Transformations.* Then in *Conditions*, click the *Add condition* button, choose shape (e.g. plane). In the *Effects*, click *Add effect* button, a pop-up table shows with list of available effects. Choose *Welding preparation*.

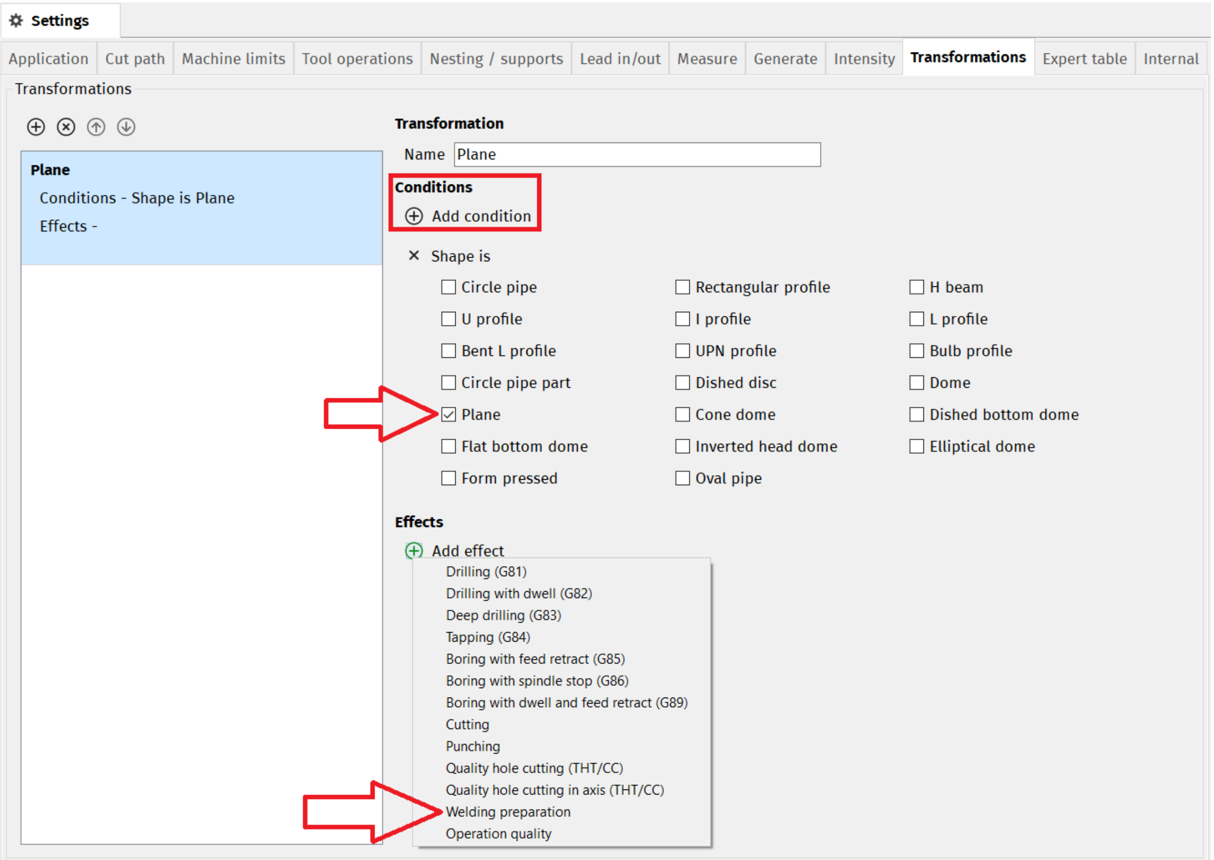


Fig. 206: Welding preparation in Transformations section of mCAM Settings

The effect can be further customized. The Welding type can be changed, Angle top and Geometry type can be adjusted and an option to Limit points to maximal material angle is available.

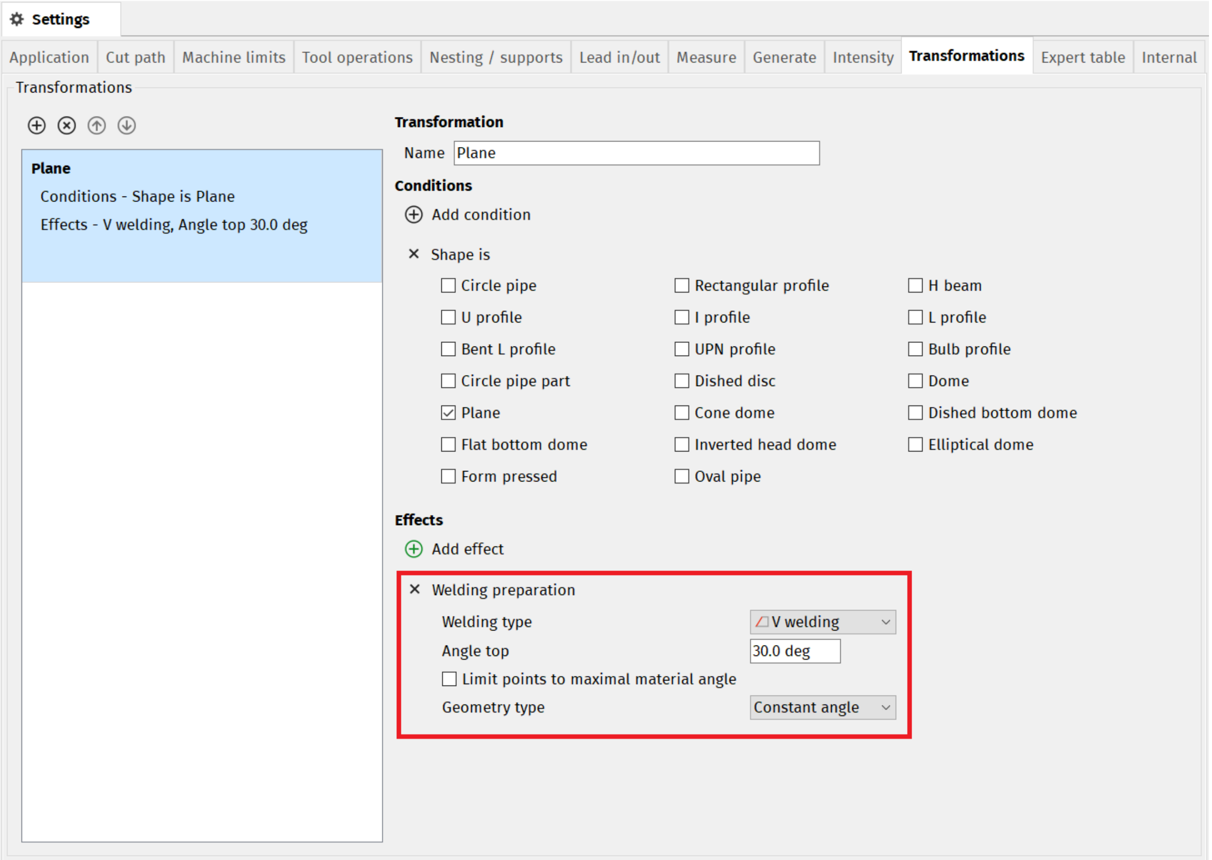


Fig. 207: Further customization of Welding preparation

The whole contour will now have all of its edges beveled in preparation for welding. As long as this function is enabled, all imported shapes that has been checked in Conditions (e.g. plane) will automatically have welding preparation.

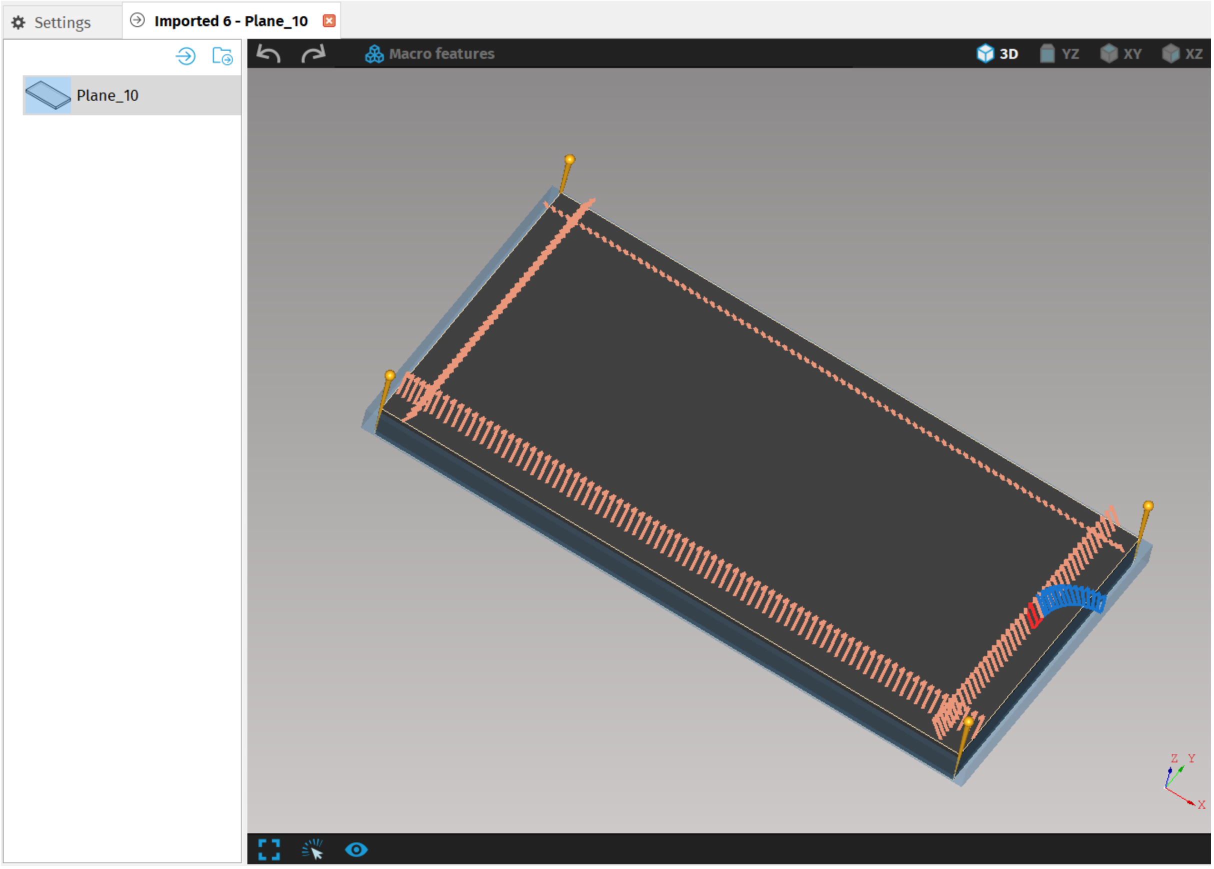


Fig. 208: Imported shape – Plane with welding preparation

Welding preparation for Hole

If a shape is chosen (e.g. plane) and only the hole needs to have welding preparation, the user can go to the *Conditions*. Click *Add condition* button, choose shape (e.g. plane), click *Add condition* button once more and choose *Hole*.

The diameter and thickness ratio of the hole can be changed. By adding the same effect, Welding preparation, now only the hole will have automatic welding preparation every time the same shape is imported and used for a job.

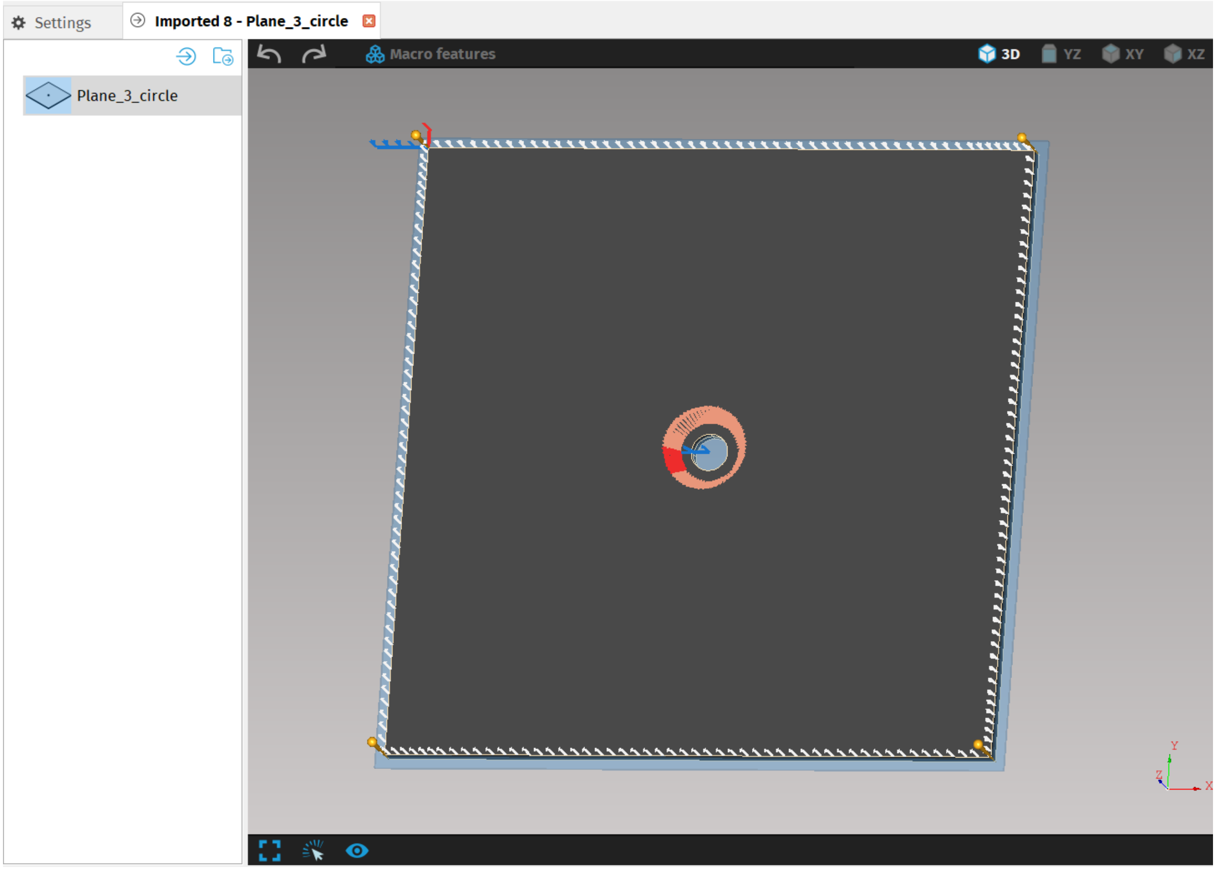


Fig. 209: Automatic Welding preparation for Hole

Circle pipe with welding denotation

Circular pipes manufactured by welding of rolled sheets may contain the welding line, where cutting of holes and other features may be prohibited for technological reasons.

When users design circle pipes in CAD, they can draw the welding line as a shallow groove into the STEP file. mCAM will then respect position of this welding line in the nesting process and pipe rotation will be disabled. When the pipe is loaded into the machine with welding line on top, produced parts will have the welding line positioned as was designed in the STEP file.

Imported STEP files which contain circle pipe with welding line, now have a **"Welding line…detected"** in the property inspector in **Technical summary** section. This shows the user that mCAM recognized a welding line in the STEP file.

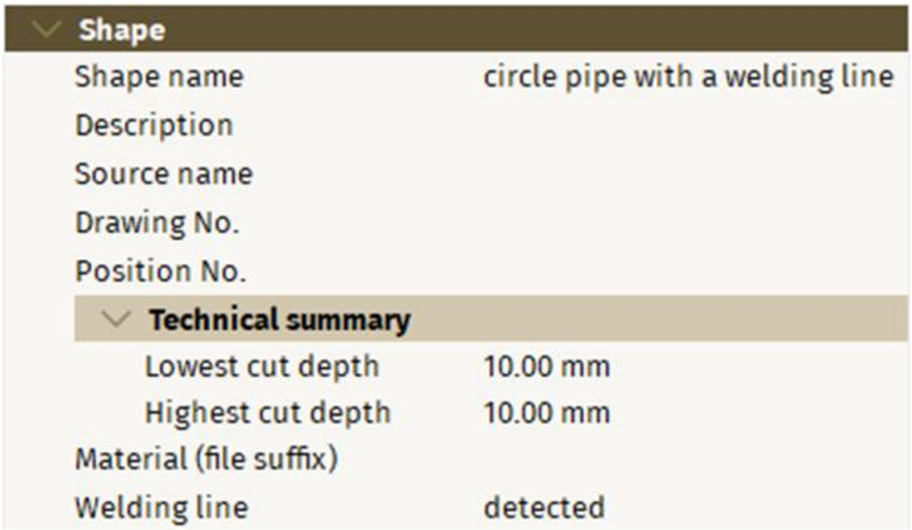


Fig. 210: Property inspector, Technical summary section

The location of the welding line is marked by a groove parallel with the pipe´s axis. **This groove cannot start or end at the edge of the pipe.**

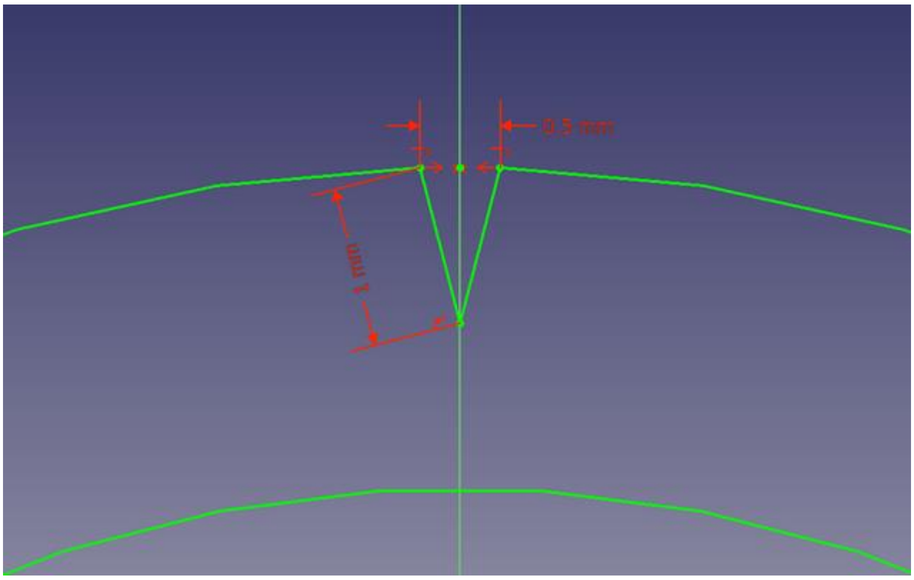


Fig. 211: Groove profile on the outer surface of circle pipe

Min. groove depth can be 0.5 mm.
Max. groove depth can be 1.5 mm.

Length of the groove should be more than 5mm and there can be at most one groove per part.

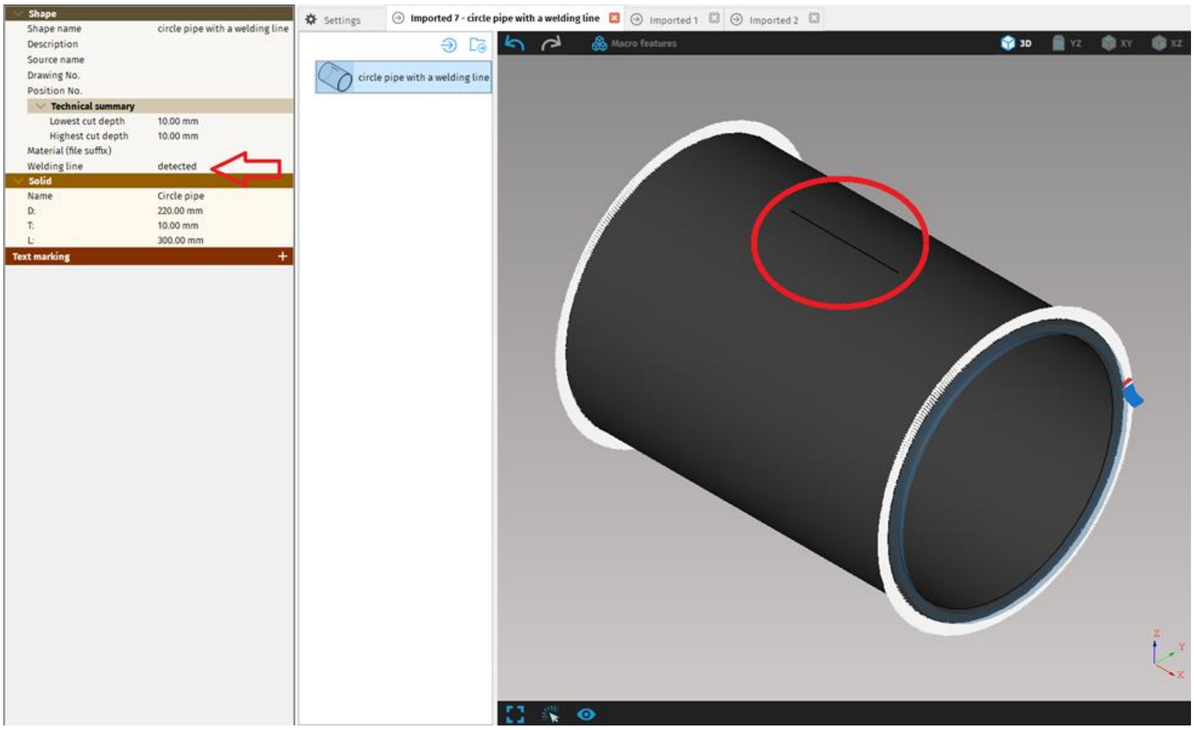


Fig. 212 Circle pipe with groove marking the welding line

It is important to load semi-products into the machine with the groove facing the top, because during the nesting process, the parts are oriented so the groove is always on top.

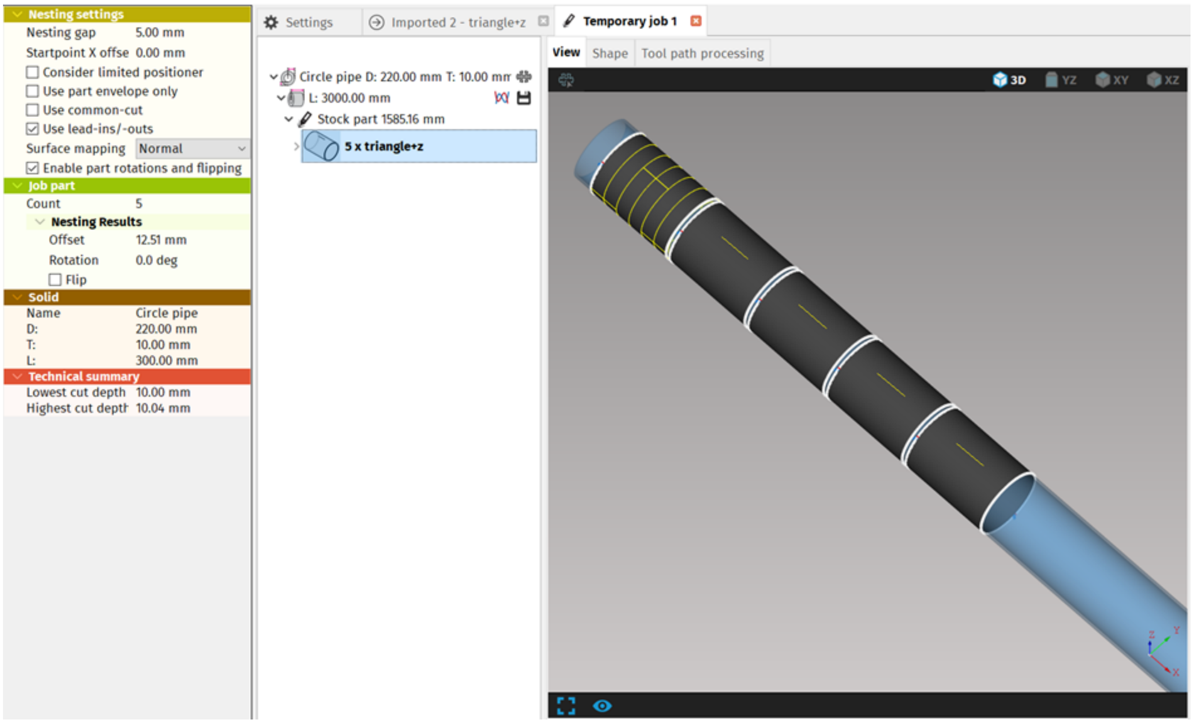


Fig. 213: Nesting of circle pipe with welding line

Pipes in one nesting job all contain the welding line or all are without it. Combination of the two is not supported.

Cutting plan preparation

This section describes major program features and functional *module*s that are utilised through whole process of cutting plan preparation. Sections describe main program *module*s as *WebService* connectivity and utilisation, *machine limits* setup, *expert tables* setup, *1D/ 2D nesting* or *ordering of cuts*.

Program tasks

This section describes all utilised *program units/ tasks* as *Import, Library* and *Job task* and their main *items* as *Part, Stock, Stock part* or *Template.*

Import

After importing some shapes, import menu appears in main menu if Import tab is selected. enables to import multiple files at once. The files intended for import can be marked by holding <Shift> and moving by arrow keys, or by simultaneous holding <Ctrl> and clicking on files. offers four options:

- **Import more files** – Imports more parts to the selected import task
- **Import complete directory** – enables to import whole directory that may contain multiple files.
- **Promote all shapes to top level** – this function is very useful especially when a user works with hundreds or thousands of parts, a lot of tasks (jobs, imports, etc.). The function sorts them and classifies them into a single integrated list.
- **Sort according types –** sorts all shapes according to shape types (with same base dimensions)
- **Close import** – the function closes active import task.

All remaining items in the File menu: Load job, Load library, Raw view, contain only one option in the main program menu – closing active task (Close job, Close library, Close view)

Job task

A *job task* contains collection of parts intended for cutting (marking, punching, drilling, etc.). *Job task* can be local, i.e. available on a storage device accessible from the computer, or it is already prepared in *MPM*. A *job task* is saved to hard drive as a database file (*.job) which saves all settings, configuration and parts included in particular *job task.*

The function *Load job* (*File – Load job*) creates a new, or loads some previously saved *job task*. A new *job task* is created by writing a name of the job that does not exists yet in the dialogue window.

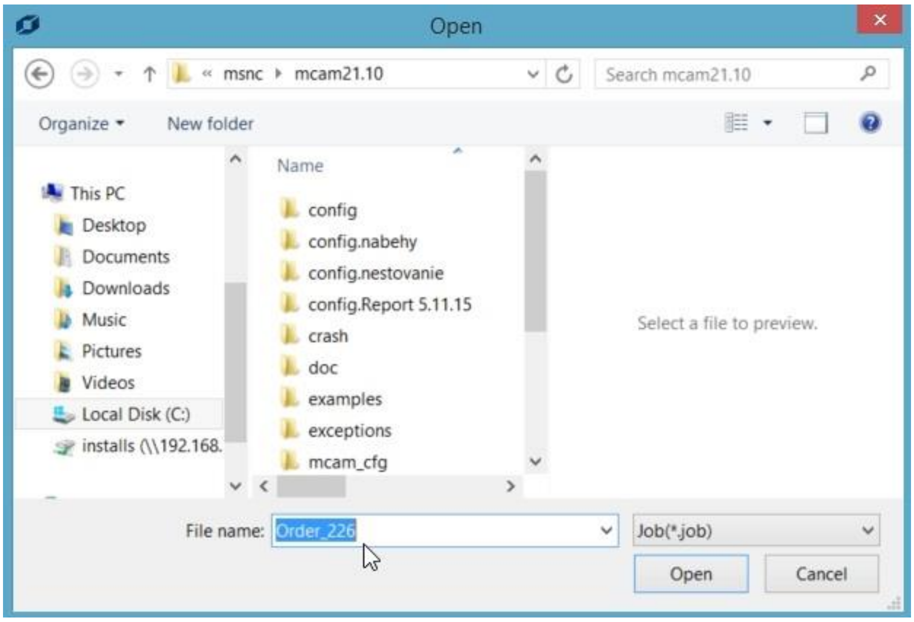


Fig. 214: Creation of new job task

A *job task* is a used as a framework for cutting plan preparation, nesting of imported parts. A *job task* contains info about *nesting settings, templates*, *stocks* and their properties as length or count of *stocks*, assigned *machine/technology/material/tool* to *template* or *stock* and all shape info (*name*, *drawing number*, *position number* etc.). A *job task* is also used to define parameters and settings important for *plan generation* such as output format, rotator mode, chuck location and CNC instructions (*cutting/marking/transformations/text marking*) for generation for selected *stock.* The work with a job is described in more details in corresponding section Job task creation.

It is possible to display or save *stock information* (nesting results) in pdf file and edit all cutting paths of shapes in particular *job task* the same way as in the *Import* or in *libraries.* In Tool path processing tab, all cutting paths are displayed with applied kerf compensation and machine and technology limitations.

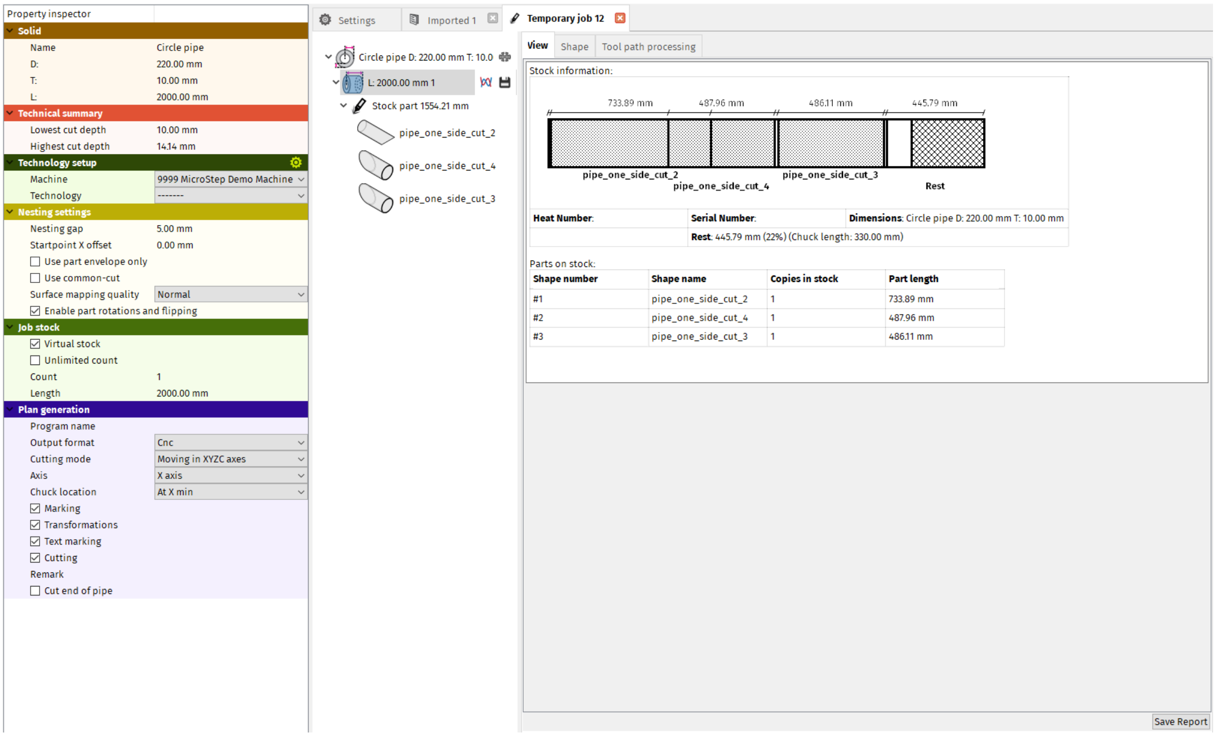


Fig. 215: Job task and stock properties

Part

A part is an object that is intended for cutting. All parts are defined by an original shape. Three or more identical parts (parts that have been duplicated) are displayed as the group with bold font style and the number of parts in that particular group. When there are multiple parts assigned to *job tasks* and *stocks*, tool path with lead ins and lead–outs are shared and the same as for the original shape. That means when e.g. lead–in for the original shape is modified, the same modification is applied for all parts originating from that shape.

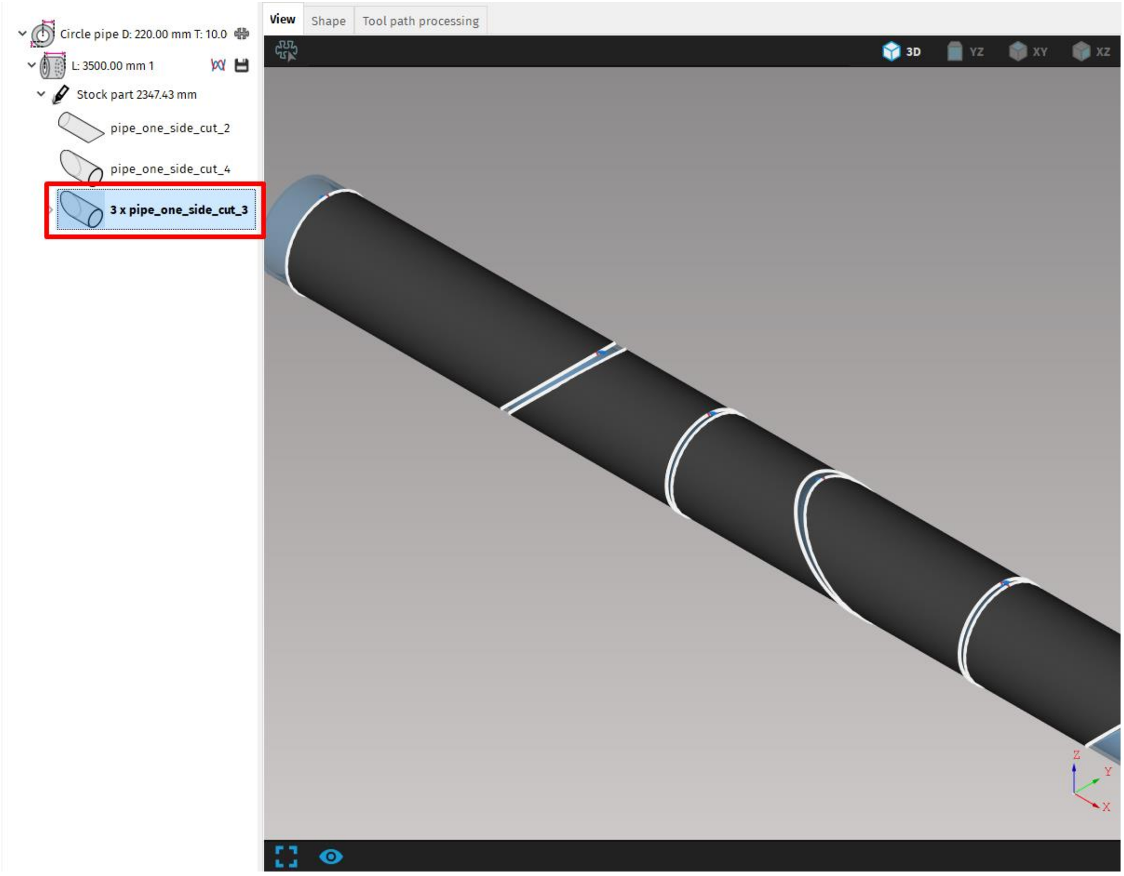


Fig. 216: Original shape defined for all multiple parts

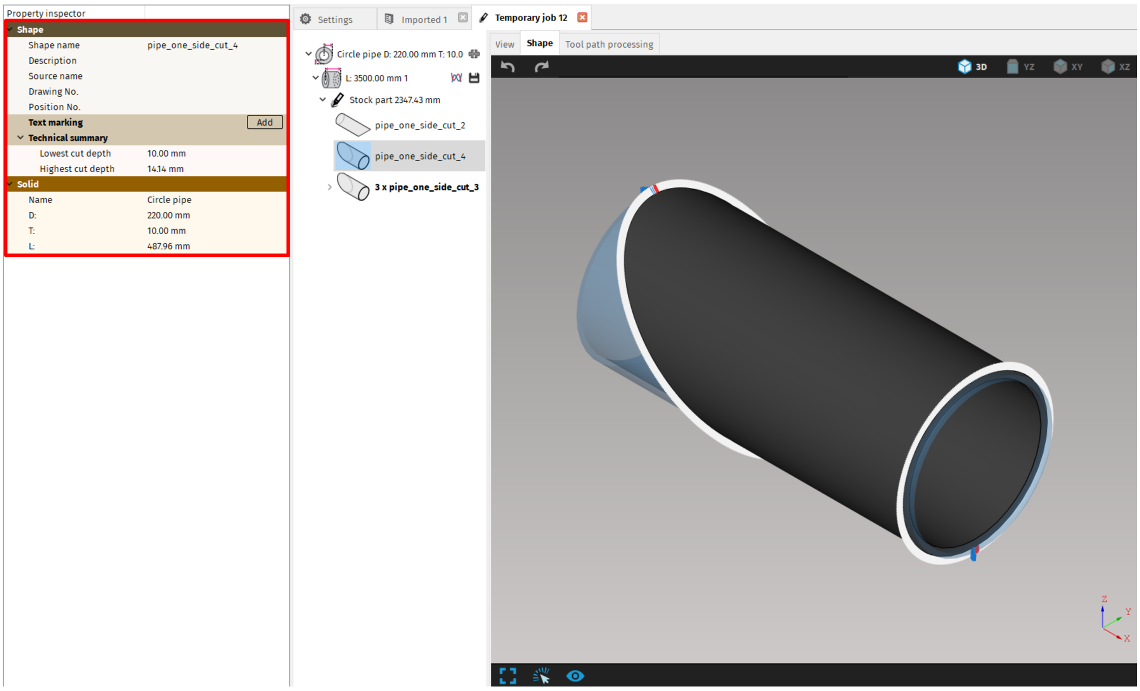


Fig. 217: Part properties

Stock (stock material)

A stock is a definition of raw material type, cross section and length. Depending on the specific stock type, the properties bar displays different parameters and dimensions. There are several types that can be machined by 3D cutting (see section Output formats).

Stock is the unit that is considered as *stock material. Stock* is related to its parent *template* (i.e. *Circle pipe D: 220.000 T: 10.000*) and can be refined with length and count. Length and count specify actual length and count of physical stock pieces (on company stock) that *mCAM* can use for *nesting.*

When utilising *mCAM* with *MPM (MicroStep Production Management)* actual *stocks* (defined in *MPM* database are used) have to be used, therefore *Virtual stock* function has to be disabled and real *stock* from database has to be loaded.

Function *Unlimited count* is used when user wants to determine, how many *stock parts* (physical stock pieces) with defined length needs to be purchased for particular order (cost calculating, etc.).

Stock properties are defined in *properties menu* on selected *stock* in *working tree* in *View tab*.

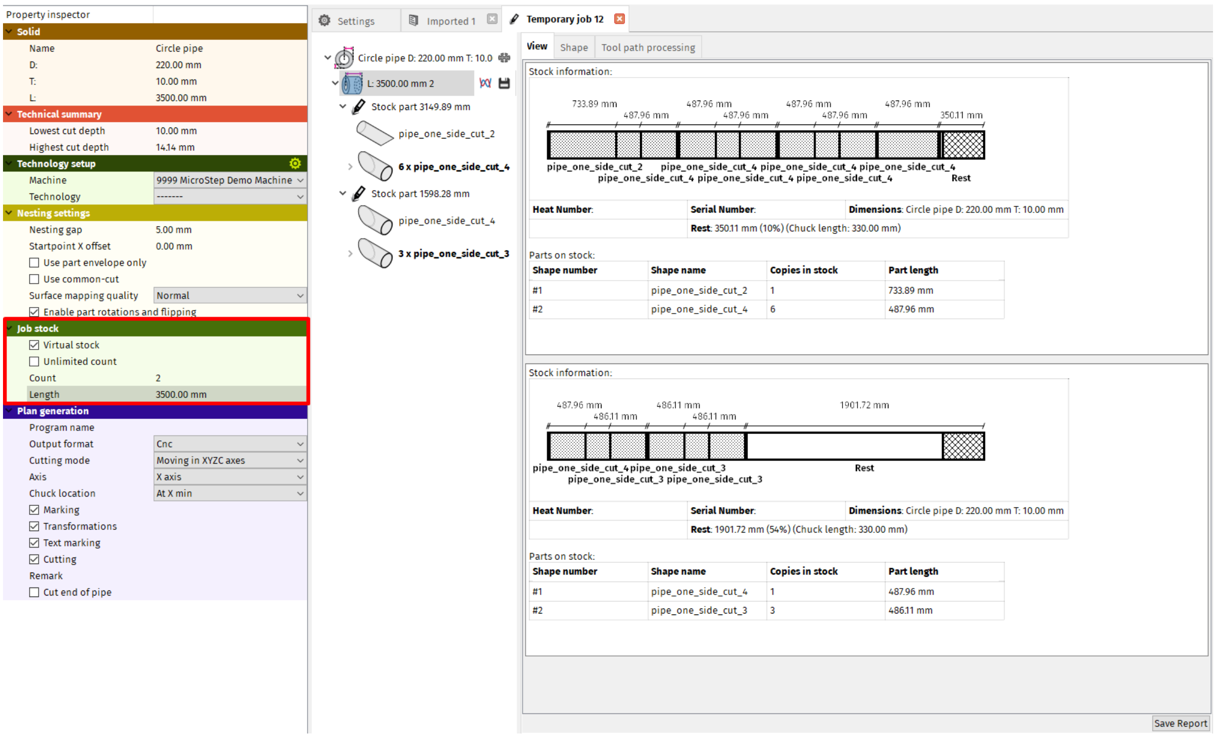


Fig. 218: Stock properties editing

Stock part (physical stock piece)

Physical *stock part* represents one physical piece of stock material. Number of physical stock piece material is given by *Count* of *stocks* defined in *Stock properties menu.* Physical *stock part* represents one *CNC program* intended for cutting on machine. As long as *stock part* is the offspring of the *stock,* all *stock parts* within the *stock* share all properties (*length, nesting settings* and all *Plan generation settings*) with the *stock. Stock part length* that is displayed nearby the *stock part* is given by exact length of nested parts.

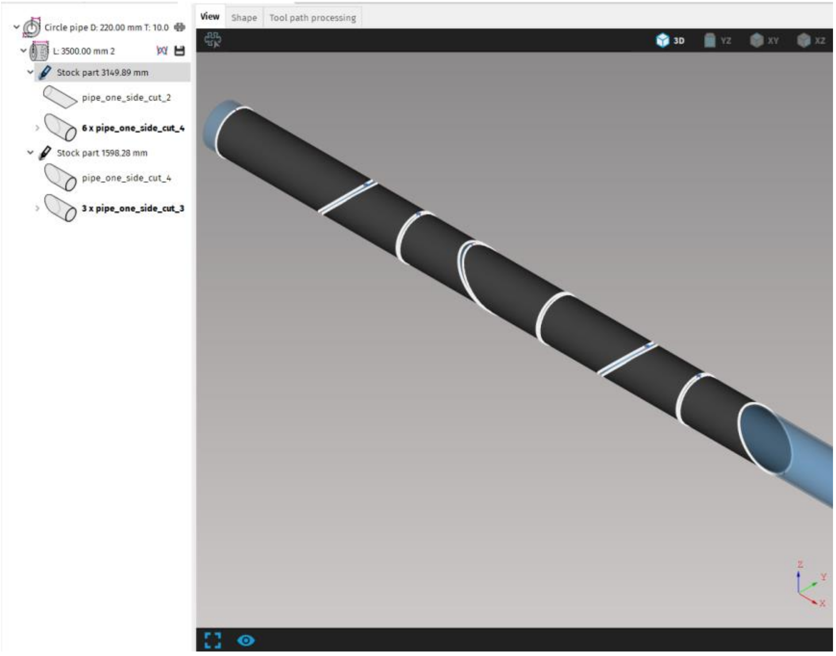


Fig. 219: Stock part – nested parts

Template

A template is a definition of a stock material pattern given by dimensions of a cross–section (material type, wall thickness, diameter, etc.) without specification of the length.

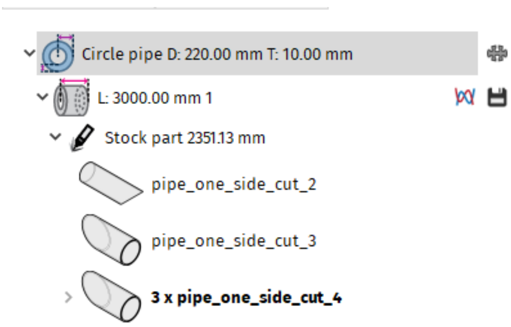


Fig. 220: Template – rectangular shaped pipe

Drag and drop

Drag and drop is a powerful tool that is used for any operations within the *working tree*, *master panel*, *job tasks*, or *imports*. The main purpose of this tool is to make the work with the imported files, *job tasks* or *database job tasks* easier by simple dragging items into a job or into any possible location in the program (*working tree, master panel*).

Function *Drag and drop* allows to:

- move *shapes* from *Import task* to opened *job task*
- assign *shapes* to *stock* within the *job task*

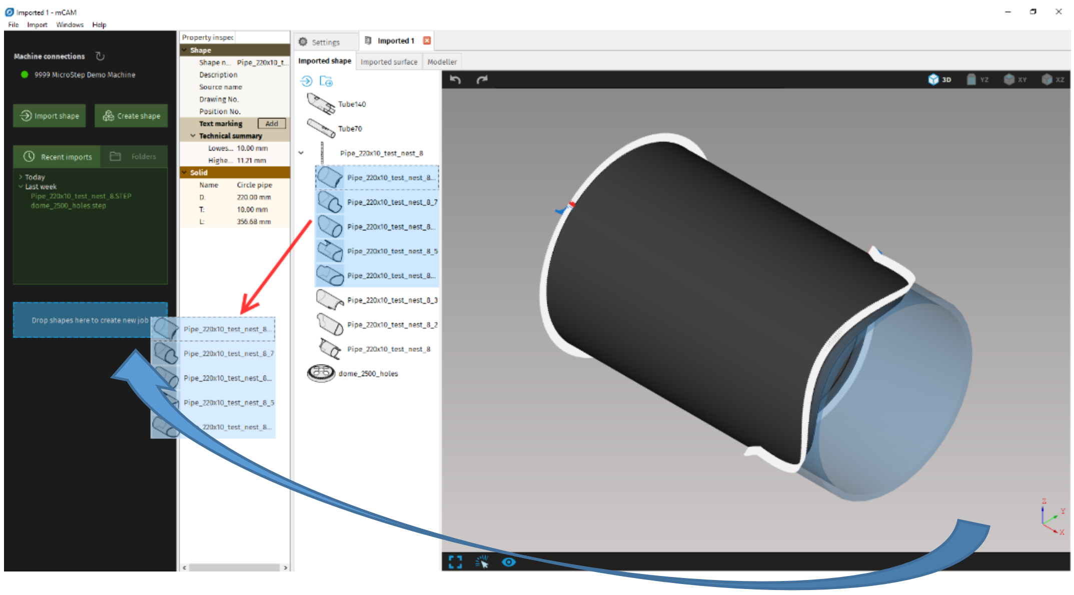


Fig. 221: Create new temporary job task

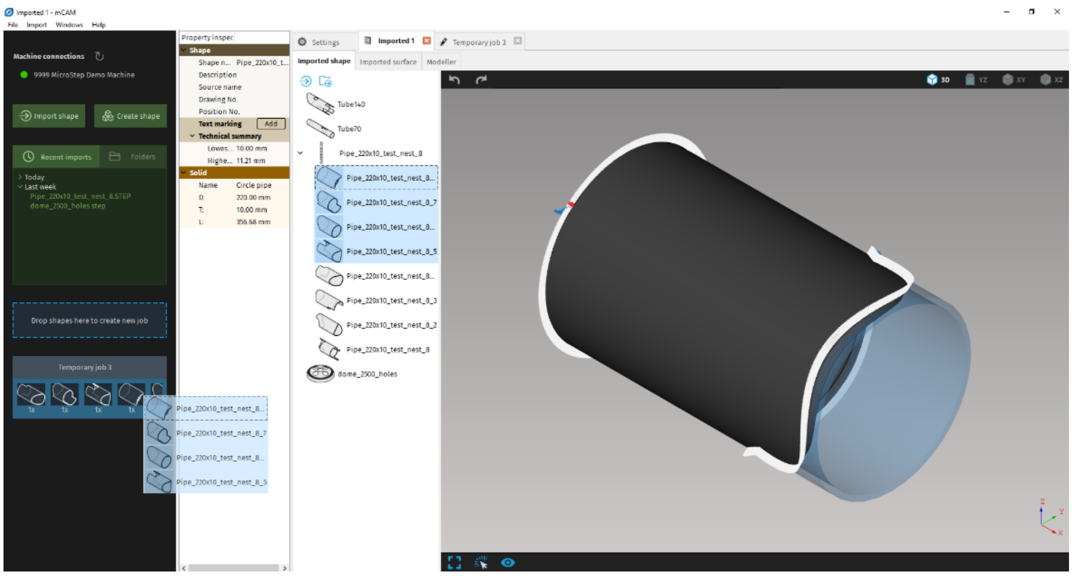


Fig. 222: Add imported shapes to Job task – option 2

WebService or Virtual machine

Cutting machine, technology and used tool has to be chosen for each stock (whole template) while generating CNC program. Therefore, connection to machine via company network and WebService needs to be established or virtual machine has to be created. WebService allows using database of tools in iMSNC (library of tools) from particular machine and work with exact cutting parameters from expert system (if machine has such expert system). Virtual machine is option for customers without WebService and for machines, that does not need such an advanced speed controlling and dynamic kerf compensation. Speed controlling and dynamic kerf width compensation are described in more details in section Kerf width compensation.

WebService setup

WebService is a network application tool that connects a machine, its technology and tools database with mCAM via company network. WebService connectivity allows using expert system of cutting parameters (if it is available for particular machine). WebService is very useful when working with several machines, several technologies or advanced cutting (which requires higher quality). WebService increases effectivity when preparing CNC programs due to iMSNC and tools library connection, so it is very easy to choose proper tool or to setup expert tables.

To establish the connection between machine and mCAM, WebService application has to be installed on machine (download WebService.zip from server and follow instructions in Installation.txt).

WebService installation – on machine (requires installation from MicroStep technician):

1. Sign in to machine computer with account with administrator privileges (account msnc500)
2. Copy all files from folder MachineWebService to directory C:\msnc\Intranet\MachineWebService
3. Start RUN (as administrator only) with INETMGR order
4. Right – click on directory in tab SIDES and use CONVERT TO APPLICATION (if needed only)
5. Check (choose if needed) Application tool (DefaultAppTool): it has to be set to FRAMEWORK 4.0,
6. Installation complete

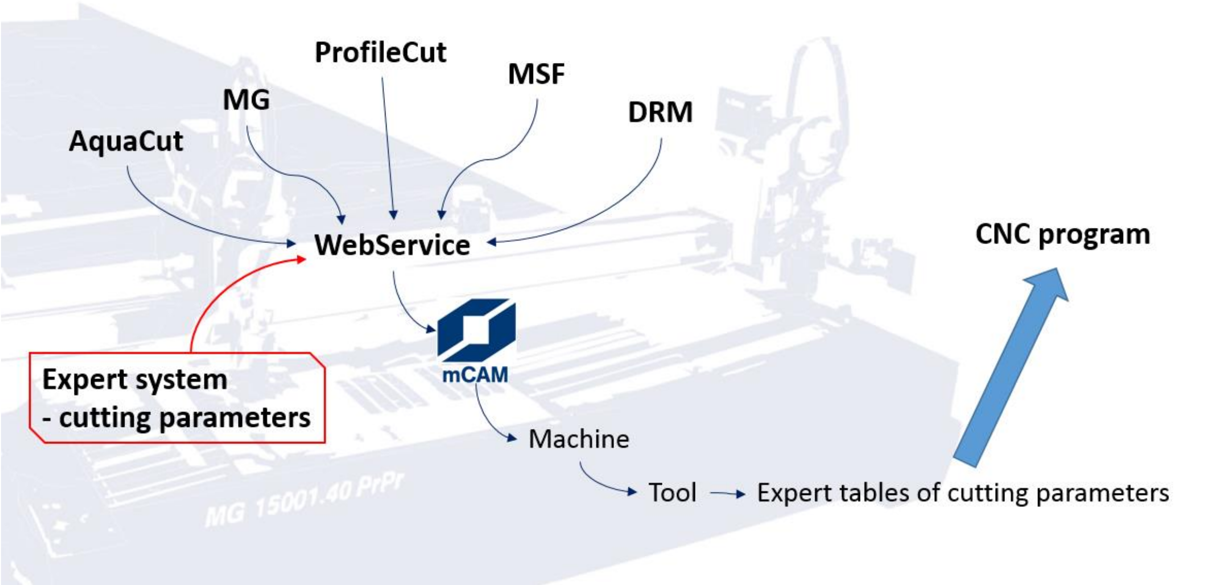


Fig. 223: WebService connection

mCAM setup for WebService machine

WebService machine connection in mCAM is defined IN Settings–> Application–>Web service. Another machines are added by „+“ button and removed by „-“ button to the left of a Web service table. Each machine is characterized by machine number and computer name. Machine computer name is standardly defined in format: msnc500wxxxx (example: msnc500w1703), where xxxx is machine number but should be checked if its name wasn't changed by user.

The list of added machines is then displayed in the table where it is possible to enable/disable them or to enable/disable EkoInfo and MPM feature by rewriting the specific cell. Alternatively, the machine can be enabled by writing a corresponding number in the cell – 0 for disabled, 1 for enabled and 2 for offline (same for EkoInfo and MPM).

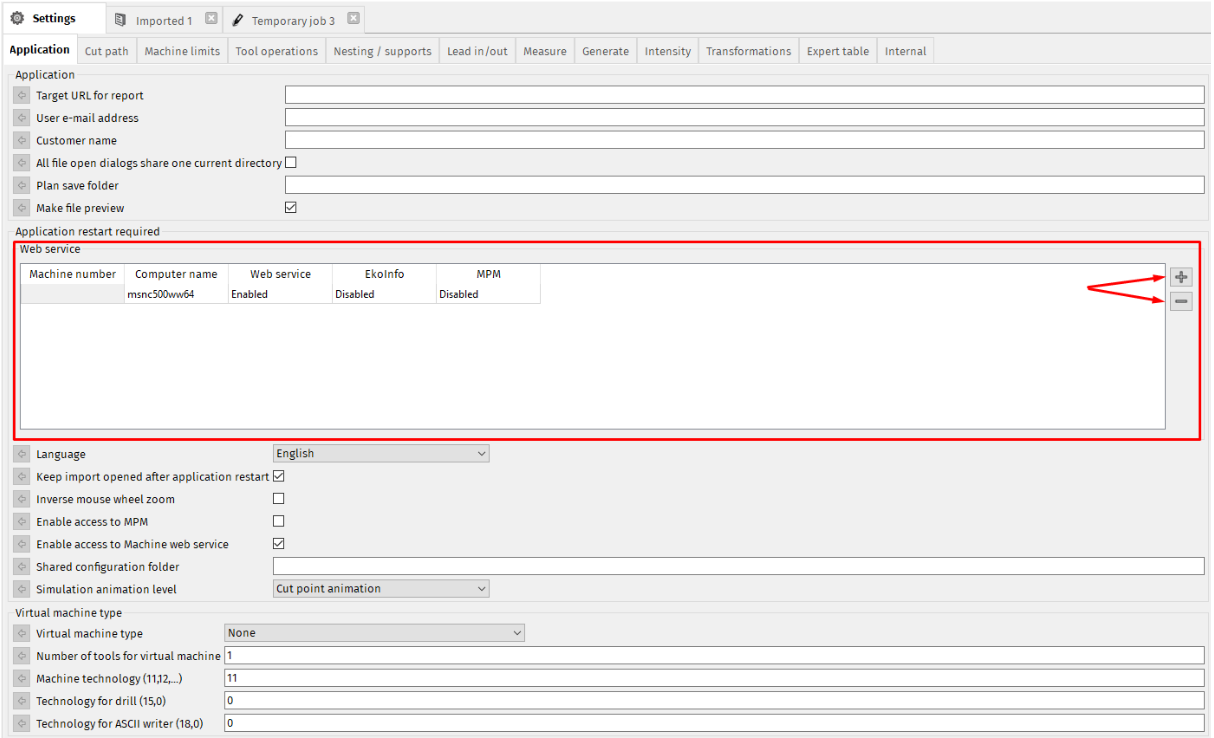


Fig. 224: Machine computers (Web – service)

mCAM setup for virtual machine

Virtual machine can be used when test cutting is performed or mCAM does not have access to any machine with WebService installed. Virtual machine simulates connection to real machine by WebService but with no access to tools from iMSNC or expert systems with cutting parameters.

To setup virtual machine in mCAM these machine data (Settings – Application – Virtual machine type) need to be defined:

- Virtual machine type (type of cutting head: straight, rotator 45°, pantographic rotator 90°, pantographic rotator 120°)
- Number of tools (created tools)
- Machine technology (number that defines used technology) standardly uses:
    - plasma = 11
    - laser = 17
    - water jet = 14
    - oxy fuel = 12)
- Drilling technology number (standard = 15)
- ASCII writer technology number (standard = 18)

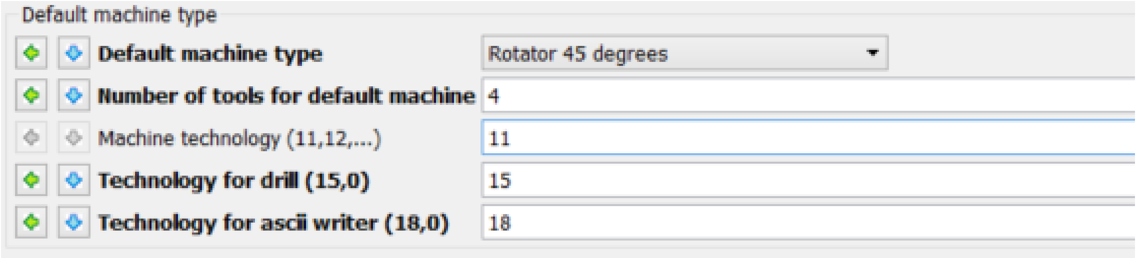


Fig. 225: Virtual machine setup

Machine limits

Machine limits define type of cutting head/rotator and are set in *Settings – Machine limits.* Parameters of Z-axis and B-axis can be seen in *ParamCNC – Coordinate System (Flash.bin).*

Parameters of A-axis (*Maximum A for cutting/marking*) and R-axis are in *ParamCNC* defined by motoric range (not angular range), so they need to be set manually according to type of rotator and rotary positioner. Standard R5 plasma, water jet or oxy-fuel rotators (including torch tip limitations) are able to reach 45 degrees (or 50 degrees for special types of torch tips). Pantographic rotators are able to reach 90 or 120 degrees according to type of the rotator (A axis ranges for this rotator are defined in angular values also in *ParamCNC*).

Standard rotary positioner on plasma (RSV 300 / 500 / 700 / 1500) are set to R min = -400 degrees and R max = +400 degrees. Special rotary positioners (hydraulic / pneumatic control) especially on laser machines may have limited range therefore this range needs to be considered as well. Initial setup of machine parameters is described in more details in section Machine

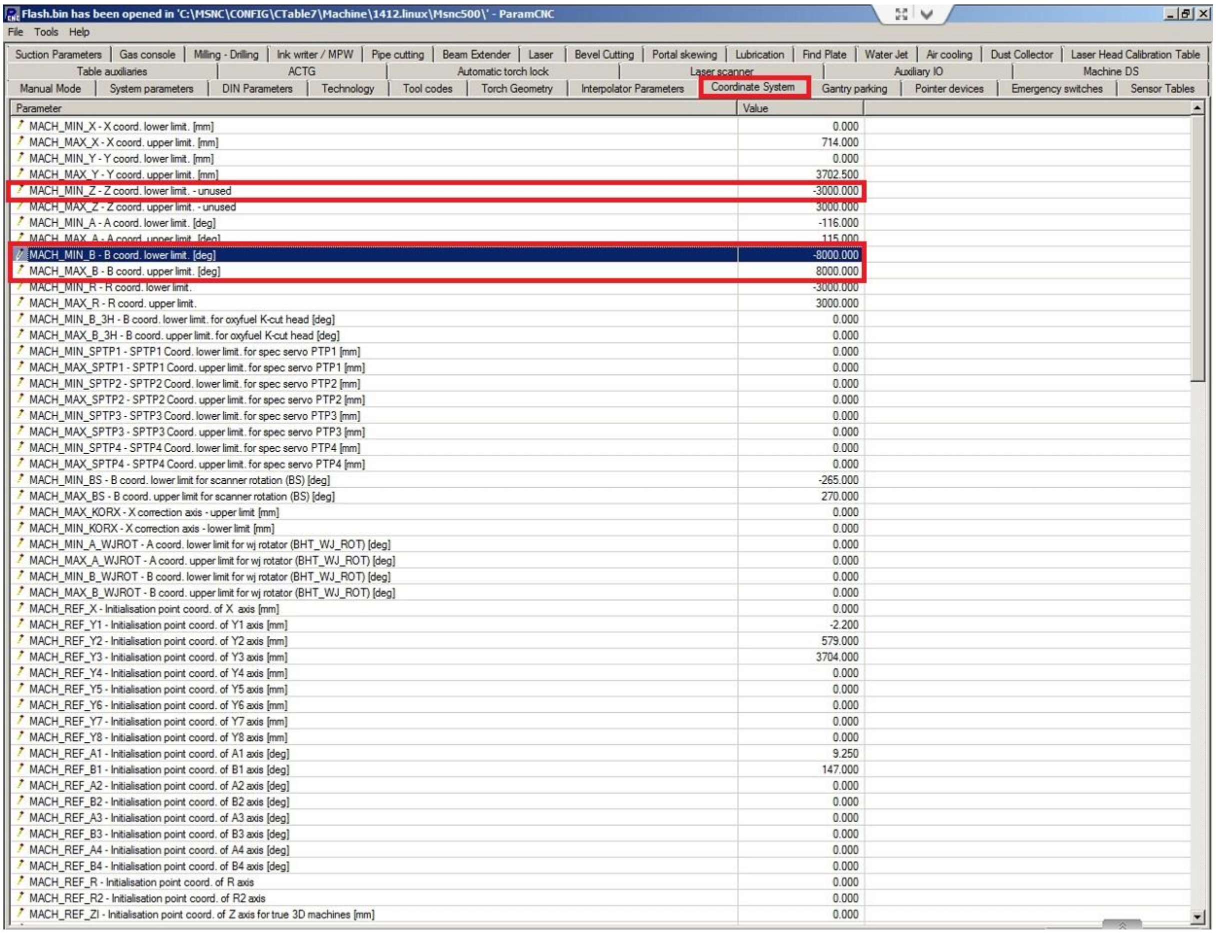


Fig. 226: Machine limits in ParamCNC

Output settings that are applied during generation of the CNC program are strictly related to *machine* and its *technology*, therefore all *machine limits*, *technology limits* and *tool operations* (technological groups of tool) needs to be set correctly. To configure any machine settings *Machine* and *Technology* in Parameter key has to be chosen.

Note: Keep in mind that mCAM always generates CNC programs according to set machine, technology and its settings, according to used tool and its expert table, so all limitations needs to be setup correctly.

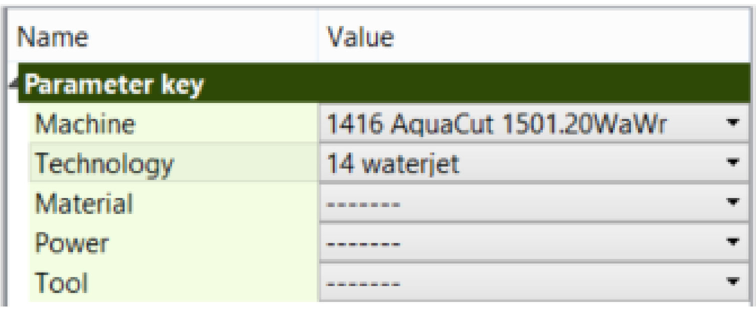


Fig. 227: Machine and technology selection

Standard machine setup

Here are some examples of machine setups for standardly used machine and rotator types.

Straight cutting head (plasma/water – jet/laser/oxy – fuel)

- with standard rotary positioner (RSV 100/300/500/1500/etc.)

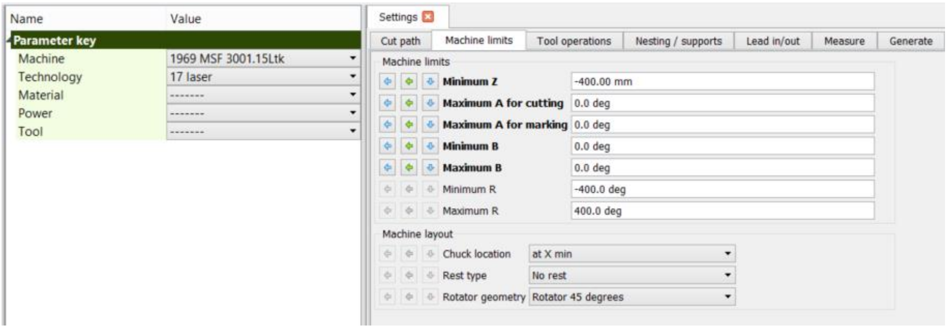


Fig. 228: Straight cutting head and static rotary positioner

Plasma rotator (45 degrees)

- with standard rotary positioner (RSV 100/300/500/1500/etc.)

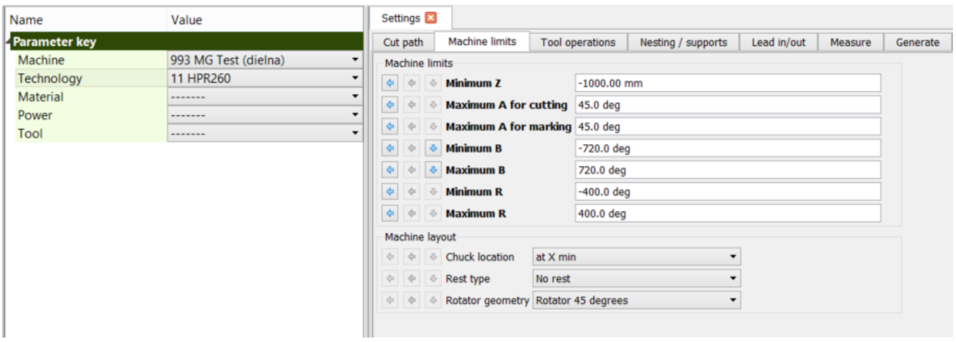


Fig. 229: Plasma rotator and static rotary positioner

Water – jet rotator (45 degrees)

- without rotary positioner

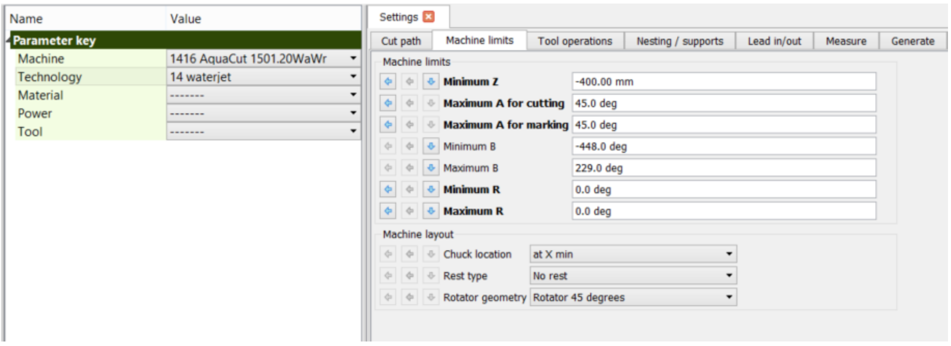


Fig. 230: Water – jet rotator without rotatry positioner

Minimum/maximum limits of B and Z-axis (of all rotators) can be found in ParamCNC of machine. Other values that define rotator and rotary positioner limits (A, R-axes) are displayed in motoric values (not angular) therefore for standard rotator A-axis has to be set to 45° (instead of 110° as set in ParamCNC).

Plasma pantograph rotator 90°

- with standard rotary positioner

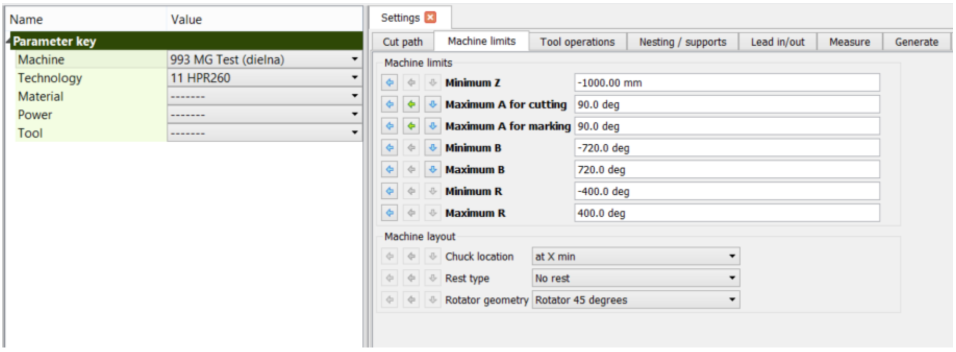


Fig. 231: Pantograph rotator 90° with standard rotary positioner

Plasma pantograph rotator 120°

- with standard rotary positioner

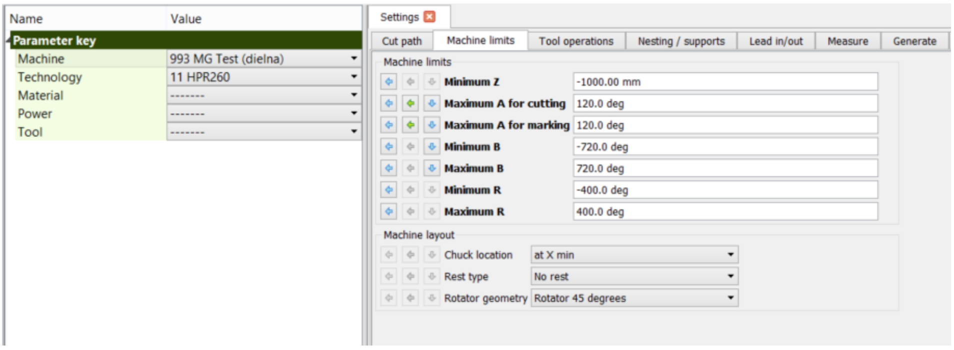


Fig. 232: Pantograph rotator 120° with standard rotary positioner

Machine layout settings (Chuck location, Rest type and rotator geometry) have to be set according to particular machine and type of rotator.

Fig. 233: Machine layout settings

B-axis reposition

Machine limits including B-axis limitations are defined in tab Machine limits in Settings. This tab contains parameters that impose limits of motion on operating range of the machine motion axes.

When generating CNC code, mCAM tries to keep the set B-axis range (depends on type of the CNC machine). In cases, that B-axis value exceeds the limit, *mCAM* displays a warning that signals the exceeded range of B-axis.

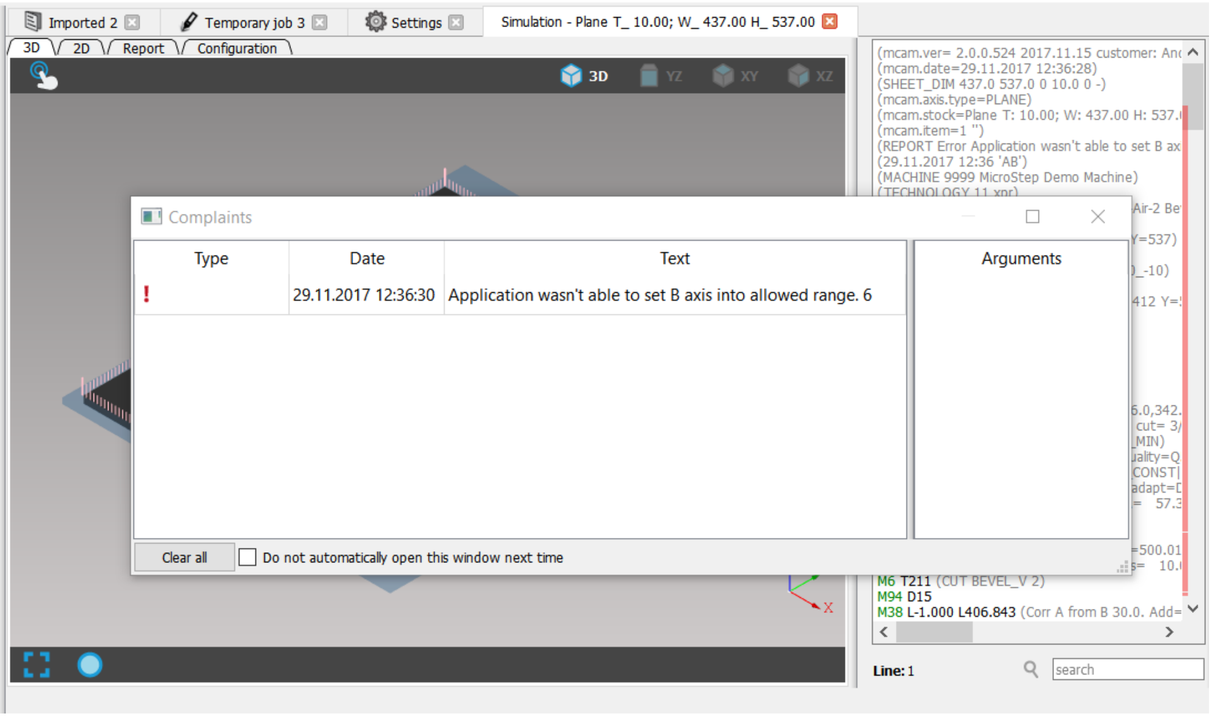


Fig. 234: Complaint about exceeded range of B-axis

There are several ways how to solve this problem:

1. **Using loops** to reduce cumulative rotation in B-axis. Loops are added manually by selecting a cut path, pressing the key L and selecting a cut point where should the loop be located. When the loop is selected, its attributes are displayed in the Properties area of mCAM. Each loop is characterized by Type, Control distance and if it is used for B axis reposition or not. Backward rotation of B-axis is represented as M38 instruction in CNC code. Using loops to reposition B axis can be problematic in plasma cutting, because the backward rotation takes place on one point in the loop and that can cause the plasma torch to turn off. It is safe to use in waterjet cutting.

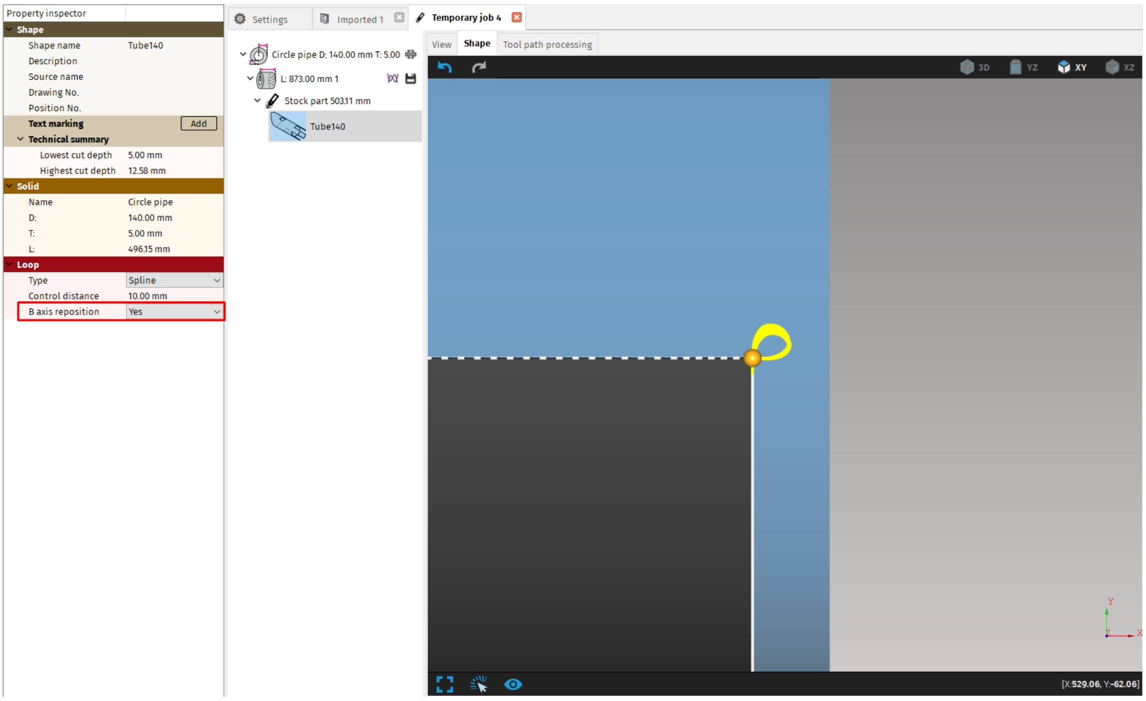


Fig. 235: Example of loop wit B axis reposition enabled

2. Dividing the cut path to smaller fragments

→ Using microjoints – making small microjoints on the cutting contour will allow the cutting head to reposition and with appropriate lead outs selected, the hole will be cut properly.

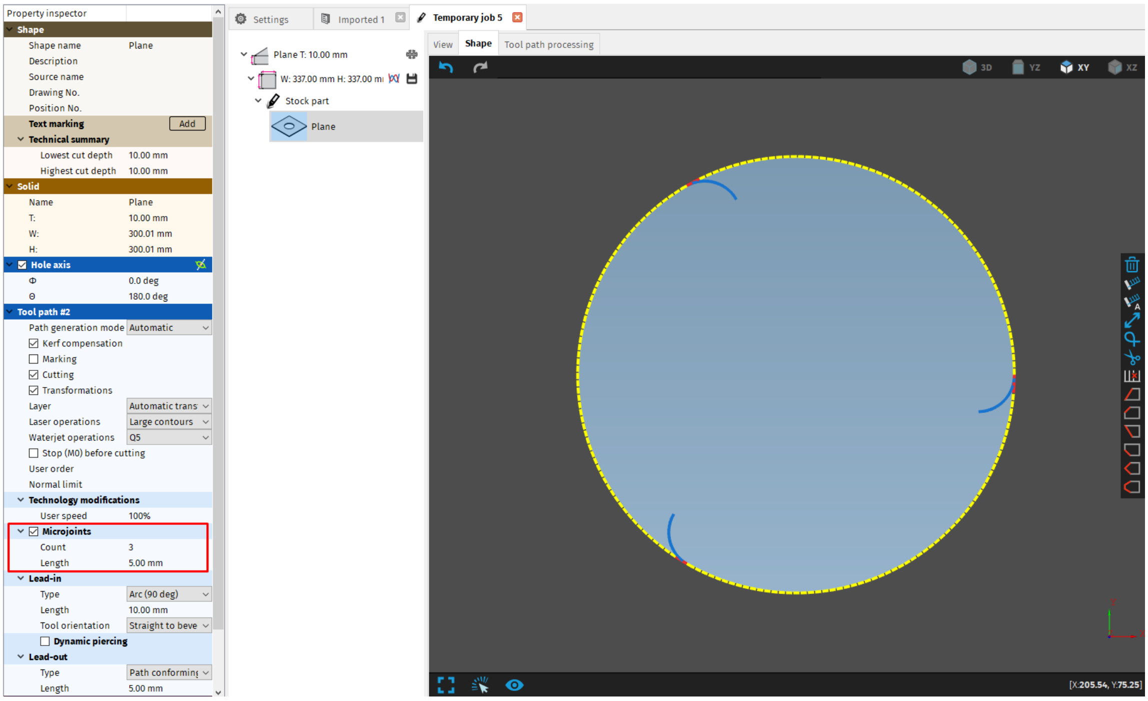


Fig. 236: Using microjoints to split path to three smaller fragments.

→ Splitting a cut path by selecting it and pressing the keyboard key B or scissors icon from the cut path editing widget and selecting a cut point where it should be split. This can be done multiple times if necessary.

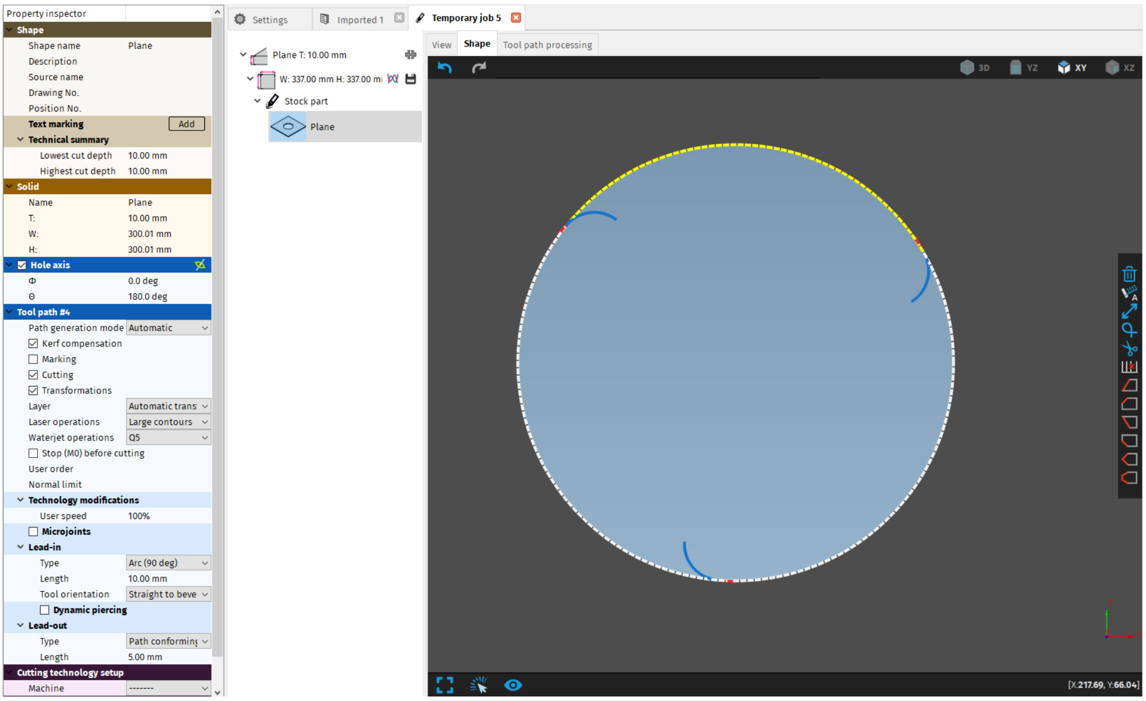


Fig. 237: Cut path split to three parts

Expert tables

Expert table of each tool operation (see section Tool operations) directly affects the processing of cutting paths, cutting speed control and kerf width compensation in each cut-point of cutting contour. mCAM processes the cutting path according to related cutting data for real effective thickness in particular cutting point. Expert tables initial setup is described in section Expert table initial setup.

Kerf width compensation

All cutting tools are characterized by kerf, that represents width of cutting beam for particular energy beam technology. To achieve accurate dimensions of parts, cutting paths need to be compensated for the tool radius. Points of particular cutting path are offset by the tool radius in the direction away from the final machined surface. During this process, some points are removed (especially in short sections of an inner corner – A) and others are added (loops on the outer edges– B).

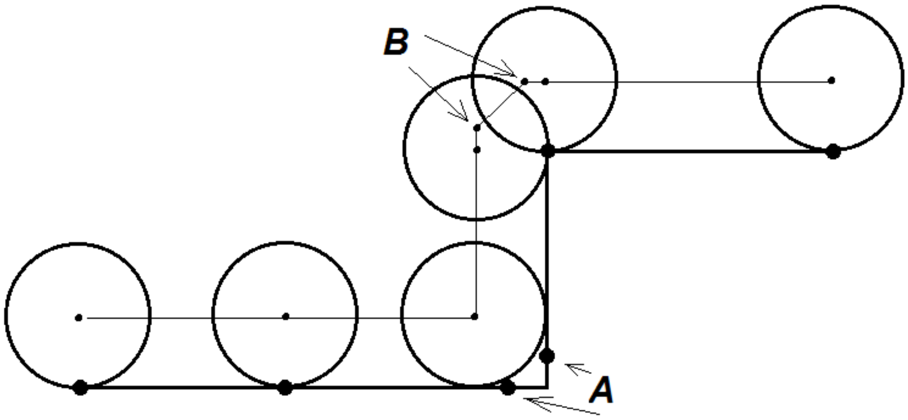


Fig. 238: Principle of kerf width compensation

In thermal beam cutting technologies where thermal affected zone and real kerf width of used tool increases with increasing of effective cutting thickness (bevel cutting), decreasing cutting feed (bevel cutting), increasing current (plasma cutting) or increasing power (laser cutting). The principle of kerf width compensation in *mCAM* consider all those fact therefore *mCAM* is able to adapt kerf width and cutting parameters according to cutting variables in each cutting point.

Dynamic kerf width compensation and *automatic speed control* means that mCAM is able to adapt final kerf width compensation according to used cutting speed, effective cutting thickness (which changes variably according to bevel, shape and type of the cut) and cutting modifiers.

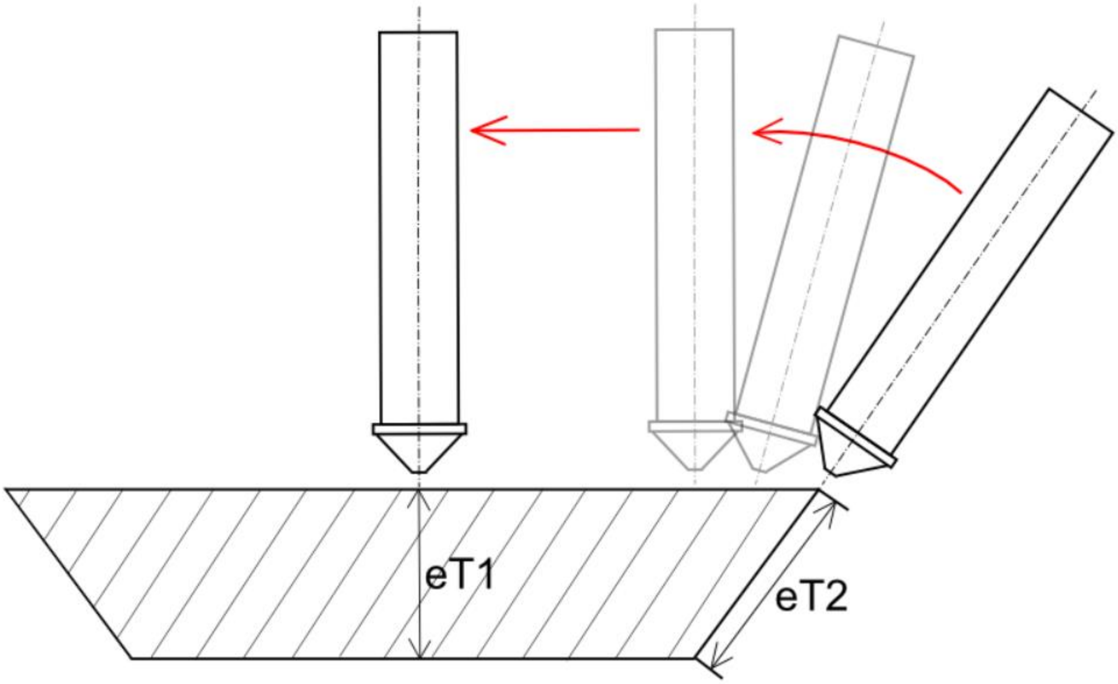


Fig. 239: Effective cutting thickness

Automatic calculation of effective cutting thickness, including applied cutting modifiers (such as Slow in corners, Lower power on profile radius, end-section slow down or other cutting modifiers) results in final used cutting speed. Final applied kerf width is related to final used cutting speed.

- effective cutting thickness => cutting speed + cutting speed modifiers => applied cutting speed
- applied cutting speed => kerf width

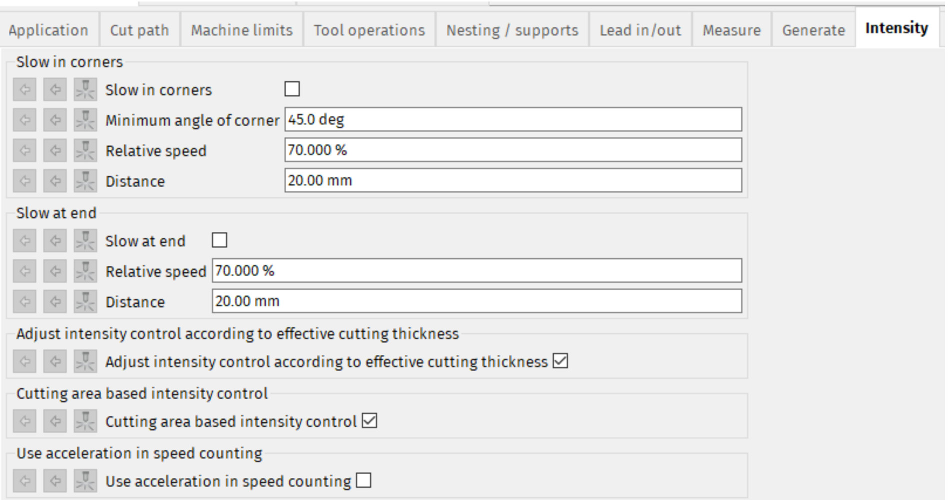


Fig. 240: Cutting speed modifiers

Note: As you can see on example of standard plasma cutting expert table, kerf width value increases with decreasing of cutting speed in whole range of material thickness. Expert table defines exact kerf width and cutting speed values in relevant range of material thicknesses that could be eventually cut by particular tool.

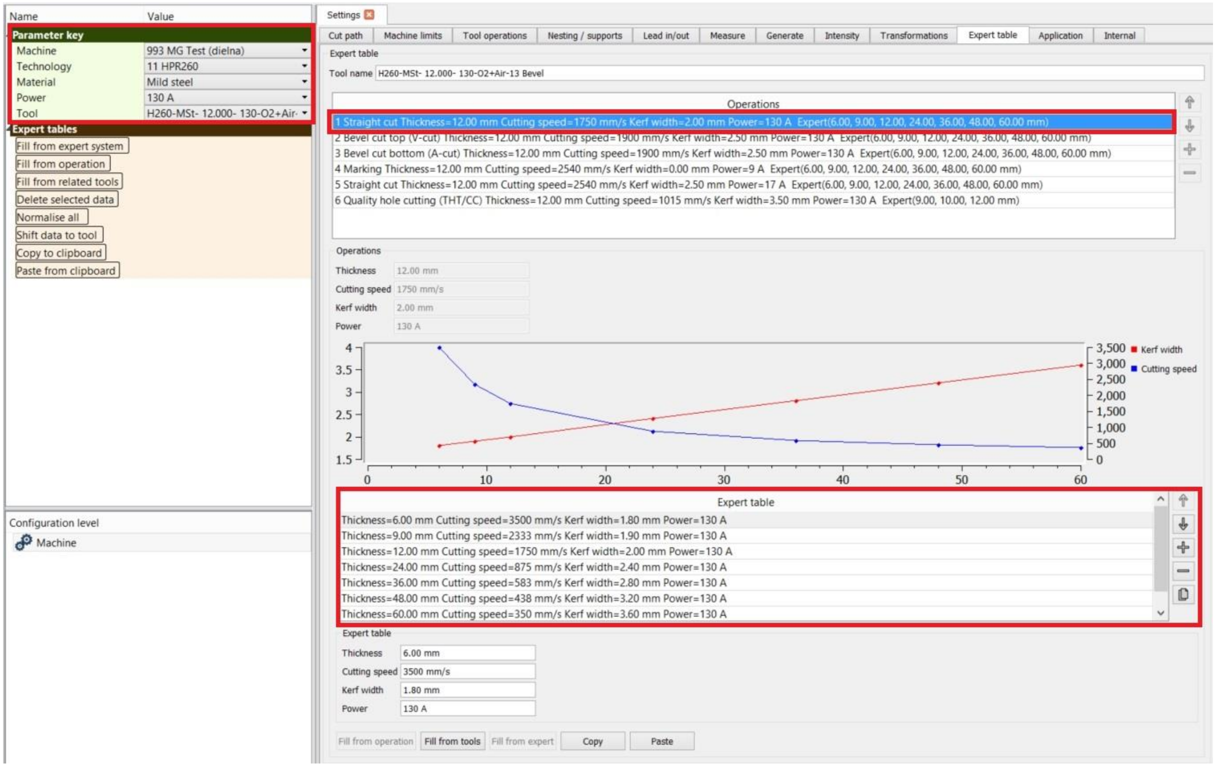


Fig. 241. Expert table of cutting parameters

Kerf width value is related to cutting speed applied on particular cut – point. If constant kerf width should be used (e.g. water jet cutting), constant kerf values in expert table of particular tool (for each tool operation) have to be defined.

> Note: Constant kerf values for water jet machines are set automatically when using WebService. In other cases, it needs to be set manually by editing expert tables.

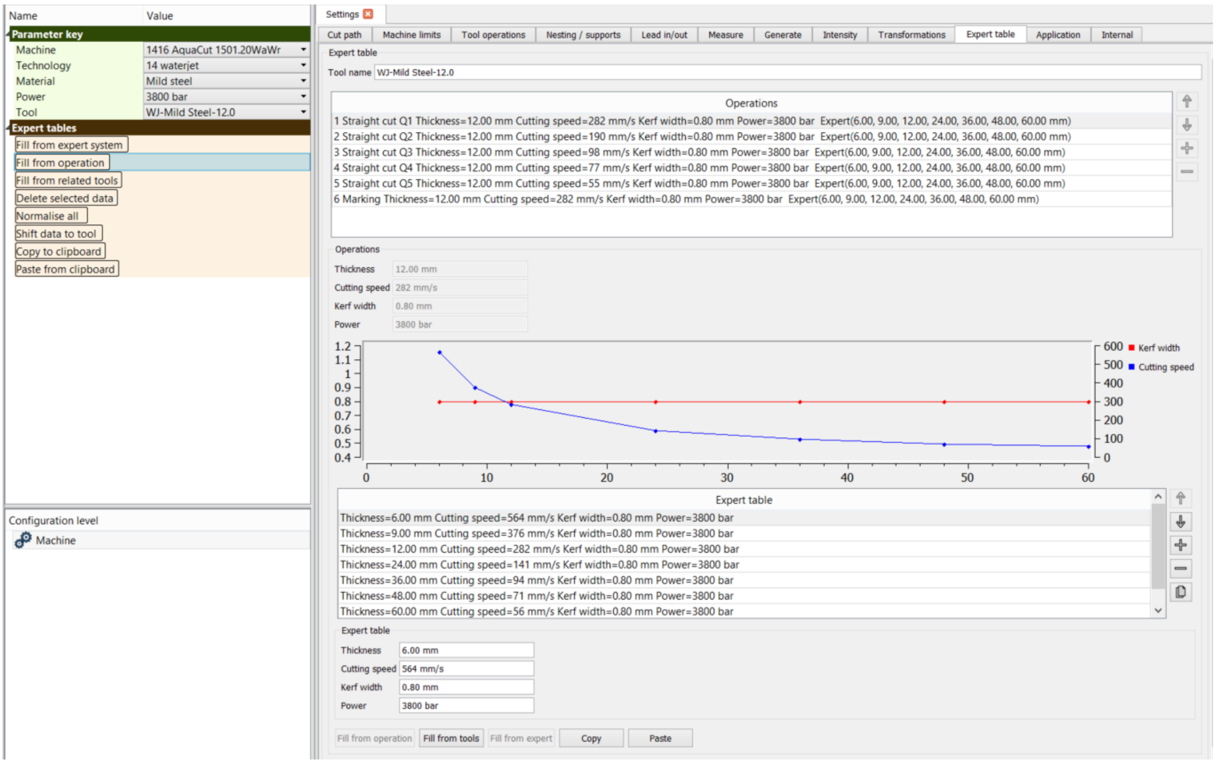


Fig. 242: Constant kerf width values

Effective cutting thickness changes more significantly especially in H – beam flange middle section where the effective cutting thickness can be 1.5 or 2 times bigger than flange thickness. Range of expert table can be easily extended by function on the right side of the screen. In case that automatically generated (fill from expert/operation/tools) range of cutting parameters does not cover the range of thicknesses to be cut by particular tool, it is necessary to extend this range by adding more rows manually. This fact is important to consider especially in H – beam cutting where a tool has only limited access to T – section where cutting speed can reach 20 – 30% of original flange cutting speed.

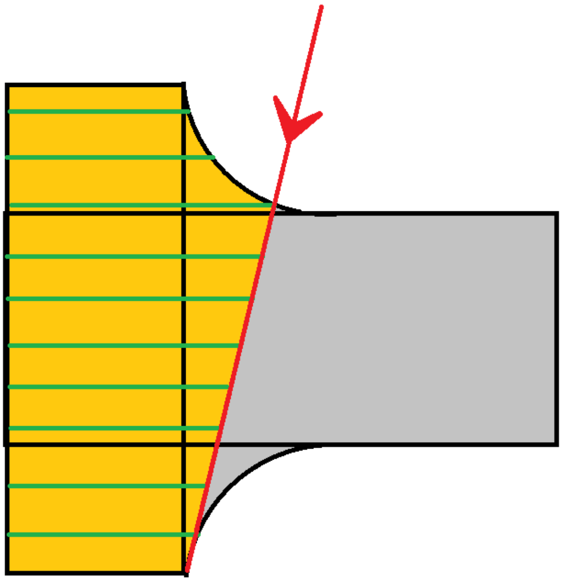


Fig. 243: H – beam middle section cutting

It is possible to enable or disable kerf width compensation for all cut–paths individually as well as all other parameters of selected tool path such as lead–in s/–outs, microjoints and used technologies.

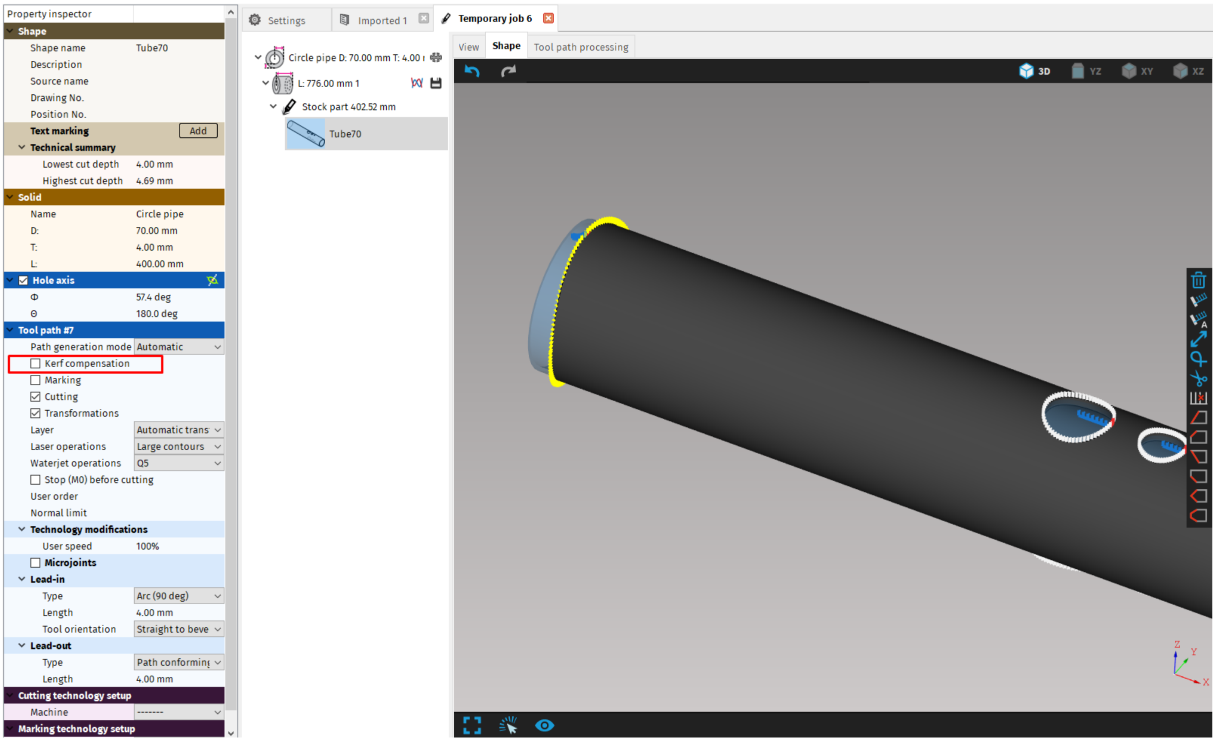


Fig. 244: Disabling kerf width compensation on selected cutting path

Tool path processing

Final processing of all cutting paths, including *kerf width compensation, machine and technology limitations,* takes place during the CNC program generation. Whole principle of cutting paths processing is based on configuration of chosen *machine* and *technology* (*machine* and *technology limits*) and *expert tables* of cutting parameters (*kerf width and cutting speed)* defined for chosen *tool. Machine, technology* and *tool settings* are defined in *Parameter key* in *properties* menu of particular *stock.* All parameters, including machine limitations, technology limitations and *kerf width compensation* values are defined in *Settings* when selected correspond *Parameter key. Kerf width compensation* values are defined in *expert tables* of selected tool. *Expert tables* are described in more details in section Expert tables.

Final cutting paths (compensated by kerf width compensation and limited by *technology* and *machine limitations*) are displayed in *Tool path processing* in *Job task*. In *properties* menu, while displaying *Tool path processing* of particular shape, kerf compensation status is displayed.

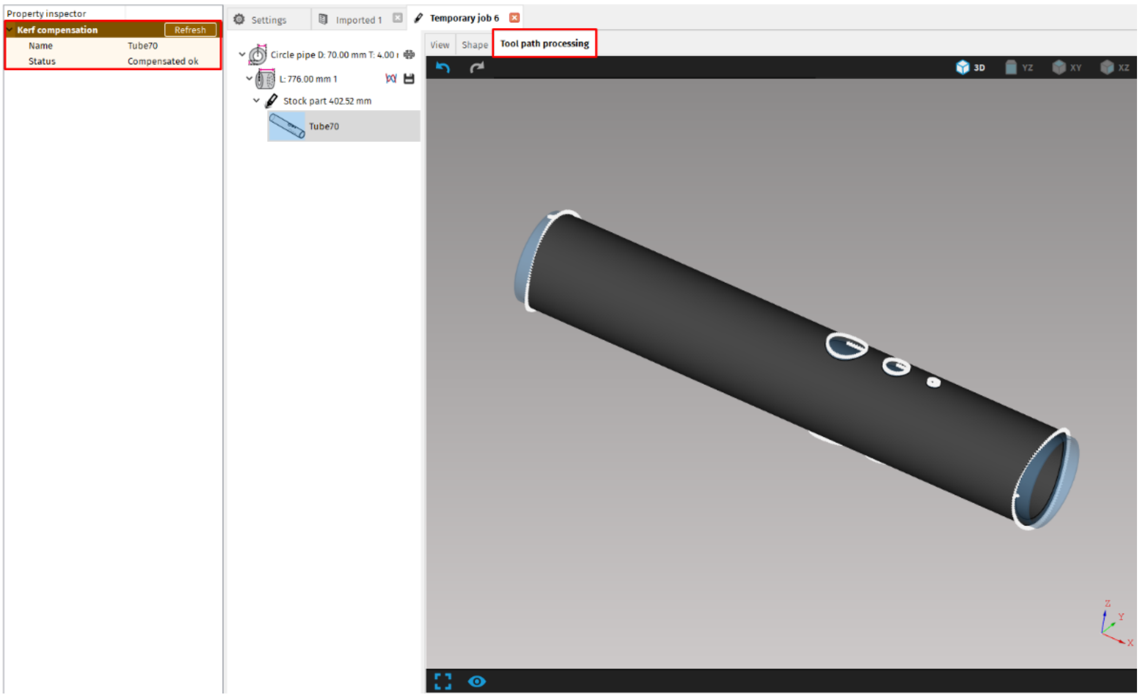


Fig. 245: Tool path processing (compensated and limited paths)

As long as cutting path processing takes place during the CNC program generation, *import*, *libraries* and *shape tab* in *Job task* display only raw–processed (non – compensated and non – limited) cutting paths. Therefore, *Import task, libraries* or *Shape tab* in *Job task* display all cutting paths as non – processed (only raw generated cutting paths – affected by applied *path generation mode*).

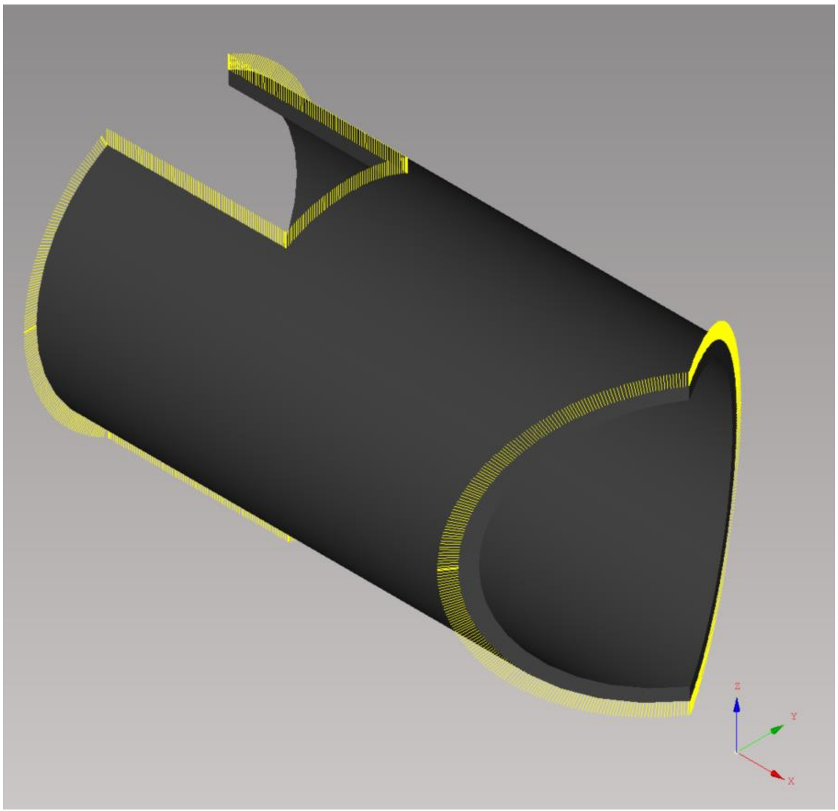


Fig. 246: Raw-processed cutting paths (according to input settings and applied path-generation mode)

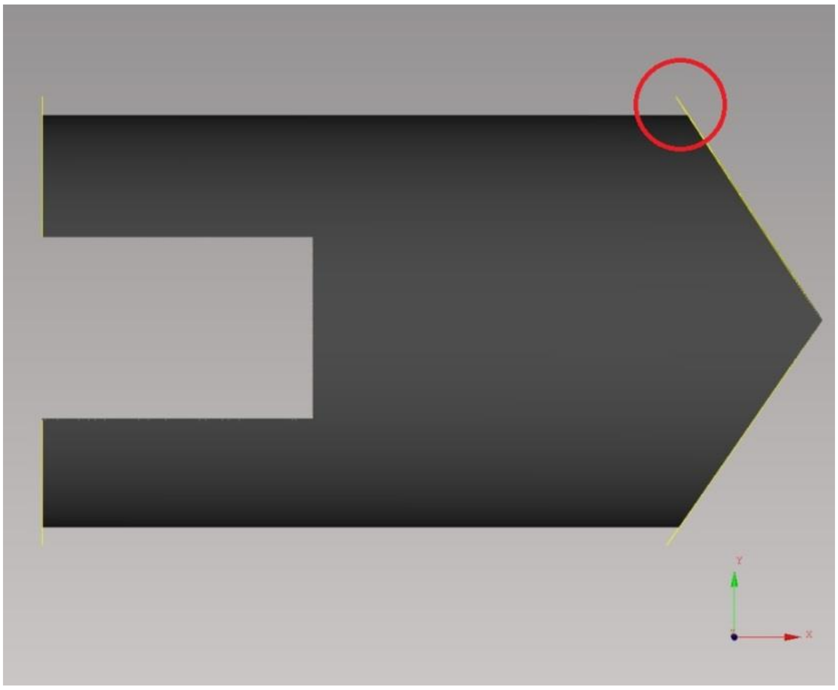


Fig. 247: Side-view on raw-processed cutting paths (bevel cutting)

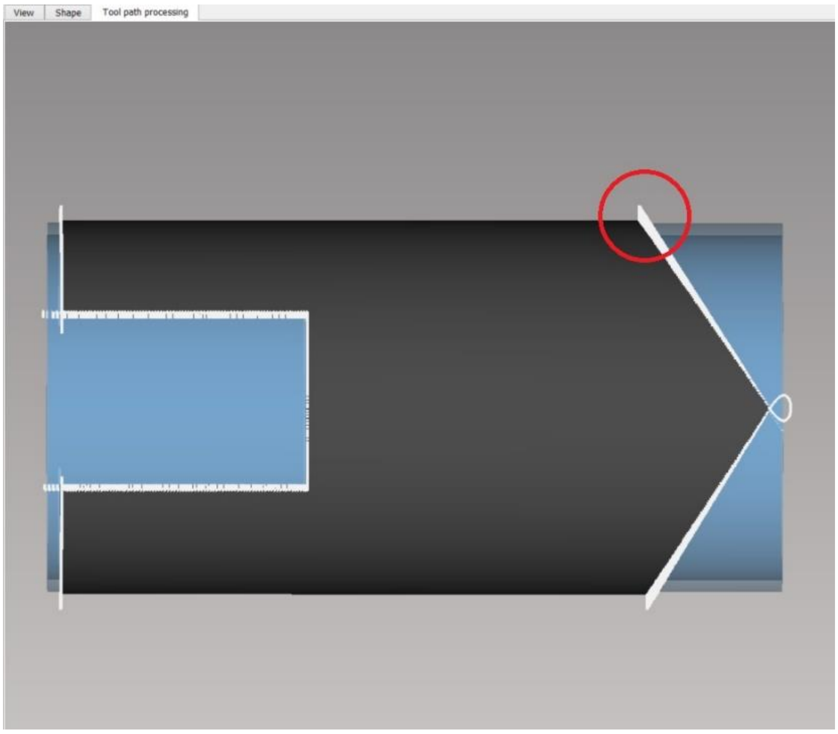


Fig. 248: Tool path processing view (processed cutting paths) – straight cutting head limitations

Expert table initial setup

Depending on type of machine, expert system disposal and WebService (installed/non – installed), it is possible to fill up expert tables in three ways:

- **expert system** (if the expert is available on machine)
- **from operation** (controlled as by machine control system – by linear extrapolation)
- **from existing tools in iMSNC** (if machine has created enough range of tools for each material to cut)

Note: It is possible to edit all expert tables by buttons on the right side of the screen regardless the way of creation. Expert table graph display an actual cutting data (kerf width and cutting speed) set for particular thickness of the material.

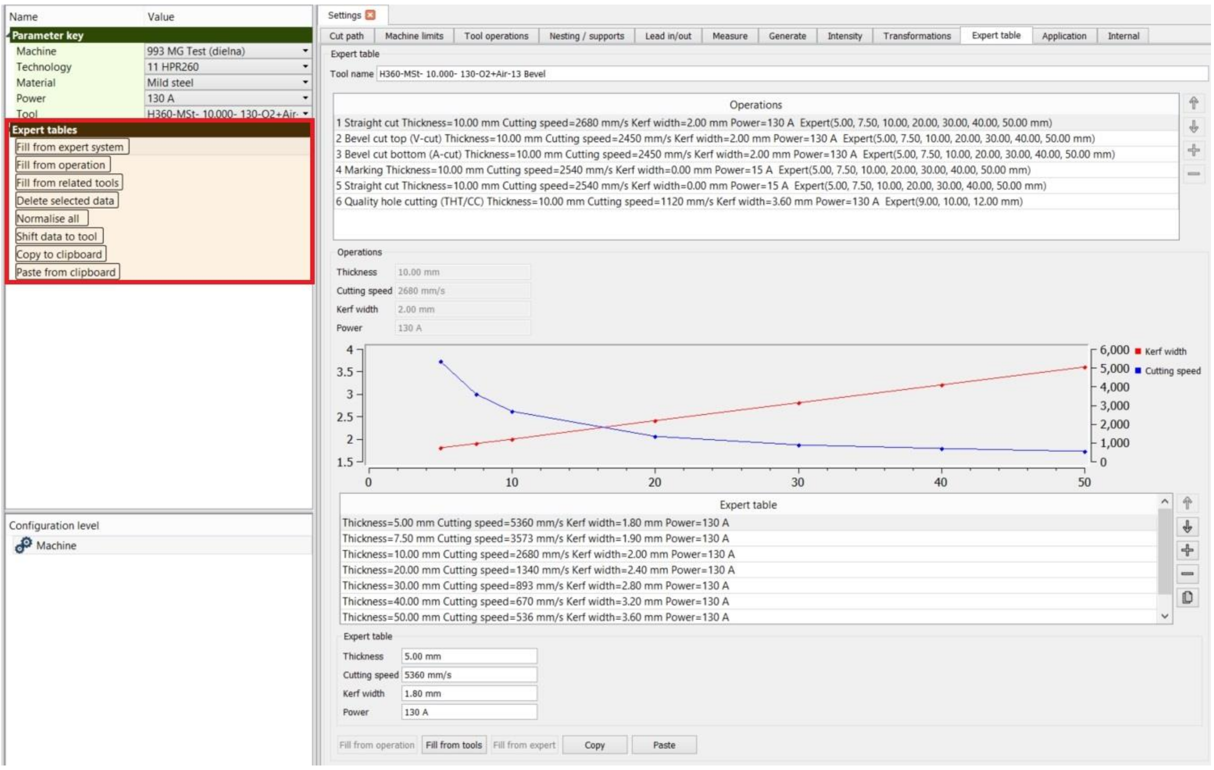


Fig. 249: Expert tables (generator)

Fill from expert system

Fill from expert system is recommended to use for plasma technology with its own expert system (if expert system is purchased) that allows to generate cutting parameters automatically. In this case, expert table is filled up from available range of parameters which could be generated from expert system in iMSNC.

Expert table of each tool operation is created automatically from expert system parameters (expert system tools that could be eventually generated for particular current, type of material and thickness). All related tools share the same identifiers (current, material and consumables). E.g. expert tables of cutting parameters for tool H260 – MSt – 10.000 – 130 – O2+Air – 13 (tool that included in iMSNC library of tools) are created from all available expert system cutting parameters which share the same identifiers: MSt (material), 130 A (current), O2+Air (used gases) and 13 (consumables).

Fill from operation (extrapolation)

Fill from operation by automatic extrapolation of reference cutting data (defined for each operation). Cutting parameters extrapolation simulates a mechanism of cutting speed control performed by machine control system on planar cutting where machine controls actual cutting speed according to used bevel angle (tilting angle) during a cut.

If this function is used when mCAM is connected to water jet cutting machine via WebService, constant kerf compensation values are generated (water jet kerf value does not change with the cut thickness of the material) and no dynamic kerf compensation is performed.

Fill from related tools (tools from iMSNC library)

Fill from tools in iMSNC (tool library) is possible to use on machine where user has created sufficient range of tools with valid cutting parameters (tested and standardly used). Expert table is filled from tools which are available in tools library in iMSNC and share the same identifiers. That means that expert table for tool H260 – MSt – 10.000 – 130 – O2+Air – 13 is created from all available tools (tools included in iMSNC tool library) with following identifiers: MSt (mild steel), 130 A (current), O2+Air (used gases) and consumables number 13.

> Note: All following tools (used for filling the expert table) are included in iMSNC tool library.

Example of grouped tools that share the same identifiers for tool H260 – MSt – 10.000 – 130 – O2+Air – 13 (chosen tool for cutting):

Expert tables are filled from actual cutting parameters of these tools:

- H260 – MSt – 5.000 – 130 – O2+Air – 13
- H260 – MSt – 10.000 – 130 – O2+Air – 13
- H260 – MSt – 15.000 – 130 – O2+Air – 13
- H260 – MSt – 20.000 – 130 – O2+Air – 13
- H260 – MSt – 25.000 – 130 – O2+Air – 13)

Keep in mind that invalid cutting data (cutting parameters and tools created cutting experiments, etc.) may refer to incorrect expert tables (non – linear curves), so expert tables need to be edited and fixed manually.

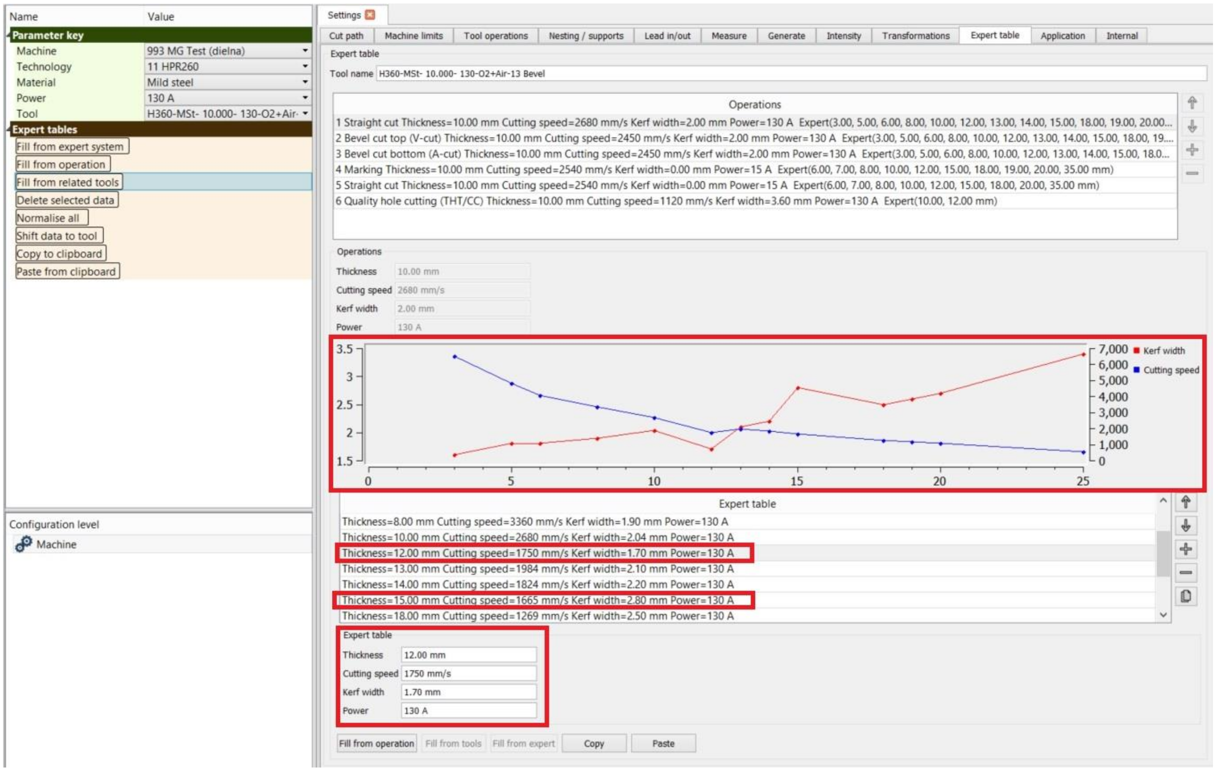


Fig. 250: Possibly invalid expert table of cutting parameters

Editing features

All functions in Expert tables (generator) are applied according to chosen level in Parameter key. If only machine and technology are selected, all tools related to selected technology are affected by used function (fill from, normalize, shift data to tool, etc.). If machine, technology and particular tool are selected, only operations related to selected tool are affected by used functions.

Note: Parameter key material and power are only filters used for displaying particular tools therefore their selection is not considered in previously mentioned functionality.

- **Delete selected expert data** – deletes all cutting expert data related to selected parameter key
- **Normalize all** – deletes non – valid cutting parameters (negative or too high kerf/cutting speed), deletes duplicated cutting parameters and sorts all parameters by thickness)

- **Shift data to tool** – if cutting speed or kerf width values in tool parameters in iMSNC have changed (and considered as valid) in iMSNC, reference parameters (downloaded from machine – displayed, and also whole range of cutting parameters in expert tables are shifted by the difference of those values.

- **Copy to clipboard/Paste from clipboard** – copy/paste all selected data (set by parameter key) to another tool or any text file.

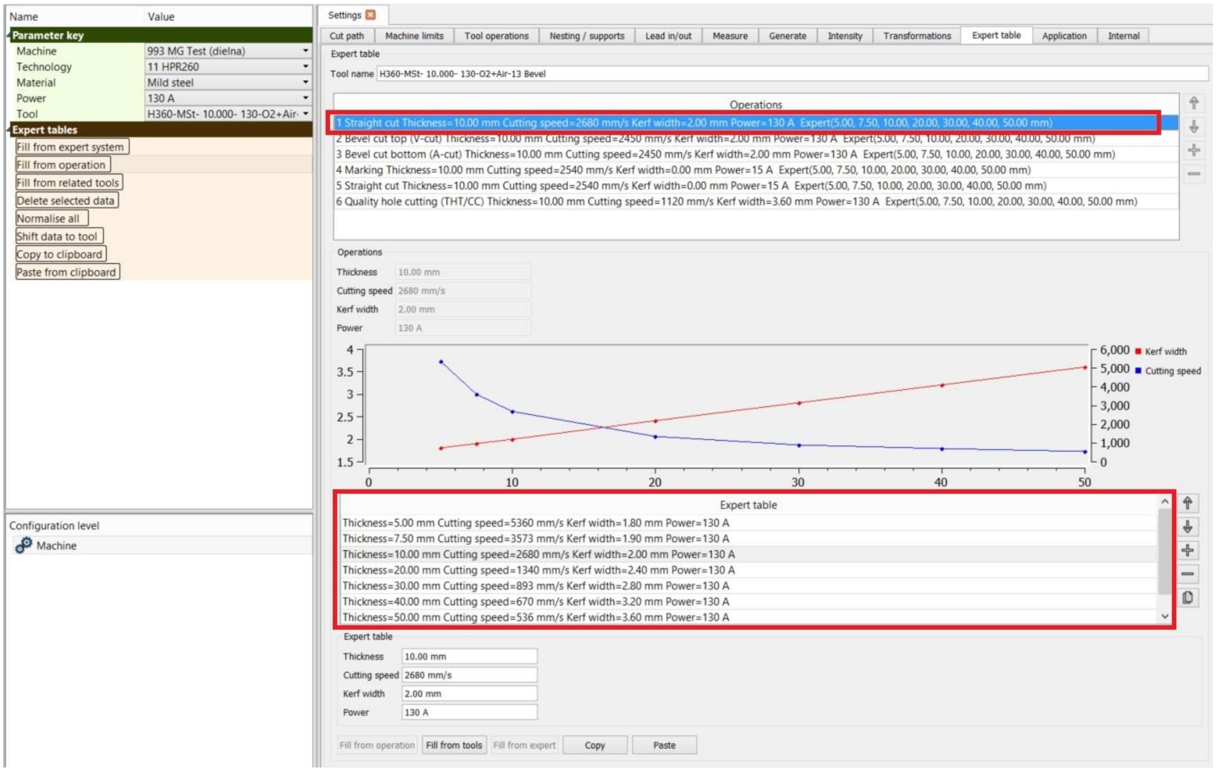


Fig. 251: Shift data to tool

Ordering

The *mCAM* uses automatic ordering method therefore any user interaction or setting should not cause any serious problem within the *NC code* and cutting. It is possible to define priority cutting for any cut, but the main ordering logic (automatic ordering managed by *mCAM*) is maintained to avoid any technological or machine–based issues in ordering defined by user.

Automatic ordering

Automatic ordering is based on several technological and optimalization principles. *Automatic ordering* managed by *mCAM* takes into consideration also a machine layout (*has rest/ no rest*), used rotator mode (cutting beams in *rotary positioner/ on supports*) or dimensions of the *stock piece*. Sophisticated algorithms of *automatic ordering* are based on these principles:

- **part at once** (cut part by part)
- **nearest cut** (move to nearest cut if possible)
- **technology priorities:** *ASCII marking → marking → drilling → punching/straight cutting/bevel cutting*
- **reduce operation switching** (i.e. drill all holes D20 → drill all holes D10 → cutting)
- **technological order of grouped cuts** (i.e. H–beam trim cut: *web → flange → flange*)
- **reduce rotation of profile** (i.e. complete all cuts on side of square pipe, H–beam, etc.)
- **sort cuts to groups** (i.e. group cuts on profiles by X coordinates, to reduce supports repositioning)
- **group measurement** (perform measurement for several grouped cuts)

Item finishing mode

Item finishing mode defines the method of ordering cutting parts, it means the sequence of technological operations (e.g. marking, cutting), or individual cutting priority within the equivalent parts.

Except for mode *Any order* when cuts are ordered by the rule of closest cut, the highest ordering priority is assigned to item *finishing mode*. The modes are described in detail in section Generate.

User ordering (priority cutting)

User ordering should be used only in case of special ordering requirements; e.g. for minimization of thermal deformation of the material or priority processing (*drilling/ cutting*) of some cuts. *User ordering* has lower priority in ordering processing than the *automatic ordering* managed by *mCAM*, therefore it is not possible to change or affect the basic principles of *automatic ordering* or affect the technological integrity of final *CNC program*.

User order number sets the priority of cutting particular cut where “1” is the highest priority. When the same *User order* priority is set for two different cuts, the principle of nearest cut manages the final order between then.

Example: *User order* (value = 1) defines that selected cut should be performed as first on that particular part. But if this cut would be performed as first on each part on nesting with using *X min Chuck location*, two highlighted parts would be cut–off completely without cutting front trim cut and two holes. Therefore, *mCAM* controls and applies *User order* priority cutting just after *Automatic ordering* processing -> **Principle of ordering: *Automatic ordering → Item finishing mode → User order***

User order of equivalent cuts can be defined by priority numbers (1–highest priority, 2–lower priority, etc.). These priority numbers are not superior to main ordering rules managed by *mCAM* automatically. Their function is activated after the primary *mCAM* ordering, so the basic ordering principles are preserved:

- cutting is executed from the edge of the semi–product so cutting off the part with unfinished holes or other parts closer to the edge of semi–product is avoided
- holes first – if any part contains holes inside the outer contour, holes have to be cut first

Note: Hole on the top of the dome is cut as last due to continuous measuring the detection point on the top of dome even if other holes in dome have higher priority.

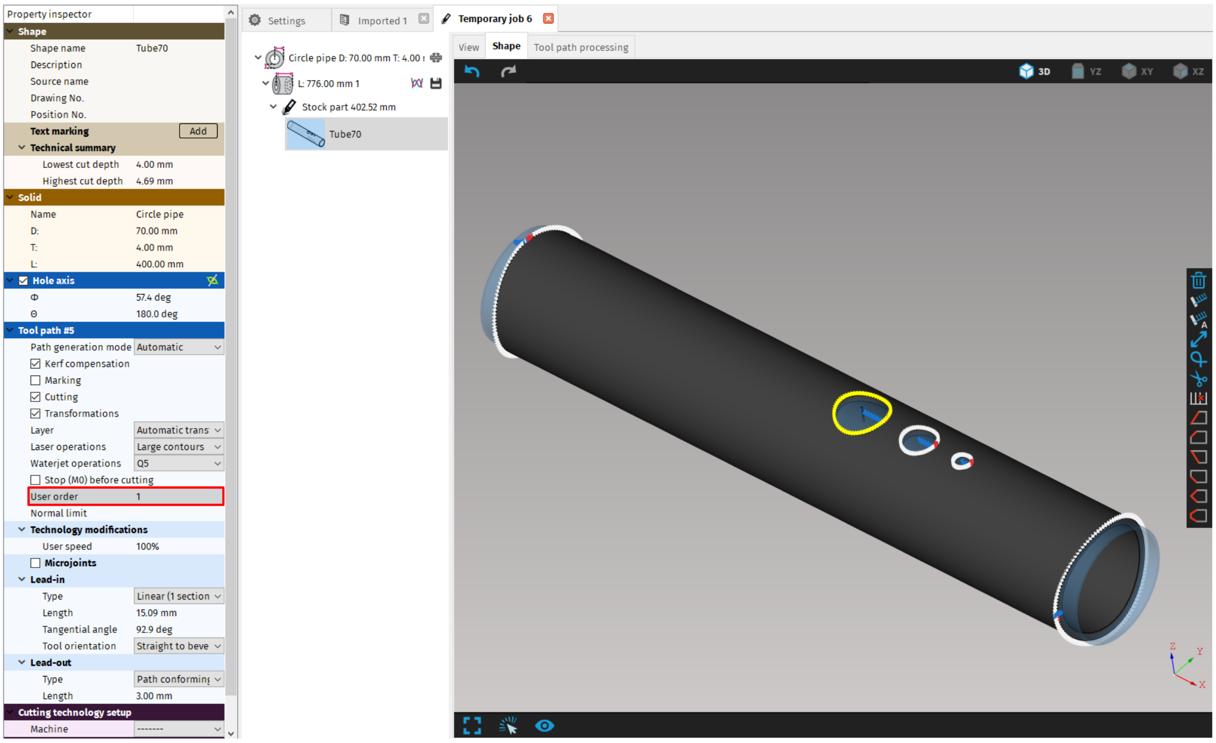


Fig. 252: Priority cutting – User order on selected cut

Complex cuts ordering (Y, X, K–cuts)

Ordering principle within the complex cuts is performed by the rule: *bottom cut → straight cut → top cut.*

Fig. 253: Complex Y–cut

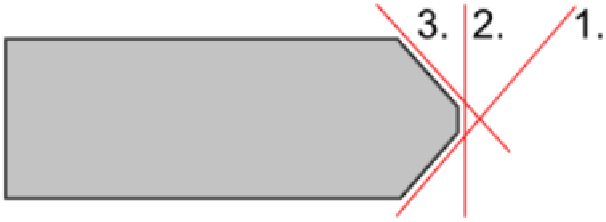


Fig. 254: Complex K–cut

Nesting

Nesting is a function used to create optimal placement of parts on the stock material. The function employs a sophisticated algorithm to place parts so that maximum utilisation of the stock material is achieved. The algorithm rotates, mirrors, shifts, and orders the parts to use the minimum amount of the stock material for particular job. Default minimum gap between neighbouring parts as well as dimensions of supports and length of non-cutting zone near chuck location are defined in *Application configuration level* in *settings* (see section Nesting

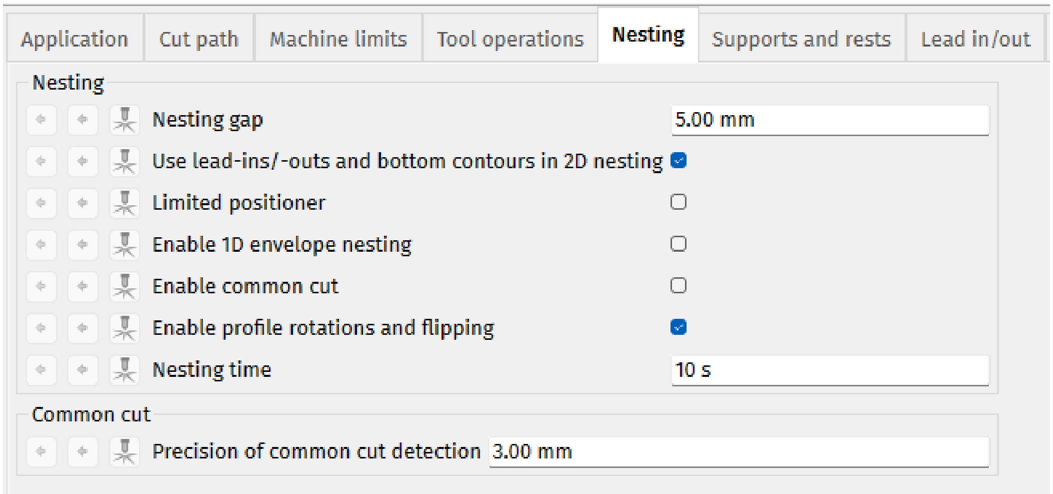
Subsection Nesting defines parameters regarding nesting of parts.

Nesting gap - represents the *distance* between individual parts. Nesting gap ignores lead-ins during the nesting so the gap must be larger than lead ins for parts.

Use lead -ins/-outs and bottom contours in 2D **nesting –** this parameter is used to prevent overlapping of adjacent parts caused by lead–in s/–outs or the bottom contours of these parts.

- **Limited positioner –** Defines if machine has a limitation for range of rotation (usually set from -398° to 39°)
- Enable 1D envelope nesting - allows to use simplified nesting process of parts that are enclosed by theoretical solid (i.e. pipes are enveloped by simple cylinders, H beam by simple cuboid, etc.)

- **Enable common cut - is a special feature that enables to** employ cutting between two adjacent edges that can be cut simultaneously by a single cut. The method is applicable only for cutting technologies that provide equal cutting quality independent on the side of the tool. Common cut can be used only on circular pipes.
- **Enable profile rotations** and flipping – if enabled, program will rotate and/or flip the material in the best position for cutting of the part.
- **Nesting time – sets maximum** time that nesting will run to prevent the algorithm running indefinitely when searching for best solution.
- Precision of common cut detection – specifies the value of precision of common cut.



- **Fig. 312:** Nesting settings

Supports1D nesting

In process of 1D nesting there are several important parameters that define the properties, quality and used nesting mode. These are the nesting properties:

- **Gap between objects –** defines the minimal perpendicular distance between nested parts.
- **Start point X offset –** defines an offset between program start point (X position of first cutting contour in NC code) and start point set by operator (or by optical sensor). Start point X offset helps to avoid piercing and cutting on the exact edge of the material by shifting start point (and whole CNC program) towards chuck location.
- **Use part envelope only** – allows to use simplified nesting process of parts that are enclosed by theoretical solid (i.e. pipes are enveloped by simple cylinders, H beam by simple cuboid, etc.).
- **Use common-cut –** is a special feature that enables to employ cutting between two adjacent edges that can be cut simultaneously by a single cut. The method is applicable only for cutting technologies that provide equal cutting quality independent on the side of the tool. The kerf width compensation is performed by a *mCAM* therefore the *expert tables* of selected tool should be defined precisely. The function can be enabled

or disabled only for the whole *Job task*, including all *stock*s and *templates* in it. Common-cut can be used only on circular pipes.

- **Surface map quality –** this setting is available only in case of nesting of non-enclosed parts (circle pipes, rectangular shaped pipes, etc.). This setting defines the quality of surface mapping (its trim contours) on parts to be nested, so it affects the final quality of nesting of complicated parts. Maximum time length is constant therefore the final nesting results (nesting quality) also depends on computer hardware performance. Surface quality mapping can be set on three levels:
  - → Fine (quality oriented)
  - → Normal
  - → Rough (performance oriented)
- **Enable part rotations and mirroring:** function enable/disable part rotations and mirroring so it increases utilisation of the material in nesting.

Note: Using the function Enable part rotations and mirroring while nesting H-beams, may cause serious problems (reverting of the cutting paths), so it is necessary to disable it.

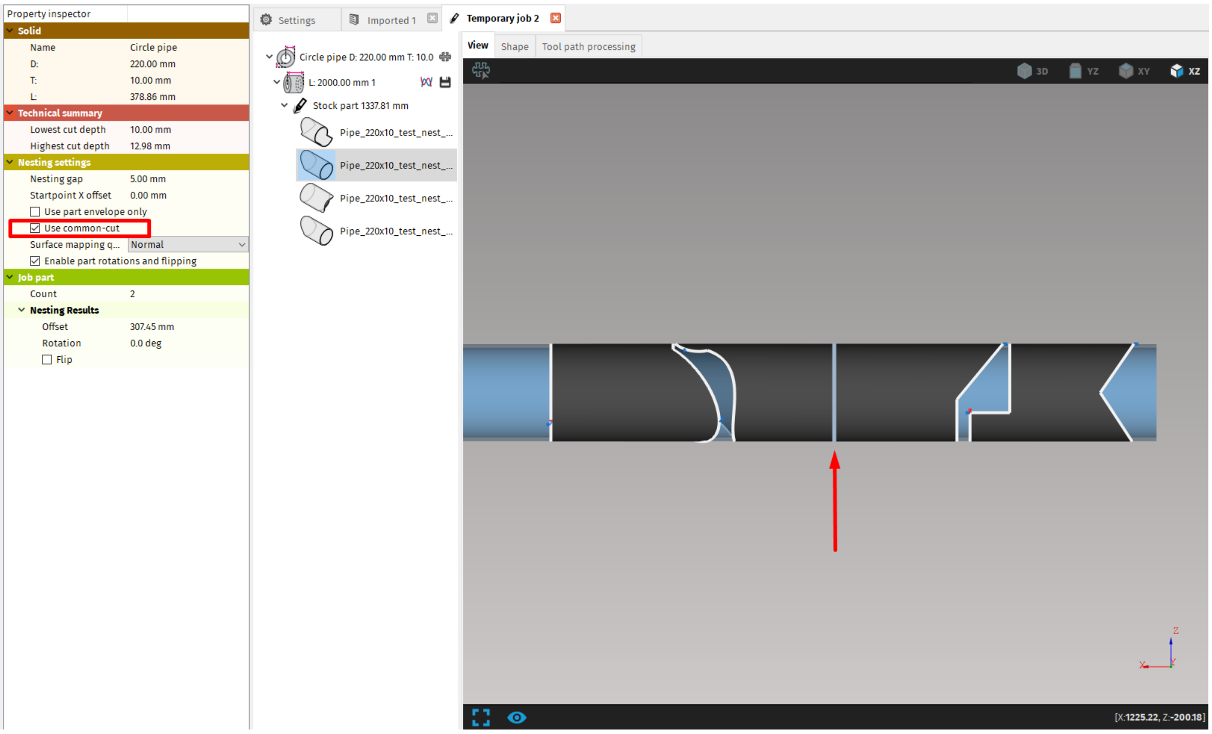


Fig. 255: Applied common-cut

There are two possible types of nesting:

- **Nesting of the whole template.** Allows to nest all parts within the same template (with the same dimensions – i.e. diameter and thickness), regardless if they have been already nested, on all available stocks within the template. Each *stock part* is a physical stock piece = one CNC program.

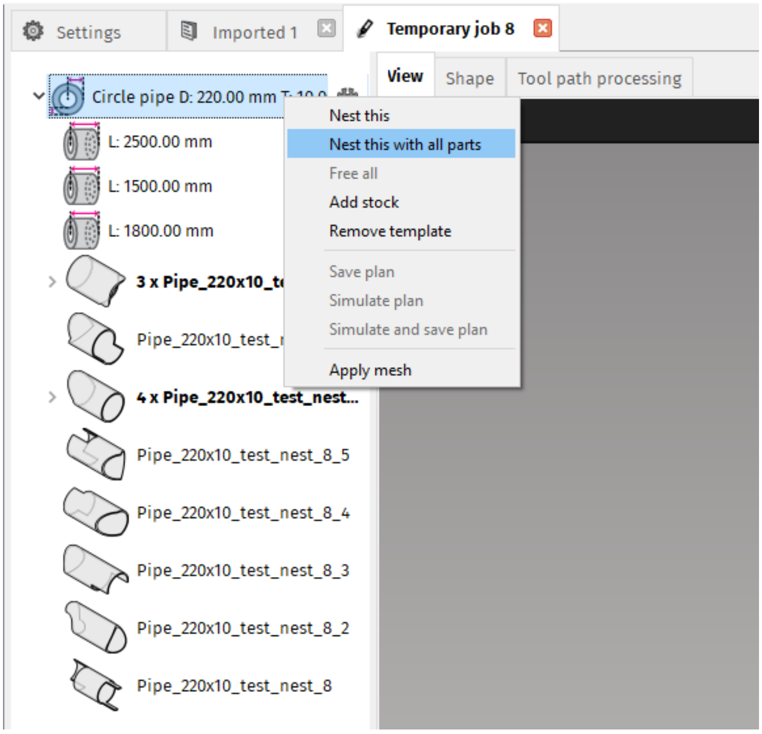


Fig. 256: Nesting of all parts within template Circle pipe D: 220 mm, T: 10 mm

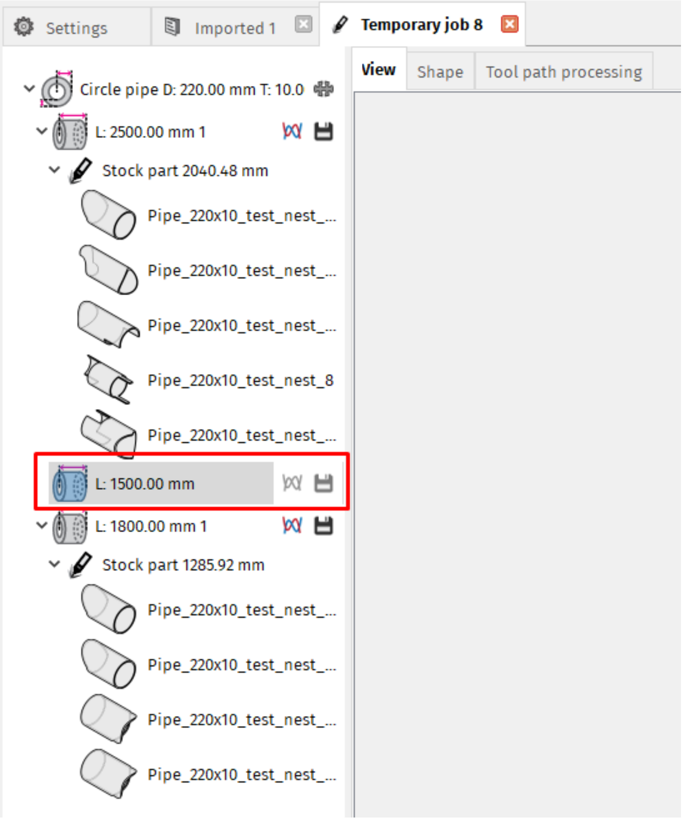


Fig. 257: Nested parts with highlighted unused stock

- **Nesting within the selected stock material.** This function allows to re-nest all parts within the selected stock material in case that cutting paths were changed in a meanwhile (after previous nesting). Other stocks and related parts remain non–affected by this nesting. Edited parts (i.e. lead–in or lead–out was changed in a meanwhile) are displayed by changed part icon.

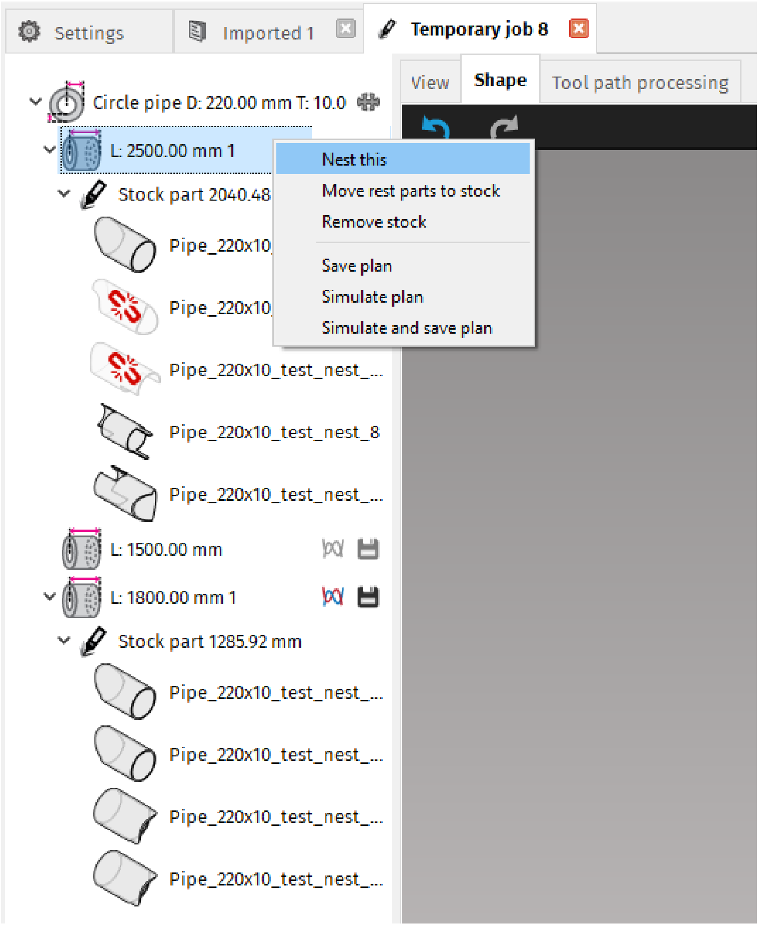


Fig. 258: Nesting within the selected stock

The function *Move rest parts to stock* allows to move all unassigned (non-nested) parts to some stock part. All unassigned parts or non–nested parts (parts that did not fit to any stock part during previous process of nesting because of short length of the stock or number of stock parts).

All unassigned parts or non-nested parts are displayed as *grey coloured parts* and are placed at the bottom of the list of parts. This feature makes it easy to identify and highlight parts that have not been processed by nesting (newly imported) or were not nested on any stock part during the nesting.

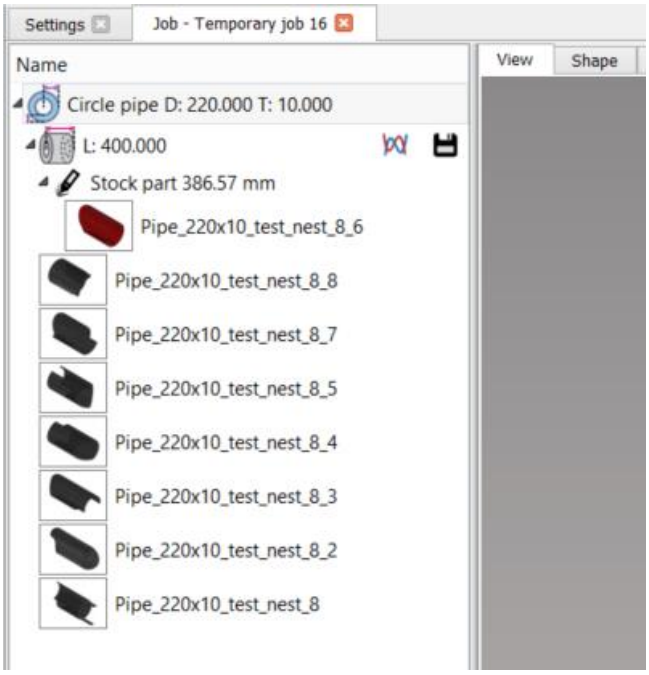


Fig. 259: Unassigned or non–nested parts (grey parts)

All templates that have been automatically created for particular shape type and dimensions (wall thickness, diameter, etc.) allow to use these functions:

- **Nest this –** this function initiates nesting process of all assigned stocks within the template without using remaining parts (all parts can be moved or transferred between all stocks within the template).
- **Nest this with all parts –** this function initiate nesting process of all assigned parts within the selected *stock* or all parts assigned on all *stocks* within selected *template* with using remaining parts (all parts, including remaining parts, can be moved or transferred between all *stocks* within the *template*).
- **Free all –** move all parts assigned to *stocks* within the template to unassigned status under the *template*. This is the same status as just after the import of these parts to the job. These unassigned parts can be assigned to any *stock* again.
- **Add stock –** add new stock piece with default length
- **Remove stock/template –** the function removes selected template including all assigned stocks and parts.
- **Save plan –** the function generates and saves cutting plans of all *stock parts* within the selected *stock* or *template*.
- **Simulated plan –** the function generates and simulates cutting plans of all *stock parts* within the selected *stock* or *template*.
- **Simulated and save plan –** the function allows to generate, simulate and save cutting plans of all *stock parts* within the selected *stock* or *template*.

Note: The nesting procedure is a complex and demanding task and duration of the nesting procedure depends on the computer performance and available resources, so it is important to meet at least minimum system requirements (see section System requirements on page 19).

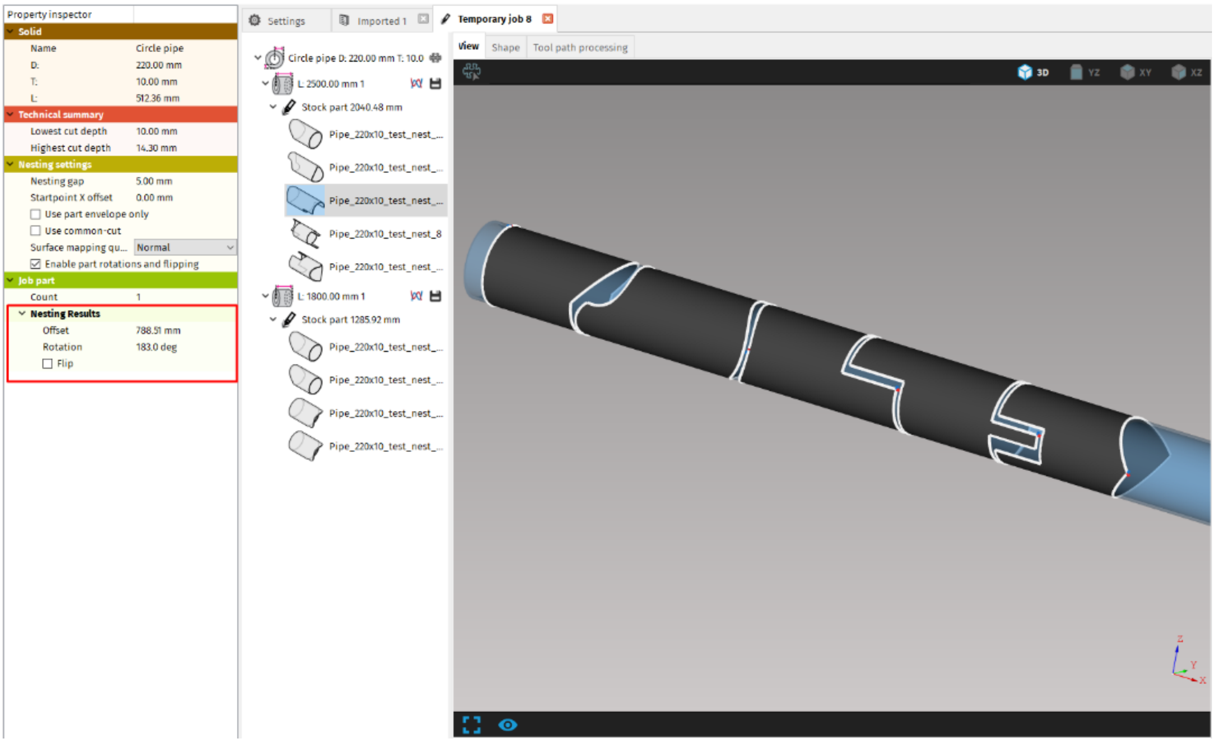


Fig. 260: Nested parts – highlighted selected part and its nesting results

2D nesting

2D nesting is used when working with surface parts and stocks (planar sheets). 2D nesting differs from 1D nesting by the method of usage and its whole nature. *mCAM* utilises so called *indic nesting* that is also implemented in *AsperWin nesting module*.

Plane nesting menu includes functions that are used to create optimal placement of parts available for particular plan. When reasonable, it is possible to modify these parameters in *Plane nesting* settings. Particular nesting settings are used for whole *job task*, which means all nesting settings that are displayed in *Properties* are applied to every *part*, *job stock*, and *template* in *Job task*.

Plane nesting menu contains these functions:

- **Nesting gap [mm]**– defines the minimum gap between nested objects (parts),
- **Bounding rectangles only** – nesting process uses only rectangular circumscription of each part (simplified nesting). This function is turned off by default (recommended).

- **Rotation step [°]** – defines the rotation step that is used in nesting process. Each part in particular can be rotated by this angular step to optimize final positioning on stock part.
- **Fit type** – defines the nesting strategy that is applied for particular job stock:
  - → *Quickest fit*
  - → *Center of gravity 1*
  - → *Grid 1*
  - → *Center of gravity 2*
  - → *Grid 2*
- **Use lead–in s/–outs and bottom contours** – this function is used to prevent overlapping of adjacent parts caused by lead–in s/–outs or the bottom contours of these parts.
- **Nest along axis** – defines a direction of parts nesting (in X axis or Y axis).

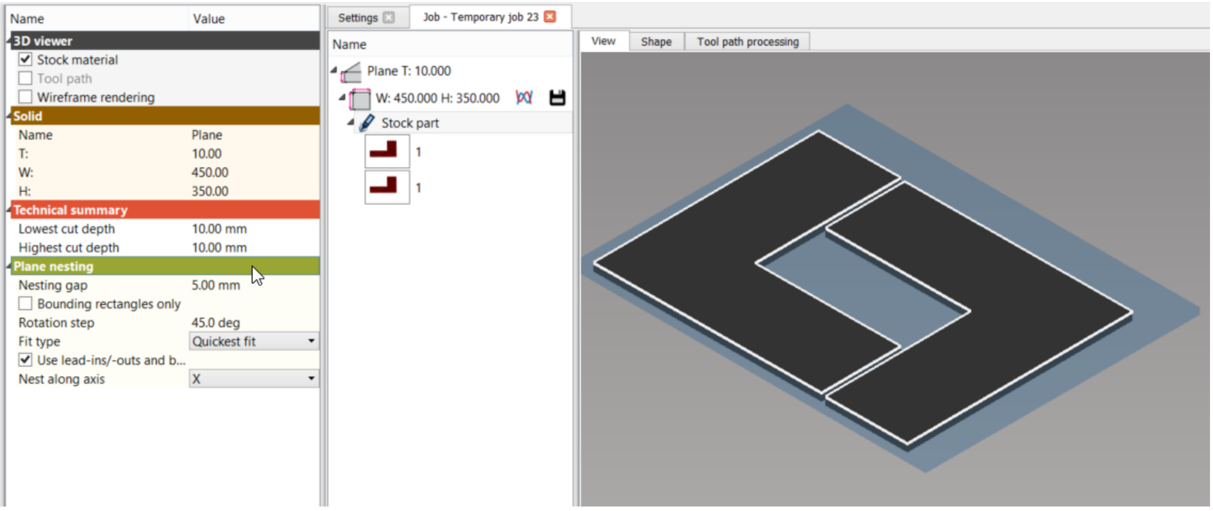


Fig. 261: 2D nesting –highlighted Plane nesting settings

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.

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/check by mCAM (when using manual nesting), therefore user takes full responsibility for possible subsequent cutting problems on machine.

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.

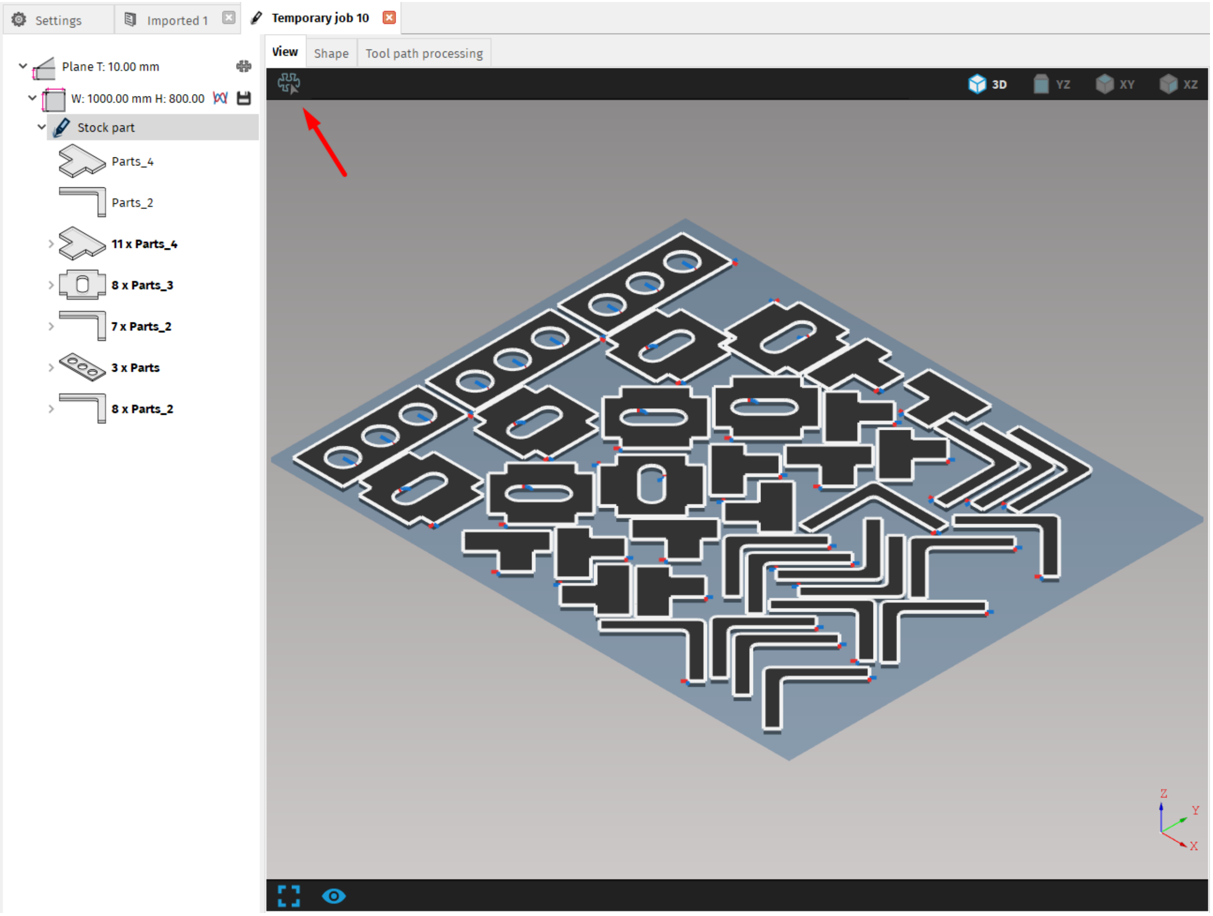


Fig. 262: Manual nesting button – manual nesting is not activated

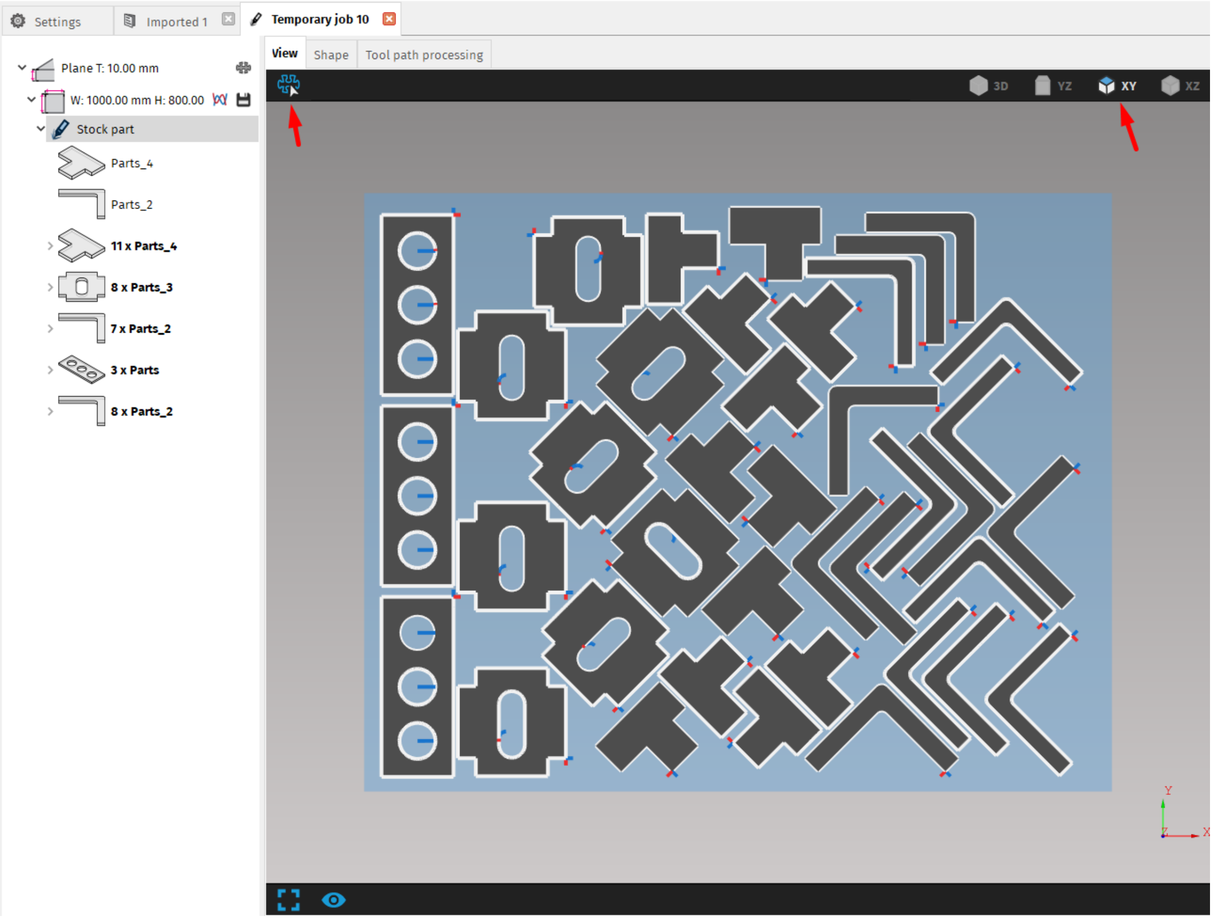


Fig. 263: Manual nesting is activated and the view is locked in 2D top view

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°. Holding the ALT key while rotating the parts will rotate them by 5°.
3. **Flipped** – only for pipes, flips the part around Z-axis (by 180°) with up and down arrow keys.

mCAM also allows to change the nesting results by manually editing *nesting results* of particular part in *Job part* settings in *properties menu* of selected part. *Nesting results* parameters differs according to nested shape type, i.e. circle pipe part position is determined by *Offset* (X – position), *Rotation* and *Mirror* status.

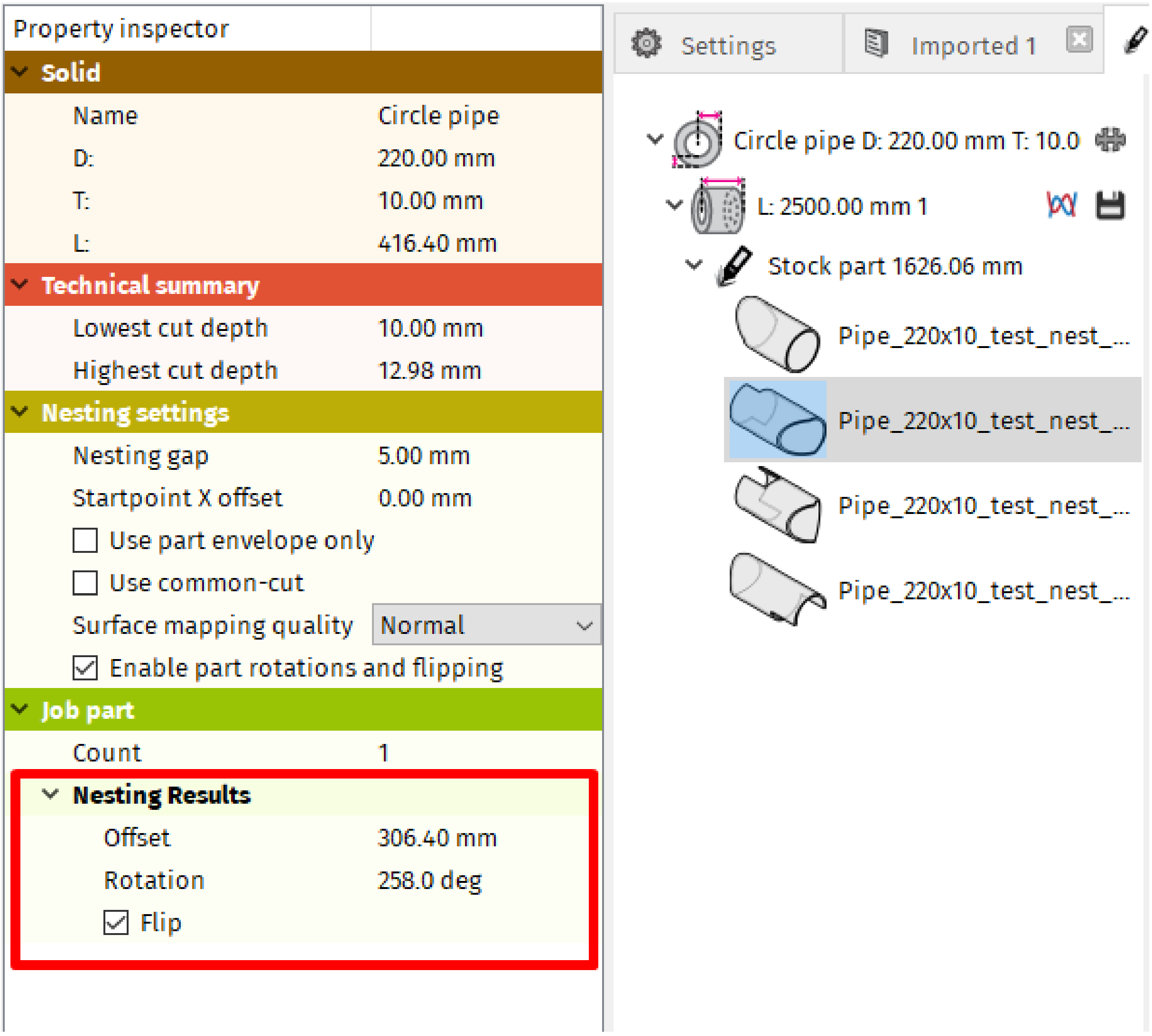


Fig. 264: Manual nesting of selected part

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
	Flip – flips the selected part (also with up and down arrow keys)
	**Move** – 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.

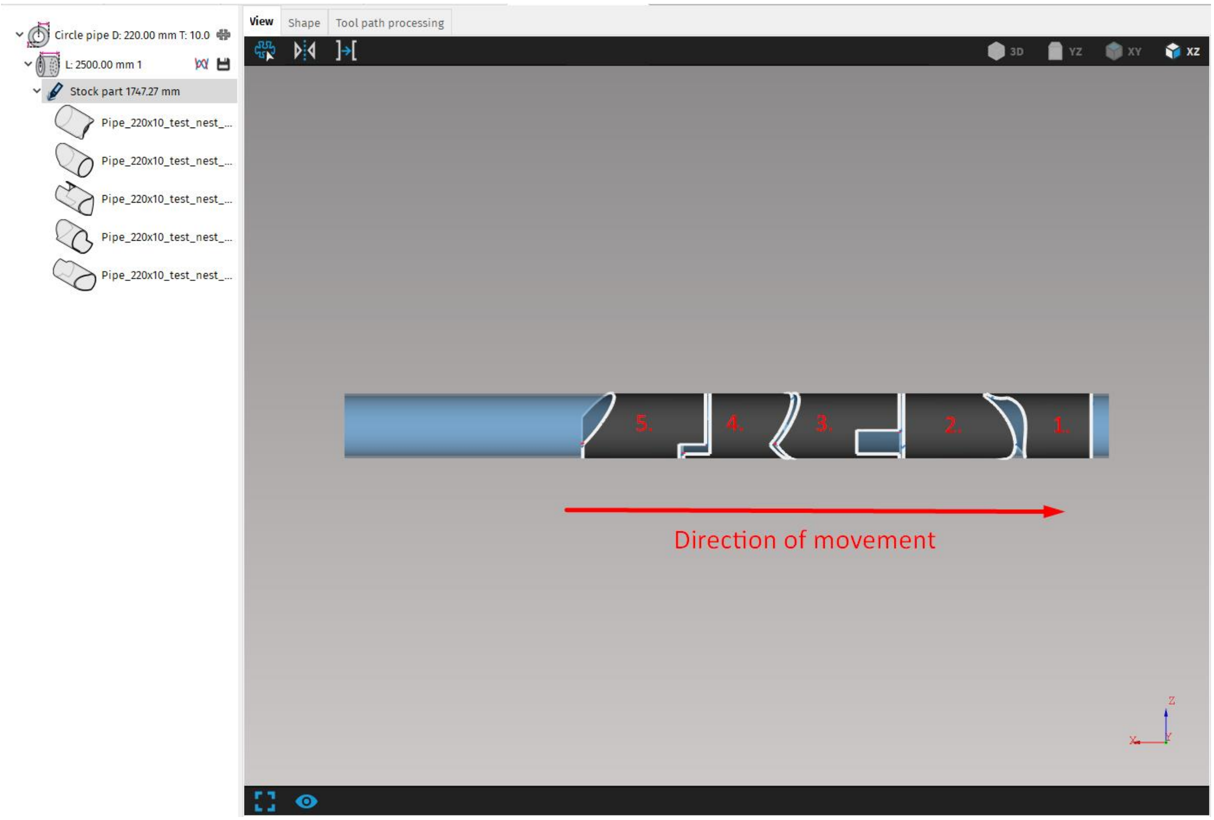


Fig. 265: Direction of movement of parts in active manual nesting mode in XZ view

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.

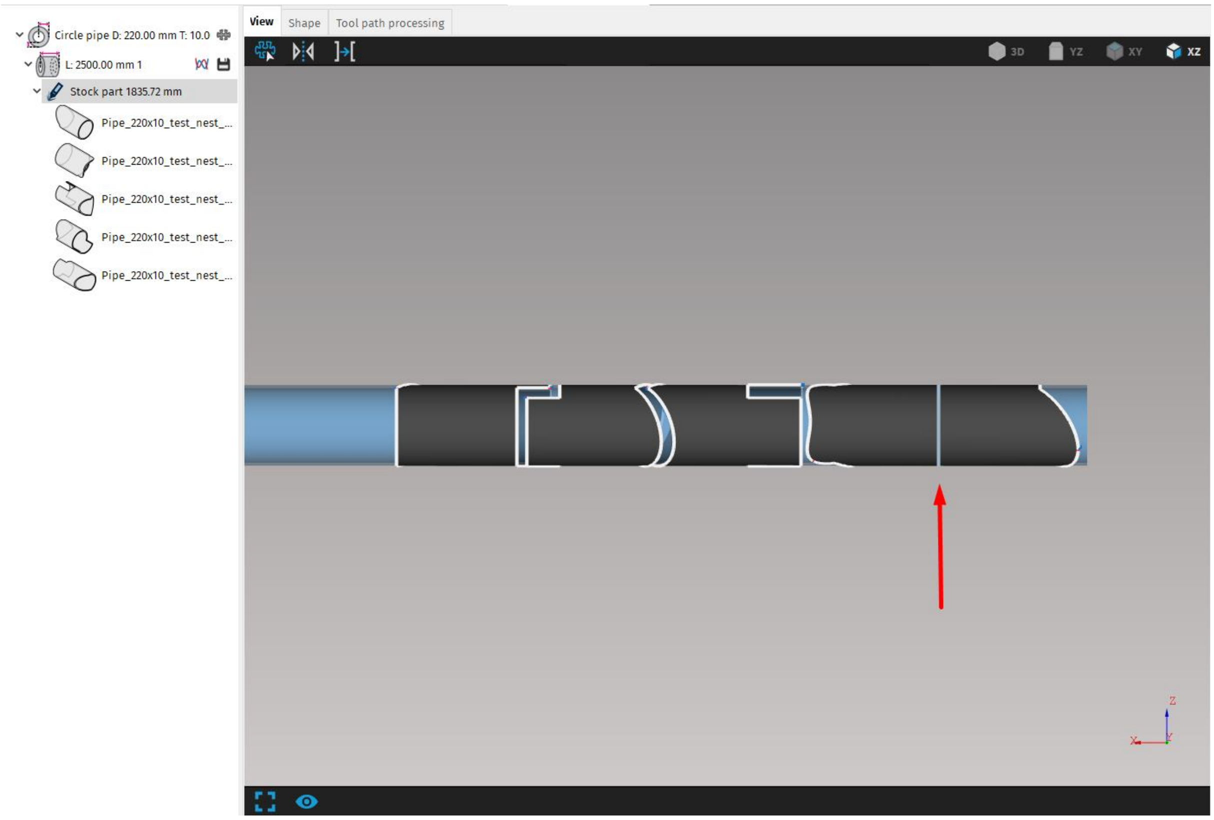


Fig. 266: Parts nested with applied common cut

Note: Selecting parts in manual nesting in visualization screen won't affect the selection in Working screen. Because the information in Properties area is based on selected part in Working tree, this 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.

Plan generation settings

Directly affects the way of NC code generating, used *Rotator mode* (3D mode/ fixed Y mode and profile cutting in rotary positioner/ on static support/ in rest), *Chuck location* (position of rotary positioner within the machine) and generated NC instructions (technological operations – *Marking/ Transformations/ Text marking/ Cutting*).

Name

Name of particular *CNC program* is defined by *Name* of *stock*. If *stock* contains more than one *stock parts,* names of all *CNC programs* within the *stock* are numbered (i.e. *name-1; name-2, name-3;* etc.). In example below, defined name results in generated *CNC programs*: Order224-1.cnc; Order224-2.cnc.

If there is not any *stock name* defined, name of *CNC program* is given by name of first *part* nested on particular *stock part* (i.e. *part-1; part-2; part-3,* etc.).

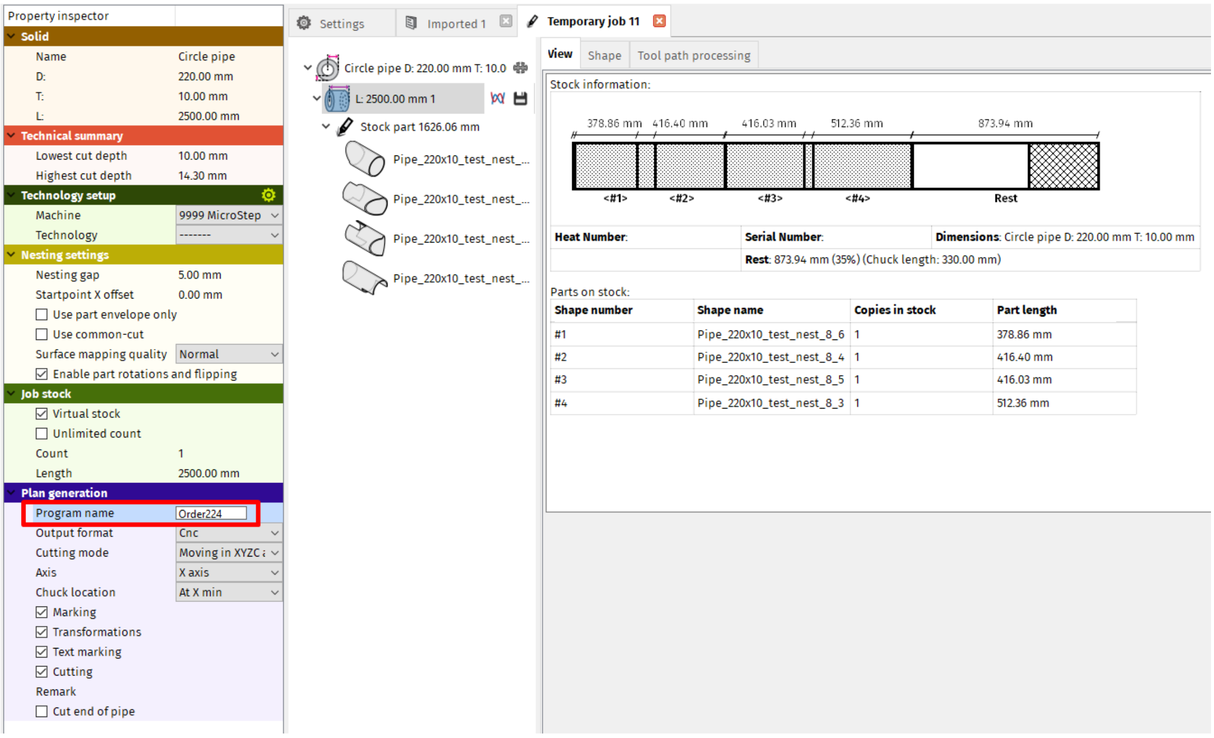


Fig. 267: Defined stock name

Chuck location

Chuck location describes location and orientation of a rotary positioning device. The positioning device can be located either at X min, or X max with respect to machine coordinate system. That means that location of the clamping chuck has to be set up to match the actual coordinate system of the particular machine. The chuck position is defined in *Settings – Machine limits – Machine layout* (as a *Machine – technology* preset) or it can be defined manually in *Plan generation* properties of each *stock* piece.

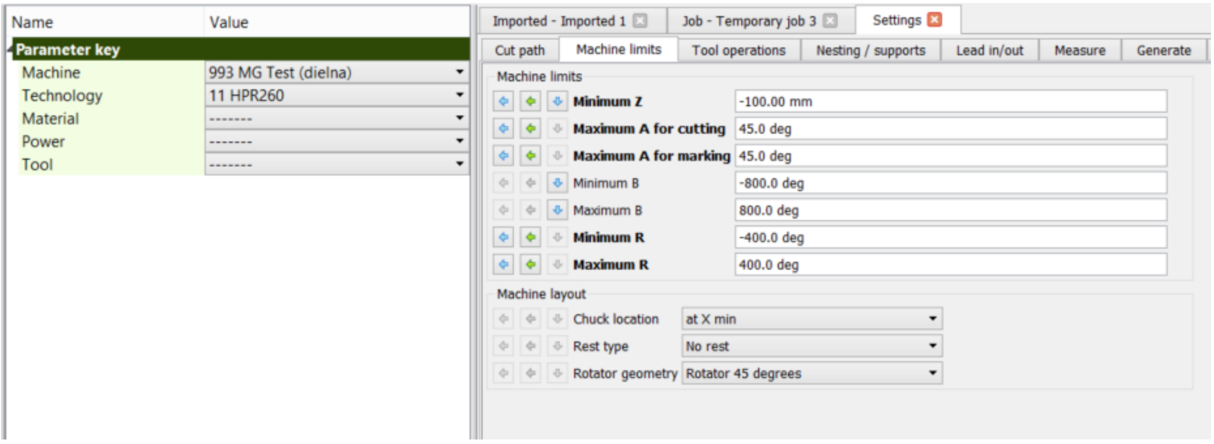


Fig. 268: Machine layout presets related to machine and technology

Output format

NC code is the only output from the *mCAM* in form of a text file containing instructions for controlling a technological device that. The *mCAM* supports DIN standard format (so called G–codes and M–codes).

Commentaries in NC code displays information regarding time and date of generating particular NC code, used cutting mode, used cutting tool or stock dimensions. Commentaries before every single cut also contains info about measurement type, applied kerf compensation values, adaptivity mode (on/off) or operation type.

mCAM does not allow to edit the generated code because the code for 3D cutting is too complex for manual editing and improper manual changes may lead to potentially dangerous behaviour of a machine.

```
(mcam.ver= 1.3.1.115 2015.10.22)
(mcam.date=29.10.2015 18:00:10)
(PIPE_DIM 477.0 220.0 0 10.0 0 -)
(mcam.axis.type=ROT_POL_MOD2)
(mcam.grip=X_MIN X)
(mcam.stock=Circle pipe D: 220.000 T: 10.000; L: 477.000)
(mcam.item=1 'Pipe_220x10_test_nest_8_5' 'C:/msnc/mcam21.10/examples/Pipe_220x10_test_nest_8.STEP' :5)
(29.10.2015 18:00 'msnc500')
(MACHINE 1 Virtual machine)
(TECHNOLOGY 11 Pantograph 90 degrees)
(TOOL 11 tool)
(SEMIPROD 1; 10 220; 477)
(SEMIPROD: CirclePipe Thickness=10 Diameter=220; Length=477)
(TRSF _0_1_0_1_0_0_0_1_0_0_0_1_0_0_0)
(SEMIPROD_MIN 1; 10 220; 425)
(SEMIPROD_MIN: CirclePipe Thickness=10 Diameter=220; Length=425)
(MODE ROTATOR XY 220.0)
(START X_MAX)

(mcam.part mode=CUT item=1 holeID=2 cut=3/DEFAULT/0/UPPER blocker=OTHER cutID=9)
(mcam.info type=MAX_X grip=X_MIN act=R_CONST measure=PLANE adaptivity=OFF)
(mcam.info2 cutHints=CUT_DIVISION_BEFORE|CUT_DIVISION_AFTER|CUT_JOIN_AFTER|CUT_END_PART quality=Q5|LARGE kerf=<2.7,2.8>)
(mcam.tech path_cut_data kerf:ON limitA=   45.0 AV=VL_AM HW=   90.0 mJoint=4; )
(PART 'Pipe_220x10_test_nest_8_5' 1 2155254)
(SIZE 1; 10 220; 416.028)
(SIZE: CirclePipe Thickness=10 Diameter=220; Length=416.028)
(mcam.expert.tool  Operation 2. thickness=   10.00 speed= 2000.00 kerf=    2.70 power=110 A)
M6 T311 (CUT BEVEL_V 3)
G0 C6.090
M94 D5
G94 X0.000 Y-11.690 Z-0.623
G94 X-30.229 Y-55.077 Z-14.781
G94 X-55.585 Y77.439 Z-31.877
M94
M38 L0.000 L179.389 (Corr A from B 0.0. Add=0.0)
G0 X-53.700 Y89.656 Z-46.268 A55.228 B0.000 C6.090 (setB=0.000)
M20 D2
M3
M90 D1.468
G1 X-55.027 Y89.290 Z-45.756 A51.900 B12.207 C6.090
M90 D1.330
G1 X-56.118 Y88.844 Z-45.140 A50.455 B18.888 C6.090
M90 K159 L95.000
M90 D1.305
G1 X-57.056 Y88.305 Z-44.409 A49.355 B25.747 C6.090
M90 D1.280
G1 X-57.814 Y87.686 Z-43.584 A48.644 B32.728 C6.090
M90 D1.321
G1 X-58.441 Y86.979 Z-42.660 A48.076 B34.224 C6.090
M90 D1.346
G1 X-58.894 Y86.197 Z-41.662 A47.446 B34.616 C6.090
```

Fig. 269: NC code generated by mCAM

Cutting mode

This option defines if the particular part (shape) is cut with rotator *Moving in XYZ* (3D rotator where all axes are controlled) or with rotator *fixed Y,* where Y-axis of rotator is not controlled or with *plane* cutting mode (cutting beams on static supports). This setting match with the setting rotator mode in iMSNC. *Rotator mode* is available to set in the section *Plan generation* in the *Job task* folder for each *stock*.

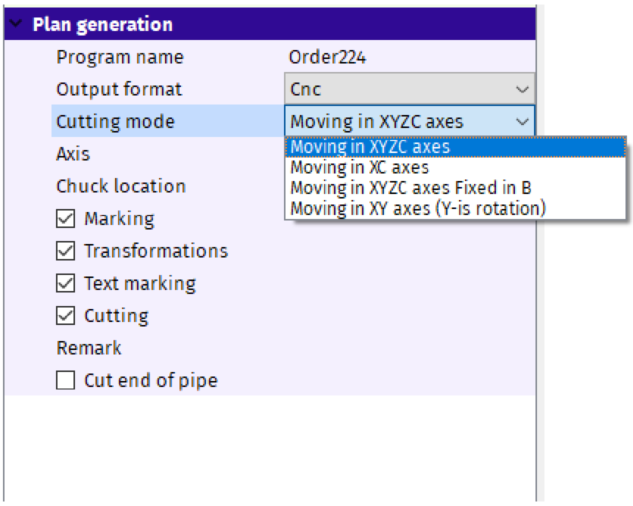


Fig. 270: Plan generations settings – Cutting mode

Default *Rotator modes* used for each shape are defined in *Settings – Generate*.

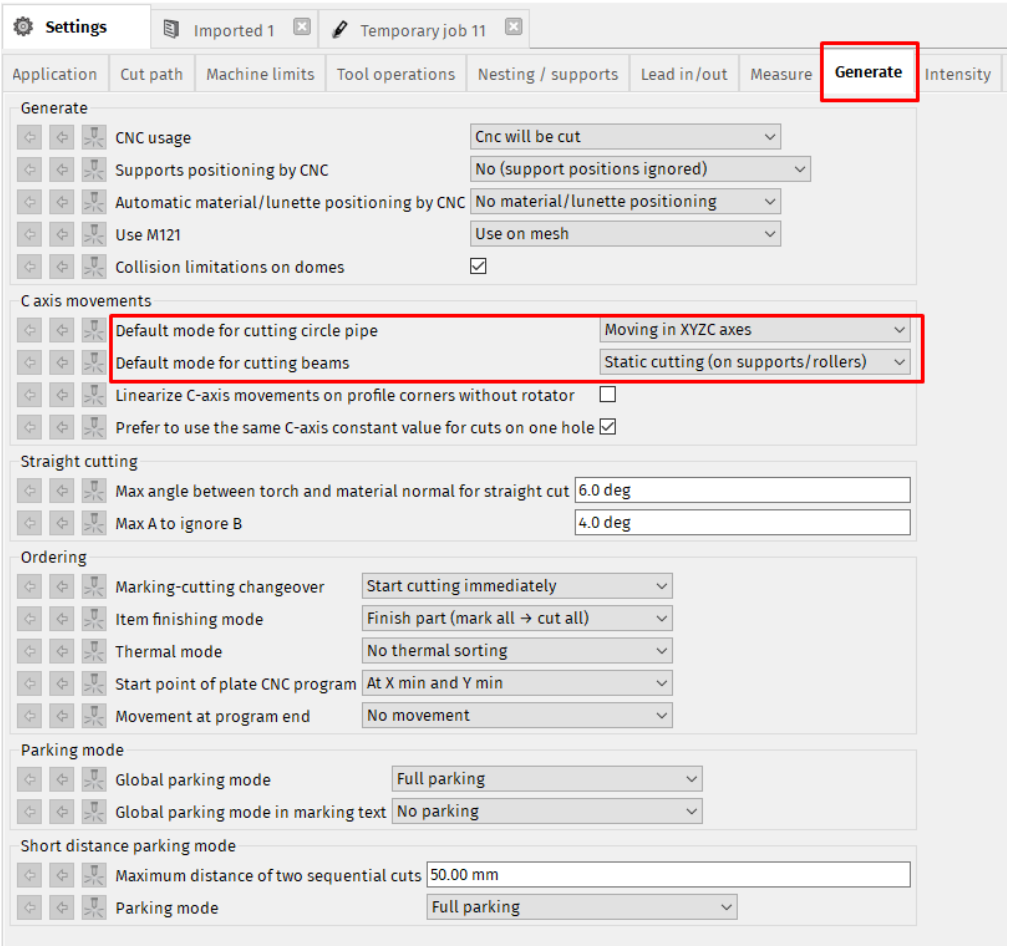


Fig. 271: Predefined rotator modes

Circle pipe cutting is able to use in XYZ mode (so called 3D mode) where all cuts (within the range of particular rotator) are performed only by rotator movements without using rotary positioner (movements of pipe). If cutting plan for circle pipe cutting is generated using *Moving in XYZ* mode, *NC code* contains all movements instructions, including rotator movements and movements of the rotary positioner (X, Y, Z, A, B, C). When using *Moving in XYZ* mode *XY* pipe cutting mode in *iMSNC* have to be enabled.

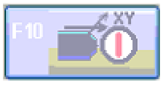


Fig. 272: Pipe cutting mode in iMSNC – Moving in XYZ

If cutting plan for circle pipe cutting is generated using mode *fixed in Y* mode, NC code contains movements without C instruction (angular movements of rotary positioner while cutting) *NC code* contains movement instructions of X, Y, Z, A and B axes. Movements of the rotary positioner are automatically calculated according to Y instructions in *NC code* and set diameter of circle pipe (defined when enabling *pipe cutting mode* in *iMSNC).* When using *fixed in Y* mode *fixed rotator pipe cutting mode* in *iMSNC* have to be enabled.

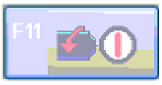


Fig. 273: Pipe cutting mode in iMSNC – fixed in Y

Note: Keep in mind that non–correctly set circle pipe diameter when enabling any pipe cutting mode may result in different cutting speed, not finished trim cuts or not correct dimensions of final parts.

Technological operations generation

Resulting *NC code* instructions (instruction that will be generated to *NC code*) are given as a combination of *cutting path properties* and *stock properties* defined in *Plan generation*. Generation of each technological operation to *NC code* needs to be confirmed (if user wants to generate particular instructions to *NC code*) by checking the checkmarks in *Plan generation*. By default, generation of all instructions to *NC code* is enabled in *stock properties.* By default, generation of all instructions, except *Marking,* in *Cutting path properties* is enabled.

The example below this section shows generation of *Marking* that is enabled for particular cutting contour, but generation of *Marking* instructions in *Plan generation* of particular *stock* is disabled, therefore no *Marking* will be generated to *CNC program*.

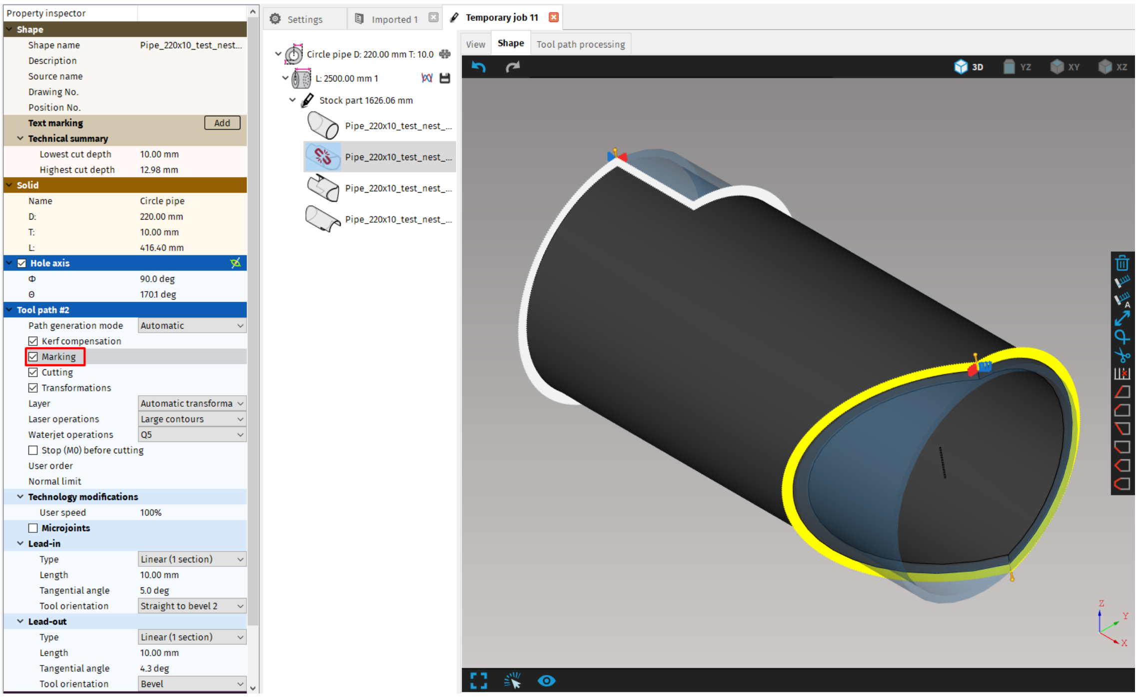


Fig. 274: Cutting path properties – marking enabled

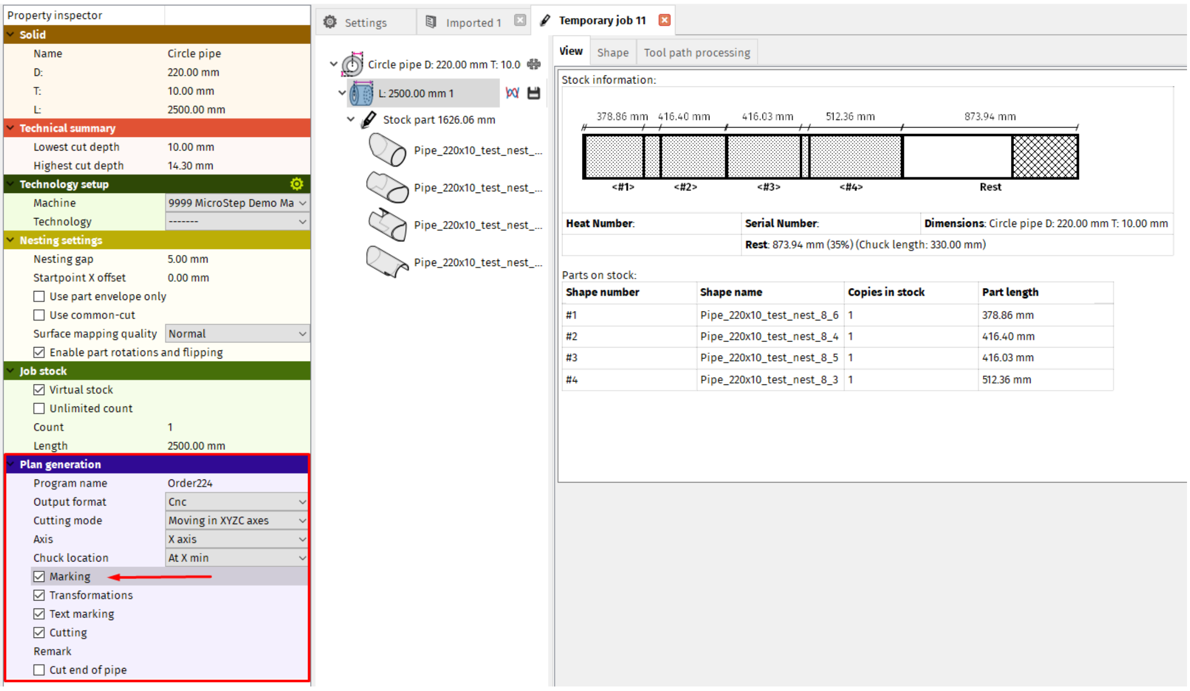


Fig. 275: Stock properties and plan generation options

Cut end *of pipe*

This function enables to perform automatic straight trim cut in the end of pipe, so the rest of used *semiproduct* (after cutting of all parts in the *CNC program*) will be suitable for next *nesting* and *cutting* by *mCAM*. Function is able to use for closed profiles (circle pipes, rectangular shaped pipes), not for open profiles (H–beams, U–beams, L–beams, etc.).

Cutting plan generation

Prior to generation of a NC code, selected stocks, or whole template that may contain several stocks, should be generated to cutting plan. Cutting plan is simulated or saved by using buttons nearby the *Stock*.

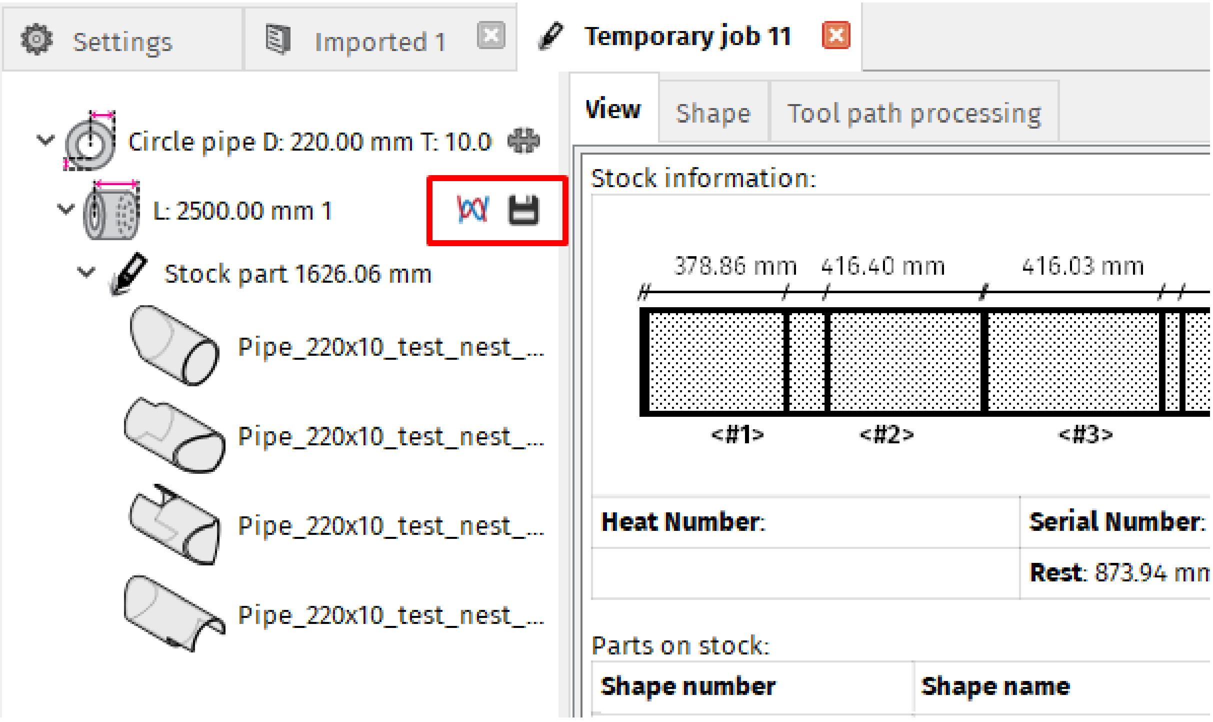


Fig. 276: Generate – simulate/save

Functions Save plan; Simulate plan; Simulate and save plan *are always applied on currently selected* job item *(*template/stock/stock part*)*

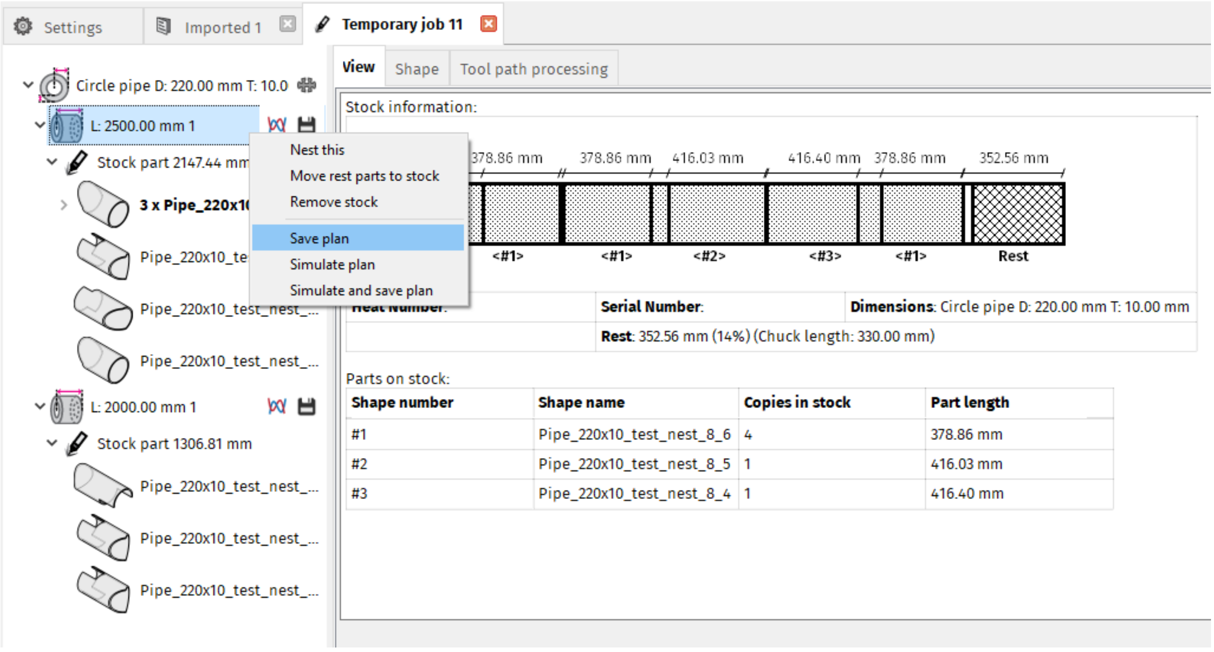


Fig. 277: Save cutting plans for stock with length 2500 mm

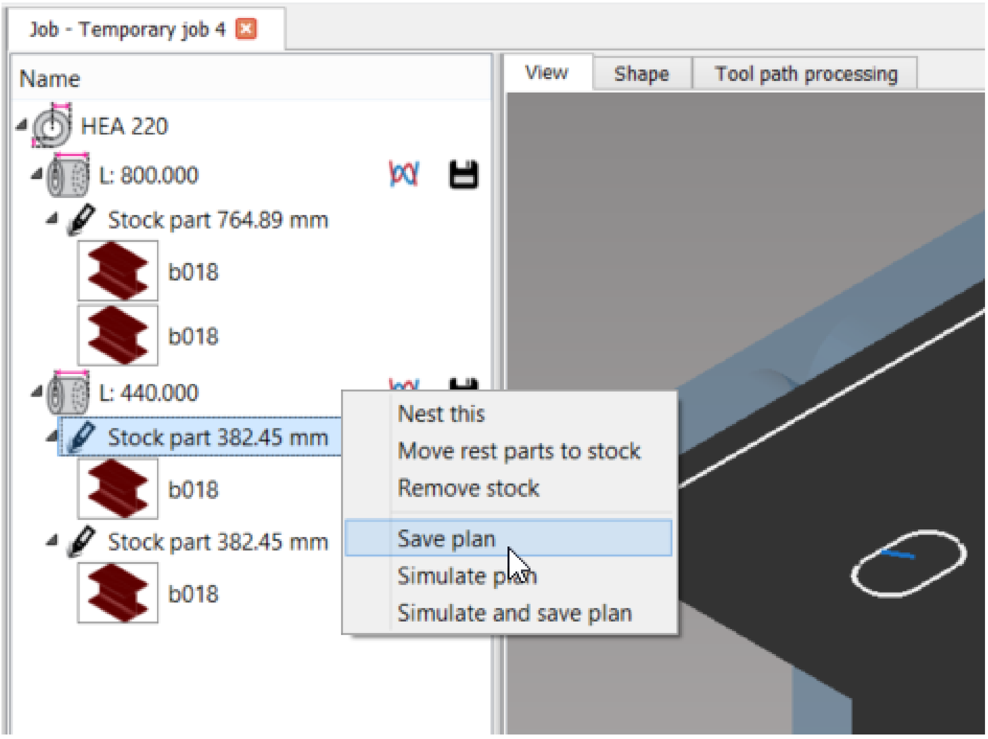


Fig. 278: Save cutting plan for first stock part in stock with 440 mm

Name of each particular plan is defined in *properties* of selected *stock part* in item *Name*. In case that *stock part* name is not defined, name is given by the first part in that particular stock part and the number of other parts in it.

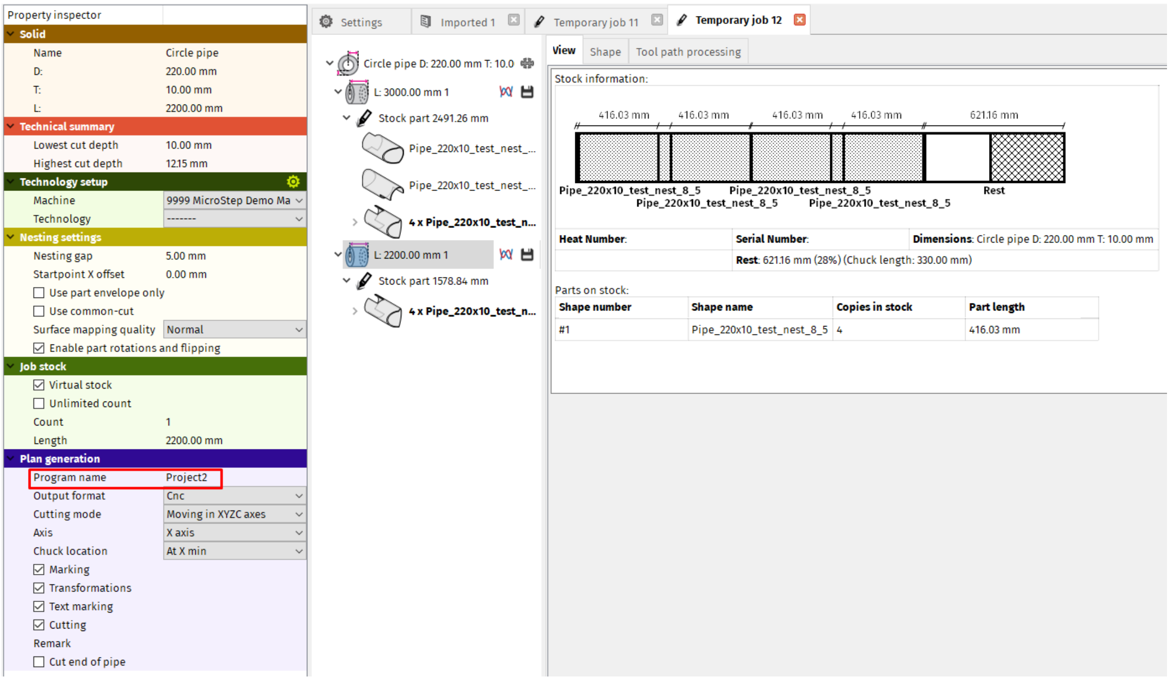


Fig. 279: Plan generation settings (name) – stock properties

Cutting simulation of CNC program

After a CNC program is generated, the simulation tab task appears. It contains four main sections: visualization screen (simulation/report/configuration), NC code viewer, watch panel (graphs) and blocks panel.

NC code viewer exposes the source code of the generated program. Each contour (feature) is highlighted by coloured line (in right side of the screen) while the line colour depends on used technology. E.g. cutting contours are highlighted by red colour, marking by blue, etc.

Visualization screen in simulation allows to display:

- ***3D simulation*** where 3D model, surface detection points, actual position and tool movements are displayed
- ***2D simulation*** where all movements are displayed by projection in XY plane
- ***Report*** where html report that is generated with CNC program
- ***Configuration*** that was used for generation of particular cutting plan

3D simulation

3D simulation displays stock, cutting paths, cutting tool, supports positions, chuck location in three – dimensional space. Simulation allows identifying potential mistakes, imperfection of cutting path, ordering of the cuts, possible collisions with supports.

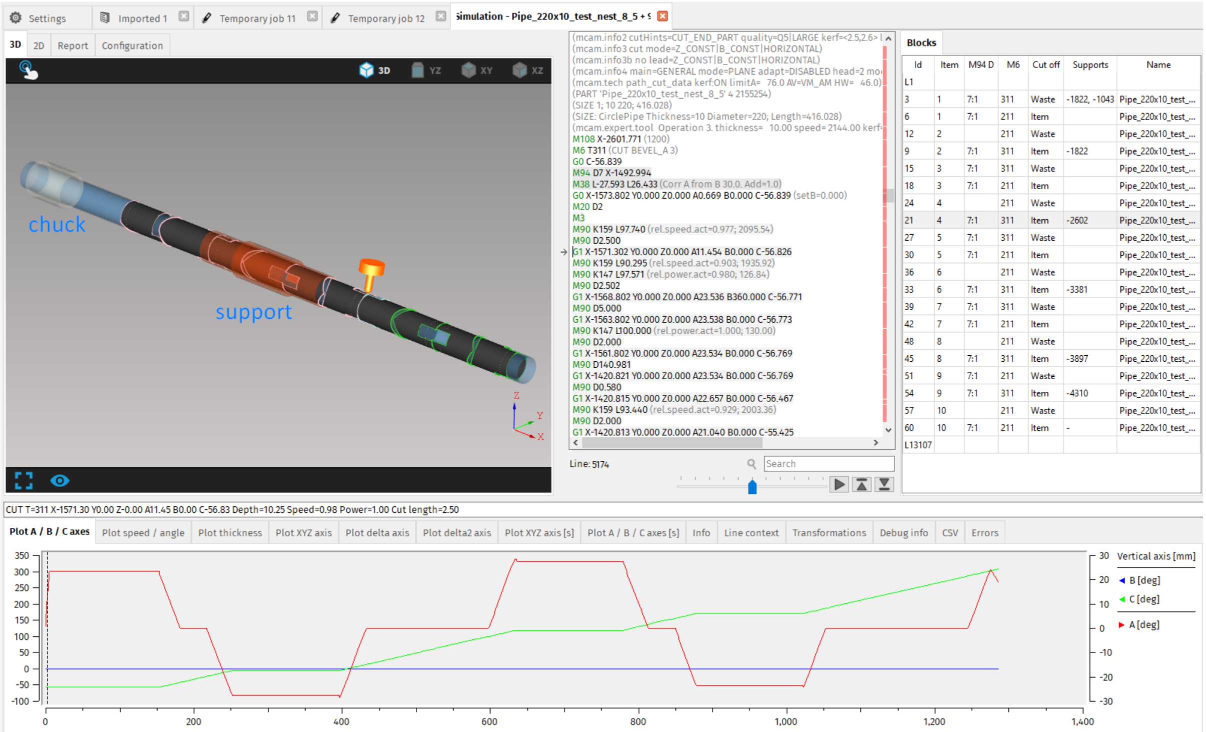


Fig. 280: 3D simulation with highlighted chuck location and support position

3D simulation controls

It is 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.

Visualization of limitation of cutting paths in simulation

Sometimes, it happens you generate a shape with cutting paths that are not possible for the machine to cut (too steep angle, limitations of material or the machine tool, and so on). When this happens mCAM automatically limits the cutting paths to fit the parameters that the machine is able to cut.

The visualization in the simulation now shows these changes by turning **the cutting paths red**. This change of color is meant to warn you, that the desired cut (hole, etc.) will be generated with different geometry and size. The paths that the machine is able to do without the limitation are showed in pink as before.

This limitation can occur in several cases:

- **A – axis hardware limitation –** is a limitation of the rotation angle of a cutting head. Every head has a maximum tilt and some can´t change their tilt at all (0°, 45°, 90°, 120°). In some cases, if the head cannot change its tilt to a specific degree, mCAM will limit it, it will change the angle of the cutting paths. This change will be seen in simulation as **red cutting paths**.

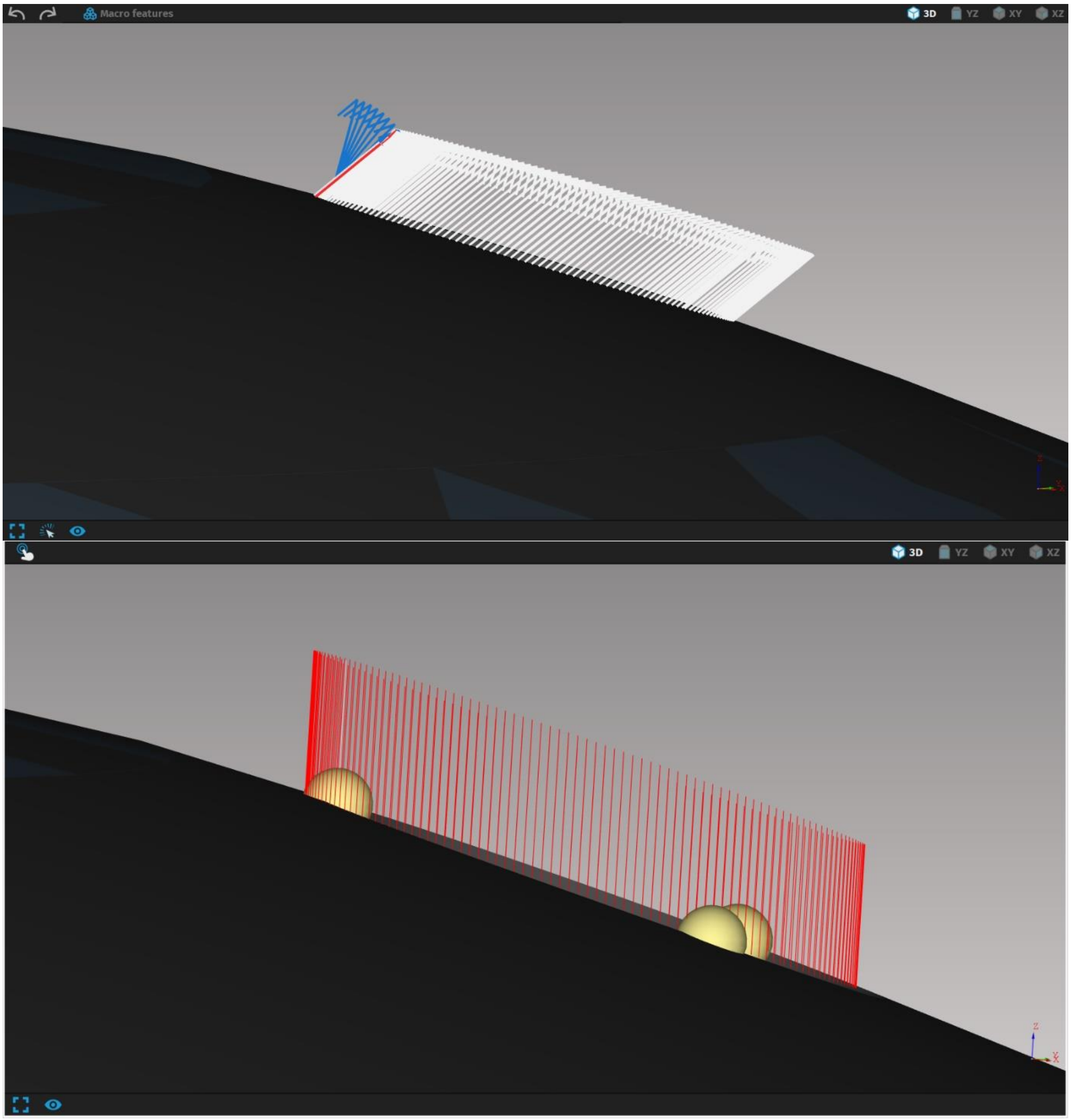


Fig. 281: A-axis hardware limitation before the simulation and after the limitation in simulation with 0° maximum tilt cutting head

- **Limitation of machine angle between tool and material surface normal –** after exceeding a maximum tilt between the tool and material surface, the cutting technology loses its effectivity and is not able to cut through the material. In this case mCAM limits the tilt of the cutting paths, so the used technology is able to cut properly. These changes are shown in simulation using **red color**.

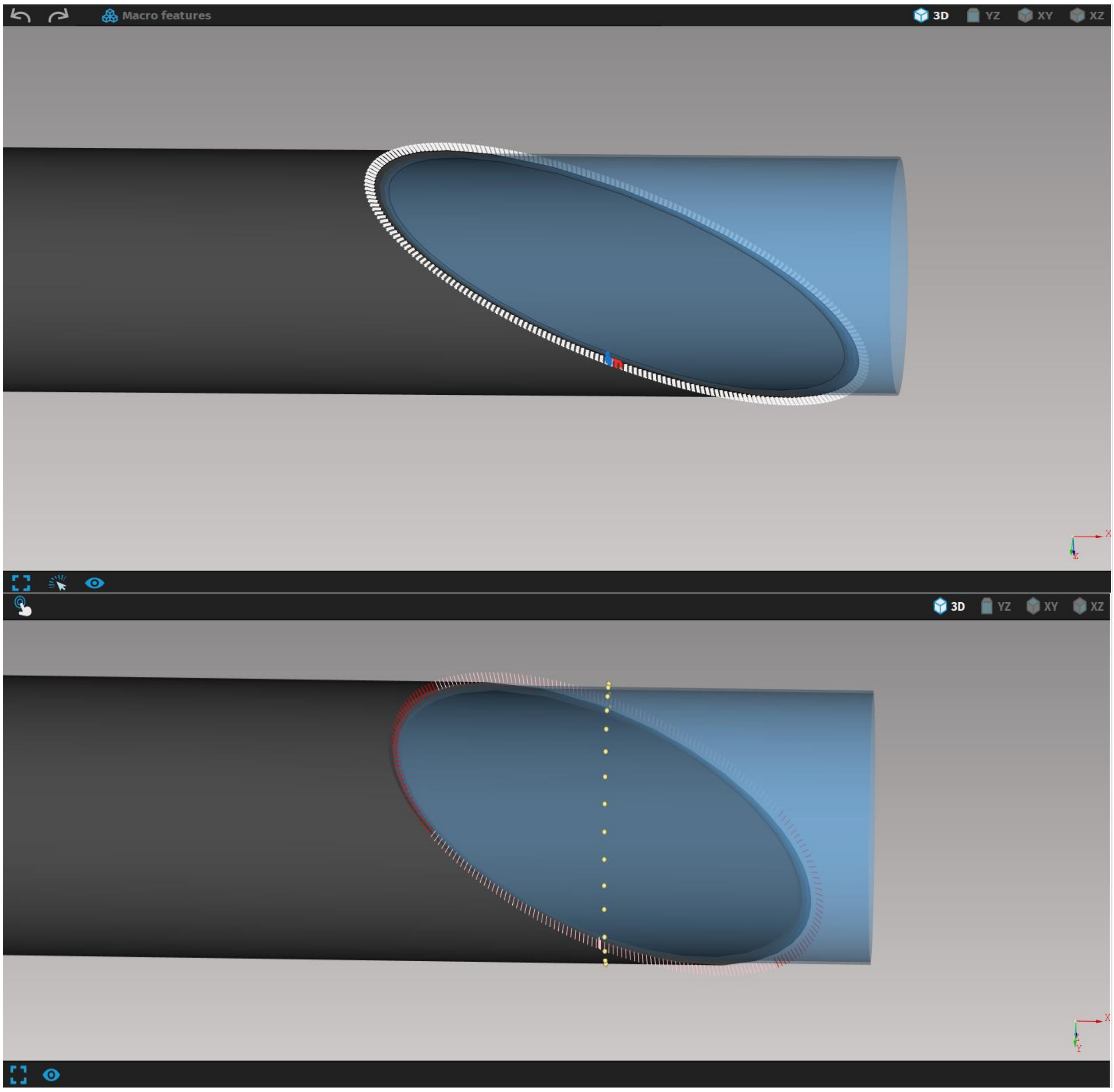


Fig. 282: Limitation of machine angle between tool and material surface before and after the limitation in simulation

- **Potential collision limitation –** this limitation is applied to avoid a collision between the cutting head and the material/ground. mCAM automatically changes, limits, the tilt of the cutting paths to avoid such situation. This change is shown in simulation by **red colored cutting paths**.

2D simulation

Cutting paths and tool paths in 2D simulation mode are displayed similarly as on the iMSNC machine system. 2D simulation allows watching CNC instructions for every particular cut–path section in details by mouse cursor on cutting plan.

If iMSNC provides information, simulation can in 2D representation show the whole frame of the machine (in yellow color).

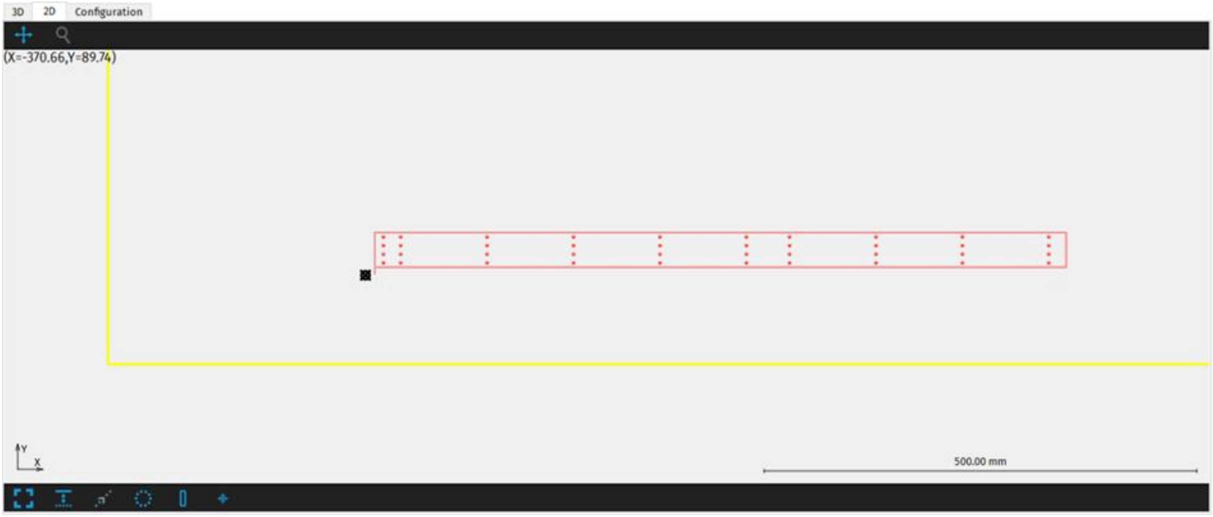


Fig. 283: Frame of the machine displayed in yellow color in 2D view in mCAM simulation

Detailed zoom of particular cut–path section is executed by mouse selection of the area under inspection. Exact line in NC code is displayed by using mouse pointer on particular contour in 2D cutting simulation. Zoom out is executed by click on random place in program desktop.

Zooming-out in 2D representation can be done even above fit-all view. With decorative data (table frame counts as decorative data) zoom-out can be done up to the 10x of their dimensions.

2D simulation controls

What is displayed in 2D view of simulation is controlled by activating or deactivating controls at the top and bottom of the screen:

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

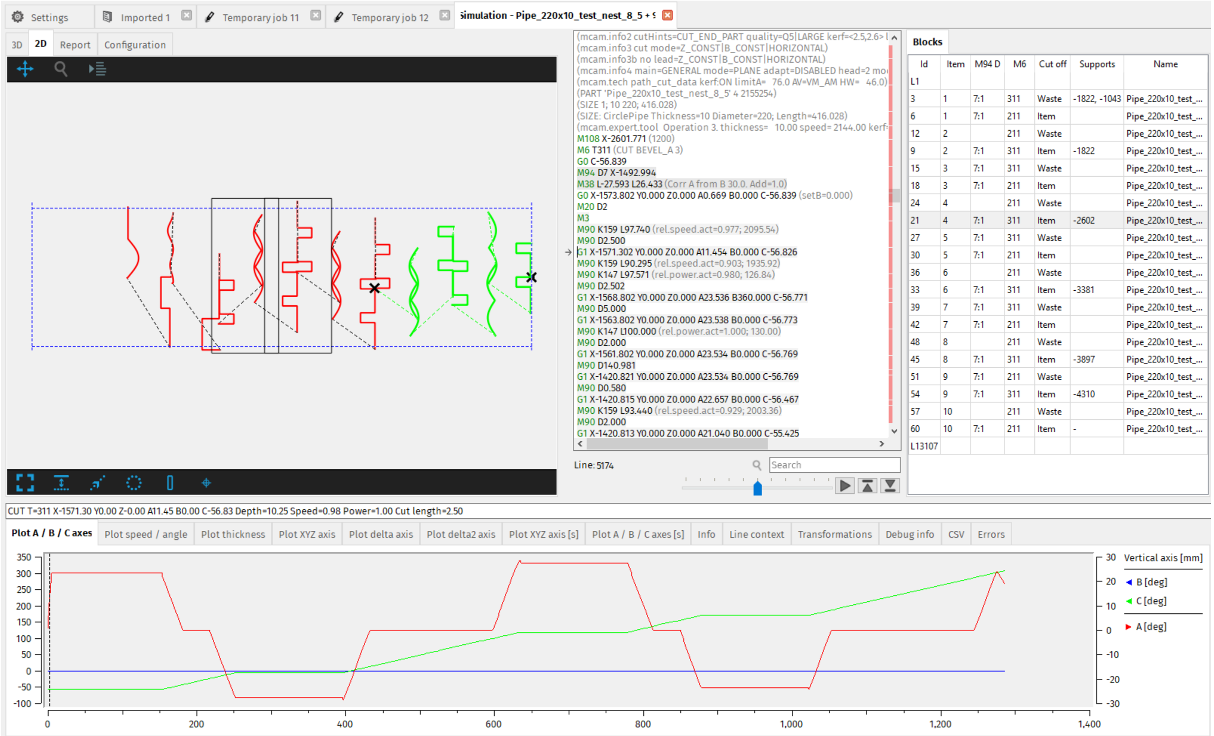


Fig. 284: 2D cutting simulation

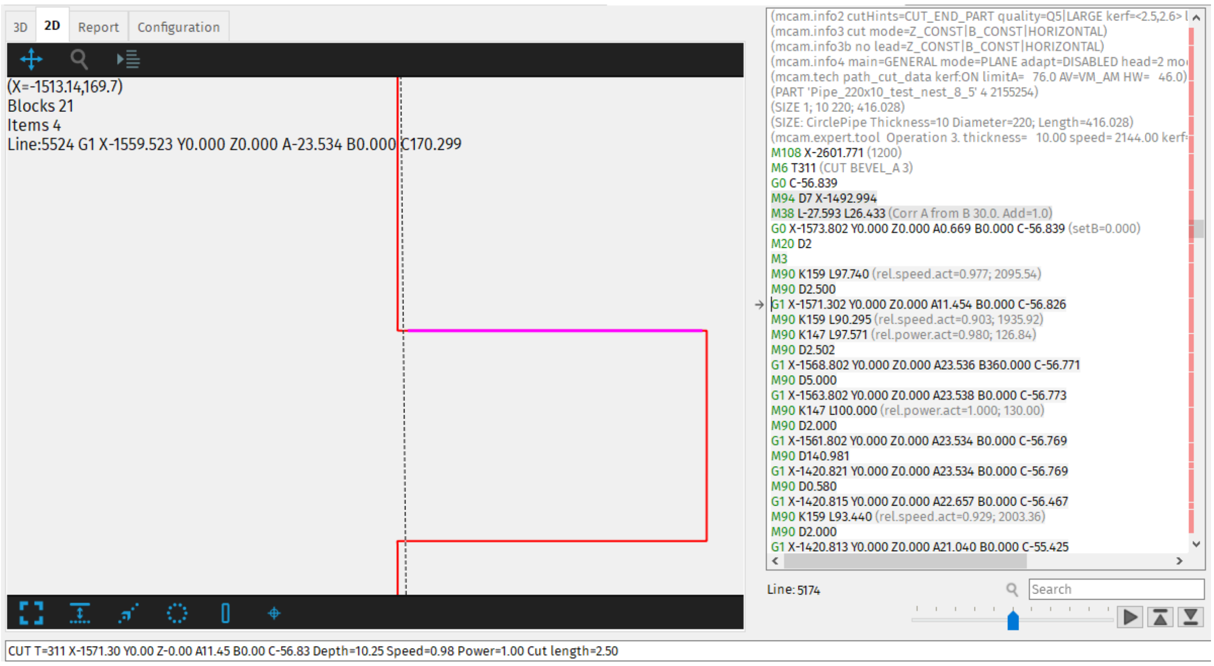


Fig. 285: Contour zoom

Tooltip shows information such as: length, angle, tool, radius, etc.

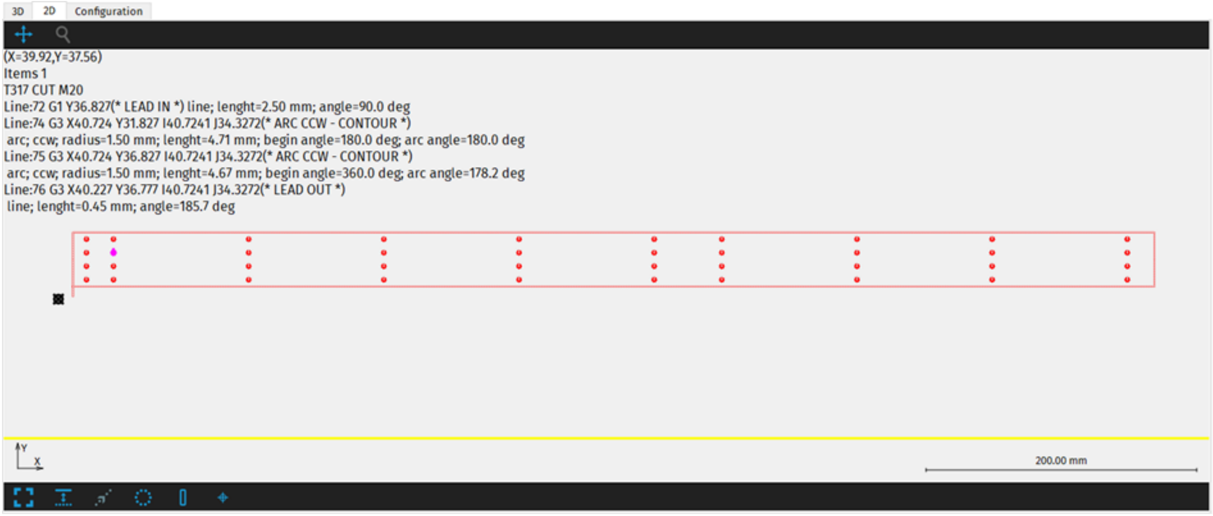


Fig. 286: Tooltip information in mCAM simulation

Some machines have different layout, because of this Simulation can rotate its view according to the iMSNC settings.

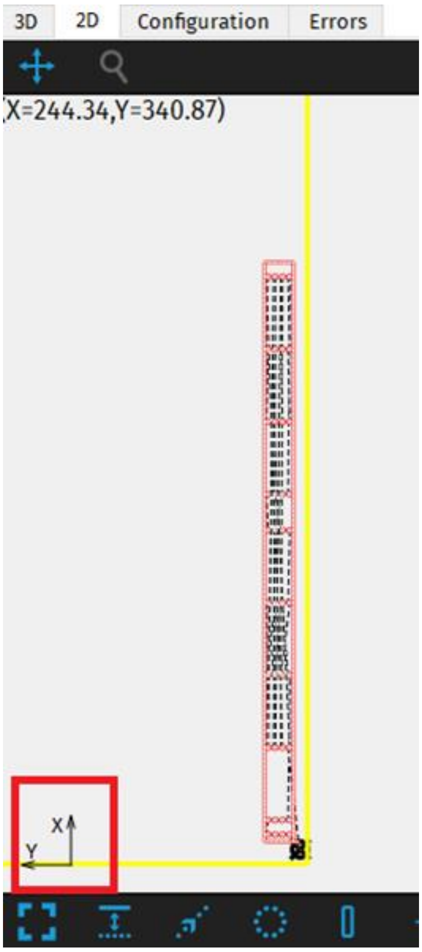


Fig. 287: Different machine layout in mCAM simulation

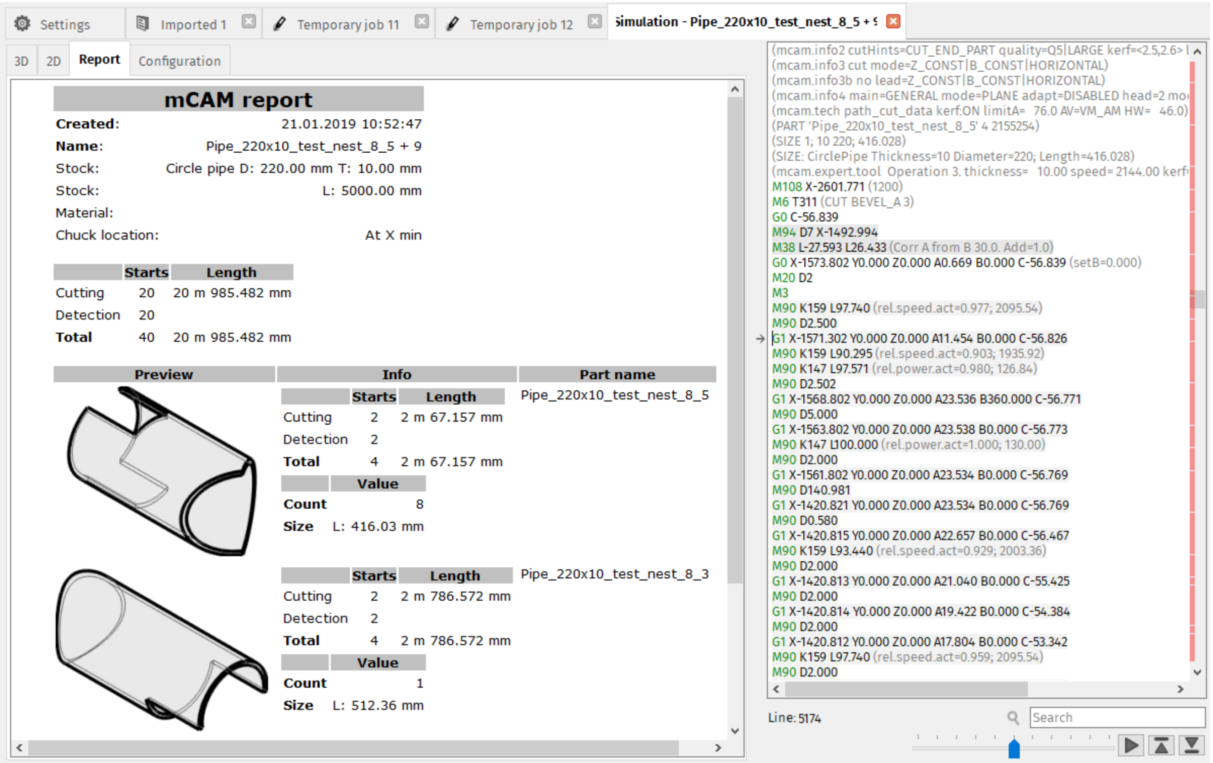


Fig. 288: HTML cutting report

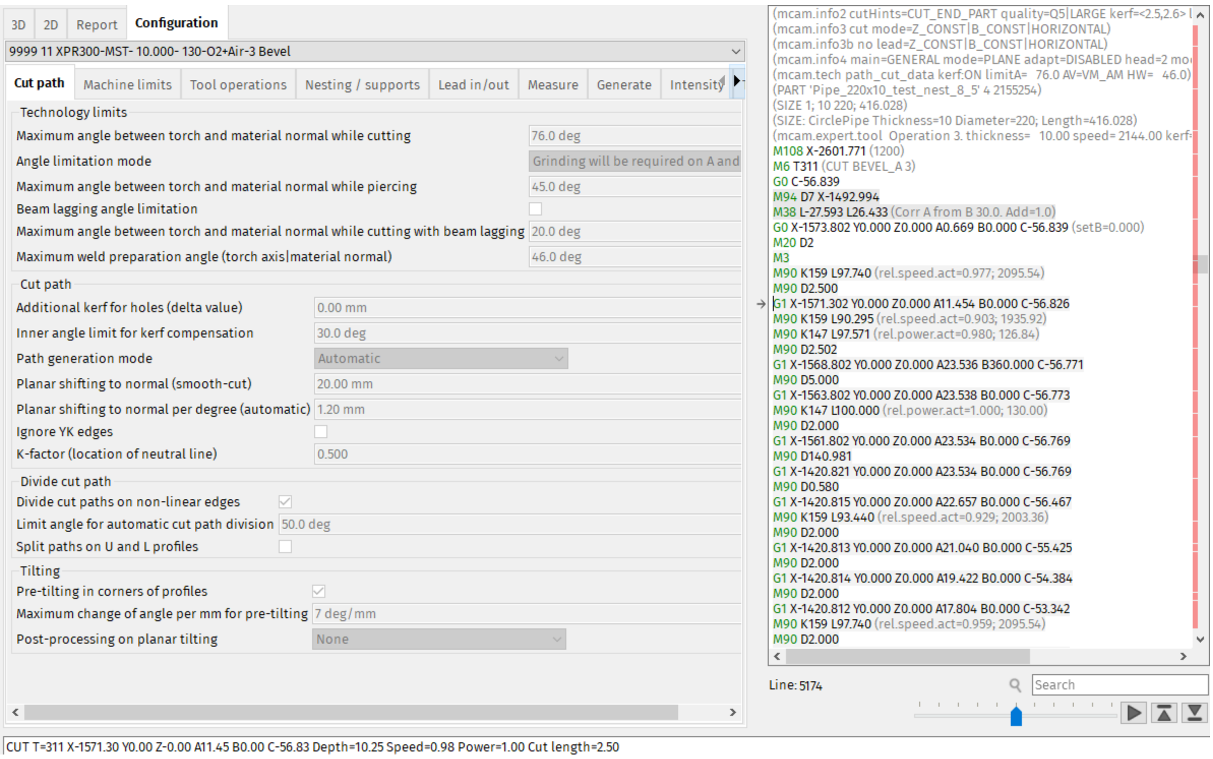


Fig. 289: Program configuration – used for CNC generation

Simulation graphs

The watch panel includes the following tabs:

- **Plot A-/B-/C-axes** – The tab displays information distilled from the generated CNC plan, namely angular coordinates of A–axis and B–axis (rotator axes) and C-axis (R-axis – rotary positioner) to reveal unexpected and unwanted defects/mistakes in NC code by excessive, sudden deviation or changes of the coordinates. The angular positions of A and B axes are expressed in degrees with respect to path tangential distance in [mm]. Limitations of A-axis, B axis and C axis (R axis) are set in *machine limits* for each *machine* and *technology* (for more detailed information see section Machine limits on page 197).
- **Plot speed / angle** – The tab provides graphical plot of percentage feed and the angle of normal to the material surface with respect to current location of the tool in terms of path tangential distance in [mm].
- **Plot thickness** – Plot of effective material thickness [mm] versus path tangential distance [mm].
- **Plot XYZ axis** – The tab displays information distilled from the generated CNC plan, namely coordinates of X–axis, Y–axis and Z–axis to reveal unexpected and unwanted defects/mistakes in NC code by excessive, sudden deviation or changes of the coordinates. The coordinates of X, Y and Z axes are expressed in millimeters.
- **Plot delta axis –** first derivation, analogy of speed according to the distance
- **Plot delta2 axis –** second derivation, analogy of acceleration according to the distance
- **Plot XYZ axes (s) -** shows changes in X, Y and Z axes in time during cutting
- **Plot A/B/C axes (s) -** shows changes in A, B and C axes in time during cutting
- **Info –** This tab displays information about stock part, type of used rotator and source of shape type.
- **Line context** – More detailed description of action performed within the current block.
- **Transformations** – Auxiliary information about geometric transformations applied in the current block.
- **Debug info** – Additional information useful for verification and debugging of the CNC program.
- **CSV –** summary of all data from simulation in one table that can be copied and exported to excel for further analysis
- **Errors**

Ranges of horizontal and vertical axes in graphical plots are adjusted automatically. The cross-hair in the plot area displays coordinates assigned to the horizontal and left vertical axis. It is also possible to zoom–in a rectangular area of the plot by selection of the area by using the left mouse button. Zooming–out is performed by <Esc> key.

Keyboard arrows controls cutting simulation manually by setting the cursor on a selected location in the NC code. Use buttons below the CNC code and Blocks panel to move through the CNC code.

	Pauses the simulation
	Moves automatically forward through simulation
	Moves through individual blocks (cuts) of simulation. Same as pressing the “-” key
	Moves through individual blocks (cuts) of simulation. Same as pressing the “+” key

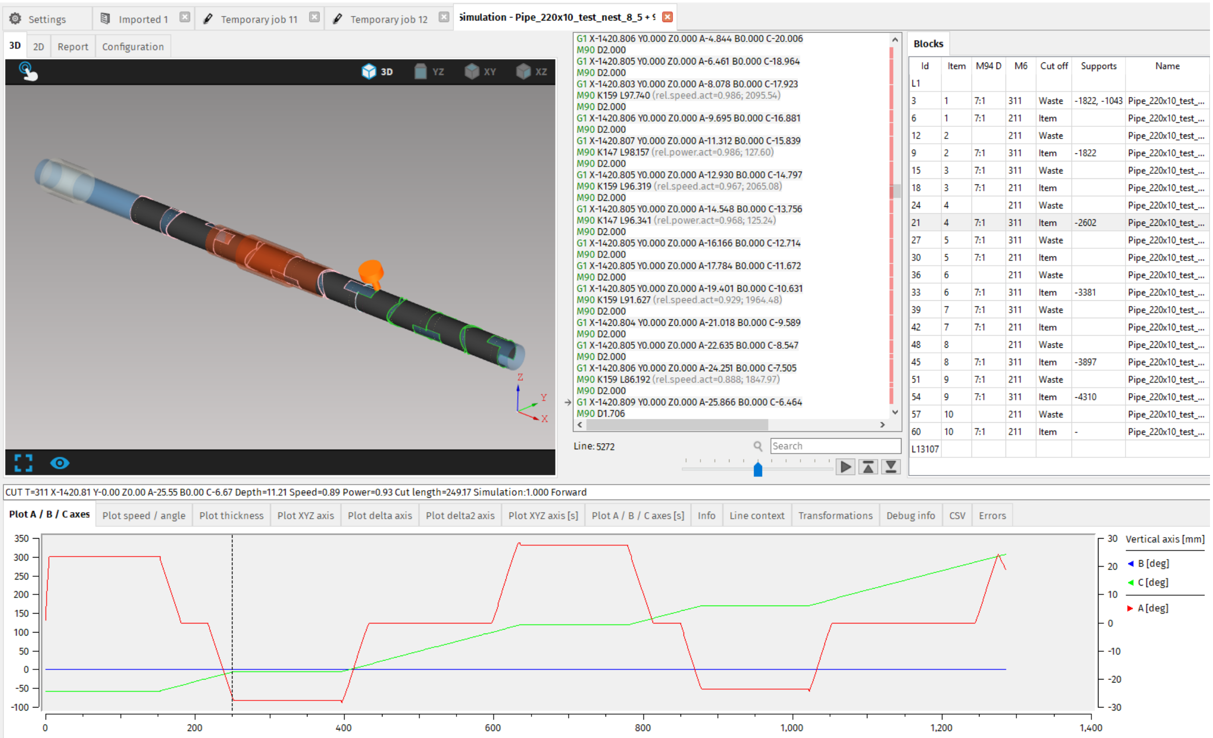


Fig. 290: Cutting simulation features and functions

Blocks panel

On the right side of the screen in Simulation is a new overview panel that displays individual blocks of cutting process. Each row in the table represents one cut from the cutting plan. For each cut, there are several columns, number of which changes in different simulations, depending on the properties of the cut and shape type. The columns depict useful summary about some CNC instructions and general information for each cut in the whole cutting plan. If the instruction is the same for every cut, this column will remain hidden.

List of possible columns in blocks panel:

- **Id –** cut ID
- **Item –** if there are more parts nested in the cutting plan, the number in this column represents to which part (in cutting order) the cut belongs to
- **Item name –** if there are different kinds of parts nested, it shows the name of the part the cut belongs to
- **M94 D –** shows the type of the instruction M94 (first number) used and number of measuring points (second number) e.g. 7:2 would mean M94 D7 was used with 2 measuring points
- **M94 K –** further specifies measuring mode and shape type that has been determined by laser measuring
- **Alignment (M94 D14) –** shows how are the holes aligned when the actual part has different dimension than CAD model
- **M117 –** shows the type of the cut – begin (first cut), hole (cut will not shorten the stock) or end (final part will fall off)
- **M20 –** shows which type of M20 instruction is used
- **M21 –** shows type of parking mode
- **M6 –** shows which tool is used for each cut
- **Cut off –** shows if the cut off is a waste or the final item
- **Supports –** displays the position of supports (instruction M108)
- **Remove (cut off part) –** shows if cut-off part needs to be removed manually (instruction M97)

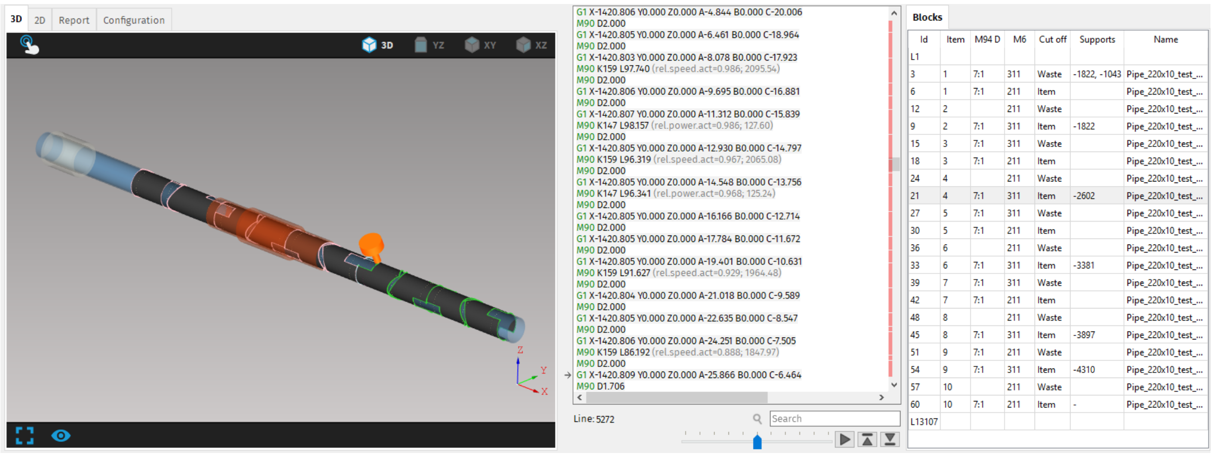


Fig. 291: Blocks panel

Settings

All settings that can be set up individually according to the machine configuration, user requirements, material and tools properties, and other production facilities can be found in: *File – Settings*.

The settings screen includes these subsections organised in tabs:

- Application
- Cut path
- Machine limits
- Tool operations
- Nesting / supports
- Lead–in/out
- Measure
- Generate
- Intensity
- Transformations
- Expert table
- Internal

Settings controls and visual cues

Following section explains controls and visual cues used for better overview of changes done by user. mCAM settings work on different levels based on selected machine, technology or tool. There are some parameters in mCAM that are not linked to a specific machine but most of them are linked to machine, technology or tool based on their properties. It is best to change them according to their respective relevant level (e.g. machine limits are relevant to change for machine or specific technology, but can't be changed for individual tools).

Each machine connected to mCAM has its own set of parameters that can be adapted according to user requirements with respect to machine/technology/tool properties.

Settings screen is divided into three sections:

1. **Technology setup** – technology setup level needs to be specified for settings that are linked to particular machine, technology or tool. Selected level affects availability of parameters that can be changed.
2. **Configuration level** – is divided into several levels:

- *Default* – read only, can't be changed. Contains default settings values set by mCAM. They are hard–coded in the source code of the program.
- *Machine* – read only. Contains settings values loaded from the machine according to parameters selected in technology setup.
- *Application* – on this level, individual parameters can be changed. Changes are applicable for whole application
- *Imports and Jobs* – individual parameters can be changed. Changes are applicable only for selected import or job.

3. **Actual settings** – is divided into 12 groups of parameters (*Application, Cut path, Machine limits, Tool operations, Nesting*

4. ______Subsection Nesting defines parameters regarding nesting of parts.

- **Nesting gap -** represents the distance between individual parts. Nesting gap ignores lead-ins during the nesting so the gap must be larger than lead ins for parts.
- **Use lead -ins/-outs and bottom contours in 2D nesting –** this parameter is used to prevent overlapping of adjacent parts caused by lead–in s/–outs or the bottom contours of these parts.
- **Limited positioner –** Defines if machine has a limitation for range of rotation (usually set from -398° to 39°)
- **Enable 1D envelope nesting -** allows to use simplified nesting process of parts that are enclosed by theoretical solid (i.e. pipes are enveloped by simple cylinders, H beam by simple cuboid, etc.)
- **Enable common cut -** is a special feature that enables to employ cutting between two adjacent edges that can be cut simultaneously by a single cut. The method is applicable only for cutting technologies that provide equal cutting quality independent on the side of the tool. Common cut can be used only on circular pipes.
- **Enable profile rotations and flipping –** if enabled, program will rotate and/or flip the material in the best position for cutting of the part.
- **Nesting time –** sets maximum time that nesting will run to prevent the algorithm running indefinitely when searching for best solution.
- **Precision of common cut detection –** specifies the value of precision of common cut.

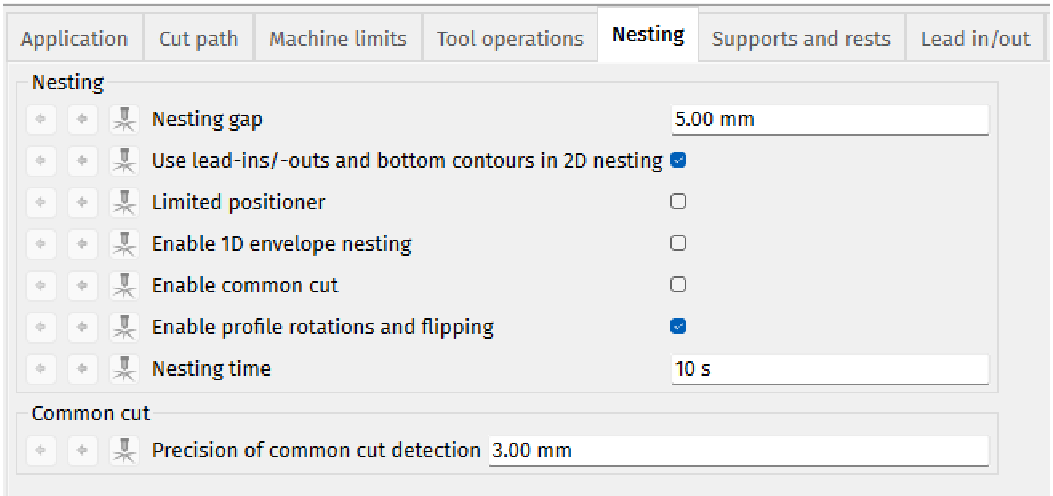


Fig. 312: Nesting settings

5. Supports, *Lead in/out, Measure, Generate, Intensity, Transformations, Expert table, Internal*) that are described in respective chapters.

To help user track the changes in settings, mCAM uses **bold font** to easily see which settings differs from value loaded from previous configuration level.

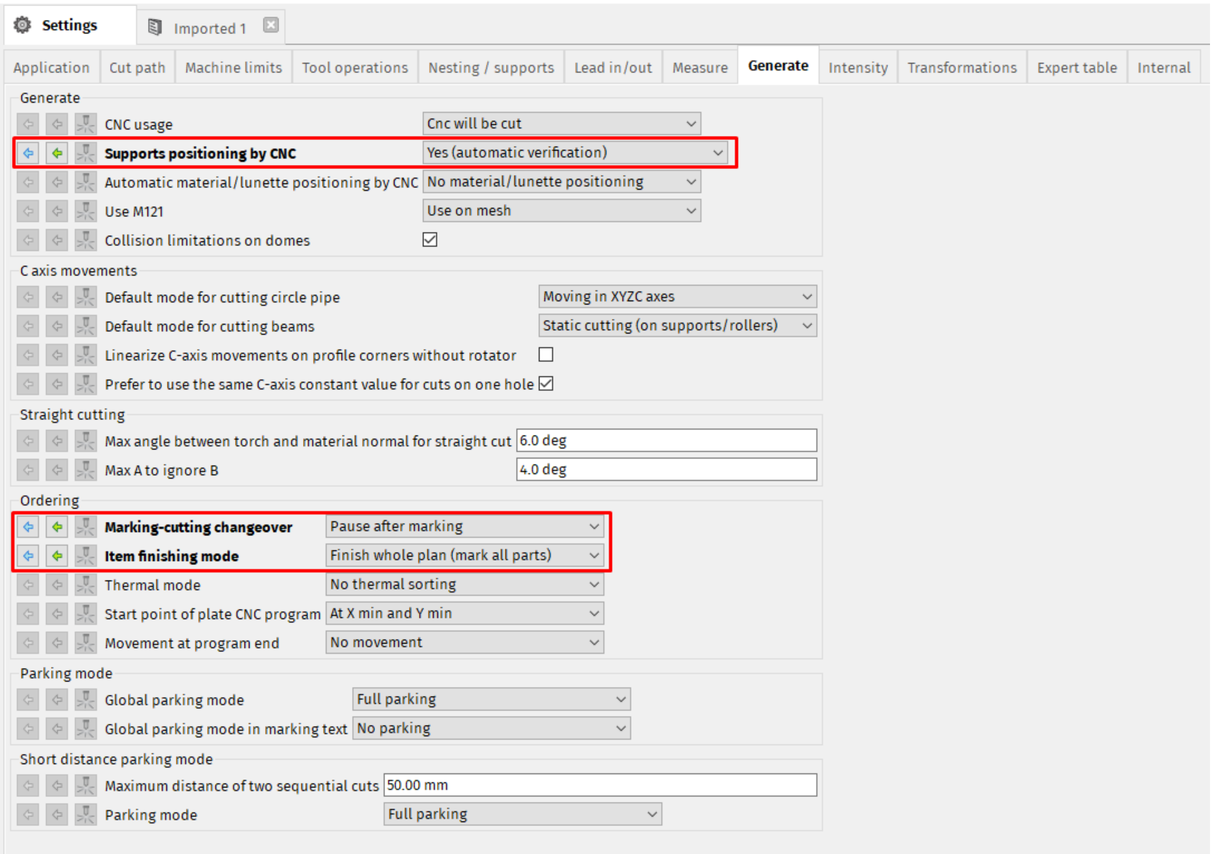


Fig. 292: Settings displayed with bold font were changed by user

Changes can be easily reverted to original value by using buttons to the left of each parameter:

Button	Description
	Use inherited value – uses the value from previous higher configuration level
	Revert all changes to start editing value – uses value that was set at the start of application
	If this button is *not* greyed out, it indicates, that on some lower technology setup level are more specific values (e.g if machine is selected, more specific values can be on technology or tool level).

Application

Subsection *Application* includes information used in user reporting such as report destination URL (sending location), user mail address, customer name.

- **All files open dialogs share one current directory** – enables to open all files, reports, CNC codes, jobs, libraries, etc. from the same directory which increases efficiency and reduces the time of cutting plan preparation. This directory path is saved after every positive opening/loading/saving of file/job/CNC code.
- **Plan save folder** – default folder for saving generated plans
- **Make a file preview** – enables a preview of listed parts in folder Directories in *Master panel* which can be inefficient when working with hundreds of parts.
- **Open simulation on CNC file**

Subsection *Application restart required* contains settings that require program restart to be applied (Language, Inverse mouse wheel zoom, etc.)

- **Web Service** – defined the connection to machine via *WebService* application. *WebService* code is defined by machine number: msnc500wxxxx, where xxxx is number of machine. Machine *WebService* connection is described in more details in section WebService or Virtual machine on page 194.
- **Keep Import opened after the restart** – the function automatically recovers opened *Import* with all imported files in case that program needs to be restart after program freeze or crash.
- **Inverse mouse wheel zoom** – is used to define behaviour (direction) of mouse wheel for zooming–in and zooming–out.
- **Enable access to *MPM* –** enables/disables connectivity to *MPM*.
- **Enable access to Machine WebService –** enables/disable connectivity to machine via *WebService* application.

- **Shared configuration folder –** defines a directory in company network for sharing all application configurations. Directory of shared folder has to be accessible and writeable (all rights/permissions need to be allowed for all users).
- **Simulation animation level –** depending on the performance level of a computer it is possible to adjust detail of simulation generation:
  - → *No animation* – colors of cut path are not changing
  - → *Cut path animation* – colors are changing for whole cut path at once
  - → *Cut point animation* – colors of cut pat in simulation are changing for each individual cut point.

Subsection *Virtual machine type* contains settings that define type of the *Virtual machine, technology number* and *number of tools. Virtual machine settings* are described in more details in section mCAM setup for virtual machine.

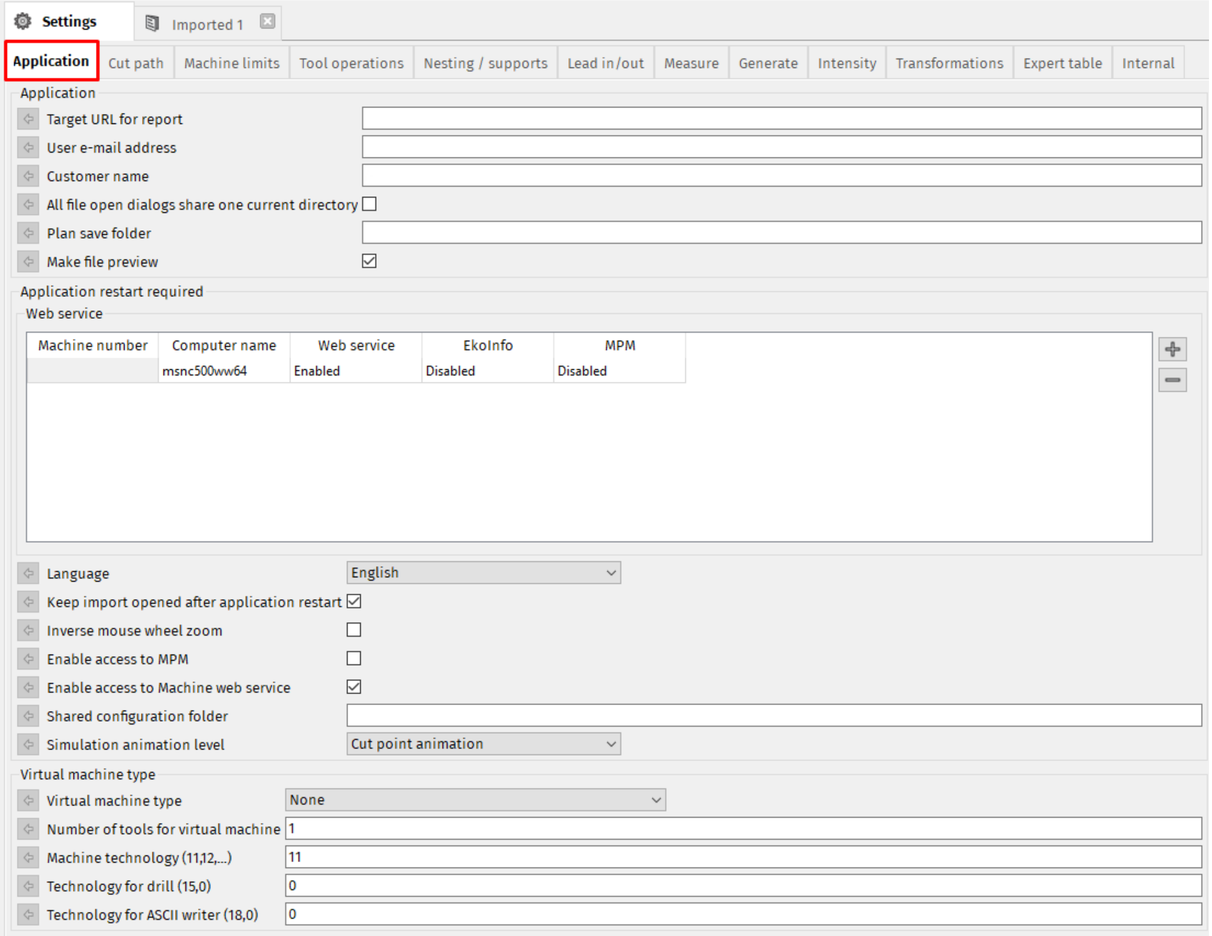


Fig. 293: Application settings

EkoInfo

EkoInfo stands for economical information and when enabled, the user can access the Ekoinfo service on machine directly from mCAM. EkoInfo is activated or deactivated for each machine in mCAM separately by manually rewriting the corresponding cell in Web Service table in Settings->Application to enabled/disabled. Alternatively, the corresponding number can be written in the cell – 0 for disabled and 1|port for enabled. Default port is 80.

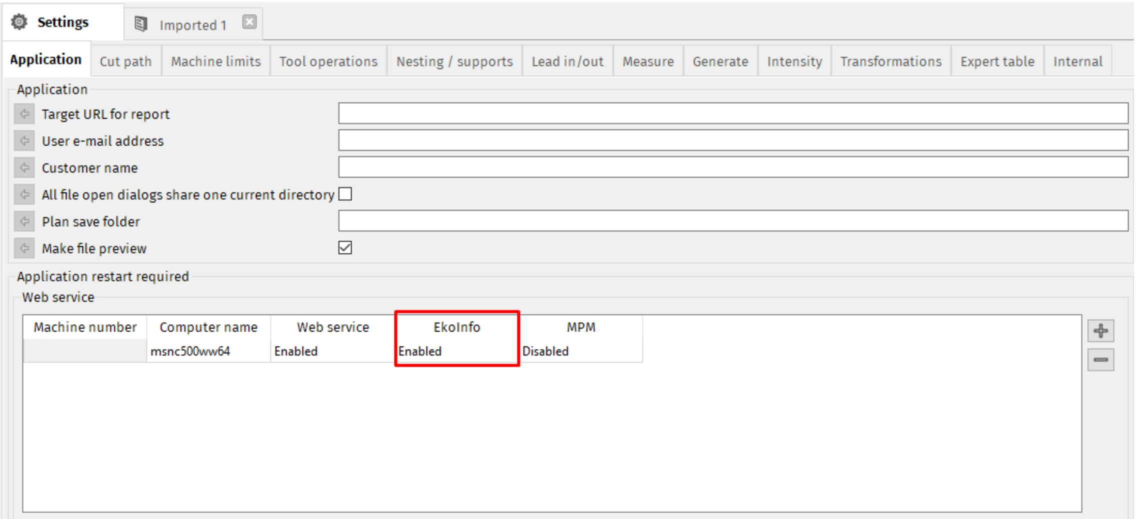


Fig. 294: Ekoinfo enabled

EkoInfo reports are accessed via Simulation menu in Simulation screen. The list contains all machines that have EkoInfo enabled in Settings. If none machine has EkoInfo enabled, the list will be empty.

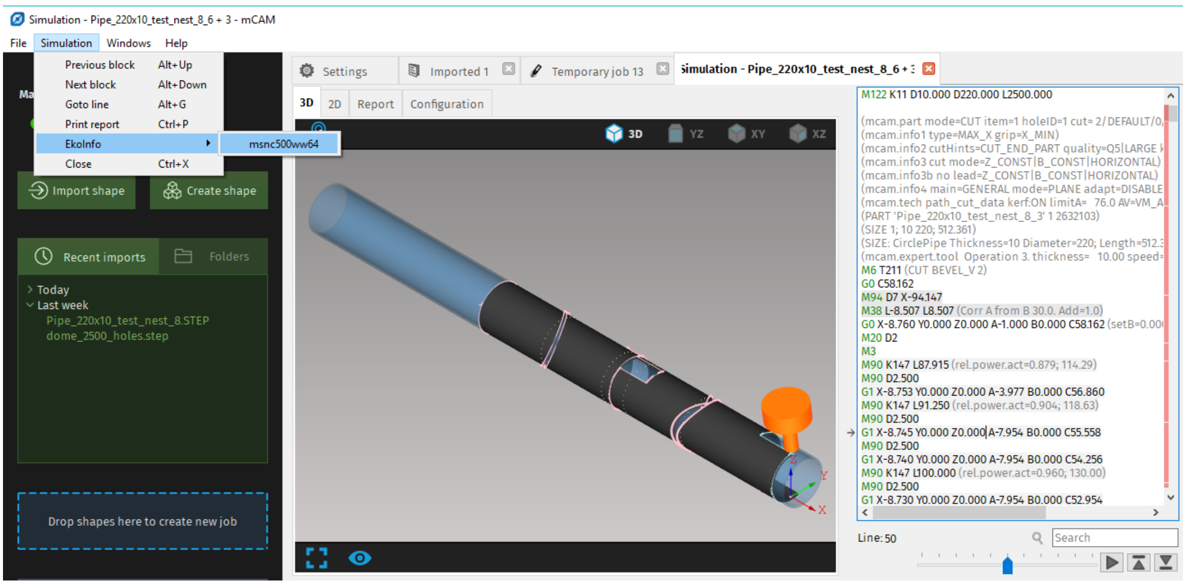


Fig. 295: List of machines with enabled EkoInfo to access EkoInfo reports

Clicking on a particular machine opens a browser and EkoInfo page with reports containing information about used technology, cutting time, prices for cutting, material, waste and other information that user can download and inspect.

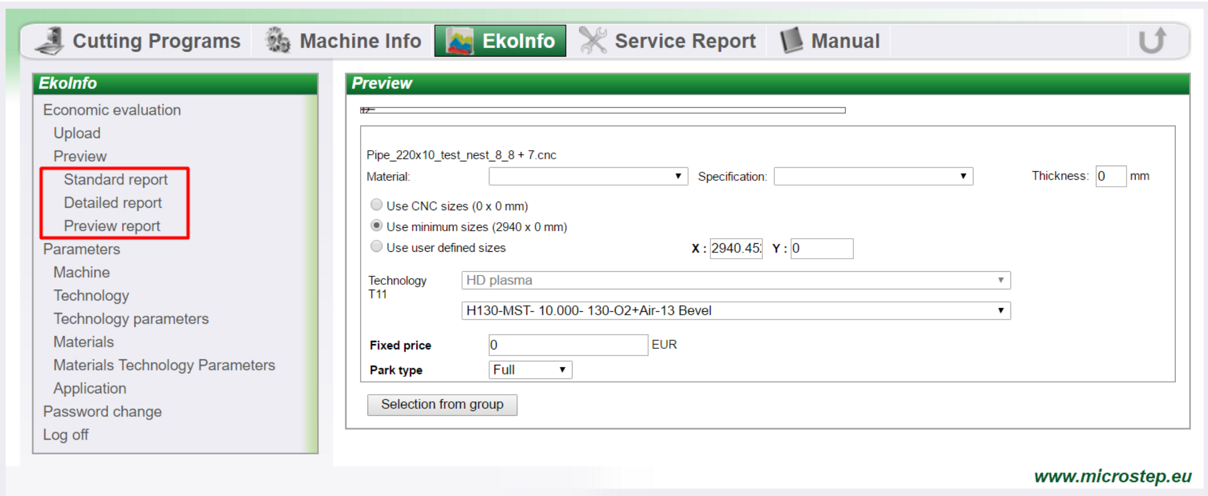


Fig. 296: EkoInfo page with reports

Cut path

Subsection *Technology limits* defines maximum angles used for cutting and marking:

- **Maximum angle between torch and material normal while cutting [°]** – upper limit of the normal angle of a material surface with respect to the vertical imposed to respect angular restrictions of bevel cutting following from applied technology (applied plasma torch tip, laser nozzle or water–jet nozzle).

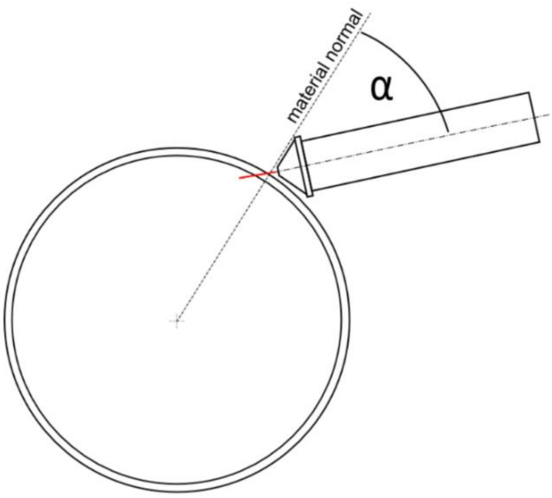


Fig. 297: Maximum angle between torch and material normal while cutting

- **Angle limitation mode** – defines the type of the angle limitation mode that is applied on paths that are limited by *mCAM* due to angle limitation of the machine and the technology (default value is set approximately to 45 degrees which is the limit value of the angle between torch axis and material normal in the process of plasma cutting.

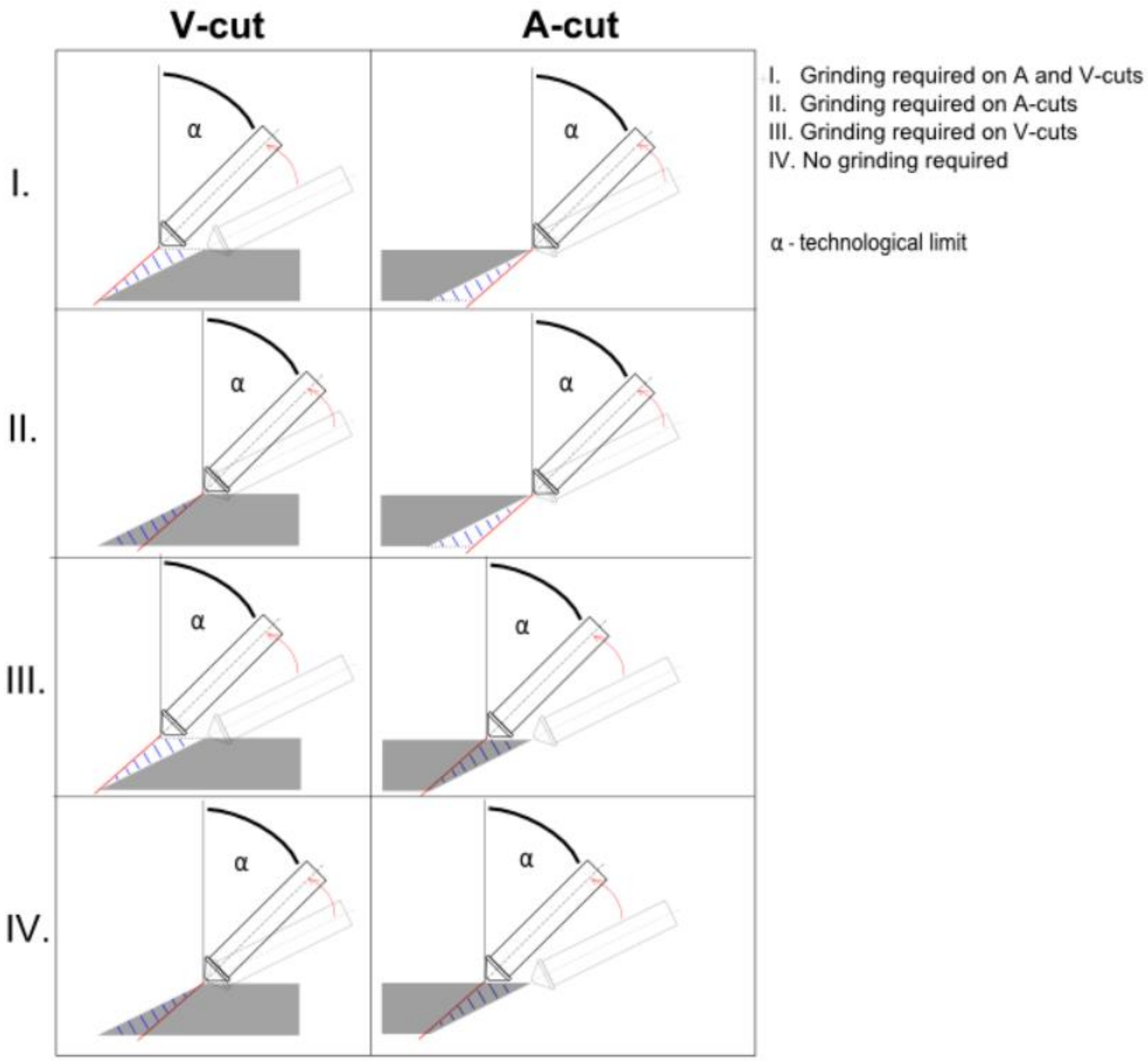


Fig. 298: Angle limitation modes

- **Maximum angle between torch and material normal while piercing [°] –** the parameter defines maximum possible angle between the tool and a material normal where it is allowed to perform piercing.

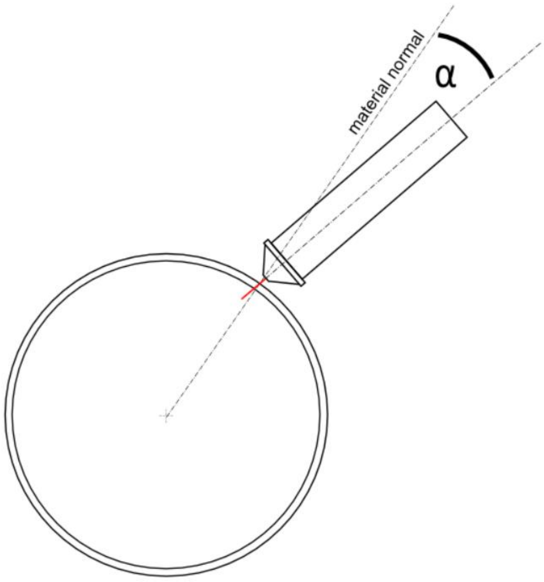


Fig. 299: Maximum piercing angle between material normal and torch

- **Beam lagging angle limitation –** this option should be checked when cutting with laser in certain direction (shown in red zone in). It activates angle limitation to ensure that the whole material is cut through

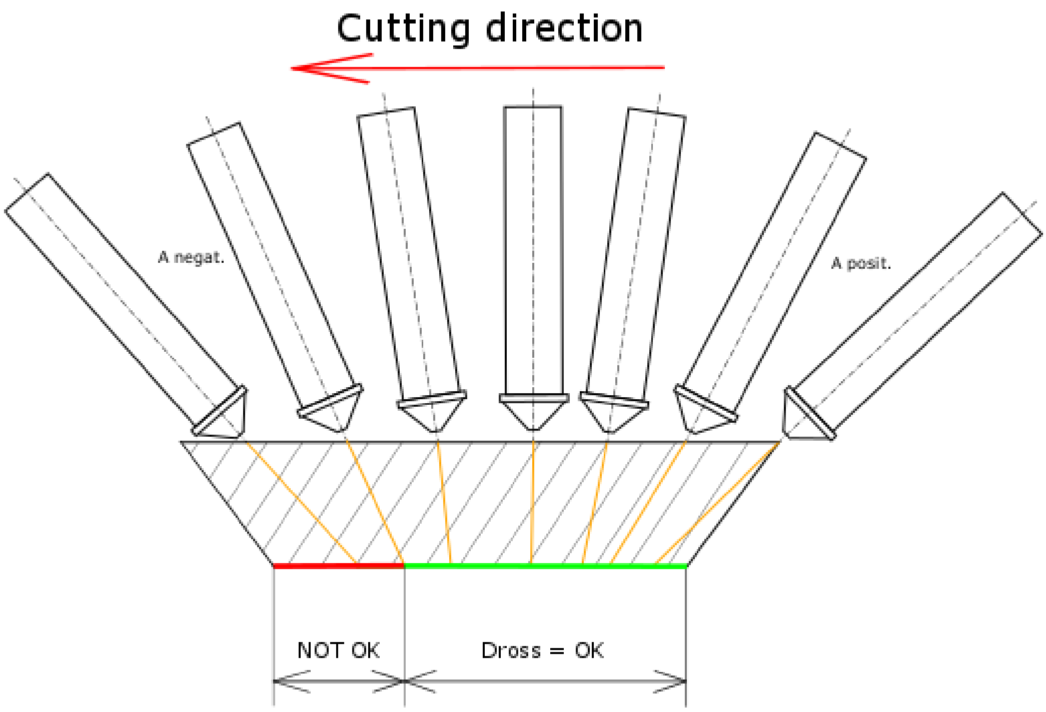


Fig. 300: Direction of cutting (red zone) requires angle limitation due to the beam lagging

- **Maximum angle between torch and material normal while cutting with beam lagging –** defines the maximum angle between torch and material normal where the limitation is not necessary.
- **Maximum weld preparation angle (torch axis/material normal)**

Subsection *Cut path* contains parameters that defines cutting path settings and properties:

- **Additional kerf for holes (delta value) [mm] –** this parameter defines total value of diametral difference of cutting beam (kerf width – defined in expert system) while cutting inner and outer contours. It is commonly used to compensate the difference between kerf width of all inner and outer contours.

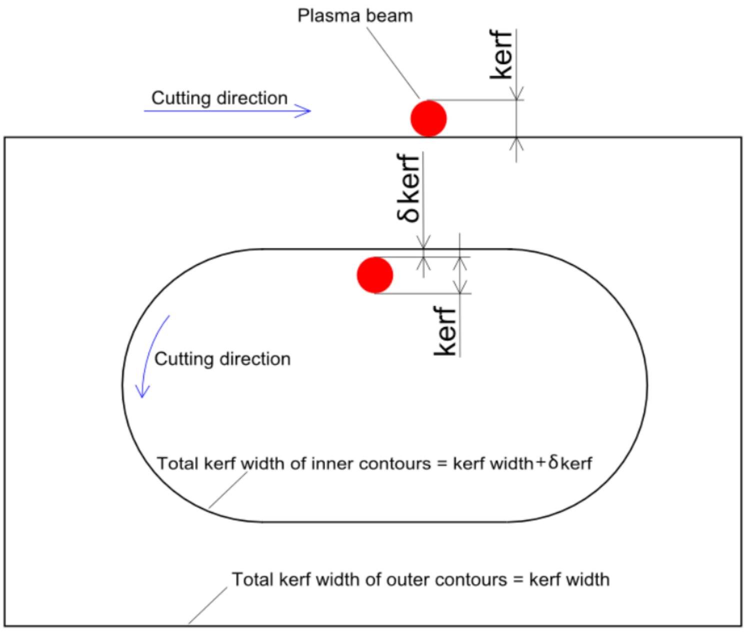


Fig. 301: Delta kerf width for inner contours

- **Inner angle limit for kerf compensation [°]** – defines the limit angle of an inner contour that ensures precise generation of kerf compensation. Limit angle can be set on much lower values (e.g. 0°) but this may cause a significant extension of generation time.
- **Path generation mode –** defines the default path generation mode that is applied on all newly imported parts. Path generation mode can be set on every cut–path individually. Characterization of all path generation modes is described in the section Path generation mode on page 144.
- **Planar shifting to normal (smooth–cut) [mm] –** defines the length of the upper cut–path contour of straight cutting edge at which the tool moves from bevel position to the position of the material normal (if possible) and continues with straightened tool. Parameter is applied only when using *Smooth–cut path generation mode.*

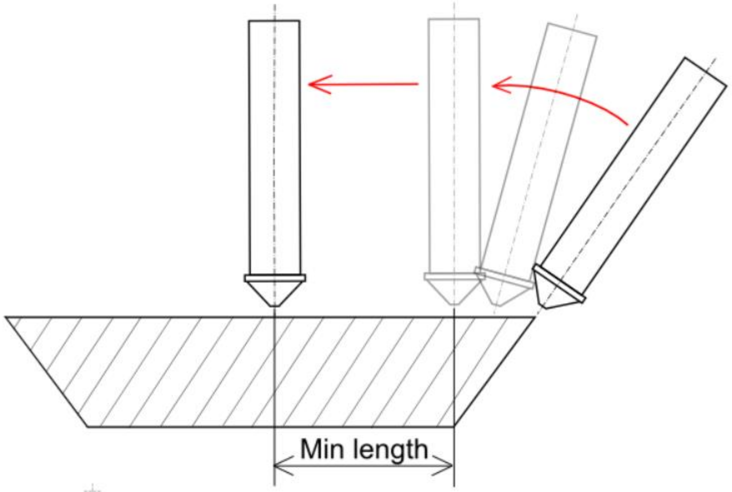


Fig. 302: Minimal length for torch shifting to material normal on planar surface

- **Planar shifting to normal per degree (automatic) [mm] –** defines the length of the upper cut–path contour at which the tool moves from bevel position to the position of the material normal (per 1 degree) and continues with straightened tool. Parameter is applied only when using *Automatic path generation mode.*
- **Ignore Y, K edges** – the option is used to ignore unwanted edges (mostly Y and K edges but also X edges) identified by *mCAM* but not intended for cutting.
- **K–factor (location of neutral line)** – K-factor is a term used for bending in metal industry and refers to the ratio of location of the neutral line to the material thickness as defined by t/T where t = location of the neutral line and T = material thickness (http://www.wikipedia.org). The neutral line is a theoretical line in the metal sheet profile that does not change its length even after the forming process or the unfolding process. The value is from the interval <0,1>, where value= 0 represents radius of the neutral line on the inner surface and value= 1 represents the radius at the outer surface. This parameter is important in cases, where the work–pieces (round pipes with large dimensions, e.g. with radius + 1 m) need to be unwrapped and cut as the sheet work–piece. The value of the parameter can be find from empiric experience and in general it is specific for each material and thickness. It includes the impact of factors affecting the location of neutral line.

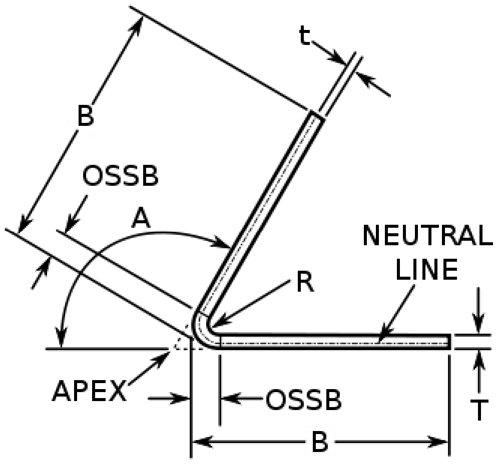


Fig. 303: Definition of K-factor for bending (http://www.wikipedia.org)

Subsection *Divide cut path* contains parameters that defines presets of cut path dividing and used cutting mode on U–beams:

- **Divide cut paths on non–linear edges** – enables/disables automatic division of cutting paths on non–linear edges according to set limit angle.
- **Limit angle for automatic cut path division [°] –** defines the limit angle of outer contour (outer edge) for automatic cut path division.

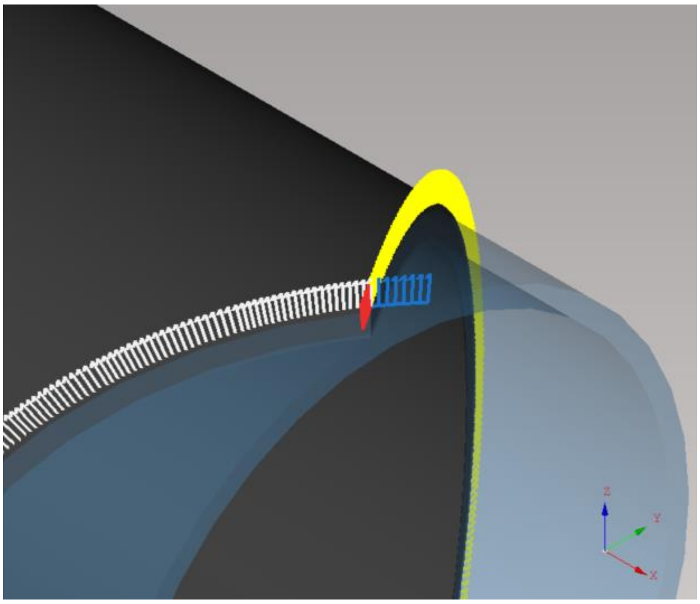


Fig. 304: Split cutting path

- **Split paths on U–profile –** enables/disables automatic advanced cutting path splitting on U–profiles to provide better detailed quality on complicated features near the beam edge (profile radius). This function allows to avoid overburnt of the material by the plasma beam.

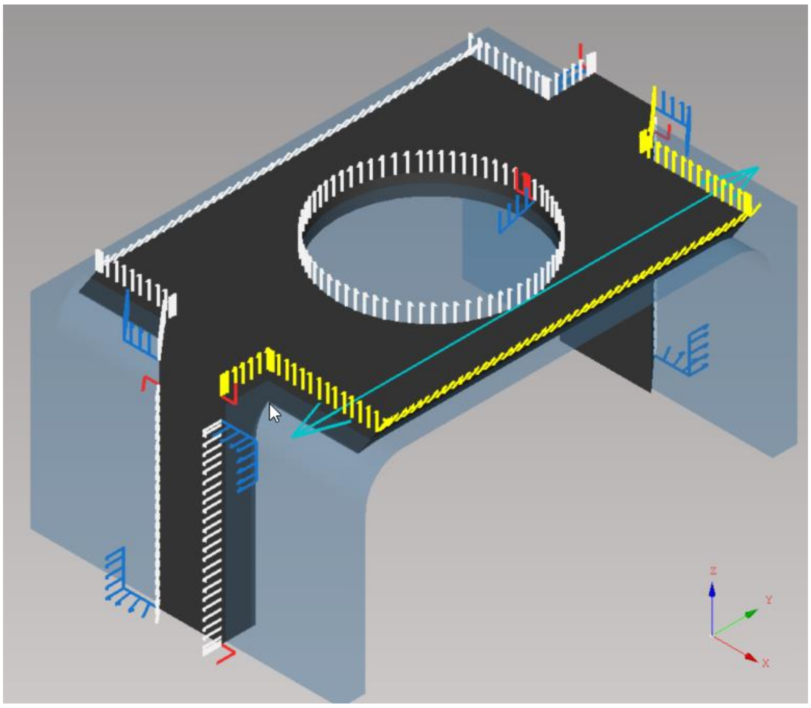


Fig. 305: Split paths on U–beam

Subsection *Tilting* contains parameters that define pre-tilting speed limit in advanced type of pre-tilting in corners of profiles (rectangular shaped pipes):

- **Pre-tilting in corners of profiles –** enables/disables pre-tilting in corners of profiles.
- **Maximal change of angle per mm for pre-tilting –** defines the maximal angle speed limit applied by pre–tilting. This function allows to avoid overburning the material by plasma or laser beam in profile corners by linear pre–tilting of cutting contour.
- **Post-processing on planar tilting**
  - → **None**
  - → **Linear**
  - → **Cubic**

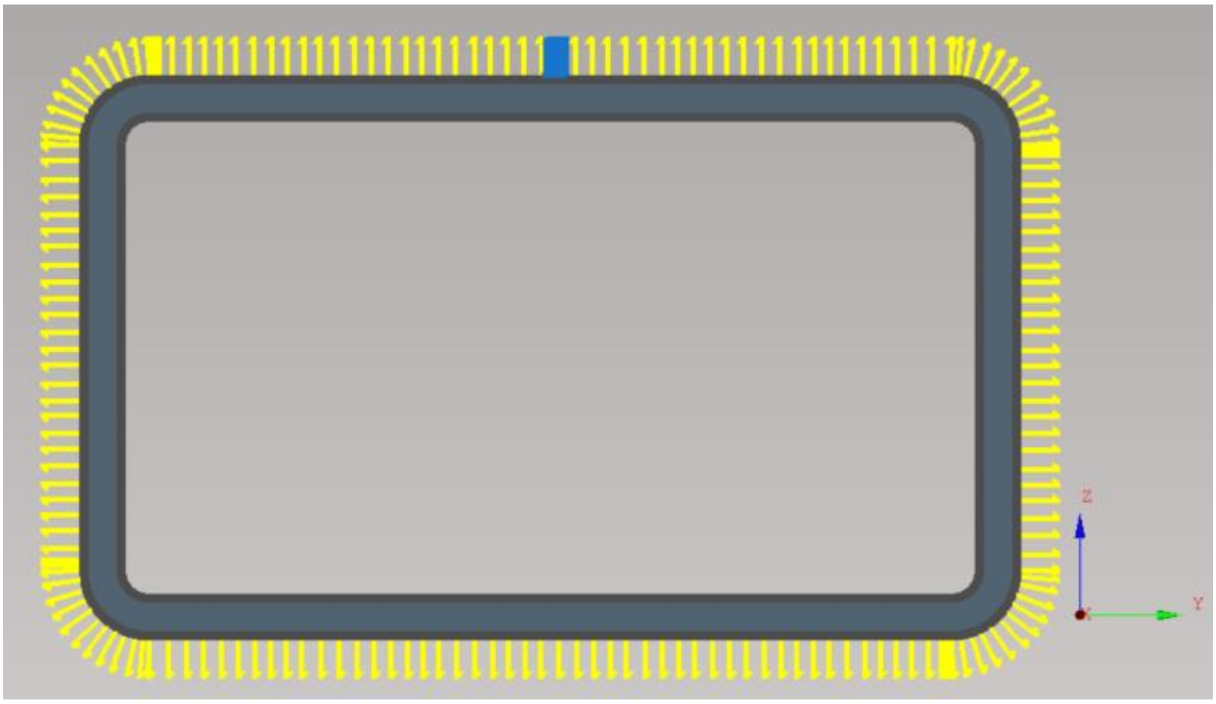


Fig. 306: Cutting path – without pre–tilting in corners

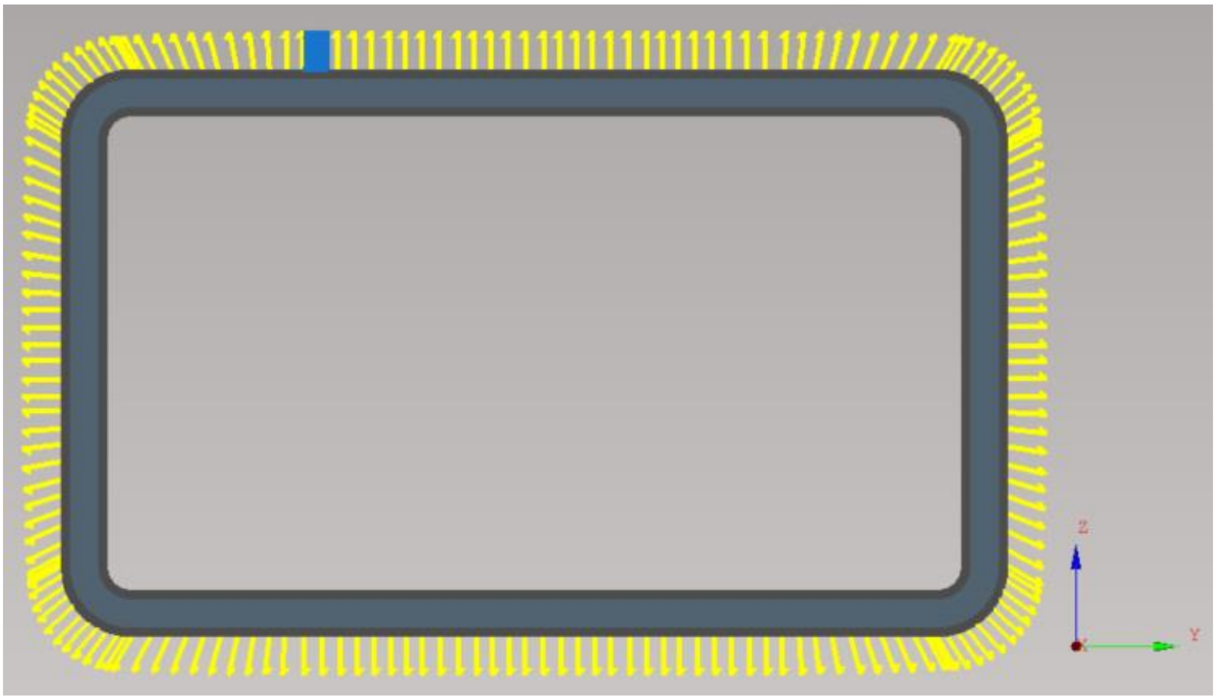


Fig. 307: Cutting path – with pre–tilting in corners

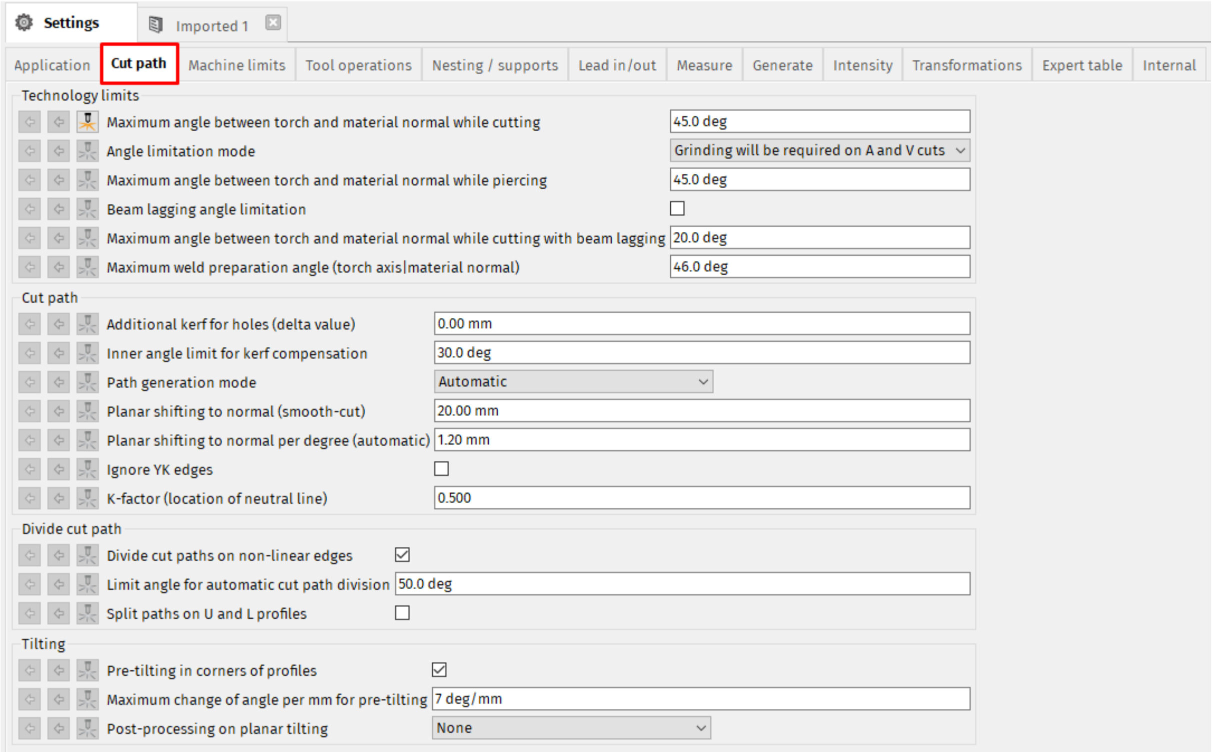


Fig. 308: Cut path settings

Machine limits

This tab contains parameters that impose limits of motion on operating range of the machine motion axes. Machine motion limits are intended to prevent collisions between a tool and any part of the machine and define the operating range of the machine, or to impose practical limitations following from the machine control system.

Subsection *Machine limits* contains these parameters:

- **Minimum Z [mm]** – motion limits of Z-axis are usually defined in the servo parameters. The limits define overall range of the axis travel. In case of bevel cutting, depending on the machine kinematics, movement of Z-axis can also be used. The minimum limit can be used to override the setting.
- **Minimum A for cutting [°]** (tilting axis) – defines the minimum value of A-axis bevel angle for cutting.
- **Maximum A for cutting [°]** (tilting axis) – defines the maximum value of A-axis bevel angle for cutting.
- **Minimum A for marking [°]** (tilting axis) – defines the minimum value of A-axis bevel angle for marking.
- **Maximum A for marking [°]** (tilting axis) – defines the maximum value of A-axis bevel angle for marking.

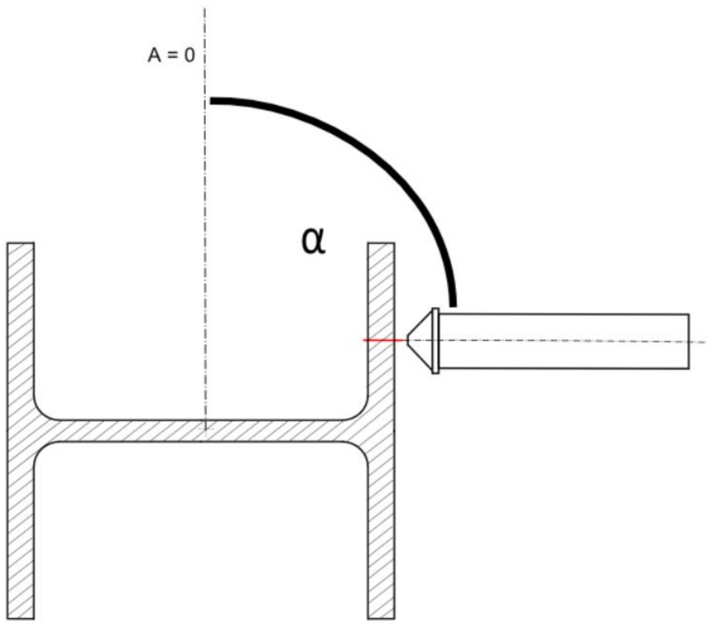


Fig. 309: Maximum A axis angle for cutting and marking

- **Minimum B [°]** (azimuth axis),
- **Maximum B [°]** (azimuth axis),

- **Minimum R [°]** (rotary positioner),
- **Maximum R [°]** (rotary positioner).
- **B axis speed limit** – the parameter defines the limitation of B-axis speed in degrees per 1 millimeter (°.mm^{-1}) that is used for backward rotation of rotator B-axis. Backward rotation of B axis is used to reduce cumulative rotation of B axis at low degree of A-axis (this limitation value of A-axis is defined in parameter Max A to ignore B. This function is useful especially in cutting paths where cumulative values of B-axis could reach the limit values (which is defined approximately on 400°– depends on type of the CNC machine).

Subsection *Machine layout* defines the parameter:

- **Chuck location (X-min/ X-max)** – defines position and orientation of clamping device for the work–piece with respect to the machine coordinate system.
- **Rest type –** specifies the machine type in order to define whether machine has rest or it uses standard rotary positioner.
- **Rotator geometry –** defines the geometry of particular rotator that is applied in limitation processing and generation of *CNC program.*

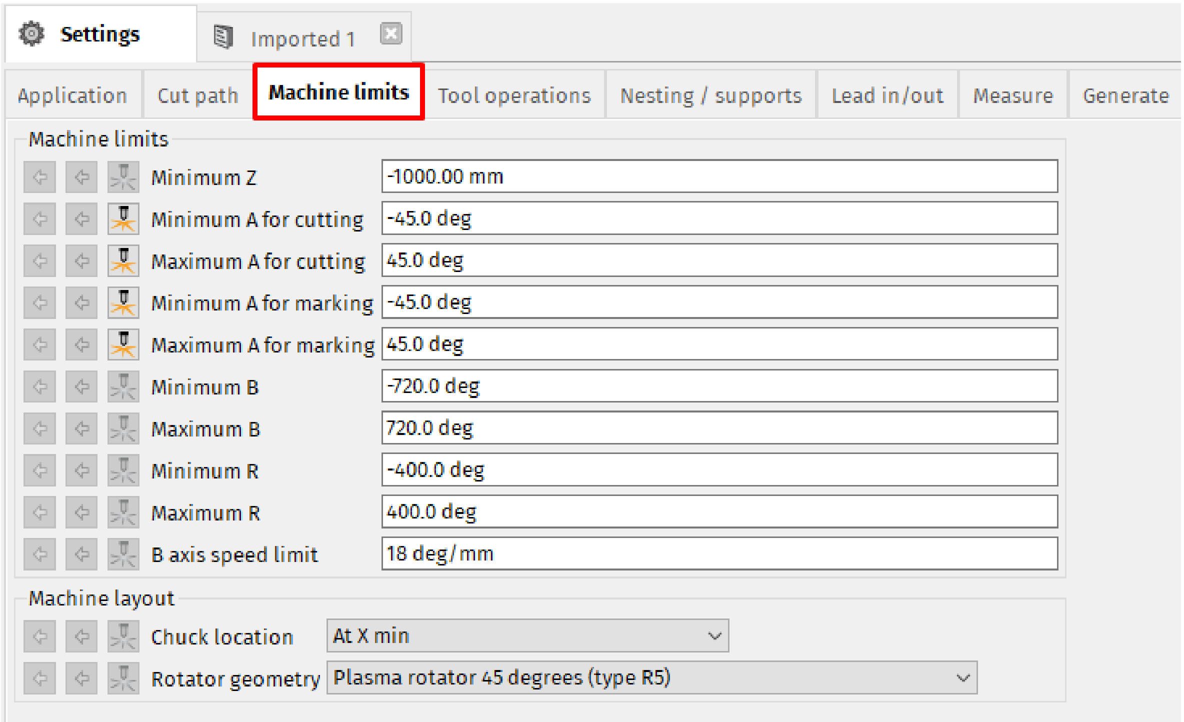


Fig. 310: Machine limits

Tools operations

This subsection contains definitions of tool operation codes used for specific operation (*straight cutting, bevel cutting, marking, etc.* or quality-oriented cutting as *Large contours, Medium contours, Q1, Q2,* etc.). One row in *tool operation type* table represents one column in *iMSNC table of cutting parameters*.

Each tool number consists of two numbers. First number (Tx11) of each tool code represents the operation (row of cutting parameters in iMSNC), known also as group of technological parameters. Second number of each tool (T1xx) code represents technology number (plasma – 11, laser – 17, waterjet – 14, oxy–fuel – 12, etc.). Tool operations are described in more details in section Tool operations)

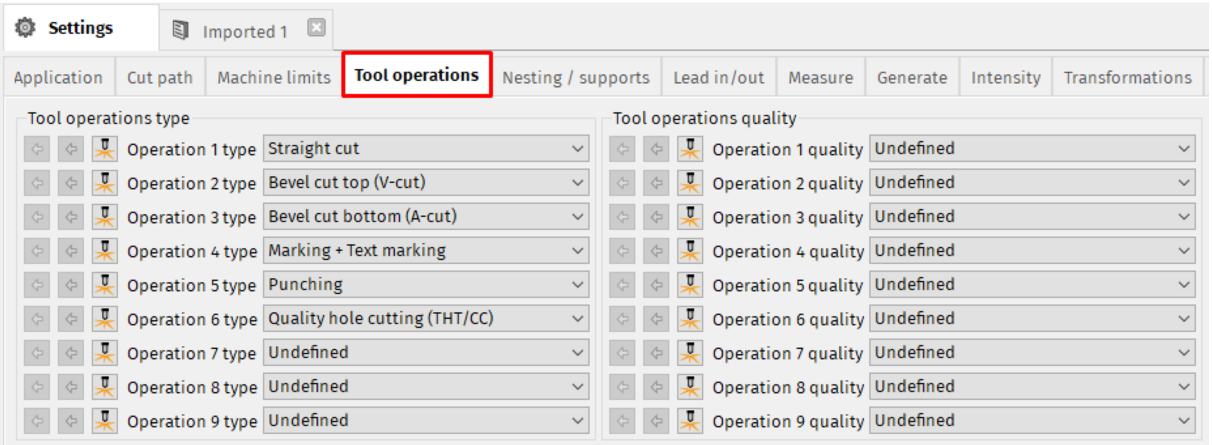


Fig. 311: Tool operation settings – tools

Nesting

Subsection Nesting defines parameters regarding nesting of parts.

- **Nesting gap -** represents the distance between individual parts. Nesting gap ignores lead-ins during the nesting so the gap must be larger than lead ins for parts.
- **Use lead -ins/-outs and bottom contours in 2D nesting –** this parameter is used to prevent overlapping of adjacent parts caused by lead–in s/–outs or the bottom contours of these parts.
- **Limited positioner –** Defines if machine has a limitation for range of rotation (usually set from -398° to 39°)
- **Enable 1D envelope nesting -** allows to use simplified nesting process of parts that are enclosed by theoretical solid (i.e. pipes are enveloped by simple cylinders, H beam by simple cuboid, etc.)
- **Enable common cut -** is a special feature that enables to employ cutting between two adjacent edges that can be cut simultaneously by a single cut. The method is

applicable only for cutting technologies that provide equal cutting quality independent on the side of the tool. Common cut can be used only on circular pipes.

- **Enable profile rotations and flipping –** if enabled, program will rotate and/or flip the material in the best position for cutting of the part.
- **Nesting time –** sets maximum time that nesting will run to prevent the algorithm running indefinitely when searching for best solution.
- **Precision of common cut detection –** specifies the value of precision of common cut.

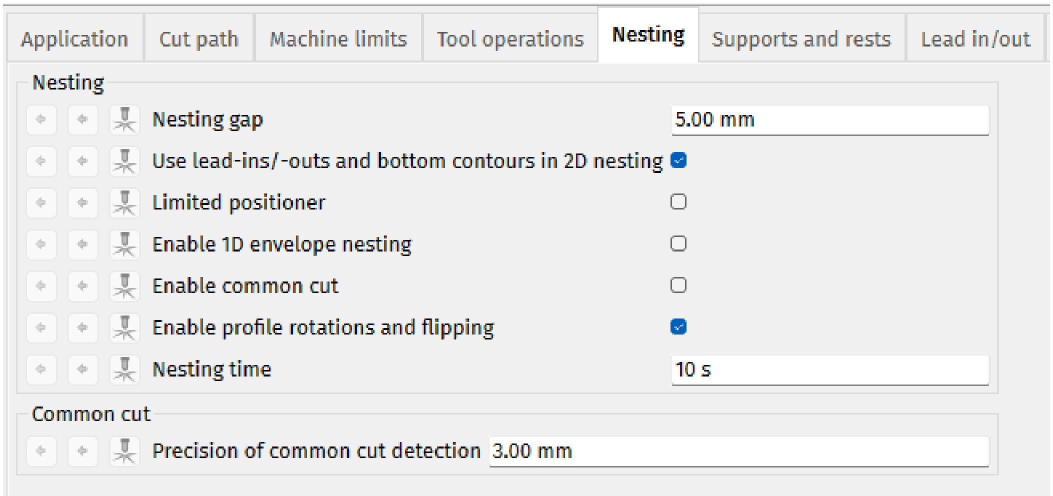


Fig. 312: Nesting settings

Supports and rests

Subsection Supports and rests defines parameters of clamping system and system of supports. It was optimized in mCAM 1.5 to make the utilization of supports more effective. Supports are used on long profiles to prevent bending of the material, to maintain precision of cutting and to ensure that the cut-off parts that can't drop freely to the ground will be removed safely. The minimal length of profiles when supports have to be used varies depending on the characteristics of the material (dimensions, mechanical properties). Generation of supports is activated in *Settings->Generate->Supports positioning by CNC*.

Described below are various settings available in mCAM that specify the usage of supports. They are located in Settings –> Nesting/Supports.

- **Support placement –** to increase the stability it is possible to support hanging parts of the stock material.

  → *Support only cut area* – only the area around cut is supported

  → *Support cut area and outer part* – this option is for machines that have special supports (props) that supports inner part of the semiproduct

→ *Support cut area, outer and inner part* – if only two supports are available and one is used to support cut area, the other one supports the free end of the pipe (outer part) preferably

- **Support cut-off part –** tells whether cut-off part should be supported with two supports or not at all.
- **Chuck length [mm] –** defines the length of semi–product that is clamped in the chuck.

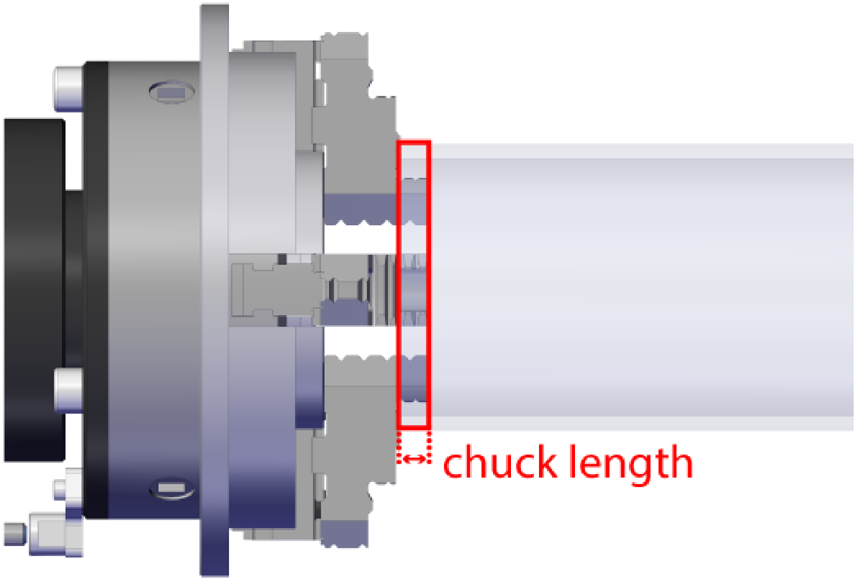


Fig. 313: Chuck length

- **Number of all supports –** defines the number of all supports used for generated *CNC programs*. (automatic and non-motorized). Since there are different kinds of supports for pipes and for profiles and it is possible to have different quantity of supports for pipes as for beams, it is necessary to adjust the number each time based on a shape that is being cut.
- **Number of motorized supports –** how many supports from all supports are motorized. Motorized supports have its own motor and reposition themselves automatically.
- **Support rails min –** minimal length of support rails
- **Support rails max –** maximal length of support rails
- **Support cylinder width –** length of support cylinder
- **Support blocked area for cutting X+ [mm] –** distance from a cut in the direction of X+ where cutting is not possible due to the danger of collision between support and cutting head.

- **Support blocked area for cutting X- [mm] –** distance from a cut in the direction of X- where cutting is not possible due to the danger of collision between support and cutting head.

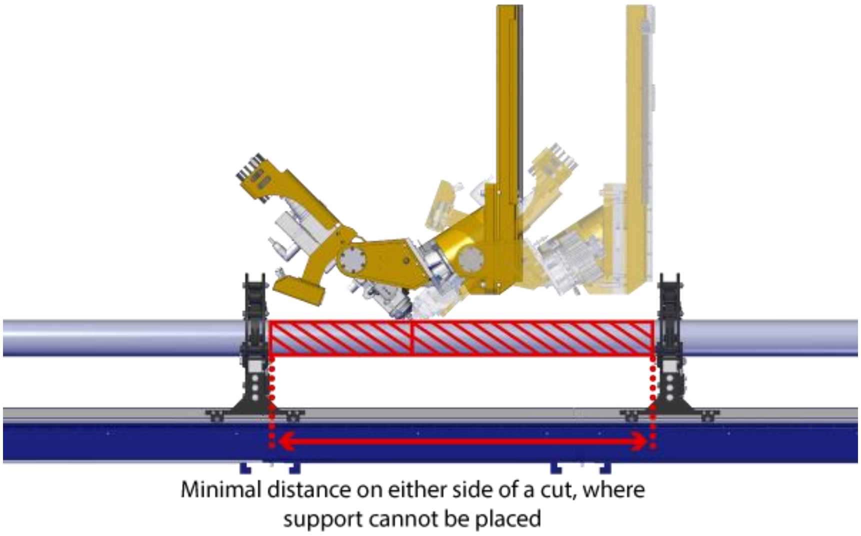


Fig. 314: Blocked areas for cutting X+ and X-

- **Has deflector –** tells if machine has deflector
- **Length of gripped part**
- **Length of support base [mm] –** defines the length of the base of each support (length of support placed on rails). This value is important to prevent collision between two supports.
- **Offset of support base**
- **Length of support up [mm] –** defines length of the support
- **Offset of support up**

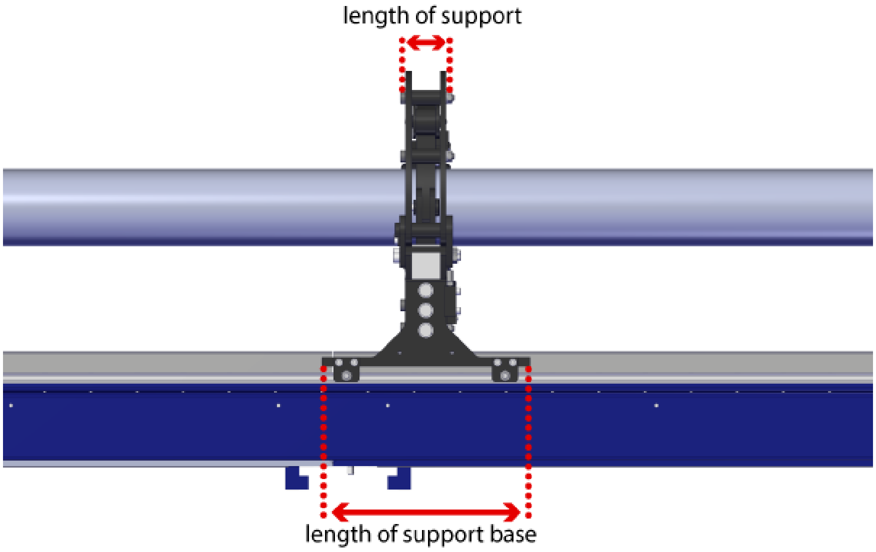


Fig. 315: Length of support and support base

- **Maximum support–support distance [mm] –** defines maximum distance between two consecutive supports. This value should respect mechanical properties of manufactured materials (compressive strength and elasticity).
- **Maximum support-cut distance –** defines maximum distance between cut and support to ensure the best precision of cutting. This value should respect mechanical properties of manufactured materials (compressive strength and elasticity) to prevent bending of the material.

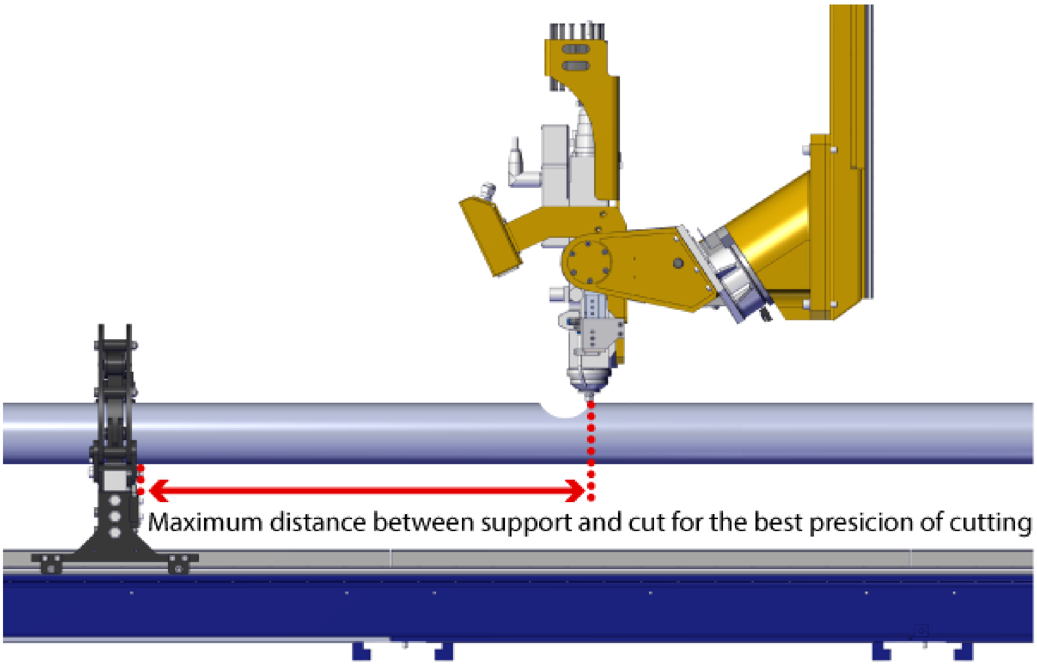


Fig. 316: Maximum support-cut distance

- **Min length of the cut-off part to be supported with two supports –** defines minimum length of cut-off section that has to be steadily supported with two supports so it does not fall-off
- **Min length of the cut-off part that must be manually removed –** defines the shortest length of a part that should not drop freely to the ground. Any cut-off part that is longer that this value will be supported with support/s so it can be safely removed.
- **Support-End of profile distance [mm] –** defines maximum distance between end of the profile and closest support while cutting.
- **Sort cuts in groups for compact support positioning –** enables/disables sorting of all cuts to groups in order to reduce number of repositioning the supports while cutting.

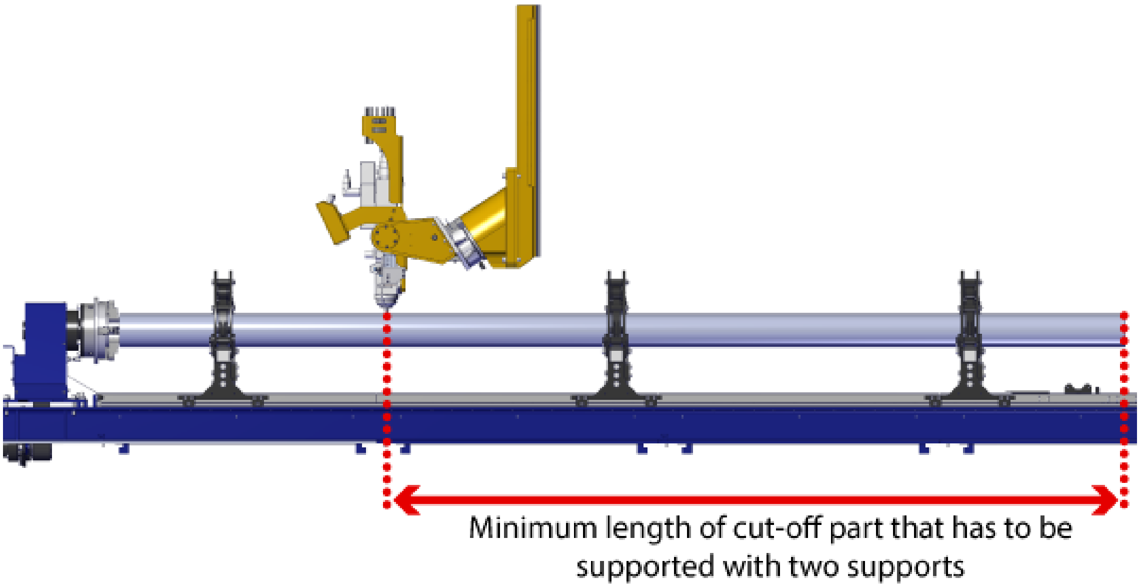


Fig. 317: Min length of the cut-off part to be supported with two supports

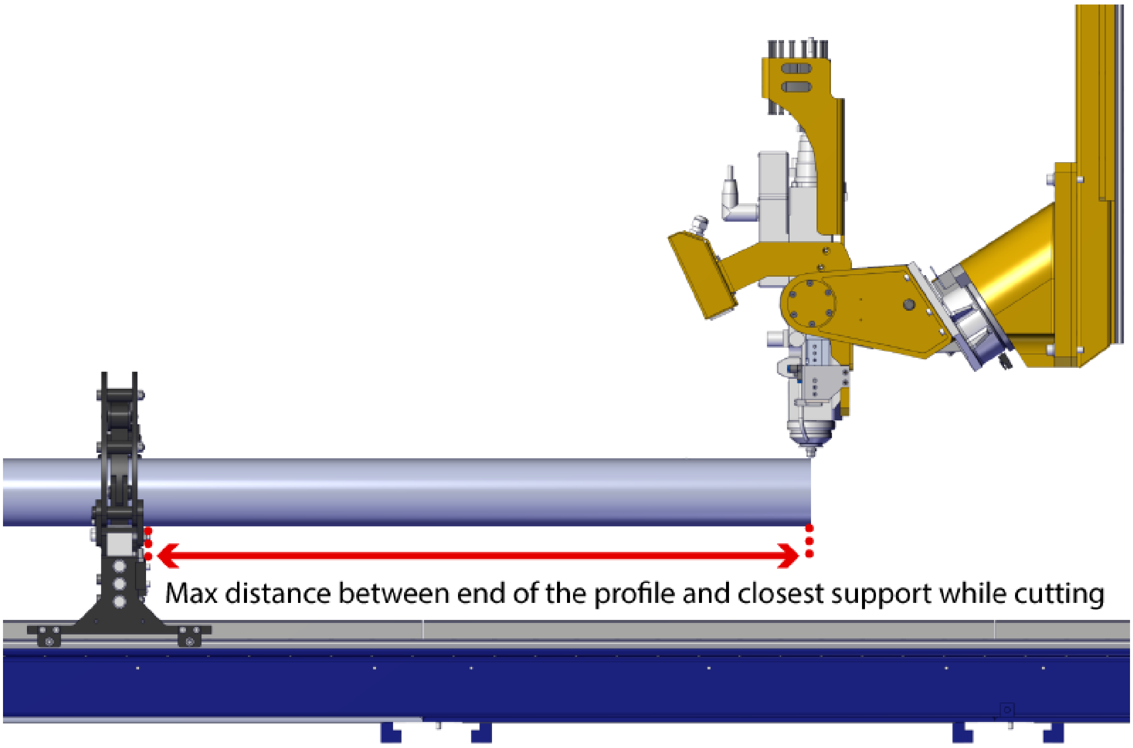


Fig. 318: Support-End of profile distance

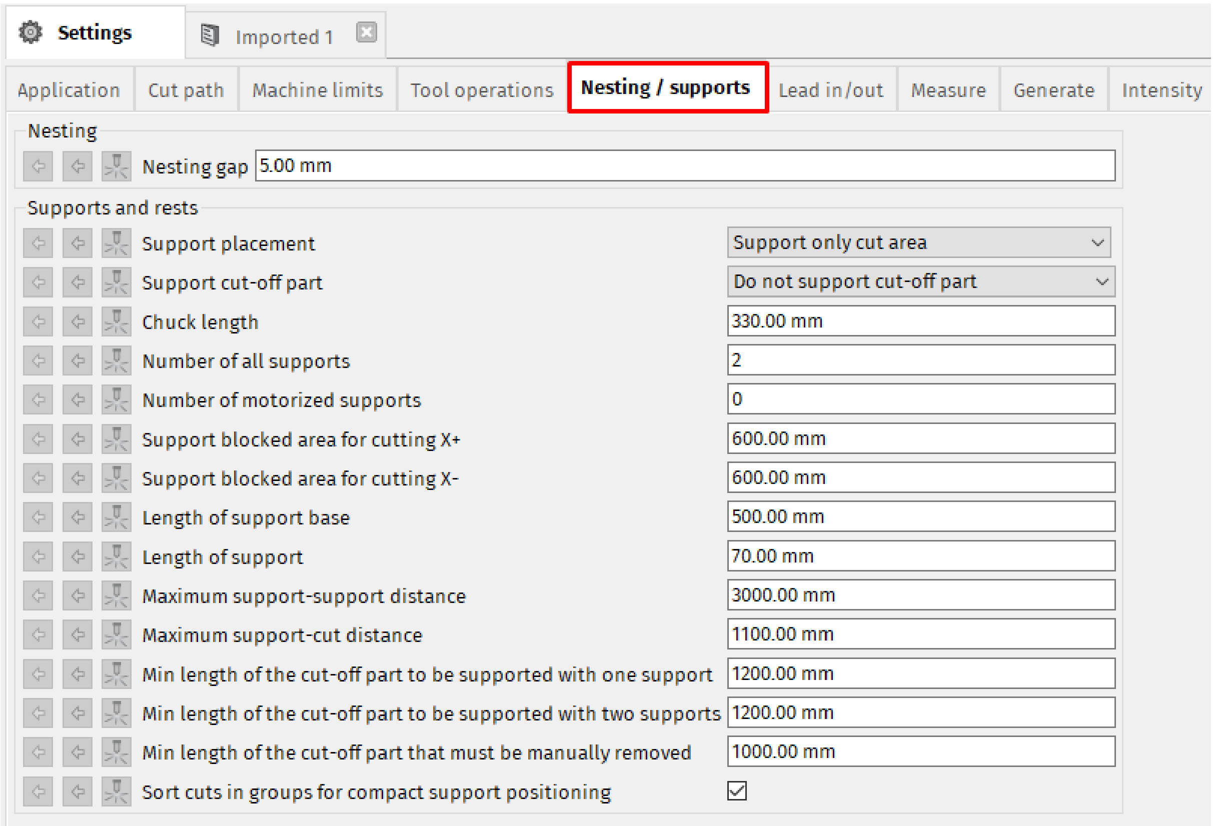


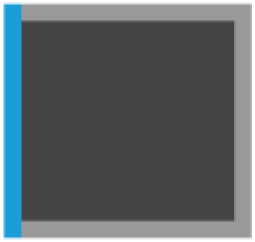
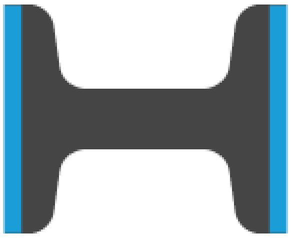
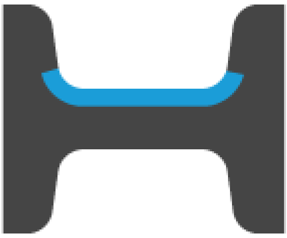
Fig. 319: Nesting/supports settings

Lead in/out

Lead-in/out tab contains definition of default parameters of lead ins and lead outs that are applied during import of parts. The parameters are defined separately for different situations identified according to machining requirements.

Lead-ins/-outs are set according to type of particular cut. *mCAM* recognizes different types of cuts and automatically apply predefined lead – in/–out:

- **Convex corner** – defines the type of lead-in/-out based on location of the lead-in/out on the contour – in this case on the convex corner of the cut
- **Contour** – defines the type of lead-in/-out based on location of the lead-in/out on the contour – in this case on the contour besides the convex corner.
- **General contour** – cutting a generally opened contour with possibly different starting and ending points (non-cyclical cut)
- **Slot** – narrow gap that is cut by one single cut
- **Flange cut –** this type of contour represents cut (e.g. H-beam trim cut), where ignition needs to be performed near the flange edge. Flange cut on beams (H/ U/ L–beam) requires a special type of lead–in to cutting contour (slide edge) that begins and ends on the edge and its lead–in /–out needs to be performed by slide edge (cross–section of I –beams, U–beams, H profiles).
- **Web cut –** this type of contour represents cuts, where ignition needs to be performed on web near the flange from inner side. Web cut of beam (H/ U/ L–beam) requires a special type of lead–in to cutting contour (web lead) that begins in the material normal position, continuously bevels to limit position and ends in the limited position of cutting contour.
- **Welding –** represents welding cutting contours

 Convex corner	 Contour	 General contour
 Slot	 Flange cut	 Web cut
 Welding		

Each section contains several parameters for setting the lead in/out:

- **Type** – based on the selected type, other parameters are active or not
- **Base length –** set by user
- **Thickness multiplier –** multiplies the material thickness by set number
- **Tangential angle –** the angle between a lead-in /-out and a tangent vector (applies for all types of linear lead-ins/outs)
- **Section angle –** the angle between the sections for linear type with two identical sections (applies for linear lead-in/out with 2 sections)
- **Tool orientation**

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.

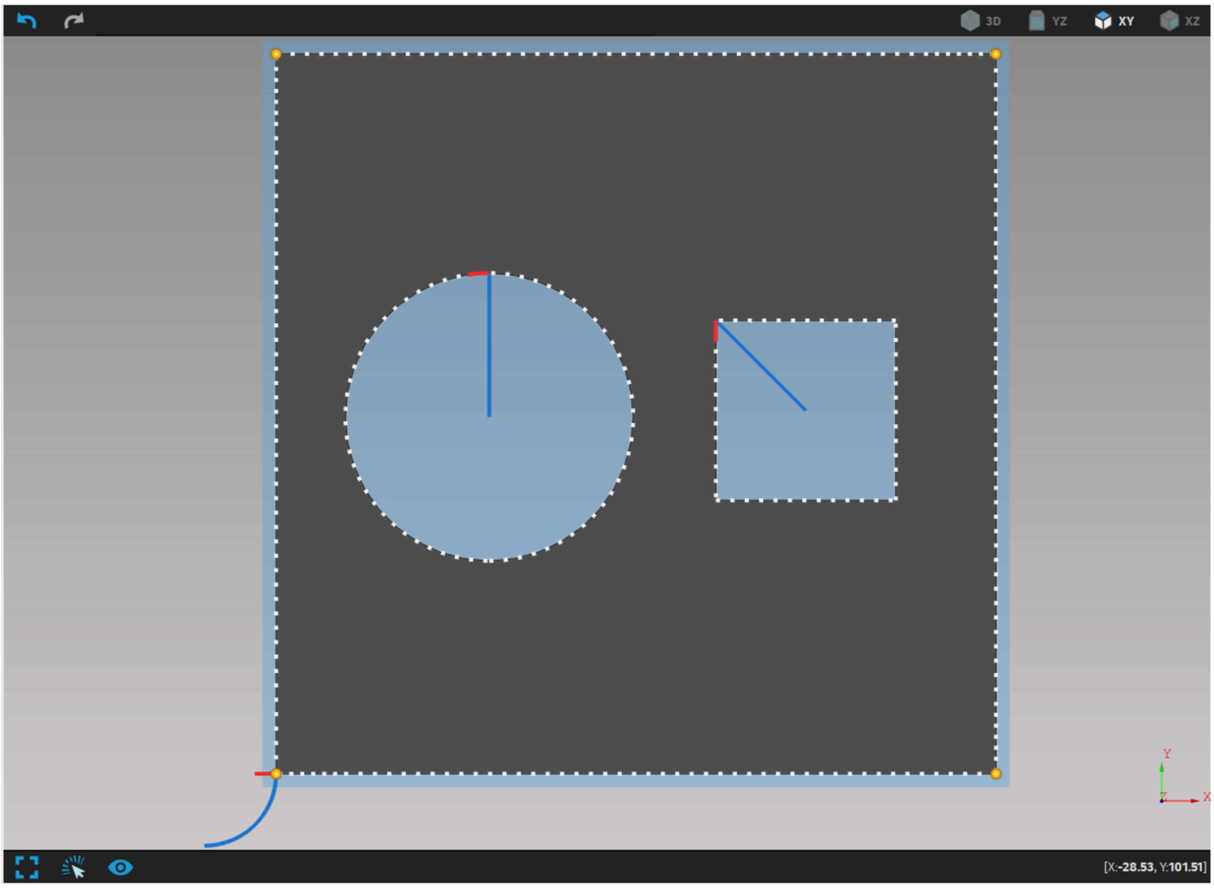


Fig. 320: Lead-ins on small holes begin in the center of the hole

There are several types of lead-ins/-outs in mCAM:

- **none**
- **path conforming**
- **path conforming & linear**
- **linear (1 section)**
- **linear (2 sections)**
- **arc (90 deg)**
- **arc (180 deg)**
- **in profile axis –** this method is used when lead-in/-out needs to be performed in X axis of profile, especially on H-beams while cutting Web section
- **Slide edge –** this method is used in cases of cross-section cuts of I-beams, H-beams, U-profiles, etc. where lead-in s/-outs have to be performed on the edge of the part (slide on the edge)
- **Web –** this method is used in all Web-cuts on H-beams where ignition near the flange from inner side is required. A tool starts in the straight position (in normal to material) followed by continuous bevelling to limit position near flange

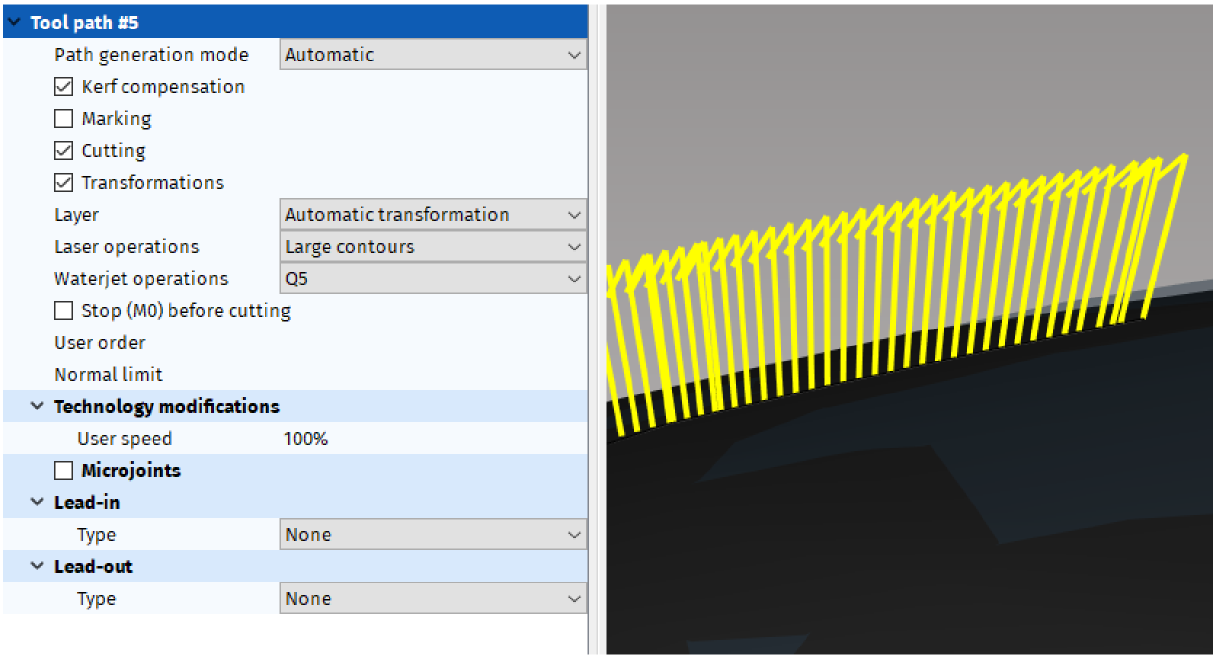


Fig. 321: Lead–in none, lead–out none on gap cutting contour

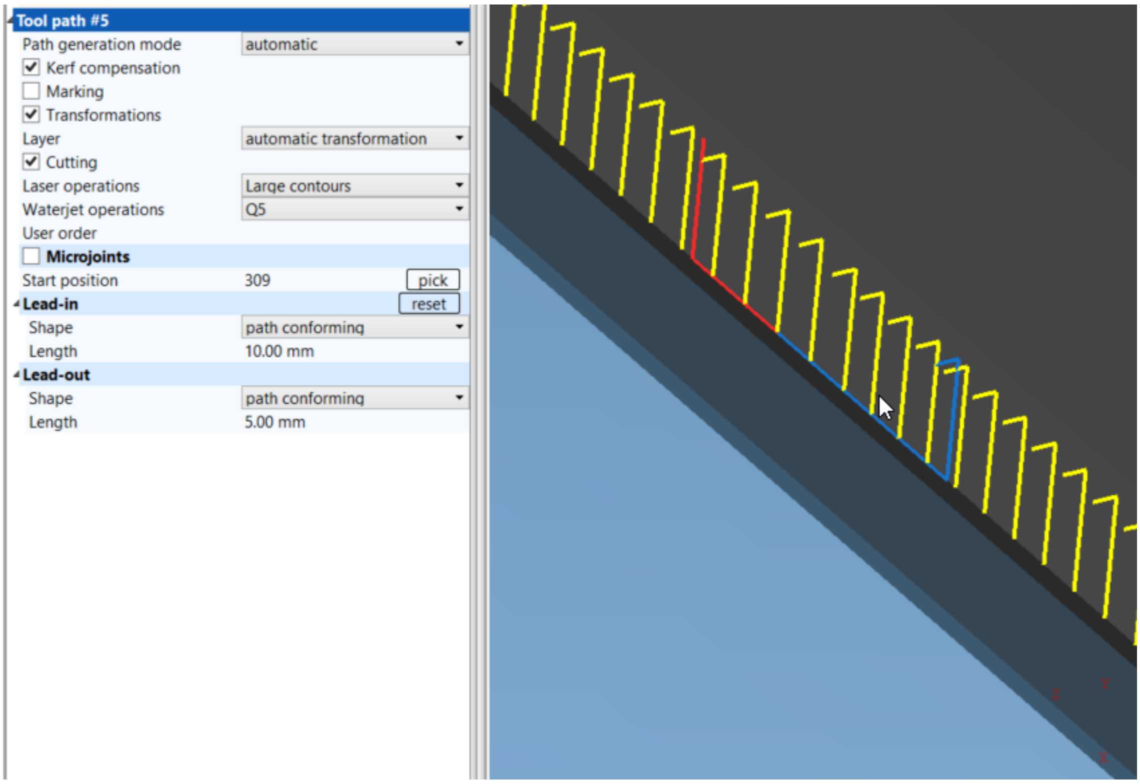


Fig. 322: Lead–in path conforming, lead–out path conforming

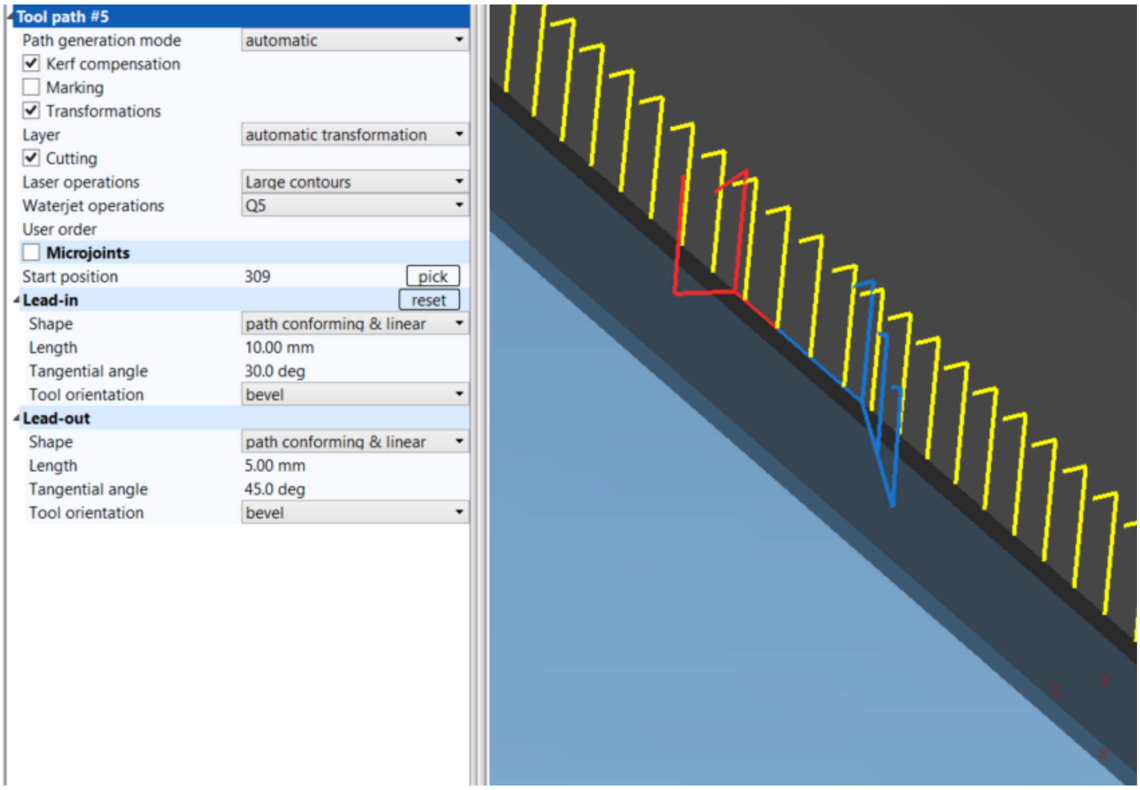


Fig. 323: Lead–in path conforming & linear, Lead–out path conforming & linear

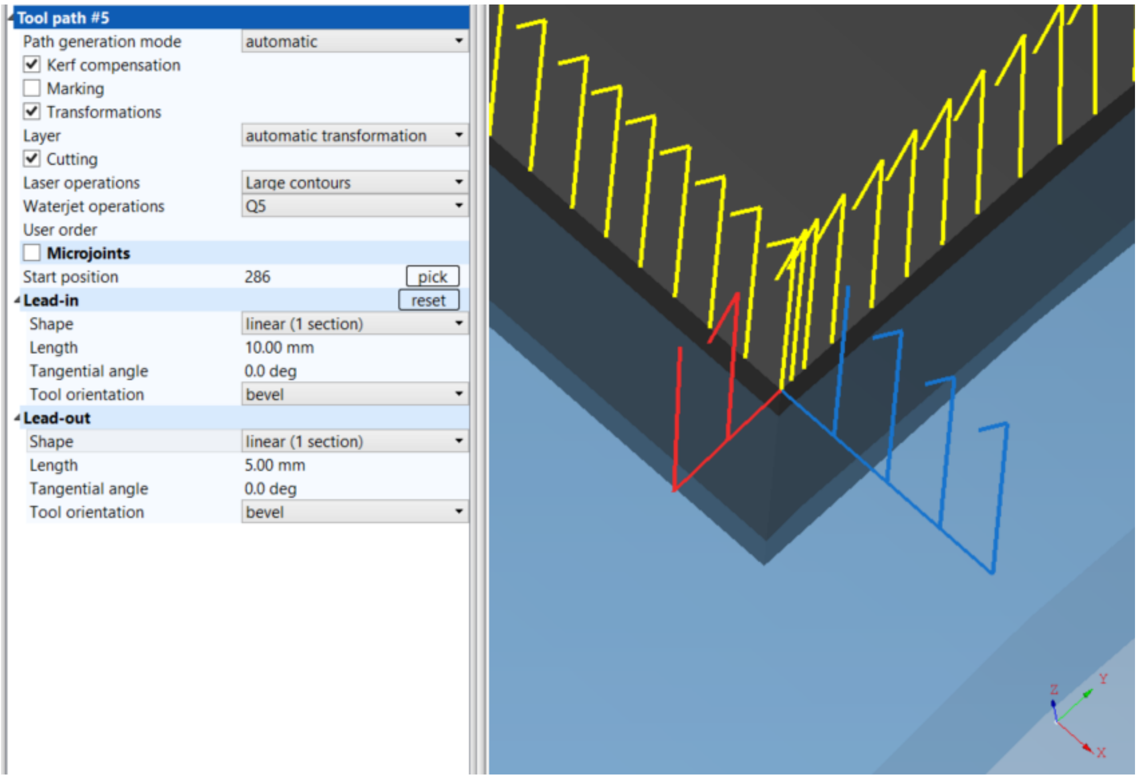


Fig. 324: Lead–in linear (1 section), Lead–out linear (1 section)

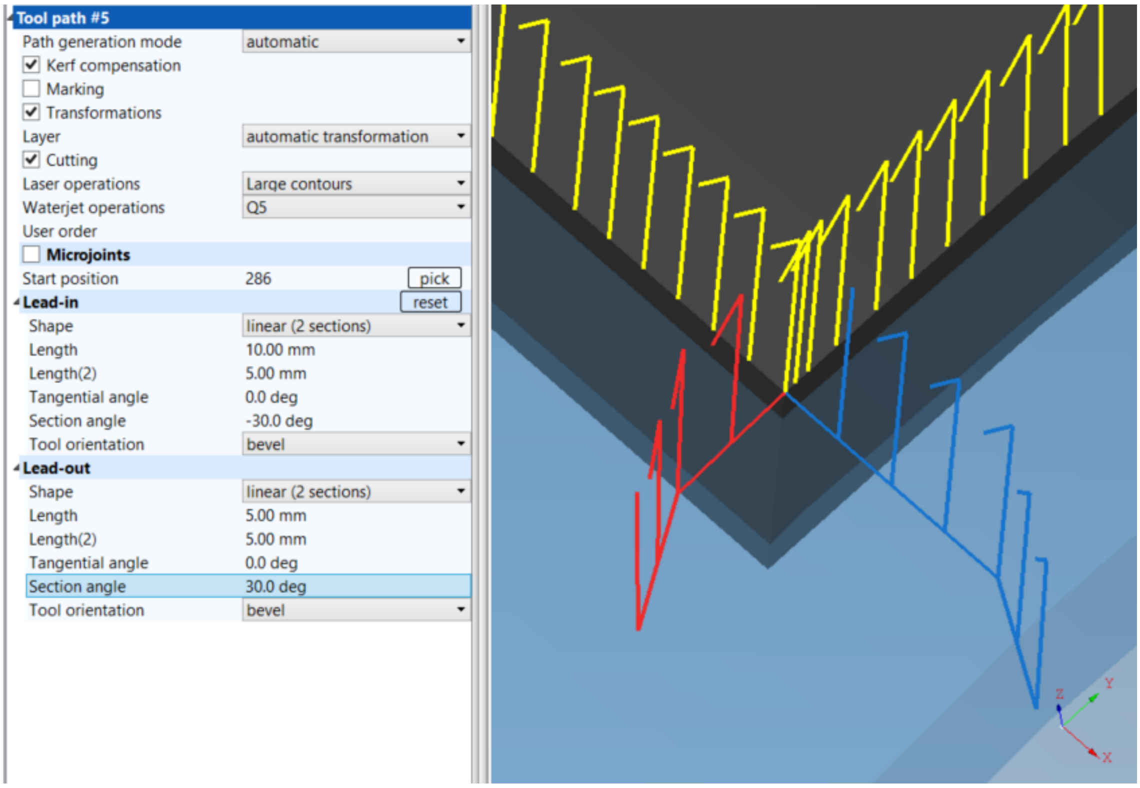


Fig. 325: Lead–in linear (2 section), Lead–out linear (2 section)

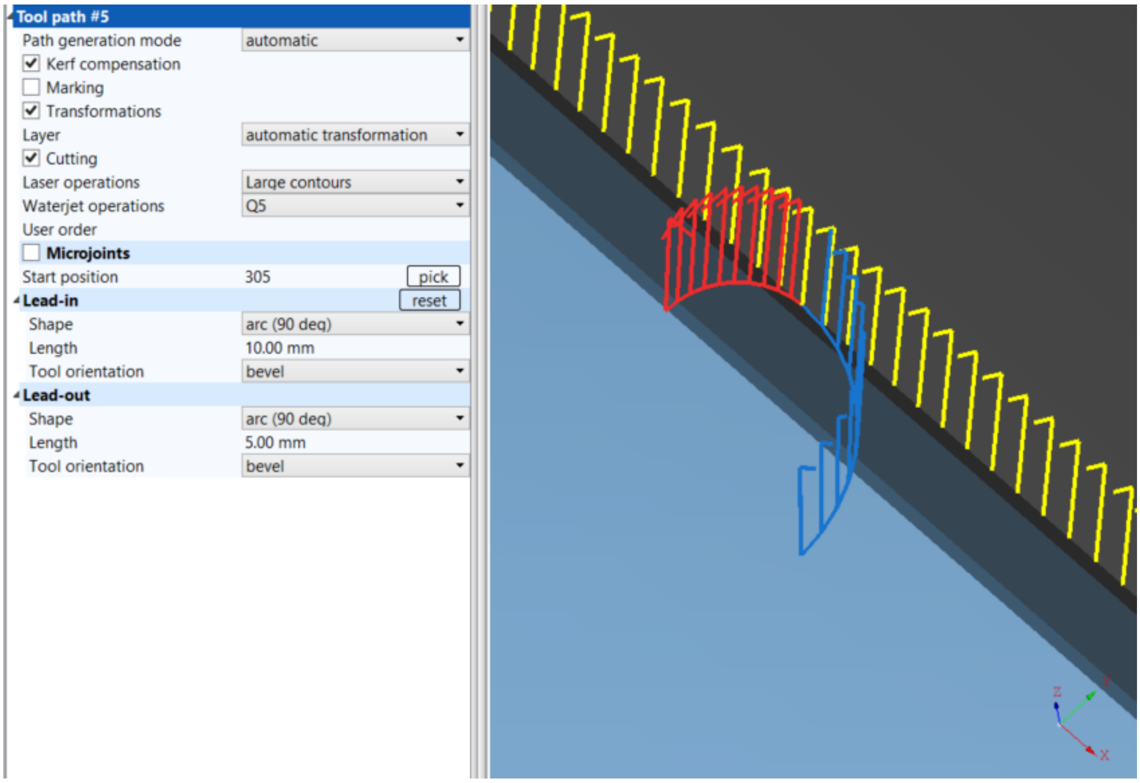


Fig. 326: Lead–in arc (90 deg), Lead–out arc (90 deg)

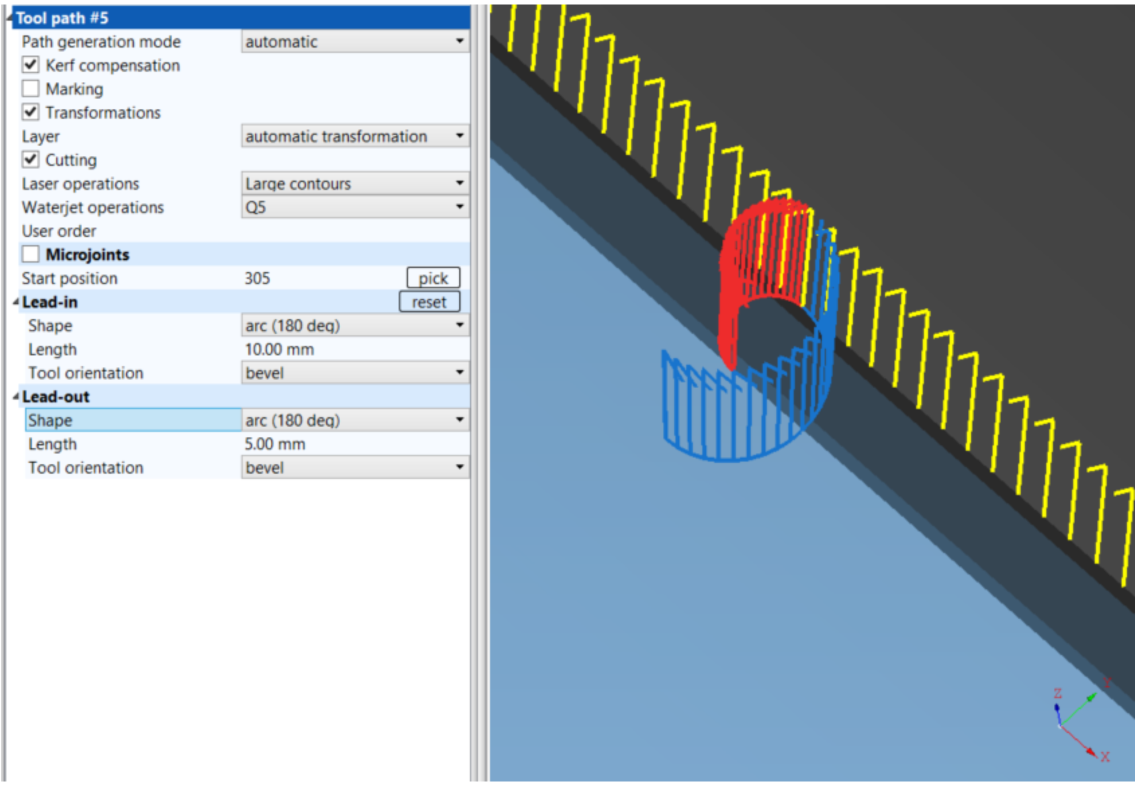


Fig. 327: Lead–in arc (180 deg), Lead–out arc (180 deg)

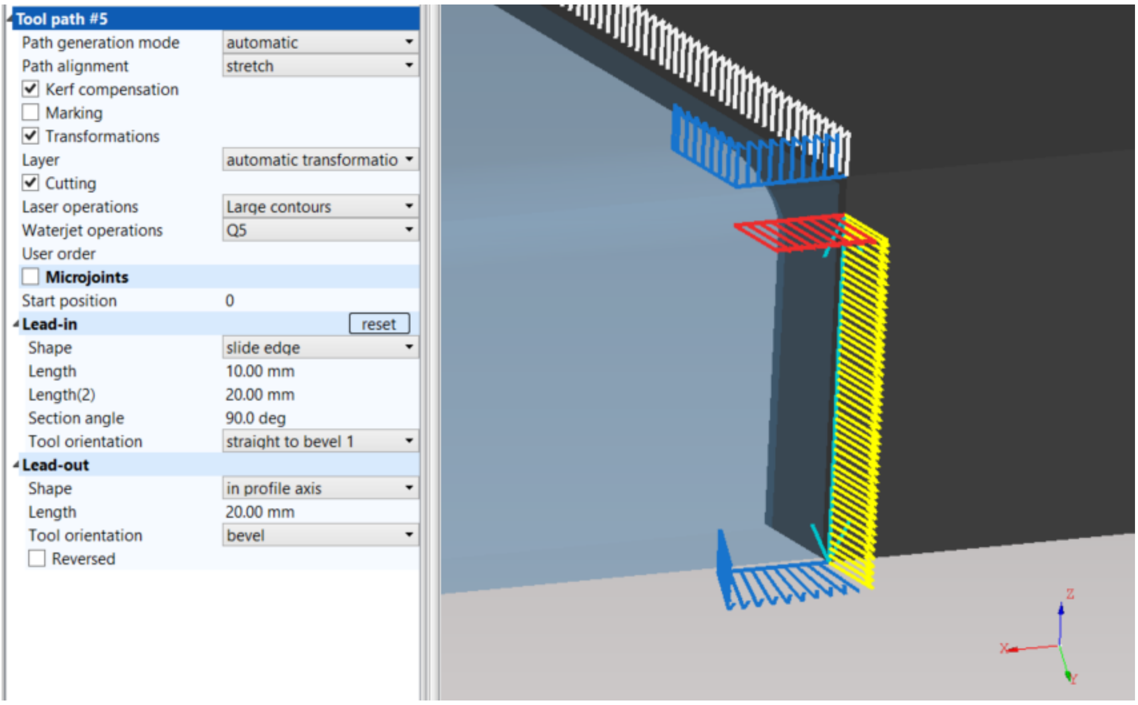


Fig. 328: Lead–in edge slide, Lead–out in profile axis

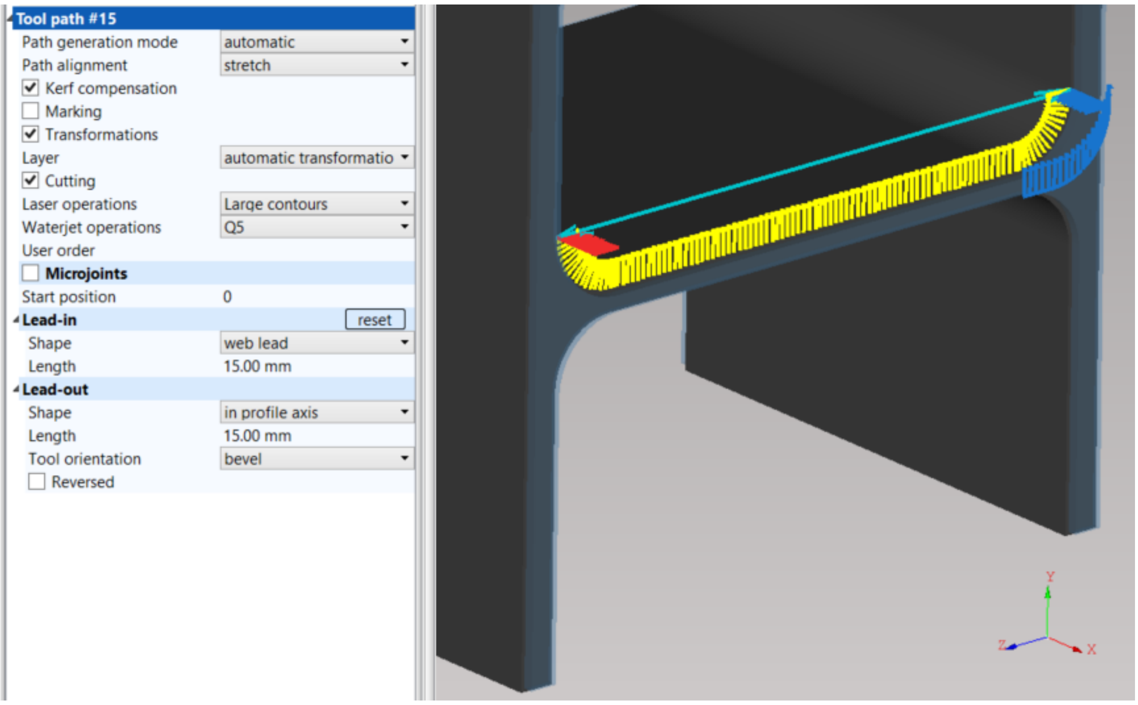


Fig. 329: Lead-in Web lead, Lead-out in profile axis

Depending on tool orientation, the lead-in s/-outs can be performed in four different ways:

- **Bevel** – a tool is maintained in a bevelled position and enters/leaves a contour with corresponding tilt angle
- **Straight** –a tool is kept in the straight position and bevelling is performed only at the points of entry and departure of a contour – suitable only for straight cutting heads
- **Straight to bevel 1** – a tool starts / finishes in the straight position (in normal to material) followed / preceded by continuous bevelling, bevelling takes place in the middle of lead–in /–out path
- **Straight to bevel 2** – a tool starts / finishes in the straight position followed / preceded by continuous bevelling, bevelling takes place in the last third of lead–in /first third of lead–out path

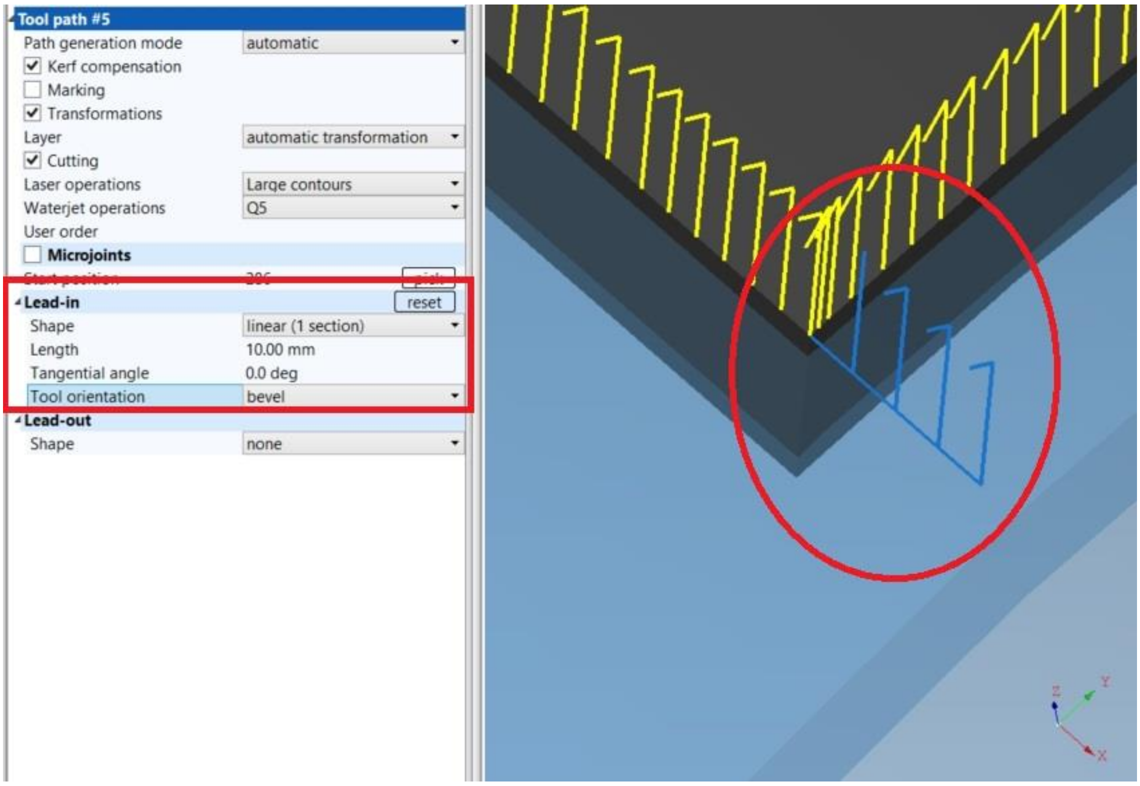


Fig. 330: Tool orientation – bevel

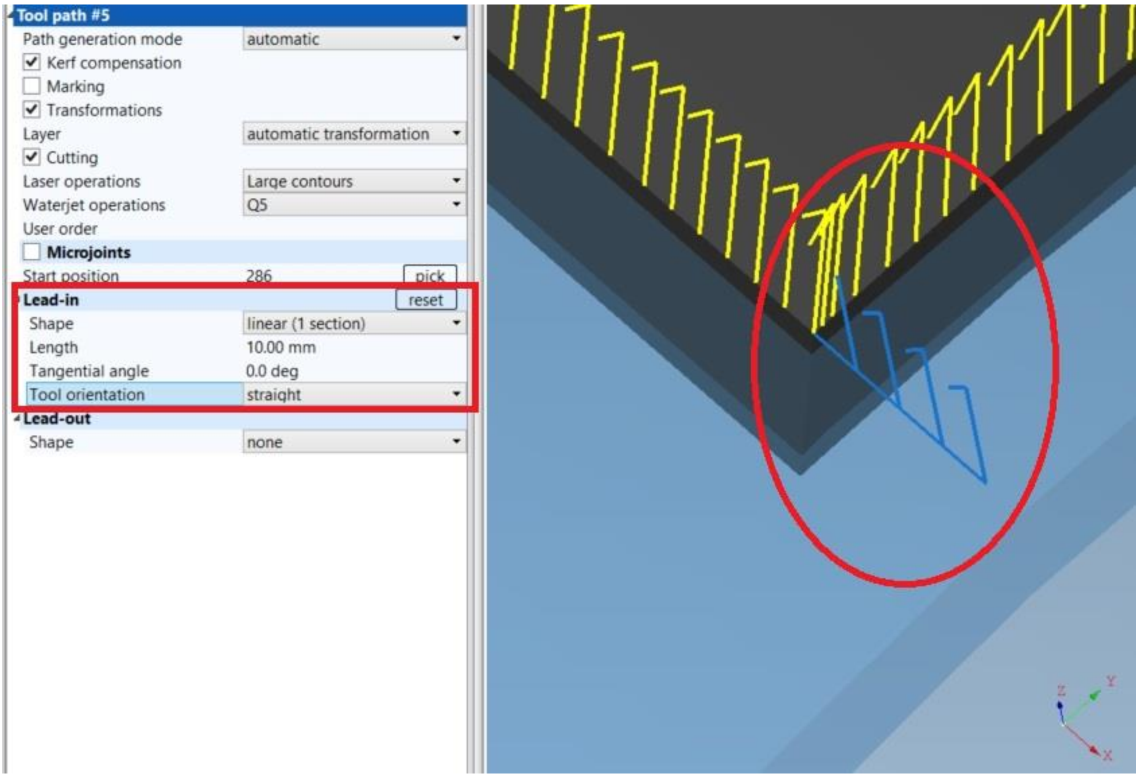


Fig. 331:Tool orientation – straight (not suitable on bevel cuts)

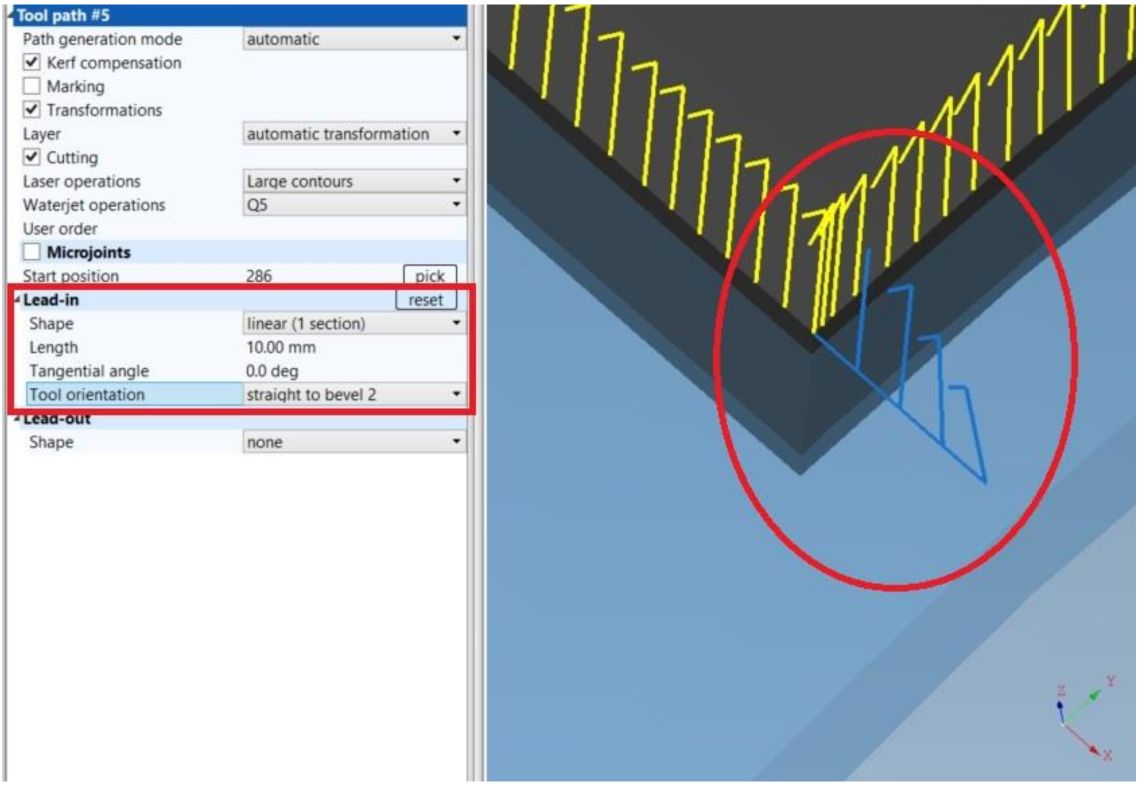


Fig. 332:Tool orientation – straight to bevel 2

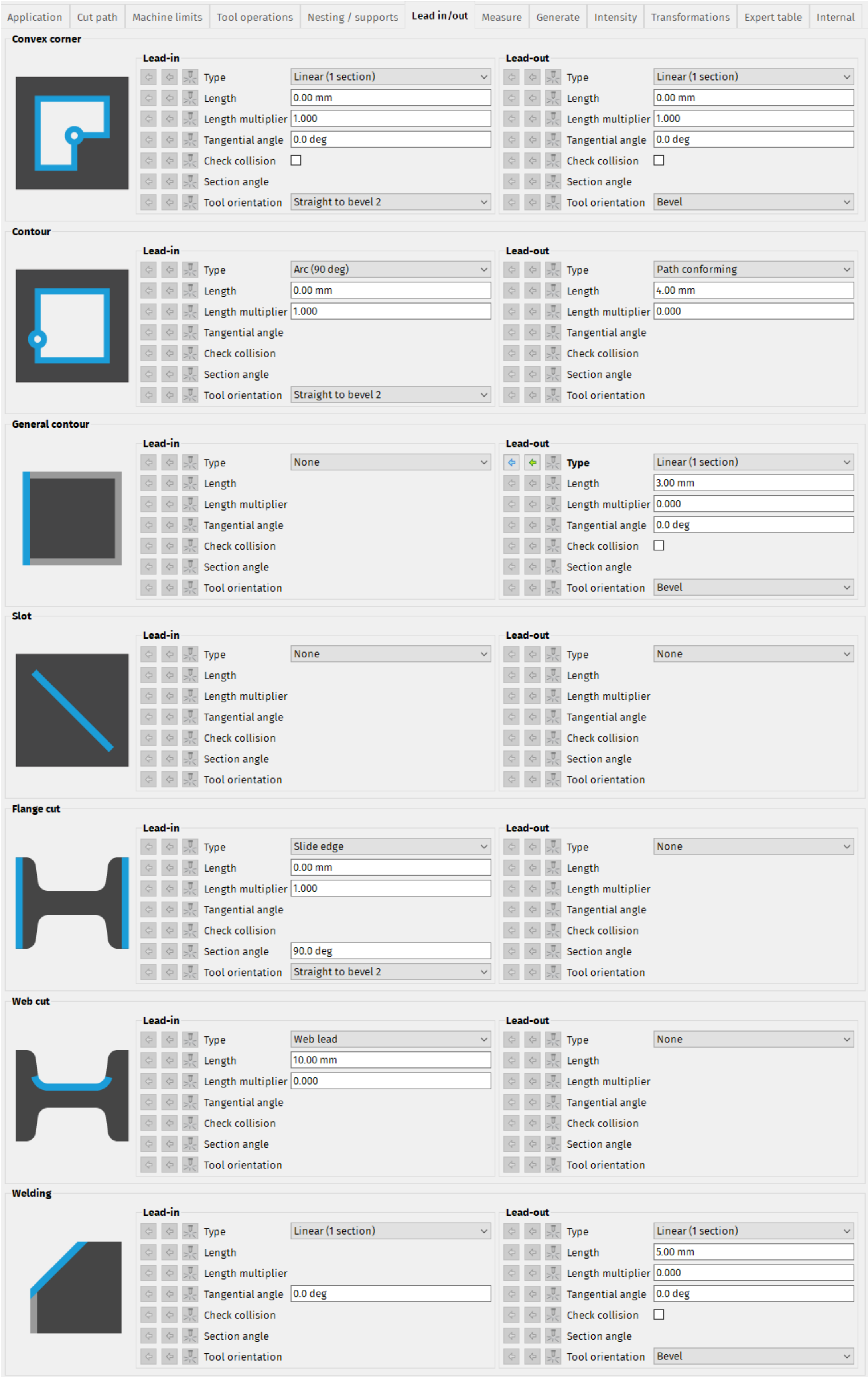


Fig. 333: Lead in/out settings – presets

Measure

All settings in this tab are applicable to surface detection for pipes as well as to laser scanner detection for profiles.

Regardless if Scanner surface detection or Default surface detection mode is selected as main surface detection mode, 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

Subsection Measurements defines parameters of surface detection:

- **Main surface detection mode –** beam surface detection is a method of coarse scanning of a material by defined measurement procedure. This function is used to determine surface detection method.
  - → **Default surface detection mode** (performs standard IHS surface detection). It is further specified by Default surface detection mode options
  - → **Laser scanner surface detection** (machines equipped by laser scanner for beam measurement – H/ U/ L-beams and rectangular shaped pipes).
  - → **Model based cutting (without any measurement)**
- **Adaptivity mode** – Adaptive mode defines the way how the tool is moved above the material surface. From tool stand–off control point of view, there are two types of cutting:
  - → **Do not use adaptivity**, so–called robotic mode (non–adaptive cutting) used in bevel cutting, and cutting of curved surfaces, pipes, domes, etc. In case of non–adaptive cutting (curved surfaces, pipes, domes, etc.), activation of the option Prefer surface detection is recommended.
  - → **Use adaptivity whenever possible** – adaptive control of tool height is preferably used for straight cutting, a common method for plasma cutting is arc voltage based adaptive control
  - → **Always use adaptivity –** adaptivity is turned on each cut regardless it is straight or bevel contour (recommended to use only water–jet machines equipped by adaptive measurement unit that detects material surface continuously while cutting all contours).
- **Default surface detection mode –** surface detection is a method of coarse scanning of a material surface by measurement of tool height in several points. This setting defines detection method for plates, domes, sphere–caps and circular pipes. Available options are:
  - → **No measurement** (water–jet machines without measurement unit),

→ **Measure only one point** (laser cutting machines),

→ **Measure plane if possible** (plasma cutting machines).

Whenever possible, it is recommended to use surface detection using three points (plane measurement) to achieve the highest possible accuracy. In specific situations, the number of detection points is reduced to two or even to one. Usually, the restrictions follow from limited space available for the detection.

Subsection _Points detection_:

- **Maximum angle between material normal and up direction while surface detection [°]** – the parameter defines maximum possible angle of the tool with respect to the normal to a material surface where it is allowed to perform measurement for surface detection. In fact, the parameter limits the area of material surface by means of maximum curvature where surface detection or IHS (initial height sensing) can be applied to prevent without a damage of a measuring tool.

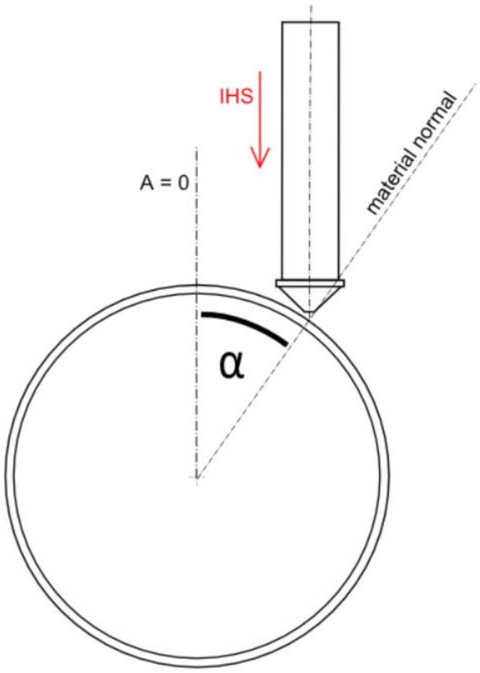


Fig. 334: Maximum angle between material normal and up direction while surface detection

- **Minimum distance of detection points [mm] –** the parameter imposes limits for minimum distance between all surface detection points.

Subsections _Profile, beam and pipe shape detection:_

- **Measure mode**

  → **Measure every cut separately –** for each cut separate measurement is made

- → **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
- **Can use two measure points -** 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 (mm) –** 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 measurement (mm) –** 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 (mm) –** when the length of cut-off material exceeds the entered value, new laser 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.

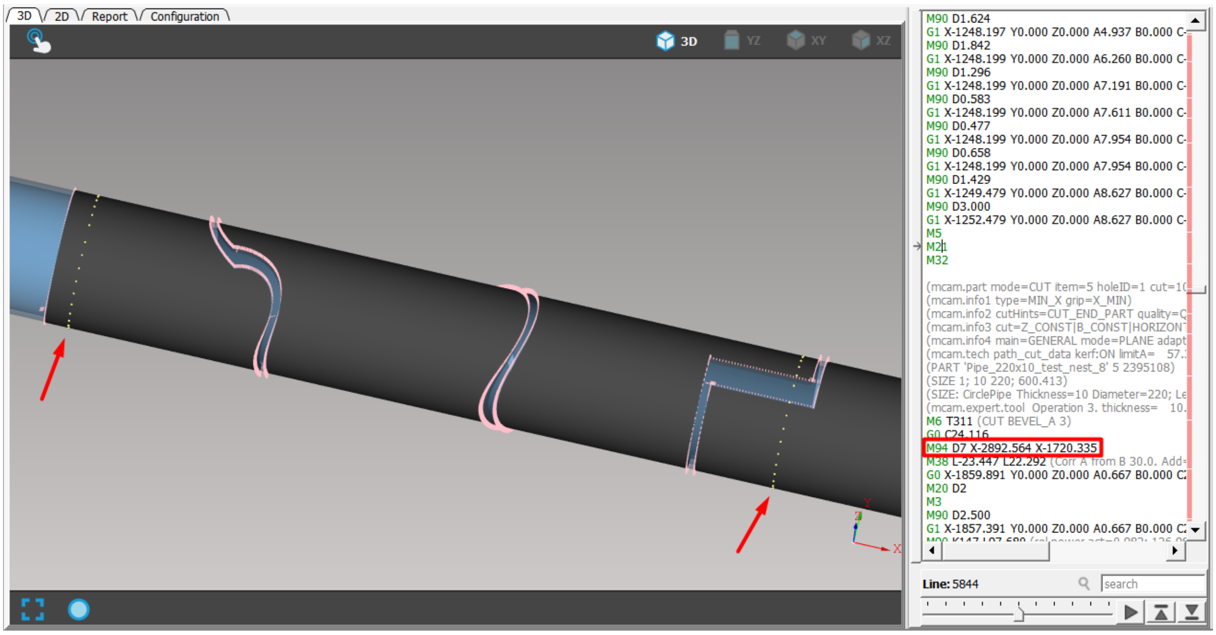


Fig. 335: Measuring on two X positions for group of cuts on pipe

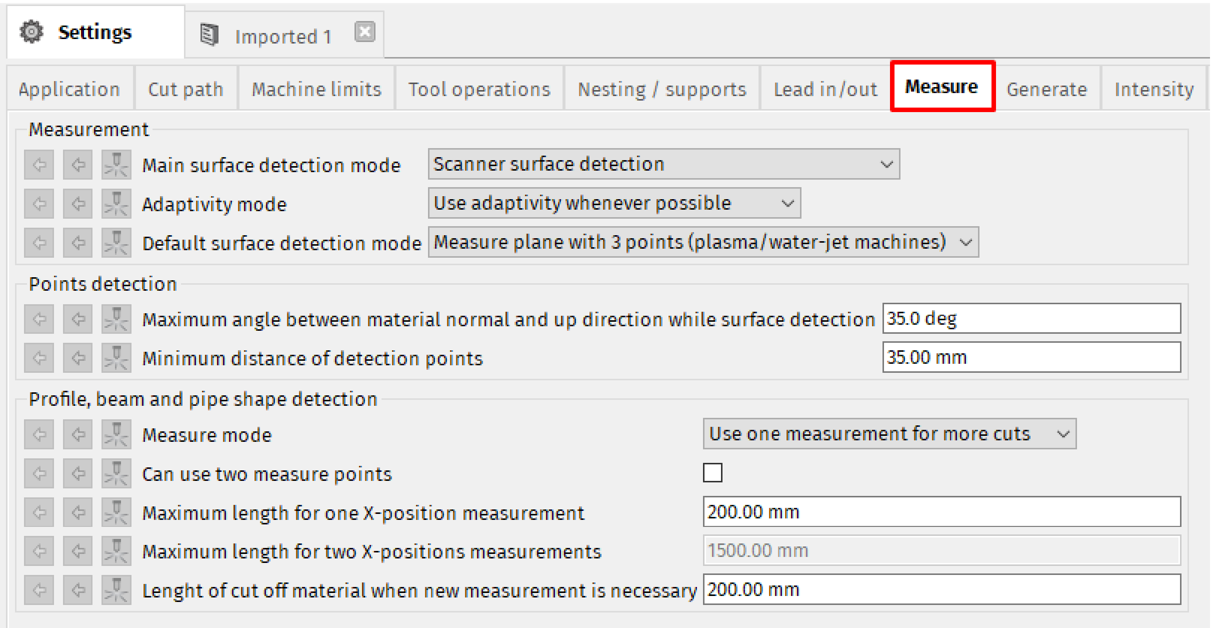


Fig. 336: Measure settings

Generate

Subsection *Generate* defines functions and parameters used during the plan generation:

- **CNC usage**
  - → **CNC will be cut –** mode without applying mesh
  - → **Before cut, mesh will be applied –** when preparing cutting plan for parts that will be scanned and mesh will be used for adjusting cutting paths
- **Supports positioning by CNC –** this function enables to generate *CNC program* with instructions of supports positions. The *mCAM* is able to generate instructions of supports positioning that are automatically recalculated to absolute machine coordinates by machine control system or relative coordinates of support positions in comments of *NC code* for manual positioning without recalculating.
- **Automatic material/lunette positioning by CNC**
  - → **No material/lunette positioning –** this option is for machines without material/lunette positioning
  - → **Positioning of every cut –** for machines with standard IHS measuring
  - → **Positioning of group of cuts –** for laser measuring, for machines that use laser scanner for pipes and beams measurement
- **Use M121 –** when applying mesh to adjust cut paths to real shape of a material, CNC code M121 moves the center of the dome to 0,0 coordinates
- **Collision limitations on domes -** 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.

Subsection *C axis movements* defines these settings:

- **Default mode for cutting circle pipe –** the parameter forces behaviour of the machine in case of cutting round pipes. Circle pipe cutting modes are described in more details in section Cutting mode.
  - → **Moving in XY axes (Y is rotation)** – setting means that relative movement of a tool with respect to the material surface in Y direction is performed by rotation of the pipe.
  - → **Moving in XYZC axes –** both the pipe and cutting head can be moved

→ **Moving in XC axes –** the cutting head is in straight position – the Z axis is fixed

→ **Moving in XYZC axes (fixed in B) –** cutting head can move only in direction of the pipe

- **Default mode for cutting beams –** the parameter defines used mode of beam cutting. Mode *Plane cutting* is used for *ProfileCut* machines equipped by 120°pantographic rotator where beams are cut on static supports. Mode *Moving in XYZ* is used for other machines equipped by standard 45° rotator where beams are clamped in rotary positioner and placed in supports while rotating the beam during the cutting.

- **Linearize C–axis movements on profile corners without rotator –** the function linearizes the movements of C–axis while cutting profiles corners (rectangular shaped pipes) without using rotator (used only for straight cutting heads)
- **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)

Subsection *Straight cutting* defines maximum A-axis angle for straight cutting and limitations used for backward rotation of B axis during cutting:

- **Max angle between torch and material normal for straight cut [°]** – a threshold for tilting angle of A-axis. If the angle is smaller than the parameter value, the mCAM considers it as straight cutting (T111). This specifies usability of the tool for straight cutting with the advantage of adaptive tool height control despite the low non–zero angle of A-axis, following from exact computations.
- **Max A to ignore B [°]** – a threshold defined for tilting angle of A-axis that enables to ignore B–axis and thus to prevent excessive rotation of B-axis in single point that possibly causes damaged contour by concentrating energy of the tool in one location.

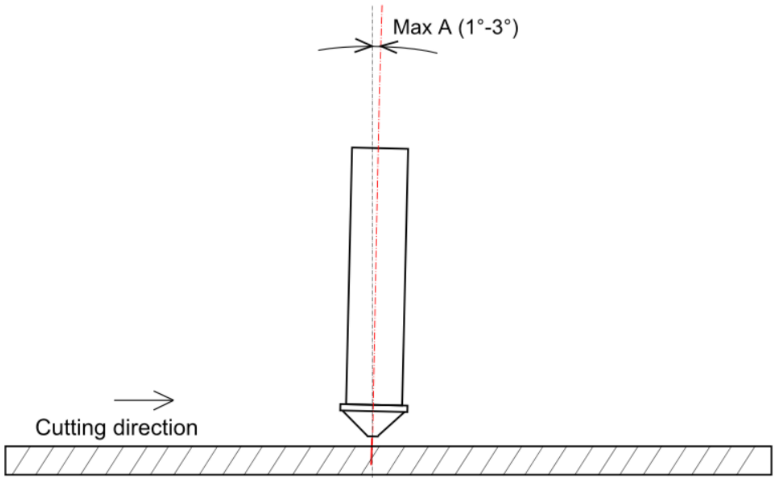


Fig. 337: Maximum A axis angle for straight cut and for B-axis repositioning

Subsection <u>*Ordering*</u> defines all ordering parameters:

- **Marking–cutting changeover** – the parameter is used to determine machine behaviour after marking is completed and prior to cutting. There are two possible settings: Immediately start cutting and Pause after marking.
- **Item finishing mode –** this function defines an ordering mode. Every mode has its own significant feature and utilisation. There are four *Item finishing modes*:
  - → *Program ordering (nearest contour)* **–** cutting order is based on the principle of the nearest cut. This mode also uses the principle Mark all which means *mCAM* generates all marking instructions and then all instructions for cutting (Marking – Cutting).
  - → *Finish whole plan (mark all parts)* – all parts in whole cutting plan are marked first and then cut
  - → *Finish part (mark all -> cut all)* **–** cutting order is based on the principle *Mark all parts at once* which means *mCAM* generates marking instructions for all parts first and then all cutting instructions. All these instructions are generated according to the principle *Part at once* which means that each part (including its holes or inner contours) must be completely finished with particular technology. After system finishes this process on one part it can move to the next position (part).
  - → *Finish part (mark+cut)* **–** cutting order is based on the principles *Part at once*, *Mark at onc*e and which means *mCAM* generates all marking instructions and all cutting instructions for each part separately. All these instructions are generated according to the principle Part at once which means that each part (including its holes or inner contours) must be completely finished (marked and then cut) first. After system finishes marking and cutting on whole part it can move to the next part.
- **Thermal mode**
- **Start point of plate CNC program –** defines the start point position of CNC program on sheet. Start point can be set to four positions: Xmin/Ymin, Xmin/Ymax, X max/Ymin, Xmax/Ymax. This setting can be useful especially when working with sheet cutting (2D cutting).
- **Movement at program end** – the parameter expresses machine behavior after cutting is completed. Available options are:
  - → No movement at program end
  - → At program end return to begin position
  - → At program end move to next free position

Subsection <u>*Parking mode*</u> defines global parking mode and parking mode for text marking:

- **Global parking mode** – this function defines the type of global parking mode used for all technological operations except text marking operation and sections controlled by Short distance parking mode. Available options are:

    - → Full parking
    - → Limited parking
- **Global parking mode in text marking** – this function enables to define parking mode used during the text marking. Available options are:
    - → Full parking
    - → Limited parking
    - → No parking.

Subsection *Short distance parking mode* defines individual parking mode for short distance sections between two separate cuts:

- **Maximum distance of two subsequent cuts** – the parameter defines the maximum distance between end and start of two subsequent cuts.
- **Parking mode –** the function defines the parking mode used between two subsequent cuts with defined distance in previous parameter.

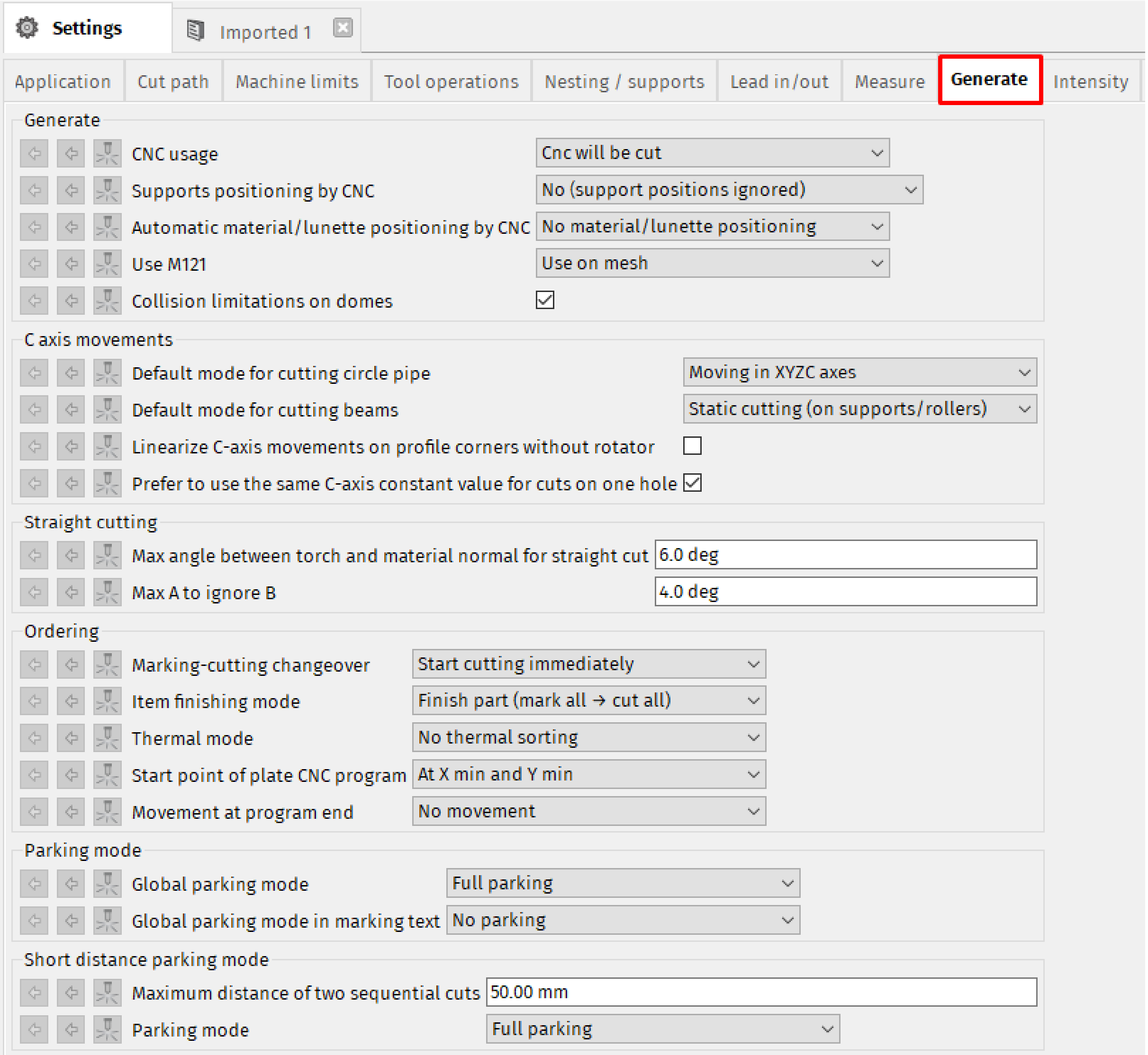


Fig. 338: Generate settings

Intensity

Cutting intensity modifiers affects the quality of cutting especially in complicated areas on plates, pipes or profiles. Cutting modifiers contain functions for slowing down in sharp corners of cutting contour, slowing down at the end of cutting contour, controlling the cutting speed according to effective cutting thickness or reduction of cutting power according to the cut area. All cutting modifiers are applied during generation of CNC program.

Subsection *Slow in corners* defines parameters of slowing–down on sharp edges on cutting contour:

- **Slow in corners** – this function enables to use relative speed on edges that meet minimum edge angle limitation.

- **Minimum angle of corner [°] –** the parameter defines minimum angle of the corner to be cut by relative speed.
- **Relative speed [%] –** the parameter defines relative cutting speed used in corners that meet minimum edge angle condition.
- **Distance [mm] –** the parameter defines the length of cutting path from the corner of material cut by defined relative speed.

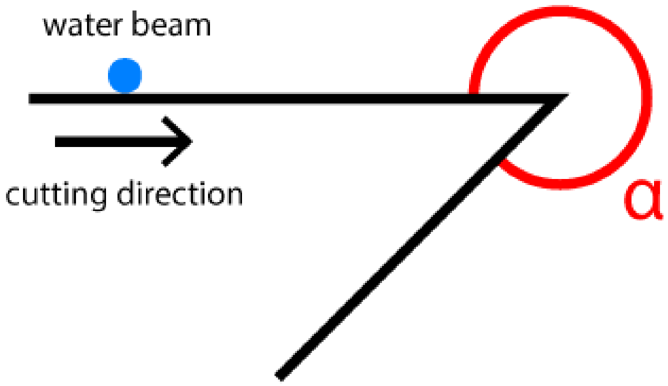


Fig. 339: Minimum angle of contour to use relative cutting speed (WaterJet cutting)

Subsection *Slow at end* defines parameters of slowing–down before each lead–out in cutting contour:

- **Slow at end** – this function enables to use relative speed at end of each cutting contour.
- **Relative speed [%] –** defines relative cutting speed used at end of cutting contour.
- **Distance [mm]** – defines the length of cutting path from the last point of cutting contour cut by defined relative speed.

Subsection *Adjust intensity control according to effective cutting thickness* enables/disables automatic control of cutting speed according to real effective thickness.

Subsection *Cutting are based intensity control* defines parameters for power reduction on edges:

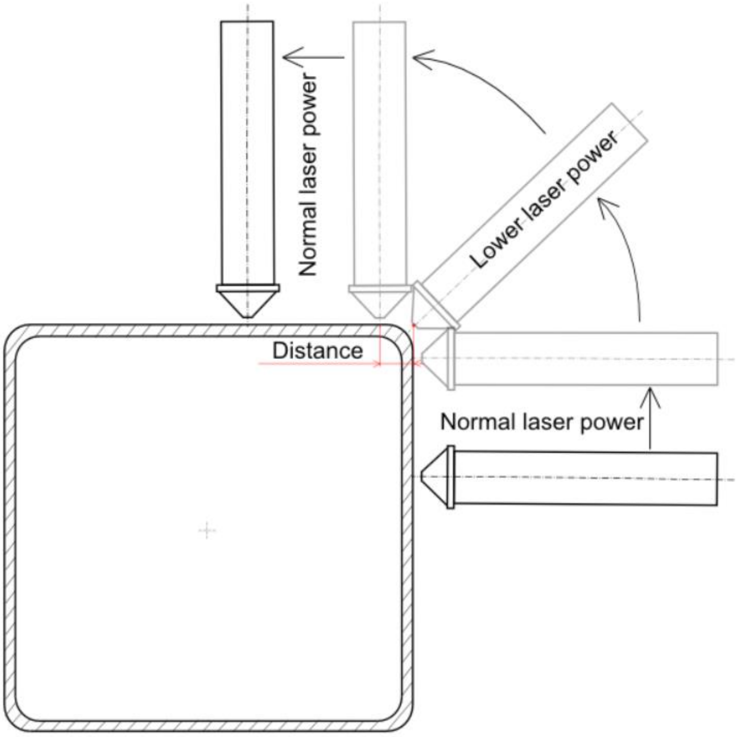


Fig. 340: Affected area by reduced laser power

- **Cutting area based intensity control** – this function enables to use relative power according to current cut area in profile radius.

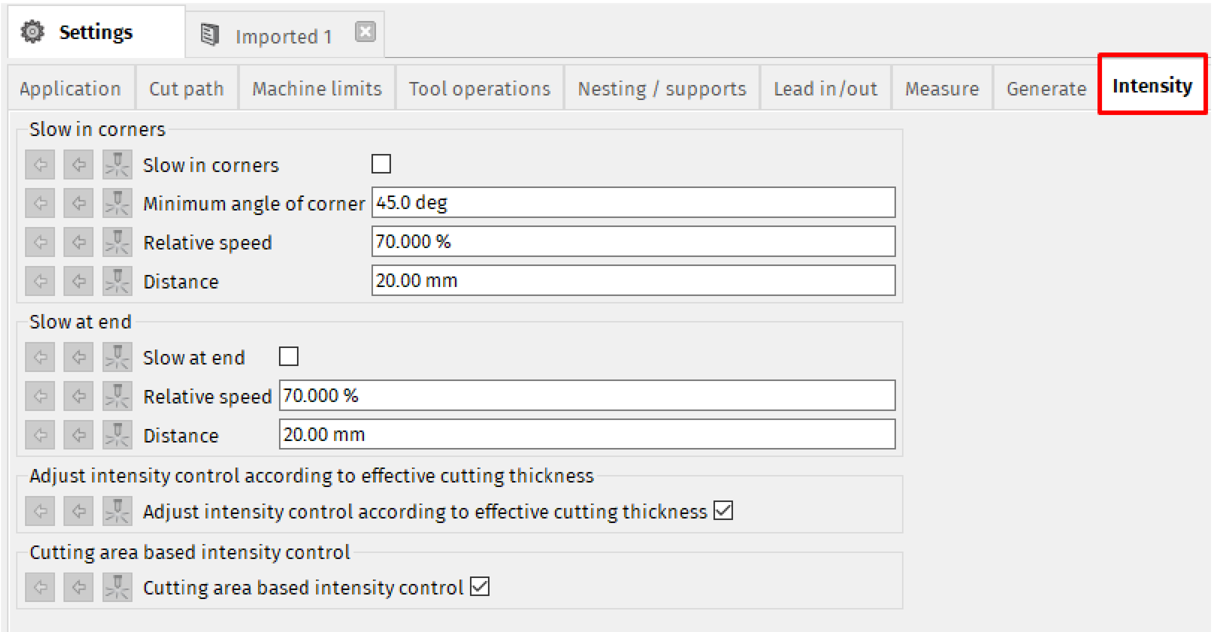


Fig. 341: Intensity settings – cutting intensity modifiers

Transformations

This section contains list of *Transformations* (rules, conditions and tasks) that can be applied on cutting paths. *Transformations* are used for *Quality hole* applications (*True hole/ Contour cut*), *Drilling* and *Punching.*

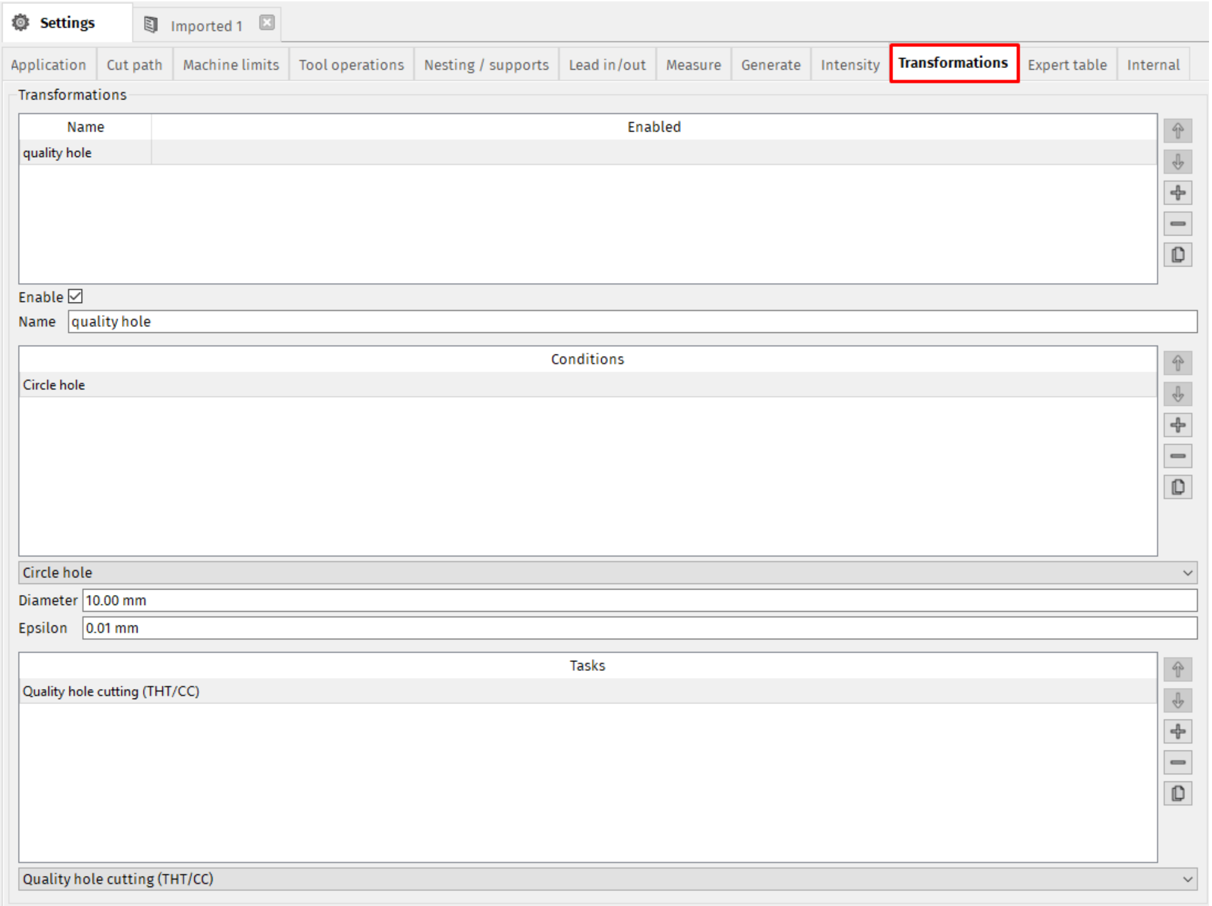


Fig. 342: Transformations settings

Setup transformations

Transformations are set for whole application (not for particular machine or technology), therefore it is not necessary to select any machine, technology or tool. All transformations can be edited in program configuration level.

Transformations are processed during generation of CNC program, therefore user is able to see whether transformations were applied as expected only in cutting simulation.

Transformation rule

Transformation rule contains set of transformations that should be performed on particular cuts (drill + tapping; punch + marking, etc.) that meet the conditions set for each transformation rule. One transformation rule may contain as many transformation conditions and as many transformation tasks as needed. It is possible to add, delete or copy transformation rules by buttons on the right side of the screen.

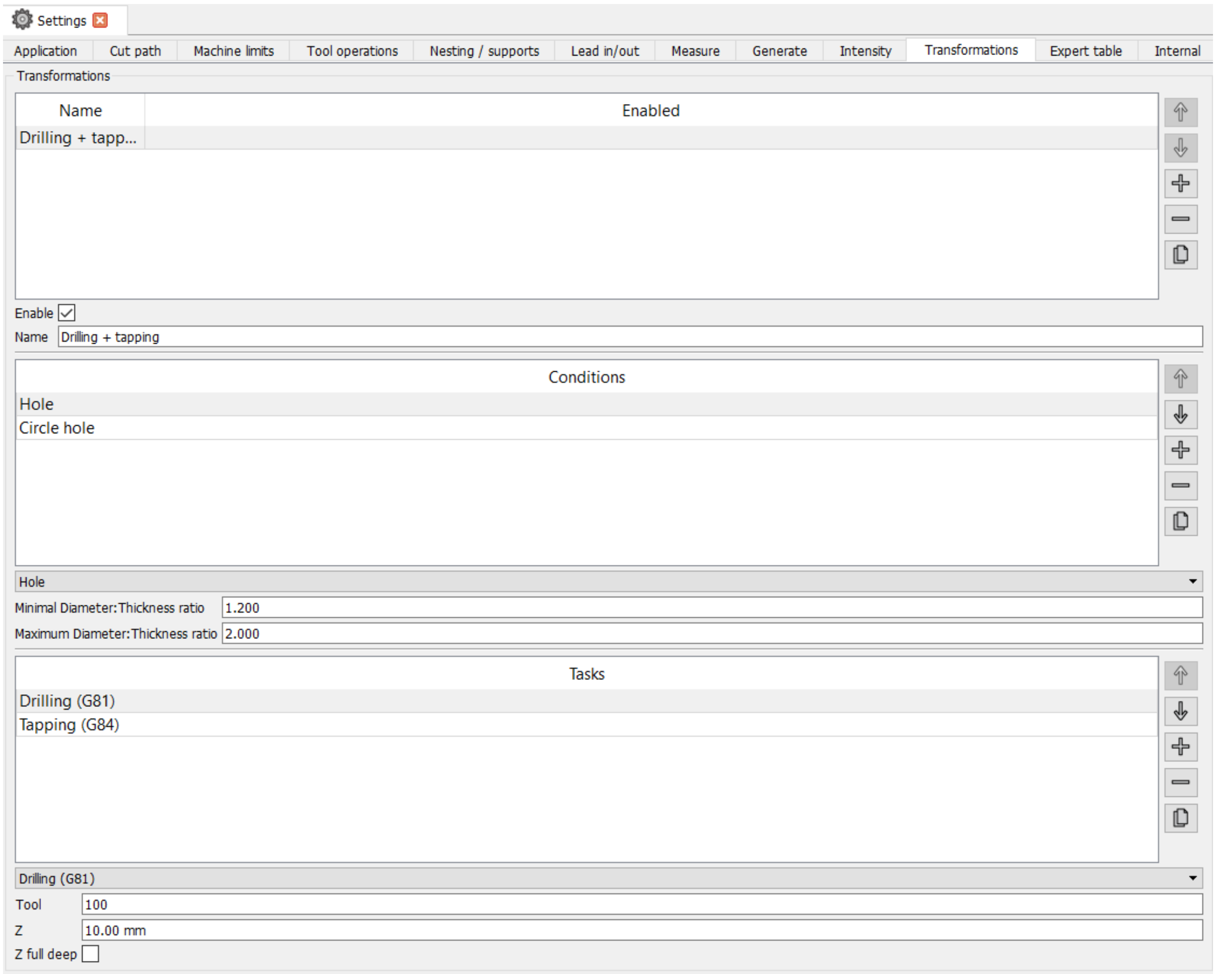


Fig. 343: Transformation rule, its conditions and tasks

Disabling transformation rules

Transformation rule can be completely disabled (for whole application) in settings of particular transformation rule by unchecking the checkbox Transformation rule enabled. Transformation then won't be performed even if some feature fulfils criteria set by disabled transformation.

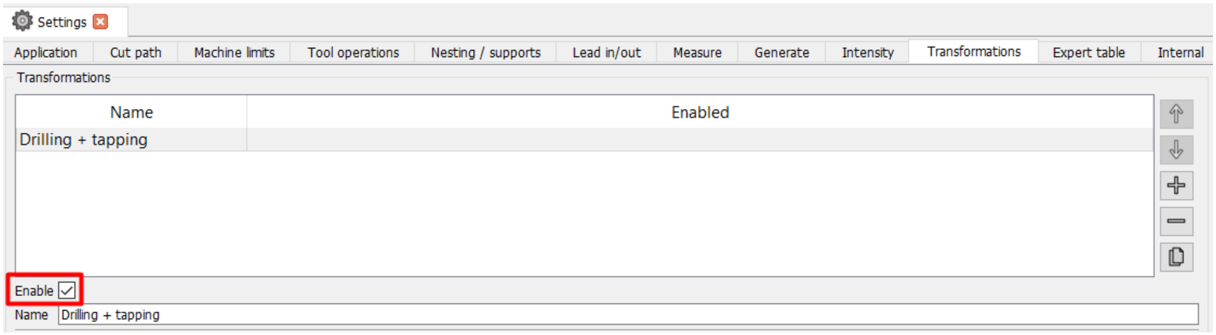


Fig. 344: Global disabling of transformation rule

Transformation condition

Transformation condition defines a condition or set of conditions (in version 1.3 only circle diameter, size of any closed hole) where transformation task should be used.

Specified transformation task is applied to features (cutting contours) that match at least one of transformation conditions defined within particular transformation rule (unless transformation rule is disabled).

Transformation conditions available in version 1.5:

- **Circle hole**; defined by:
    1) Diameter of hole [mm]
    2) Epsilon [mm] (tolerance)

Set transformation task or tasks are applied on all circle holes that match at least one of the transformation condition (hole diameter + tolerance).

- **Hole** (any closed hole); defined by:
    1) Minimum diameter: thickness ratio
    2) Maximum diameter: thickness ratio

Transformation task is applied on holes where distance of two points of contour (two points with maximum distance) is smaller than multiple of thickness of the material and thickness multiplier (d < t. coef).

- **Constant thickness –** transformation task is applied only when the condition of constant thickness is met (defined by epsilon = tolerance)
- **Path in plane –** transformation task is applied only when whole path is in plane

- **Always false –** alternative way to disable transformation task, but it still remains accessible in dropdown menu in properties area for selected cut path (Tool path -> Layer) and can be manually applied

Transformation task

Transformation task defines the operation (e.g. drilling or punching) or set of operations (e.g. drilling + tapping) that should be performed on matching feature (cutting contour) defined by transformation condition.

Transformation of any contour automatically disables standard cutting (unless cutting is also included in transformation task) and applies set of transformation tasks.

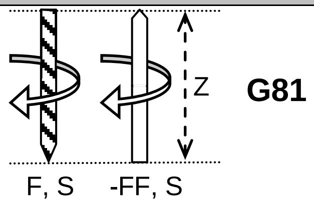
Transformation tasks available in version 1.5:

- **Drilling cycles** (G81, G82, G83, G84, G85, G86, G89) (all drilling cycles are described in more details in section Drilling)
- **Cutting**
- **Punching**
- **Quality hole cutting (THT/CC)**
- **Quality hole cutting in axis (THT/CC)**

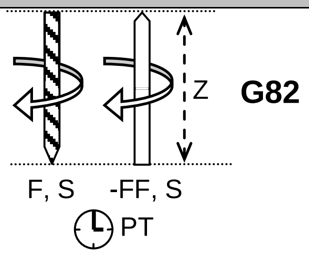
Drilling cycles

A *drilling cycle* is an elementary drilling operation represented by a dedicated instruction in CNC code. Arguments of *drilling cycles* include *depth*, direction of rotation, *time dwell*. The *mCAM* supports the following drilling cycles: "*G81*", "*G82*", "*G83*", "*G84*", "*G85*", "*G86*", and "*G89*". All *drilling cycles* are defined and used in *Transformations* (*Settings – Transformations*). *Transformations* are described in more details in section Transformations.

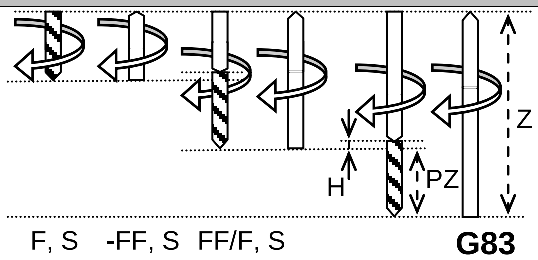
G81 – Drilling

<table>
<tr><th colspan="4">G81 – Drilling</th></tr>
<tr><td>Parameters</td><td>Z</td><td>Drilling depth [mm]</td><td>
</td></tr>
<tr><td>Comment</td><td colspan="3">Feed F [mm/rev] and spindle rotation speed S [rpm] are set in tool parameters</td></tr>
<tr><td>Example</td><td colspan="3">G81 Z30 (Simple drill cycle with 30 mm depth)</td></tr>
</table>

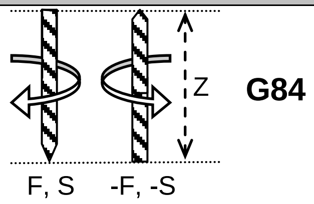
G82 – Drilling with dwell

<table>
<tr><th colspan="4">G82 – Drilling with dwell</th></tr>
<tr><td rowspan="2">Parameters</td><td>Z</td><td>Drilling depth [mm]</td><td rowspan="2">
</td></tr>
<tr><td>PT</td><td>Dwell time in the hole [s]</td></tr>
<tr><td>Comment</td><td colspan="3">Feed F [mm/rev] and spindle rotation speed S [rpm] are set in tool parameters</td></tr>
<tr><td>Example</td><td colspan="3">G82 Z30 PT5 (Drilling cycle with 30 mm depth and 5 s dwell after drilling)</td></tr>
</table>

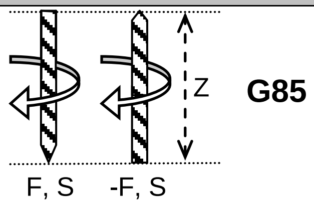
G83 – Deep drilling

<table>
<tr><th colspan="4">G83 – Deep drilling (peck drilling)</th></tr>
<tr><td rowspan="2">Parameters</td><td>Z</td><td>Drilling depth [mm]</td><td rowspan="2">
</td></tr>
<tr><td>PZ</td><td>Drill step [mm]</td></tr>
<tr><td>Comment</td><td colspan="3">Feed F [mm/rev] and spindle rotation speed S [rpm] are set in tool parameters.
Correction for retract step H [mm] is set in servo in parameter KorekStep</td></tr>
<tr><td>Example</td><td colspan="3">G82 Z30 PZ5 (Drilling with depth 30 mm performed in 5 mm steps)</td></tr>
</table>

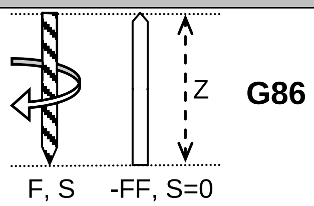
G84 – Tapping

G84 – Tapping (making threads)			
Parameters	**Z**	Drilling depth [mm]	
Comment	Feed *F* [mm/rev] and spindle rotation speed *S* [rpm] are set in tool parameters		
Example	G84 Z30 (*Tapping with depth 30 mm*)		

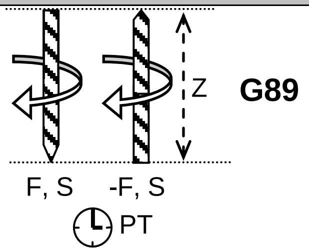
G85 – Boring with feed retract

G85 – Boring with feed retract			
Parameters	**Z**	Drilling depth [mm]	
Comment	Feed *F* [mm/rev] and spindle rotation speed *S* [rpm] are set in tool parameters		
Example	G85 Z30 (*Boring depth 30 mm with slow return*)		

G86 – Boring with spindle stop

G86 – Boring with spindle stop			
Parameters	**Z**	Drilling depth [mm]	
Comment	Feed *F* [mm/rev] and spindle rotation speed *S* [rpm] are set in tool parameters		
Example	G86 Z30 (*Boring depth 30 mm with spindle turning switched off when returning*)		

G89 – Boring with dwell and feed retract

G89 – Boring with dwell and feed retract			
Parameters	**Z**	Drilling depth [mm]	
	PT	Time dwell in the hole [s]	
Comment	Feed *F* [mm/rev] and spindle rotation speed *S* [rpm] are set in tool parameters		
Example	G89 Z30 PT5 (*Boring depth 30 mm with spindle kept turning and 5 s dwell*)		

Expert table

This section is used for *Expert tables* generation and editing. *Expert tables* contain cutting parameters (*thickness – cutting speed – kerf width*) used for kerf width compensation and cutting speed control. *Expert tables* initial setup as well as subsequent editing is described in section Expert tables.

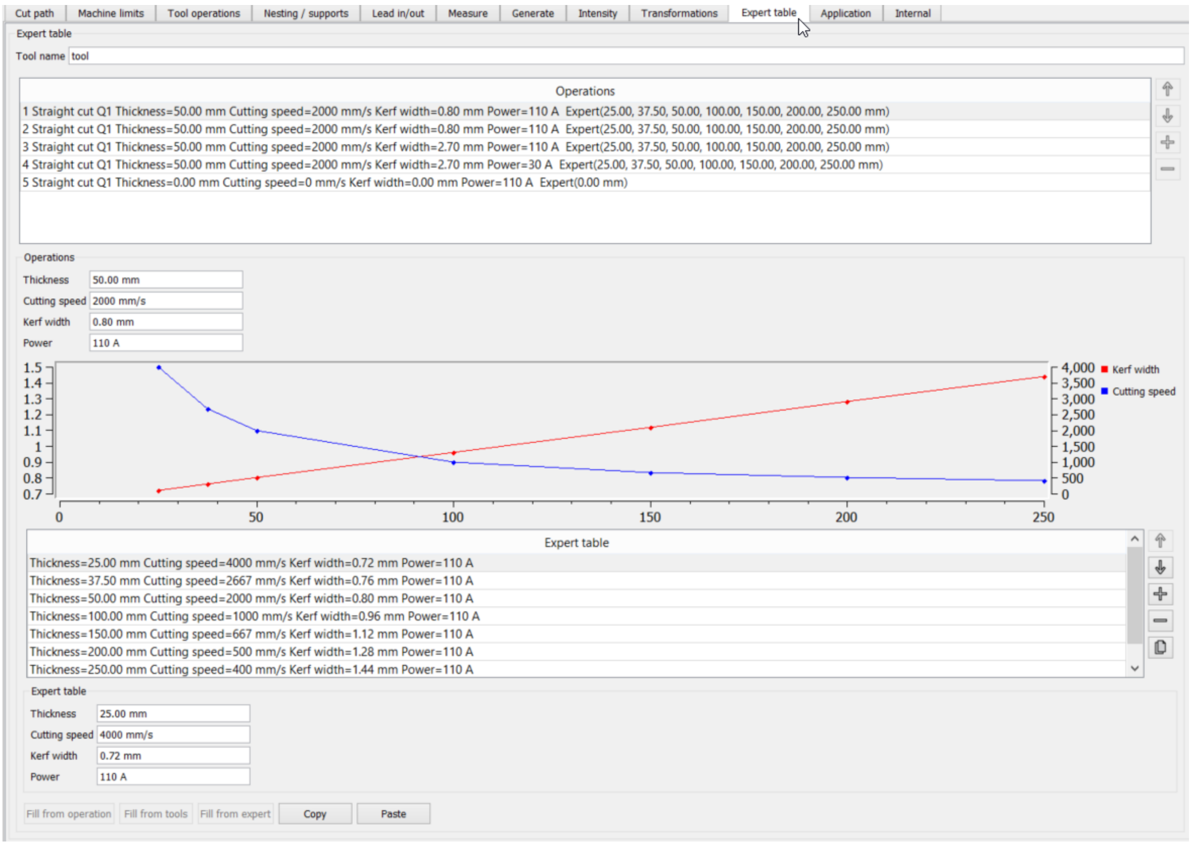


Fig. 345: Expert table settings

Internal

Parameters in the folder are intended to be set by MicroStep staff. Correctness of the values is a necessary condition for trouble–free performance of the machine. The parameters define fundamental conditions for generation of cutting plans and due to the internal nature they are not explained in this document.

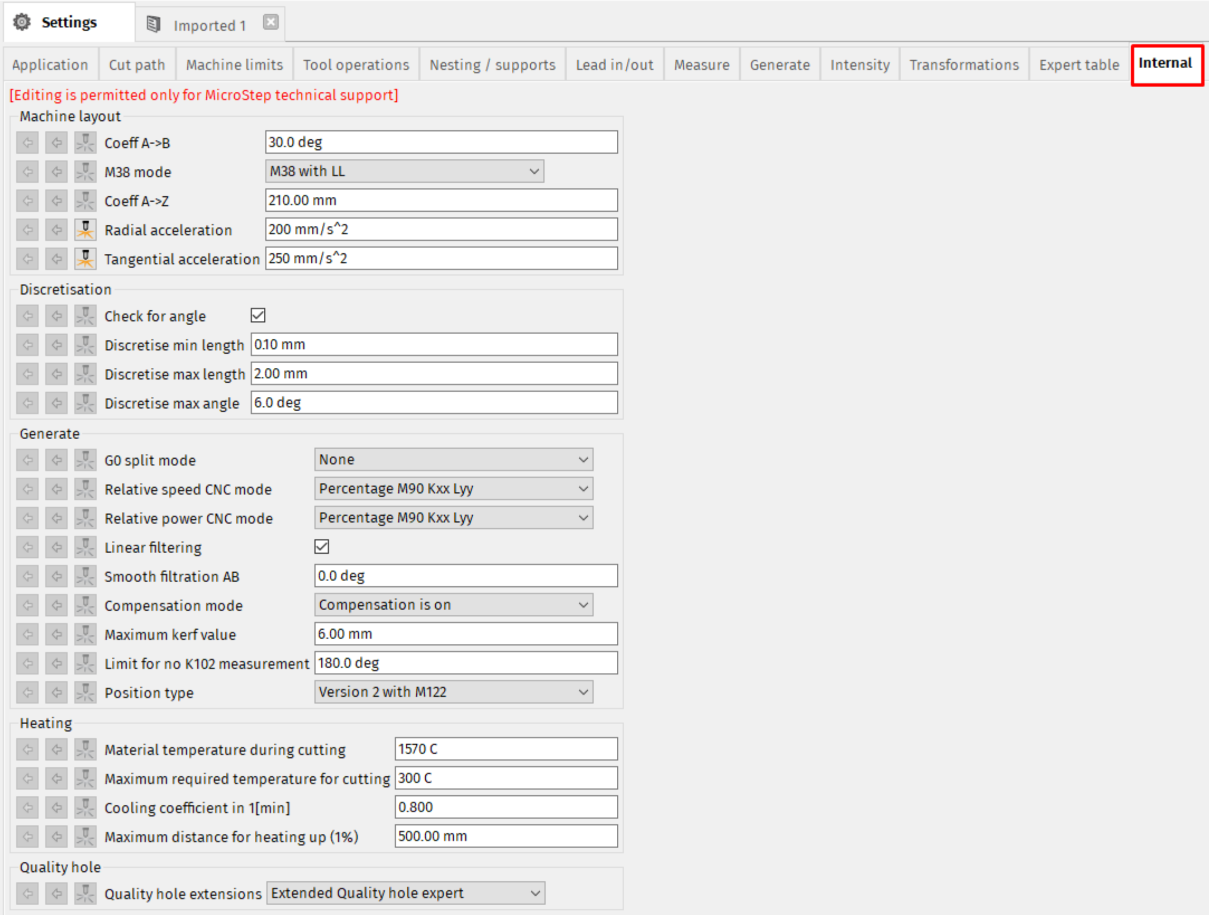


Fig. 346: Internal settings

Troubleshooting

This section contains several troubleshooting options, reporting system and basic information about the program (version, used libraries, etc.), view help (manual guide), console, tips and keyboard shortcuts.

Help

This subsection contains basic information about program manuals, console usage, useful paths, tips and keyboard shortcuts.

About

This option displays the basic information about program, version of the executable file and version of dllcore file, and about used libraries: Qt, Open Cascade, etc.

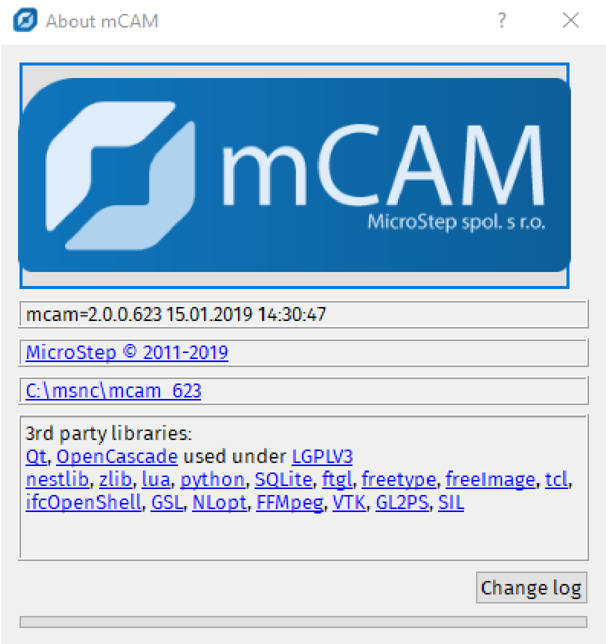


Fig. 347: About program

View new features

This function opens the PDF version of user manual – upgrade to version 2.0.

View help

This function opens the PDF version of user manual, keyboard shortcut: <F1>.

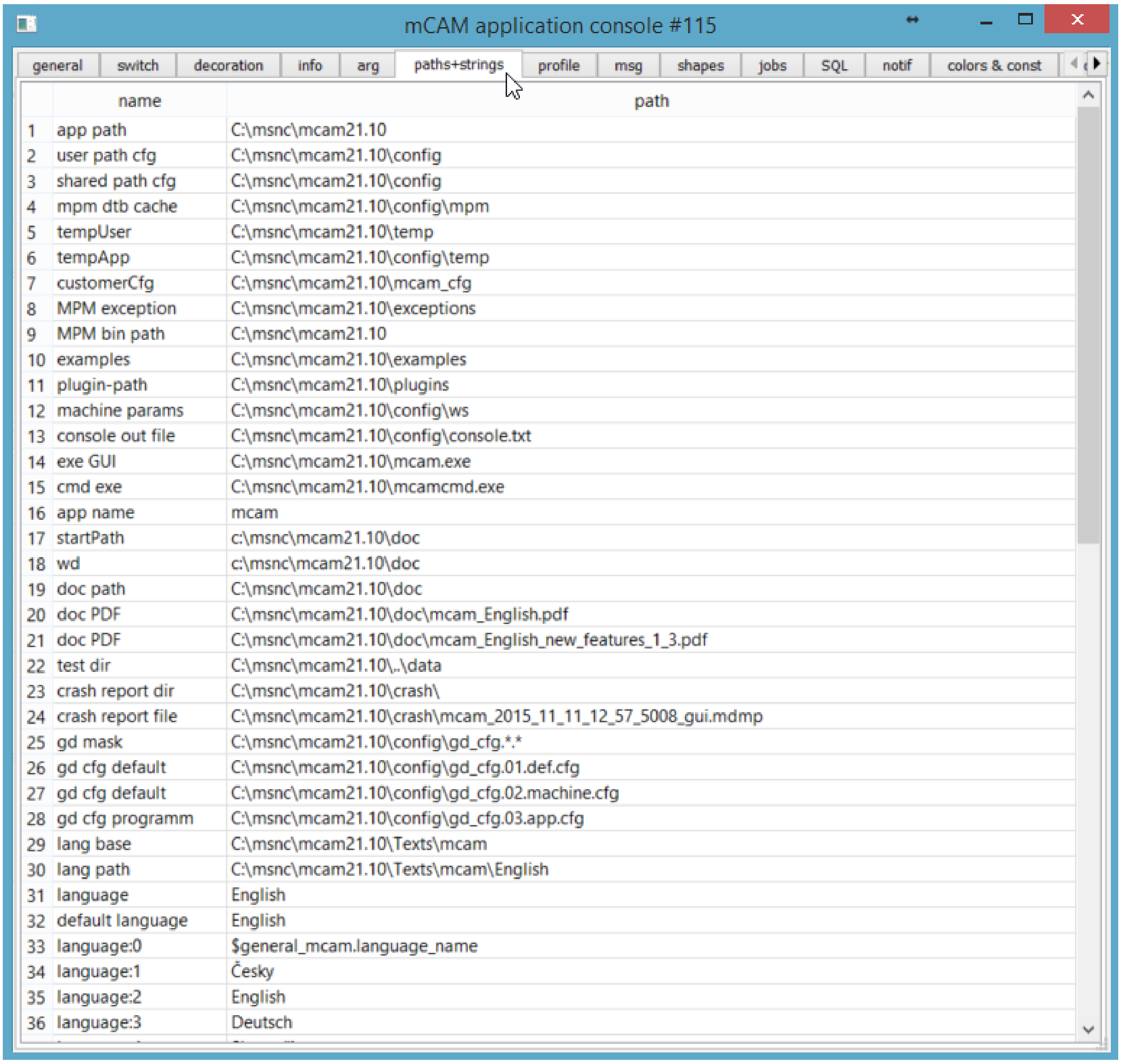
Console

This section contains commonly used options, commands and directions (keyboard shortcut: <F1>). Two main subsections of console are switch and paths+strings. Almost all sections included in the console are hard–coded, so it is not possible to change them. The most important and useful settings are presented below.

Paths and strings

This section displays paths and strings to important files, configurations, and other data that could be modified. It contains directories for these useful data:

- common program user data
- configurations and libraries
- *MPM* local databases
- temporary directories for traces
- temporary directory application
- import libraries
- reports
- executable file name
- documentation path
- files for all languages
- crash report directories



	name	path
1	app path	C:\msnc\mcam21.10
2	user path cfg	C:\msnc\mcam21.10\config
3	shared path cfg	C:\msnc\mcam21.10\config
4	mpm dtb cache	C:\msnc\mcam21.10\config\mpm
5	tempUser	C:\msnc\mcam21.10\temp
6	tempApp	C:\msnc\mcam21.10\config\temp
7	customerCfg	C:\msnc\mcam21.10\mcam_cfg
8	MPM exception	C:\msnc\mcam21.10\exceptions
9	MPM bin path	C:\msnc\mcam21.10
10	examples	C:\msnc\mcam21.10\examples
11	plugin-path	C:\msnc\mcam21.10\plugins
12	machine params	C:\msnc\mcam21.10\config\ws
13	console out file	C:\msnc\mcam21.10\config\console.txt
14	exe GUI	C:\msnc\mcam21.10\mcam.exe
15	cmd exe	C:\msnc\mcam21.10\mcamcmd.exe
16	app name	mcam
17	startPath	c:\msnc\mcam21.10\doc
18	wd	c:\msnc\mcam21.10\doc
19	doc path	C:\msnc\mcam21.10\doc
20	doc PDF	C:\msnc\mcam21.10\doc\mcam_English.pdf
21	doc PDF	C:\msnc\mcam21.10\doc\mcam_English_new_features_1_3.pdf
22	test dir	C:\msnc\mcam21.10\..\data
23	crash report dir	C:\msnc\mcam21.10\crash\
24	crash report file	C:\msnc\mcam21.10\crash\mcam_2015_11_11_12_57_5008_gui.mdmp
25	gd mask	C:\msnc\mcam21.10\config\gd_cfg.*.*
26	gd cfg default	C:\msnc\mcam21.10\config\gd_cfg.01.def.cfg
27	gd cfg default	C:\msnc\mcam21.10\config\gd_cfg.02.machine.cfg
28	gd cfg programm	C:\msnc\mcam21.10\config\gd_cfg.03.app.cfg
29	lang base	C:\msnc\mcam21.10\Texts\mcam
30	lang path	C:\msnc\mcam21.10\Texts\mcam\English
31	language	English
32	default language	English
33	language:0	$general_mcam.language_name
34	language:1	Česky
35	language:2	English
36	language:3	Deutsch

Fig. 348: Paths and strings

Shape geometry data

In order to understand process of shape type recognition, several functions need to be specified. Next sections describe several functions for rescanning shape type/ cutting faces or tool paths that are used when user deals with some problem or wants to apply changed presets. The *mCAM* also contains functions that helps user (or MicroStep technician/support) to determine possible problem.

Shape data rescanning

In–depth analysis of raw geometrical data *mCAM* is able to recognize shape type and process cutting faces automatically.

Rescan shape type

If the detection procedure fails or in cases of ambiguity, several possibilities of shape interpretation are offered for manual selection, i.e. ring shape can be created from a round pipe or as an annulus from a flat sheet.

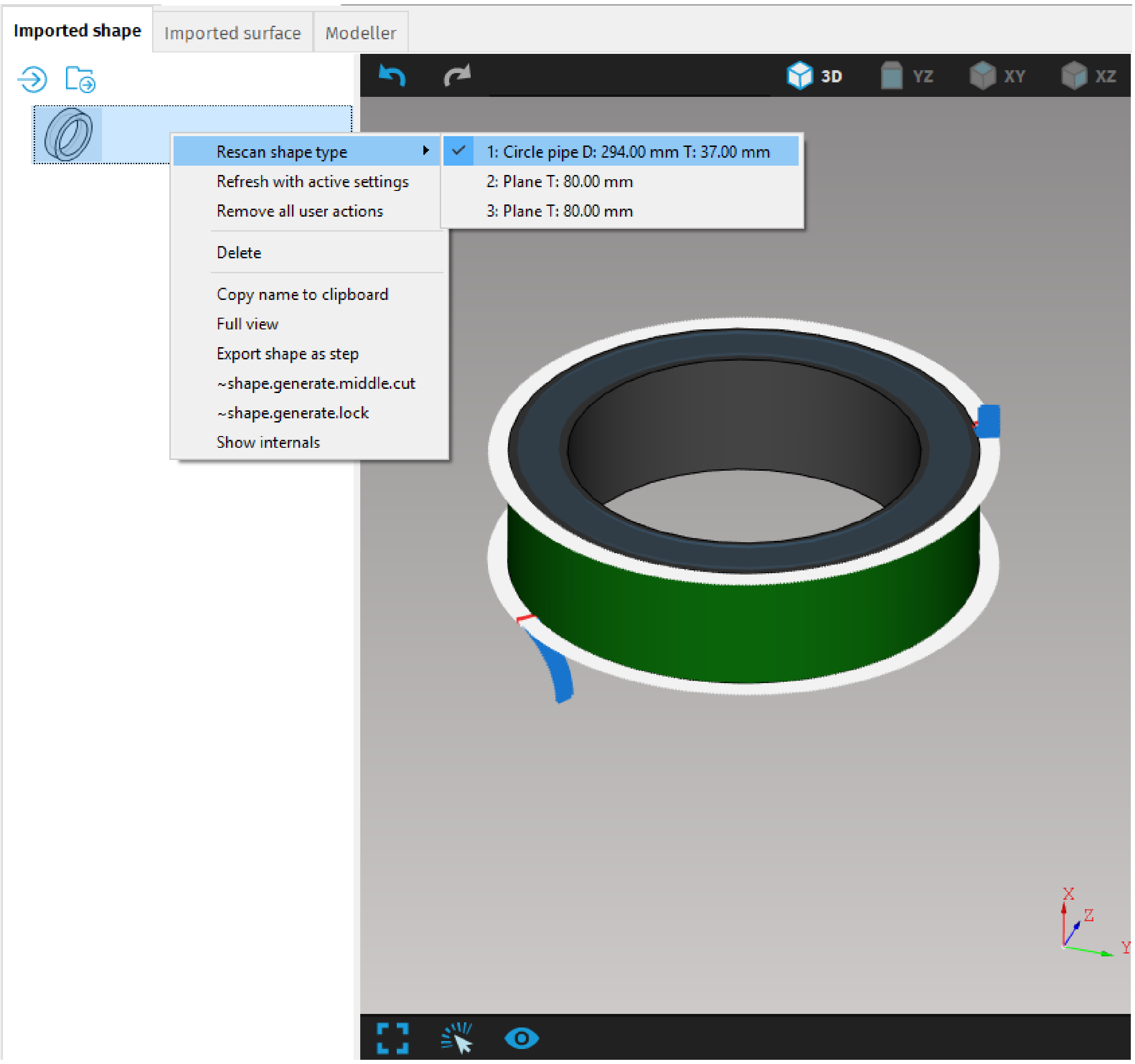


Fig. 349: Rescan shape type to Circle pipe

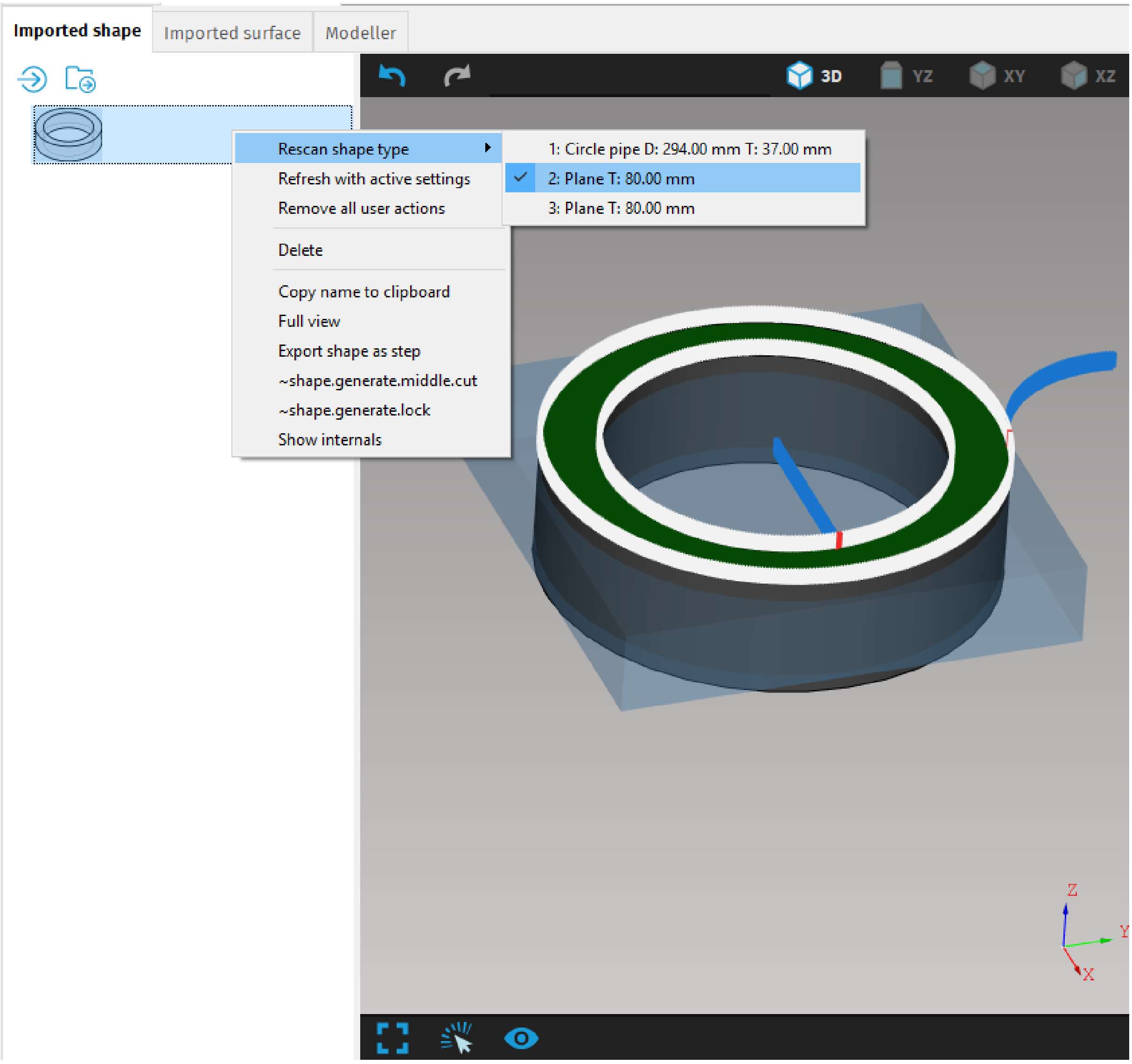


Fig. 350: Rescan shape type to Plane

Geometrical inspection

3D geometrical inspections allows to display a 3D CAD drawing with all geometry data of processed/unprocessed 3D solid. This function allows user to view tree structure of shape geometry, including recognized cutting faces and cutting paths (in *Full view*). Depending on function *mCAM* is able to view raw data of solid geometry in *Raw view* or processed solid geometry in *Full view.*

Full view

This option allows to display a detailed specification of selected part. Detailed specification includes shape information that have been processed by *mCAM* during the import of particular part. The function can be found in the list that is displayed by using right–

click on the part in the list of parts within any shape in *Import task* or in the list of parts in the *Job task*.

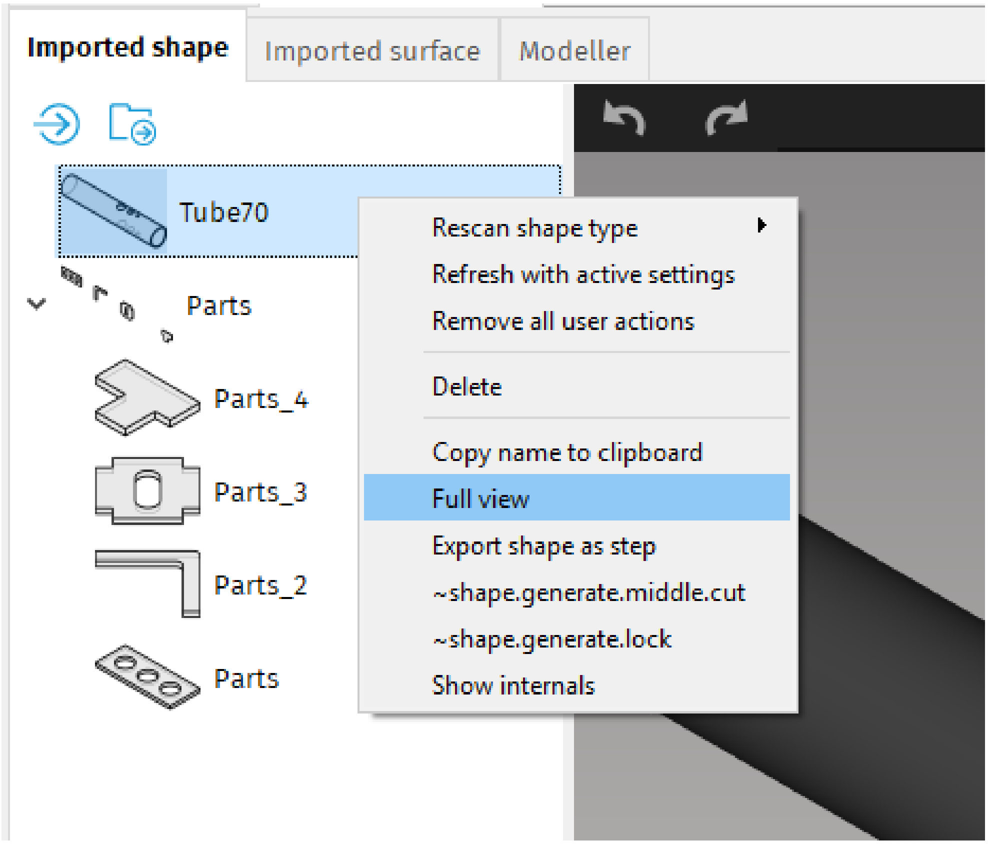


Fig. 351: Full view function

Full view displays detailed shape data processed by *mCAM*. All geometry data are structured by the definition of *B-rep model structure*. Shape data are displayed in tree structure that describes whole part. A solid is divided into five elements:

- shell
- faces
- wires
- edges
- vertexes

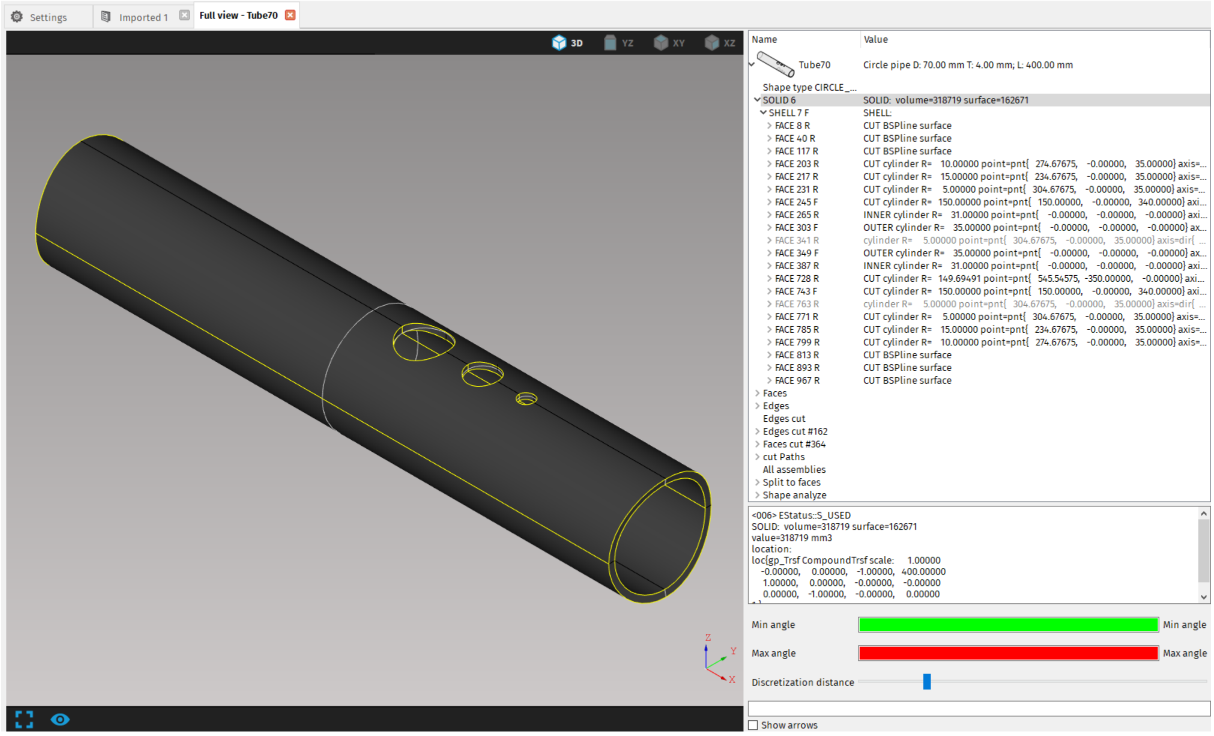


Fig. 352: Geometrical structure of 3D solid (B-rep model)

More information about the listed elements can be found in section Boundary representation. Full view displays shape data information divided to few groups according to their type:

- faces: inner, outer or cut faces
- edges: smooth (also known as auxiliary edges) or outer
- edges cut
- faces cut
- cutting paths

Face-cut recognition directly affects the way and quality of cutting path processing. It is important especially in features and cuts with weld – edge preparation where complex cuts (Y, K, X-cuts) have to be automatically recognized. Therefore, it is important to have those cut – faces identified and processed correctly. For debugging those kind of problems face-cuts need to be visually analysed.

It is important to check whether cutting edges were processed correctly:

- top edge of cutting contour (line on the top of the material) has to be highlighted by yellow color
- bottom edge of cutting contour (line on the bottom of the material) has to be highlighted by light – blue color
- starting edge of cutting contour has to be highlighted by green color
- ending edge of cutting contour has to be highlighted by purple color

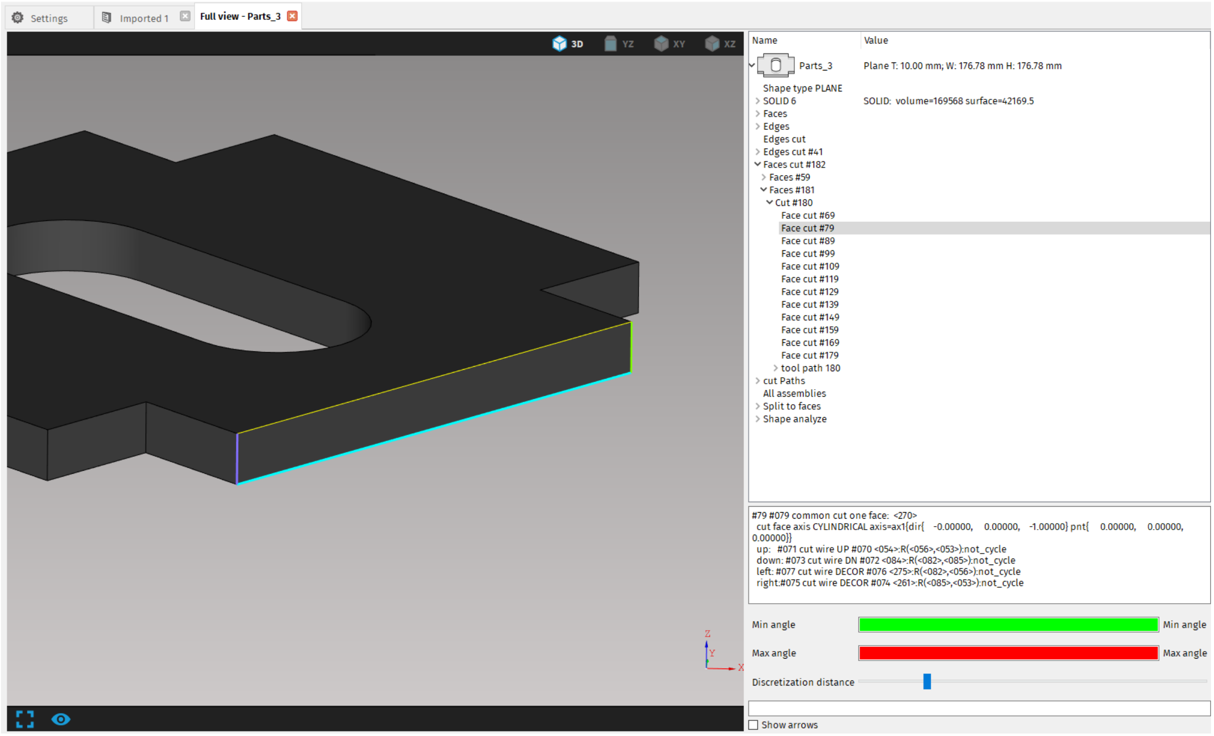


Fig. 353: Face – cut recognition and processing

Note: In case that some cut is not recognized correctly (i.e. cutting – points do not copy the face – cut) or not recognized at all (complex cuts like Y, K or X – cut) the only way to find out the problem is to go through geometry inspection.

Raw view

This option displays a raw view i.e. unprocessed CAD drawing of *mCAM* file. All info and data that are defined in the imported file (STEP, IGES) with all geometrical data are displayed in a tree structure of the file (on the right side of the screen). *Raw view* of any 3D shape can be displayed by using function in *File – Raw* and selection of particular part (key shortcut <Ctrl + S>).

The raw view is useful in cases when the file was not processed correctly and a user wants to check whether the model, complete info, geometry, dimensions, and data from the file are properly and correctly defined.

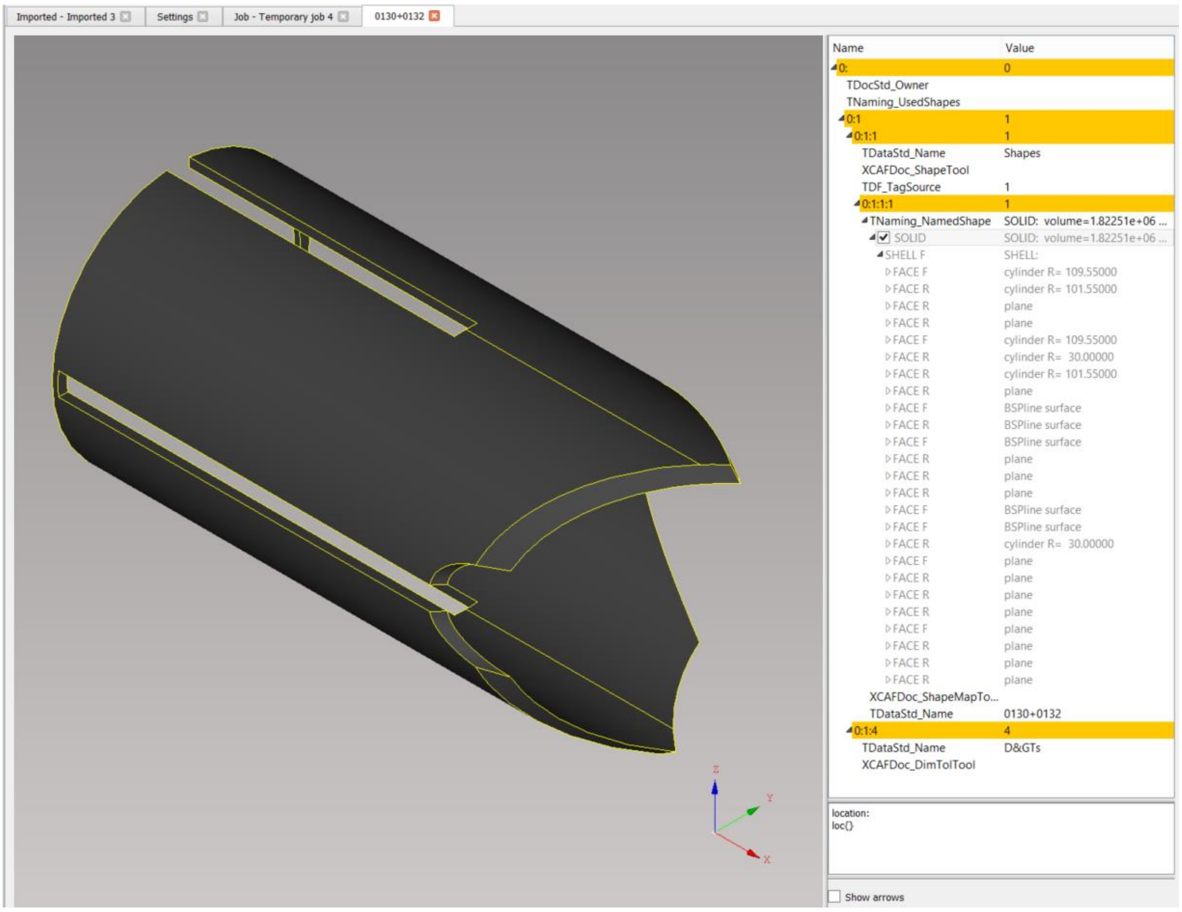


Fig. 354: Raw geometrical shape data – Raw view

Reporting

This option displays the dialogue where user can describe any issue or problem with *mCAM*, that he encounters. There is also possibility to add user's email address, error description, problematic shapes, select data that could be useful for the solving of the problem (see) or add any files that are considered to be helpful for software development. It is possible to load or save the report. The report has to be sent directly to the *mCAM* support team who will try to solve the problem as soon as possible.

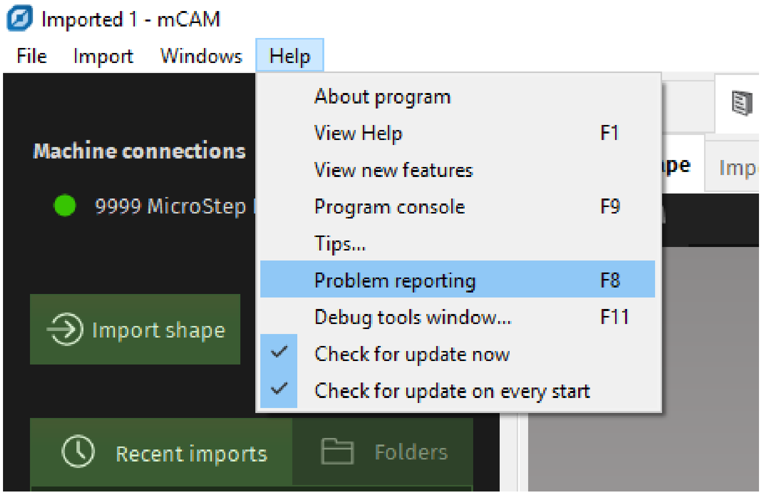


Fig. 355: Open Reporting dialogue window

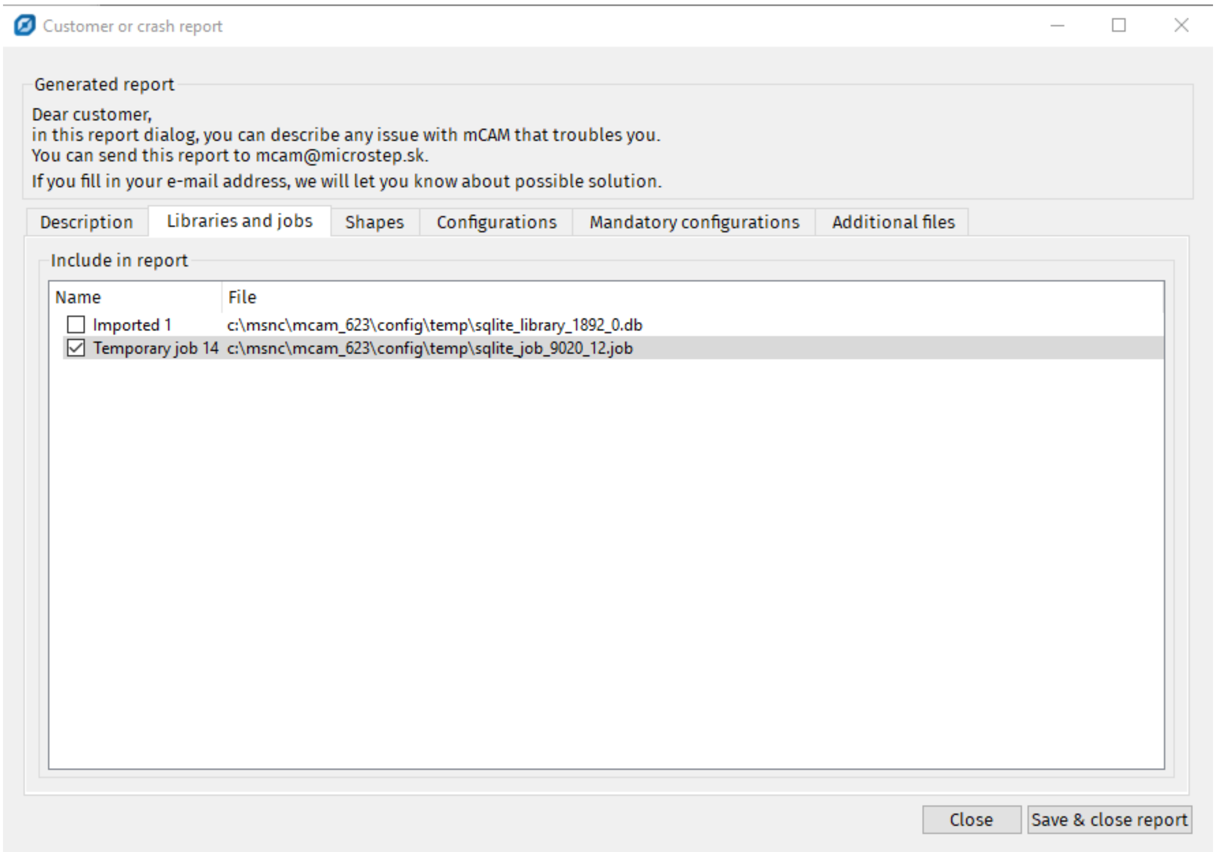


Fig. 356: Saving of .zip report with attached job task and generated CNC program

Working with MPM

Integration the *mCAM* with the *MPM module* that manages parts, semi–products and orders, afford to create cutting plans, manage production requirements and monitor the status of orders effectively in real–time.

MicroStep Production Management module (*MPM*) is an application of a SQL database with an intranet interface that allows to access to data to all authorized users of a company network via any internet browser. This *module* serves to manage all materials, stock material, parts, products, orders, cutting plans, and reports data.

To activate MPM module within mCAM, it has to be first enabled in Settings -> Application by checking the checkbox beside *"Enable access to MPM"* which will provide access to MPM globally for mCAM. mCAM has to be restarted afterwards. MPM is then activated or deactivated for each machine in mCAM separately by manually rewriting the corresponding cell in Web Service table in Settings->Application to enabled/disabled. Alternatively, the corresponding number can be written in the cell – 0 for disabled and 1 for enabled.

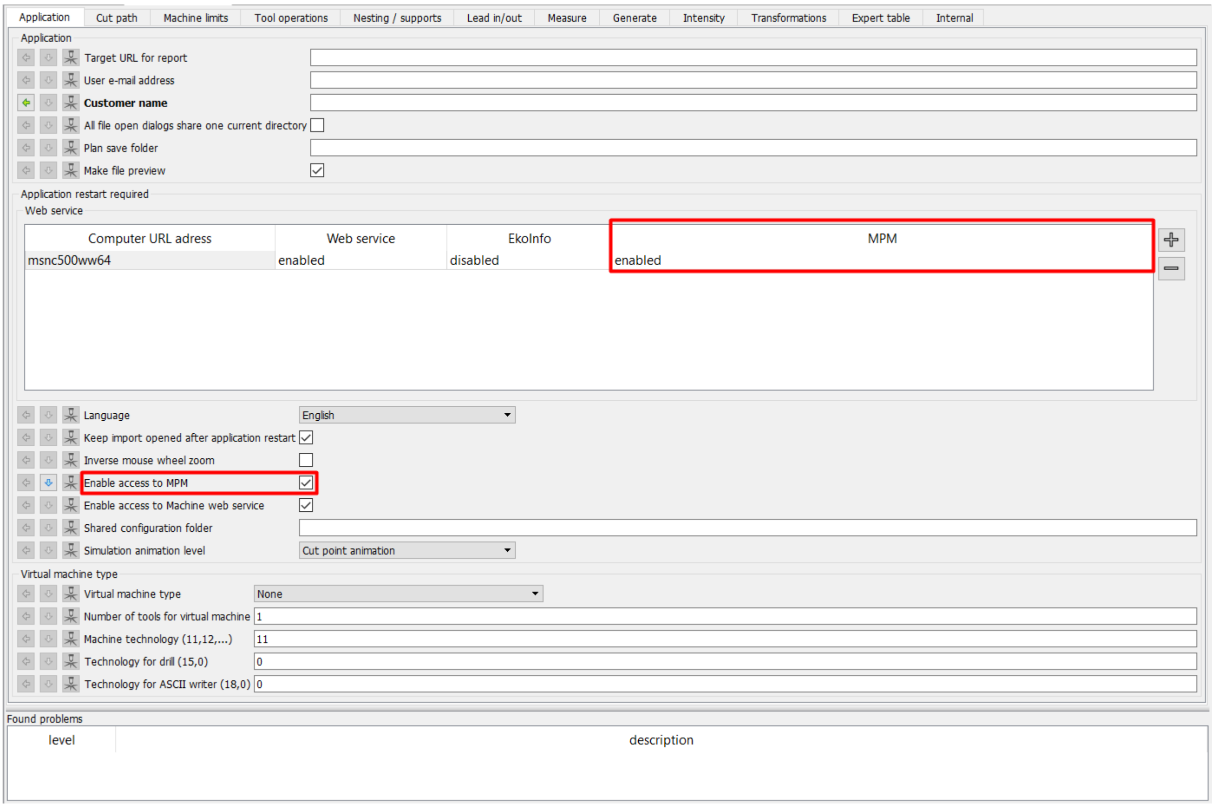


Fig. 357: MPM

Standard working procedure

The procedure of working with *mCAM* integrated with *MPM module* can be defined in several steps:

1. Load parts into *mCAM*
2. Import loaded parts to *MPM* through *mCAM*
3. Create semi–product in *MPM*
4. Create *MPM job* in *mCAM*
5. Add semi–product into job task from *MPM*
6. Add parts into job task from *MPM*
7. Assign parts to semi–product and nest them
8. Generate cutting plan

Loading parts into mCAM

Loading 3D or su rface parts into *mCAM* while working with *MPM module* is exactly the same as without *MPM module*. Loading parts into *mCAM* is performed by the function *Import* or *Import surface*. Importing procedure is described in more details in section Importing.

Importing of loaded parts into MPM

Importing of loaded parts into *MPM* is executed by the function Import all to *MPM* or by dragging the selected parts into the *MPM* folder in *Master panel* (function Drag and drop).

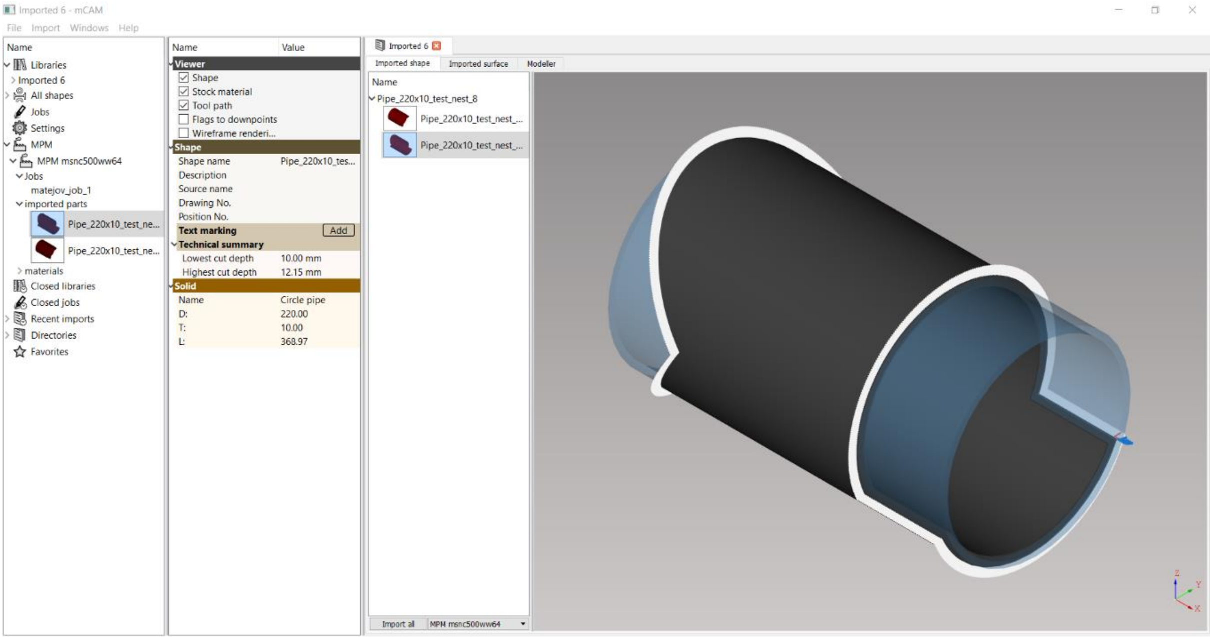


Fig. 358: Import of selected parts in the MPM database

Parts that have been imported in the *MPM* database are displayed in *Master panel* in *MPM* folder. Imported parts also appear in *MPM* intranet database where they can be displayed using the filter.

Fig. 359: Imported parts in MPM database

Creation of semi–product in MPM

To create a cutting program with real semi–product, it has to be created in *MPM module*. Afterwards, the created semi–product can be used to nest the parts and for cutting.

The semi–product can be added into the *MPM database* using function *Add* on the bottom of the window. Parameters that define the semi–product are as follows:

- name (material)
- thickness
- size
- specification (optional)

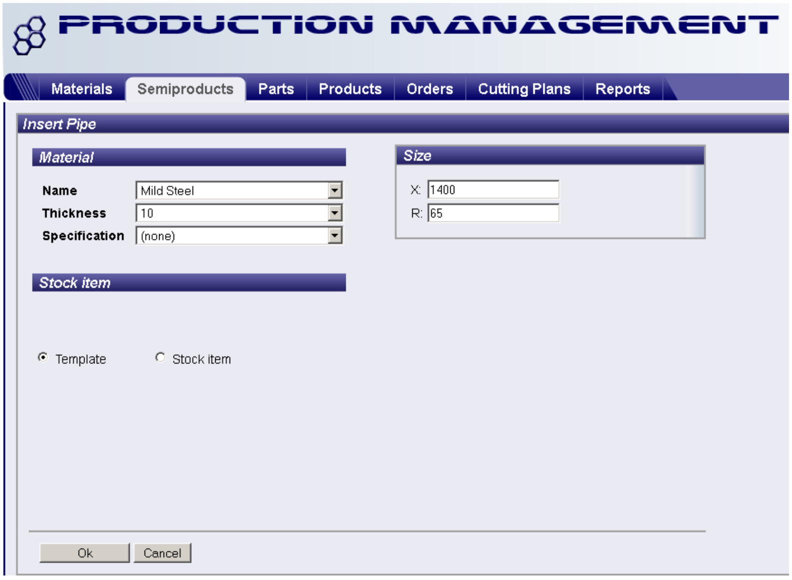


Fig. 360: Creation of the semi-product with specified size and material

Creation of MPM job in mCAM

A new *MPM* job can be created using the function Open *MPM* job. In next dialogue window user defines the type of semi–product (circular pipe, rectangular pipe, etc.) and its name.

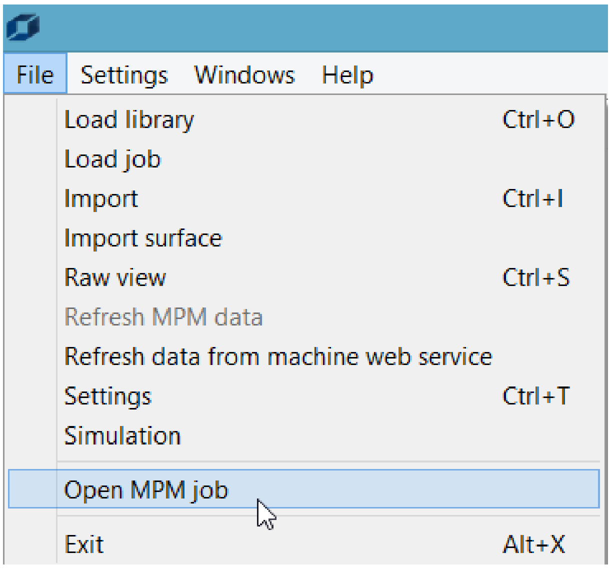


Fig. 361: Create MPM job task

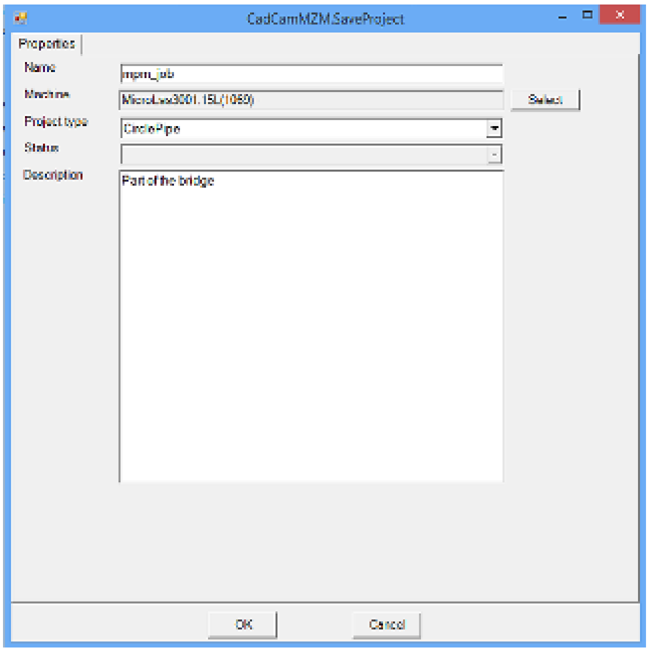


Fig. 362: Creation of MPM job

Adding semi–products from MPM database

The next step is to add defined semi–product from *MPM* database into Job task in *mCAM*. To add a semi–product with defined dimensions and material type from *MPM* database, use the function Add stock from *MPM* in Job task that appears by using right–click in the *Working tree.*

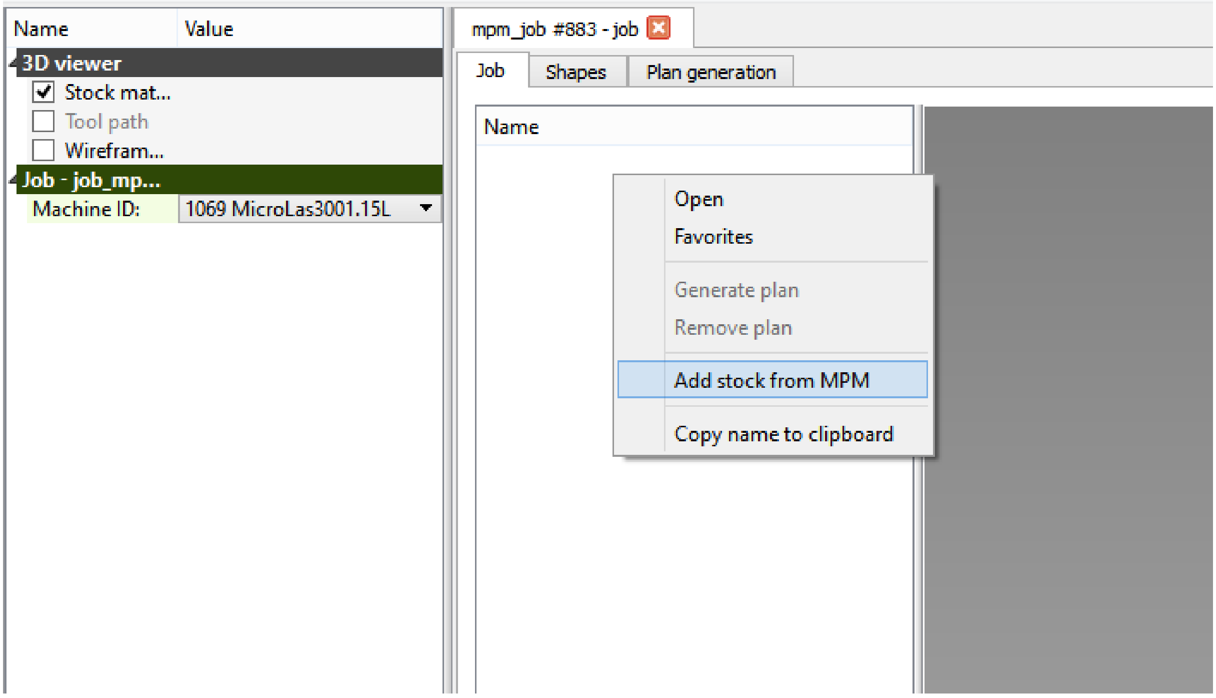


Fig. 363: Add stock from MPM function

The function opens the *MPM* database with all semi–products and by using the filter it is possible to find specified semi–product. Specified semi–product can be added to the *MPM* job by simple selection and the button OK.

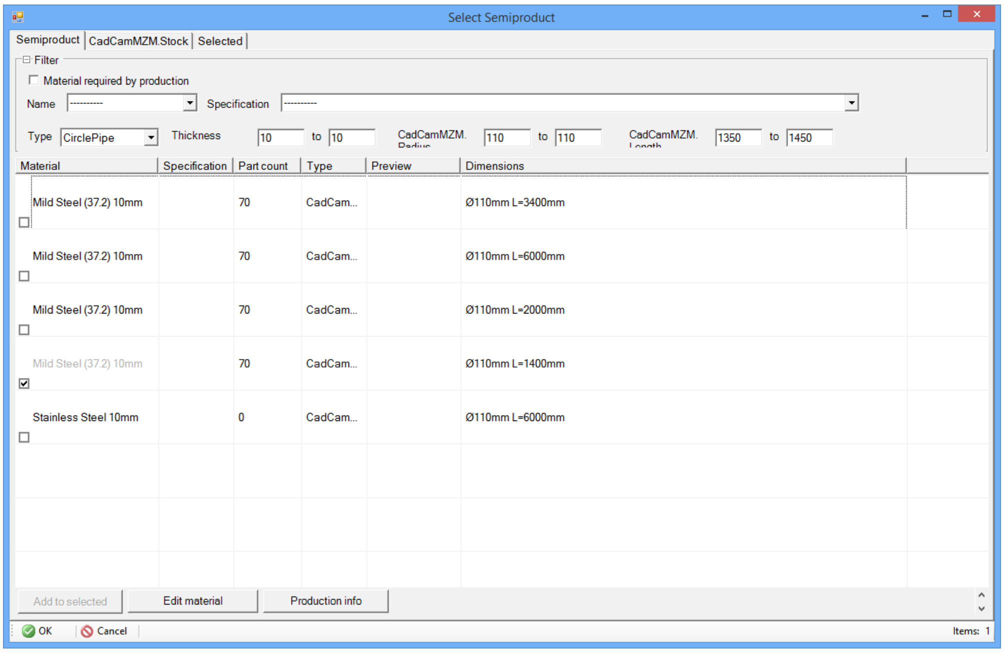


Fig. 364:Adding semi-product from MPM database to mCAM

Adding parts to mCAM from MPM database

To add specified parts to *MPM* job in *mCAM* the particular parts from *MPM* database should be selected by using the function Add part from *MPM*. The function is displayed by right–click on the semi–product (stock) which have been added to the job in previous step. To add parts that have not been added into some Order in the *MPM module*, the tab Template should be opened. In case of parts that have been included in some orders, the Production tab should be opened. Both tabs, Template and Production, contain the filter for more effective searching of specific parts.

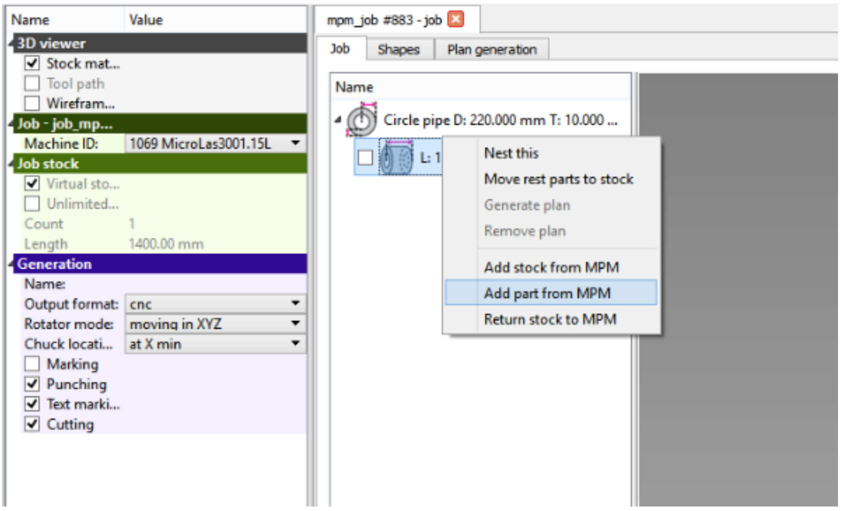


Fig. 365: Add parts from MPM database

1

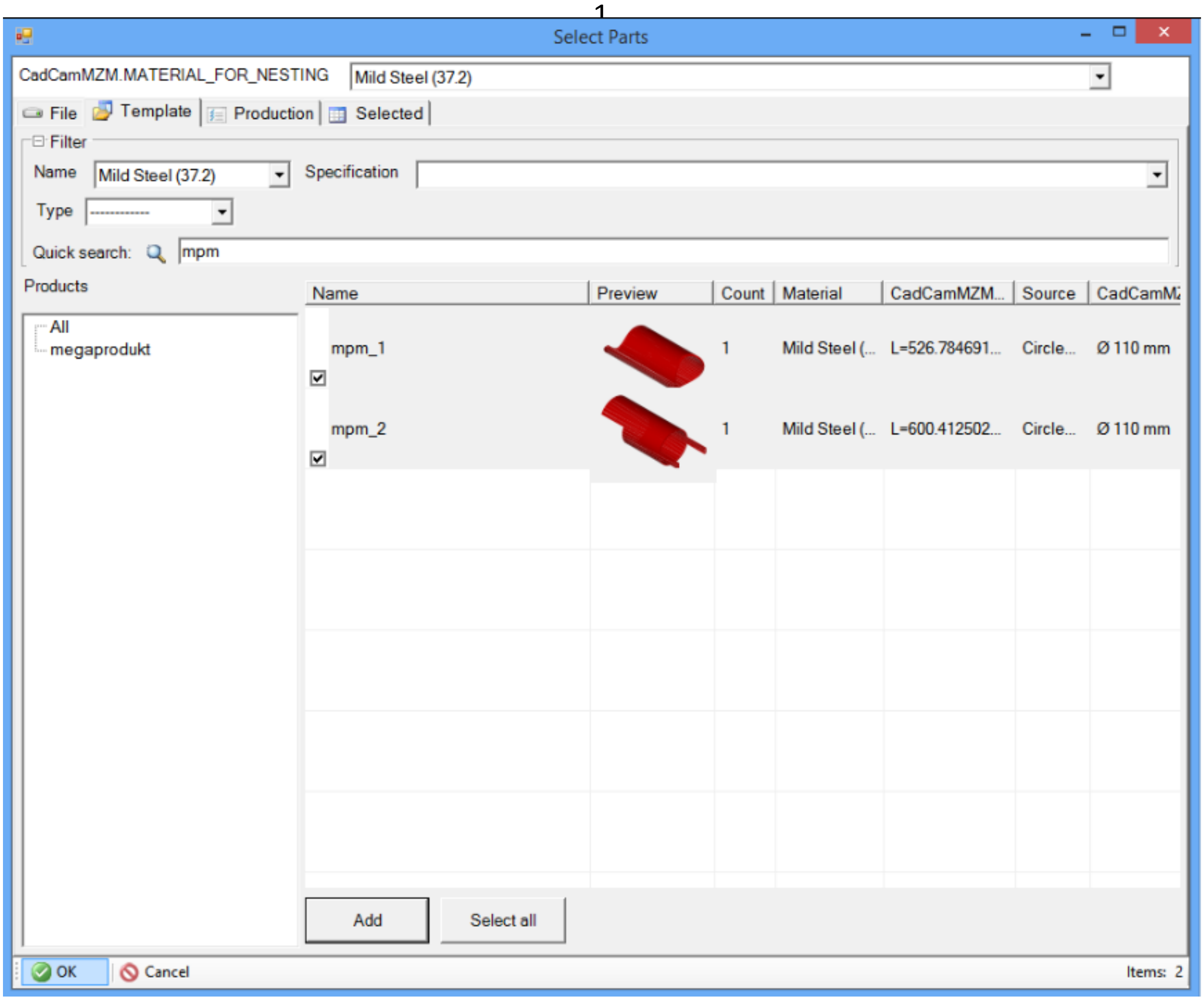


Fig. 366: Adding selected parts to MPM job task

Generating a cutting plan

A cutting plan within the work with *MPM module* can be generated in exactly the same way as the cutting plan that is generated without the connection to the *MPM module*. That means that the cutting plan can be generated for whole template (including all stocks in its structure) or for particular stocks (semi–products). The process of cutting plan generation is described in more details in section Simulation of CNC program and graphical visualization.

After generating the cutting plan (CNC code), a report is displayed in the visualization screen. The report contains basic information about particular cutting plan such as: name, date, time of generation, type and size of semi–product, chuck location and information about all parts that are included in the cutting plan.

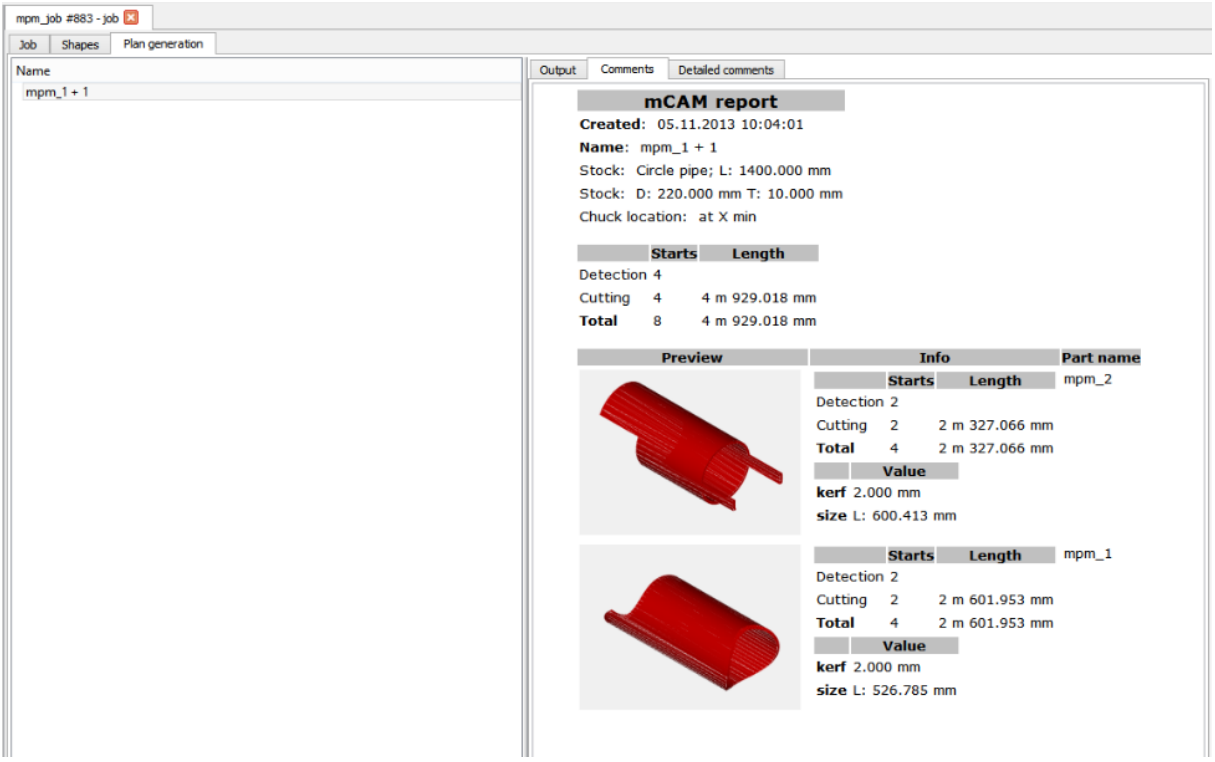


Fig. 367:Report of generated cutting plan

Note: After generation of cutting plan, it is automatically sent to *MRP module* (module of cutting plans). This module works as a local database of all generated cutting plans. Cutting machines are directly connected to this database, so the *MRP* works as a kind of a source. In case that particular CNC code (cutting plan) has not been used for cutting (no machine has taken it) yet, the cutting plan can be modified in *mCAM* and re-generated. Otherwise, if CNC has been already taken by some CNC machine, CNC code (cutting plan) cannot be modified and therefore it cannot be re-generated. If a cutting plan has been already taken from MRP to some machine and a user tries to edit that particular cutting plan in *mCAM*, *mCAM* displays the report that permission to modify the cutting plan is denied.

Supporting documents

Petal cutting workflow in mCAM

Introduction

MicroStep machines have all the means for precise cutting of dome petals from semi-product. Dome petals are prefabricated from pressed sheets of material and cut into their final form, with the use of a petal cutting process on a MicroStep machine. This document represents the recommendations during the mLIVE program preparation.

Dome petals shapes

Dome petals are divided into 3 categories based on their shape and geometrical parts:

- **spherical petals** (single radius -> spherical surface)
- **torispherical petals** (dual radius -> spherical, toroidal, cylindrical surface)
- **conical petals** **type 1** - cylindrical surface + radius + spherical surface
  type 2 - conical surface
  type 3 - conical surface + radius + cylindrical surface
  type 4 - cylindrical surface + radius + conical surface + radius + cylindrical surface

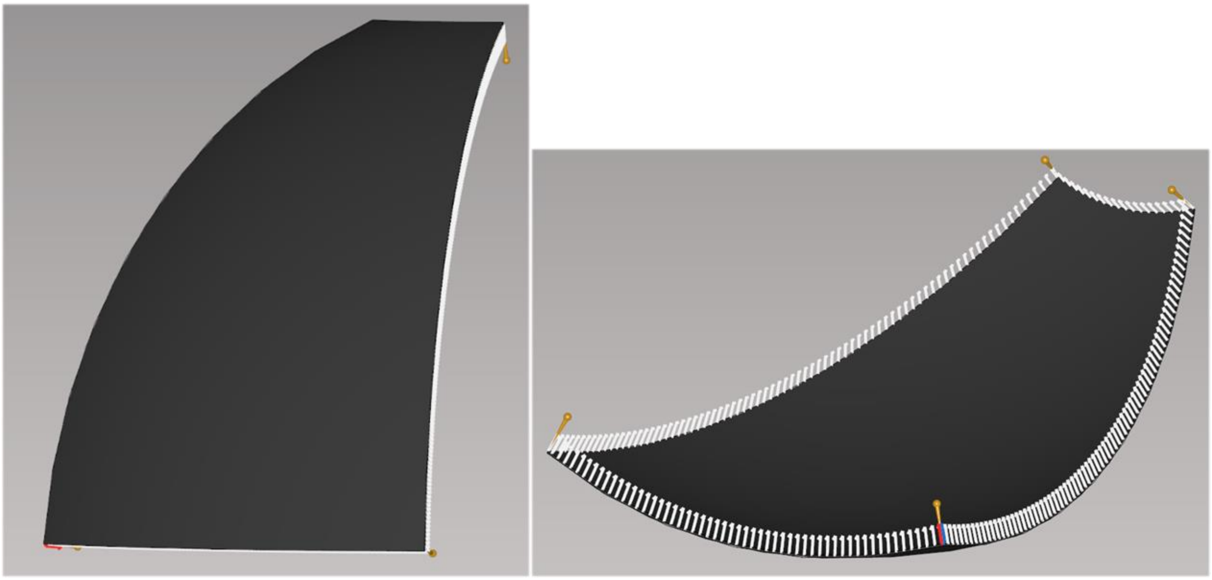


Fig. 368: Spherical dome petal

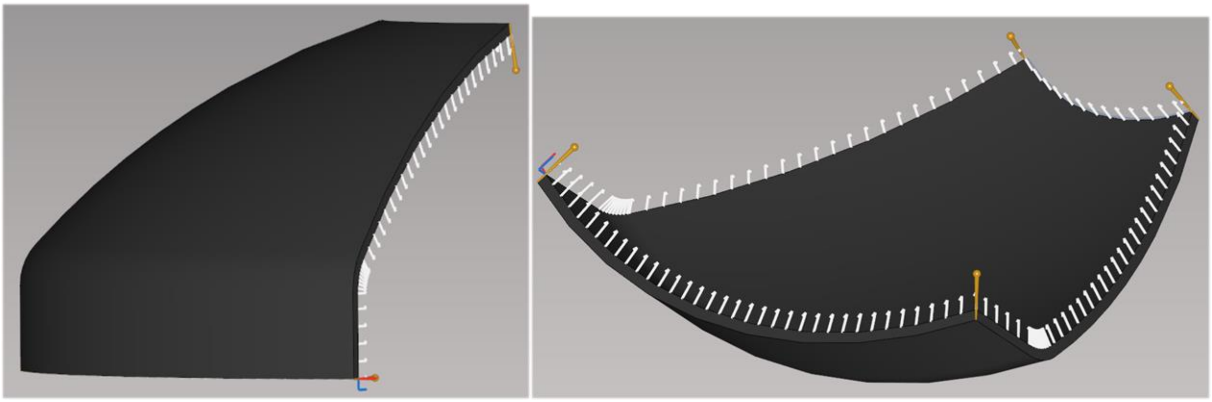


Fig. 369: Torispherical dome petal

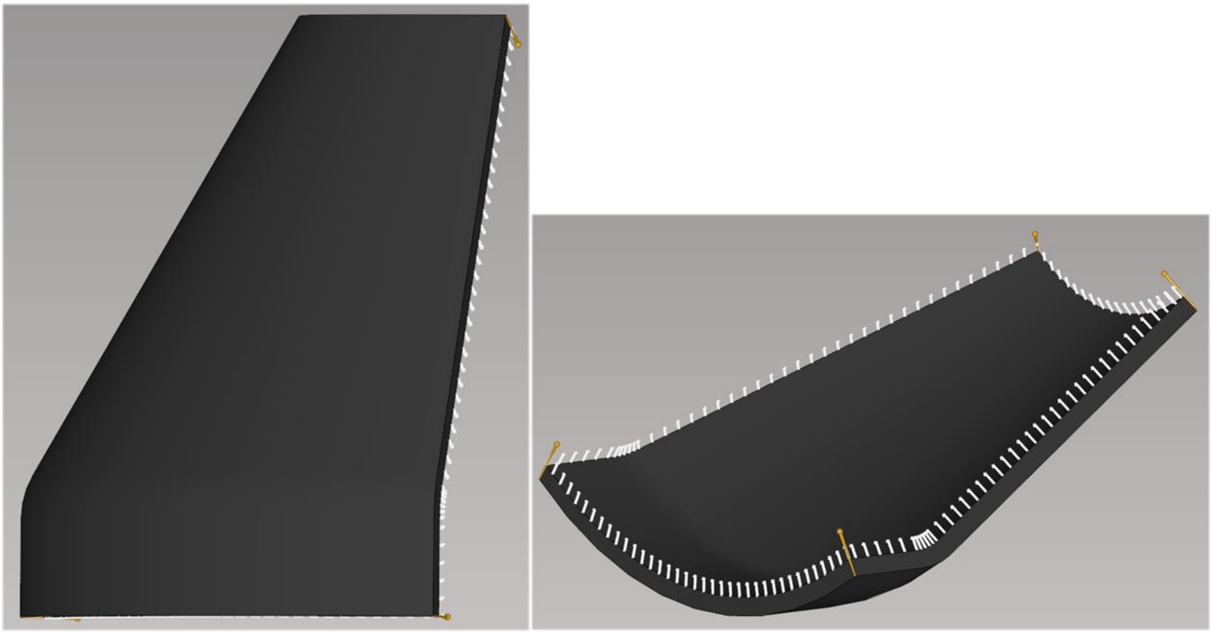


Fig. 370: Conical dome petal type 1

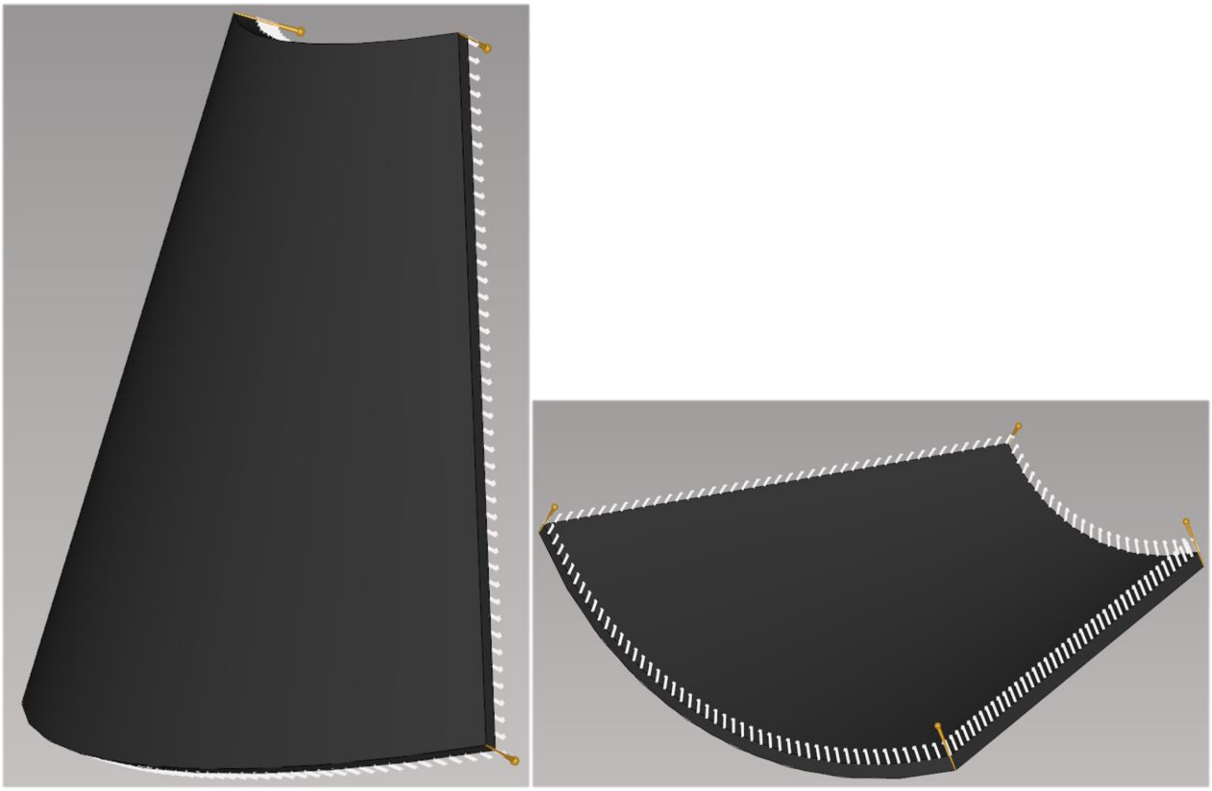


Fig. 371: Conical dome petal type 2

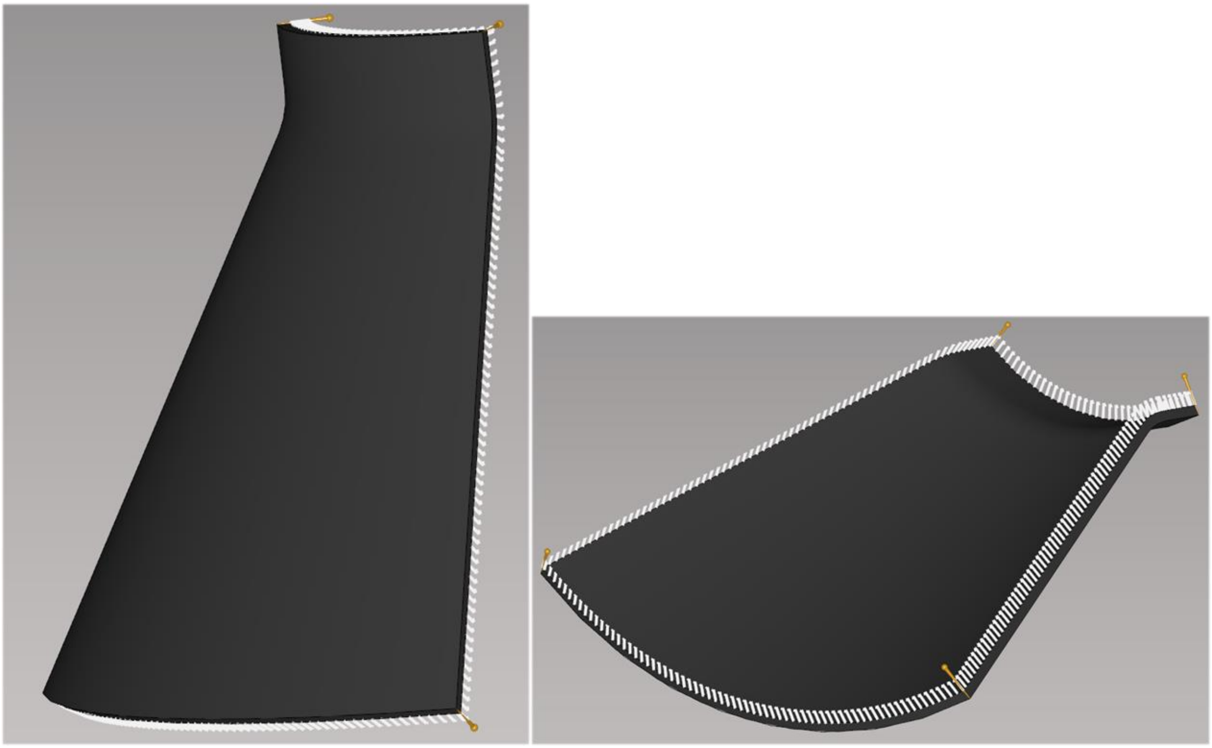


Fig. 372: Conical dome petal type 3

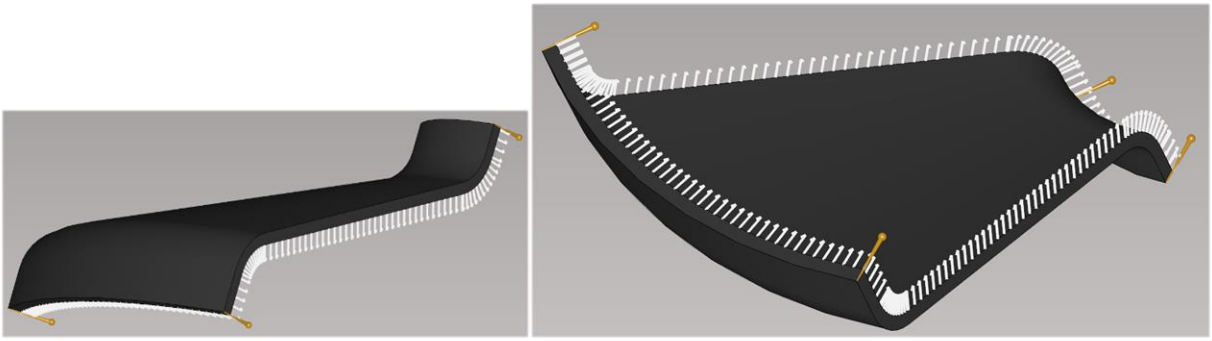


Fig. 373: Conical dome petal type 4

Importing dome petals in the mCAM program

__Import shape in mCAM__

To import the desired petal shape in a suitable format (*.step / *.stp) it is possible to use the **Import shape** button on the left side of the main mCAM screen. Pressing the button will allow you to browse computer files and choose, which should be loaded in mCAM.

Alternatively, the user can choose **File -> Import shape** or press the shortcut key **Ctrl + I.**

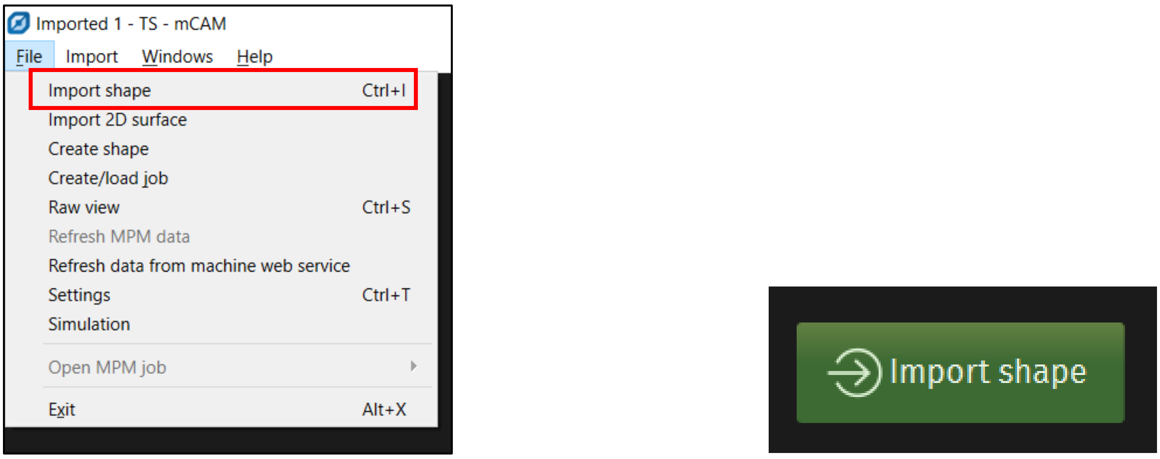


Fig. 374: Import shape in mCAM

Cutting paths of the dome petal

Cutting of the petal itself can be done either by:

- using a single cut (lead in -> cut -> lead out)
- split the cutting path into multiple paths, which is a recommended option

To split the cutting path simply click the Tool path area on the petal (displayed on the edge of the material) and in the **Split cut paths** box choose **Standard.** After splitting the cut paths user is able to highlight each path separately.

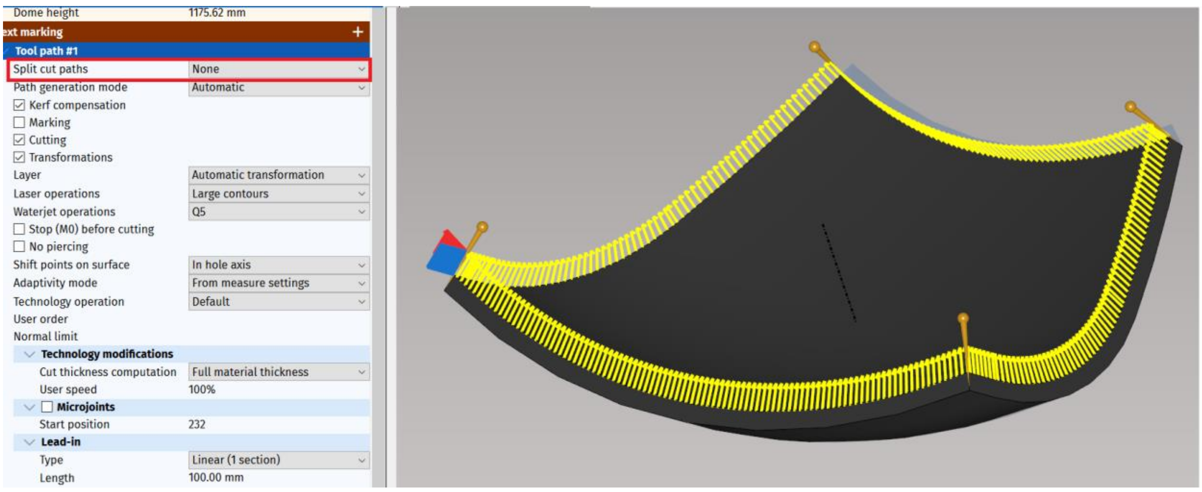


Fig. 375: Cut paths before splitting in mCAM

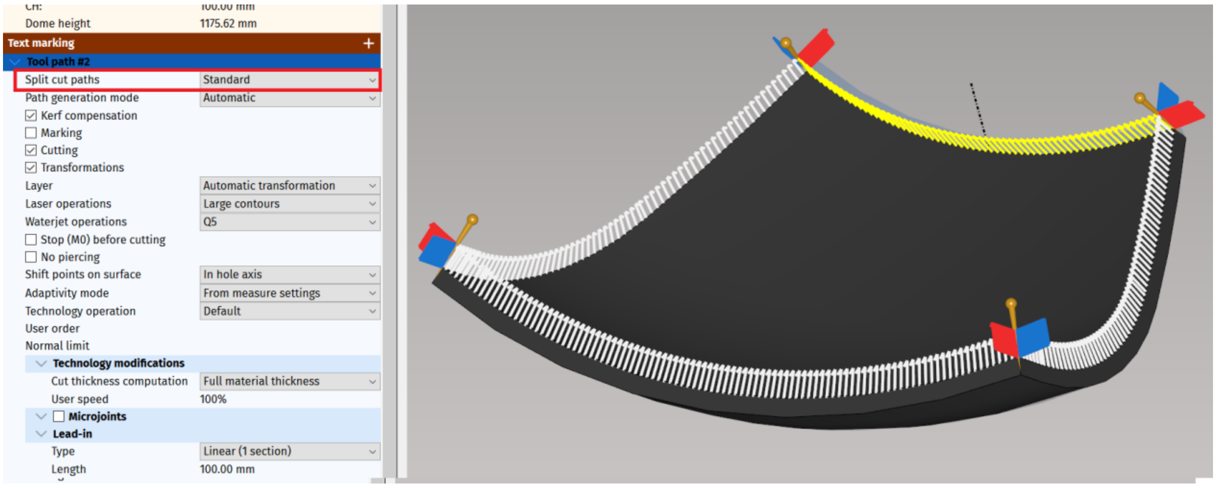


Fig. 376: Cut paths after splitting in mCAM

For each cut paths, it is recommended to set lead in and lead out. It is even desired to set larger lead-ins/outs (relative to the size of the material) to ensure that the cutting path is complete.

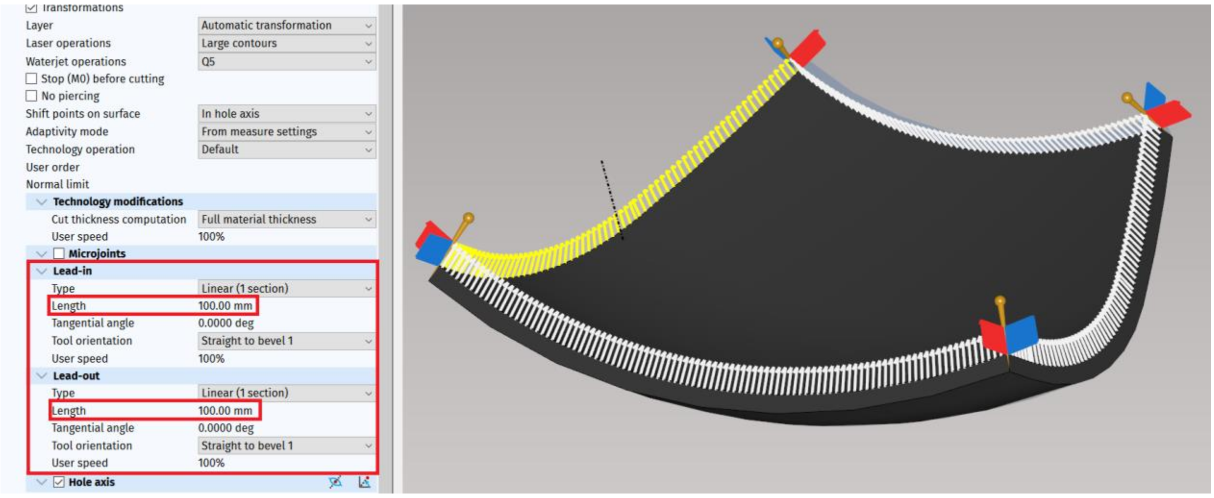


Fig. 377: Setting the lead ins and lead outs for each cutting path in mCAM

MicroStep recommends choosing "**Linear (1 section)**" or "**Path conforming**" for lead-in and lead-out types.

Order of cuts

Side cuts are usually performed first. To set the order, click on the specific cutting path and enter a number in the **User order box** in the Tool path category (for example numbers 1-4 in case the path will be split into 4 different cutting paths).

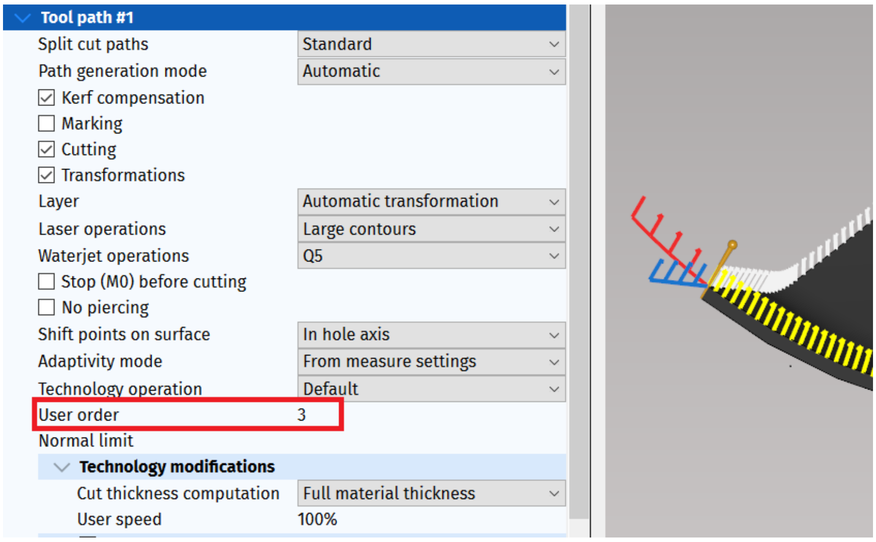


Fig. 378: Setting the cutting order for each cutting path in mCAM

In case when the cutting of a specific side is not required and should be skipped, simply highlight that path by mouse-click and delete it by clicking the trash bin located in the panel on the right side of the mCAM screen.

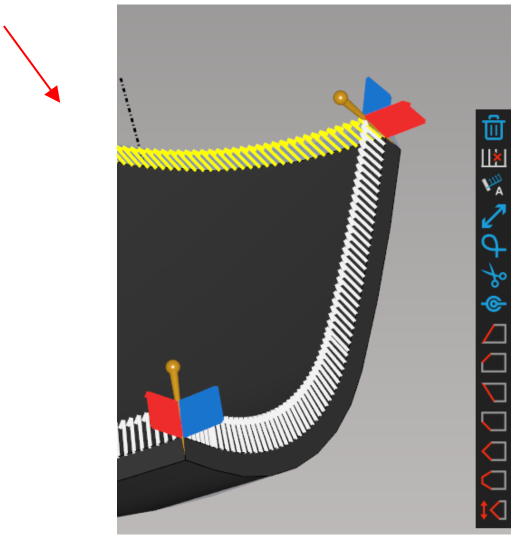


Fig. 379: Deleting the selected cutting path in mCAM

Marking

If user wants to take advantage of the marking function before cutting, it can be done by checking the box **Marking** in Tool path settings for each cutting path separately.

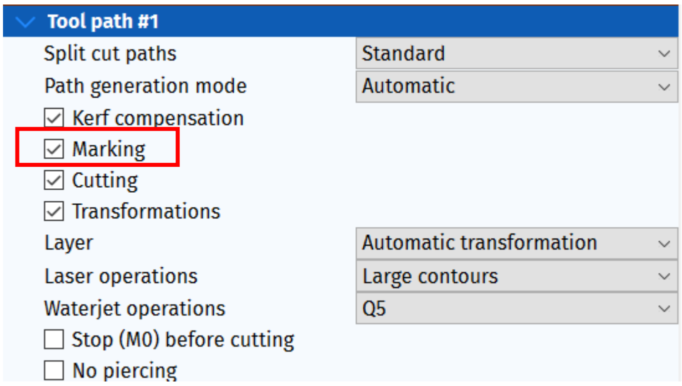


Fig. 380: Marking checkbox in mCAM

To ensure that the machine will stop before cutting, after the marking has been done, check "**Stop before cutting"** box in the Tool path options. Otherwise, the machine will perform the cut right after the marking.

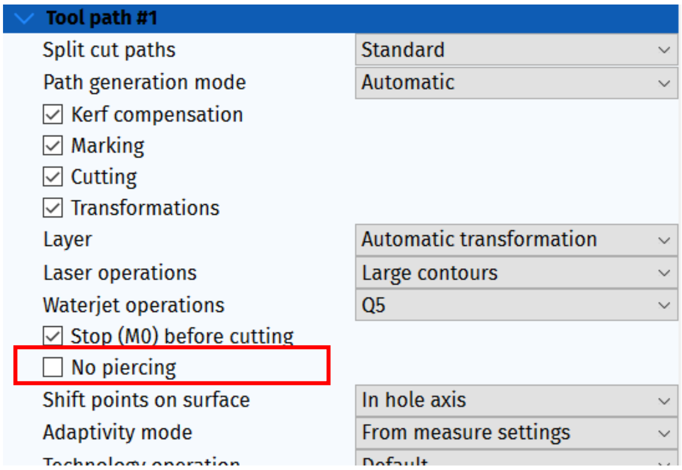


Fig. 381: Stop before cutting the checkbox in mCAM

There is a possibility to select more than one cutting path for the settings of cutting paths purposes (e.g., settings of lead-ins and lead-outs, marking, stop after marking function, etc.). To choose multiple cutting paths hold SHIFT and select all desired cutting paths. Performed settings will apply to all the selected paths.

Path generation mode

For **Path generation mode** settings MicroStep recommends:

- In case of a **single cutting path** -> Linear partial mapping
- In case of **multiple cutting paths** -> Min distance

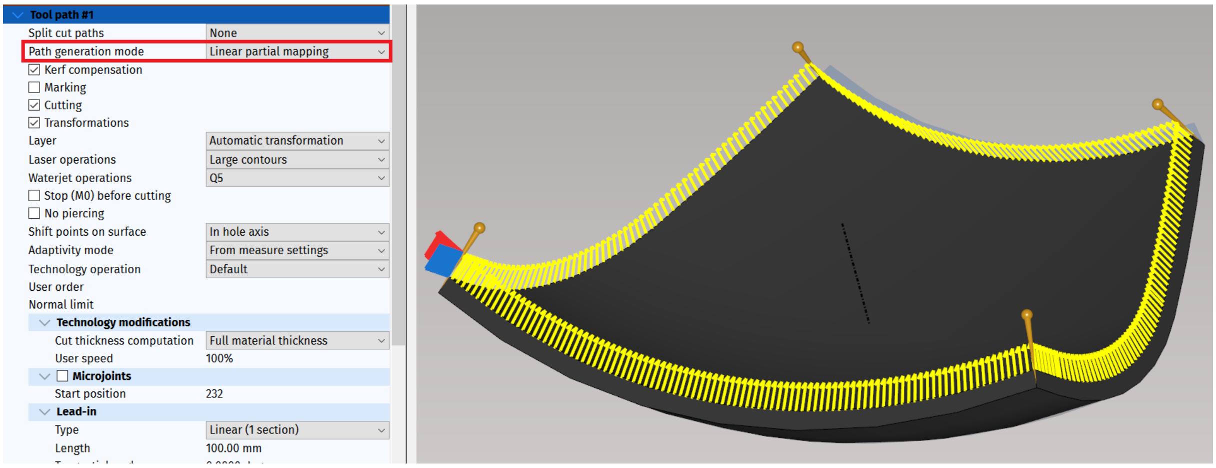


Fig. 382: Single cutting path

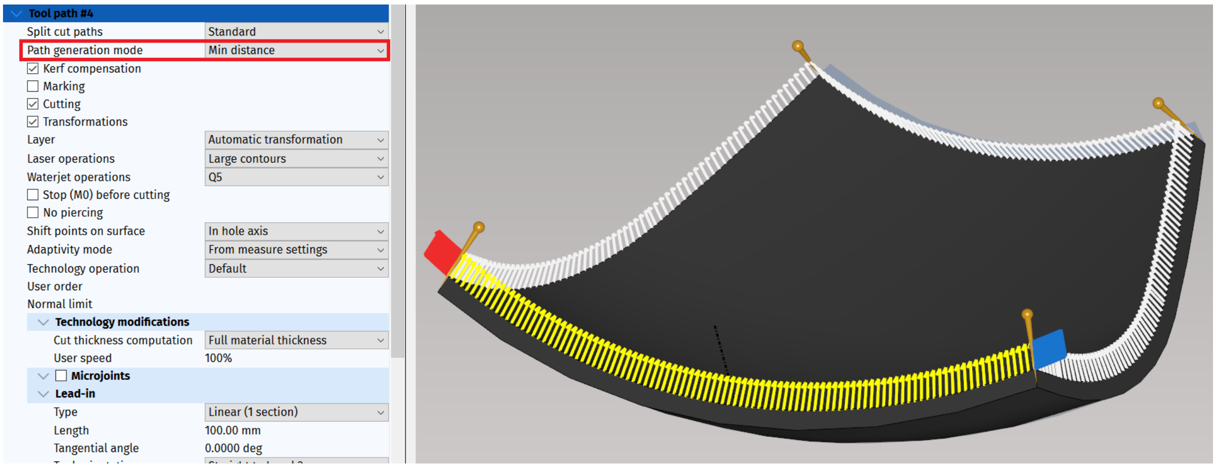


Fig. 383: Multiple split cutting paths

Shift points on the surface

Due to the precise projection of the cutting path on the surface of the scanned semi-product, the user is prompted to choose the method for “Shift points on surface”. This means the way how is the direction of the cutting path shifted on the scanned material. MicroStep recommends using one of these options:

- **In hole axis**

In hole axis method means, that each one of the cutting points shifts in the exact same direction depending on the hole axis.

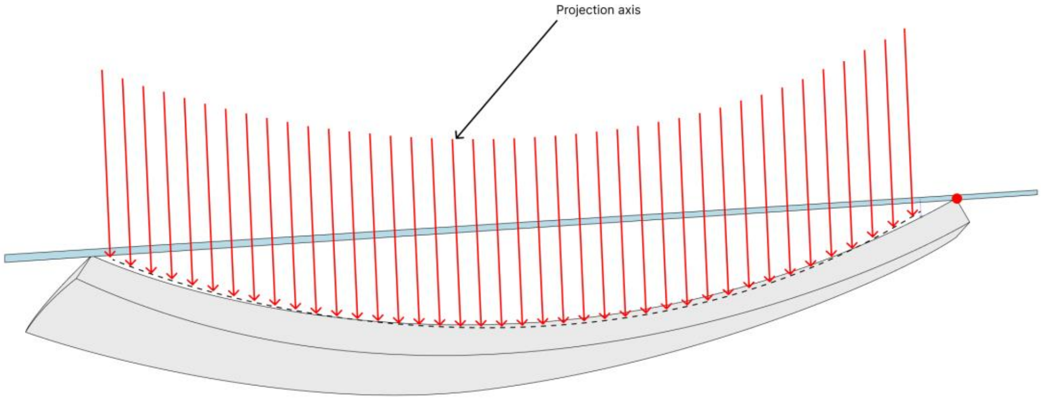


Fig. 384: Shift points on surface – in hole axis type

To select this method, click on the desired path and choose “In hole axis” in Shift points on surface area in the Tool path options.

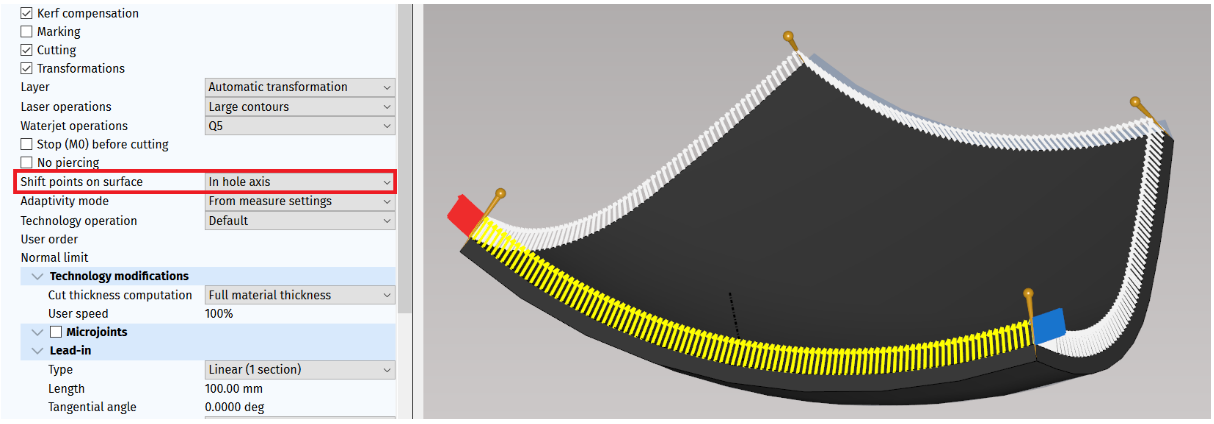


Fig. 385: Shift points on surface – in hole axis type in mCAM

- **In cutting direction**

In this method each one of the cutting points shifts in a defined direction -> perpendicular to the material.

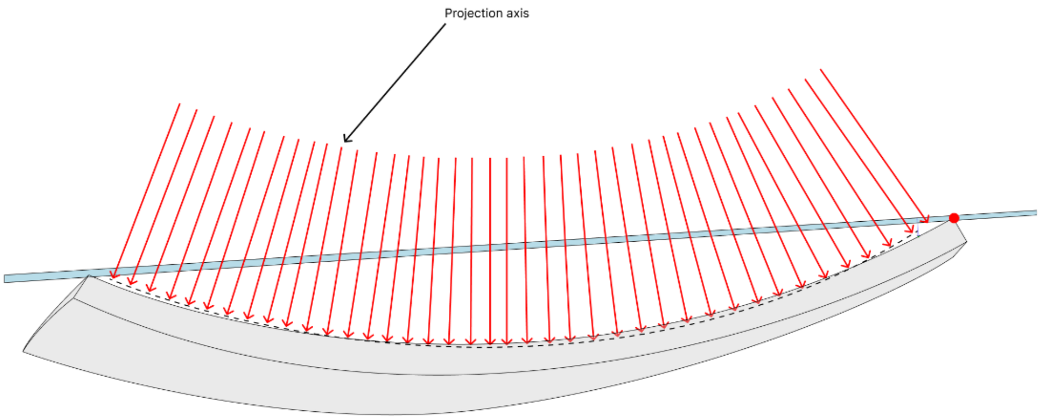


Fig. 386: Shift points on surface – in cutting direction type

To select this method, click on the desired path and choose “In cutting direction” in Shift points on surface area in the Tool path options.

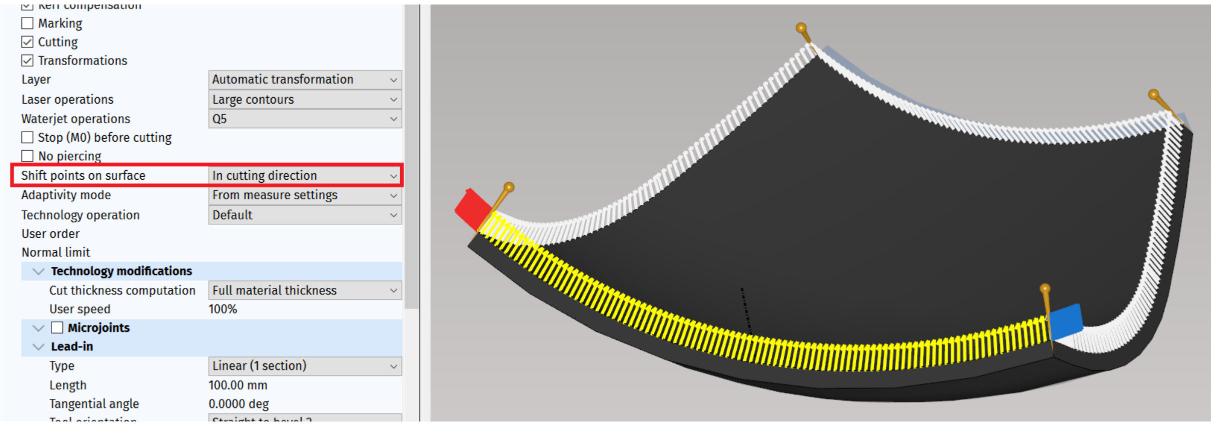


Fig. 387: Shift points on surface – in cutting direction type in mCAM

Finally, to generate the CNC, simply create a Job by dragging the desired shapes in the Jobs area and press the diskette icon to choose a folder for placing generated CNC file.

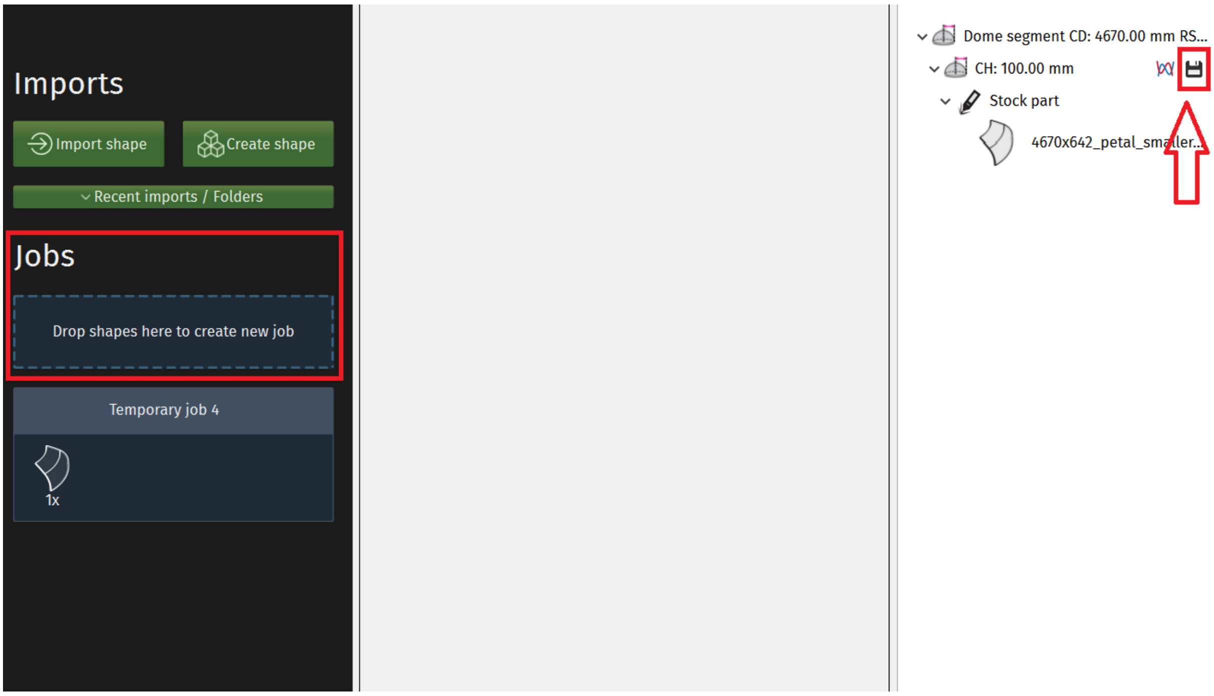


Fig. 388: Generating CNC from created Job

Pipe nesting in mCAM

Introduction

Pipe nesting process is a complex task of nesting individual parts on a stock in all forms of pipe and profile shapes. This document introduces settings, possibilities and recommendations during Pipe nesting process in mCAM.

For customer based production

- Import shapes
- Place them to temporary job
- Cut result
- Discard all data

For own production:

- Import shapes
- Place them to permanent job
- Cut result and check result
- Store job for later usage

Importing pipe shapes in the mCAM program

Import shape in mCAM

To import the desired pipe shapes in a suitable format (*.step / *.stp) it is possible to use the **Import shape** button on the left side of the main mCAM screen. Pressing the button will allow to browse computer files and choose those, that should be loaded in mCAM.

Alternatively, the user can choose **File -> Import shape** or press the shortcut key **Ctrl + I**

For import whole directory use – right icon .

Fig. 389: Import shape in mCAM

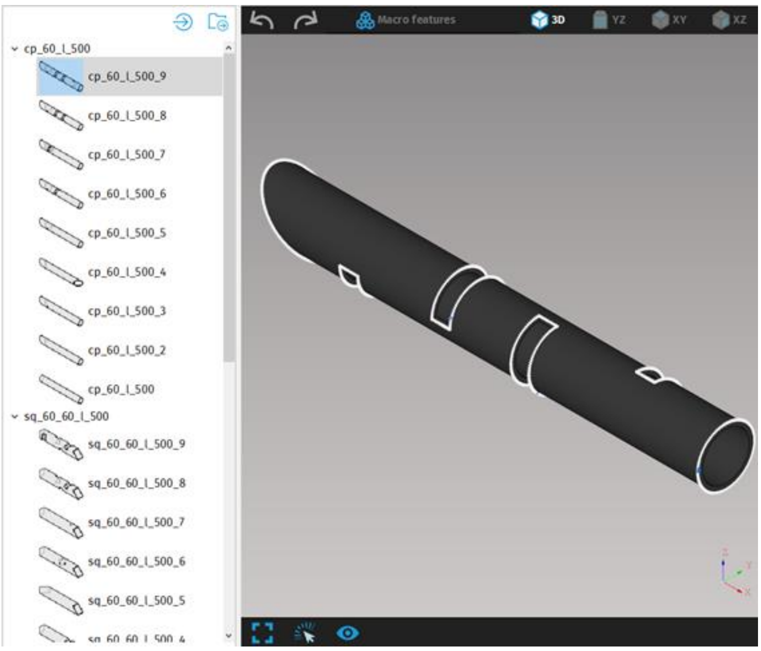


Fig. 390: Imported pipe shapes in mCAM

Creating a Job in mCAM

To create Job in mCAM simply drag and drop your selected shapes into the Jobs area in the left part of the mCAM screen. Shapes can be imported to a newly created Job or in already existing Job. Job can be saved to disk as a single file. This job can easily be transferred to different mCAM.

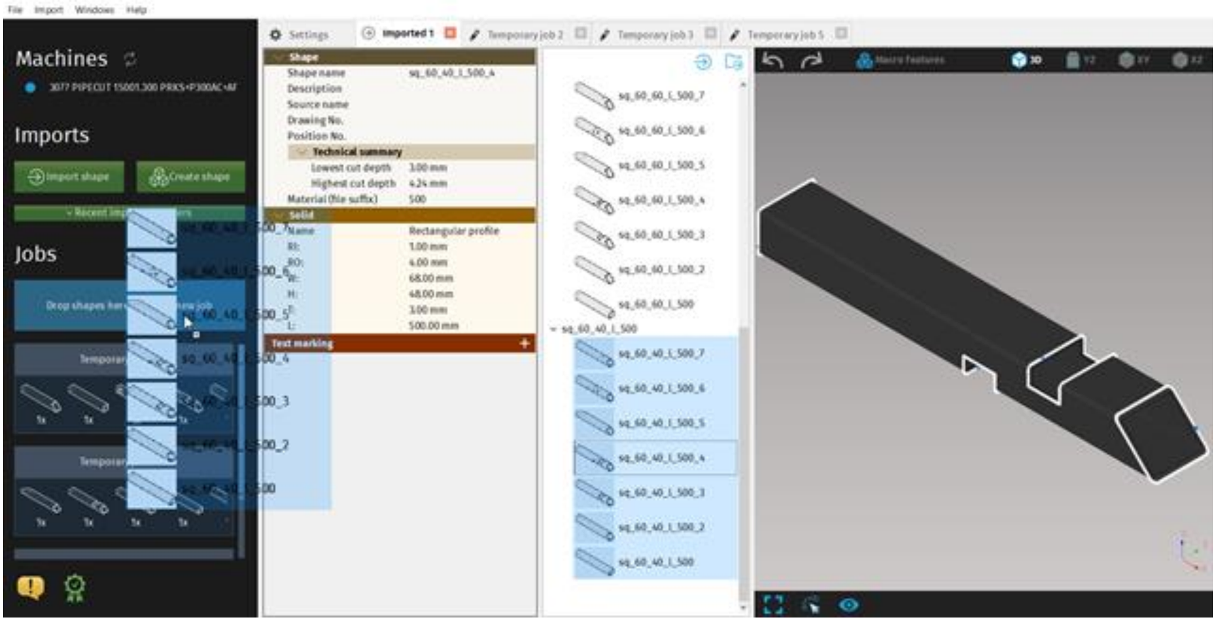


Fig. 391: Drag shapes to create a new job in mCAM

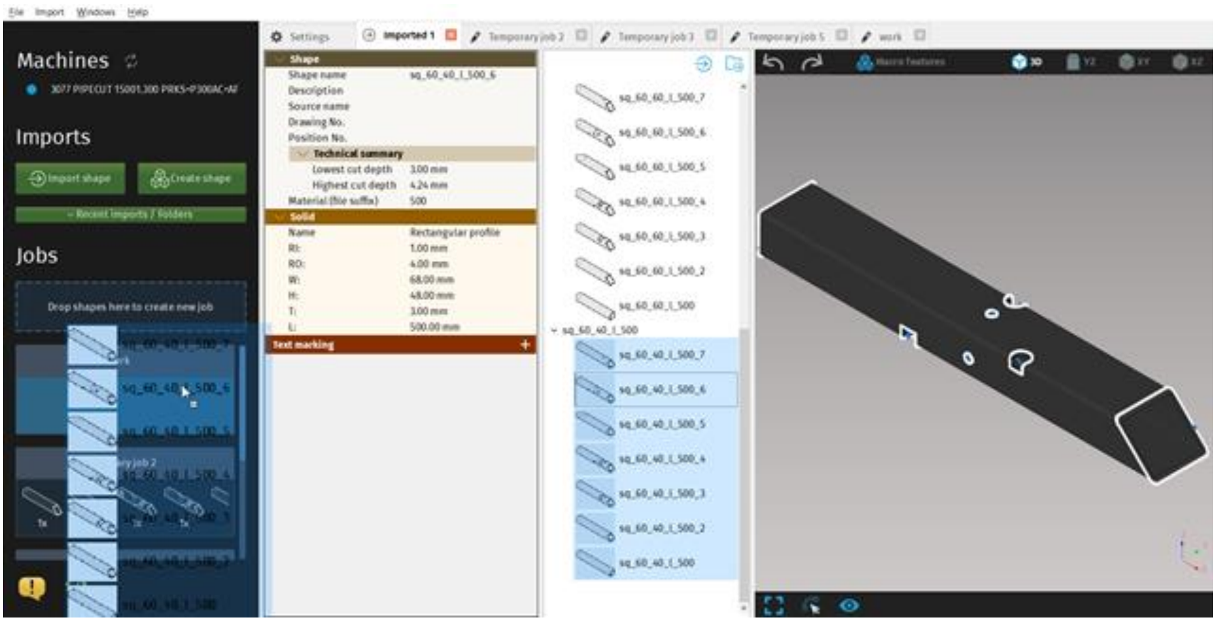


Fig. 392: Drag shapes into already created Job in mCAM

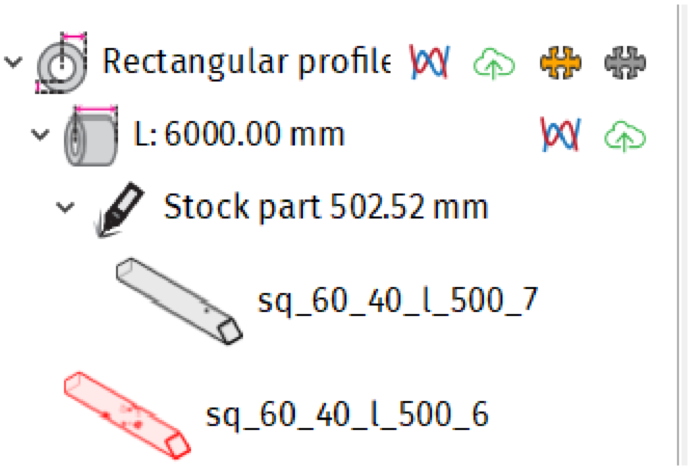


Fig. 393: Example of imported job in mCAM

After creation of job, mCAM automatically sets a template and stock with default length. User can run simulation and save CNC using the corresponding icons. Template is a single size of stock pipe or profiles.

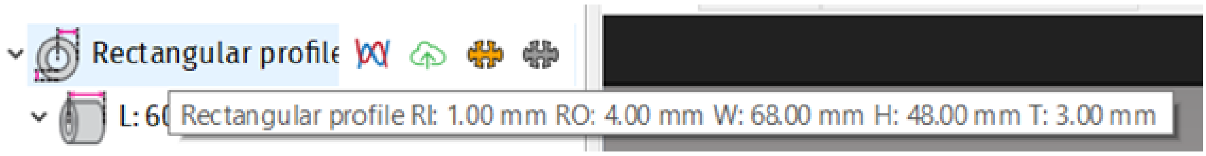


Fig. 394: Template

Stock settings in mCAM

Stock part is a specific piece of material where the parts have been nested. To add stock with specific dimension, use right mouse click and choose "**Add stock**". In the Job stock area set desired length. The number of stocks can be multiple.

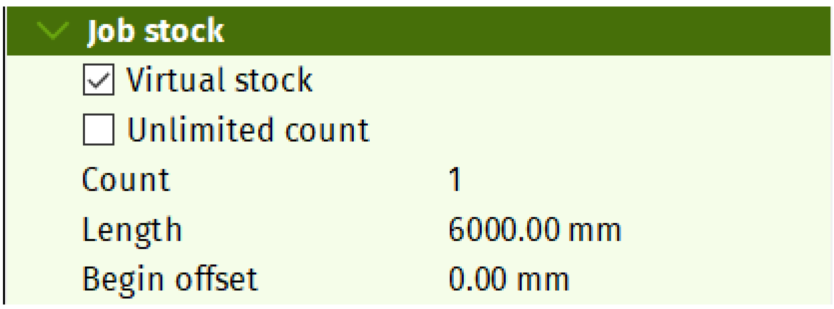


Fig. 395: Job stock properties in mCAM

The objects with grey contours in the list (for example "**sq_60_40_l_500_7**") are parts that have already been placed on the stock.

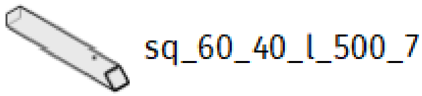


Fig. 396: Part that has been placed on stock

The objects with red contours (for example “**sq_60_40_l_500_6**”) are the parts that have not been placed on the stock yet.

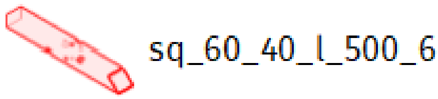


Fig. 397: Part that is waiting to be placed on stock

mCAM allows to multiply number of parts. If you want to create multiple parts with the same shape and dimensions navigate to Job part area and set a desired number of parts. Take into consideration that change of the cutting paths that has been done on the original part will apply for all of the copies as well. To avoid this, it is necessary to drag & drop the part multiple times.

Fig. 398: Setting number of Job parts

Nesting of parts in mCAM

Icons for nesting of the parts are shown on the right side from the template as orange or grey “puzzle pieces” as shown below.

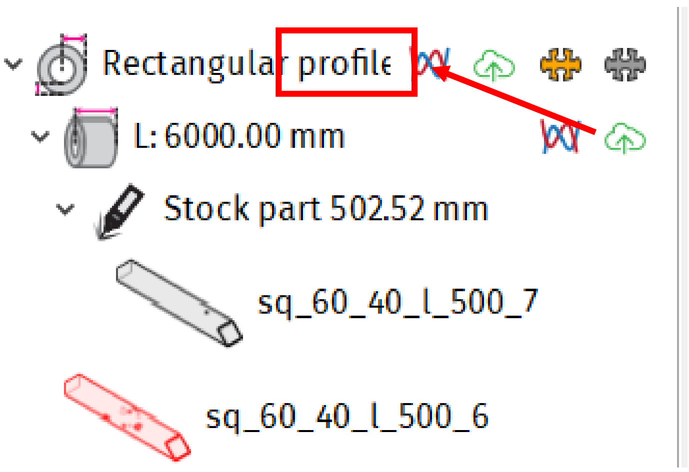


Fig. 399: Nesting icons

Grey icon represents the (original) full nesting. The way it works, is that it takes all the parts and stacks them according to its own consideration (randomly). Nesting is a complicated task; by using this method, the results might not always be optimal. The drag and drop function will order the parts according to the requirements within the job.

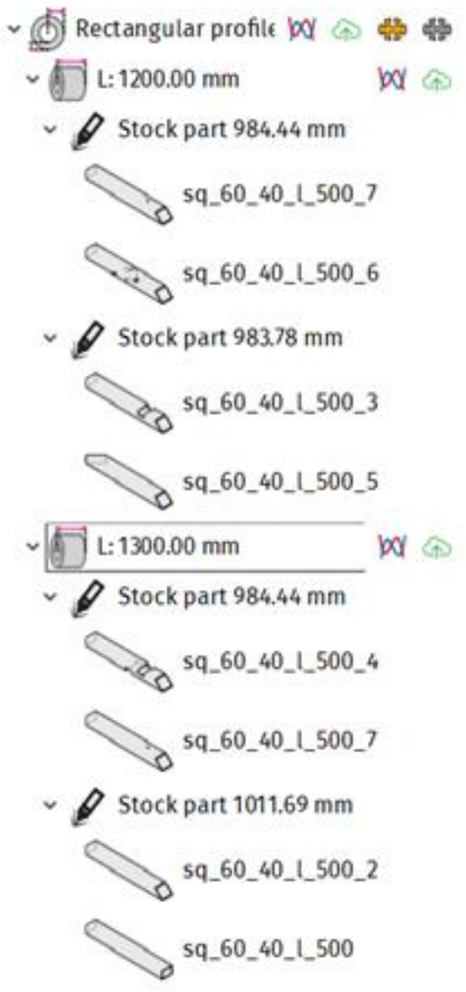


Fig. 400: Ordering the parts within the job

Orange icon represents the local (arranging nesting). It allows user to manually set the rotation and to optimize nesting manually. This way is considered optimal for using mainly for these reasons:

- does not change the order of the parts
- does not transfer the parts to other pipes
- if the part does not fit on the pipe, nesting will exclude this part

Limited positioner in mCAM

Due to reasons of power supply wiring leading to the positioner and supports, the positioner has a usual limitation for range of rotation set from -398° to 39°. To take this matter into account, check button -> **Consider limited positioner** in Nesting settings for template.

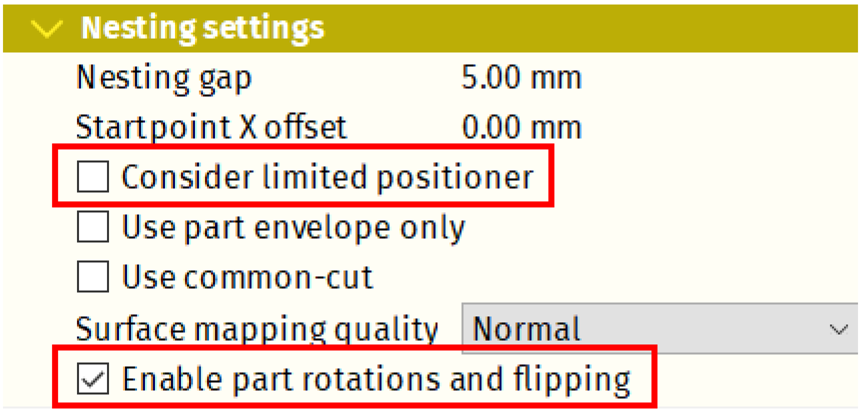


Fig. 401: Consider Limited positioner option in Nesting settings in mCAM

Rotating the part might be a solution for cases when the positioner limitation can be an obstacle during the cutting process. To enable this feature check **Enable part rotation and flipping** in the Nesting settings for template. If it is possible program will rotate and/or flip the material in the best position for cutting of the part.

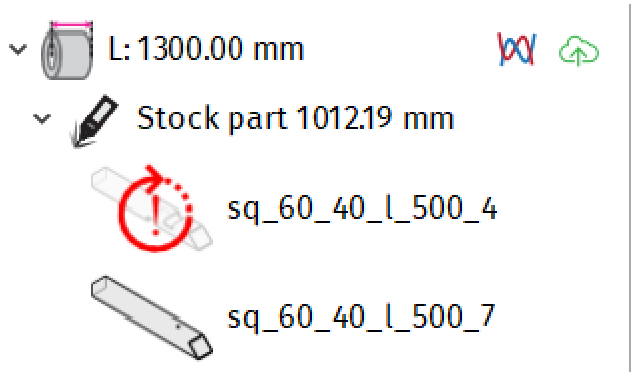


Fig. 402: mCAM will alert the operator in case when the part can't be cut due to a limitation of the positioner

Another way how to solve the obstacle related to positioner limitation is to use microjoints, which will divide the whole cut to smaller sections.

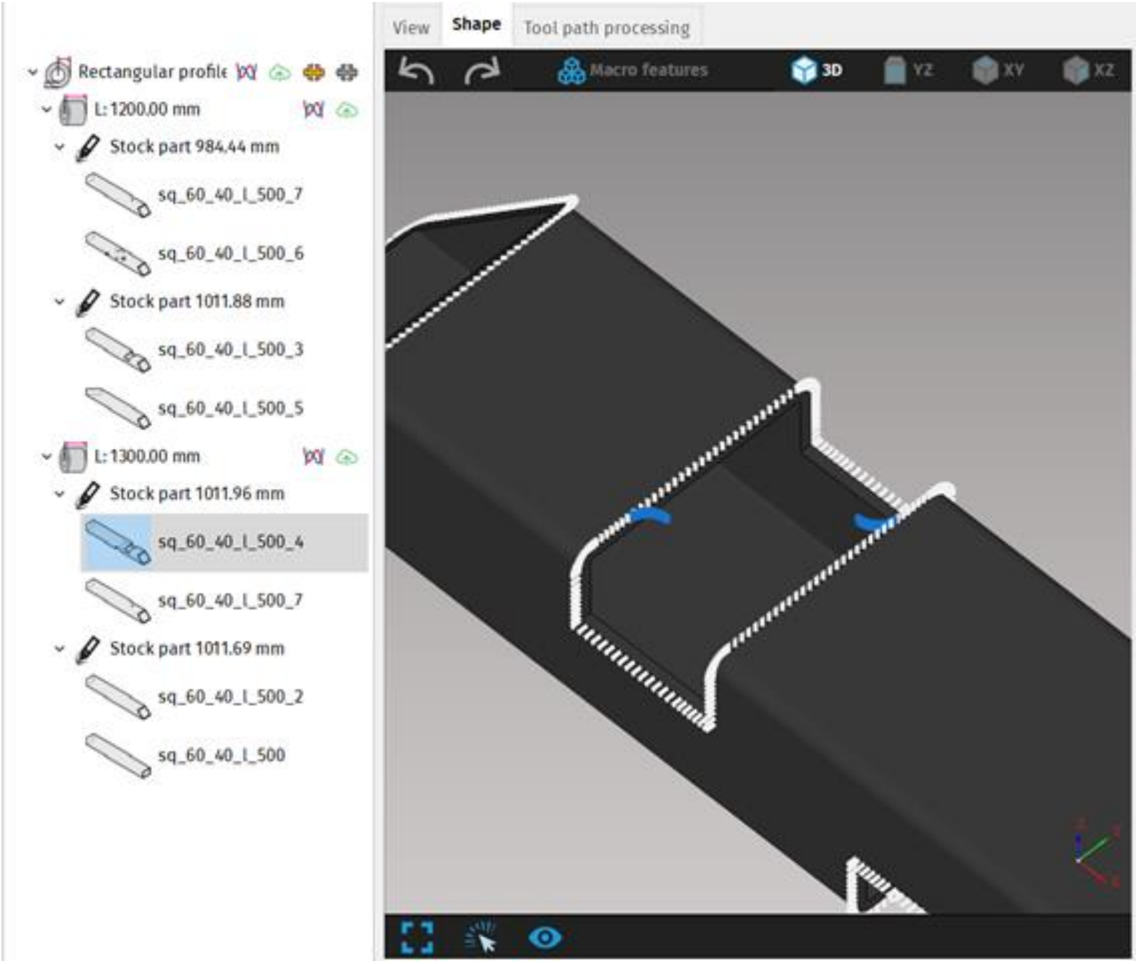


Fig. 403: Microjoints applied to cut

For certain machines (in case there is a mechanical limitation on the machine), it is suitable to specify a small clearance at the beginning of the pipe. For these reasons there is a "**Start point X offset"** option, that can be found in nesting settings. To set the length of the offset start, enter the value specified in millimeters. "Start point X offset" and "Begin offset" represents the same entry.

Start point X offset is defined for template; i.e., all pipes specified in the given template (one type of shape). Begin offset is defined for a specific stock. The resulting offset is the sum of these two values.

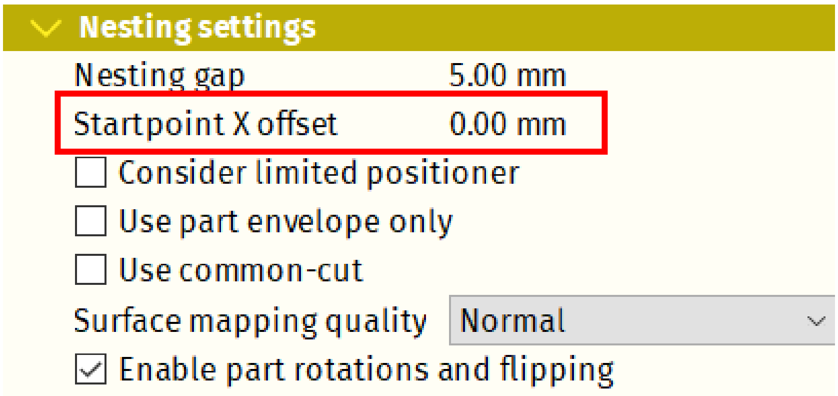


Fig. 404: Start point X offset option in mCAM

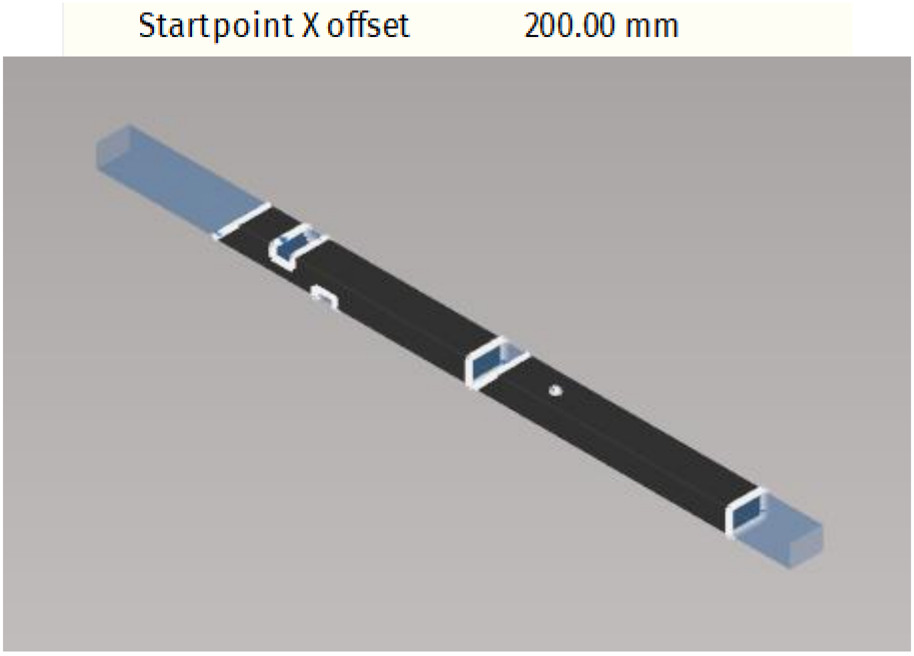


Fig. 405: Start point X offset option applied on the material

Use part envelope only –

Using the function "**Use part envelope only**" in mCAM, will results in the parts being nested next to each other without the possibility to slide them into each other when bevel cuts are present. On the other hand, it allows the user to freely rotate the profile without the need to run the nesting again (i.e., the part is always considered as it does not have a diagonal cut on the edges).

Use part envelope only option can be enabled/disabled by using checkbox in Nesting settings in mCAM.

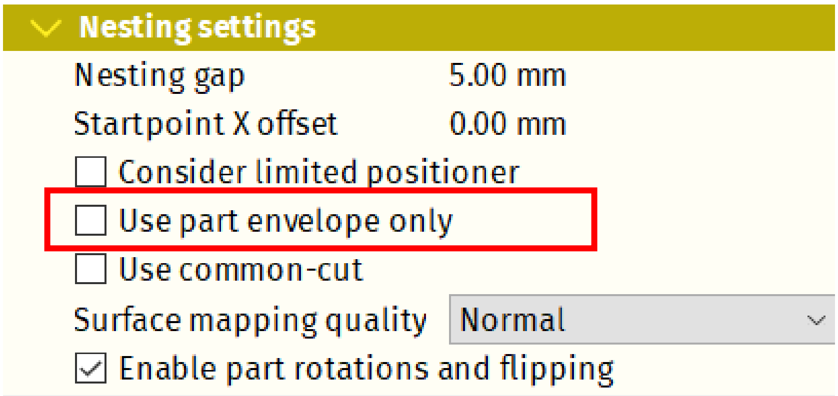


Fig. 406: Option "Use part envelope only" in mCAM

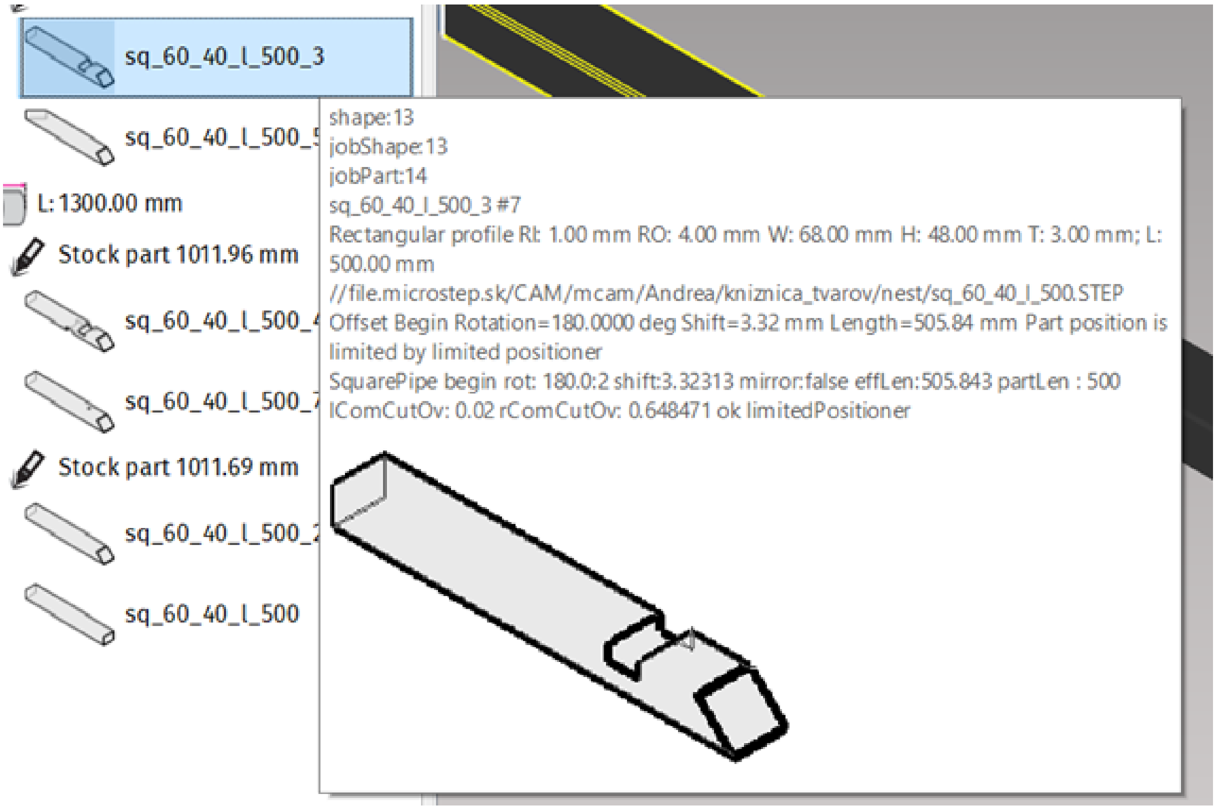


Fig. 407: Nesting results shown in a tooltip

Using the common-cut.

In cases when two parts are placed to each other and the cuts of these parts are identical (with given precision), they can be combined into a single common-cut; provided that this option is enabled in settings. To enable/disable common-cut and to set a precision of the common-cut, use checkbox “**Use common-cut**” in nesting settings.

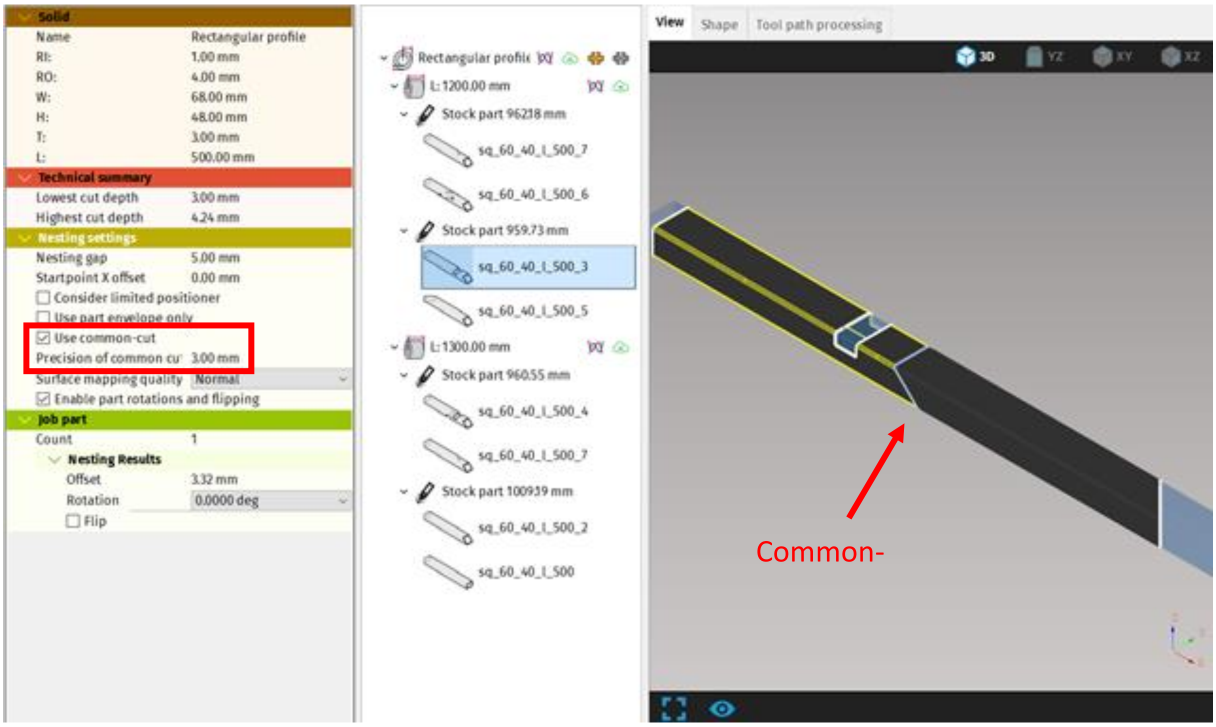


Fig. 408: Common cut in mCAM

If the specified value of precision is too small (0,0001 mm), there is a risk that common cut cannot be created. If the specified value of precision is too big (3 mm), the common cut will be created, although the resulting precision of the cut might be insufficient.

Nesting gap in mCAM

After the initial nesting and after all the necessary manual corrections, the field “Enable part rotations and flipping” can be unchecked. Nesting will automatically add the new parts one after another. To set the gap between each part, set “Nesting gap” value, which represents the distance between individual parts. Nesting gap ignores lead-ins during the nesting; therefore, the gap must be larger than lead in for each part.

Fig. 409: ***Nesting gap in mCAM***

Saving the CNC in mCAM

To **save** the desired CNC use one of the options below:

1. <u>Using the diskette icon to save CNC</u>

Diskette icon - save the CNC file to disk.

This option is suitable for machines that don’t have the MRP feature turned on.

Fig. 410: Saving file to disk

2. <u>Using the cloud icon to save CNC</u>

Cloud icon - save CNC file to disk and also to MRP.

Using this option results to sending the CNC file directly to the machine and to the database.

Fig. 411: ***Saving file to disk and MRP***

MRP must be supported by particular machine. In that case, user can select “use MRP” in mCAM settings, so mCAM can upload cutting programs directly to machine database – until the network to machine connection is available.

Split stock in mCAM

In case when a single stock has multiple physical parts, it is possible to use "Split stock" function and separate individual stocks in the list of stocks in mCAM. Split stock function can be found in mCAM, after right click on the specific stock part.

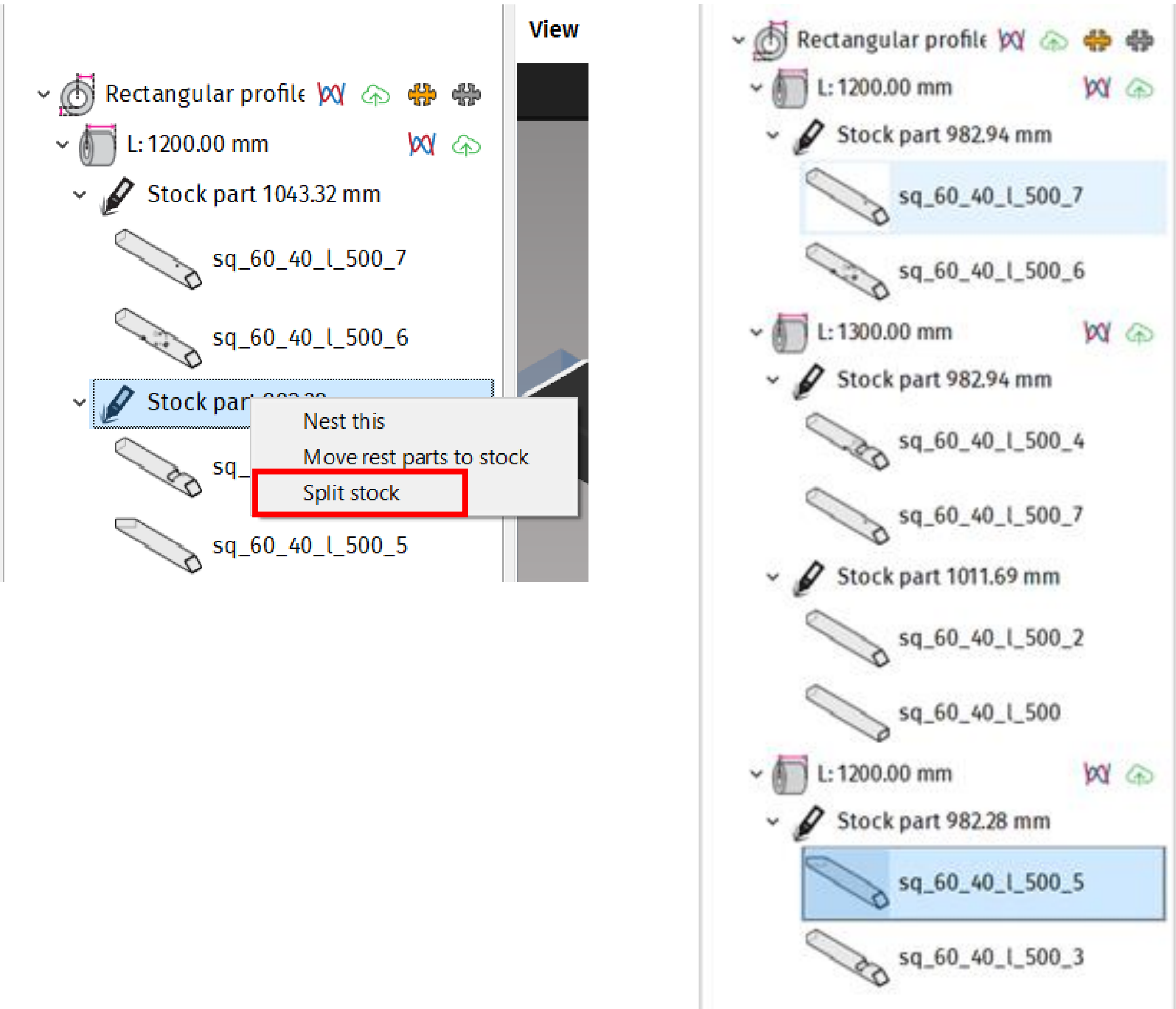


Fig. 412: Before and after "Split stock" results in mCAM