myNCT SMART DIALOG

OPERATOR'S MANUAL

Version:	3.3.1
Compiled by:	NCT Ipari Elektronikai Zrt.
Contact:	mynct@nct.hu

Contents

1 OBJECTIVE

The aim of myNCT is to provide graphical support for writing CNC programs. Using it, it is less necessary to be good at G-code NCT CNC programming language based on the standard DIN 66025. Its goal is to significantly reduce program writing time and provide more support for users.

The ideal users are those who are competent in machining technology, and:

- who manufacture parts in unit and series production;
- who are less skilled in NCT programming language.

myNCT runs on the control of the machine tool which is an advantage over other CAM systems.

Functions supported by myNCT are the following:

- Selection of stock
- Holes (centre drilling, drilling, tapping, reaming, thread milling)
- Milling:
  - Plane milling
  - Pocket milling
  - Boss milling
  - Letter milling / engraving
  - Indexing
  - Contour milling
- Turning:
  - Facing
  - Contour turning (drawing contour, roughing + finishing)
  - Grooving
  - Threading
  - Parting off

2 STARTING THE MYNCT AND ITS STRUCTURE

2.1 Starting

There are two different ways to start the myNCT.

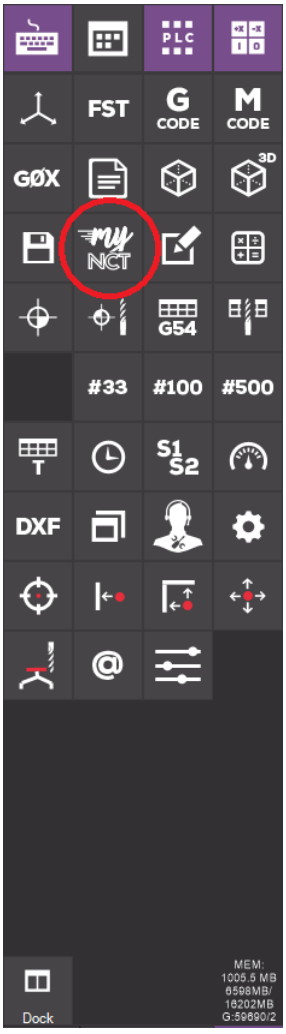


Figure 1 Starting myNCT from icon

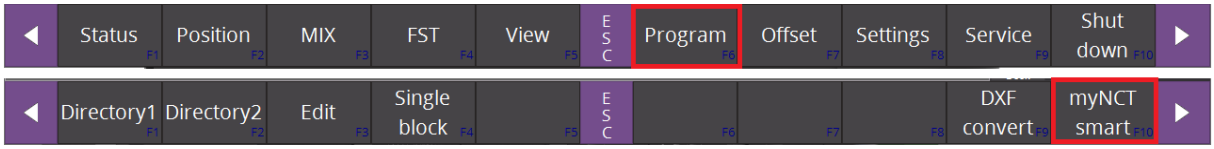


Figure 2 Starting myNCT from softkey menu (Program → myNCT smart)

2.2 Structure of the myNCT Smart Dialog

In addition to the softkey menu, it has the following four main parts:

1. Operation plan
2. Explanatory/Help window
3. Data specification for selected operation

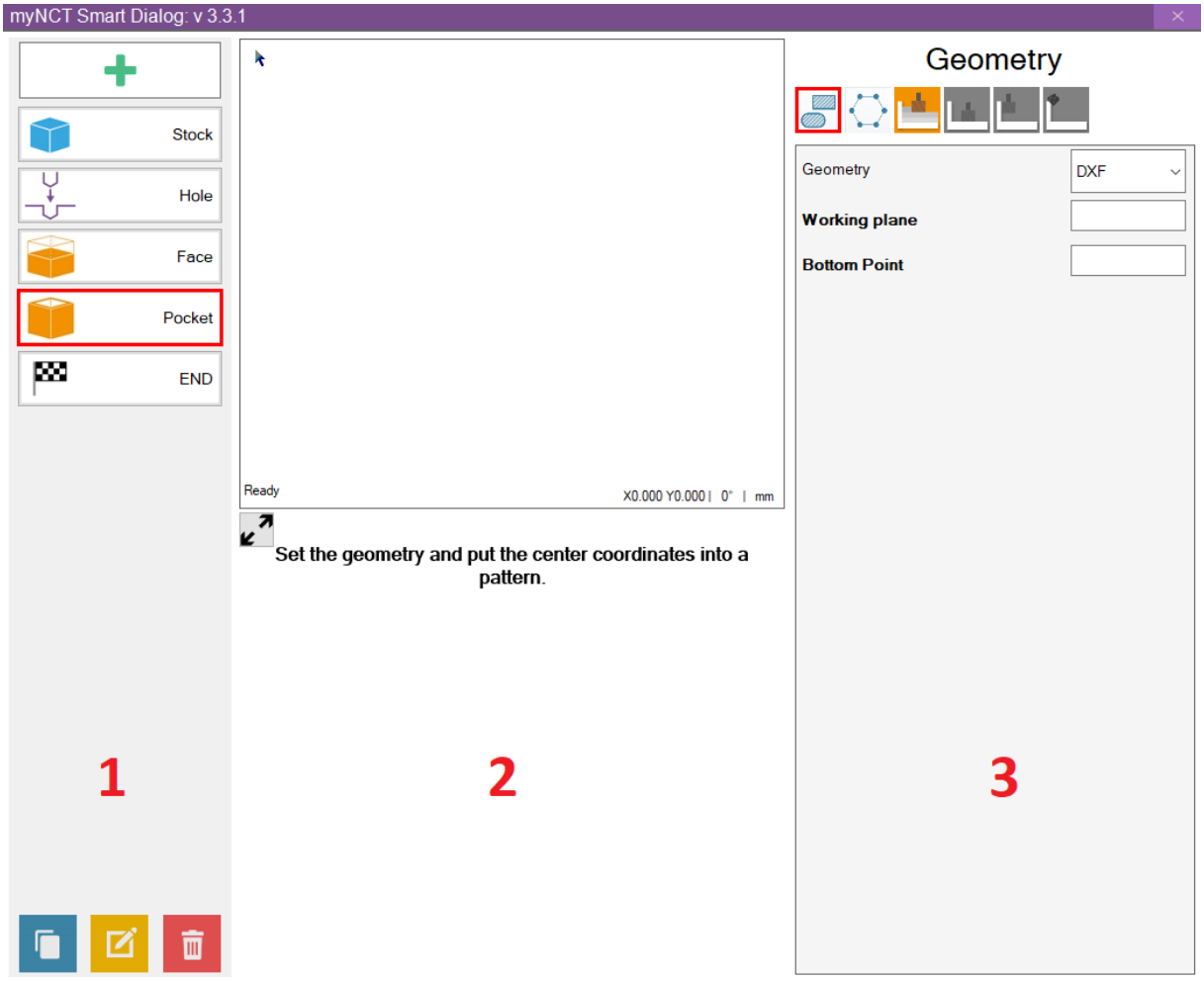


Figure 3 Main parts of myNCT

3 USING MYNCT

3.1 Softkey menu of myNCT

If myNCT Smart Dialog is the active window on the screen of the control, its own softkey menu will appear.

There are the following six options at start:

- F1: Create a new program
- F2: Open an existing myNCT program
- F3: Add new operation
- F4: Save
- F8: Export
- F9: Update macros
- F10: Help

If a new file has been created or a file has been opened, all the menus will appear on the softkey bar:

- F5: Simulation
- F6: Save as
- F7: Import

3.1.1 Creating new file

The extension of the file created with myNCT Smart Dialog is "my.nct".

The structure of the file generated by myNCT is as follows:

- It begins with <myNCT> identifier.
- The first part of the file contains G code that will be executed by the control.
- The second part of the file contains necessary data to read back the myNCT file.
- It ends with </myNCT> identifier.

3.1.2 Opening

It is possible to re-open files created with myNCT. If the file was created in myNCT, it will be opened in myNCT automatically from the directory window.

3.1.3 Saving

It is for saving the content of the edited file to a file. G code will not be generated if there is no licence or the cycles are specified incorrectly or imperfectly, but content of input fields and operations will be saved to a file. In case of valid licence and perfect specification, the G code generation will also be valid for files created earlier.

3.1.4 Adding new operation

A new operation is added to the operation plan right behind the active (selected) operation – but in between "Stock" and "End" operations. On click, selectable operations appear in a window.

3.1.5 Simulation

By clicking on Simulation, the following things happen:

- It is checked whether all macro files are available used by myNCT. In case of problem, the link of recent macro files will be copied to the clipboard (= right button of mouse, copy command).
- The windows on screen will be rearranged for simulation.

3.1.6 Save as

Program may be saved in a new file with new name so that it may be changed without modifying the original one.

3.1.7 Import

It is for importing content of another myNCT file. "Stock" operation is not imported.

3.1.8 Export

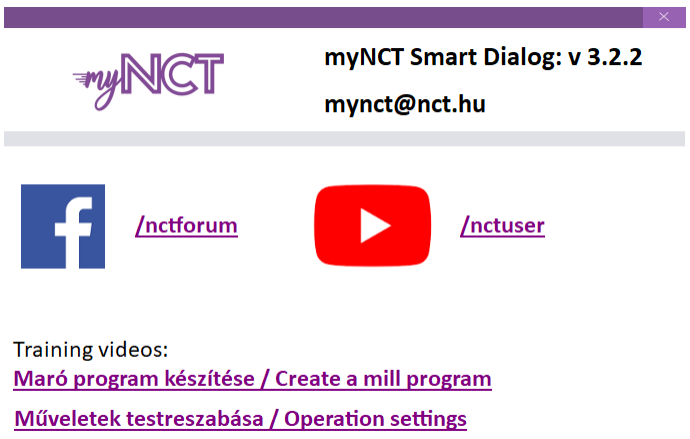
It is for copying G code of a myNCT file to an independent file. There is no relation between exported file and myNCT.

3.1.9 Update macros

When all macros are up-to-date, a wrench icon is displayed on the button. If macros may be updated, it changes to a red exclamation mark and clicking on it will update the macros.

3.1.10 Help

Contains contact and useful links.



4 OPERATIONS

4.1 Stock operation

Each program begins with the stock operation.

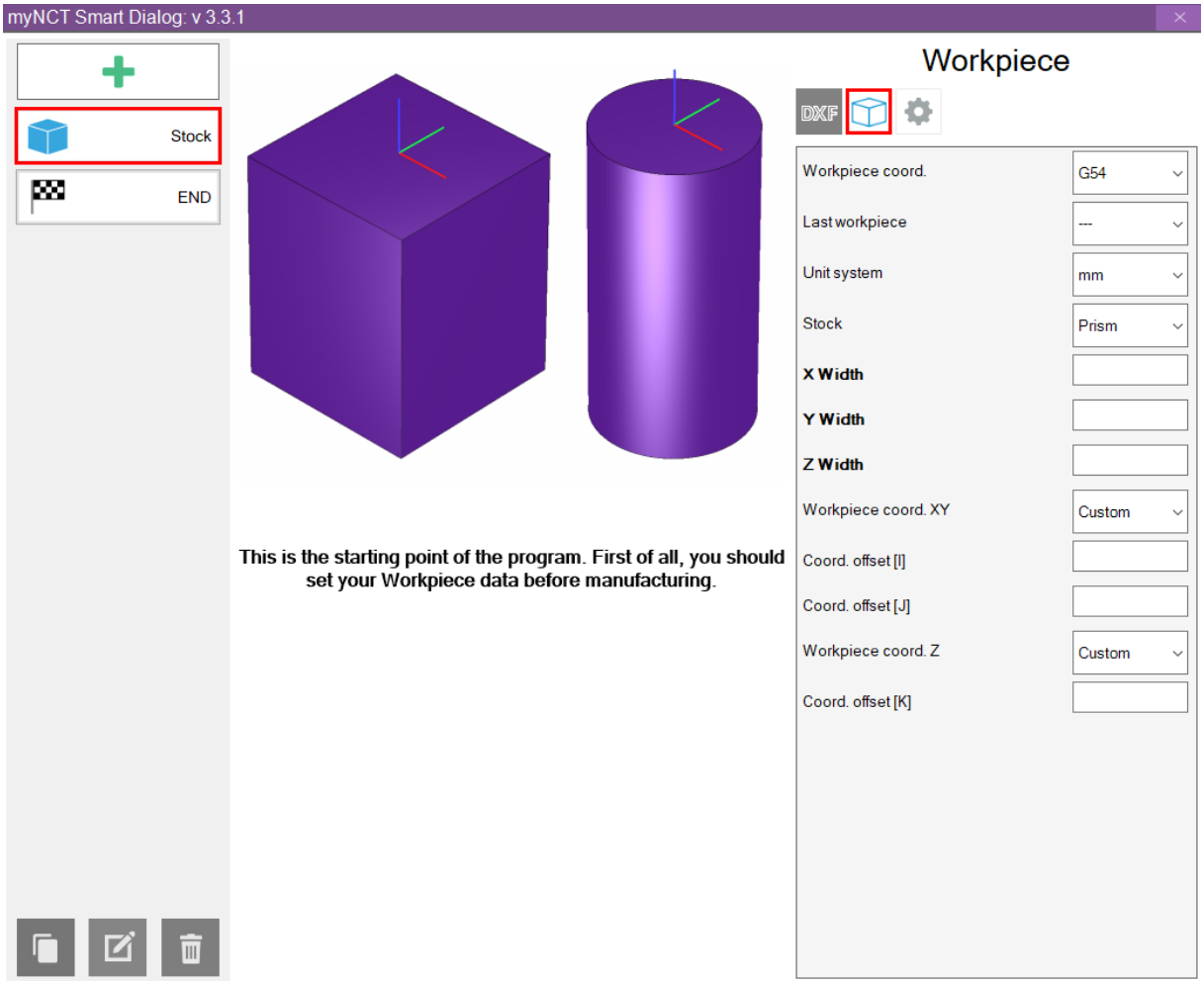


Figure 4 The stock operation

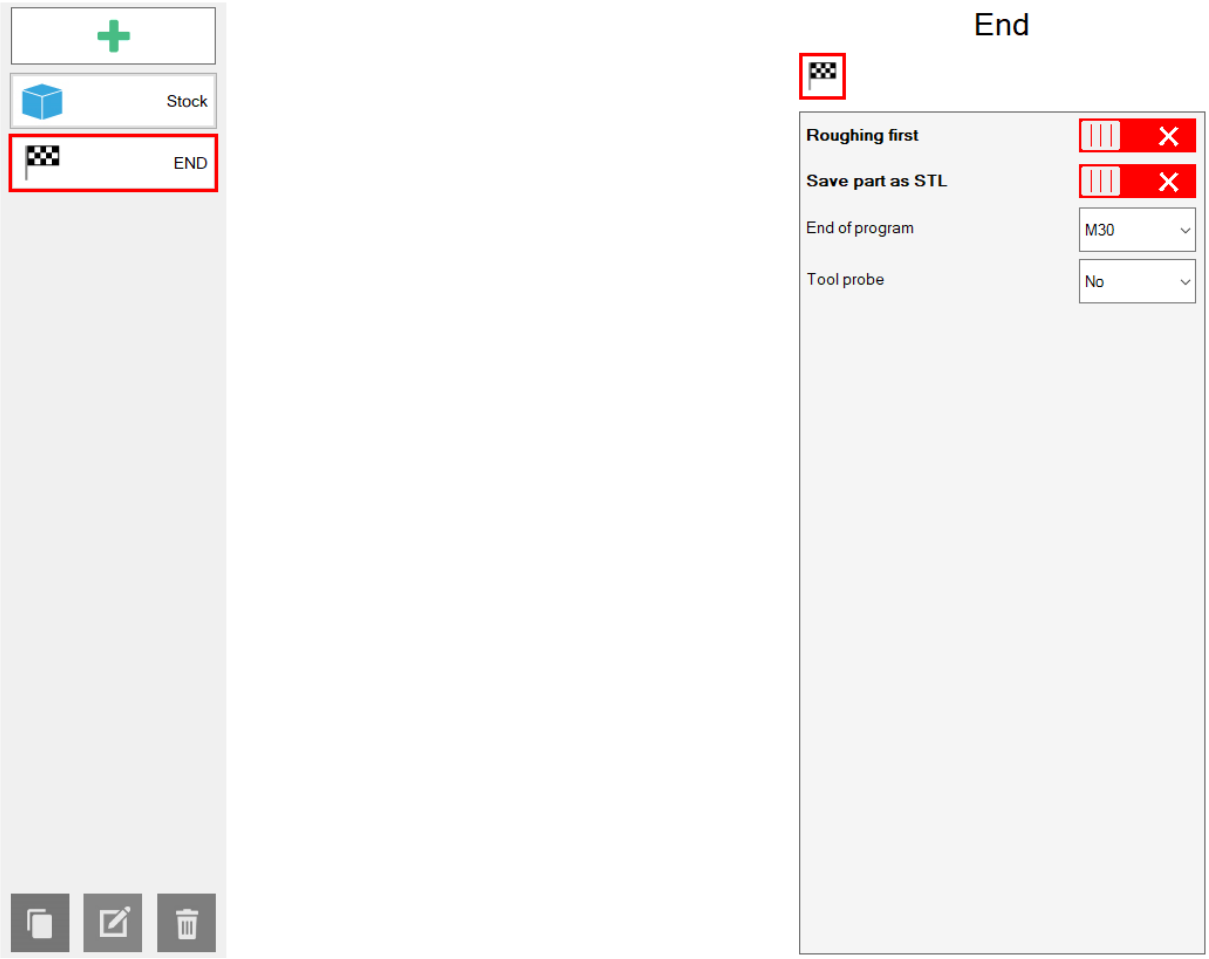
The stock operation is mandatory at the beginning of the program. Unlike other operations, this operation cannot be:

- moved;
- copied;
- renamed;
- deleted.

In the stock operation on the first tab a DXF file or a geometry may be set as stock for operations. On the second, workpiece coordinate system, unit system, type of stock for solid graphics and translation of workpiece zero point may be chosen. The third tab contains

settings to change Z rapid plane, an option to stop myNCT program at every toolchange or at the end of every suboperation, and also the way of that stop may be chosen.

4.2 End operation



The end operation is mandatory at the end of the program. Unlike other operations, this operation cannot be:

- moved;
- copied;
- renamed;
- deleted.

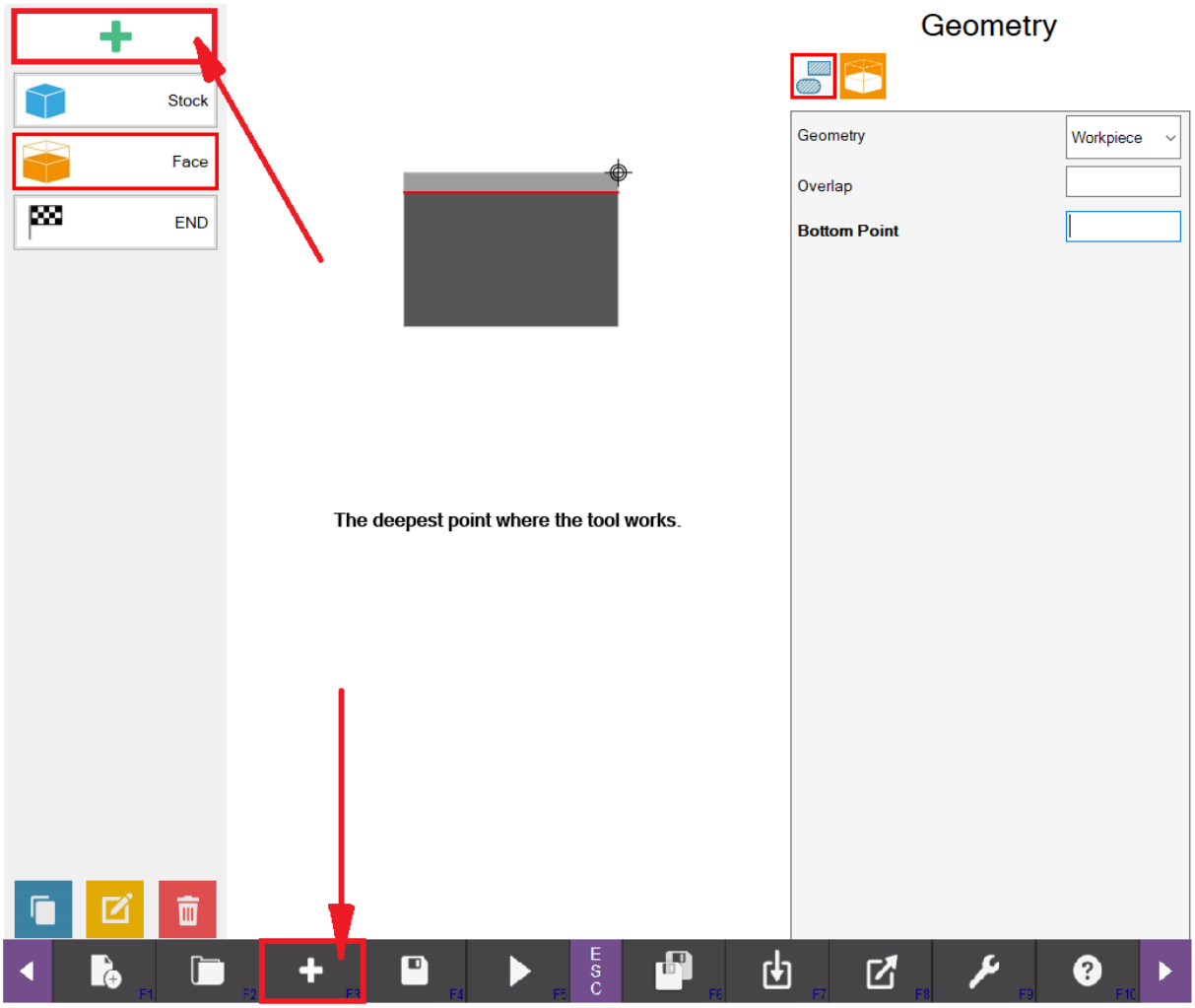
In the end operation panel order of suboperations may be changed - when multiple roughing operations are used, it is possible to execute roughing operations first.

An STL file of the created part can be generated.

The end of the program can be set to M30 (runs only once) or M99 (repeats the program).

It may be selected if that there should be tool probing during program execution, and if so, should every tool be probed at the beginning of the program or just before using it for the first time in the program.

4.3 Adding new operation



On the left side, clicking on the button a window pops up containing selectable operations. By clicking on any of them, the selected operation will be inserted in the position following the active operation in any case.

Figure 5 There is a red frame around the selected operation

4.4 Moving operations

The order of operations can be changed using 'Drag and Drop' method.

4.5 Copying operations

When the selected operation is being copied, the full content of the operation is being copied. The name of the copied operation corresponds to the original operation, and '+' character is added to its end. It is useful in a variety of ways:

- Bottom and side finishing has a fixed order. If side finishing is desired to be done first, then bottom finishing should be deactivated (see at 4.9.1) in the first operation and side finishing has to be active. In the copied operation, vice versa. *Warning*: after side finishing, bottom finishing allowance is usually not left– this may be evaded by modifying geometry (changing bottom plane coordinate).

4.6 Deleting operations

Pushing the delete button, the selected operation and its full content will be deleted.

4.7 Renaming operations

Custom and short name to the selected operation can be given. Use of the <''> character is not allowed.

4.8 Statuses of an operation

There are three different status of an operation.

4.8.1 Active operation

It is the status of the operation created recently and specified correctly. Checking the latter one happens when save button is pushed and G code is generated.

Figure 6 Icon of the active operation

4.8.2 Active operation specified incorrectly or imperfectly

If the operation is specified incorrectly or imperfectly, myNCT will not generate G code of this operation when saving. The content of the operation will be saved in the part of the file that follows G code.

Figure 7 Incorrect operation

4.8.3 Inactive operation

The operations can be activated/deactivated by double click. The G code of the inactive operation will not be issued when saving.

Figure 8 Inactive operation

4.9 Panels of the operations

The selected operation has a list of forms displayed on the right side. As an example, a pocket milling operation is illustrated below.

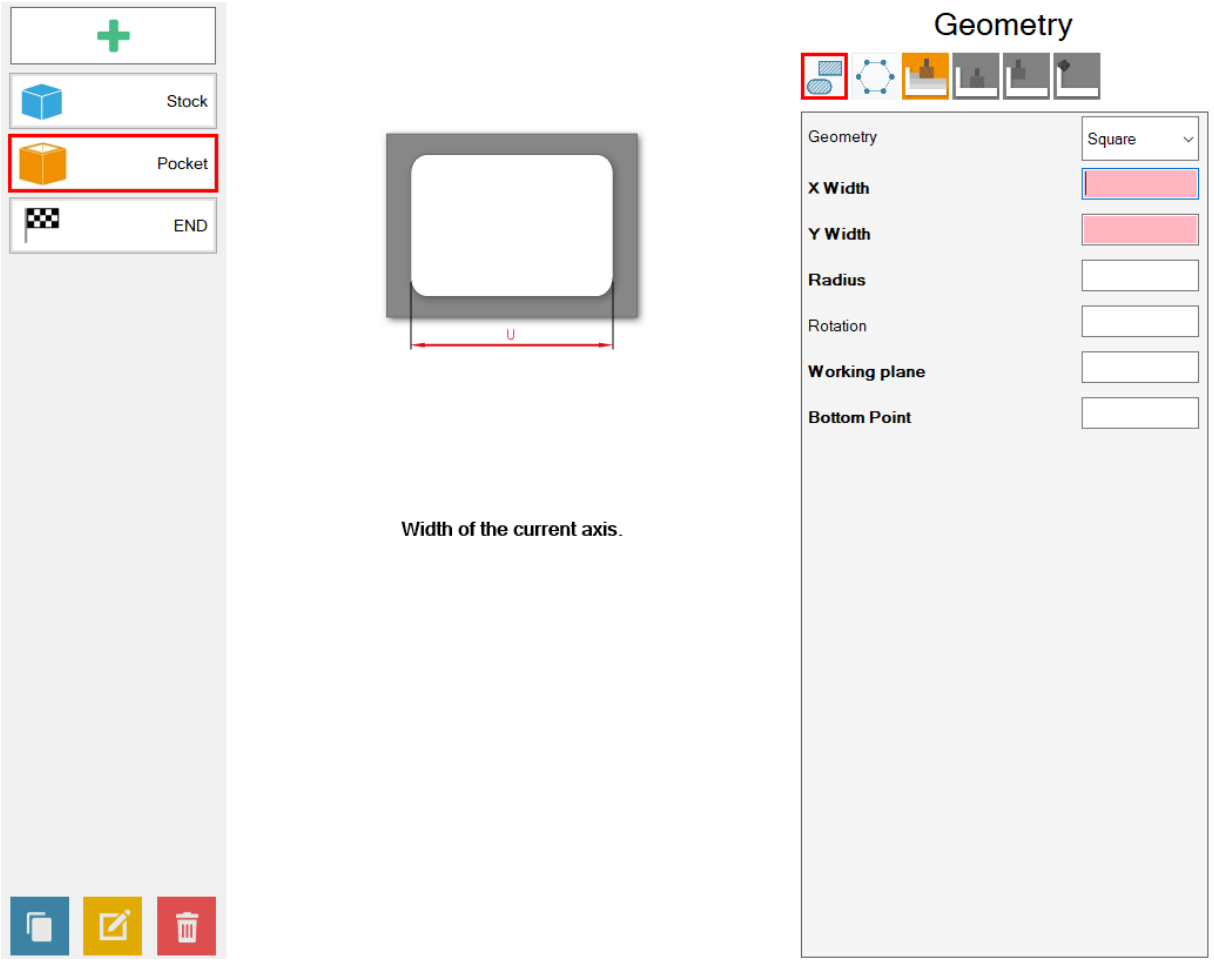


Figure 9 Forms and input fields of a pocket milling operation

4.9.1 Selecting geometry, positions and machining operations

The pushbuttons located horizontally are related to the forms belonging to the operation. Appearance of selected, incorrect and active form is similar to those of the buttons of operations.

Figure 10 The forms of a pocket milling operation can be reached by clicking on the pushbuttons

- The order of them cannot be changed (there is no Drag and Drop).

- In case of incorrect/imperfect specification, the pictures do turn red (as well as the incorrect elements of the form).
- The form might be activated/deactivated by a switch under the active pushbutton.

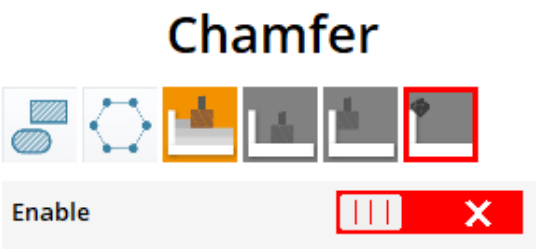


Figure 11 Deactivated chamfer

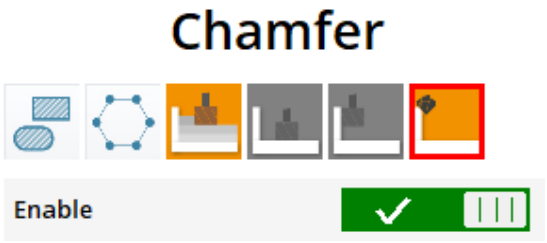


Figure 12 Activated chamfer

4.9.2 Forms

On forms there are different fields which can be selected and filled. Names of the fields that are mandatory to fill are written in bold.

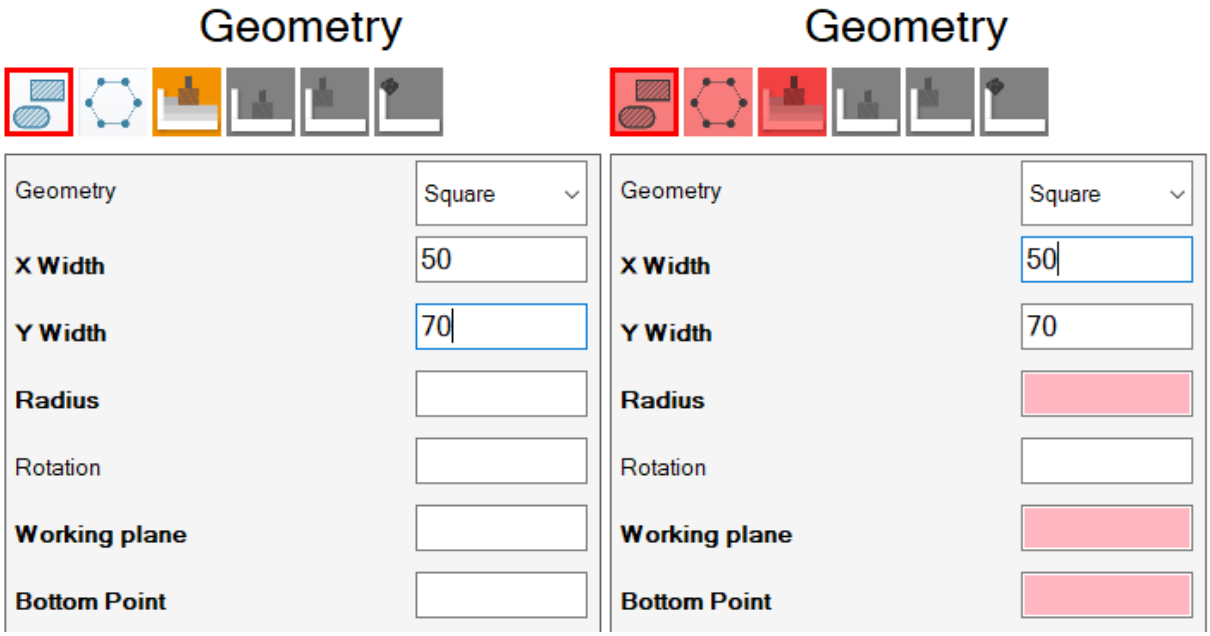
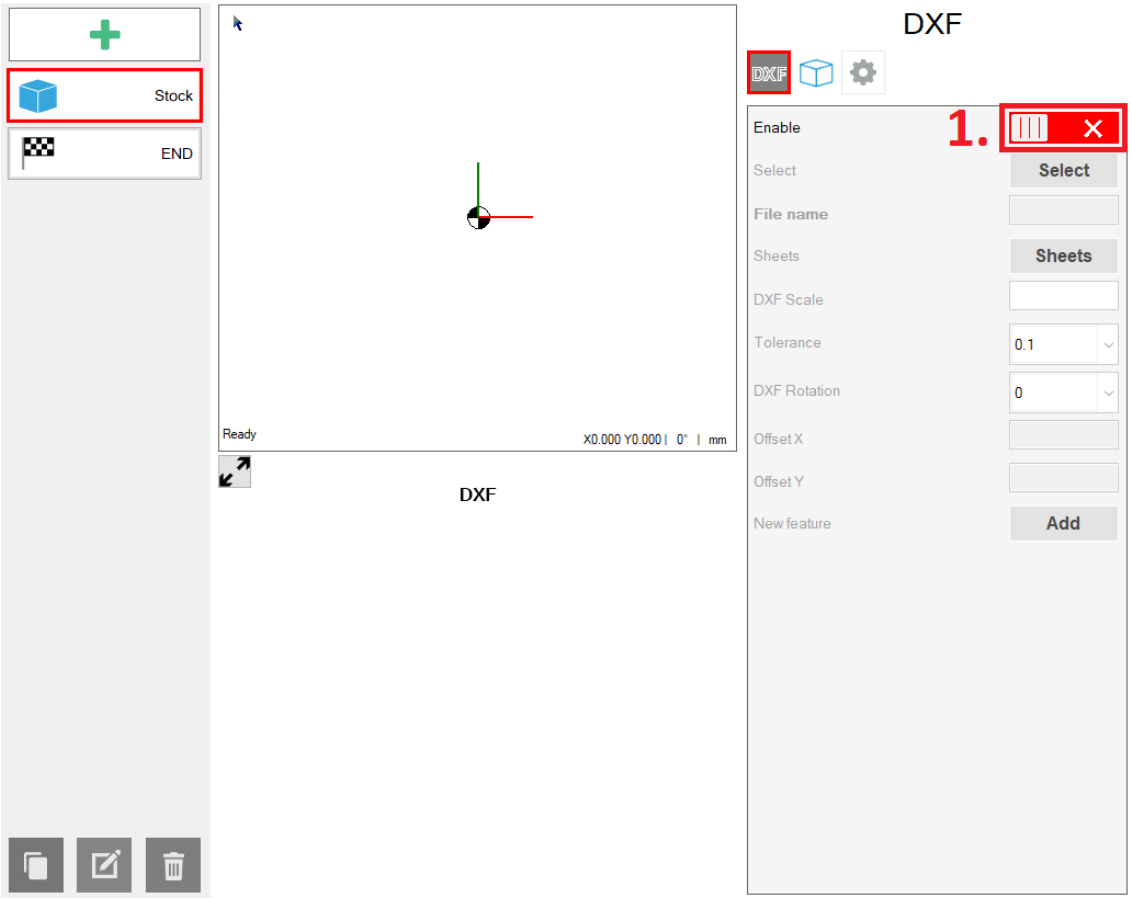


Figure 13 Statuses of the fields to be filled mandatory:
1 – At beginning; 2 – After save command

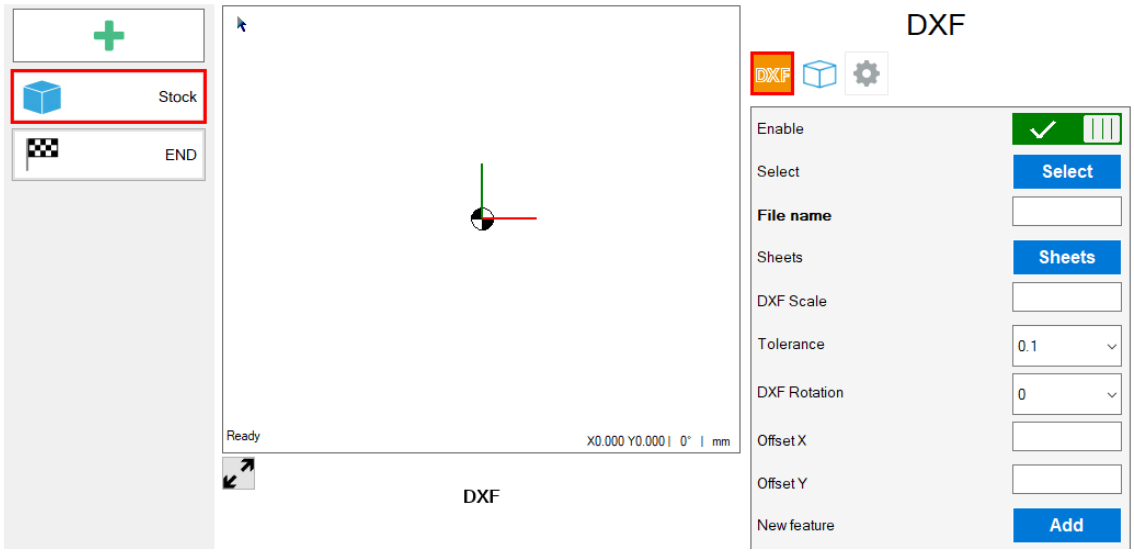
5 WORKING WITH DXF FILES

5.1 Use of DXF Panel

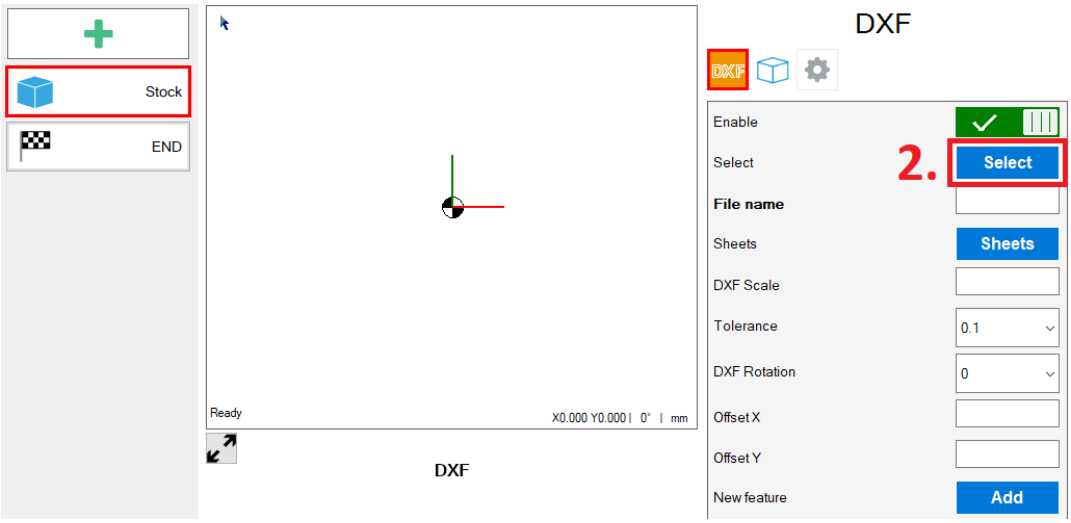
5.1.1 Open a DXF file



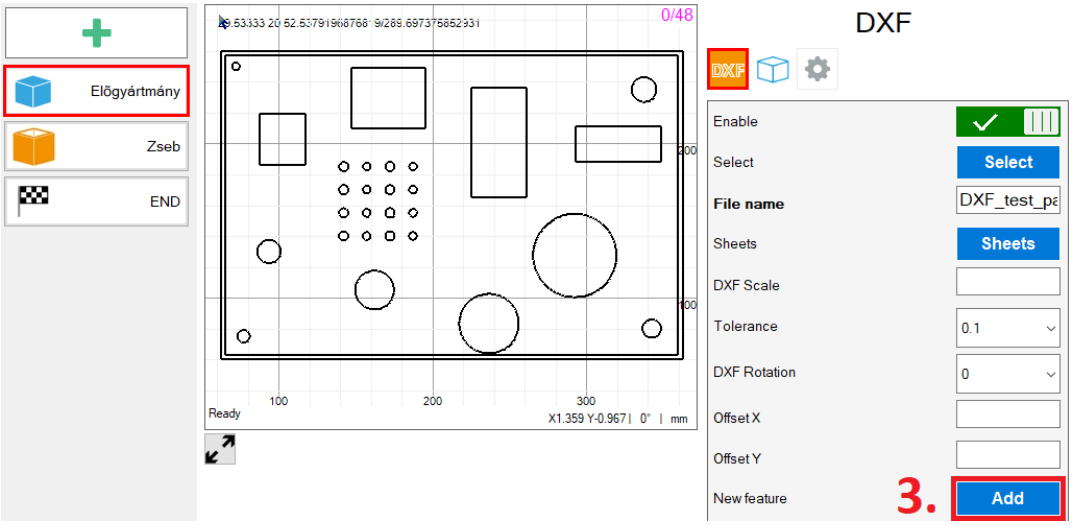
5-1. Figure Activate DXF panel



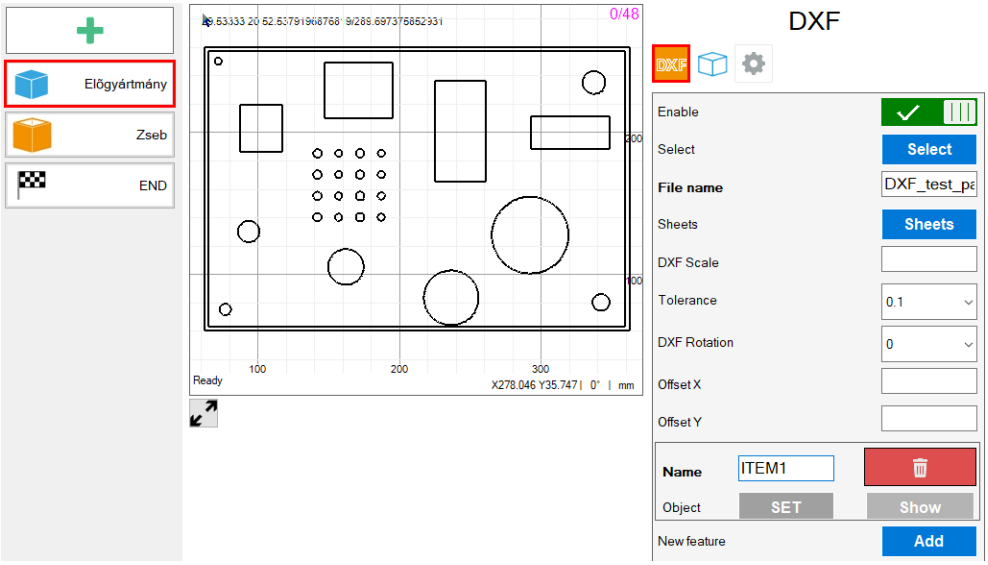
5-2. Figure Activate DXF panel is activated



5-3. Figure Select DXF File



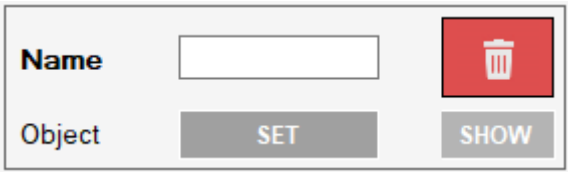
5-4. Figure: Add Feature



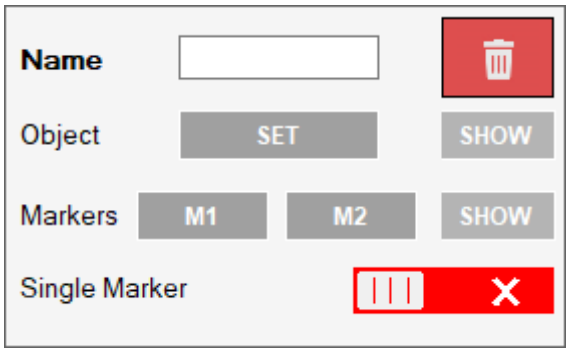
5-5. Figure: A „Feature" added to the screen. Use unique name for each one.

5.1.2 Feature Properties

Features contain information of the selected geometries.



5-6. Figure Feature with its values



5-7. Figure Feature with markers (optional)

Name: helps identifying different Features

Object: contains the graph objects (i.e. points, lines, circles…) for manufacturing

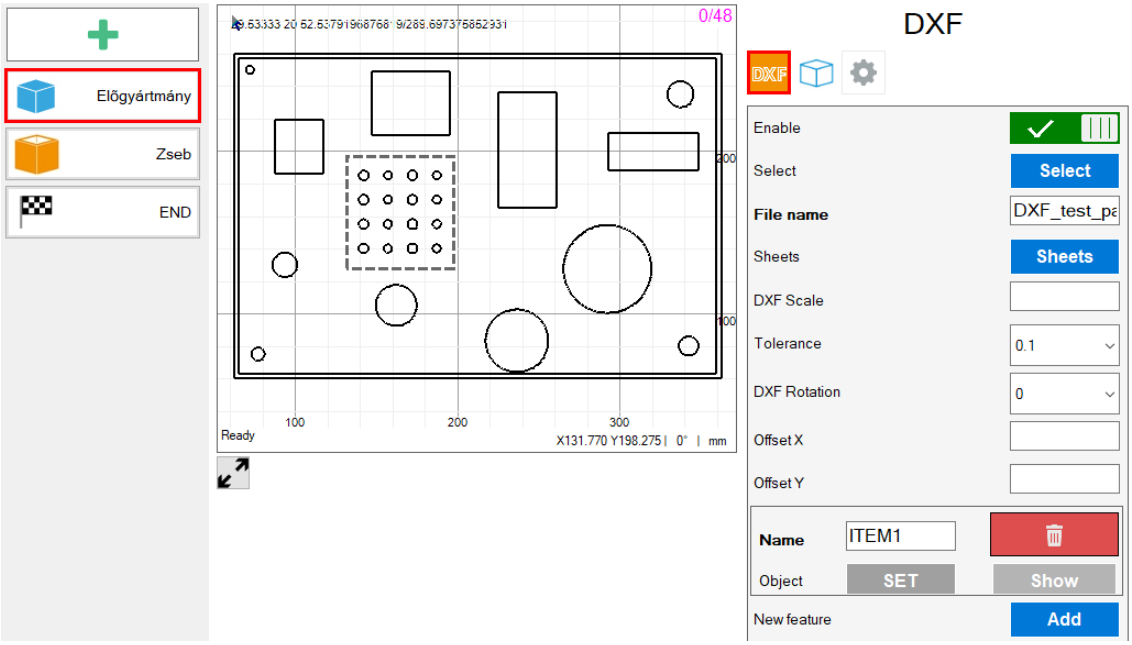
Markers: these contain 2 positions for calculating feature coordinate system

Single Marker: switch on/off single marker mode, when each closed geometry has its own marker. It works with circles.

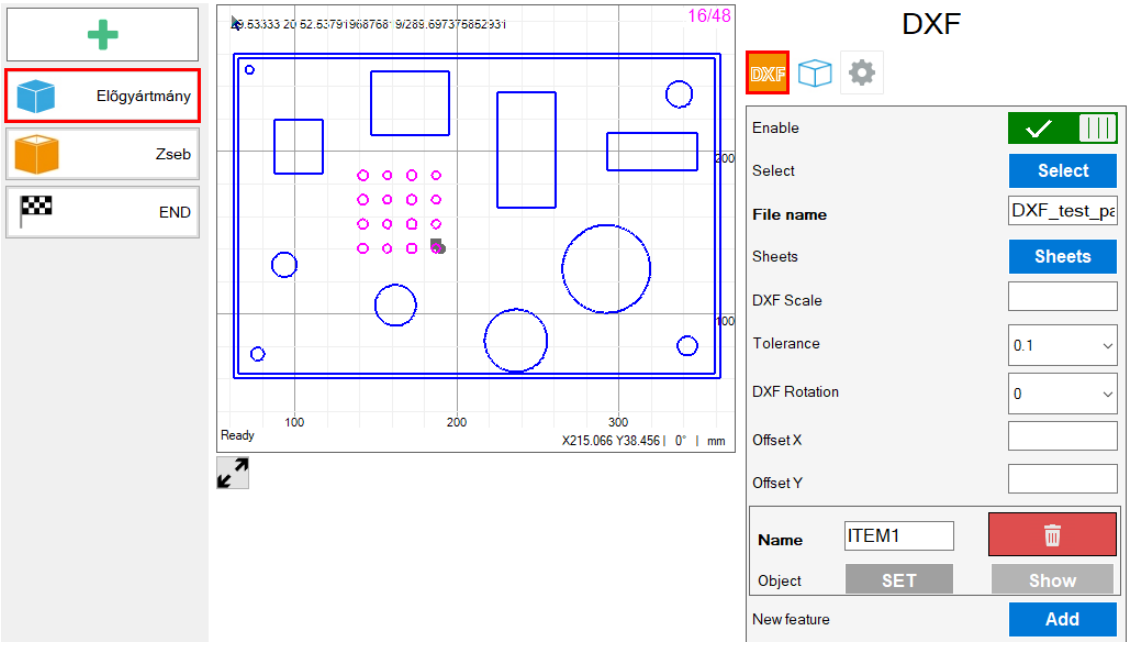
Remove: button with bin removes this feature

5.1.3 Multiple Selection

An element should be clicked on or an area should be bounded (drag) to select geometry (circle or line) in DXF control.



5-8. - - - signs indicate the selected area



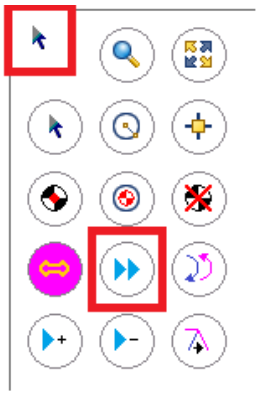
5-9. Figure The selected element's color is pink

5.1.4 Selecting items one by one

Click the mouse button in the upper left corner on DXF control. By default, the "Auto select connected elements" button is active. In this way, if a line is clicked, connected line(s) will also be selected.

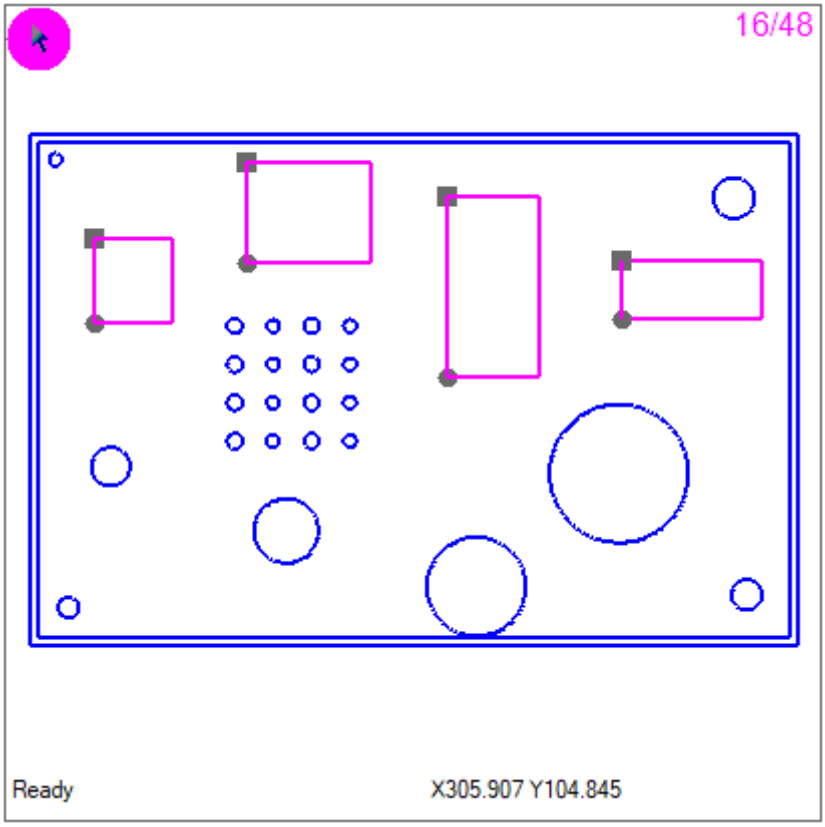
5-10. Figure „Auto select connected elements" on.

To select multiple not connected geometries, switch on the following icon:



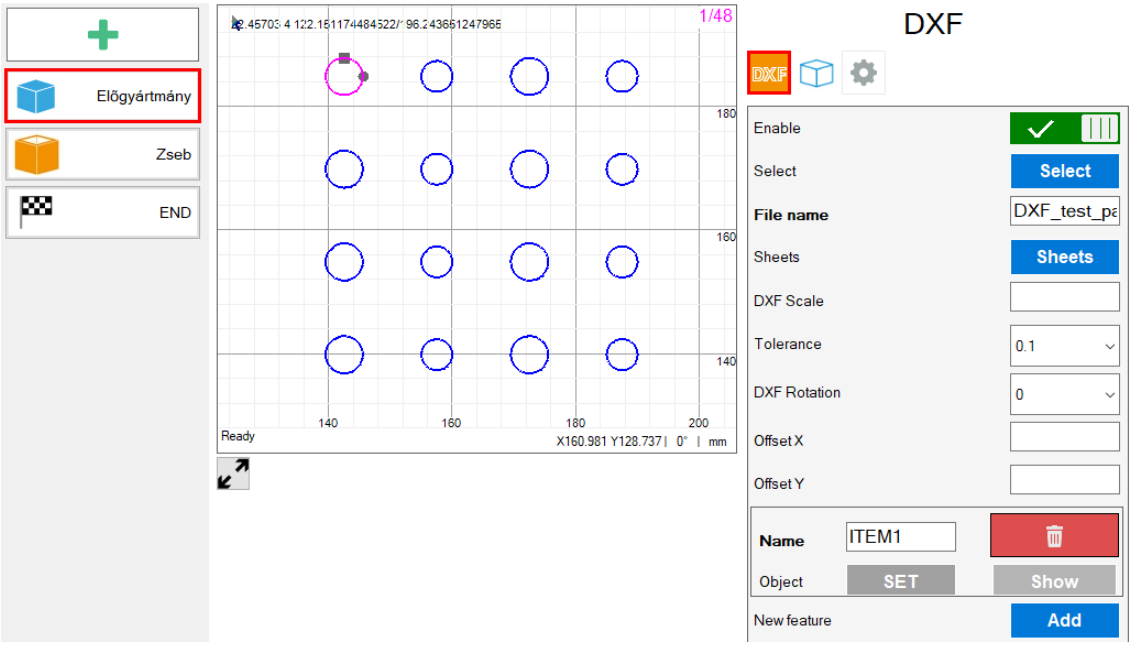
5-11. Figure Double click the cursor icon or click multi selection button

In multi selection mode, the mouse cursor's background color is pink.



5-12. Figure Active multi selection mode - the cursor's background is pink.

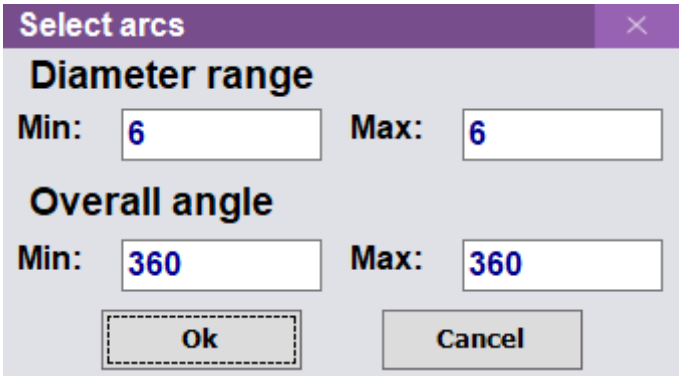
5.1.5 Selecting similar circles/arcs



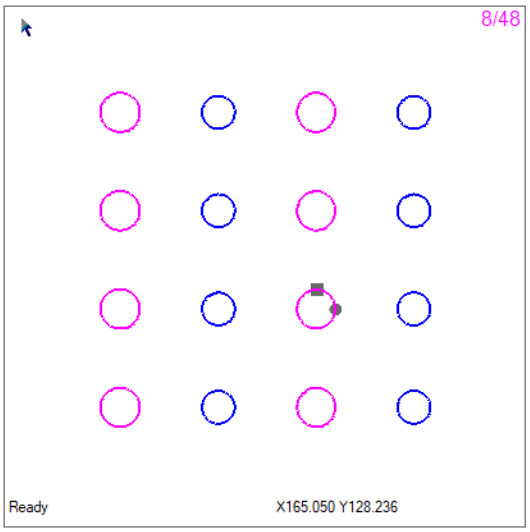
5-13. Figure Selecting a circle on the DXF content by touching or clicking it

5-14. Figure Click to cursor button to make DXF menu visible

5-15. Figure Search circles with same diameter

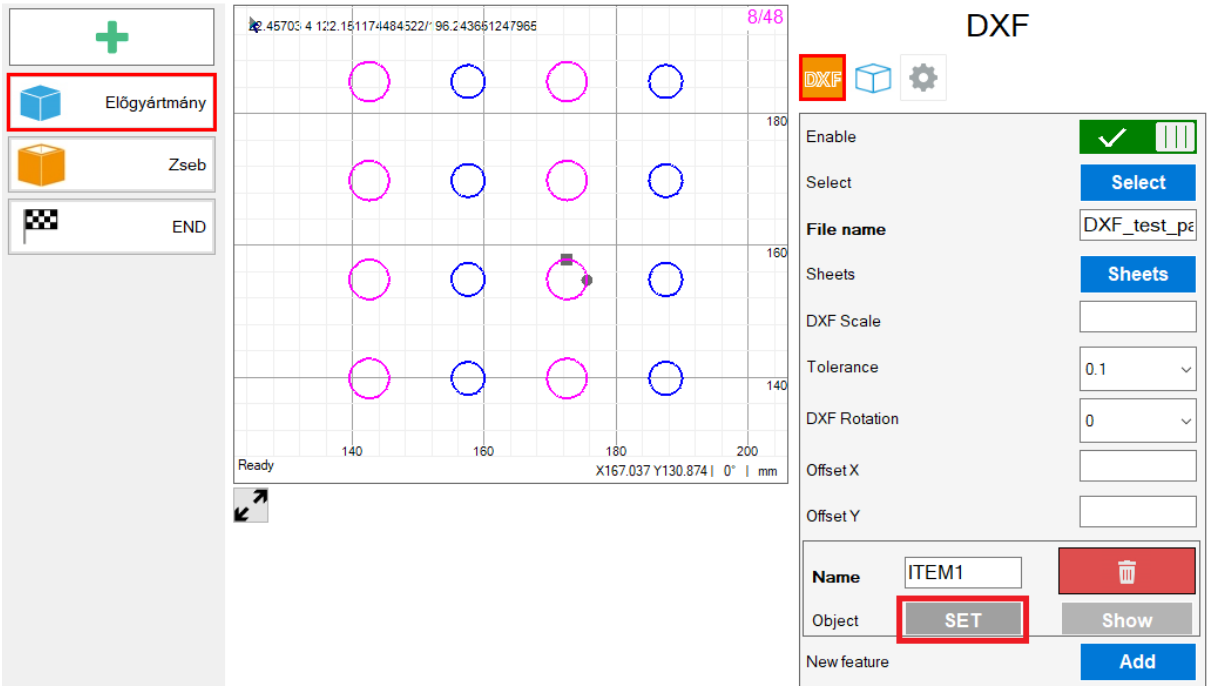


5-16. Figure The selected circles parameter automatically filled. Press „Ok".

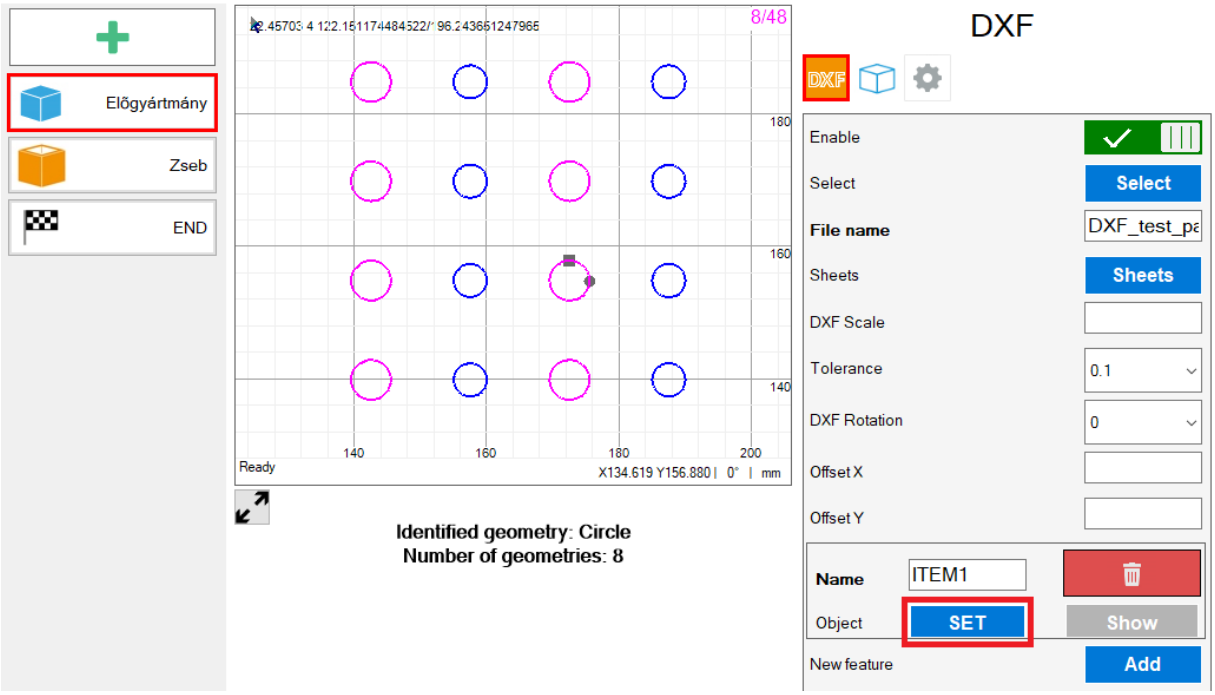


5-17. Figure All similar circles are selected.

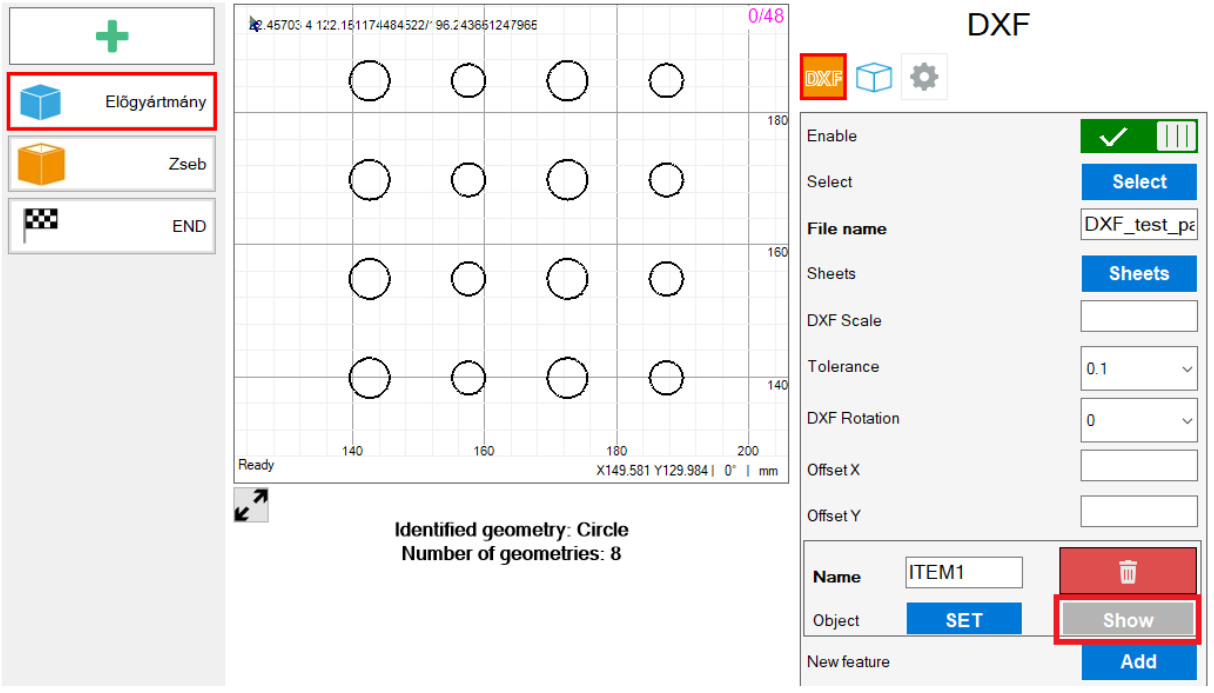
5.1.6 Set Objects



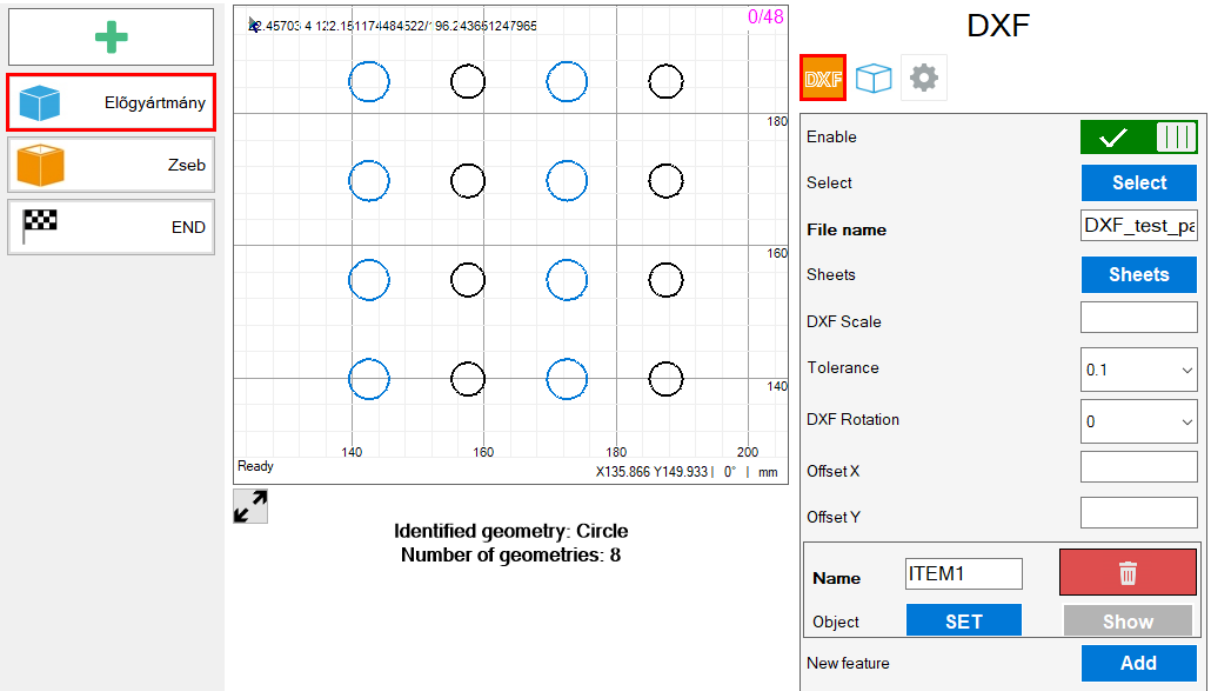
5-18. Figure Push „SET" to save the selected circles into the „ITEM1" named feature



5-19.Figure If an object is not empty, the SET button is blue



5-20.Figure Press Show to check which geometries are in this object



5-21. Figure DXF shows the object (blue color)

5.1.7 Set Markers (optional)

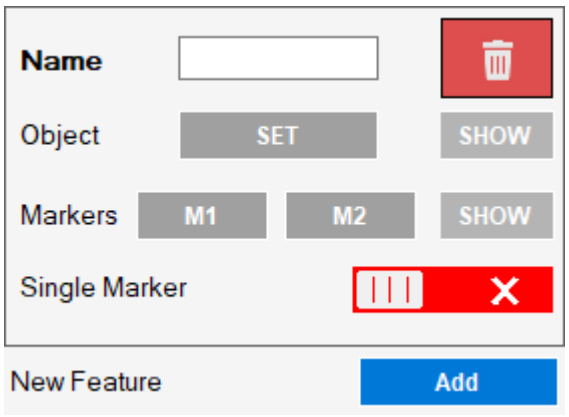
To use markers, the machine has to be equipped with a feature recognition camera system.

There are two ways of using markers. If single marker is chosen, each selected circle in an object has its own center point as a marker.

To use different marker points, you can set one ("M1" button), or two ("M1" and "M2" buttons) circles as markers.

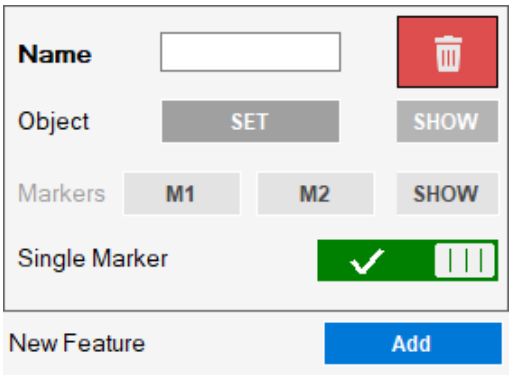
Set Single Marker:

Firstly, Single Marker is turned off and "Markers" buttons (M1, M2 & SHOW) are enabled.



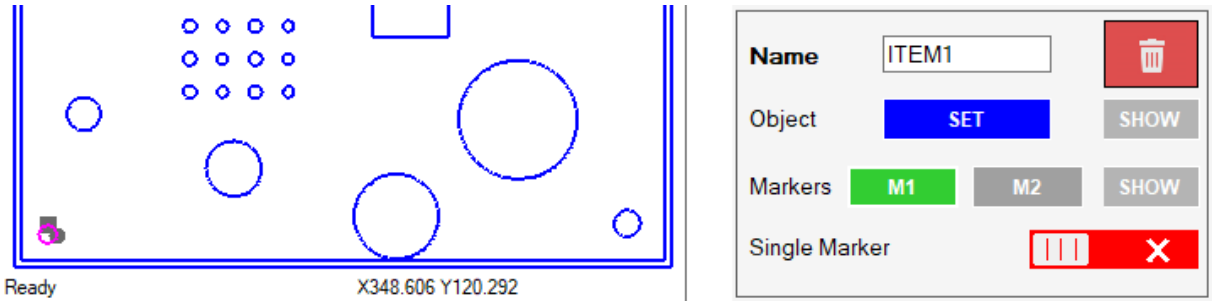
5-22. Figure

If „Single marker" is switched on, then each geometry is also a marker. When this mode is turned on, the „Markers" buttons (M1, M2 & SHOW) are disabled.

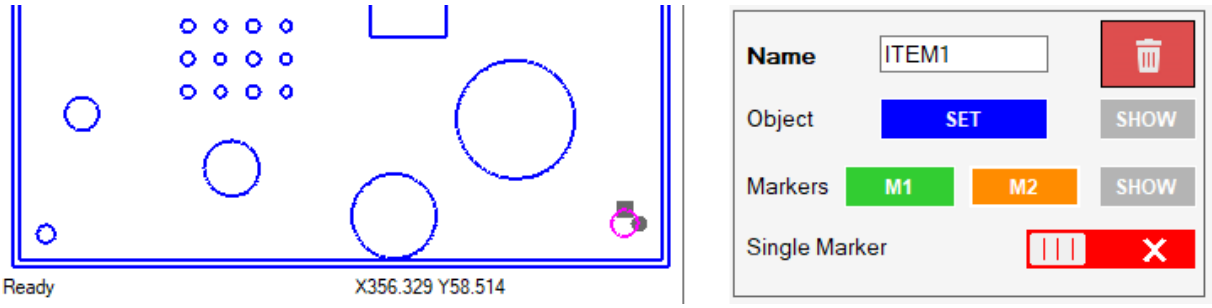


5-23. Figure Single Marker mode selected: objects are markers.

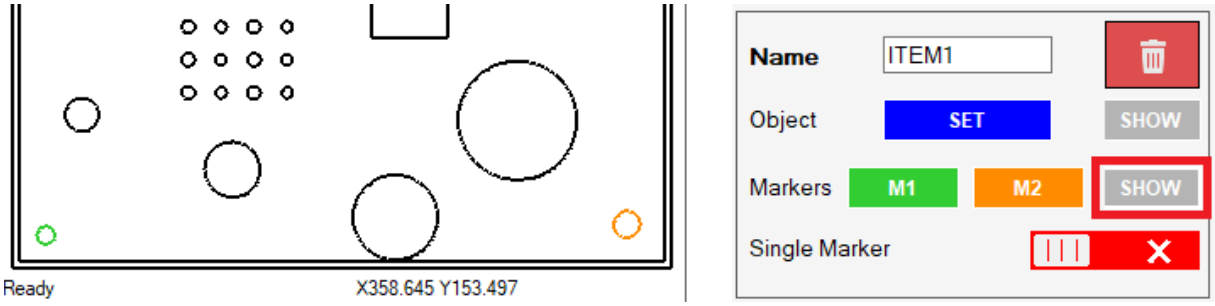
Set „M1" and „M2" markers:



5-24. Figure Select a circle and set it as the first marker („M1" button).



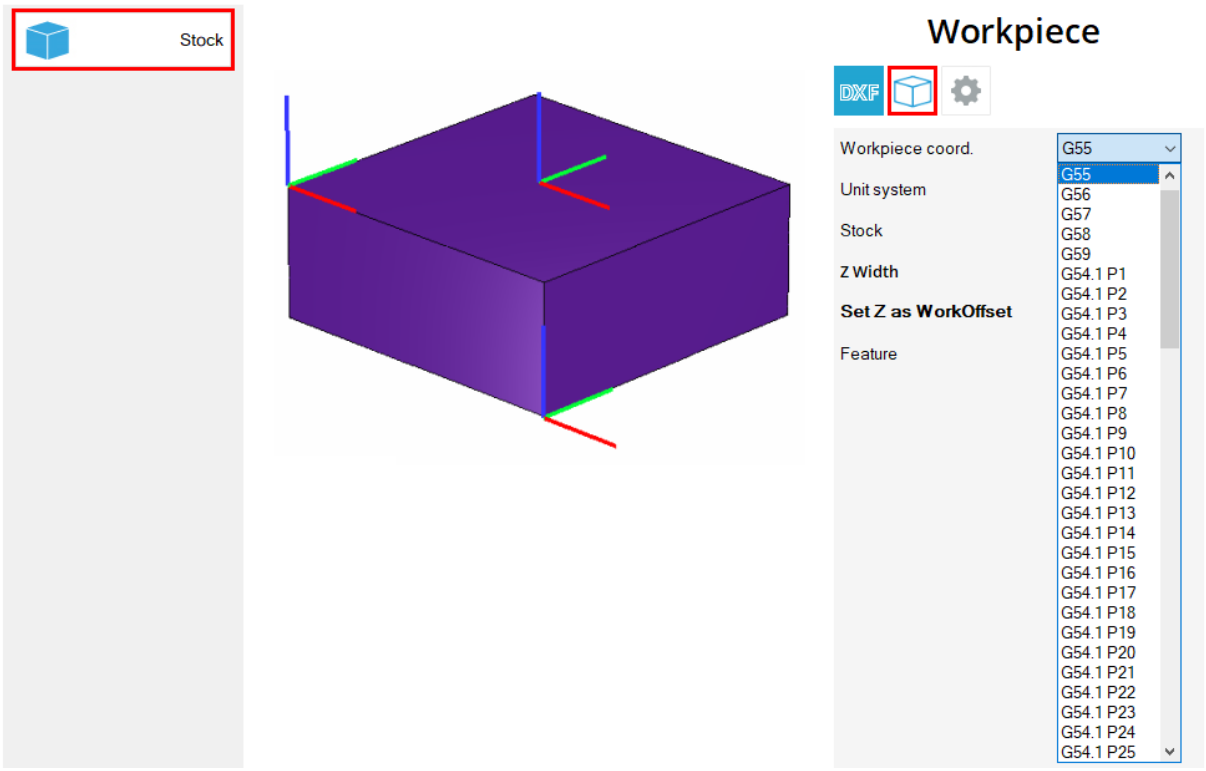
5-25. Figure Select another circle and set it as the second marker („M2" button).



5-26. Figure Clicked „SHOW" at the „Markers" line shows the saved items.

5.1.8 Features without marker but with option turned on

In case the software is desired to be used with marker option but without markers, do not use G54 because it is for measurement with camera. Select G55 or higher G value.

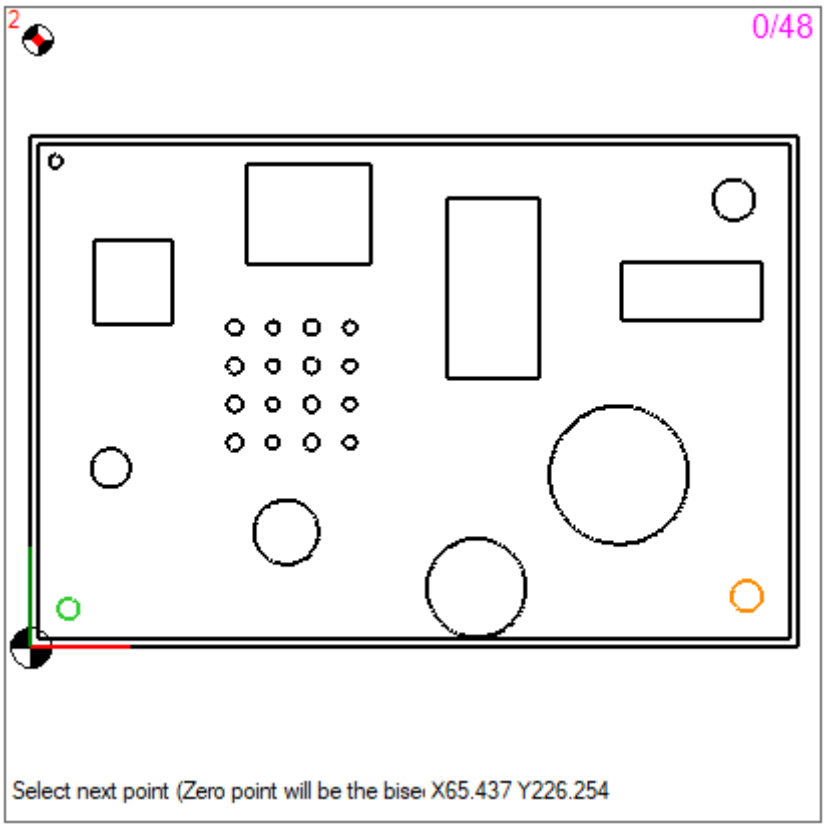


5-7. Figure Select G55 or higher

5.1.9 Modify DXF workpiece coordinate system

5-8. Figure Set the Coordinate System to (left to right):

1. selected point or midpoint (Details in the next picture)
2. a circle center point
3. original (DXF file) point



5-9. Figure Mind deselecting this function by using the right mouse button or select another menu in case midpoint is not needed

5.1.10 Rotation of DXF workpiece

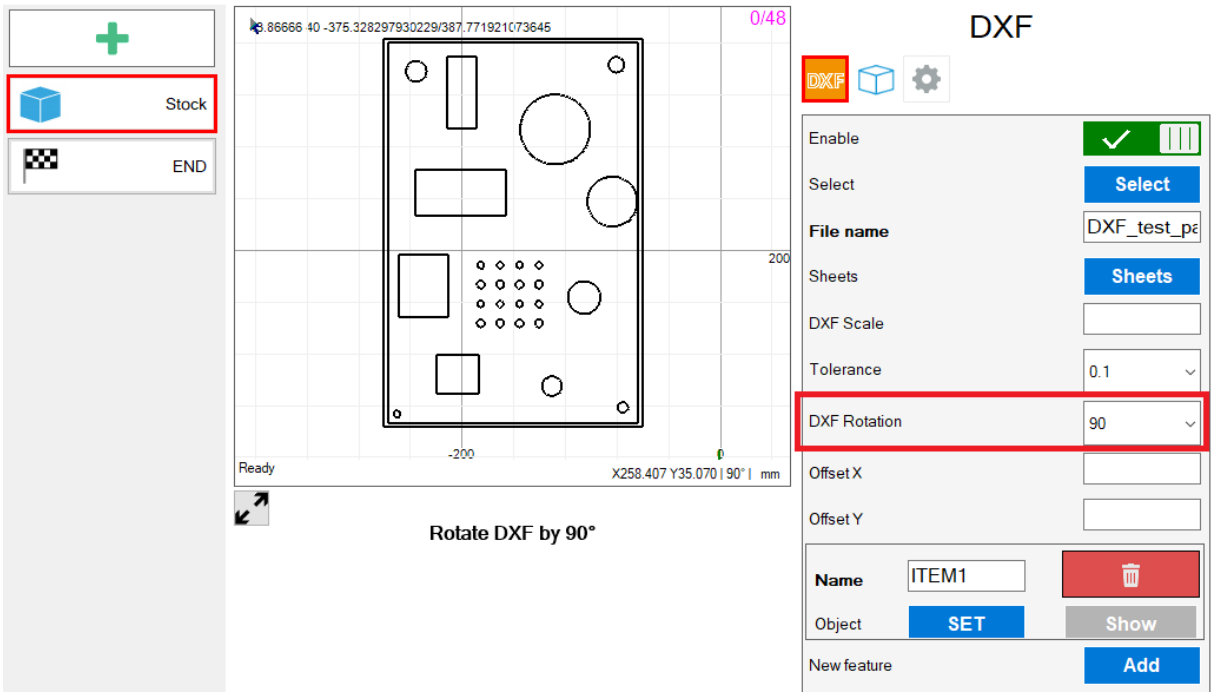


Figure 5-10 DXF after rotation

5.1.11 Displaying DXF in greater size

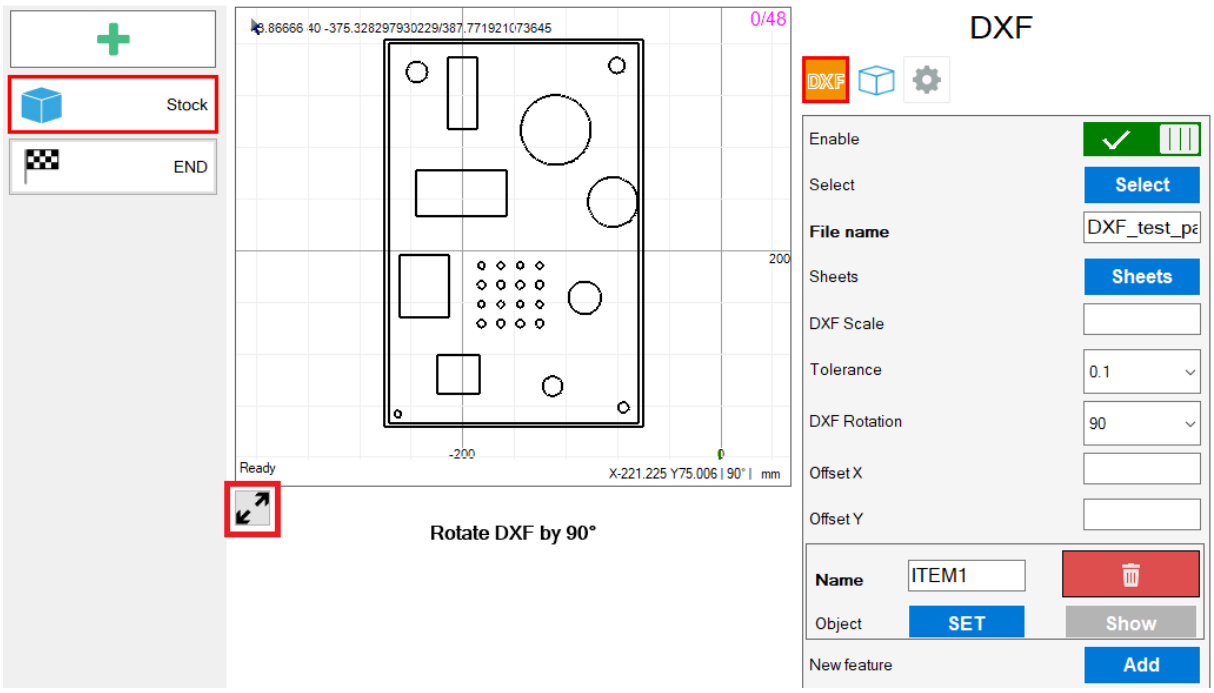


Figure 5-11 Button for opening DXF in a greater window

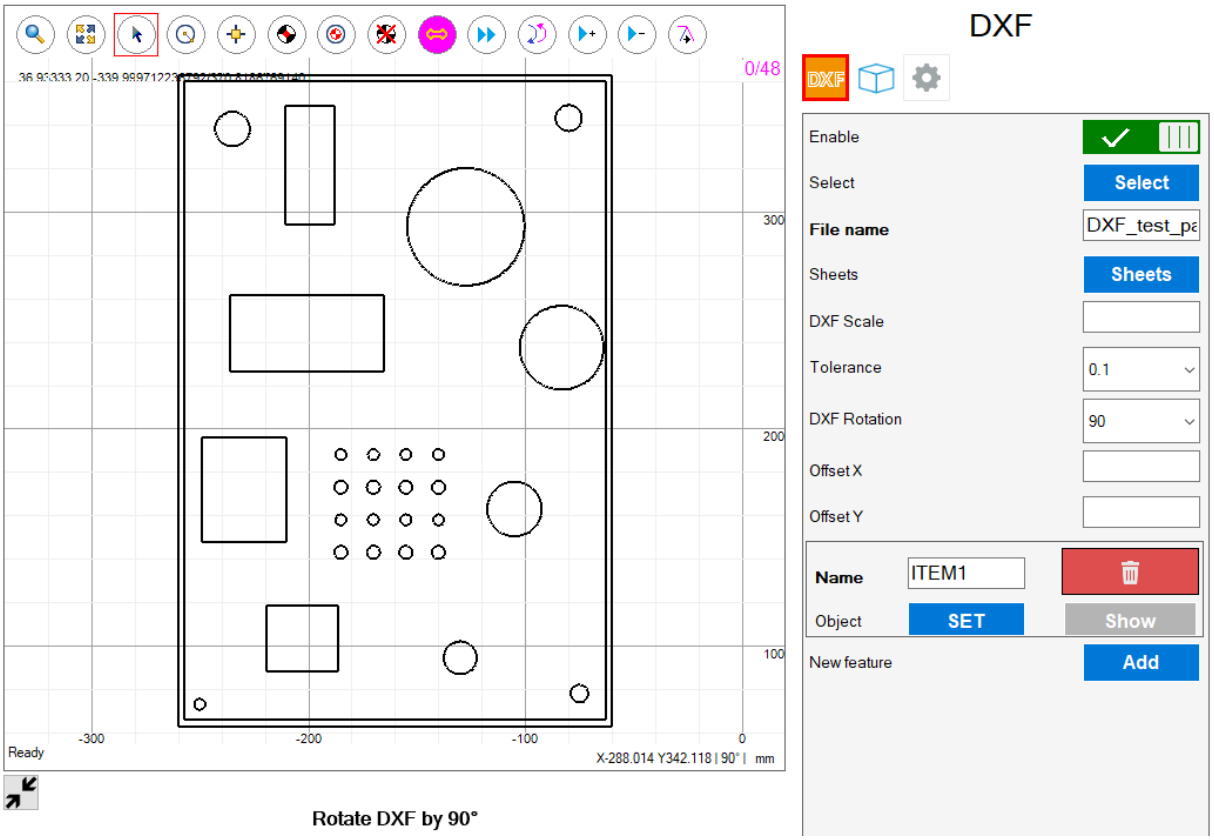
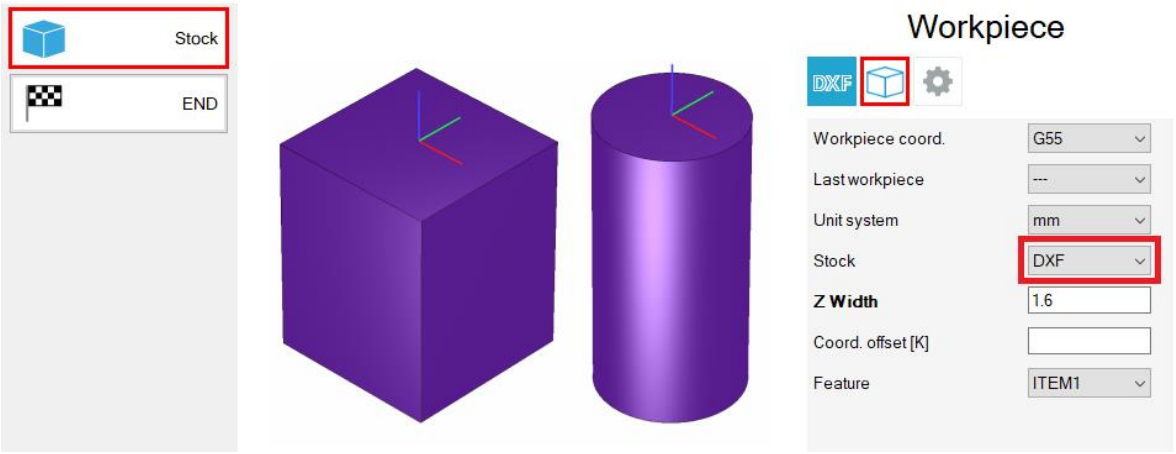


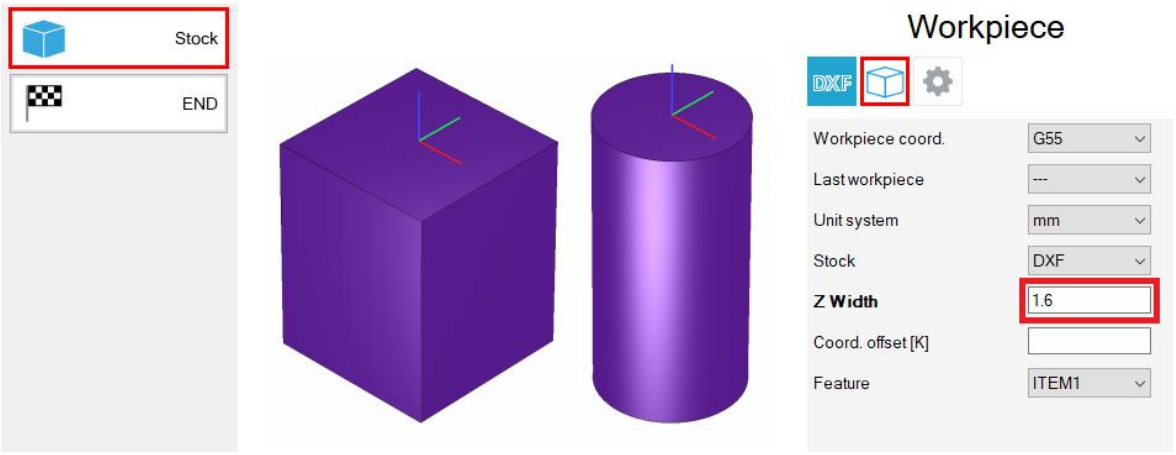
Figure 5-12 DXF in greater size

5.2 Use features

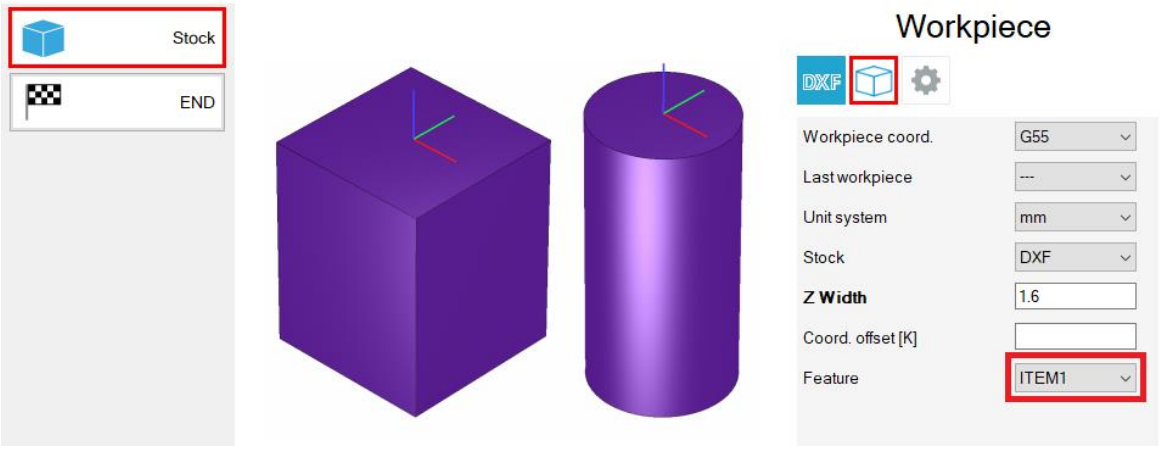
5.2.1 Features as stock



5-13. Figure DXF Stock is available in Workpiece panel

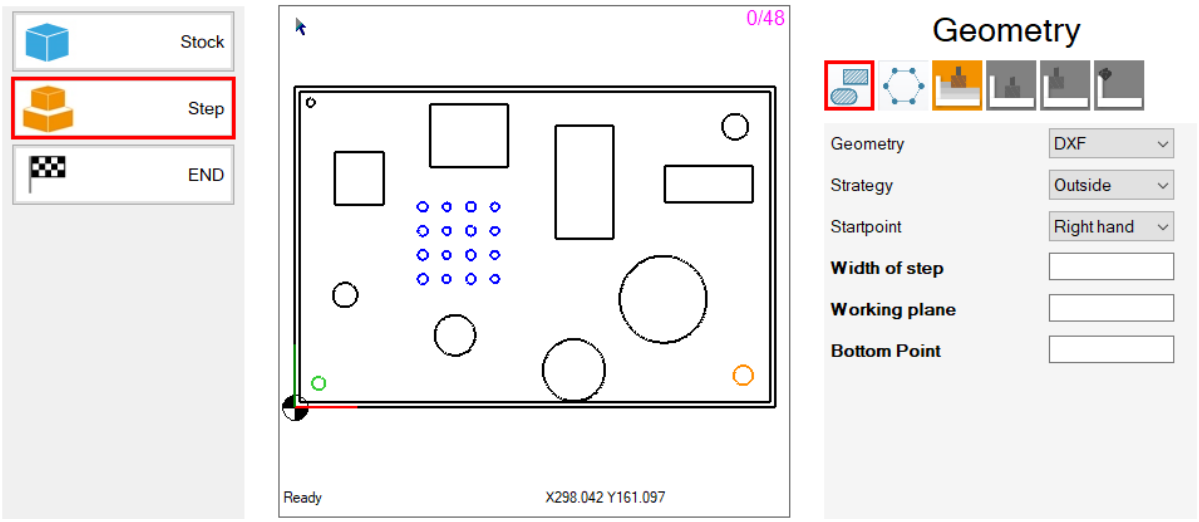


5-14. Figure Z Width of stock



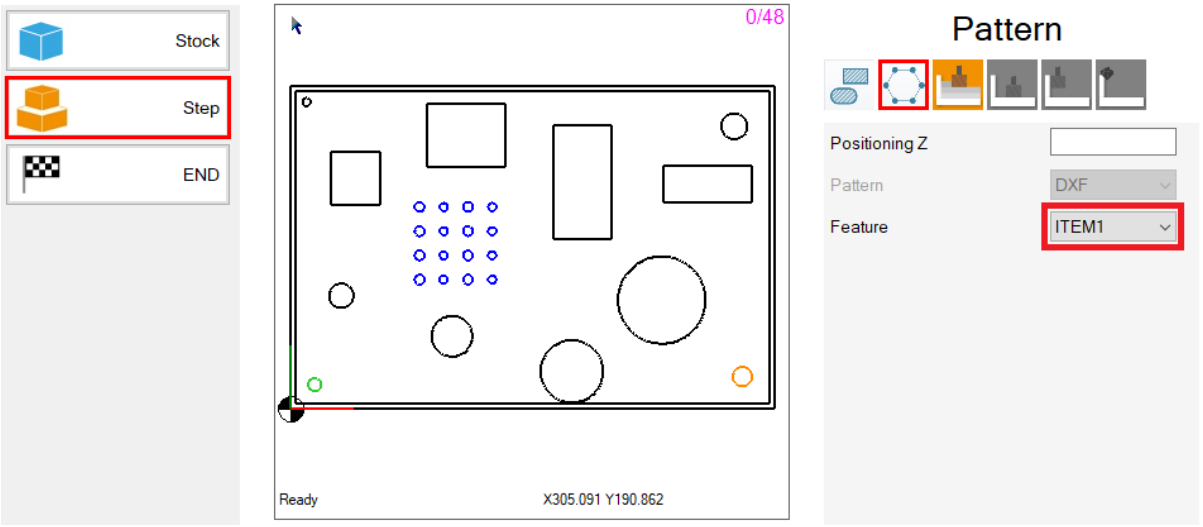
5-15. Figure Select which feature is used as Stock

5.2.2 Features as geometries



5-16. Figure Select DXF as geometry.

5.2.3 Features as patterns



5-17. Figure Select Features pattern and a feature. The selected feature is shown.

6 OPERATIONS

6.1 Common panels

6.1.1 Geometry panel

Geometry panel is for creating some simple geometries, like square, circle, track. Additionally, geometric size and other properties of these features may be set here.

6-1. Figure Geometry panel

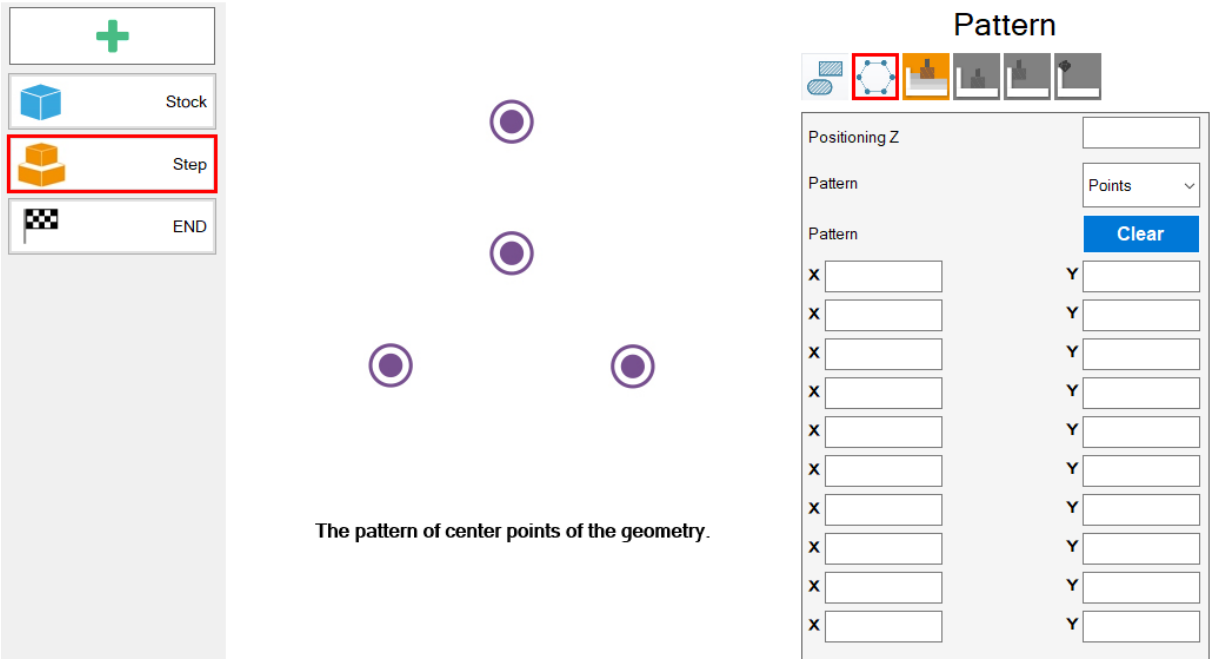
6.1.2 Pattern panel

Pattern panel is for calculating geometries' center points. It works in the same way in every operation.

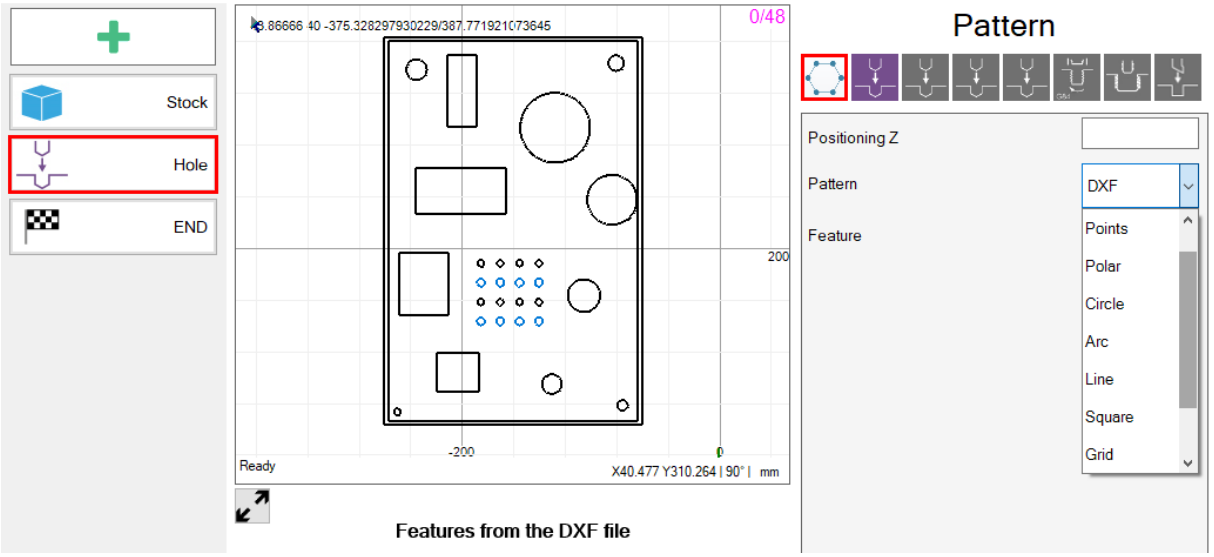
Common parameter:

Positioning Z: the safety position in Z axis, where the tool moves the next point. To avoid collisions, it is recommended to set this value so that the tool can move above other geometry features while travelling between pattern points. During positioning motion, rapid feed is applied.

Clone pattern should be highlighted as pattern of a former operation can be linked to the active operation. Application example: creating counterbore holes, holes may be drilled with *Drilling* (see at 6.2) operation, then cloning the pattern to *Pocket* (see at 6.4) or *Helical pocket* (see at 6.6) operation, counterboring can be quickly programmed.



6-1. Figure Pattern panel



6-2. Figure Available patterns.

6.1.3 Tool parameters

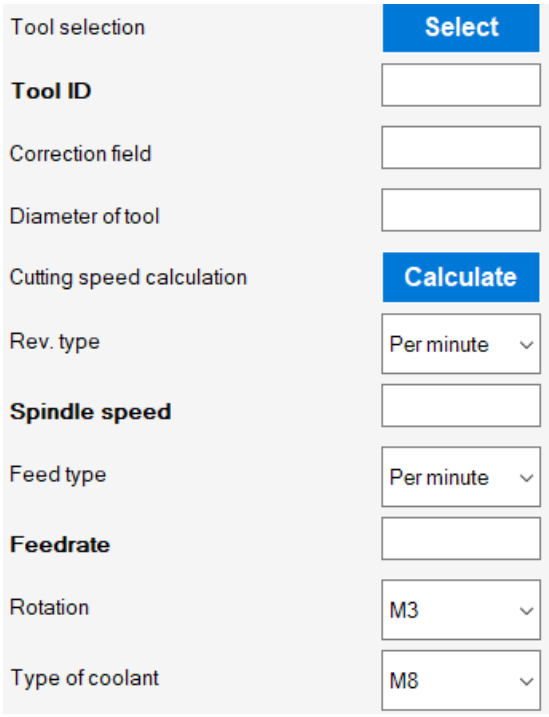
Drilling, milling, turning type panels has tool parameters. A tool with its details may be selected from tool offset table. In case tool management is used in a mill controller, default feedrate and spindle will also be set with the tool ID.

Required parameters:

- Tool selection,
- Tool ID (programmed as T)
- Spindle speed (programmed as S)
- Feedrate (programmed as F)
- Rotation direction
- Coolant type

Filling out correction field, the selected tool's correction may be altered from its default value which is stored under the same ID as the corresponding tool in the tool correction table.

Clicking on "Calculate" pops up a window for cutting speed calculation. By clicking on "OK", calculated parameters are written in their respective fields automatically.



6-3. Figure Tool parameters in a panel

6.2 Drilling

6-4. Figure Drilling panels

Drilling operation includes:

- 4 drilling cycles
- 1 tapping panel
- 1 thread milling panel
- 1 boring panel

Common and required parameters:

Start point [R]: Motion to this position is rapid, cutting feed is applied from here. Absolute coordinate programming. It is expedient determining a point above the hole's start to avoid rapid feed collisions.

End point [Z]: The end point of drilling. Absolute coordinate programming. Drill's tip moves to this point. As drills have nose angle, the hole's depth where its diameter is equal to the drill's diameter is shorter than the full depth of the hole.

6.2.1 Drilling panel

Drill type:

- One cut: Drilling holes in one cut. The drill moves uninterrupted to end point with cutting feed. It's a quick cycle but chips may get stuck and tool cooling is not really executable (except for internal tool cooling). It is not recommended for deep holes.
- Peck: After each cut, tool moves to the rapid point then moves back to an approaching point above the recently done cut with rapid feed. After that next cut is taken. Relatively low efficiency as takes more time thanks to more movements but chip breaking and tool cooling is superior.
- High-speed: Compromise between one cut and peck drilling. After each cut, tool is sent up to approaching distance only, then next cut is taken immediately. Short cycle time and chip breaking.
- Reaming: Tool is moved to end point in one cut. Contrary to the previous cycles, extraction is executed with programmed feedrate, not rapid feed.

Dwell [P]: Waiting quantity at the end point of the hole. Finished geometry (accurate, good surface quality) on the bottom is created after the tool rotates around for a few times. Measurement unit is second [s] if G94 is active, revolution if G95 is active.

Depth of Cut [Q]: Depth of cut in the current unit system. Chip breaking can be tuned with this parameter. In deep holes, smaller value is advised. Incremental value.

Distance to depth [E]: Approaching distance before cuts. In case of *High-speed* drilling, extraction between cuts is executed to this point up. Incremental value.

Extraction override [I]: Extraction feed may differ from cutting feed in reaming operation. Measurement unit is percent [%] that defines extraction feed to cutting feed ratio. At more than 100%, extraction feed is higher than cutting feed thus cycle time is decreased. Default value: 100%.

6.2.2 Tapping panel

Type of clamping:

- Rigid: tool is rigidly clamped into the tool holder
- Spring: tool is able to move in tool holder thanks to its design

Depth of Cut [Q]: Depth of cut in the current unit system. Chip breaking can be tuned with this parameter. In deep holes, smaller value is advised. Incremental value.

Distance to depth [E]: Approaching distance before cuts. In case of *High-speed* drilling, extraction between cuts is executed to this point up. Incremental value.

Extraction % [I]: Retraction feed may differ from cutting feedrate. Measurement unit is percent [%] that defines extraction feed to cutting feed ratio. At more than 100%, extraction feed is higher than cutting feed thus cycle time is decreased. Default value: 100%.

Drill type:

- Peck (see at 6.2.1)
- High-speed (see at 6.2.1)

6.2.3 Thread milling panel

Place of thread:

- Internal
- External

Type:

- Whitworth
- Whitworth Fine
- Metric
- Metric Fine
- UNC
- UNF
- UNEF
- BSPP

If thread type is chosen, corresponding nominal thread sizes are shown.

Absolute Nominal size: The nominal size in standard steps. If size is chosen, thread pitch, nominal size and core diameter are written into their input fields respectively.

Thread:

- Right hand
- Left hand

Type of tool:

- Single point: creates one pitch of thread at a time
- More: creates more pitches of thread at a time

Check distortion: If tool's diameter is close to the core diameter of the desired thread, a warning message is sent upon choosing "Yes".

Pitch of thread [F]: Pitch value, i.e. axial distance between neighboring crests or roots.

Absolute Nominal size: Nominal size of thread, e.g. M16 = 16 mm

Core diameter [X]: Also known as minor diameter for external threads. Radial distance of roots on external and radial distance of crests on internal threads.

Radial depth of cut: Depth of cut in radial direction. Finished thread roots may be created with more cuts. Tool load is reduced and thread quality is improved using smaller depth of cut although cycle time is increased. Default value: one cut

6.2.4 Boring panel

Extraction:

- Automatic: Extraction to starting point is done automatically.
- Manual: When bottom point is reached, the controller enters STOP state, then the operator can pull the tool away from the hole surface and pull the tool out of the hole after entering manual operation mode.

Spindle at extraction:

- Stopped: Spindle stops when it reaches bottom point and restarts after extracion.
- Rotating: Tool is extracted while spindle is rotating.

Shifting the tool:

- Yes: When bottom is reached, the tool is moved away from the hole surface before extraction according to the specified values [I, J].
- No: When bottom is reached, the tool is not moved away from the surface of the hole.

Direction of cut:

- Forwards: Boring starts from the top and goes downwards
- Backwards: Boring starts from the bottom and goes upwards

Dwell [P] (see at 6.2.1)

Relief X,Y [I,J]: Relief of the tool at the cutting bottom. Rapid feed is applied.

Extraction % [I]: Retraction feed may differ from cutting federate in case automatic retraction is enabled with rotating tool. Measurement unit is percent [%] that

defines extraction feed to cutting feed ratio. At more than 100%, extraction feed is higher than cutting feed thus cycle time is decreased. Default value: 100%.

6.3 Face

Used for face milling with different strategies.

6-5. Figure Face operation panels

Panels:

- geometry
- face milling

6.3.1 Geometry panel

Additional parameters:

Geometry:

- Workpiece: facing on the top plane of the stock
- Square
- Circle
- DXF: facing on selected feature

Bottom point: The deepest point where the tool works

Geometry: Square

X Width: Expansion in X direction.

Y Width: Expansion in Y direction.

Center X: X coordinate of center point of rectangle.

Center Y: Y coordinate of center point of rectangle.

X Frame: An extension can be drawn around the rectangle to create safety margins around it. The machining starts and ends at the edge of the rectangle which can be modified using this parameter to prevent high speed approaching (rapid feed is applied approaching start point and abandoning endpoint). Extension dimension in X direction.

Y Frame: An extension can be drawn around the rectangle to create safety margins around it. The machining starts and ends at the edge of the rectangle which can be modified using this parameter to prevent high speed approaching (rapid feed is applied approaching start point and abandoning endpoint). Extension dimension in Y direction.

Working plane: The start plane of the geometry.

Geometry: Circle

Center X: X coordinate of circle's center point.

Center Y: Y coordinate of circle's center point.

Width of ring: An extension can be drawn around the circle to create safety margins around it. The machining starts and ends at the edge of the circle which can be modified using this parameter to prevent high speed approaching (rapid feed is applied approaching start point and abandoning endpoint). Extension dimension in radial direction.

Radius: Radius of the circle to be machined.

Working plane: The start plane of the geometry.

6.3.2 Face milling

Direction of cut:

- X: cutting on paths parallel to X axis
- Y: cutting on paths parallel to Y axis

Tool path:

- Uni: Cutting in one way. Extraction between radial depth of cuts. Positioning to start point of next cut is often executed above non-machined planes.
- Bi: Cutting in two ways, back and forth. No extraction between the radial depth of cuts. Positioning to start point of next cut is executed along the same side of the workpiece where the last cut ended.
- Ring: Cutting in a spiral pattern. The most productive, tool is constantly in cut.

Approach position:

- E1: starting from the bottom right corner
- E2: starting from the top right corner
- E3: starting from the bottom left corner
- E4: starting from the top left corner

Radial depth of cut [C]: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%).

Depth of cut [J]: Axial depth of cut. Default value: one cut.

Clearance [K]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 3 mm.

Approach, abandon [M]: Entry and leaving distance. Tool approaches the border of the plane from this distance and stops after this much later. Default value: 5 mm.

6.4 Pocket

Used for milling the selected geometry from inside.

6-6. Figure Pocket operation panels

Panels:

- geometry
- pattern
- rough
- bottom finish
- side finish
- chamfer

6.4.1 Geometry panel

Geometry:

- Square
- Circle
- Track
- Groove
- DXF

Working plane: The start plane of the geometry.

Bottom point: The deepest point where the tool works, bottom coordinate of pocket.

Geometry: Square

X Width: Expansion in X direction.

Y Width: Expansion in Y direction.

Radius: Fillet radius in corners.

Rotation: Angle measured from X axis with positive angles in counter-clockwise direction. The geometry is rotated around its center by the given angle value. Measurement unit: degree. Default value: 0°. At 0°, sides of rectangle will be parallel to the axes.

Geometry: Circle

Radius: Radius of the circle to be machined.

Geometry: Track

Center distance: Distance of closing half circles' center points.

Radius: Radius of closing half circles. Double of this radius equals to the width of track.

Rotation: Angle measured from X axis with positive angles in counter-clockwise direction. The geometry is rotated around its center by the given angle value. Measurement

unit: degree. Default value: 0°. At 0°, straight sides of track will be parallel to X axis.

Geometry: Groove

Difference from Square: opposite two sides are open, no material is expected there.

Length: Length of closed sides.

Groove width: Length of open sides.

Rotation: Angle measured from X axis with positive angles in counter-clockwise direction. The geometry is rotated around its center by the given angle value. Measurement unit: degree. Default value: 0°. At 0°, closed sides of groove will be parallel to X axis.

Startpoint: Position of start point relative to groove.

- Left hand
- Right hand

Abandon: Overtravel of tool crossing open sides. After each cut, rapid feed is applied while moving to next cut.

6.4.2 Rough

Milling direction:

- Down
- Up

Radial depth of cut [C]: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

Depth of cut [J]: Axial depth of cut. Default value: one cut.

Approach radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Helical depth [S]: Axial depth of cut may be executed on straight or helical paths. Helical depth is the distance the tool lowers while completing one circle. Default value: inactive function, i.e. straight path is used.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Bottom finish [H]: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Side allowance [T]: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Z Feedrate [E]: Feed rate on vertical paths which is used when axial depth of cut is taken. Default value: value given in *Feedrate* input field.

6.4.3 Bottom finish

Milling direction:

- Down
- Up

Radial depth of cut [C]: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%).

Approach radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Helical depth [S]: Axial depth of cut may be taken on straight or helical paths. Helical depth is the distance the tool lowers while completing one circle. Default value: inactive function, i.e. straight path is used.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Side allowance [T]: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Z Feedrate [E]: Feed rate on vertical paths which is used when axial depth of cut is taken. Default value: value given in *Feedrate* input field.

6.4.4 Side finish

Milling direction:

- Down
- Up

Depth of cut [J]: Axial depth of cut. Default value: one cut.

Approach radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

6.4.5 Chamfer

Milling direction:

- Down
- Up

Chamfer amount [I]: Size of the chamfer.

Tool angle [Angle]: Tool's nose angle.

Out of depth [H]: Tool length which is below the chamfer's bottom edge. It can be tuned which part of the tool is cutting.

Approach radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

T small diameter [T]: The small diameter of the tool.

Z Feedrate [E]: Feed rate on vertical paths which is used when axial depth of cut is taken. Default value: value given in *Feedrate* input field.

6.5 Step

Used for milling the selected geometry from outside or a geometry which has an empty center region.

6-7. Figure Step operation panels

Panels:

- geometry
- pattern
- rough
- bottom finish
- side finish
- chamfer

6.5.1 Geometry Panel

Width of step: Step size created. Material removal is expected on this area around the final geometry. From another point of view, starting geometry size is defined by this parameter.

Working plane: The start plane of the geometry.

Bottom point: The deepest point where the tool works.

Geometry: Square

Strategy: Location of material removal.

- Outside: Step will be machined from outside, thus creating an island of rectangle shape.
- Inside: Step will be machined from inside, thus creating a frame of rectangle shape. Size means the length of inner sides in this case. Material is not expected to be in the very middle, otherwise *Pocket* operation is recommended.

Startpoint: Position of start point relative to feature.

- Right hand
- Left hand

X Width: Expansion in X direction.

Y Width: Expansion in Y direction.

Corner: Corner type.

- Fillet
- Chamfer

Amount of corner: Radius of fillet or size of chamfer on corners, according to the selected type in *Corner*. Sharp edge can be created writing zero.

Rotation: Angle measured from X axis with positive angles in counter-clockwise direction. The geometry is rotated around its center by the given angle value. Measurement unit: degree. Default value: 0°. At 0°, sides of rectangle will be parallel to the axes.

Geometry: Circle

Strategy: Location of material removal.

- Outside: Step will be machined from outside, thus creating an island of circle shape.
- Inside: Step will be machined from inside, thus creating a ring shape. Size means the radius of inner circle in this case. Material is not expected to be in the very middle, otherwise *Pocket* operation is recommended.

Startpoint: Position of start point relative to feature.

- Right hand
- Left hand

Radius: Radius of the circle to be machined.

Geometry: Track

Strategy: Location of material removal.

- Outside: Step will be machined from outside, thus creating an island of track shape.
- Inside: Step will be machined from inside, thus creating a frame of track shape. Size in this case means the size of a track that can be fitted in the frame. Material is not expected to be in the very middle, otherwise *Pocket* operation is recommended.

Startpoint: Position of start point relative to feature.

- Right hand
- Left hand

Center distance: Distance of closing half circles' center points.

Radius: Radius of closing half circles. Double of this radius equals to the width of track.

Rotation: Angle measured from X axis with positive angles in counter-clockwise direction. The geometry is rotated around its center by the given angle value. Measurement unit: degree. Default value: 0°. At 0°, straight sides of track will be parallel to X axis.

Geometry: One side

Length: Length of closed side.

Rotation: Angle measured from X axis with positive angles in counter-clockwise direction. The geometry is rotated around its center by the given angle value. Measurement unit: degree. Default value: 0°. At 0°, closed side of step will be parallel to X axis.

Approach: Start point may be moved away from the bounds of the feature thus preventing high speed approach. This is the distance between start point and edge of feature to be machined.

Abandon: Overtravel of tool at the end of cut. This is the distance between endpoint and edge of feature to be machined. Rapid feed is applied from then on.

6.5.2 Rough

Milling direction:

- Down
- Up

Radial depth of cut [I]: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

Depth of cut [J]: Axial depth of cut. Default value: one cut.

Approach radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Finishing allowance [H]: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Side allowance [T]: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Z Feedrate [E]: Feed rate on vertical paths which is used when axial depth of cut is taken. Default value: value given in *Feedrate* input field.

6.5.3 Bottom finish

Milling direction:

- Down
- Up

Radial depth of cut [I]: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

Approach radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Side allowance [T]: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological

parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Z Feedrate [E]: Feed rate on vertical paths which is used when axial depth of cut is taken. Default value: value given in *Feedrate* input field.

6.5.4 Side finish

Milling direction:

- Down
- Up

Depth of cut [J]: Axial depth of cut. Default value: one cut.

Approach radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

6.5.5 Chamfer

Milling direction:

- Down
- Up

Chamfer amount [I]: Size of the chamfer.

Tool angle [Angle]: Tool's nose angle.

Out of depth [H]: Tool length which is below the chamfer's bottom edge. It can be tuned which part of the tool is cutting.

Approach radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

T small diameter [T]: The small diameter of the tool.

6.6 Helical

It is used for milling the selected geometry from inside. The tool is moving on a helical path.

6-8. Figure Helical operation panels

Panels:

- geometry
- pattern
- rough
- bottom finish
- side finish
- chamfer

6.6.1 Geometry Panel

Radius: Value of the radius of pocket which always has a circle base.

Working plane: The start plane of the geometry.

Bottom point: The deepest point where the tool works.

6.6.2 Rough

Strategy:

- Helical: Milling down helically. With more radial depth of cuts, more helical paths are used.
- Depth: Cutting at full depth. After a single helical movement to the bottom, material removal is executed staying there.

Milling direction:

- Down
- Up

Radial depth of cut [C]: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

First cut [J]: Value of first radial depth of cut relative to the tool diameter as a percentage value (max value: 99%).

Helical depth [S]: Axial depth of cut may be taken on straight or helical paths. Helical depth is the distance the tool lowers while completing one circle. Default value: inactive function, i.e. straight path is used.

Abandon radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Bottom finish [H]: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Side allowance [T]: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

6.6.3 Bottom finish

Milling direction:

- Down
- Up

Radial depth of cut [C]: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

First cut [J]: Value of first radial depth of cut relative to the tool diameter as a percentage value (max value: 99%).

Helical depth [S]: Axial depth of cut may be taken on straight or helical paths. Helical depth is the distance the tool lowers while completing one circle. Default value: inactive function, i.e. straight path is used.

Abandon radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Side allowance [T]: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

6.6.4 Side finish

Milling direction:

- Down
- Up

Helical depth [S]: Axial depth of cut may be taken on straight or helical paths. Helical depth is the distance the tool lowers while completing one circle. Default value: inactive function, i.e. straight path is used.

Abandon radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Bottom finish [H]: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

6.6.5 Chamfer

Milling direction:

- Down
- Up

Chamfer amount [I]: Size of the chamfer.

Tool angle [Angle]: Tool's nose angle.

Out of depth [H]: Tool length which is below the chamfer's bottom edge. It can be tuned which part of the tool is cutting.

Approach radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

T small diameter [T]: The small diameter of the tool.

Z Feedrate [E]: Feed rate on vertical paths which is used when axial depth of cut is taken. Default value: value given in *Feedrate* input field.

6.7 Helical step

It is used for milling the selected geometry from outside or a geometry which has an empty center region. The tool is moving on a helical path.

6-9. Figure Helical step operation panels

Panels:

- geometry
- pattern
- rough
- bottom finish
- side finish
- chamfer

6.7.1 Geometry panel

Starting radius: Step is formed from this radius size of a circle. Step shape is always a circle.

Finished radius Step is formed to this radius size of a circle. Step shape is always a circle.

Working plane: The start plane of the geometry.

Bottom point: The deepest point where the tool works.

6.7.2 Rough

Strategy:

- Helical: Milling down helically. With more radial depth of cuts, more helical paths are used.
- Depth: Cutting at full depth. After a single helical movement to the bottom, material removal is executed staying there.

Milling direction:

- Down
- Up

Radial depth of cut [C]: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

First cut [J]: Value of first radial depth of cut relative to the tool diameter as a percentage value (max value: 99%).

Helical depth [S]: Axial depth of cut may be taken on straight or helical paths. Helical depth is the distance the tool lowers while completing one circle. Default value: inactive function, i.e. straight path is used.

Abandon radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Bottom finish [H]: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Side allowance [T]: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

6.7.3 Bottom finish

Milling direction:

- Down
- Up

Radial depth of cut [C]: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

Helical depth [S]: Axial depth of cut may be taken on straight or helical paths. Helical depth is the distance the tool lowers while completing one circle. Default value: inactive function, i.e. straight path is used.

Abandon radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Side allowance [T]: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

6.7.4 Side finish

Milling direction:

- Down
- Up

Helical depth [S]: Axial depth of cut may be taken on straight or helical paths. Helical depth is the distance the tool lowers while completing one circle. Default value: inactive function, i.e. straight path is used.

Abandon radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Bottom finish [H]: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

6.7.5 Chamfer

Milling direction:

- Down
- Up

Chamfer amount [I]: Size of the chamfer.

Tool angle [Angle]: Tool's nose angle.

Out of depth [H]: Tool length which is below the chamfer's bottom edge. It can be tuned which part of the tool is cutting.

Approach radius [K]: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance [M]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

T small diameter [T]: The small diameter of the tool.

Z Feedrate [E]: Feed rate on vertical paths which is used when axial depth of cut is taken. Default value: value given in *Feedrate* input field.

6.8 Engrave

It is used for engraving characters, serial number or a contour of a DXF file

6-10. Figure Engrave operation panels

Panels:

- geometry
- pattern
- engrave

6.8.1 Geometry Panel

Clearance: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that.

Height: Height of characters in the current unit system.

Working plane: The start plane of the geometry.

Bottom point: The deepest point where the tool works.

6.8.2 Engrave

Type:

- Character: Engraved text consisting of letters, numbers and other marks.
- Serial number: Serial number between 1 and 999 999 999.
- DXF: Engraving selected as a contour from DXF drawing.

Text: Text to be engraved.

Place of text:

- Left hand: Text start point is in the selected coordinate.
- Center: Text center is in the selected coordinate.

First number: The first and smallest value of the serial number

Highest value: The last and greatest value of the serial number

Count of characters: Number of digits of the smallest number (Value: 1 .. 9)

Global macro variable: The global variable contains the actual number. (Value between #200-#499 or #500-#999)

0 strikethrough:

- Yes (Ø)
- No (0)

Arc:

- No: Characters are laid on a line next to each other.
- Yes: Characters are laid on an arc with their baseline on the smaller circle radius.
- Oppositely: Characters are laid on an arc with their baseline on the bigger circle radius.

Radius: Characters are laid on an arc of this radius. Radius is measured on the centerline of characters.

First angle: Angle of the first character. Character's center will be on this point but its left side will be to the left even more. Measurement: in degrees starting from X axis with positive angles in counter-clockwise direction.

Last angle: Angle of the last character. Character's center will be on this point but its right side will be to the right even more. Measurement: in degrees starting from X axis with positive angles in counter-clockwise direction.

6.9 DXF Contour

It is used for milling DXF contour.

6-11. Figure DXF Contour operation panels

If step contour is disabled, then pocket milling will be executed.

If both contours are enabled and correct, then milling will be executed inside the pocket contour but outside the step contour. In other words, a pocket is milled in the area bounded by pocket and step contours.

Panels:

- pocket
- step
- pattern
- contour roughing
- finishing
- chamfer

6.9.1 Pocket panel

Working plane: The start plane of the geometry.

Bottom point: The deepest point where the tool works.

6.9.2 Step panel

Working plane: The start plane of the geometry.

Bottom point: The deepest point where the tool works.

6.9.3 Contour rough

Extraction between cuts: Setting to extract the tool 0.2 mm from the bottom during connecting moves. Without extraction, cycle time is decreased but bottom plane may be scratched.

- Yes
- No

Clearance: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Helical start depth: Below this point, vertical milling is done on a helical path. Incremental value measured from the bottom plane. Default value: given value in *Working plane* input field.

Minimum cutting radius: The smallest permitted tool path radius. Having value greater than zero may leave material in corners. However, in corners with same radius as the tool radius, vibrations may occur.

First cut radius: Radius of helical path during vertical movement.

Prebore diameter: Optionally, a prebore may be drilled where the first vertical milling will be started. Consequently, this hole will be handled as an empty volume during this operation's machining.

Ramp angle: Slope of tool path during vertical helical movement. Measurement unit: degree.

Radial depth of cut: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Default value: 10%.

Depth of cut: Axial depth of cut. Default value: one cut.

Bottom finish: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Side allowance: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Helical feedrate: Applied feed during helical movement.

Rapid feedrate: Applied feed during connecting moves (near the bottom). Default: feedrate.

Approach radius: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

6.9.4 Contour finish

Contour type:

- Closed
- Opened

Approach radius: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

In case of closed contour:

Approach type:

- Line
- Arc

Milling location:

- Inside
- Outside
- Contour

Milling direction:

- Down
- Up

Approach ID: The ID of the contour element to approach.

Approach overlap: Overlap between approach and abandon.

In case of opened contour:

Contour tracking:

- Left hand
- Right hand
- On contour

To selection: Relation of machining direction and order of contour elements.

- Same way
- Opposite way

Extension: Extend contour with straight lines before the first and after the last element.

Tolerance of straight lines: If arcs are split into lines, tolerance is the greatest permitted distance of created lines and original arc.

Interpolation: Joining short lines that are near to each other.

- Off
- Smoothing: Interpolated curve strictly goes through every end point of straight lines. Advantage of this method is that there aren't sudden movement direction shifts and so results in better surface quality and less wear of mechanical components and tool. However, tool path differs from contour defined by straight lines.
- Fine smoothing: Interpolated curve only approaches end points. In other aspects, it is like *Smoothing* on an even more ideal curve and greater difference from original contour.

Strategy:

- One radial cut
- Multiple radial cuts

Radial depth of cut: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

Width of step: In case of multiple radial cuts is active. (Default value: 0 mm)

Tool correction pos.: Tool diameter correction position temporarily used to store diameter for multiple radial cuts

Bottom finish: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Side allowance: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Clearance: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Depth of cut: Axial depth of cut. Default value: one cut.

6.9.5 Chamfer

Contour type:

- Closed
- Opened

Approach radius: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

In case of closed contour:

Approach type:

- Line
- Arc

Milling direction:

- Down
- Up

Milling direction:

- Inside
- Outside
- Contour

In case of opened contour:

To selection: Relation of machining direction and order of contour elements.

- Same way
- Opposite way

Contour tracking:

- Left hand
- Right hand
- On contour

Chamfer amount: Size if the chamfer.

Tool angle: Tool's nose angle.

Contact point: The part of the tool that works on the bottom edge of the chamfer. 100% is the greatest, 0% is the smallest diameter of the tool.

Approach radius: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

T small diameter: The small diameter of the tool.

6.10 Simple groove

Operation for creating simple grooves with both ends open.

6-12. ábra Simple groove operation panels

Panels:

- rough
- side finish

6.10.1 Rough

Clearance: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Working plane: The start plane of the geometry.

Bottom point: The deepest point where the tool works.

Depth of cut: Axial depth of cut. Default value: one cut.

Rapid feedrate: Applied feed during connecting moves.

- G0 (rapid feed)
- G1 (programmed cutting feed)

Milling direction:

- Down
- Up

Startpoint [X]: Groove machining starts at this X coordinate. Startpoint is on the longitudinal centerline of the groove. It is recommended to determine this point before the groove's real longitudinal startpoint as connecting movement are executed along this coordinate.

Startpoint [Y]: Groove machining starts at this Y coordinate. Startpoint is on the longitudinal centerline of the groove. It is recommended to determine this point before the groove's real longitudinal startpoint as connecting movement are executed along this coordinate.

Endpoint [X]: Groove machining ends at this X coordinate. End point is on the longitudinal centerline of the groove. It is recommended to determine this point after the groove's real longitudinal end point as connecting movement are executed along this coordinate.

Endpoint [Y]: Groove machining ends at this Y coordinate. End point is on the longitudinal centerline of the groove. It is recommended to determine this point after the

groove's real longitudinal end point as connecting movement are executed along this coordinate.

Groove width: Size perpendicular to the groove's length. Default value: equal to tool's diameter.

Radial depth of cut: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

6.10.2 Side finish

Used for finishing the sides of the groove.

Input fields: see at *6.10.1*.

6.11 Curved groove

It can be used to make curved grooves. The groove may be specified by its length or its central angle.

6-13. Figure Curved groove operation panels

Panels:

- geometry
- pattern
- rough
- bottom finish
- side finish

6.11.1 Geometry panel

Geometry: Submission type of curved groove geometry.

- With angle
- With length

Angle: Specify the groove with center angle. If the angle is more than 360 °, a complete annular groove is machined. Measurement unit: degree.

Length: Specify the groove with its length. Length of a circular arc cannot be less than or equal to the width of groove.

Division circle radius: Radius of the centerline of the groove.

Groove width: Distance between closed sides of the groove, size perpendicular to its length.

Working plane: The start plane of the geometry.

Bottom point: The deepest point where the tool works.

6.11.2 Pattern panel

It is worth highlighting that pitch circle's center point of each groove will be programmed on the circle/arc determined on the pattern panel. So, to create more grooves on the same pitch circle, then input field *Radius* should be set zero on this panel.

6.11.3 Rough panel

Radial depth of cut [E]: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

Depth of cut [Q]: Axial depth of cut. Default value: one cut.

Clearance [K]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Bottom finish [W]: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Side allowance [U]: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Feed override (%): Used for drilling-type movements and for milling longitudinally while the tool is loaded on both sides. This speed is relative to cutting feed as a percentage value. Default value: 50%.

6.11.4 Bottom finish panel

Radial depth of cut [E]: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

Clearance [K]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Side allowance [U]: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Feed override (%): Used for drilling-type movements and for milling longitudinally while the tool is loaded on both sides. This speed is relative to cutting feed as a percentage value. Default value: 50%.

6.11.5 Side finish panel

Depth of cut [Q]: Axial depth of cut. Default value: one cut.

Clearance [K]: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Bottom finish [W]: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Feed override (%): Used for drilling-type movements and for milling longitudinally while the tool is loaded on both sides. This speed is relative to cutting feed as a percentage value. Default value: 50%.

6.12 Simple geometry

Features bounded by one, two or three planes may be machined using this operation.

Figure 6-14 Simple geometry panels

Panels:

- Geometry
- Rough
- Bottom finish
- Side finish
- Chamfer

6.12.1 Geometry panel

Geometry:

- One wall
- Two walls
- Three walls

Geometry (position):

- Right hand
- Down
- Left hand
- Up

Startpoint X: X coordinate of a specific point of the geometry, according to help image.

Startpoint Y: Y coordinate of a specific point of the geometry, according to help image.

Fillet: Corner fillet radius in case of two or three walls is selected.

Length: Geometric size along the first axis.

Width: Geometric size along the second axis.

Working plane: The start plane of the geometry.

Bottom point: The deepest point where the tool works.

6.12.2 Rough panel

Radial depth of cut: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

Depth of cut: Axial depth of cut. Default value: one cut.

Clearance: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Bottom finish: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Side allowance: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

6.12.3 Bottom finish panel

Radial depth of cut: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

Clearance: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Side allowance: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

6.12.4 Side finish panel

Depth of cut: Axial depth of cut. Default value: one cut.

Clearance: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Bottom finish: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

6.12.5 Chamfer panel

Chamfer amount: Size of the chamfer.

Tool angle: Tool's nose angle.

Out of depth: Tool length which is below the chamfer's bottom edge. It can be tuned which part of the tool is cutting.

Clearance: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

T small diameter: The small diameter of the tool.

6.13 Contour

It is used for drawing and milling free contour.

6-15. Figure Contour operation panels

If step contour is disabled, then pocket milling will be executed.

If both contours are enabled and correct, then milling will be executed inside the pocket contour but outside the step contour. In other words, a pocket is milled in the area bounded by pocket and step contours.

Panels:

- pocket
- step
- pattern
- contour roughing
- finishing
- chamfer

6.13.1 Pocket panel

The contour can be edited on this panel.

On the left side, the contour may be edited with the following buttons:

 Add a new point to the end of the contour.

 Remove selected contour from the list.

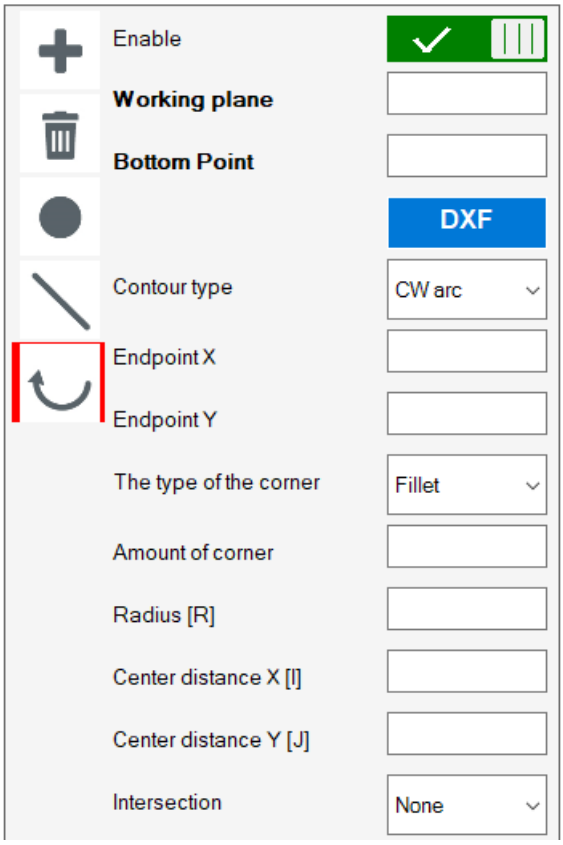
The contour points take place under these two buttons listed above. The icons of the contour point indicate its type.

 The point is start point of the contour.

 The point is endpoint of a line segment.

 The point is endpoint of an arc.

When a point is selected, inputs for that point appear on the right side of the panel.



6-16. Figure Contour panel

Working plane: The start plane of the geometry.

Bottom point: The deepest point where the tool works.

Contour type:

- Straight (G1)
- CW arc: clockwise (G2)
- CCW arc: counter-clockwise (G3)

Intersection: Line-line, line-arc, arc-arc intersection calculation.

6.13.2 Step panel

See at: *6.13.1 Pocket panel*

6.13.3 Contour rough

Extraction between cuts: Setting to extract the tool 0.2 mm from the bottom during connecting moves. Without extraction, cycle time is decreased but bottom plane may be scratched.

- Yes
- No

Clearance: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Helical start depth: Below this point, vertical milling is done on a helical path. Incremental value measured from the bottom plane. Default value: given value in *Working plane* input field.

Minimum cutting radius: The smallest permitted tool path radius. Having value greater than zero may leave material in corners. However, in corners with same radius as the tool radius, vibrations may occur.

First cut radius: Radius of helical path during vertical movement.

Prebore diameter: Optionally, a prebore may be drilled where the first vertical milling will be started. Consequently, this hole will be handled as an empty volume during this operation's machining.

Ramp angle: Slope of tool path during vertical helical movement. Measurement unit: degree.

Radial depth of cut: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Default value: 10%.

Depth of cut: Axial depth of cut. Default value: one cut.

Bottom finish: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Side allowance: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Helical feedrate: Applied feed during helical movement.

Rapid feedrate: Applied feed during connecting moves (near the bottom). Default: feedrate.

Approach radius: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

6.13.4 Contour finish

Contour type:

- Closed
- Opened

Approach radius: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

In case of closed contour:

Approach type:

- Line
- Arc

Milling location:

- Inside
- Outside
- Contour

Milling direction:

- Down
- Up

Approach ID: The ID of the contour element to approach.

Approach overlap: Overlap between approach and abandon.

In case of opened contour:

Contour tracking:

- Left hand
- Right hand
- On contour

To selection: Relation of machining direction and order of contour elements.

- Same way
- Opposite way

Extension: Extend contour with straight lines before the first and after the last element.

Tolerance of straight lines: If arcs are split into lines, tolerance is the greatest permitted distance of created lines and original arc.

Interpolation: Joining short lines that are near to each other.

- Off
- Smoothing: Interpolated curve strictly goes through every end point of straight lines. Advantage of this method is that there aren't sudden movement direction shifts and so results in better surface quality and less wear of mechanical components and tool. However, tool path differs from contour defined by straight lines.
- Fine smoothing: Interpolated curve only approaches end points. In other aspects, it is like *Smoothing* on an even more ideal curve and greater difference from original contour.

Strategy:

- One radial cut
- Multiple radial cuts

Radial depth of cut: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

Width of step: In case of multiple radial cuts is active. (Default value: 0 mm)

Tool correction pos.: Tool diameter correction position temporarily used to store diameter for multiple radial cuts

Bottom finish: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Side allowance: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Clearance: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Depth of cut: Axial depth of cut. Default value: one cut.

6.13.5 Chamfer

Contour type:

- Closed
- Opened

Approach radius: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

In case of closed contour:

Approach type:

- Line
- Arc

Milling direction:

- Down
- Up

Milling direction:

- Inside
- Outside
- Contour

In case of opened contour:

To selection: Relation of machining direction and order of contour elements.

- Same way
- Opposite way

Contour tracking:

- Left hand
- Right hand
- On contour

Chamfer amount: Size of the chamfer.

Tool angle: Tool's nose angle.

Contact point: The part of the tool that works on the bottom edge of the chamfer. 100% is the greatest, 0% is the smallest diameter of the tool.

Clearance: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

T small diameter: The small diameter of the tool.

6.14 Turning

It is used for generating longitudinal (G77), simple thread (G78) and face (G79) turning cycles

6.14.1 Turning Panel

Direction of cut:

- Face turning
- Longitudinal
- Thread turning

X Startpoint: X coordinate of the point from where cut is taken. Movements to this point are at rapid feed.

Z Startpoint: Z coordinate of the point from where cut is taken. Movements to this point are at rapid feed.

X Endpoint: X coordinate of the point where machining ends.

Z Endpoint: Z coordinate of the point where machining ends.

Depth of cut: Depth of cut in the current unit system and direction. Measured in diameter if it is written behind in parenthesis (*Depth of cut (Diameter)*), otherwise in radius.

6.15 Contour turning

It is used for generating G71, G72, G73 roughing contour turning cycles and contour finishing.

6-17. Figure Contour turning operation panels

Panels:

- contour
- rough
- finishing

6.15.1 Contour Panel

The contour can be edited on this panel.

On the left side, the user can press buttons to:

 Add a new point to the end of the contour.

 Remove selected contour from the list.

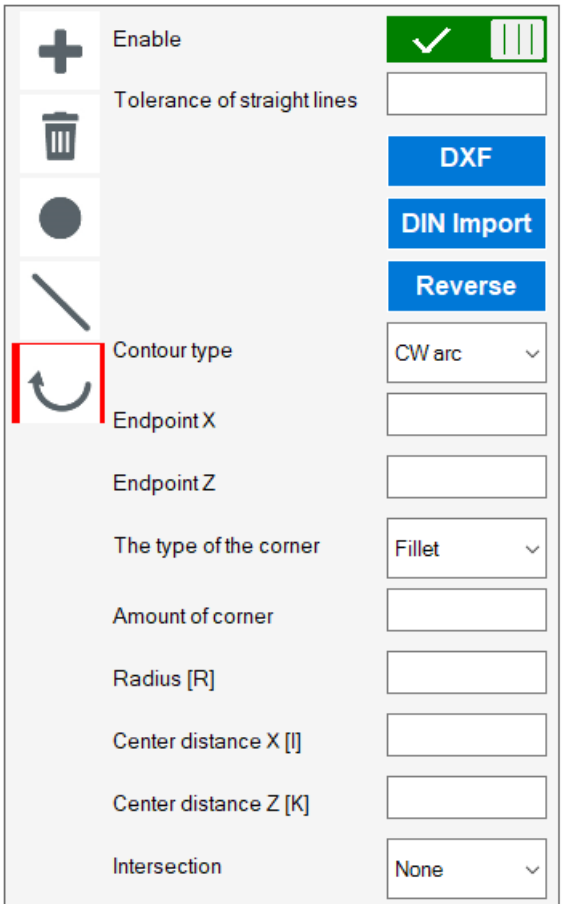
The contour points take place under these two buttons listed above. The icons of the contour point indicate its type.

 The point is a start point of the contour.

 The point is an end point of a line segment.

 The point is an end point of an arc.

When a point is selected, the inputs for that point appear on the right side of the panel.



6-18. Figure Contour panel

Tolerance of straight lines: Arcs may be split into line segments so that the distance between the newly created lines and the original arcs is the given value at max. Default: inactive function, arcs are used.

DXF: Opening DXF contour.

DIN Import: DIN509-E, DIN509-F standard undercuts at shoulders, DIN76 standard thread undercuts may be pasted quickly into edited contour.

Reverse: Reverse order of contour elements as well as machining direction.

Contour type:

- Straight (G1)
- CW arc: clockwise (G2)
- CCW arc: counter-clockwise (G3)

Start chamfer: A 45° chamfer is pasted before the contour's start point with the given size (*Start chamfer value*).

Intersection: Line-line, line-arc, arc-arc intersection calculation.

6.15.2 Rough Panel

On this panel the user can specify the type and the parameters of the roughing.

Direction of cut:

- G71: Longitudinal roughing cycle
- G72: Face roughing cycle
- G73: Pattern repeating roughing turning cycle

Rough:

- Complete
- Skip pockets
- Rest rough

X Startpoint: X coordinate of the point from where cut is taken. Movements to this point are at rapid feed.

Z Startpoint: Z coordinate of the point from where cut is taken. Movements to this point are at rapid feed.

Depth of cut (Radius): Depth of cut in the current unit system.

Escape: After completing a cut, the tool is moved away from the machined plane, then to the point where the next depth of cut is taken. It is a positive number given in radius. Default value: 0.5 mm.

X Allowance (Diameter): Finishing allowance in radial direction. Value should be given in radius.

Z Allowance: Finishing allowance in axial direction.

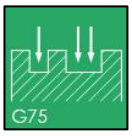
6.15.3 Finishing Panel

On this panel, myNCT generates only a finishing pass on the contour edited earlier.

Rapid approach of axes:

- Together: approach before the operation and abandon after the operation with simultaneous use of X and Z axis.
- X axis first: during approach, X axis moves to start point first. During abandon, X axis moves later.
- Z axis first: during approach, Z axis moves to start point first. During abandon, Z axis moves later.

6.16 Grooving



It is used for generating axial (G74) and radial (G75) grooving cycles.

6.16.1 Grooving Panel

Direction of cut:

- Radial (G75)
- Axial (G74)

Rapid approach of axes:

- Together: approach before the operation and abandon after the operation with simultaneous use of X and Z axis.
- X axis first: during approach, X axis moves to start point first. During abandon, X axis moves later.
- Z axis first: during approach, Z axis moves to start point first. During abandon, Z axis moves later.

X Startpoint: X coordinate of one of the upper corner of the groove.

Z Startpoint: X coordinate of one of the upper corner of the groove.

X Endpoint: X coordinate of the corner diagonally opposite to the start point. If G74 is active, default width of groove equals to tool's width.

Z Endpoint: Z coordinate of the corner diagonally opposite to the start point. If G75 is active, default width of groove equals to tool's width.

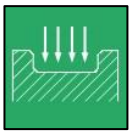
X Depth of cut (Radius): Depth of cut in direction of X axis. Default: one cut.

Z Depth of cut (Radius): Depth of cut in direction of Z axis. Default: one cut.

Escape: After completing a cut, the tool is moved away from the machined plane, then to the point where the next depth of cut is taken. It is a positive number given in radius. Default value: 0.5 mm.

Relief: Amount with which the tool is moved away from the machined plane at the bottom. Default value: 0.1 mm.

6.17 Conical groove



Radial (external and internal) and axial (face or rear plane) groove can be made with customized and independent chamfers and fillets on all four corners of the groove, with slanted (conical) sides.

Firstly, the middle part of the groove is machined, then its sides with chamfers/fillets. In case of the optional finishing, the tool moves alongside the contour.

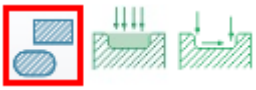


Figure 6-19 Conical groove panels

Panels:

- Geometry
- Rough
- Finishing

6.17.1 Geometry panel

Geometry:

- External: radial groove, depth direction: *-X*
- Internal: radial groove, depth direction: *+X*
- Axial front: axial groove, depth direction: *-Z*
- Axial back: axial groove, depth direction: *+Z*

X corner point/Finished diameter: X coordinate of the corner on the bottom of groove. Value is measured as diameter or radius values according to the axis' setup.

Z corner point: Z coordinate of the corner on the bottom of groove.

Groove initial coordinate: Coordinate of top plane of groove. It is an X coordinate in case of radial groove, and Z coordinate in case of axial groove.

Groove width: Groove width measured on its bottom. If sides are slanted, this differs from the width measured on the initial plane. Default value: tool width.

First chamfer or fillet: Chamfer or fillet on one top corner of the groove. If it's a positive value then fillet, if it's negative then chamfer is programmed. Default value: 0 (sharp edge).

Second chamfer or fillet: Chamfer or fillet on one top corner of the groove. If it's a positive value then fillet, if it's negative then chamfer is programmed. Default value: 0 (sharp edge).

Third chamfer or fillet: Chamfer or fillet on one bottom corner of the groove. If it's a positive value then fillet, if it's negative then chamfer is programmed. Default value: 0 (equal to tool's corner radius).

Fourth chamfer or fillet: Chamfer or fillet on one bottom corner of the groove. If it's a positive value then fillet, if it's negative then chamfer is programmed. Default value: 0 (equal to tool's corner radius).

Left/Lower side banking: Angle between vertical (radial groove) or horizontal (axial groove) axis and the groove's left (radial groove) or lower (axial groove) side, measured in degrees. Default value: 0 degrees.

Right/Upper side banking: Angle between vertical (radial groove) or horizontal (axial groove) axis and the groove's right (radial groove) or upper (axial groove) side, measured in degrees. Default value: 0 degrees.

6.17.2 Rough panel

Tool cutting width: Width of used grooving tool.

Tool cutting length: Maximal allowed material thickness to be removed with the tool's side edge.

Safety margin: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Always measured as a radius value. Default value: 2 mm.

Depth of cut in direction of depth: Depth of cut used when the middle part of the groove is machined. Always measured as a radius value. Default value: one cut.

Depth of cut in direction of width: Depth of cut taken parallel to the tool's width. Always measured as a radius value. Default value: 95% of the tool's width subtracted the two tool corner radius values.

Side finishing allowance: Allowance on the sides of the groove. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Bottom finishing allowance: Allowance on the bottom of the groove. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Rapid approach of axes:

- Together: approach before the operation and abandon after the operation with simultaneous use of X and Z axis.
- X axis first: during approach, X axis moves to start point first. During abandon, X axis moves later.
- Z axis first: during approach, Z axis moves to start point first. During abandon, Z axis moves later.

6.17.3 Finishing panel

Tool cutting width: Width of used grooving tool.

Tool cutting length: Maximal allowed material thickness to be removed with the tool's side edge.

Safety margin: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Always measured as a radius value. Default value: 2 mm.

Rapid approach of axes:

- Together: approach before the operation and abandon after the operation with simultaneous use of X and Z axis.
- X axis first: during approach, X axis moves to start point first. During abandon, X axis moves later.
- Z axis first: during approach, Z axis moves to start point first. During abandon, Z axis moves later.

6.18 Thread

It is used for generating straight or tapered thread. The panel has predefined thread types and sizes, which when chosen, automatically fill the appropriate input fields.

6.18.1 Thread Panel

Place of thread:

- Internal
- External

Type:

- Whitworth
- Whitworth Fine
- Metric
- Metric Fine
- UNC
- UNF
- UNEF
- BSPP

Absolute Nominal size: The nominal size in standard steps. If thread type is chosen, several options are listed.

X Startpoint: Automatically filled if thread type and size are chosen.

X Endpoint: Automatically filled if thread type and size are chosen.

Height of thread: Automatically filled if thread type and size are chosen.

Z Startpoint: It is recommended selecting a point before the thread's real start point by a few pitches. Tool arrives here at rapid feed.

Z Endpoint: At this point the tool is pulled out completely.

Thread profile angle: Automatically filled if thread type is chosen.

Minimum cutting depth: Cutting quality can be improved by degressive depth of cuts (i.e. decreasing cutting depth with each cut). Minimum value is needed to prevent too little cutting depths. It affects the number of cuts required.

The first infeed: This is the biggest cut. From here, cuts decrease in value. It affects the number of cuts required.

Retract: Retraction after finishing a cut and before connecting moves. Default value: 2 mm.

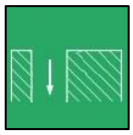
Thread pullout: Tool is pulled out on a distance proportional to the given value. This value determines the ratio between pullout and the one tenth of thread pitch (i.e. pullout length is one tenth of pitch multiplied by the given value). Default value: 0 mm (perpendicular pullout).

Finishing count: Number of finishing passes. Default value: 0.

Finishing allowance: Finishing allowance in radial direction. Should be given in radius. Default value: 0 mm.

Tapering: Distance of start and endpoint in X direction, given in radius. If value is positive, endpoint is on a smaller radius than the start point, if it is negative, vice versa. In other words, for X coordinates, start point is equal to endpoint plus tapering value. Default value: 0, i.e. cylindrical thread.

6.19 Parting off



Used for separating the finished part from the clamped part.

6.19.1 Parting off panel

X Startpoint: The X coordinate where the tool arrives at rapid feed, and from where feedrate is applied.

Z Startpoint: The Z coordinate where the tool arrives at rapid feed, and from where feedrate is applied.

2. region startpoint: From this X coordinate, modified movement parameters are applied. Default value: no modification validated.

Modified feed: Feedrate in 2nd region. Default value: *Feedrate*.

Modified spindle: Spindle speed in 2nd region. Default value: *Spindle speed*.

Modified cutting speed: Cutting feed in 2nd region. Default value: *Cutting speed*.

Modified max spindle: Maximum spindle speed in 2nd region. Default value: *Max spindle*.

X End point: Tool is moved to this point. It is advised writing a value a bit less then zero in case of complete parting off.

Clearance: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Always measured as a radius value. Default value: 0 mm.

Chamfer amount: Chamfer may be made at the beginning. Given value's sign defines chamfer direction (positive: *+Z* direction, negative: *-Z* direction) and absolute value determines its size. Default value: 0 mm, no chamfer.

Retract: Relief along Z axis after end point has been reached, before tool is pulled out at a rapid feed. Default value: 0 mm.

6.20 Polygon

Used for milling regular polygons with any number of sides and one or two side flattening using driven tool. Centerpoint of polygon is on *X=0* straight, on the axis of rotation.

Figure 6-20 Polygon panels

Panels:

- Geometry
- Rough
- Bottom finish
- Side finish
- Chamfer

6.20.1 Geometry panel

Geometry:

- Polygon
- Flat

Geometry: Polygon

Number of sides: Number of sides of a regular polygon is three, there is no upper limit.

First Angle: One corner's angle measured from X axis. Angle is positive in counter-clockwise direction. Measurement unit: degree. Defalut value: 0°.

Starting diameter: Diameter of initial cylinder, from which the polygon will be machined.

Type:

- Incircle: Size given by diameter of incircle. If polygon has an even number of sides, this value is equal to the distance of oppposite sides.
- Excircle: Size given by diameter of excircle. If polygon has an even number of sides, this value is equal to the distance of oppposite corners.

Size: Geometric size of polygon according to chosen *Type*.

Corner:

- Fillet: creating fillets on each corner.
- Chamfer: creating chamfers on each corner.

Amount of corner: Geometric size of element chosen in Corner – radius of fillet or size of chamfer. Default value: 0 (sharp edge).

Working plane: The start plane of the geometry.

Bottom point: The deepest point where the tool works.

Geometry: Flat

Type:

- One side
- Two sides: Creating two sides opposite to each other.

First Angle: Angle between X axis and midpoint/centerpoint of side. Angle is positive in counter-clockwise direction. Measurement unit: degree. Default value: 0°.

Starting diameter: Diameter of initial cylinder, from which the sides will be machined.

Type:

- Flat distance: Distance of flattened plane and centerline of cylinder.
- Thickness: Distance of flattened plane and the point opposite to it, on the starting diameter.
- Width: Distance of sides (expansion visible from axial point of view).

Size: Geometric size according to selected size *Type*.

Working plane: The start plane of the geometry.

Bottom point: The deepest point where the tool works.

6.20.2 Rough

Milling direction:

- Down
- Up

Radial depth of cut: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

Depth of cut: Axial depth of cut. Default value: one cut.

Abandon radius: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance: Approaching distance in axial (Z) direction. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Radial Clearance: Approaching distance in radial (X) direction. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Bottom finish: Allowance on the bottom of the pocket. Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Side allowance: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Rapid approach of axes:

- Together: approach before the operation and abandon after the operation with simultaneous use of X and Z axis.
- X axis first: during approach, X axis moves to start point first. During abandon, X axis moves later.
- Z axis first: during approach, Z axis moves to start point first. During abandon, Z axis moves later.

6.20.3 Bottom finish

Milling direction:

- Down
- Up

Radial depth of cut: Radial depth of cut relative to the tool diameter as a percentage value (max value: 99%). Tool paths are parallel to the sides of the geometry, so the distance between these paths is the radial depth of cut.

Abandon radius: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance: Approaching distance in axial (Z) direction. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Radial Clearance: Approaching distance in radial (X) direction. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Side allowance: Allowance on the sides of the geometry Makes it possible creating more precise products with better surface quality using different technological parameters to achieve these instead of higher material removal rate. Default value: 0 mm.

Rapid approach of axes:

- Together: approach before the operation and abandon after the operation with simultaneous use of X and Z axis.
- X axis first: during approach, X axis moves to start point first. During abandon, X axis moves later.
- Z axis first: during approach, Z axis moves to start point first. During abandon, Z axis moves later.

6.20.4 Side finish

Milling direction:

- Down
- Up

Depth of cut: Axial depth of cut. Default value: one cut.

Abandon radius: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Clearance: Approaching distance in axial (Z) direction. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Radial Clearance: Approaching distance in radial (X) direction. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Rapid approach of axes:

- Together: approach before the operation and abandon after the operation with simultaneous use of X and Z axis.
- X axis first: during approach, X axis moves to start point first. During abandon, X axis moves later.
- Z axis first: during approach, Z axis moves to start point first. During abandon, Z axis moves later.

6.20.5 Chamfer

Approach radius: Radius of approach and abandon. Tool is led on a quarter circle onto and down from the cutting path. Default value: 1 mm.

Milling direction:

- Down
- Up

Chamfer amount: Size of the chamfer.

Tool angle: Tool's nose angle.

Contact point: The part of the tool that works on the bottom edge of the chamfer. 100% is the greatest, 0% is the smallest diameter of the tool.

Clearance: Approaching distance. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

T small diameter: The small diameter of the tool.

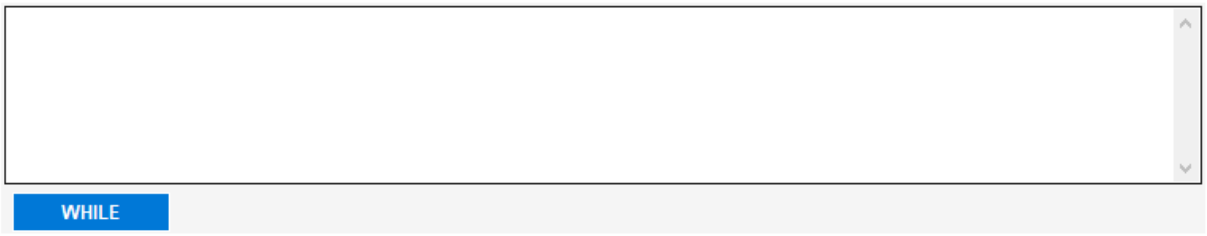
Radial Clearance: Approaching distance in radial (X) direction. Above this height, rapid feed is applied, cutting feed is used below that. Default value: 2 mm.

Rapid approach of axes:

- Together: approach before the operation and abandon after the operation with simultaneous use of X and Z axis.
- X axis first: during approach, X axis moves to start point first. During abandon, X axis moves later.
- Z axis first: during approach, Z axis moves to start point first. During abandon, Z axis moves later.

6.21 Your code

It is used for inserting your own G code into the myNCT operation list



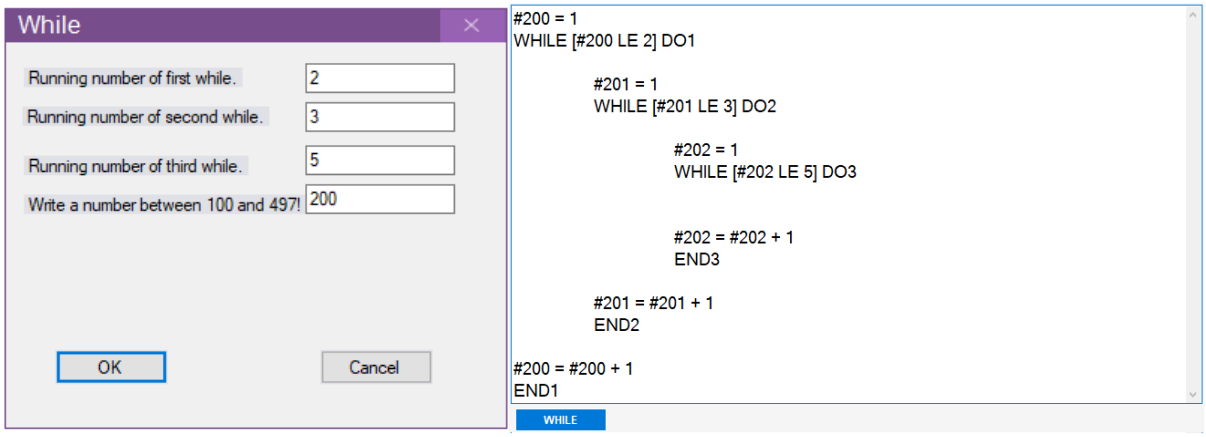
6-21. Figure Input for own G code

6.21.1 While cycle

The While button is used to generate a *while* cycle from 4 parameters.

The number of iterations of each cycle will be saved in variables signed with the given number (between 100 and 497) and following two numbers.

It is advised choosing a number for variable to be at least 250 to prevent collision and overwriting with macro variables that might be used during the program in certain operations.

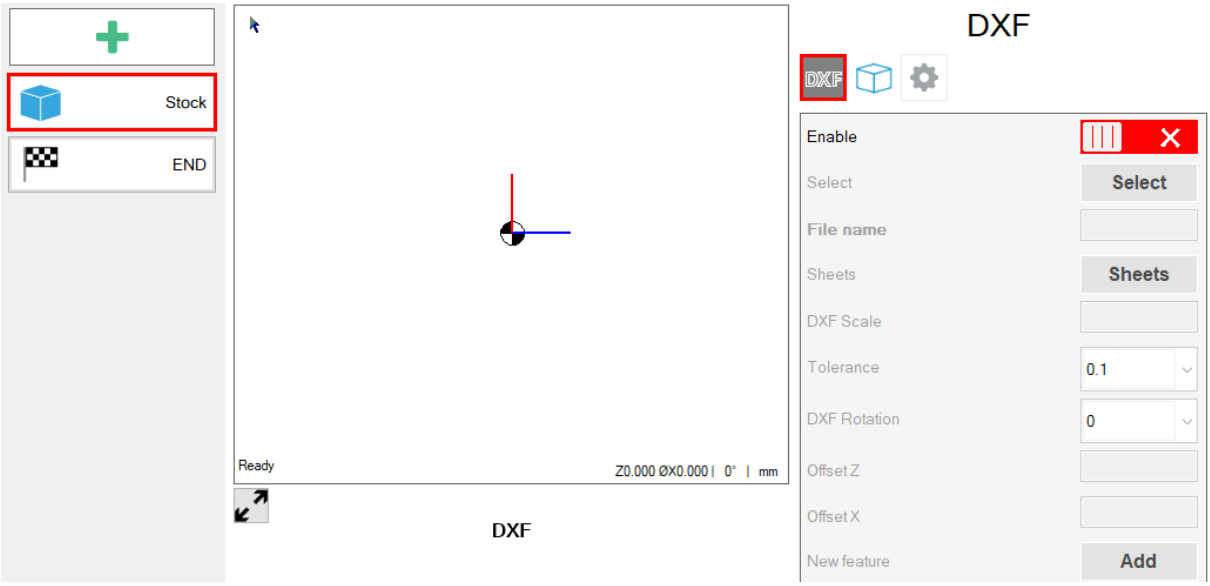


6-22. Figure While cycle form and generated code

7 CREATE STOCK AND GEOMETRIES WITHOUT DXF

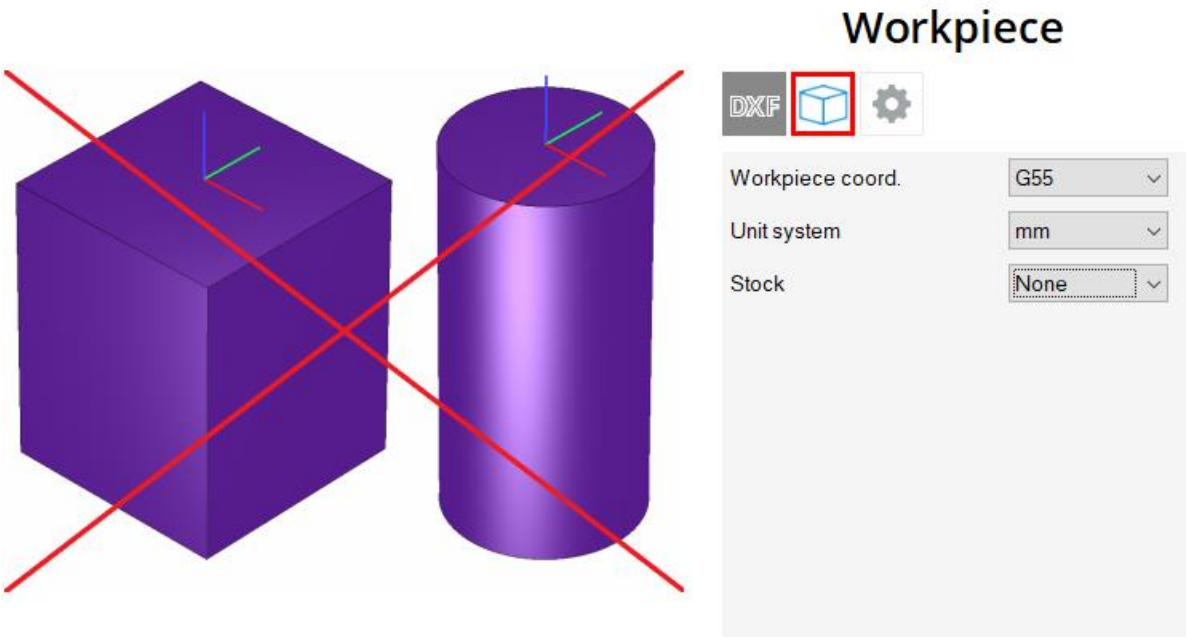
Creating and machining geometries available without DXF file.

7.1 Stock Settings

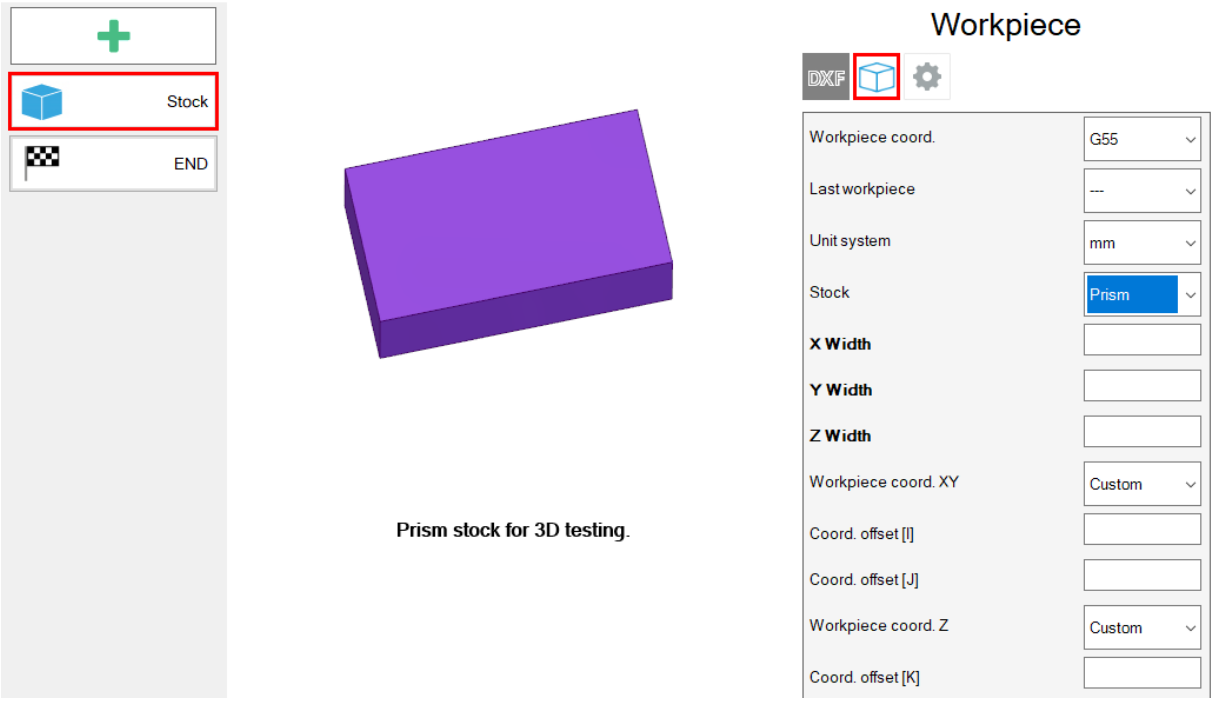


7-1. Figure DXF disabled

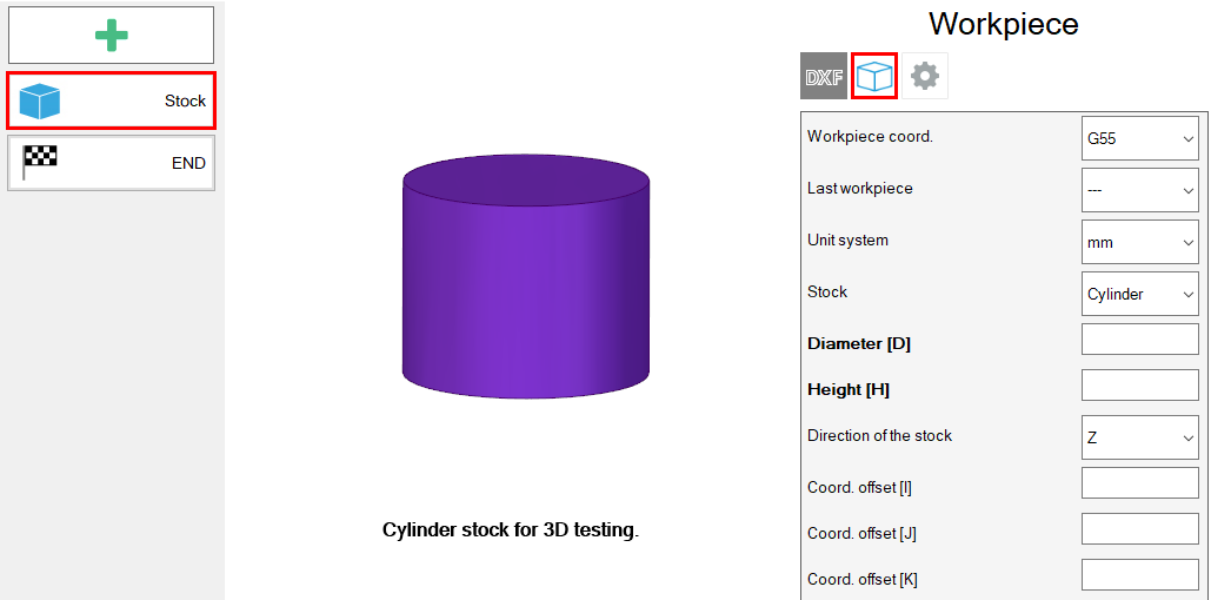
7-2. Figure Select a workpiece coordinate system (not G54)



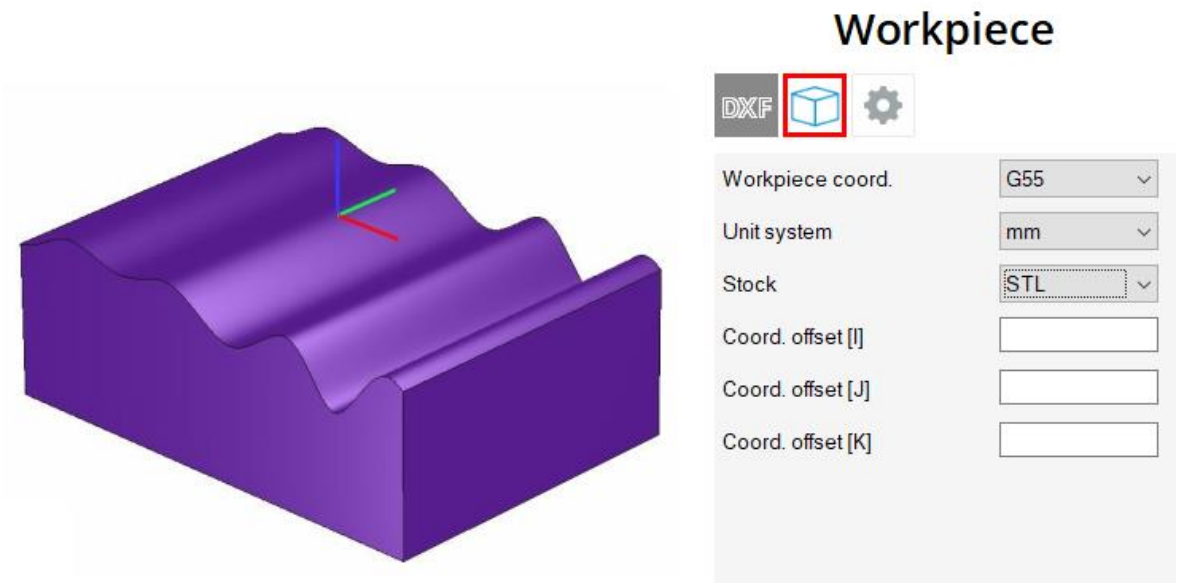
7-3. Figure No stock



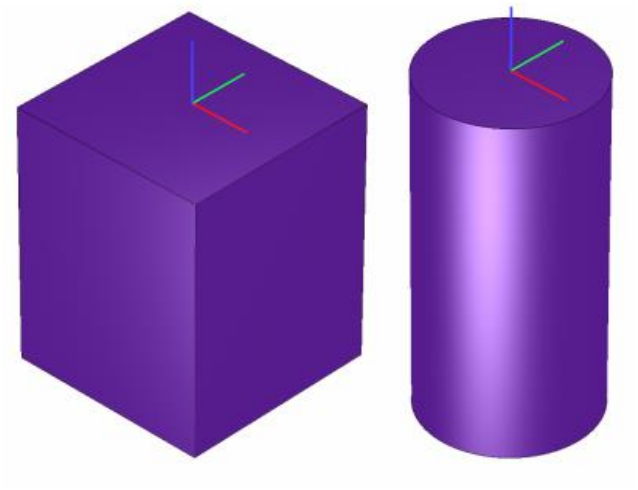
7-4. Figure Stock is prism



7-5. Figure Stock is Cylinder

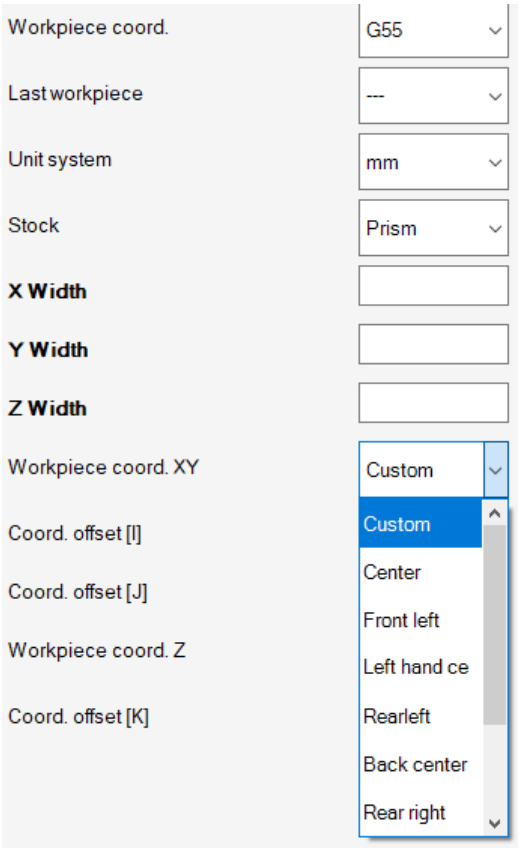


7-6. Figure Stock is STL file



7-7. Figure The workpiece coordinate system is in the middle „XY" point of the stock in the highest point in „Z".

To move the workpiece coordinate system set Coord. offset I (X), J (Y) or K (Z). IJK positive directions are the same as the XYZ axes directions. A quick solution is to move the coordinate system to specific points of the stock (corners, center points).



7-8. Figure Clicked W to corner

7.2 Geometry panel

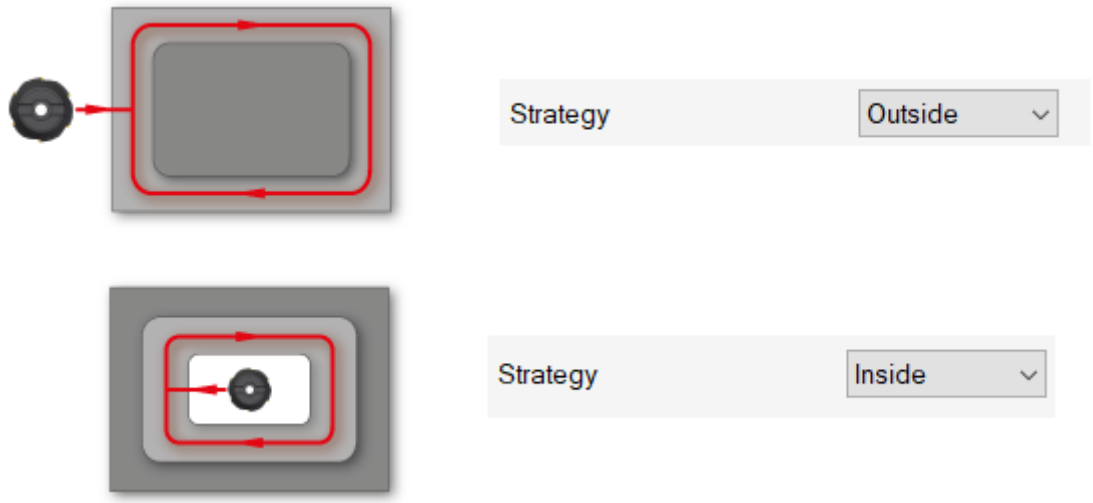
A geometry should be selected and created to be machined.

7.2.1 Pocket

7-9. Figure Pocket geometries

7.2.2 Step

7-10. Figure Step geometries



7-11. Figure Strategy: milling from outside or from inside

7.3 Pattern panel

In the pattern panel, you set the center points of the geometries.