

InventorCAM 2015



InventorCAM
iMachining – The Revolution in CAM!



2015
the iMachining Edge



What's new in InventorCAM 2015



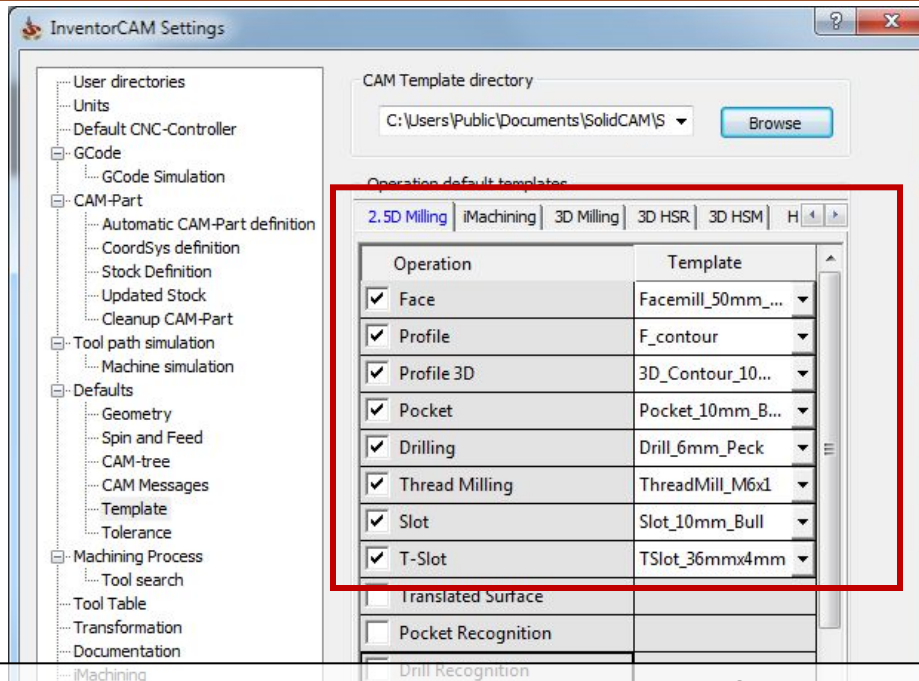
iMachining 2D & 3D | 2.5D Mill | HSS | 3D HSR/HSM | Indexial Multi-Sided | Sim. 5X | Turning | Advanced Mill-Turn | Solid Probe

© 1995-2014 SolidCAM Ltd. All rights reserved.

www.inventorcam.com

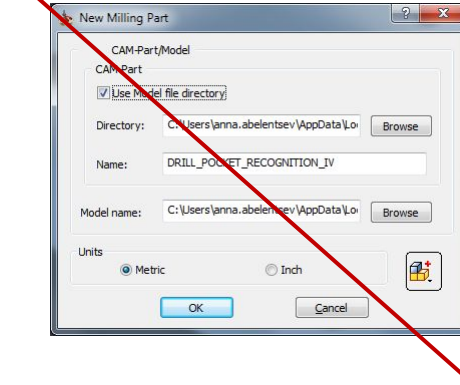
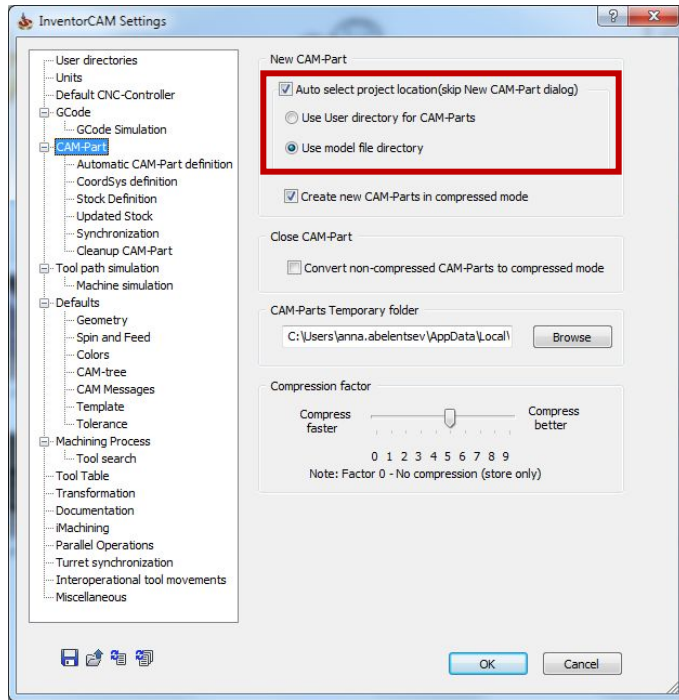
General

Templates: Default templates for 2.5D Mill operations



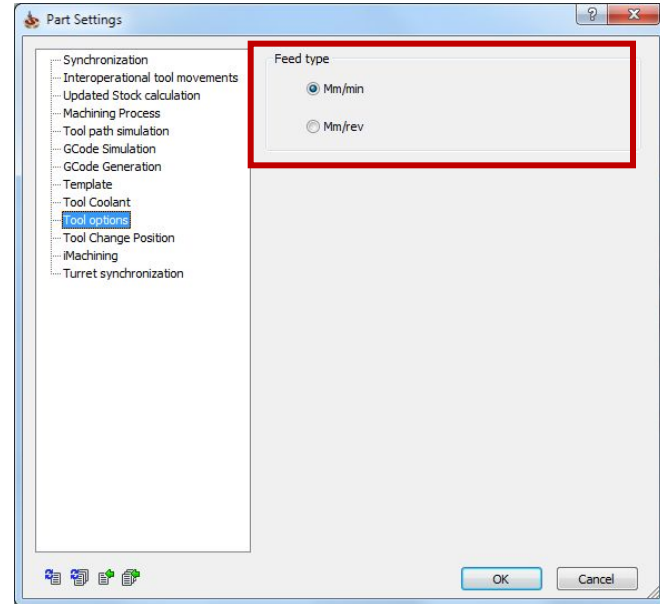
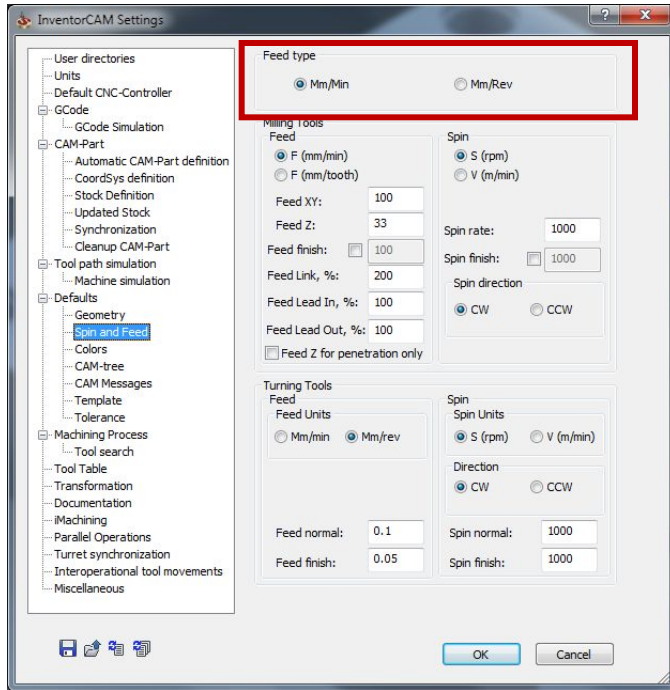
- Default templates are set in InventorCAM settings when new operation is created these templates are used
- Useful for example to have a default starting Tool

Quick Start settings - Skip "New CAM-part" dialog



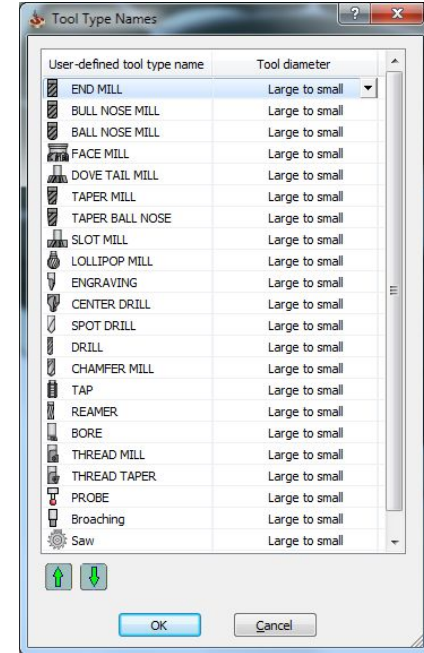
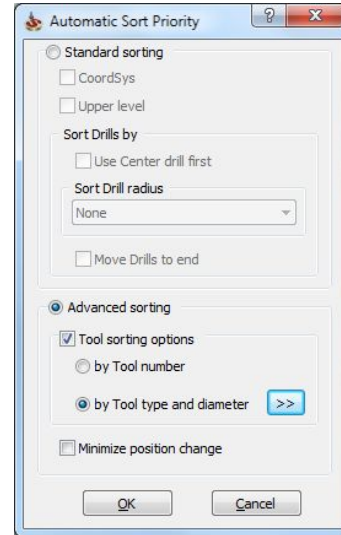
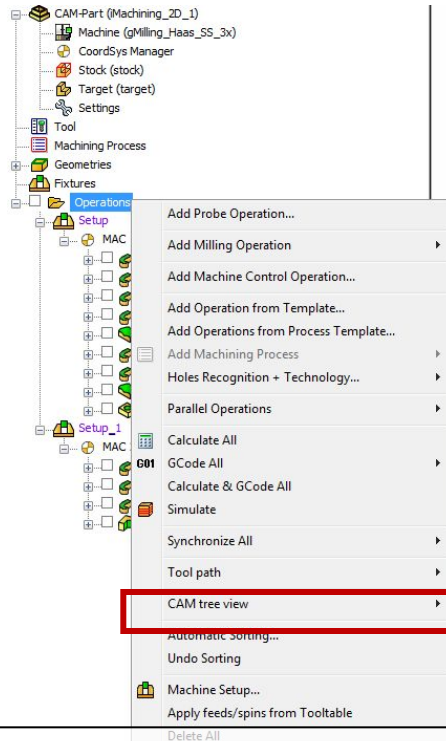
- Options in SolidCAM settings to skip the first New CAM-part dialog with default values
- Enables the user to start directly adding operations in a new Part

Feed units: mm/rev or mm/min by default



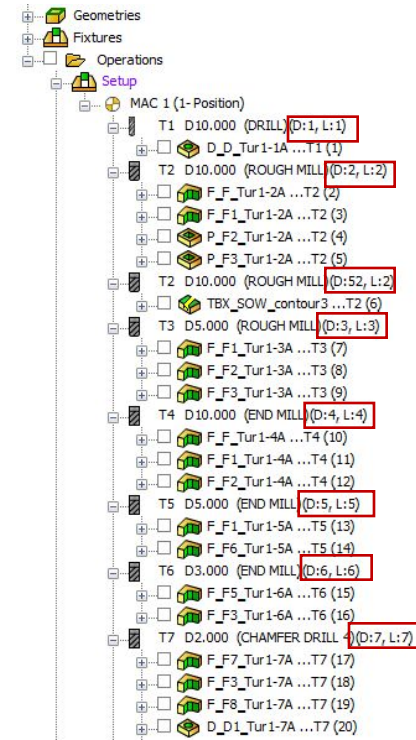
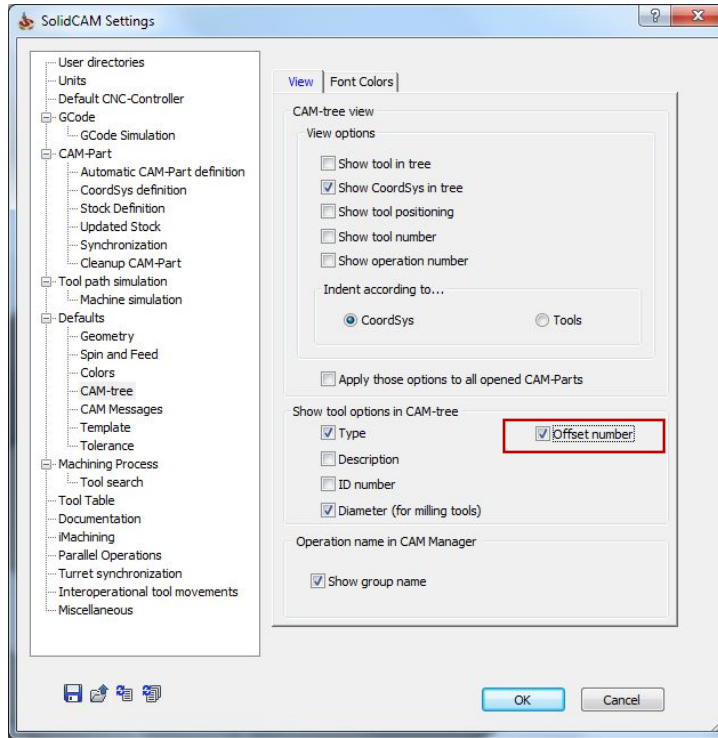
- Define default feed type for new CAM-parts in InventorCAM settings

CAM tree: Advanced sorting of operations



- Possibility to sort operations in CAM-tree by tool number and tool properties (Diameter, Tool Type)

CAM-tree: Show Tool Offset numbers



• Show Tool Offset numbers in CAM-tree

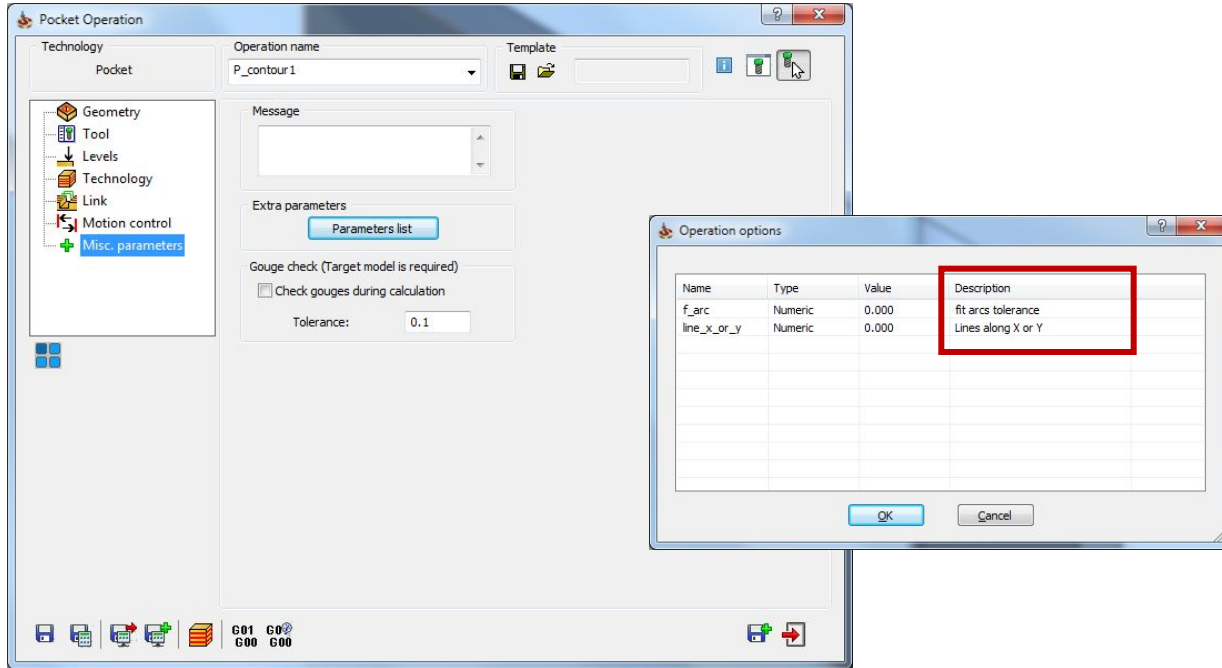
Operations: Additional parameters to INFO dialog

The screenshot shows the 'HSM Constant Z machining operation' dialog box. The 'Info' dialog is open, displaying the following parameters:

Info	
Name	HSM_CZ_target
Name	23 END MILL D16
Diameter	16.000000 mm
Corner radius	0.000000 mm
Taper angle	0.000000 deg
Outside holder	55.000000 mm
Cutting length	30.000000 mm
Feed cutting	2000.000000
Feed rapid	10000.000000
Feed ramp down	600.000000
Feed ramp up	4000.000000
Spin	8000.000000
Machining time	0:41:23
Z-Top	0.000000
Z-Bottom	-349.067993

- Show Cutting depth information and Additional tool data in Info dialog

Operations: Description of parameters in GUI



- Description of user-defined parameters is now visible inside the operation

Generate G-code per operation

The screenshot illustrates the process of generating G-code per operation in SolidCAM. On the left, the software's interface shows a tree view of operations. The 'Generate' menu is open, and the 'Generate per operation' option is highlighted with a red box. A red arrow points to the right, where three Notepad windows are shown, each containing G-code for a different operation (G21, G291, M30).

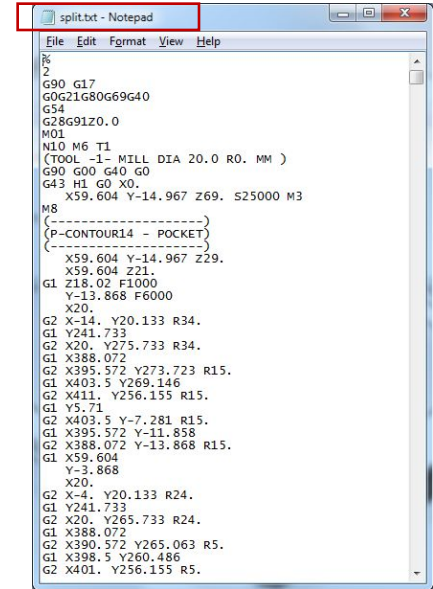
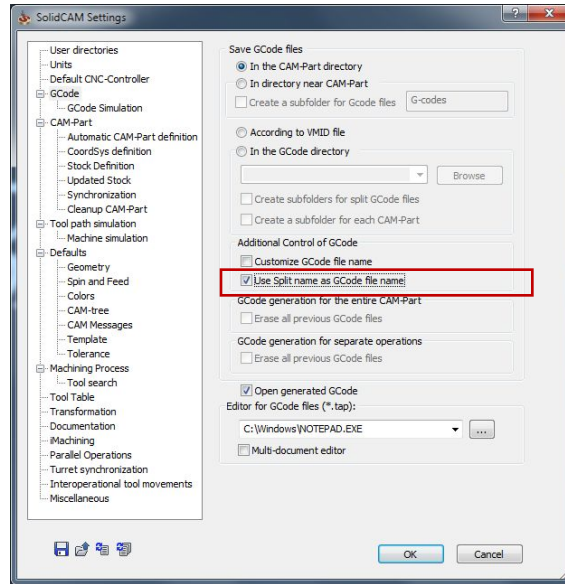
```
lower.mpf - Notepad
File Edit Format View Help
%N_2000_MPF
;SPATH=/_N_WKS_DIR/_N_MILL_TURN_ADVANCED_WPD
G21
G291
( T2 * HE
( * OP4 C
G50 S3000
G96 S120 M3
M107 M150
G0 X104.
Z-55.211
X56.3
G1 X50.8
G0 X57.3
X104.
Z1.11
M30

upper.mpf - Notepad
File Edit Format View Help
%N_1000_MPF
;SPATH=/_N_WKS_DIR/_N_MILL_TURN_ADVANCED_WPD
G21
G291
M30

lower.mpf - Notepad
File Edit Format View Help
%N_2000_MPF
;SPATH=/_N_WKS_DIR/_N_MILL_TURN_ADVANCED_WPD
G21
G291
( T2 * HELIR 1616-3T12 * )
( * OP3 TR-CONTOUR33 * )
G50 S3000
G96 S120 M3
M107 M150
G0 X104. Z1.11
Z-43.663
X57.188
G1 X53.952 F0.2
X55.188 Z-44.282
X55.588
Z-43.663
X33.952
X52.714 Z-43.045
Z-42.426
X51.478
X52.714 Z-43.045
X53.114
Z-42.426
X51.478
X50.242 Z-41.808
Z-19.98
G0 X51.84
Z-17.78
G1 X50.242
Z-13.731
X50.13
X50.23 Z-15.771
G3 X50.24 Z-16.211 R18.02
G1 Z-18.411
G3 X48.701 Z-23.621 R18.02
G1 Z-41.038
X50.242 Z-41.808
X50.642 Z-41.608
Z-41.038
X48.7
X47.162 Z-40.269
Z-25.698
G2 X48.701 Z-23.621 R18.02
```

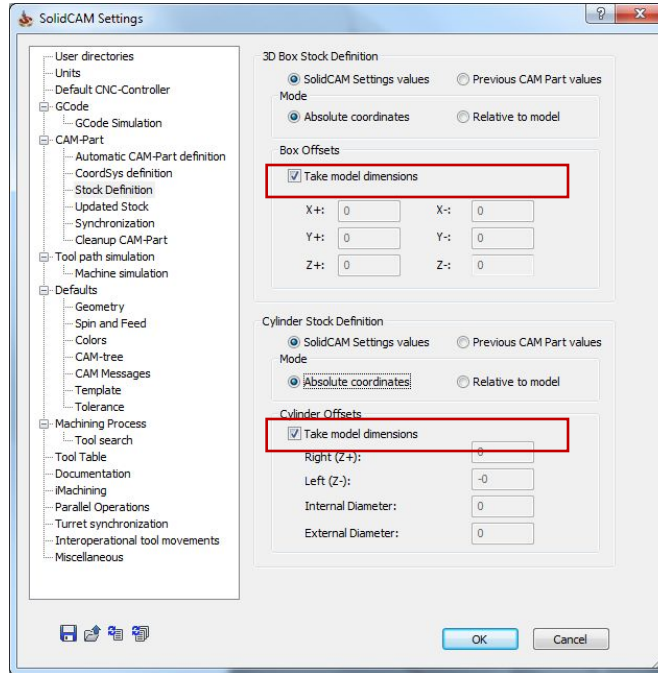
- **Generate separate file of G-code per each operation**

Use split name as G-code file name



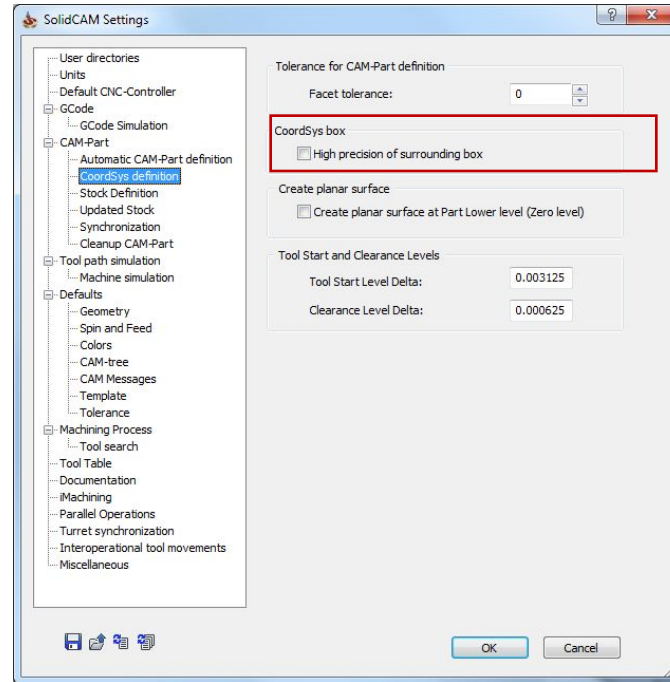
- Use name of Split as name of G-code file

Stock: Take target model dimensions by default



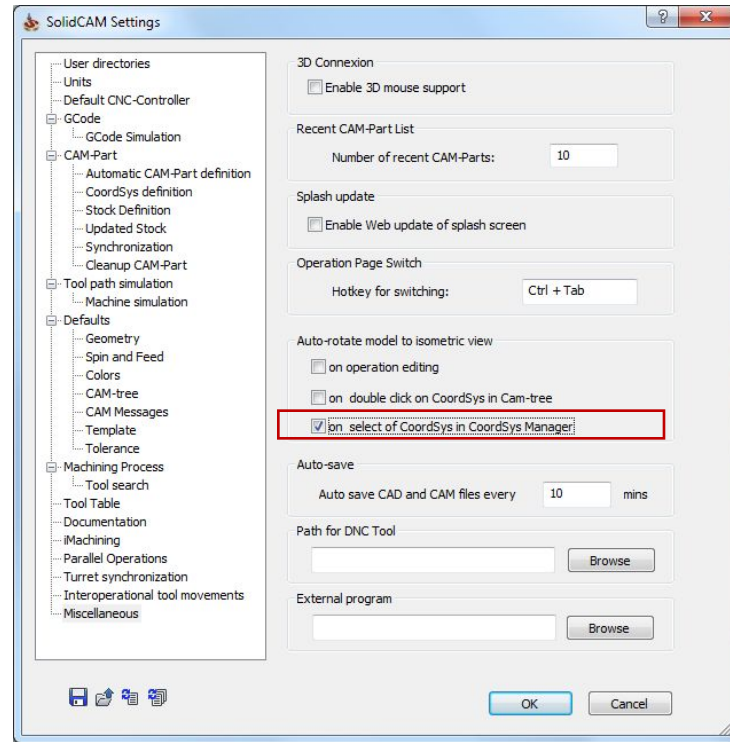
- When Stock definition mode is set to Absolute Coordinates – dimensions of Target model will be taken automatically

High precision of box for CoordSys definition (facetting)



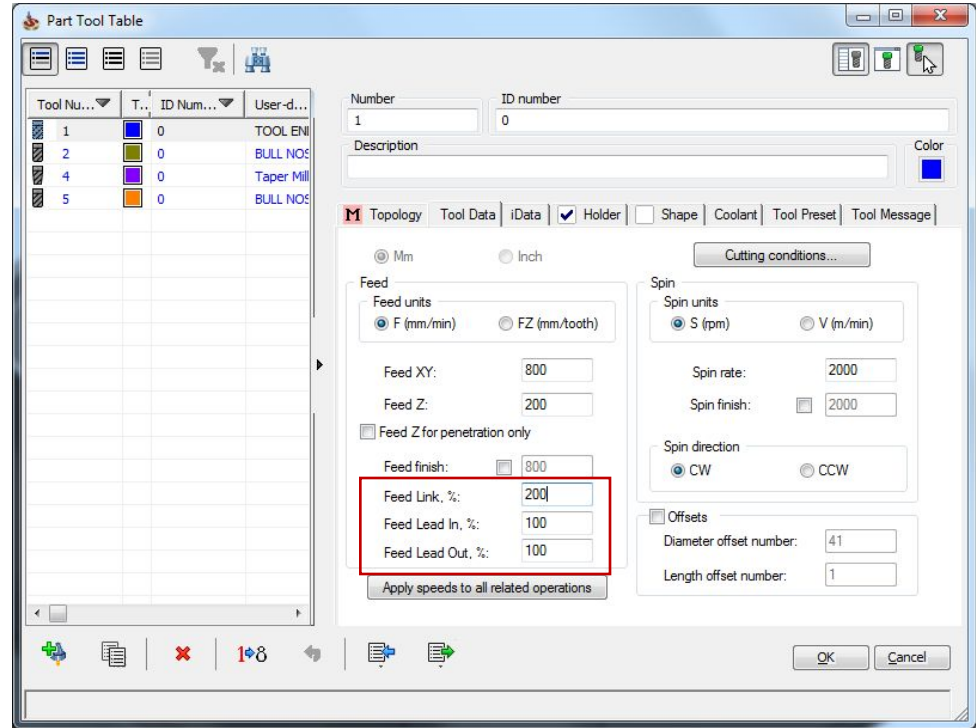
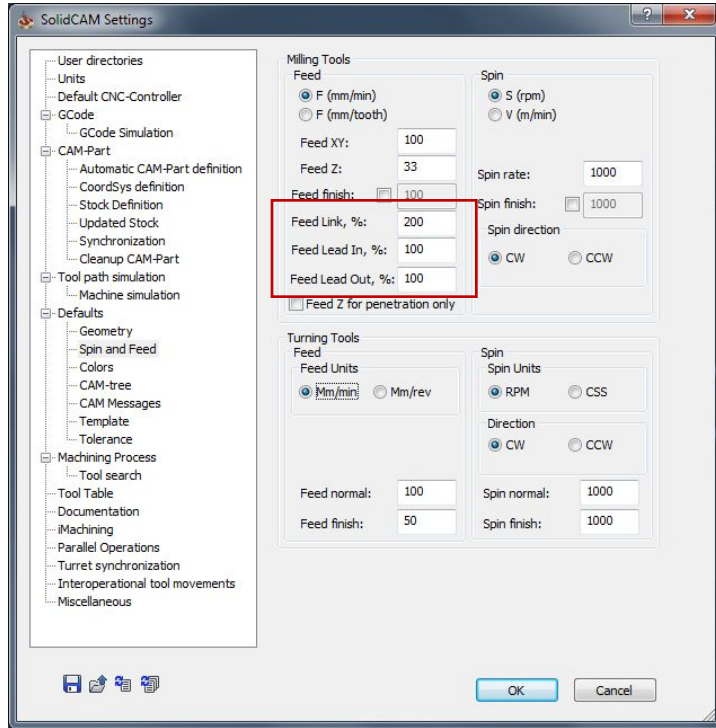
- “ High precision“ for box (facetting) set to ON always during CoordSys definition

Rotate model to isometric in CoordSys manager



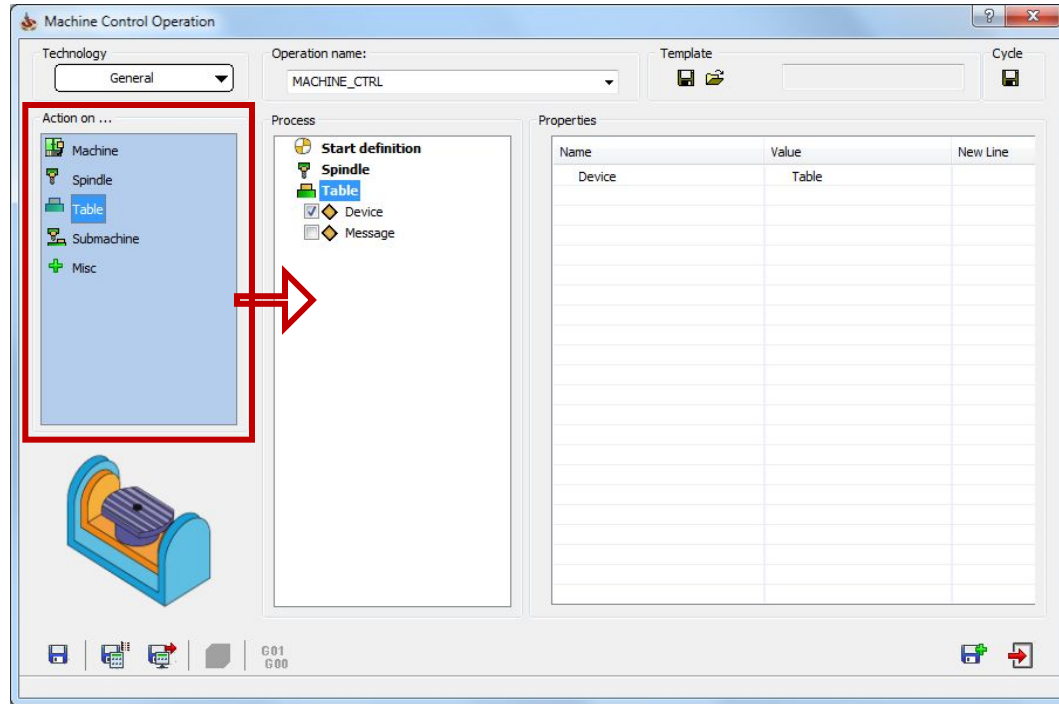
- Rotate model to isometric view when clicking on CoordSys in CoordSys manager

Defaults through Settings for Feed Link, Lead in, Lead out



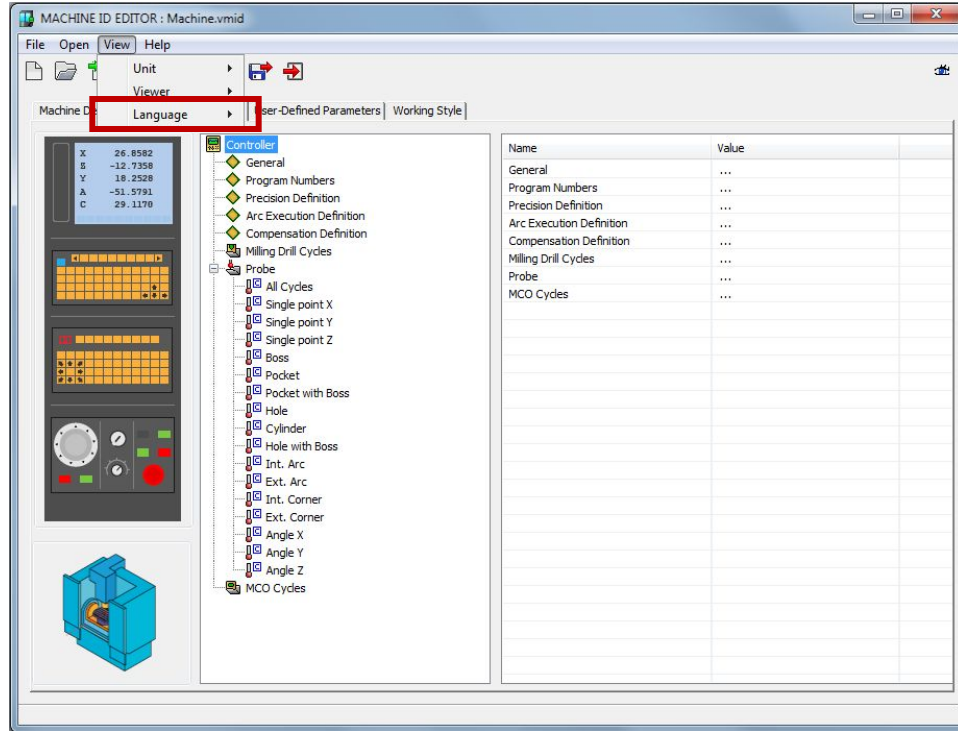
- Possibility to define defaults through Settings for Feed Link, Lead In and Lead Out

MCO: Faster action definition



- Double click on the item in the „Action on...” list automatically adds this item to the Process list.

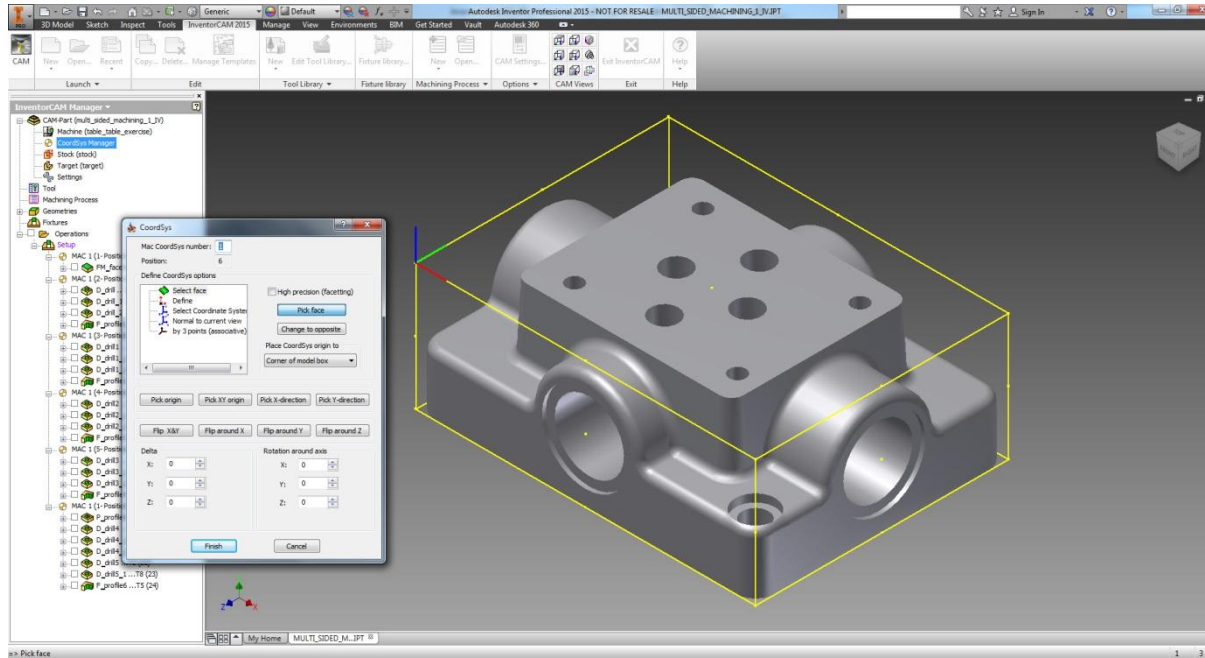
Machine ID: Change language of VMID



- Possibility to show MachineID editor fields in any language, independent from main installation language

CoordSys

CoordSys: Always build box around target model



- Create by default CoordSys envelope box around the target
- Useful for CoordSys Origin definition

CoordSys: Changes in CoordSys dialog

CoordSys Data

Machine CoordSys number: 1

Position: 2 X: 0 Y: 0 Z: 0

Shift
X: -120 Y: 429.076 Z: 230

Rotation around
X: 0 Y: 0 Z: 0

Default machining levels

Front | **Radial** | Rear

<input type="button" value="Tool start level"/>	300
<input type="button" value="Clearance level"/>	200
<input type="button" value="Part Upper level"/>	0
<input type="button" value="Part Lower level"/>	-10
<input type="button" value="Tool Z-level"/>	250

Create planar surface at Part Lower level



CoordSys Data

Machine CoordSys number: 1 Position: 2

Shift
X: -120 Y: 429.076 Z: 230

Rotation around
X: 0 Y: 0 Z: 0

Main | **Rear**

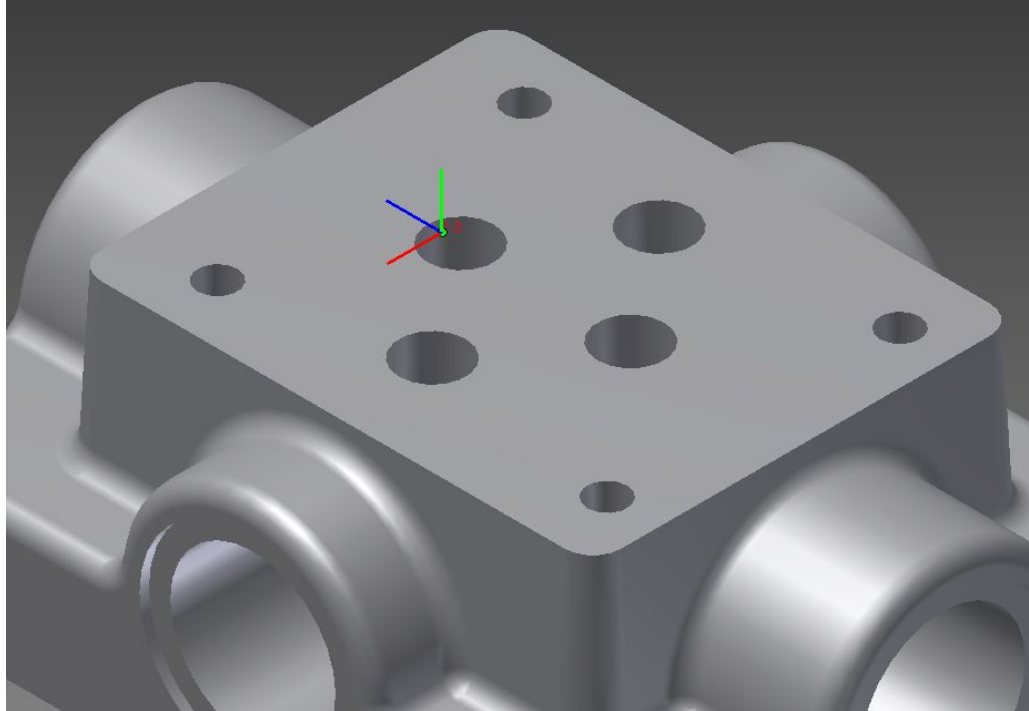
Default machining levels

Plane movements		Radial movements	
<input type="button" value="Tool start level"/>	300	<input type="button" value="Tool start level"/>	300
<input type="button" value="Clearance level"/>	200	<input type="button" value="Clearance level"/>	200
<input type="button" value="Part Upper level"/>	0	<input type="button" value="Part Upper level"/>	0
<input type="button" value="Part Lower level"/>	-10	<input type="button" value="Part Lower level"/>	-10
<input type="button" value="Tool Z-level"/>	250		

Create planar surface at Part Lower level

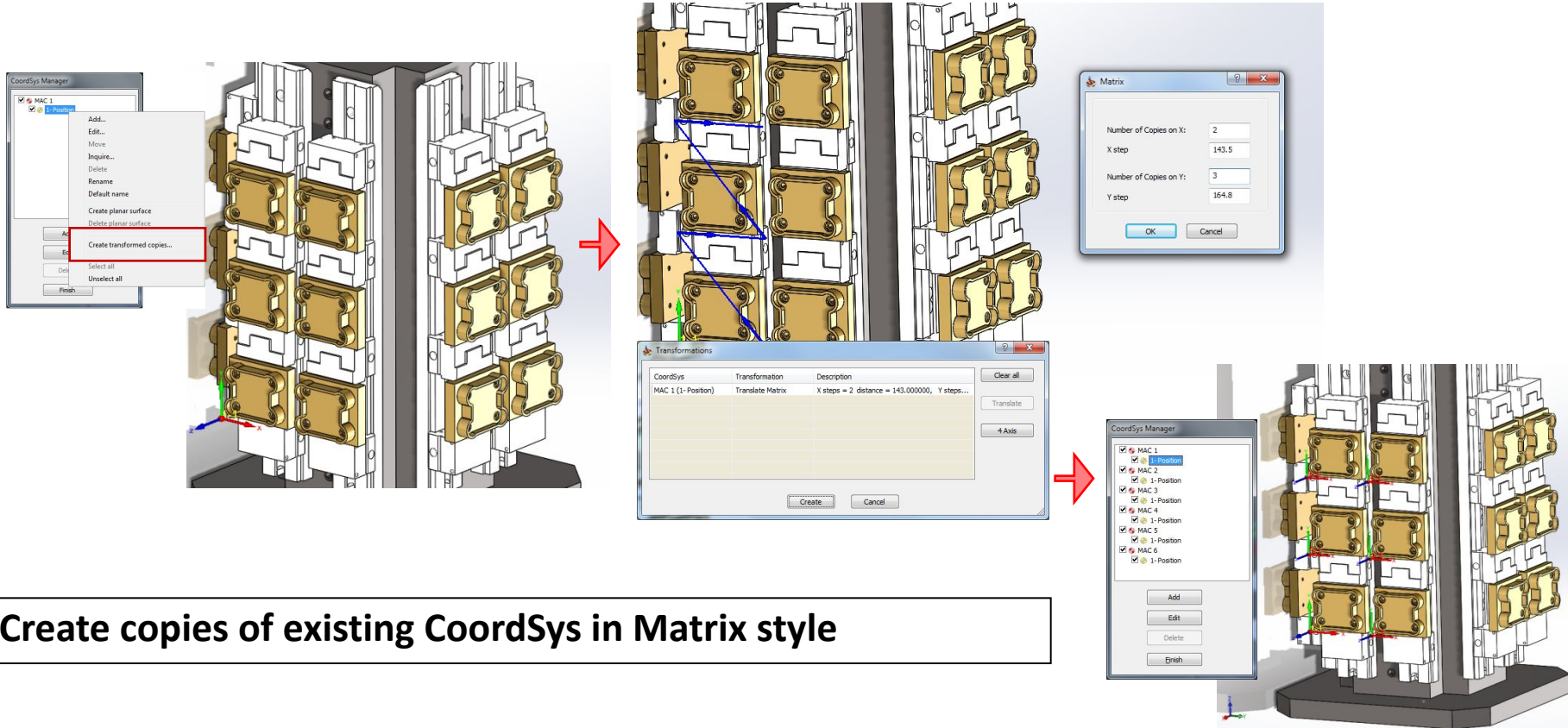
- Radial movement levels are now on the main dialogue (instead of TAB)
- Better noticed by user

CoordSys: Associativity with Inventor Coordinate System



- **Support of associativity of a CAM CoordSys, that is built on a Inventor CoordSys**

Copying of CoordSys in transformation style: Matrix



• Create copies of existing CoordSys in Matrix style

Copying of CoordSys in transformation style: Custom Axis

CoordSys Manager

- MAC 1
- 1-Position**
- Add...
- Edit...
- Move...
- Inquire...
- Delete
- Rename
- Default name
- Create planar surface
- Delete planar surface
- Create transformed copies...
- Select all
- Unselect all
- Finish

Transformations

CoordSys	Transformation	Description
MAC 1 (1-Position)	4th Axis	3 Rotations in List

4 Axis

Rotate List

Add angle

Angle: 90

Steps: 2

4x rotation axis

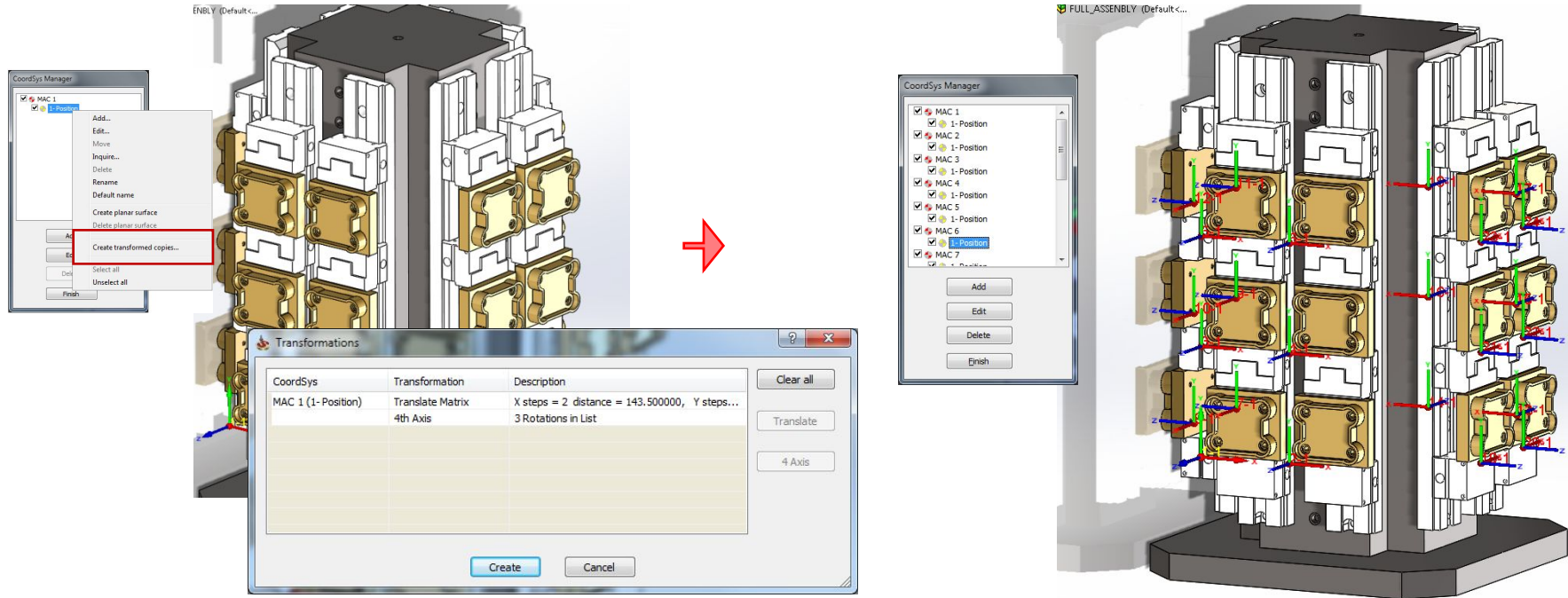
Axis + Point: MAC 1 (1-POS1)

Angle list

Num.	Angle	Offset
1	90.000000	
2	180.000000	

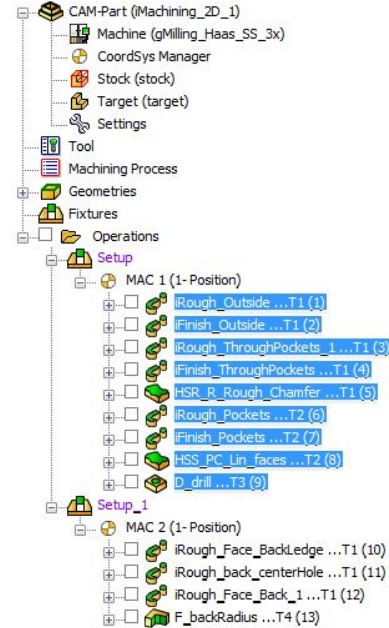
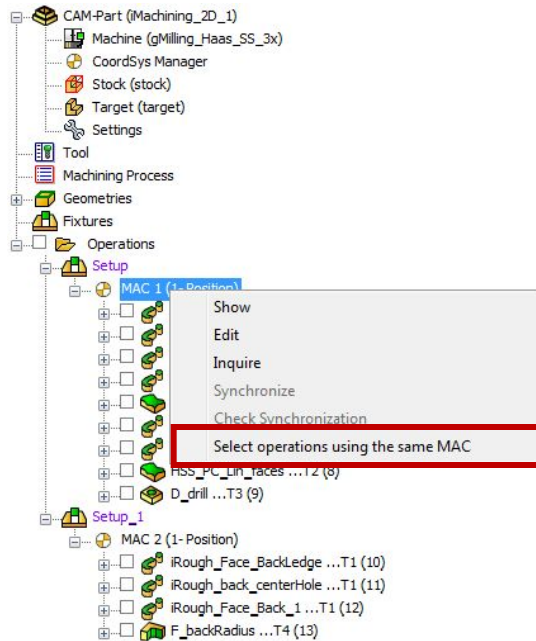
• Create copies of existing CoordSys around user-defined axis

Copying of CoordSys in transformation style: Combined transformation



- Combine 2 styles of CoordSys copying
- User can use each style of transformation only once

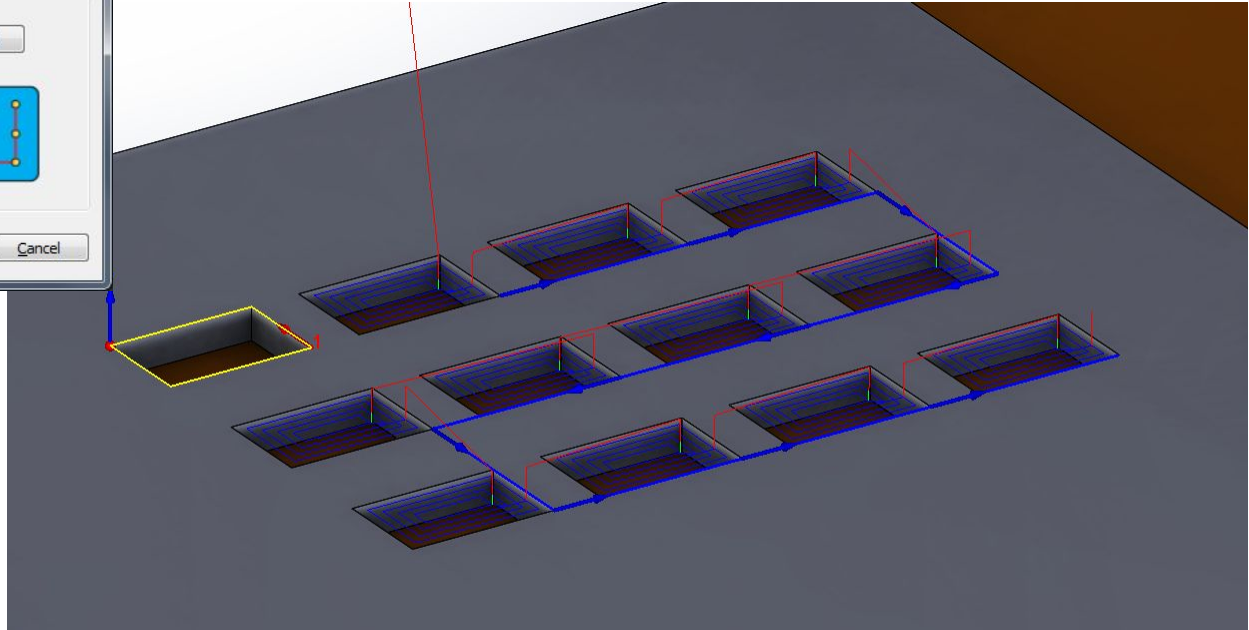
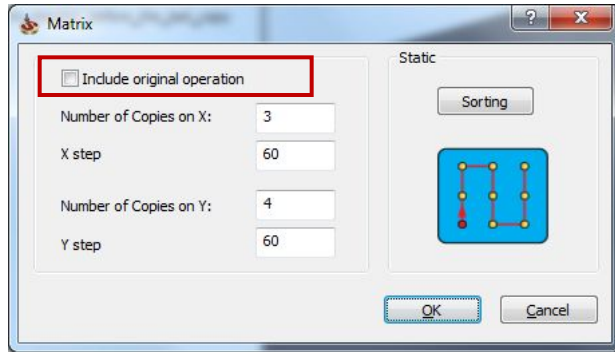
Select all operations of the same CoordSys



- Fast selection of all operations defined in the Coordsys (MAC) and it's positions

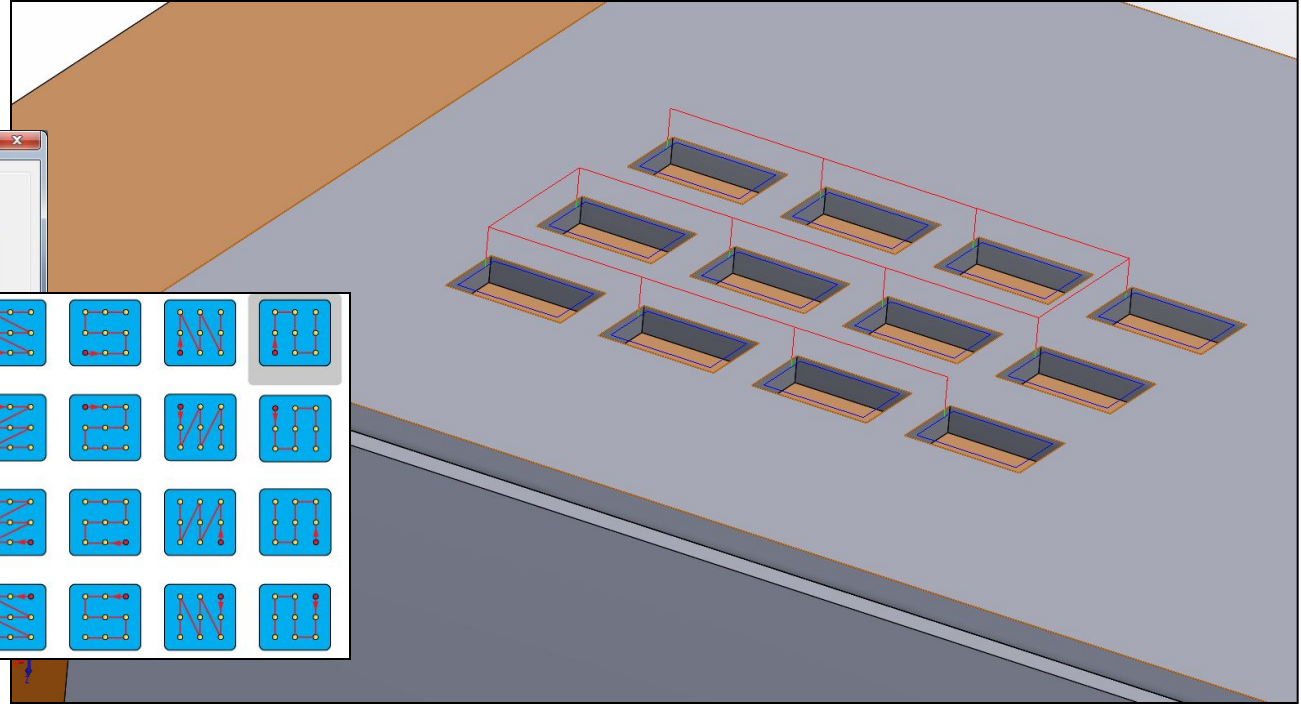
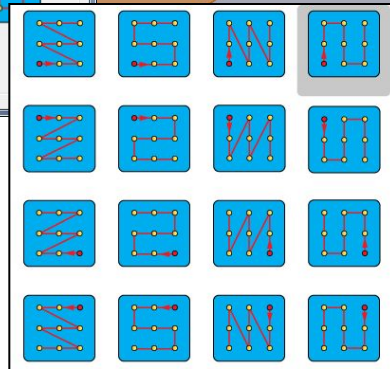
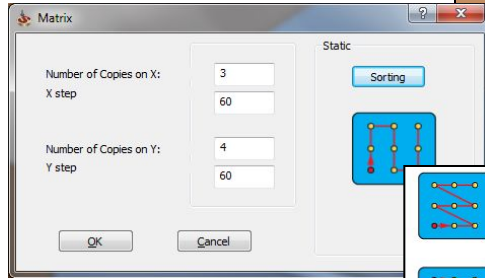
Transformation

Transform: Matrix without original operation



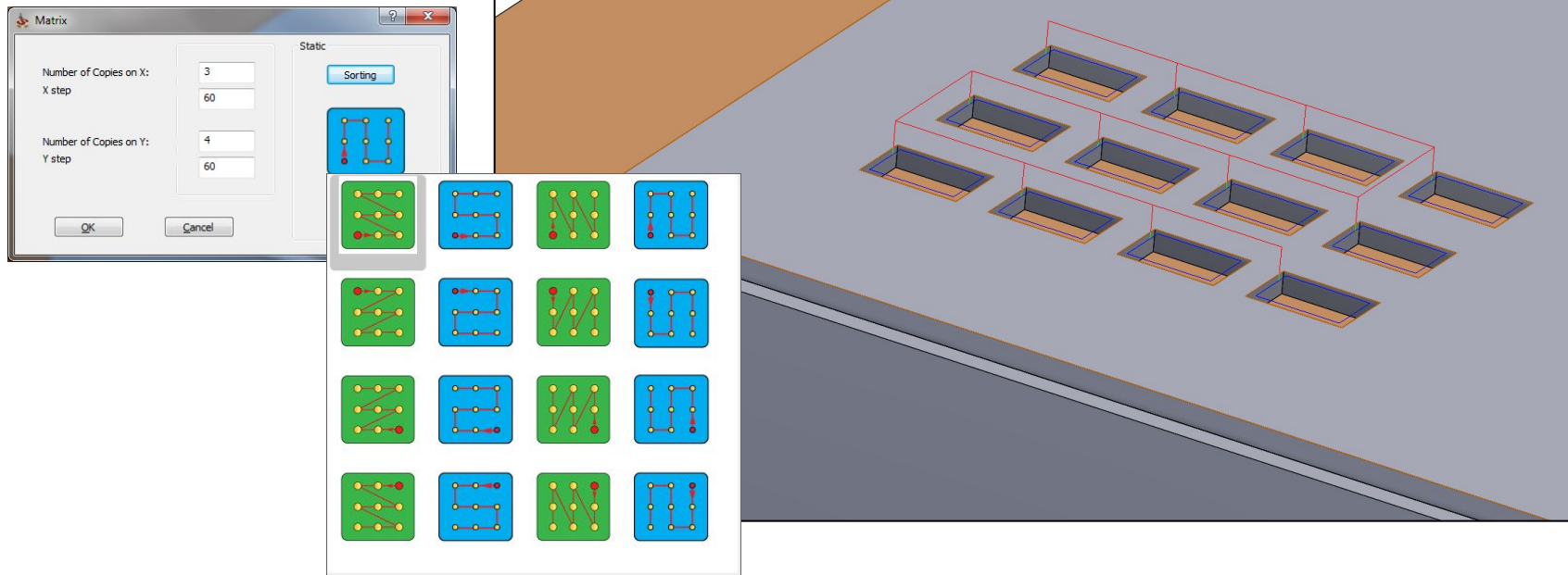
- Possibility to make Matrix without original operation toolpath
- Useful for postponed transformed operations

Transform: Matrix sorting options



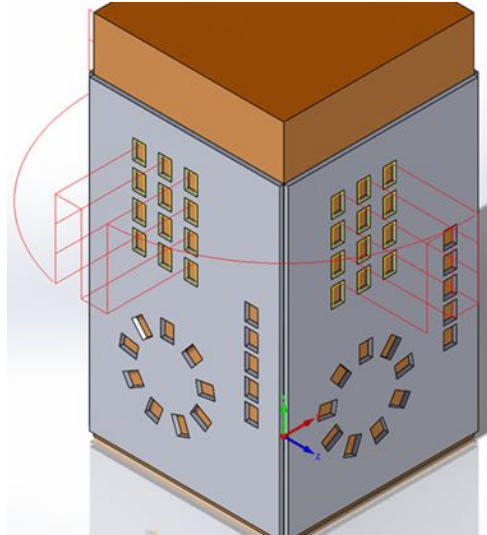
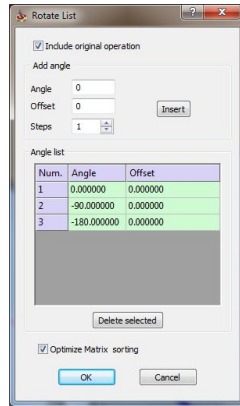
- **Additional Sorting options in Matrix transformation**

Transform: Marking of sorting types affected by optimization



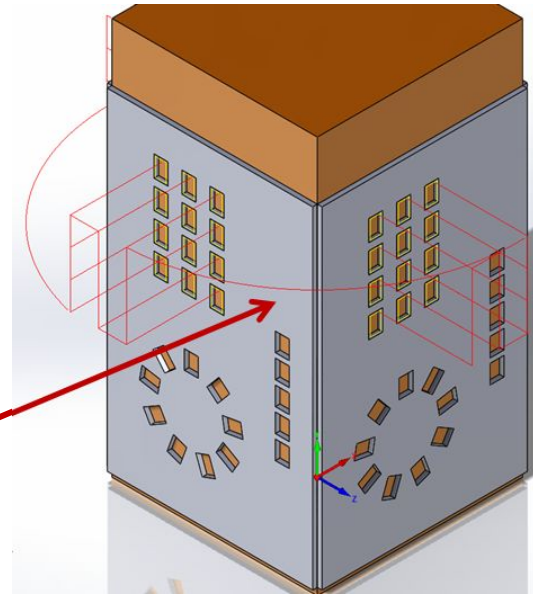
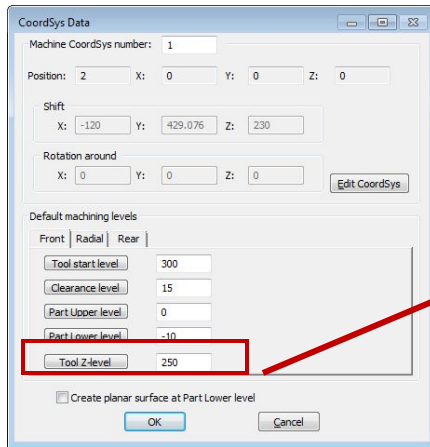
- If Optimization of operation loops is turned on in *.VMID, the sorting types which will be affected by optimization, are marked by green color

Transform: Optimize Matrix Sorting in 4x transformation



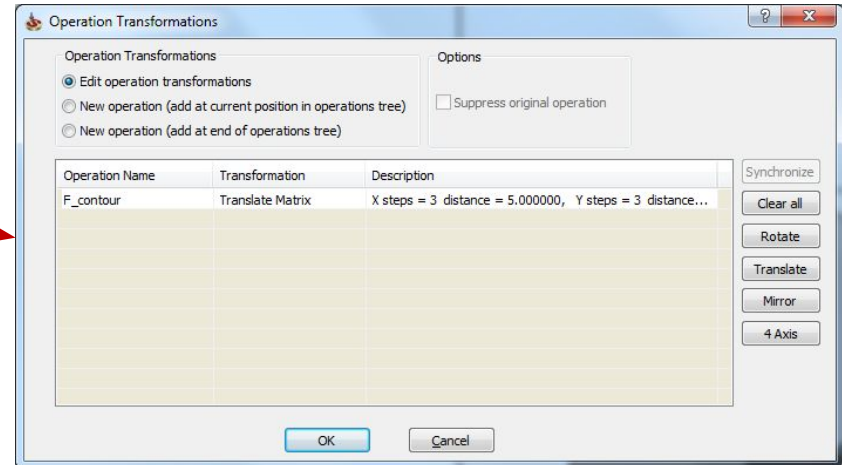
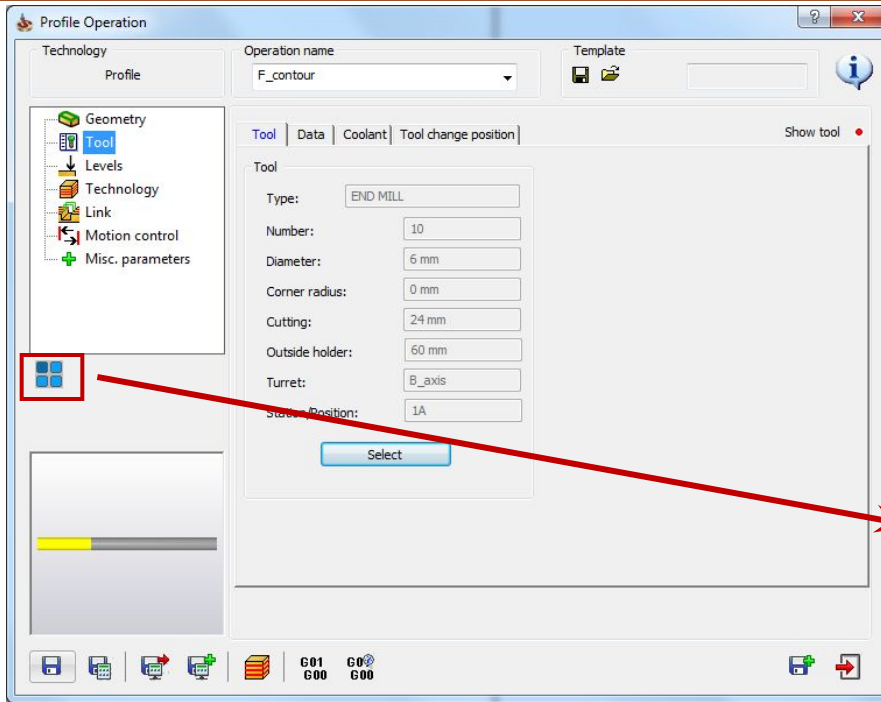
- If “optimize Matrix sorting” is checked – movements between 4th axis positions are done to minimize tool movements

Transform: Clearance radius for movements between 4th axis positions



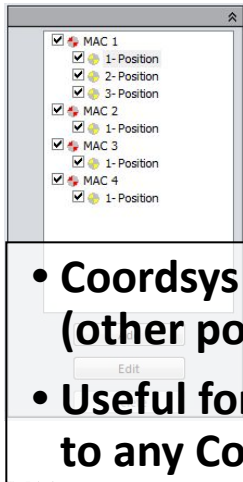
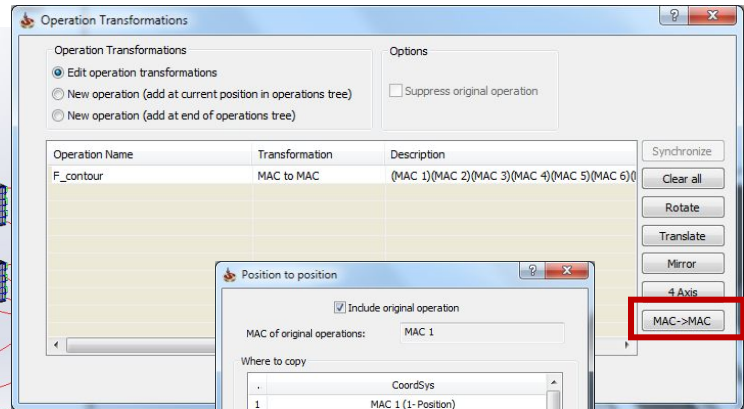
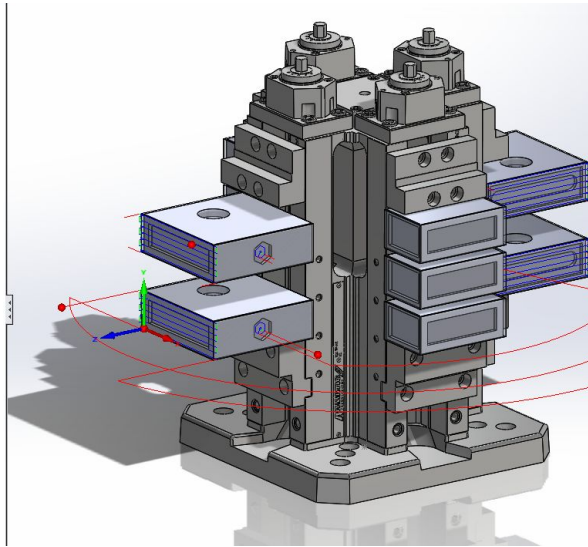
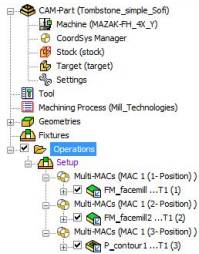
- Use Tool Z level from MACx-posN for movements between planes in Transformation around 4x

Transform: Access to transform from operation



- Possibility to open Transform dialog straight from the Operation dialog

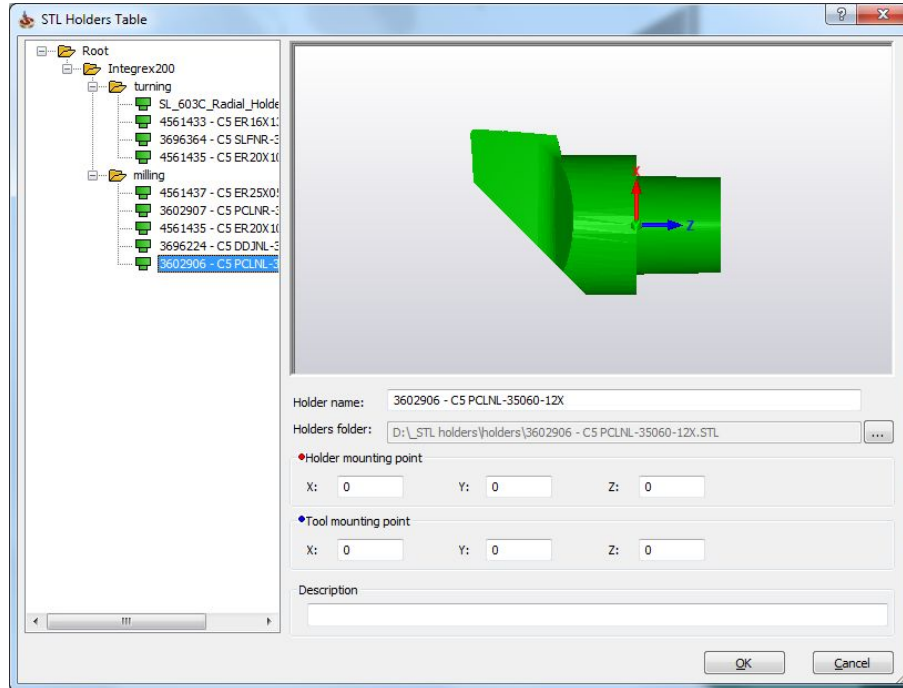
Transformation of operations from one CoordSys to another



- Coordsys where operations should be copied to , should contain only 1 position (other positions will be created automatically)
- Useful for Tombstone operations as it enables the simple transfer of an operation to any Coordinate system, created on any face of the Tombstone.

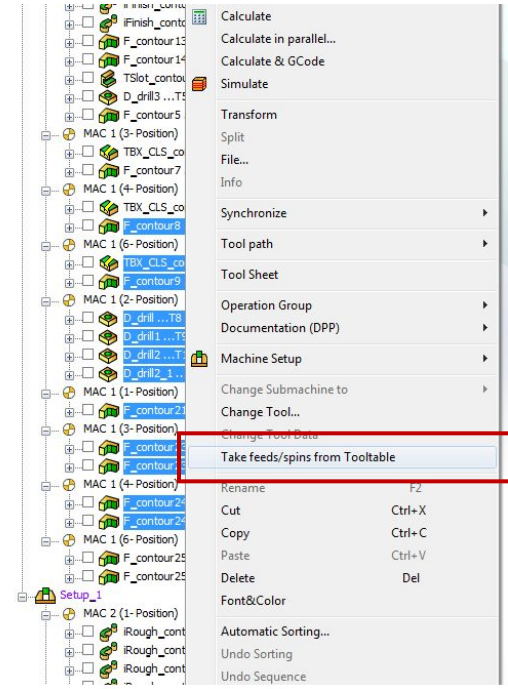
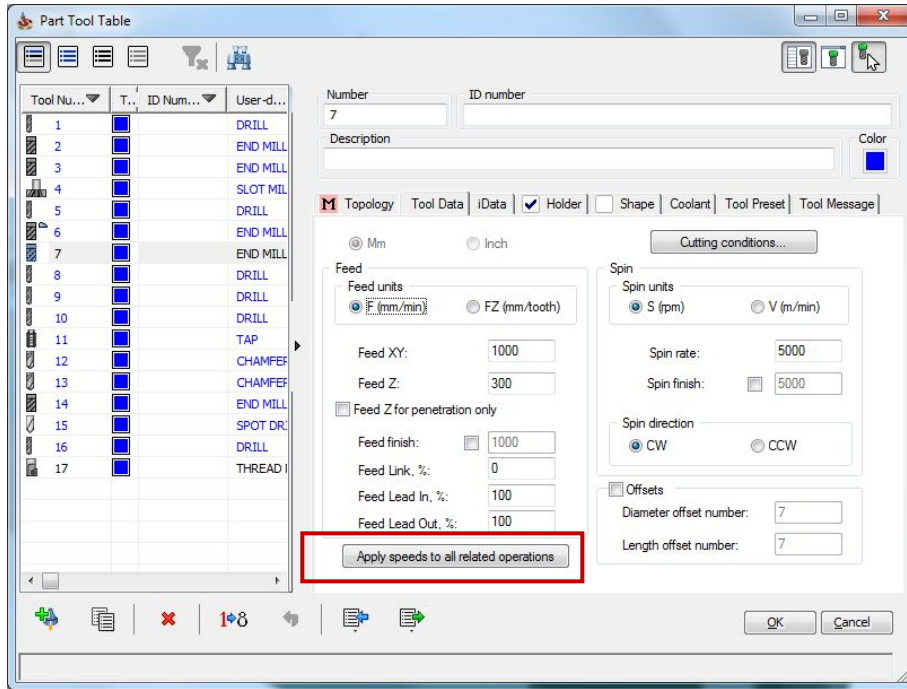
Tooltable

Tooltable: Changes in STL holders library



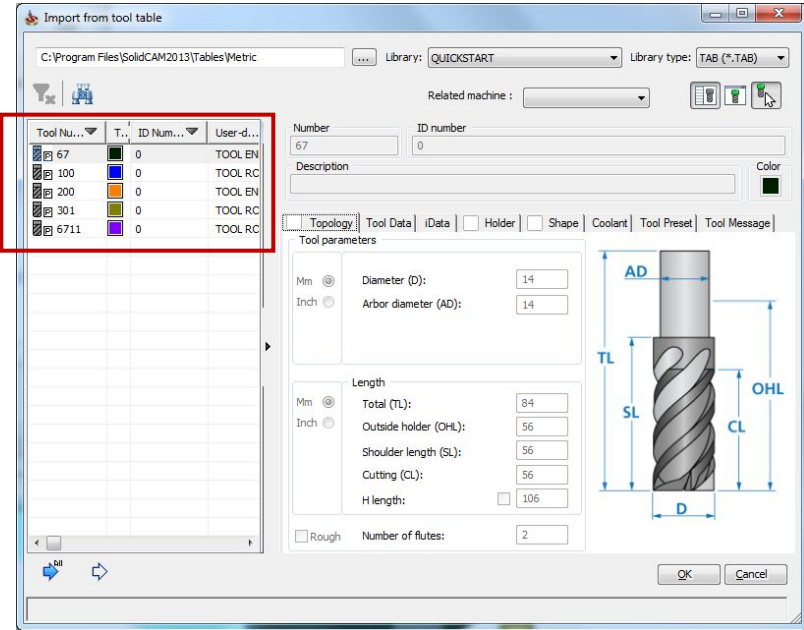
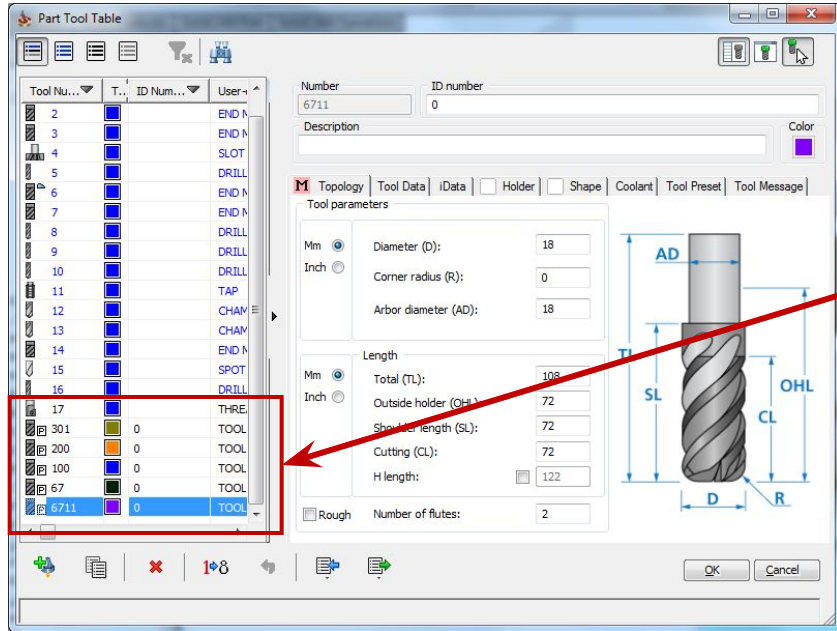
- Combine Milling and Turning STL Holders under one machine library

Tooltable: Update operation feeds/spins according to tooltable



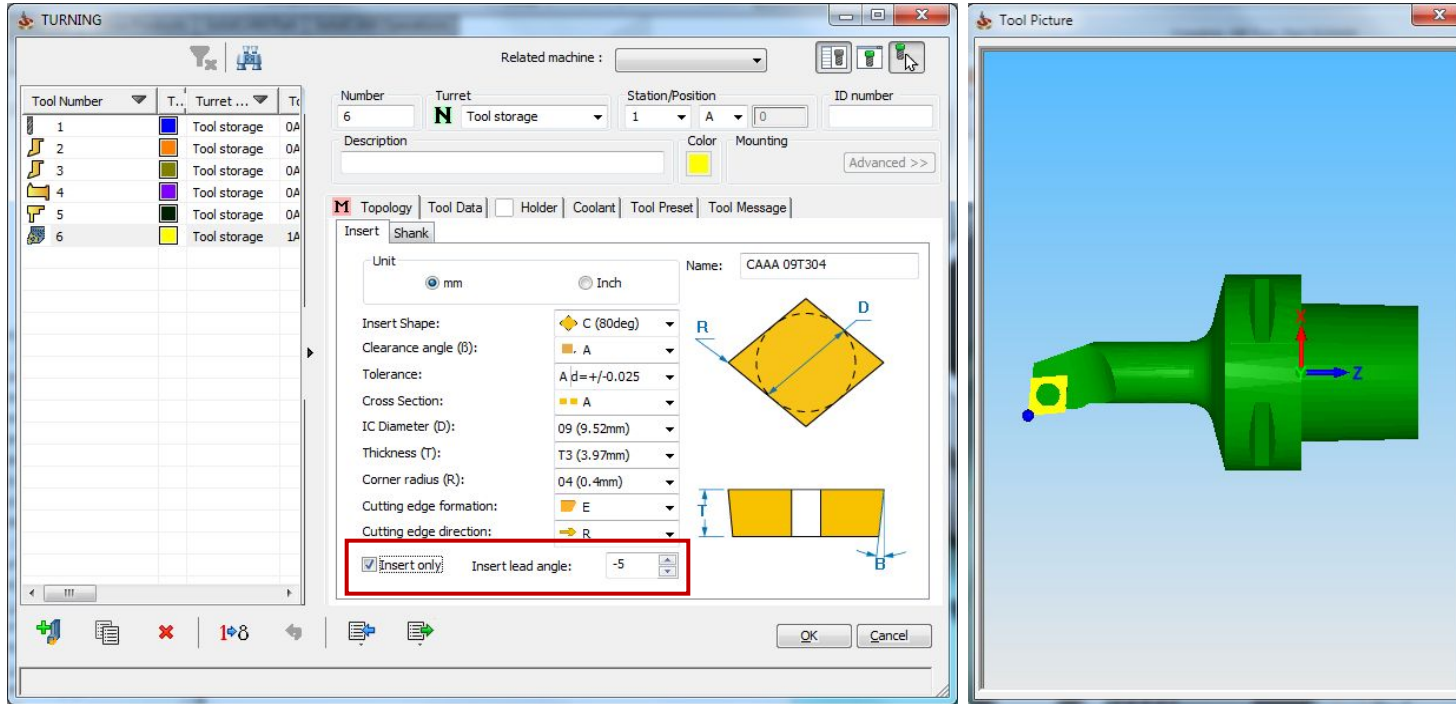
- Possibility to update feeds and spins in operations according to changes in tooltable
- Available from tooltable and from CAM-tree

Tooltable: Permanent tool keeps it's number during import



- If tool is marked as permanent – it will be imported without any change in any field

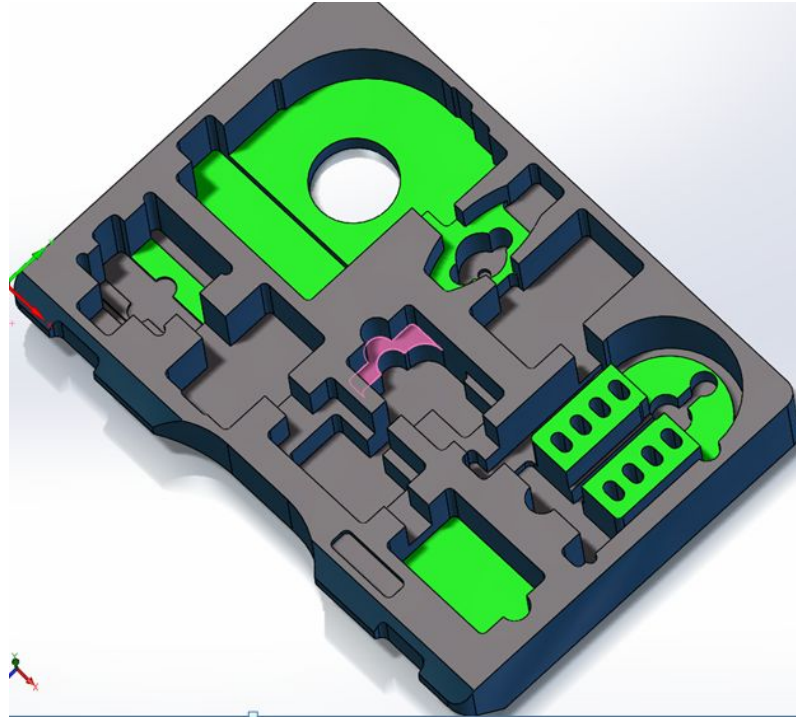
Tooltable: Insert only – add Lead angle



- Option to rotate insert, when working in Insert only mode, by setting Insert Lead angle

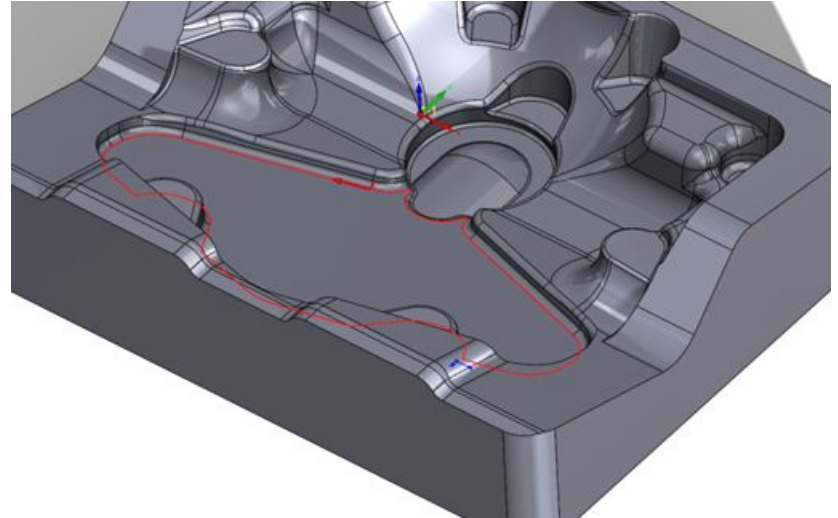
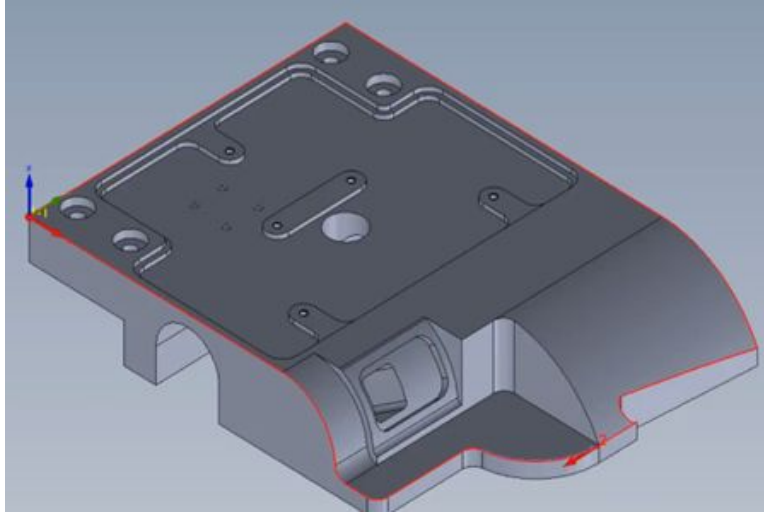
Geometry

Geometry: Filtering of Pocket Recognition faces by color



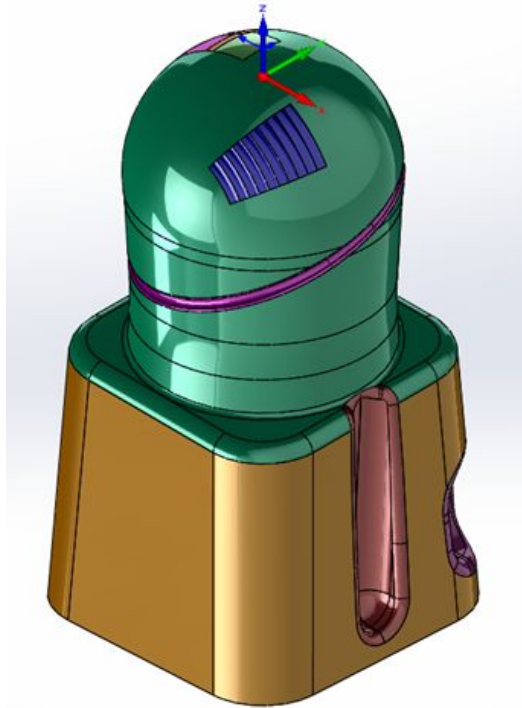
- Select only faces of specified color in Pocket Recognition geometry

Geometry: Automatic Curve propagation with Tangent and Delta-Z



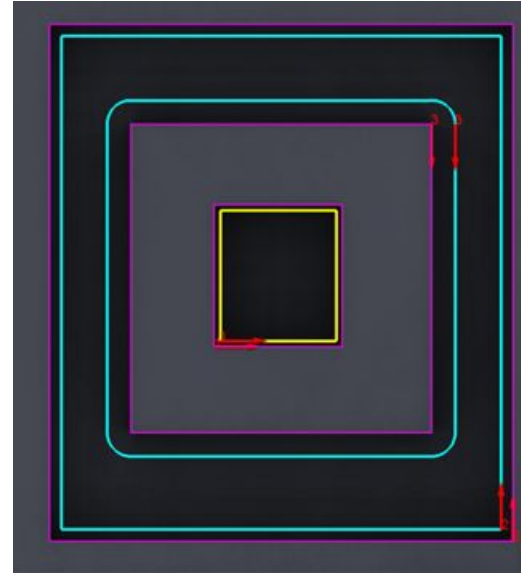
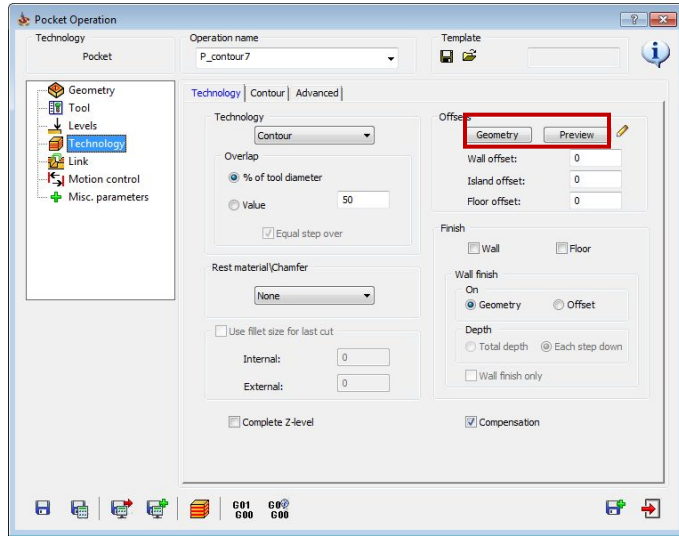
- **Creating chain geometries has been made a lot easier - this option enables you to use a combination of Tangent and Delta Z to create Multi level Chains.**

Geometry: Selection of faces by color



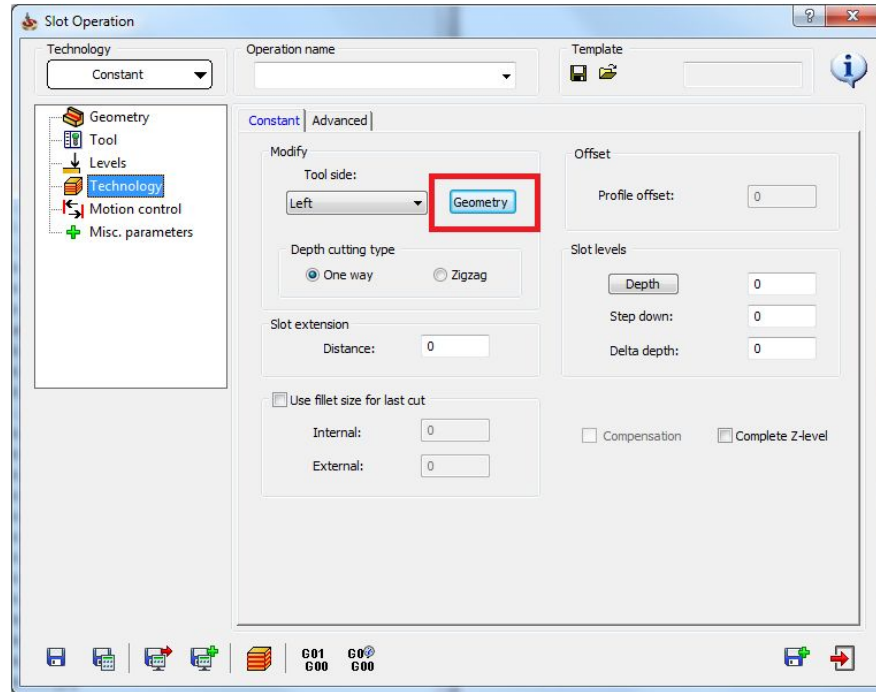
- Select only faces of specified color in HSS and 5x geometries

Pocket geometry: Option to add Offset



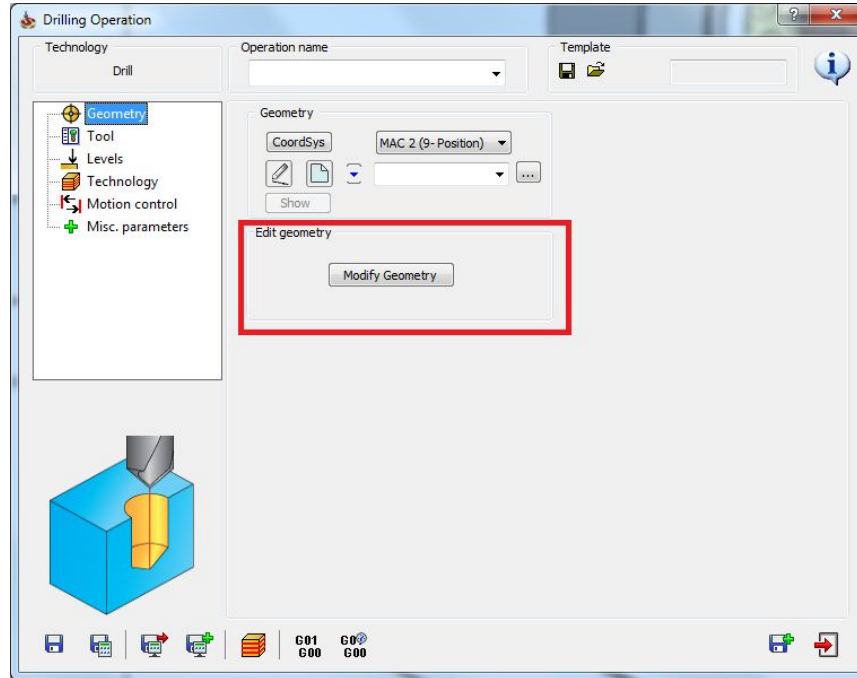
- Option to add offset to geometry of Pocket operation
- Enable user to handle tolerances, without defining new geometry

Slot geometry: Option to add Offset



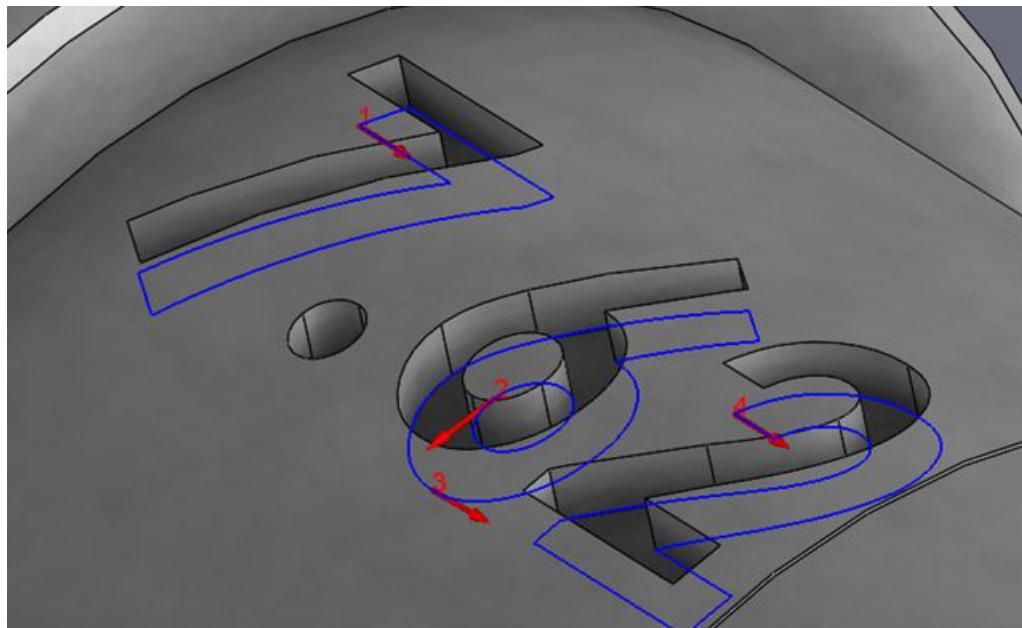
- Option to add offset to geometry of Slot operation
- Enable user to handle tolerances, without defining new geometry

Drill geometry: Modify option



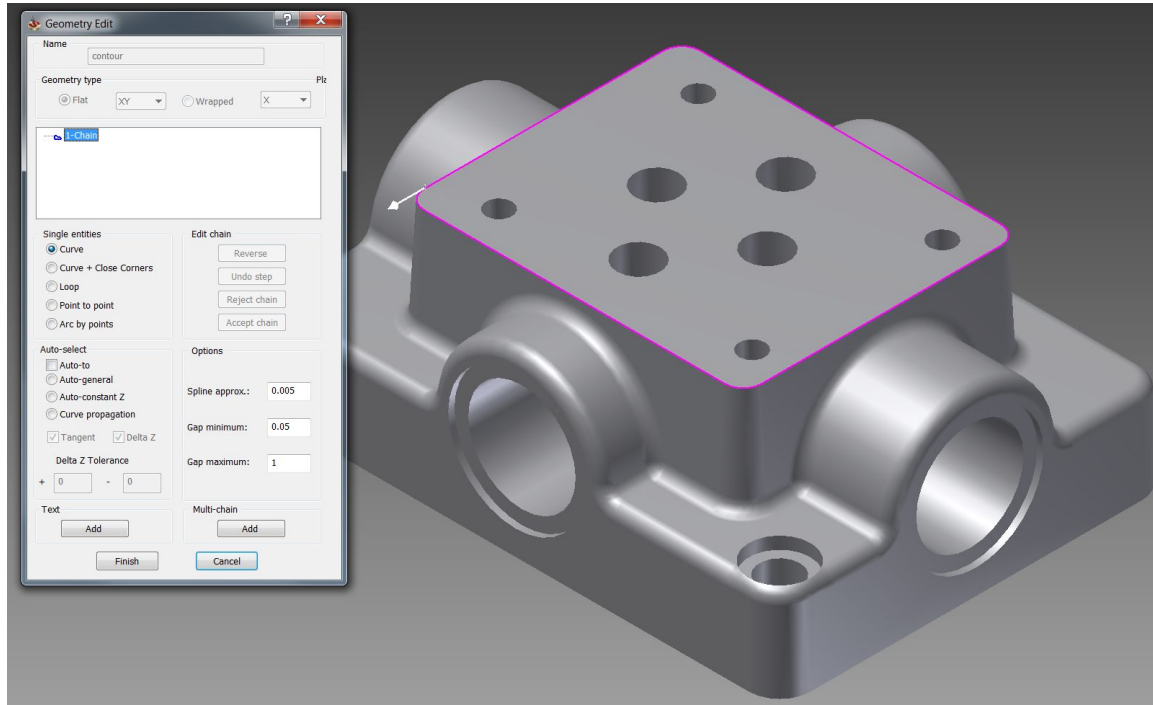
- X and Y shiftings available per hole for Drilling geometry

2D Geometry: Changes in wrapped geometry definition



- **User Interface changes in Wrapped geometry definition**

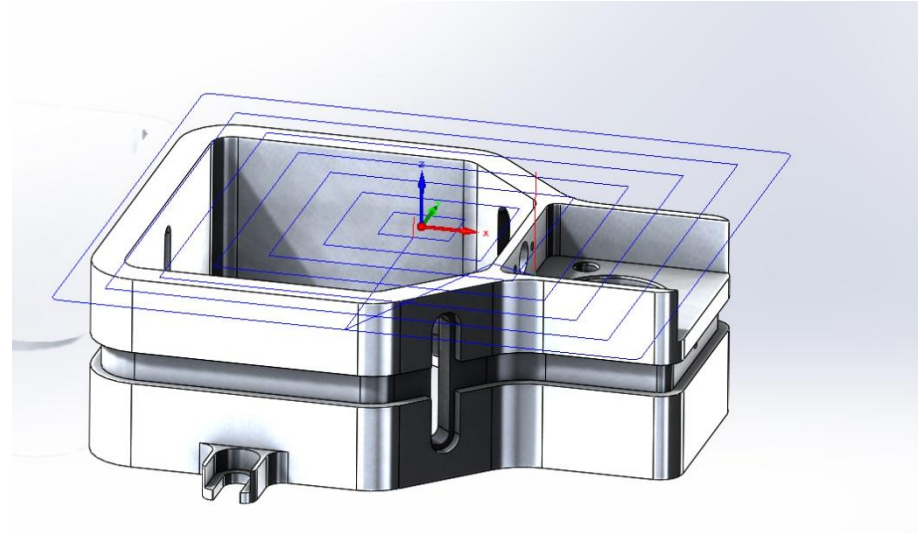
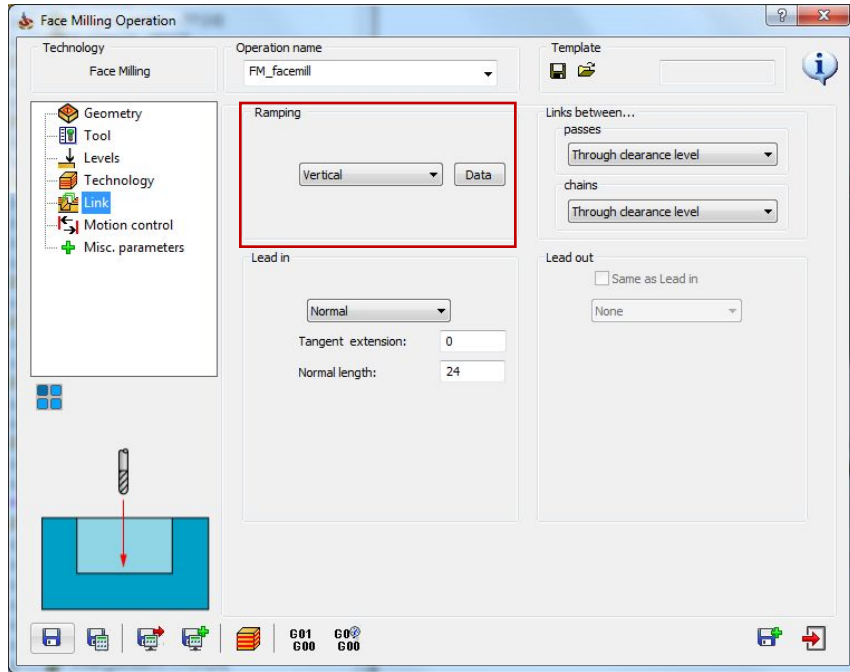
2D Geometry: Reverse geometry by F5 button



- Easy way to reverse geometry – press F5 button on keyboard

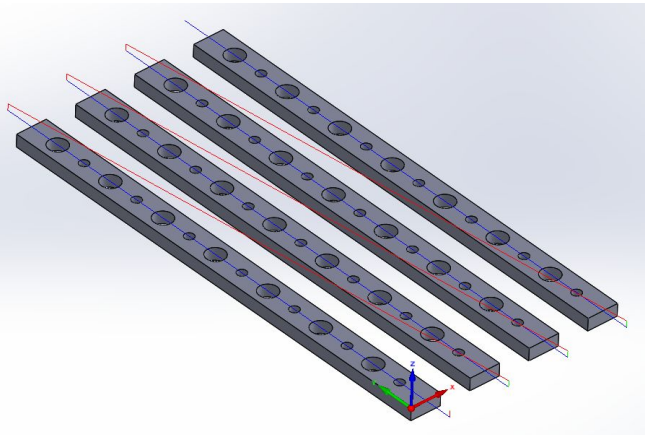
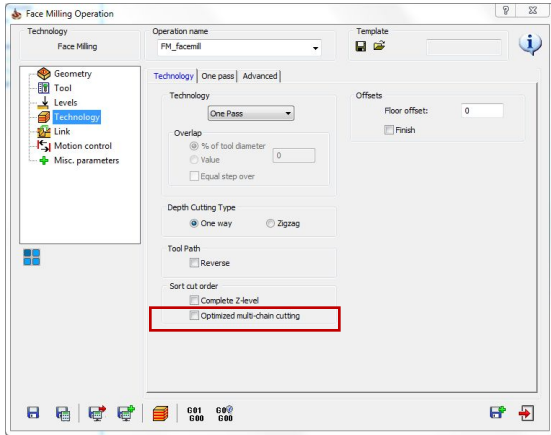
2.5D Mill

Face milling: Vertical ramping option

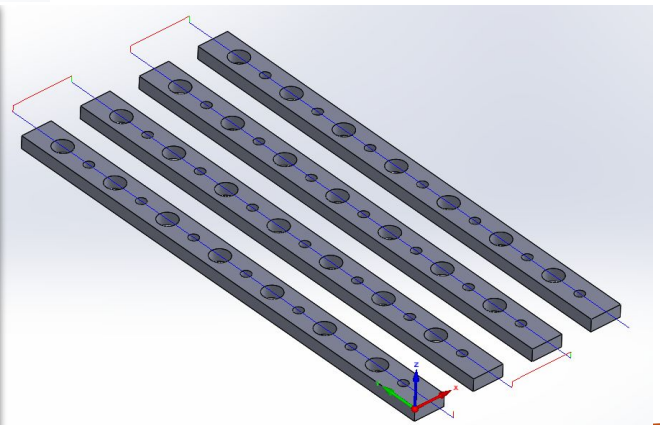
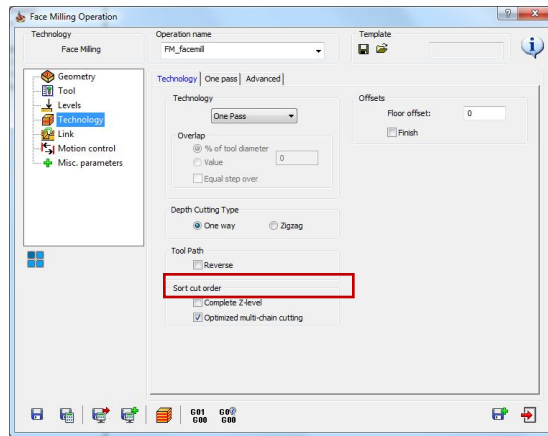


- New option “Vertical” for Ramping options in Face milling operation

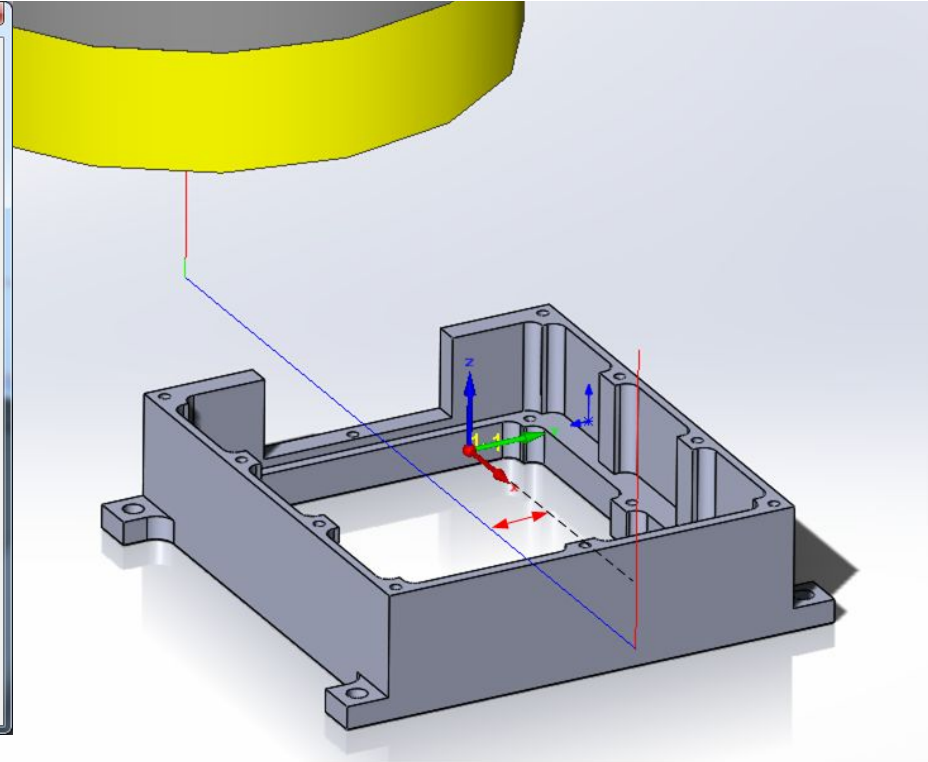
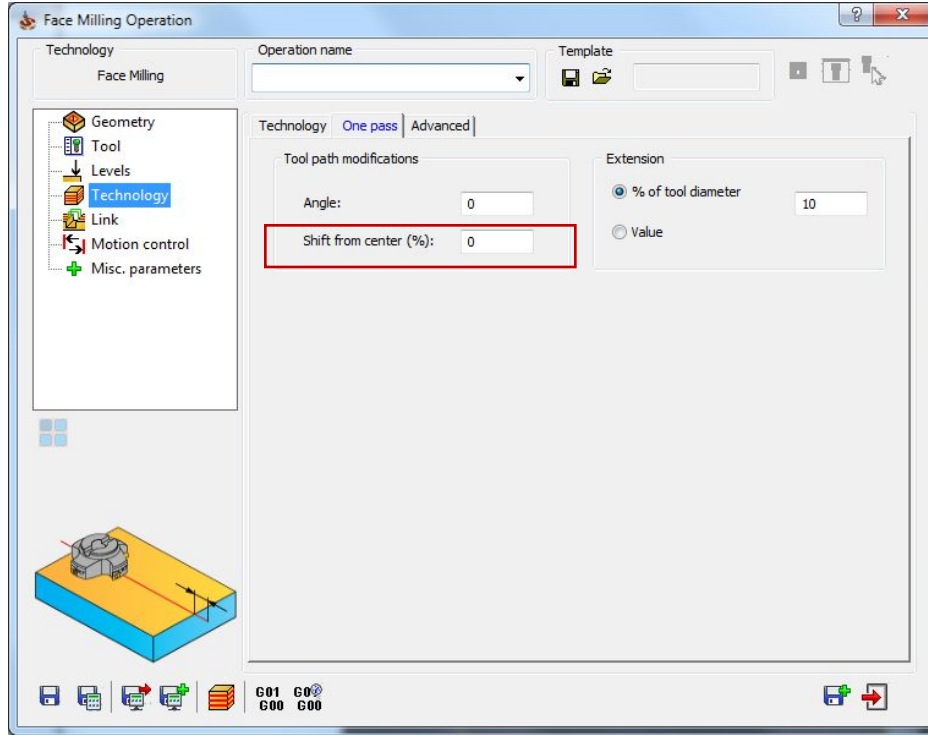
Face milling: Cutting direction optimization



- For long geometries that require one way face milling – there is option to skip long non-cutting tool moves

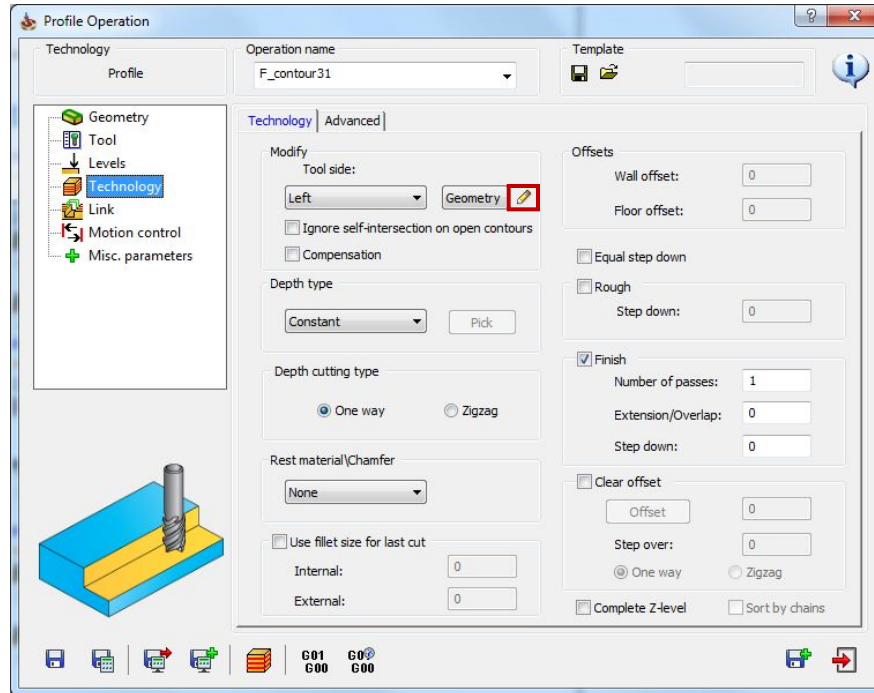


Face milling: Shifting from center



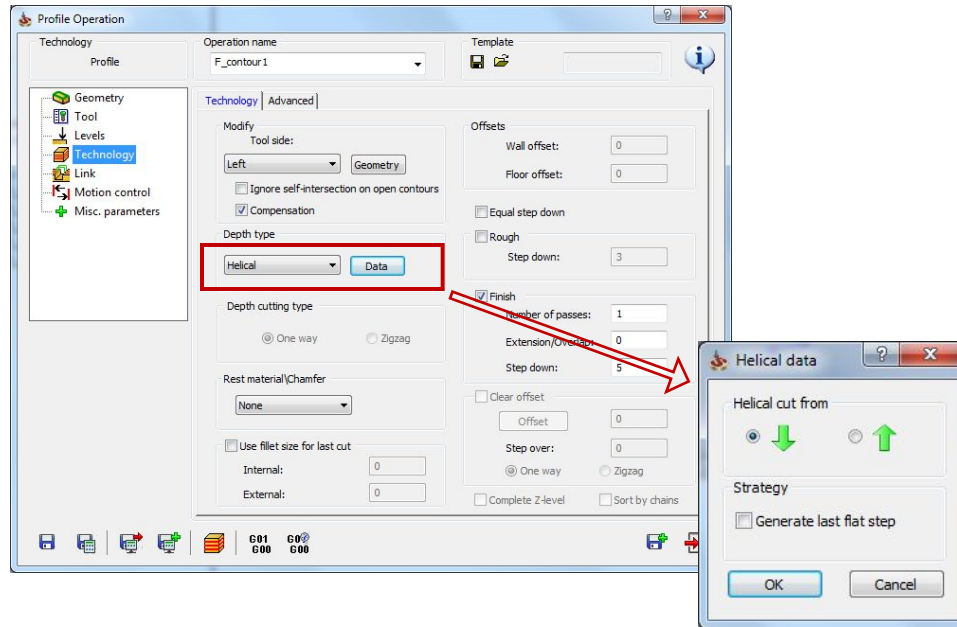
- Shift the single cutting pass by applying % of tool diameter

2.5D Mill: Mark by icon changes of geometry offsets



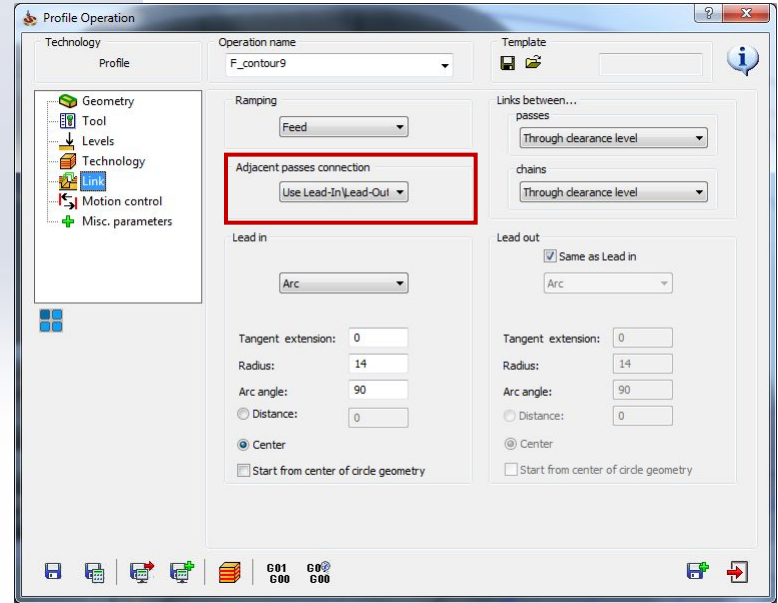
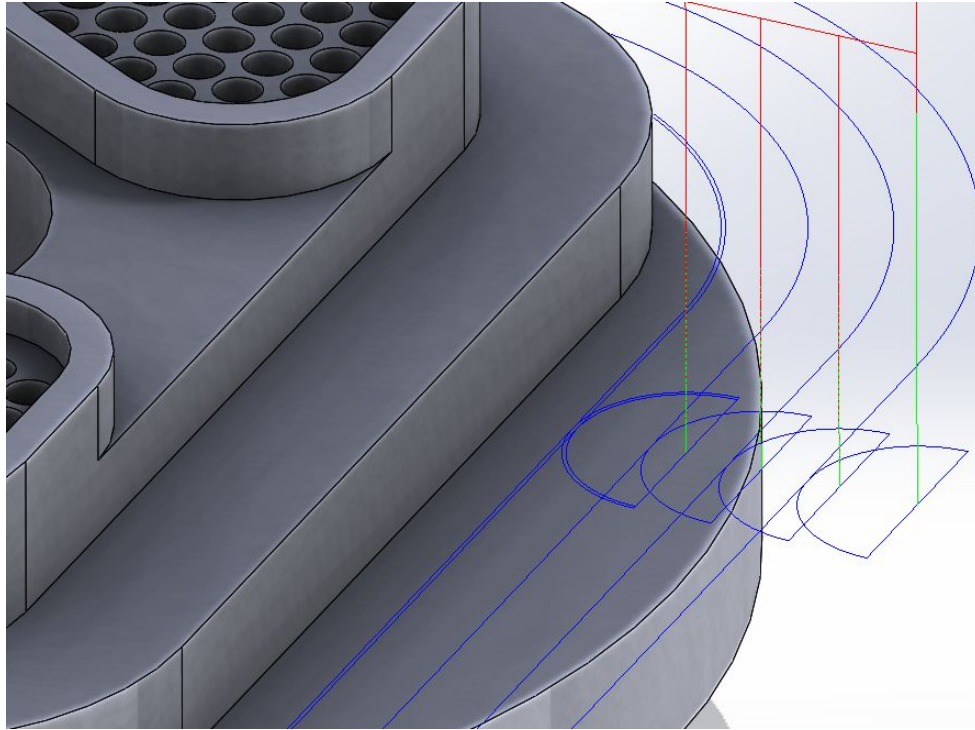
- If in an operation, modification of geometry was applied – an icon indicating a change occurred appears near Geometry button.

Profile: Helical movement improvement



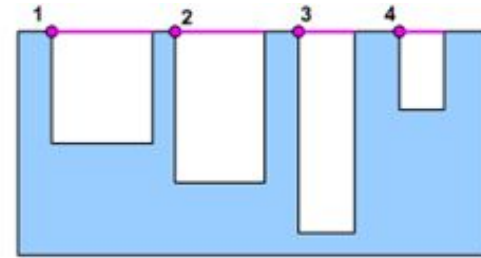
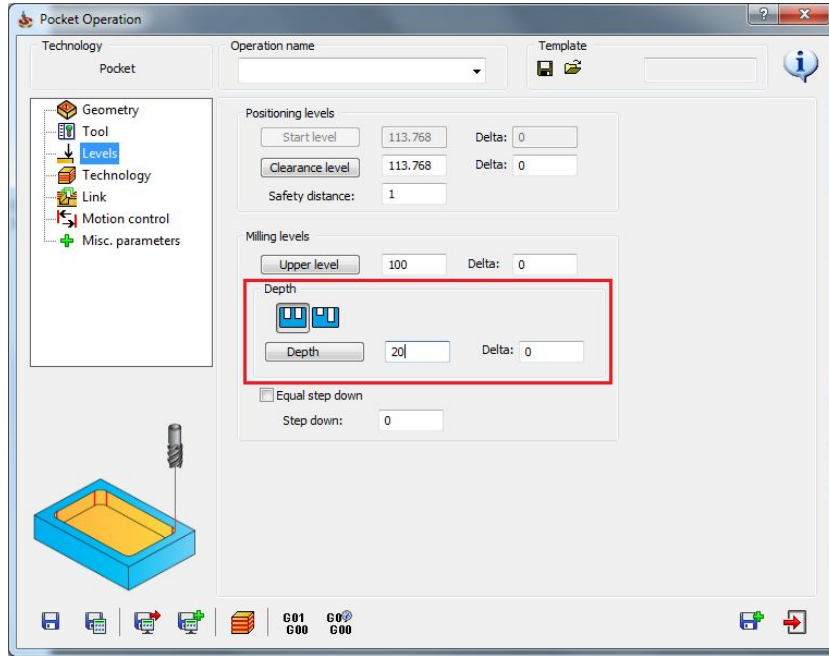
- Different start positions: From top to bottom, From bottom to top
- Optional flat circular movement at the bottom of helical cut

Profile: Lead in/out on each pass of Clear offset



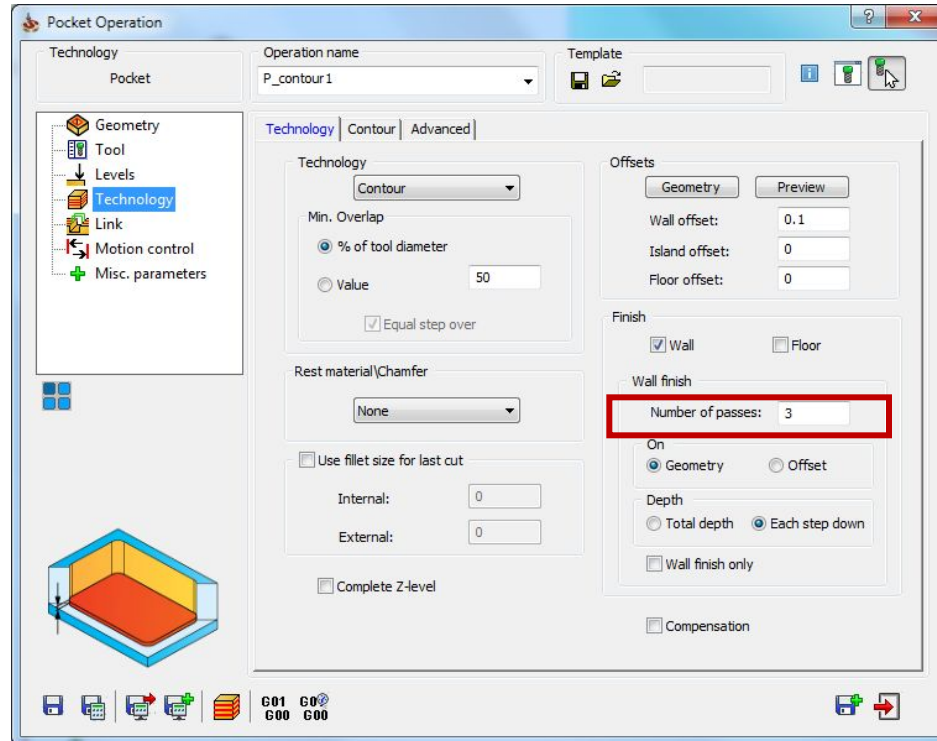
- Use lead in/out on each cut of Clear offset strategy

Pocket: Variable depth



- Define depth of pocket per chain => possibility to machine pockets with same start level but different depths, in same operation

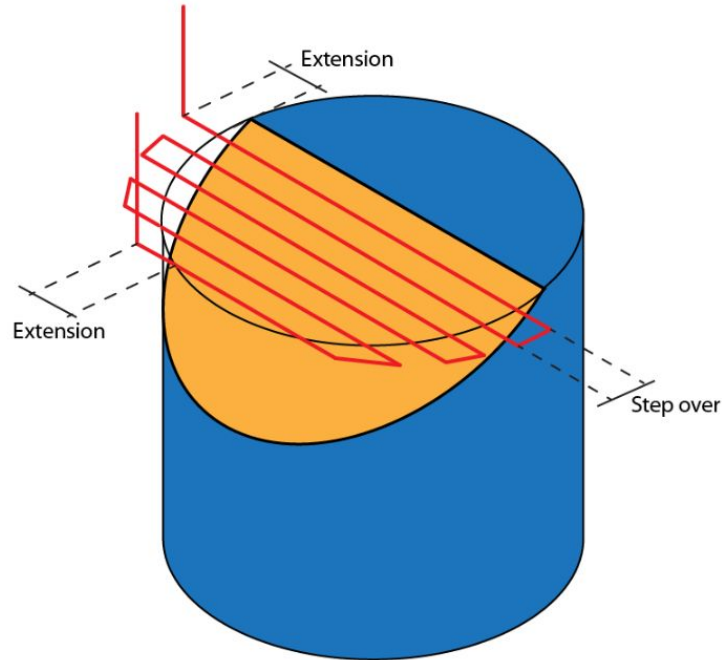
Pocket: Several wall finish passes



- Several finish passes at the same place in Pocket operation

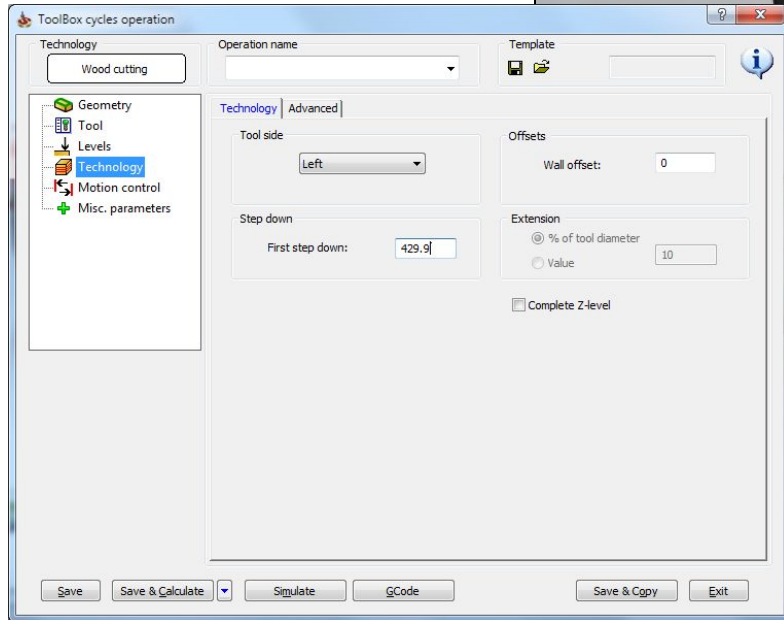
Toolbox

Toolbox: Angled cylinder



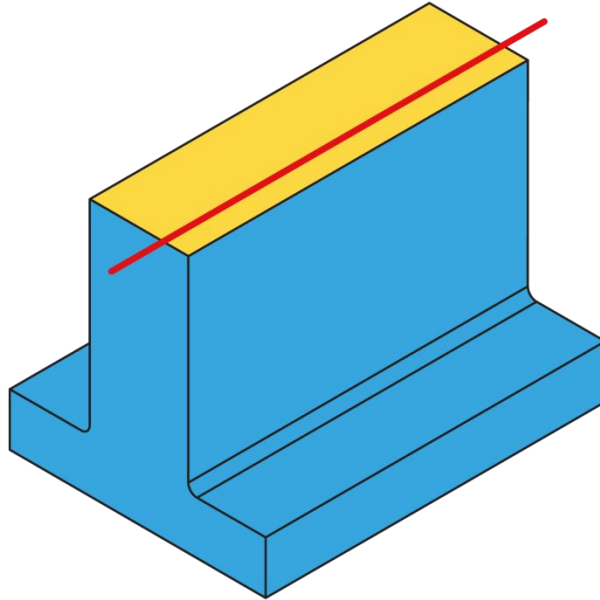
- Machining of angled cylinder
- Minimizing air cuts

Toolbox: Saw machining



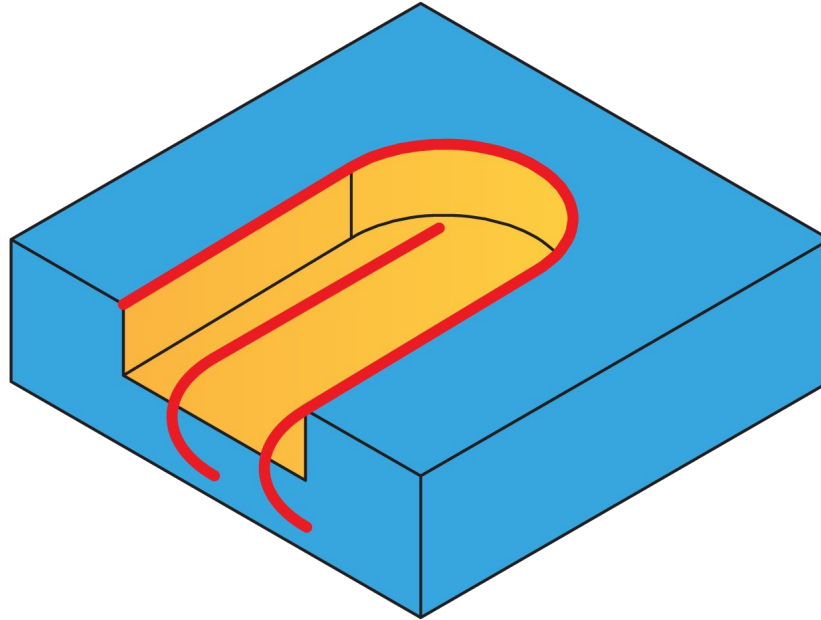
- Special strategy for wood cutting

Toolbox: Rib face milling



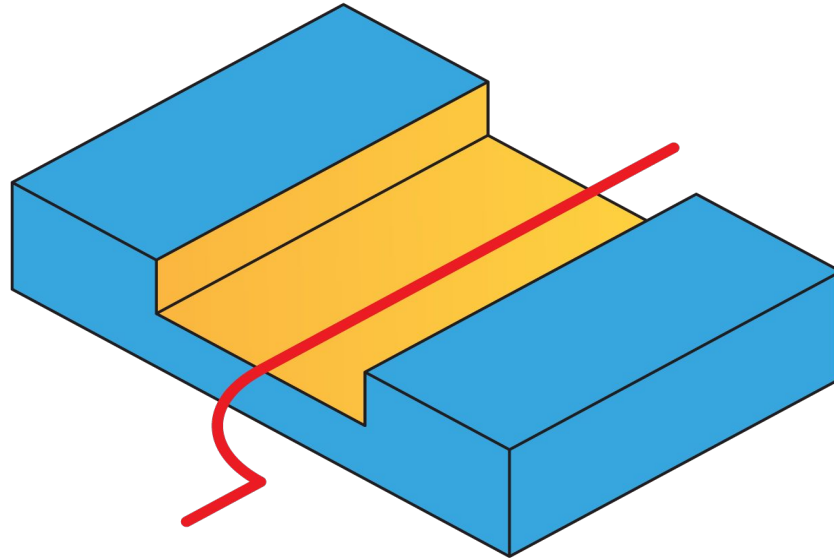
- **Cleaning of rib faces**

Toolbox: Roll into closed slot



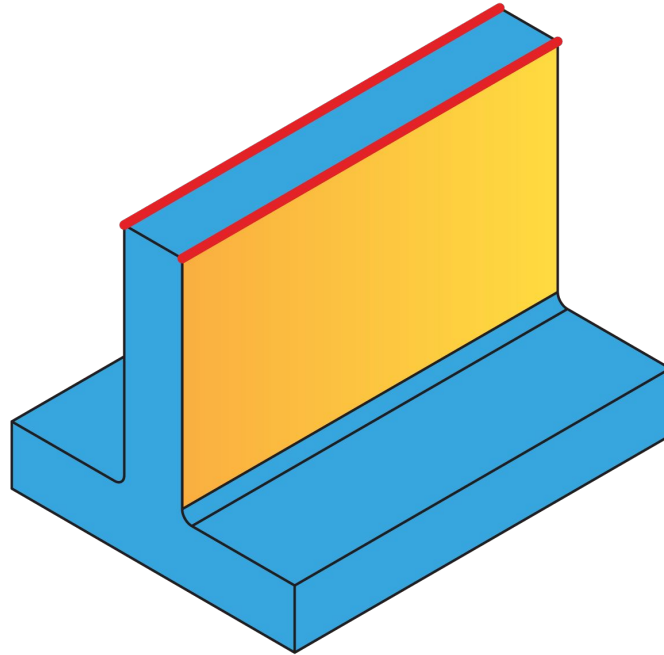
- **Constant tool loading, when entering a closed slot**

Toolbox: Roll into open slot

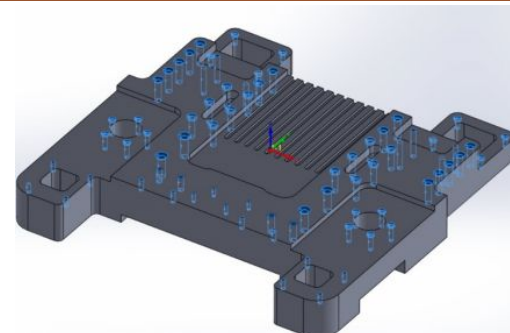
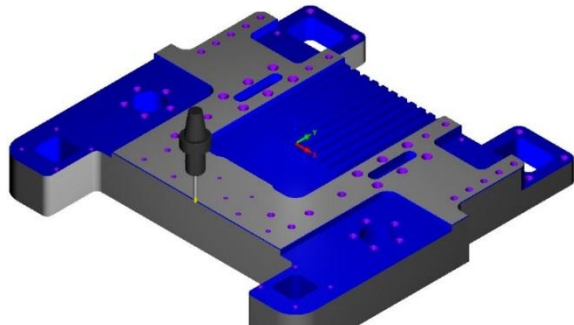


- **Constant tool loading when entering an open slot**

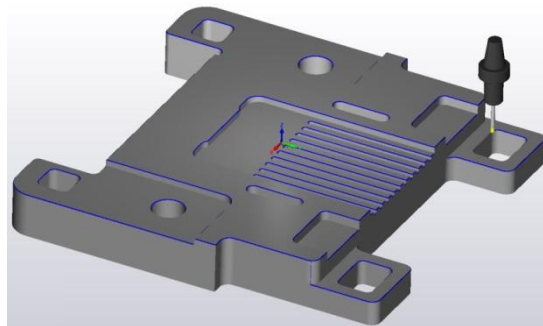
Toolbox: Thin wall machining



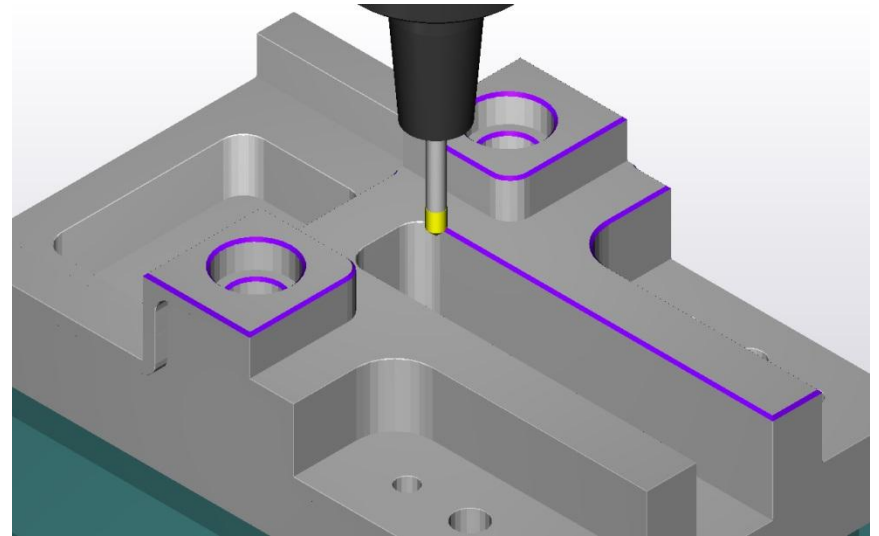
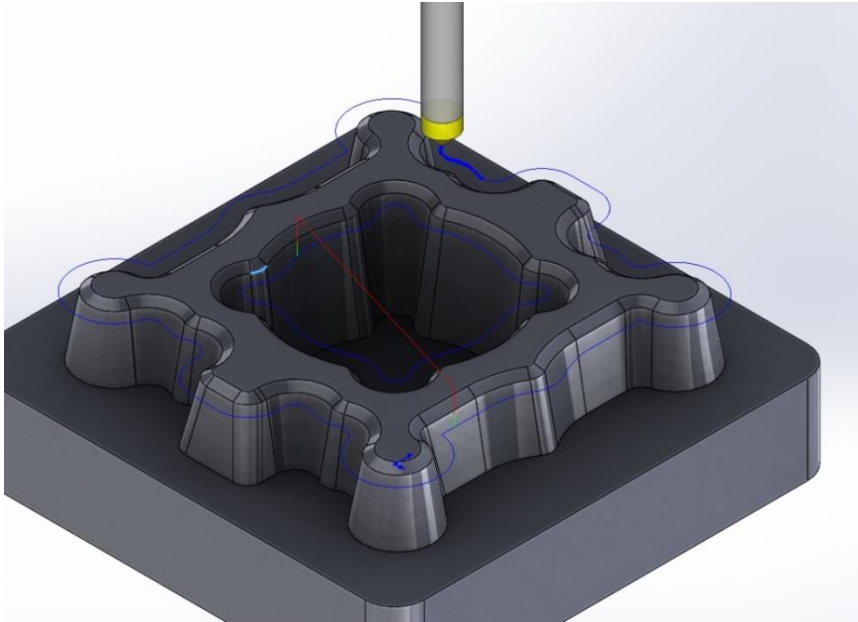
- **Special strategy for 2.5D thin wall machining**



Automatic Feature Recognition and Machining (AFRM)

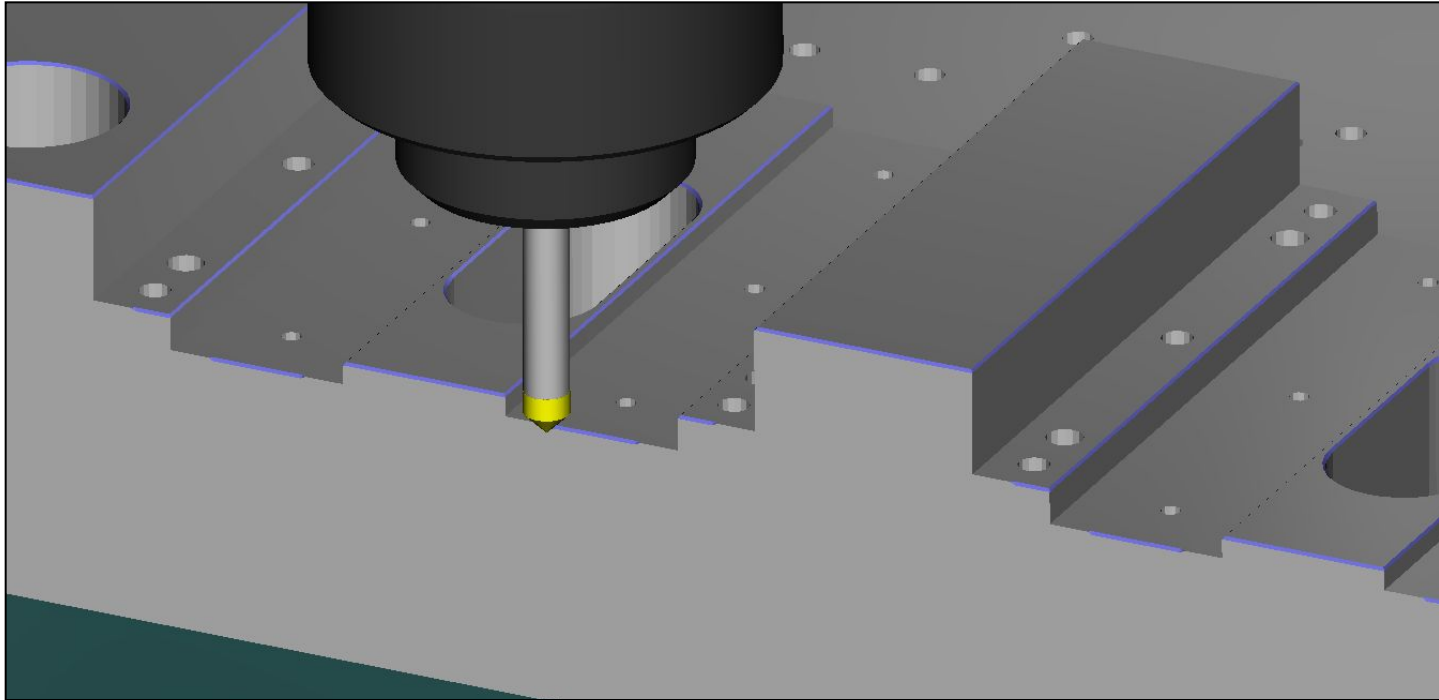


AFRM: Chamfer recognition and machining

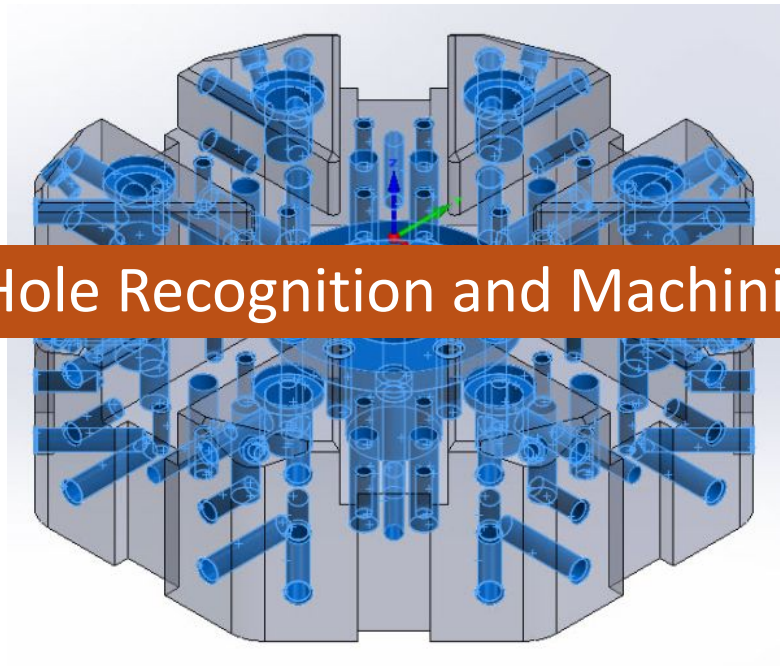


- Automatic recognition and machining of edges where it is possible to apply chamfer

AFRM: Chamfer recognition and machining



- The Automatic recognition of edges is smart and avoids gouging the walls

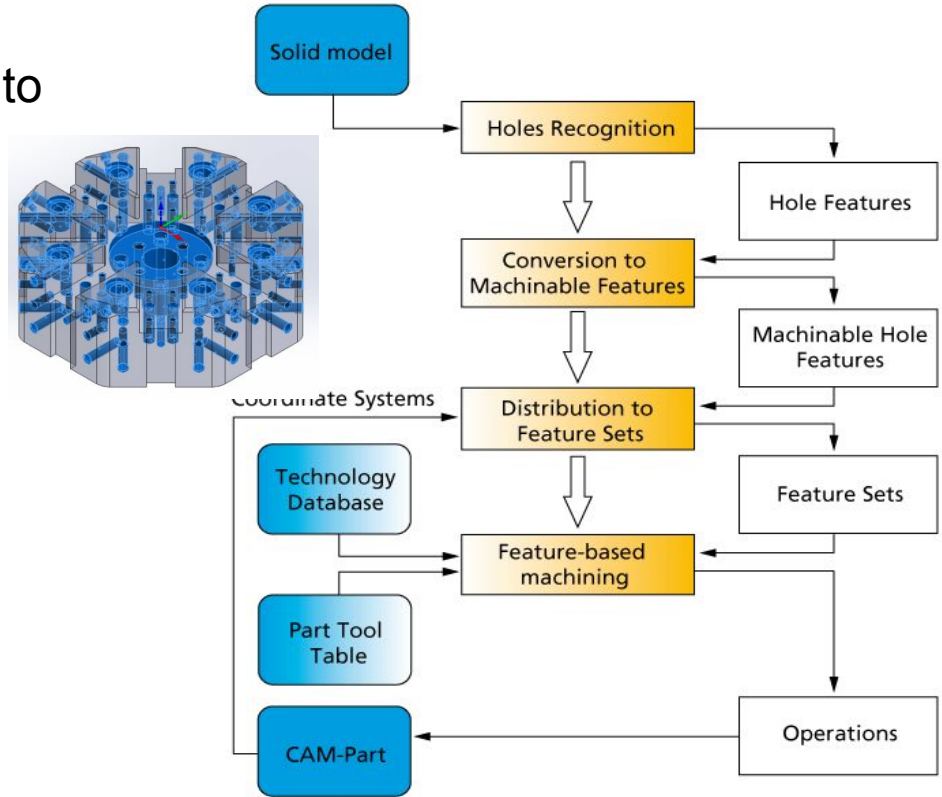


Automatic Hole Recognition and Machining (AHRM)

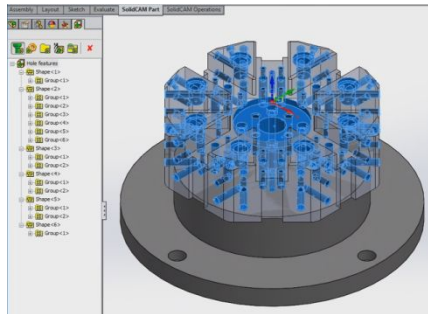
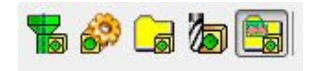
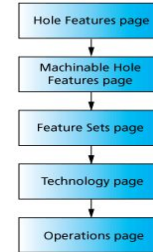
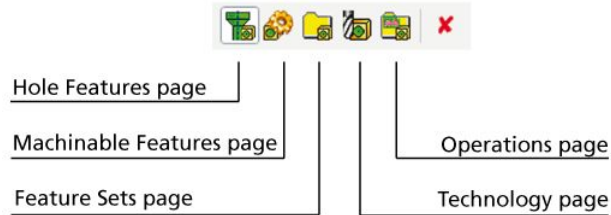
AHRM Review

InventorCAM's AHRM module is designed to automatically:

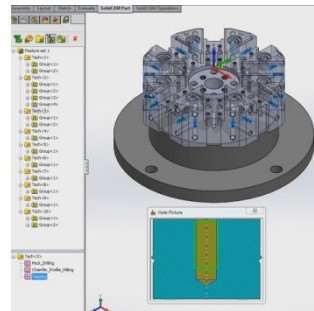
- Classify Shapes and Groups of Holes
- Convert to Machinable Features
- Select and/or Create All Required Tools
- Build Machining Technologies
- Generate Machining Operations
- Support all Work Position Orientations



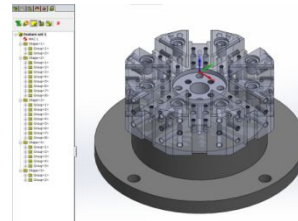
AHRM review – Process steps



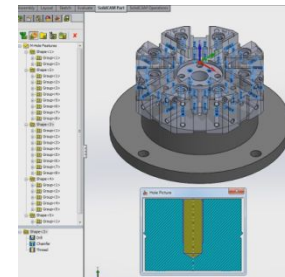
Step 1: Recognize Holes (Shapes & Groups)



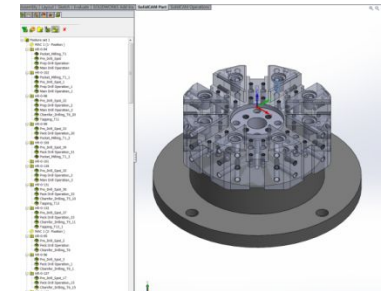
Step 2: Convert Holes to Machinable Segments



Step 3: Distribute Machinable Segments to Feature Sets

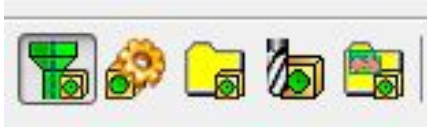


Step 4: Choose Technological Solution for Machinable Segments

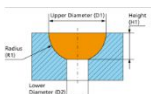
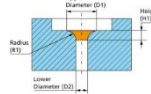
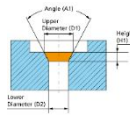
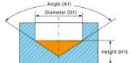
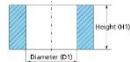


Step 5: Generate all Machining Operations

AHRM review - Step 1: Recognize Holes

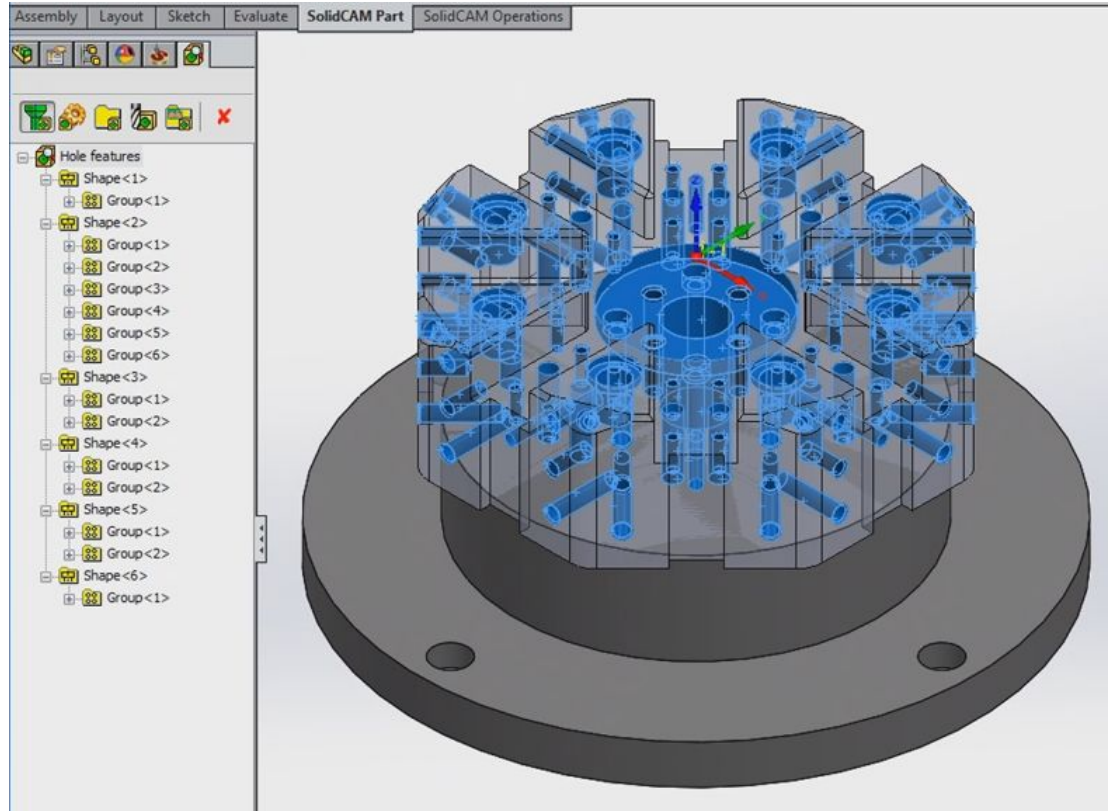


Step 1: Recognize Holes (Shapes & Groups)



Types of Hole Segments:

- **Cylindrical**
- **Cone**
- **Chamfer**
- **Planar**
- **Torus**
- **Sphere**

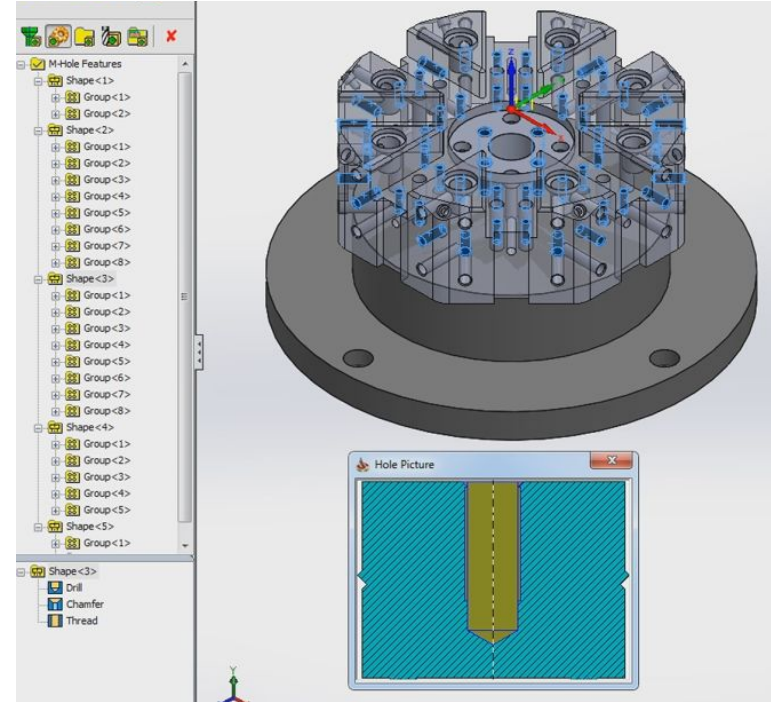
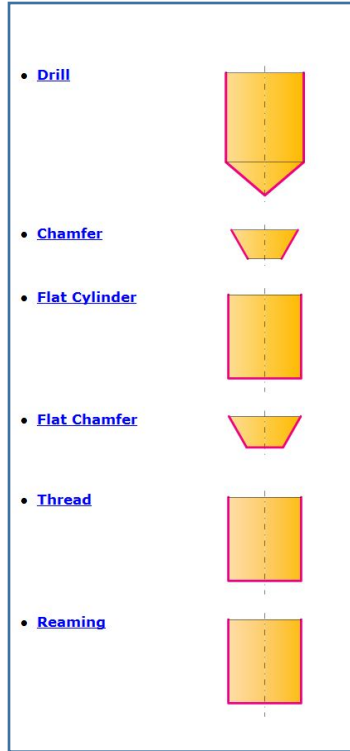


AHRM review - Step 2: Machinable segments

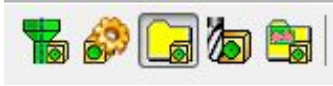


Step 2: Convert Holes to Machinable Segments

The **Machinable Hole Feature** consists of one or more **Machinable Hole Feature segments** that can be machined in one operation with the same tool.

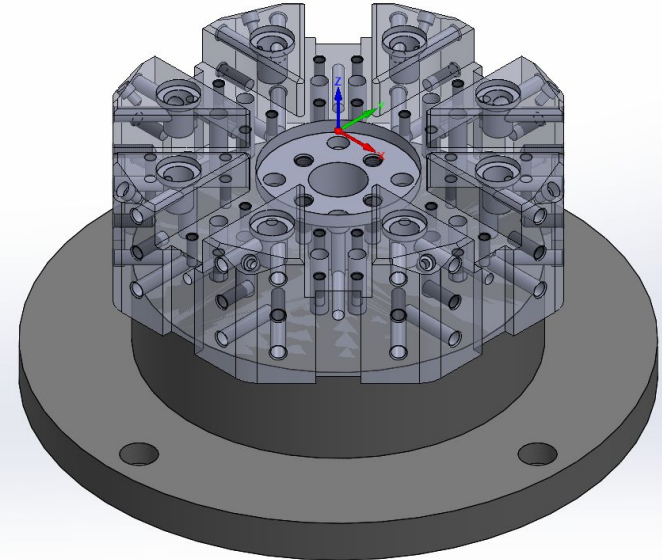
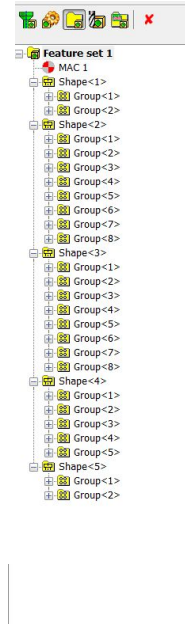
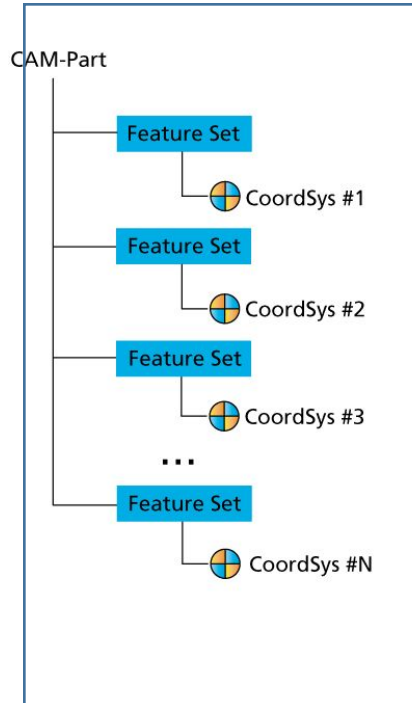


AHRM review - Step 3: Feature Sets



Step 3: Distribute Machinable segments to Feature sets

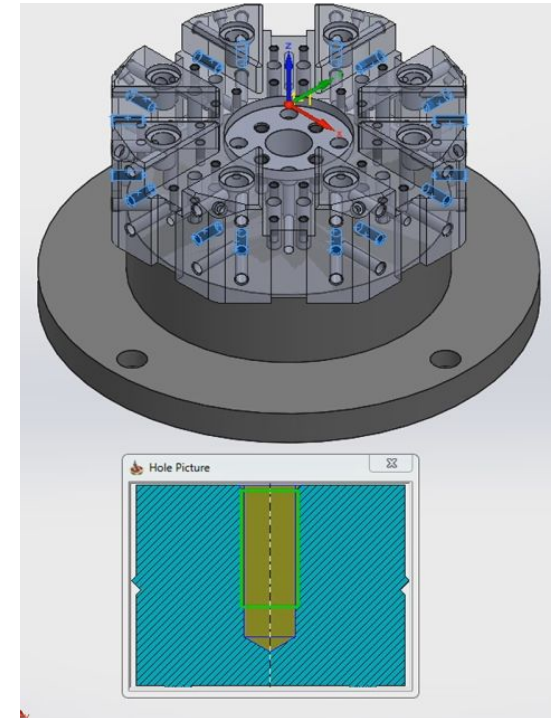
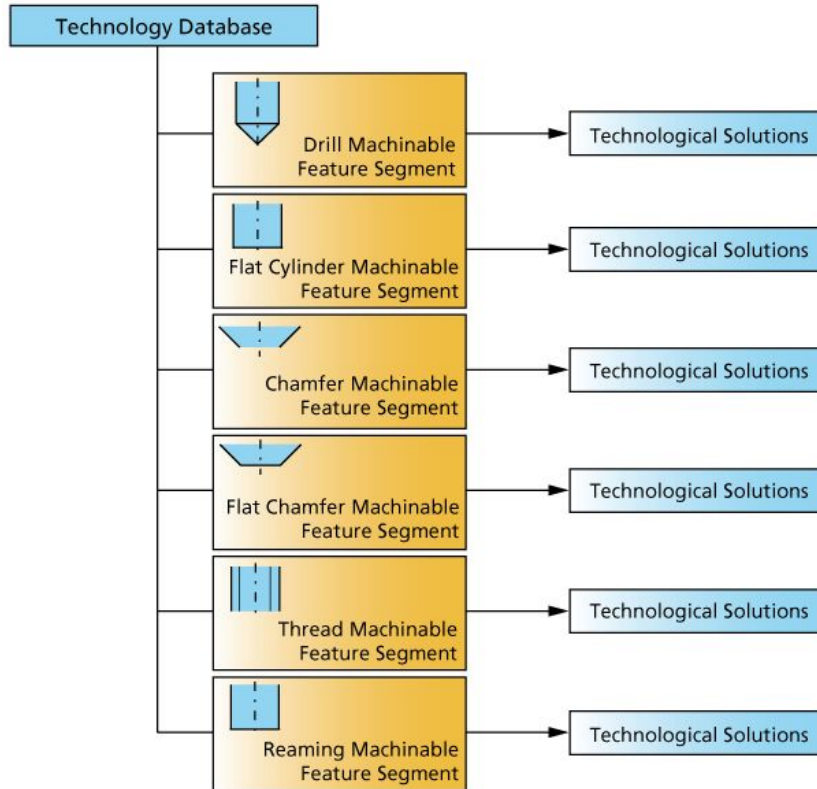
Feature Set is a number of Machinable segments that will be machined within the same setup using one Coordinate System.



AHRM review - Step 4: Technology Database Solution



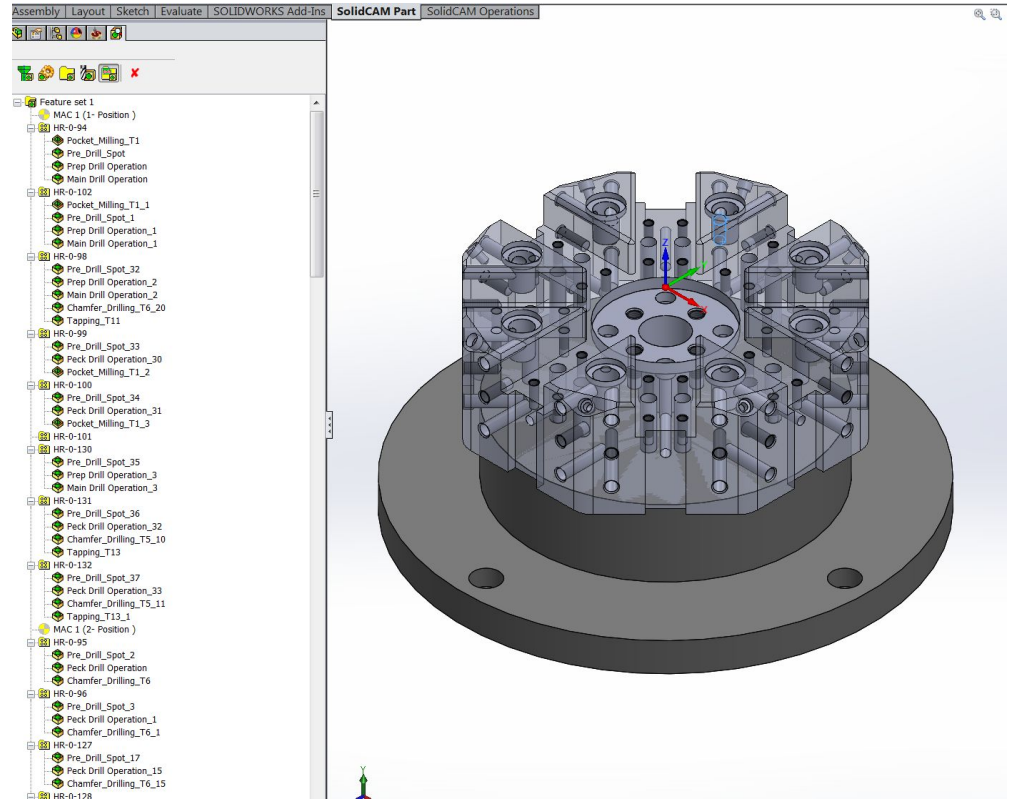
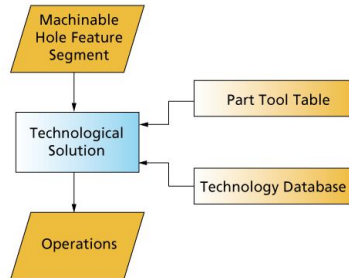
Step 4: Choose technological solution for Machinable Segments from Technology Data base



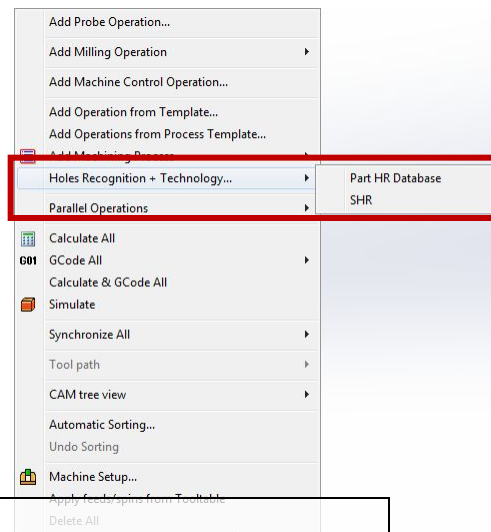
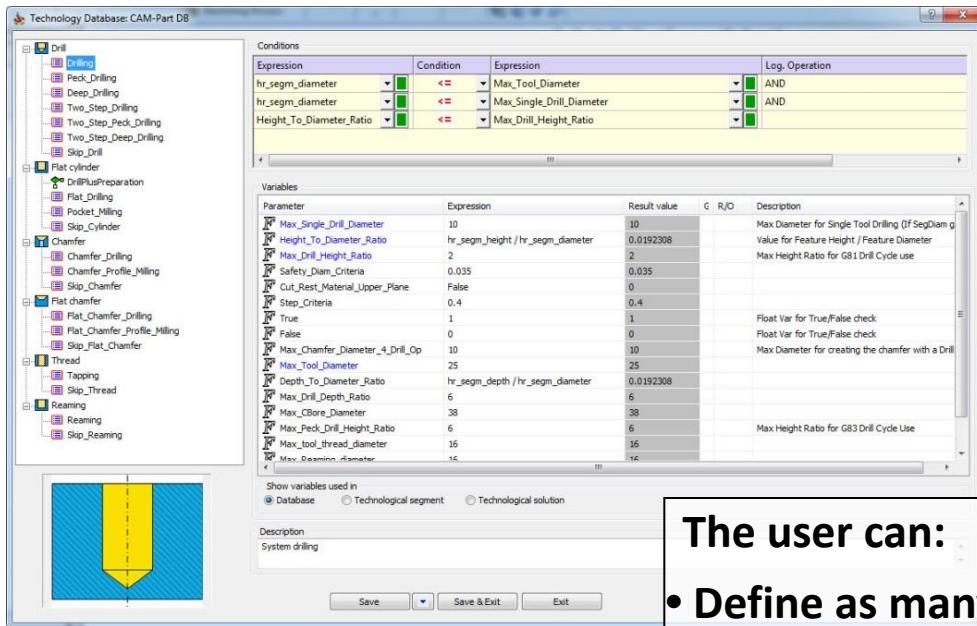
AHRM review - Step 5: Machining Operations



Step 5: Generate all machining operations

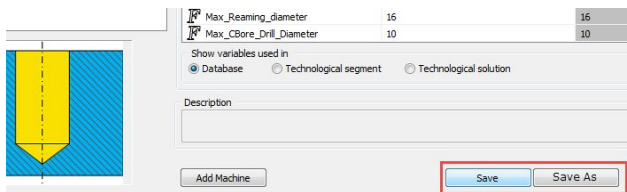


AHRM New: Limitless number of DataBase configurations

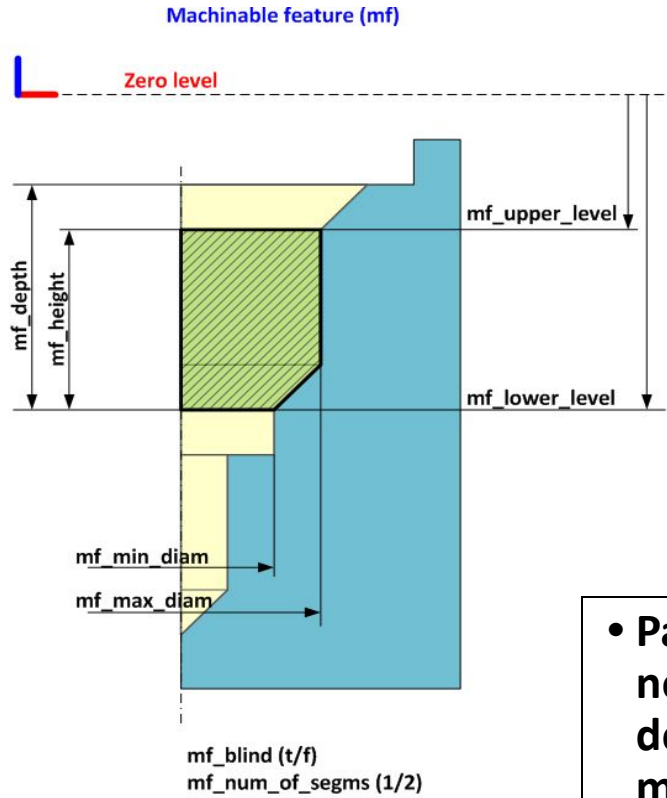


The user can:

- Define as many DataBases as he wants
- Select any of them for the process
- Edit the “Part DataBase ” Independently
- Save active DataBase As a NEW one



AHRM New: Machinable feature parameters (mf_xxx)



The screenshot shows the "Machining Process Define Manager" window. The "Used parameters" table lists various parameters and their expressions. A red box highlights the "Holes Recognition parameters..." menu item. Another red box highlights the "Holes Recognition parameters" dialog box, which lists parameters such as "hr_segms_upper_plane", "hr_segms_depth", "hr_prev_segms_diameter", "hr_segms_diameter", "hr_segms_angle", "hr_segms_height", "hr_segms_color", "hr_target_color", "mf_upper_plane", "mf_lower_plane", "mf_depth", "mf_height", "mf_Num_segms", "mf_Color", "mf_max_diameter", and "hr_segms_blind".

Parameter	Expression	Result value	G	R/O	Descri...
home_number	MAC 1 (1-Position)	MAC 1 (1-Position)	G	R/O	
Drill_Diameter	hr_segms_diameter				Diam...
Drill_Angle	hr_segms_angle		G	R/O	Tip an...
Drill_OHL	Drill_CL * 1.2				
Drill_CL	IF hr_segms_diam...				
Drill_Total_Length	IF hr_segms_diam...				
Drill_Depth	hr_segms_height				
Drill_Upper_Level	hr_segms_upper_p...				
Drill_Feed	75				
Drill_Spin	IF Drill_Diameter <				
Geometry					
Drill_Description	Main Drill				
Drill_St	Drill_CL				
Spot_Drill_Diam...	IF hr_segms_diam...				
Spot_Drill_Angle	90				
Spot_Drill_Arbor...	Spot_Drill_Diameter				
Spot_Drill_Total...	Spot_Drill_Diameter...				Ove
Spot_Drill_OHL	Spot_Drill_Diameter...				Spot

Unused parameters

Parameter	Expression	Result value
-----------	------------	--------------

Add new parameter

Type: Float

Name: Add Delete

- Parameters of machinable features are now available with prefix "mf_", describing all needed dimensions of a machinable feature (which is not always equal to the hole segment)

AHRM New: Conditional Logic Support

The screenshot displays the 'Machining Process Define Manager' window, which is used for configuring machining operations. The 'Current expression set' is set to 'GENERIC'. The 'Used parameters' table lists various parameters and their expressions, with a red box highlighting the 'spot_Drill_Diam...' parameter and its associated condition. The 'Conditions...' dialog box is open, showing a table of conditions and their values.

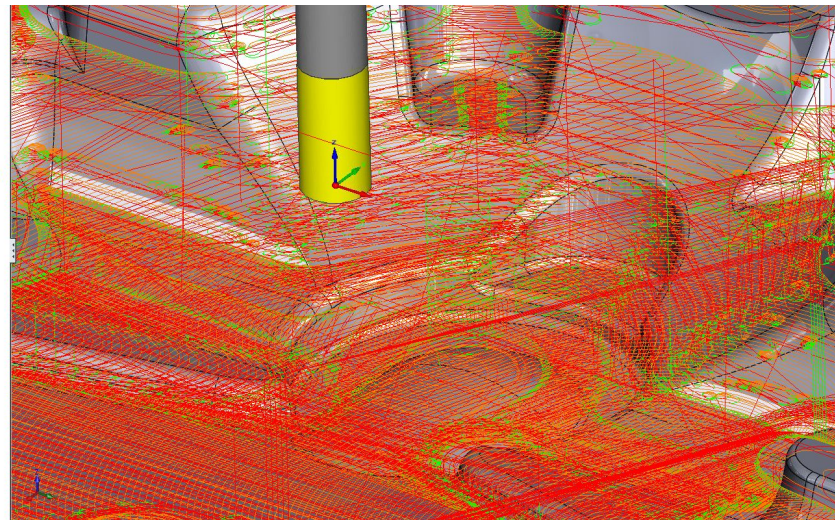
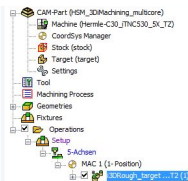
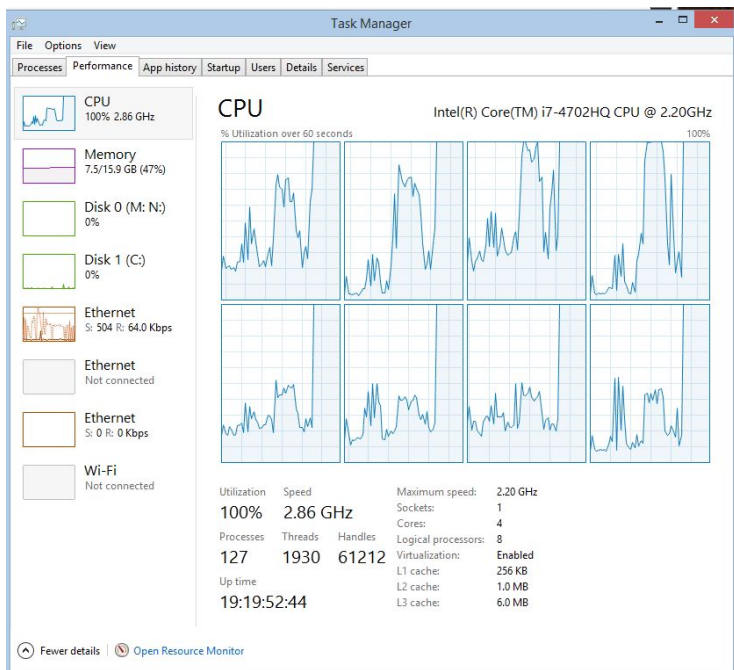
Parameter	Expression	Result value	G	R/O	Description
Drill_SL	Drill_CL				Shoulder length of Drill (Currently set to Cut Length)
spot_Drill_Diam...	IF hr_segm_diameter <= 5 THAN 6 ELSE_IF hr_segm_diameter > 5 AND hr_segm...				Diameter of Spot Drill Tool used for Pre Drill prep
spot_Drill_Angle	90				User-defined parameters...
spot_Drill_Diameter				F5	lgle
spot_Drill_Total...	IF hr_segm_diameter <= 3 THAN Drill_Diameter*2 ELSE_IF hr_segm_diamet...			F6	Diameter (Currently set to Spot Drill Diameter)
spot_Drill_OhtL	Spot_Drill_SL * 1.2			F7	ut from tool holder
spot_Drill_SL	IF hr_segm_diameter <= 3 THAN Drill_Diameter*13 ELSE_IF hr_segm_diamet...			F8	der Length... (Currently "Hard Coded" to 1"
spot_Drill_Descr...	Spot Drill			F9	scription on the Spot Drill
spot_Drill_Spn	IF Spot_Drill_Diameter == 6 THAN 2000 ELSE_IF Spot_Drill_Diameter == 12...			F10	Speed (RPM) for Spot Drill

Conditions	Expression	Condition	Expression	Log. Operation	Value
IF	hr_segm_diameter	<=	5		6
ELSE_IF	hr_segm_diameter	>	11	AND	12
ELSE_IF	hr_segm_diameter	>	11	AND	20
ELSE	hr_segm_diameter	<=	18		25

- Apply different values to parameters, according to user-defined condition in the Machining Process of the AHRM (e.g. Enables Spot drills to be chosen according to hole size)

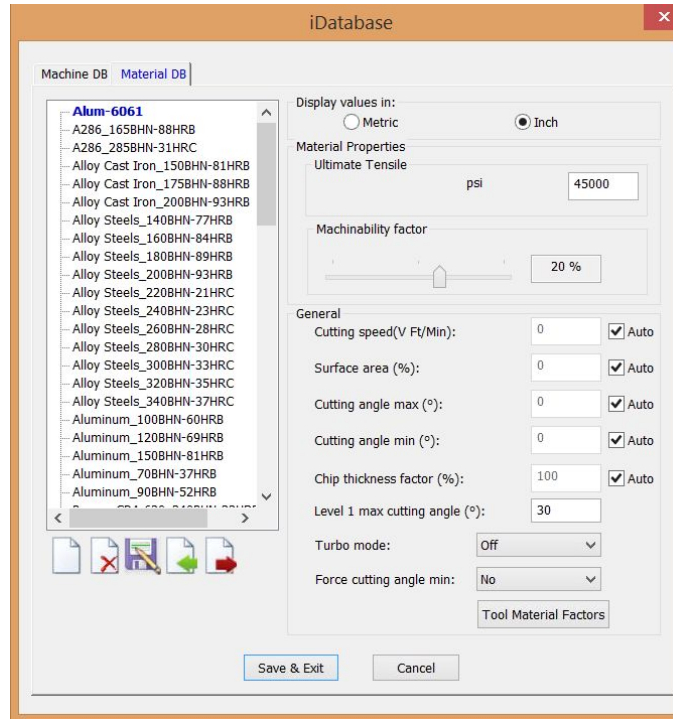
iMachining 2D & 3D

Parallel calculation in iMachining



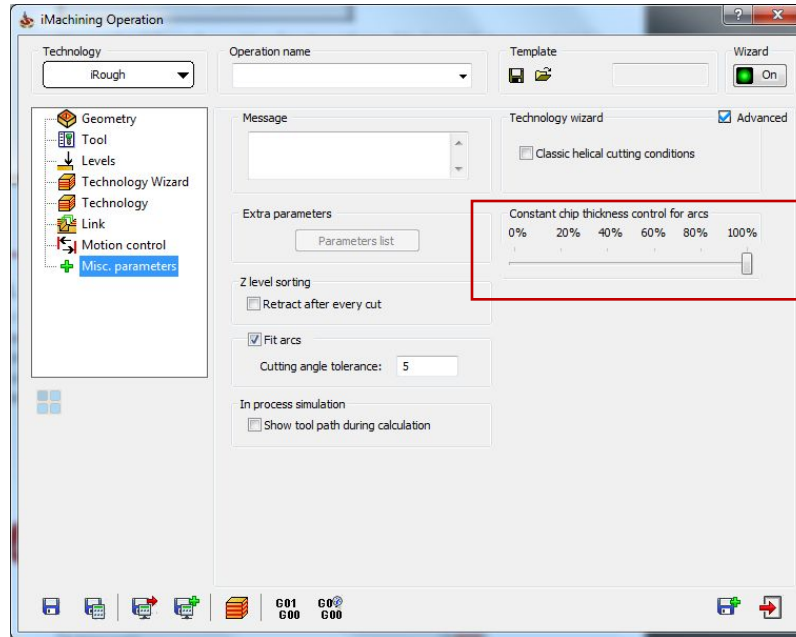
- Speeding up iMachining calculation by using multi cores & multi threading for parallel calculation

iDatabase Material: Machinability factor



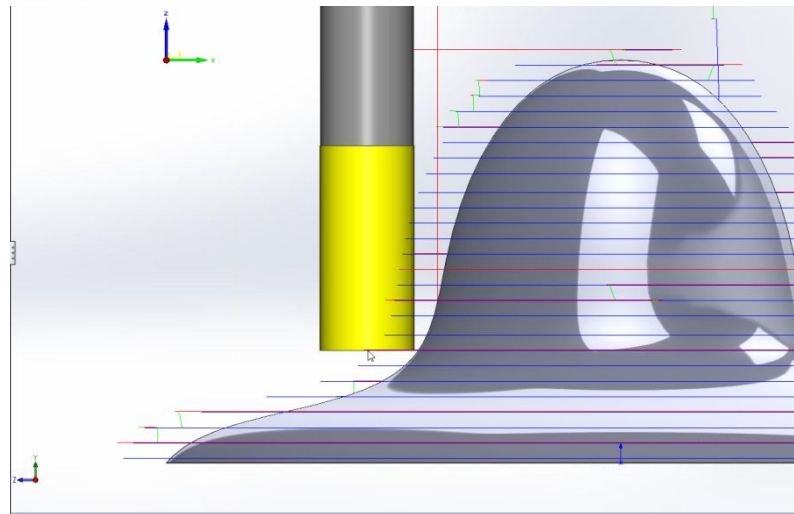
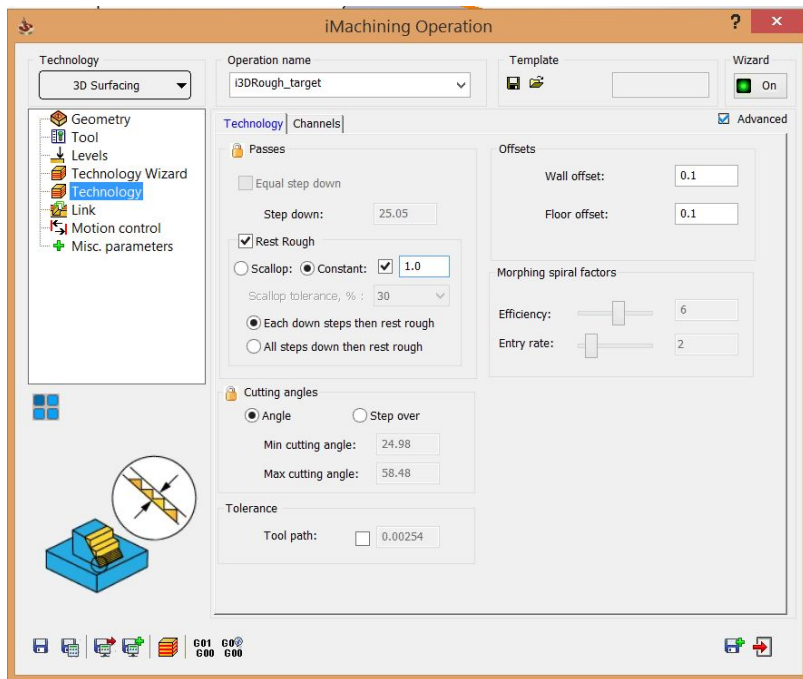
- Increase/Decrease Cutting Conditions, based on Machinability of specific material

iMachining: Constant chip thickness control for arcs



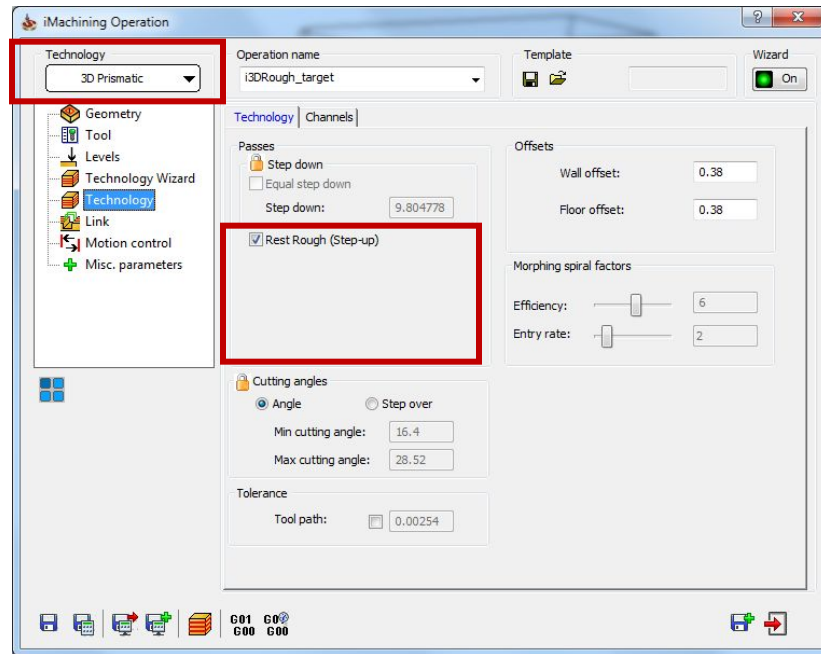
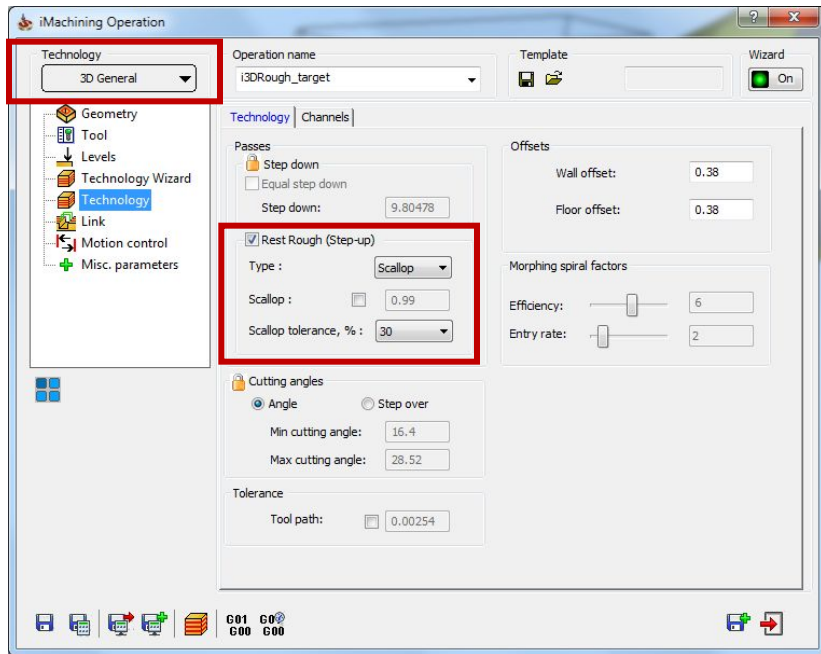
- Controls the feed correction for arcs
- Value of 0 means no feed rate correction, resulting in faster cutting and higher tool wear
- Value of 100 means complete correction, slower cutting but less tool wear

iMachining 3D: Constant Step up



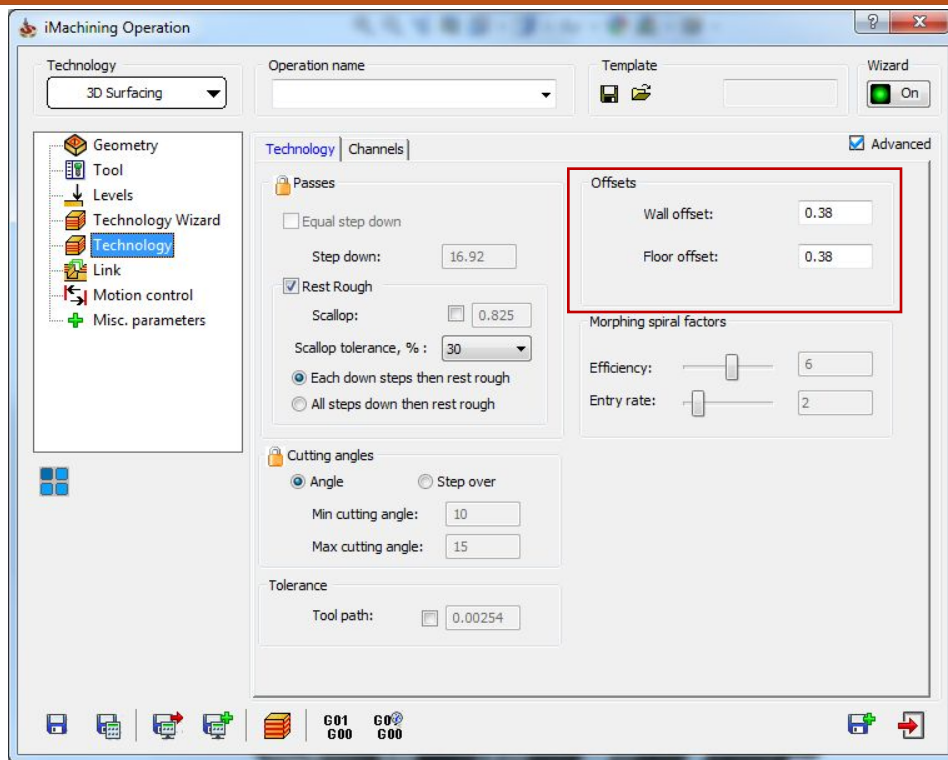
- iMachining 3D option: Constant Set up (as alternative to Scallop)

iMachining 3D: Prismatic parts machining technology



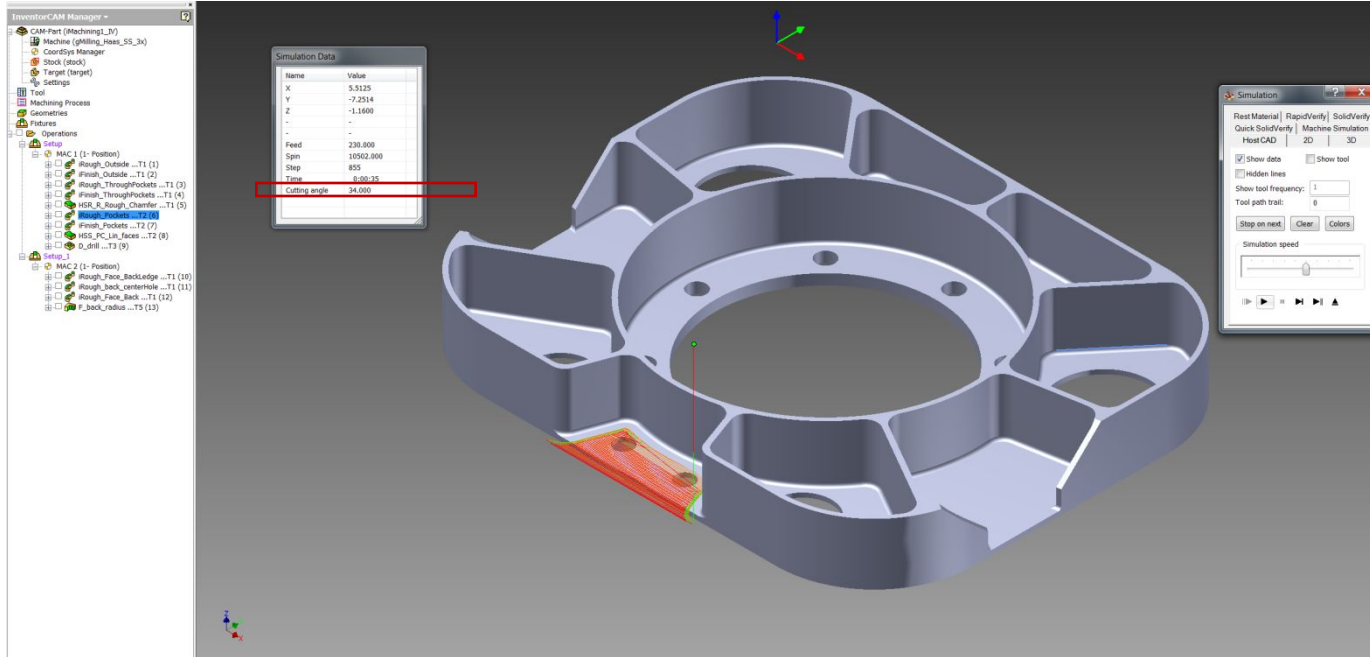
- iMachining 3D technology for Prismatic Part Machining with automatic scallop calculation

iMachining 3D: Floor offset



- Enables you to define a Floor offset that is separate from the Wall offset

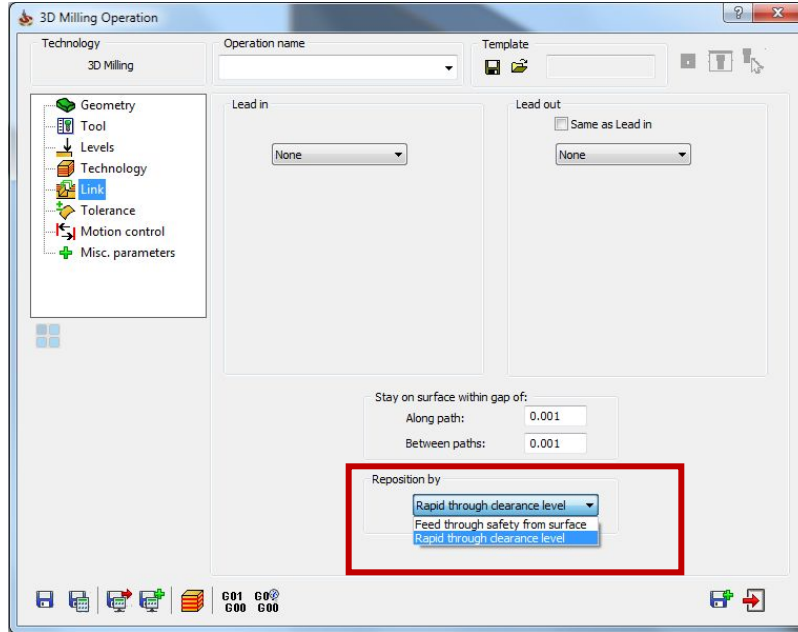
Show Cutting angle in simulation of iMachining



- Showing the cutting angle in iMachining simulation

3D Milling

3D Milling: Reposition options

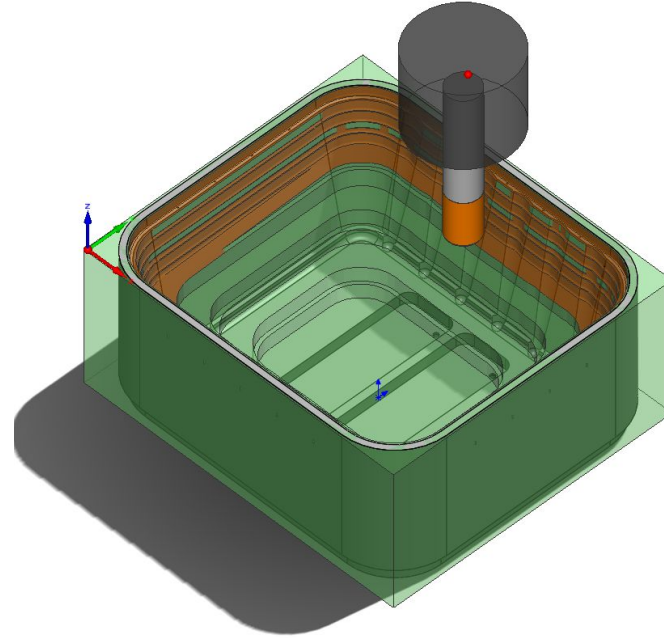
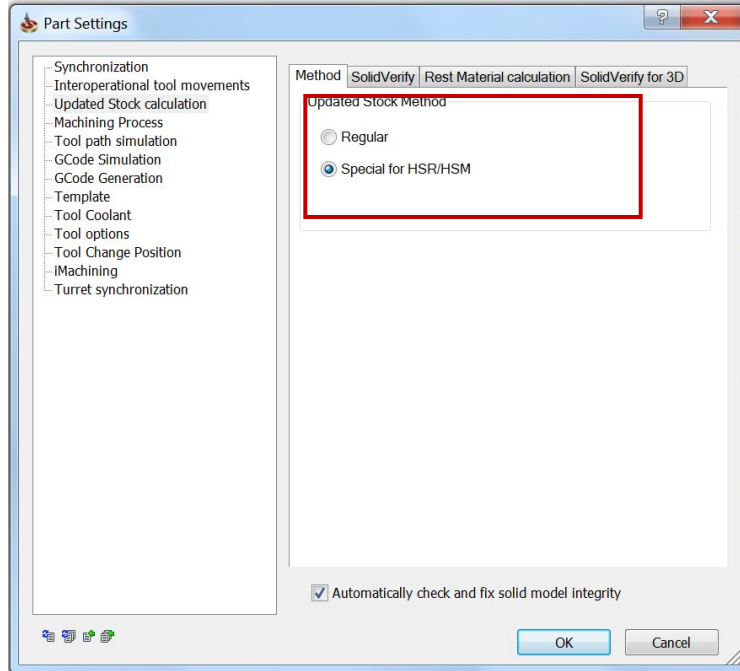


2 options of tool reposition:

- By Rapid move through clearance plane
- By Feed move through safety distance from the surface

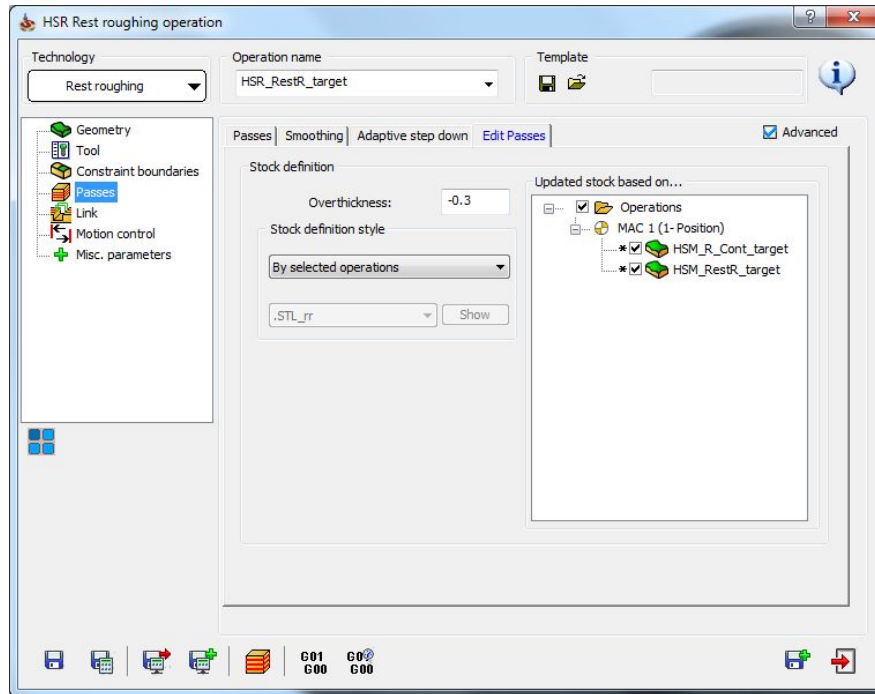
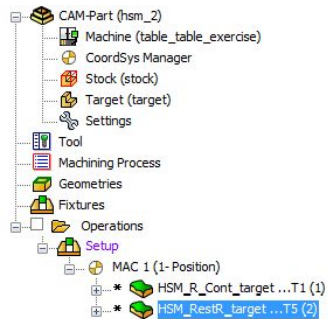
HSR/HSM

HSR/HSM: Major Speeding up of updated stock calculation



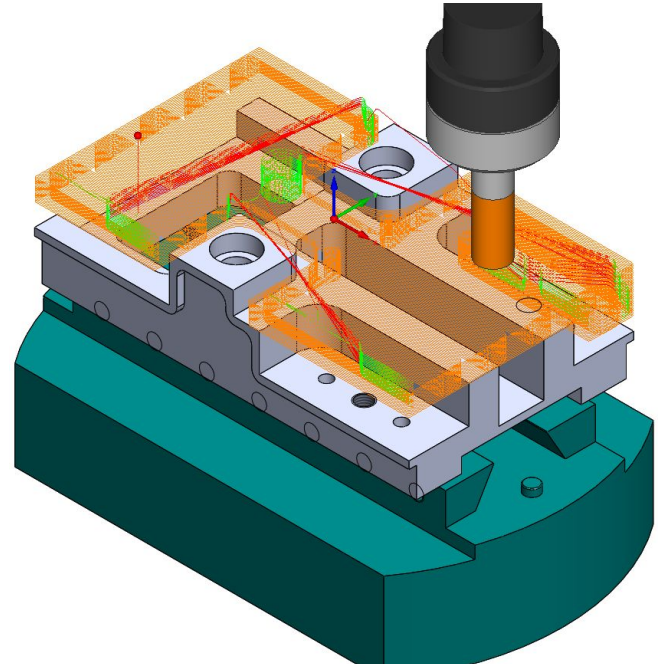
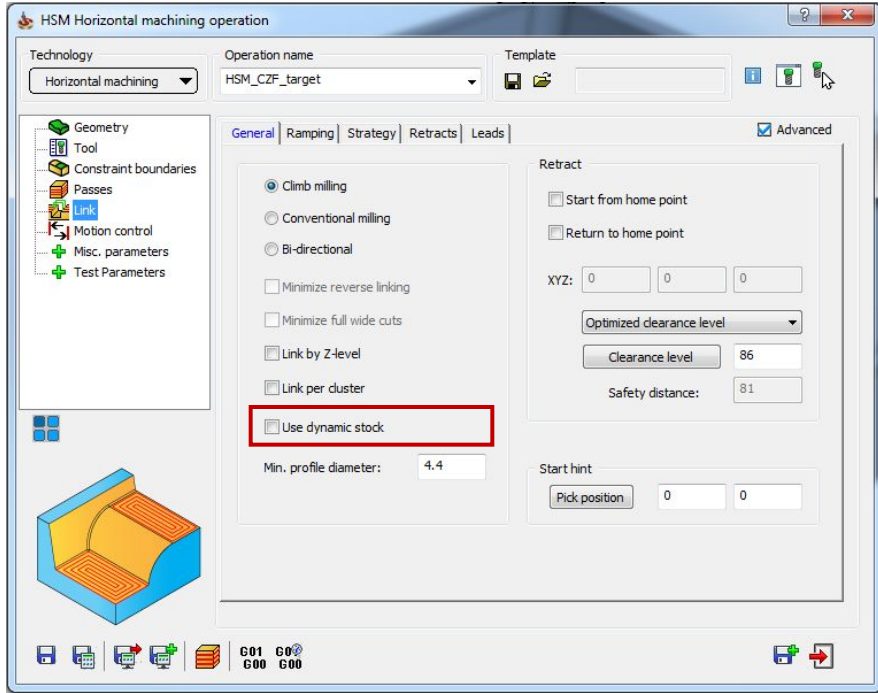
- Choose in Settings option of speeding up Updated stock calculation, if all previous operations are HSR/HSM
- Saving on average 40% in updated stock calculation time

HSR/HSM: Selection of non-calculated operations for updated stock



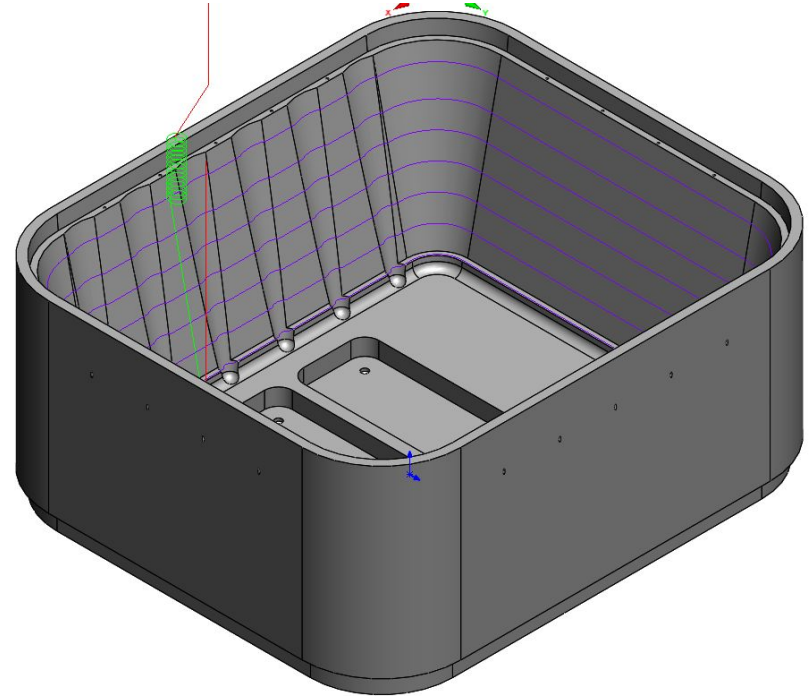
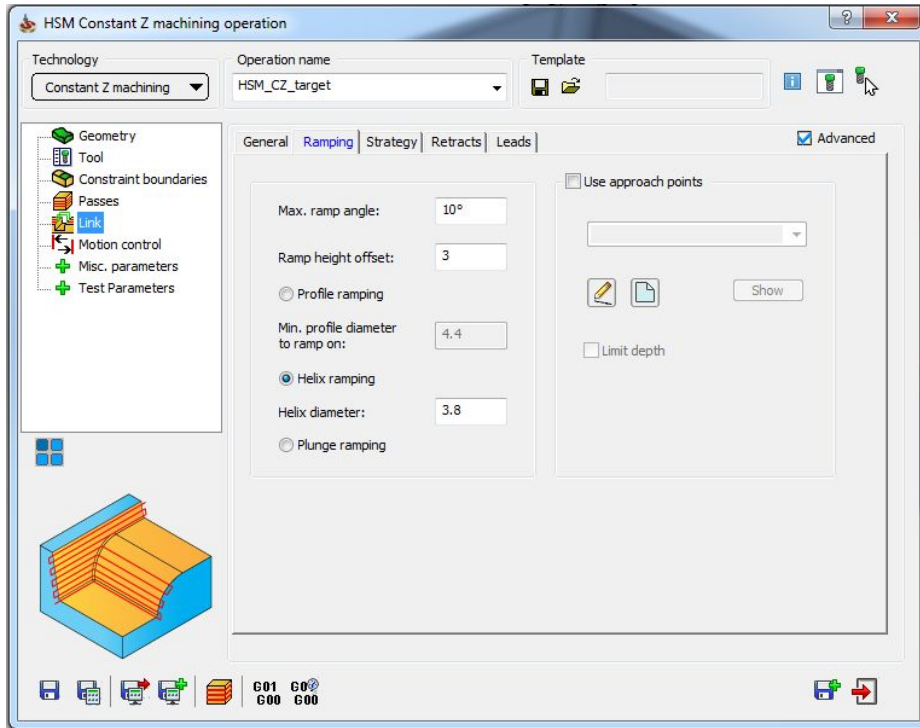
- Even if previous operations are not calculated , it's possible to select them for further updated stock calculation in HSR Rest Roughing

HSR/HSM: Use updated Stock for linking



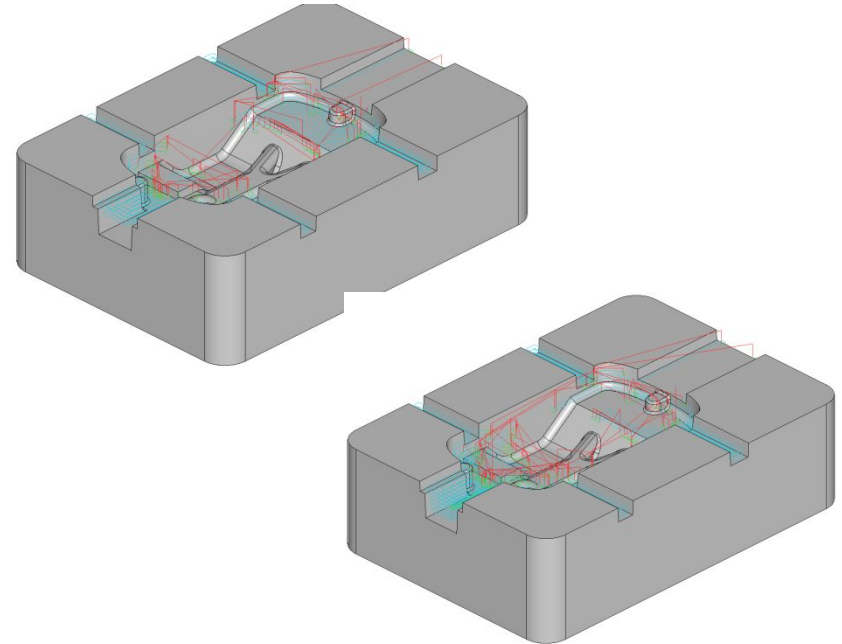
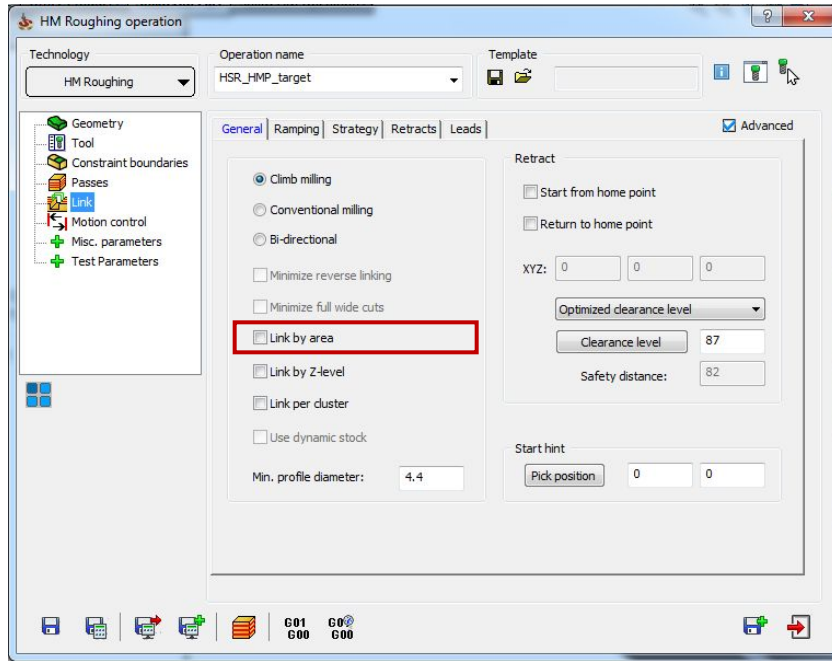
- Links the passes using dynamic stock(updated stock), instead of static stock (the initial stock), resulting in a very efficient toolpath

HSM: Ramping options added to Constant-Z machining



- Ramping options, added to HSM Constant Z machining, similar to HSR operations, are useful to increase tool life, when finishing is done immediately after roughing

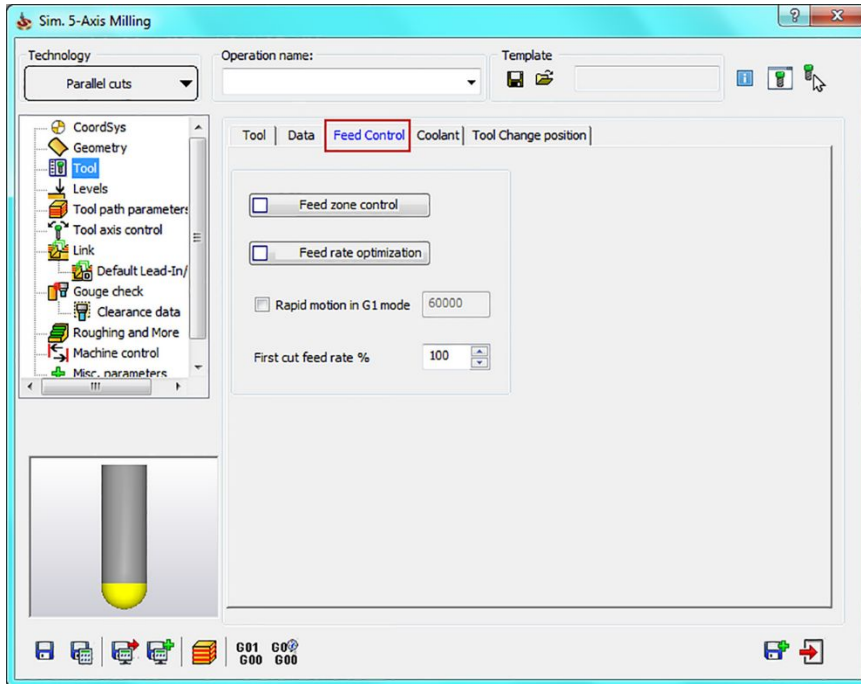
HSR/HSM: Link by area added to HM Roughing



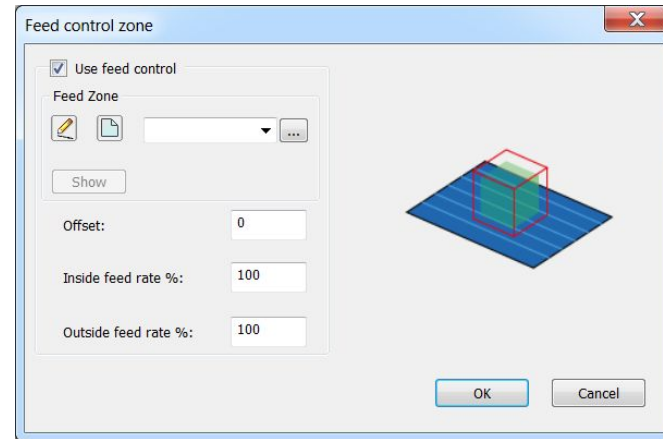
- Pockets are machined independently, but within a pocket, strict Z level ordering is enforced

Sim 5X Milling & HSS

Feed Control

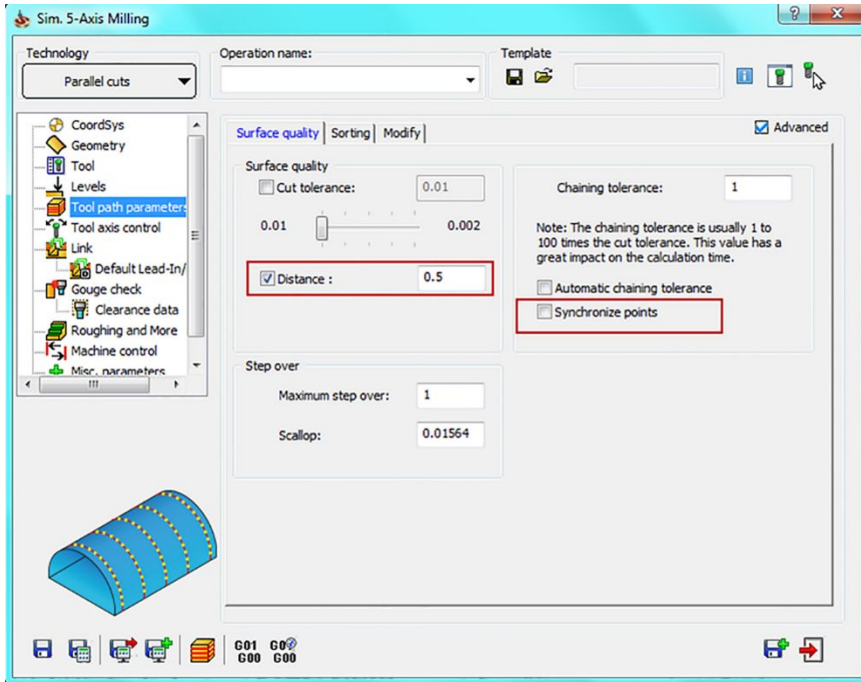


- New Feed Control tab added.
- Feed Control enables the user to reduce/increase cutting feed inside & outside defined volumes.

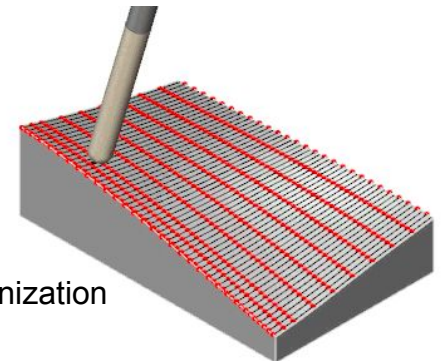
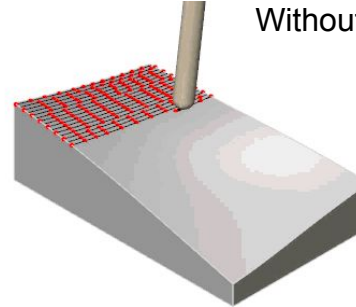


Toolpath parameters: Synchronize points

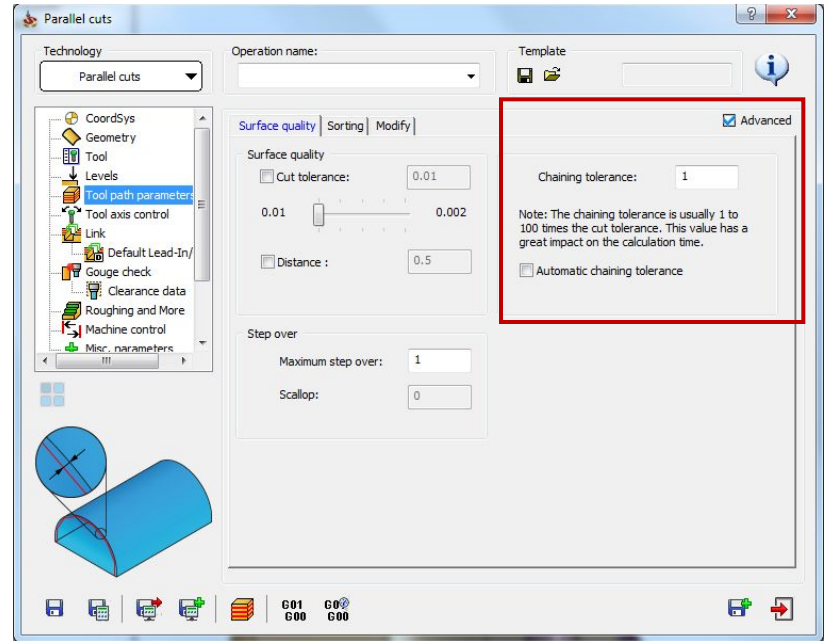
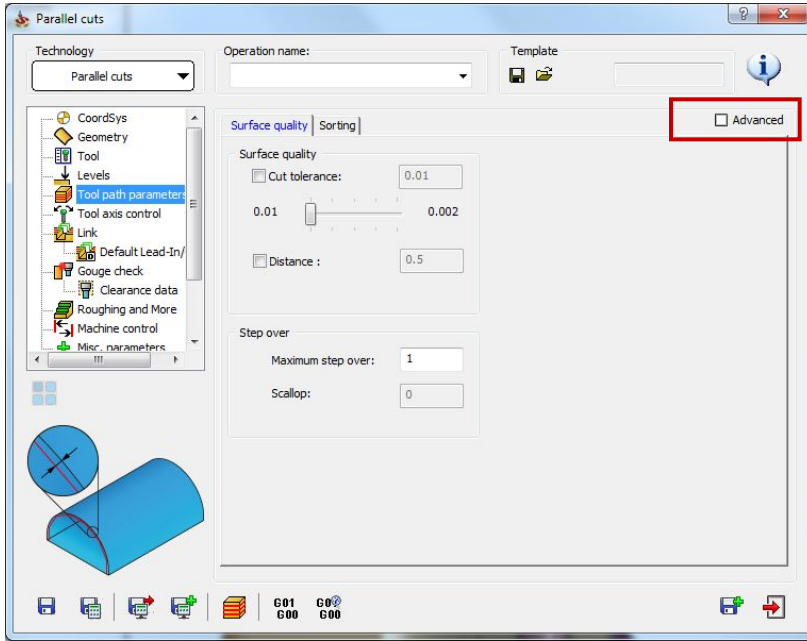
- Surface quality tab: New option of Synchronize points added.
- Option is available when the Distance check box is ON.



- The Synchronize points option enables you to equalize the spacing and number of points on all contours, thus getting better surface quality



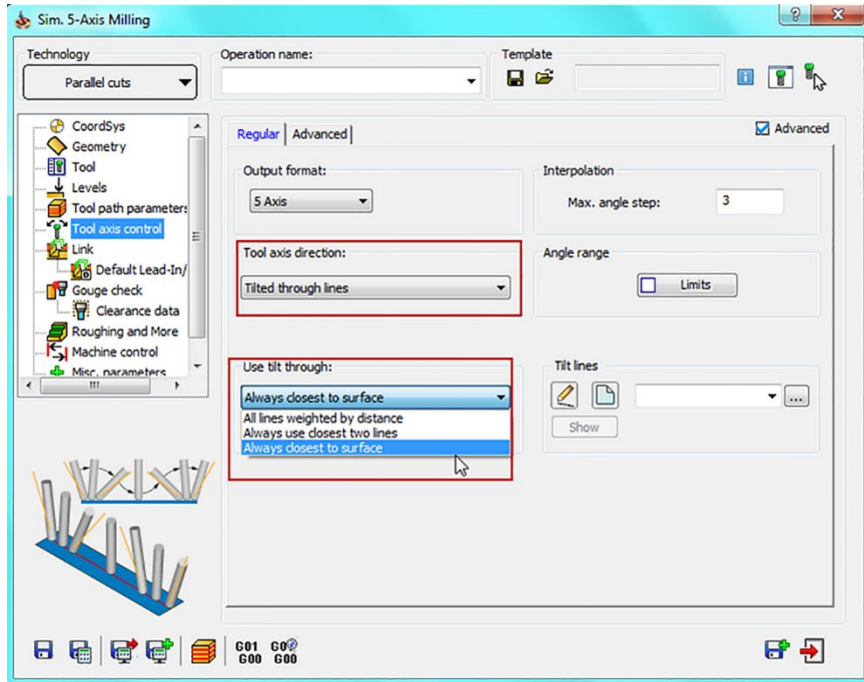
Sim 5x: Advanced mode button



• Only most used options are open in the standard interface, making it easier for customers

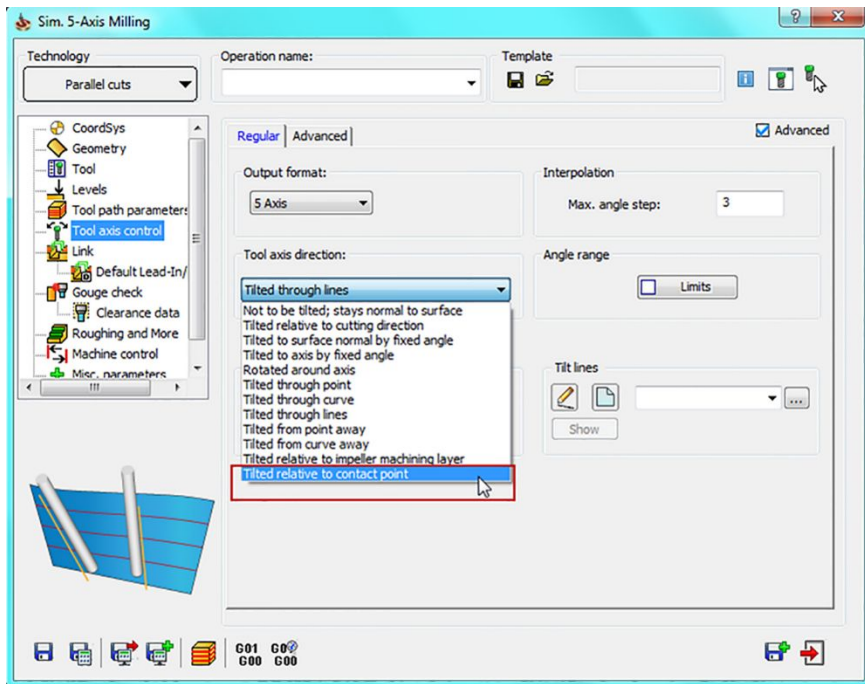
• Advanced button opens additional control options, needed by advanced users

Sim5X - Tool axis control: Always closest to surface



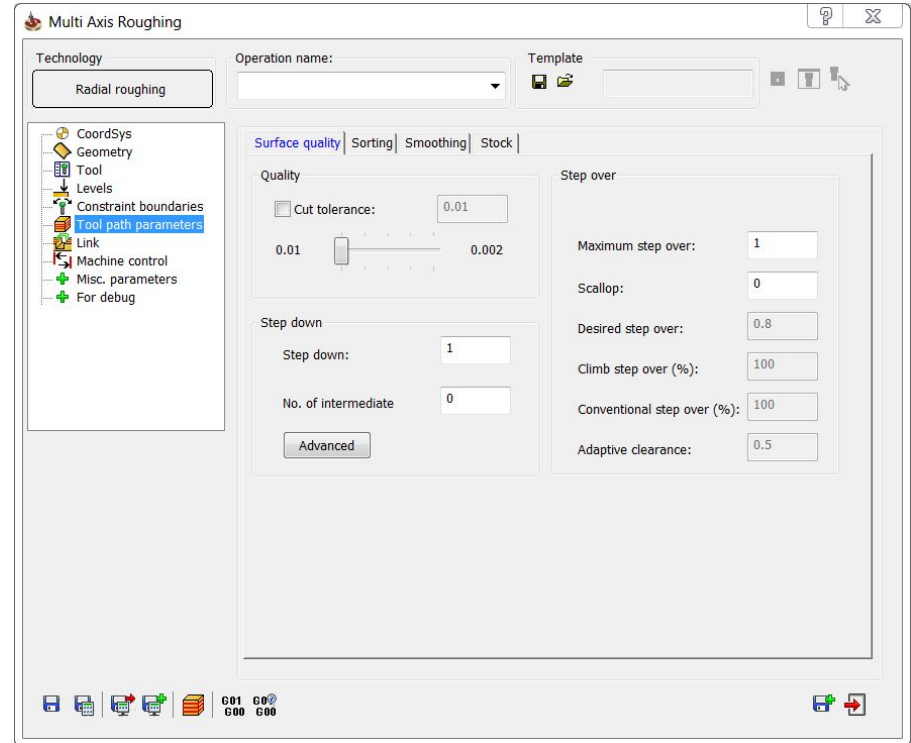
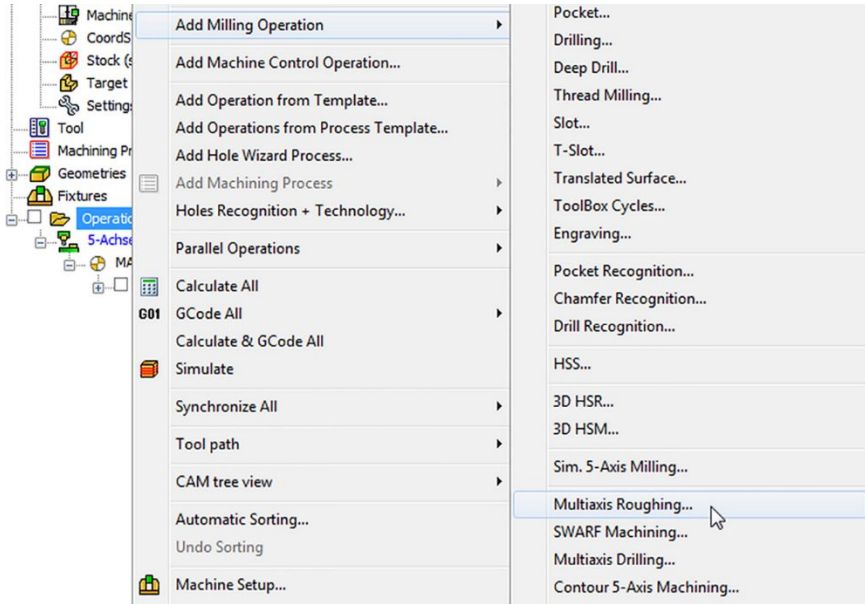
- ❑ In Tool axis direction/Tilted through lines: New option of Always closest to surface is added in the *Use tilt through*.
- ❑ This option enables you to tilt the tool as defined in the Tilt lines section, maintaining always the tool at closest distance to the surface, avoiding sudden tilting
- ❑ This option maintains the tilt by using the tilt lines that are at the closest distance to the surface.

Sim5X - Tool axis control: Tilted relative to contact point



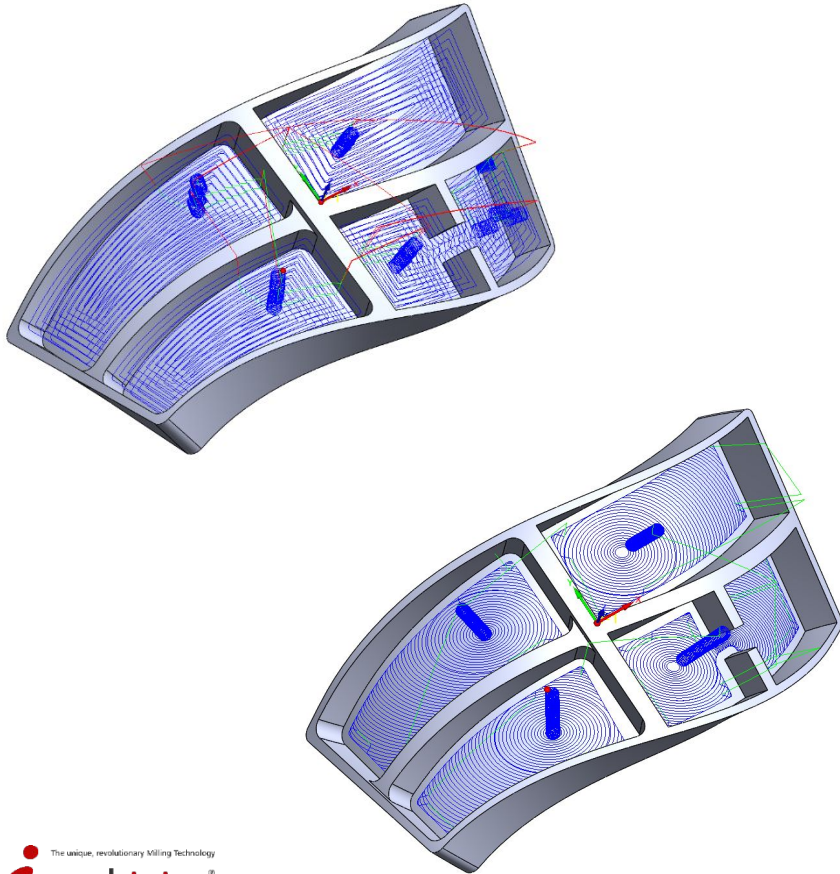
- ❑ In Tool axis direction: New option of Tilted relative to contact point is added.
- ❑ This option is similar to “Tilted Relative to Cutting Direction”, however in this option instead of cutting direction, the tool tilting will be relative to the contact point of the tool with the surface.

New operation: Multiaxis Roughing



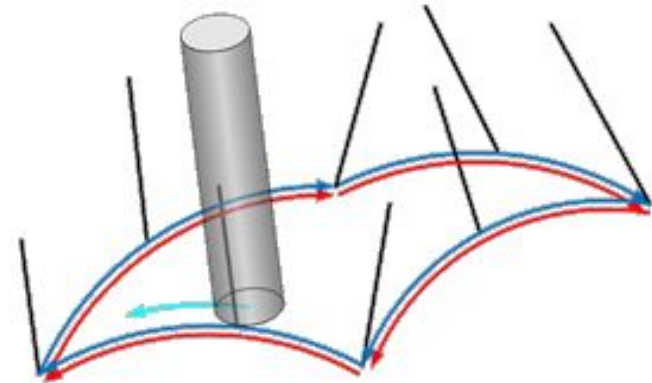
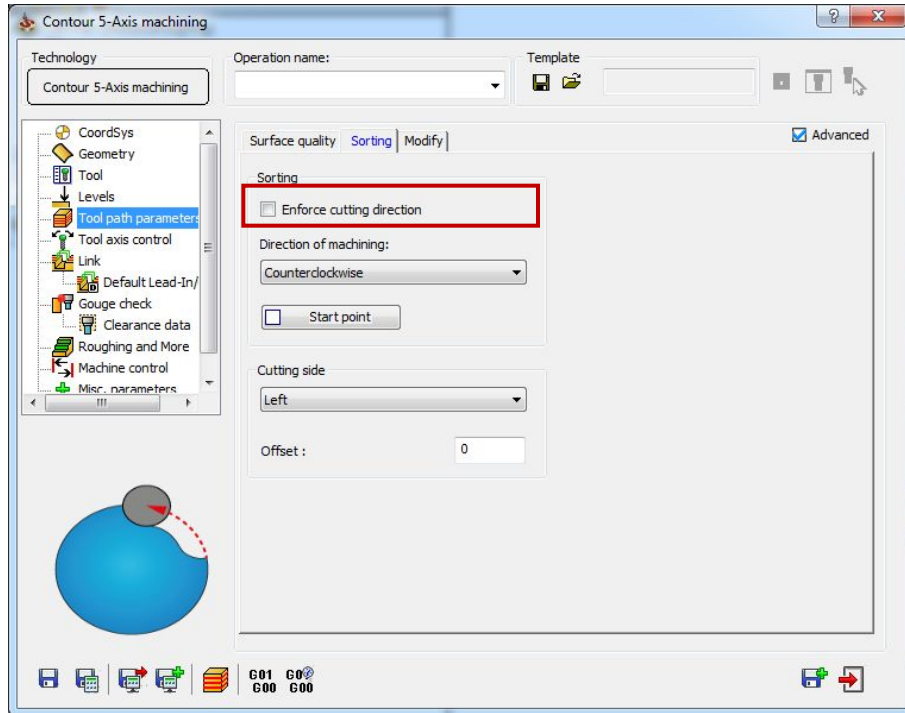
□ New Multiaxis Roughing operation is added for Roughing of parts, that need multiple setup if done in 3X.

New operation: MultiAxis Roughing



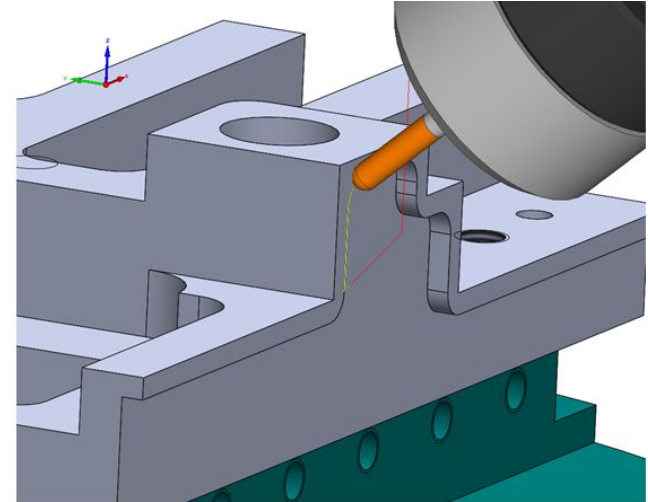
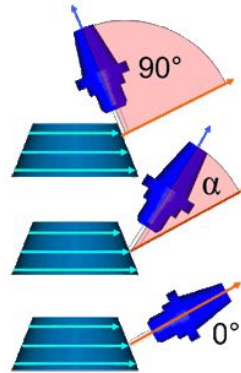
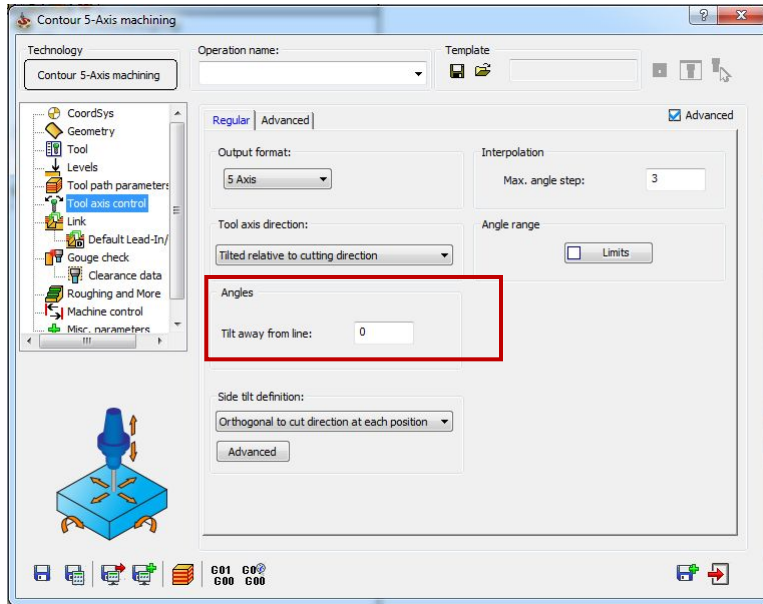
- ❑ This operation creates a multiaxis tool path that can be used to rough out pocket shaped geometries in full 5 Axis.
- ❑ The user has to specify the floor, wall and ceiling surfaces and the system automatically creates the roughing tool path.
- ❑ Adaptive roughing feature also available.

Contour 5x machining: Enforce cutting direction



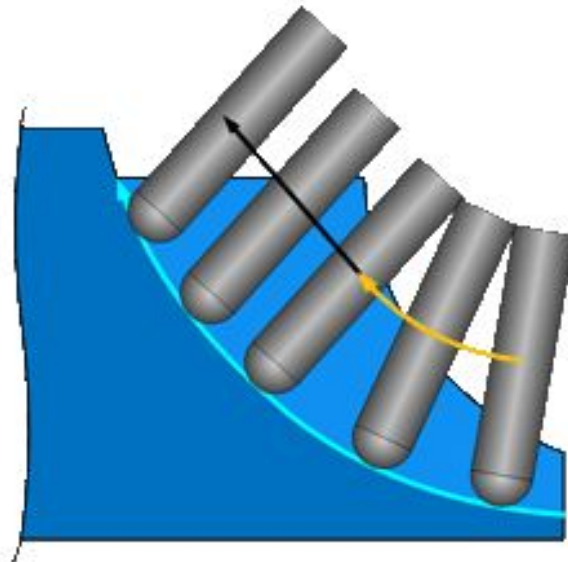
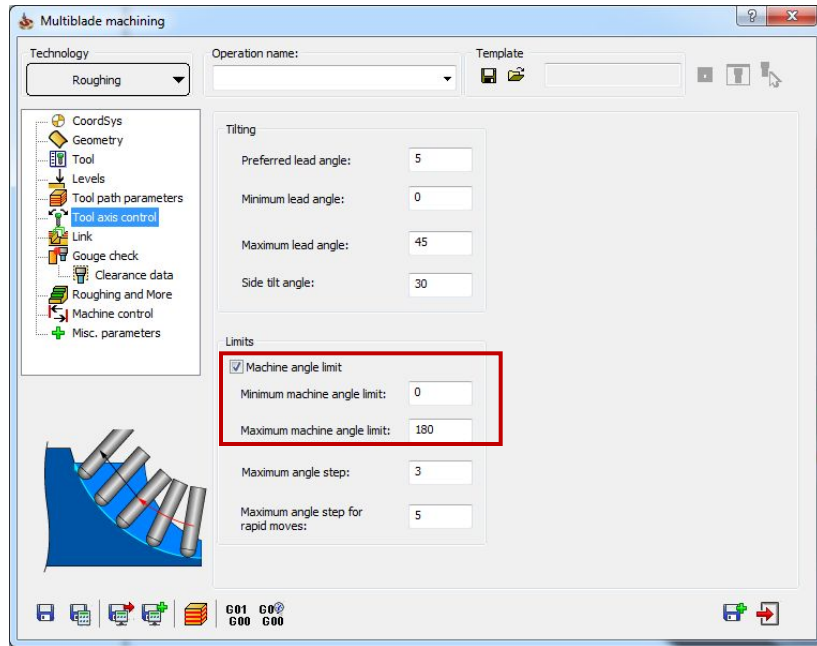
- **Direction is set according to direction of machining, ignoring the selected chain direction**

Contour 5x Machining: Tilt away from line



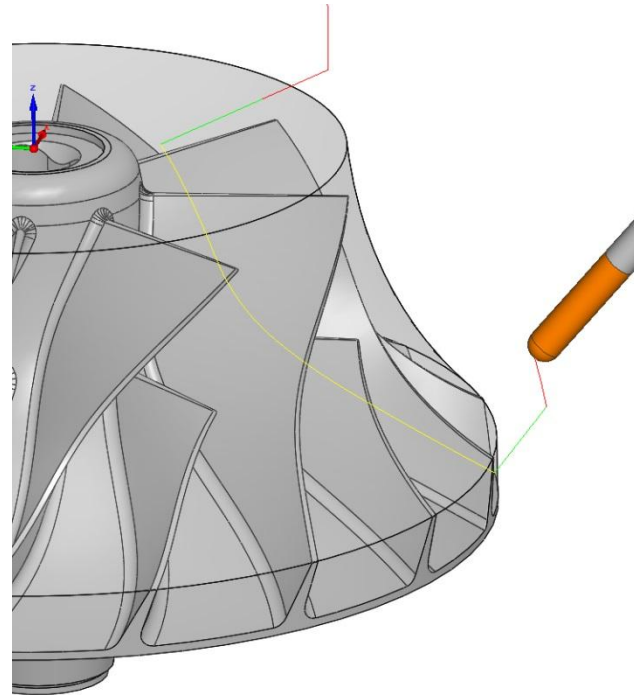
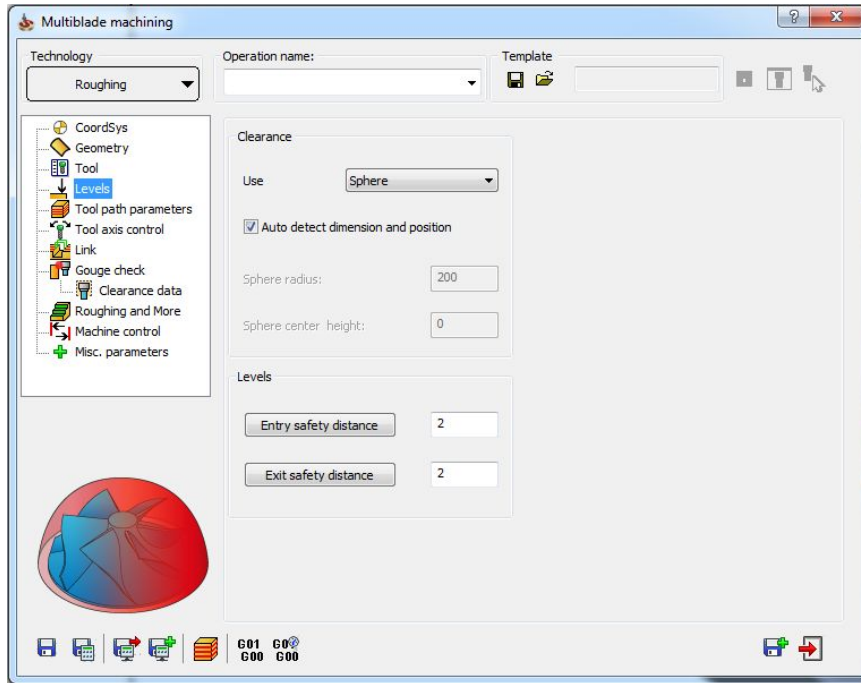
- Tilt away from line defines the side tilt of the tool relative to the contour

Multiblade Machining: Machine angle limits – Min & Max



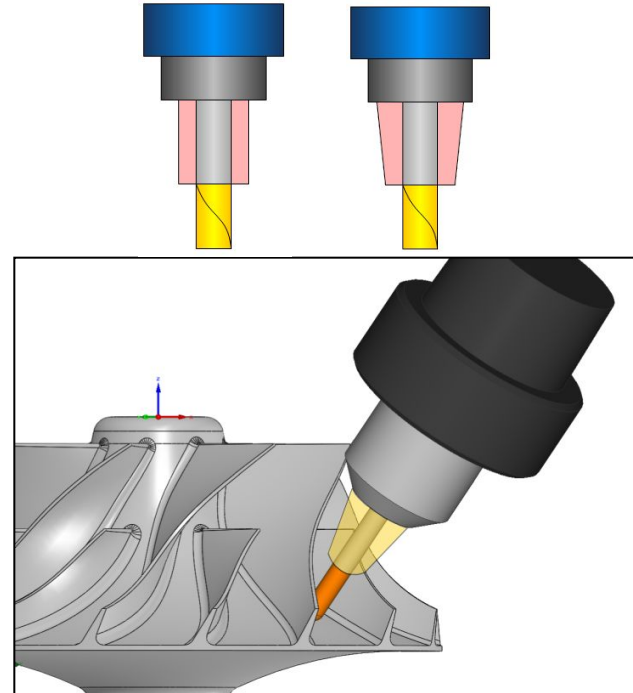
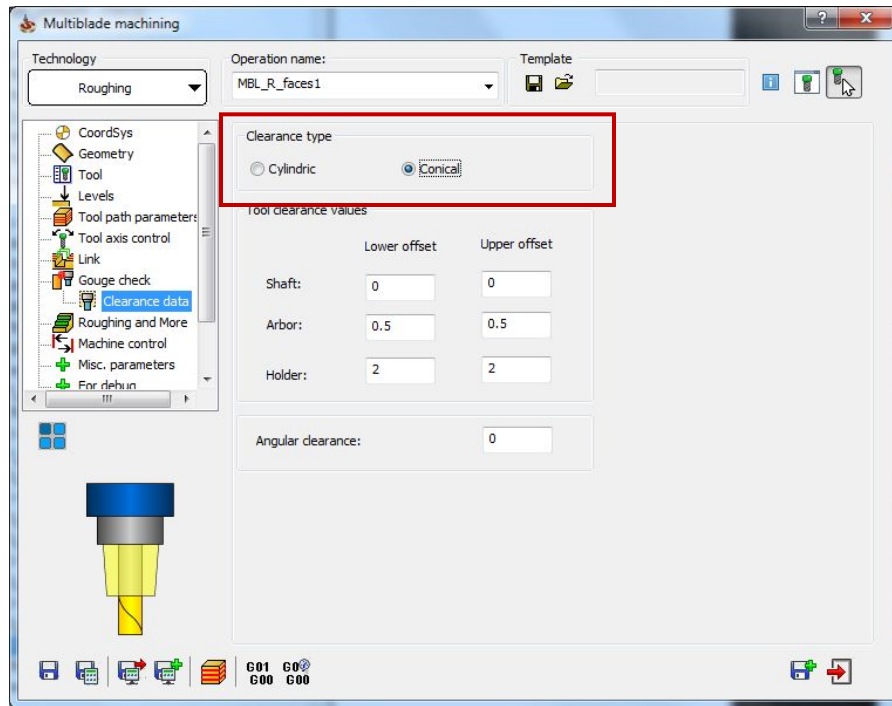
- Machine angle limit is now controlled by two parameters: minimum and maximum angle, rather than by only one limit angle in older versions, giving much better control of angles in CNC machines that have less swivel

Multiblade Machining: Entry & Exit Safety distance



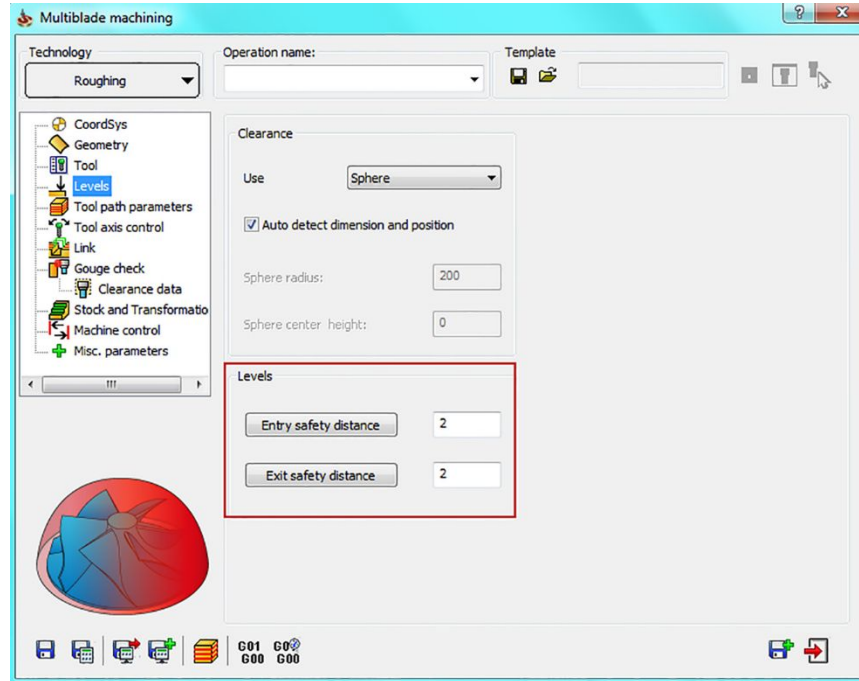
- **Safety distance is divided into two fields: Entry & Exit safety distance, providing better control**

Multiblade Machining: Additional Clearance type - Conical



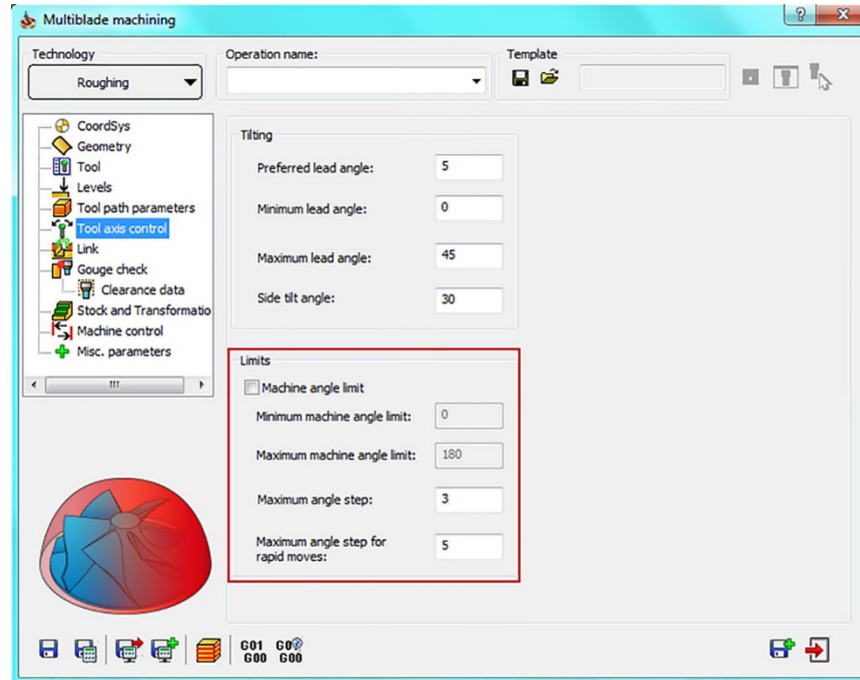
- A new Tool Clearance type, Conical, is added to enable cutting more material and working deeper, while avoiding gouging

Multiblade Machining: Levels section



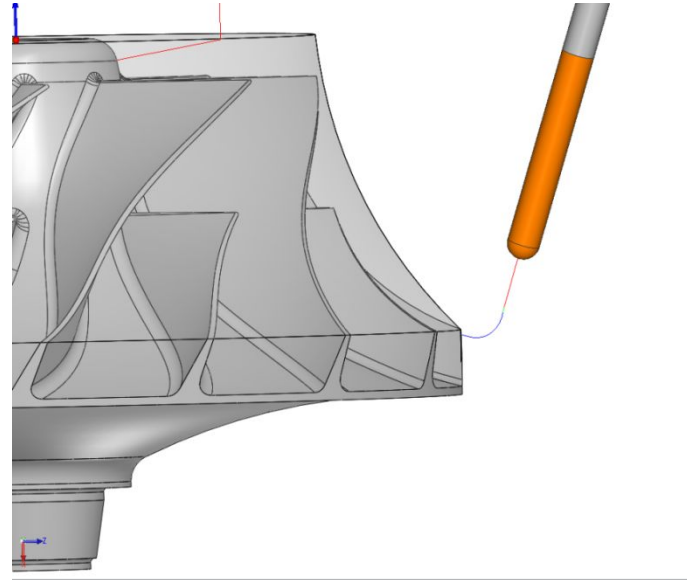
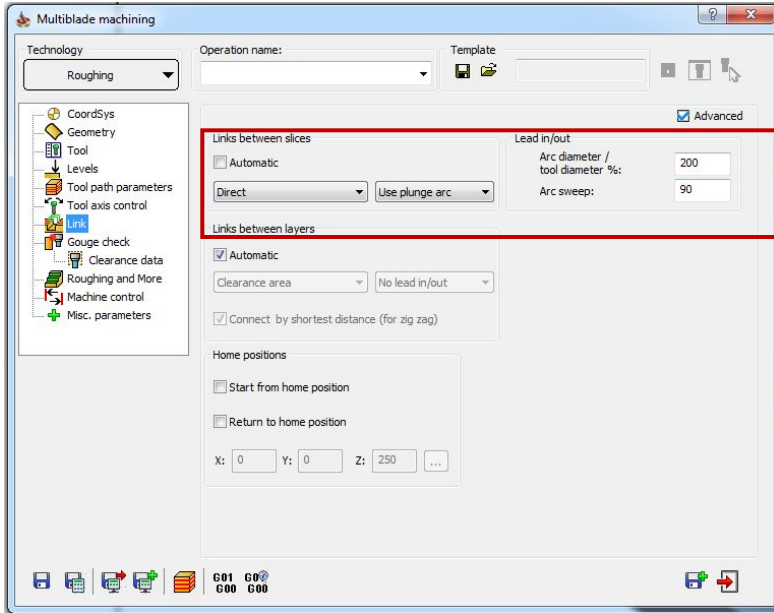
□ Levels page: New Levels section is added to enable entry and exit safety distance.

Multiblade Machining: Tool Axis Control Limits



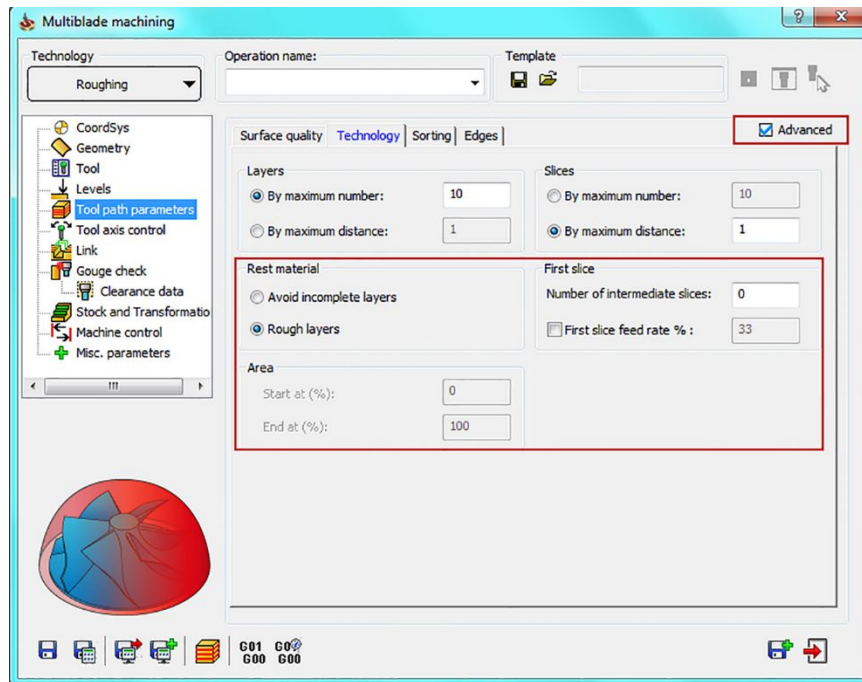
□ Tool axis control page: Limits section has enhancements.

Multiblade Machining: Plunge arc for link between slices or layers



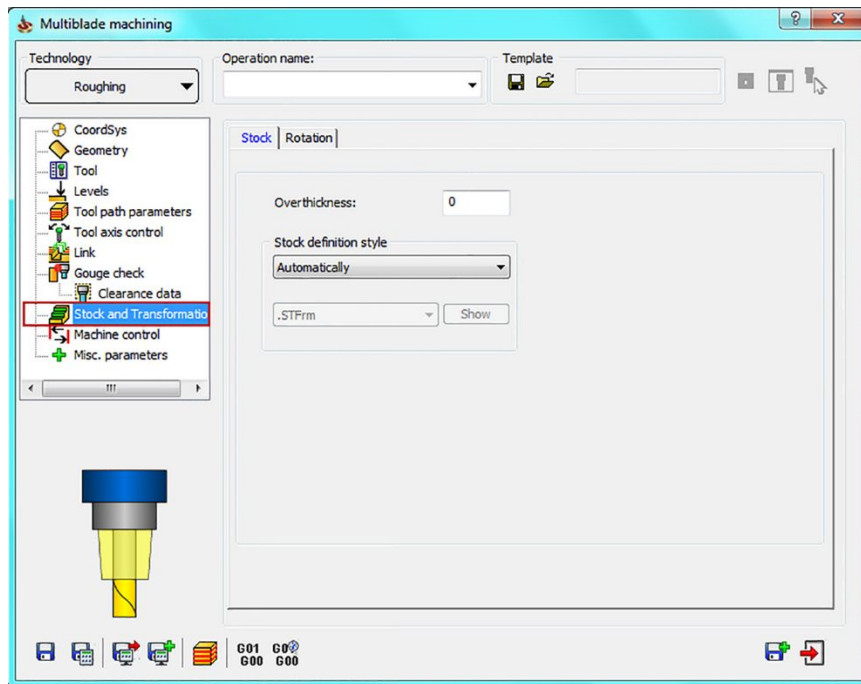
- **Option to use a plunge arc, while performing link between slices or layers, in order to provide gradual entry into the material**
- **Enables you to specify the diameter of the approach & retreat arc, using the ratio of the Arc diameter to the Tool diameter**

Multiblade Machining: GUI Improvements



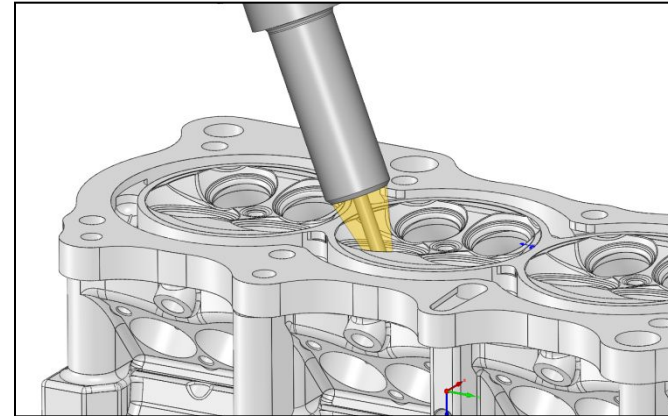
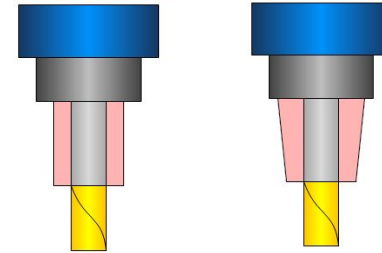
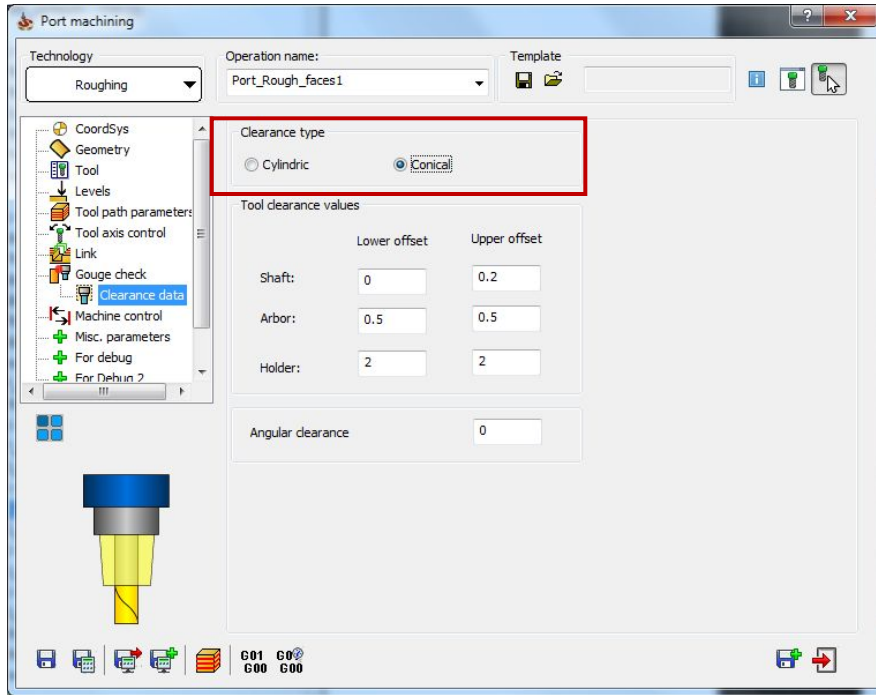
- ❑ Tool path parameters page/Technology/ Options of Rest material, First slice, and Area are available only when the Advanced check box is selected.

Multiblade Machining: GUI Improvements



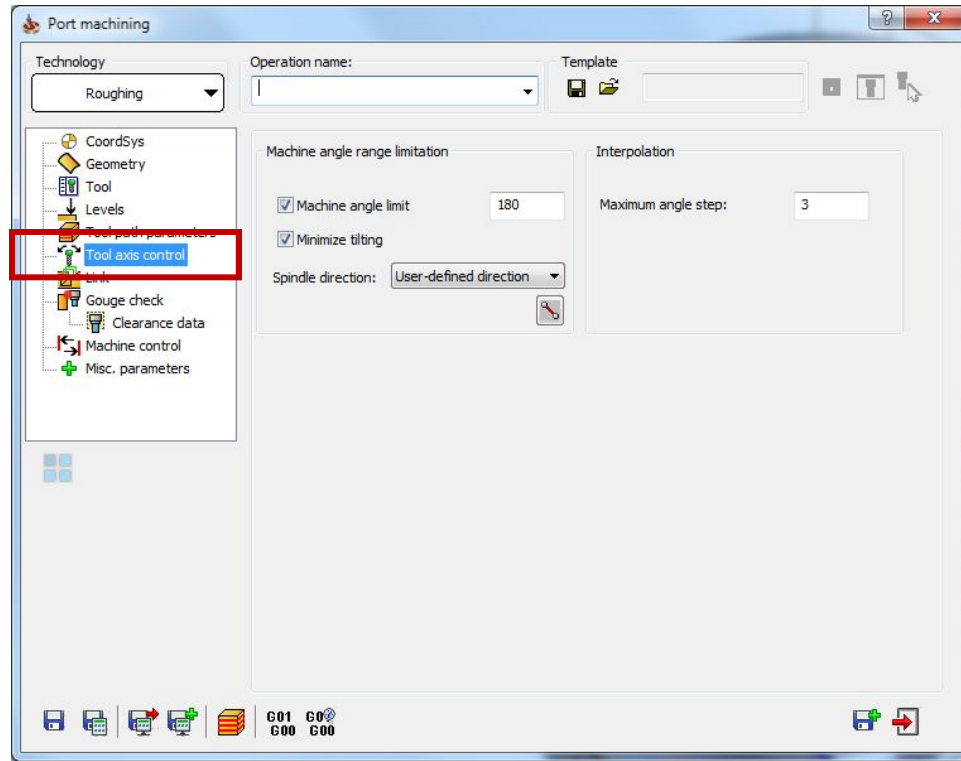
❑ Roughing and More is renamed as Stock and Transformation.

Port Machining: Additional Clearance type - Conical



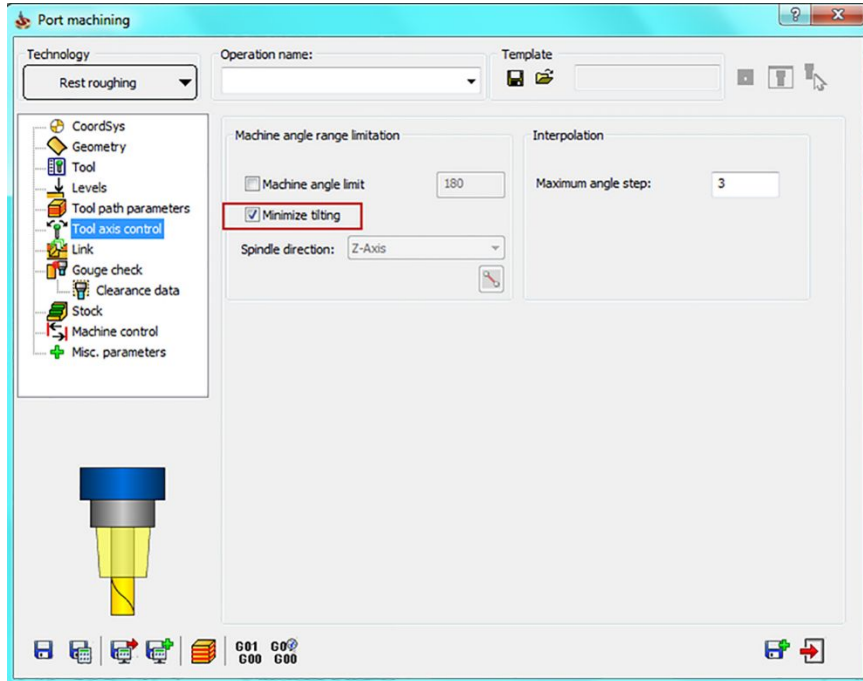
- A new Tool Clearance type, Conical, is added to enable cutting more material and working deeper, while avoiding gouging

Port Machining: Tool axis control page



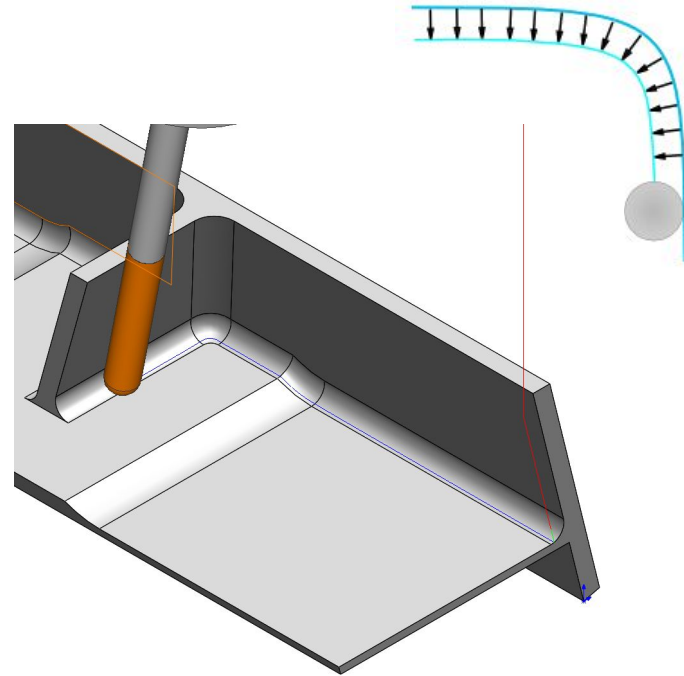
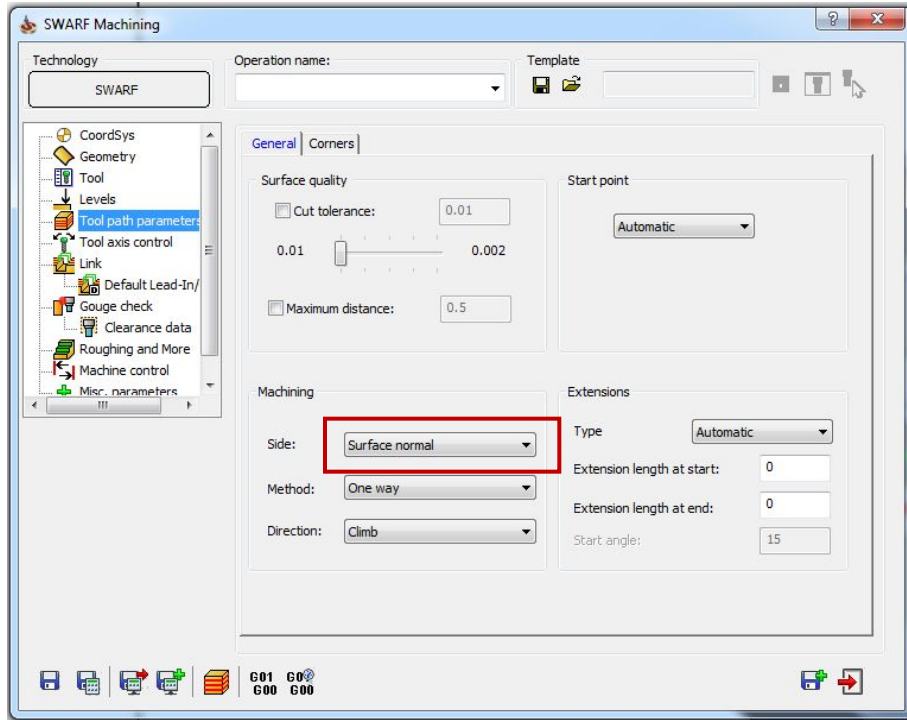
□ Tool axis control page added in order to provide smoother tool tilting during cutting

Port Machining: Tool Axis Control - Minimize Tilting



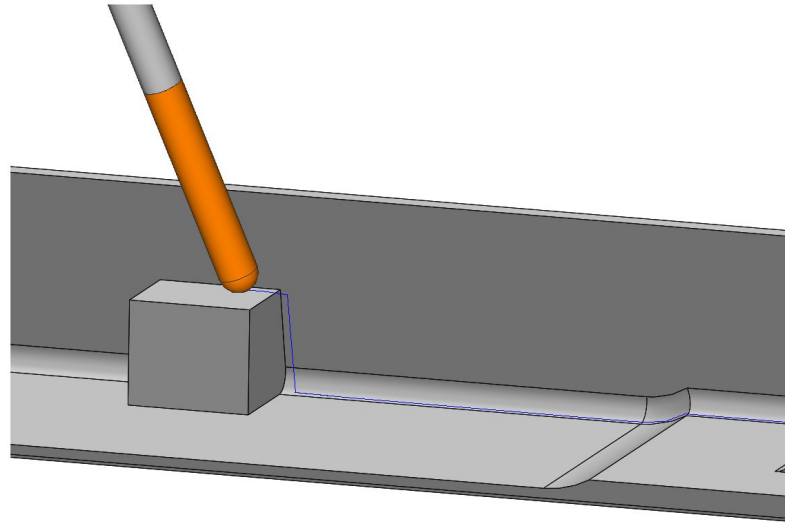
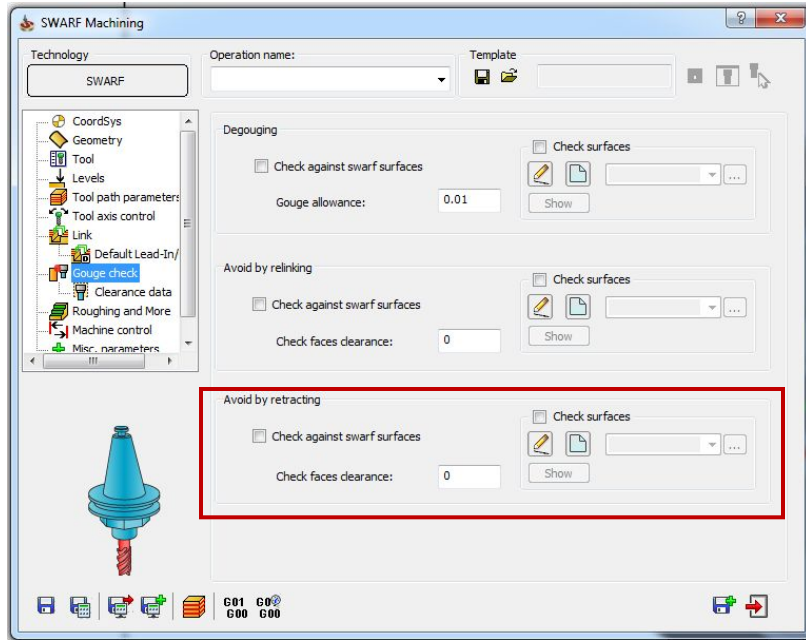
- Tool axis control: Minimize tilting is added.
- This option improves tool tilting by minimizing angle changes and keeping machine tilt motions to minimum.

SWARF Machining: Surface normal defines machining side



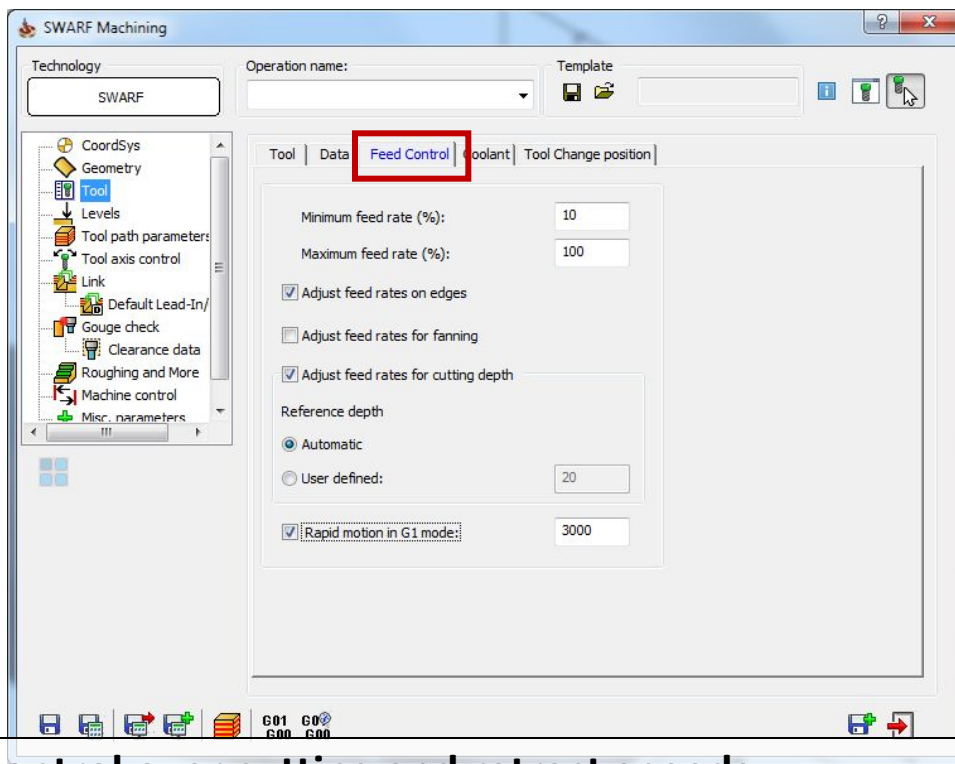
- Side of machining is determined according to the surface normal, simplifying the determination of the machining side when machining multiple surfaces

SWARF Machining: New Avoid gouge strategy and new Layout for gouge page



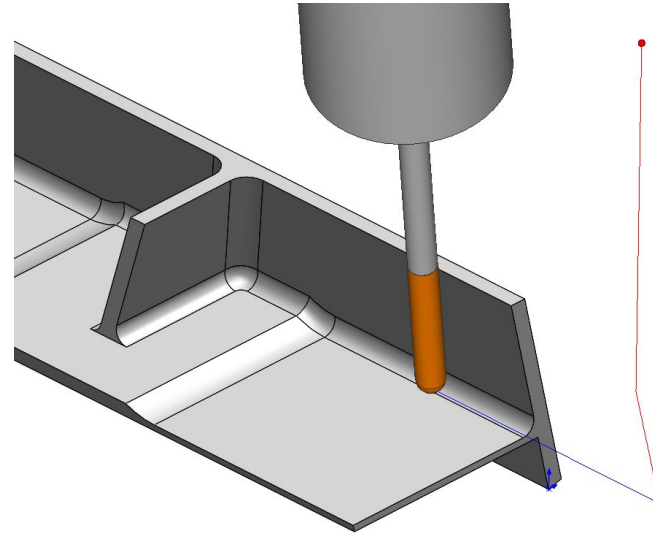
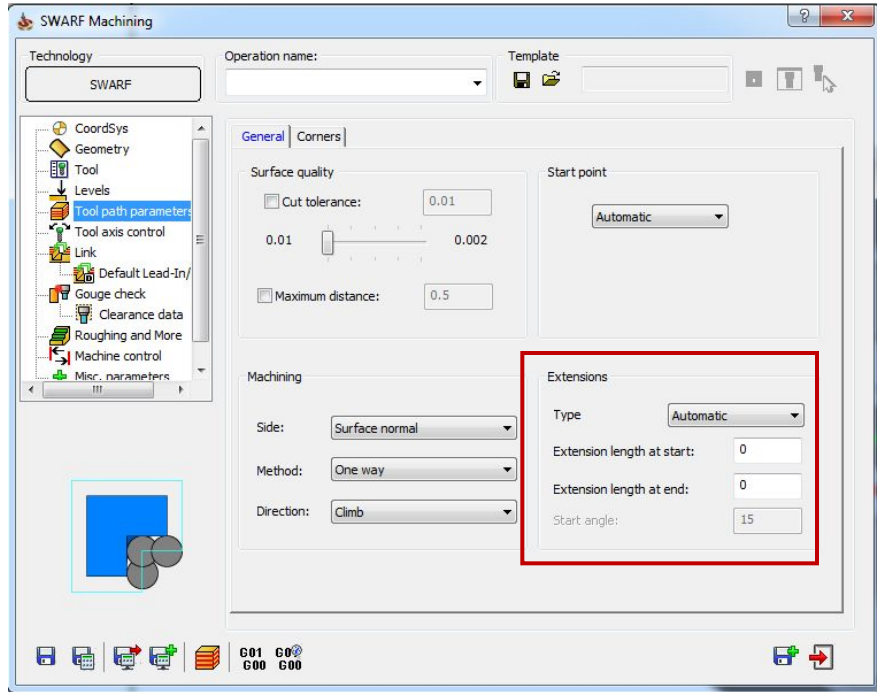
- **New Avoid Gouge strategy, Avoid by Retracting, enables the user to avoid obstacles by retracting the tool.**
- **New simplified layout for gouge page.**

SWARF Machining: Feed control



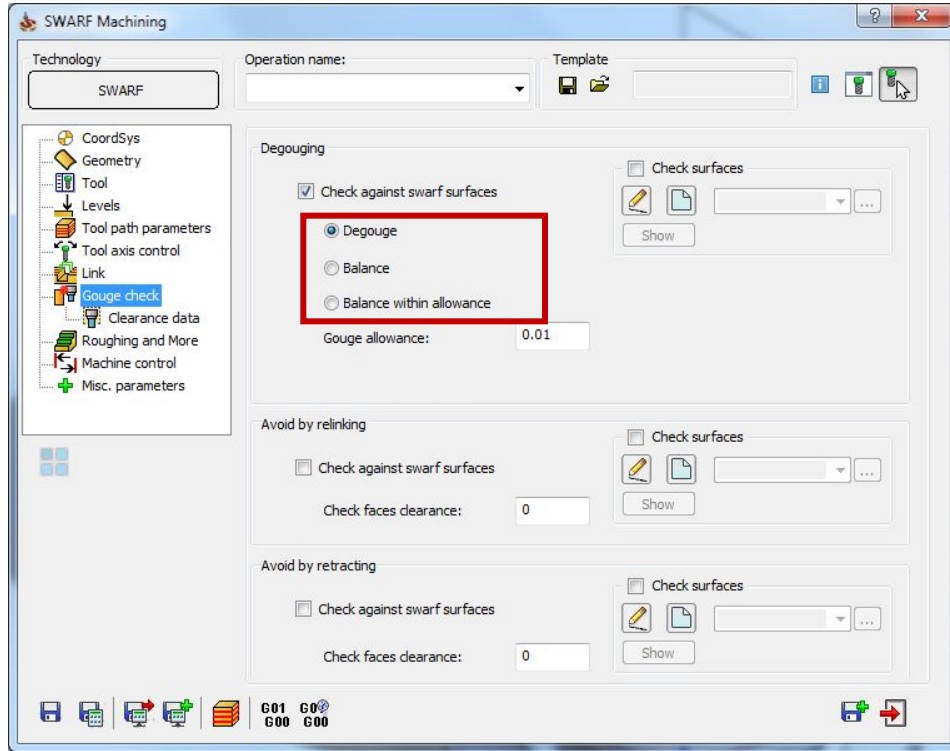
- More flexible control over cutting and retract speeds
- Possibility to replace Rapid movements by G1 moves

SWARF Machining: Adding extensions to toolpath

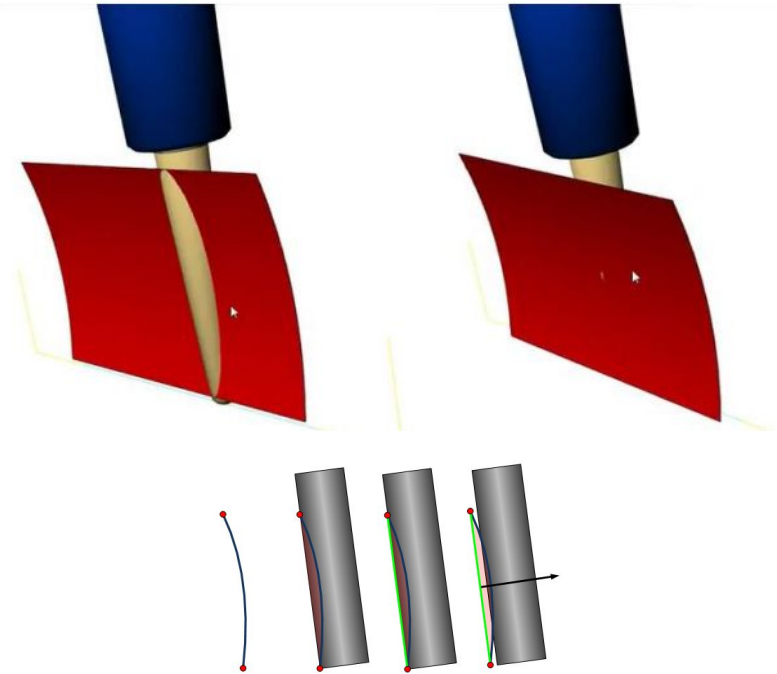


- **New option to add extensions to the tool path, to avoid direct entry into the material, thereby increasing tool life**

SWARF Machining: Degouging strategy



Degouge option



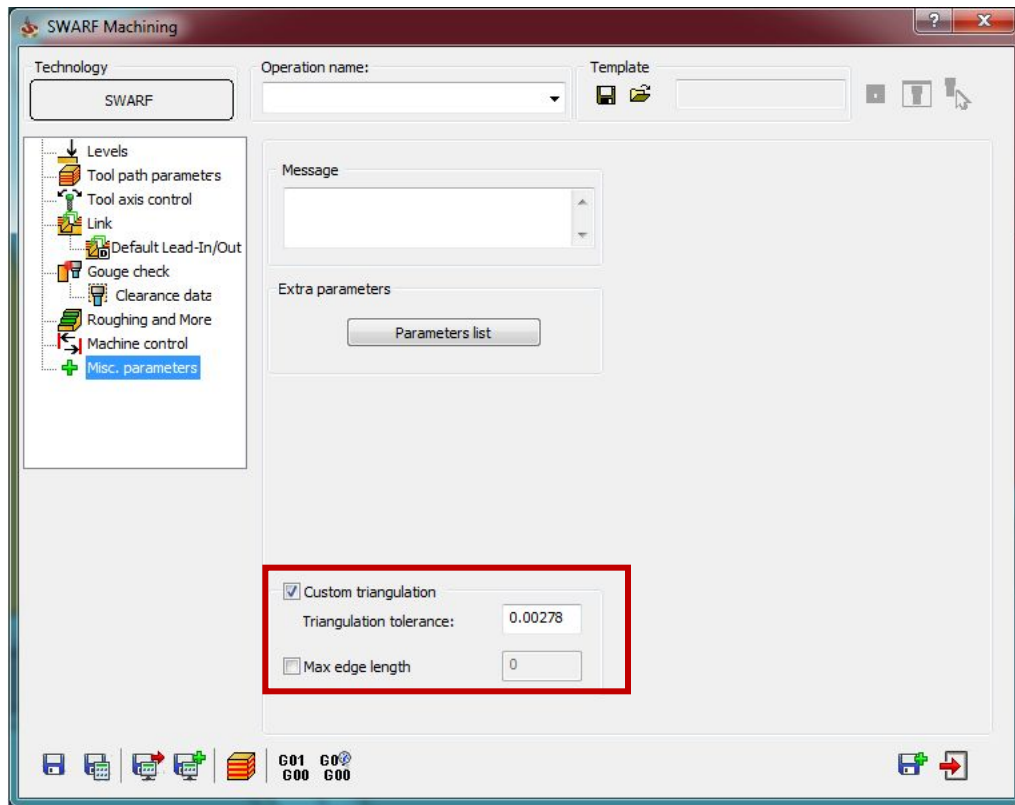
- New options to avoid gouges with drive surface

SWARF Machining: Rotate & Translate

The screenshot displays the SWARF Machining software interface. The main window shows the 'SWARF Machining' title bar and a sidebar with a tree view containing 'CoordSys', 'Geometry', 'Tool', 'Levels', 'Tool path parameters', 'Tool axis control', 'Link', 'Gouge check', 'Clearance data', 'Roughing and More', 'Machine control', and 'Misc. parameters'. The 'Roughing and More' option is selected. The main area contains several configuration panels: 'Operation name', 'Template', 'Pattern slices' (with 'Depth steps' set to 'By number of slices', 'Distance' set to 1, and 'Number' set to 1), 'Pattern layers' (with 'Number of layers' set to 1 and 'Layer distance' set to 1), 'Sorting' (with 'Sequence' set to 'By layer'), 'Links between passes' (with 'Small moves' set to 'Direct' and 'Large moves' set to 'Clearance area'), and 'Tool shift' (set to 'Constant for each slice'). A red box highlights the 'Rotate & Translate' checkbox, which is checked. The 'Rotate / Translate Tool Path' dialog box is open, showing the following settings: 'Use Rotation / Translation' is checked, 'Orientation' is 'Z-Axis', 'Rotary axis base point' is 'Select point', 'Number of steps' is 4, 'Rotate' section has 'Start angle' set to 0 and 'Rotation angle' set to 90, and 'Translate' section has 'Start distance' set to 0 and 'Step over distance' set to 0. The dialog box also features 'OK' and 'Cancel' buttons. To the right of the dialog box, there is a 3D model of a blue tool cutting a grey part, with a yellow line indicating the tool path.

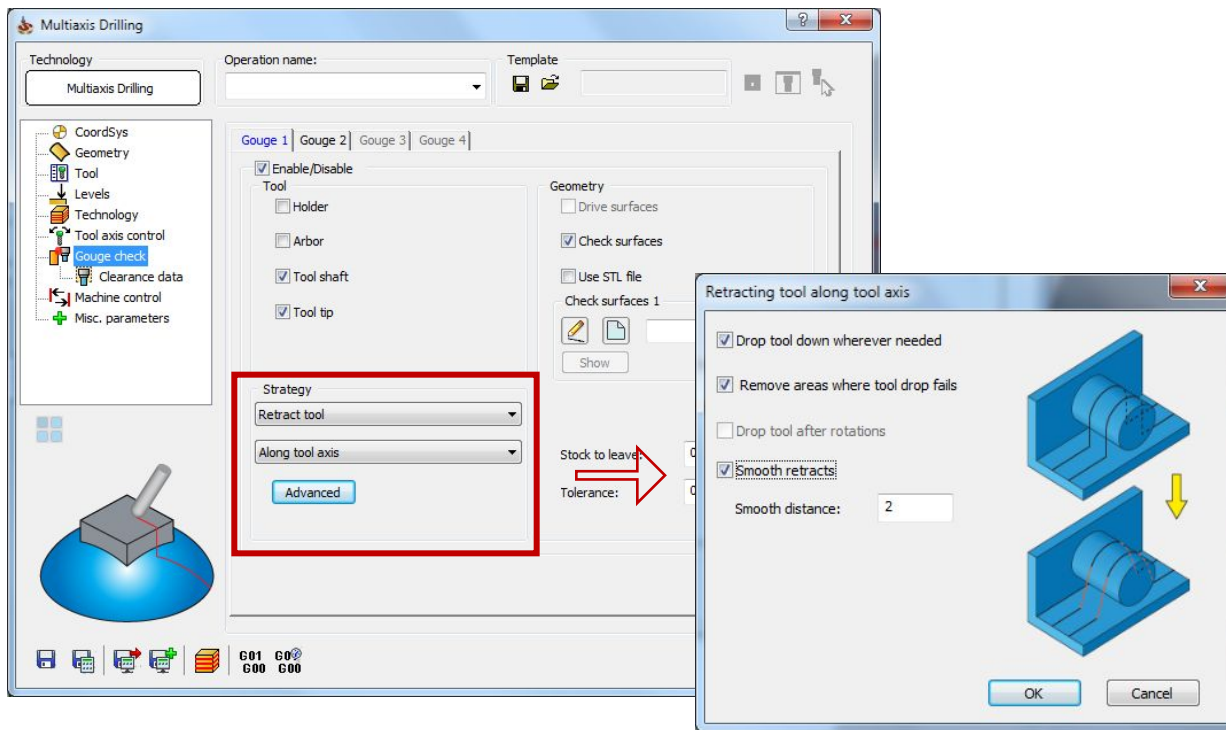
- Rotate & Translate option is available for SWARF operation also

SWARF/MultiBlade/Port/MultiAxis Roughing: Custom triangulation



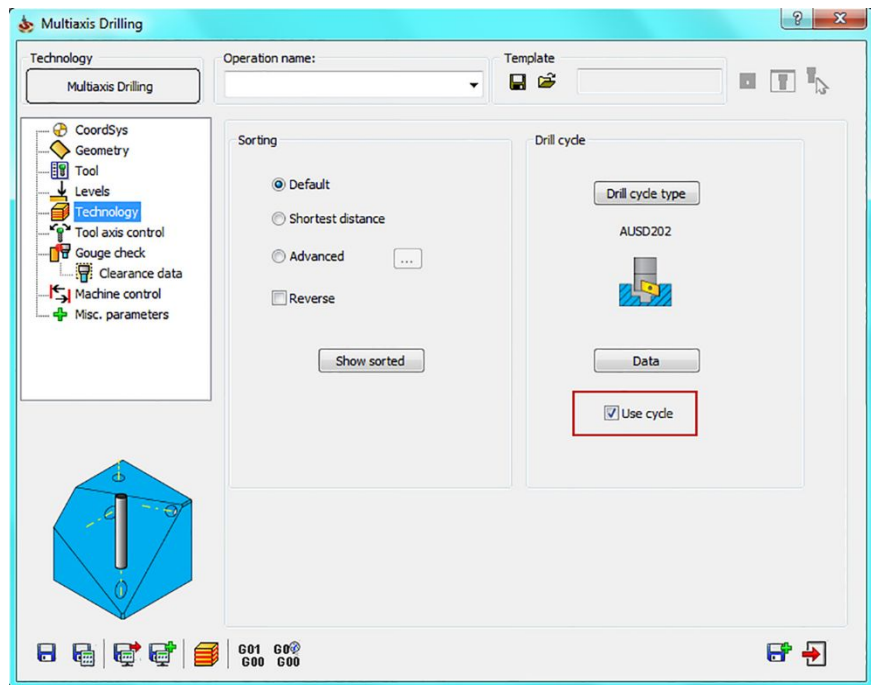
- ❑ Provide more control over surface triangulation
- ❑ Custom triangulation check box = OFF, SolidCAM uses the native CAD triangulation method.
- ❑ Custom triangulation check box = ON, 5-Axis triangulation method is used to define the Triangulation tolerance and Max. edge length.

Multiaxis Drilling: retract tool along tool axis



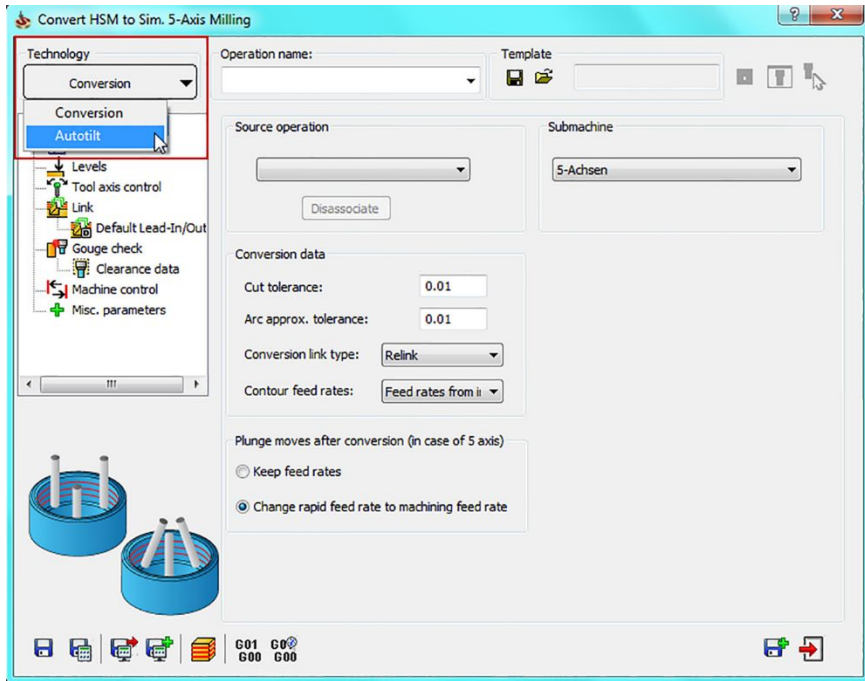
- Avoid collisions by preventing sudden tool „jumps“

Multiaxis Drilling: Use Cycle option



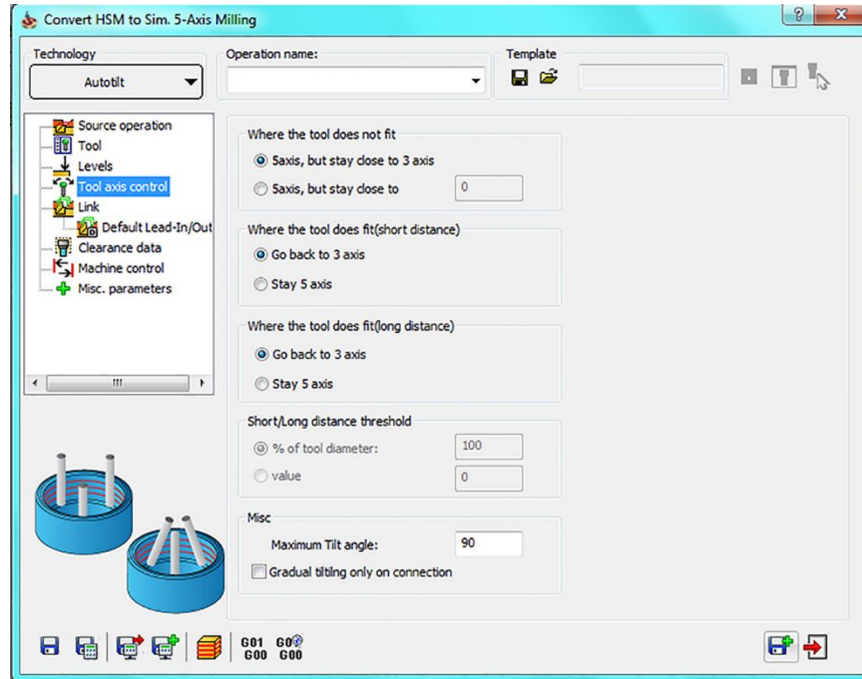
- ❑ The Multiaxis Drilling Technology page has a new option: Use cycle.
- ❑ When this check box is ON, the generated G-code uses canned drill cycles, if CNC machine enables it.
- ❑ If this check box is OFF, the output is in the form of linear movements.

Convert HSM to Sim 5-Axis Milling: Autotilt



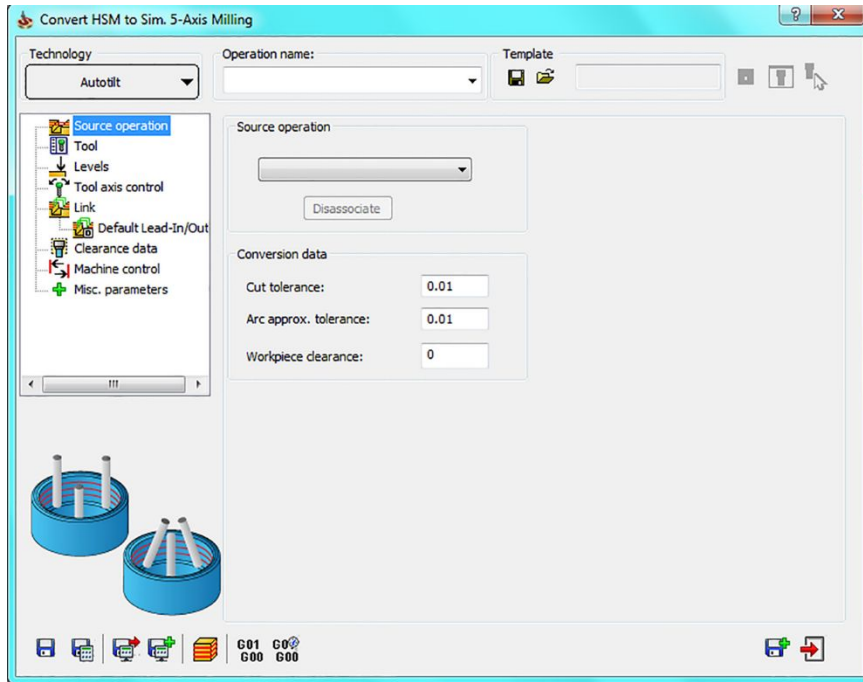
- ❑ New Technology of Autotilt is added.
- ❑ This option enables the conversion of a 3-axis input tool path into a full automatic collision-checked 5-axis tool path.
- ❑ The main aim is to take the 3-axis tool path and use it with a much shorter tool.
- ❑ The automatic tilting now does compensate the holder with the geometry and tilts it away.

Convert HSM to Sim 5-Axis Milling: Autotilt



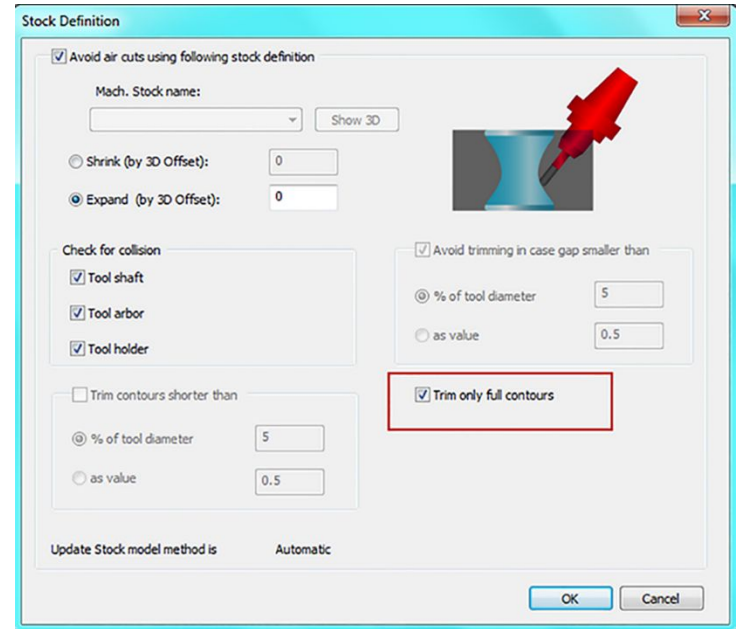
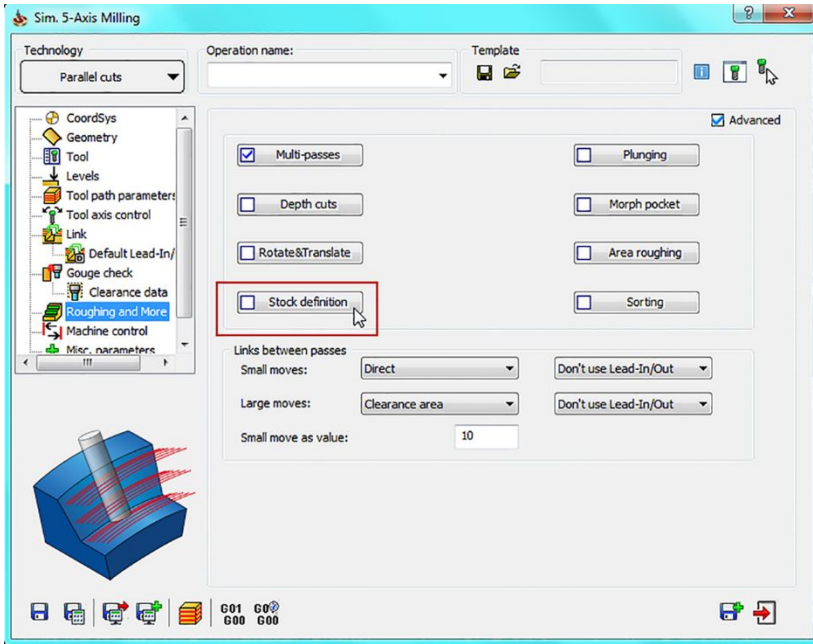
□ Tool axis control page has new options when Autotilt is selected as Technology.

Convert HSM to Sim 5-Axis Milling: Autotilt - Workpiece Clearance



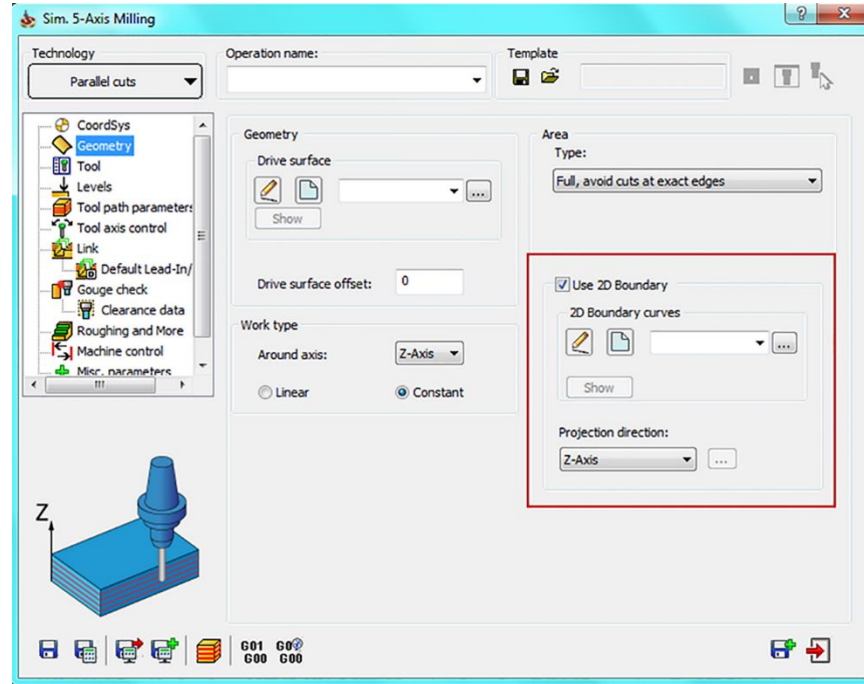
- ❑ Source operation page has the new option of Workpiece clearance.
- ❑ The Workpiece clearance option enables you to set a value by which the tool clears the workpiece when moving between two positions.
- ❑ Note: This option is available only with the Autotilt technology.

Roughing and More Page: Trim only full contours



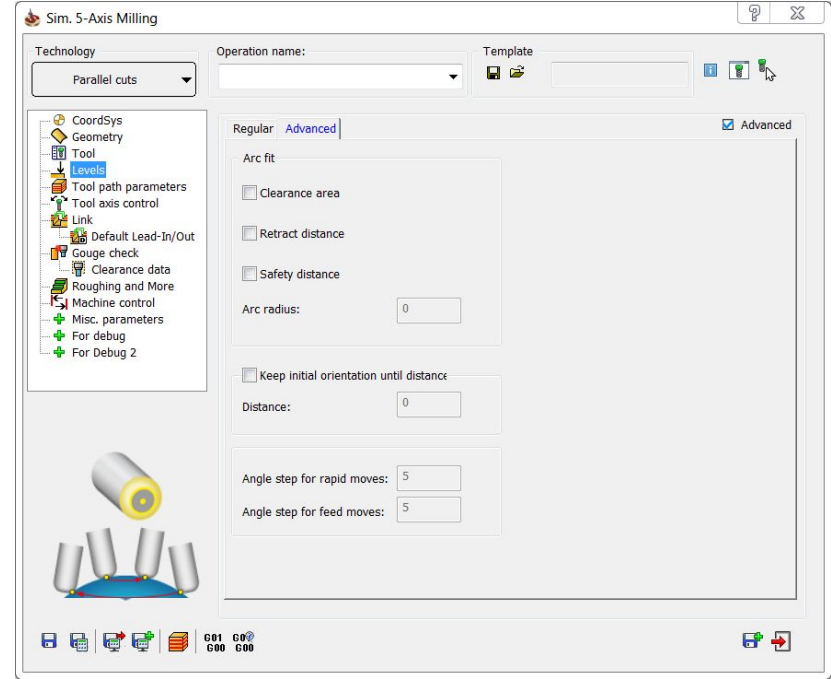
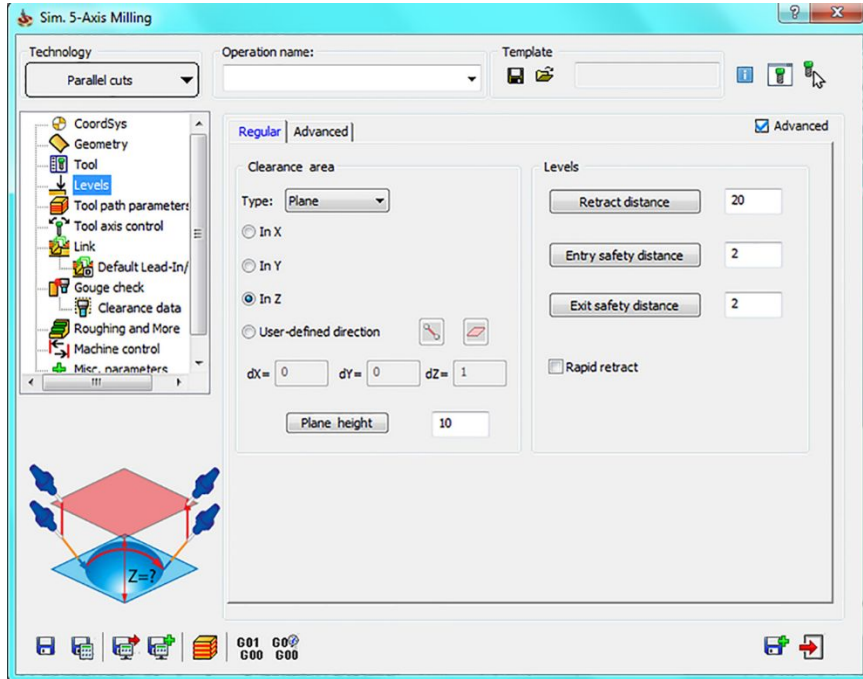
- **Stock definition: New option of Trim only full contours is added.**
- **Enables you to keep the cuts that are partially within the stock and remove the cuts which are completely outside the stock - enabling much smoother cuts and less jumps.**

GUI Changes - Geometry Page



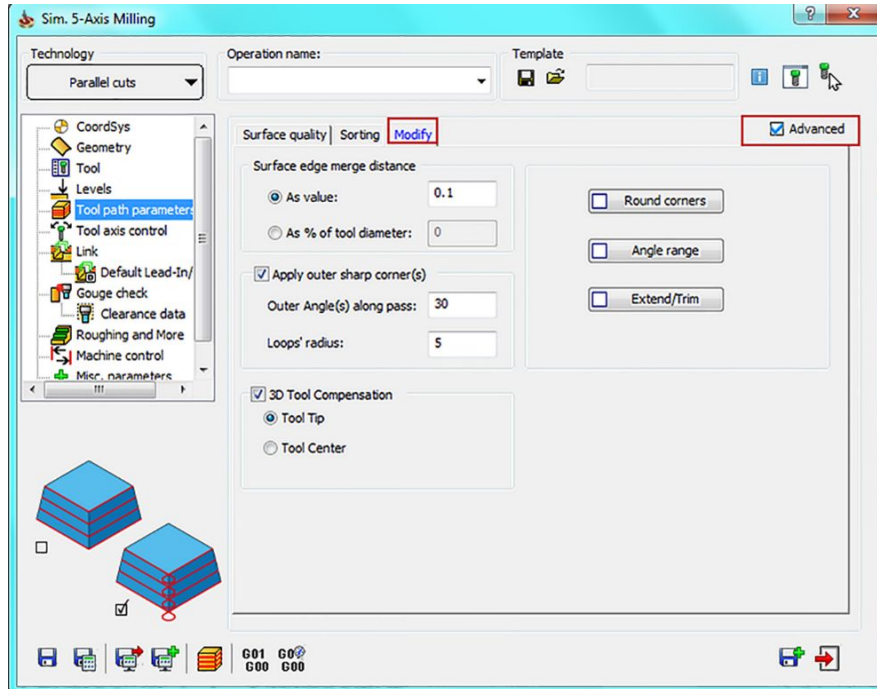
□ 2D Boundary curves content is placed inside Geometry page.

GUI Changes - Levels Page



□ Parameters divided into two tabs: Regular (most used) and Advanced.

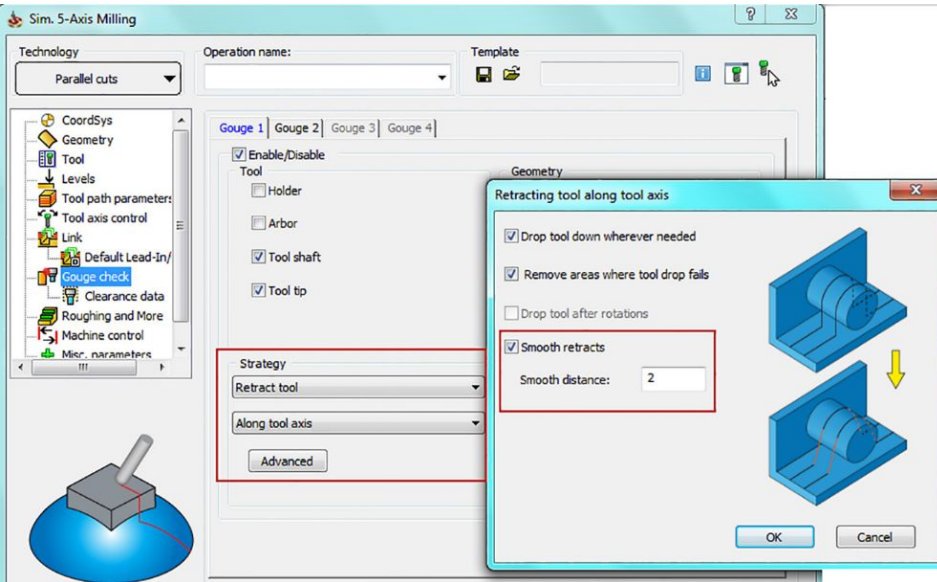
GUI Changes - Toolpath parameters Page



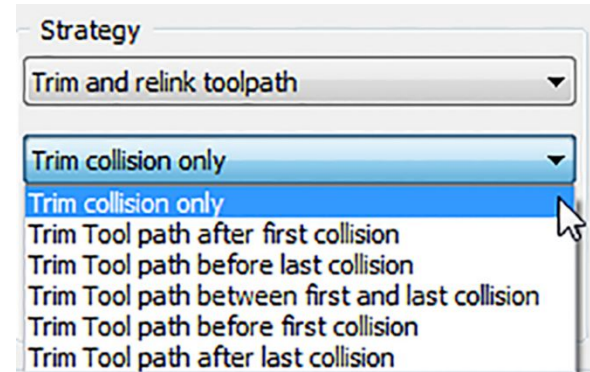
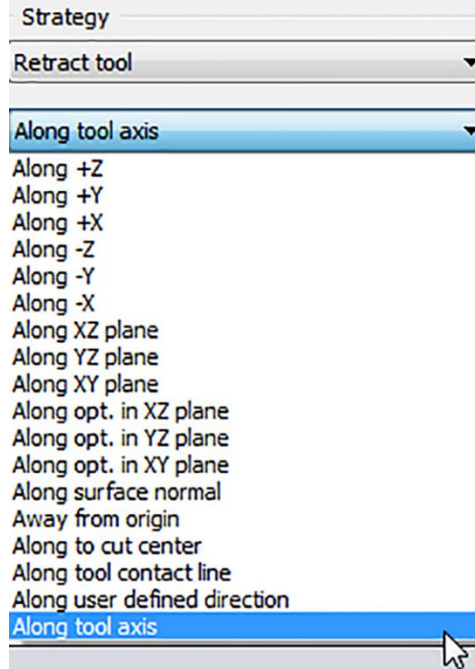
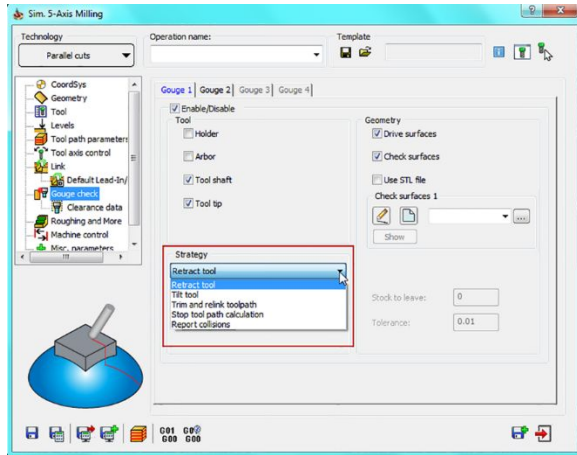
- New Modify tab is added.
- Round corners, Extend/Trim, and Angle range moved to this page from Geometry page since they are related to toolpath.
- Modify tab is available only when the Advanced check box is ON.

Gouge check Page: Smooth Retracts

- Strategy: Retract along tool axis/ Advanced: New option of Smooth retracts is added.
- The Smooth retracts check box enables you to smooth the transition from the collision free area to the tool retraction area by avoiding sudden axis jumps.
- The Smooth distance field determines the start distance of the smoothing to the collision area.

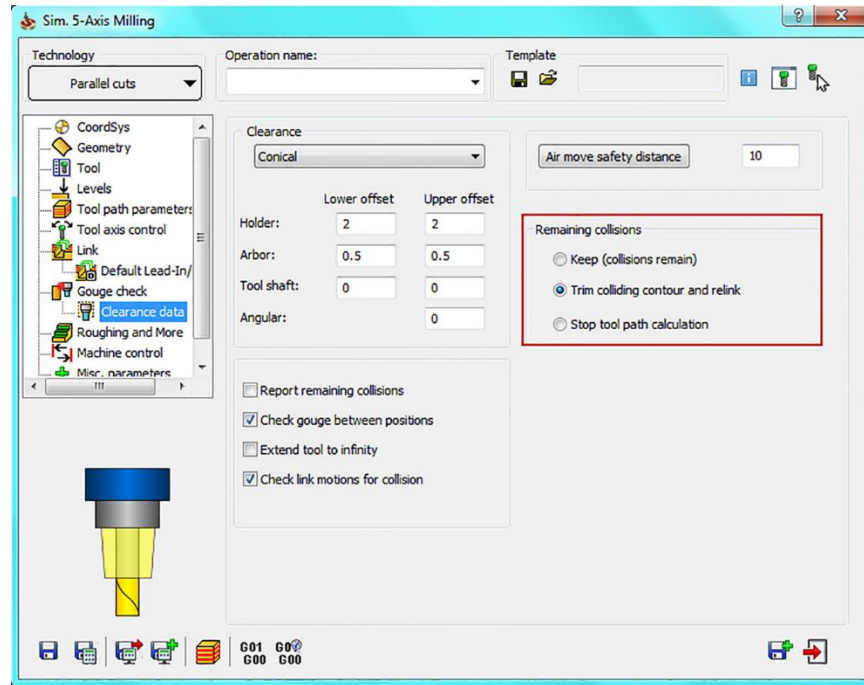


Gouge check Page: Renaming of strategies



 The Gouge check strategies are logically named

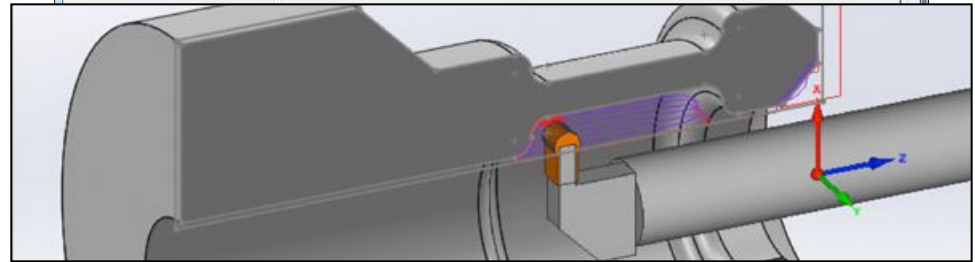
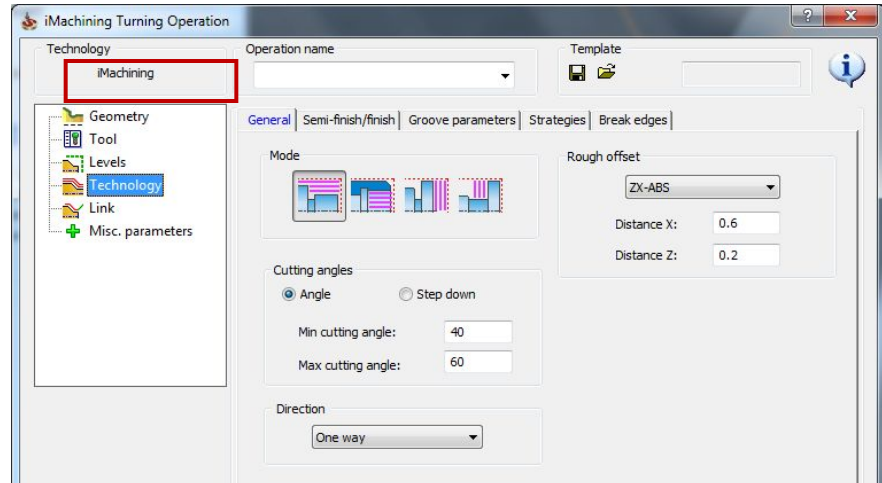
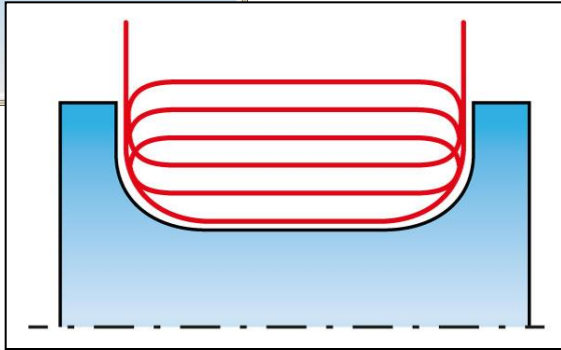
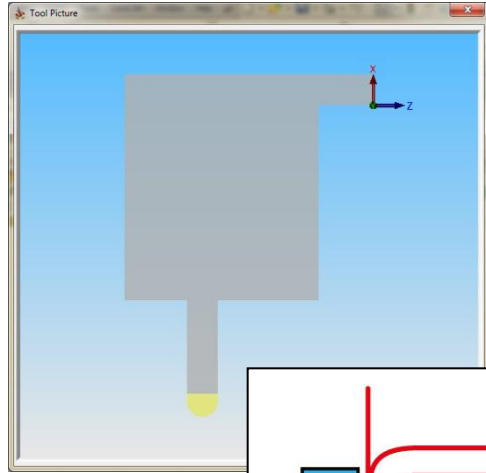
Gouge check Clearance data Page



□ Remaining collisions section is updated – one option removed.

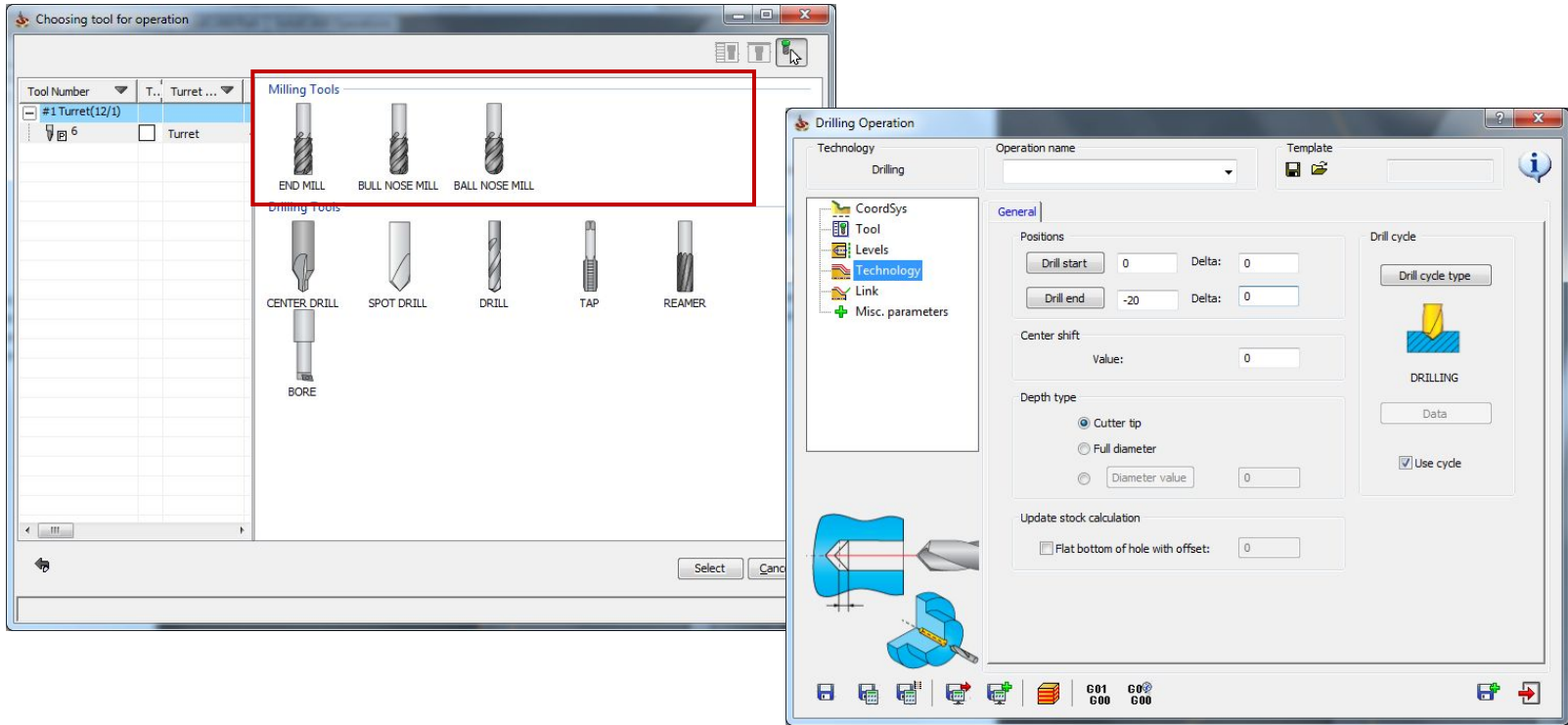
Turning

Turning: Torochoidal Turning operation



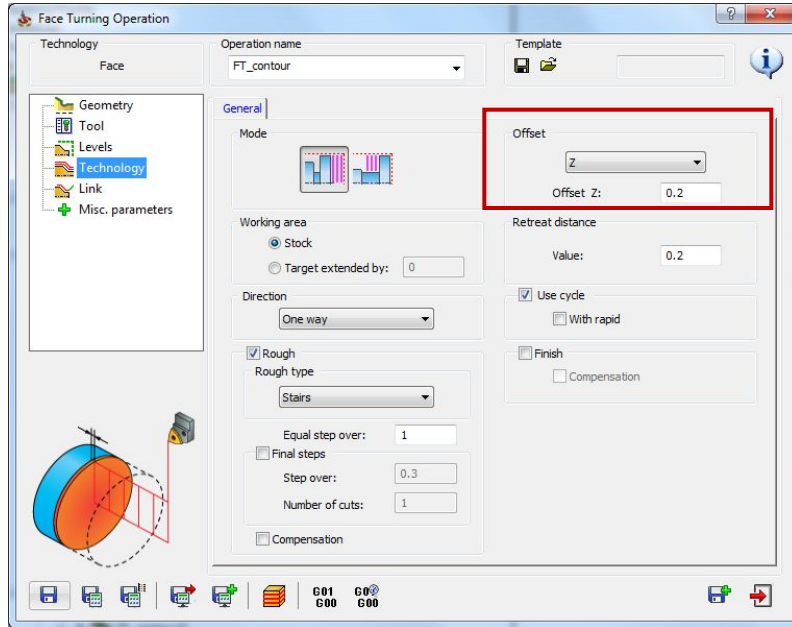
- iMachining-style torochoidal moves of round grooving tool in turning

Turning: Use all milling tools in turning drilling



- Possibility to use Milling tools in Turning Drilling operation

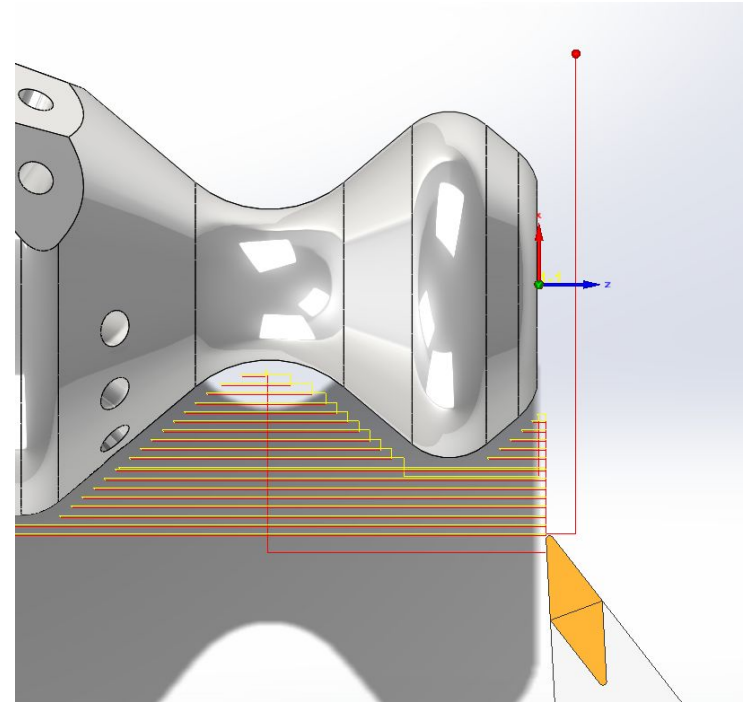
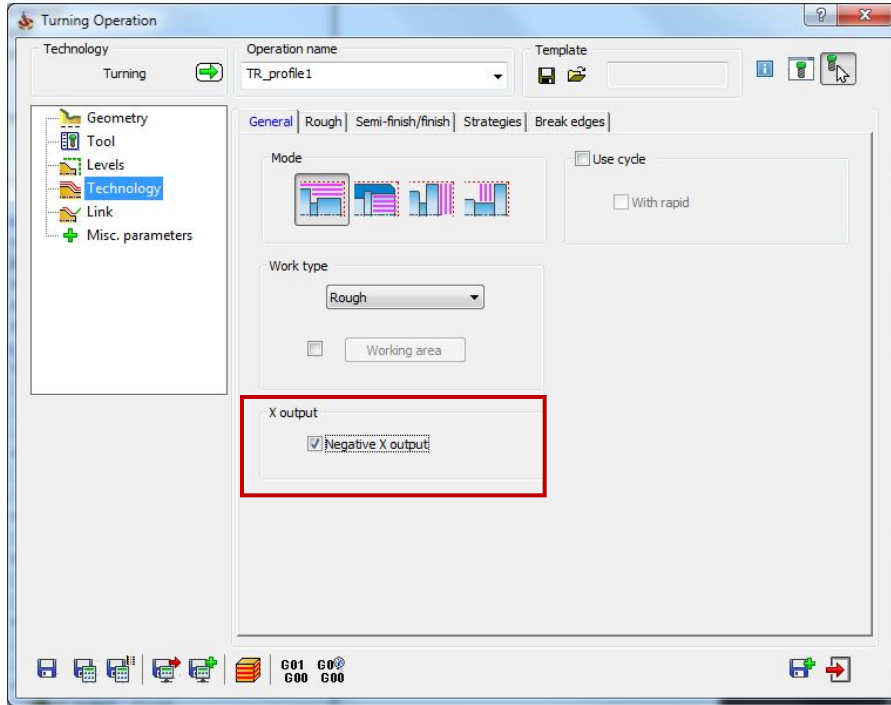
Turning: Offset types in face turning



@turning ==> work_type:rough semi_finish:false finish:false
..> label:5002 start_line:10 end_line:12
..> process_type:face turning_mode:external
..> is_line:false num_points:0
..> rough_offset_x:0.000 **rough_offset_z:0.200**
..> semi_offset_x:0.000 semi_offset_z:0.000
..> first_pos_x:26.600 first_pos_z:0.000
..> last_pos_x:5.600 last_pos_z:0.000
..> down_step:1.000 safety:2.000
..> retreat_distance:0.200

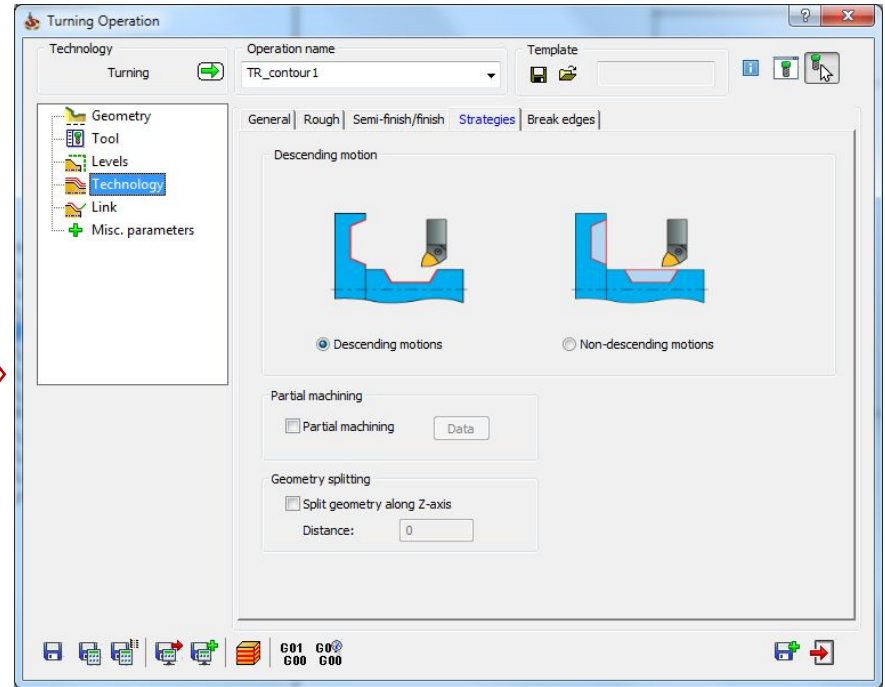
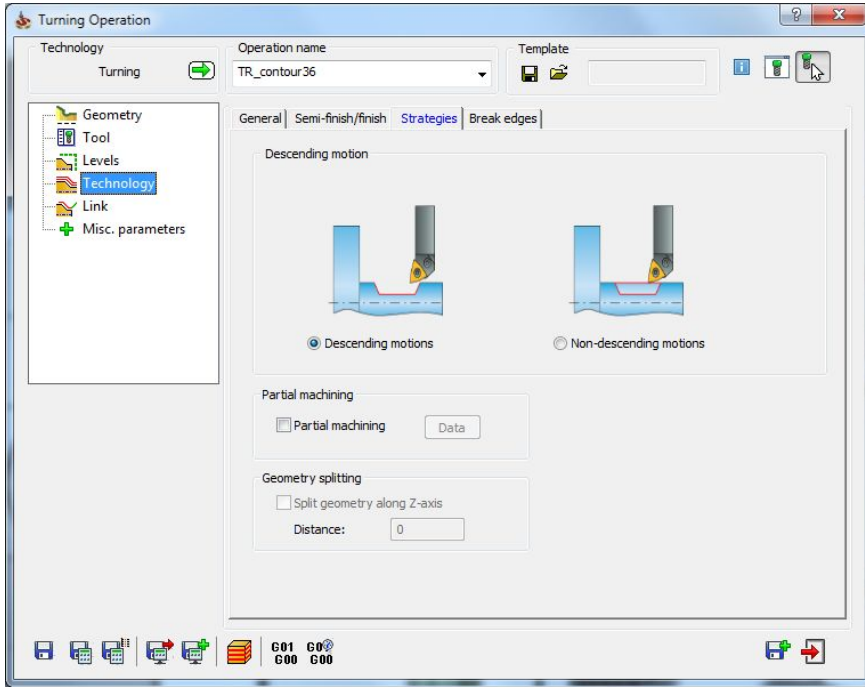
- Output offset value in Face turning cycle

Turning: Negative X output



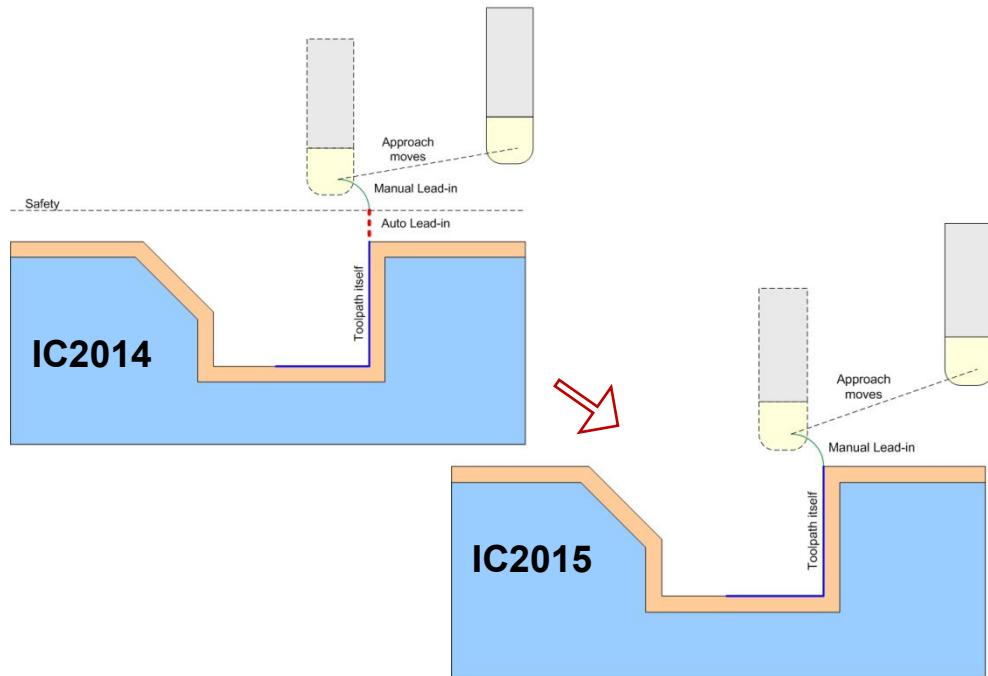
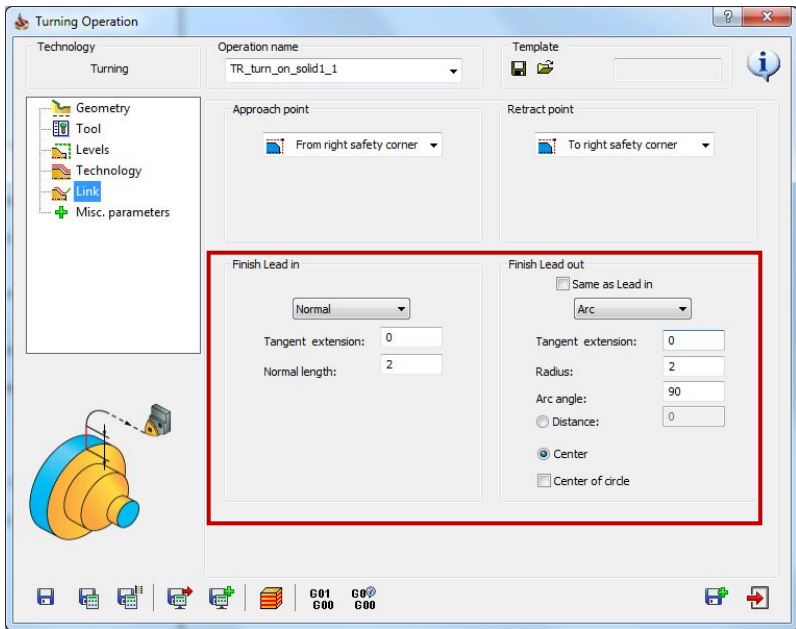
- Possibility to get (-X) cutting toolpath with start/end points in (+X)

Turning: Non-descending motion on face surfaces also



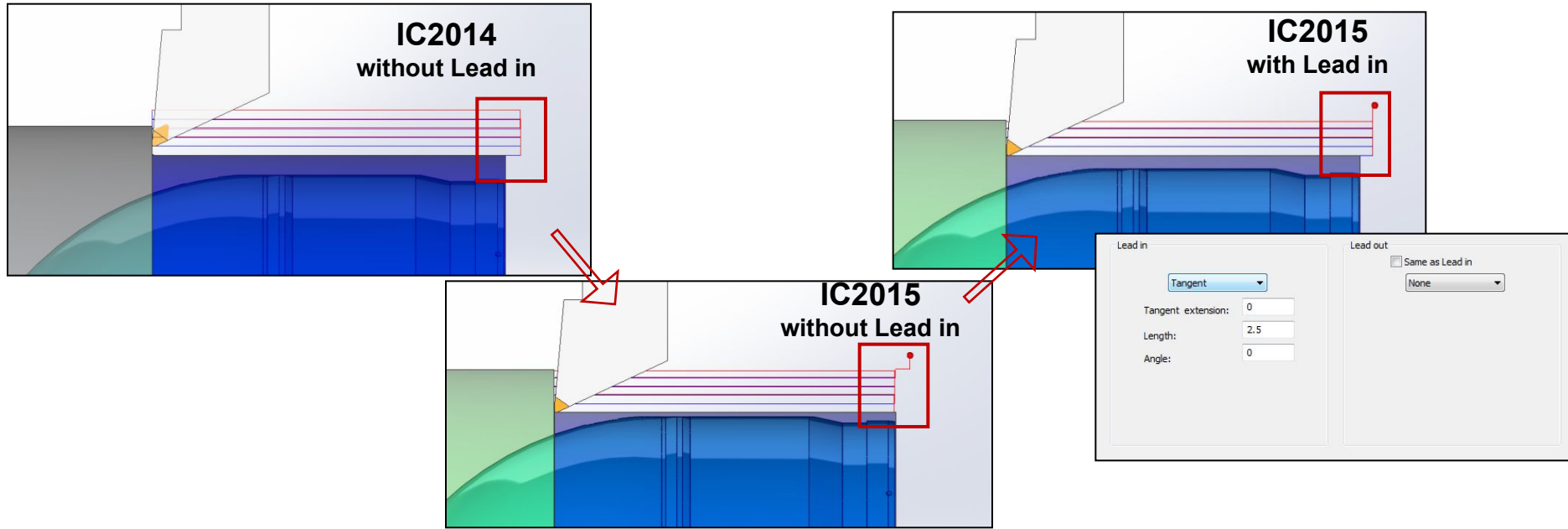
- Option to avoid penetration to slots on face surfaces also

Turning: Auto Lead in/out cancelled



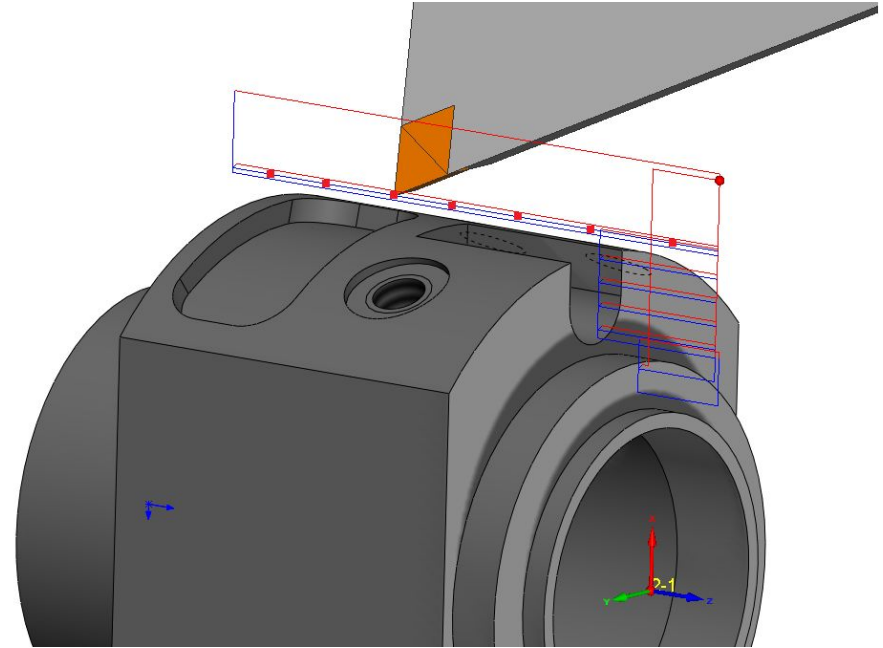
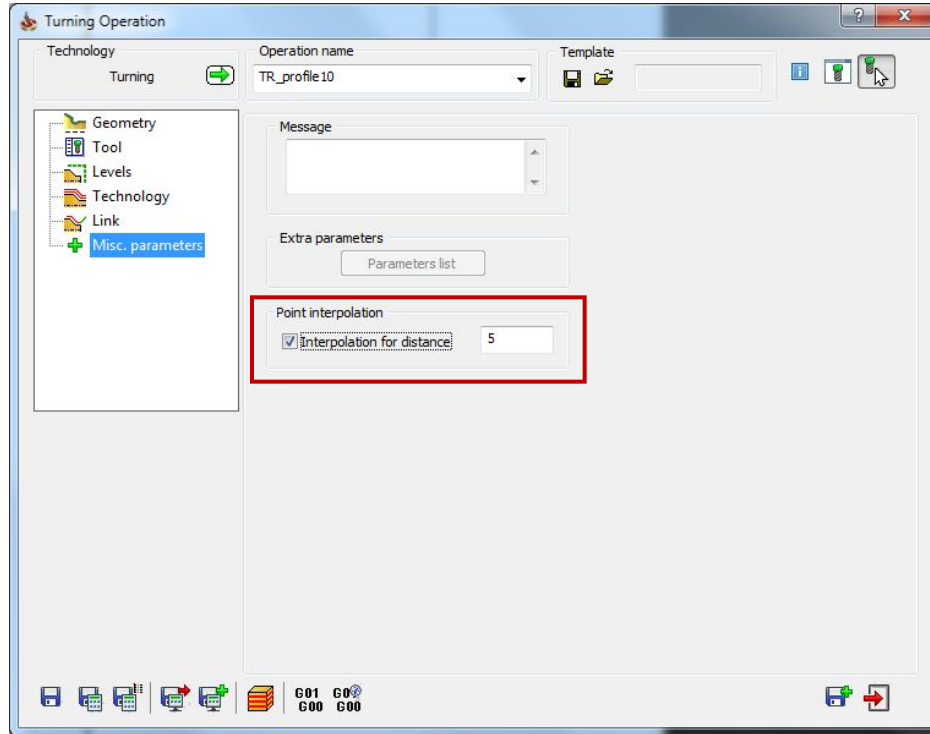
- In previous versions, an additional movement before Lead in/out was added automatically. In this version they are not added; user defines lead-in and lead-out

Turning: Lead in/out Used in rough turning also



- In previous versions, we had only an automatic lead in/out for roughing
- Now the automatic lead in/out is cancelled and manual lead in/out, if defined, will be applied to Rough toolpath also

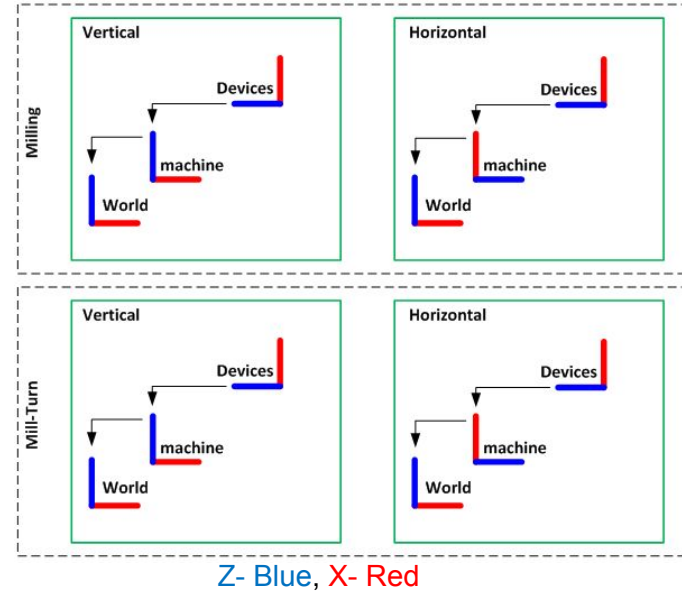
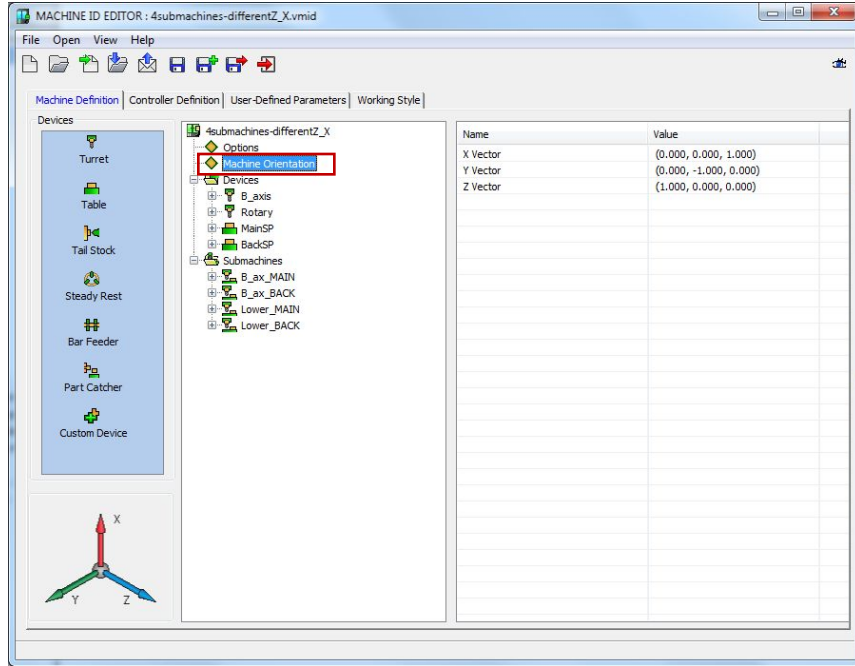
Turning: Split long lines of turning toolpath



- Split long lines of turning toolpath to short lines, according to the distance defined

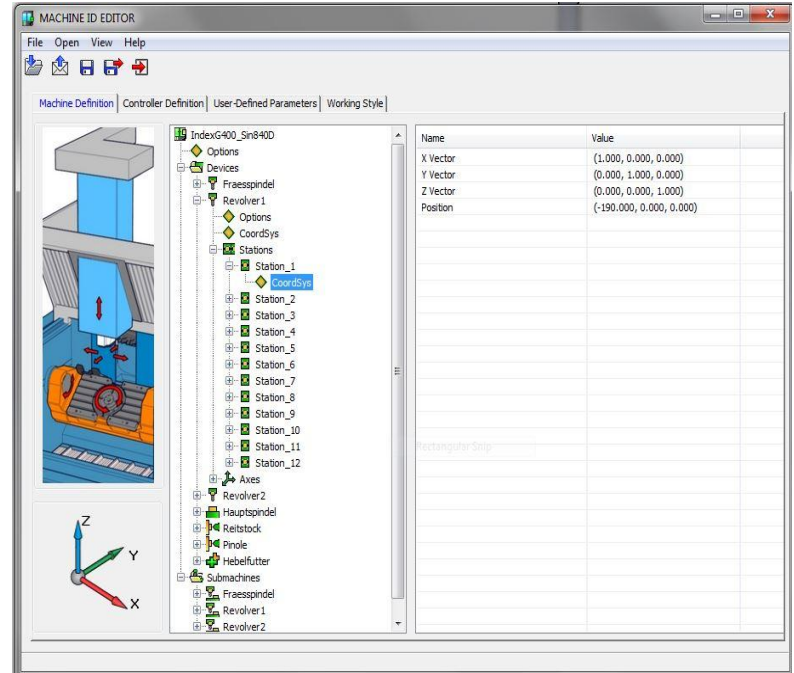
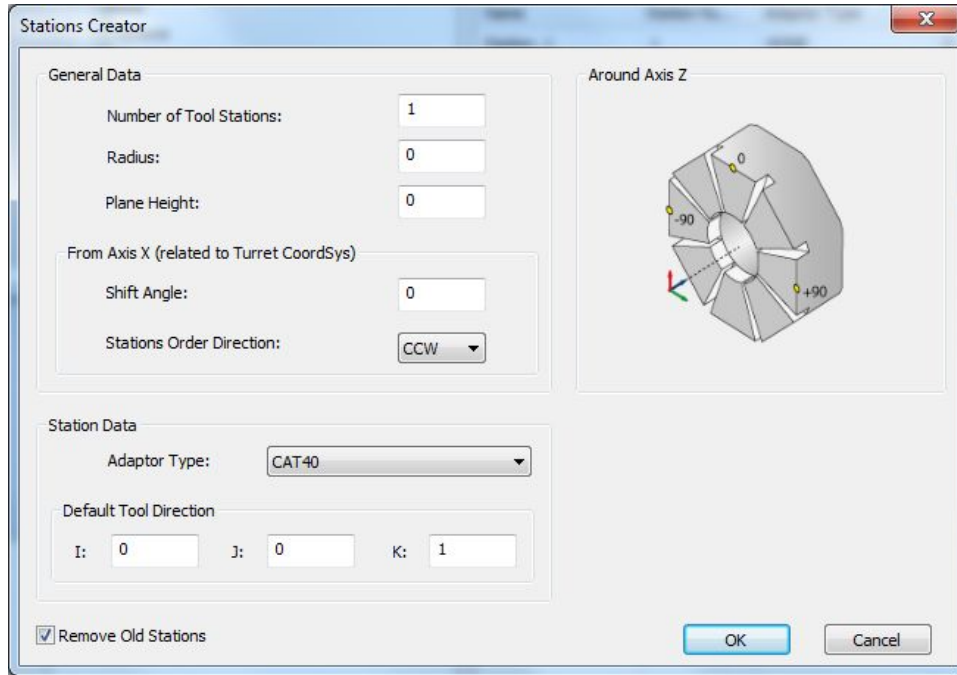
Mill-Turn

Machine ID: Machine orientation



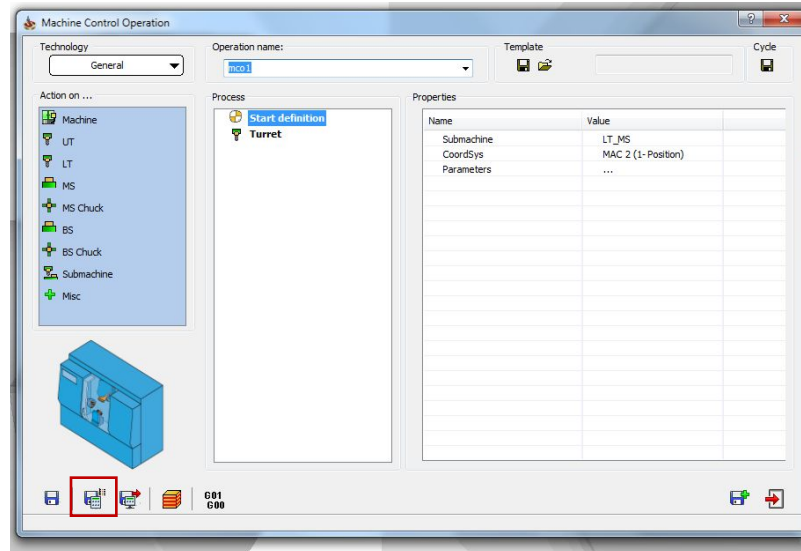
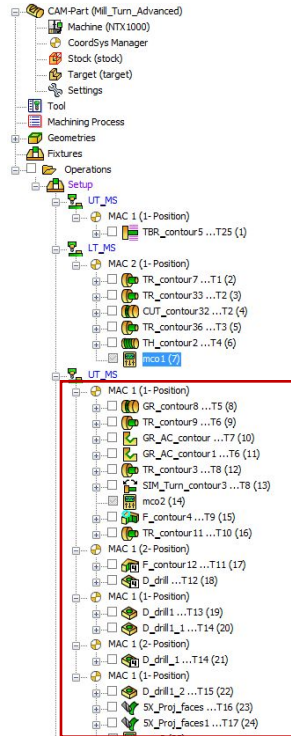
- World CoordSys is always with Z vertically UP
- Machine orientation (vertical, horizontal) is defined in World CoordSys

Turret: New style of station creation in VMID



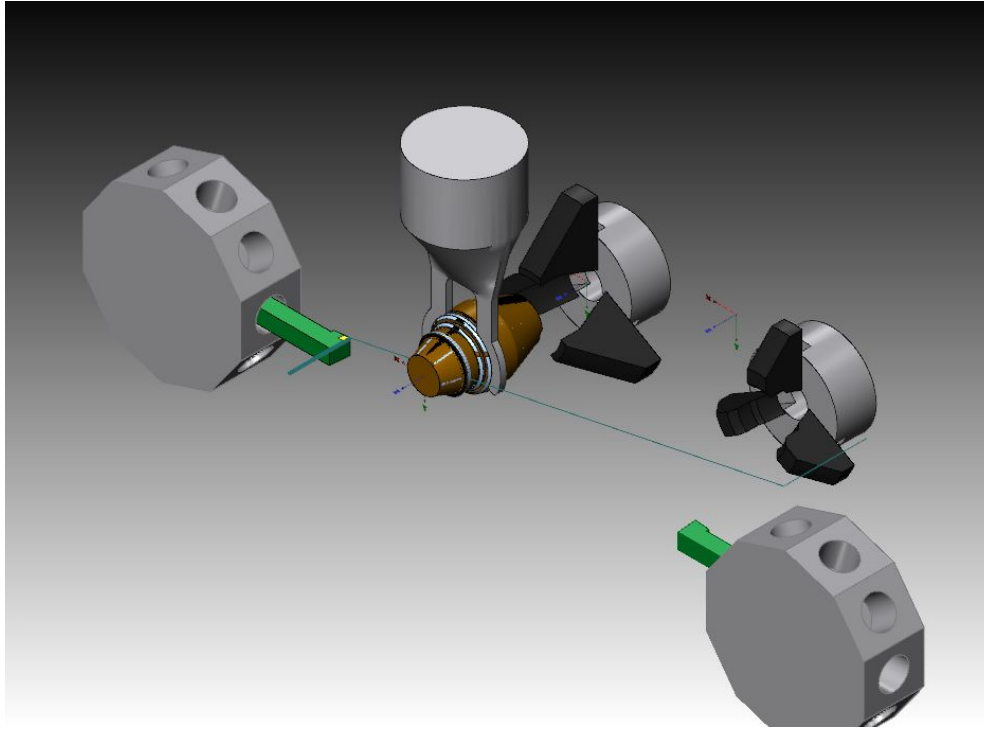
- Easy way to create multiple stations in VMID

MCO: Calculate all related operations when MCO changes



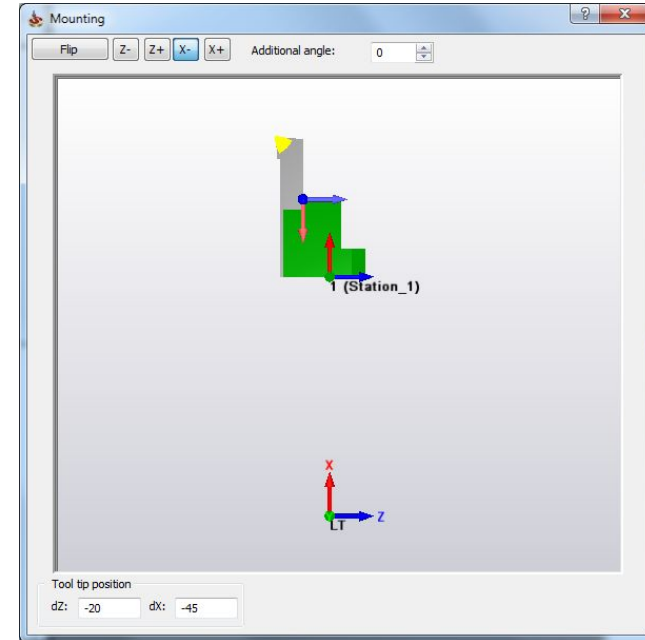
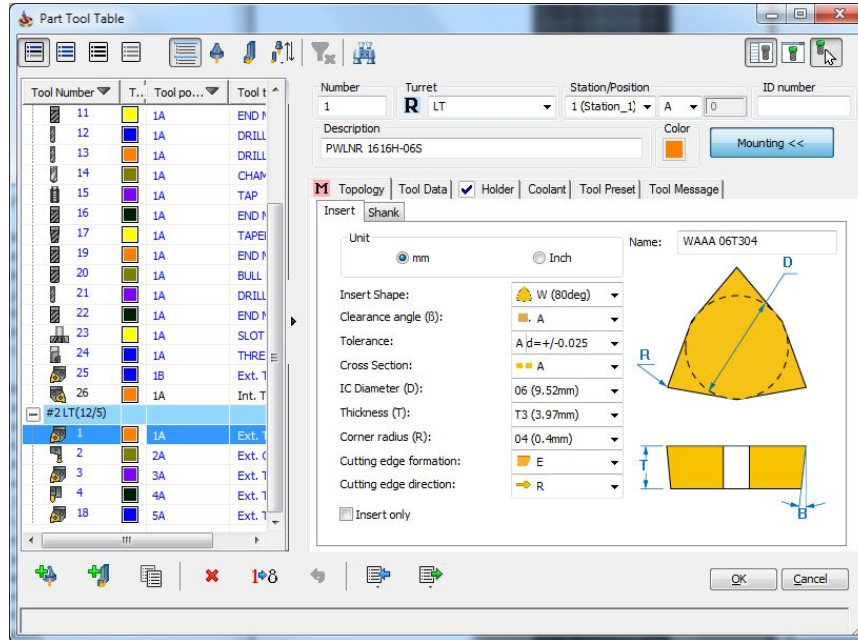
- Automatic calculation of changed MCO's related operations

MCO: Move Part by turret or any other device



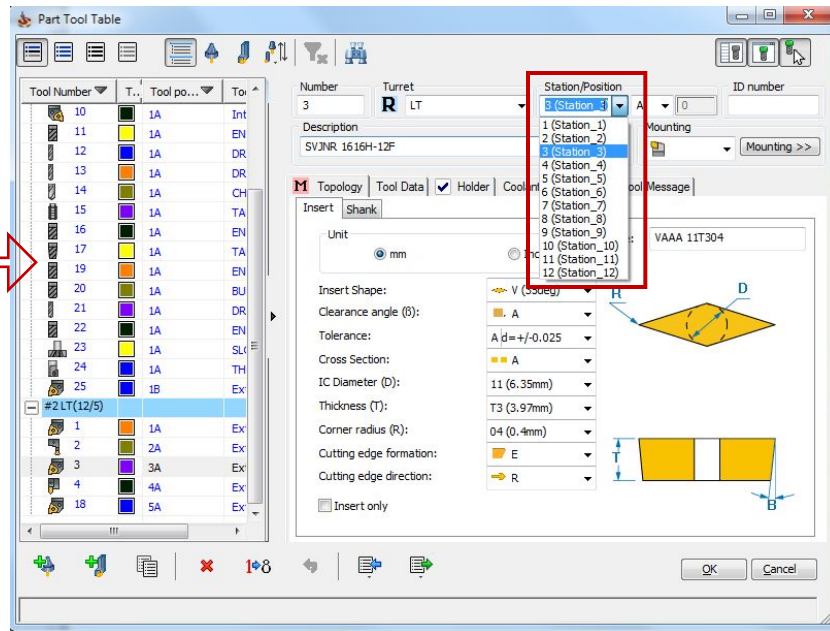
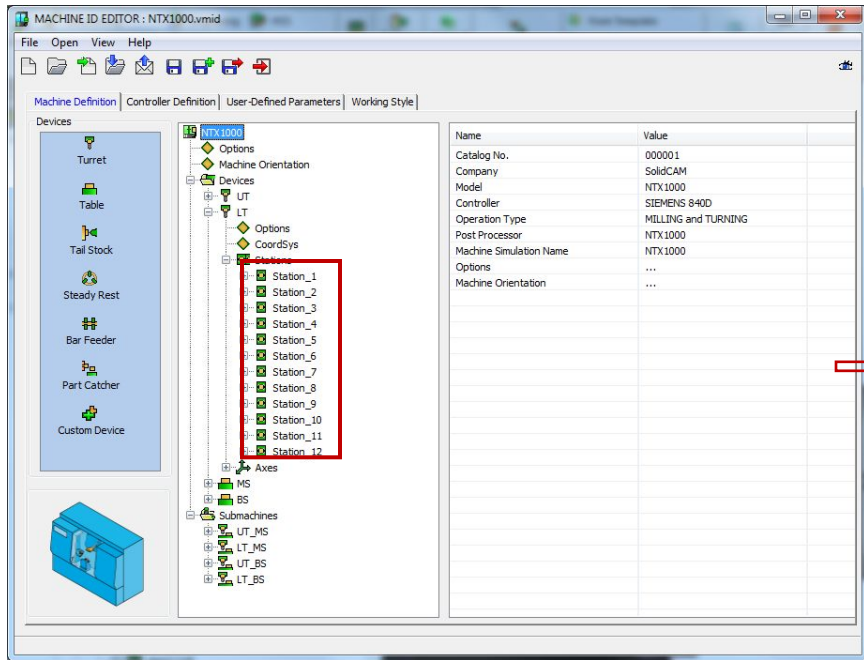
- **Possibility to move the CAM-part from Table to Table by turret or any other device**

Tooltable: new mounting interface



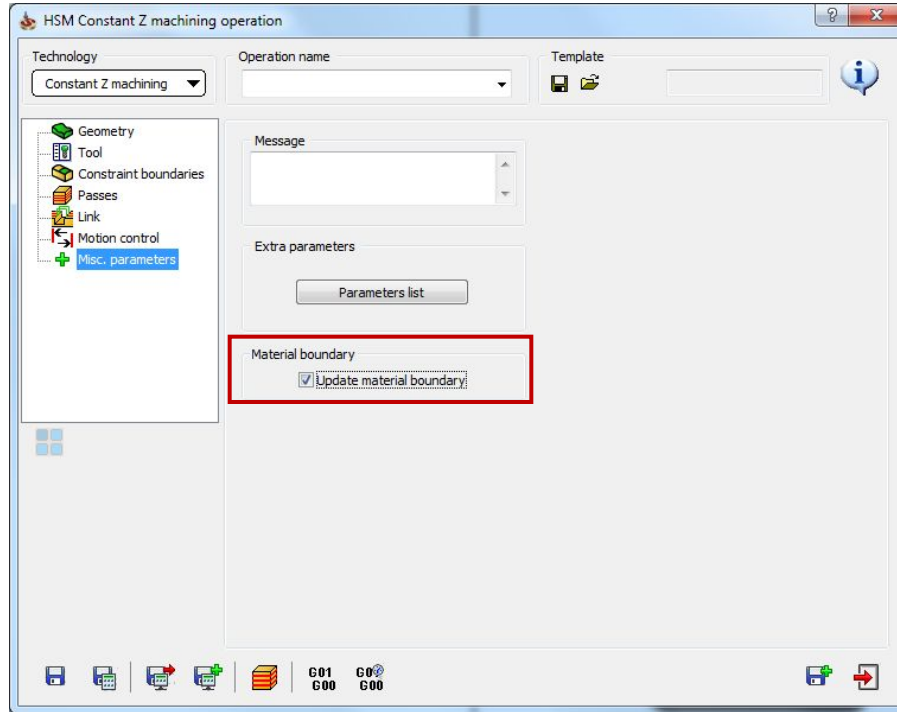
- New tool mounting interface – based on station and device CoordSys
- Shown as it actually looks in CNC Machine

Tooltable: Show station name



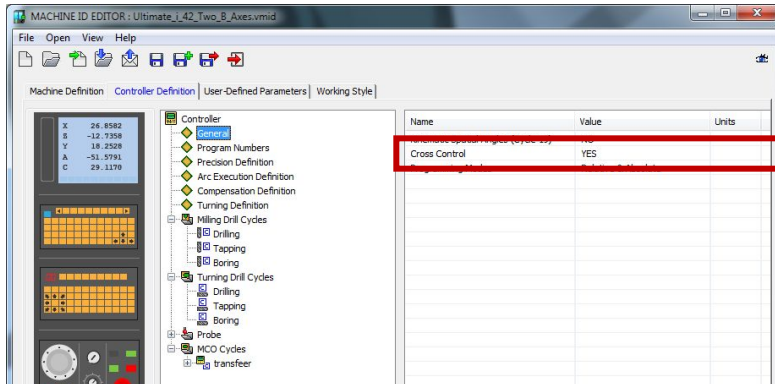
- Show station name in the List of stations in Tooltable

Update material boundary (for mill-turn)

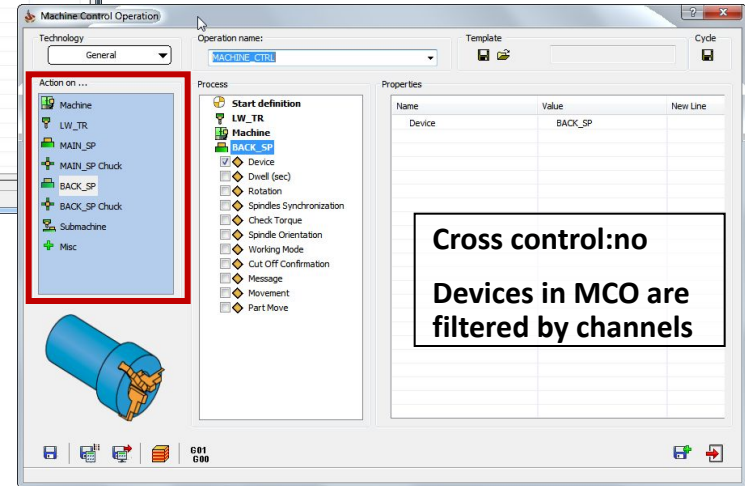
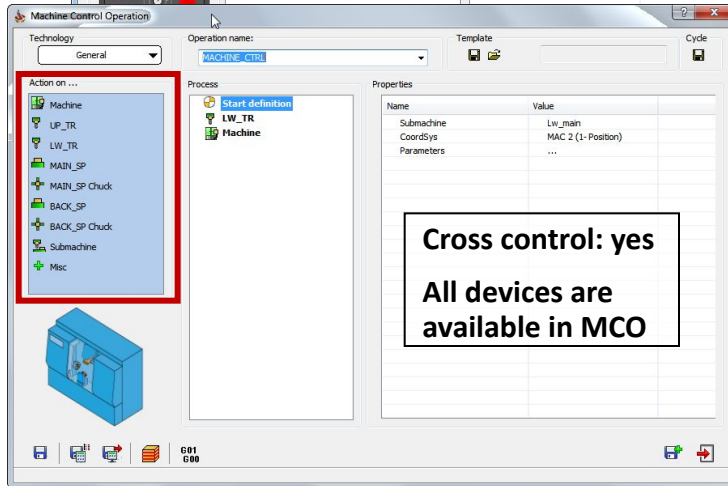


- Take into account HSR/HSM operations when updating stock for further turning operations

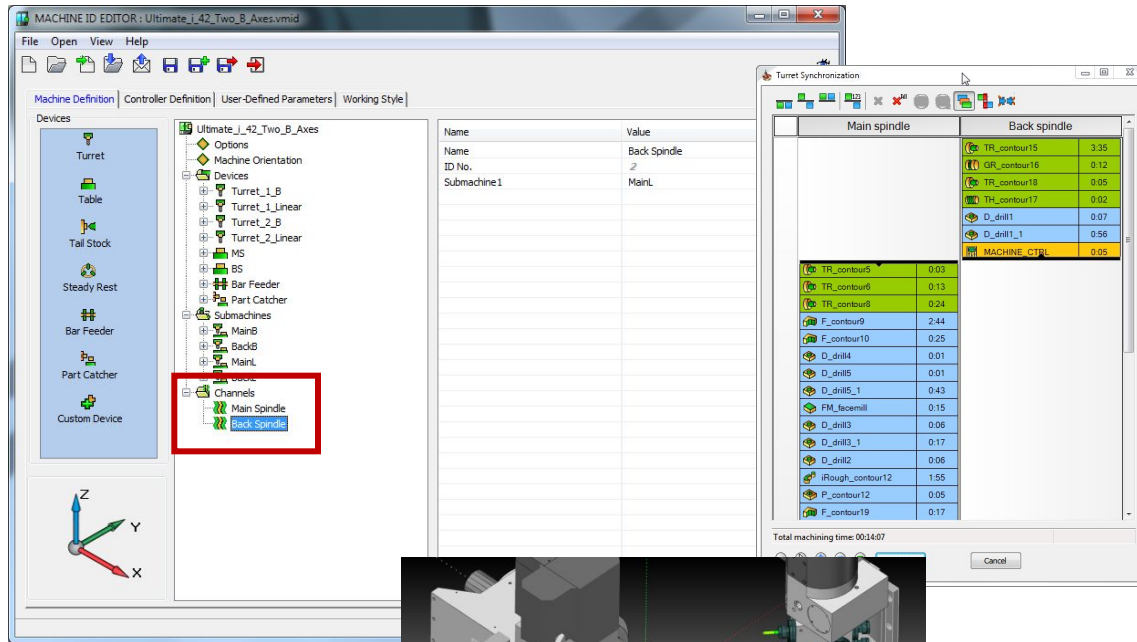
Definition of Cross Control



- By default – cross control is always turned on
- *.VMID allows to cancel Cross Control (access to devices from opposite channel), if it is not supported by the machine.



Channels definition in VMID



- Custom channels definition
- Definition of channels by sub-machines, not turrets
- Enables synchronisation process between tables (Swiss Type lathes)

Full support of Mazak Mill Turn machines



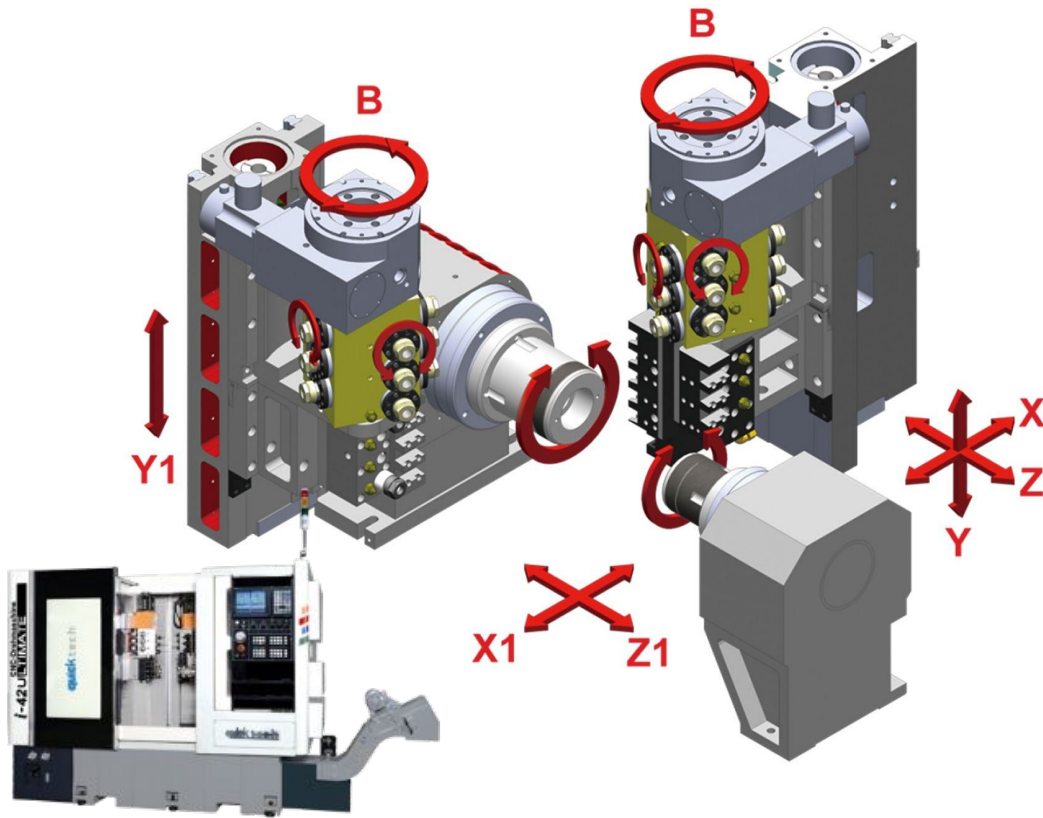
- **Opposite spindle**
- **Rotary turret**
- **Simultaneous 5 axes**
- **Balanced turning**
- **Part transfer**
- **Additional devices**

Full support of Fanuc Mill Turn machines



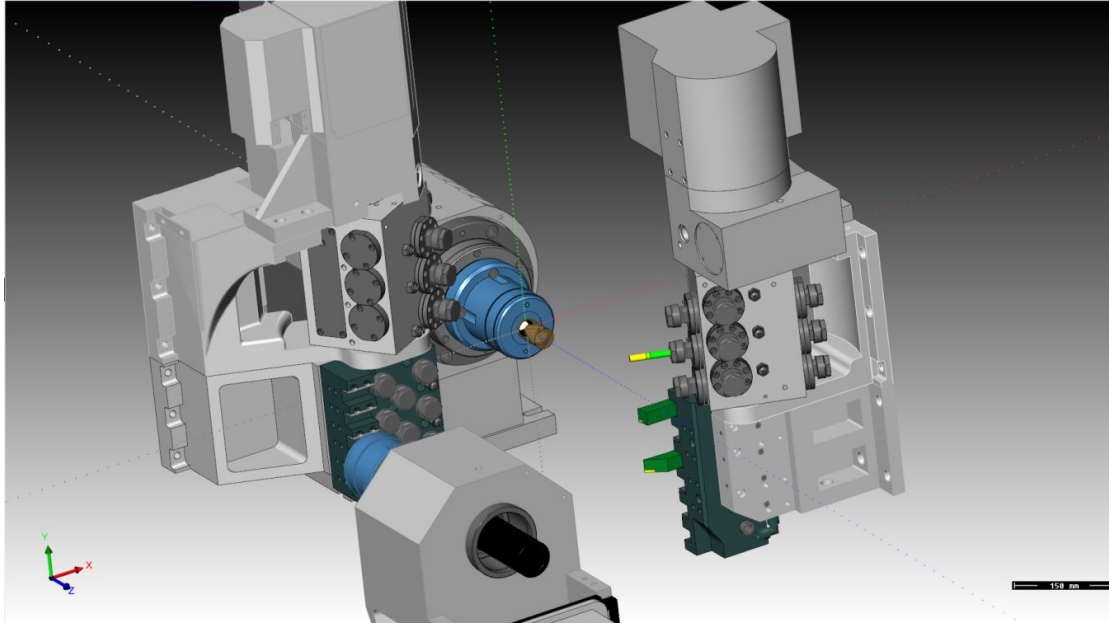
- **Opposite spindle**
- **Rotary turret with milling functionality**
- **Simultaneous 5 axes**
- **Balanced turning**
- **Part transfer**
- **Additional devices**

Support of machines with combined turrets

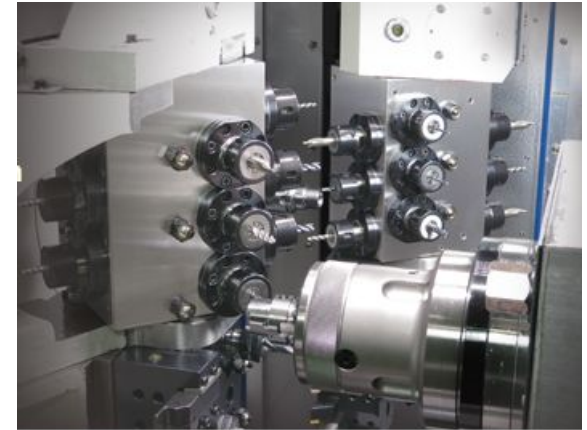


- Opposite spindle
- Rotary turret as B-axis
- Linear turrets
- Simultaneous 5 axes
- Synchronization
- Part transfer
- Combined turrets

Support of Multi-station Combined-turrets Mill-Turn machines

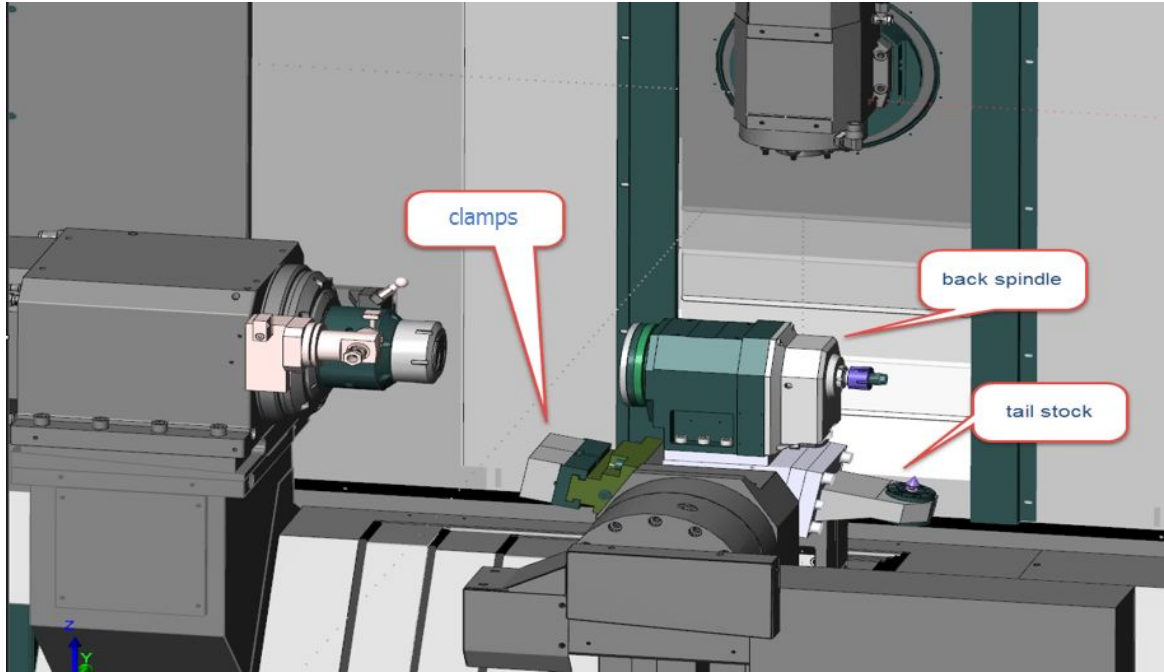


Quicktech Ultimate I42



- **Two spindles, Each turret is a combined turret (Rotary type and Linear type)**
- **Channel synchronization by tables**

Support of Willemin-style Mill-Turn machines



[Willemin 508 MT](#)

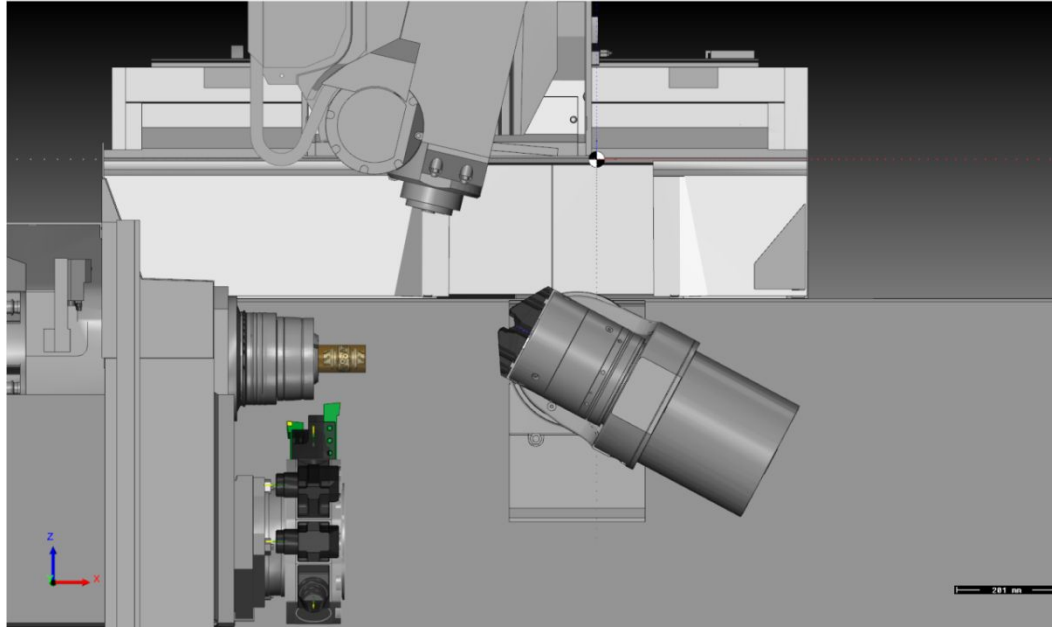
[Cutting Video 1](#)

[Cutting Video2](#)



- **Willemin-style: Two spindles, B-axis, Additional devices (Clamps and tail Stock)**
- **Additional devices support by MCO, with full machine simulation**

Support of Chiron-style Mill-Turn machines

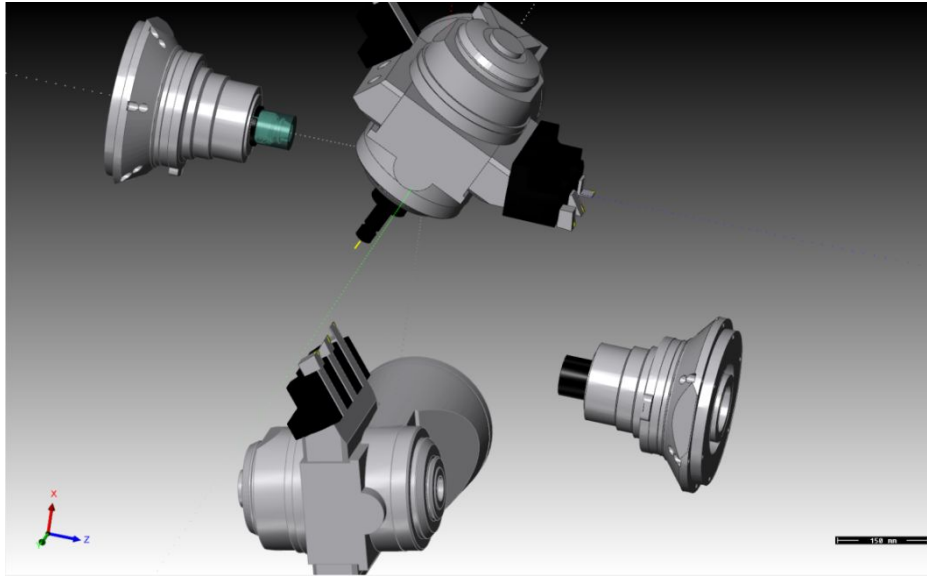


Chiron FZ 12 MT



- **Chiron-style: Main spindle, Rotary Turret, B-axis, Tilting of back spindle**
- **Full support of MCO operations with full machine simulation**
- **Working with Synchronization (Multi-Channel)**

Support of Index-style Mill-Turn machines



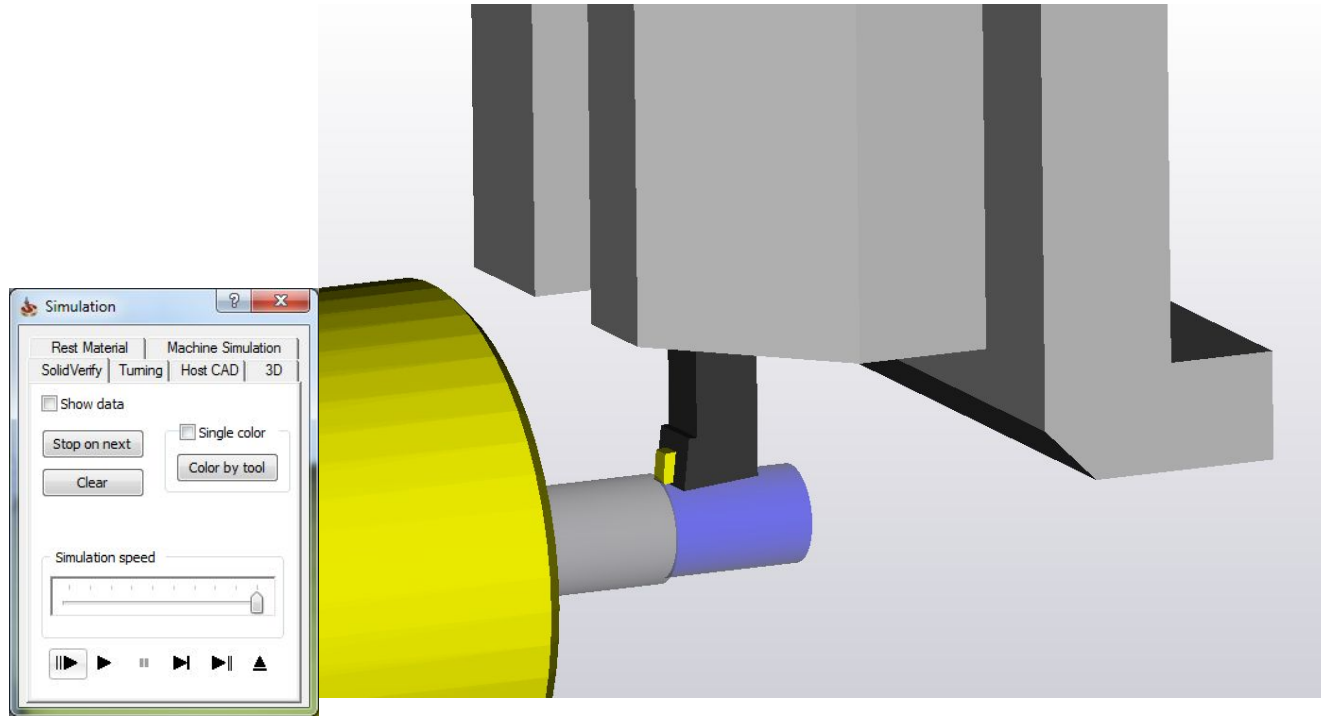
[Index R200](#)



- **Index-style: Two spindles, Each turret is a combined turret (B-axis type and Linear type)**
- **Full support of MCO operations, with full machine simulation**
- **Programming by Channels (not by Turrets)**

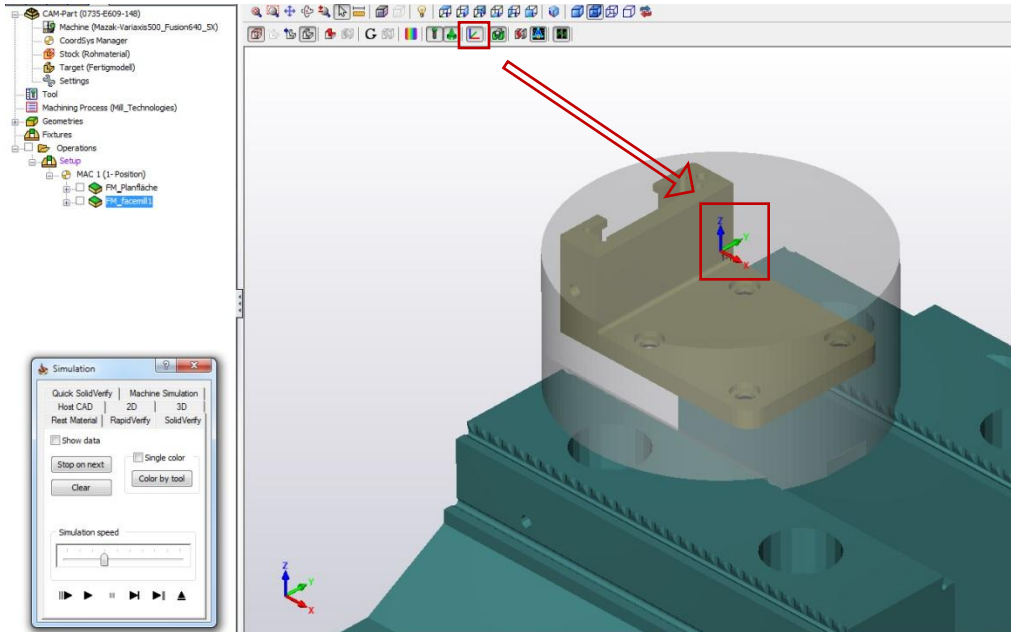
Simulation

Simulation: Show STL holders in simulation



- Show STL holders in simulation and in gouge checking

Simulation: Show current CoordSys in SolidVerify simulation



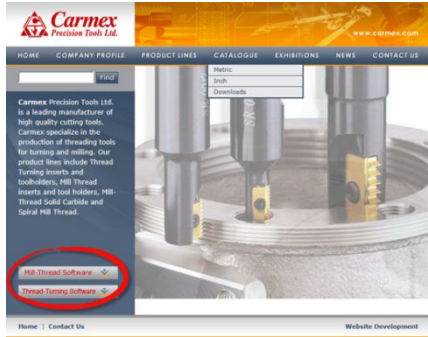
- Show current CoordSys in SolidVerify simulation
- Useful for Shop Floor Editor, where SolidCAM runs standalone.

Tool Libraries integrations

Tool Libraries



Iskar



Carmex



Vardex

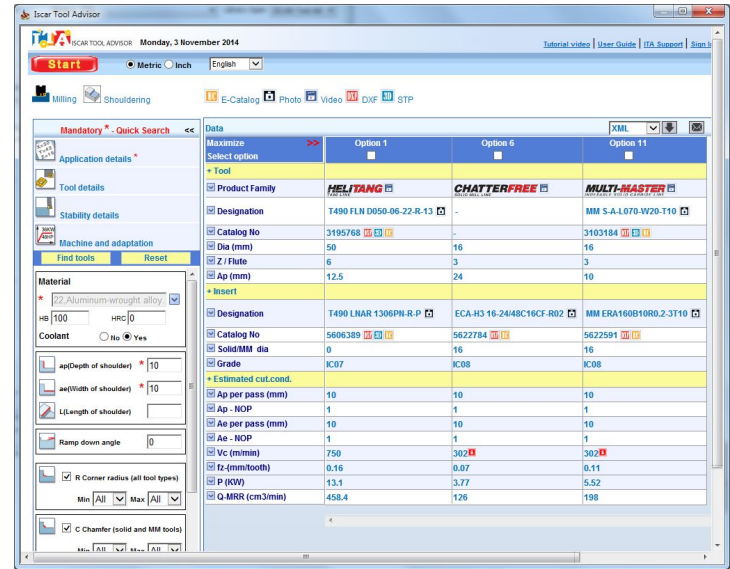
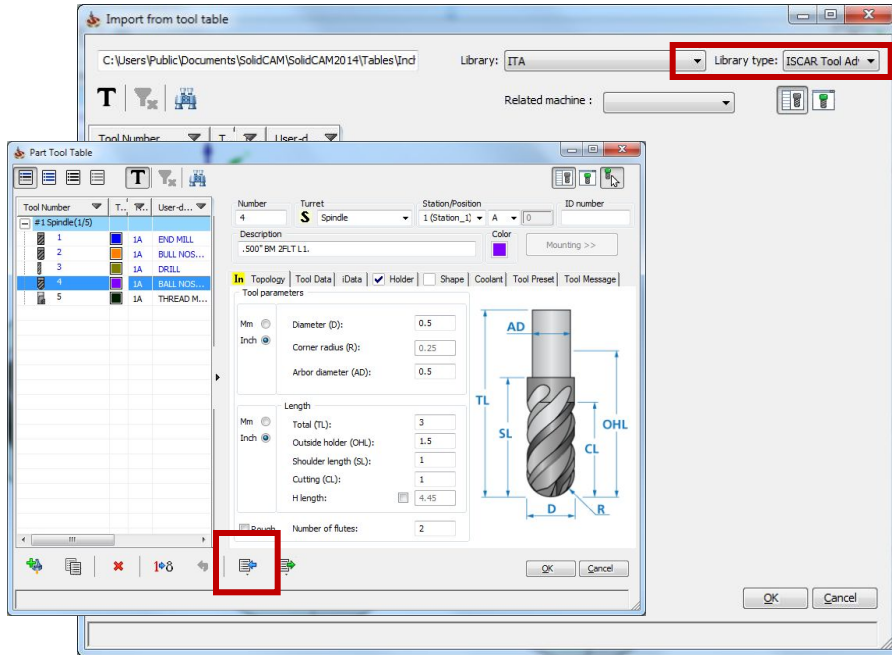
- Online / Offline Databases from Tool Manufacturer
- Offering 2D/3D CAD Models with exact Tool Dimensions
- .. and recommended Cutting Conditions for their Tools

ISCAR Tool Advisor (ITA)



- **Online Catalog with assistance to guide users to the best ISCAR Tools**

Integration with ISCAR Tool Advisor (ITA)



<http://www.iscar.com/ita/MainPage.aspx>

- At the Import Option of the Tool Table, select ISCAR Tool Advisor to launch it

Carmex



<http://www.carmex.com>

Carmex Precision Tools Ltd.
Precision Tools Ltd.

HOME COMPANY PROFILE PRODUCT LINES CATALOGUE EXHIBITIONS NEWS CONTACT US

Find

Metric
Inch
Downloads

Carmex Precision Tools Ltd. is a leading manufacturer of high quality cutting tools. Carmex specialize in the production of threading tools for turning and milling. Our product lines include Thread Turning inserts and toolholders, Mill Thread inserts and tool holders, Mill-Thread Solid Carbide and Spiral Mill Thread.

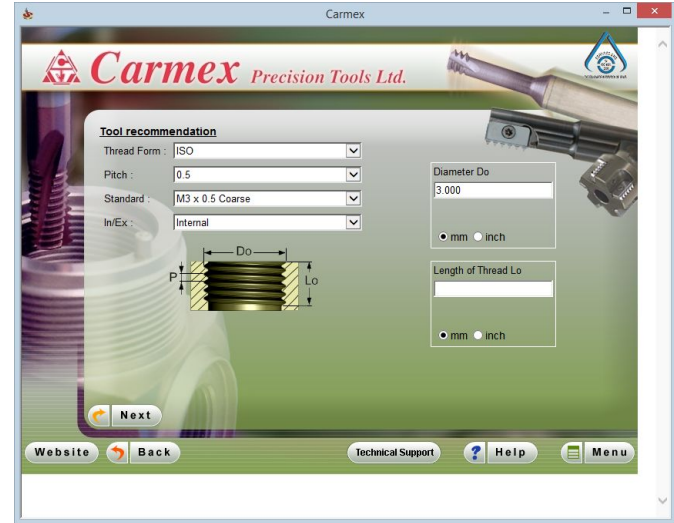
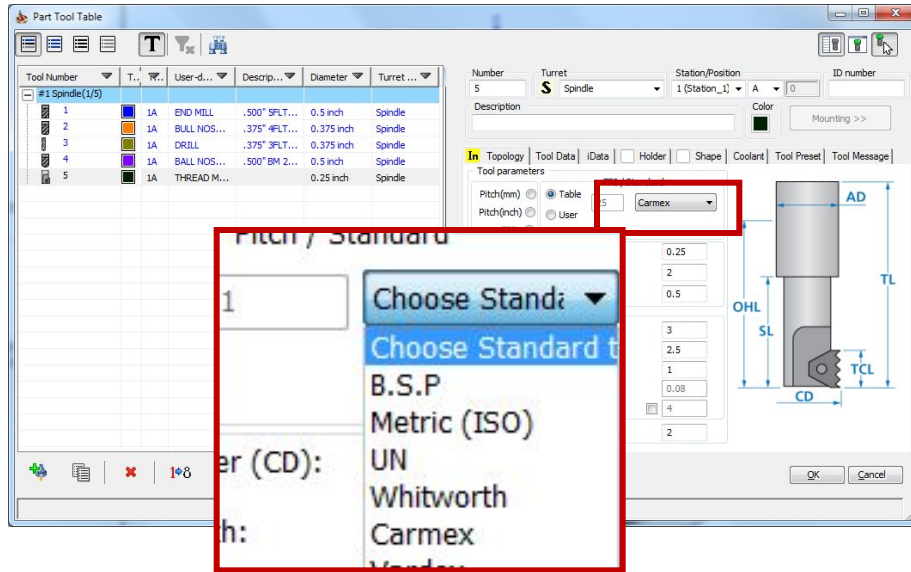
Mill-Thread Software
Thread-Turning Software

Home | Contact Us Website Development



• Carmex is specialized in the Production of Threading Tools for Turning and Milling

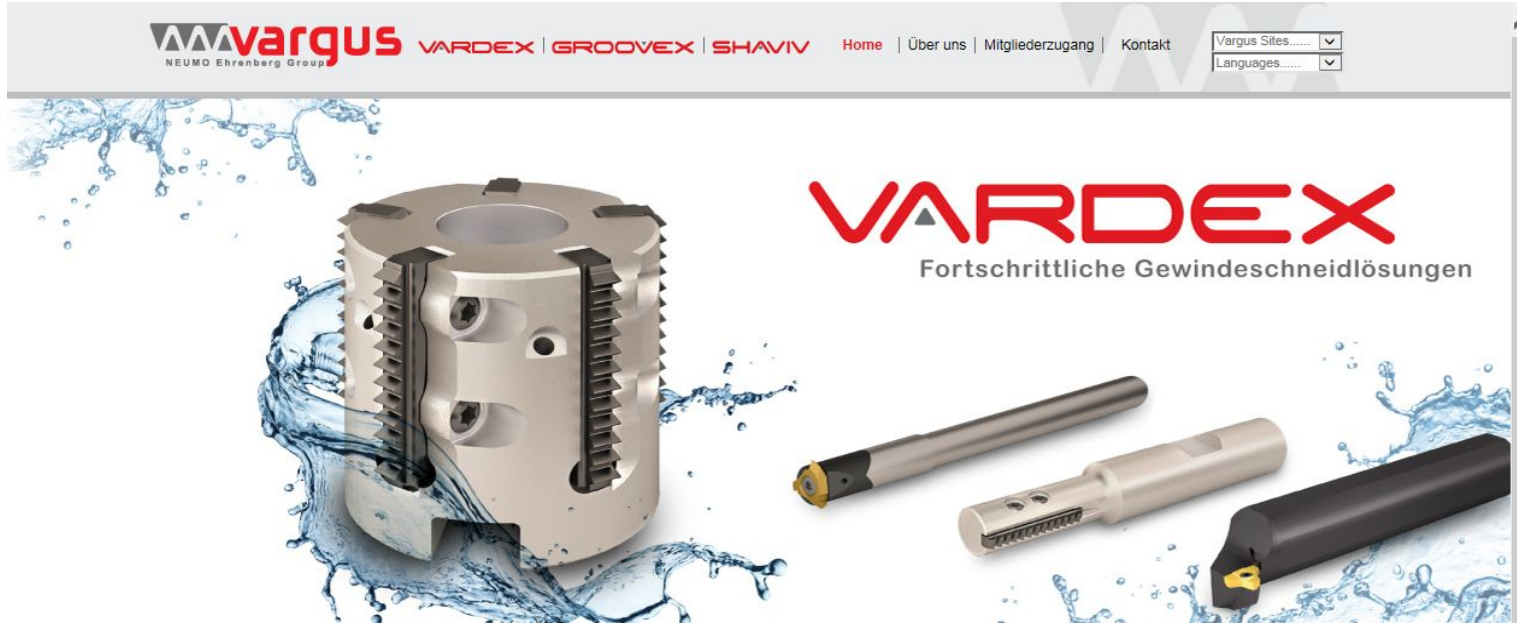
Integration with Carmex Tool Library



(Carmex software should be installed)

- Integration at the Tool Type Thread Mill in the Table of Pitch / Standard
- Select Carmex from the List - Carmex Tool Recommendation Software starts

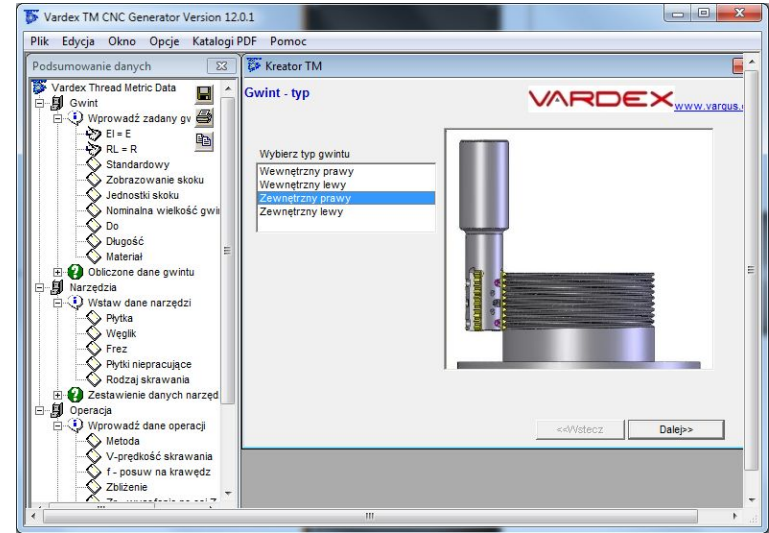
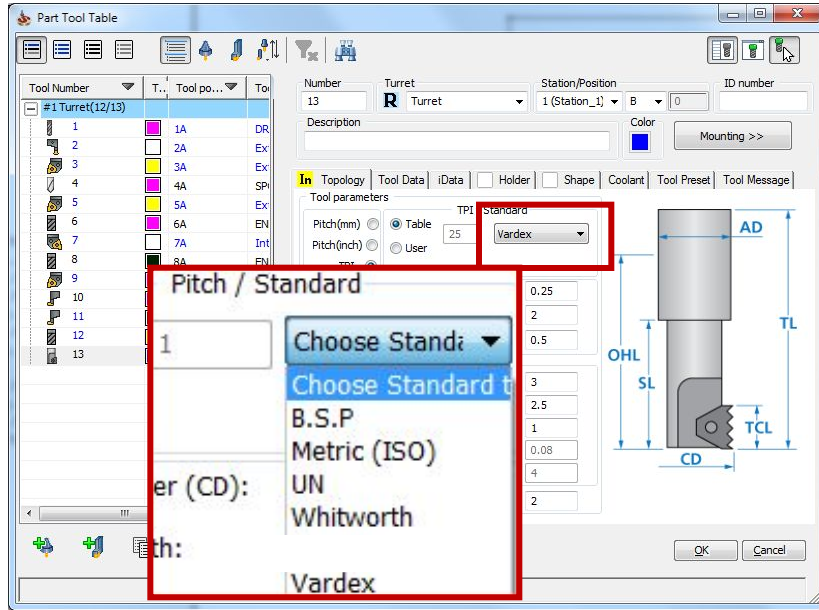
VarDEX



<http://www.vargus.de/>

- Vardex is a Product Line from Vargus for Threading Tools

Integration with Vardex Tool Library



(Vardex software should be installed)

- Integration at the Tool Type Thread Mill in the Table of Pitch / Standard
- Select Vardex from the List - The Vardex TM Assistant starts

Tool Management Softwares Integration

Tool Data Management Softwares

Software for Tool Data Management



TDM Base Module



TDM Global Line



TDM Gauge & Calibration Management



TDM Facility & Maintenance Management



TDM Fixture Management



TDM Multi Plant Management

TDM

TDM



Management of Tools and Manufacturing Data

With process integration.



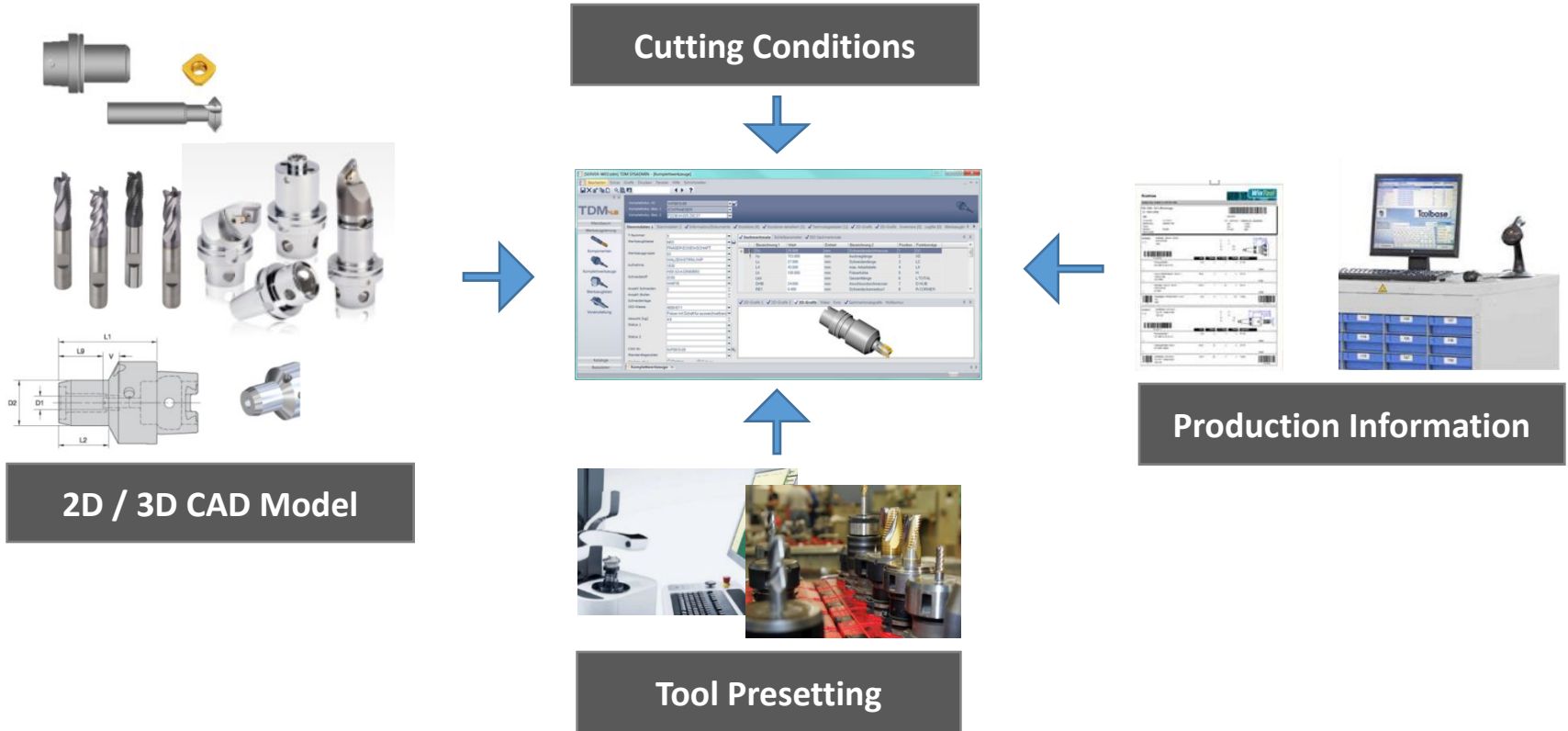
WinTool AG - www.wintool.com

21

WinTool

- Software to manage the overall information of Tools

Tool Data Management (TDM) Software

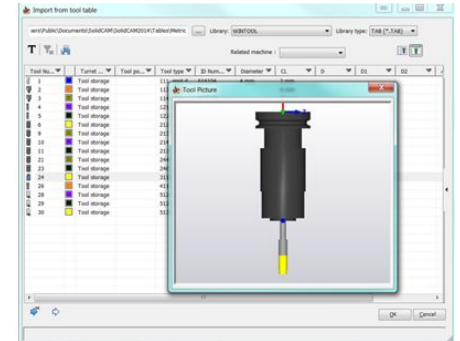
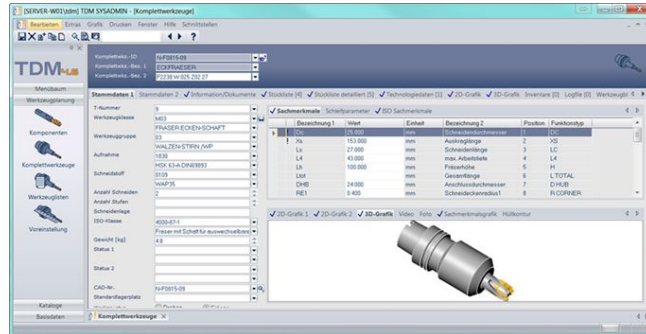


Workflow using TDM Software

Collect Data

Create a Tool with a combination of Data

Use in InventorCAM Import Tool Option



e.g. downloaded 2D/3D CAD Models

Build a Tool as a combination of different CAD Models and Information

Tool with Holder imported from TDM Software

TDM Version 4.6



tdmsystems

Software for Tool Data Management



TDM Base Module



TDM Global Line



TDM Gauge & Calibration Management



TDM Facility & Maintenance Management



TDM Fixture Management



TDM Multi Plant Management

TDM

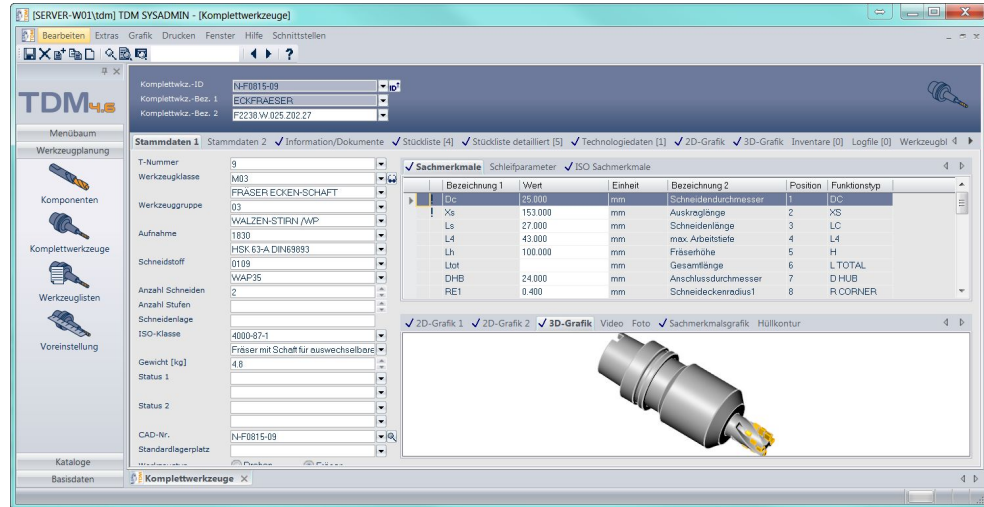
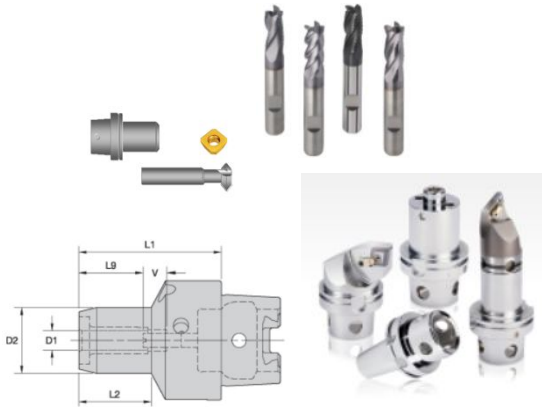
YouTube <http://youtu.be/A3a-iyhMmUY>

Management of Tools and Manufacturing Data in TDM

Collect components

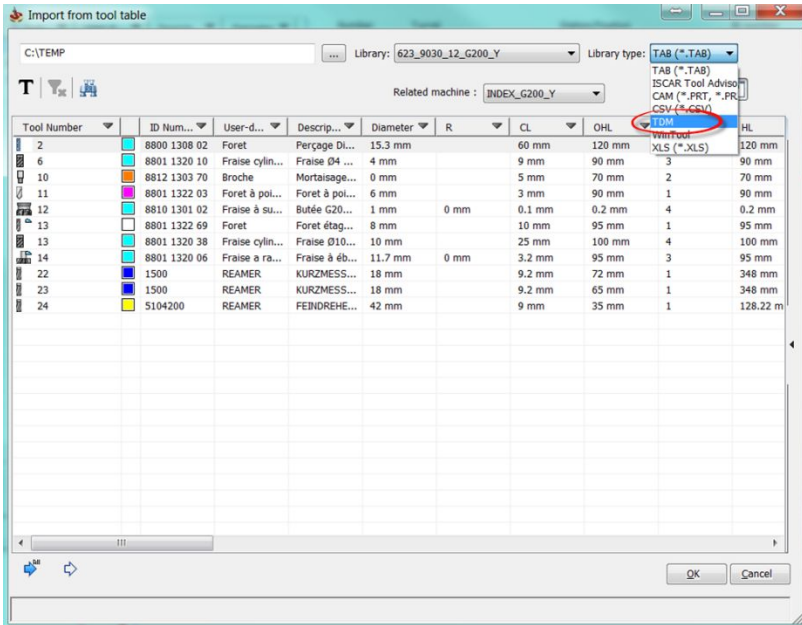
Assemble it

Export to CAM

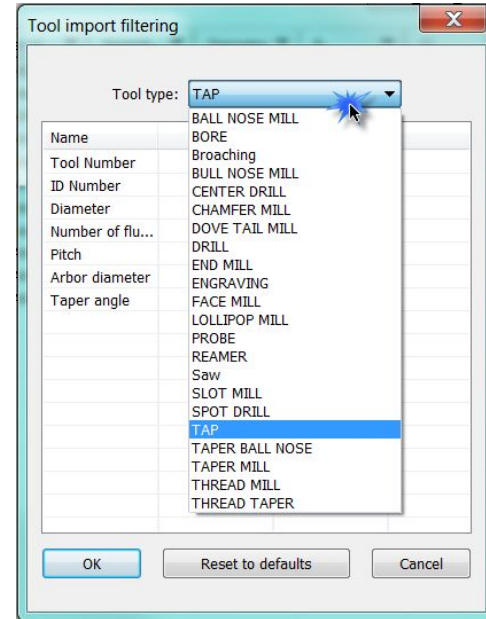


- Users collect the models form online Recourses and build the Assembly of the whole Tool inside TDM
- TDM generates automatically ready to use Data for the InventorCAM Tool Table

Integration with TDM



(TDM should be installed at the Client / Server)



Tool Import filtering dialog

- At the InventorCAM Import Option of Tool table, select TDM – tool import filtering dialog opens



Management of Tools and Manufacturing Data



For more information
click to the Logo

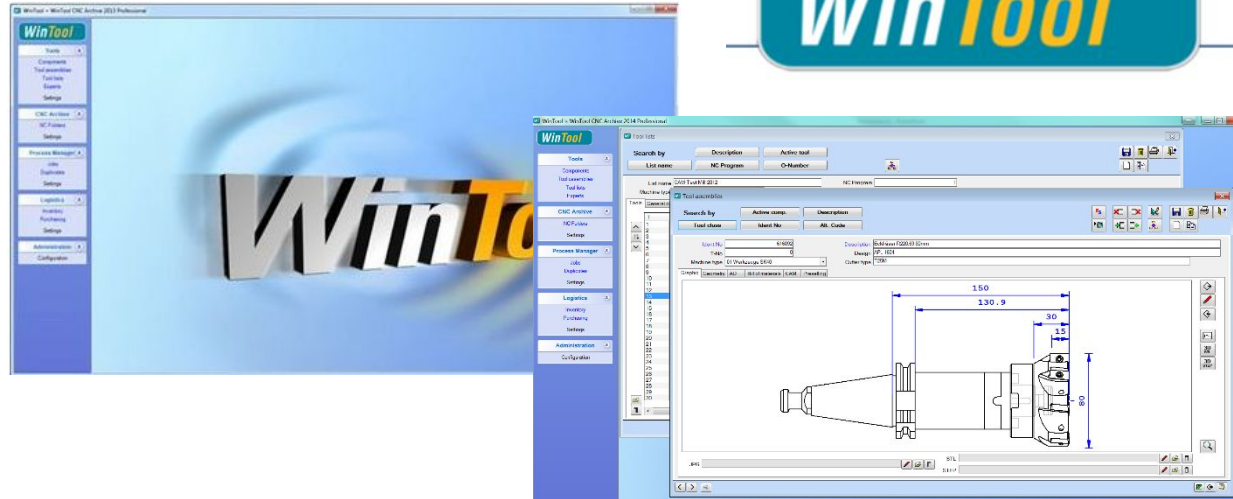
• WinTool - Software for Tool Data Management

Management of Tools and Manufacturing Data

Collect components

Assemble it

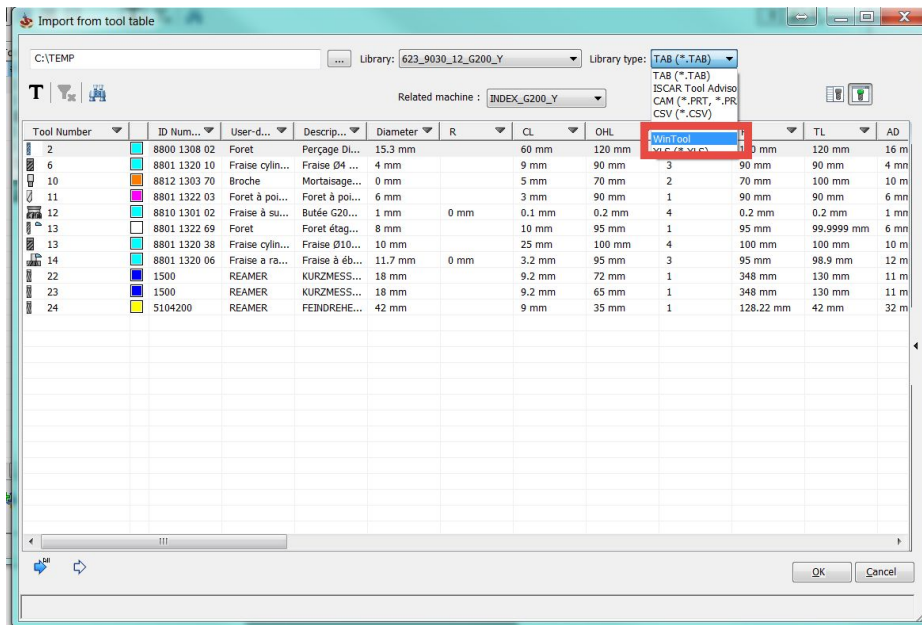
Export to CAM



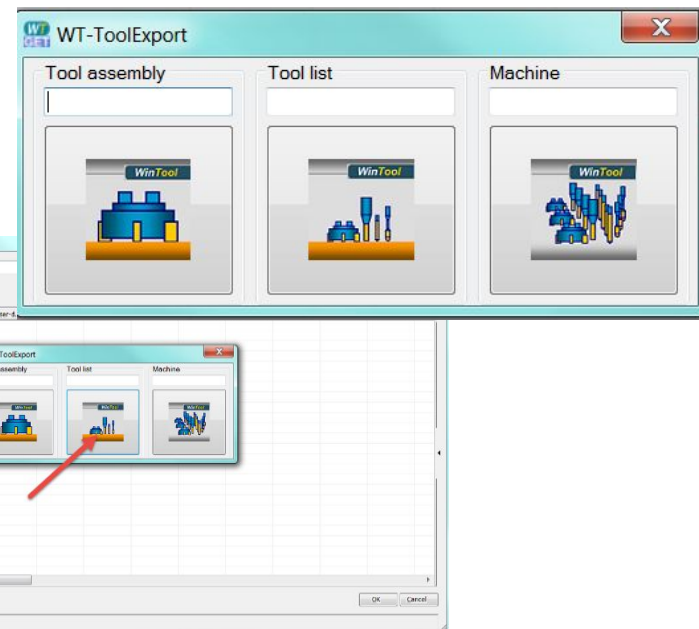
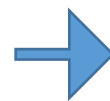
- Users collect the Models form Online Recourses and build the Assembly of the whole Tool inside WinTool
- WinTool generates ready to use Tool Data to import to InventorCAM ToolTable

The vision, revolutionary Milling Technology

Integration with WinTool



(WinTool Professional should be installed and started)



- At the InventorCAM Import Option of the Tool table, select Wintool - Tool Export Dialog opens