InventorCAM 2015

The unique, revolutionary Milling technology

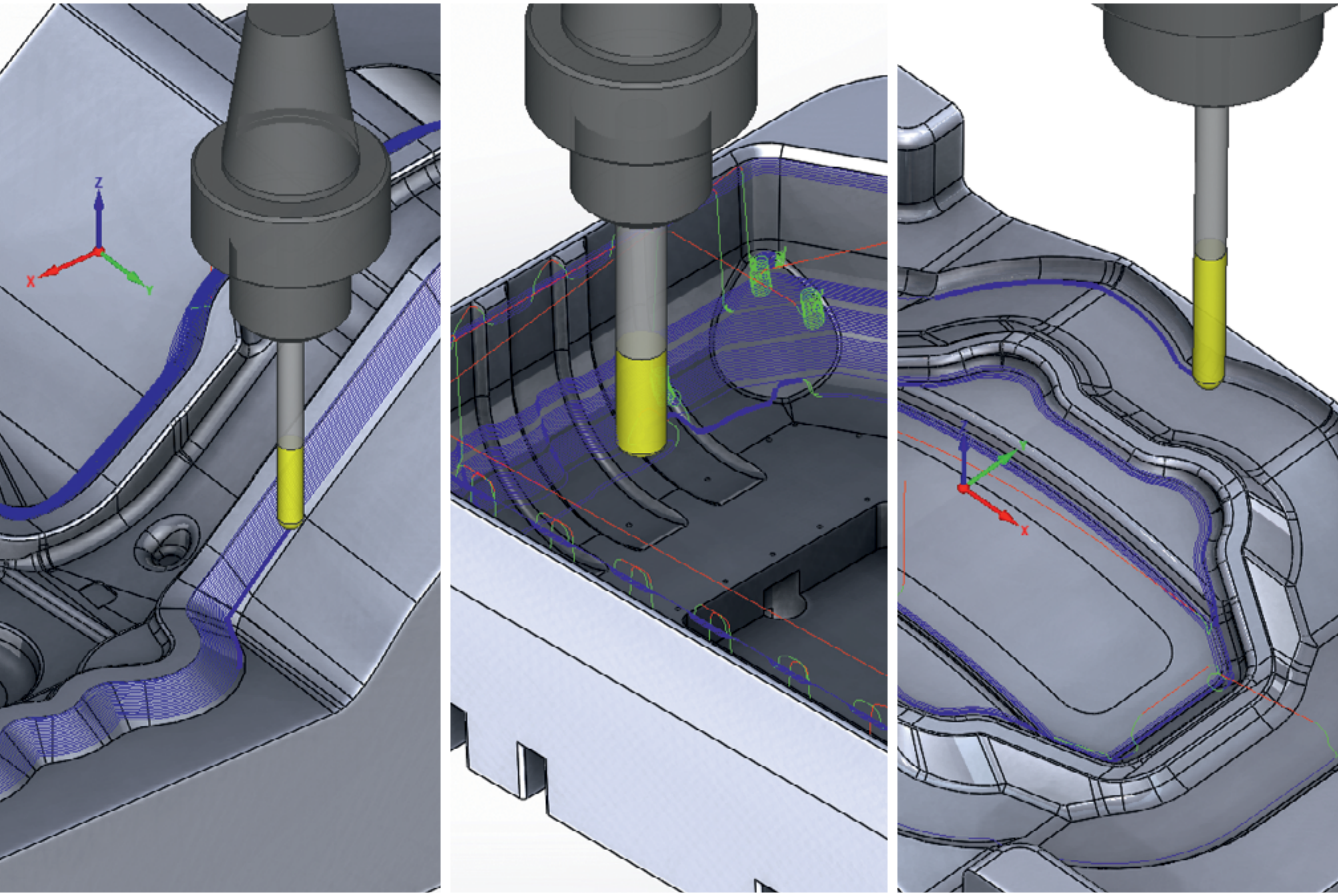
iMachining®

patent by SolidCAM

TIME SAVINGS

70%

... AND MORE!



InventorCAM User Guide:

HSR/HSM

iMachining 2D & 3D | 2.5D Milling | HSS | **HSM** | Indexial Multi-Sided | Simultaneous 5-Axis | Turning & Mill-Turn | Solid Probe

CERTIFIED AUTODESK® INVENTOR® 2015 AUTODESK

InventorCAM + Inventor

The Complete, Integrated Manufacturing Solution

InventorCAM

iMachining – The Revolution in CAM!

InventorCAM
iMachining – The Revolution in CAM!

InventorCAM 2015 HSR/HSM Module User Guide

Contents

Introduction and Basic Concepts

1

Welcome to InventorCAM HSR/HSM!

InventorCAM HSR/HSM is a very powerful and market-proven high-speed machining (HSM) and high-speed roughing (HSR) module for molds, tools and dies and complex 3D parts. It offers unique machining and linking strategies for generating high-speed tool paths.

InventorCAM HSR/HSM module smooths the paths of both cutting moves and retracts wherever possible to maintain a continuous machine tool motion–an essential requirement for maintaining higher feed rates and eliminating dwelling.

With InventorCAM HSR/HSM module, retracts to high Z-levels are kept to a minimum. Angled where possible, smoothed by arcs, retracts do not go any higher than necessary, thus minimizing air cutting and reducing machining time.

The result of HSR/HSM is an efficient, smooth, and gouge-free tool path. This translates to increased surface quality, less wear on your cutters, and a longer life for your machine tools.

With demands for ever-shorter lead and production times, lower costs and improved quality, HSR/HSM is a must in today's machine shops.

About this book

This book is intended for experienced InventorCAM users. If you are not familiar with the software, start with the examples in the **Getting Started** manual and then contact your reseller for information about InventorCAM training classes.

About the Examples

The CAM-Parts used for this book are attached in a ZIP archive. Extract the content of the **Examples** archive into your hard drive. The Autodesk Inventor files used for the exercise were prepared with **Autodesk Inventor 2015**.

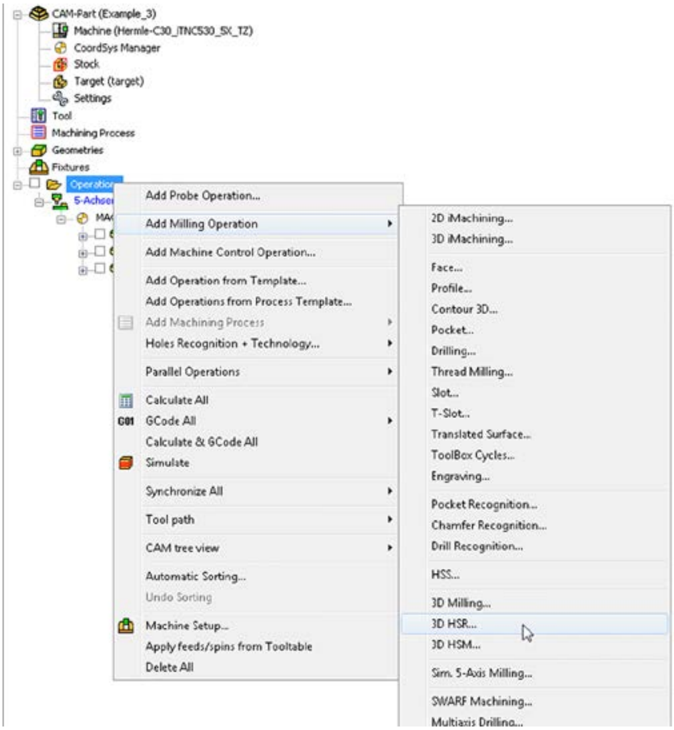
Windows 7

The screenshots in this book were made using **InventorCAM 2015** integrated with Autodesk Inventor 2015 running on Windows 7. If you are running on a different version of Windows, you may notice differences in the appearance of menus and windows. These differences do not affect the performance of the software.

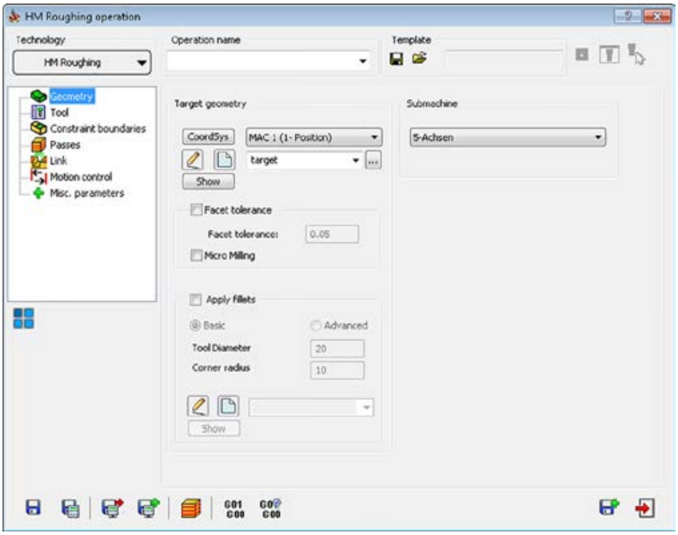
The contents of this book and the examples can be downloaded from the InventorCAM website at **http://www.inventorcam.com**.

1.1 Start HSR/HSM Operation

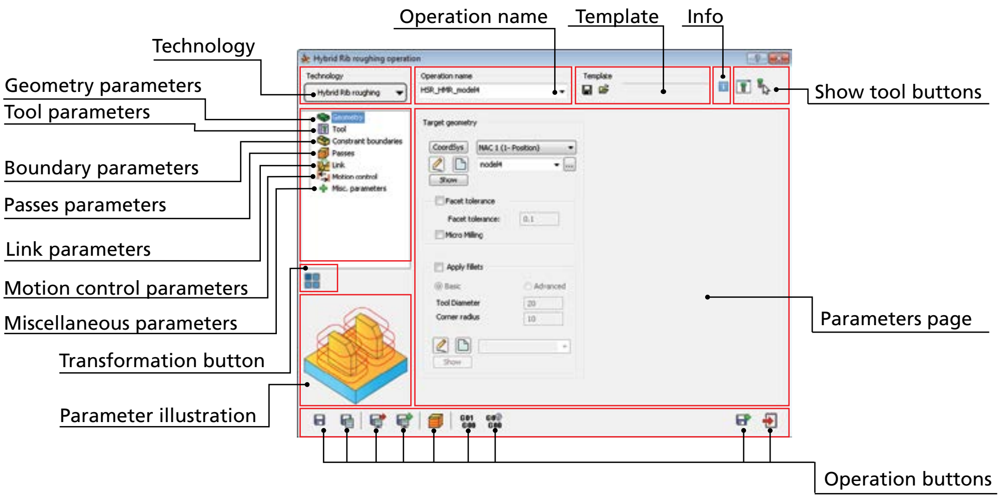
To add an **HSR Operation** or **HSM Operation** to the CAM-Part, right-click the **Operations** header in **InventorCAM Manager** and choose either **3D HSR** or **3D HSM** command from the **Add Milling Operation** submenu.



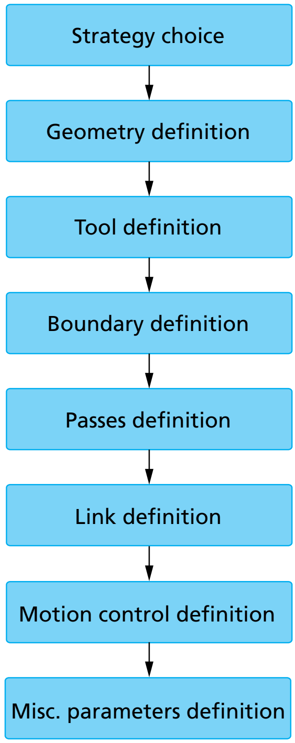
The corresponding operation dialog box is displayed.



1.2 InventorCAM HSR/HSM Operation Overview



The definition of an InventorCAM HSR/HSM operation consists of the following stages:



At the first stage, you have to choose one of the available **machining strategies**. The machining strategy defines the technology that will be used for the machining. For more information on the machining strategies, refer to chapter **2**.

At the **Geometry definition** stage, you have to specify the 3D model geometry that will be machined. For more information on the geometry definition, refer to chapter **3**.

The next stage enables you to choose from the **Part Tool Table** a cutting tool that will be used for the operation. For more information on the tool definition, refer to chapter **4**.

The **Boundaries definition** page enables you to limit the operation machining to the specific model areas. For some machining strategies an additional boundary defines the drive curve of the operation tool path. For more information on the boundary definition, refer to chapter **5**.

In the **Passes definition**, InventorCAM enables you to specify the technological parameters used for the tool passes calculation. For more information on the passes definition, refer to chapter **6**.

The **Link parameters** page enables you to define the tool link moves between cutting passes. For more information on the link definition, refer to chapter **7**.

The **Motion control parameters** page enables you to optimize the calculated tool path according to the kinematics and special characteristics of your CNC-machine. For more information on these parameters definition, refer to chapter **8**.

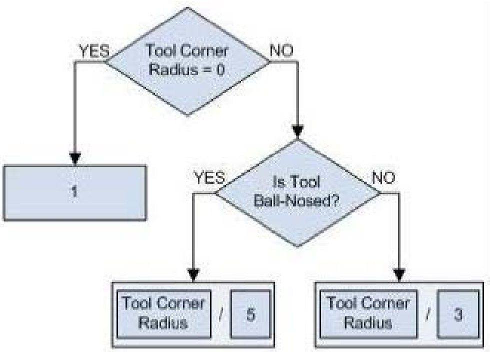
The **Miscellaneous parameters** page enables you to define the non-technological parameters related to the HSR/HSM operation. For more information on the miscellaneous parameters definition, refer to chapter **9**.

1.3 Parameters and Values

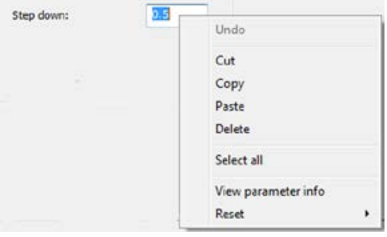
Most of the parameters used in the InventorCAM HSR/HSM operation receive default values according to built-in formulas that define dependencies between the parameters. When a number of basic parameters such a tool diameter, corner radius, offsets, etc., are defined, InventorCAM updates the values of dependent parameters.

For example, the **Step down** parameter for **Contour roughing** is defined with the following formula:

> If the tool corner radius is 0 (end mill), the **Step down** parameter default is set to 1. If a ball-nosed tool is chosen, the **Step down** value is equal to the tool corner radius value divided by 5; for bull-nosed tools the **Step down** value is equal to the tool corner radius value divided by 3.

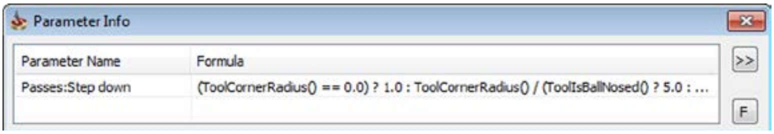


InventorCAM provides you with a right-click edit box menu for each parameter.

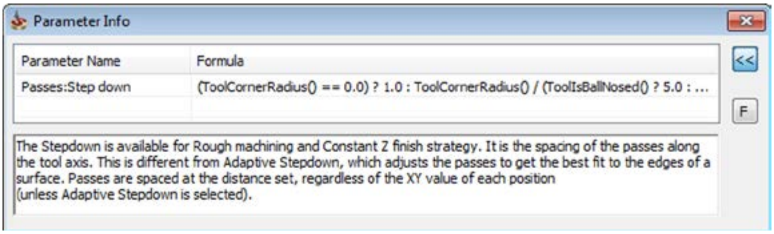


View parameter Info

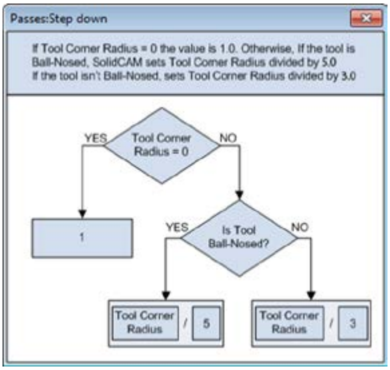
This command displays the **Parameter Info** dialog box. This dialog box shows the internal parameter name and the related formula (if exists) or a static value.



The **Unfold**  button displays a brief explanation of the parameter.



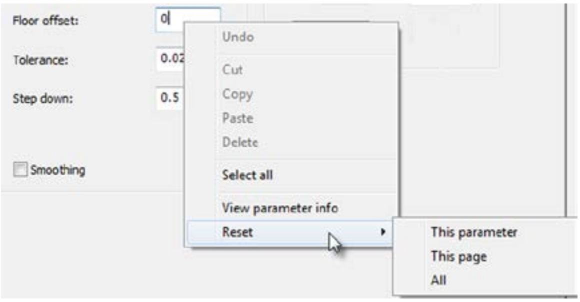
The  button displays the flow chart of the parameter value calculating.



Reset

When you manually change a parameter default value, the formula assigned to the parameter is removed.

The **Reset** commands enable you to reset parameters to their default formulas and values.

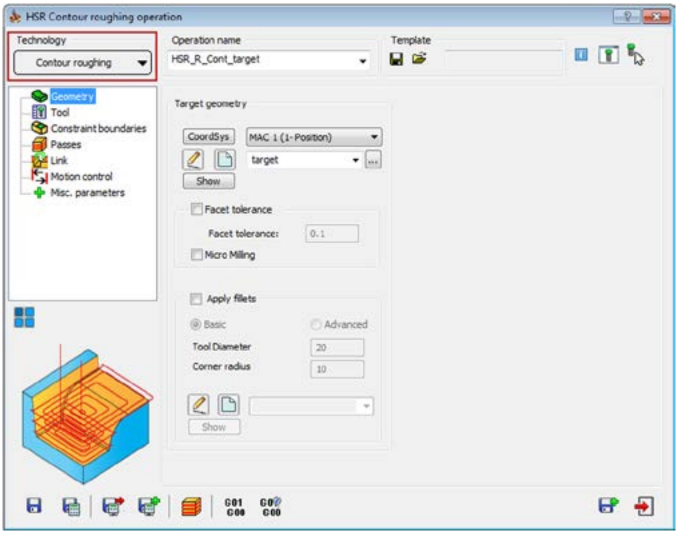


- **This parameter**. This option resets the current parameter.
- **This page**. This option resets all the parameters at the current page.
- **All**. This option resets all the parameters of the current HSR/HSM operation.

Technology

2

The **Technology** section enables you to choose a rough (HSR) or finish (HSM) machining strategy to be applied.



The following strategies are available:

Roughing strategies (HSR):

- HM Roughing
- Contour roughing
- Hatch roughing
- Hybrid Rib roughing
- Rest roughing

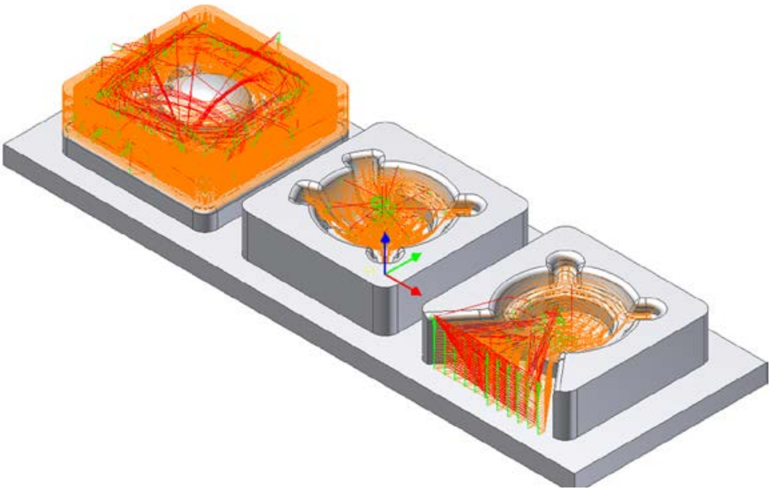
Finishing strategies (HSM):

- Constant Z machining
- Hybrid Constant Z
- Helical machining
- Horizontal machining
- Linear machining
- Radial machining

- Spiral machining
- Morphed machining
- Offset cutting
- Boundary machining
- Rest machining
- 3D Constant step over
- Pencil milling
- Parallel pencil milling
- 3D Corner offset
- Prismatic Part machining
- **Combined strategies:**
    - Constant Z with Horizontal machining
    - Constant Z with Linear machining
    - Constant Z with 3D Constant step over machining
    - Constant Z and 3D Corner offset machining

2.1 HM Roughing

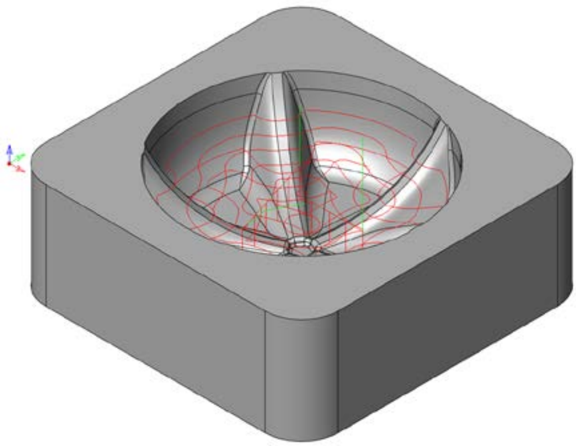
The **HM roughing** strategy massively reduces rapid moments by controlling the tool movement so that it remains on the part, following previous cut paths instead of rapid feeding to the new position. Additional tool path optimization features include the ability to automatically machine flat areas only instead of adding extra Z-levels to clean up planar zones. The roughing algorithm also permits to the use of a large step over (greater than 50%), where an offset algorithm ensures total coverage of the machining area by adding smooth transition corner pips to clean up any remaining areas.

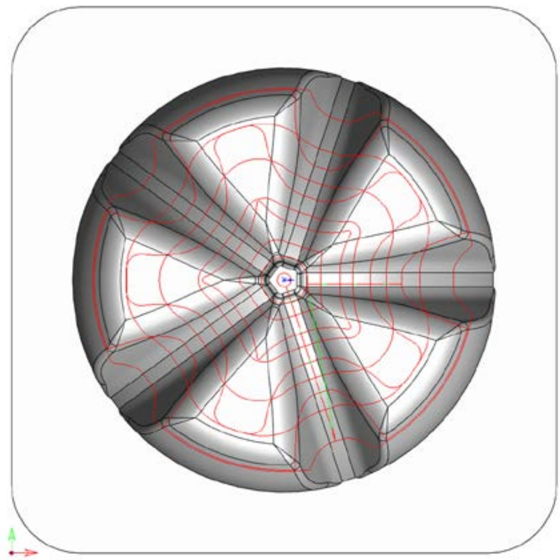


The strategy can be easily modified to work in different modes: cavity, core and hybrid spiral passes. These modes provide different approaches to machine a part.

2.2 Contour Roughing

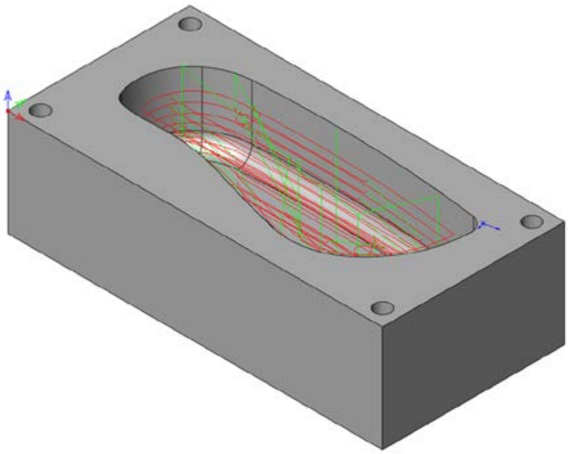
With the **Contour roughing** strategy, InventorCAM generates a pocket-style tool path for a set of sections generated at the Z-levels defined with the specified **Step down** (see topic **6.1.4**).

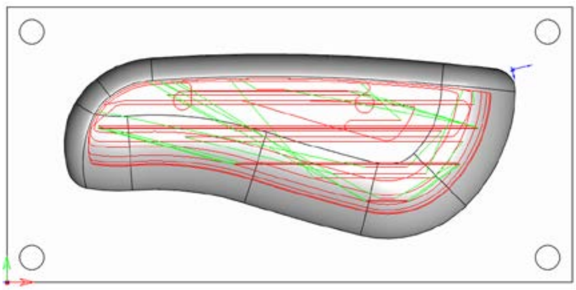




2.3 Hatch Roughing

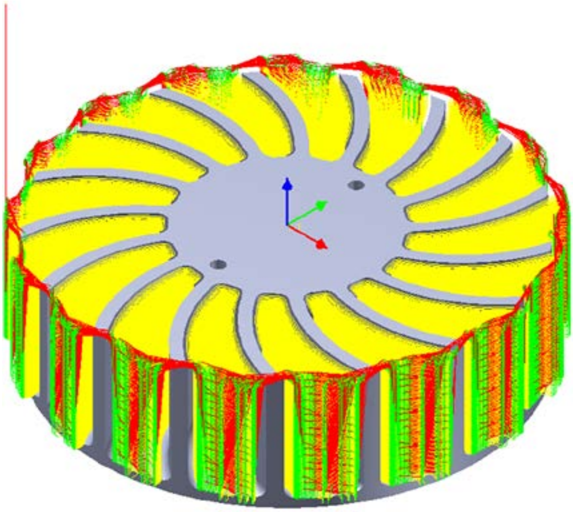
With the **Hatch roughing** strategy, InventorCAM generates linear raster passes for a set of sections generated at the Z-levels defined with the specified **Step down** (see topic **6.1.4**). Hatch roughing is generally used for older machine tools or softer materials because the tool path predominantly consists of straight line sections.

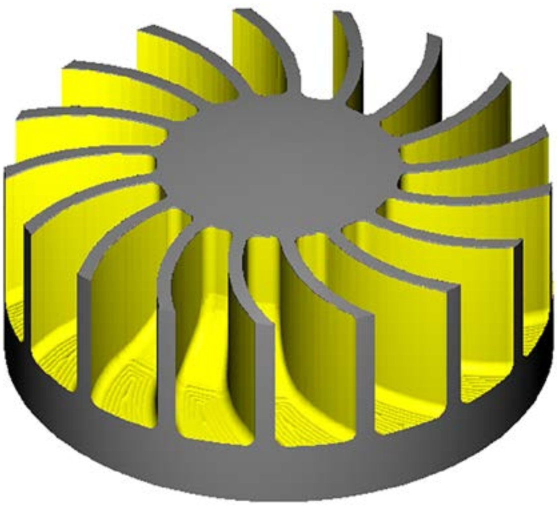




2.4 Hybrid Rib Roughing

The **Hybrid Rib** roughing is a strategy designed to machine very thin walls. These walls are made of exotic materials (titanium, graphite) and therefore a traditional approach to their machining can be difficult and risky. This strategy combines a new roughing and finishing tool path, creating a unique tool path that should preserve the highest possible rigidity of the part.

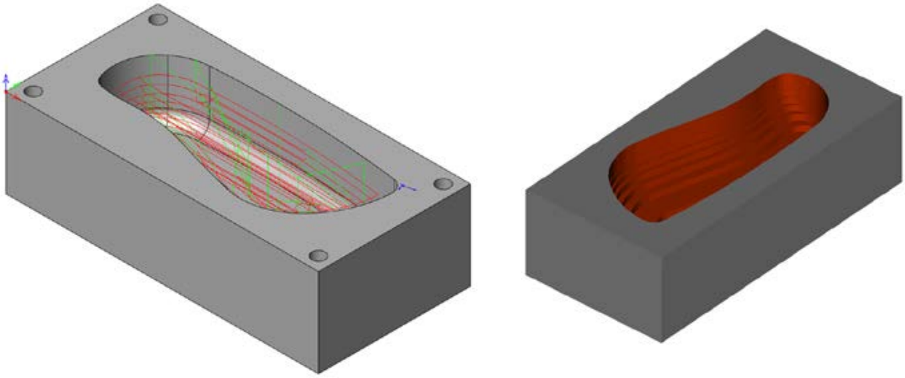




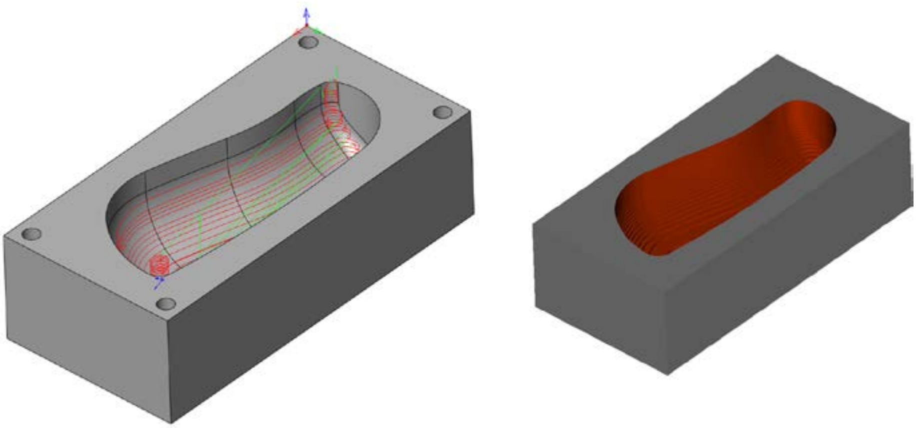
2.5 Rest Roughing

The **Rest roughing** strategy determines the areas where material remains unmachined after the previous machining operations (the "rest" of the material) and generates a tool path for the machining of these areas. The tool path is generated in the **Contour roughing** (see topic **2.2**) manner. Rest roughing operation uses a tool of smaller diameter than that used in previous roughing operations.

The following image illustrates the hatch roughing tool path performed with an **End mill** of **Ø20**.

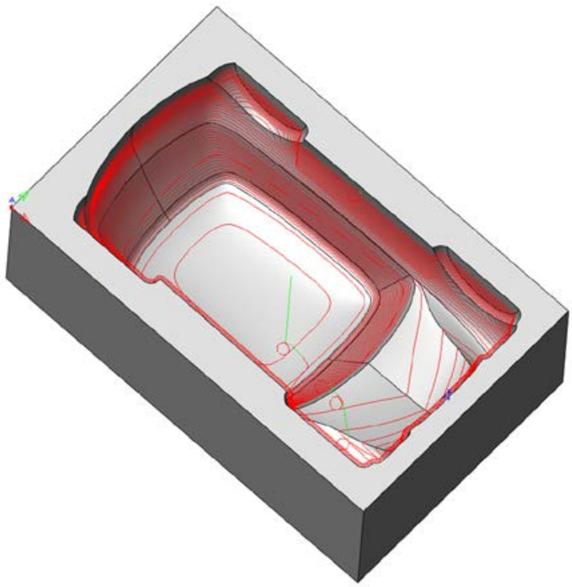


After the hatch roughing, a Rest roughing operation is performed with an **End mill** of **Ø10**. The tool path is generated in the contour roughing manner.



2.6 Constant Z Machining

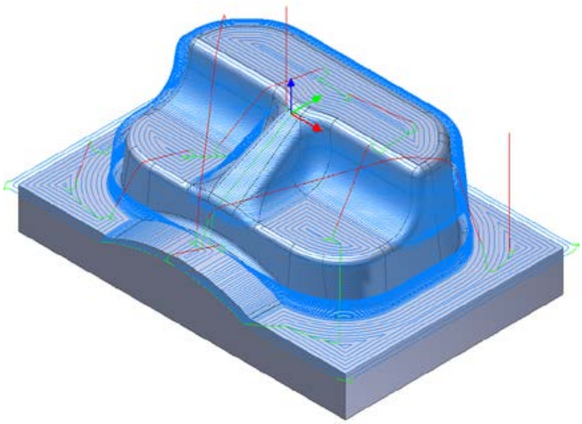
Similar to **Contour roughing**, the Constant Z tool path is generated for a set of sections created at different Z-heights determined by the **Step down** (see topic **6.1.4**) parameter. The generated sections are machined in a profile manner. The **Constant Z** strategy is generally used for semi-finishing and finishing of steep model areas with the inclination angle between 30 and 90 degrees. Since the distance between passes is measured along the Z-axis of the Coordinate System, in shallow areas (with smaller surface inclination angle) the **Constant Z** strategy is less effective.



The image above illustrates the **Constant Z** finishing. Note that the passes are densely spaced in steep areas. Where the model faces get shallower, the passes become widely spaced, resulting in ineffective machining. Therefore, the machining should be limited by the surface inclination angle to avoid the shallow areas machining. These areas can be machined later with a different InventorCAM HSM strategy, e.g. **3D Constant step over** (see topic **2.17**).

2.7 Hybrid Constant Z

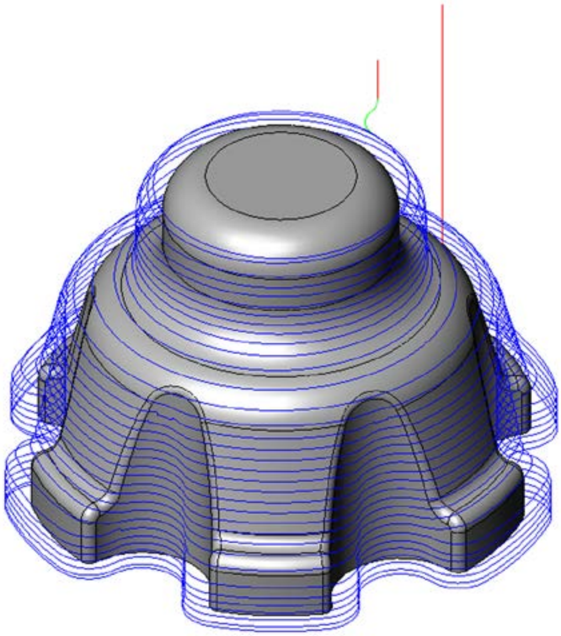
The **Hybrid Constant Z** is a finishing strategy that combines all the benefits of a traditional **Constant Z** operation with a 3D pocketing routine that interacts with the main strategy whenever the shallow area between consecutive passes allows the insertion of additional passes. This method allows an optimal finish over the entire part.



2.8 Helical Machining

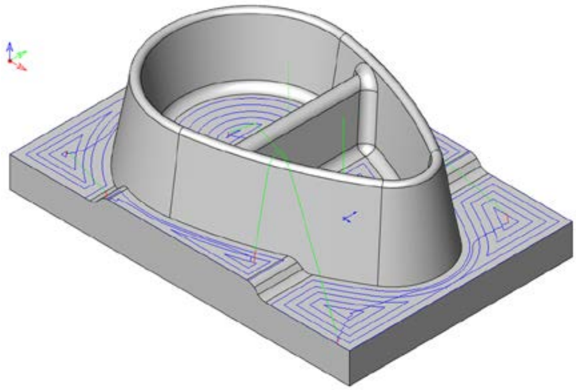
With this strategy, InventorCAM generates a number of closed profile sections of the 3D Model geometry located at different Z-levels, similar to the **Constant Z** strategy. Then these sections are joined in a continuous descending ramp in order to generate the **Helical machining** tool path.

The tool path generated with the **Helical machining** strategy is controlled by two main parameters: **Step down** and **Max. ramp angle** (see topic **6.6.5**).



2.9 Horizontal Machining

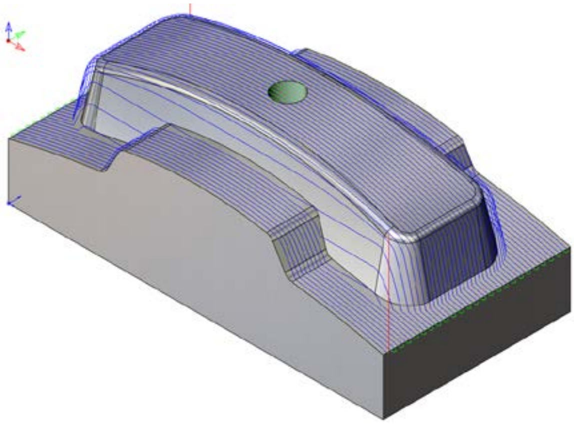
With the **Horizontal machining** strategy, InventorCAM recognizes all the flat areas in the model and generates a tool path for machining these areas.



This strategy generates a pocket-style (a number of equidistant profiles) tool path directly at the determined horizontal faces (parallel to the XY-plane of the current Coordinate System). The distance between each two adjacent passes is determined by the **Offset** (see topic **6.1.7**) parameters.

2.10 Linear Machining

Linear machining generates a tool path consisting of a set of parallel passes at a set angle with the distance between the passes defined by the **Step over** (see topic **6.1.5**) parameter.

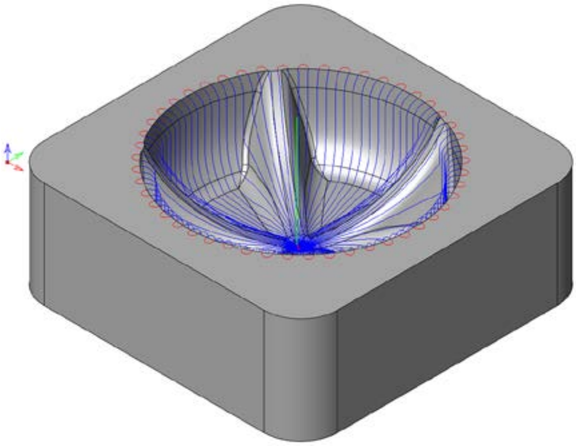


With the **Linear machining** strategy, InventorCAM generates a linear pattern of passes, where each pass is oriented at a direction defined with the **Angle** value. This machining strategy is most effective on shallow (nearing horizontal) surfaces, or steeper surfaces inclined along the passes direction. The Z-height of each point along a raster pass is the same as the Z-height of the triangulated surfaces, with adjustments made for applied offsets and tool definition.

In the image above, the passes are oriented along the X-axis. The passes are evenly spaced on the shallow faces and on the faces inclined along the passes direction. The passes on the side faces are widely spaced; **Cross Linear machining** (see topic **6.6.4**) can be used to finish these areas.

2.11 Radial Machining

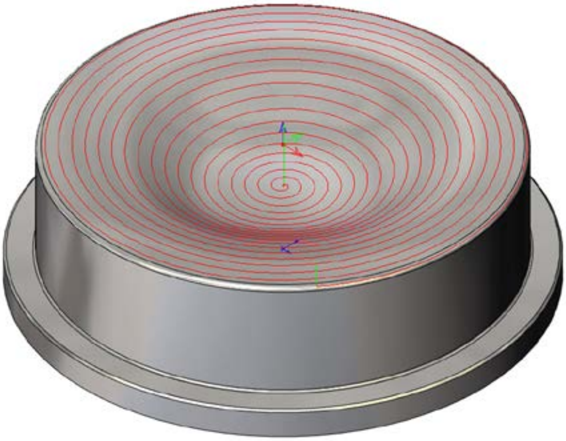
The **Radial machining** strategy enables you to generate a radial pattern of passes rotated around a central point.

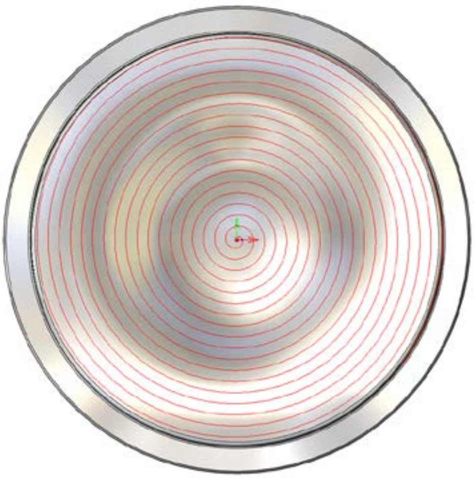


This machining strategy is most effective on areas that include shallow curved surfaces and for model areas formed by revolution bodies, as the passes are spaced along the XY-plane (step over), and not the Z-plane (step down). The Z-height of each point along a radial pass is the same as the Z-height of the triangulated surfaces, with adjustments made for applied offsets and tool definition.

2.12 Spiral Machining

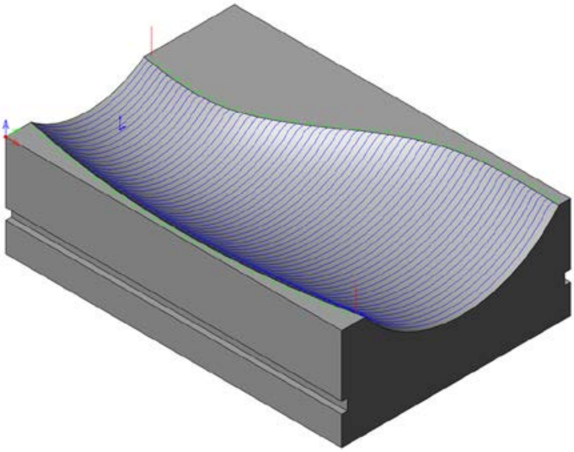
The **Spiral machining** strategy enables you to generate a 3D spiral tool path over your model. This strategy is optimal for model areas formed by revolution bodies. The tool path is generated by projecting a planar spiral (located in the XY-plane of the current Coordinate System) on the model.



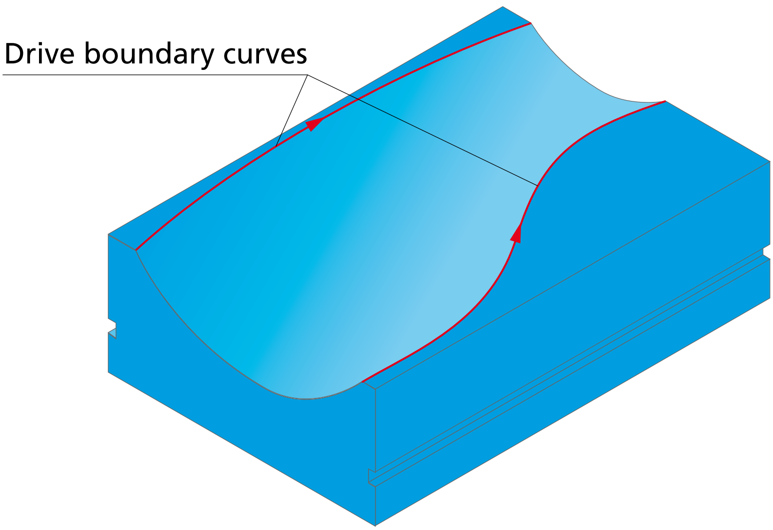


2.13 Morphed Machining

Morphed machining passes are generated across the model faces in a close-to-parallel formation, rather like **Linear machining** passes (see topic **2.10**); each path repeats the shape of the previous one and takes on some characteristics of the next one, and so the paths "morph" or gradually change shape from one side of the patch to the other.

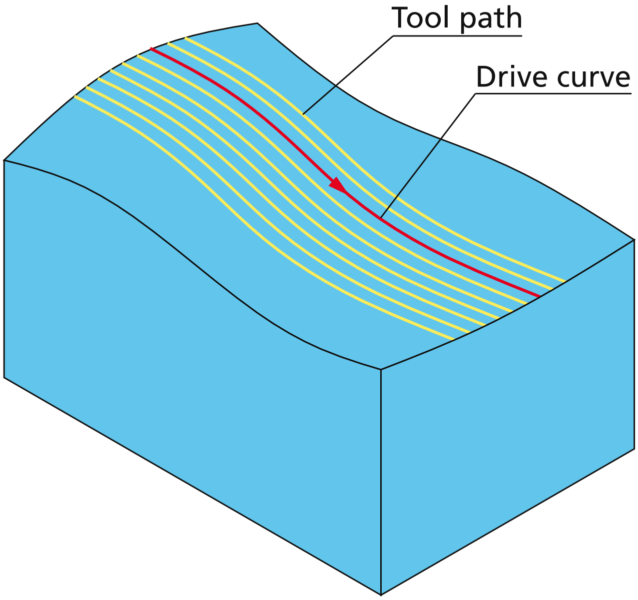


The shape and direction of the patch is defined by two drive boundary curves.



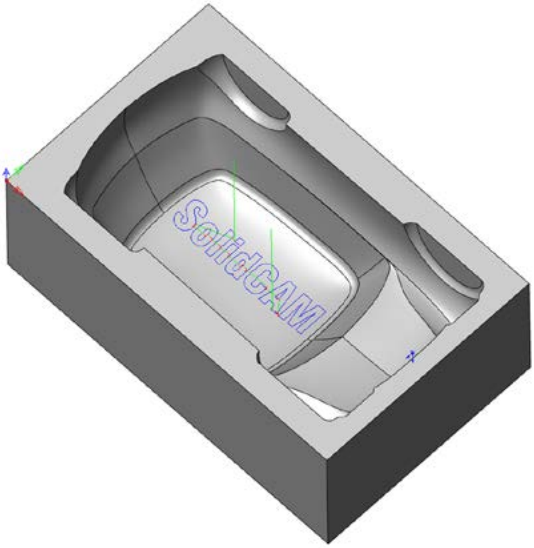
2.14 Offset Cutting

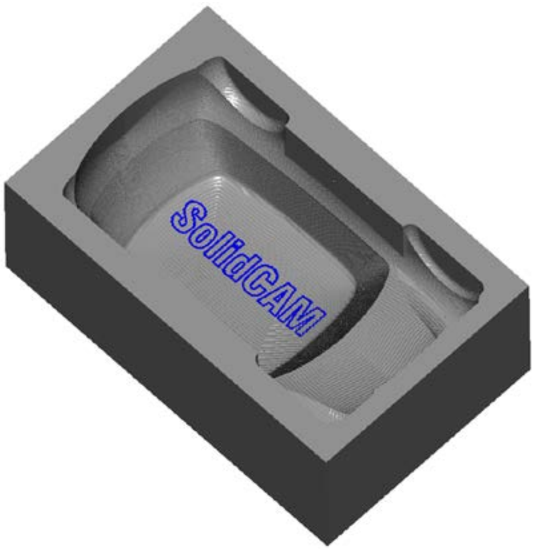
This strategy is a particular case of the **Morphed machining** strategy (see topic **2.13**). The **Offset cutting** strategy enables you to generate a tool path using a single Drive curve. The tool path is generated between the Drive curve and a virtual offset curve, generated at the specified offset from the Drive curve.



2.15 Boundary Machining

A **Boundary machining** strategy enables you to create the tool path by projecting the defined **Drive boundary** (see topic **5.1.1**) on the model geometry. The Machining depth is defined relative to the model surfaces with the **Wall offset** (see topic **6.1.1**) parameter. The tool path generated with the **Boundary machining** strategy can be used for engraving on model faces or for chamfer machining along the model edges.

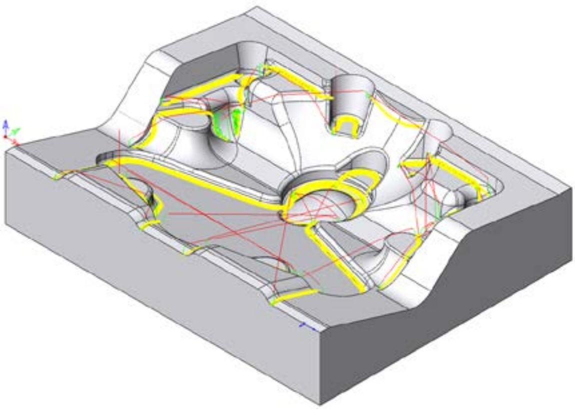




2.16 Rest Machining

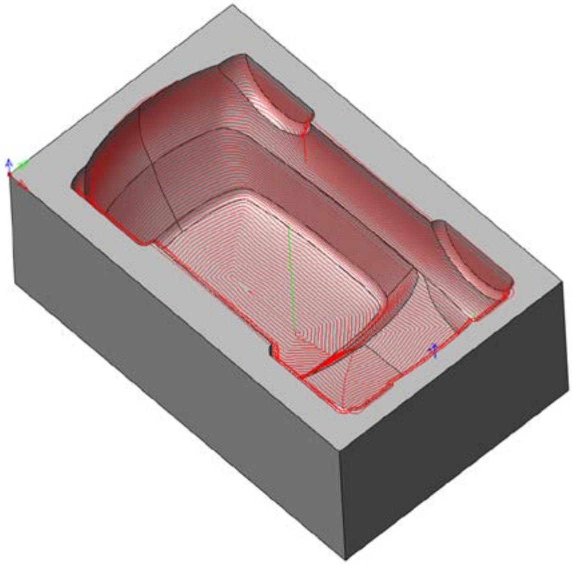
Rest machining determines the model areas where material remains after the machining by a tool path, and generates a set of passes to machine these areas.

Pencil milling vertical corners can cause both the flute of the tool and the radius to be in full contact with the material, creating adverse cutting conditions. **Rest machining** picks the corners out from the top down, resulting in better machining technique. Steep and shallow areas are both machined in a single tool path, with different rest machining strategies.



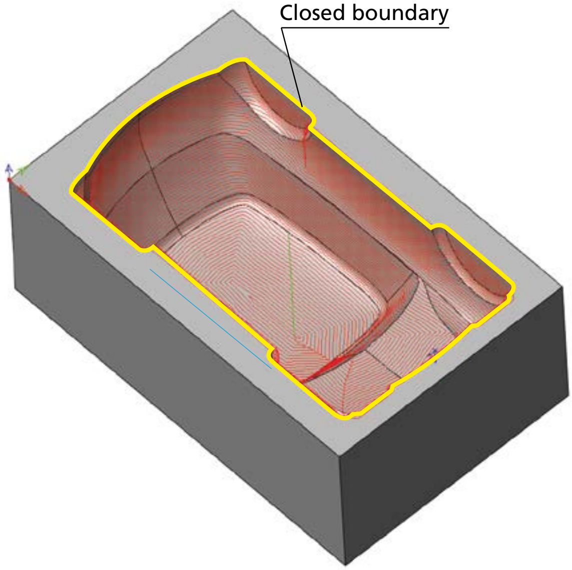
2.17 3D Constant Step Over Machining

3D Constant step over machining enables you generate a 3D tool path on the CAM-Part surfaces. The passes of the tool path are located at a constant distance from each other, measured along the surface of the model.



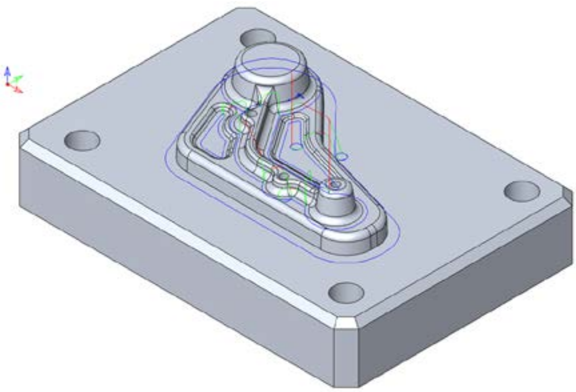
This is an ideal strategy to use on the boundaries generated by rest machining or in any case where you want to ensure a constant distance between passes along the model faces.

Constant surface step over is performed on a closed profile of the **Drive boundary** (see topic **5.1.1**). InventorCAM creates inward offsets from this boundary.



2.18 Pencil Milling

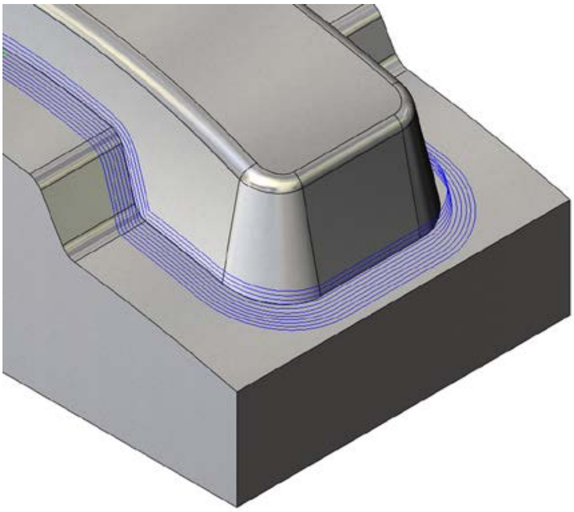
The **Pencil milling** strategy creates a tool path along internal corners and fillets with small radii, removing material that was not reached in previous machining. This strategy is used to finish corners which might otherwise have cusp marks left from previous machining operations. This strategy is useful for machining corners where the fillet radius is equal to or smaller than the tool radius.



2.19 Parallel Pencil Milling

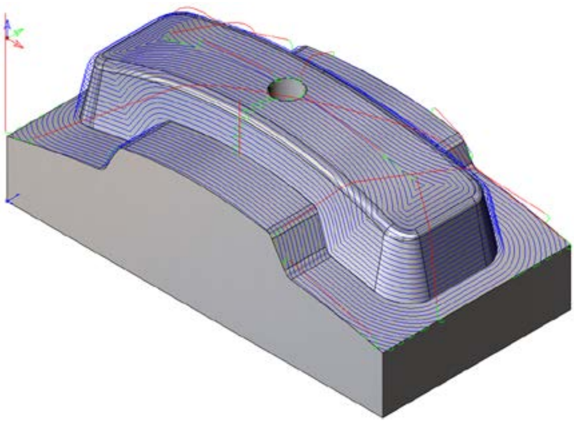
Parallel pencil milling is a combination of the **Pencil milling** strategy and the **3D Constant step over** strategy. At the first stage, InventorCAM generates a **Pencil milling** tool path. Then the generated pencil milling passes are used to create **3D Constant step over** passes; the passes are generated as a number of offsets on both sides of the pencil milling passes. In other words, the **Parallel pencil milling** strategy performs **3D Constant step over** machining using **Pencil milling** passes as drive curves to define the shape of passes.

This strategy is particularly useful when the previous cutting tool was not able to machine all the internal corner radii to size. The multiple passes generated by this strategy will machine from the outside in to the corner, creating a good surface finish.



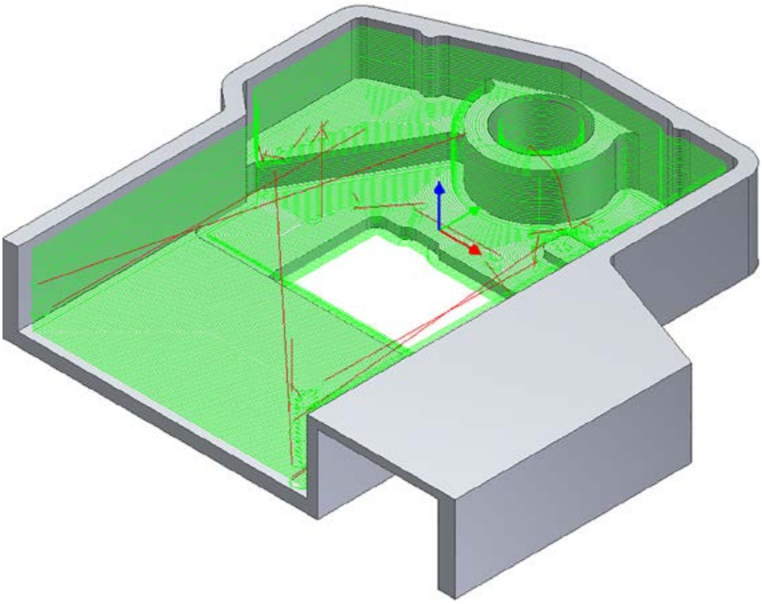
2.20 3D Corner Offset

The **3D Corner offset** strategy is similar to the **Parallel pencil milling** strategy. This strategy is also a combination of the **Pencil milling** strategy and the **3D Constant step over** strategy. InventorCAM generates a **Pencil milling** tool path and uses it for the **3D Constant step over** passes generation. These passes are generated as offsets from the **Pencil milling** passes. In contrast to the **Parallel pencil milling** strategy, the number of offsets is not defined by user but determined automatically in such a way that all the model within the boundary will be machined.



2.21 Prismatic Part Machining

The **Prismatic Part machining** strategy is designed especially for high-speed finishing of prismatic parts. This strategy comprises the technology of the **Constant Z** (see topic **2.6**) and **3D Constant step over** (see topic **2.17**) strategies by integrating these two strategies into one smart functionality of prismatic part machining. The difference from the **Combined Constant Z** with **3D Constant step over** strategy (see topic **2.22**) is as follows: in the **Combined** strategy, the subsequent strategies are performed successively one after the other. In the **Prismatic Part machining** strategy, the machining is performed consistently according to the order of the walls and flat faces along the Z-axis.

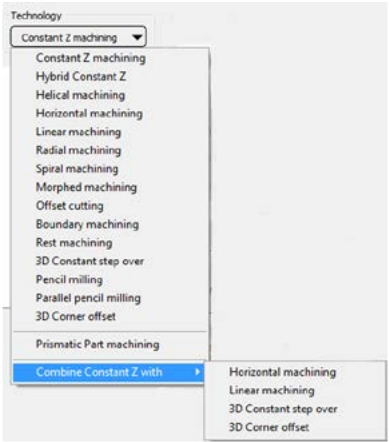


The user-defined geometry parameters are taken into account by the system for calculation of default values for technological parameters to provide you with the optimized machining solution. For example, the minimal and maximal Z-levels of the defined geometry are used for calculation of the surface inclination angle, and so on.

The tool choice also affects the automatic calculation of defaults for technological parameters.

2.22 Combined Strategies

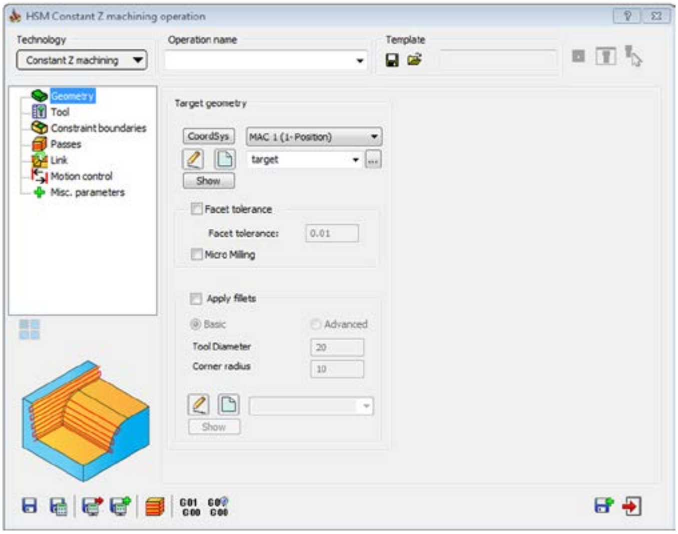
InventorCAM enables you to combine two machining strategies in a single HSM operation: **Constant Z** with **Horizontal**, **Linear**, **3D Constant step over** or **3D Corner Offset** machining. Two combined machining strategies share the **Geometry**, **Tool** and **Constraint boundaries** data. The technological parameters for the passes calculation and linking are defined separately for each strategy.



Geometry

3

The **Geometry** page enables you to define the 3D model geometry for the InventorCAM HSR/HSM operation.



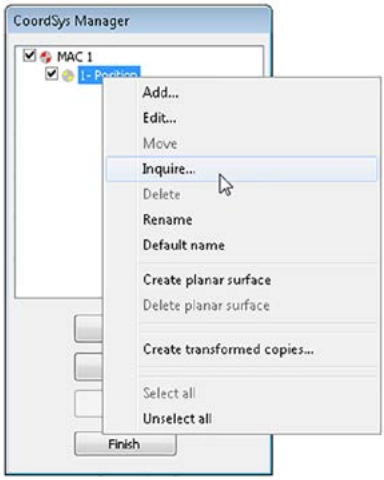
3.1 Geometry Definition

The **Target geometry** section enables you to specify the appropriate Coordinate System for the operation and to define the machining geometry.

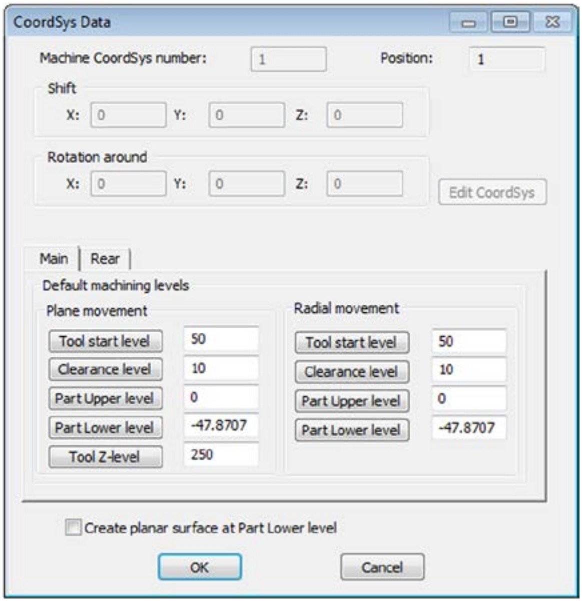
3.1.1 CoordSys

InventorCAM enables you to define the Coordinate System for the operation by choosing it from combo-box or by selecting it from the graphic screen by clicking the **CoordSys** button. The **CoordSys Manager** dialog box is displayed. Together with this dialog box, InventorCAM displays the location and axis orientation of all Coordinate Systems defined in the CAM-Part.

To get more information about the Coordinate System, right-click the CoordSys entry in **CoordSys Manager** and choose the **Inquire** option from the menu.



The **CoordSys Data** dialog box is displayed.

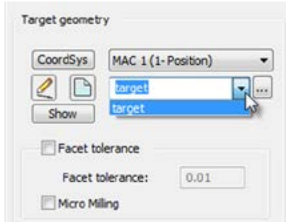


When the CoordSys is chosen for the operation, the model is rotated to the appropriate orientation.

The CoordSys selection operation must be the first step in the geometry definition process.

3.1.2 Target geometry

After the Coordinate System is chosen, define the 3D Model geometry for the InventorCAM HSR/HSM operation.



If you have already defined 3D Model geometries for this CAM-Part, you can select a geometry from the list.

The **Show** button displays the chosen 3D model geometry in the Autodesk Inventor window.

The **New** button enables you to define a new 3D Model geometry for the operation with the **3D Model Geometry** dialog box. The **Edit** button enables you to edit an existing geometry.

The **Browse** button enables you to view the available geometries on the model and choose the relevant one from the list.

For more information on 3D Geometry selection, refer to the **InventorCAM Milling Online Help**.

When you choose the geometry from the list, the related Coordinate System is chosen automatically.

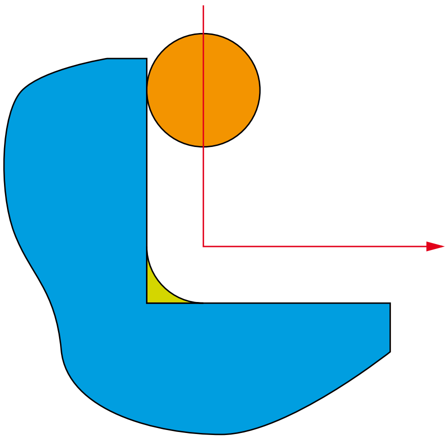
3.1.3 Facet tolerance

Before the machining, InventorCAM generates a triangular mesh for all the faces of the 3D model geometry used for the operation. **Facet tolerance** is the accuracy to which triangles fit the surfaces. The smaller the value the more accurate the triangulation is, but the slower the calculation.

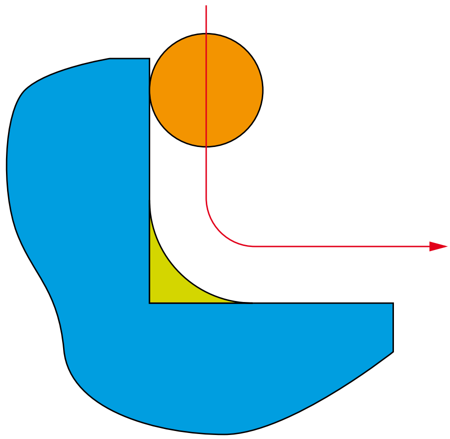
The 3D model geometry will be triangulated and the resulting facets will be saved. The triangulation is performed on the 3D model geometry when you use it for the first time in a InventorCAM HSR/HSM operation. If you use the 3D geometry in another operation, InventorCAM will check the tolerance of the existing geometry. It will not perform another triangulation as long as the facets have been created with the same surface tolerance.

3.1.4 Apply fillets

This option automatically adds fillets to the internal model corners. Therefore, the tool does not have to dramatically change direction during the machining, preventing damage to itself and to the model surfaces and enabling faster feed rates and eventually better surface quality.

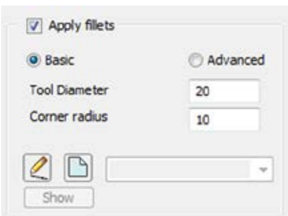


When the corner radius is smaller than or equal to the tool radius, the tool path consists of two lines connected with a sharp corner; at this corner point the tool sharply changes its direction.



By adding fillets, the corner radius becomes greater than the tool radius and the tool path lines are then connected with an arc, resulting in a smooth tool movement without sharp changes in direction.

Select the **Apply fillets** check box to automatically add fillets for the tool path generation.



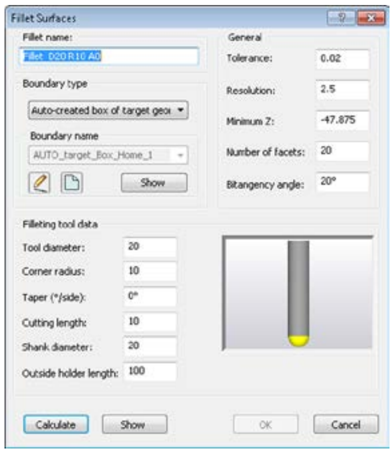
Click  to create a new fillets geometry. The **Fillet Surfaces** dialog box is displayed.

The **Show** button displays the chosen fillet geometry directly on the solid model.

Model without fillets

Model with fillets

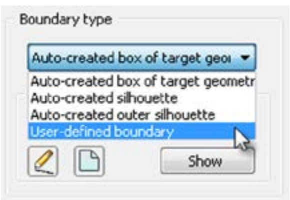
3.1.5 Fillet Surfaces dialog box



The **Fillet Surfaces** dialog box enables you to generate fillet geometry for the current 3D Model geometry used for the HSM operation.

Boundary

The **Boundary type** section enables you to specify the boundary geometry for the fillet generation. The fillets will be generated inside the specified 2D boundary.



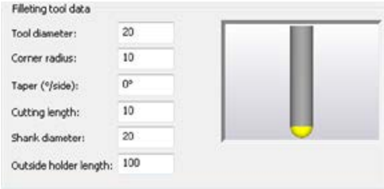
InventorCAM enables you to choose the 2D boundary type from the list. 2D boundaries of the following types are available: **Auto-created silhouette** (see topic **5.3.3**), **Auto-created outer silhouette** (see topic **5.3.4**), **User-defined boundary** (see topic **5.4.3**), and **Auto-created box of target geometry** option. The latter option automatically generates a planar box surrounding the Target geometry.

The **Boundary name** section enables you to choose a 2D boundary geometry from the list or define a new one using the **New** button. The appropriate dialog box will be displayed. You can edit an existing geometry with the **Edit** button.

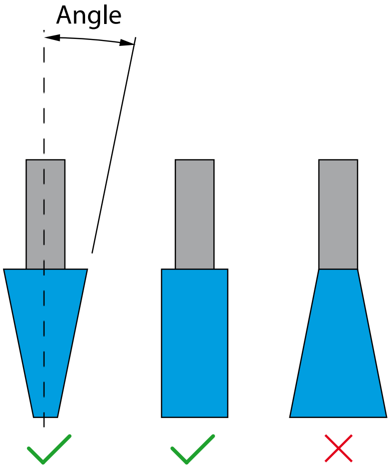
The **Show** button displays the **Select Chain** dialog box, and the chains are displayed and highlighted in the graphic window. If needed, you can unselect some of the automatically created chains.

Filleting Tool Data

For the calculation of fillets, InventorCAM uses a virtual tool. The **Filleting tool data** section enables you to specify the geometry parameters of this tool.

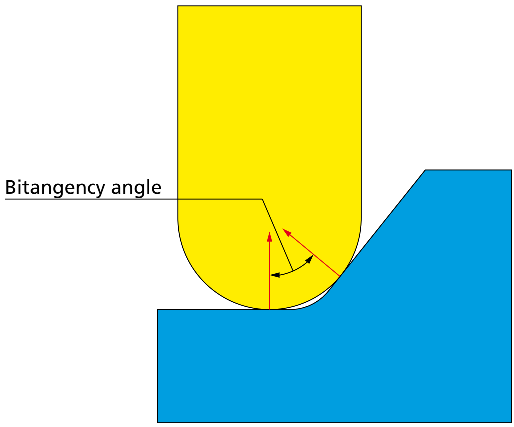


- **Tool diameter**. This field enables you to specify the cutting diameter of the virtual tool.
- **Corner radius**. This field enables you to specify the corner radius of the virtual tool.
- **Taper (°/side)**. This field enables you to specify the taper angle of the tool's flank. InventorCAM does not support tool with a back taper, like a **Dove tail** tool.
- **Cutting length**. This field enables you to specify the length of the cutting edge of the tool.
- **Shank diameter**. This field enables you to specify the shank diameter.
- **Outside holder length**. This field enables you to specify the length of the visible part of the tool, from the tip to the start of the tool holder.



General

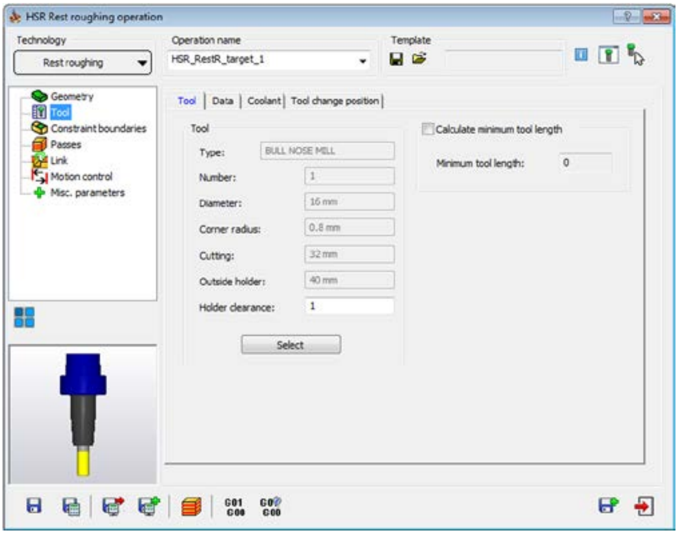
- **Tolerance**. This parameter defines the tolerance of fillet surfaces triangulation. A lower value will give more accurate results, but will increase the calculation time.
- **Resolution**. This is the "granularity" of the calculation. Using a smaller value will give finer detail but will increase the calculation time.
- **Minimum Z**. This option sets the lowest Z-level the tool can reach.
- **Number of facets**. This is the number of flat faces (triangles) across the radially curved section of the fillet.
- **Bitangency angle**. This is the minimum angle required between the two normals at the contact points between the tool and model faces, in order to decide to generate the fillet.



Tool

4

In the **Tool** section of the InventorCAM HSR/HSM operation dialog box, the following tool parameters are displayed:



- **Type**
- **Number**
- **Diameter**
- **Corner radius**
- **Cutting length**
- **Outside holder**
- **Holder clearance**

4.1 Calculate Minimum Tool Length

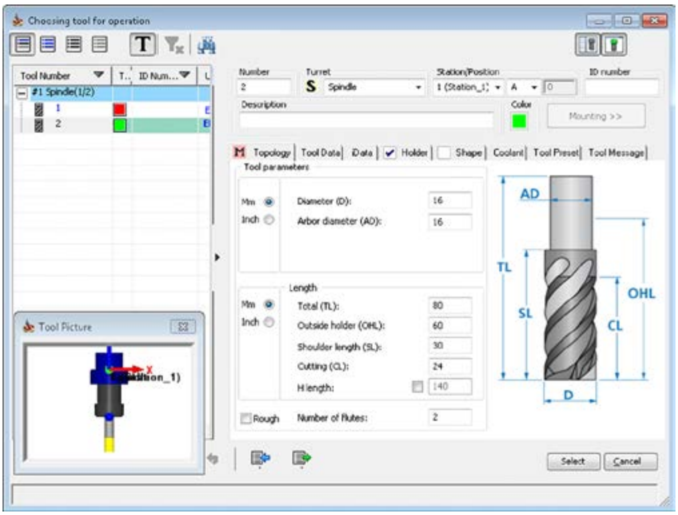
This option enables you to calculate the minimal **Outside holder length** of the tool to avoid the holder gouging with the model. When you select the **Calculate minimum tool length** check box and click the **Save & Calculate** button , InventorCAM calculates the recommended value and displays it in the **Minimum tool length** field.

This option is not available for the **Rest machining** strategy.

4.2 Tool Selection

The **Select** button enables you to edit tool parameters or define the tool you want to use for this operation.

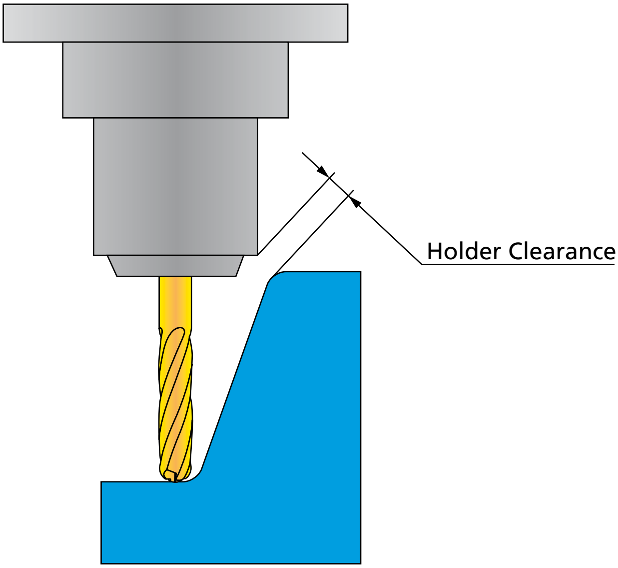
This button displays the **Edit** page of the **Part Tool Table**. You can also add a new tool to be defined for the operation or choose another tool from the **Part Tool Table**.



For more information on the tool definition, refer to the **InventorCAM Milling Online Help**.

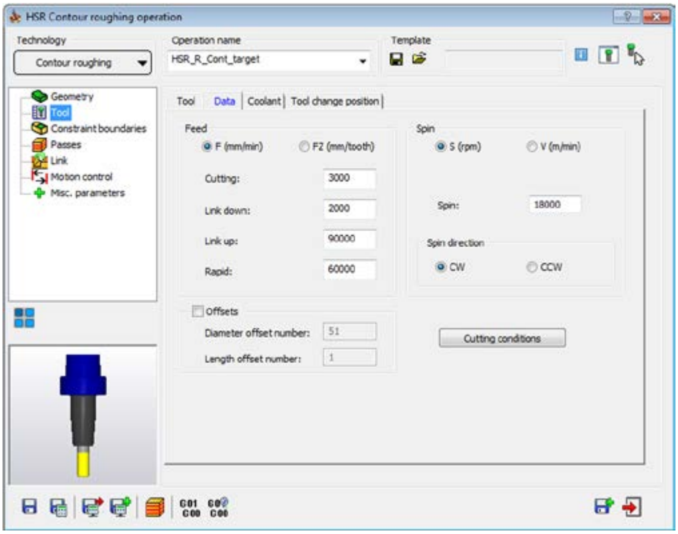
4.3 Holder Clearance

The **Holder clearance** parameter enables you to define how close the holder can approach the material during the machining.



4.4 Spin & Feed Rate Definition

The **Data** tab displays the spin and feed parameters that you can edit.



Spin

This field defines the spinning speed of the tool.

The spin value can be defined in two types of units: **S** and **V**.

S is the default and signifies **Revolutions per Minute**. **V** signifies Material cutting speed in **Meters/Minute** in the **Metric** system or in **Feet/Minute** in the **Inch** system; it is calculated according to the following formula:

V = (S * PI * Tool Diameter) / 1000

Feed Rate

The feed value can be defined in two types of units: **F** and **FZ**. **F** is the default that signifies **Units per minute**. **FZ** signifies **Units per tooth** and is calculated according to the following formula:

FZ = F/(Number of Flutes * S)

The **F**/**FZ** buttons enable you to check the parameter values.

- **Cutting**. This field defines the feed rate of the cutting section of the tool path.
- **Link down**. The feed rate to be set for lead in moves.
- **Link up**. The feed rate to be set for lead out moves.
- **Rapid**. This parameter enables you to define a feed rate for the retract sections of the tool path, where the tool is not contacting with the material.

Boundaries

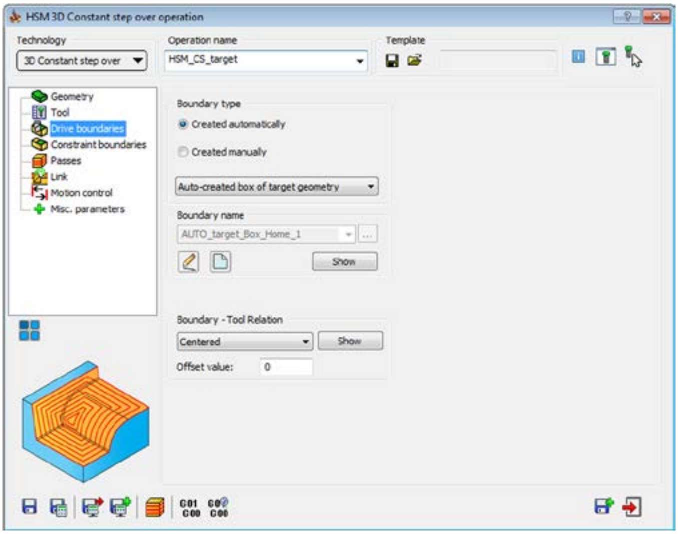
5

5.1 Introduction

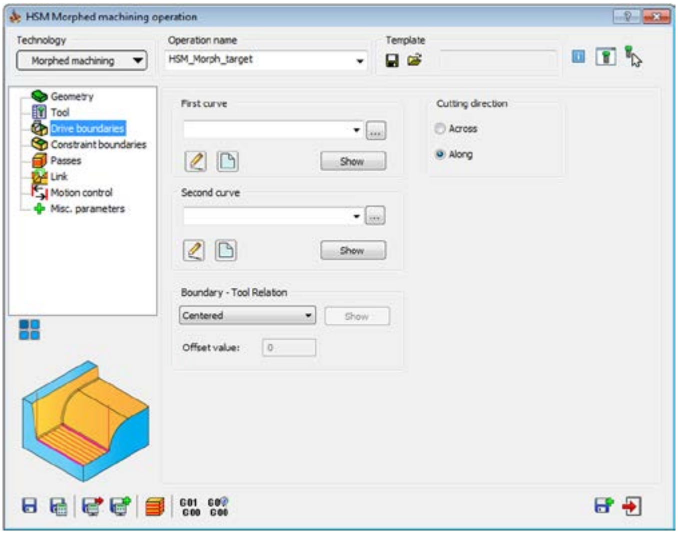
InventorCAM enables you to define two types of boundaries for the InventorCAM HSR/HSM operation tool path: drive boundaries and constraint boundaries.

5.1.1 Drive boundaries

Drive boundaries are used to drive the shape of the tool path for the following strategies: **3D Constant step over**, **Morphed machining**, **Offset cutting** and **Boundary machining**.



Drive boundaries for Morphed machining

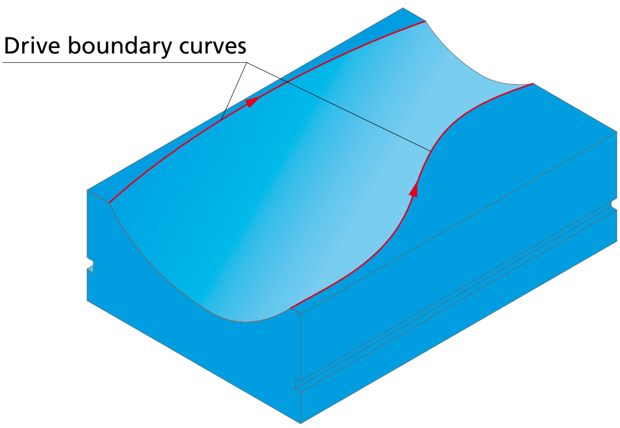


InventorCAM enables you to define drive boundary curves for the **Morphed machining** strategy (see topic **2.13**).

You can choose existing geometries for the first and second drive curves from list or define a new one by clicking  . The **Geometry Edit** dialog box is displayed. For more information on geometry selection, refer to the **InventorCAM Milling Online Help**.

The **Show** button displays the chosen drive curve geometry directly on the solid model.

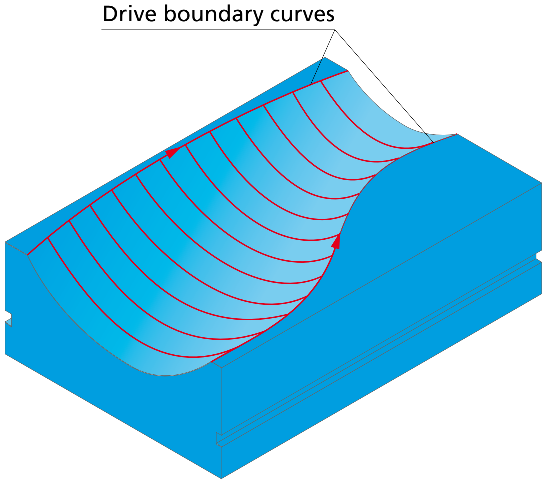
Make sure that the directions of both drive curves are the same in order to perform the correct machining.



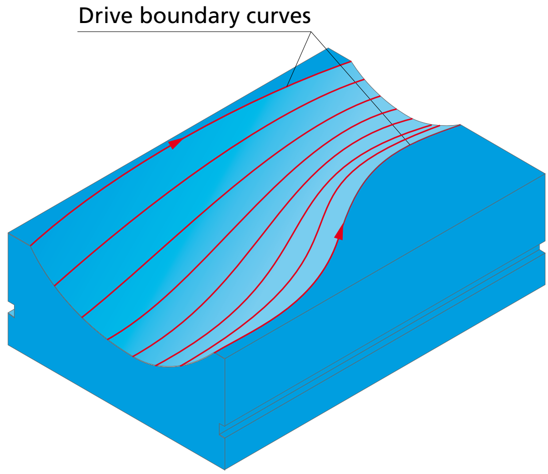
Cutting direction

This option enables you define the tool path direction between the drive curves.

- **Across**. The morphed tool path is performed across the drive curves; each cutting pass connects the corresponding points on the drive curves.

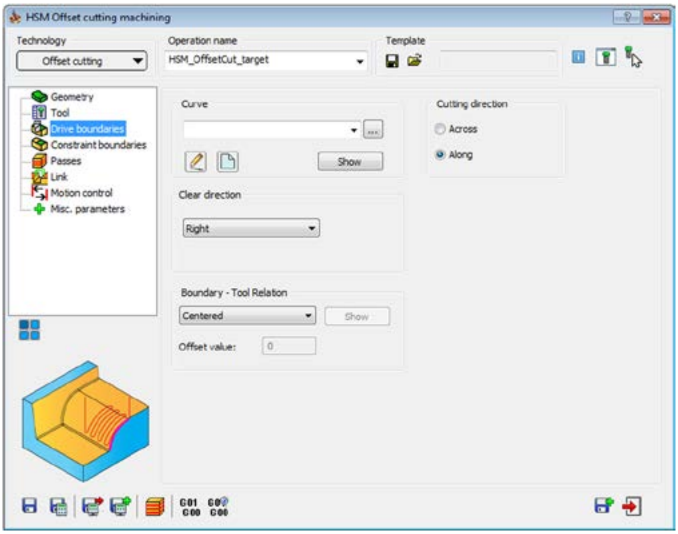


- **Along**. The morphed tool path is performed along the drive curves. The tool path morphs between the shapes of the drive curves gradually changing shape from the first drive curve to the second.



Drive boundaries for Offset cutting

The **Drive boundaries** page of the **HSM Offset cutting machining** dialog box enables you to define the curve and the related parameters.

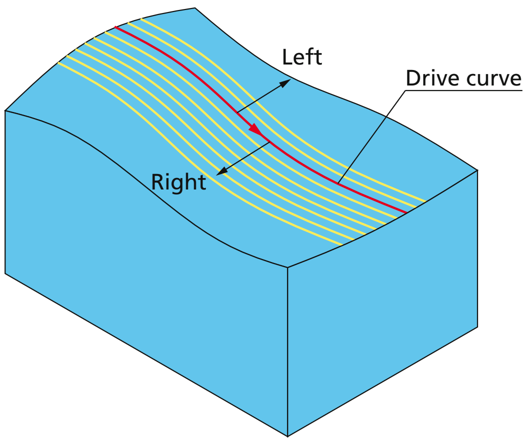


Curve

This section enables you to define the Drive curve used for the tool path definition.

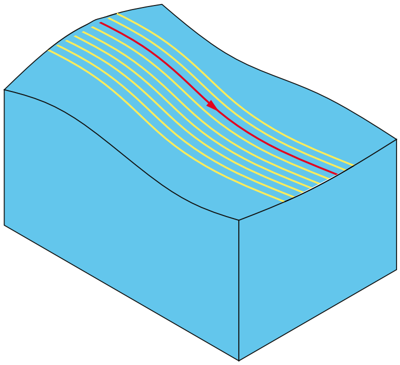
Clear direction

This section enables you to specify the direction in which a virtual offset from the Drive curve is created. The offset can be generated in the **Right**, **Left** or **Both** directions from the Drive curve.

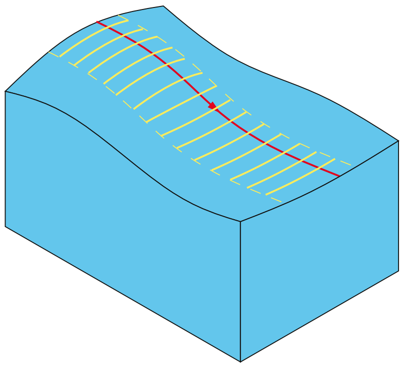


Cutting direction

This section enables you to determine how the machining is performed. When the **Along** option is chosen, the machining is performed along the Drive curve. The tool path morphs between the shapes of the Drive curve and the offset curve, gradually changing shape from the first Drive curve to the offset curve. When the **Across** option is chosen, the tool path is performed across the Drive curve; each cutting pass connects the corresponding points on the Drive curve and offset curve.



Along



Across

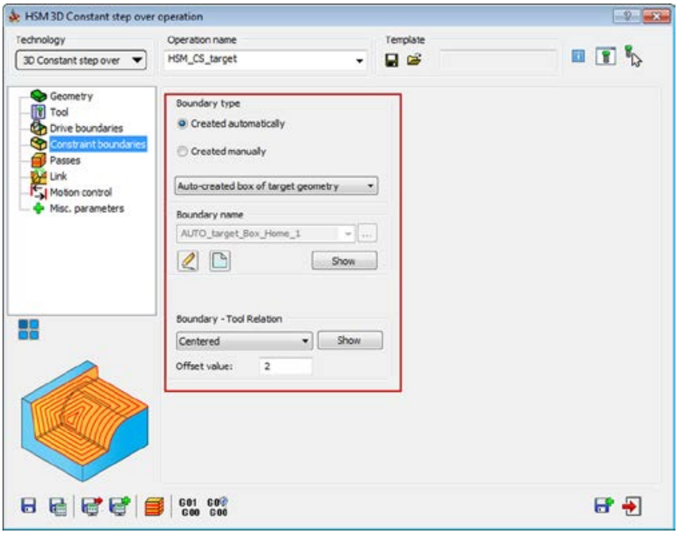
Boundary – Tool Relation

The **Boundary – Tool Relation** section enables you to define the position of the tool relative to the defined boundary and the related parameters.

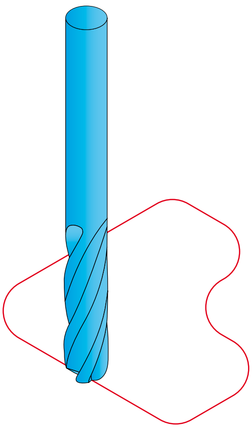
For more information, see topic **5.2.1**.

5.1.2 Constraint boundaries

A constraint boundary enables you to limit the machining to specific model areas.

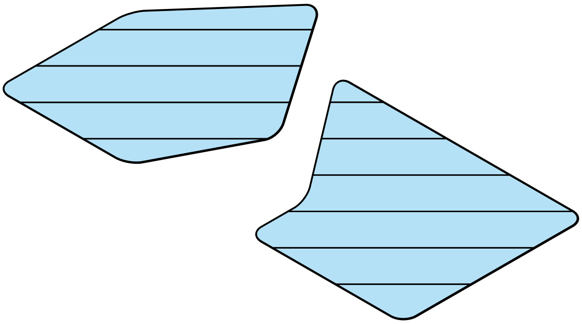


Machining always takes place within a boundary or a set of boundaries. The boundaries define the limits of the tool tip motion. The area actually machined can be extended beyond the boundary by as much as the tool shaft radius.

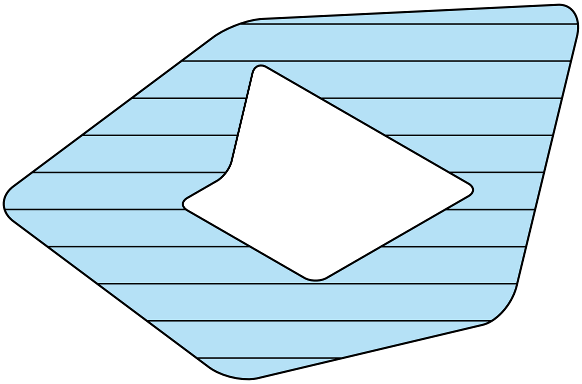


In the image above, the tool center is located at the edge of the boundary, therefore the tool extends beyond the edge by tool radius. You can use the **Offset** (see topic **6.1.7**) feature to offset the tool inside by a certain distance.

If there are several boundary contours then the operation will use all of them.



If one boundary is completely inside another, then it will act as an island. The area enclosed by the outer boundary, minus the area defined the inner boundary, will be machined.

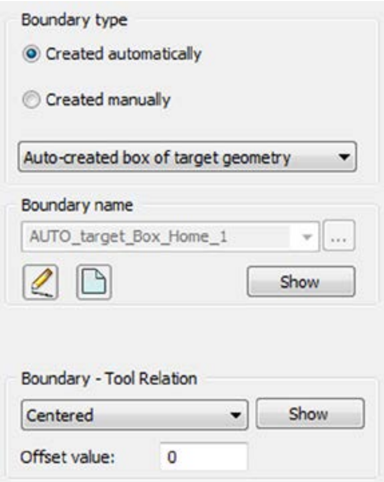


You can extend this to define more complicated shapes by having islands within islands.

5.2 Boundary Definition

5.2.1 Boundary type

The following boundary types are available:



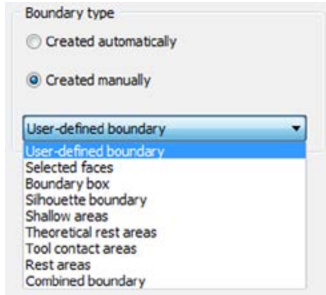
Created automatically

This option enables you to automatically create the boundary using the stock or target models.

The following types of automatically created boundaries are supported in InventorCAM:

- **Auto-created box of target geometry**
- **Auto-created box o stock geometry**
- **Auto-created silhouette**
- **Auto-created outer silhouette**

Created manually



This option enables you to define the constraint boundary that limits the tool path by creating a 2D area above the model in the XY-plane of the current Coordinate system or by an automatically generated 3D curve mapped on the surface.

The following types of 2D boundaries are supported:

- **Boundary box**
- **Silhouette boundary**
- **User-defined boundary**
- **Profile geometry**
- **Combined boundary**

The following types of 3D boundaries are supported:

- **Selected faces**
- **Shallow areas**
- **Theoretical rest areas**
- **Tool contact areas**
- **Rest areas**

5.2.2 Boundary name

This section enables you to define a new boundary geometry or choose an already defined one from the list.

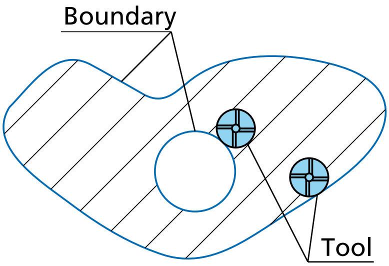
- The **New** button  displays the appropriate dialog box for the geometry definition.
- The **Edit** button  displays the **Select Chain** dialog box (see topic **5.4.7**) enabling you to choose the necessary chains for the boundary. The chosen boundaries are displayed and highlighted in the graphic window.

5.2.3 Boundary – tool relation

This option controls how the tool is positioned relative to the boundaries. This option is relevant only for 2D boundaries.

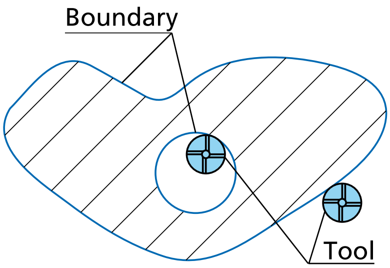
Internal

The tool machines inside the boundary.



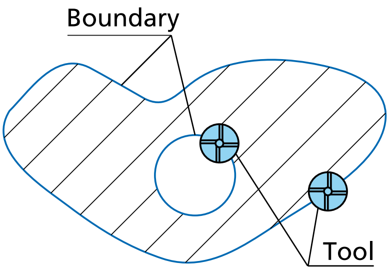
External

The tool machines outside the boundary.



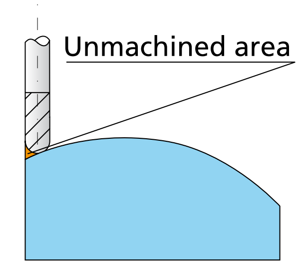
Centered

The tool center is positioned on the boundary.

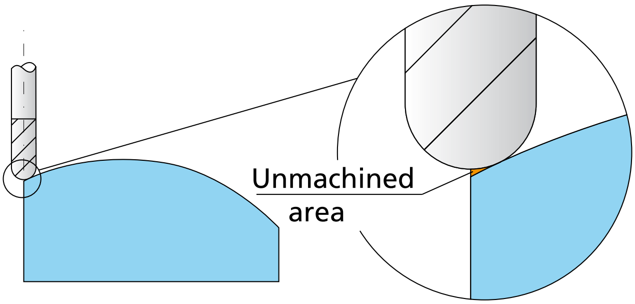


Tangent

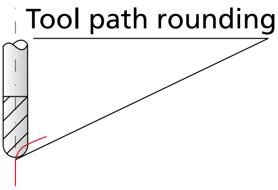
The **Internal/External/Centered** methods of the boundary definition have several limitations. In some cases, the limitation of the tool path by planar boundary results in unmachined areas or corners rounding.



Boundary – Tool Relation: Internal



Boundary – Tool Relation: Centered

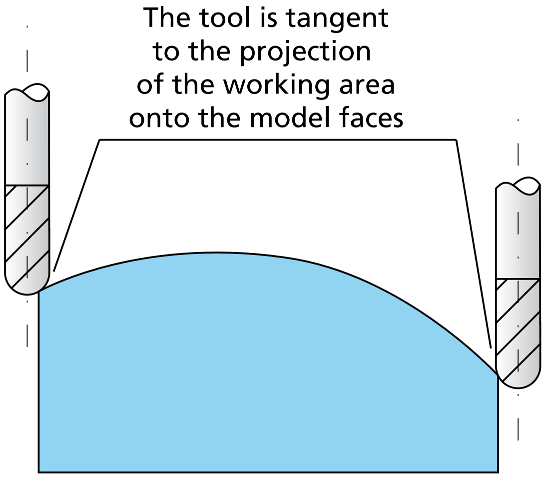


Boundary – Tool Relation: External

The **Tangent** option enables you to avoid these problems.

When this option is chosen, InventorCAM generates the tool path boundaries by projecting the planar working area on the 3D model. The tool path is limited in such a way that the tool is tangent to the model faces at the boundary.

This option enables you to machine the exact boundary taking the geometry into account.

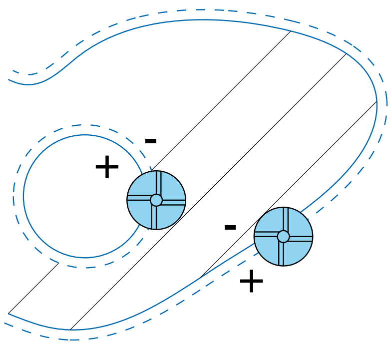


Boundary – Tool Relation: Tangent

Offset value

This value enables you to specify the offset of the tool center.

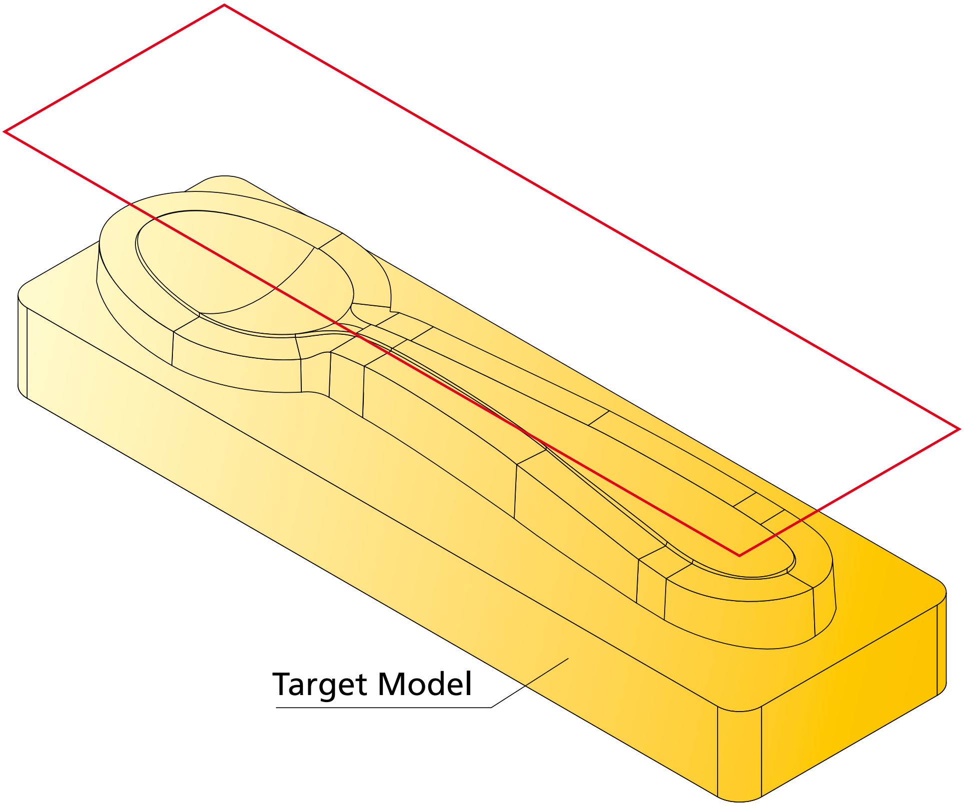
A positive offset value enlarges the boundary; a negative value reduces the boundary to be machined.



5.3 Automatically Created Boundaries

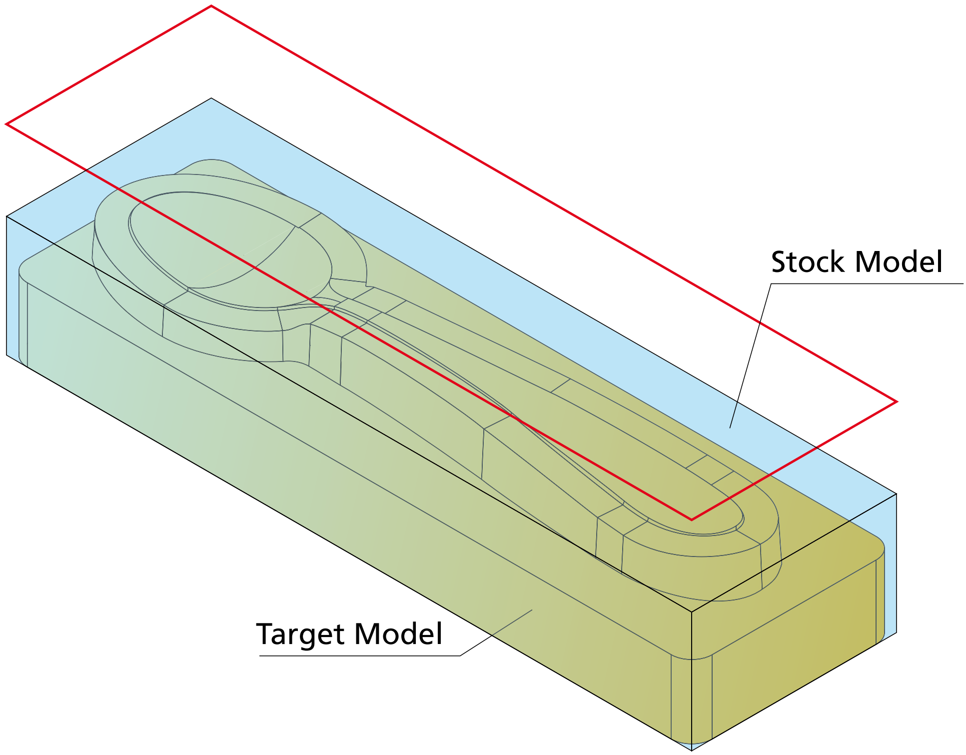
5.3.1 Auto-created box of target geometry

With this option InventorCAM automatically generates a rectangular box surrounding the target model. The tool path is limited to the area contained in this box.



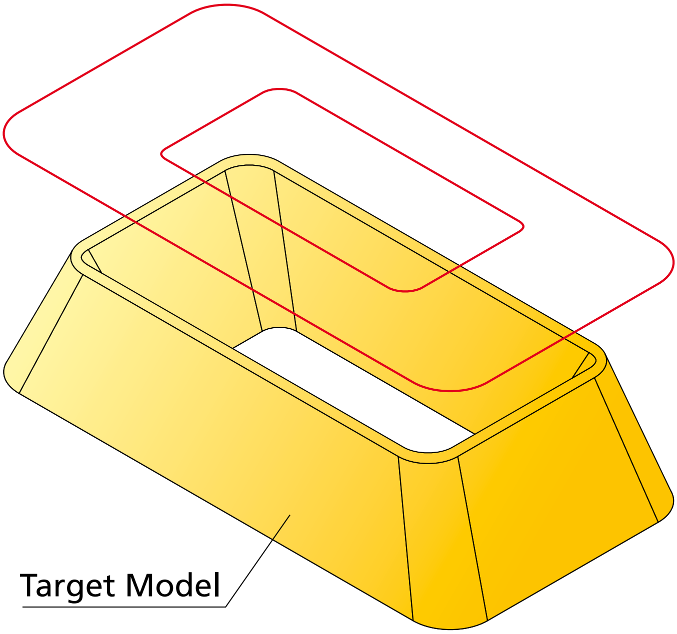
5.3.2 Auto-created box of stock geometry

With this option InventorCAM automatically generates a rectangular box surrounding the stock model. The tool path is limited to the area contained in this box.



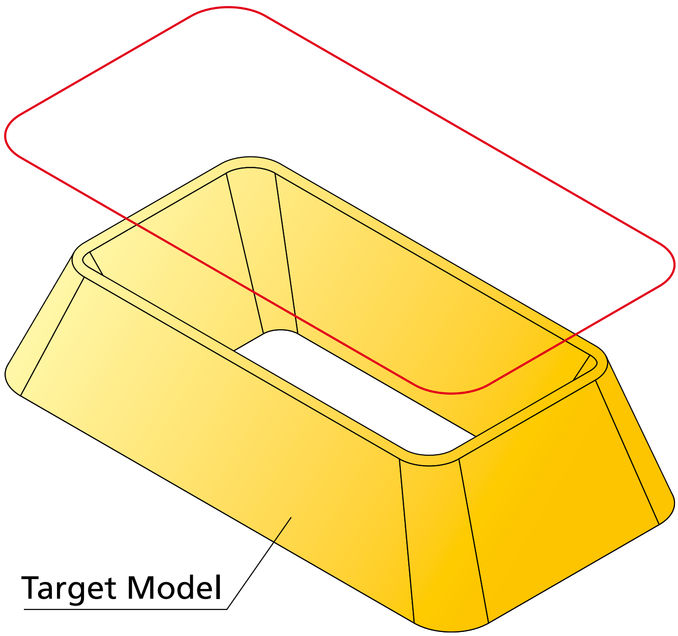
5.3.3 Auto-created silhouette

With this option, InventorCAM automatically generates a silhouette boundary of the target model. A silhouette boundary is a projection of the outer and inner contours of the target model onto the XY-plane.



5.3.4 Auto-created outer silhouette

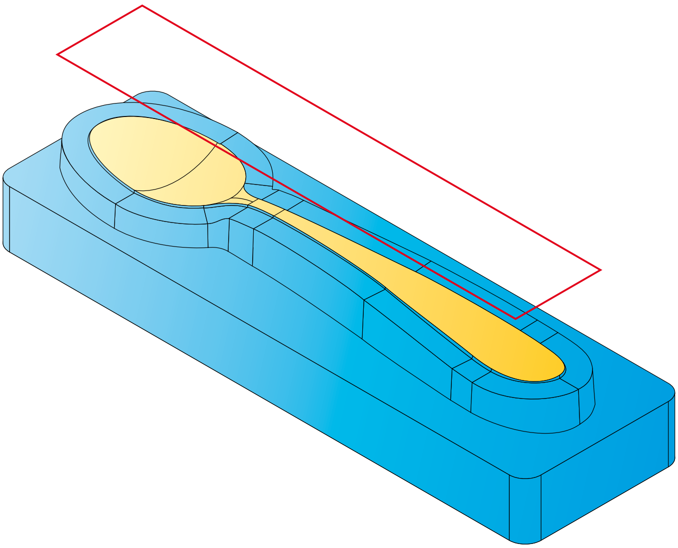
With this option, InventorCAM automatically generates an outer silhouette boundary of the target model. In this case, an outer silhouette boundary is a projection of the outer contours only onto the XY-plane.



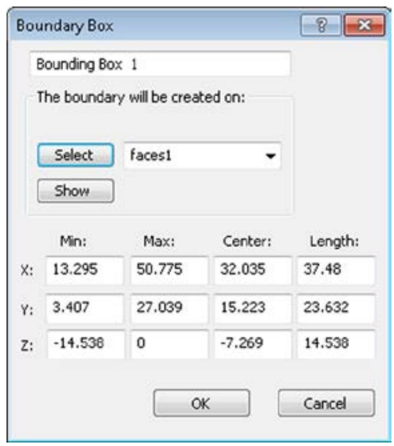
5.4 2D Manually Created Boundaries

5.4.1 Boundary box

A **Boundary Box** is a rectangular box surrounding the selected model geometry. InventorCAM enables you to limit the machining passes to the area contained in the Boundary box.



The **Boundary Box** dialog box enables you to define a necessary parameters and choose the model elements for the bounding box calculation.



The boundary will be created on

This option enables you to select the faces for which a bounding box is generated. Click the **Select** button to display the **Select Faces** dialog box (see topic **5.4.6**).

The **Show** button displays the already selected faces geometry.

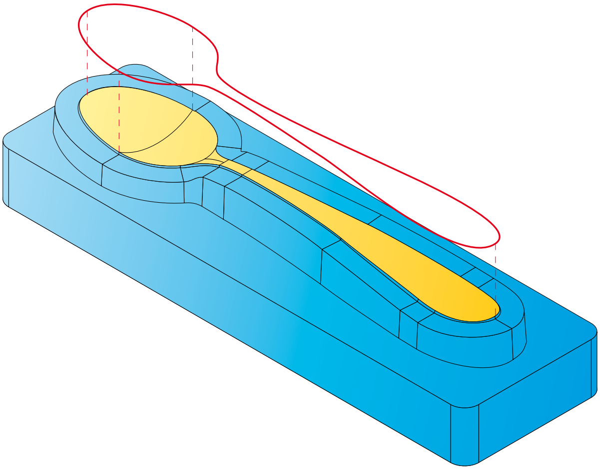
The table section displays the automatically calculated minimum and maximum coordinates, center and length of the bounding box.

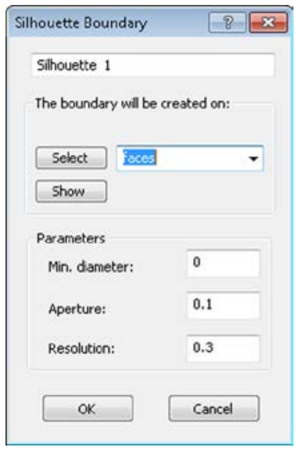
InventorCAM enables you to change the XY-coordinates of the minimum and maximum coordinates of the bounding box.

When the geometry for the bounding box generation is defined, click the **OK** button. The boundary chains will be generated and the **Select Chain** dialog box (see topic **5.4.7**) will be displayed.

5.4.2 Silhouette boundary

A **Silhouette** boundary is a projection of the face edges onto the XY-plane. In other words, it is the shape that you see when you looking at a set of surfaces down the tool axis.





The **Silhouette Boundary** dialog box enables you to define the parameters and choose the solid model elements for the silhouette boundary calculation.

The boundary will be created on

This option enables you to choose a faces geometry to generate a silhouette boundary. InventorCAM enables you either to choose an already existing **Faces geometry** from the list or define a new one with the **Select** button. The **Select Faces** dialog box (see topic **5.4.6**) will be displayed. The **Show** button displays the selected face geometry.

- **Min. diameter**

  This value defines the spanning of the boundary, the distance between two points on either side. Boundaries that have a diameter smaller than this are discarded.

- **Aperture**

  This value defines the fuzziness of the Silhouette. Decrease the value to bring it into sharper focus; increase it to close up unwanted gaps between boundaries.

- **Resolution**

  This value defines the granularity of the calculation: a small value results in a more detailed boundary, but it is slower to calculate.

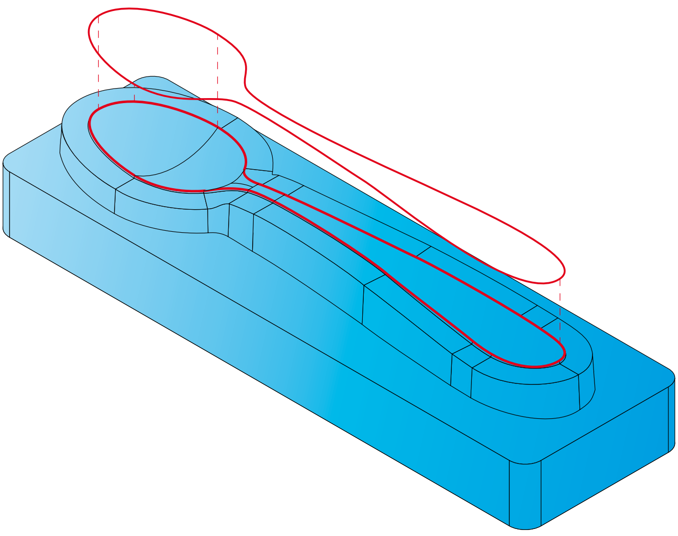
When the geometry for the silhouette boundary generation is defined, click the **OK** button. The boundary chains will be generated and the **Select Chain** dialog box (see topic **5.4.7**) will be displayed.

5.4.3 User-defined boundary

InventorCAM enables you to define a user-defined boundary based on a **Working area** geometry (closed loop of model edges as well as sketch entities).

For more information on Working area geometry, refer to the **InventorCAM Milling Online Help**.

InventorCAM automatically projects the selected geometry on the XY-plane and defines the 2D boundary.



The **Geometry Edit** dialog box enables you to define the geometry.

5.4.4 Profile geometry

InventorCAM enables you to define a user-defined boundary based on a **Profile geometry**. All the HSM strategies enable you to use closed profile geometries. The **Boundary machining** strategy (see topic **2.15**) enables you to use also open profiles for the boundary definition; this feature is useful for single-contour text engraving or for chamfering.

For more information on Profile geometry, refer to the **InventorCAM Milling Online Help**.

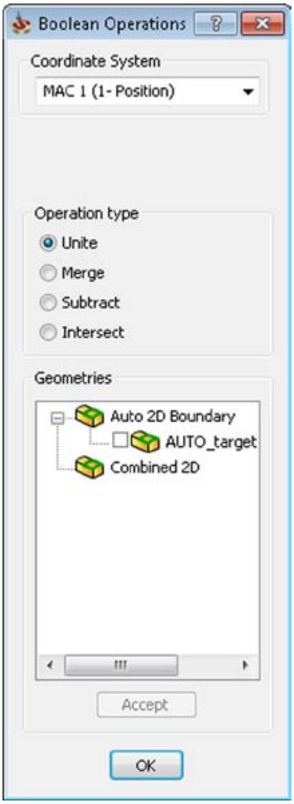
InventorCAM automatically projects the selected geometry on the XY-plane and defines the 2D boundary.

The **Geometry Edit** dialog box enables you to define the geometry.

5.4.5 Combined boundary

This option enables you to define the boundary by performing a number of boolean operations between working area geometries and boundaries.

The **Boolean Operations** dialog box is displayed.



Coordinate System

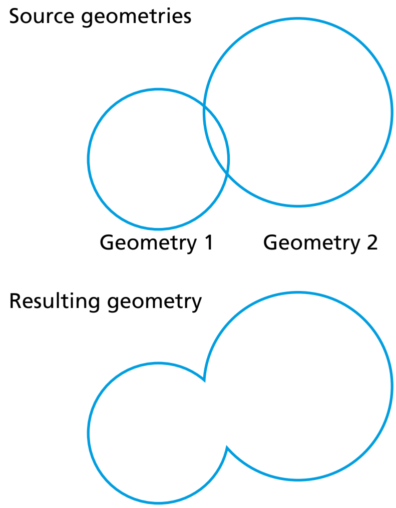
This field enables you to choose the Coordinate System where the source geometries for the boolean operation are located. The resulting combined geometry will be created in the chosen coordinate system.

Operation type

This field enables you to define the type of the boolean operation. The following boolean operations are available:

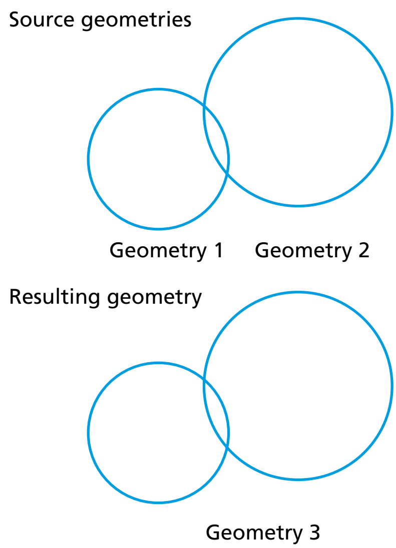
- **Unite**

  This option enables you to unite selected geometries into a single one. All internal segments are removed; the resulting geometry is outer profile.



- **Merge**

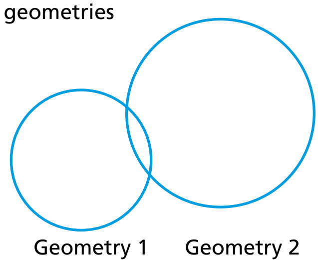
  This option enables you to merge a number of geometries, created by different methods, into a single one.



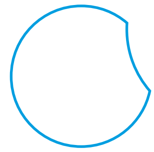
- **Subtract**

  This option enables you to perform subtraction of two geometries. The order of the geometry selection is important; the second selected geometry is subtracted from the first selected one.

Source geometries



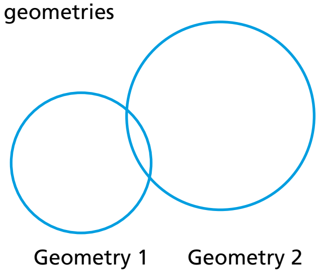
Resulting geometry



- **Intersect**

  This option enables you to perform intersection of two geometries.

Source geometries



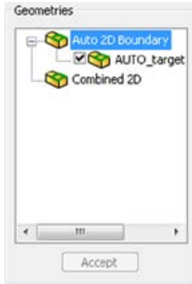
Resulting geometry

The **Accept** button performs the chosen operation with the geometries chosen in the **Geometries** section.

Geometries

The **Geometries** section displays all the available working area geometries classified by the definition method.

This section enables you to choose the appropriate geometries for the boolean operation. Select the check box near the geometry name in order to choose it for the boolean operation.



When you click the **Accept** button, the resulting geometry is displayed in the list under the **Combined 2D** header. InventorCAM enables you to edit the name of the created geometry. The newly created geometry is automatically chosen for the further boolean operation.

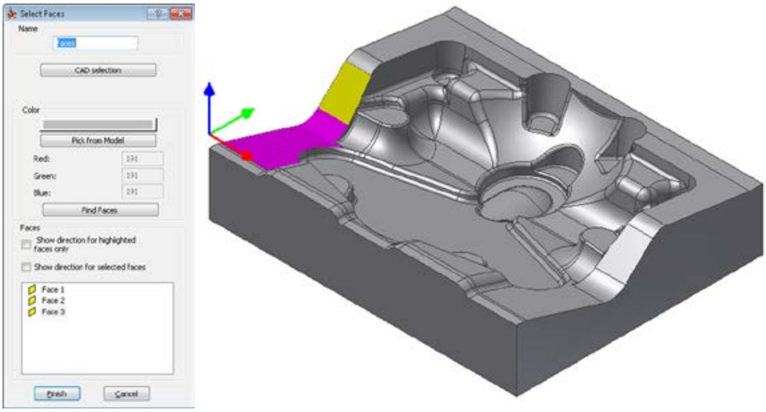
The resulting combined geometry is always a 2D geometry even if one or more of the input geometries is a 3D boundary.

The right-click menu available on the list items enables you to perform the following operations:

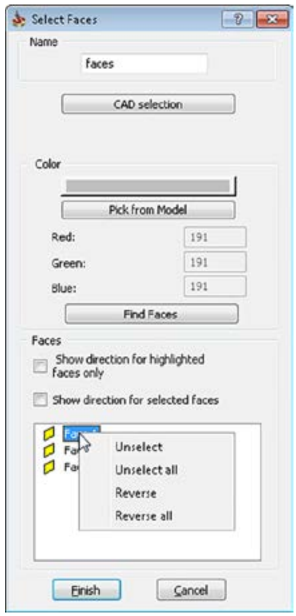
- **Accept**. This button enables you to perform the chosen boolean operation with the selected geometries.
- **Unselect All**. This option unselects all the chosen geometries.
- **Delete**. This option enables you to delete combined geometries generated in the current session of the **Boolean Geometries** dialog box.

5.4.6 Select Faces dialog box

This dialog box enables you to select one or several faces of the Autodesk Inventor model. The selected Face tags will be displayed in the dialog box.



If you have chosen wrong entities, use the **Unselect** option to undo your selection. You can also right-click on the entity name (the object will be highlighted) and choose the **Unselect** option from the menu.

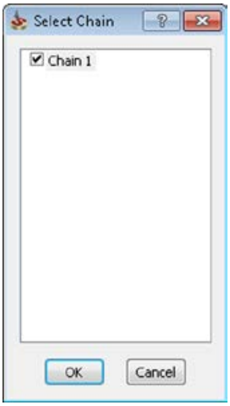


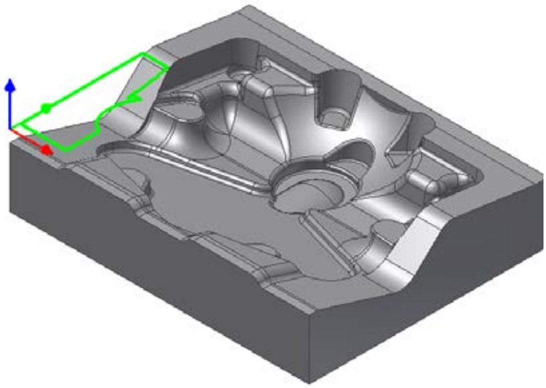
The **Reverse**/**Reverse all** option enables you to change the direction of the normal vectors of the selected faces.

The **CAD Selection** option enables you to select faces with the Autodesk Inventor tools.

5.4.7 Select Chain dialog box

Depending on the boundary type, InventorCAM generates a number of chains for the selected faces. The **Select Chain** dialog box enables you to select the chains for the boundary.

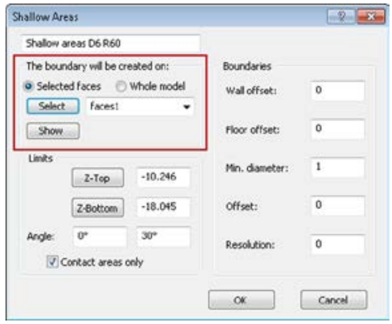




5.5 3D Manually Created Boundaries

5.5.1 Common parameters

The boundary will be created on



- **Selected faces**. This option enables you to choose a faces geometry to generate a boundary of the defined type. InventorCAM enables you either to choose an already existing **Faces geometry** from the list or define a new one with the **Select** button. The **Select Faces** dialog box (see topic **5.4.6**) will be displayed. The **Show** button displays the selected face geometry.

- **Whole model**. With this option, InventorCAM generates boundaries of the chosen type for all the model faces.

Limits

- **Z Limits**

  Set the machining range along the Z-axis by definition of upper and lower limits. Boundaries will be generated within this range.

- **Angle**

  Set the contact angle range of your tool by setting the minimum and maximum contact angle. Boundaries will be generated around areas where the angle is within that range. For **Shallow Area boundaries** (see topic **5.5.3**), the range should typically be between 0 and 30 degrees, but where surfaces are very close to the minimum or maximum angle, you may get an undesirably jagged edge so you may want to alter the range slightly. Alternatively, you can sometimes prevent formation of jagged edges by giving the boundary a small offset.

- **Contact Areas Only**

  This option should be selected to choose only boundaries that are in contact with the model surface.

Boundaries

- **Wall offset**

  This value defines the distance at which the boundaries and therefore the tool will be away from the surface, similar to the **Wall offset** parameter on the **Passes** page (see topic **6.1.1**).

  For roughing and semi-finishing operations, this value must be greater than zero. The calculations are based on a modified tool, the surface of which is offset to be larger than the actual tool. This will leave material on the part.

  For finishing operations, the value must be set to zero. The calculations are based on the dimensions of the defined tool with no offset.

  In special circumstances, such as producing electrodes with a spark gap, this value can be smaller than zero. The tool will remove material at a level below the designated surface. The calculations are based on a modified tool, offset smaller than the one used.

- **Floor offset**

  This value defines the distance away from the surface at which the boundaries will be in the tool axis direction.

  The boundary is calculated using the **Wall offset**. The resulting boundary is updated by offsetting along the tool axis by a distance equal to the **Floor offset**.

- **Min Diameter**

  This value defines the spanning of the boundary, the distance between two points on either side. Boundaries that have a diameter smaller than this are discarded.

- **Offset**

  The boundaries are calculated and then offset by this amount.

  It may be advantageous sometimes to set a small offset value to prevent jagged boundary edges where surface area is at angle similar to the **Contact Angle**.

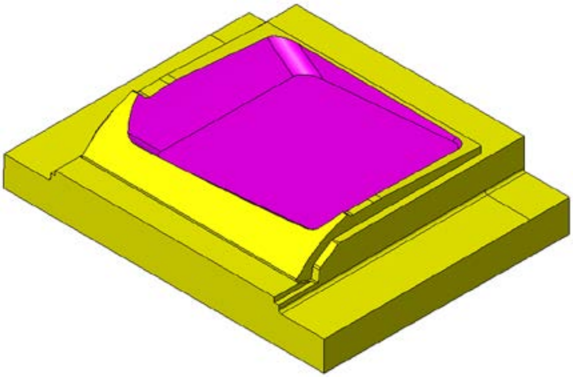
  In **Rest areas** (see topic **5.5.6**) with no offset, the exact boundary area would be machined, resulting in marks or even cusps around the edge. For **Theoretical rest areas** (see topic **5.5.4**), the boundaries are offset outwards along the surface by this amount after they have been made; a good surface finish is ensured at the edges of the rest areas. Without offsetting, the exact **Theoretical rest area** would be machined, probably leaving marks or even cusps (of just under the **minimum material depth** value) around the edge. The offsetting makes the boundaries smoother, so a tool path made using them is less jagged.

- **Resolution**

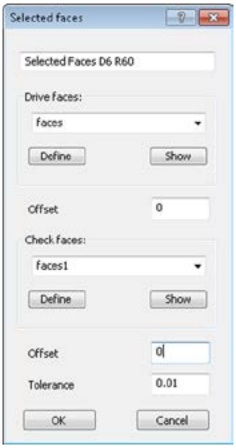
  This value defines the granularity of the calculation. A small value results in a more detailed boundary but it will be slower to calculate.

5.5.2 Selected faces

This option enables you to define the boundary by selecting drive and check faces similar to the Working area definition for 3D Milling operations.



Under **Boundary name**, click the **Define** button to start the boundary definition. The **Selected faces** dialog box enables you to define the drive and check faces.

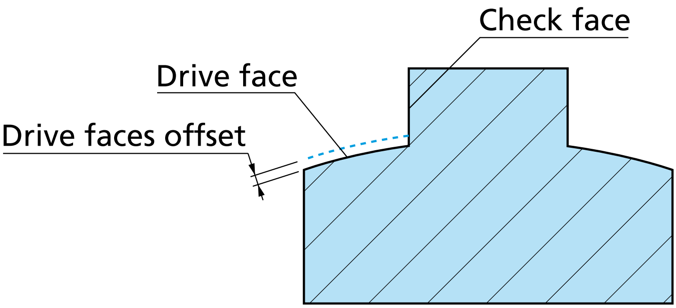


Name

This section enables you to define the boundary name and the tolerance that is used for the boundary creation.

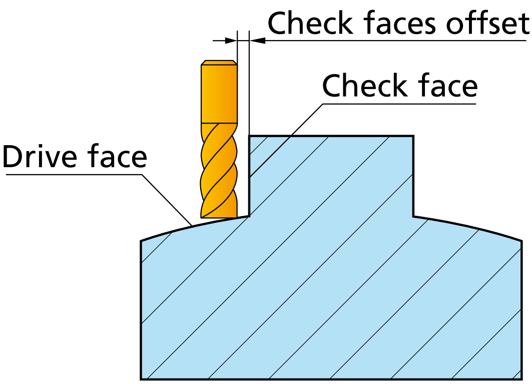
Drive faces

This section enables you to define **Drive faces** – the set of faces to be milled. The tool path is generated only for machining of these faces. The **Define** button displays the **Select Faces** dialog box used for the faces selection. The **Offset** edit box enables you to define the offset for the **Drive faces**. When the offset is defined, the machining is performed at the specified offset from the **Drive faces**.



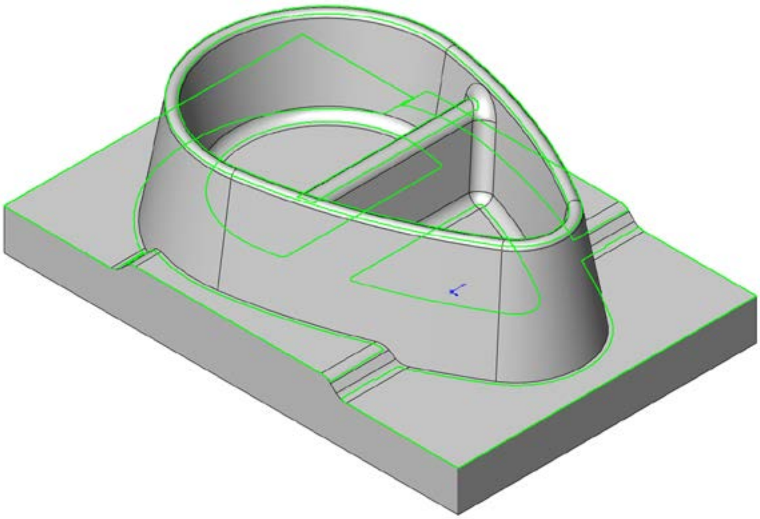
Check faces

This section enables you to define **Check faces** – the set of faces to be avoided during the generation of the tool path. The **Define** button displays the **Select Faces** dialog box used for the faces selection. The **Offset** edit box enables you to define the offset for the **Check faces**. When the offset is defined, the machining is performed at the specified offset from the **Check faces**.



5.5.3 Shallow areas

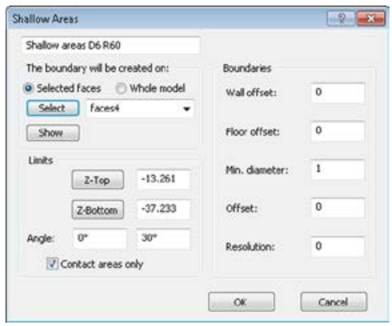
With this option, InventorCAM enables you to automatically determine shallow areas in the model and define boundaries around them.



The tool has to be chosen for the operation before the shallow areas boundary definition.

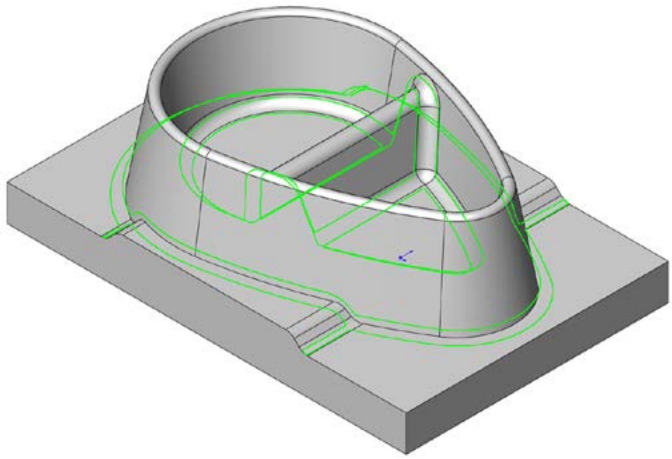
The **Select faces** dialog box enables you to choose the necessary model faces. When the faces are chosen and the dialog box is confirmed, the **Shallow Areas** dialog box is displayed.

This dialog box enables you to define a number of parameters for the shallow areas boundary generation.

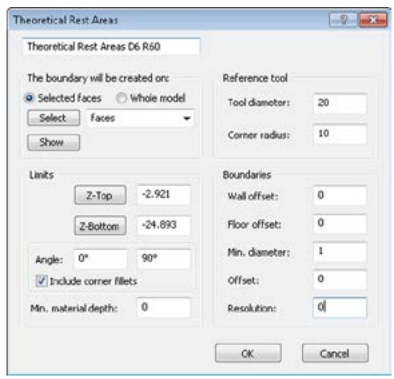


5.5.4 Theoretical rest areas

You can create 3D boundaries from rest areas left by an imaginary reference tool. This gives good results when used for semi-finish and finish machining operations. You can then use these boundaries to limit another InventorCAM HSM operation performed with a tool of an equal or smaller size.



The **Select faces** dialog box enables you to choose the necessary model faces. When the faces are chosen and the dialog box is confirmed, the **Theoretical Rest Areas** dialog box is displayed.



This dialog box enables you to define a number of parameters for the theoretical rest material areas generation.

Limits

- **Include Corner Fillets**

  In corner area, the angle is degenerate. Use this option to include or exclude all corner areas from the rest area boundaries.

- **Min material depth**

  The smallest amount of material to be found in areas included in the rest area boundary prior to rest machining. If the reference tool left parts of the material with less than this amount, those material areas would not be included in the rest area boundaries.

  The **Min material depth** should be greater than the cusp height left by the passes of the imaginary reference tool path. If the **Min material depth** is smaller than the cusp height left by the passes of the imaginary reference tool path the whole area machined by the reference tool will be included in the rest area boundary.

Reference Tool

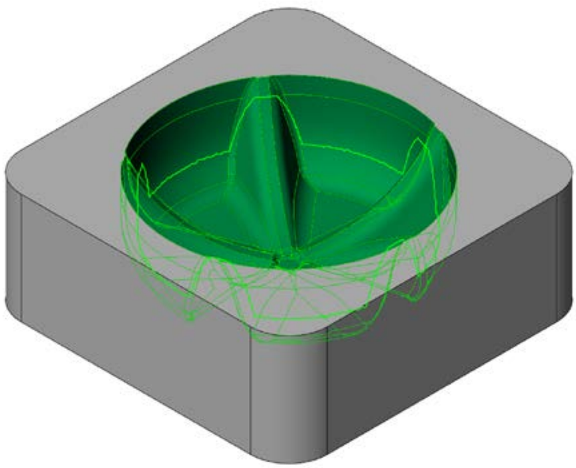
This section enables you to specify the tool with which the theoretical rest areas will be calculated. This tool is usually larger than the tool that will be used to cut the rest areas. The reference tool is used to represent an imaginary tool path, and the rest areas are created assuming that the tool path had been created.

Define the size of the tool by inserting values into the **Tool diameter** and **Corner radius** fields.

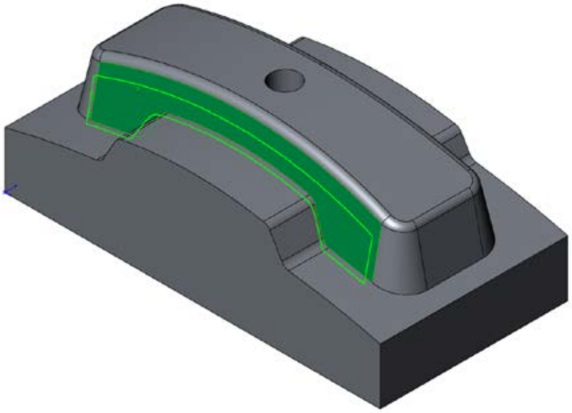
5.5.5 Tool contact area

Tool contact area detection enables you to create 3D boundaries around areas where the tool is in contact with the selected surface or surfaces.

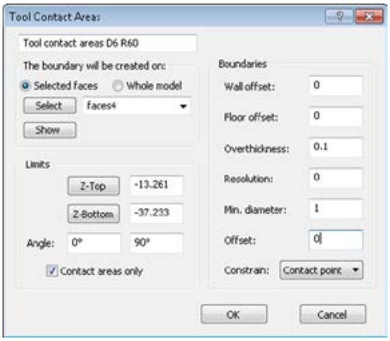
Tool contact area boundaries do not work on vertical or near-vertical surfaces. The steepest angle you should use for best results is 80 degrees.



The surface has to be selected as shown below.



If a **Tool contact area** boundary is created from this selection, there will be offset from the edges where the selected surface is adjacent to another surface. The tool can only reach the edges where there are no other surfaces to hinder its movement.



The **Select faces** dialog box enables you to choose the necessary model faces. When the faces are chosen and the dialog box is confirmed, the **Tool Contact Areas** dialog box is displayed.

This dialog box enables you to define the parameters for the boundary calculation.

Boundaries

Overthickness

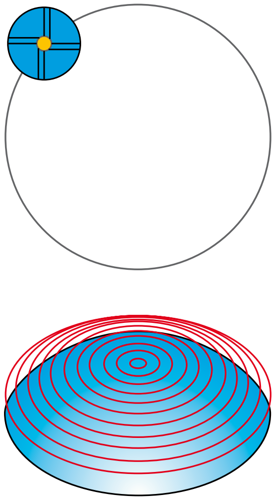
This option is available only for **Tool contact area** boundaries. **Overthickness** is an extra offset that can be applied to the tool in addition to the set **Wall offset** when you wish to calculate with a tool slightly larger than the one you intended to use, to create smooth filleted edges.

Constrain

This option enables you to limit the tool motion in two ways:

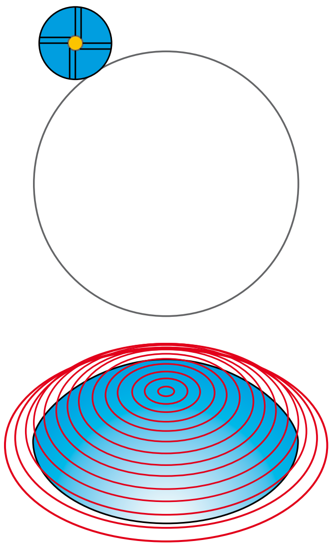
- **Center point**

  The point of contact between the tool and the surfaces always lies within the boundary.



- **Contact point**

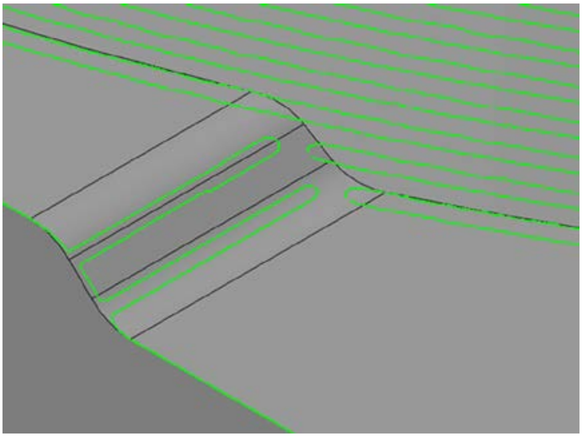
  The edge of the tool always lies within the boundary.



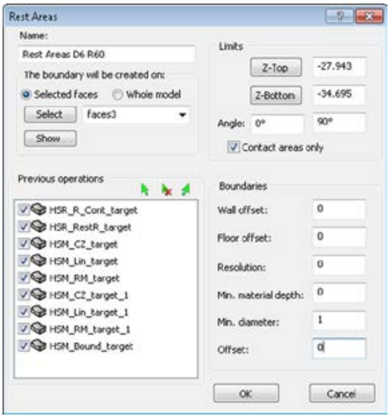
5.5.6 Rest areas

This option enables you to define rest material left unmachined after any machining strategy to create 3D boundaries. You can then use these boundaries to limit the operation tool path, made with a tool of an equal or smaller size to these specific areas.

The **Select faces** dialog box enables you to choose the necessary model faces. When the faces are chosen and the dialog box is confirmed, the **Rest Areas** dialog box is displayed.



This dialog box enables you to define the parameters for the rest areas calculation.



Previous operations

InventorCAM enables you to choose any previous HSR/HSM operation for the Rest areas calculation.

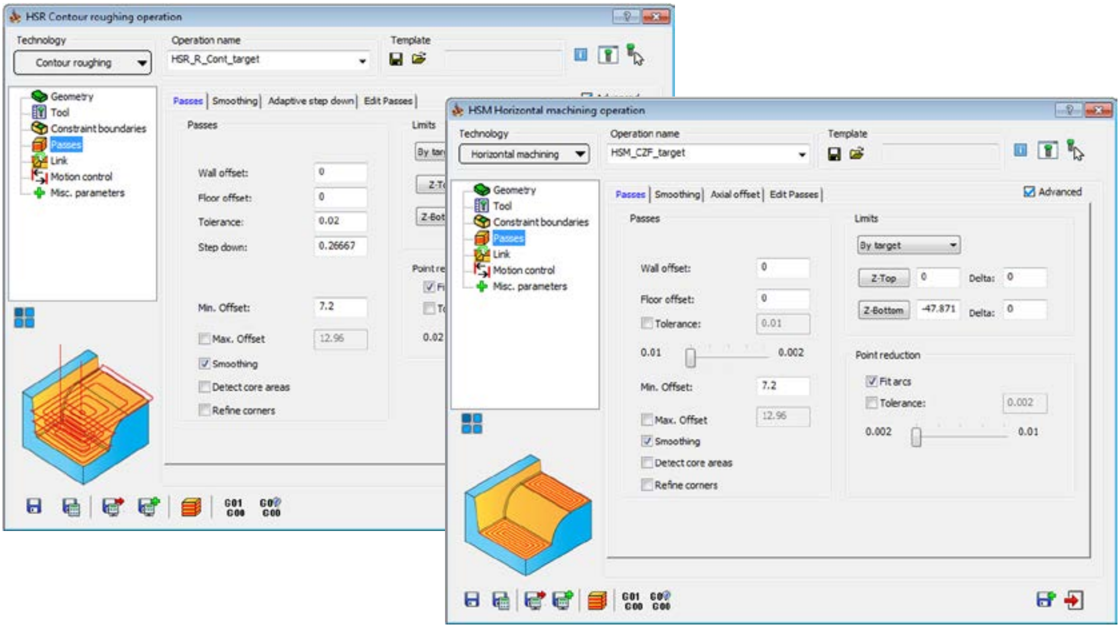
Min. material depth

This value defines the granularity of the calculation. A small value results in a more detailed boundary but it will be slower to calculate.

Passes

6

The **Passes** page enables you to define the technological parameters needed to generate the tool path for the InventorCAM HSR/HSM operation.



Common Parameters

The **Passes** parameters for the various machining strategies vary slightly, but most of them are the same. The following section is a general overview of the common parameters for all the InventorCAM HSR/HSM strategies.

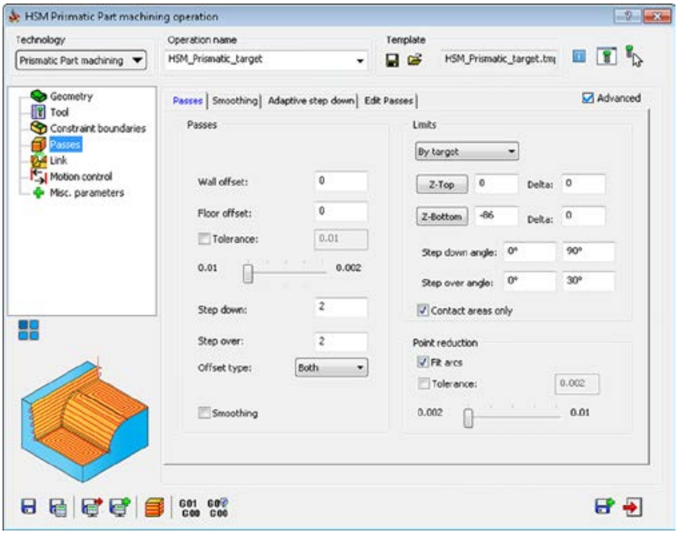
The following tabs are available:

• **Passes parameters**	**HSR/HSM**
• **Smoothing parameters**	**HSR/HSM**
• **Adaptive step down parameters**	**HSR/HSM**
• **Edit Passes parameters**	**HSR/HSM**
• **Cross parameters**	**HSM**
• **Axial offset parameters**	**HSM**

The **Advanced** check box displays the additional tabs with advanced parameters.

6.1 Passes Parameters

The **Passes** tab displays the major parameters that affect the generation of tool path passes.

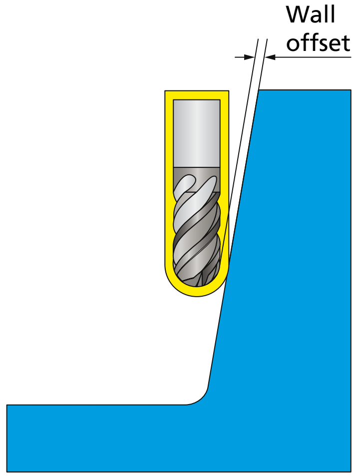


- **Wall offset**
- **Floor offset**
- **Tolerance**
- **Step down**
- **Step over**
- **Pass extension**
- **Offsets**
- **Refine corners**
- **Limits**
- **Point reduction**

6.1.1 Wall offset

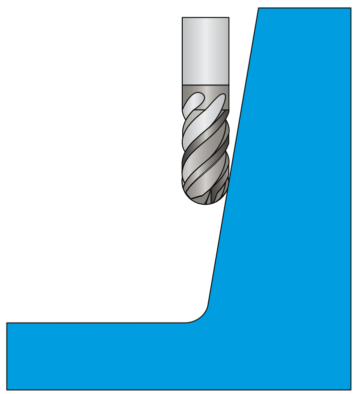
This option enables you modify the tool diameter. The machining is performed using the modified tool.

Positive value



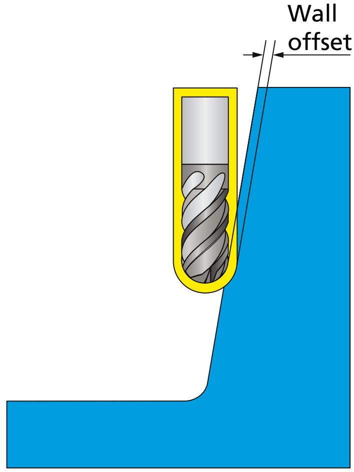
The tool is moved away from the machining surface by the defined value. The offset is left unmachined on the surfaces. Generally, positive values are used for roughing and semi-finishing operations to leave an allowance for further finishing operations.

No offset



The tool with the specified diameter is used for the tool path calculation. The machining is performed directly on the model surfaces. Generally, zero offset is used for finishing operations.

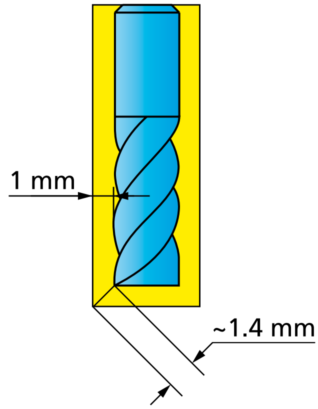
Negative value



The tool is moved deeper into the material penetrating the machining surface by the specified value.

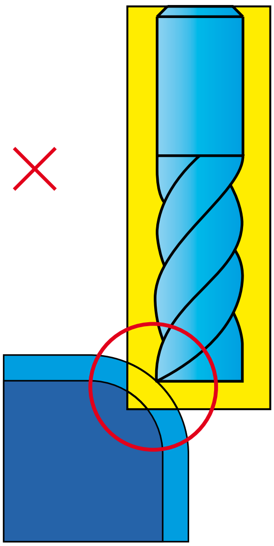
This option is used in special circumstances, such as producing electrodes with a spark gap. The tool will remove material at a level below the designated surface. The calculations are based on a modified tool smaller than the one used.

Since the calculations for negative values are based on a modified tool which is smaller than the one used, the offset must be equal to or smaller than the corner radius of the tool. Where the offset is greater than the corner radius of the tool, surfaces at angles near to 45° are unfavorably affected as the corner of the tool impacts on the machined surface, since the offset at the corners is in fact greater than the value set (see image). Surfaces that are horizontal or vertical are not affected.

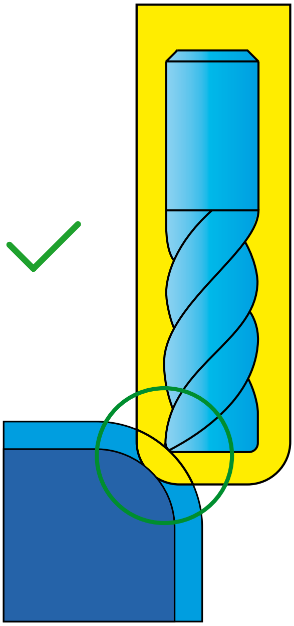


If a negative value (e.g. –1 mm) is applied to a tool without corner radius, the real offset at the corners of the tool will be considerably greater than 1 mm (–1.4 mm appx.). This is obviously incorrect. If you want to simulate a negative offset with a slot mill, start by defining a bull-nosed tool with the corner radius equal to the negative value of the offset – corner radius of 1 mm is used with negative offset of –1 mm.

If you define an end mill, the offset will be greater than the value set on surfaces nearing 45 degrees.



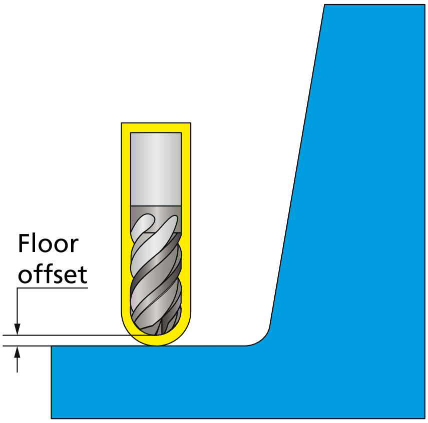
Using a bull-nosed tool with a positive corner radius equal to the desired negative offset, you will achieve better and more accurate results.



6.1.2 Floor offset

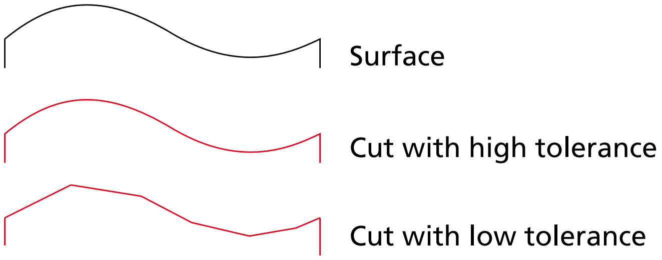
This offset is applied to the tool and has the effect of lifting (positive value) or dropping (negative value) the tool along the tool axis. As a result, **Floor offset** has its greatest effect on horizontal surfaces and no effect on vertical surfaces. By default, this value is equal to that of **Wall offset** (see topic **6.1.1**).

The tool path is calculated taking into account the tool plus **Wall offset**. The resulting tool path is calculated by offsetting along the tool axis by the distance equal to the specified **Floor offset** value.



6.1.3 Tolerance

All machining operations have a tolerance, which is the accuracy of the calculation. The smaller is the value, the more accurate is the tool path.

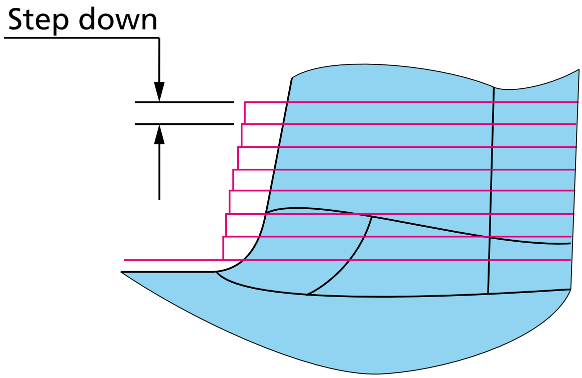


The tolerance is the maximum amount that the tool can deviate from the surface.

Using the slider, you can adjust the tolerance within the range defined on the **3D HSM/3D HSR** page in **InventorCAM Settings > Defaults > Tolerance**.

6.1.4 Step down

The **Step down** parameter defines the spacing of the passes along the tool axis. This parameter is different from **Adaptive step down** (see topic **6.3**), which adjusts the passes to get the best fit to the edges of a surface. The passes are spaced at the distance set, regardless of the XY-value of each position (unless the **Adaptive step down** check box is selected).



This parameter is available for the **Constant Z** finishing strategy.

Step down type

- **Constant**

  This parameter is the default option that allows constant step down along all model levels.

- **Constant & flats**

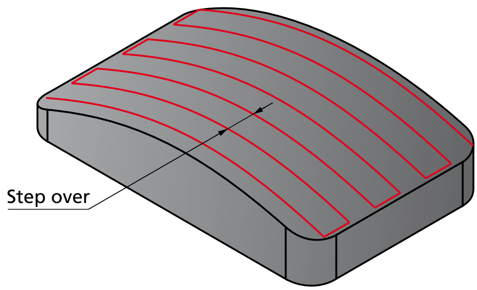
  This parameter allows adding extra passes between step downs to machine material left on flats.

This parameter is available for the **HM Roughing** strategy.

6.1.5 Step over

Step over is the distance between two adjacent passes.

In the **3D Constant step over** strategy, the value of the **Step over** is constant.



In **Morphed machining** and **Offset cutting** strategies, the actual **Step over** can vary, and this parameter defines the maximal step over value.

For all the strategies, **Step over** is measured in the XY-plane, but for the **3D Constant step over** strategy (see topic **2.17**), **Step over** is measured along the surface.

This parameter is available for **Linear machining**, **Radial machining**, **Spiral machining**, **Morphed machining**, **Offset cutting**, **3D Constant step over**, **Hatch roughing** and **HM Roughing**.

Step over type

- **Core**

  All passes are offsets of the machinable area. All areas are machined from the outside inwards.

- **Cavity**

  All passes are offsets of the machinable area. All areas are machined from the center outwards.

- **HM spiral**

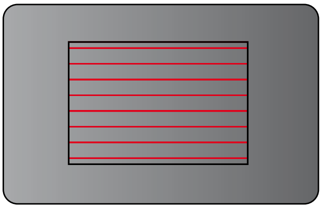
  All passes are offsets of the part area. Core areas are machined from the outside inwards. Cavity areas are machined from the center outwards.

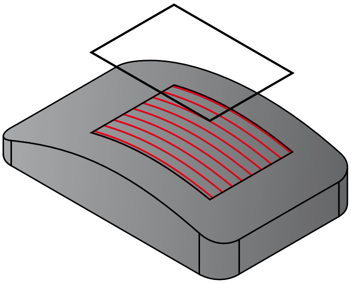
  This parameter is available for the **HM Roughing** strategy.

6.1.6 Pass extension

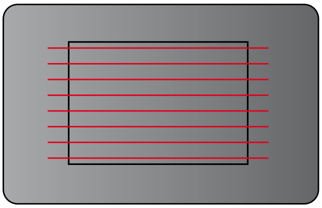
This option enables you to extend the tool path beyond the boundary to enable the tool movement into the cut with machining feed rather than rapid feed.

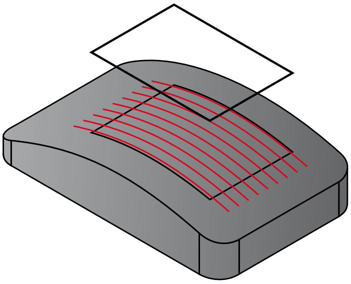
The Linear tool path shown below is created with the zero pass extension:





The Linear tool path shown below is created with 5 mm pass extension:

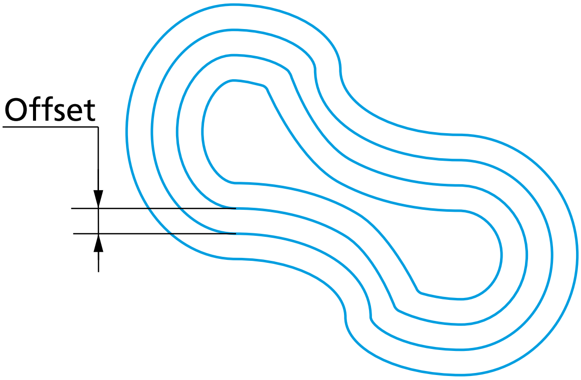




The **Pass extension** parameter is available for **Linear machining** and **Radial machining** strategies.

6.1.7 Offsets

Each Z-level comprises a "surface profile" and a series of concentric offset profiles. The minimum and maximum offset values define the range of the size of spaces between the passes. InventorCAM will choose the largest value possible within that range that does not leave unwanted upstands between the passes.

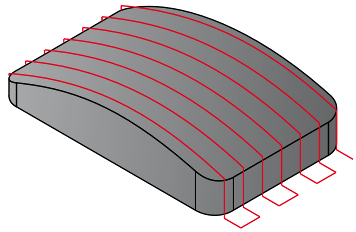


A set of **Contour roughing** passes, for example, is created from a series of offset profiles. If each profile is offset by no more than the tool radius then the whole area will be cleared. In certain cases where the profile is very smooth it is possible to offset the profiles by up to the tool diameter and still clear the area. Obviously, offsetting by more than the tool diameter will leave many upstands between the passes. Between these two extremes, the radius and the diameter, there is an ideal offset where the area will be cleared leaving no upstands. InventorCAM uses an advanced algorithm to find this ideal offset.

The **Min. Offset** value should be greater than the **Offset tolerance** (see topic **6.2.3**) parameter and smaller than the tool shaft radius; the **Max. Offset** value is calculated automatically.

This parameter is used for **Contour roughing**, **Hatch roughing** and **Horizontal finishing**.

6.1.8 Limits



The limits are the highest and lowest Z-positions for the tool – the range in which it can move.

- **Z-Bottom limit**. This parameter enables you to define the lower Z-level of the machining. The default value is automatically set at the lowest point of the model.

  This limit is used to limit the passes to level ranges or to prevent the tool from falling indefinitely if it moved off the edges of the model surface. When the tool moves off the surface, it continues at the **Z-Bottom** limit and falls no further.

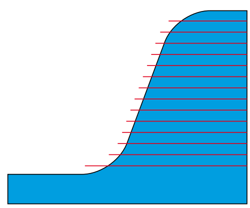
- **Z-Top limit**. This parameter defines the upper machining level. The default value is automatically determined at the highest point of the model.

- **Delta**. This parameter allows you to set a value for machining above or below the values specified in **Z-Top** and **Z-Bottom** limits.

- **CoAngle**. This parameter defines the contact angle alignment to be used when making cross machining passes.

  This option is available only for the **Linear machining** strategy.

- **Angle**. InventorCAM enables you to limit the surface angles within a range most appropriate to the strategy. The **Constant Z** strategy, for example, is most effective on steeper surfaces, because the spaces between the passes are calculated according to the **Step down** value, and on surfaces where there is little Z-level change, the spaces between the passes are greater, therefore you may get unsatisfactory results. You can limit the work area to surface angles between, for example, 30 and 90 degrees.

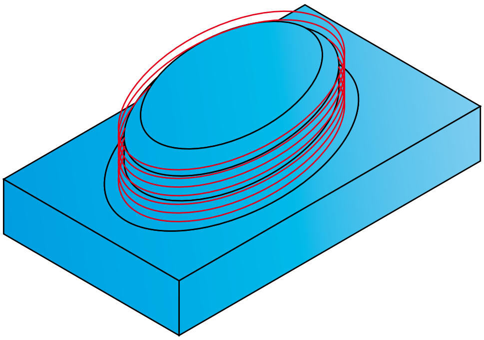
  The angle is measured between the two normals at the contact points between the tool and model faces. The angle of 0 means coincidence of surface normal and tool axis (horizontal surface).

  This option is available for the **Constant Z**, **Linear**, **Radial**, **Spiral**, **Morphed**, **Boundary**, **3D Constant Step over**, and **Pencil milling** strategies.

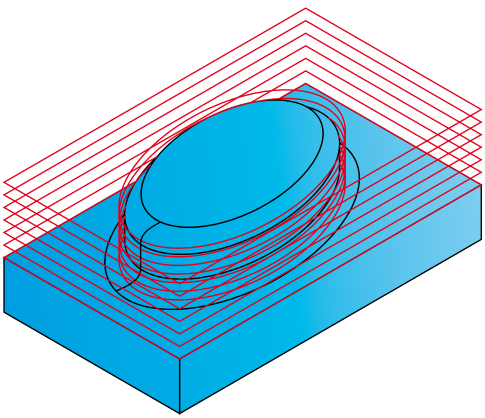
- **Contact areas only**

When this check box is selected, the tool path is only created where the tool is in contact with model faces. The examples below show the result of **Constant Z** strategy with and without the **Contact areas only** option.

When this check box is selected, the machining is limited to the actual surfaces of your geometry.

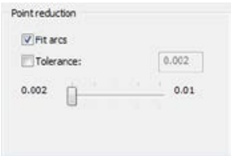


When this check box is not selected, the outer edge of the base surface is machined as well as the central boss.



6.1.9 Point reduction

InventorCAM enables you to optimize the tool path by reducing the number of points.



The **Fit arcs** option enables you to activate the fitting of arcs to the machining passes according to the specified **Tolerance** value.

The **Tolerance** value is the chordal deviation to be used for point reduction and arc fitting.

Using the slider, you can adjust the tolerance within the range defined on the **3D HSM/3D HSR** page in **InventorCAM Settings > Defaults > Tolerance**.

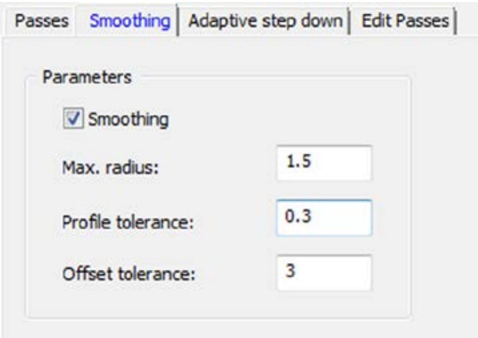
6.1.10 Refine corners

This option enables you to refine corner positions to provide a smoother tool path.

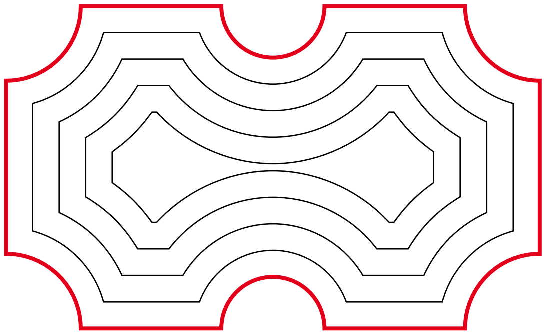
This option is available for **Contour roughing, Rest roughing** and **Horizontal machining**.

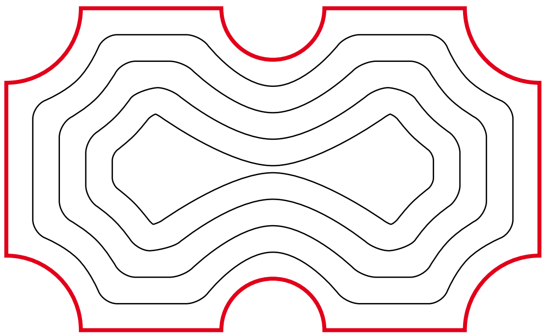
6.2 Smoothing Parameters

The **Smoothing** option enables you to round the tool path corners.



This option enables the tool to maintain a higher feed rate and reduces wear on the tool. This feature is often used in rough machining.


Tool path without smoothing


Tool path with smoothing

6.2.1 Max. radius

A curve can be approximated as an arc. The **Max. radius** parameter defines the maximum arc radius allowed.

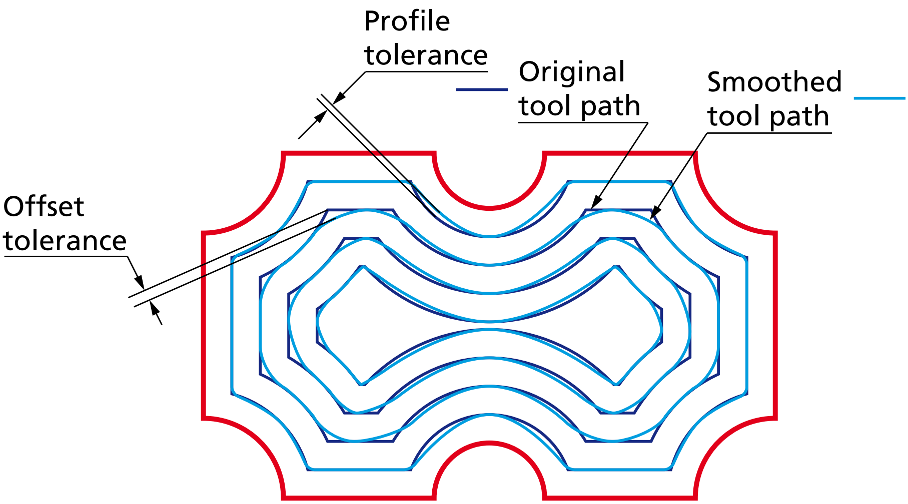
6.2.2 Profile tolerance

This value is the maximum distance that the smoothed outer profile will diverge from the actual profile. Set the **Profile tolerance** to a low or zero value to reduce the amount of material missed.

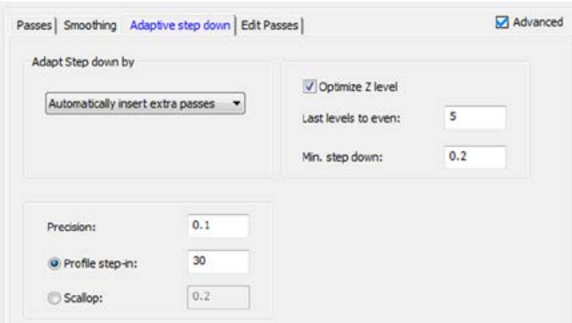
6.2.3 Offset tolerance

This value is the maximum distance that the smoothed profile offset will diverge from the inner (offset) profiles. This parameter is identical to the **Profile tolerance**, except that it refers only to the inner (offset) profiles and not to the outer profile. The **Offset tolerance** is measured between any given smoothed profile (excluding the outermost one) and the sharp corner of an imaginary profile drawn without smoothing, but at the same offset as the smoothed one.

Unlike the **Profile tolerance** parameter, above, changing this value does not mean you miss material.

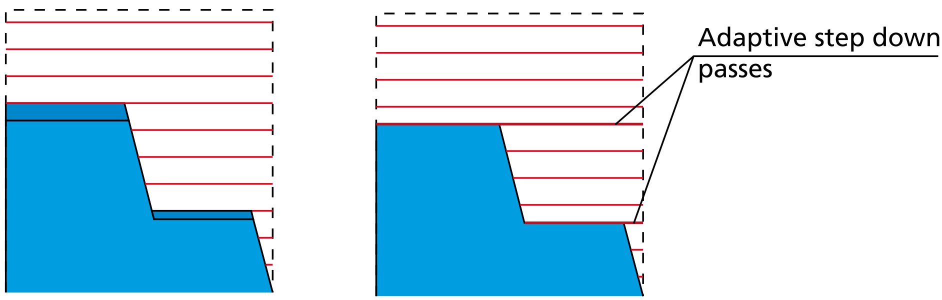


6.3 Adaptive Step Down Parameters

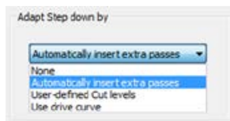


In areas where the horizontal distance between the passes is significant, **Adaptive step down** can be used to insert extra passes and reduce the horizontal distance.

In areas where the passes on the topmost edges of a surface would fall too close or too far away from that edge, **Adaptive step down** will add extra passes to compensate. So the **Step down** value controls the maximum Z-distance between the passes for the entire surface, while **Adaptive step down** adjusts those values for best fitting the surfaces.



Adaptive step down is not chosen   Adaptive step down is chosen



The **Adapt Step down by** list enables you to select the mode of the adaptive step down:

- **None**

  Passes are applied without **Adaptive step down**, and some material may be left on the top faces.

- **Automatically insert extra passes**

  A pass is inserted to cut the top face; the next step down will be calculated from this pass.

  - **Minimum step down**

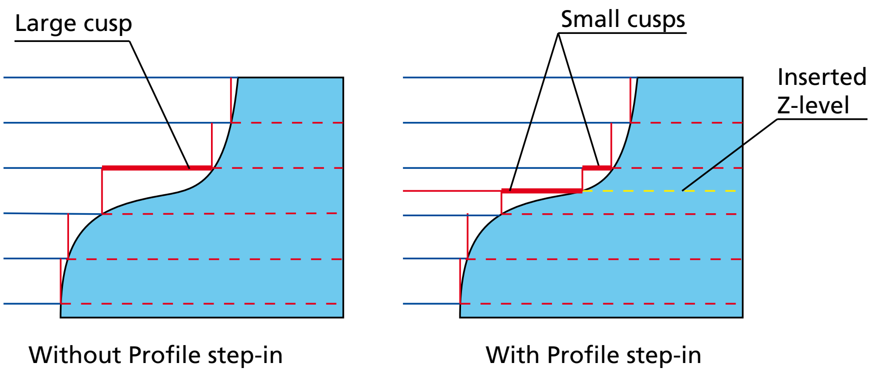
    This parameter specifies the minimum step down value to be used, which means that passes will be no less than this distance from each other.

  - **Precision**

    This parameter controls how accurately the system finds the appropriate height to insert a new slice.

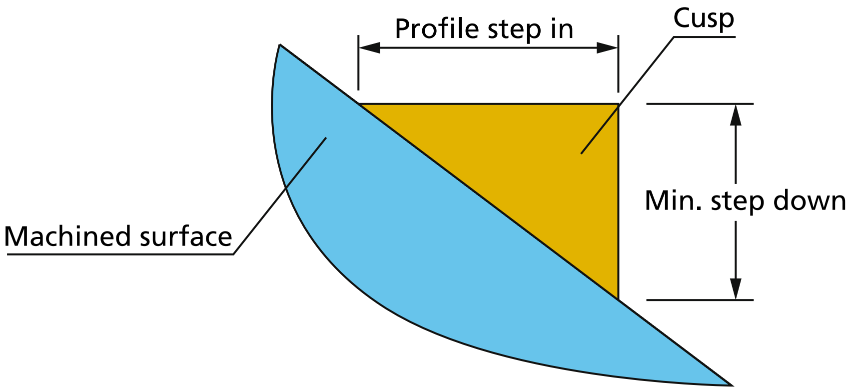
  - **Profile step-in**

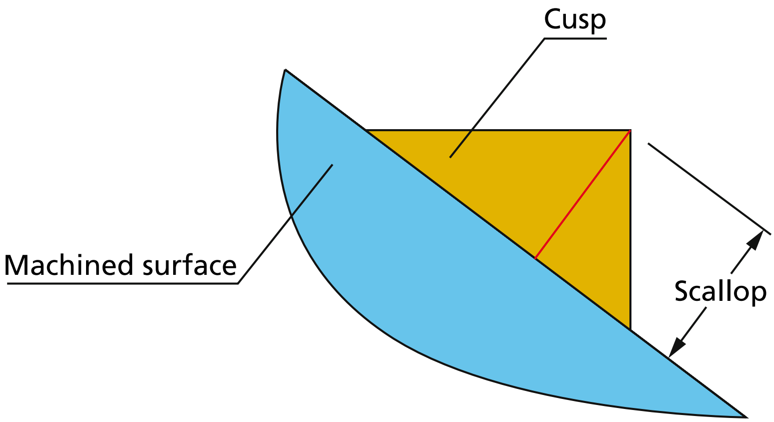
    This parameter defines the maximal XY-distance between cutting profiles located on two successive Z-levels. When InventorCAM calculates the cutting profile at a given Z-level, the distance to the cutting profile on the previous Z-level is calculated. If the calculated value is greater than the defined **Profile step-in**, InventorCAM inserts an additional Z-level and calculates the cutting profile in such way that the distance between cutting profiles located on two successive Z-levels will be smaller than the specified **Profile Step in** value.



- **Scallop**

The cusp that remains after the machining can be defined either by combination of the **Min. step down** and **Profile step in** parameters or by combination of the **Min. step down** and **Scallop** parameters. Therefore, the **Profile step in** and **Scallop** parameters are mutually exclusive.

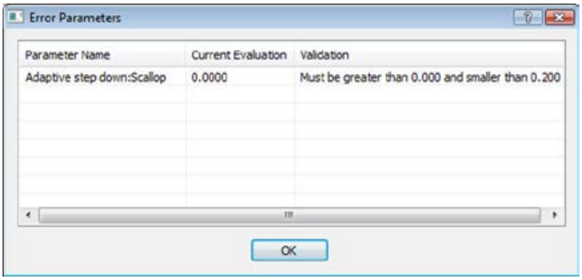




When the combination of the **Scallop** and **Min. step down** parameters is used for the operation definition, InventorCAM performs the parameters validation according to the criteria below.

- The **Scallop** value must be positive;
- The **Scallop** value must be smaller than that of the **Min. step down** parameter.

If the **Scallop** parameter does not match the validation criteria, the **Error Parameters** dialog box is displayed during the operation calculation. This dialog box specifies the parameters defined incorrectly and prompts you to edit these parameters definition.



- **User-defined Cut levels**

  The cut levels can be edited manually and inserted in the **User-defined cut levels** table. Type the Z-levels of your choice into the **Z** column of the table. The values will be sorted in the decreasing order.

  Passes are applied without **Adaptive step down**, and some material may be left on the top faces.

  This option is available for HSR strategies of **Contour** and **Hatch roughing** and for HSM strategies of **Hybrid Constant Z**, **Prismatic Part machining** and **Helical machining**.

- **Use drive curve**

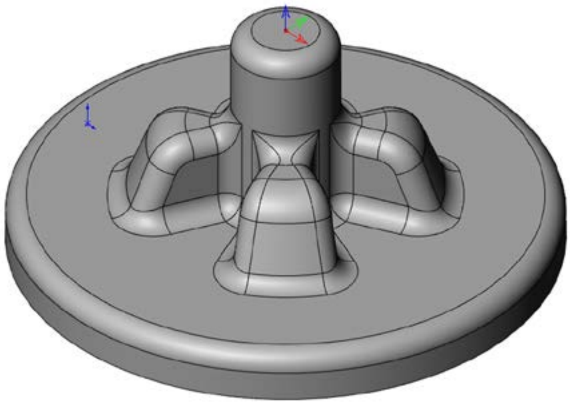
  This option enables you to use a drive curve to determine the heights at which passes are calculated. Click and choose any open or closed curve to define the geometry.
  Using this option maintains consistency while machining a curve, automatically giving a smaller step down in the shallower areas.

  When you choose the **Use drive curve** option, the **Step down** parameter cannot be defined manually.

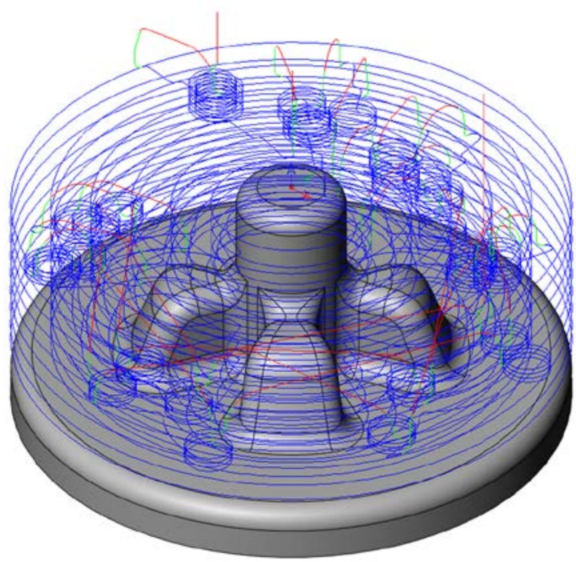
6.4 Edit Passes Parameters

If you start the machining with a formed stock instead of a rectangular or cylindrical block of material, you could trim the passes to the formed stock faces to avoid unnecessary air cutting. The tool path trimming is used either when you use a casting as stock for the part machining or you use the updated stock resulting from a number of previous operations.

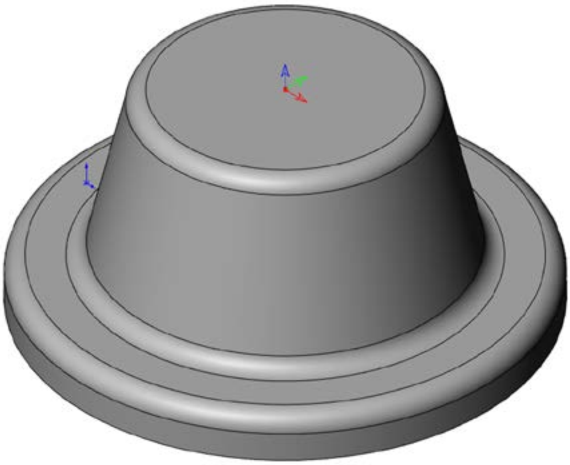
For example, suppose you want to machine by **Contour roughing** the following model:



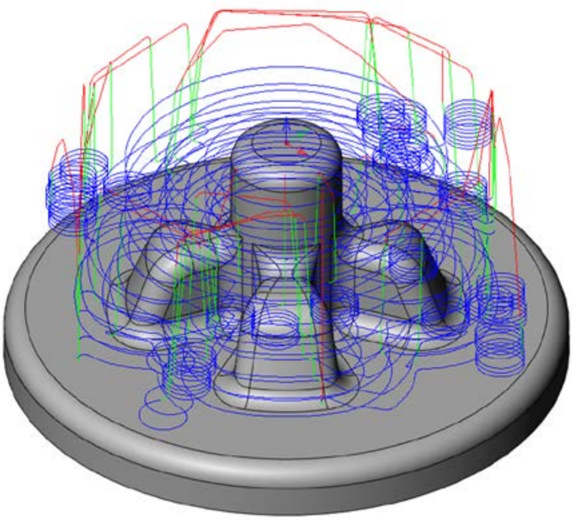
Using the **Contour roughing** strategy, you get the following tool path.



Rather than starting from a cylindrical block of material, you start with the casting shown below.

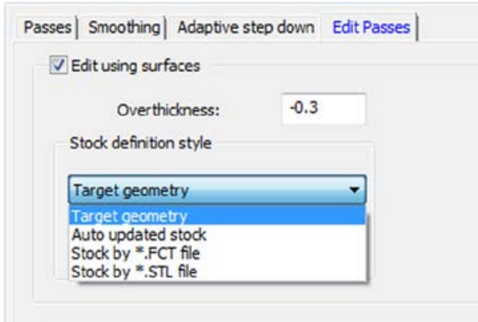


The resulting trimmed tool path is shown below.



The **Edit Passes** page enables you to define the parameters for the trimming of passes.

Edit using surfaces



By selecting this check box, you can limit the machining by using the Updated Stock model or by defining an offset from the operation geometry.

Overthickness

This parameter defines an extra thickness that can be temporarily applied to the tool and can be set when editing passes. The use of this parameter can help create better trimmed passes. A negative value will cause the system to select only passes that are below the model faces by the specified amount, while a positive value will select all passes that are within the specified distance from the model faces.

Stock definition style

This option enables you to specify the method of machining area definition.

- When the **Target geometry** option is chosen, InventorCAM adds the **Overthickness** value as an offset to the target geometry of current operation. This offset target is used as a stock.
- When the **Auto updated stock/Automatically** option is chosen, InventorCAM calculates the updated stock model after all the previous operations. The **Overthickness** value is added as offset to the stock, which will be used as stock for the current operation.
- When the **Stock by *.FCT file** option is chosen, machining is performed in the area defined by an offset from the updated stock, defined in FCT file located in the CAM-Part folder. The offset is defined by the **Overthickness** parameter.
- When the **Stock by *.STL file** option is chosen, machining is performed in the area defined by an offset from the updated stock, defined in STL file located in the CAM-Part folder. The offset is defined by the **Overthickness** parameter.
  - **Show**

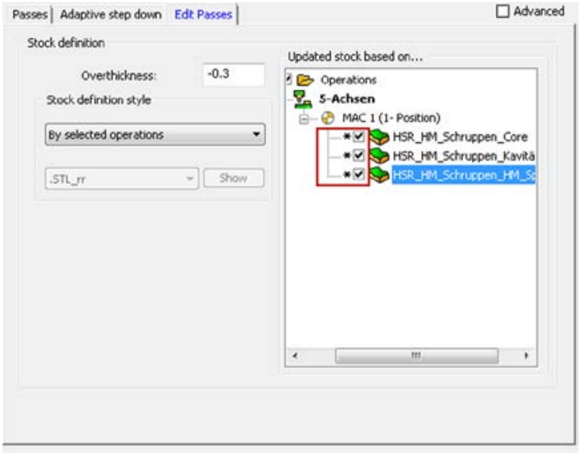
    This button displays the difference between the updated stock model and the target geometry used in the operation.

Stock definition in Rest roughing

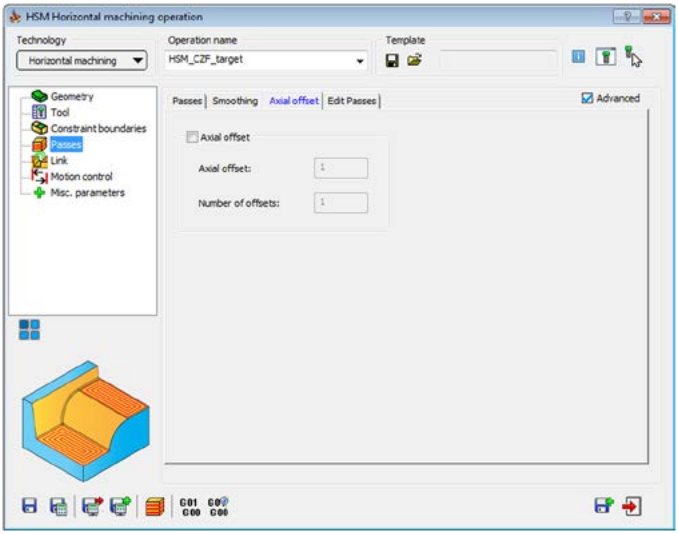
In the **Rest roughing** operation, you can control the number of recalculations and thus save time.

You can choose how the stock for the operation is defined: either by selected operations or automatically.

- When you choose the **Automatically** option from the list, the updated stock is obtained after processing all previous operations.
- When you choose the **By selected operations** option, you can see all operations preceding the current operation in a fragment of the CAM-tree. Any previous operation, whether calculated or not, can be selected for updated stock definition. Select the operations by clicking the check box near the operation name. You can also select all operations by clicking the check box near the **Operations** header.



6.5 Axial Offset



This page enables you to axially offset the tool path (one or several times). The tool path can be generated by any of the HSM finish strategies, except for **Constant Z** and **Rest machining**.

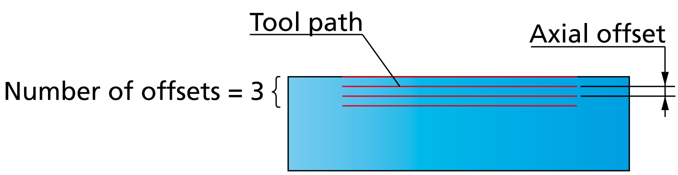
When the **Axial offset** check box is selected, you have to define the following parameters:

- **Axial offset**

  This parameter defines the distance between two successive tool path passes.

- **Number of offsets**

  This parameter enables you to define how many times the offset of the tool path is performed. This final number of tool path passes is equal to **Number of offsets +1**.



The tool path passes are generated in the positive Z-direction. The machining is performed from the upper instance to the lower.

The **Axial offset** feature enables you to perform the semi-finish and finish machining in a number of equidistant vertical steps. It can be used for engraving in a number of vertical steps with the **Boundary machining** strategy or for removing the machining allowance by a finishing strategy in a number of vertical steps.

6.6 Strategy Parameters

In addition to the common parameters relevant for all of the machining strategies, InventorCAM provides you with options and parameters that enable you to control specific features of various machining strategies.

Roughing strategies (HSR):

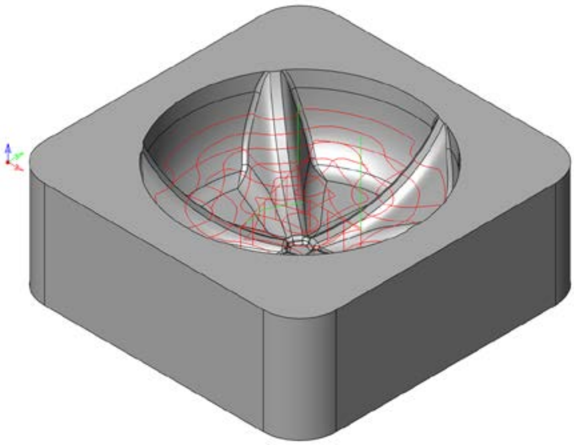
- HM Roughing
- Contour roughing
- Hatch roughing
- Hybrid Rib roughing
- Rest roughing

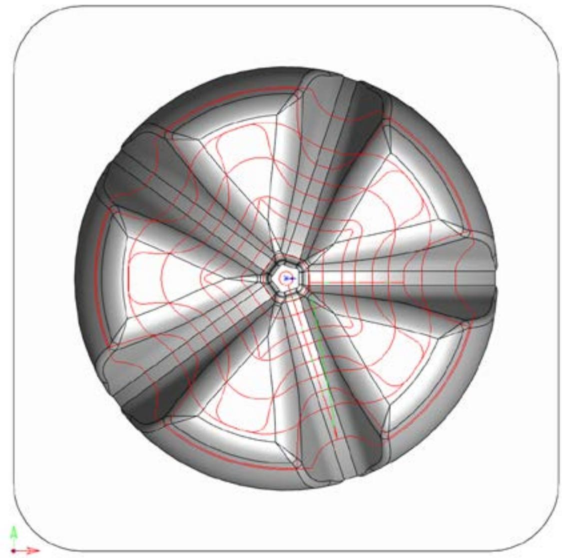
Finishing strategies (HSM):

- Constant Z machining
- Hybrid Constant Z
- Helical machining
- Horizontal machining
- Linear machining
- Radial machining
- Spiral machining
- Morphed machining
- Offset cutting
- Boundary machining
- Rest machining
- 3D Constant step over
- Pencil milling
- Parallel pencil milling
- 3D Corner offset
- Prismatic Part machining
- Combined strategies

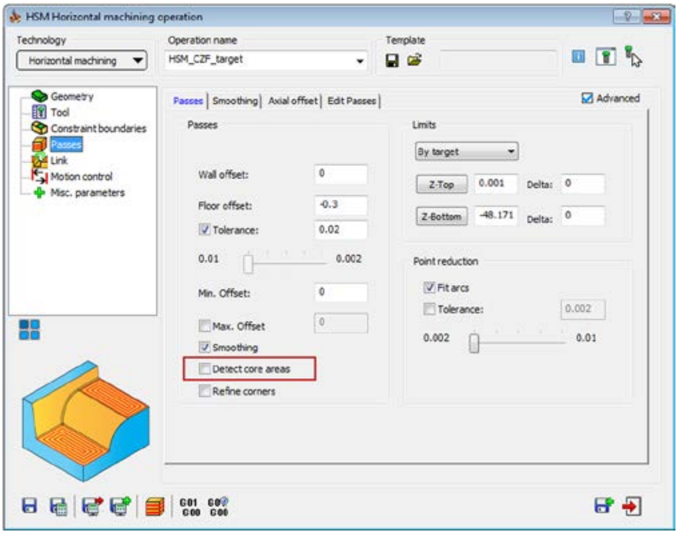
6.6.1 Contour roughing

With the **Contour roughing** strategy, InventorCAM generates a pocket-style tool path for a set of sections generated at the Z-levels defined with the specified **Step down** (see topic **6.1.4**).





Detect core areas



This option causes the tool to start from the outside of the model rather than take a full width cut in the center of the component.

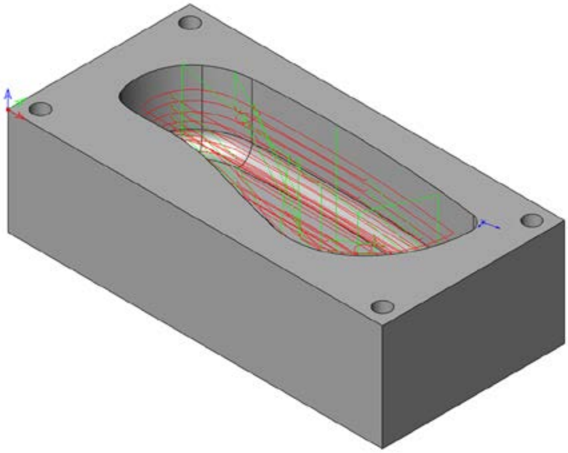
If your model includes both core and cavity areas, the system will automatically switch between core roughing and cavity roughing within the same tool path.

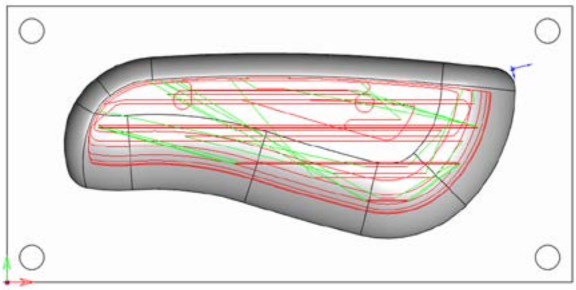
When these passes are linked to create a **Contour roughing** tool path, the areas are machined from the top downwards. Obviously, material has to be machined at one level before moving down to the next one.

The passes for the **Z-Top level** machining are not usually included in the operation tool path. Adjust the **Z-Top level** by adding the **Step down** value to the current **Z-Top level** value when you want to include the top level passes in the operation tool path.

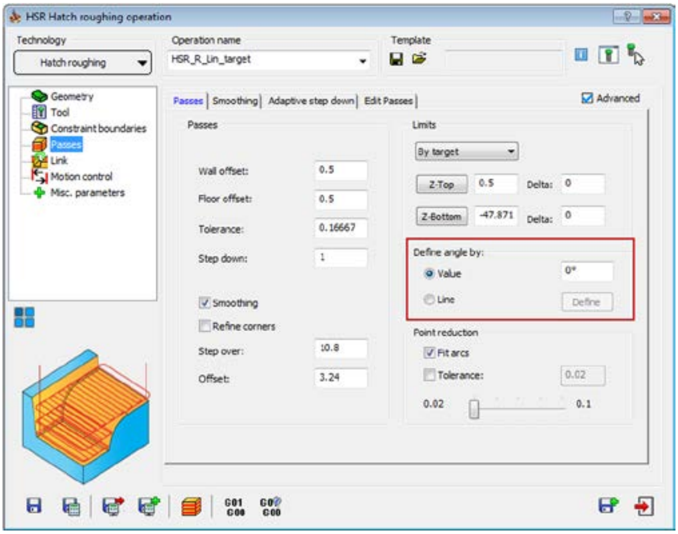
6.6.2 Hatch roughing

With the **Hatch roughing** strategy, InventorCAM generates linear raster passes for a set of sections generated at the Z-levels defined with the specified **Step down** (see topic **6.1.4**). **Hatch roughing** is generally used for older machine tools or softer materials because the tool path predominantly consists of straight line sections.

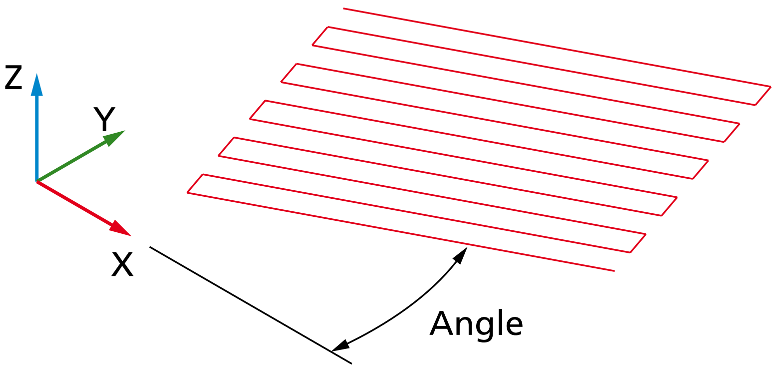




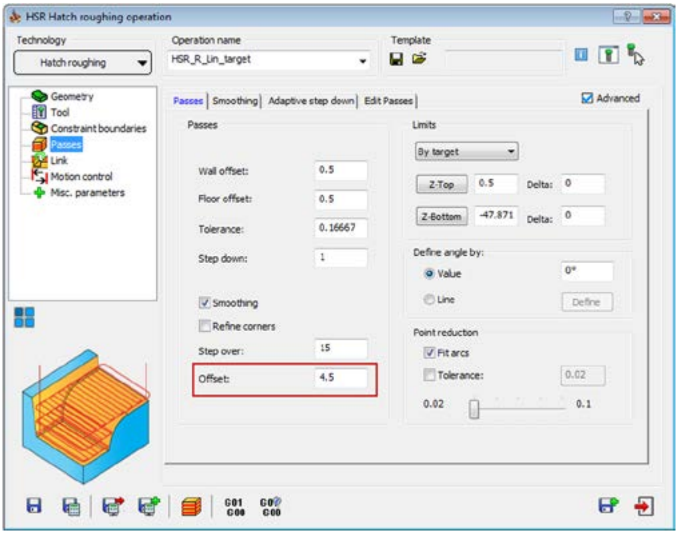
Angle



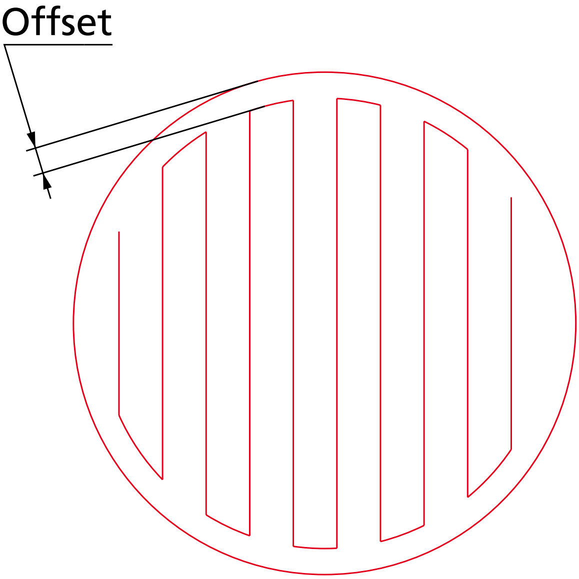
This option enables you to define the angle of the hatch passes relative to the X-axis of the current Coordinate System.



Offset

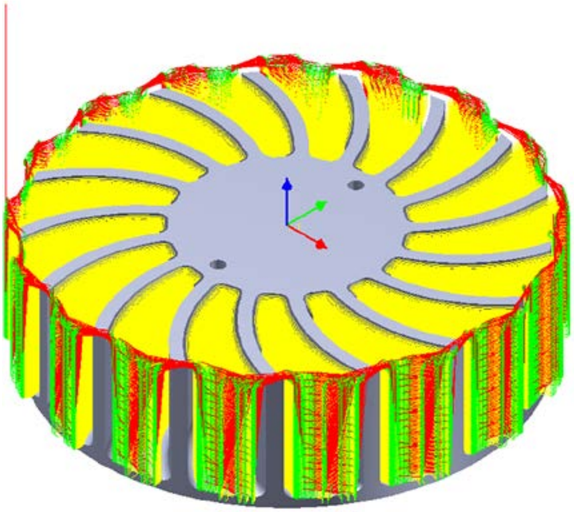


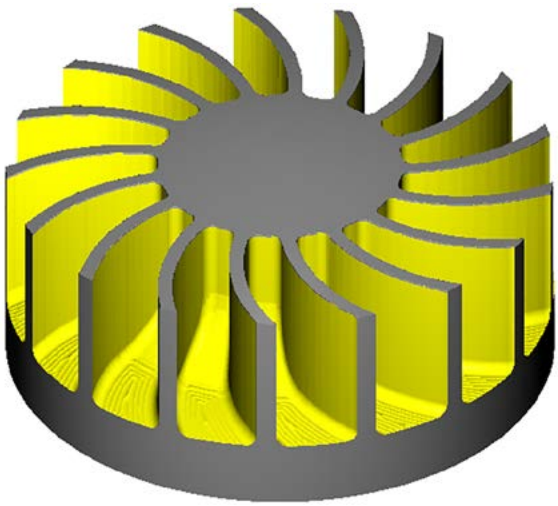
The **Offset** parameter defines the distance between the hatch passes and the outer/inner profiles.



6.6.3 Hybrid Rib roughing

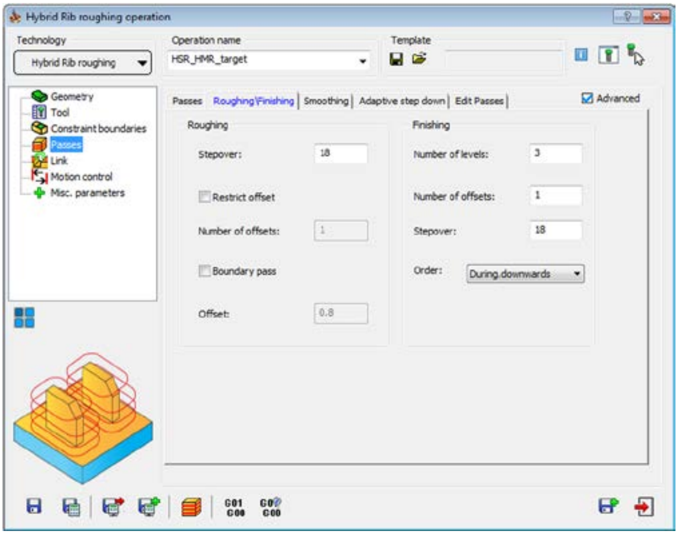
The **Hybrid Rib** roughing is a strategy designed to machine very thin walls. These walls are made of exotic materials (titanium, graphite, etc.) and therefore a traditional approach to their machining can be difficult and risky. This strategy combines a new roughing and finishing tool path, creating a unique tool path that should preserve the highest possible rigidity of the part.





Roughing/Finishing parameters

In **Hybrid Rib** operation, each Z-level is machined with roughing and finishing passes after which machining of a lower level is performed.



The **Roughing** section enables you to control roughing passes and levels performed against a rib.

- **Stepover**

  This parameter defines the distance between two parallel tool path passes.

- **Restrict offset**

  When the **Restrict offset** option is selected, you can limit the number of parallel passes by specifying the value in the **Number of offsets** field.

- **Boundary pass**

  When this option is selected, an additional pass is performed around the outside perimeter of the constraint boundary on each roughing level.

- **Offset**

  This parameter enables you to define an offset applied to the constraint boundaries from outside.

The **Finishing** section enables you to control finishing passes and levels performed against a rib.

- **Number of levels**

  This parameter enables you to define the number of finishing levels required between step downs.

- **Number of offsets**

  This parameter enables you to define the number of finishing offsets required at each level.

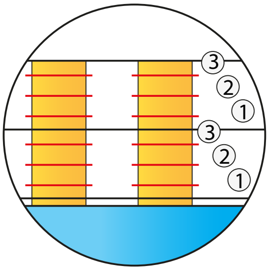
- **Stepover**

  This parameter defines the distance between two parallel finishing passes.

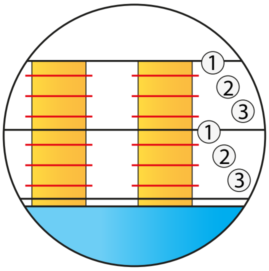
- **Order**

  This parameter enables you to control the order of linking the finishing passes. The following options are available:

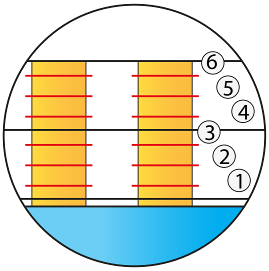
  - **During, upwards**: after each roughing level, the finishing levels are machined from lowest to highest



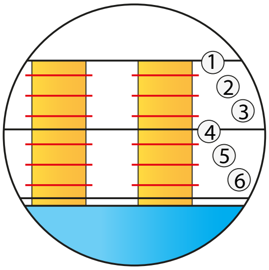
  - **During, downwards**: after each roughing level, the finishing levels are machined from highest to lowest



  - **After, upwards**: all roughing levels are machined first, followed by the finishing levels from lowest to highest



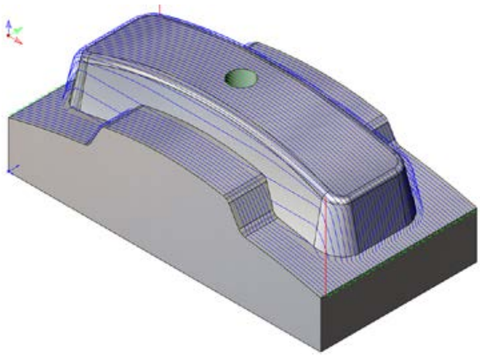
  - **After, downwards**: all roughing levels are machined first, followed by the finishing levels from highest to lowest.



6.6.4 Linear machining

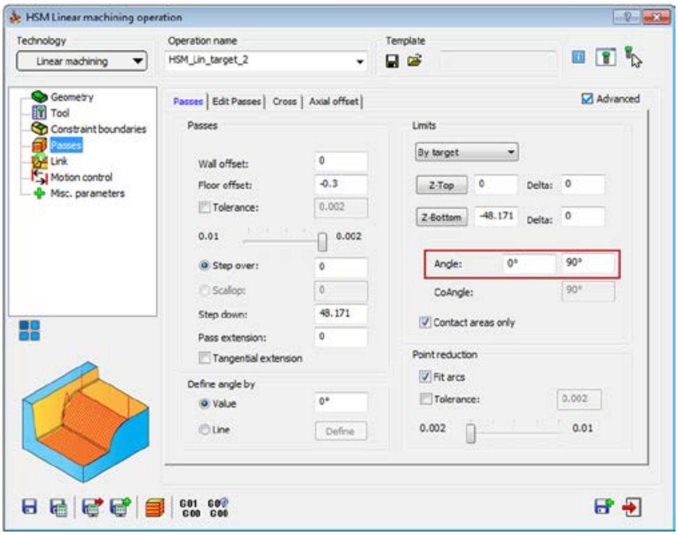
Linear machining generates a tool path consisting of a set of parallel passes at a given angle with the distance between the passes defined by the **Step over** parameter (see topic **6.1.5**).

With the **Linear machining** strategy, InventorCAM generates a linear pattern of passes, where each pass is oriented at a direction defined with the **Angle** value. This machining strategy is most effective on shallow (nearly horizontal) surfaces, or steeper surfaces inclined along the passes direction. The Z-height of each point along a raster pass is the same as the Z-height of the triangulated surfaces, with adjustments made for applied wall offset and tool definition.



In the image, the passes are oriented along the X-axis. The passes are evenly spaced on the shallow faces and on the faces inclined along the passes direction. The passes on the side faces are widely spaced; **Cross linear machining** can be used to finish these areas.

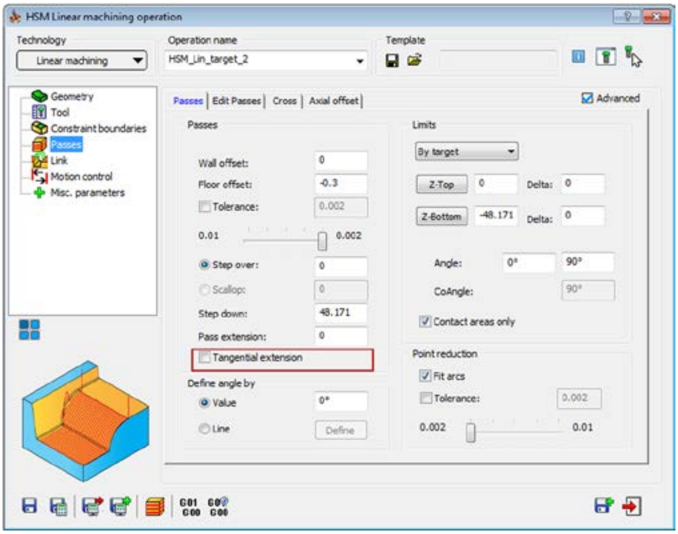
Angle



The **Angle** parameter enables you to define the angle of the passes direction. The value of this parameter is within the range of **–180°** to **180°**. If **Angle** is set to **0°**, the direction of passes is parallel to the X-axis of the current Coordinate System. The order of the passes and the direction of the machining is controlled by the link settings.

The defined angle affects the step over calculation. If you are machining vertical surfaces, **Linear machining** works best where the angle is perpendicular to these surfaces.

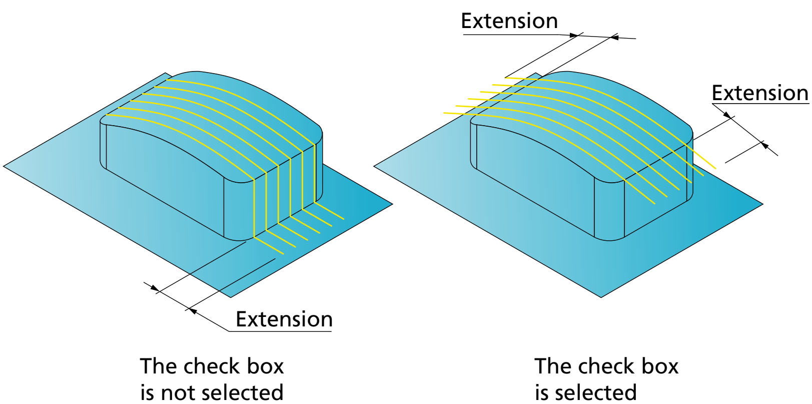
Tangential extension



This option enables you to extend the passes tangentially to the model faces by a length defined by the **Pass extension** parameter.

When the check box is not selected, the extension passes are generated as a projection of the initial pattern (either linear or radial) on the solid model faces.

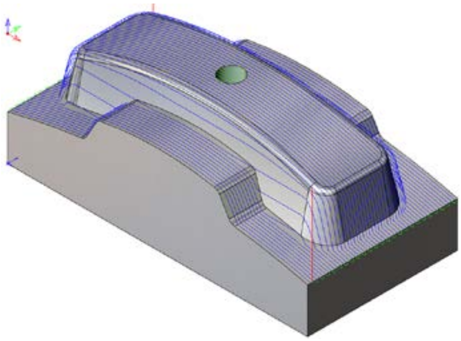
When the check box is selected, the extension passes are generated tangentially to the solid model faces.



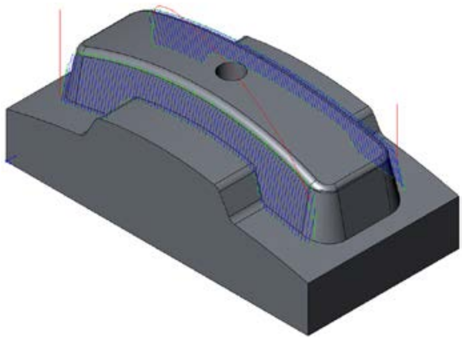
Cross linear machining

InventorCAM automatically determines the areas where the **Linear machining** passes are sparsely spaced and performs in these areas an additional Linear tool path in a direction perpendicular to the direction of the initial Linear tool path. The passes parameters used for the **Cross linear machining** definition are the same as those used for the initial **Linear machining**.

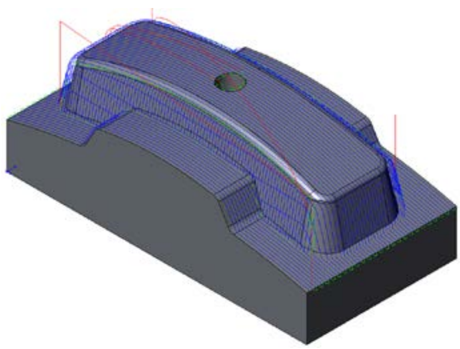
Initial Linear machining tool path



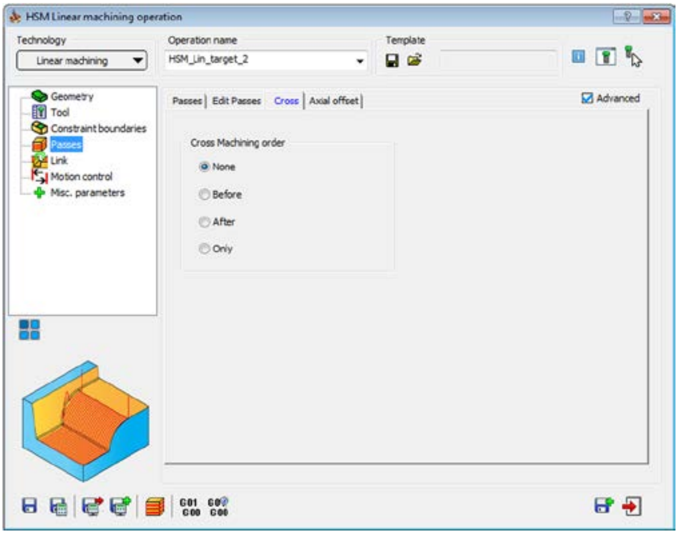
Cross linear machining tool path



Combined Linear and Cross linear machining tool path



Cross page



The **Cross** page enables you to define the order of performing **Linear** and **Cross linear machining**.

- **None**

  Cross linear machining is not performed.

- **Before**

  Cross linear machining is performed before the main Linear machining.

- **After**

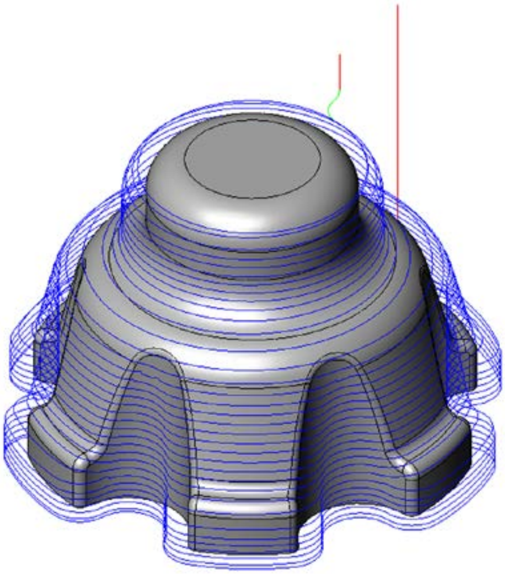
  Cross linear machining is performed after the main Linear machining.

- **Only**

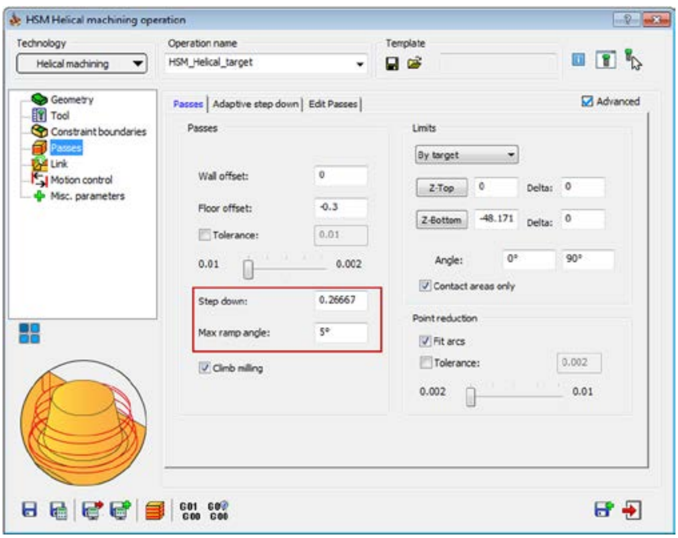
  Only Cross linear machining is performed; the main Linear machining is not performed.

6.6.5 Helical machining

This strategy enables you to generate a number of closed profile sections of the 3D Model geometry located at different Z-levels, similar to the **Constant Z** strategy. Then these sections are joined in a continuous descending ramp in order to generate the Helical machining tool path.



The tool path generated with the **Helical machining** strategy is controlled by two main parameters: **Step down** and **Max. ramp angle**.

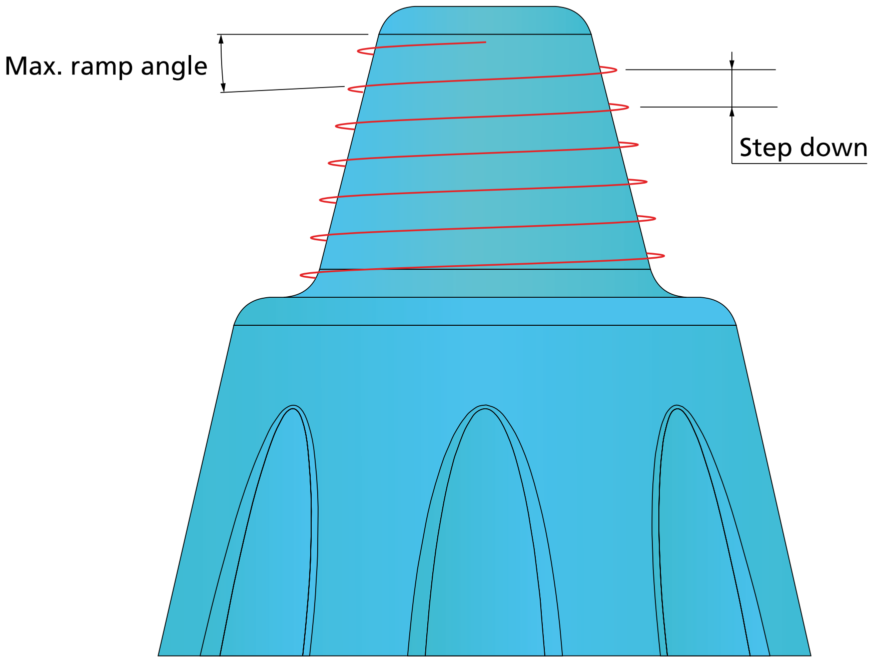


Step down

This parameter defines the distance along the Z-axis between two successive Z-levels, at which the geometry sections are generated. Since the **Step down** is measured along the Z-axis (similar to the **Constant Z** strategy), the **Helical machining** strategy is suitable for steep areas machining.

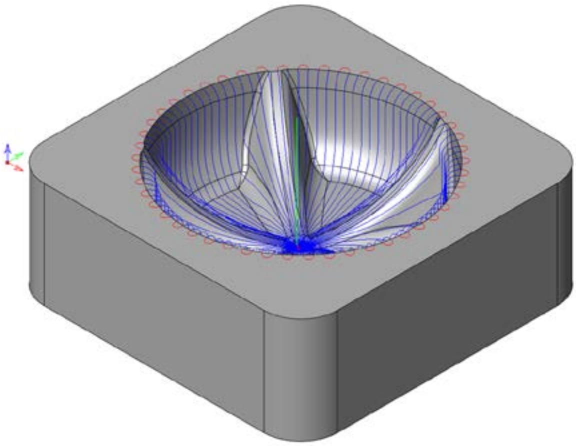
Max. ramp angle

This parameter defines the maximum angle (measured from horizontal) for ramping. The descent angle of the ramping helix will be no greater than this value.

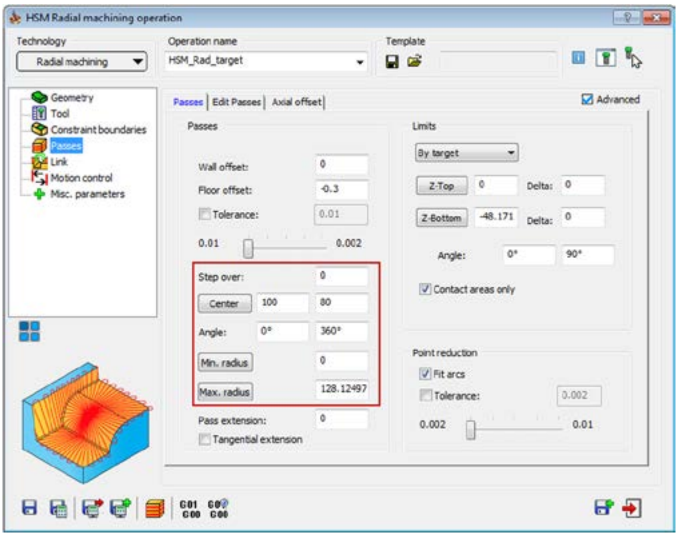


6.6.6 Radial machining

The **Radial machining** strategy enables you to generate a radial pattern of passes rotated around a central point.



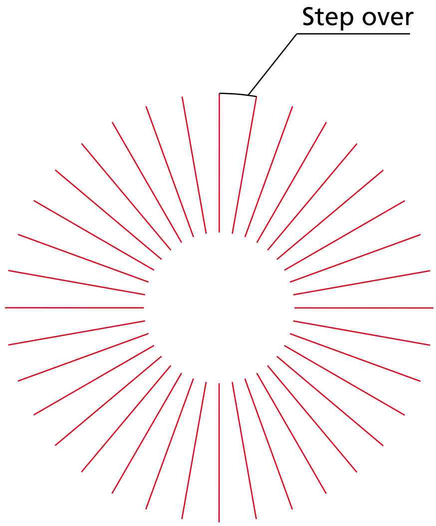
This machining strategy is most effective on areas that include shallow curved surfaces and for model areas formed by revolution bodies, as the passes are spaced along the XY-plane (**Step over**), and not the Z-plane (**Step down**). The Z-height of each point along a radial pass is the same as the Z-height of the triangulated surfaces, with adjustments made for applied offset and tool definition.



Step over

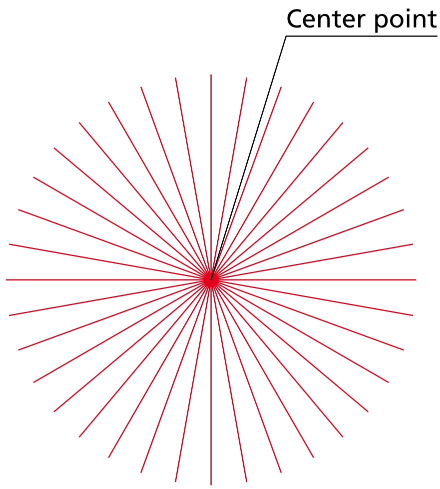
Step over is the spacing between the passes along the circumference of the circle.

The passes are spaced according to the **Step over** value measured along the circle defined by the **Maximum radius** value.



Center

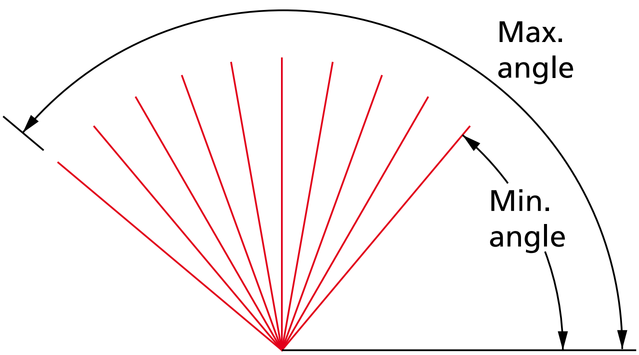
You must specify the XY-position of the center point of the radial pattern of passes. The **Radial passes** will start or end in this center point.



Angle

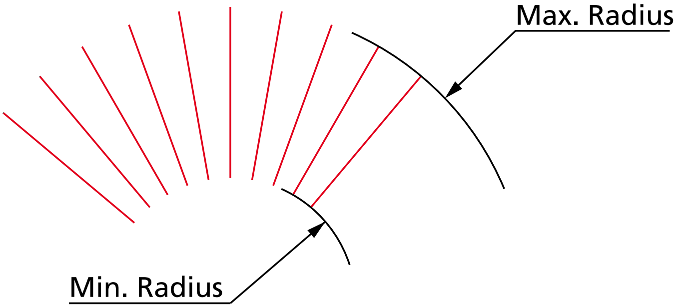
The minimum and maximum angles enables you to define start and end of the pattern passes. These parameters control the angle span of the operation, that is, how much of a complete circle will be machined.

The angles are measured relative to the X-axis in the center point in the counterclockwise direction.



Radii

The **Min. radius** and **Max. radius** values enable you to limit the tool path in the radial direction.



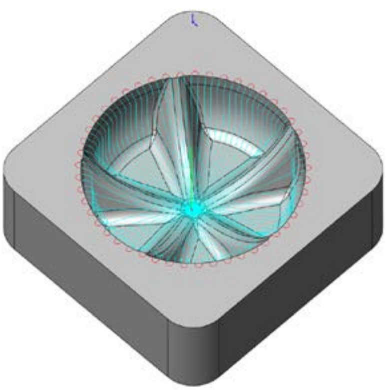
The diagram above shows the effect of different minimum and maximum radii on Radial passes.

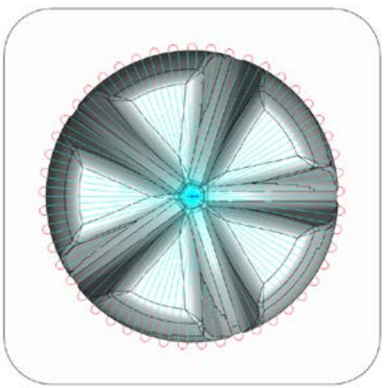
You can define the radii by entering the values or by clicking the buttons and picking points on the model. The X- and Y-coordinates of this point are displayed in the **Select a coordinate** dialog box.

When this dialog box is confirmed, the radius value appears in the appropriate edit box calculated as the distance from the defined center point of the radial pattern of passes.

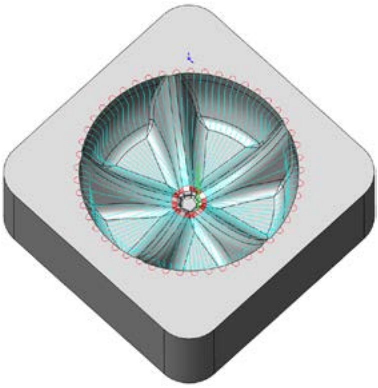
You can use the **Min. radius** value to protect the part faces from over-machining in the central point and around it. Alternatively, you can define boundaries to limit the machining.

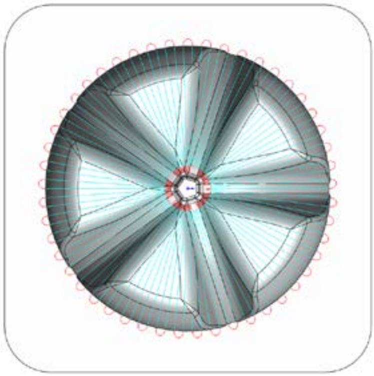
Over-machining is visible in the center point:





The tool path is limited at the center point area using a boundary, or by increasing the **Min. radius** value:





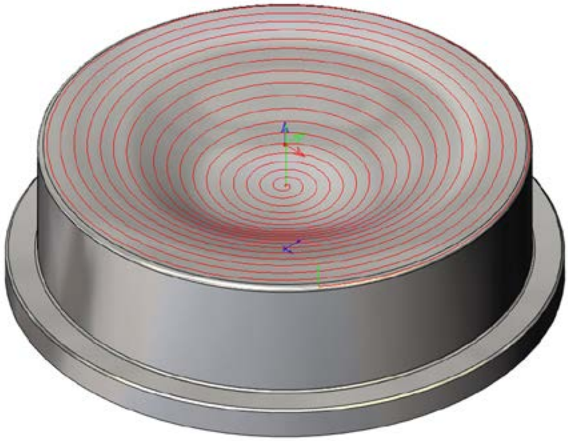
You can use another strategy (e.g. **3D Constant step over)** to machine the central area.

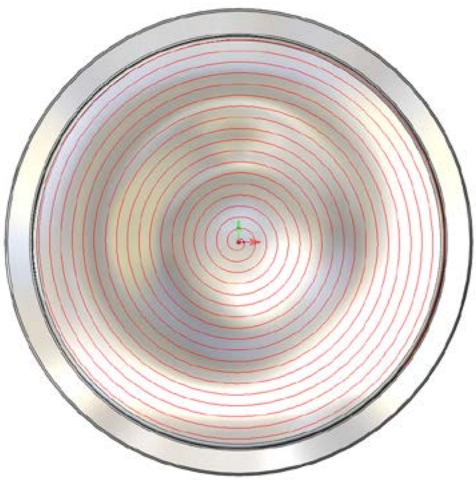
Tangential extension

This option enables you to extend the passes tangentially to the model faces by a length defined by the **Pass extension** parameter (see topic **6.6.4**).

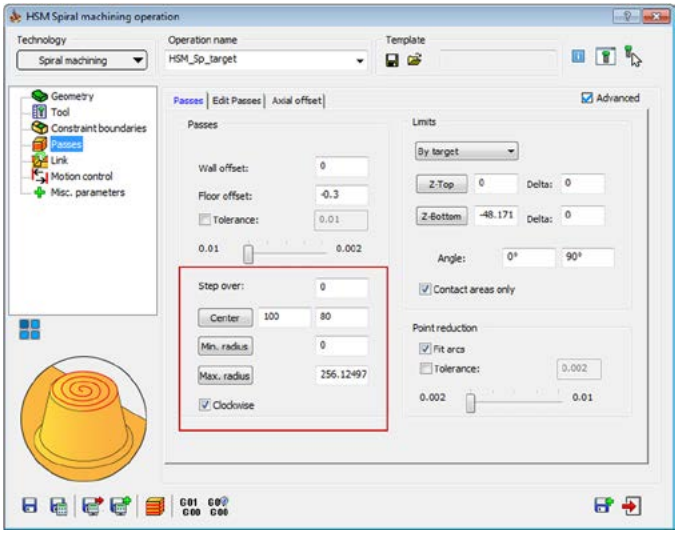
6.6.7 Spiral machining

The **Spiral machining** strategy enables you to generate 3D spiral tool path over your model. This strategy is optimal for model areas formed by revolution bodies. The tool path is generated by projecting a planar spiral (located in the XY-plane of the current Coordinate System) on the model.

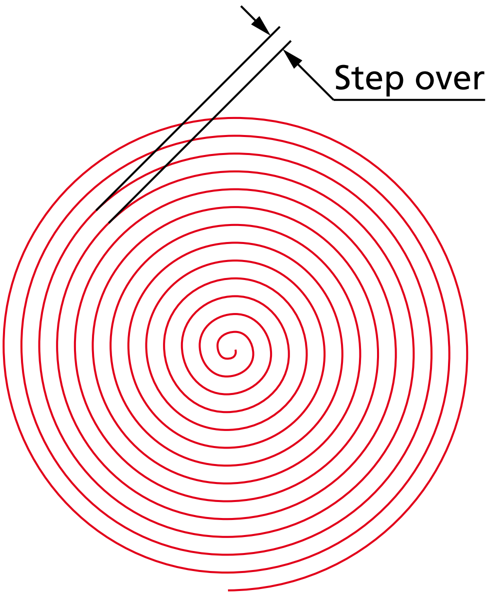




Step over

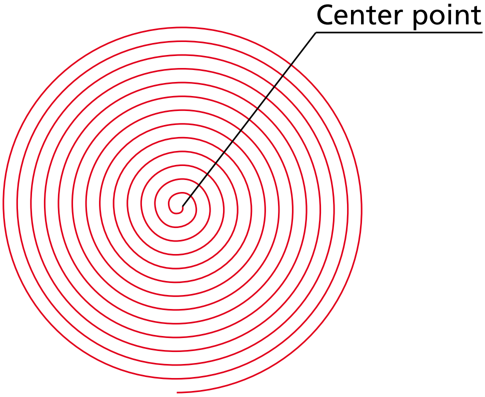


The **Step over** parameter defines the distance between two adjacent spiral turns in the XY-plane of the current Coordinate System.



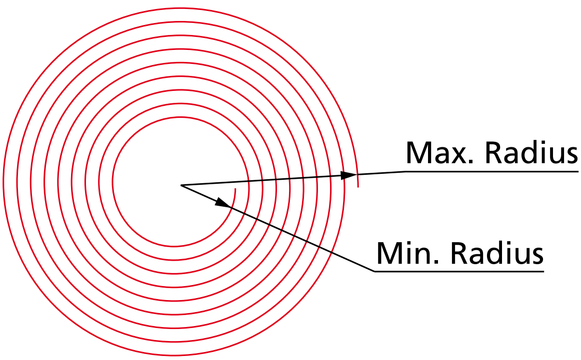
Center

You have to specify the XY-position of the center point of the spiral. The spiral tool path is calculated from this point, even if it does not actually start from there (minimum radius may be set to a larger value).

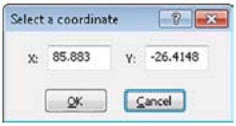


Radii

Define the area to be machined by the spiral by setting the minimum and maximum Radii. If the spiral is to start from the center point, set the **Minimum radius** value to 0. When the spiral is to start further from the center, enter the distance from the center point by setting the **Minimum radius** to a higher value. Control the overall size of your spiral with the **Maximum radius** value.



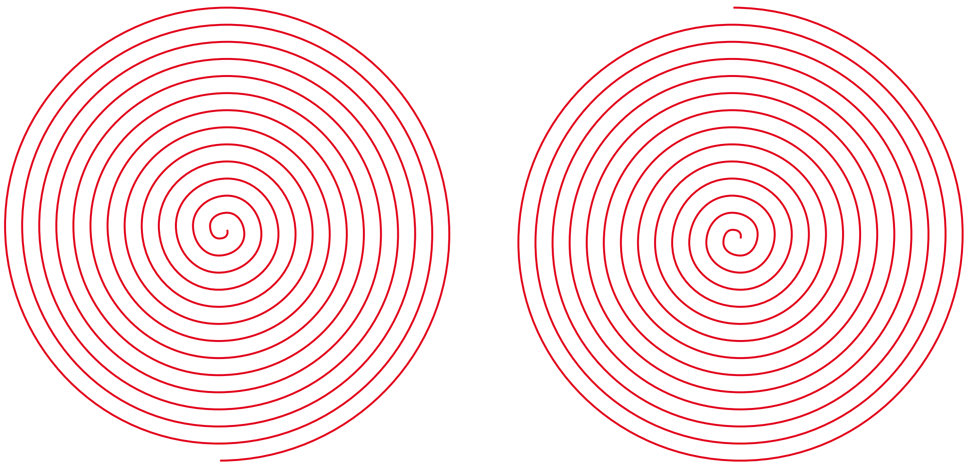
You can define the radii by entering the values or by clicking the buttons and picking points on the model. The X- and Y-coordinates of this point are displayed in the **Select a coordinate** dialog box.



When this dialog box is confirmed, the radius value appears in the appropriate edit box calculated as the distance from the defined center point of the spiral pattern of passes.

Clockwise

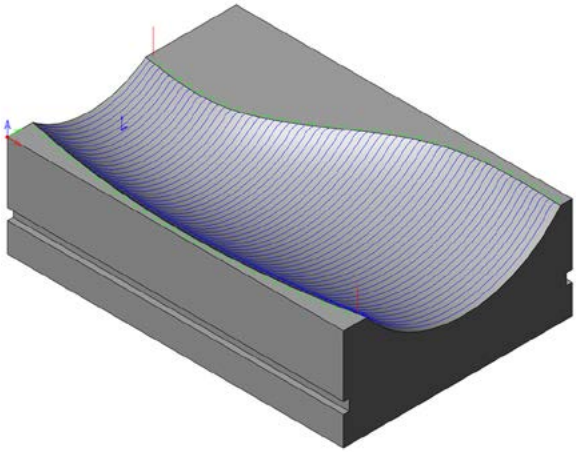
This option enables you to define the direction of the spiral. When this check box is selected, InventorCAM generates a spiral tool path in the clockwise direction. When this check box is not selected, InventorCAM generates a spiral tool path in the counterclockwise direction.



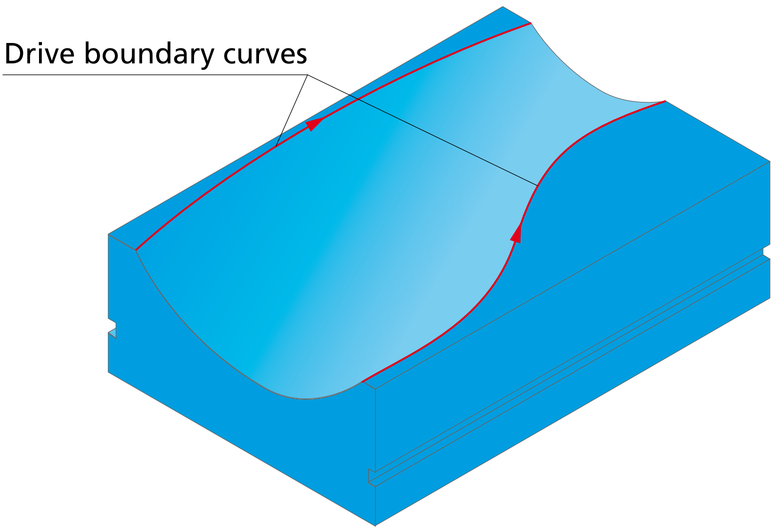
Clockwise direction   Counterclockwise direction

6.6.8 Morphed machining

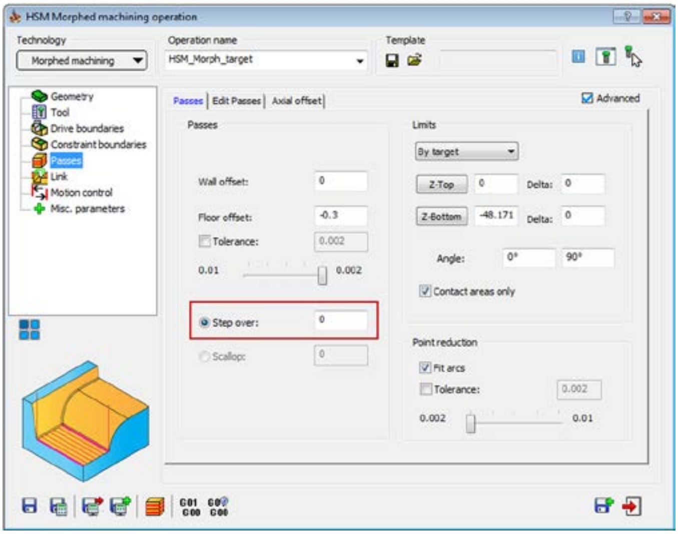
Morphed machining passes are generated across the model faces in a close-to-parallel formation, rather like **Linear machining** passes (see topic **2.10**); each path repeats the shape of the previous one and takes on some characteristics of the next one, and so the passes "morph" or gradually change shape from one side of the patch to the other.



The shape and direction of the patch is defined by two drive boundary curves.



Step over

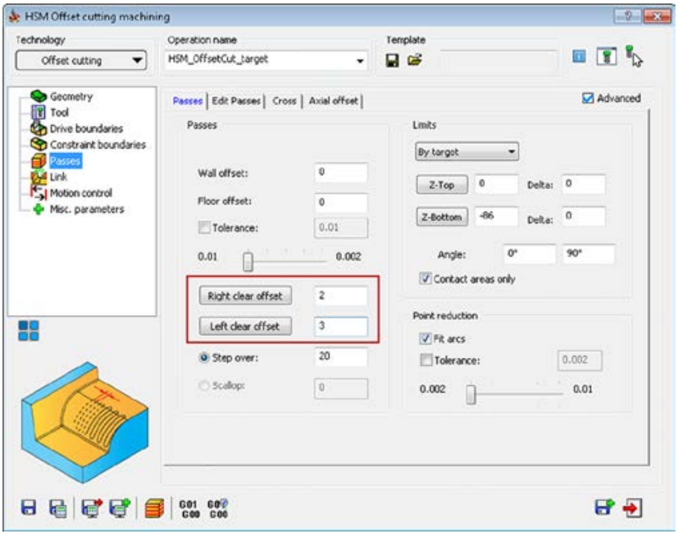


This parameter defines the distance between each two adjacent passes and is measured along the longest drive boundary curve; for the other drive boundary curve the step over is calculated automatically. For best results, the two drive boundaries should be as close in length as possible.

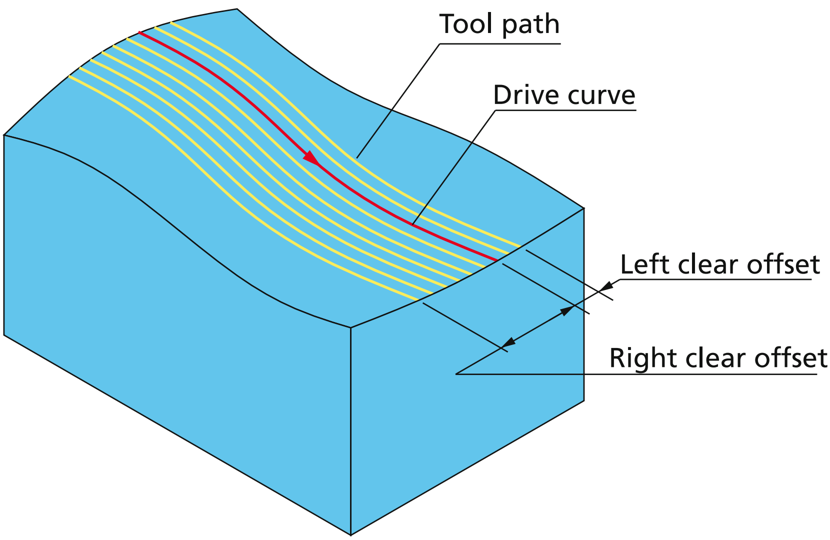
This machining strategy is most effective on areas that include shallow surfaces as the passes are spaced along the XY-plane (**Step over**) and not the Z-plane (**Step down**).

6.6.9 Offset cutting

The **Clear offset** parameters enable you to define the offset distance used for the virtual offset curve calculation.



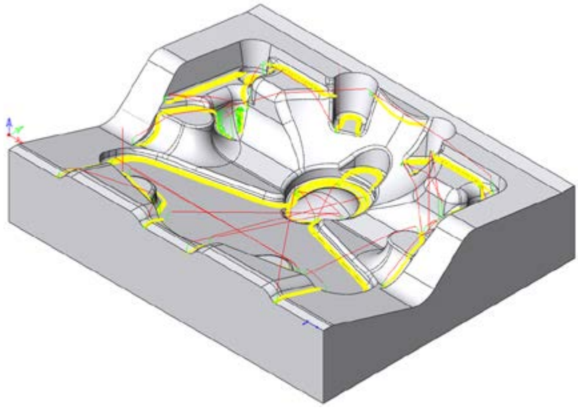
InventorCAM enables you to define separate values for the **Left clear offset** and **Right clear offset**. These offsets are activated when you choose a **Clear direction** in the **Drive boundaries** page.



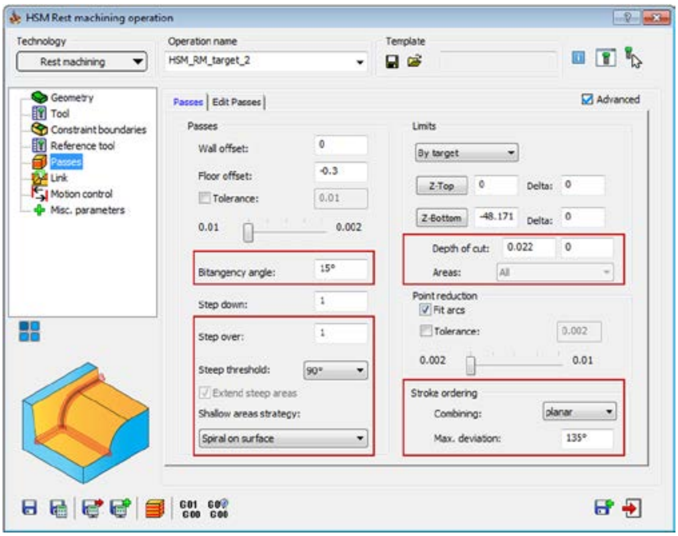
6.6.10 Rest machining

Rest machining determines the model areas where material remain after the machining by a tool path, and generates a set of passes to machine these areas.

Pencil milling vertical corners can cause both the flute of the tool and the radius to be in full contact with the material, creating adverse cutting conditions. **Rest machining** machines the corners from the top down, resulting in better machining technique. Steep and shallow areas are both machined in a single tool path, with different **Rest machining** strategies.

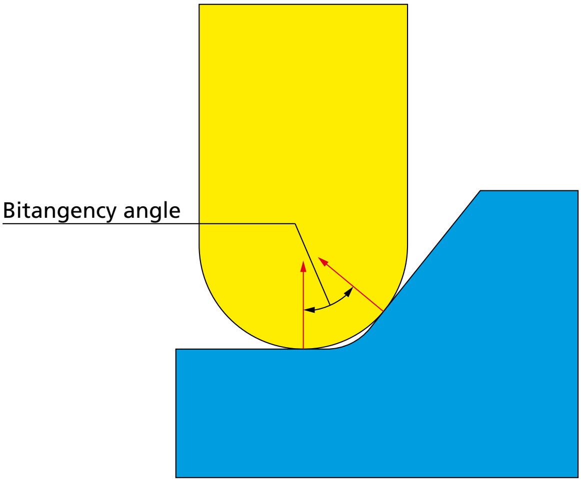


InventorCAM determines the rest material areas using a **Reference tool** (the tool that is assumed to have already been used in the CAM-Part machining) and a Target tool (the tool that is used for the Rest machining). Both tools must be ball-nosed.



Bitangency angle

This parameter defines the minimum angle required between the two normals at the contact points between the tool and model faces in order to perform the **Rest machining**.

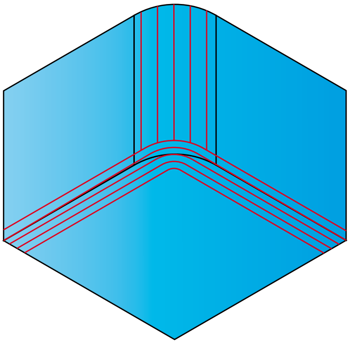


This value enables you to control the precision with which rest material areas are found. Reducing the value will typically cause the system to find more areas due to the triangle variations, however the most appropriate value will depend on the geometry of the machined piece.

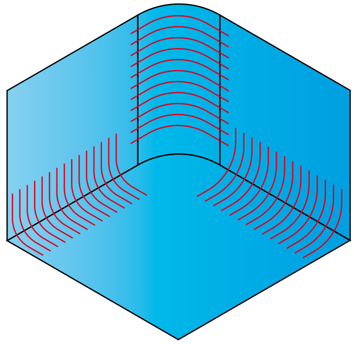
Steep threshold

This parameter enables you to specify the angle range at which InventorCAM splits steep areas from shallow areas. The angle is measured from horizontal, so that 0° represents a horizontal surface and 90° represents a vertical face.

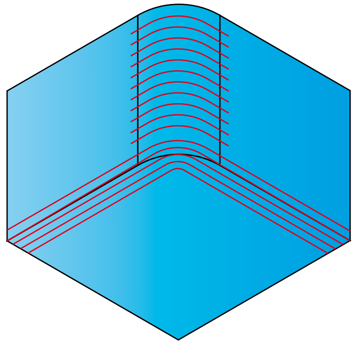
Setting the value to **90°** will mean that all areas in this range will be treated as shallow and the passes in the rest material areas will run along the corner.



Setting the value to **0°** will mean that all areas in this range will be treated as steep and the passes in the rest material areas will run across the corner.



Setting the value to **45°** will mean that areas where the slope is between 0 and 45° will be treated as shallow and the passes will run along the corner. Areas where the slope is between 45 and 90° will be treated as steep and the passes will run across the corner.



Shallow strategy

This option enables you to choose the machining strategy to be used in shallow areas (i.e. those below the **Steep threshold** value). The following options are available:

- **Linear**. This option enables you to perform links between passes using straight line motions.
- **Spiral**. This option joins some passes using smooth curved paths. This results in passes that are continuous, and reduces the use of linking moves. The spiral linking move will cut across the corner, avoiding the large volume of material that lies in the center of the rest area. Corner areas may not be fully finished.
- **Spiral on surface**. This option links the passes with smooth curved paths resulting in continuous passes and reducing the rapid moves. The spiral linking move is projected into the rest corner up to the maximal depth of the cut specified.

Min. depth of cut

This parameter specifies the minimum depth of material to be removed from the areas to be machined. Areas in which the depth of material to be cut is smaller than this will be ignored.

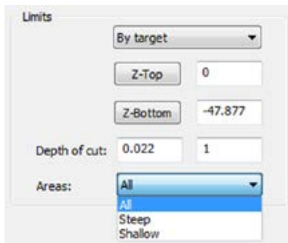
Min. depth of cut can also be useful in situations where a fillet radius of the part is approximately equal to the radius of the reference tool, i.e. places where, in theory, there is no material to be removed. If unwanted passes are created in such areas, increasing the value of **Min. depth of cut** may improve the situation.

Max. depth of cut

This parameter specifies the maximum depth of material that can be cut. Areas in which the depth of material is greater than this value will be ignored. This parameter is used to avoid situations where the cutter may otherwise attempt to make deep cuts. This may result in some rest area material not being removed; by creating further sets of Rest machining passes, using smaller reference tools, you can clear such areas.

Areas

This option enables you to decide whether to perform the machining in the steep areas only, in the shallow areas only or in both of them.



- **Shallow**

  The machining is performed only in the shallow areas (the surface inclination is smaller than the **Steep threshold** value).

- **Steep**

  The machining is performed only in the steep areas (the surface inclination is greater than the **Steep threshold** value).

- **All**

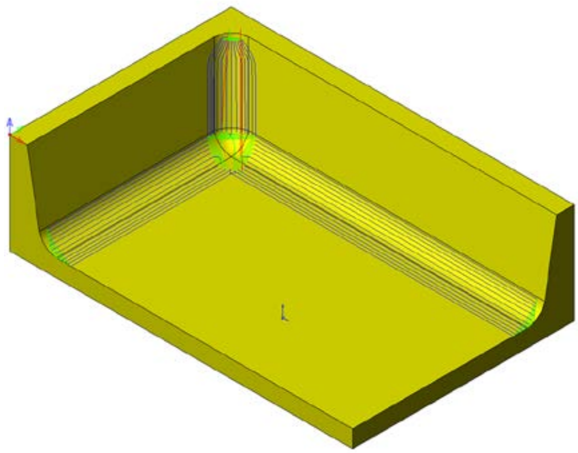
  The machining is performed in both steep and shallow areas.

Stroke ordering

This option enables you to control how the passes are merged, in order to generate better Rest machining passes. The available strategies are:

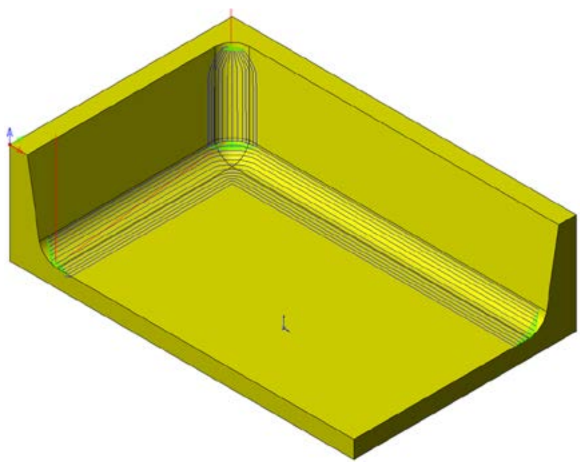
- **None**

  Passes are not combined; uncut material might be left in corners where several sets of passes converge.



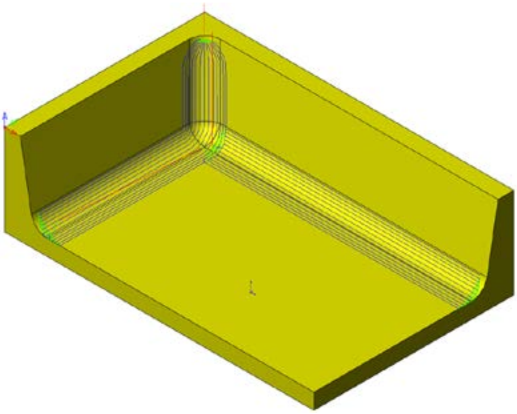
- **Planar**

  InventorCAM looks at the passes from the tool axis direction (from +Z) and connects passes that have a direction change with an angle smaller than the **Max. deviation** value.



- **Angular**

  The system looks at the passes in 3D and connects passes that have a direction change with an angle smaller than the **Max. deviation** value.



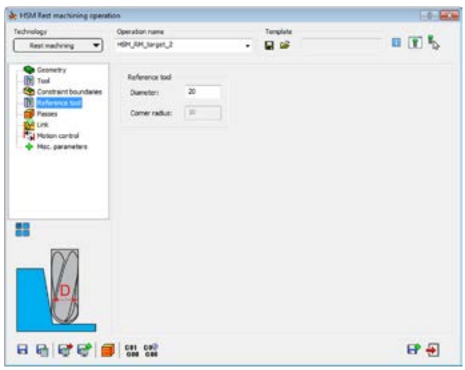
- **User-defined**

  The passes are neither combined nor split into steep and shallow sections. The option of **Max. deviation** is disabled in this strategy.

Max. deviation

When Rest machining passes approach a sharp change of direction, they can be made continuous round the corner, or can be split into separate segments. The value of **Max. deviation** is used to determine whether the passes are split (if the angle of deviation of the passes is larger than the **Max. deviation** value) or continuous (if the angle of deviation of the passes is smaller than the **Max. deviation** value).

Reference tool page

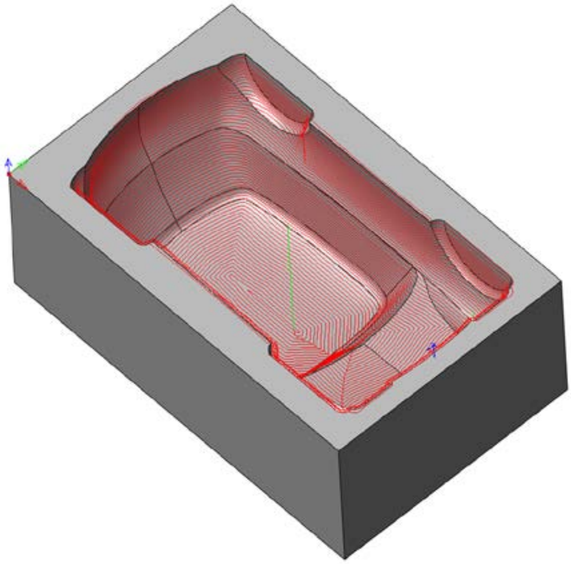


This page enables you to define the reference tool used for the Rest machining tool path calculation.

- The **Diameter** field defines the diameter of the reference tool.
- The **Corner radius** field defines the corner radius of the reference tool. Since the reference tool is ball-nosed, the corner radius is equal to half of the reference tool diameter.

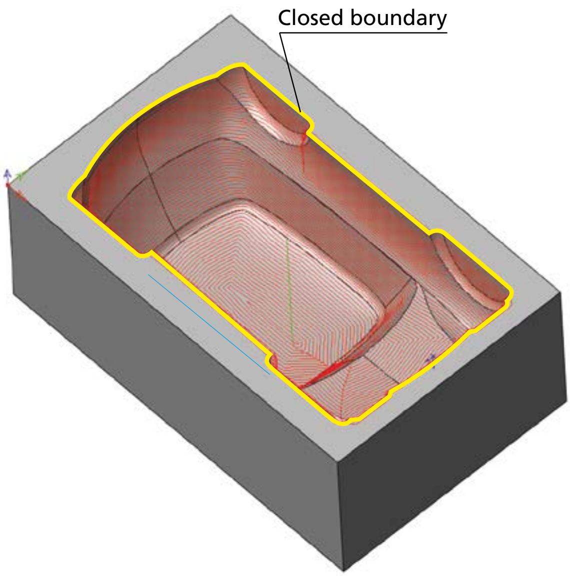
6.6.11 3D Constant step over

3D Constant step over machining enables you generate 3D tool path on the CAM-Part surfaces. The passes of the tool path are located at a constant distance from each other, measured along the surface of the model.

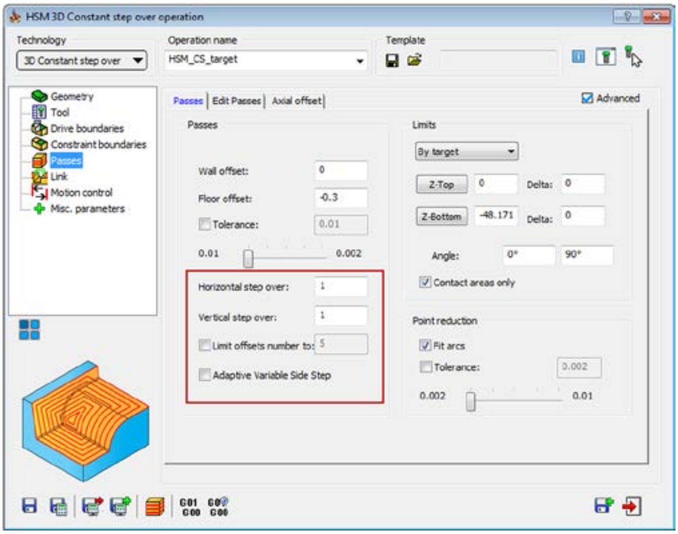


This is an ideal strategy to use on the boundaries generated by Rest machining or in any case where you want to ensure a constant distance between passes along the model faces.

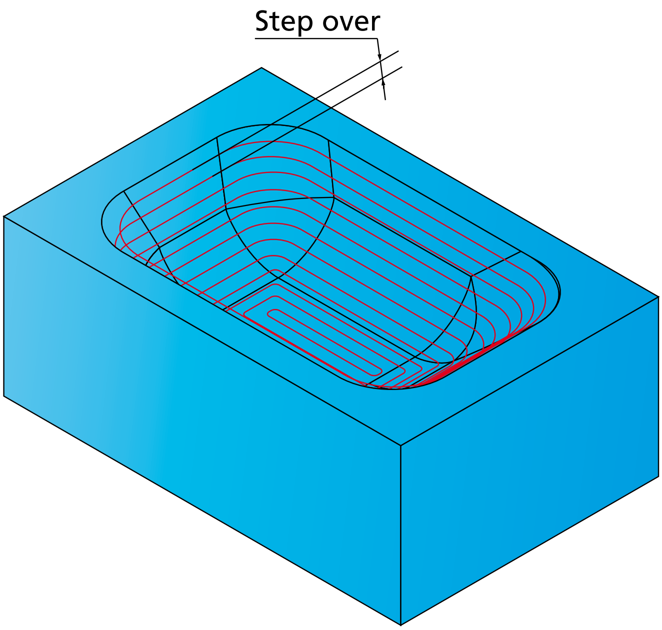
Constant surface step over is performed on a closed profile of the **Drive boundary** (see topic **5.1.1**). InventorCAM creates inward offsets from this boundary.



Step over



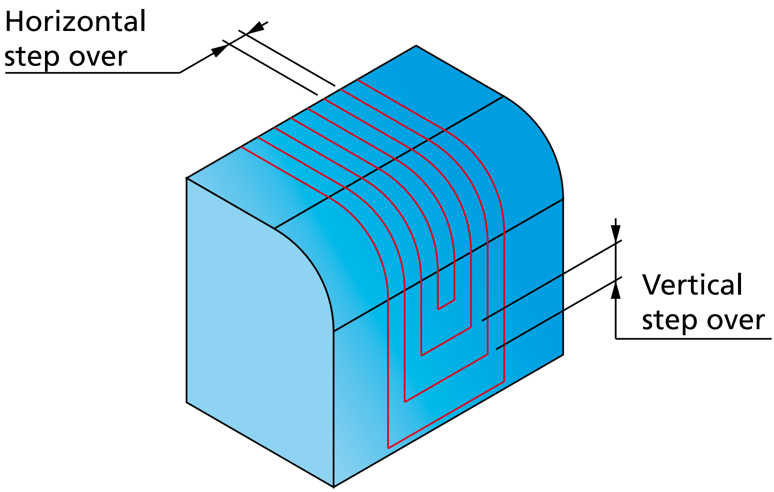
This parameter enables you to define the distance between cutting passes. In **3D Constant step over machining**, the **Step over** value is calculated in such a way that all passes are equidistant along the surface.



The **Horizontal** and **Vertical step over** parameters determine the distance between passes. The two step over types relate to the direction in which the step over is being measured. Where passes are offset horizontally, the **Horizontal step over** distance is used while for passes that are offset vertically, the **Vertical step over** distance is used. Where the step direction is neither vertical nor horizontal, the an average of the two values is used.

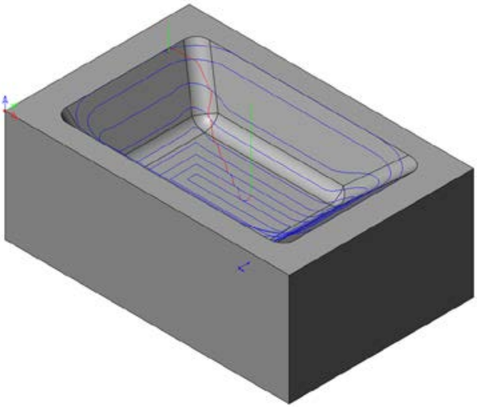
Limit offsets number to

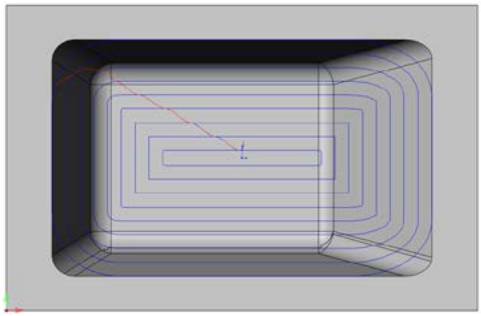
The **Limit offsets number to** parameter enables you to limit the number of offsets of a drive boundary profile. Select the **Limit offsets number to** check box and set the offsets number.



Adaptive Variable Side Step

If the **Adaptive Variable Side Step** check box is selected, the step over will be taken from the horizontal plane only, that is, a 2D offset. With this option, only the **Horizontal step over** value is used, the **Vertical step over** value is not relevant.

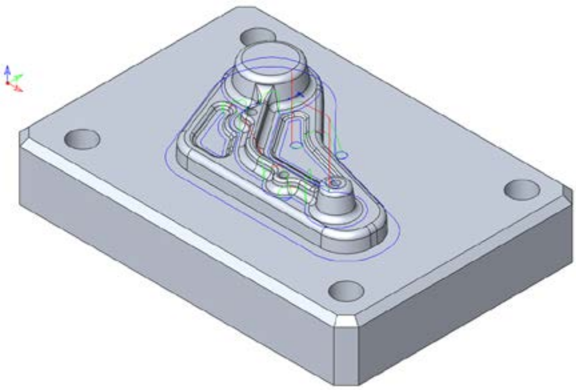


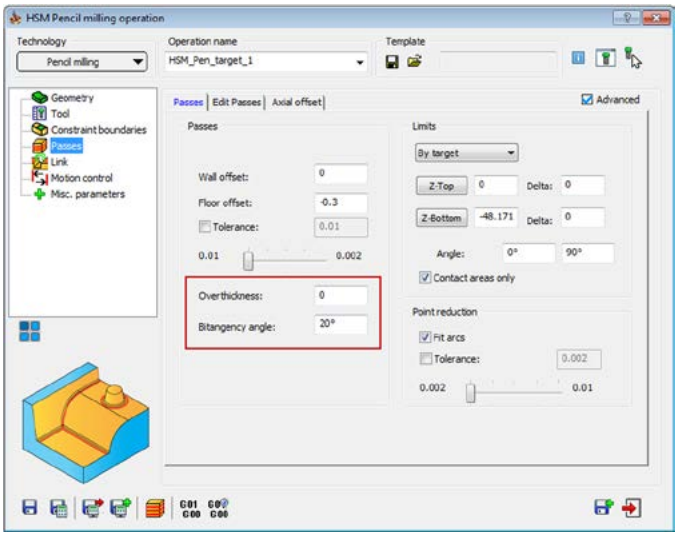


You can see from the illustration above that using this option on this model creates only few passes on steep areas since the spacing is calculated only along the horizontal plane; using this option is therefore not recommended for such models.

6.6.12 Pencil milling

The **Pencil milling** strategy creates a tool path along internal corners and fillets with small radii, removing material that was not reached by previous machining. This strategy is used to finish corners which might otherwise have cusp marks left from previous machining operations. This strategy is useful for machining corners where the fillet radius is the same or smaller than the tool radius.

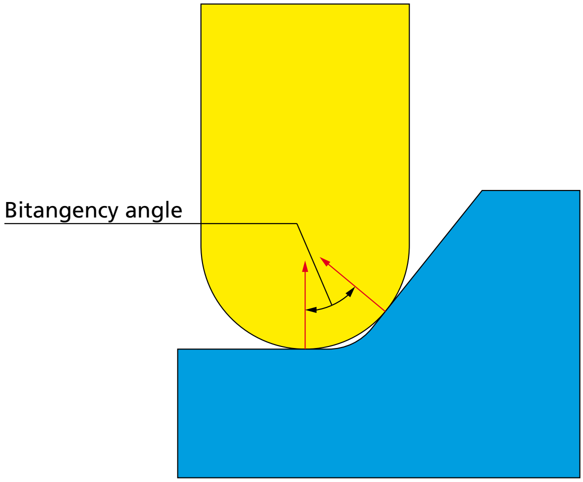




Bitangency angle

This is the minimum angle required between the two normals at the contact points between the tool and model faces, in order to decide to perform the pencil milling.

The default value of the **Bitangency angle** parameter is 20°. Generally, with this value InventorCAM detects all the corners without fillets and with fillet radii smaller than the tool radius. To detect corners with fillets radii greater than the tool radius you can either use the **Overthickness** parameter or decrease the **Bitangency angle** value. Note that decreasing the **Bitangency angle** value can result in the occurrence of unnecessary passes.



Overthickness

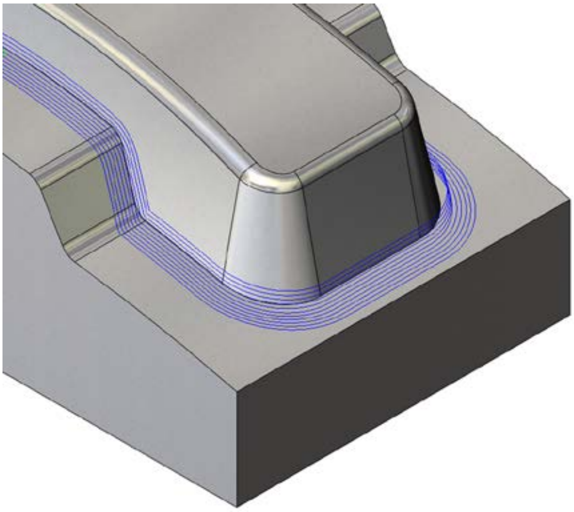
This parameter enables you to define an extra thickness that can be temporarily applied to the tool in addition to the normal **Wall offset**.

You can use the **Overthickness** parameter to generate a tool path along fillets whose radius is greater than the tool radius. For example, if you have a filleted corner of radius 8 mm and you want to create a Pencil milling tool path along it with the 10 mm diameter ball-nosed tool, you can set the **Overthickness** value to 4 mm. The Pencil milling tool path is calculated for a ball-nosed tool with the diameter of 18 mm (which will detect this fillet), and then projected back onto the surface to make a tool path for the 10 mm diameter tool.

Since this is an offset value, it is specified in exactly the same manner as other offsets, except that it is added to the defined tool size, in addition to any surface offset, during calculations.

6.6.13 Parallel pencil milling

Parallel pencil milling is a combination of the **Pencil milling** strategy and the **3D Constant step over** strategy. At the first stage, InventorCAM generates a **Pencil milling** tool path. Then, the generated pencil milling passes are used to create **3D Constant step over** passes; the passes are generated as a number of offsets on both sides of the pencil milling passes. In other words, the **Parallel pencil milling** strategy performs **3D Constant step over** machining using **Pencil milling** passes as drive curves to define the shape of passes.



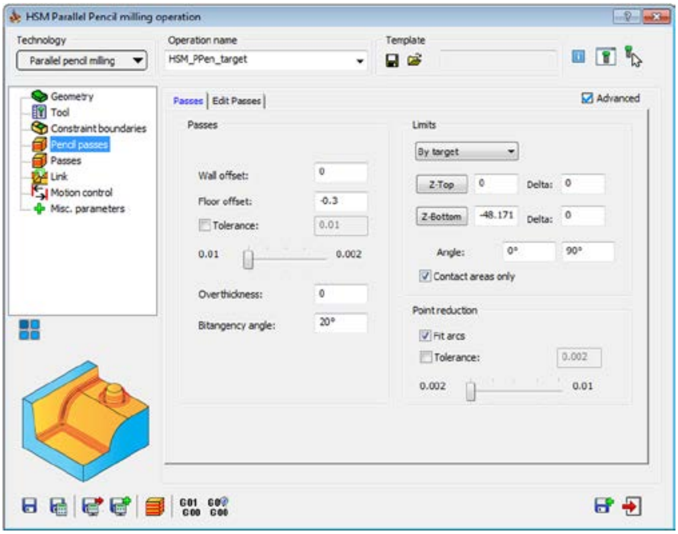
This is particularly useful when the previous cutting tool has not been able to machine all the internal corner radii to size. The multiple passes generated by this strategy will machine from the outside in to the corner, creating a good surface finish.

The order of passes machining is determined by the **Order** parameters (see topic **7.1.2**).

In this combined strategy, you define the **Pencil milling** parameters and the **3D Constant step over** parameters in two separate pages.

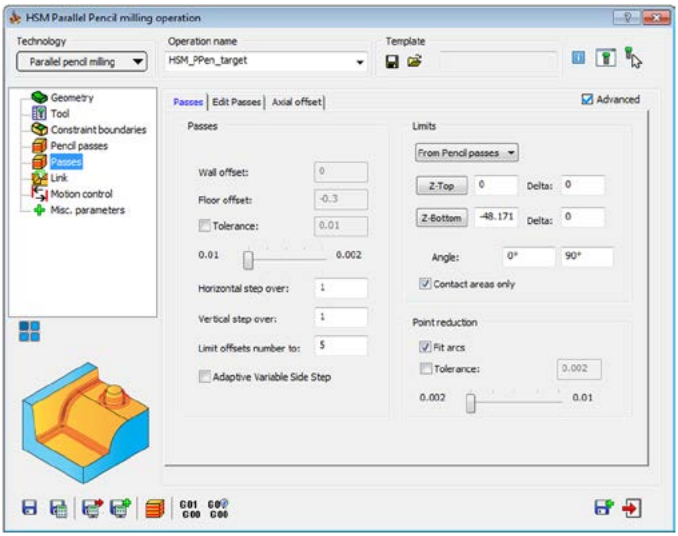
Pencil milling parameters

The **Pencil passes** page enables you to define the parameters of the **Pencil milling** passes (see topic **6.6.12**).



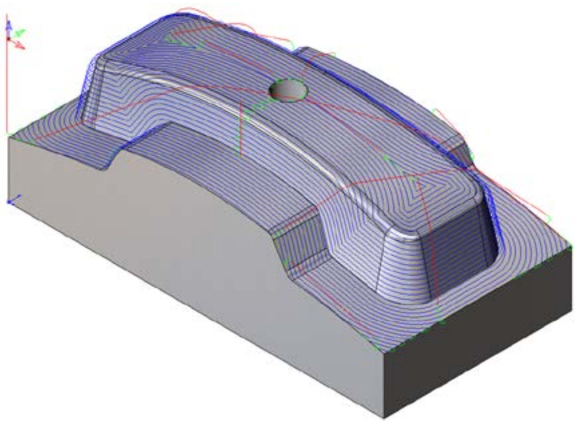
3D Constant step over parameters

The **Passes** page defines the parameters of the **3D Constant step over** passes (see topic **6.6.11**).



6.6.14 3D Corner offset

The **3D Corner offset** strategy is similar to the **Parallel pencil milling** strategy. This strategy is also is a combination of **Pencil milling** strategy and **3D Constant step over** strategy. InventorCAM generates a **Pencil milling** tool path and uses it for the **3D Constant step over** passes generation. These passes are generated as offsets from the **Pencil milling** passes. In contrast to the **Parallel pencil milling** strategy, the number of offsets is not defined by user, but determined automatically in such a way that all the model inside a boundary will be machined.

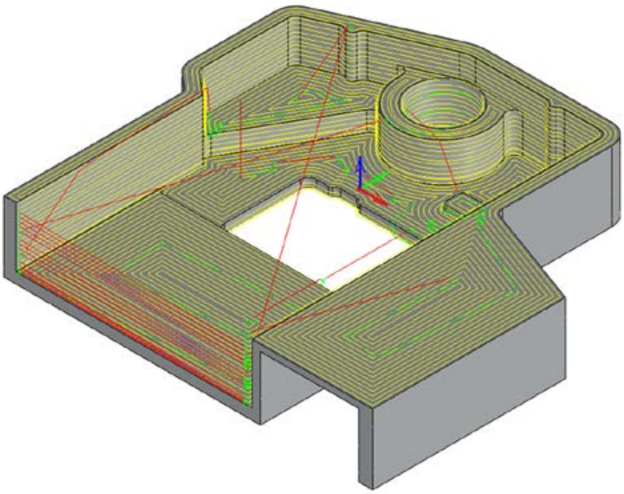


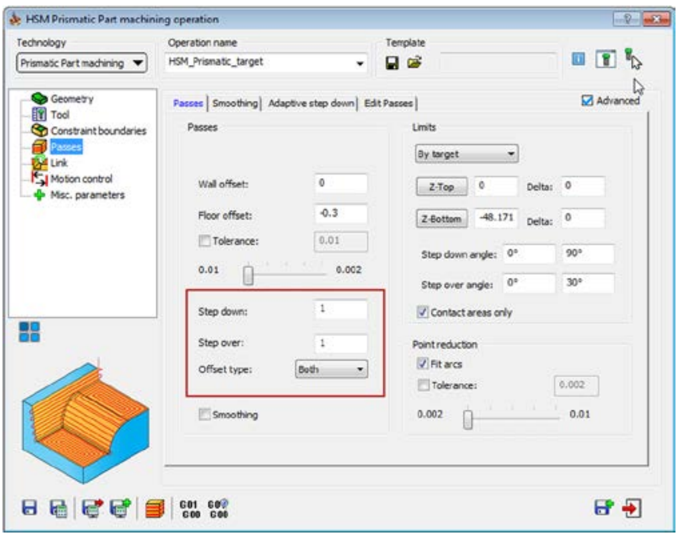
The order of passes machining is determined by **Order** parameters (see topic **7.1.2**).

In this combined strategy, you define the **Pencil milling** parameters and the **3D Constant step over** parameters in two separate pages.

6.6.15 Prismatic Part

The **Prismatic Part machining** strategy is designed especially for high-speed finishing of prismatic parts. This strategy comprises the technology of the **Constant Z** and **3D Constant step over** strategies by integrating these two strategies into one smart functionality of prismatic part machining. The difference from the **Combined Constant Z with 3D Constant step over** strategy is as follows: in the **Combined** strategy, the subsequent strategies are performed successively one after the other. In the **Prismatic Part machining** strategy, the machining is performed consistently according to the order of the walls and flat faces along the Z-axis.





Step down

This parameter defines the height of the passes spacing along the tool axis in the **Constant Z** finishing.

Step over

This parameter defines the distance between the adjacent tool passes measured along the surface.

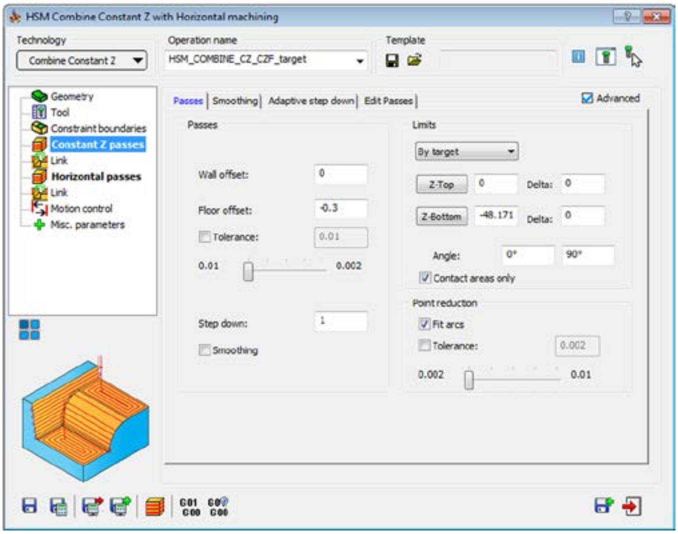
Offset type

This option enables you to define an offset between adjacent **Constant Z** passes. When the **Upper** option is chosen, the upper **Constant Z** pass is offset and trimmed to the lower **Constant Z** pass. When the **Lower** option is chosen, the lower **Constant Z** pass is offset and trimmed to the upper **Constant Z** pass. When the **Both** option is chosen, both passes are offset together.

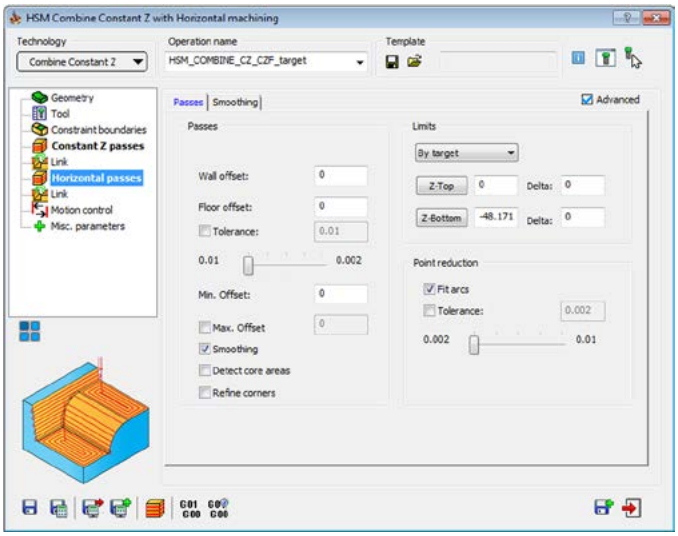
6.6.16 Combined strategy

Constant Z combined with Horizontal strategy

The **Constant Z passes** page defines the parameters of the **Constant Z** machining strategy.



The **Horizontal passes** page defines the parameters of the **Horizontal** machining strategy.



The following parameters defined on the **Constant Z Passes** page are automatically assigned with the same values on the **Horizontal passes** page:

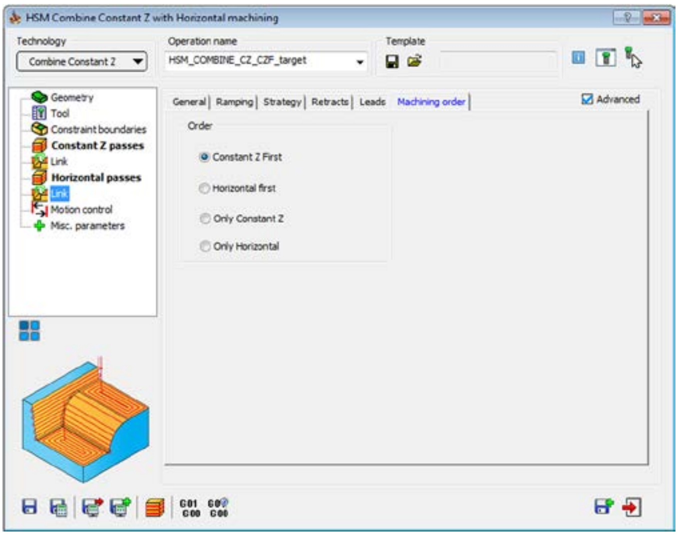
- **Wall offset (see topic 6.1.1)**
- **Floor offset (see topic 6.1.2)**
- **Tolerance (see topic 6.1.3)**
- **Limits (see topic 6.1.8)**
- **Smoothing parameters (see topic 6.2)**
- **Adaptive step down parameters (see topic 6.3)**
- **Edit passes parameters (see topic 6.4)**

When these parameters are edited on the **Constant Z passes** page, their values are updated automatically on the **Horizontal passes** page. But when edited on the **Horizontal passes** pages, the values in the **Constant Z passes** page remain unchanged.

Two **Link** pages located under the **Constant Z passes** and **Horizontal passes** pages define the links relevant for each of these strategies.

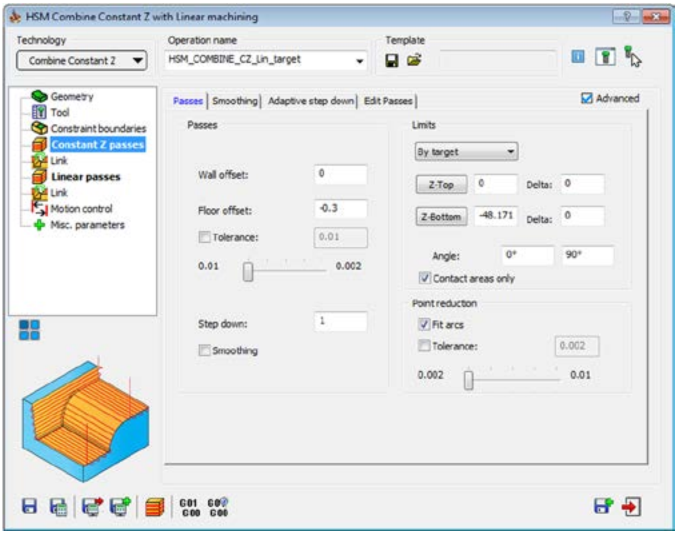
On the **Link** page for **Horizontal** passes, there is the **Machining order** tab that enables you to define the order in which the **Constant Z** and **Horizontal** machining will be performed. The default option is **Constant Z First**.

When the tool has finished performing the passes of the first machining strategy, it goes up to the **Clearance level**, then descends back to the machining surface to continue with the next strategy.

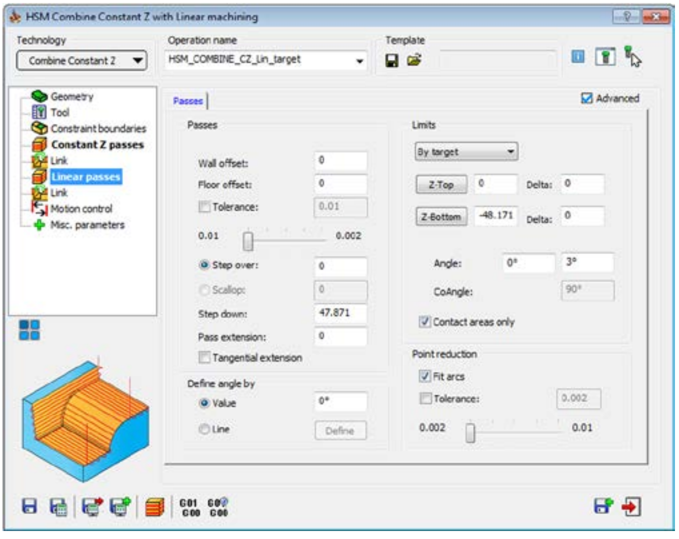


Constant Z combined with Linear strategy

The **Constant Z passes** page defines the parameters of the **Constant Z** machining strategy.



The **Linear passes** page defines the parameters of the **Linear** machining strategy.



The following parameters defined on the **Constant Z Passes** page are automatically assigned the same values on the **Linear passes** page:

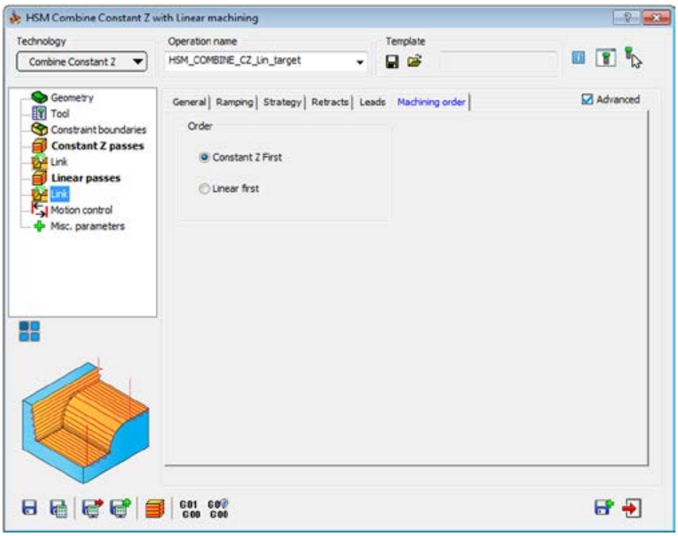
- **Wall offset (see topic 6.1.1)**
- **Floor offset (see topic 6.1.2)**
- **Tolerance (see topic 6.1.3)**
- **Limits (see topic 6.1.8)**
- **Smoothing parameters (see topic 6.2)**
- **Adaptive step down parameters (see topic 6.3)**
- **Edit passes parameters (see topic 6.4)**

When these parameters are edited on the **Constant Z passes** page, their values are updated automatically on the **Linear passes** page. But when edited on the **Linear passes** page, the values in the **Constant Z passes** page remain unchanged.

Two **Link** pages located under the **Constant Z passes** and **Linear passes** pages define the links relevant for each of these strategies.

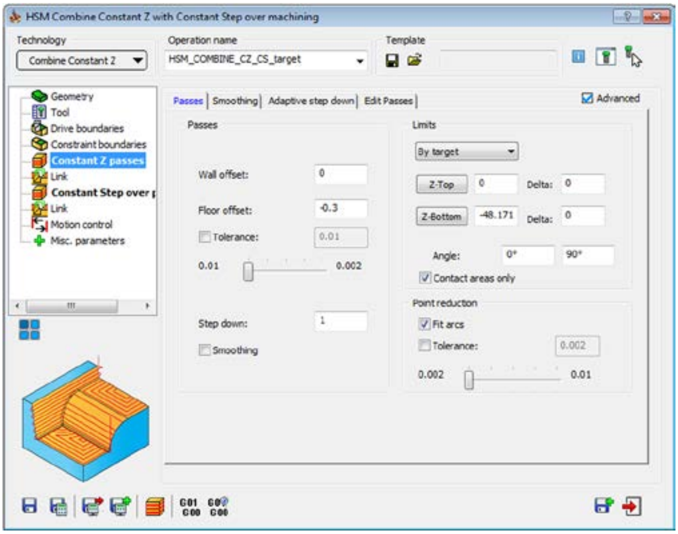
On the **Link** page for **Linear** passes, there is the **Machining order** tab that enables you to define the order in which the **Constant Z** and **Linear** machining will be performed. The default option is **Constant Z First**.

When the tool has finished performing the passes of the first machining strategy, it goes up to the **Clearance level**, then descends back to the machining surface to continue with the next strategy.

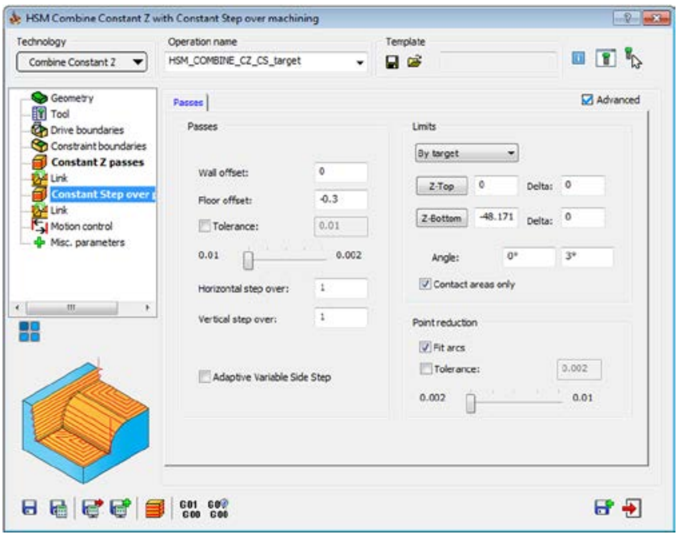


Constant Z combined with 3D Constant Step over strategy

The **Constant Z passes** page defines the parameters of the **Constant Z** machining strategy.



The **Constant Step over passes** page defines the parameters of the **3D Constant Step over** machining strategy.



The following parameters defined on the **Constant Z Passes** page are automatically assigned the same values on the **Constant Step over passes** page:

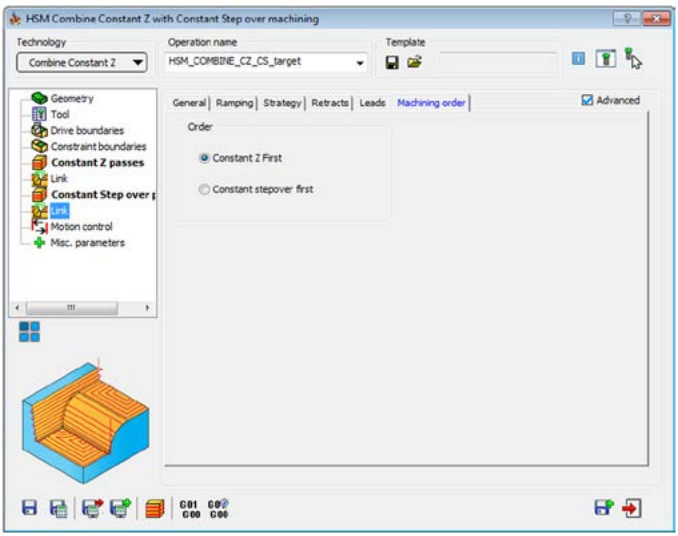
- **Wall offset (see topic 6.1.1)**
- **Floor offset (see topic 6.1.2)**
- **Tolerance (see topic 6.1.3)**
- **Limits (see topic 6.1.8)**
- **Smoothing parameters (see topic 6.2)**
- **Adaptive step down parameters (see topic 6.3)**
- **Edit passes parameters (see topic 6.4)**

When these parameters are edited on the **Constant Z passes** page, their values are updated automatically on the **Constant Step over** page. But when edited on the **Linear passes** page, the values in the **Constant Z passes** page remain unchanged.

Two **Link** pages located under the **Constant Z passes** and **Constant Step over passes** pages define the links relevant for each of these strategies.

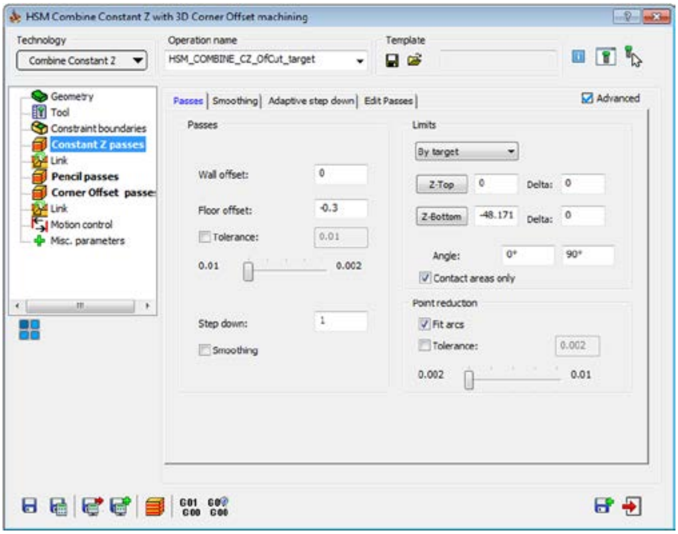
On the **Link** page for **Constant step over** passes, there is the **Machining order** tab that enables you to define the order in which the **Constant Z** and **Constant step over** machining will be performed. The default option is **Constant Z First**.

When the tool has finished performing the passes of the first machining strategy, it goes up to the **Clearance level**, then descends back to the machining surface to continue with the next strategy.

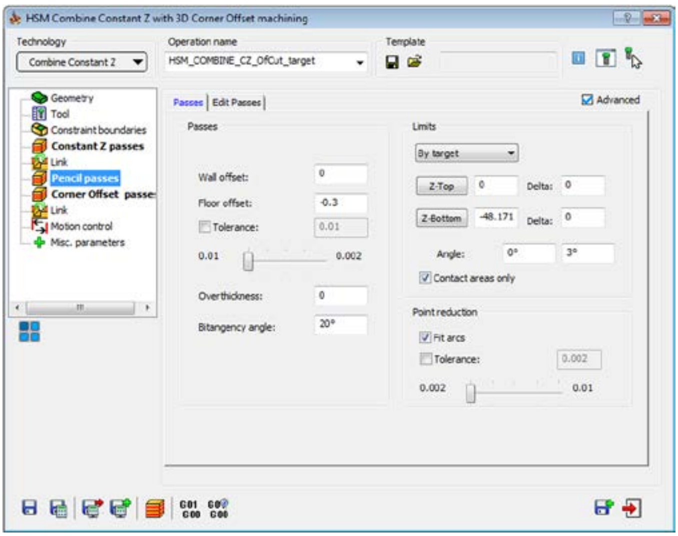


Constant Z combined with 3D Corner offset strategy

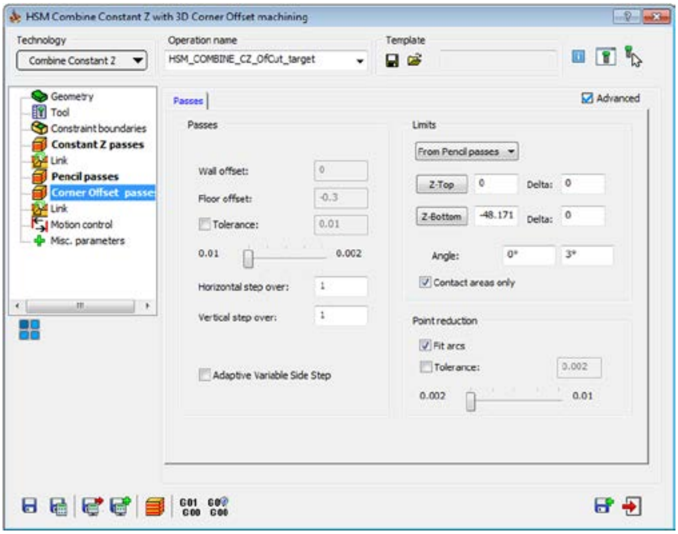
The **Constant Z passes** page defines the parameters of the **Constant Z** machining strategy, which is used for semi-finishing and finishing of steep model areas.



The **Pencil passes** page defines the parameters of the **Pencil** milling strategy, which is used for removing material along internal corners and fillets with small radii that was not reached in previous operations.

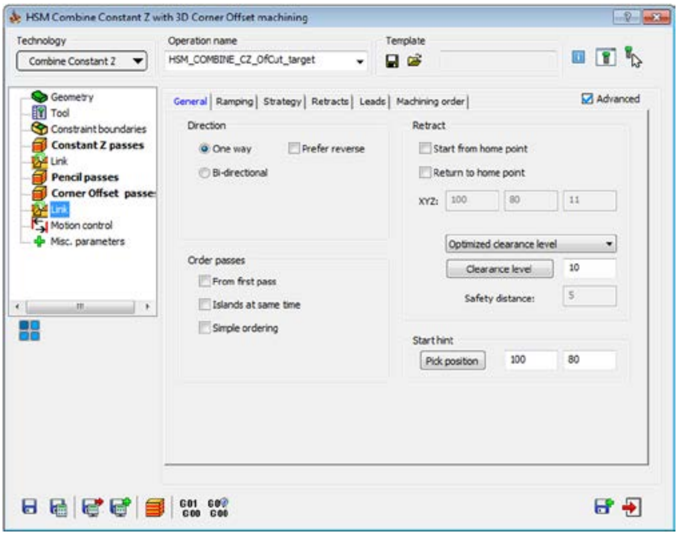


The **Corner Offset passes** page defines the parameters of the **3D Corner offset** machining strategy, which is used in order to machine all the internal corner radii to size.



Two **Link** pages, which are located under the **Constant Z passes** and the **Corner Offset passes** pages, define the linking of the tool path for each of these strategies.

In the **Link** page for **Corner Offset passes**, the **Machining order** tab enables you to define the order in which the **Constant Z** and **Corner offset** passes will be performed.



When the tool has finished performing the passes of the first machining strategy, it goes up to the **Clearance level**, then descends back to the machining surface to continue with the next strategy.

6.7 Calculation Speed

Tool paths for three basic tool types (end mill, ball-nosed and bull-nosed) are calculated by completely different machining algorithms. This means that calculation speed may differ for the same operation and geometry, but with a different tool type. For example, using a bull-nosed tool with a smaller corner radius will result in longer calculation time.

Calculation speed depends also on the tolerance. Setting a tolerance for a tool path defines the worst case value; the actual tolerance may, in some circumstances, be significantly tighter. That is particularly true for **Contour roughing** and **Constant Z machining** operations when a bull-nosed tool with a small corner radius is used. The results are often more accurate than required and calculation is slower.

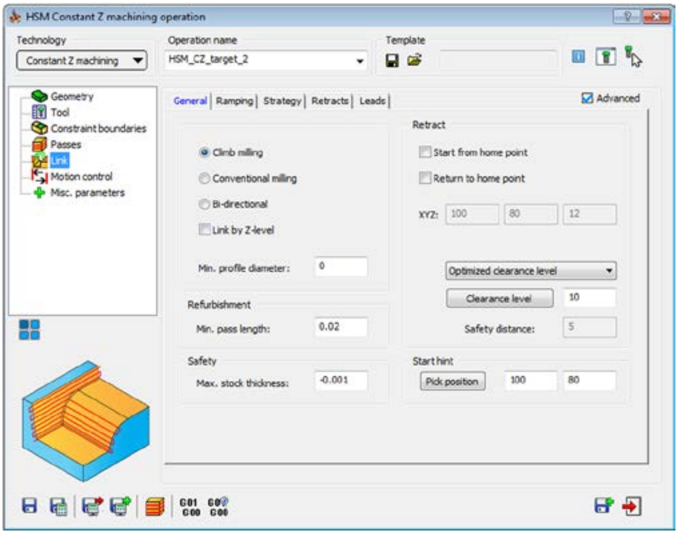
When a positive **Wall offset** is defined, the machining algorithm uses a tool with larger corner and shaft radii than the original one. When a small **Wall offset** is applied to an end mill, the tool used by the algorithm is bull-nosed with a small corner radius. That tool with applied **Wall offset** has different algorithmic characteristics, as mentioned above, and calculation time may change.

The only other instance in which the tool type may change when applying a **Wall offset** is when a negative **Wall offset** equal to or exceeding the corner radius is applied to a bull-nosed tool. Then an end mill is used in the machining algorithm, and a result may be reached faster. However, there are instances where applying a negative **Wall offset**, which is significantly greater than the corner radius, does not produce satisfactory results.

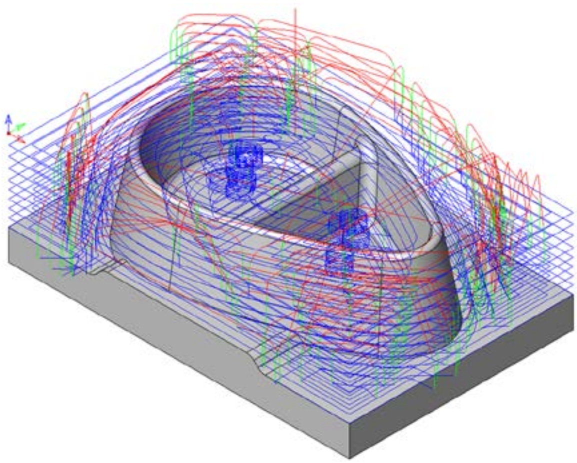
Links

7

The **Link** page in the **HSR/HSM Operation** dialog box enables you to define the way how the generated passes are linked together into a tool path.



In the image, the link movements are in green, the rapid movements are in red and the machining passes are in blue.



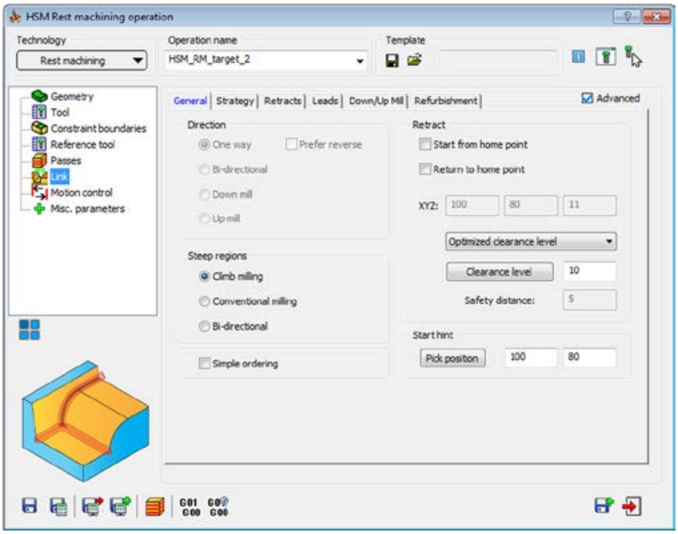
Following are the linking parameters that can be defined by the user:

- **General parameters**
- **Ramping parameters**
- **Strategy parameters**
- **Retracts parameters**
- **Leads parameters**
- **Down/Up Mill parameters**
- **Refurbishment parameters**

Some of the parameters pages can be disabled unless you check the **Advanced** option in the right upper part of the dialog box.

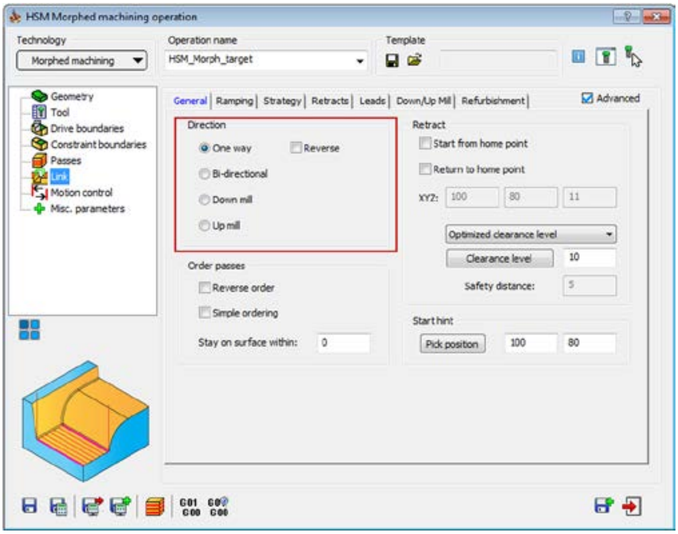
7.1 General Parameters

The **General** page enables you to set the general parameters of the tool path linking



- **Direction**
- **Order passes**
- **Retract**
- **Start hint**
- **Minimize reverse linking**
- **Minimize full wide cuts**
- **Link by area**
- **Link by Z level**
- **Link per cluster**
- **Min. profile diameter**
- **Refurbishment**
- **Safety**

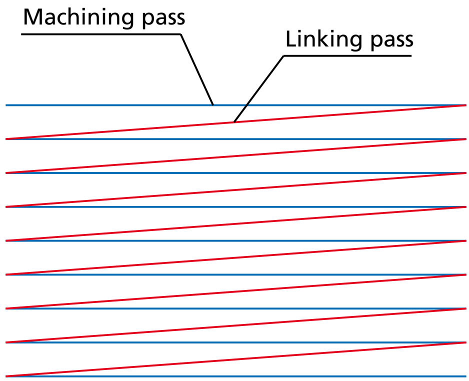
7.1.1 Direction options



This parameters group enables you to define the direction of the machining.

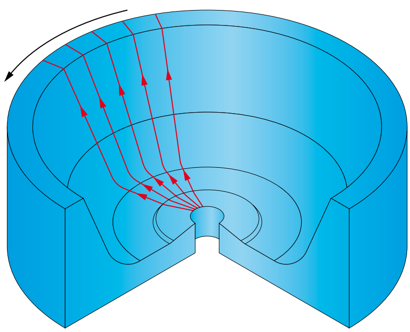
One way

With this option, machining is performed in one direction, but there is no guarantee that this will be consistently climb or conventional milling. It is up to you to check the tool path and respond by selecting the **Reverse** check box, if needed, for the desired milling style.

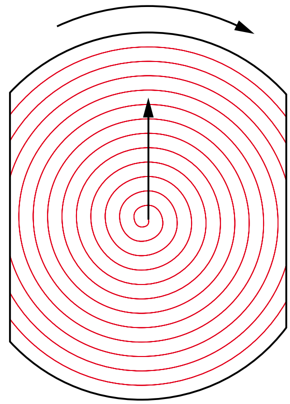


A one way hatch path has many retractions; after the machining pass the tool has to perform air movement to the start point of the next pass (shown in red).

- **One way cutting with Radial Machining strategy.** The radial arrows indicate the direction of the passes themselves, while the circular arrow indicates the ordering of the passes.

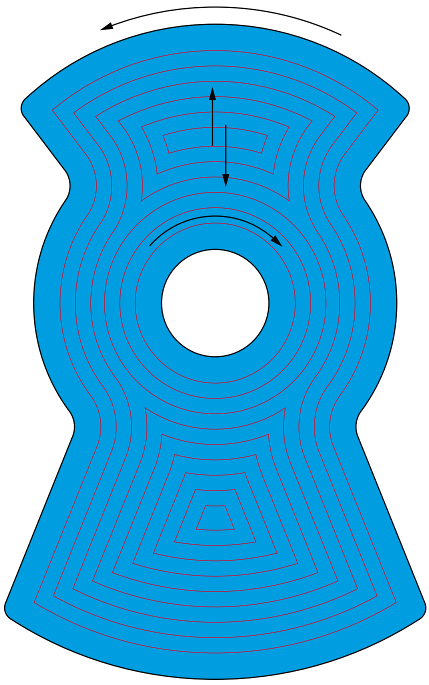


- **One way cutting with Spiral machining strategy.** The spiral pass is limited by a boundary. The circular arrow indicates the direction of the passes themselves, while the radial arrow indicates the ordering of the passes. Passes are machined in the clockwise direction, moving outwards.



- **One way cutting with 3D Constant Step over strategy.** The passes are limited by a boundary, with another boundary inside it. The passes are ordered in a one way direction to perform climb milling. The inner circular arrow indicates the direction for the passes adjacent to the inner boundaries.

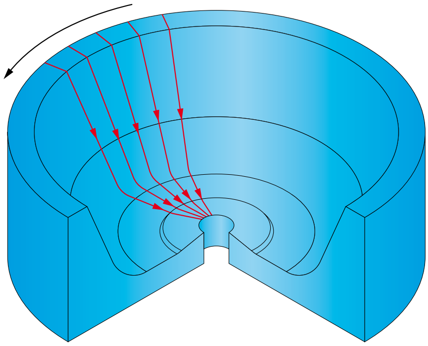
  The outer circular arrow indicates the direction for outer boundaries. In this example, most machining passes are performed in the counterclockwise direction, working from the farthest offsets outwards to the outer boundary, then rapidly moving to machine the farthest offset of the inner boundary and working inwards towards the inner boundary.



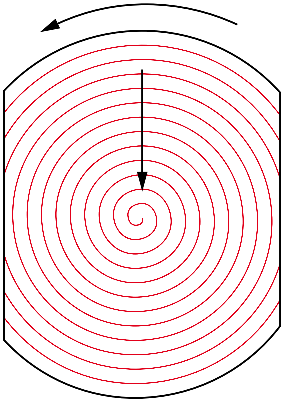
Reverse

The **Reverse** option results in the direction of passes being reversed.

The example below shows one-way radial passes with the reversed direction; the passes will be climb milled.

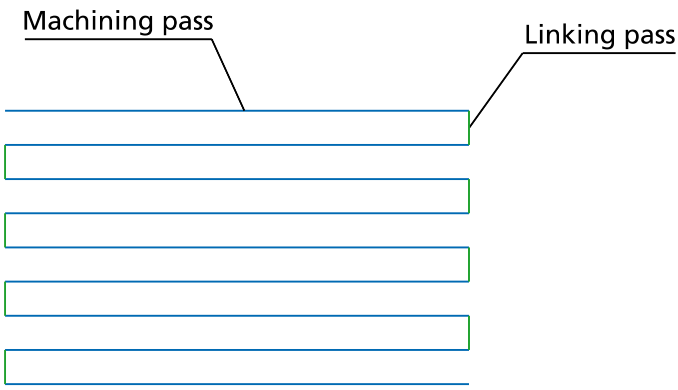


The example below shows a reversed one way spiral passes.

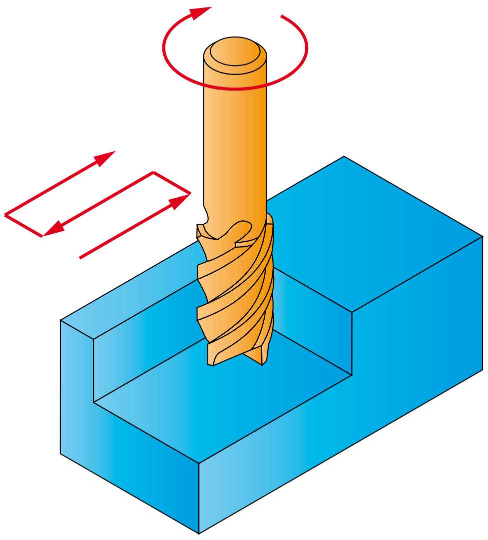


Bi-directional

With this option, each pass is machined in the opposite direction to the previous pass. A short linking motion (shown in green) connects the two ends - this is often called zigzag machining.

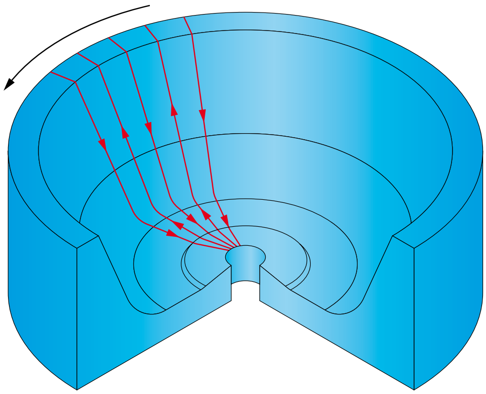


Both **Climb milling** and **Conventional milling** methods are used in the bi-directional tool path.

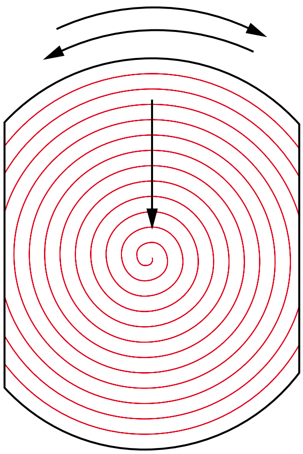


Bi-directional milling

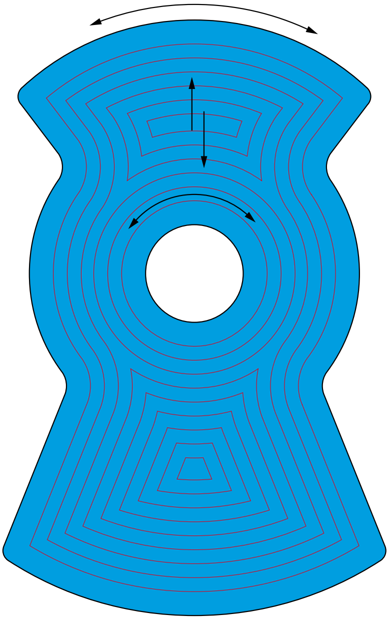
Bi-directional Radial machining:



Bi-directional Spiral machining:



Bi-directional 3D Constant Step over machining:



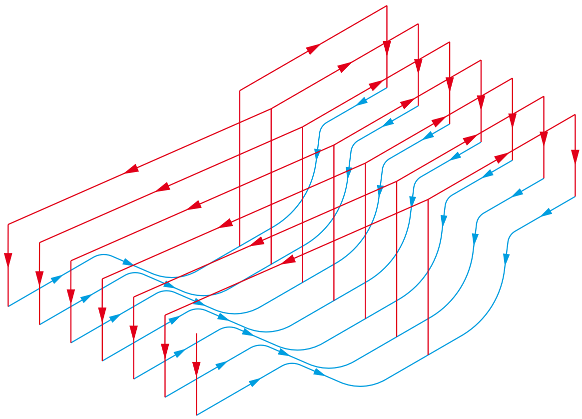
Down mill/Up mill

These options enables you to perform the machining downwards or upwards. Flat pieces are machined in the direction defined by the **Reverse** parameter.

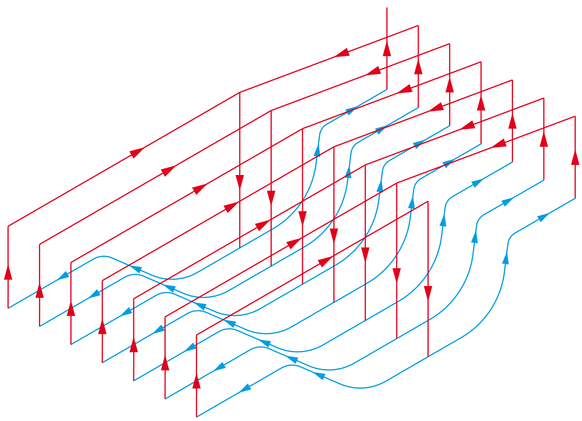
This option is available for strategies where the Z-level varies along a pass. This option is not available for the **Constant Z** and **Horizontal** strategies.

The **Down/Up mill** page (see topic **7.6**) enables you to define the parameters of the down and up milling.

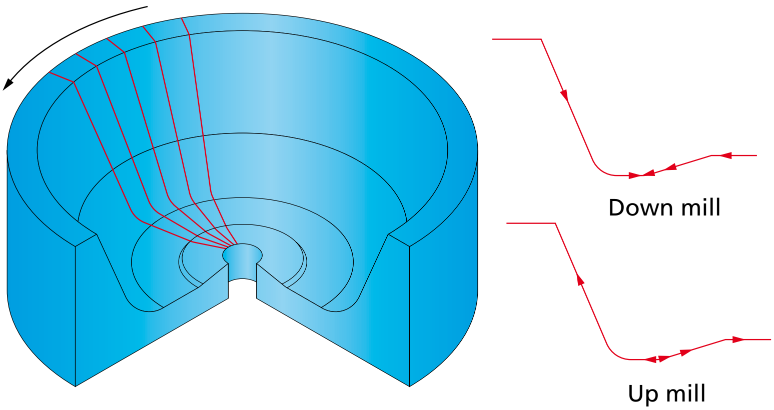
- **Down mill direction**



- **Up mill direction**

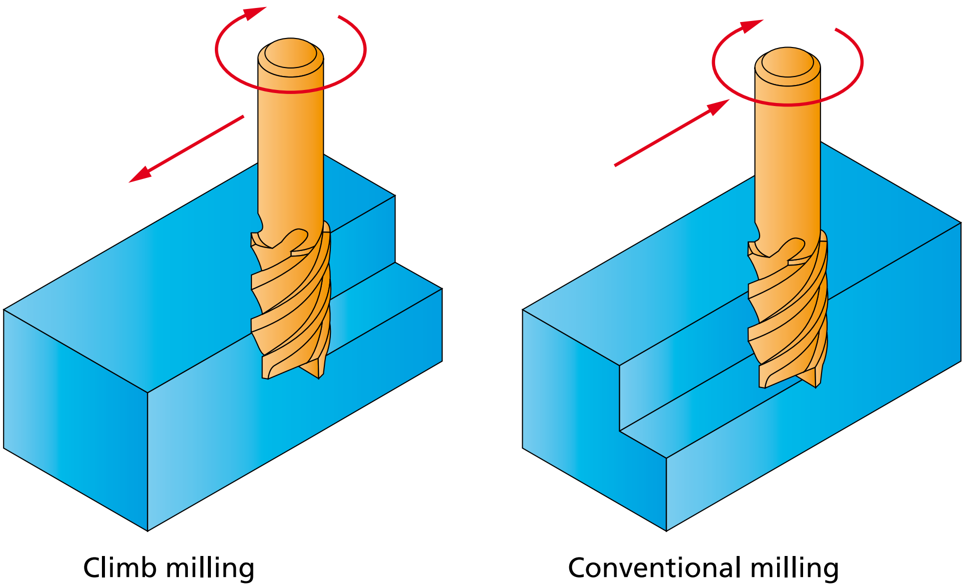


The image below shows the direction of the **Radial machining** passes when the **Down/Up Mill** options are used.



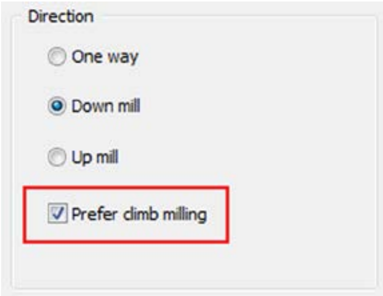
Climb/Conventional milling

These options enables you to set the tool path direction in such a manner that the climb/conventional milling will be performed.



These options are available for the **Constant Z** and **Horizontal** strategies.

Prefer climb milling

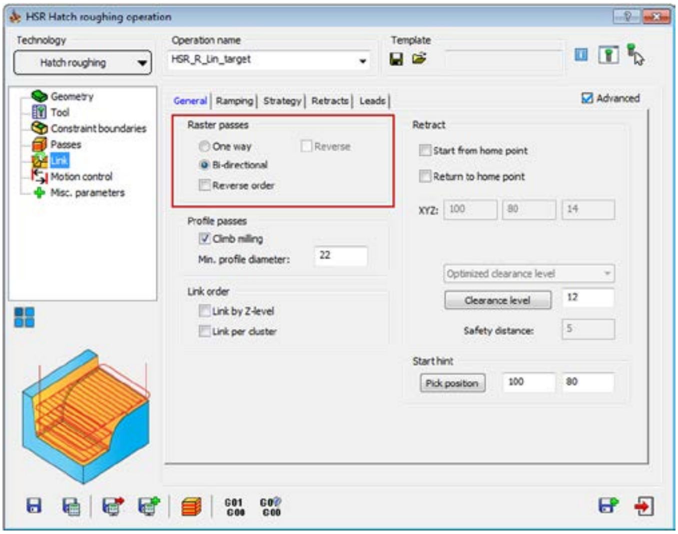


This option is available for the **Pencil milling** strategy.

If this option is selected, the Pencil milling passes will usually be climb milled. A decision is made as to whether the material is mainly on the left or the right of the tool as it goes along a pass. The direction is then chosen so that most material is on the right.

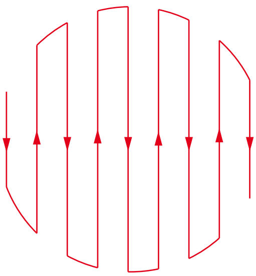
When this option is not selected, the milling direction for all the passes is reversed, so that they will probably be conventionally milled.

Direction for Hatch roughing

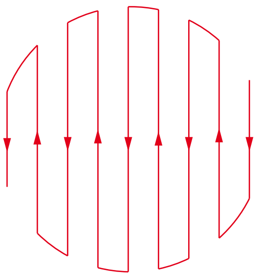


Raster passes

- This section enables you to define the direction for the hatch (raster) passes.
- InventorCAM enables you to choose **One way** or **Bi-directional** direction for the raster passes.



Initial order



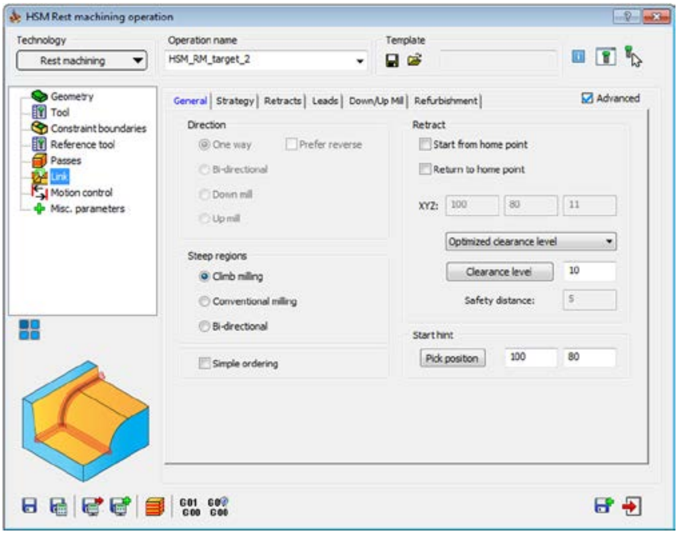
Reversed order

- The **Reverse order** option enables you to reverse the order of the hatch passes machining.

Profile passes

- This section enables you to define the direction for profile passes. InventorCAM enables you to choose the **Climb** or **Conventional** direction of the Profile passes.

Direction for Rest machining

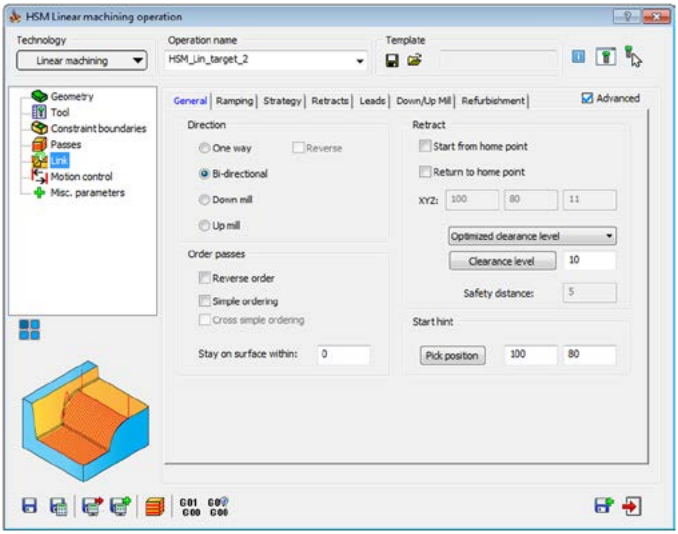


Steep regions

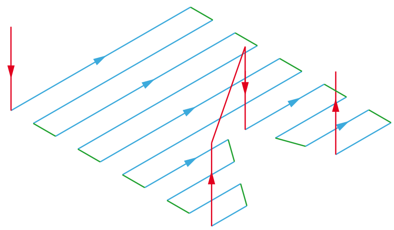
This section enables you to define the direction of the steep areas machining. InventorCAM enables you to choose the following options:

- **Climb milling**
- **Conventional milling**
- **Bi-directional**

7.1.2 Order passes



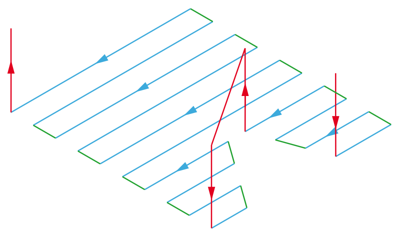
Some passes allow you to specify the direction of the pass ordering. When no option is selected, the passes are linked in an efficient way to limit the rapid travel between passes. Where several separate areas are machined, each area will be machined to completion, before machining of the next area is started.



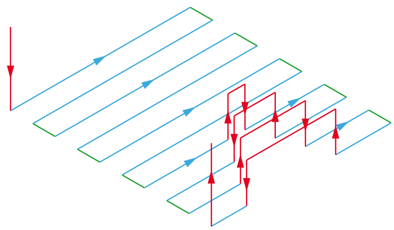
The passes will be linked in the most efficient way. Below is shown a set of **Linear** passes, linked in the default order (starting from the top left-hand corner) to minimize the rapid travel between the passes.

Reverse order

This option enables you to reverse the order of the tool path relative to the default order.



Simple ordering

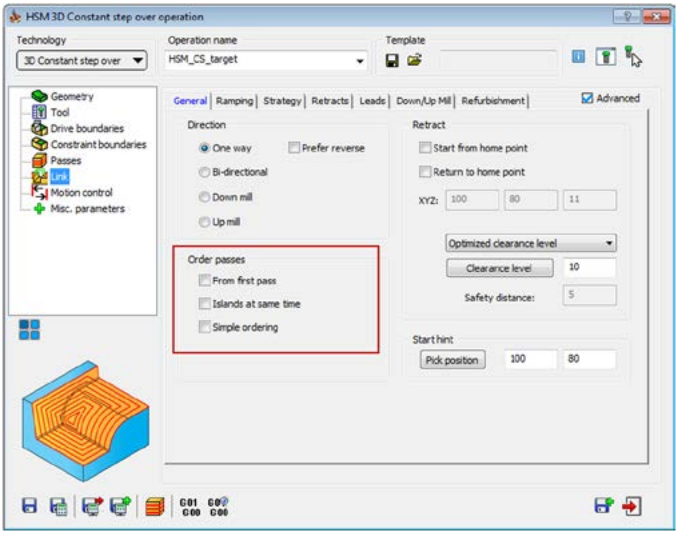


Passes will be linked in the order of their creation. Parts of a specific pass divided by a boundary will be linked together with a rapid movement. This option enables you to maintain the order of the passes, but increases the number of air movements.

Cross simple ordering

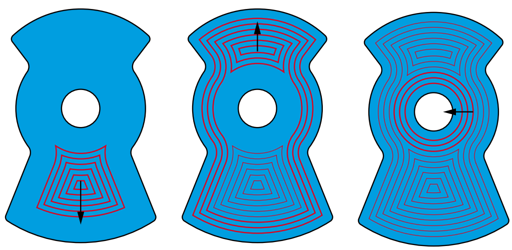
When the **Cross linear machining** is performed (see topic **6.6.4**), simple ordering is applied to the cross linear passes.

Order 3D Constant Step over passes

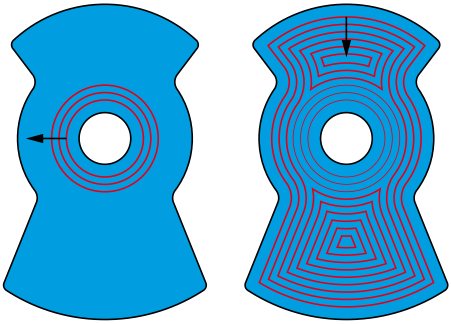


From first pass

When this option is not chosen, the passes are machined from the smallest of the outside boundary offsets to the outer boundary and then from the largest offset of the internal boundary to the inside.



When this option is chosen, the machining is performed in the reverse order. The machining starts from the internal boundary outside. After that the machining is performed from the outer boundary inside.



If you reverse the order or the direction, conventional milling will be performed. If you reverse both, climb milling will be performed again.

Islands at same time

If the original boundaries have islands, InventorCAM will normally machine inwards from the outer boundary, then outwards from the island boundary.

When this option is chosen, InventorCAM performs machining while swapping between the outer and the island boundary offsets, ensuring that each is never more than one pass ahead of the other.

Simple ordering

Passes will be linked in the order of their creation. Parts of a specific pass divided by a boundary will be linked together with a rapid movement. This option enables you to maintain the order of the passes, but increases the number of air movements.

Optimize lead positions

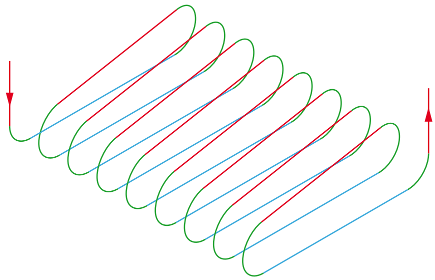
When this option is chosen, the start point of the pass is adjusted to minimize the length of the lead move.

When it is not selected, the lead move connects to the natural start point of the pass.

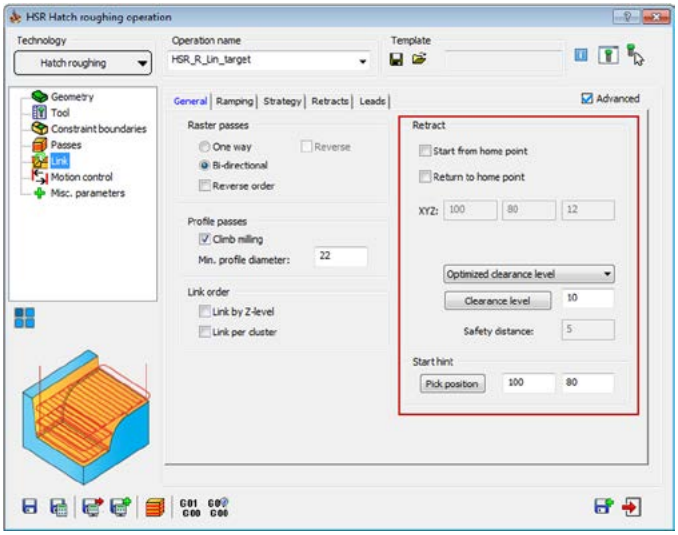
7.1.3 Retract

The image below shows a set of linked one way Hatch roughing passes along a flat horizontal surface.

The tool path starts from the **Start hint** point that is set at the **Safety distance** level. The rapid movements (shown in red) are performed at the **Clearance level** and above it. The tool moves along the green lines towards, away from, or along the surface, without cutting (link movements). The blue lines show the tool path when cutting is performed. The tool path is finished in the end point located at the **Safety distance** level.



The **Retract** section enables you to define a number of parameters of the start and end of the tool path.



Start from home point/Return to home point

These options enable InventorCAM to start/finish the operation tool path in the specified home point. The **XYZ** boxes defines the location of this point.

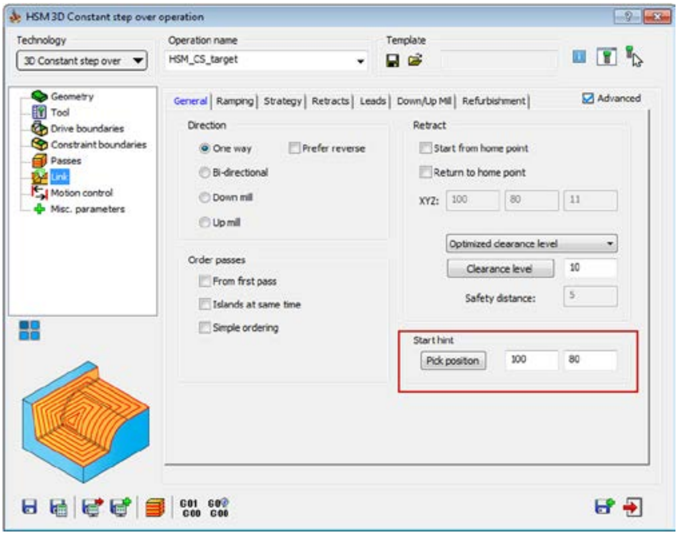
Clearance level

This field defines the plane where the rapid movements of the operation (between passes) will be performed. The **Clearance level** value can be set according to the **Part clearance level** or according to the automatically calculated **Optimized clearance level**.

Safety distance

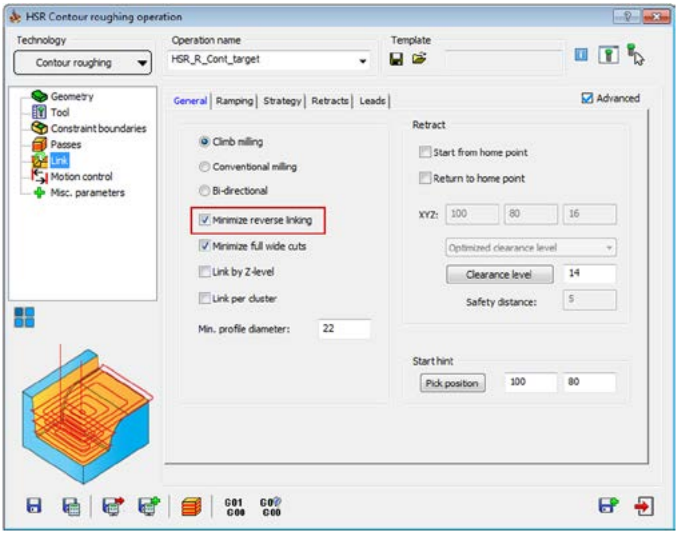
This field defines the distance to the **Upper level** at which the tool will start moving at the Z feed rate you have entered for the tool. Movements from the **Clearance level** to this height are performed in rapid mode.

7.1.4 Start hint



Enter the XY-coordinates of the starting position of the tool; the tool will move to this position at the beginning of the tool path. The default value for the **Start hint** is the center of your model. On larger models, where there is a great distance from the center of the model and your current work area, you may want to change these values. If there is more than one set of passes to be linked, the linking will start with the passes closest to the **Start hint** point.

7.1.5 Minimize reverse linking



This option will reduce the amount of reverse linking on the tool path. It will also ensure that the tool cutting direction is maintained when linking passes.

If this option is chosen, the linking moves within a Z-level will be adjusted to maintain climb or conventional milling.

If this option is not chosen, linking moves may conventionally mill even though climb milling is maintained for the passes and vice versa.

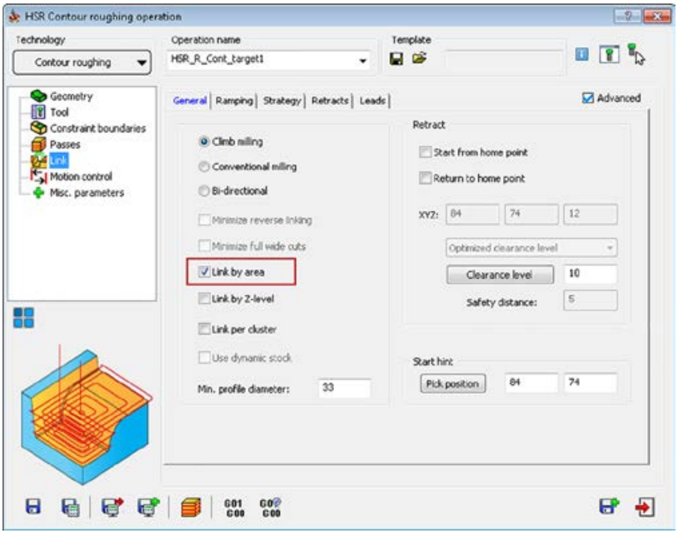
This option is only available if the **Detect Core areas** option (see topic **6.6.1**) of the **Contour roughing** strategy is enabled.

7.1.6 Minimize full wide cuts

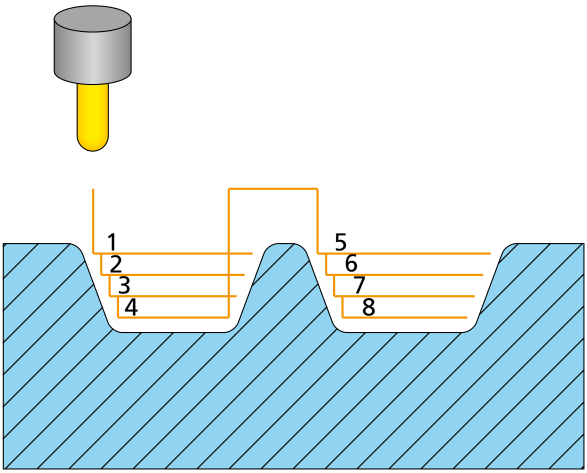
This option will reduce full width cuts wherever possible. This is useful because full width cuts (those which have equal width to the tool diameter) are not recommended in most machining situations.

This option is only available if the **Detect Core areas** option (see topic **6.6.1**) of the **Contour roughing** strategy is enabled.

7.1.7 Link by area



The **Link by area** option enables you to perform all the passes with linking within the same area, so that each area is machined completely before moving to the next one.

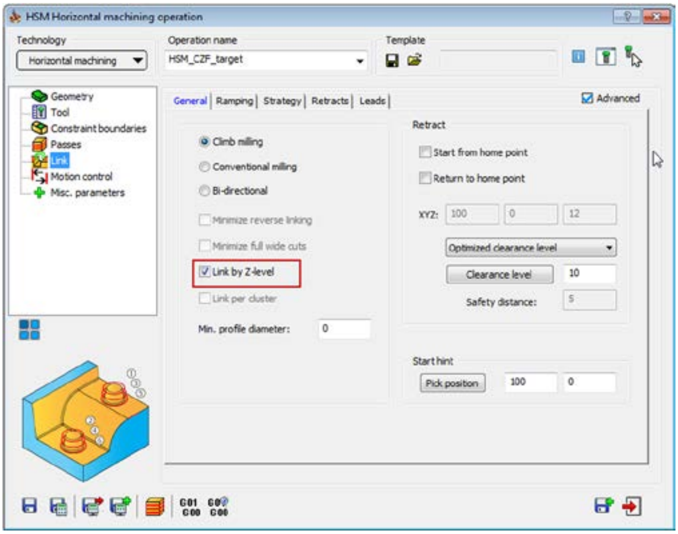


This option forces all passes on one Z-level within a single cavity/core to be machined before either moving down to the next Z-level or across to another cavity/core.

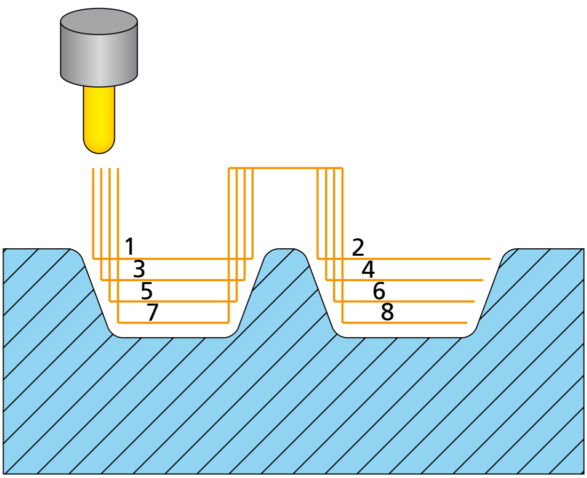
For **Contour roughing**, this parameter does not have effect if the passes are not edited, since all the passes on one Z-level within the area are machined before moving to the next level/area.

This option is available for the **Contour roughing** and **HM Roughing** strategies.

7.1.8 Link by Z-level



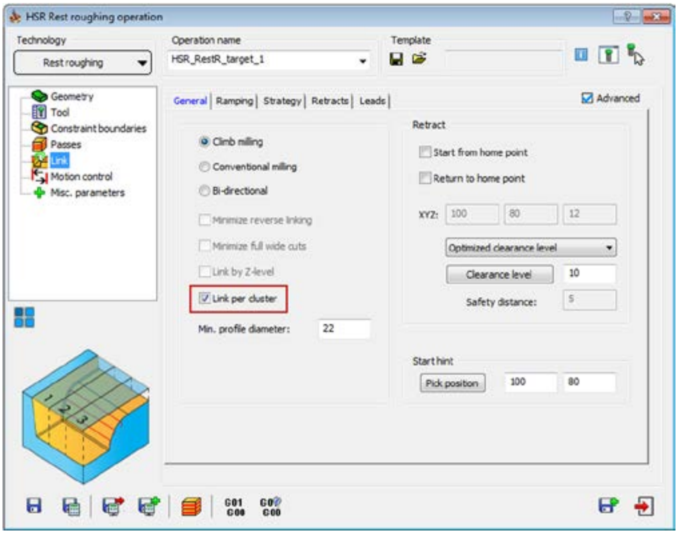
The **Link by Z-level** option enables you to perform all the passes at a specific Z-level before moving to the next one. This will frequently result in occurrence of air movements between different areas of the same Z-level.



By default the option is not chosen. It means that the passes are linked in such a manner that each area is machined completely before moving to the next one.

This option is available for the **Constant Z** and **Horizontal** strategies, and for all HSR operations.

7.1.9 Link per cluster



When you link machining passes that are made up of several different clusters of passes, in corners, for example, the **Link per cluster** option allows each corner to be machined before the tool moves to another corner. If you do not select this option, the machine may need to make a number of rapid feed rate moves to connect the clusters of passes.

This option is available for the **Contour roughing**, **Hatch roughing**, **Rest roughing** and **Horizontal** strategies.

7.1.10 Use dynamic stock

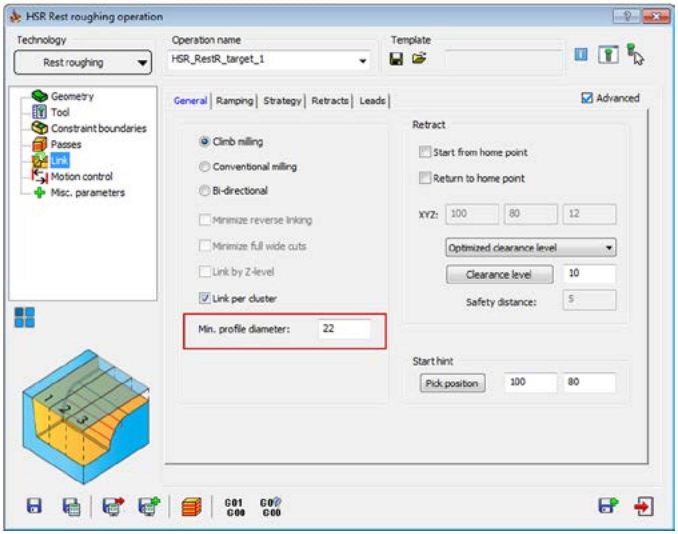
The **Use dynamic stock** option enables you to use the dynamic stock slices.

For all tool path linking operations, the stock lying outside the boundary is automatically protected when calculating linking motions. For roughing operations, uncut stock within the constraint boundary is also protected. The uncut stock can either be modelled as a static stock resulting in faster but less optimal tool path or a dynamic stock, which works slower but produces a better tool path.

A dynamic stock is the updated stock, therefore when avoiding collisions, linking is calculated according to the machined stock resulting in a more accurate tool path.

This option is available for the **Contour roughing**, **HM Roughing**, **Hatch roughing**, and **Horizontal** machining strategies.

7.1.11 Min. profile diameter

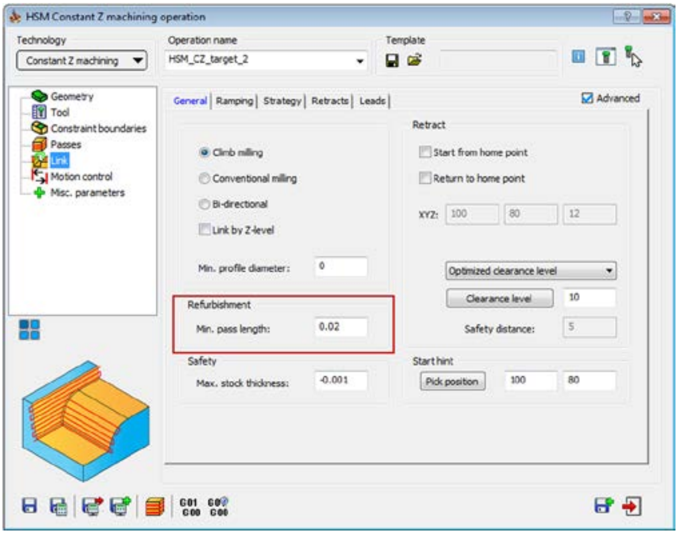


The diameter of a profile is its "span", which is the largest distance between two points of the profile. Any profile that is smaller than this value will not be machined to avoid difficulties in ramping the tool into this space. The default **Min. profile diameter** value is slightly smaller than that of the flat part of the end mill tool (and zero for ball-nosed tools).

For example, if the set of surfaces has a hole about the size of the tool you want to use, you will get a column of profiles that appear to "fall" through the hole down to the lowest Z-level. If you do not want these profiles, you can use the **Min. profile diameter** parameter.

This option is available for the **Constant Z** and **Horizontal machining** strategies.

7.1.12 Refurbishment

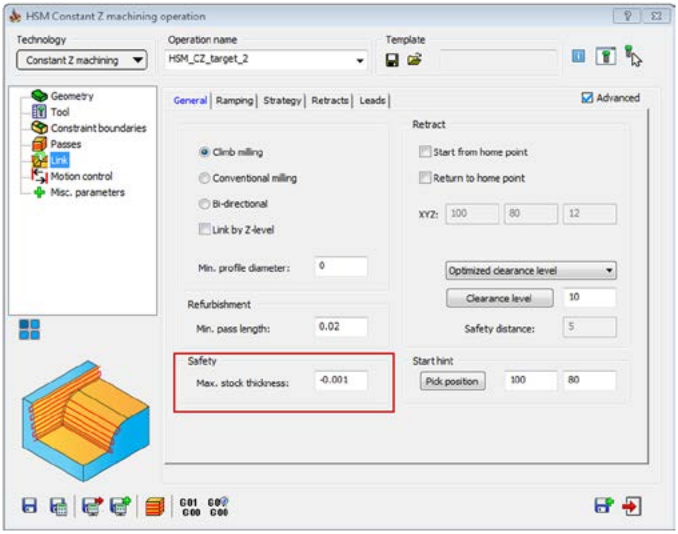


Min. pass length

The **Min. pass length** parameter enables you to define the minimal length of the pass that will be linked. Passes with length smaller than this parameter will not be linked. This option enables you to avoid the machining of extremely short passes and increases the machining performance.

This option is available for the **Constant Z** machining strategy.

7.1.13 Safety



Max. stock thickness

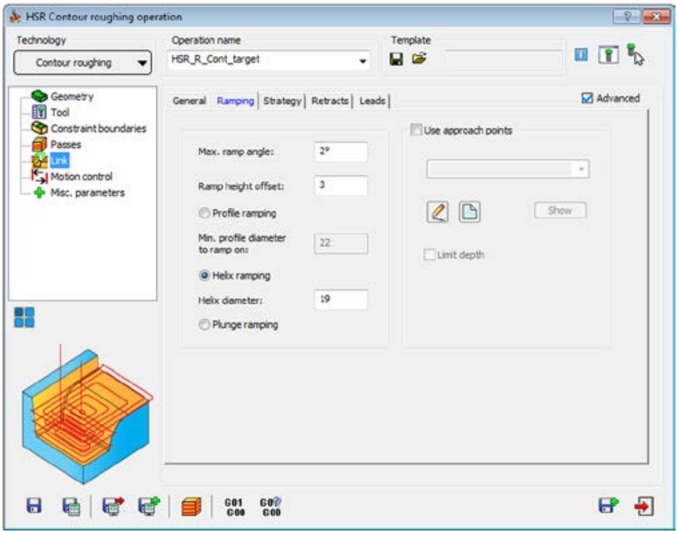
The **Max. stock thickness** parameter enables you to control the order of Constant Z machining of several cutting areas.

When the distance between cutting areas is smaller than the specified **Max. stock thickness** value, the machining is ordered by cutting levels. In this case InventorCAM machines all of these cutting areas at the specific cutting level, and then moves down to the next level.

When the distance between cutting areas is greater than the specified **Max. stock thickness** value, the machining is ordered by cutting areas. In this case InventorCAM machines a specific cutting area at all of the cutting levels, and then moves to the next cutting area.

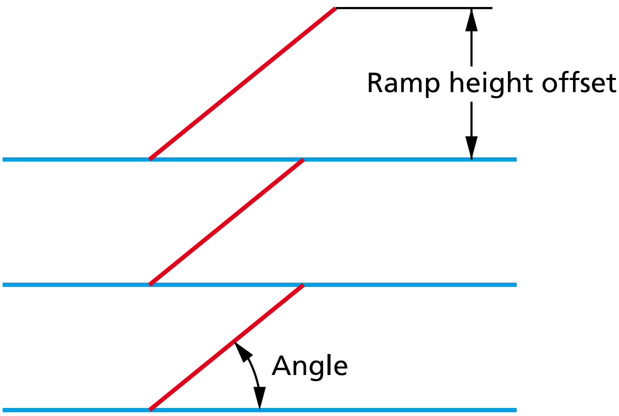
This option is available for the **Constant Z** machining strategy.

7.2 Ramping Parameters



The **Ramping** page enables you to control the ramping aspects of the tool path.

Ramping is used when the tool moves from one machining level down to the next one; the tool moves downwards into the material at an angle.



This page is available for all roughing strategies and **Constant Z**, **Horizontal**, and **3D Constant step over** strategies.

Max. ramp angle

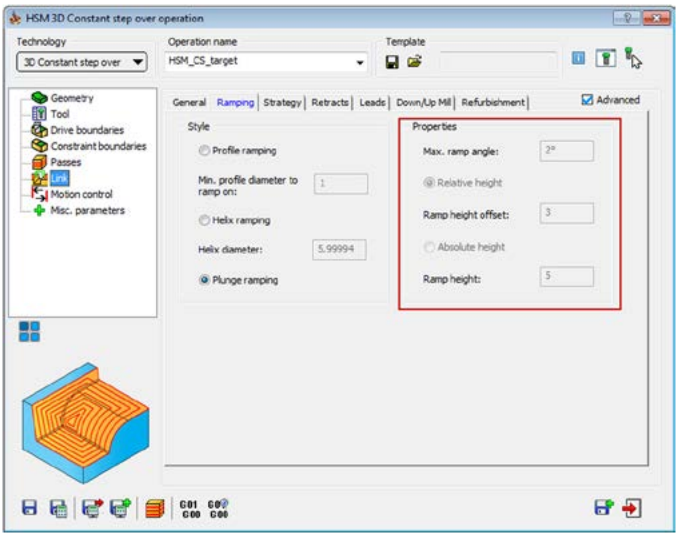
The Ramp angle is calculated automatically and depends on the model geometry and the tool type. The **Max. ramp angle** parameter enables you to limit this angle.

The dimensions and type of tool you are using and the power of your machine tool will determine an appropriate ramp angle. The angle used on a profile will often be shallower than this, as the ramp always steps forward by at least the shaft radius of the tool.

If a profile is very small, then the angle used might have to be larger than you specify. In this case you can avoid the machining of short profiles with the **Min. profile diameter** (see topic **7.1.11**) parameter located in the **General** page.

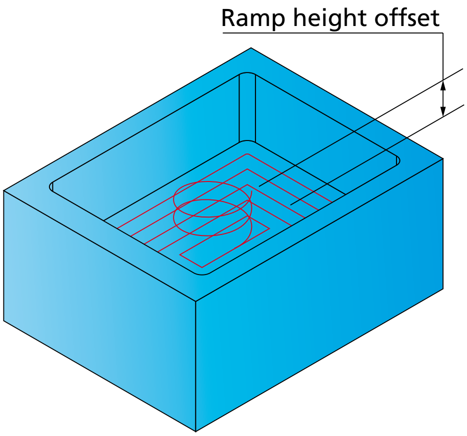
Relative and absolute ramp height

InventorCAM enables you to define also the relative or absolute start position for the ramp motion with the **Ramp height offset**/**Ramp height** parameter measured from the Coordinate System origin.



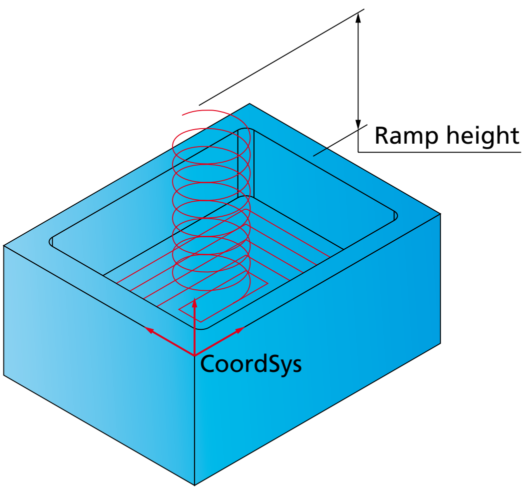
- **Relative height**

  With this option, the start position of the ramp motion for the upper Constant Step over pass is defined relative to the first point of the pass using the **Ramp height offset** parameter.



- **Absolute height**

  With this option, the start position of the ramp motion is defined with the absolute **Ramp height** value measured from the Coordinate System origin.



These options are available only for the **3D Constant step over** machining, when **Helix** and **Profile** ramping strategies are used.

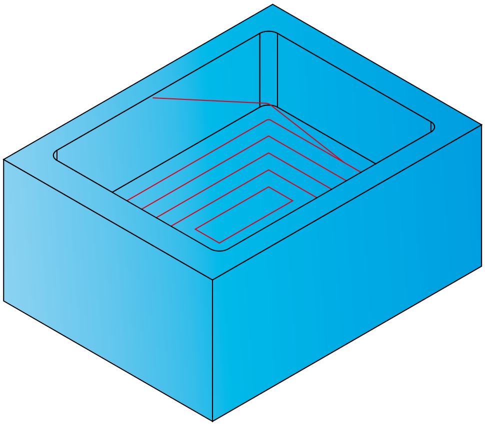
Ramp height offset

This parameter defines the height used in the ramping motion to the first upper profile. It ensures that the tool has fully slowed down from rapid speeds before touching the material so that it enters the material at a ramping angle.

InventorCAM enables you to perform the ramp movement either with a profile, or with a helix (spiral).

Profile ramping

The tool performs the downward movements to the specific Z-level around the contour of the profile.

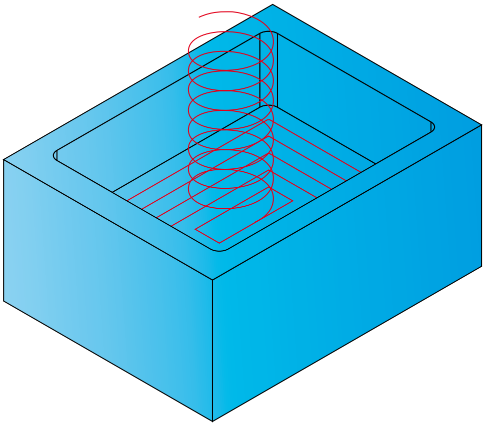


Min. profile diameter to ramp on

InventorCAM enables you to avoid ramp movements along small profiles, as a very tight tool motion would counterbalance any advantages gained by ramping for the smoothness of transition; by setting a minimum profile diameter ("span") you will be able to ensure that small profiles will not be ramped down to.

Helix ramping

The tool performs the downward movements to the specific Z-level in a corkscrew fashion, ensuring a smooth movement. Helix ramping also puts less load on the tool than profile ramping.

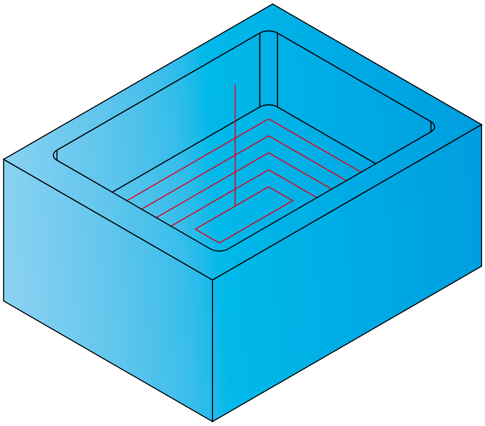


Helix diameter

This is the diameter of the ramping helix. In cases where the profile is too small for a helix ramp of this diameter, Profile ramping is used.

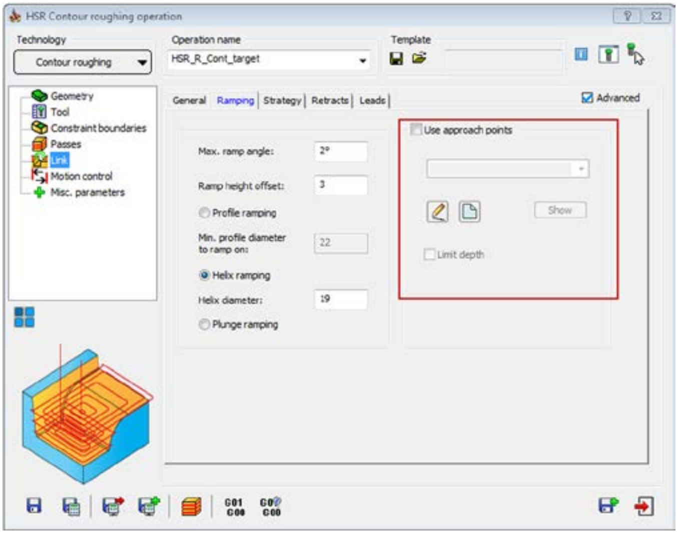
This option is available for the **Morphed machining**, **Radial machining**, **Linear machining**, **Spiral machining**, and **3D Constant step over** strategies.

Plunge ramping



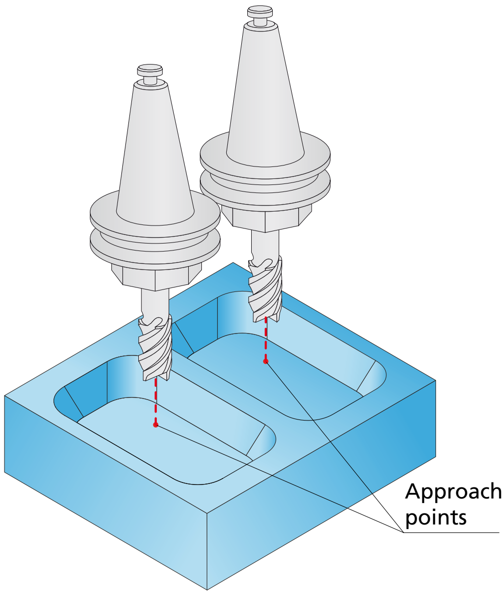
The tool performs the downward movements to the specific Z-level in a vertical movement.

Use approach points

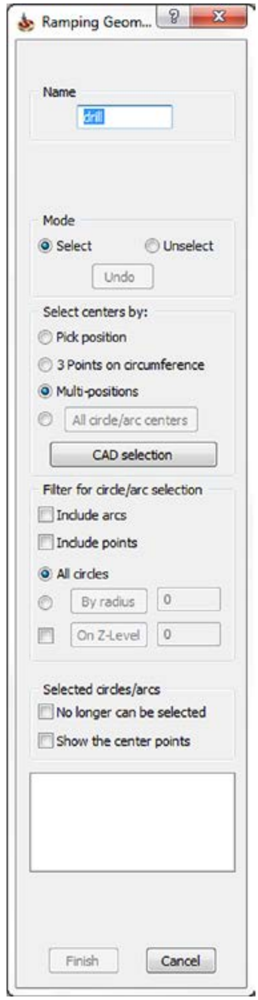


This section enables you to define the ramping by approach points in pre-drilled holes. The centers of the holes are defined as approach points.

No tool path for a specific area is generated, if the approach point is defined outside of the area.



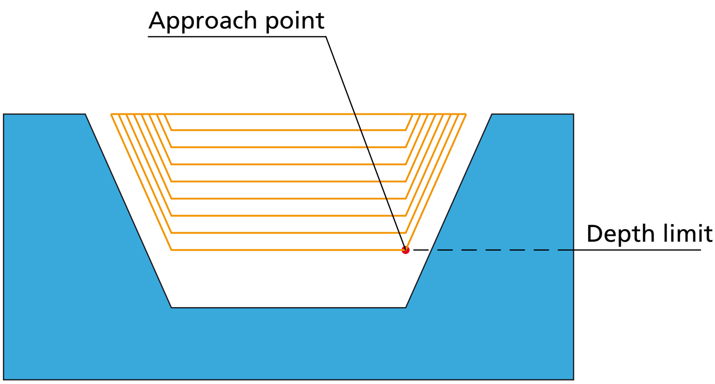
To define the approach points, click the **Define** button. The **Ramping Geometry Selection** dialog box is displayed. This dialog box enables you to define the approach points by picking on the solid model.



The content of this dialog box is similar to that of the **Drill Geometry Selection** dialog box used for drilling geometry definition (for more information, refer to **InventorCAM Milling Online Help**).

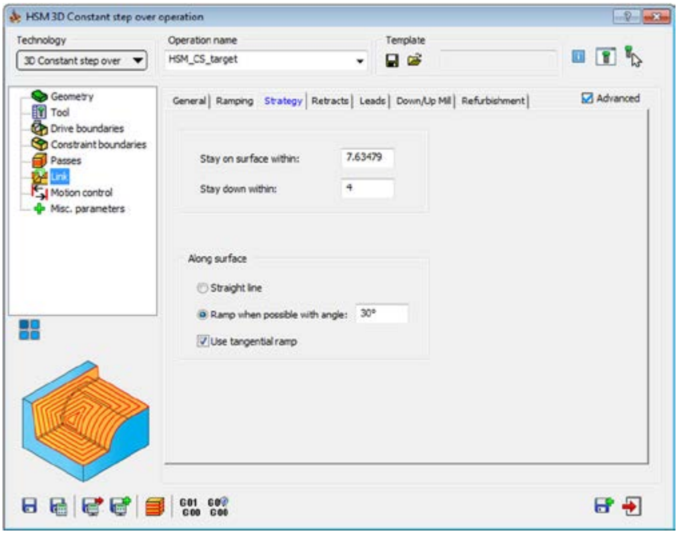
Limit depth

This check box controls the maximum depth at which the cutting in the current area will be performed. When this check box is selected, the Z-coordinate of the lowest approach point defines the maximum depth at which the cutting will be performed; the tool will not descend lower than the Z-level of this point.



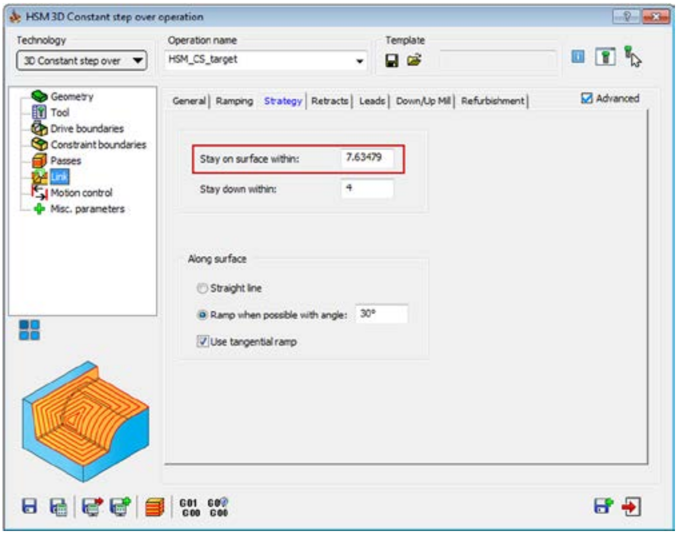
7.3 Strategy Parameters

The **Strategy** pages enables you to define the following parameters related to the linking strategy.

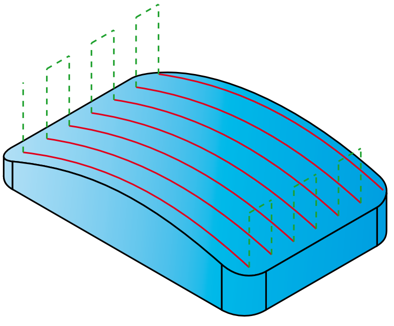


- **Stay on surface within**
- **Stay down within**
- **Along surface**
- **Linking radius**
- **Link clearance**
- **Horizontal link clearance**
- **Trim to ramp advance**

7.3.1 Stay on surface within

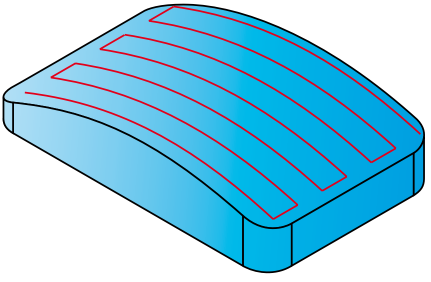


The **Stay on surface within** parameter enables you to control the way how the tool moves from the end point of a pass to the start point of the next one. When the distance between these points is greater than the specified parameter value, the tool movement is performed at the **Clearance plane**, using rapid feed.



When the distance between the points is smaller than the parameter value, the tool moves with cutting feed directly on the model face.

This option enables you to decrease the number of air movements between the passes of the tool path.



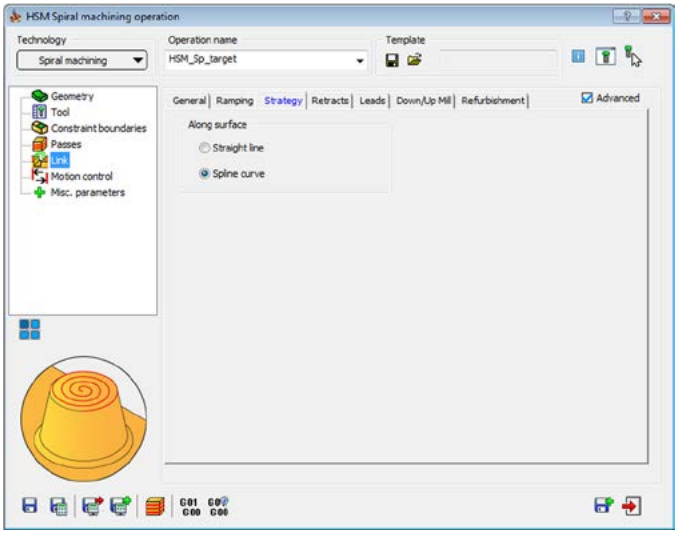
To control the manner of the link movement between passes when the tool moves on surface, use the **Along surface** option (see topic **7.3.2**).

7.3.2 Stay down within

The **Stay down within** parameter enables you to control the point of choosing between retracting or staying on the surface. When the distance between the end point of a pass and the start point of the next one is less than the specified parameter value, the cutting tool stays on the surface. If the distance is greater than the specified parameter value, the cutting tool retracts to the clearance plane.

This option is disabled when **Shortest route** is selected from the **Style** section in the **Retracts** tab. This option is available for the **Constant Z machining**, **Helical machining**, **Horizontal machining**, **Boundary machining**, **Rest machining**, **Pencil milling**, and **Parallel pencil milling** strategies.

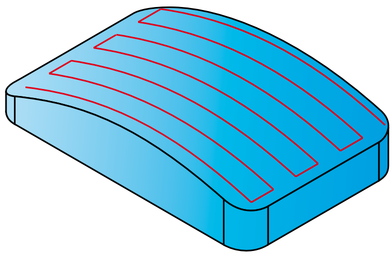
7.3.3 Along surface



When the tool moves on the surface, links between passes can be:

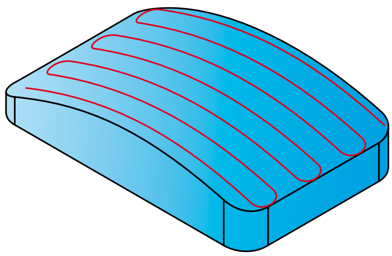
- **Straight line**

  When this option is active, a direct connection is made on the surface in a straight line.



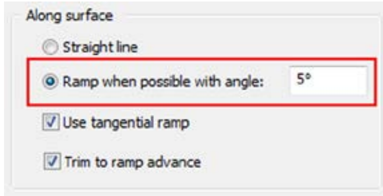
- **Spline curve**

  When this option is active, a spline connection is made along the surface. The movement is smooth; there are no sharp corners so there is little change of speed of the tool throughout the length of the link.

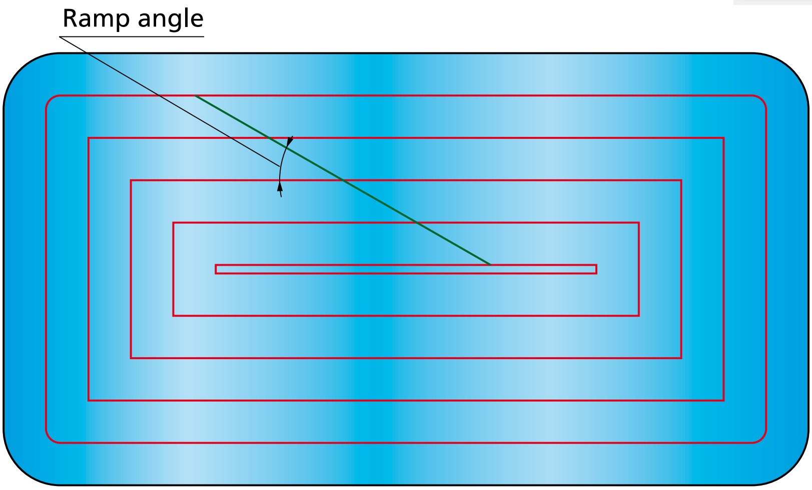


These options are available for the **Linear machining**, **Spiral machining**, **Radial machining**, **Boundary machining**, and **Pencil milling** strategies.

Ramp when possible with angle

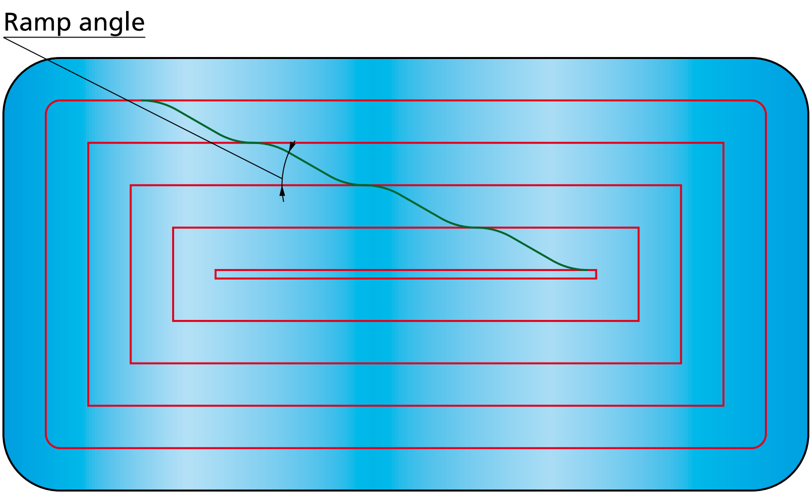


The **Ramp when possible with angle** option enables you to perform the connection along the surface at the specified angle.



- **Use Tangential Ramp**

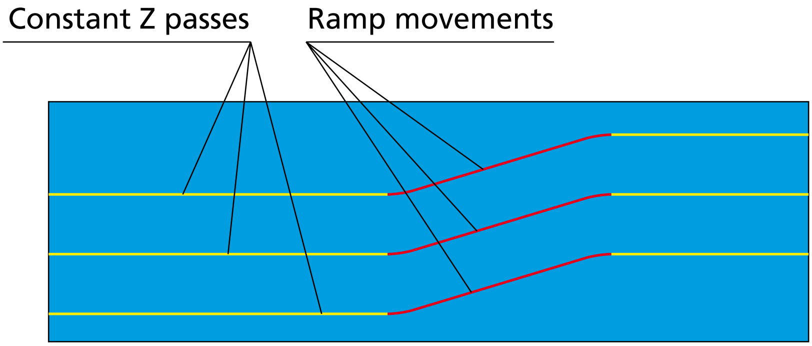
  This option enables you to perform the angled link movements in a smooth S-curve. With this option the transition between passes is performed smoothly without sharp corners.



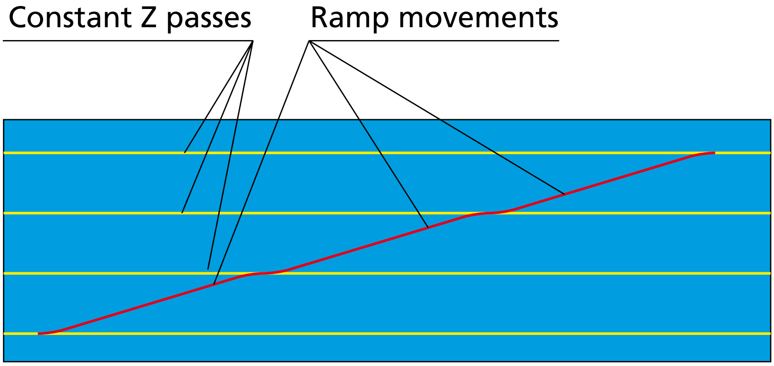
- **Trim to ramp advance**

This option enables you generate a helical style finish when linking Constant Z passes.

When this check box is selected, the Constant Z pass above which a ramp linking movement is performed is trimmed by the length of the ramping move. In such a way a helical style tool path is generated, avoiding the unnecessary cutting moves at the already machined areas and maintaining a constant tool load.

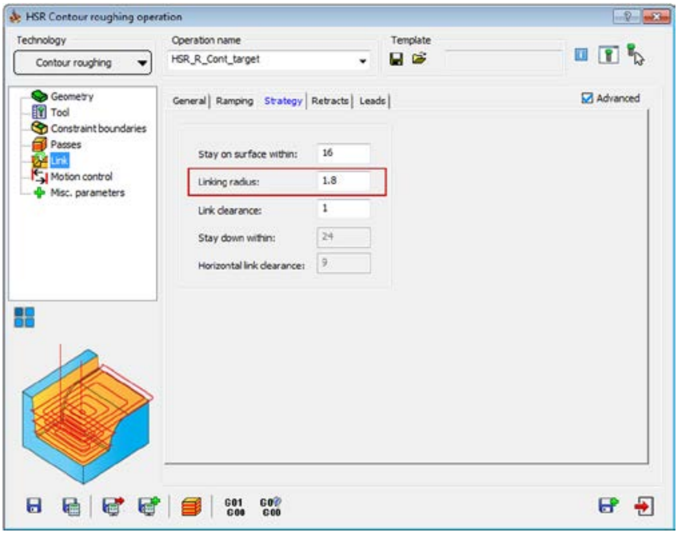


When this check box is not selected, the whole Constant Z passes are linked with the ramp movements.

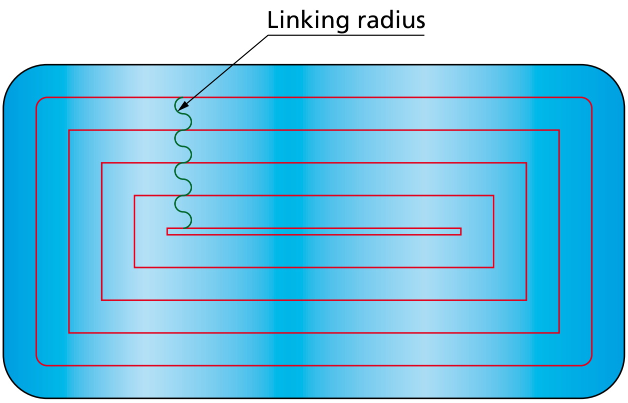


The **Ramp when possible with angle** option only has effect on passes that consist of closed loops at different Z-heights, such as **Constant Z** and **3D Constant step over** passes.

7.3.4 Linking radius



Using this parameter, InventorCAM enables you to generate S-shaped curves linking the adjacent closed passes of the contour machining. The value defines the radius of the link arc. If you set the **Linking radius** to **0** or turn off **Smoothing** then a simpler, straight-lined route will link each loop.

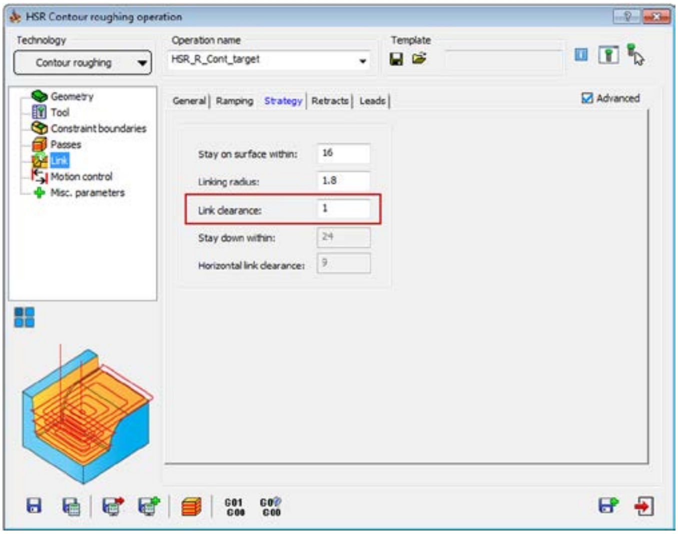


When the radius is set to zero, straight line link movements are performed.

In the **HM Roughing** strategy, the default value of the **Linking radius** is set as 20% of the defined **Step over**.

These options are available for **Contour roughing, Horizontal machining,** and **HM Roughing** strategies.

7.3.5 Link clearance



With this parameter, InventorCAM enables you to maintain a horizontal clearance from the bounding profile when moving horizontally from one location to another. The value defines the minimal distance from the bounding profile.

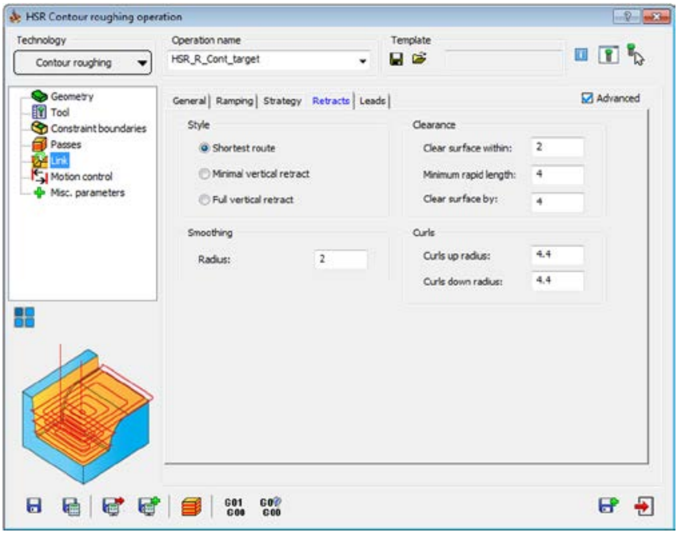
This option is available for **Contour roughing, Hatch roughing, Rest roughing** and **Horizontal machining**.

7.3.6 Horizontal link clearance

When the **Detect core areas** (see topic **6.6.1**) option is used, the **Horizontal link clearance** parameter defines the distance outside of the material to perform plunging.

This option is available for **Contour roughing** and **Rest roughing**.

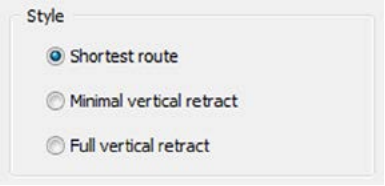
7.4 Retracts Parameters



This page enables you to control retract movements between passes of the tool path.

- **Style**
- **Clearance**
- **Smoothing**
- **Curls**

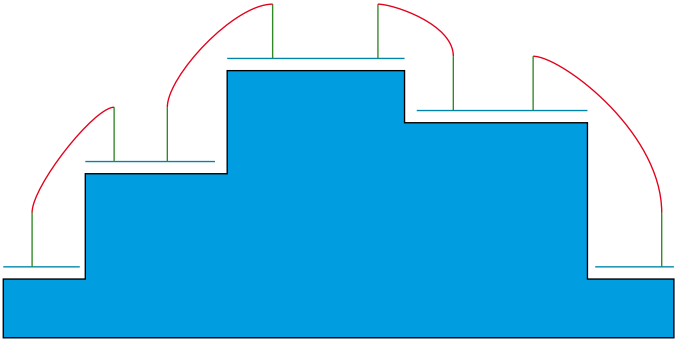
7.4.1 Style



The **Style** options enable you to define the way how the retract movements are performed between passes.

Shortest route

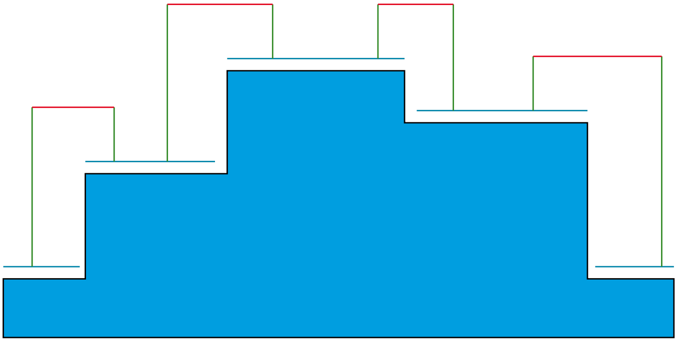
The tool performs a direct movement from one pass to another. InventorCAM generates a curved retract movement trajectory. The minimum height of the retract movement is controlled by the **Clear surface by** parameter, and the curve's profile is controlled by the **Smoothing** and **Curls** parameters.



This style is chosen by default, as it creates the shortest retract movements. However, some machine tools are unable to perform the rapid movement effectively along a curved path; in these cases you can choose one of the other two retract styles.

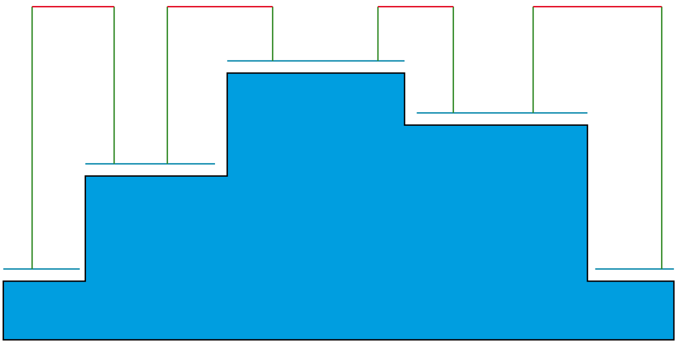
Minimal vertical retract

The tool moves vertically to the minimum Z-level where the safe rapid movement can be performed, moves along this plane in a straight line and drops down vertically to the start point of the ramp movement to the next pass. The minimum height of the retract is controlled by the **Clear surface by** parameter.



Full vertical retract

The tool moves vertically up to the clearance plane, rapidly moves at this level in a straight line, and drops down vertically to the start point of the ramp movement to the next pass.



7.4.2 Clearance

The **Clearance** parameters apply both to the lead in and the lead out components of retract motions.

Clear surface within

This option affects the tool path when the **Shortest route** style is chosen. It specifies the distance the tool moves away from the surface with the cutting feed rate, before the rapid movement starts.

The distance is measured from the end of the lead out arc to the point where the tool is guaranteed to be clear of the surface.

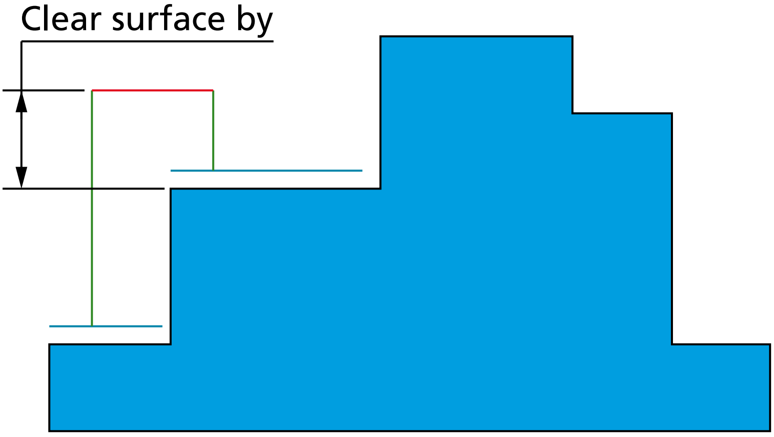
Minimum rapid length

This option is used to specify a limit. If the 2D distance between the ends of the lead moves is less than this value, a 3D connecting move at lead feed rate is used. Otherwise a retract move is used.

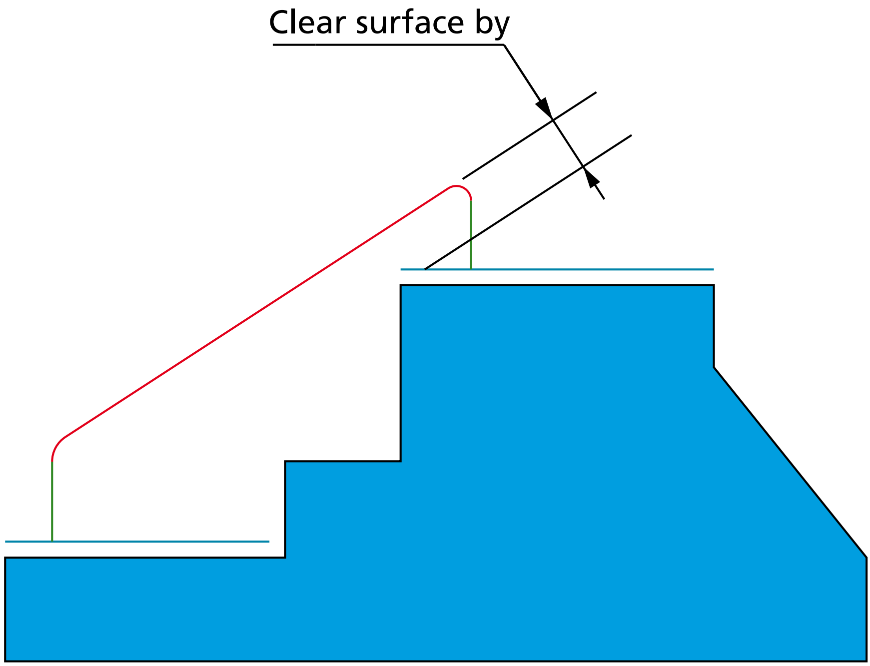
Clear surface by

This is the minimum distance by which the tool will be clear of the surface during its rapid linking motion. All points of the tool – on both the tip and the side have to avoid the surface by this distance.

For **Minimal vertical retract** motions, the tool lifts up to a height that ensures clearance.

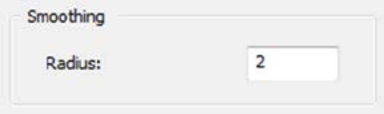


For **Shortest route** motions, the tool is lifted up above the surface to ensure the clearance, then it performs rapid motion maintaining the **Clear surface within** distance.



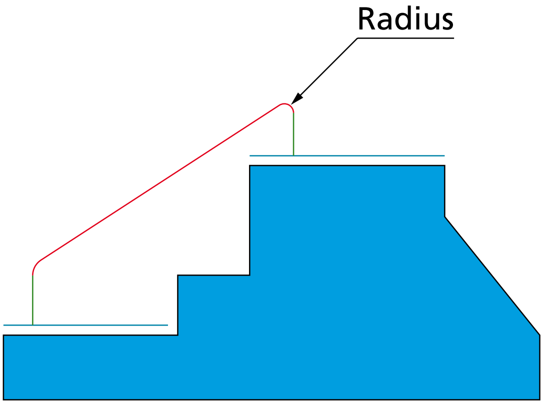
This clearance is applied in addition to any wall offset that you have already specified for the tool. In particular, with a negative wall offset, the clearance is above the reduced surface and not the real surface – so you should set this value higher to prevent the tool from gouging the surface.

7.4.3 Smoothing



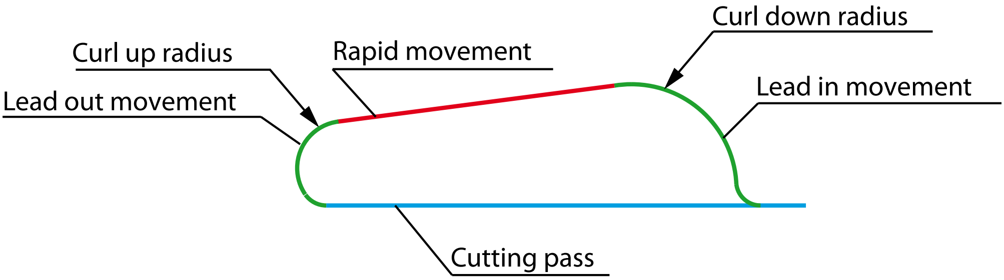
Radius

InventorCAM enables to round sharp corners of the retract motions when the **Shortest route** option is used by adding a vertical curve of a defined radius. This makes the tool movement smoother and enables higher feed rates.



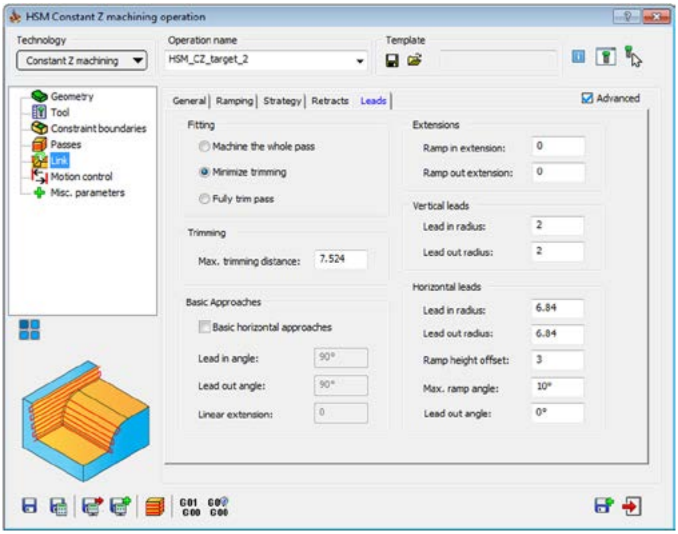
7.4.4 Curls

InventorCAM enables you to add arcs in the end if the lead-out movements and in the beginning of the lead in movements. The **Curl over radius** and **Curl down radius** define the radii of these arcs.



The **Curls** options affect the tool path when the linking style is **Shortest route**.

7.5 Leads Parameters



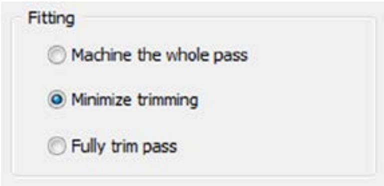
The parameters located on this page enable you to control the lead in and lead out motions.

- **Fitting**
- **Trimming**
- **Basic Approaches**
- **Extensions**
- **Vertical leads**
- **Horizontal leads**

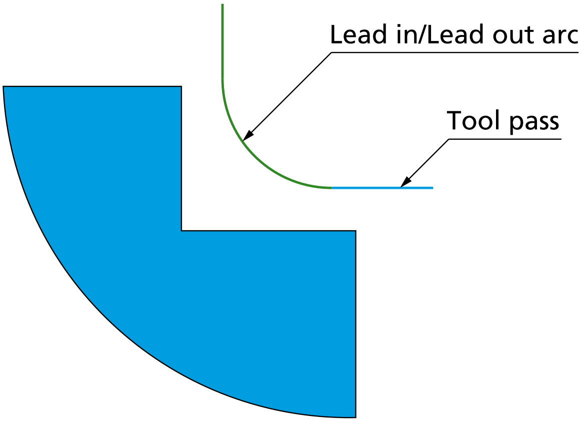
The **Stay on surface within** parameter located on the **Strategy** page enables you to define the maximum distance between passes to stay on the surface and when to perform a retract movement.

The style of the retract movement can be defined on the **Retracts** page.

7.5.1 Fitting

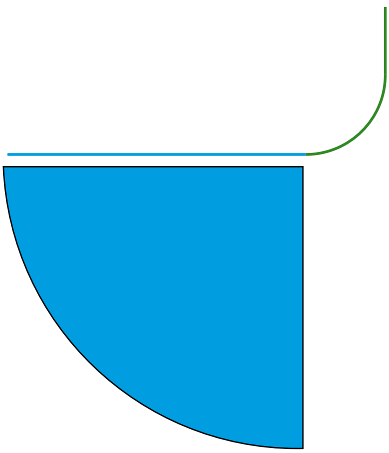


You define here how the lead in and lead out arcs of the retract movements fit to the machining pass.

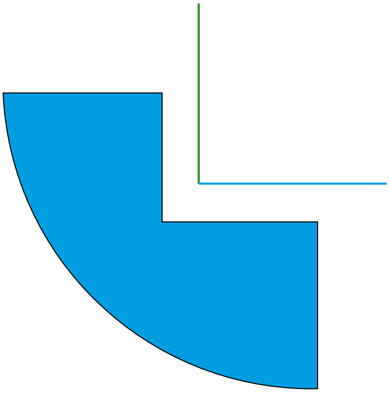


Machine the whole pass

With this option the complete pass is machined. The arc can be applied at the end of the pass, without trimming of the pass.



The arc can be inserted only if it can be done safely without gouging the part faces. When the arc is conflicting with the model geometry, a straight vertical lead in/out movement is performed.

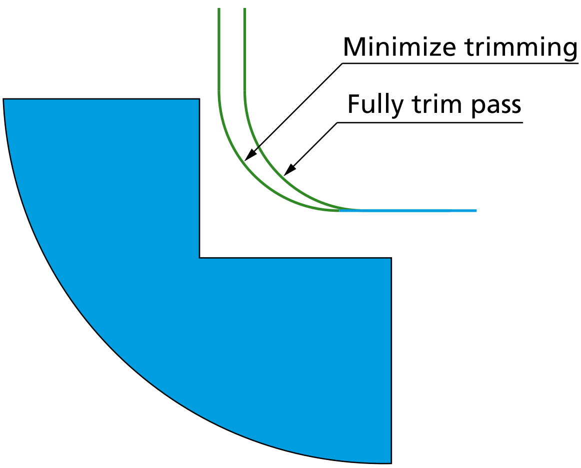


Minimize trimming

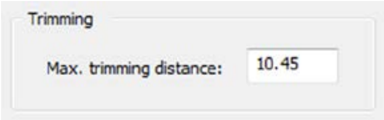
This option enables you to perform the arc retract movement with minimal possible trimming of the cutting tool pass. The retract pass is as close to the surface as possible, maintaining a minimum distance from the surface to fit the arc of the defined radius.

Fully trim pass

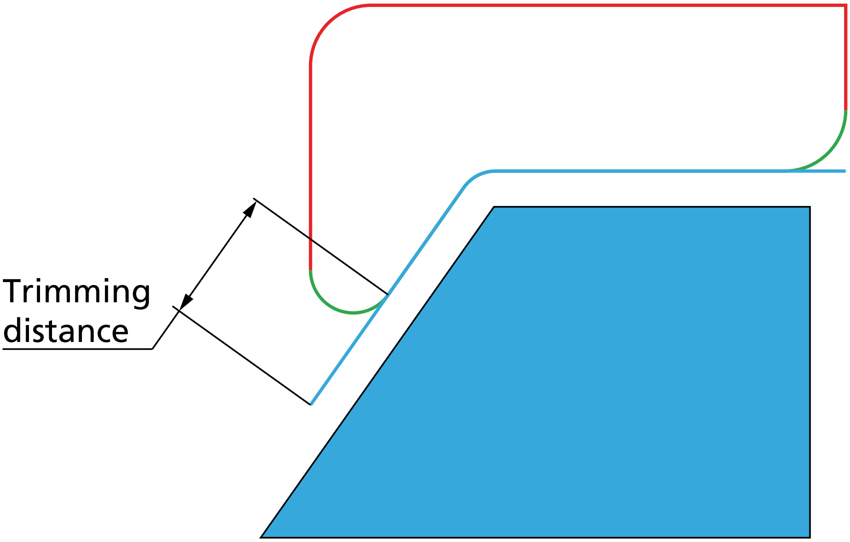
In cases where it is crucial to prevent over-machining, this is a good and cautious strategy modification. The pass is trimmed back so the entire arc fits into it, but no nearer than a full machine pass link would be.



7.5.2 Trimming



When a lead arc is added to a horizontal machining pass, the length of pass trimmed off will be at most the radius of the arc. However, when adding an arc to a steep finishing pass, the length of pass trimmed (trimming distance) will be much greater.

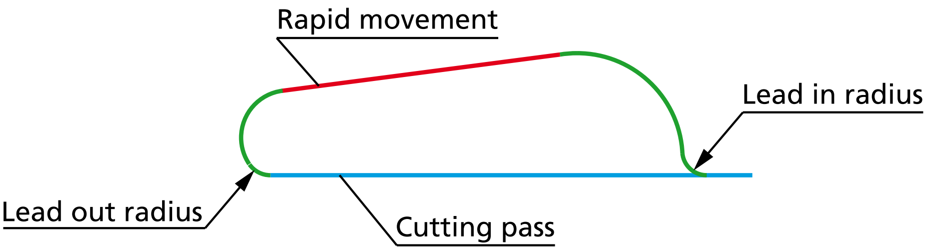


Such trimming of the passes can result in the occurrence of large unmachined areas. To avoid this, InventorCAM enables you to limit the trimming distance with the **Max. trimming distance** parameter. If the trimming distance exceeds this value, then no arc is used; the whole pass is machined, and a straight vertical motion is added.

This option affects the path when the **Lead fitting** is **Minimize trimming** or **Fully trim pass**.

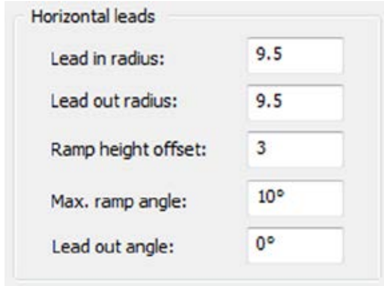
7.5.3 Vertical leads

The **Vertical leads** parameters enable you to define the radius of the arcs located in a vertical plane used to enter and leave the machining pass.



7.5.4 Horizontal leads

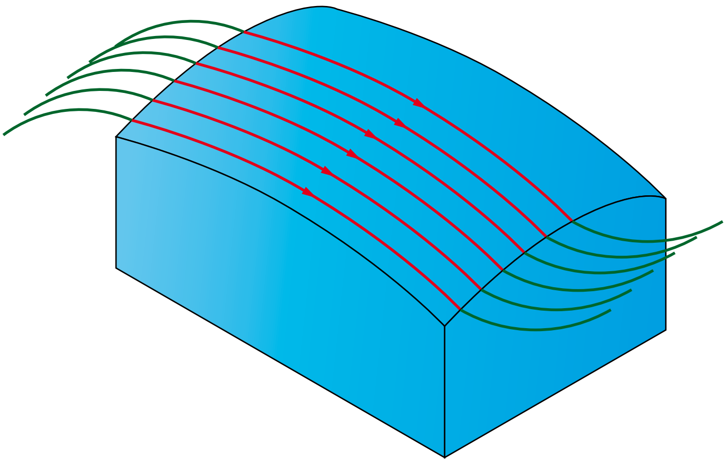
InventorCAM enables you to perform Horizontal lead in/out movements to provide you with smooth entering/exiting from the material.

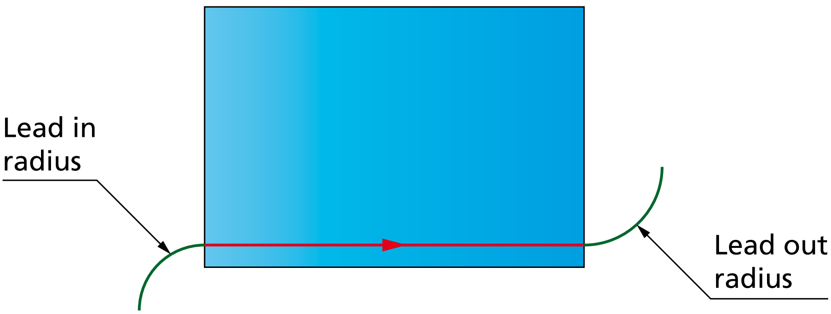


Using horizontal leads the tool path can be set up so that the tool approaches and leaves machining passes tangentially using helical moves. Note that if the requested radius (Lead in or Lead out) is too large, then the horizontal lead is omitted, and only vertical leads are used.

Lead in/out radius

These parameters enable you to define the radius of the arcs, located in a horizontal plane, used to enter and leave the machining pass.



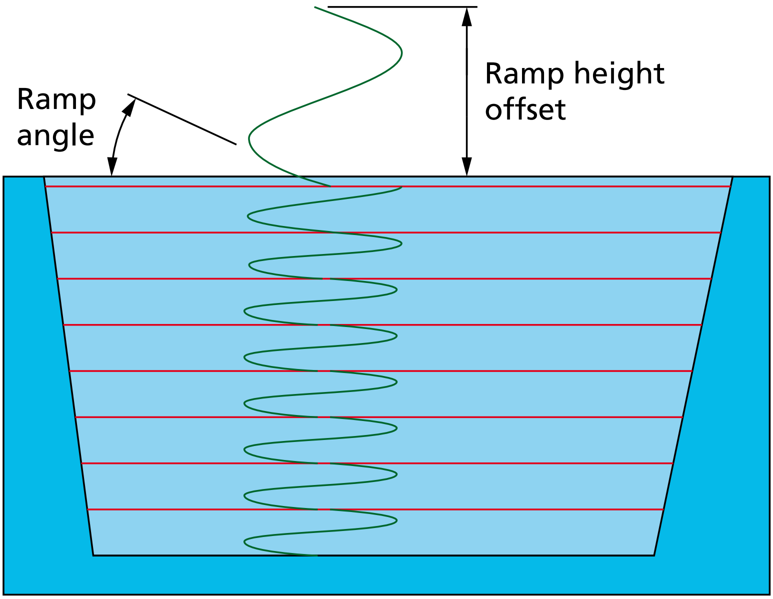


Max. ramp angle

InventorCAM enables you to perform ramp down movements during the arc lead in. The **Max. ramp angle** parameter enables you to limit the maximum angle (measured from the horizontal plane) for ramping.

Ramp height offset

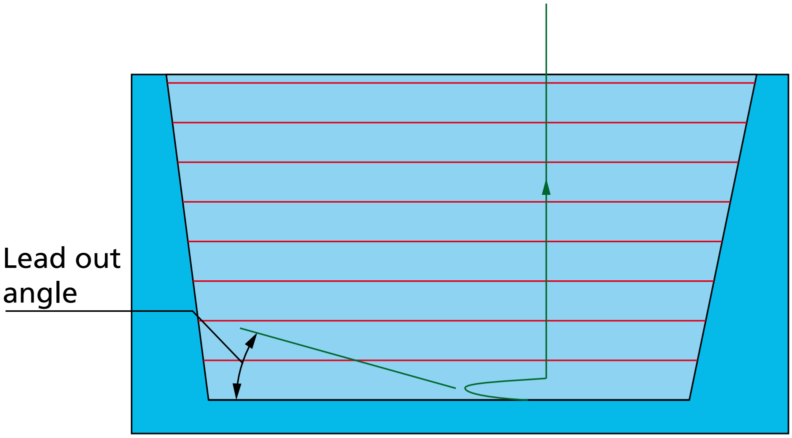
The ramp height offset is an extra height used in the ramping motion down from a top profile. It ensures that the tool has fully slowed down from rapid speeds before touching the material, and that it enters the material smoothly at the ramping angle.



The **Max. ramp angle** and **Ramp height offset** parameters are available for the **Contour roughing**, **Hatch roughing**, **Rest roughing** and **Constant Z** strategies.

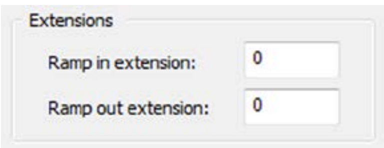
Lead out angle

InventorCAM enables you to perform inclined upwards movements during the arc lead out. The **Lead out angle** parameter enables you to define the angle of inclined lead out movement. The angle is measured from horizontal plane.



The **Lead out angle** parameter is available for the **Contour roughing**, **Hatch roughing**, **Rest roughing** and **Constant Z** strategies.

7.5.5 Extensions



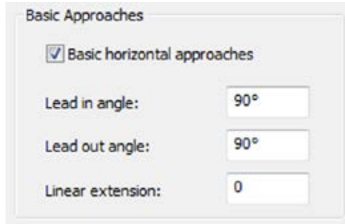
Ramp in extension

The ramp in height offset is an extra height used in the ramping motion down from a top profile. It ensures that the tool has fully slowed down from rapid speeds before touching the material so that it enters the material smoothly at the ramping angle.

Ramp out extension

The ramp out height offset is an extra height used in the ramping motion. It ensures that the tool speeds up to rapid speeds gradually.

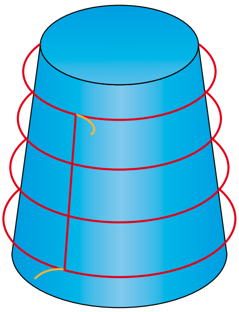
7.5.6 Basic approaches



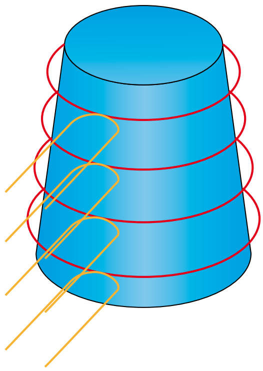
The **Basic horizontal approaches** check box enables you to perform a separate lead in/lead out movement for each Z-level pass in **Constant Z** machining. This functionality is useful for smooth entrance into the material from outside in semi-finish machining of steep model areas.

When this check box is not selected, the tool enters the material according to the defined approach/retreat parameters, and the Z-level passes are machined successively one after the other without exits from the material.

When this check box is selected, the tool exits from the material after machining of each Z-level pass and enters again on the next Z-level pass.



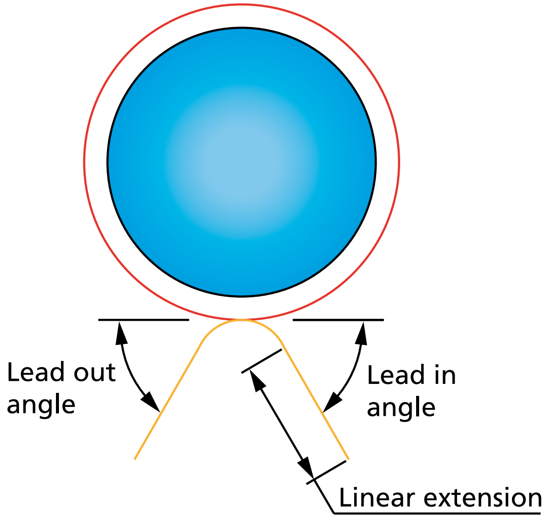
Basic horizontal approaches check box is not selected



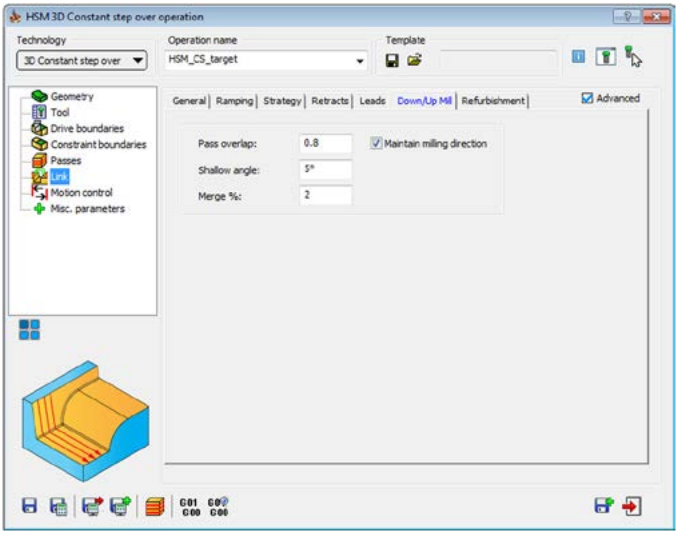
Basic horizontal approaches check box is selected

- The **Lead in angle** parameter defines the angle between the tool approach line and the line tangent to the Z-level pass in the start point of the machining.
- The **Lead out angle** parameter defines the angle between the tool retreat line and the line tangent to the Z-level pass in the end point of the machining.

- The **Linear extension** parameter defines the length of the tool horizontal approach line.



7.6 Down/Up Mill Parameters



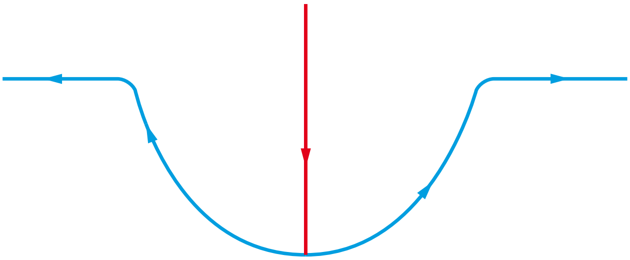
This page enables you to define the parameters of the **Down/Up** milling.

This page is available for all strategies, except for **Contour roughing**, **Hatch roughing**, **Rest roughing**, **Hybrid Rib roughing**, **Horizontal Machining** and **Constant Z** machining.

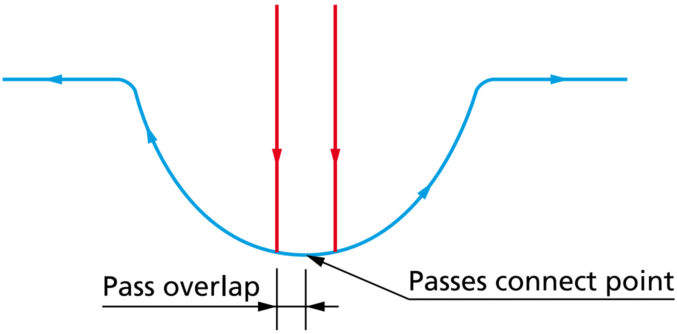
Down/Up milling options should be first selected on the **General** page, to have the parameters on this page operable.

Pass overlap

When a pass is broken in order to perform down and up movements, each segment can be extended, from the point where pass segments are connected, so that they overlap. This ensures a smoother finish.



No Pass overlap

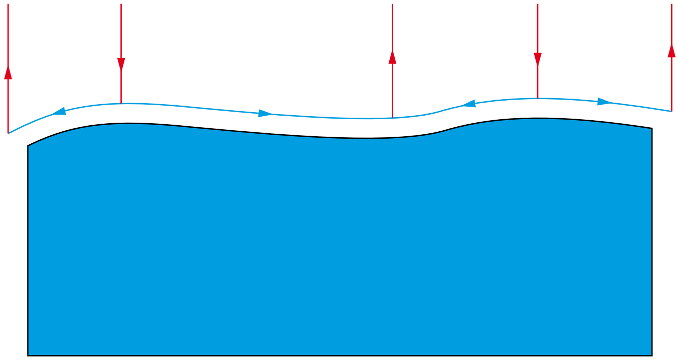


Since both pass segments are extended by the **Pass overlap** value, the actual length of overlap is twice the defined value.

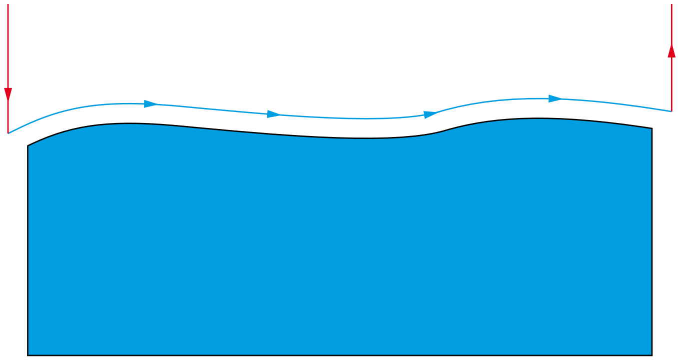
Shallow angle

Model areas with the inclination angles less than the **Shallow angle** value are considered as shallow. Such areas can be machined in either direction, as obviously **up** or **down** milling is irrelevant, and in these areas the tool path will be less broken up.

The image below illustrates the case when the inclination angles of the model faces are greater than the defined **Shallow angle** value.



In the image below, the **Shallow angle** value has been increased resulting in no break up of the tool path.



Merge %

InventorCAM enables you to machine some segments of the tool path upwards where downward movement is preferred, and vice versa, to avoid too much fragmentation.

The **Merge %** parameter defines the limit length of the opposite segments as a relative percentage of the whole pass. When the percentage of the segments length where the direction of the machining to be changed is less than the defined value, the direction will not be changed.

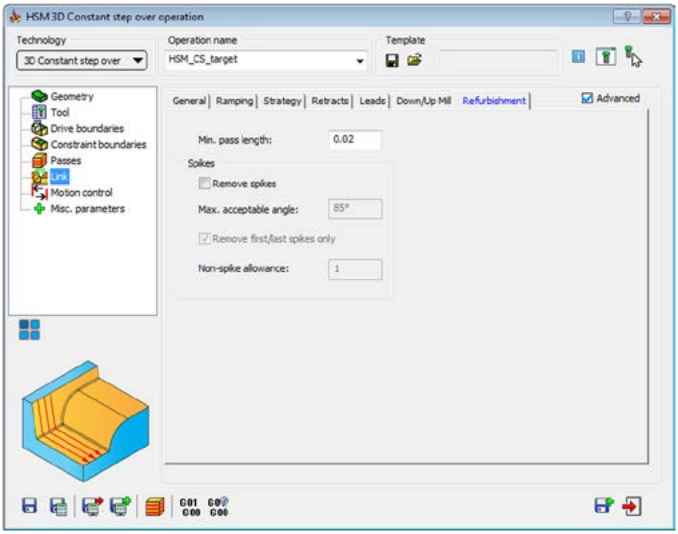
Maintain milling direction

This option affects the ordering of **Linear**, **Radial**, **Spiral** and **3D Constant Step over** passes. It ensures that all segments will either be climb milled or conventionally milled, if selected.

When the **Maintain milling direction** check box is not selected, passes will be either climb or conventional passes, depending on the relative position of the tool at the time.

7.7 Refurbishment Parameters

This page enables you to define a number of parameters of the tool path refurbishment.



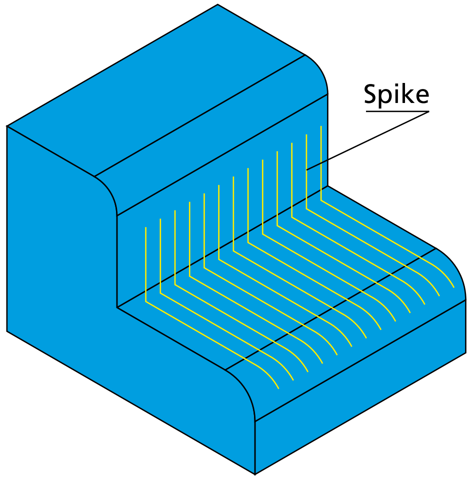
7.7.1 Min. pass length

The **Min. pass length** parameter enables you to define the minimal length of the pass that will be linked. Passes with length less than this parameter will not be linked.

This option enables you to avoid the machining of extremely short passes and increases the machining performance.

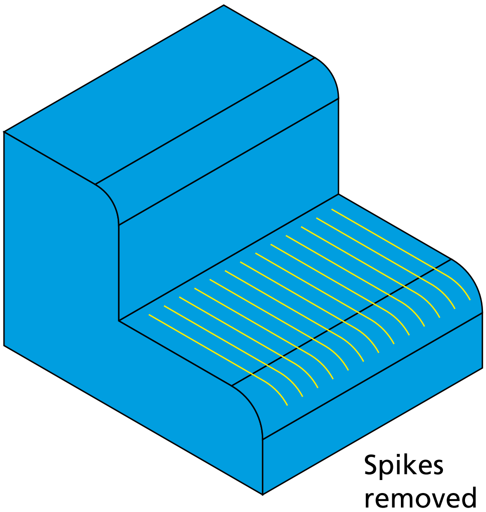
7.7.2 Spikes

Sometimes at the end of a pass, where one surface is adjacent to another at a very steep angle, there is a sharp jump. This can happen where the tool touches a steep wall and is lifted to the top, or where it "falls off" a high ledge and drops to the bottom. InventorCAM enables you to remove these spikes.



Remove Spikes

This option enables you to remove sharp jumps (spikes) from the tool path.



- **Max. acceptable angle**

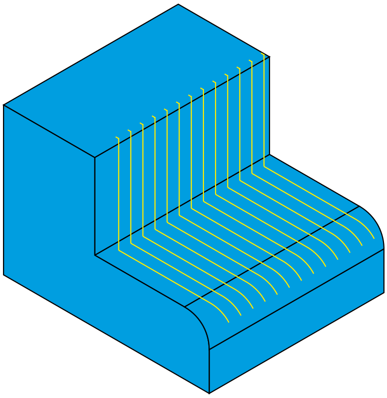
  Spikes or jumps with an angle greater than this value are removed from the tool path. The angle is measured from the horizontal plane.

Remove first/last spikes only

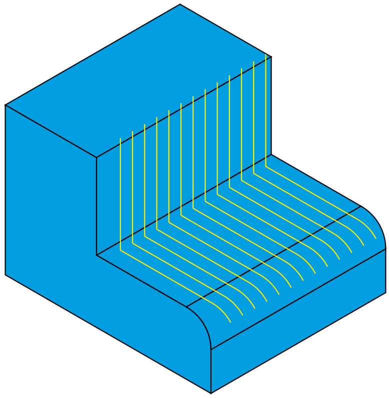
When this option is active, only spikes at the beginning and end of passes are removed. There will be no spike removal on a looped pass if this option is active, as there is no pass end.

- **Non-spike allowance**

  You can trim off any small horizontal areas left at the top or bottom of the spike. The value here is the maximum length of horizontal pass that will be removed from the tool path.



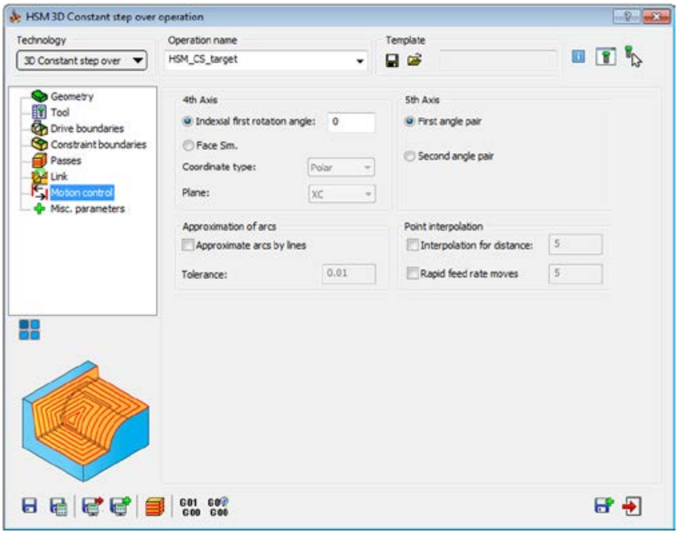
Horizontal passes
at the top of spikes



Horizontal passes
trimmed

Motion Control 8

Using the parameters of the **Motion control** page, you can optimize the calculated tool path according to the kinematics and special characteristics of your CNC-machine.



The default values of these parameters are defined in the VMID file of your CNC-machine.

The interface of the **Motion control** page may vary depending on the parameters of your CNC-machine.

8.1 Approximation of Arcs

This option enables you to control the process of arcs approximation at the operation level.

The **Approximate arcs by lines** check box enables you to control the existence of arcs in the GCode output for the current operation.

When this check box is selected, InventorCAM approximates all tool path arcs by lines. The precision of the approximation depends on the `Max Chord Length` and `Max Arc Angle` parameters defined on the **Controller Definition** page of the VMID file; the resulting GCode does not contain arcs.

When this check box is not selected, the resulting GCode might contain arcs.

8.2 Point Interpolation

The point interpolation provides the ability to create intermediate points by setting a certain maximum angle step distance (for 5-axis motions) or by splitting long linear motions (3-axis and 5-axis tool paths) for feed rate moves and rapid rate moves.

Interpolation for distance

Using this option, InventorCAM enables you to perform interpolation for the linear tool movements. When this option is active, a new interpolated tool position is defined at each distance, defined by the **Interpolation for distance** parameter.

E.g. when the linear tool movement is performed from 0, 0, 0 to 0, 0, 100 and the **Interpolation for distance** option is used with the **Distance** value of 10, InventorCAM adds 9 tool positions between start and end positions (0, 0, 10, then 0, 0, 20 etc.).

Rapid feed rate moves

When this option is activated, all rapid motions (on clearance area and rapid distance) are converted into a feed motion with the given value.

8.3 5th Axis

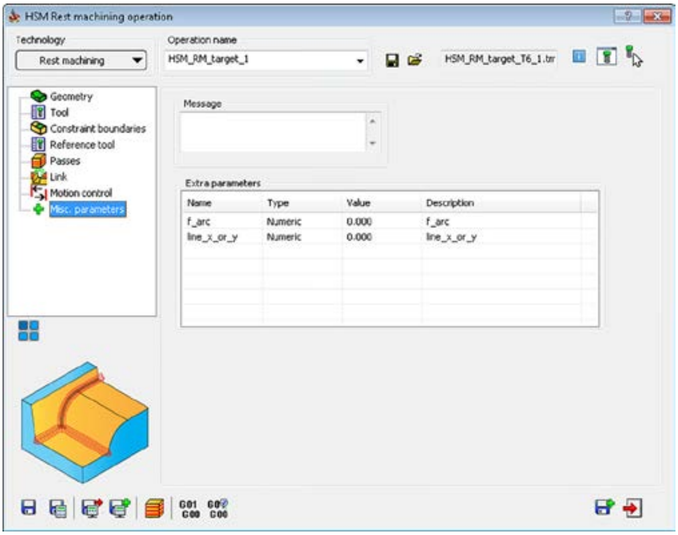
For a 5-axis machine, the tool axis vector can always be mapped into two different angle pairs. During the tool path generation, InventorCAM calculates for each tool axis orientation both of these two angle pairs; only one of the two has to be chosen for the GCode generation.

Some machines can only use one of the angle pairs due to mechanical limitations. In this case the angle pair will then be chosen as the **First angle pair** or **Second angle pair**.

Miscellaneous Parameters

9

This page displays the non-technological parameters related to the HSR/HSM operations.



9.1 Message

This field enables you to type a message that will appear in the generated GCode file.


```
G43G0 X-49.464 Y-38.768 Z12. S1000 M3
(Upper Face Milling)
(--------------------------)
(P-POCK-T2 - POCKET)
(--------------------------)
G0 X-49.464 Y-38.768
Z10.
```

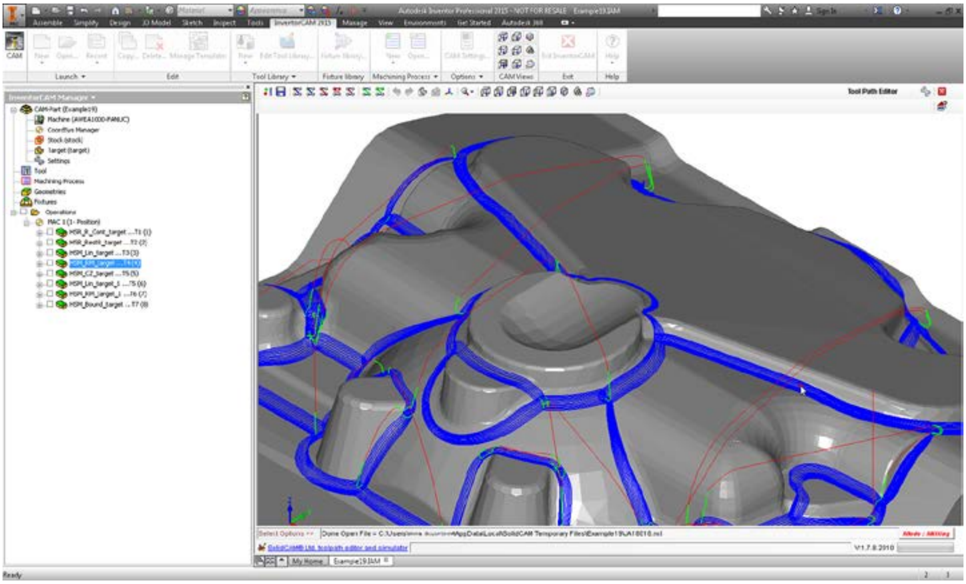
9.2 Extra Parameters

This table contains certain parameters only when special operation options have been implemented in the postprocessor you are using for this CAM-Part.

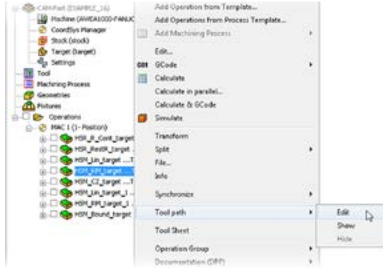
For more information on additional parameters definition, refer to the **InventorCAM Machine ID Reference Guide**.

Tool Path Editor 10

The InventorCAM **Tool Path Editor (TPE)** allows you to apply editing actions to a calculated tool path.



10.1 Opening the Tool Path Editor



To open the **Tool Path Editor**, in the **InventorCAM Manager**, right-click the operation which tool path you are going to modify and select **Tool Path > Edit** command from the menu.

The **Tool Path Editor** window is displayed.

10.2 TPE Toolbar

The toolbar enables you to control the editing process and the model visualization in the graphic area.

-  **Open new operation from tree:** this command enables you to edit the tool path of an operation chosen in the **InventorCAM Manager tree**. First, select the operation name and then click this icon. Your current operation tool path is closed, and the new tool path is opened for editing.

-  **Save output file:** this command enables you to save all changes made in the current tool path.

-  **Delete point:** this command enables you to delete a tool path point.

-  **Modify point:** this command enables you to delete a tool path point.

-  **Delete from (Pt) to (Pt):** this command enables you to delete the tool path between two selected points.

-  **Trim tool path:** this command enables you to trim the tool path either by a point or by a plane.

-  **Delete segment:** this command enables you to delete the tool path segments.

-  **Modify Range segment:** this command enables you to modify a tool path segment.

-  **Undo Last Operation:** this command enables you to cancel the last action.

-  **Redo Last Operation:** this command enables you to redo the last cancelled change.

-  **Show/Hide Target model:** this command enables you to show or hide the target model.

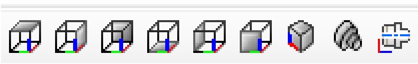
-  **Show/Hide Fixture File:** this command enables you to show or hide the fixture.

-  **Zoom**: this command enables you to choose the zoom options.

-  **Full Screen:** this command enables you to display the model on the screen in full size.

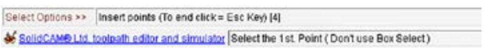
-  **Zoom In/Out:** this command enables you to zoom the model in and out with the left mouse button.

-  **Zoom by Box:** this command enables you to zoom a part of the model limited by the box.

-  **CAM Views**: these commands are identical to those on the main **InventorCAM** ribbon.

10.3 Editing Options

When you click a command icon on the toolbar, a hint message is displayed in the lower part of the window instructing you on your next step.



Follow the instruction to complete the tool path editing.

When you select a point or segment, the corresponding option menu is displayed in the lower part of the window.

Choose the relevant options and click **OK** to save changes and exit or **Apply** to save changes and continue editing with the same option.

Editing the tool path should be done carefully, otherwise you can break your tool or part. Always check the results before actual machining.

Examples

11

The archive supplied together with this book contains various CAM-Parts illustrating the use of the InventorCAM 3D HSR/HSM Module.

Examples #1–#3 illustrate the usage of specific HSR strategies.

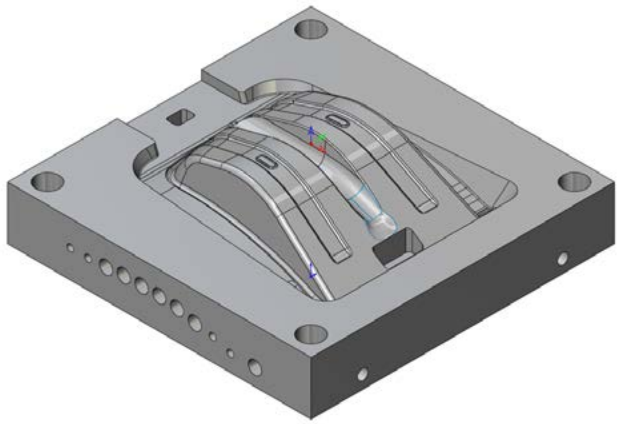
Examples #4–#13 illustrate the usage of specific HSM strategies.

Examples #14–#19 illustrate the usage of both HSR and HSM strategies to completely finish a part.

Extract the **Examples** archive to your hard drive. The Autodesk Inventor files used for exercises were prepared with **Autodesk Inventor 2015**.

Example #1: Rough machining and Rest roughing

This example illustrates the use of InventorCAM HSR strategies for the mold core machining.



- **HSR_R_Cont_target**

  This operation performs **Contour roughing** of the core model. The **Detect core areas** option is used to perform the approach into the material from outside.

- **HSR_RestR_target**

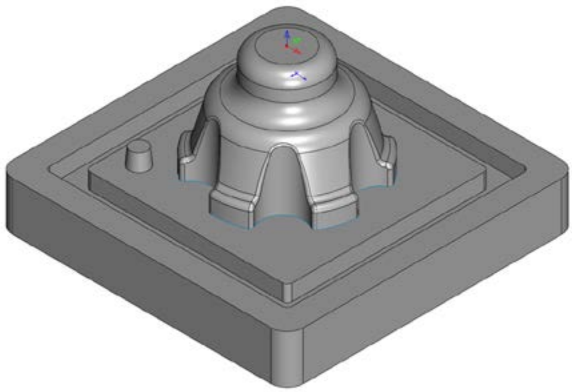
  This operation performs **Rest roughing** of the core model in the areas where material is left after the previous **Contour roughing** operation.

- **HSR_R_Lin_target**

  This operation performs **Hatch roughing** of the core model; this strategy can be used as an alternative to contour roughing for older machine tools or softer materials.

Example #2: Hybrid Rib roughing

This example illustrates the use of InventorCAM **Hybrid Rib roughing** for the impeller machining.



- **HSR_HMR_target**

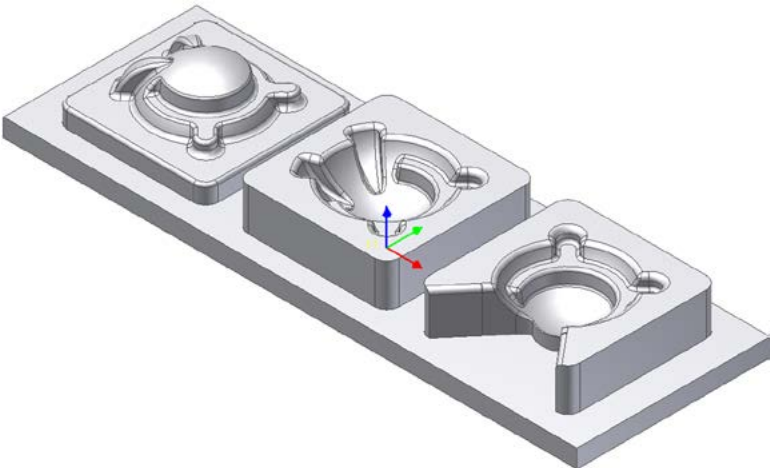
  This operation performs **Hybrid Rib roughing** of the core model with constant Step down. The number of parallel passes is limited by the **Number of offsets** value.

- **HSM_Morph_target**

  This operation performs **Morphed machining** of the model faces.

Example #3: HM Roughing

This example illustrates the use of InventorCAM **HM Roughing** strategy with different types of step over for machining of various models: core, cavity and hybrid (HM spiral).



- **HSR_HM_Roughing_Core**

  This operation performs **HM Roughing** of the core model with constant **Step down**. The **Step over type** is set as **Core** to machine the area from outside in.

- **HSR_HM_Roughing_Cavity**

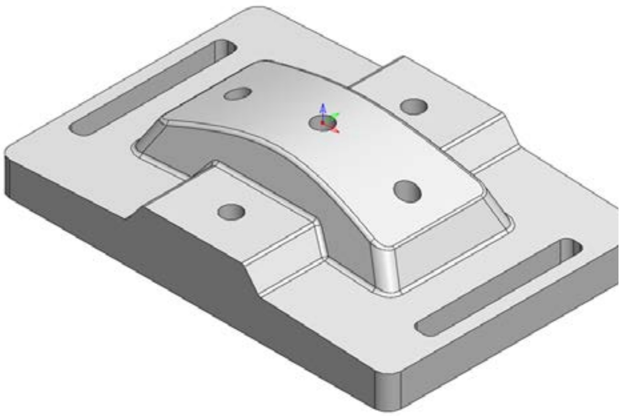
  This operation performs **HM Roughing** of the part containing a cavity with constant **Step down**. The **Step over type** is set as **Cavity** to machine the closed pocket area from inside out.

- **HSR_HM_Roughing_HM_Spiral**

  This operation performs **HM Roughing** of the hybrid area containing both cavity and open edges. The **Step over type** is set to **HM spiral** and **Step down type** is defined as **Constant & flats** to enable machining of flat areas.

Example #4: Constant Z, Helical and Horizontal machining

This example illustrates the use of InventorCAM **Constant Z**, **Helical** and **Horizontal** strategies for the machining of a mold core part.



- **HSM_CZ_target**

  This operation performs **Constant Z** machining of the part with constant Step down. The **Max. Stock thickness** parameter enables you to perform the separate machining of the forming faces and the boss faces.

- **HSM_CZ_target_1**

  This operation is a variation of the previous operation with the **Adaptive Step down** option set.

- **HSM_Helical_target**

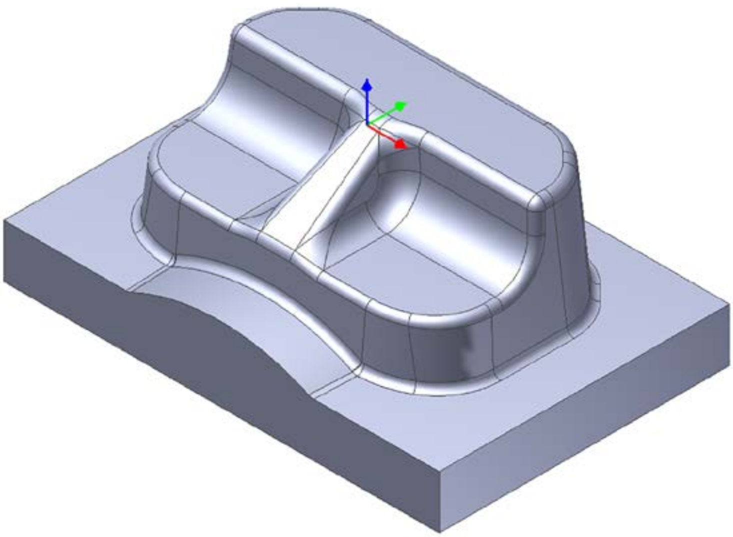
  This operation performs **Helical** machining of the core faces.

- **HSM_CZF_target**

  This operation performs **Horizontal** machining of the flat faces of the part.

Example #5: Hybrid Constant Z machining

This example illustrates the use of InventorCAM **Hybrid Constant Z** strategy for the machining of a part.

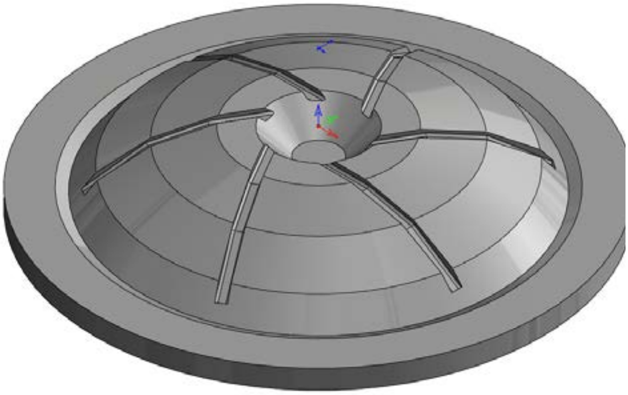


- **HSM_HMC_target**

  This operation performs machining of steep areas with constant step down and of shallow areas with 3D constant step over.

Example #6: Linear machining

This example illustrates the use of InventorCAM **Linear** strategy for the machining of a mold core part.

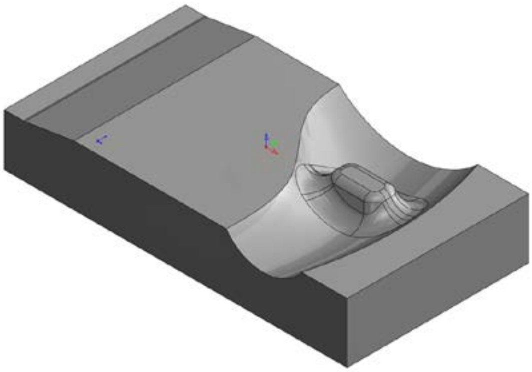


- **HSM_Lin_target**

  This operation performs **Linear** machining of the forming faces of the mold core. This operation illustrates the use of **Cross Linear finishing** in order to completely machine the model faces where the Linear passes are sparsely spaced.

Example #7: Radial and Spiral machining

This example illustrates the use of InventorCAM **Radial** and **Spiral** machining strategies for the machining of a bottle-bottom mold insert.



- **HSM_Rad_target**

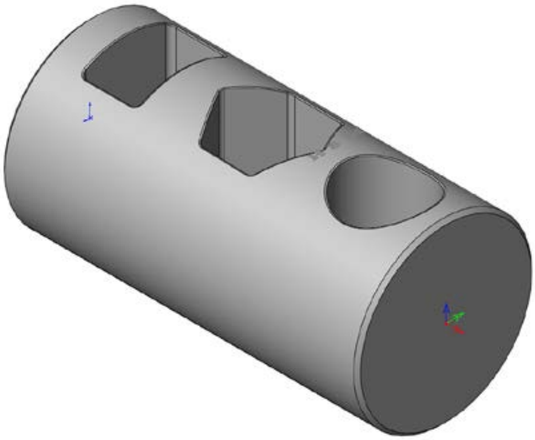
  This operation performs **Radial machining** of the forming faces of the insert. The user-defined boundary is used to limit the tool path.

- **HSM_Sp_target**

  This operation performs **Spiral machining** of the forming faces of the insert. The user-defined boundary is used to limit the tool path. The **Simple ordering** option is used to perform optimal ordering and linking of the tool path.

Example #8: Morphed machining and Offset cutting

This example illustrates the use of InventorCAM **Morphed machining** and **Offset cutting** strategies for the machining of a cavity part.



- **HSM_Morph_target**

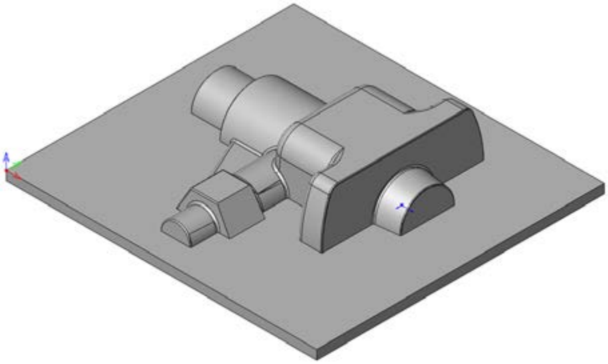
  This operation performs **Morphed machining** of the model faces.

- **HSM_OffsetCut_target**

  This operation illustrates the **Offset Cutting** strategy use for the parting surface machining.

Example #9: Boundary machining

This example illustrates the use of InventorCAM **Boundary machining** strategy for the machining of the cylindrical part shown below.

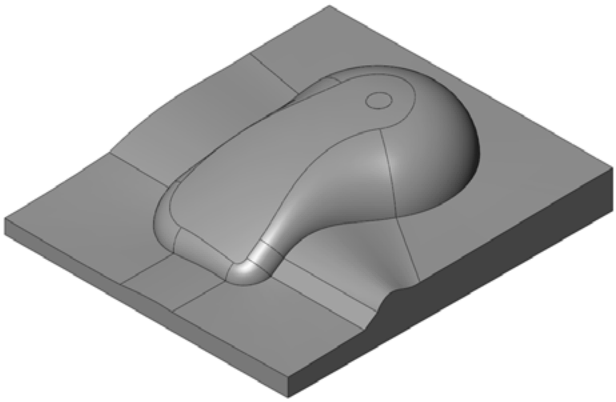


- **HSM_Bound_target**

  This operation illustrates the use of the **Boundary machining** strategy for the chamfering of model edges.

Example #10: Rest machining

This example illustrates the use of InventorCAM **Rest machining** strategy for the electrode part shown below.



- **HSM_RM_target**

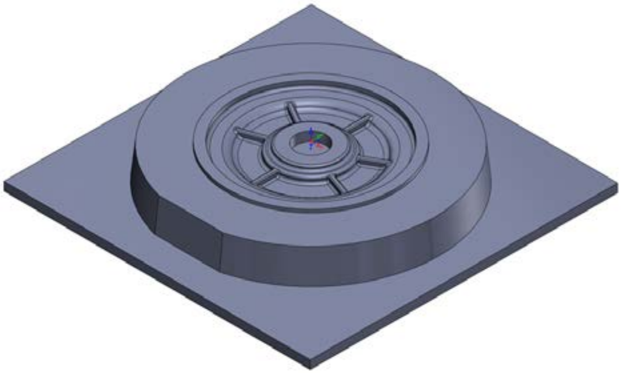
  This operation illustrates the use of the **Rest machining** strategy for the machining of model corners.

- **HSM_Bound_target**

  This operation illustrates the use of the **Boundary machining** strategy for optimal finishing of filleted corners.

Example #11: 3D Constant Step over machining

This example illustrates the use of InventorCAM **3D Constant Step over** machining strategy for the machining of the mold core shown below.

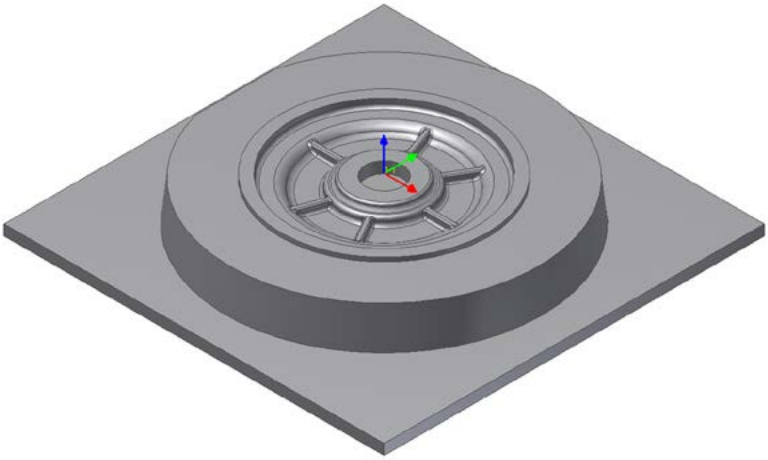


- **HSM_CS_target**

  This operation illustrates the use of the **3D Constant Step over** strategy for the machining of the parting face of the core.

Example #12: Pencil, Parallel Pencil and 3D Corner Offset

This example illustrates the use of InventorCAM **Pencil**, **Parallel Pencil** and **3D Corner Offset** strategies for the mold cavity shown below.



- **HSM_Pen_target**

  This operation illustrates the use of the **Pencil milling** strategy for the machining of cavity corners in a single pass.

- **HSM_PPen_target**

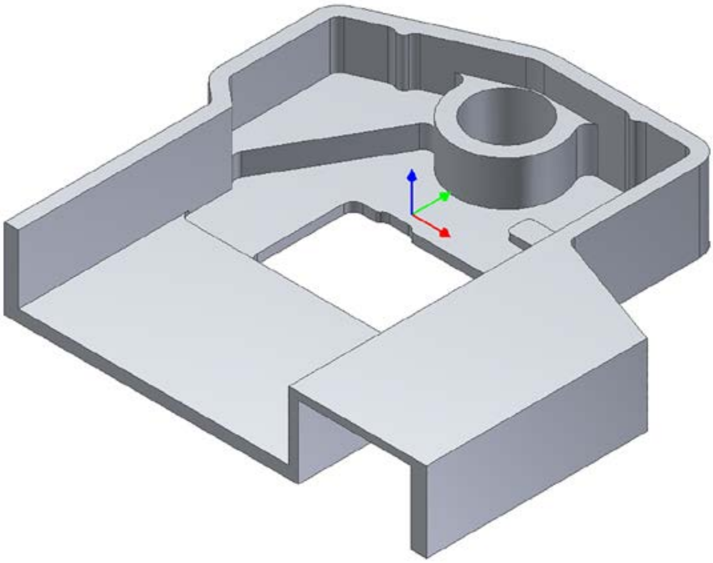
  This operation illustrates the use of the **Parallel Pencil milling** strategy for the machining of cavity corners in a number of passes.

- **HSM_Crn_Ofs_target**

  This operation illustrates the use of the **3D Corner Offset** strategy for the machining of the cavity part.

Example #13: Prismatic Part machining

This example illustrates the use of InventorCAM HSR and HSM strategies to complete machining of the prismatic part shown below.



- **HSR_R_Cont_target**

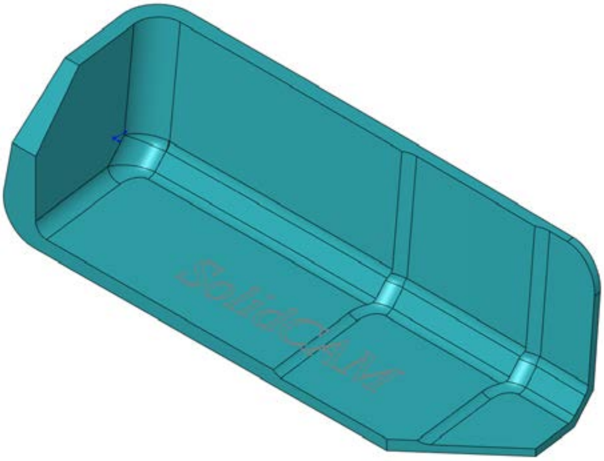
  This operation performs the **Contour roughing** of the part. An end mill of Ø32 is used with the step down of 3 mm. The machining allowance of 0.5 mm remains unmachined for further finish operation. The **Detect core areas** option is used to perform the approach into the material from outside.

- **HSM_Prismatic_target**

  This operation uses the **Prismatic Part machining** startegy for finishing of the part. performs rest roughing of the core. An end mill of Ø16 is used with the step down of 2 mm and step over of 2 mm to remove the material left after the roughing.

Example #14: Mold cavity machining

This example illustrates the use of several InventorCAM HSR and HSM strategies to complete machining of the mold cavity shown below.



- **HSR_R_Cont_target**

  This operation performs **Contour roughing** of the cavity. An end mill of Ø20 is used with the step down of 2 mm to perform fast and productive roughing. The machining allowance of 0.5 mm remains unmachined for further semi-finish and finish operations.

- **HSR_RestR_target**

  This operation performs **Rest roughing** of the cavity. A bull nosed tool of Ø12 and corner radius of 1 mm is used with a step down of 1 mm to remove the steps left after the roughing. The same machining allowance as in the roughing operation is used.

- **HSM_CS_target_1**

  This operation performs **3D Constant Step over semi-finishing** of the forming faces of the cavity. A ball nosed tool of Ø10 is used. The machining allowance of 0.2 mm remains unmachined for further finish operations.

- **HSR_RestR_target_1**

  This operation uses the **Rest roughing** strategy for the semi-finish machining of the model areas left unmachined after the previous operations. A ball nosed tool of Ø4 is used with the step down of 0.4 mm. The machining allowance of 0.2 mm remains unmachined for further finish operations.

- **HSM_RM_target**

  This operation uses the **Rest machining** strategy for finishing the model corners. A ball nosed tool of Ø6 is used for the operation. A reference tool of Ø10 is used to determine the model corners.

- **HSM_Crn_Ofs_target**

  The **3D Corner Offset** strategy is used for the finish machining of the cavity faces that are inside the constraint boundaries. The shape of pencil milling passes, generated by this strategy, is used for the constant step over machining of the cavity faces. A ball nosed tool of Ø6 is used for the operation.

- **HSM_Lin_target**

  The **Linear** strategy is used to complete the finish machining of the planar faces of the cavity that were not machined by the previous operation. A ball nosed tool of Ø6 is used for the operation.

- **HSM_HMC_target_1**

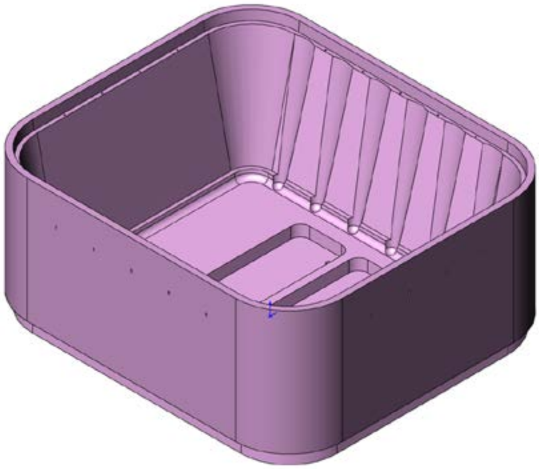
  The **Hybrid Constant Z** strategy is used for the finish machining of the model faces. A ball nosed tool of Ø4 is used for the operation.

- **HSM_PPen_target**

  The **Parallel Pencil milling** strategy is used for the finish machining of the cavity corners in a number of steps. A ball nosed tool of Ø3 is used for the operation.

Example #15: Aerospace part machining

This example illustrates the use of several InventorCAM HSR and HSM strategies to complete machining of the aerospace part shown below.



- **F_profile**

  This operation performs preliminary roughing using the **Profile** operation. An end mill of Ø12 is used.

- **HSR_R_Cont_target**

  This operation performs the **Contour roughing** of the part. An end mill of Ø12 is used with the step down of 2 mm to perform fast and productive roughing. The machining allowance of 0.5 mm remains unmachined for further semi-finish and finish operations.

- **HSM_CZ_target**

  This operation performs **Constant Z** finishing of the steep model faces. A bull nosed tool of Ø8 and corner radius of 0.5 mm is used for the operation.

- **HSM_CZF_target**

  This operation performs **Horizontal** machining of the flat faces. A bull nosed tool of Ø8 and corner radius of 0.5 mm is used for the operation.

- **HSM_CZ_target**

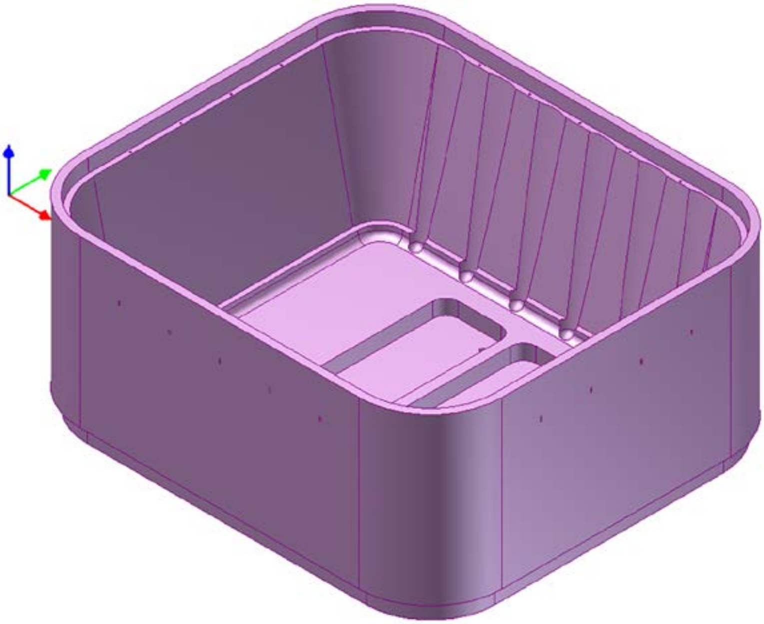
  This operation performs **Constant Z finishing** of the side fillet and chamfer faces using the **Adaptive Step down** option to perform the machining with the necessary scallop. A ball nosed tool of Ø4 is used for the operation.

- **HSM_Bound_target**

  This operation uses the **Boundary machining** strategy for the engraving on the model faces with a chamfer mill.

Example #16: Electronic box machining

This example illustrates the use of several InventorCAM HSR and HSM strategies to complete machining of the electronic box shown below.



- **HSM_R_Cont_target1**

  This operation performs the **Contour roughing** of the part. An end mill of Ø30 is used with the step down of 10 mm to perform fast and productive roughing. The machining allowance of 0.5 mm remains unmachined for further semi-finish and finish operations.

- **HSR_RestR_target1**

  This operation performs the **Rest roughing** of the part. A bull nosed tool of Ø16 and corner radius of 1 mm is used with the step down of 5 mm to remove the steps left after the roughing. The same machining allowance as in the roughing operation is used.

- **HSM_CZ_target1**

  This operation performs **Constant Z finishing** of the upper vertical model faces up to a certain depth. A bull nosed tool of Ø12 and corner radius of 0.5 mm is used.

- **HSM_CZ_target1_1**

  This operation performs **Constant Z finishing** of the bottom vertical model faces. A bull nosed tool of Ø12 and corner radius of 0.5 mm is used.

- **HSM_CZF_target1**

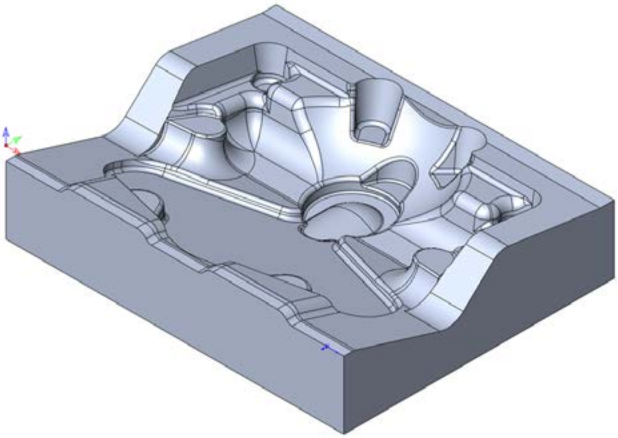
  This operation performs **Horizontal machining** of the flat faces. A bull nosed tool of Ø12 and corner radius of 0.5 mm is used.

- **HSM_CZ_target1_2**

  This operation performs **Constant Z machining** of the inclined faces. A taper mill of 12° angle is used to perform the machining of the inclined face with large step down (10 mm). Using such a tool enables you to increase the productivity of the operation.

Example #17: Mold insert machining

This example illustrates the use of several InventorCAM HSR and HSM strategies to complete machining of the mold insert.



- **HSR_R_Cont_target**

  This operation performs the **Contour roughing** of the part. An end mill of Ø25 is used with the step down of 3 mm. A machining allowance of 0.5 mm remains unmachined for further semi-finish and finish operations. The **Detect core areas** option is used to perform the approach into the material from outside.

- **HSR_RestR_target**

  This operation performs the **Rest roughing** of the part. A bull nosed tool of Ø12 and corner radius of 2 mm is used with the step down of 1.5 mm to remove the steps left after the roughing. The same machining allowance as in the roughing operation is used.

- **HSM_CZ_target**

  This operation performs the **Constant Z semi-finishing** of the steep faces (from 40° to 90°). A ball nosed tool of Ø8 is used for the operation. The machining allowance of 0.2 mm remains unmachined for further finish operations.

- **HSM_Lin_target**

  This operation performs the **Linear semi-finishing** of the shallow faces (from 0° to 42°). A ball nosed tool of Ø8 is used for the operation. The machining allowance of 0.2 mm remains unmachined for further finish operations.

- **HSM_RM_target**

  This operation uses the **Rest machining** strategy for semi-finishing of the model corners. The semi-finishing of the model corners enables you to avoid tool overload in the corner areas during further finishing. A ball nosed tool of Ø6 is used for the operation. A reference tool of Ø8 is used to determine the model corners. The machining allowance of 0.2 mm remains unmachined for further finish operations.

- **HSM_CZ_target_1**

  This operation performs **Constant Z finishing** of the steep faces (from 40° to 90°). A ball nosed tool of Ø6 is used for the operation. The **Apply fillet surfaces** option is used to generate a smooth tool path and to avoid a sharp direction changes in the model corners.

- **HSM_OffsetCut_target**

  This operation performs **Offset cutting** of the upper face edge.

- **HSM_CZF_target**

  This operation performs **Horizontal machining** of the flat faces. An end mill of Ø16 is used.

- **HSM_CS_target**

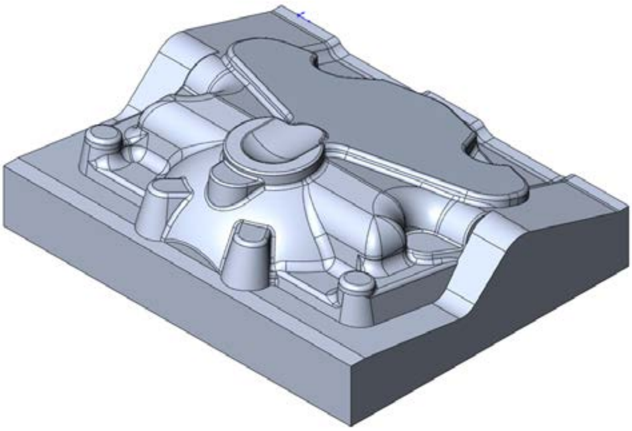
  This operation performs **3D Constant Step over machining** of the insert bottom faces; since these faces are horizontal, the machining is limited to an angle range from 0° to 2°. A ball nosed tool of Ø4 is used for the operation.

- **HSM_RM_target**

  This operation uses the **Rest machining** strategy for finishing of the model corners. A ball nosed tool of Ø4 is used for the operation. A reference tool of Ø7.5 is used to determine the model corners.

Example #18: Mold cavity machining

This example illustrates the use of several InventorCAM HSR and HSM strategies to complete machining of the mold cavity shown below.



- **HSR_R_Cont_target**

  This operation performs **Contour roughing** of the cavity. An end mill of Ø20 is used with the step down of 3 mm. A machining allowance of 0.5 mm remains unmachined for further semi-finish and finish operations.

- **HSR_RestR_target**

  This operation performs **Rest roughing** of the cavity. A bull nosed tool of Ø12 and corner radius of 2 mm is used with the step down of 1.5 mm to remove the steps left after the roughing. The same machining allowance as in roughing operation is used.

- **HSM_CZ_target**

  This operation performs **Constant Z semi-finishing** of the steep faces (from 40° to 90°). A ball nosed tool of Ø10 is used for the operation. The machining allowance of 0.25 mm remains unmachined for further finish operations. The **Apply fillet surfaces** option is used.

- **HSM_Lin_target**

  This operation performs **Linear semi-finishing** of the shallow faces (from 0° to 42°). A ball nosed tool of Ø10 is used for the operation. The machining allowance of 0.25 mm remains unmachined for further finish operations. The **Apply fillet surfaces** option is used.

- **HSM_RM_target**

  This operation uses the **Rest machining** strategy for semi-finishing of the model corners. The semi-finishing of the model corners enables you to avoid tool overload in the corner areas during further finishing. A ball nosed tool of Ø6 is used for the operation. A reference tool of Ø12 is used to determine the model corners. A machining allowance of 0.25 mm remains unmachined for further finish operations.

- **HSM_CZ_target_1**

  This operation performs **Constant Z finishing** of the steep faces (from 40° to 90°). A ball nosed tool of Ø8 is used for the operation. The **Apply fillet surfaces** option is used.

- **HSM_Lin_target_1**

  This operation performs **Linear finishing** of the shallow faces (from 0° to 42°). A ball nosed tool of Ø8 is used for the operation. The **Apply fillet surfaces** option is used.

- **HSM_RM_target_1**

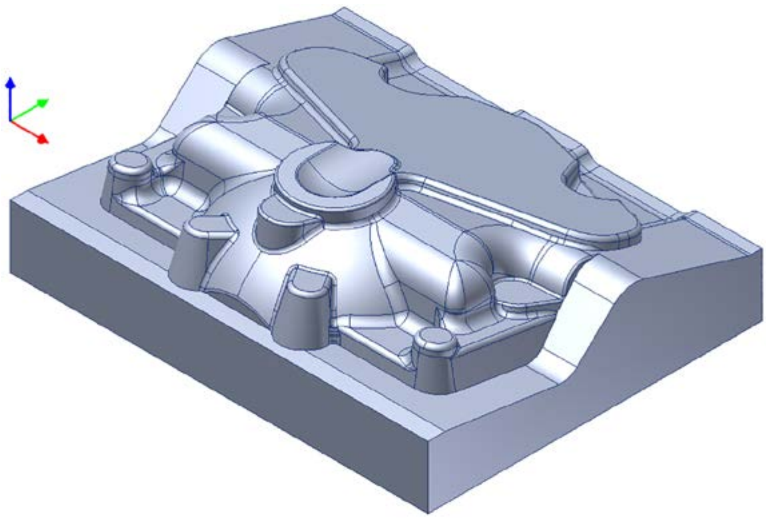
  This operation uses the **Rest machining** strategy for finishing of the model corners. A ball nosed tool of Ø4 is used for the operation. A reference tool of Ø10 is used to determine the model corners.

- **HSM_Bound_target**

  This operation uses **Boundary machining** strategy for the chamfering of upper model edges. A chamfer drill tool is used for the operation. The chamfer is defined by the external offset of the drive boundary and by the **Floor offset** parameter.

Example #19: Mold core machining

This example illustrates the use of several InventorCAM HSR and HSM strategies to complete the machining of the mold core shown below.



- **HSR_R_Cont_target**

  This operation performs **Contour roughing** of the core. An end mill of Ø20 is used with the step down of 4 mm to perform fast and productive roughing. The machining allowance of 0.5 mm remains unmachined for further semi-finish and finish operations. The **Detect core areas** option is used to perform the approach into the material from outside.

- **HSR_RestR_target**

  This operation performs rest roughing of the core. A bull nosed tool of Ø12 and corner radius of 2 mm is used with the step down of 2 mm to remove the steps left after the roughing. The same machining allowance as in roughing operation is used. The **Detect core areas** option is used to perform the approach into the material from outside.

- **HSM_Lin_target**

  This operation performs Linear semi-finishing of the core faces. A ball nosed tool of Ø10 is used for the operation. The machining allowance of 0.2 mm remains unmachined for further finish operations. The **Apply fillet surfaces** option is used.

- **HSM_RM_target**

  This operation uses the **Rest machining** strategy for semi-finishing of the model corners. The semi-finishing of the model corners enables you to avoid tool overload in the corner areas during further finishing. A ball nosed tool of Ø6 is used for the operation. A reference tool of Ø12 is used to determine the model corners. The machining allowance of 0.2 mm remains unmachined for further finish operations.

- **HSM_CZ_target**

  This operation performs **Constant Z finishing** of the steep faces (from 30° to 90°). A ball nosed tool of Ø8 is used for the operation. The **Apply fillet surfaces** option is used.

- **HSM_Lin_target_1**

  This operation performs **Linear finishing** of the shallow faces (from 0° to 32°). A ball nosed tool of Ø8 is used for the operation. The **Apply fillet surfaces** option is used.

- **HSM_RM_target_1**

  This operation uses the **Rest machining** strategy for finishing of the model corners. A ball nosed tool of Ø4 is used for the operation. A reference tool of Ø10 is used to determine the model corners.

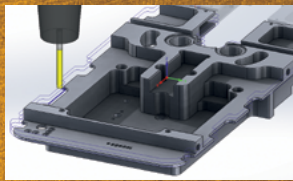
- **HSM_Bound_target**

  This operation uses **Boundary machining** strategy for the chamfering of upper model edges. A chamfer drill tool is used for the operation. The chamfer is defined by the external offset of the drive boundary and by the **Wall offset** parameter.

InventorCAM 2015

InventorCAM User Guide:
HSR/HSM

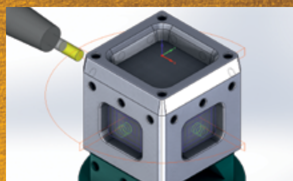
iMachining 2D



2.5D Milling

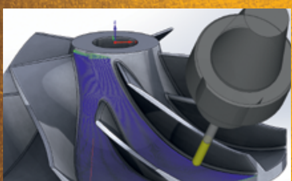
HSS (High-Speed Surface Machining)

iMachining 3D

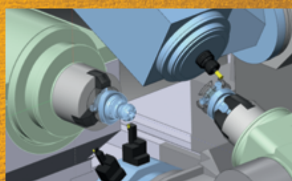


Indexed Multi-Sided Machining

HSM (High-Speed Machining)



Simultaneous 5-Axis Machining



Turning & Advanced Mill-Turn

Solid Probe

YouTube Broadcast Yourself™

www.youtube.com/SolidCAMProfessor
www.youtube.com/SolidCAMiMachining

www.facebook.com/SolidCAM

www.inventorcam.com

InventorCAM
iMachining – The Revolution in CAM!