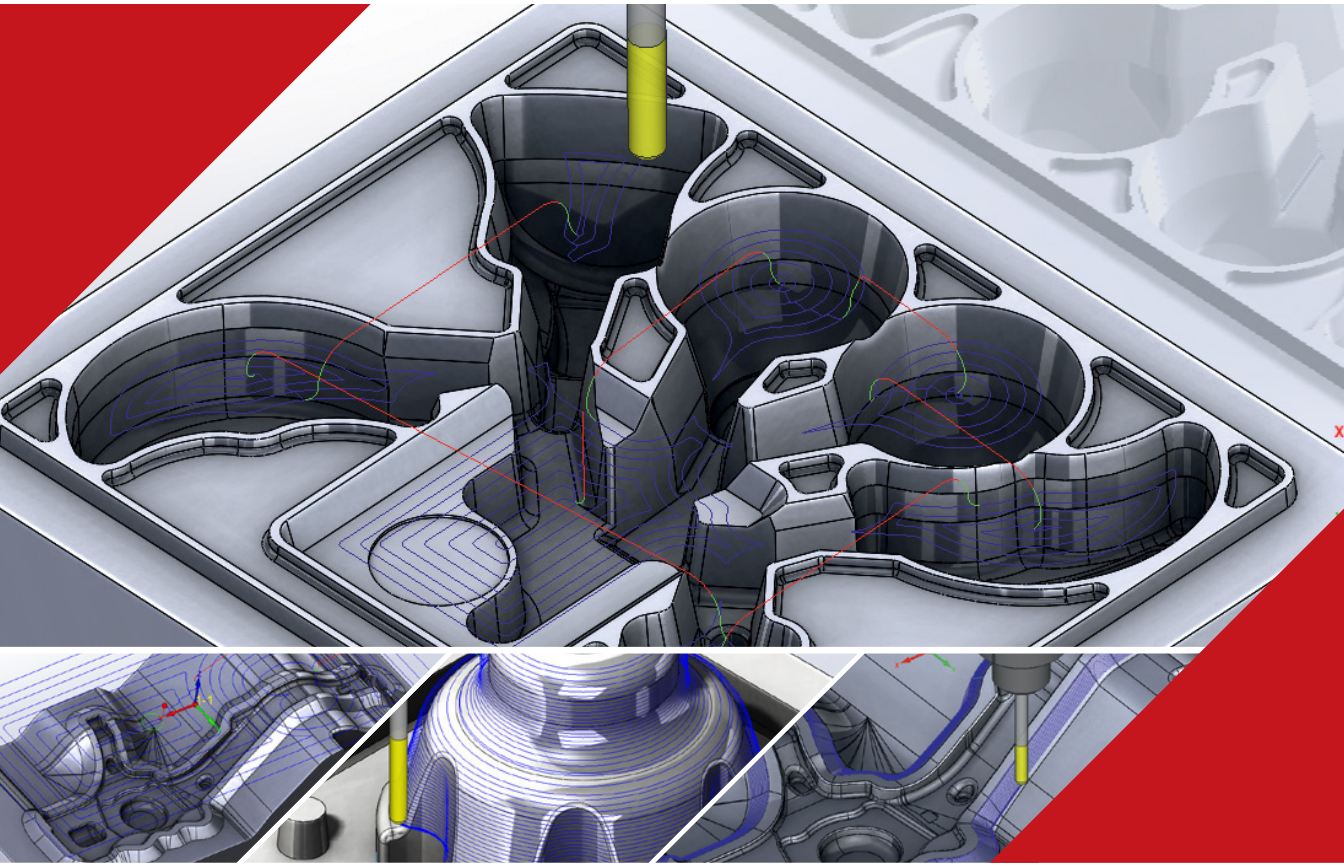
SolidCAM 2018

The unique, revolutionary Milling technology

iMachining®

patent by SolidCAM

TIME SAVINGS 70% ... AND MORE!



SolidCAM User Guide

Turbo 3D HSM

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

CERTIFIED Gold Product SOLIDWORKS

SolidCAM + SOLIDWORKS

The Complete Integrated Manufacturing Solution

SolidCAM

iMachining – The Revolution in CAM!

SolidCAM
iMachining – The Revolution in CAM!

SolidCAM 2018
Turbo 3D HSM Module
User Guide

Contents

Introduction and Basic Concepts

1

Welcome to SolidCAM 3D HSM!

SolidCAM's Turbo 3D HSM is a very powerful HSM module for quick and faster calculations than the normal HSM module. It offers unique machining and linking strategies for generating high-speed tool paths. This 3-Axis calculation engine recalculates the tool path at a lightning speed. Its 64-bit architecture completely utilizes all the cores for tool path calculations.

SolidCAM's Turbo 3D HSM module is simpler with limited parameters and advanced gouge checking strategies.

About this book

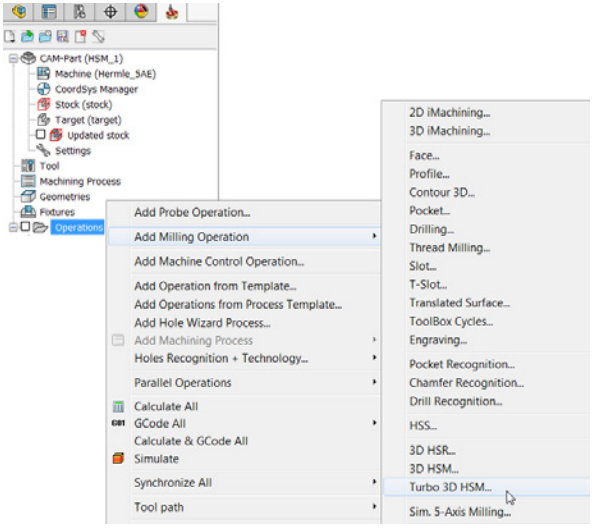
This book is intended for experienced SolidCAM users. If you are not familiar with the software, contact your reseller for information about SolidCAM training classes.

Windows 7

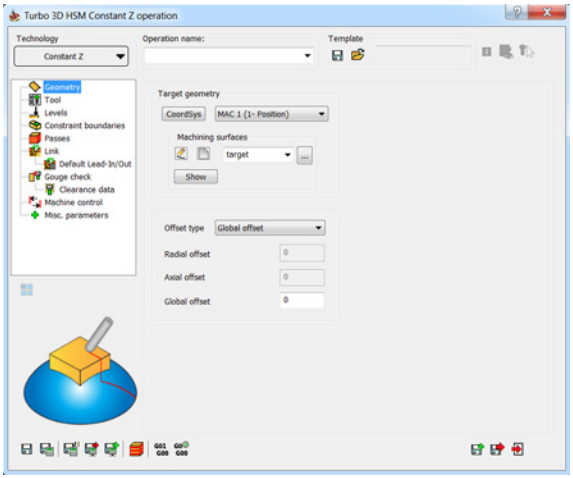
The screenshots in this book were made using **SolidCAM 2018** integrated with SOLIDWORKS 2018 running on Windows 7. If you are running on a different version of Windows, you may notice differences in the appearance of menus and windows. These differences do not affect the performance of the software.

1.1 Start Turbo 3D HSM Operation

To add a **Turbo 3D HSM Operation** to the CAM-Part, right-click the **Operations** header in **SolidCAM Manager** and choose **Turbo 3D HSM** command from the **Add Milling Operation** submenu.



The Turbo 3D HSM Constant Z operation dialog box is displayed.



Technology

This section enables you to define the type of the operation. SolidCAM provides you with six types of 3D HSM operation: **Constant Z**, **Linear**, **Constant Step Over**, **Pencil**, **Constant Z Rest Finish**, and **Constant Step Over Rest Finish**.

1.2 Constant Z Machining

The Constant Z tool path is generated for a set of sections created at different Z-heights. 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.

1.3 Linear Machining

With this strategy, SolidCAM 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.

1.4 Constant Step Over Machining

This strategy creates an equidistant cut pattern on the machining surfaces. A constant distance between each contour is maintained so that the created cusps have the same height. This strategy is generally used to semi-finish or finish a component. It is best suited to machine steep as well as shallow areas.

1.5 Pencil Machining

This strategy is beneficial in providing fast corners and fillets processing. It is performed via single- or multi-pencil cuts.

1.6 Constant Z Rest Finish

This strategy is used for performing the rest finishing of areas using the constant Z method. This strategy determines the model areas where material remains after the machining by a tool path, and generates a set of passes to machine these areas.

1.7 Constant Step Over Rest Finish

This strategy is used to generate a tool path to remove all the non-machined areas left by the previous bigger finishing tool. This strategy enables you to use smaller step over and smaller cutter size to machine only those areas that are left out by the previous tool.

Parameter pages

The parameters of the Turbo 3D HSM operation are divided into a number of subgroups. The subgroups are displayed in a tree format on the left side of the **Turbo 3D HSM machining operation** dialog box. When you click a subgroup name in the tree, the parameters of the selected subgroup appear on the right side of the dialog box.

- **Geometry**

  Choose geometry for machining and define the related parameters.

- **Tool**

  Choose a tool for the operation and define the related parameters such as feed and spin.

- **Levels**

  Define the Clearance area and the machining levels.

- **Constraint boundaries**

  Define the boundaries.

- **Reference tool**

  Define the reference tool used for the Rest Machining tool path calculation. This page is available only when the **Technology** is selected as **Constant Z Rest Finish** or **Constant Step Over Rest Finish**.

- **Passes**

  Define the passes parameters.

- **Link**

  The **Link** and **Default Lead-In/Out** pages enable you to define how the Turbo 3D HSM cutting passes are linked to the complete tool path.

- **Gouge check**

  The **Gouge check** and **Clearance data** pages enable you to avoid the tool gouging of any selected drive and check surfaces.

- **Machine control**

  Define the parameters related to the kinematics and special characteristics of your CNC-machine.

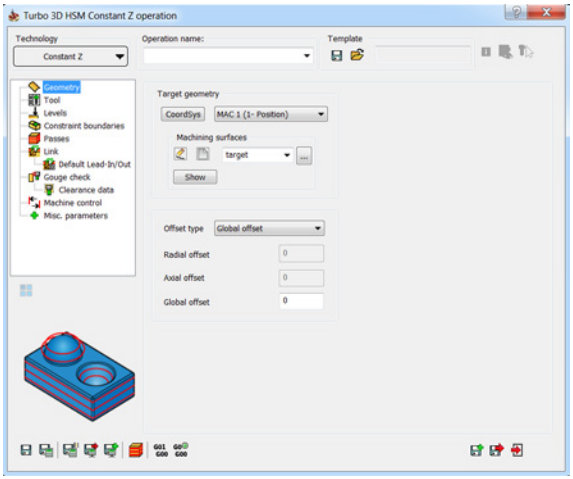
- **Misc. parameters**

  Define a number of miscellaneous parameters and options related to the tool path calculation.

Geometry

2

The **Geometry** page enables you to define the 3D model geometry for the SolidCAM 3DHSM operation.



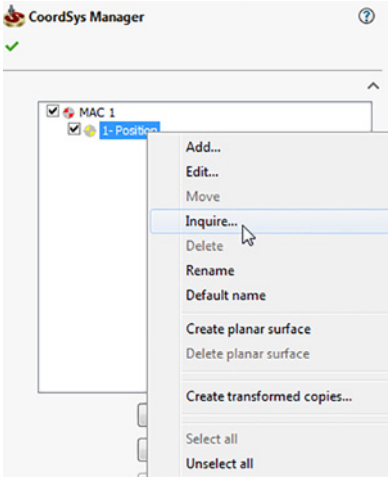
2.1 Geometry Definition

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

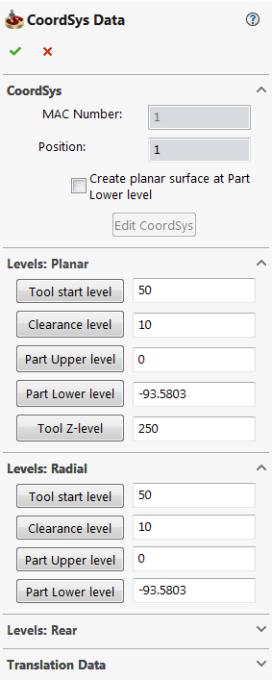
CoordSys

SolidCAM 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, SolidCAM 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.



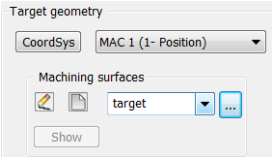
When the CoordSys is chosen for the operation, the model is rotated to the appropriate orientation.

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

Machining surfaces

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

The Machining surfaces are the entire model or any surface(s) of the design model.

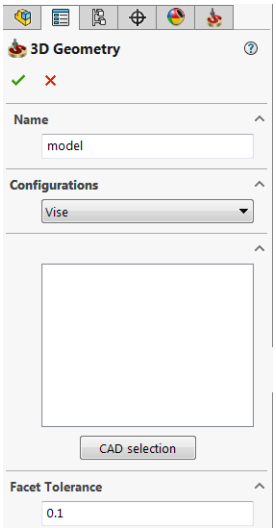


If you have already defined 3D Model geometries for this CAM-Part, you can select a geometry from the list.

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

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

The 3D geometry dialog box has the following components:



Name: This option enables you to define the name of the geometry. SolidCAM offers you the Default Geometry name that can be edited.

Configurations: This option enables you to switch between SOLIDWORKS configurations. Choose the relevant configuration for the geometry definition.

CAD selection: This option enables you to select the 3D geometry with the SOLIDWORKS tools.

When an object is selected in the CAD selection mode, the CAD selection button changes to Resume.

Facet Tolerance: This tolerance controls the maximum deviation of the mathematical representation from the original solids and surfaces of your model. 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 SolidCAM 3D HSM operation. If you use the 3D geometry in another operation, SolidCAM 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.

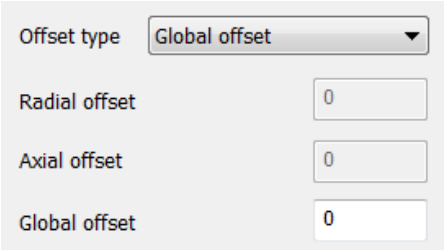
The button enables you to edit an existing geometry.

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

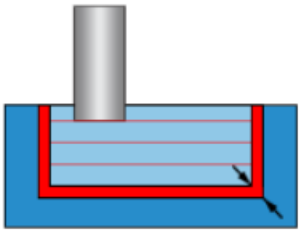
When you choose the geometry from the list, the related Coordinate System is chosen automatically.

2.2 Offset type

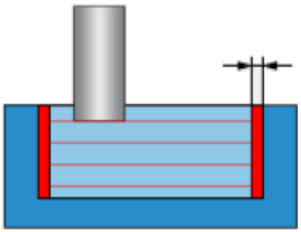
This section provides two options. You can set the offset as **Global** or **Radial and Axial**.



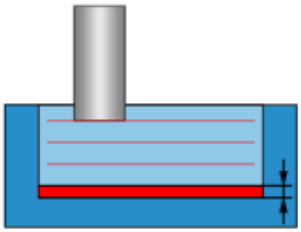
Global offset: This is a 3-dimensional offset which expands the machining faces in all directions.



Axial and Radial offset: The radial offset is a 2-dimensional offset which expands the machining faces in the radial direction.



The axial offset is a 1-dimensional offset which expands the machining faces in the axial direction.



When you choose **Global offset** from the **Offset type** list, the fields of **Axial and Radial offset** are disabled and when you select **Axial and Radial offset** from the list, the field of **Global offset** is disabled.

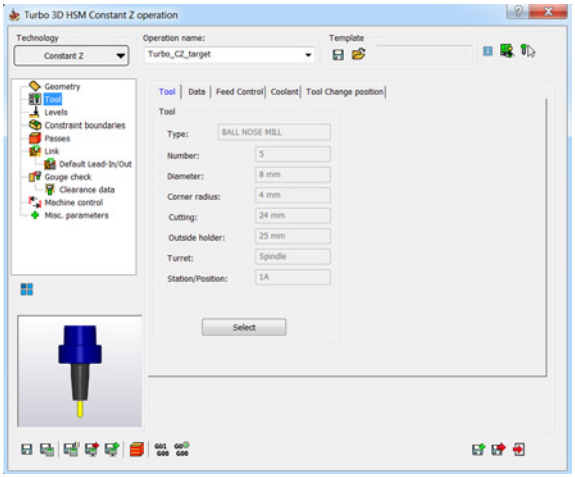
2.3 Drive curves

When the **Technology** is selected as **Constant Step Over**, this section enables you to morph the constant step over pattern between multiple drive curves. Selecting allows you to select new drive curves. Using you can edit the selected curve. The **Show** button displays the selected curve.

Tool

3

In the **Tool** page of the SolidCAM 3D HSM operation dialog box, the following tool parameters are displayed:



- **Type**
- **Number**
- **Diameter**
- **Corner radius**
- **Cutting length**
- **Outside holder**
- **Turret**
- **Station/Position**

In 3D HSM operations the following tools are available for selection:

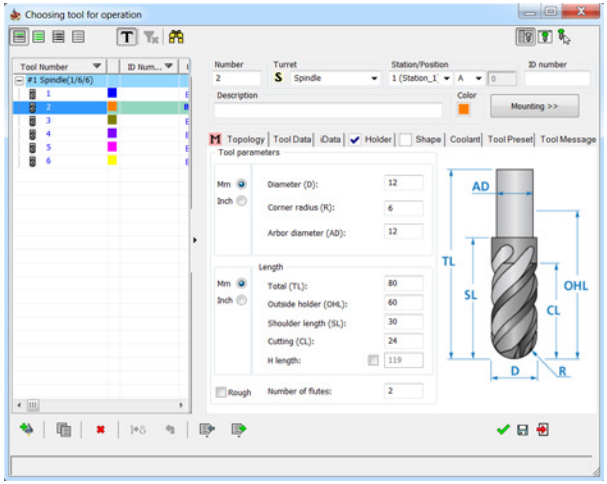
- **End Mill**
- **Bull Nose Mill**
- **Ball Nose Mill**
- **Face Mill**
- **Taper Mill**
- **Taper Ball Nose**
- **Lollipop Mill**

3.1 Tool Selection

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

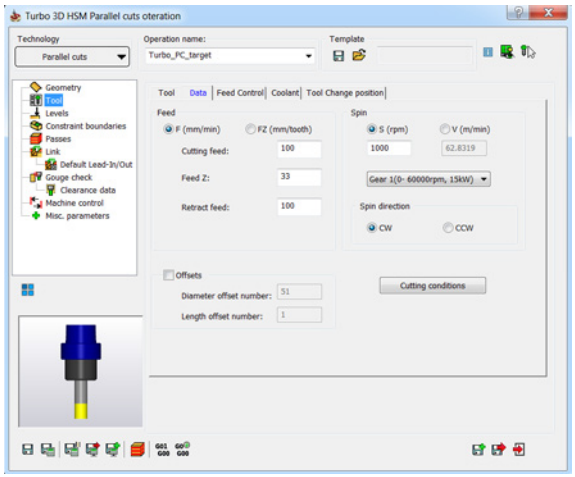
This button displays the **Edit** page of the **Part Tool Table**. You can also add a new tool to be defined for the operation or choose another tool from the **Part Tool Table**.

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



3.2 Feed Rate & Spin Definition

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



Feed

The Feed section enables you to define the tool feed for Turbo 3D HSM operation. 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 Feed: This field defines the cutting feed rate of the tool.
- Feed Z: This field defines the feed of the tool movements from the safety position to the cutting depth.
- Retract Feed: This field defines the feed of the tool movements from the material to the retract level. The default value of the Retract Feed is calculated according to the following formula:

  *Retract Feed=Cutting Feed * 2*

Spin

The Spin section enables you to define the spinning speed of the tool.

The spin value can be defined in two types of units: **S** and **V**. **S** is the default that signifies Revolutions per Minute. **V** signifies material cutting speed in Meters per Minute in the Metric system or in Feet per Minute in the Inch system; it is calculated according to the following formula:

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

The **Gear** list enables you to select a Gear producing the spinning speed. The first parameter in the parentheses is a spin range; the second parameters is the power. The gear is selected automatically according to the defined spin. Only gears having the current spin value within their range are shown in the list.

The **Spin direction** section enables you to choose between the clockwise (**CW**) or counterclockwise (**CCW**) direction.

Offsets

Diameter offset number

This parameter defines the number of the **Diameter Offset Register** of the current tool in the **Offset** table of the CNC-machine.

Length offset number

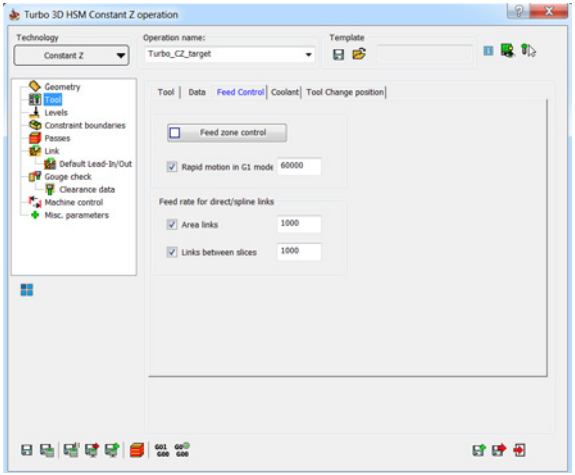
This parameter defines the number of the **Length Offset Register** of the current tool in the **Offset** table of the CNC-machine.

Cutting conditions

This button enables you to update the cutting conditions defined for use of the current tool on the chosen CNC-machine according to the parameters set in the Part Tool Table.

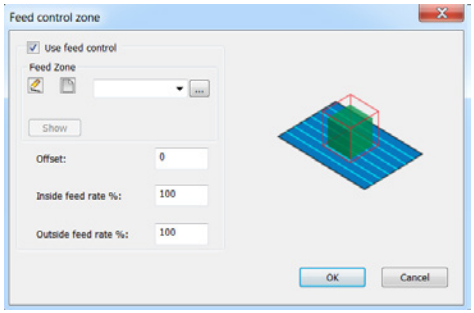
3.3 Feed Control

The **Feed Control** tab allows you to define the feed control parameters.



3.3.1 Feed zone control

The **Feed zone control** parameter allows you to define a volume to determine an area in which a higher or lower percentage of the cutting feed rate can be set. Select the check box. The **Feed control zone** dialog box is displayed.



Feed Zone

You can choose the appropriate geometry from the list or define a new one by clicking . The **Select Faces** dialog box is displayed. This dialog box enables you to define the geometry for feed zone.

Offset

This field enables you to put an additional offset value on the feed control geometry in order to let the tool slow down before it enters the actual feed control geometry.

Inside/Outside feed rate %

The inside and outside feed rate percentage fields define the feed rate inside and outside of the feed control geometry as a percentage of the actual cutting feed rate.

3.3.2 Rapid motion in G1 mode

When the **Rapid motion in G1 mode** check box is selected, the resulting GCode does not contain **G0** commands. The rapid movements are performed using the feed rate defined by the **Rapid feed rate** parameter.

Example: `G1 X-2.942 Y75.567 Z24.402 A-88.436 B-26.482 F9998 M116`

3.3.3 Feed rate for direct/spline links

This section enables you to define determined feed rates for direct links and spline links that correspond with the machining feed rate.

When you select the **Area links** check box, you can set the value of connection moves between the offset cuts in a single group or, connection moves between different groups inside a single region on the same cutting layer.

When you select the **Links between slices** check box, you can set the value of connection moves between the cutting layers on multiple heights.

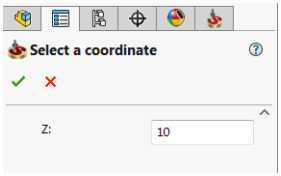
Levels

4

In the **Levels** page of the SolidCAM 3D HSM operation dialog box, you can define the Clearance area. It is the area where the tool movements can be performed safely without contacting the material. The tool movements in the Clearance area are performed with the rapid feed.

In 3D HSM operations, only **Plane height** can be defined. The **Plane height** parameter defines the distance between the appropriate Coordinate System plane and the Clearance area plane.

Selecting the **Plane height** button displays the **Select a coordinate** dialog box.



You can define the value by picking points on the model. The Z coordinate of this point is displayed in the Select a coordinate dialog box.

Click ✔ to confirm the set value.

Constraint Boundaries

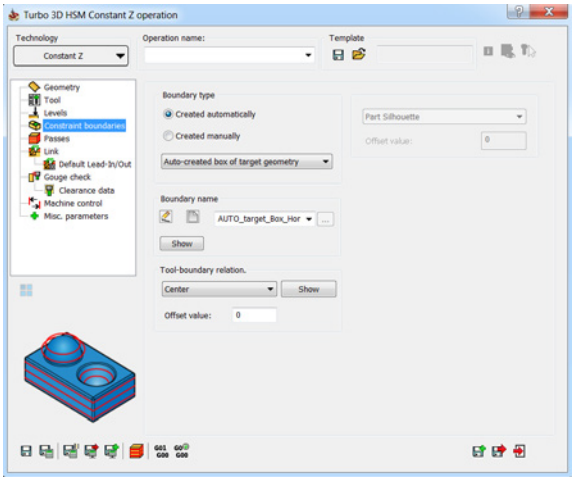
5

5.1 Introduction

SolidCAM enables you to define the constraint boundaries for the SolidCAM 3D HSM operation tool path.

5.1.1 Constraint boundaries

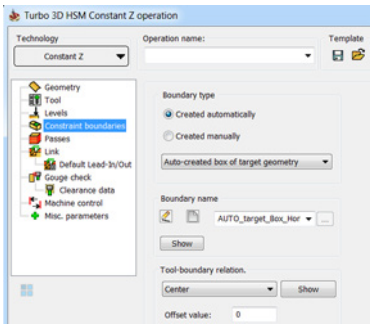
A constraint boundary enables you to limit the machining to specific model areas.



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.

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 SolidCAM:

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

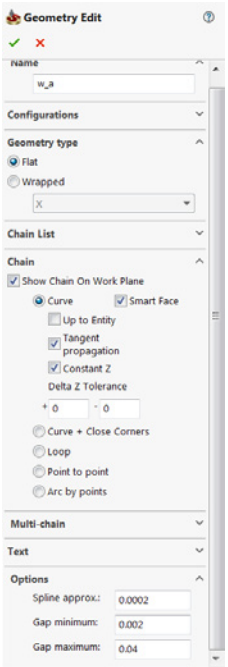
Created manually

The option of **User-defined boundary** enables you to define a user-defined boundary based on a Working area geometry.

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 **Geometry Edit** dialog box enabling you to choose the necessary chains for the boundary. The chosen boundaries are displayed and highlighted in the graphic window. The following sections are available in the **Geometry Edit** dialog box:



Geometry Name

This field enables you to define the name of the geometry. SolidCAM offers you a default geometry name that can be edited.

Configurations

This option enables you to switch between SOLIDWORKS configurations. Choose the relevant configuration for the geometry definition.

Geometry type

Flat: This option enables you to select a geometry on a flat surface.

Wrapped: This option enables you to wrap the selected profile geometry around any axis of the Coordinate System. The axes list offers you the choice of axis around which the geometry is wrapped.

Adding a Chain

You can add a chain to the existing set of chains in the current geometry. The new chain is added under the next sequential number.

Select the chain geometry using the **Chain** options.

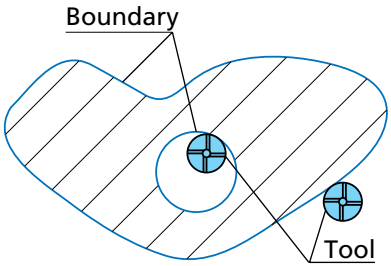
For more information on chain options and other Geometry Edit dialog box sections, refer to the SolidCAM Milling Online Help.

5.2.3 Tool – boundary relation

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

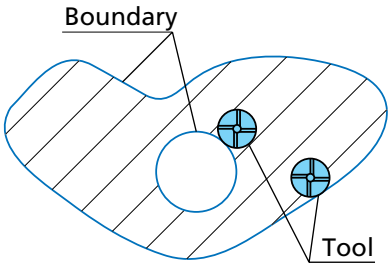
External

The tool machines outside the boundary.



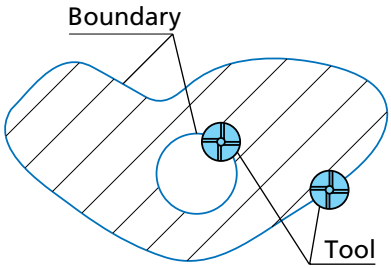
Internal

The tool machines inside the boundary.



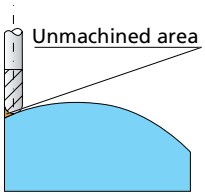
Center

The tool center is positioned on the boundary.

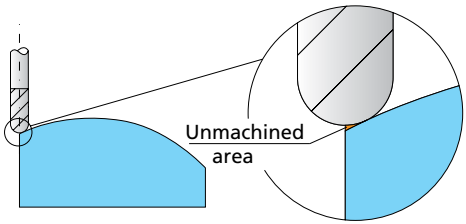


Tangent

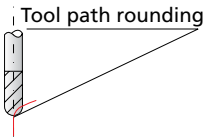
The **Internal/External/Centered** methods of the boundary definition have several limitations. In some cases, the limitation of the tool path by planar boundary results in unmachined areas or corners rounding.



Boundary – Tool Relation: Internal



Boundary – Tool Relation: Centered

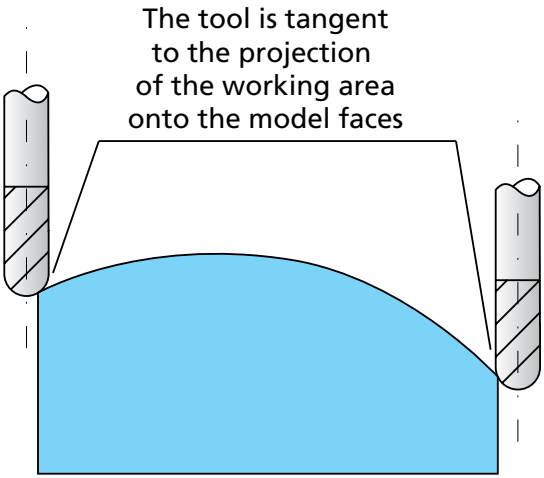


Boundary – Tool Relation: External

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

When this option is chosen, SolidCAM 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.

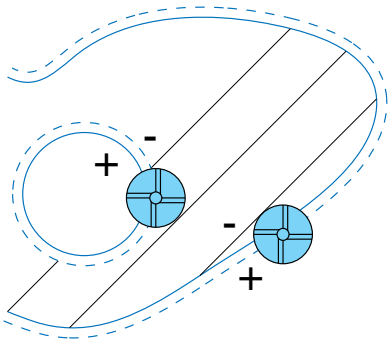


Boundary – Tool Relation: Tangent

Offset value

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

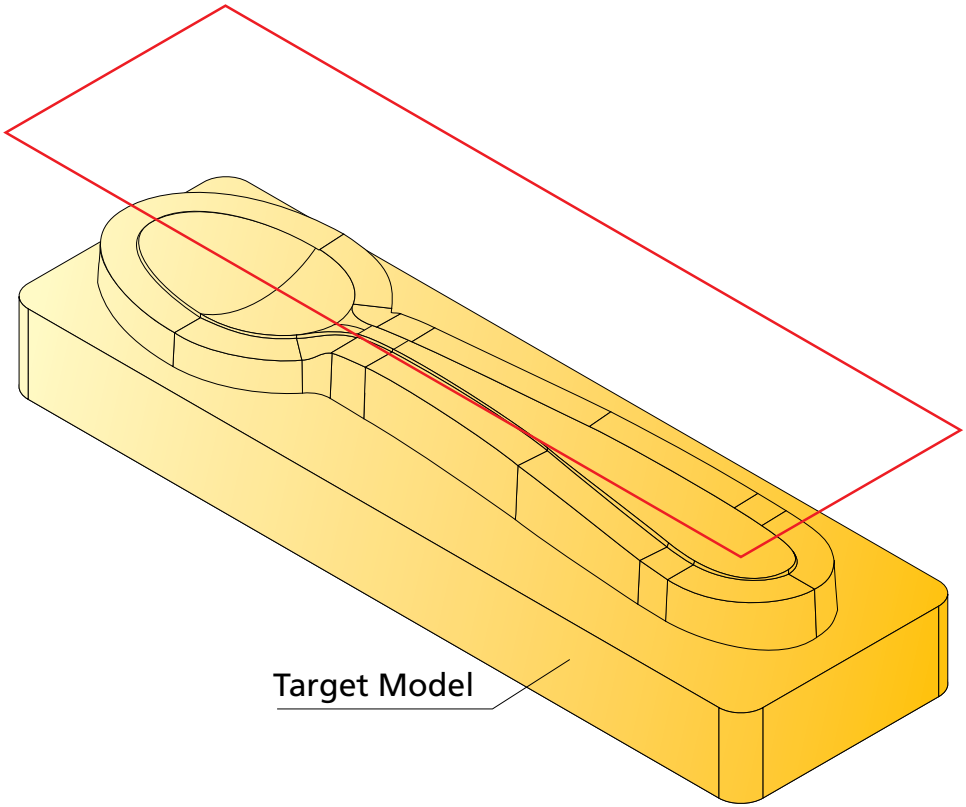
A positive offset value enlarges the boundary; a negative value reduces the boundary to be machined.



5.3 Automatically Created Boundaries

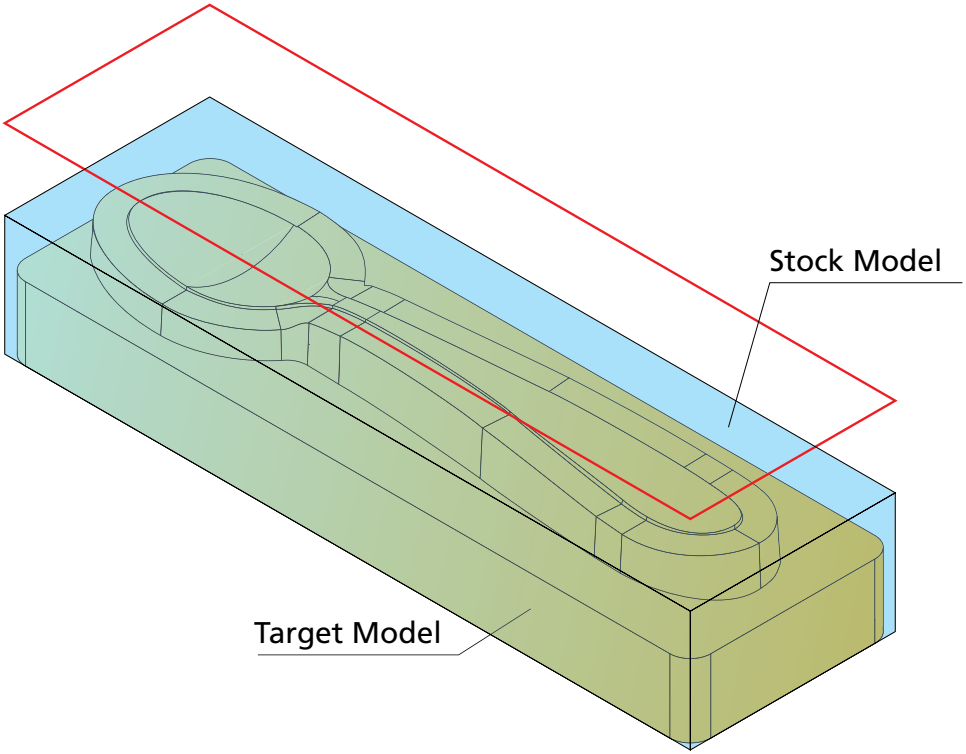
5.3.1 Auto-created box of target geometry

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



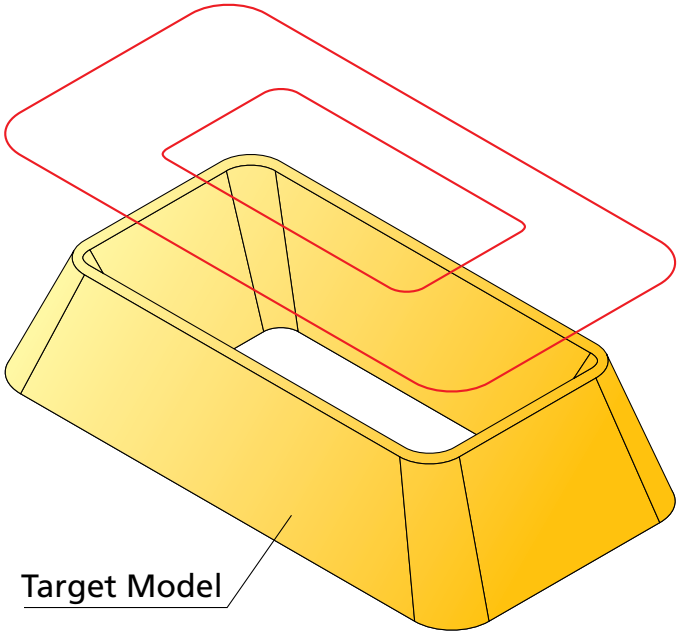
5.3.2 Auto-created box of stock geometry

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



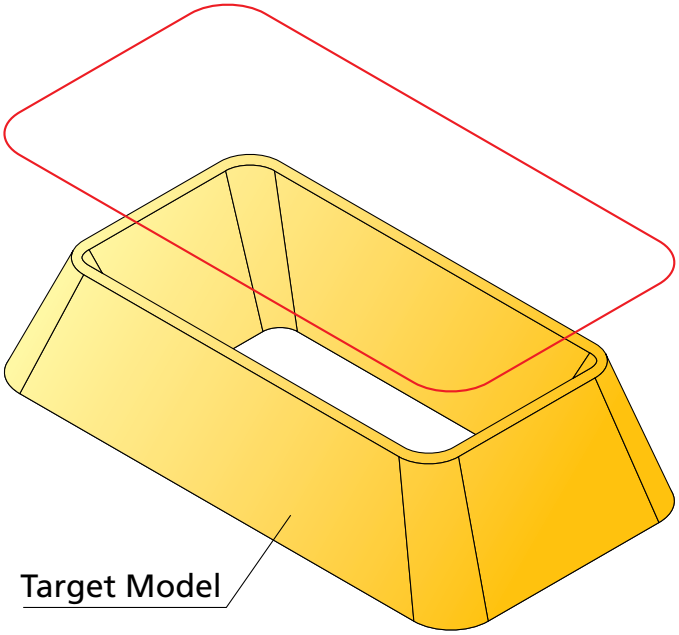
5.3.3 Auto-created silhouette

With this option, SolidCAM 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.



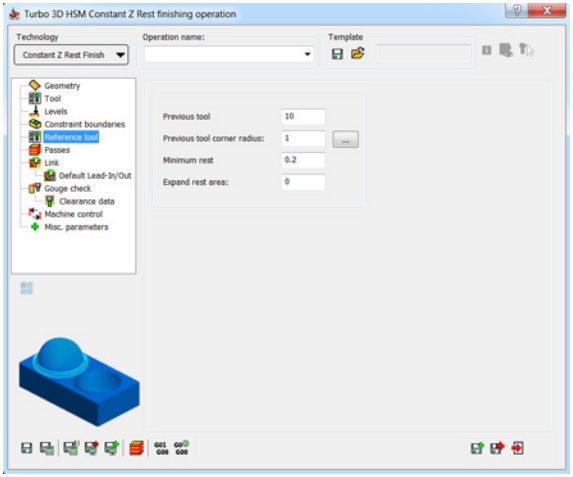
5.3.4 Auto-created outer silhouette

With this option, SolidCAM 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.



Reference Tool 6

The **Reference tool** page in the **3D HSM Operation** dialog box enables you to define the reference tool used for the Rest Machining tool path calculation.



Previous tool

The diameter of the tool used in the previous operation. This option enables you to use the previous tool or enter the value in the text field.

Previous tool corner radius

This parameter defines the Corner radius of the reference tool. Select ... to display the tool table and choose the relevant tool.

Min rest

This parameter defines the amount of material to be left prior to rest machining.

Expand rest area

This option enables you to expand the rest-finishing areas boundaries in order to process more material, and clean-up the remain regions more efficiently.

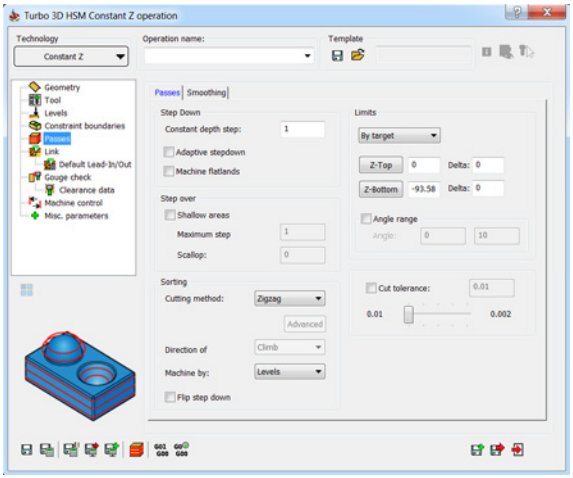
This page is available only when the **Technology** is selected as **Constant Z Rest Finish** or **Constant Step Over Rest Finish**.

Passes

7

The **Passes** page enables you to define the technological parameters needed to generate the tool path for the SolidCAM 3D HSM operation. For each **Technology**, the **Passes** page has different parameters to use.

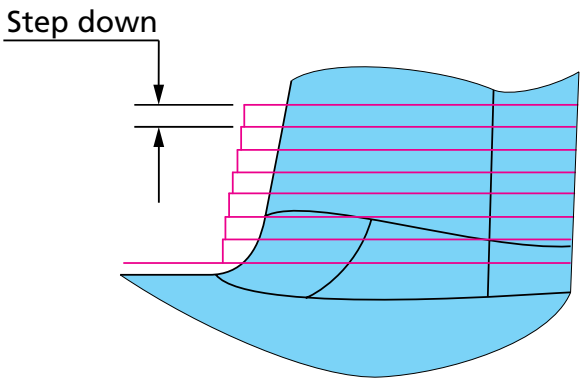
7.1 Passes Parameters for Constant Z



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

7.1.1 Step down

This option allows you to define intermediate slices based on depth distance between them. When the **Technology** is selected as **Constant Z**, the **Step down** parameter defines the spacing of the passes along the tool axis. The passes are spaced at the distance set, regardless of the XY-value of each position (unless the **Adaptive stepdown** check box is selected). Selecting the **Adaptive step down** check box adjusts the passes to get the best fit to the edges of a surface.

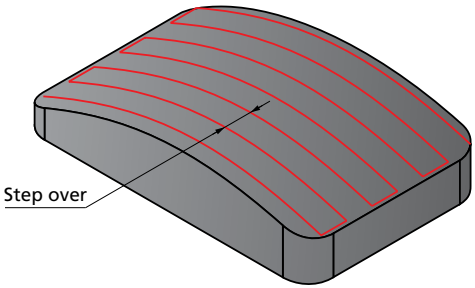


Selecting the **Machine flatlands** check box enables you to cut an intermediate slice at the height of the flat area when the generated Z level slices do not match the height of the flat area which lies in between the step over of two Z slices.

At a time you can either select the option of **Adaptive stepdown** or **Machine flatlands**.

7.1.2 Step over

Step over is the distance between two adjacent passes. Select the **Shallow areas** check box. This option enables to create combined tool path which consists of Constant Z slices for steep areas and Constant step over cuts for shallow areas of a part.



The option of **Maximum step** enables you to define the distance between two adjacent cuts.

The option of **Scallop** enables you to define the value to control the distance between the tool path lines by the cusp height you want to achieve on the finished model.

7.1.3 Sorting

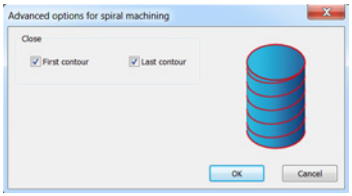
The **Sorting** section displays the options that enable you to define the sorting of the tool path passes.

Cutting method

This option enables you to define how the cuts are connected.

- **One way:** When this option is chosen, all cuts are machined in the same direction. The tool performs the machining of a cut in the specified direction, then moves to the start of the next cut and machines it in the same direction.
- **Zigzag:** When this option is chosen, the machining direction changes from cut to cut. The tool performs the machining of a cut in the specified direction, then moves to the next cut and machines it in the opposite direction.
- **Spiral:** When this option is chosen, a spiral tool path is generated around the drive surface according to the chosen pattern. Clicking the **Advanced** button displays the **Advanced options for spiral machining** dialog box. This dialog box enables you to define how the spiral cuts are performed.

When the **First contour** check box is selected, machining of the first slice is performed in a closed contour. The spiral machining motions start with the second slice. When this check box is not selected, the spiral machining motions start with the first slice.



When the **Last contour** check box is selected, machining of the last slice is performed in a closed contour.

The option of **Spiral** is available only with the **Constant Z Technology**.

Direction of

This option enables you to define the direction of the machining.

The options of **Climb** and **Conventional** set the tool path direction in such a manner that the climb or conventional milling is performed. This option is not available if **Zigzag** is selected as the **Cutting method**.

Machine by

SolidCAM enables you to define the machining order for a 3D HSM operation. The **Machine by** list enables you to choose the order of machining of certain areas; it defines whether the surface will be machined by **Lanes** or by **Regions**.

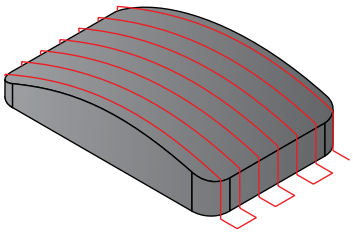
The generated tool path usually has a topology of multiple contours (lanes) on the drive surfaces. When the tool path is generated in many zones, it might be preferable to machine all the regions independently.

Flip step down

When this check box is not selected, the machining starts from top to bottom. When you select this check box, the machining will start from bottom to top.

7.1.4 Limits

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



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

- **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.

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

- **Angle range**. This parameter allows you to set the contact angle range of your tool by setting the minimum and maximum contact angle.

7.1.5 Cut tolerance

The **Cut tolerance** parameter defines the tool path accuracy. This parameter defines the chordal deviation between the machining surface and the tool path; the tool path can deviate from the surface in the range defined by the **Cut tolerance**.

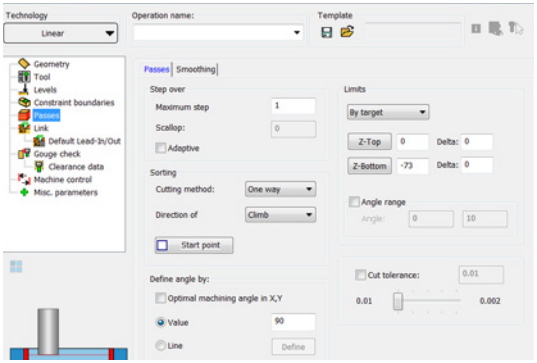
You can type the value manually or adjust it using the slider.

A smaller **Cut tolerance** value gives you more tool path points on the drive surface resulting in more accurately generated tool path. The result is a better surface quality, but the calculation time is increased.

A greater **Cut tolerance** value generates less points on the tool path. After the machining, the surface finish quality is lower but the calculation is much faster.

The sections of **Limits**, **Angle range**, and **Cut tolerance** are same for every **Technology** of 3D HSM.

7.2 Passes Parameters for Linear



7.2.1 Step over

Step over is the distance between two adjacent passes. In the **Linear Technology**, selecting the **Adaptive** check box enables you to adjust the step over between tool path passes in an adaptive way, in order to ensure an acceptable distance between adjacent passes.

7.2.2 Sorting

The **Sorting** parameters are same as for **Constant Z Technology** except for the options of **Start point** and **Define angle by**.

Start point

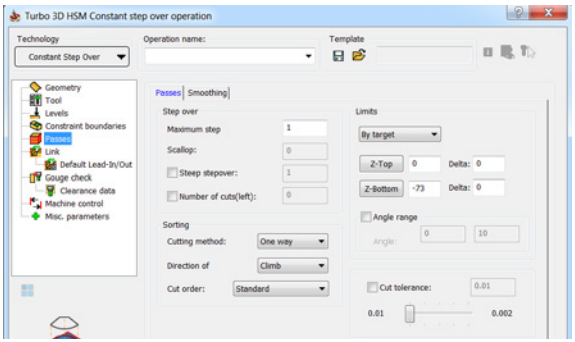
Clicking the **Start point** check box displays the **Start Point Parameters** dialog box. For closed contours, the Start point option enables you to define a new position of the start point of the first cut. The position is defined along a cut. The start points of the next cuts are determined automatically, taking into account the start point location and the cutting strategy. Select the **Use Start point** check box to enable the dialog box options. Vector coordinates can be set by numeric values, or the direction can be picked on the model using ...

7.2.3 Define angle by

When you select the **Optimal machining angle in X,Y** check box, SolidCAM automatically defines the angle required for machining.

When **Optimal machining angle in X,Y** check box is not selected, the angle can be defined by entering its **Value** in the edit box or by picking a **Line** on the model.

7.3 Passes Parameters for Constant Step Over



7.3.1 Step over

Step over is the distance between two adjacent cuts. The **Maximum step** distance can be defined as 'value for the side step' or as 'value for the cusp height'.

The option of **Scallop** enables you to specify a value to control the distance between the tool path lines by the cusp height you want to achieve on the finished model. This option is available only when the **Steep stepover** check box is not selected.

The option of **Steep stepover** enables you to specify the distance between adjacent cuts located on the steep areas of a part.

The **Number of cuts(left)** allows you to limit the number of cuts or set a determined number of cuts. The value 0 will create a full pattern until the edge of the machining surfaces or a given containment.

7.3.2 Cut order

Cutting method

This option enables you to define how the cuts are connected.

- **One way:** When this option is chosen, all cuts are machined in the same direction. The tool performs the machining of a cut in the specified direction, then moves to the start of the next cut and machines it in the same direction.
- **Zigzag:** When this option is chosen, the machining direction changes from cut to cut. The tool performs the machining of a cut in the specified direction, then moves to the next cut and machines it in the opposite direction.
- **Spiral:** When this option is chosen, a spiral tool path is generated around the drive surface according to the chosen pattern.

Direction of

This option enables you to define the direction of the machining.

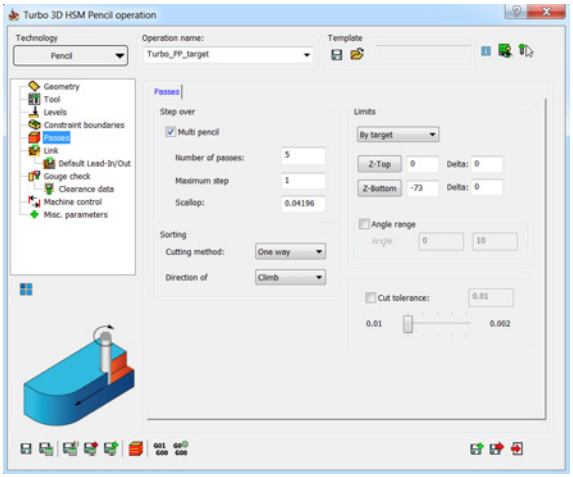
The options of **Climb** and **Conventional** set the tool path direction in such a manner that the climb or conventional milling is performed. This option is not available if **Zigzag** is selected as the **Cutting method**.

Cut order

The cut order defines the cut sequence for Constant Step Over machining.

- **Standard:** When there are no islands in the tool path, the machining starts from outside and progresses inwards. When islands are built in the tool path, the machining starts from outside and progresses inwards for all islands. The machining starts at the outer islands.
- **From center away:** With one outer containment, the machining starts from inside and progresses outwards. With outer and inner containment, the machining starts from the island center and progresses outwards both the sides.
- **From outside to center:** With outer and inner containment, the machining starts from the island outer containments and progresses inwards towards the island center. With outer containment, the machining starts from outside and progresses inwards.
- **Top to bottom:** Within all regions the machining starts at the top slice and progresses towards the floor.
- **Bottom to top:** Within all regions the machining starts at the floor and progresses towards the top.

7.4 Passes Parameters for Pencil



7.4.1 Step over

Multi pencil

This option creates multiple offset cuts from the initial cut. Selecting the check box to enables the following:

The option of **Number of passes** defines the number of offset passes.

The option of **Maximum step** defines the maximum distance between two adjacent multipencil cuts.

The option of **Scallop** enables you to specify a value to control the distance between the tool path lines by the cusp height you want to achieve on the finished model.

7.4.2 Sorting

Cutting method

This option enables you to define how the cuts are connected.

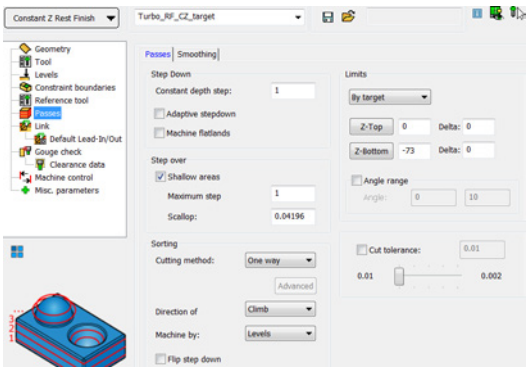
- **One way:** When this option is chosen, all cuts are machined in the same direction. The tool performs the machining of a cut in the specified direction, then moves to the start of the next cut and machines it in the same direction.
- **Zigzag:** When this option is chosen, the machining direction changes from cut to cut. The tool performs the machining of a cut in the specified direction, then moves to the next cut and machines it in the opposite direction.

Direction of

This option enables you to define the direction of the machining.

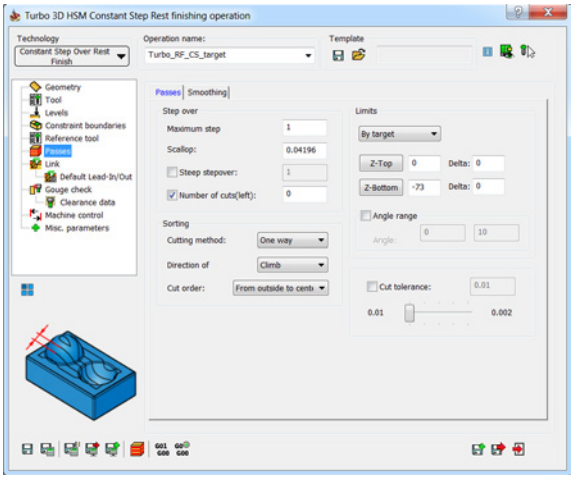
The options of **Climb** and **Conventional** set the tool path direction in such a manner that the climb or conventional milling is performed. This option is not available if **Zigzag** is selected as the **Cutting method**.

7.5 Passes Parameters for Constant Z Rest Finish



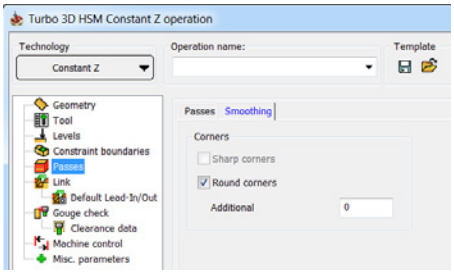
The **Passes** tab for this **Technology** is the same as for the **Technology** of **Constant Z**. For more information, refer to **7.1**.

7.6 Passes Parameters for Constant Step Over Rest Finish



The **Passes** tab for this **Technology** is the same as for the **Technology** of **Constant Step Over**. For more information, refer to **7.3**.

7.7 Smoothing Parameters



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

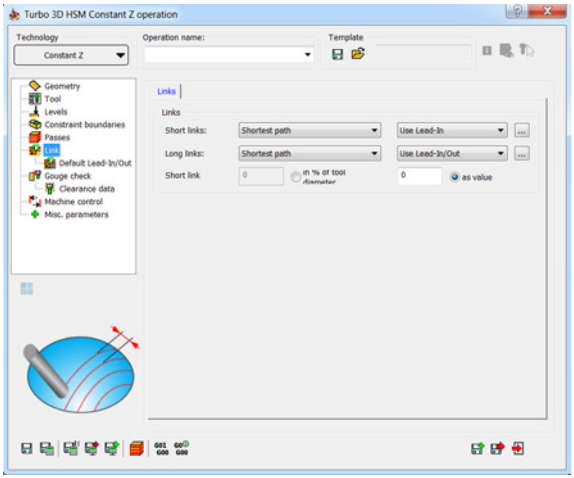
When you select the **Sharp corners** check box, it enables you generate sharper tool path in the corners.

When you select the **Round corners** check box, it enables you round out the inner corners for machining. You can define a corner radius in the **Additional** field.

This tab is not available when **Pencil** is selected from the **Technology** list.

Link 8

The **Link** page in the **3D HSM Operation** dialog box enables you to define the way how the generated passes are linked together into a tool path.



Short/Long Links

The options of **Short** and **Long Links** enable you to define connection type between the cuts. Both the list have the options of: **Blend spline**, **Shortest path**, and **Retract to clearance area**.

- **Blend spline**. This option allows you to connect the pass segments with a spline tangential to both the segments.
- **Shortest path**. This option enables the use of shortest distance connection considering shape of the part and safety conditions.
- **Retract to clearance area**. This option enables the use of straight line connection between the passes with retracting tool to the specified clearance area.

In the **Short link** field, you can define the value as **in % of tool diameter** or **as value**.

Lead-In/Lead-Out

SolidCAM enables you to use pre-defined **Lead-In**/**Lead-Out** strategies to perform the movements between segments of a pass divided by a gap.

- **Use Lead-In**. SolidCAM performs the approach movement to the drive surface after the gap using the specified **Lead-In** options.
- **Use Lead-Out**. When a gap is detected, SolidCAM performs the retreat movement using the specified **Lead-Out** options.

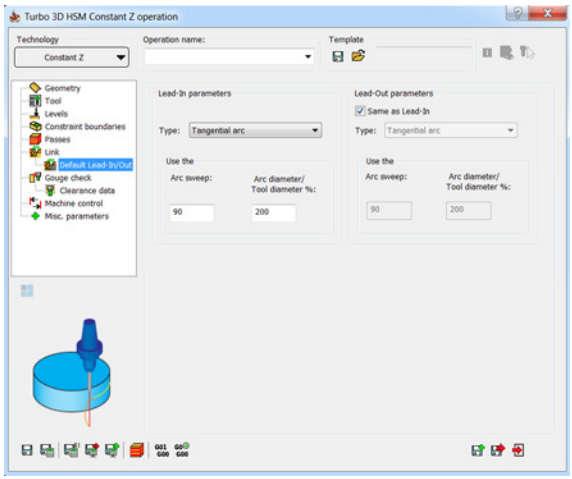
Select ... in front of **Lead-In** or **Lead-Out** to display the parameters for these options.

SolidCAM enables you to choose the following types of lead-in/lead-out.

- **Tangential arc**. This option enables you to connect the tool path start point tangential to the contour. The orientation is along the normal to machining surface.
- **Vertical tangential arc**. This option enables you to connect the tool path start point tangential to the contour. The orientation is orthogonal to the machining plane.
- **Horizontal tangential arc**. This option enables you to connect the tool path start point tangential to the contour. The orientation is on the machining plane.

8.1 Default Lead-In/Out

The **Default Lead-In/Out** parameters page enables you to define the parameters used for the approach/retreat movements.



Arc sweep

This option enables you to define the angle of the approach/retreat arc segment.

Arc diameter/Tool diameter

This option enables you to specify the diameter of the approach/retreat arc using the percentage of the Arc diameter to the Tool diameter. For example, when a tool with the diameter of 10 mm is used and the percentage parameter is set to 200%, the resulting approach/retreat arc diameter is 20 mm.

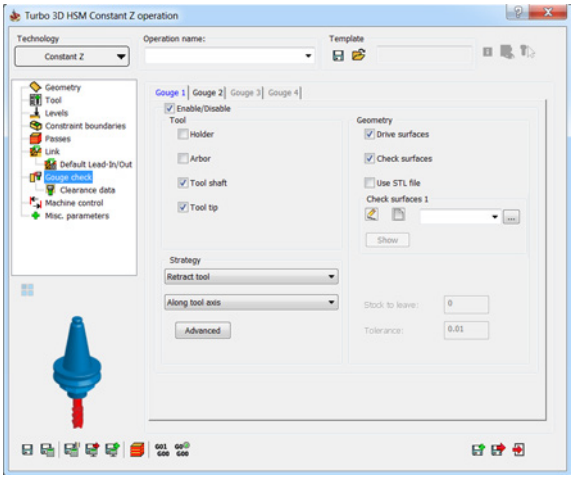
Gouge check 9

The **Gouge check** page enables you to automatically detect and avoid the possible collisions between the tool (with the tool holder) and the workpiece.

9.1 Gouge checking

SolidCAM enables you to define four different sets of gouge checking parameters. In each set you have to choose components of the tool holding system and model faces to check the possible collisions between them. You also have to define the strategy how to avoid the possible collisions. Combining these sets, SolidCAM enables you to choose different strategies for avoiding the different types of possible collisions.

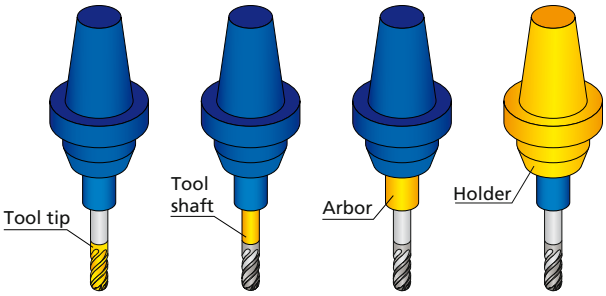
Select the **Enable/Disable** check box to activate a set of gouge checking parameters.



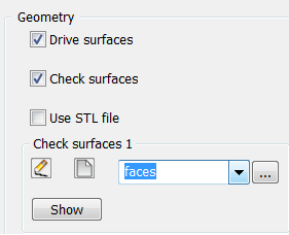
9.1.1 Tool

This section enables you to choose both tool and tool holding system components to perform the gouge check for them. The following parts of the components are available for the gouge check:

- **Holder**
- **Arbor** (shank)
- **Tool shaft** (between flute length and arbor)
- **Tool tip** (flute length)



9.1.2 Geometry



The **Geometry** section enables you to choose the model faces for which the gouge checking is performed.

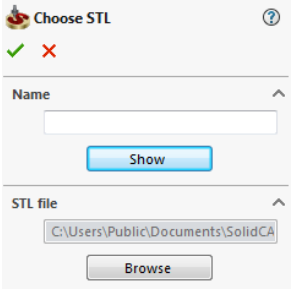
Drive surfaces

When this option is chosen, SolidCAM performs the gouge checking for the **Drive surfaces** avoiding the possible collisions.

Check surfaces

With this option, SolidCAM enables you to choose a number of non-drive surfaces on the model as the check surfaces and perform the gouge checking for them.

The **Check surfaces 1** section enables you either to choose the check surfaces geometry from the list or define a new one with the  button displaying the **Select Faces** dialog box.



When the **Use STL file** check box is activated, the **Check surfaces** section enables you to choose a check surfaces geometry from an STL file. The  button enables you to display the **Choose STL** dialog box.

The **Browse** button in the dialog box enables you to choose the necessary STL file. The full name (including the path) of the chosen STL file is displayed in the **STL file** edit box. The **Show** button enables you to display the chosen STL file in a separate window.

When only the **Check surfaces** option is chosen, SolidCAM enables you to define two additional parameters:

- **Stock to leave**. This parameter enables you to define an allowance for the check surfaces. The tool will reach the check surface by the specified value. For example, if the **Stock to leave** value is set to 1, SolidCAM checks that the tool is kept away from check surfaces by 1 mm.

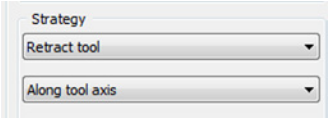
This parameter is used only in case when only the **Check surfaces** option is chosen for the gouge checking (the **Drive surfaces** option is not chosen). In case when both **Drive surface** and **Check surface** options are turned on, SolidCAM uses **Drive surface offset** to define the machining allowance for both drive and check surfaces.

- **Tolerance**. This parameter enables you to define the accuracy of the gouge checking for the check surfaces. The value defines the chordal deviation between the tool path and the check surfaces.

This parameter is used just in cases when only the **Check surfaces** option is chosen for the gouge checking (**Drive surfaces** option is not chosen). In cases when both **Drive surface** and **Check surface** options are turned on, SolidCAM uses **Cut tolerance** value (see topic **6.1.1**) to define the gouge checking tolerance for both drive and check surfaces.

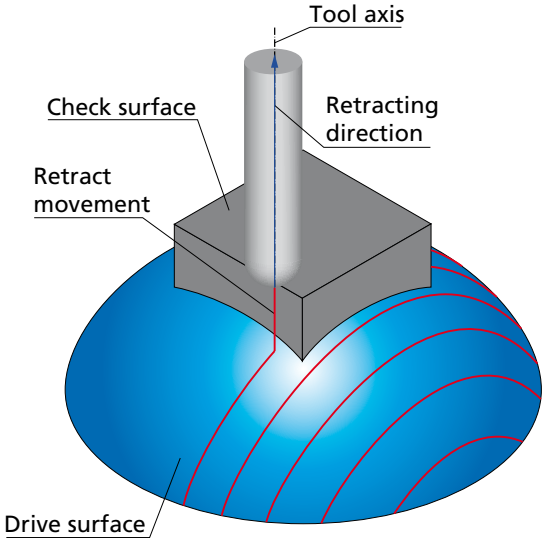
9.1.3 Strategy

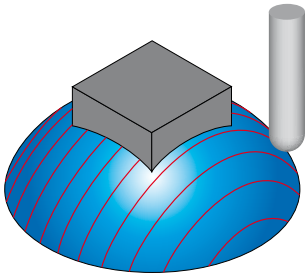
SolidCAM provides you with a number of strategies enabling you to avoid possible gouges.



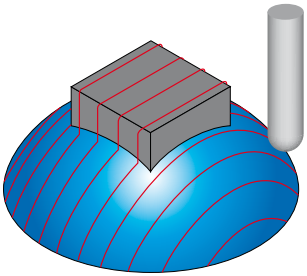
Retract tool

When this option is chosen, SolidCAM enables you to avoid the possible collisions by retracting the tool. When a possible collision is detected, the tool performs a retract movement at the automatically calculated distance and then "flows" around check faces avoiding the gouge. The initial gouging tool path is substituted with a new one free of gouges.





Initial tool path gouging the check surfaces



Updated tool path not gouging the check surfaces

Advanced

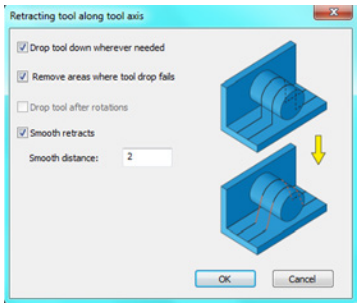
SolidCAM provides you with advanced parameters for projection of the tool path from the drive surface plane onto the model to be machined.

In certain machining cases, it is convenient to define the required tool path on a flat surface, which facilitates the definition of parameters such as distance between cuts, machining angle that is set in the XY-plane, etc. Then this tool path can be projected onto a 3D model that needs to be machined.

Consider the example of the following part: all of its surfaces need to be machined in simple parallel cuts. Instead of selecting all of its faces as drive faces to process the tool path on, you can create a new flat surface, on which the tool path will be defined. This surface will be defined as the drive surface, to which the actual tool path will be applied. You can easily check the resulting tool path on the flat surface, edit if necessary, and then to project it onto the faces of the model to be machined. Note that these faces will be defined as check surfaces.

To project the obtained tool path on the check surfaces, click the **Advanced** button.

The **Retracting tool along tool axis** dialog box is displayed. This dialog box enables you to define the parameters of tool path projection.



The **Drop tool down wherever needed** check box enables you to activate the projection of the tool path on the required 3D geometry. When you select this check box, the tool path applied to the drive surface is projected onto the defined drive surfaces.

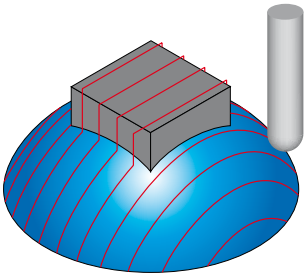
The **Remove areas where tool drop fails** check box enables you to exclude the areas, where the projection cannot be performed, from the tool path. When this check box is selected, the positions where the projection failed are removed.

The **Smooth retracts** check box enables you to smooth the transition from the collision free area to the tool retraction area by avoiding sudden axis jumps. The **Smooth distance** field determines the start distance of the smoothing to the collision area.

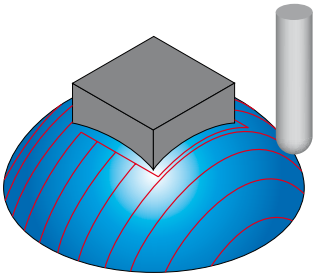
SolidCAM enables you to choose the following options to define the retract direction:

- **Along +X, -X, +Y, -Y, +Z, -Z**

  The retraction movement is performed along the chosen axis.



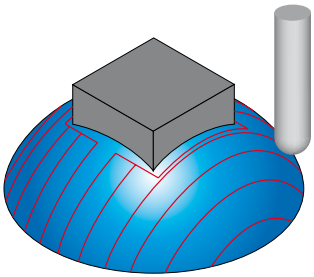
Retract tool in +Z



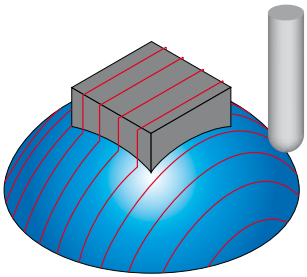
Retract tool in +X

- **Along ZX, YZ, XY plane**

  The retraction movement is performed in the chosen plane. The retraction movement is performed in the direction defined by the projection of the drive surface normal vector on the chosen plane.



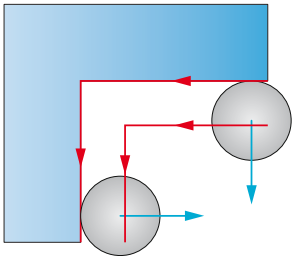
Retract tool in XY plane



Retract tool in ZX plane

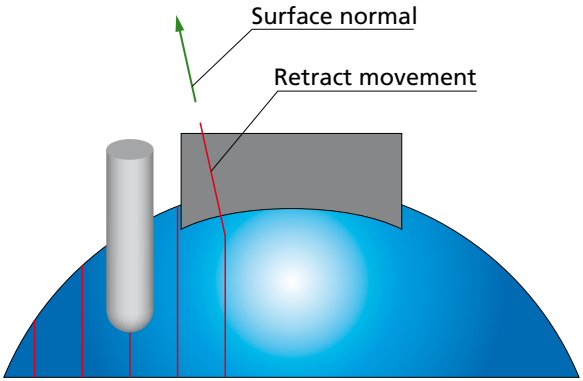
- **Along-optimized in ZX, YZ, XY plane**

  The retraction movement is performed in the chosen plane, similar to the **Along ZX, YZ, XY plane** plane options; the differences are in the direction of the retract movements in the chosen plane. The contact points, at which collisions are detected, are projected on the chosen plane and connected into a contour. This contour is offset outwards by a distance equal to the sum of the tool radius and the **Stock to leave** values. This option enables you to perform the retract movements in optimal directions, generating the shortest tool path.



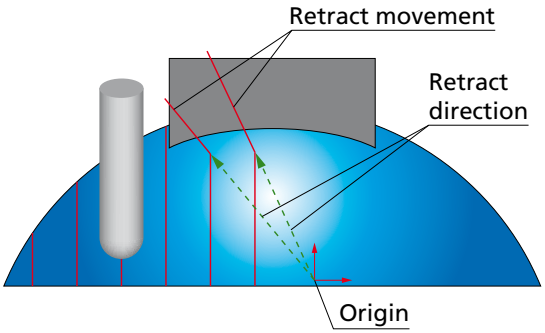
- **Along surface normal**

  The retraction movement is performed in the direction of the drive surface normal at the contact point.



- **Away from origin**

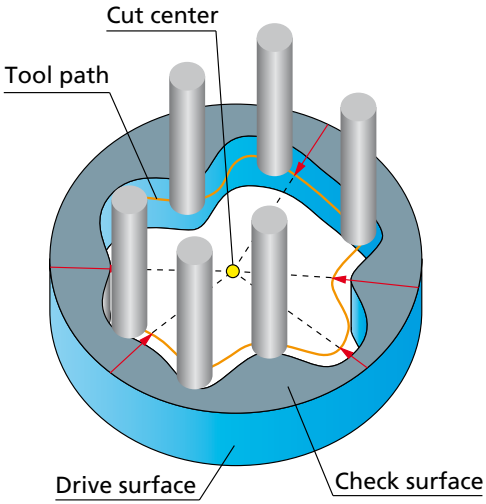
  The retraction movement is performed in the direction of the vector from the Coordinate System origin to the tool contact point.



- **Along to cut center**

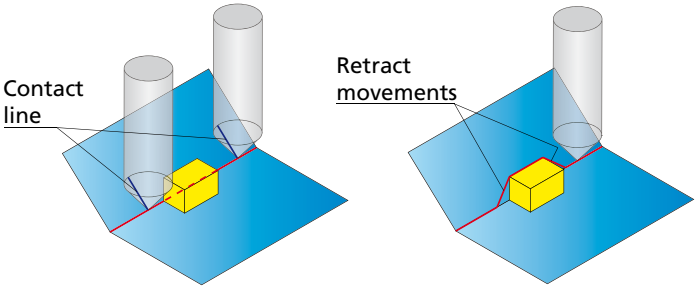
  The retraction movement is performed in the direction of the center of the cutting pass. This option is useful for tube milling.

  On the illustration, the machining of the drive surface is performed with the parallel Z cutting passes. When the gouging of the check surface occurs, the retract movement is performed to the cutting pass center, avoiding the gouge.



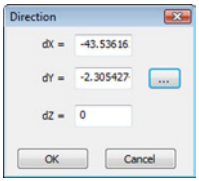
- **Along tool contact line**

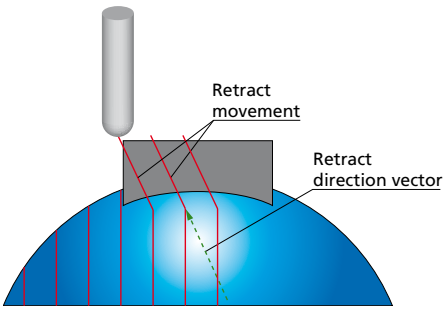
  The retraction movements are performed along the contact line between the tool and the drive surface.



- **Along user-defined direction**

  This option enables you to define the direction of the retraction movements by a vector. The **Direction** dialog box (available with the button) enables you to define the direction vector by its coordinates (**dX**, **dY** and **dZ** parameters). Using the button, SolidCAM enables you to pick the start and end points of the vector directly on the solid model.





- **Along tool axis**

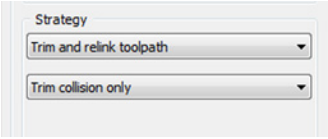
  This option enables you to avoid the possible collisions by retracting the tool in the direction of the tool axis.

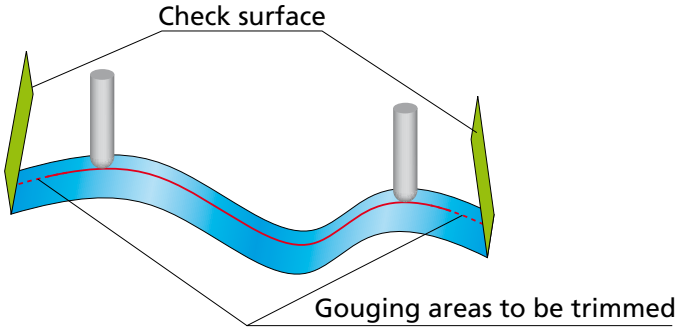
- **Along tool plane**

  This option enables you to move the tool in its tool plane. The tool plane is the plane that is normal to the tool axis. This option avoids pushing tool into a certain single direction, eliminating possible collisions while the tool orientation and height are maintained during the machining.

Trim and relink toolpath

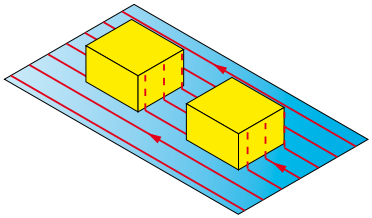
When this strategy is used, SolidCAM trims the segments of the tool path where the collisions are detected. The tool path updated by trimming does not contain gouges.



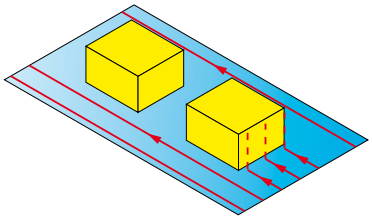


The following trimming options are available:

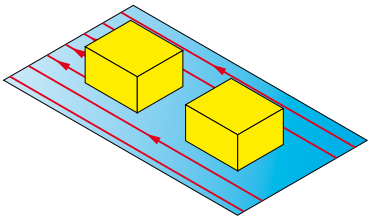
- **Trim collision only**. With this option, only the colliding segments of the tool path are trimmed out.



- **Trim tool path after first collision**. With this option, SolidCAM trims the whole cutting pass, after the first detected collision.

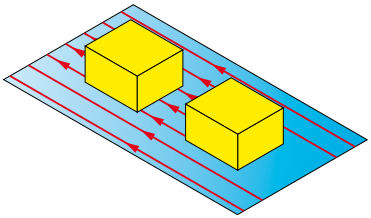


- **Trim tool path before last collision**. With this option, SolidCAM trims the whole cutting pass, before the last detected collision.

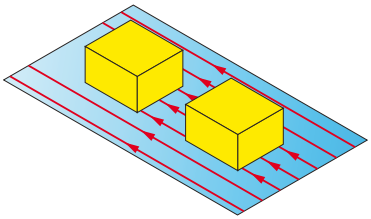


- **Trim tool path between first and last collision**. With this option, SolidCAM trims the cutting path between the first and last detected collisions.

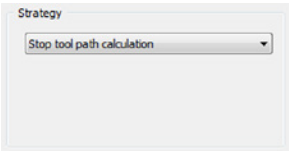
- **Trim tool path before first collision**. When a collision is detected, the tool path is trimmed in such a manner that the portion of the current cut from the beginning till the first collision is removed from the tool path.



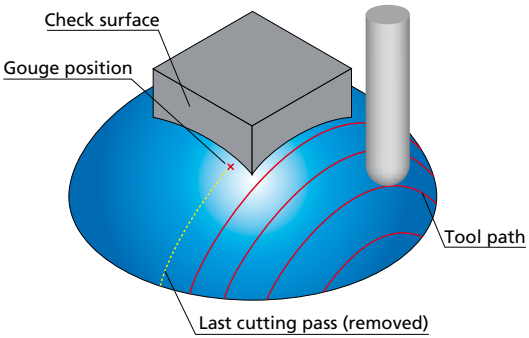
- **Trim tool path after last collision**. When a collision is detected, the tool path is trimmed in such a manner that the portion of the current cut from the last collision till the end of the cut is removed from the tool path.



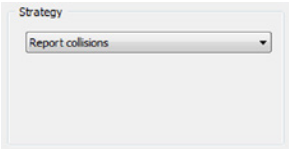
Stop tool path calculation



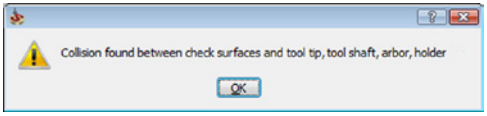
When this option is chosen, the tool path is generated until the position where the first gouge occurs. At this point the tool path calculation is stopped. The last cutting pass (where the gouge is detected) is not included into the operation tool path. You have to edit the machining parameters and calculate the tool path again to avoid the gouge.



Report collisions



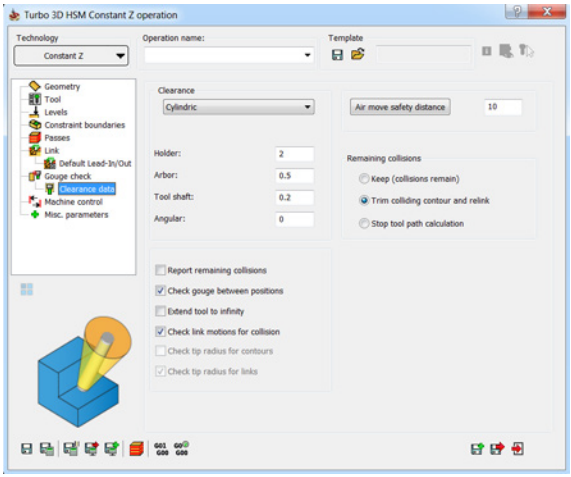
SolidCAM checks only for collision between the tool and the check faces, without trying to avoid the collision; a warning message is displayed.



Using the simulation, you can check the collision areas and choose the appropriate method to avoid gouging.

9.2 Clearance data

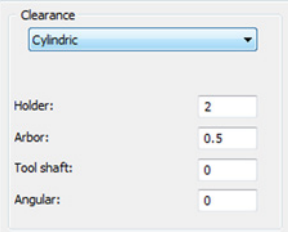
The **Clearance data** page enables you to define the clearance offsets for arbor and tool holder in order to get a guaranteed clearance gap between arbor, tool holder and workpiece.



9.2.1 Clearance

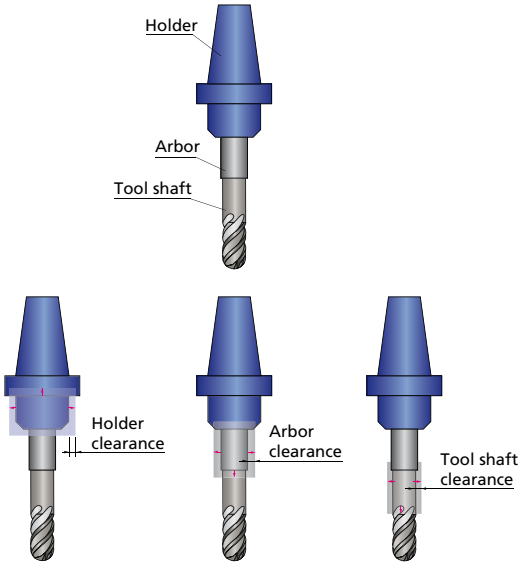
SolidCAM enables you to choose either **Cylindric**or **Conical** shape of the tool holder, arbor and tool shaft clearance.

Cylindric clearance



The **Holder** parameter defines the offset applied to the tool holder cylinder from all sides. The **Arbor** defines the offset applied to the arbor cylinder from all sides. The **Tool Shaft** parameter defines the offset applied to the tool shaft cylinder from all sides. The **Angular** parameter defines the angular offset applied to the tool.

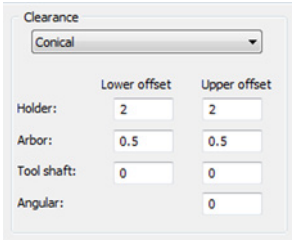
Generally, an arbor is the tool extension located between the tool shaft and its holder. Lollipop and Slot End Mills do not have tool shaft, the cylindrical Connection between tool and holder is considered as Arbor.



Conical clearance

Conical clearance is applied similarly to the cylindrical one being defined with **Upper** and **Lower offset** values.

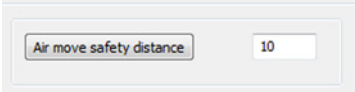
Angular conical clearance is applied between the tool and collision surface. It is spanned between the contact point of the tool, the drive surface, and the collision point.



The **Upper offset** value must be greater than the **Lower offset** value.

Air move safety distance

This parameter enables you to define the minimal distance between the clearance area and the **Drive surface** (see topic **3.1.1**).



9.2.2 Report remaining collisions

This option enables you to generate a report about possible collisions that remain in the tool path after gouge checking. When this option is selected, SolidCAM checks the tool path using the tolerance two times greater than the specified value (see topic **9.1.2**) to detect collisions.

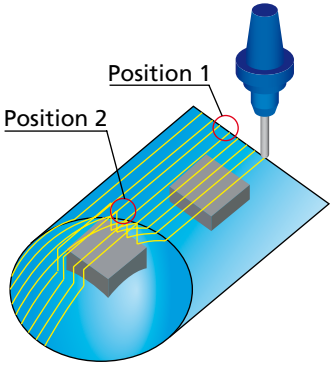
You can turn off the collision checking between the tool path positions (see topic **9.2.3**). In such case the tool path calculation is accelerated, but the possibility of remaining collisions is present. The **Report remaining collisions** option is helpful to notify about possible collisions in the resulting tool path.

In case of engraving or trimming operations used together with the **Report remaining collisions** option, SolidCAM notifies you about collisions. The reason for such notification is that the technology of such operations requires that the tool tip be inside the machined surfaces.

The **Report remaining collisions** option enables you to detect too small retract and approach distances or too low clearance levels. In such case, the report about collisions enables you to solve the potential problems.

9.2.3 Check gouge between positions

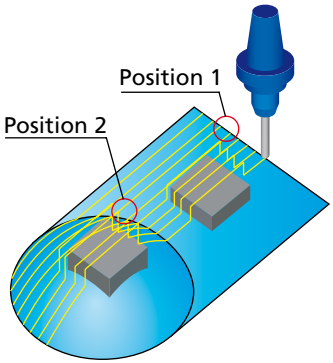
The **Check gouge between positions** option enables you to avoid the possible gouges between tool path positions. When the movement is performed between two successive tool path positions, this option enables you to check for possible collisions of the tool and tool holder with drive and check surfaces.



This option is useful especially for flat faces machining, where the tool path positions are generated only at the drive surface edges.

When the **Check gouge between positions** option is not used, the gouge checking of the tool path on the flat face is not performed because of absence of tool path positions on the face. The gouging of a boss may occur.

When the **Check gouge between positions** option is used, the gouge checking between tool path positions on the flat surface is performed. The gouging of a boss is avoided.



The **Check gouge between positions** option has no effect on the gouge checking of the tool path spherical surface, because of the many tool path positions that were generated on this face. The gouge checking for this face is performed for these positions avoiding possible collisions.

9.2.4 Extend tool to infinity

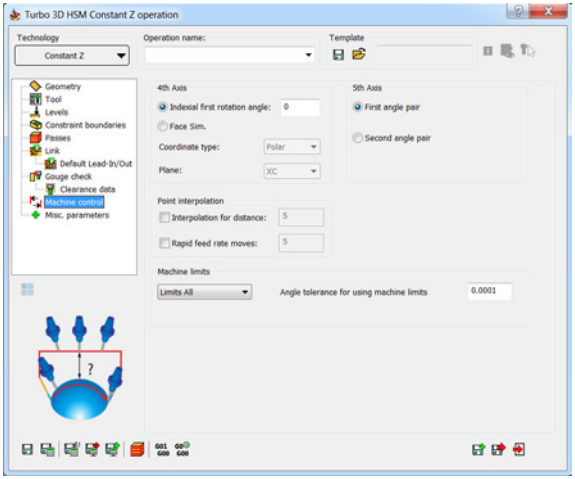
This option enables you to consider the tool as being extended to infinity during collision check in order to make sure that all active surfaces are checked for collision, no matter where they are located in space.

9.2.5 Check link motions for collision

When this option is chosen, SolidCAM automatically performs the gouge checking for link movements in order to avoid possible collisions.

Machine Control 10

Using the parameters of the **Machine 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 **Machine control** page may vary depending on the parameters of your CNC-machine.

10.1 4th Axis

This option enables you to project the portion of the 5th axis back onto the selected 4th axis.

Selecting the **Indexial first rotation angle** option allows you to set a rotational angle value to extend the working area of machines having a limitation in one of the linear axis in the working plane.

Selecting the **Face Sim** option enables you to perform milling using the rotary axis by translating the linear movements in the XY-plane into the rotary XC-movements.

Note that the technology offered by SolidCAM is universal: there is no limitations in terms of machines on which the part can be milled. But if you use the GCode output in the XYZ coordinates (the Diameter option), the usability of this GCode is limited to machines with possibility of movements along three linear axes (X, Y, Z). Therefore, to enable the possibility of milling the part on 3-axis Mill-Turn CNC-machines that support only the XZC coordinates where the movements along the Y-axis is impossible, you need to choose the Face option.

- The option of **Coordinate type** enables you to determine whether the GCode will consist of split blocks or blocks in polar/Cartesian coordinates. When you select the option of **Polar**, the tool path is calculated in polar coordinates. When you select the option of **Cartesian**, the tool path lines and arcs are calculated in Cartesian coordinates; the CoordSys position is zero for linear coordinates. The milling is processed using the rotary axis by translating the linear movements into the rotary-linear movements according to the plane.
- The option of **Plane**, allows you to choose the appropriate plane i.e, **XC**,**YC**, or **ZC**.

10.2 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, SolidCAM 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, SolidCAM adds 9 tool positions between start and end positions (0, 0, 10, then 0, 0, 20 etc.).

Rapid feed rate moves

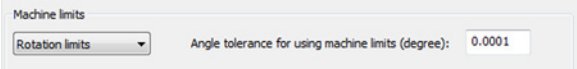
When this option is activated, all rapid motions (on clearance area and rapid distance) are converted into a feed motion with the given value.

10.3 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, SolidCAM 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**.

10.4 Machine limits



With this option, SolidCAM enables you to use the machine limits defined within the machine definition to limit the tool path movements in translation and/or rotation axis.

The following options of machine limits use are available:

- **No limits**

  All the machine limits defined in the machine definition are ignored.

- **Translation limits**

  SolidCAM uses the machine limits defined in the machine definition for translation movements.

- **Rotation limits**

  SolidCAM uses the machine limits defined in the machine definition for rotation movements.

- **All limits**

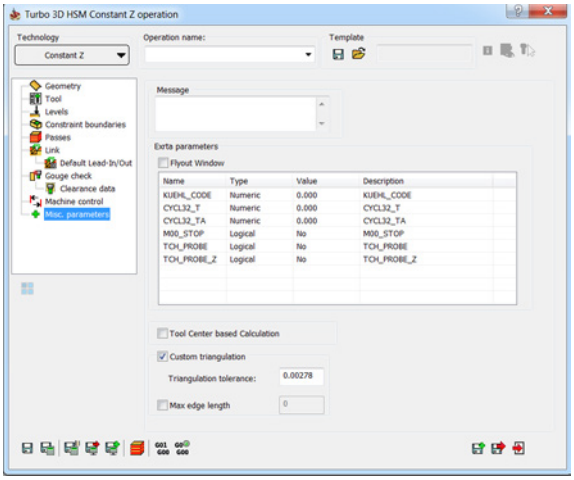
  SolidCAM uses the machine limits defined in the machine definition for both translation and rotation movements.

When machine limits are used, the calculated tool path is checked in order to avoid exceeding the machine limits. The check is performed using the angle tolerance defined by the **Angle tolerance for using machine limits** parameter.

Miscellaneous Parameters

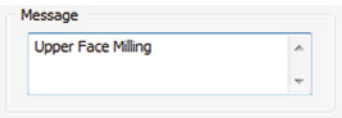
11

This page displays the non-technological parameters related to the 3D HSM operation.



11.1 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.
```

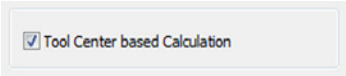
11.2 Extra Parameters

The **Extra parameters** option displays the list of additional parameters defined in the post-processor and enables you to use special operation options implemented in the post-processor for the current CAM-Part. If you prefer working with a larger window, selecting the **Flyout Window** option displays the Operation Option window.

11.3 Flyout Window

If you prefer working with a larger window, the Flyout Window option displays the Operation Option window.

11.4 Tool center based calculation



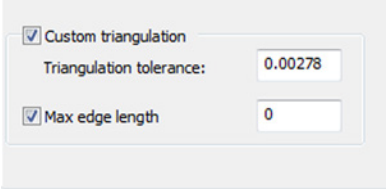
This option enables you to perform the tool path calculation based on the tool center.

When the option is turned off, the contact points between the tool and machined surface are located at the specified Z-levels. When the **Tool Center based Calculation** check box is selected, the tool center points are located at the specified Z-levels.

11.5 Custom triangulation

The **Custom triangulation** option enables you to achieve higher rate of accurate triangulation.

When the **Custom triangulation** check box is not selected, SolidCAM uses the native CAD triangulation method. When the **Custom triangulation** check box is selected, triangulation method is used to define the **Triangulation tolerance** and **Max. edge length**.



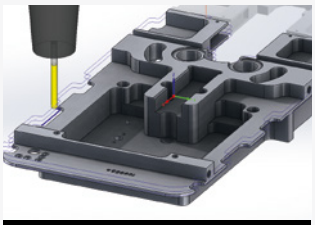
When the **Max. edge length** check box is not selected, the 5-Axis triangulation method is used, however, the results achieved are similar to the native CAD triangulation results. When the **Max. edge length c**heck box is selected, it allows you to control the maximum edge length.

SolidCAM 2018

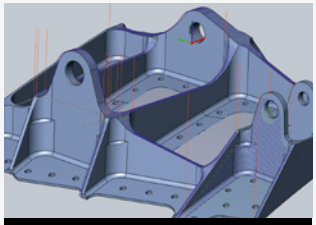
SolidCAM User Guide

Turbo 3D HSM

iMachining 2D

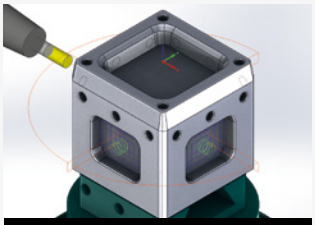


2.5D Milling



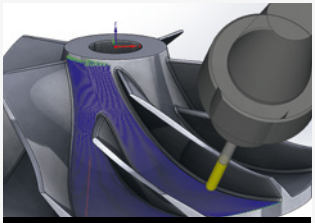
HSS (High-Speed Surface Machining)

iMachining 3D

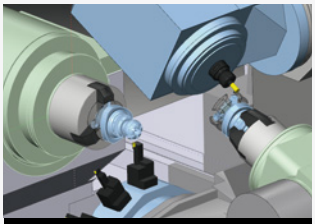


Indexial Multi-Sided Machining

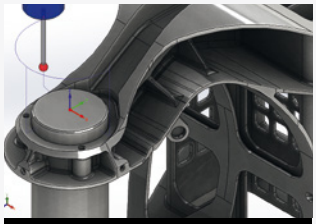
HSM (High-Speed Machining)



Sim. 5-Axis Machining



Turning & Advanced Mill-Turn



Solid Probe

www.youtube.com/SolidCAMProfessor
www.youtube.com/SolidCAMiMachining

www.facebook.com/SolidCAM

www.solidcam.com

SolidCAM
iMachining – The Revolution in CAM!