

worknc



Training Guide

Auto5 - V2017 R2

Shanghai Qianghu Informatin Technology Co.,Ltd
上海强互信息科技有限公司 WorkNC 中国总代理



1 - Table Of Contents

1 - Introduction	2
1.1 - Training Guide Objectives	2
1.2 - Auto5	2
2 - Converting a 3-Axis Toolpath	6
3 - 3 to 5 Axis Machining Strategies	21
3.1 - Vertical Strategy	25
3.2 - Constant to Axis Strategy	39
3.3 - No Change Strategy	48
3.5 - Constant (Attractive) Strategy	76
3.6 - Constant (Repulsive) Strategy	90
3.7 - Guide (Attractive) Strategy	98
3.8 - Guide (Repulsive) Strategy	106
3.9 - Guide (Surface) Strategy	110
3.10 - Vertical Rotation Strategy	126
4 - 3 to 5 Machining Additional, Advanced and Holder Parameters	133
4.1 - Holder Parameters for Collision Avoidance	133
4.2 - Advanced Parameters	144
4.2.1 - Adjusting Tilt Movements on Narrow Regions	148
4.3 - Keep Axis Along Machining Direction	162
4.4 - Smoothing Distance	169
4.5 - Surface Sensitivity	173
5 - Auto3+2 Strategy	180
5.1 - Programming an Auto3+2 Strategy	193
6 - Machine Limit Check	215
6.1 - Selecting a Machining Context	217
6.2 - Defining the Main Settings	224
6.3 - Defining Additional Settings	229
6.4 - Defining Start and End Points	237
6.5 - Defining Start and End Angles	241
6.6 - Saving Machine Settings	249
6.7 - Defining Advanced Parameters for the Machine Limit Check Function	251

1 - Introduction

1.1 - Training Guide Objectives

This training session and its related manual require that the user has a good working knowledge of **WorkNC**. He must know how to activate parts, create and edit toolpaths. The particular examples given in the manual may not concern all users or applications of **WorkNC**. The main goal of this manual is to provide users with the tools (knowledge and concepts) that may be applied to specific problems that they may encounter when working with the **Auto5** Module.

At the end of the training session, the user should be able to:

- Run the **Auto5** module,
- Select a strategy to convert a 3-axis toolpath into a 5-axis toolpath,
- Define the appropriate parameters to convert 3-axis toolpaths,
- Program an **Auto3+2** strategy,
- Select the machine in the **Auto5** module,
- Define the appropriate parameters in the **5 to Machine** section of the **Auto5** module.

Materials Required

Workzones

Auto5_Initial
Auto5_Keep_Axis
Auto5_v23
Fender_Auto5
Mold_moule_form
Start-end_angles
Pumpe_auto5
Auto5_slim_regions
Auto5_retract
Auto5_retracts2

Holder

Fidia_HSC_Hydro_D6.hld
Holder_606373_07.hld
Fidia_HSC_Hydro_D6_spindle.hld
Outil1.hld

CAD Parts

Multi-sided_part.xdw

This training manual is not a comprehensive manual. For more help you can go to our **Online Help** or read our other training manuals.

1.2 - Auto5

While 3 axis toolpaths are mostly enough for "usual" parts, they may sometimes pose problems on parts with deep areas and thin pockets. One solution may be to change the orientation of the tool and machine areas which were not visible. But when the number of deep areas and pockets is too high, the time spent in preparing views and toolpaths is too long.

The **Auto5** module has been developed to meet these demands.

Methodology

There are two principal phases in the **Auto5** methodology :

1. The first step uses the **3 to 5 Axis** module which takes a 3-axis or 3+2-axis toolpath and calculates a 5-axis toolpath by avoiding collisions. If a collision can not be avoided this position is marked. This process does not depend on the machine.

2. The second step uses the **Machine Limits Collision Check** module which takes the output of **3 to 5 Axis** and considers the machine angle limits by selecting a particular 5-axis machine. All positions of the toolpath which have a collision are replaced by retracts and lead-ins in the resulting toolpath.

Auto5 Workflow

2 - Converting a 3-Axis Toolpath

The conversion of a 3-axis toolpath into a 5-axis toolpath is made through the **Workzone Manager** by selecting the toolpath in the list and clicking the



icon on the right side of the user interface.

1. Open the *Auto5_initial/workzone*.

Auto 5: Reference 3-Axis Toolpath

The only toolpath that has been calculated is a 3-axis **Planar Finishing** toolpath.

2. Open the **Toolpath Parameters** menu of this toolpath: note that it has been calculated with a 6 mm ball-end cutter and a machining view to limit the toolpath.
3. Make a copy of the toolpath.

We are now going to transform this newly created toolpath into a 5-axis toolpath.

4. Select the new toolpath, click on the



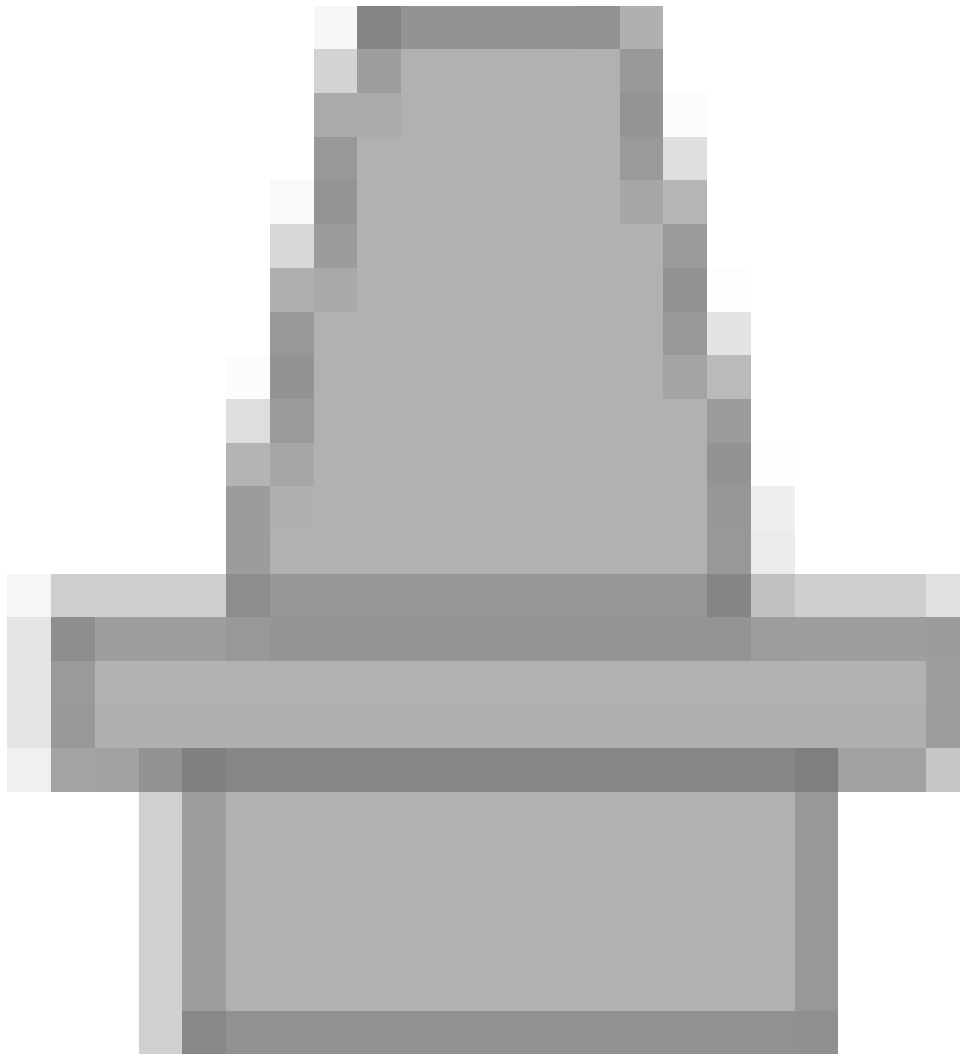
icon and select the *Fidia_HSC_Hydro_D6* tool holder.

Collision Detection

As you do not need to run collision detection, you can select the **No Collision Detection** option.

5. Click **OK** to validate.

In the **Workzone Manager**, the



that a tool holder has been selected.

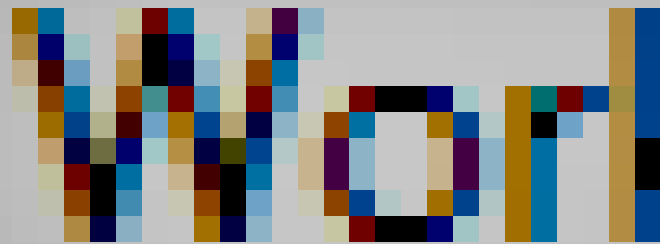
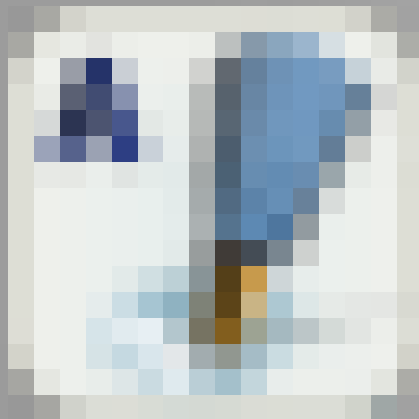
icon indicates

6. Make sure that the toolpath is selected and click on the



icon.

The **Auto5** module is displayed.



No.

2

A

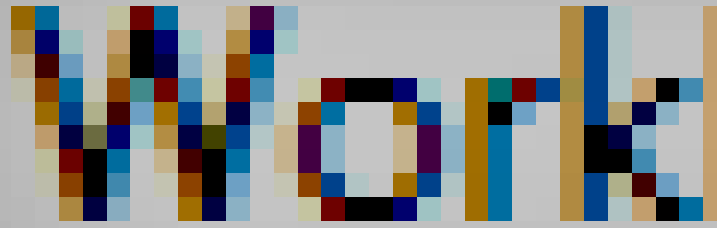
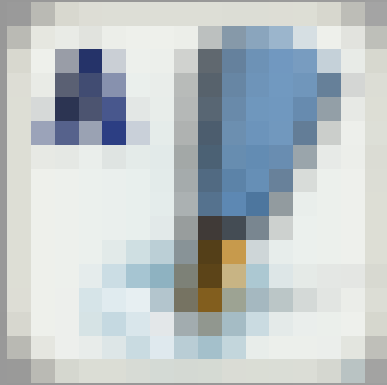


The upper table () shows information about the toolpath: the used tool, the calculation status, etc. Note that you can also select several toolpaths in the **Workzone Manager** to have all of them converted with the **Auto5** module. In this case, the upper table shows data concerning each toolpath.

7. Click on the **Choose Strategy** button



The **Strategy Selection** dialog box is displayed:



Auto 5: Strategy Selection Dialog Box

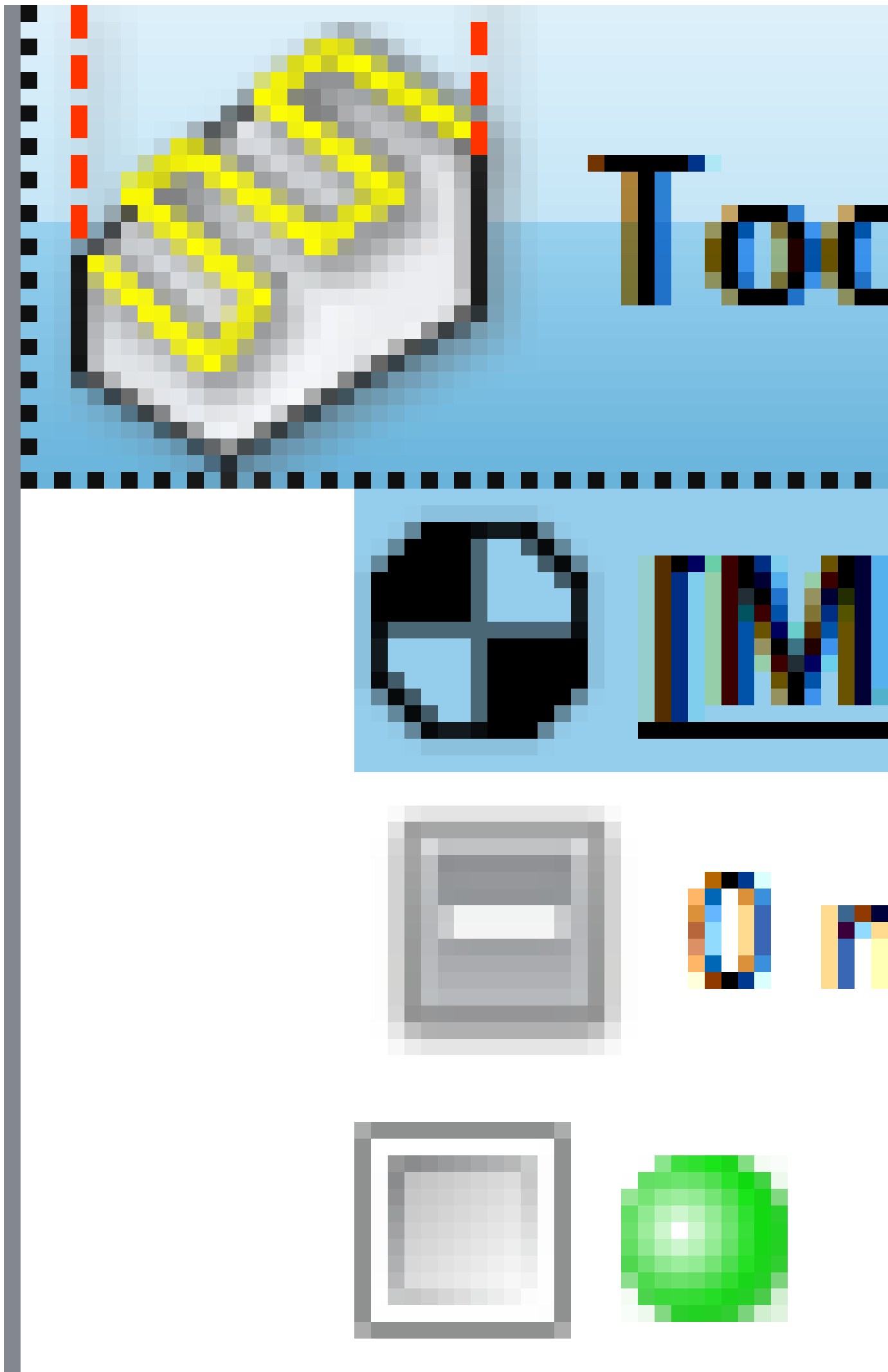
8. For this example, select the **Vertical** strategy and click **OK** in the **Auto5** dialog box.

In the **Workzone Manager**, the



icon indicates

that a conversion to 5-axis has been requested but not yet calculated:



Workzone Manager: 3 to 5 Conversion requested

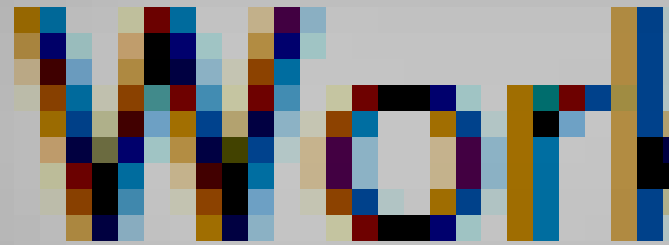
9. Run the calculations for toolpath conversion.

Once finished, you can check the result with the progressive display:

Vertical Strategy

3 - 3 to 5 Axis Machining Strategies

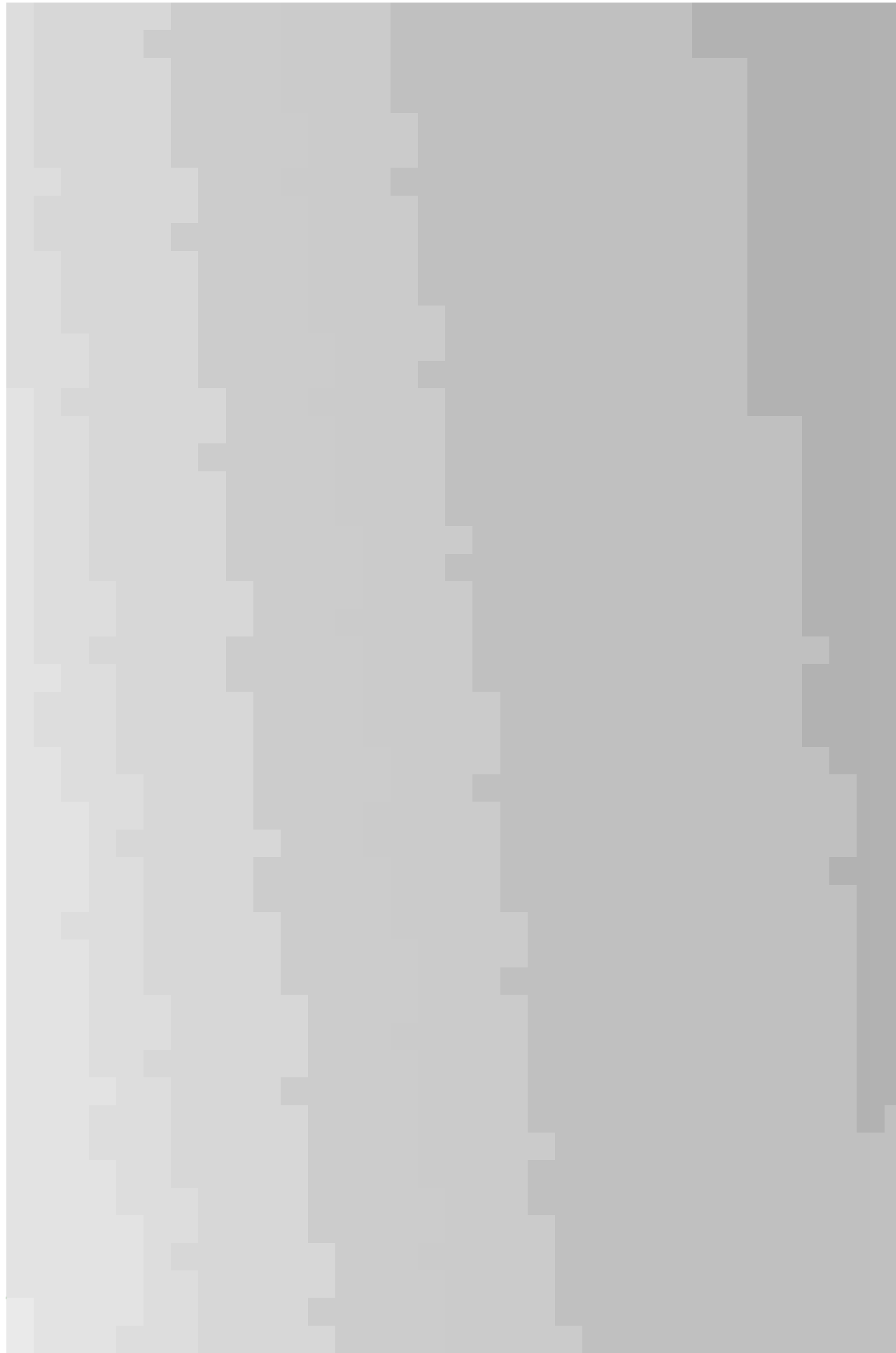
Nine different strategies are available in the 3 to 5axis module as shown in the 3 to 5 Axis Strategy Selection dialog box.



Auto 5: Strategy Selection Dialog Box

Strategy Restrictions

It is important to note that each of these strategies is only used as a recommendation during calculations. If it turns out that using this recommendation results in the tool or holder colliding with the geometry, calculations search for the closest non colliding position. If such a position cannot be found the point is marked as a colliding point. The user can see these indications with a red ring displayed around the toolpath point in the **Viewing Area** as shown below.



Toolpath with Points (red circles) where Collisions could not be avoided

Collision detection and avoidance is a complex and time consuming process which has to be done for every point of the toolpath. When correctly prepared, the whole calculation process can be sped up by choosing the optimal strategy and the optimal values for the applicable parameters.

For example, if machining along a vertical wall and the initial recommendation is to start with a vertical axis, each point would have a collision and several collision avoidance tests would need to be done for each of these points.

By directly starting with an angled axis the process could avoid this and calculation times would be significantly reduced.

3.1 - Vertical Strategy

Introduction

This strategy implies that the tool will remain vertical with respect to the Z axis of the View (or the Z axis of the milling machine if no View has been defined) whenever possible. Any deviations from the vertical position are made only where collisions may occur.

1.2 - Auto5

3 - 3 to 5 Axis

XE "3 to 5 Axis Machining Strategies"

3 - 3 to 5 Axis

XE "Auto5"

1.2 - Auto5

XE "3 to 5 Axis Machining Strategies"

3 - 3 to 5 Axis

XE "Auto5"

1.2 - Auto5

XE "3 to 5 Axis Machining Strategies"

3 - 3 to 5 Axis

XE "Auto5"

1.2 - Auto5

XE "3 to 5 Axis Machining Strategies"

3 - 3 to 5 Axis

XE "Auto5"

1.2 - Auto5

XE "3 to 5 Axis Machining Strategies"

3 - 3 to 5 Axis

XE "Auto5"

1.2 - Auto5

XE "3 to 5 Axis Machining Strategies"

3 - 3 to 5 Axis

XE "Auto5"

1.2 - Auto5

The cutter axis is inclined at (1) because a vertical wall is machined.

Use

This strategy is typically used on parts where there are few potential holder collision areas and where a large majority of the surfaces can be machined with a vertically positioned cutter.

Example

If you complete a progressive display of the **Planar Finishing** toolpath we converted earlier, you can note that the tool is kept vertical along the Z axis of the view, except when machining vertical walls:

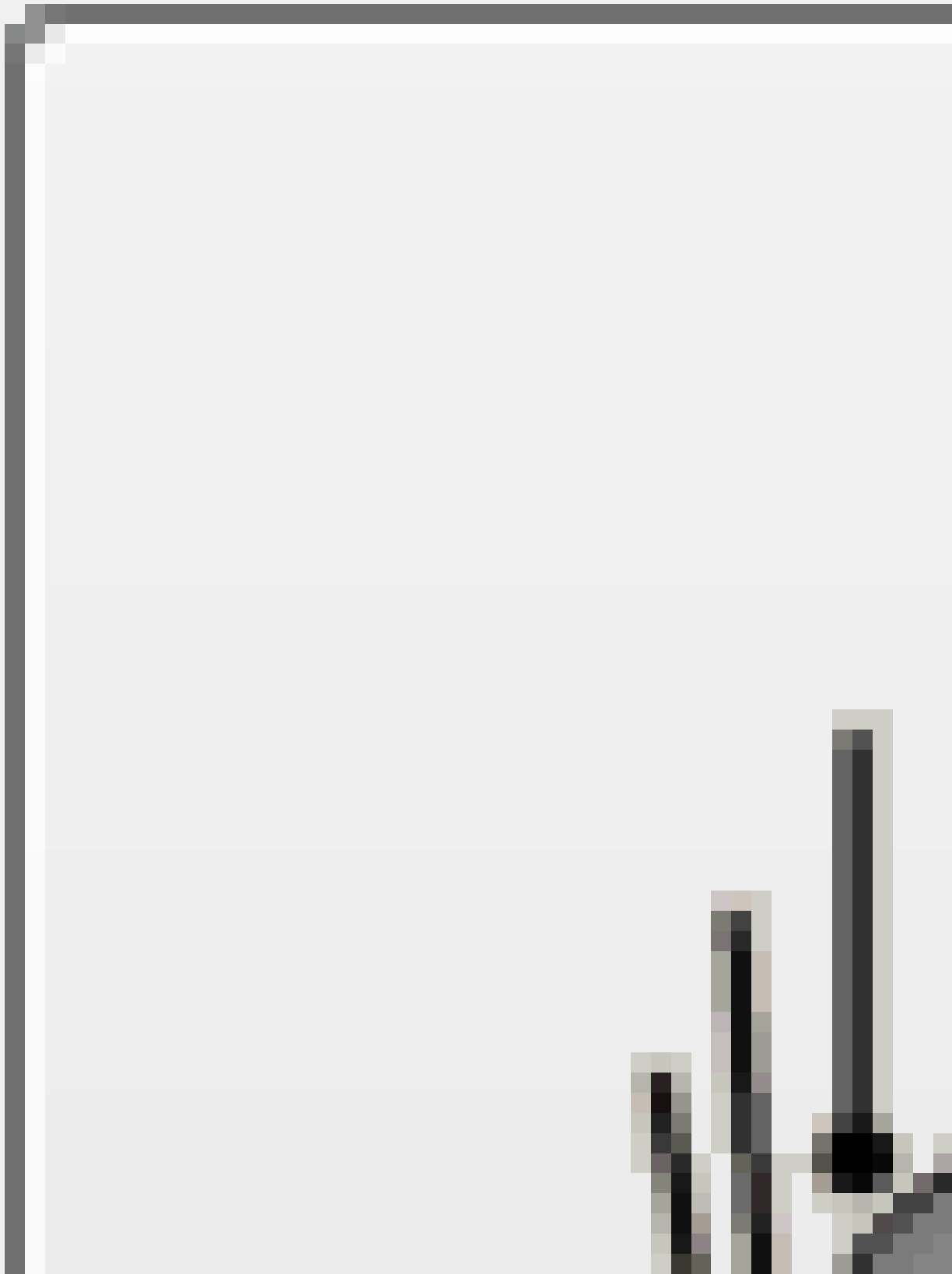
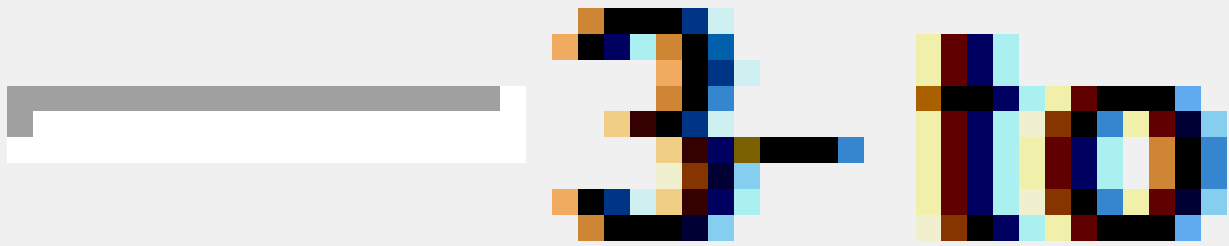
Auto5: Vertical Strategy

1. Make a copy of the **Planar Finishing** toolpath we converted earlier.
2. Select the new toolpath and click on the



icon.

3. Since the toolpath is a copy of a previously converted toolpath, click on the **Recalculate (A-)** button to activate the **Vertical Strategy** parameters.



Auto5: Recalculate Button

You can make the **Vertical** strategy results vary according to three basic parameters: the **Maximum Allowed Angle**, the **Lead Angle** and the **Maximum Angle Difference**.

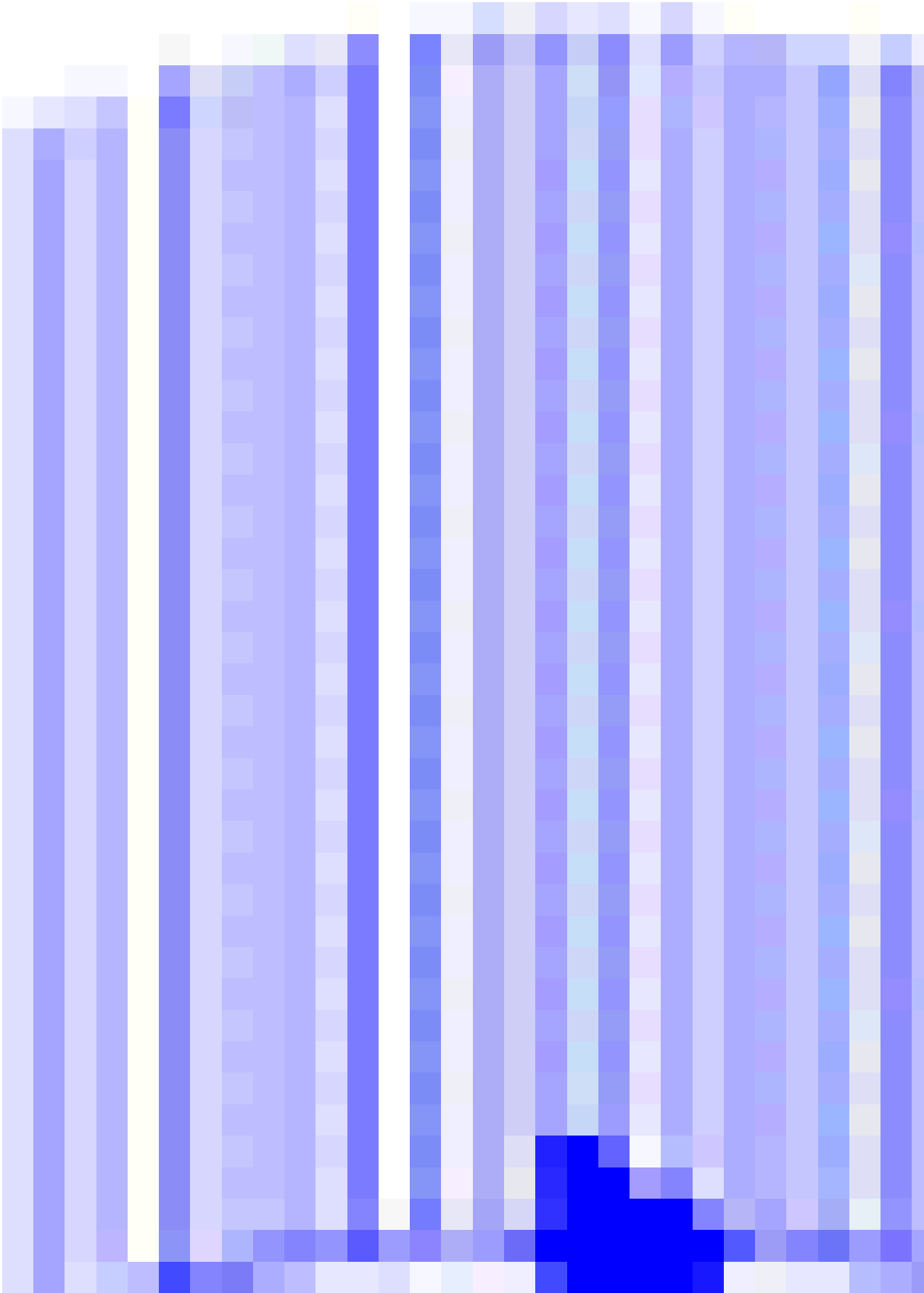
Maximum Machining Context Angle

Any angle which is obtained during the **3 to 5 Axis** calculation process is limited by the **Maximum Machining Context Angle (A)**. As a result the 5-axis toolpath will not have any axis having an angle greater than the **Maximum Machining Context Angle (A)**. All other angles must be smaller than A. If a collision could only be avoided by taking a bigger angle the corresponding point will be marked and removed by **5 to Machine**.

Lead Angle

Choosing a positive lead angle implies that the tool axis is inclined in the direction of the tool movement for the complete toolpath. This is to obtain optimal cutting conditions and extend tool life.

The following examples illustrate the use of this parameter.



Identical toolpaths with Lead Angles of 0° and 15°

Maximum Angle Difference

Allows toolpath points to be added to ensure that no two consecutive points have an axis angle which exceed this value (in degrees).

Applies to both **3 to 5 Axis** and **5 to Machine** modules.

Note that these three parameters apply to all 3-axis to 5-axis strategies.

1. For our example, we suggest that you set the lead angle to 15° then start the conversion by clicking **OK**.
2. Check the difference using the progressive display.

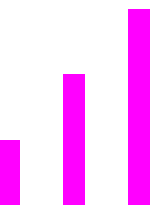
Auto5: Vertical Strategy with a 15° Lead Angle

Vertical Axis Direction

You can orientate the tool according to the Z axis of a view. To do so, activate the **From View** option and select the View from the corresponding drop-down list.

- Open the *Pumpe_auto5* workzone and compare toolpaths #2 and #3.

Both are copies of a **Planar Finishing** toolpath (#1), which have been converted to 5-axis toolpaths using the **Vertical** strategy. In toolpath #2, the axis direction is given by the Z axis of the initial toolpath.



Vertical Strategy: From Toolpath Option

In toolpath #3, the axis direction is given by the Z axis of an inclined view.

Vertical Strategy: From View Option

See also...

- "Converting a 3-Axis Toolpath" on page 6

3.2 - Constant to Axis Strategy

Introduction

This strategy implies that the cutter will maintain a constant angle with respect to the Z axis of the View or the machine if no View has been defined.

Use

This strategy is typically used for **Z-Level Finishing** toolpaths and for machines where the A or B axis can remain constant in the majority of the zone to be machined.

Example

1. Open the *Auto5_examples* workzone.
2. Copy the first **Planar Finishing** toolpath.

We are now going to convert this toolpath into a 5-axis toolpath using the **Constant to Axis** strategy.

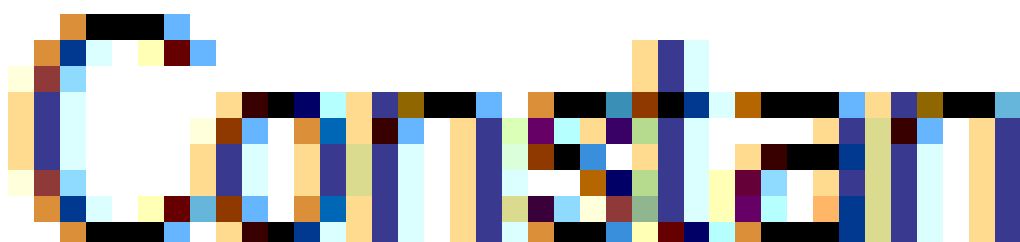
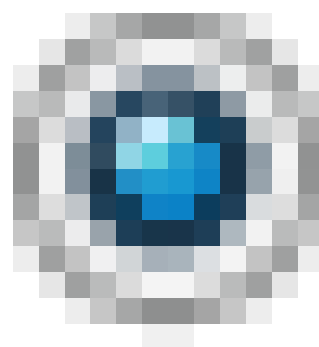
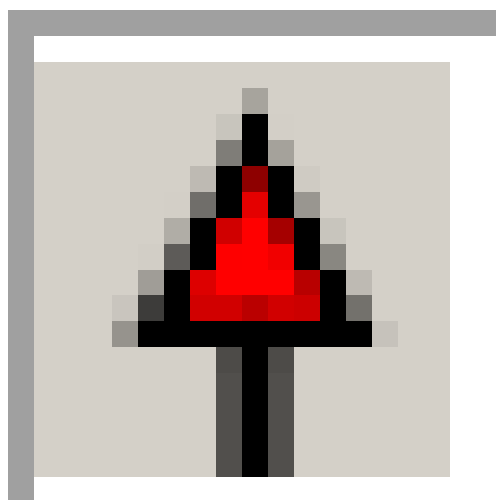
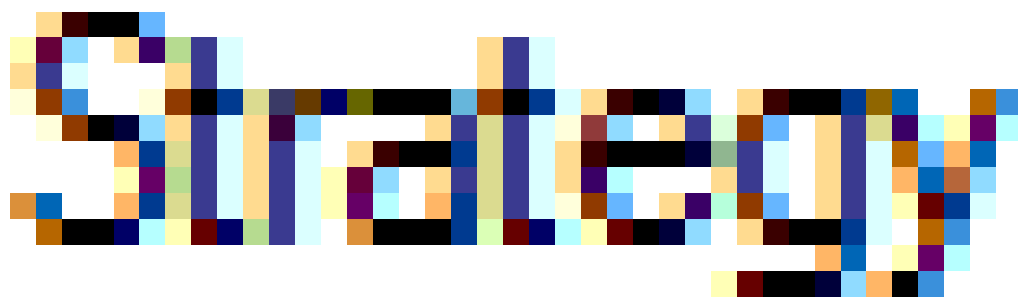
3. Click on the



icon.

4. Click on the **Choose Strategy** button and select **Constant to Axis** strategy.

The number of parameters available for this strategy is higher:



Constant to Axis Strategy Interface

Reference Axis System

Allows you to define the Reference Axis System which is either the **View Axis** or the **Machine Axis**. All angle values will be referenced and measured with respect to the selected Axis.

1. Enter a 25° constant angle.
2. Make sure that the **Force Constant Angle** and **Keep Axis for Horizontal Regions** options are deactivated.
3. Convert the toolpath by clicking **OK** and running the calculation.

Constant Angle: 25°

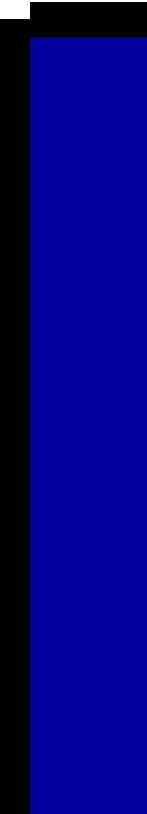
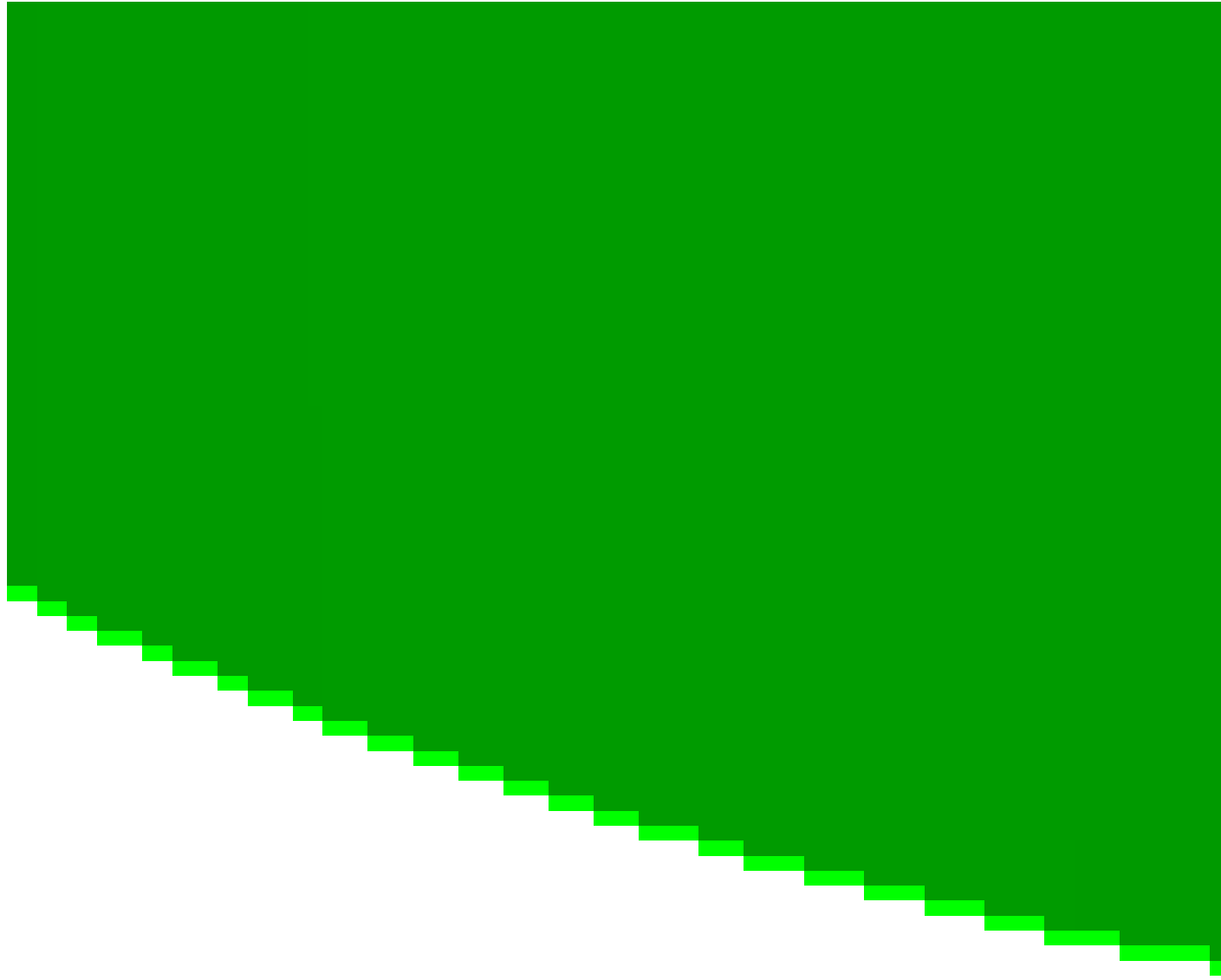
Force Constant Angle (4-Axis Machining)

If the constant condition cannot be maintained, **Auto5** tries to avoid the collision by keeping a constant A/B angle. If such a position cannot be found the point is marked as colliding. The resulting toolpath always maintains a constant A/B angle.

Keep Axis for Horizontal Regions

1. Make a copy of the **Planar Finishing** toolpath.
2. Open the **Auto5** module and click on the **Recalculate** button.
3. Activate the **Keep Axis for Horizontal Regions** option.
4. Start the toolpath conversion and check the result.

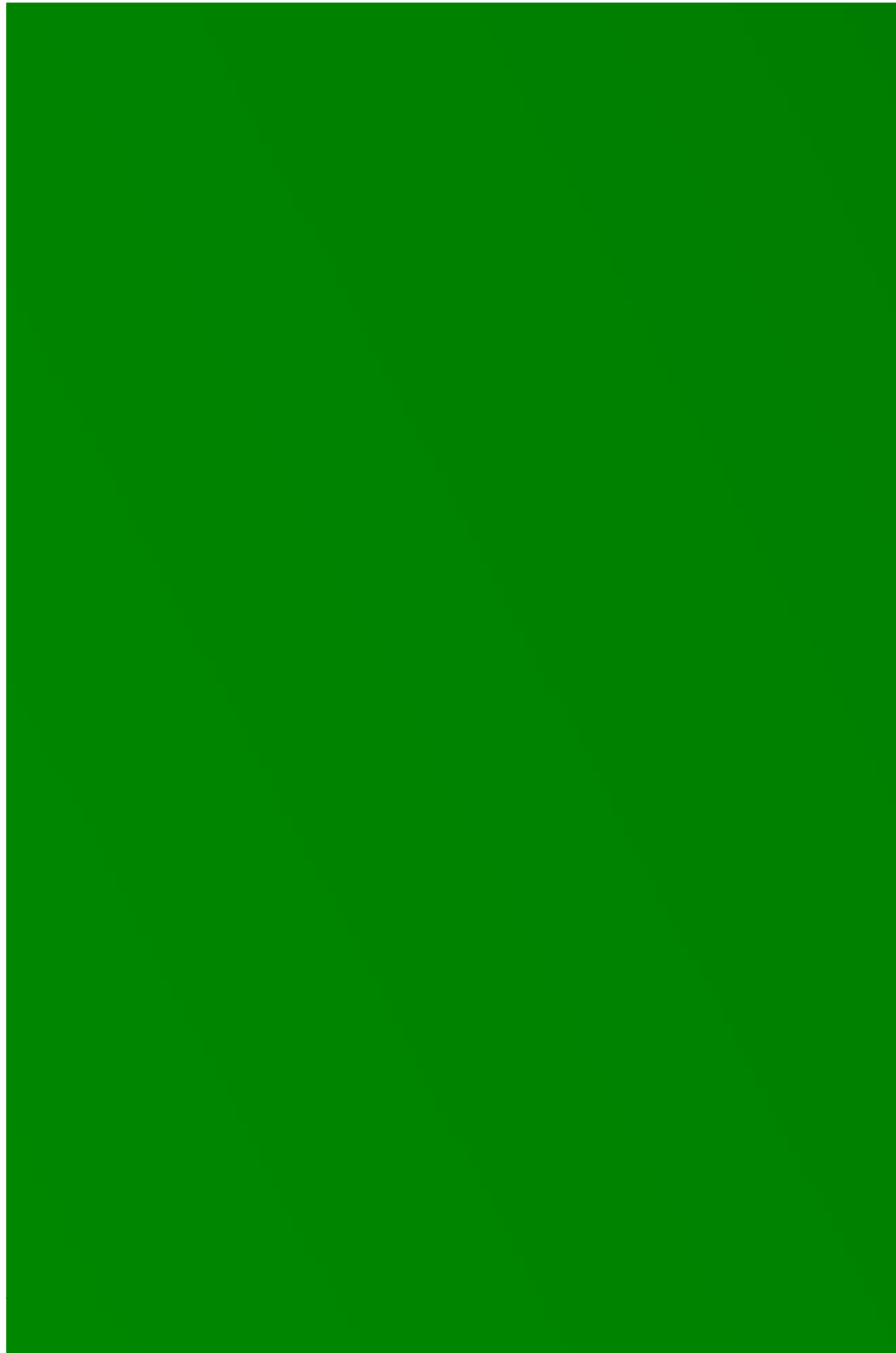
If this option is deactivated, the cutter will be positioned along the reference axis in horizontal regions as shown below.



Keep Reference Axis

The toolpath illustrated above is the first of the **Planar Finishing** toolpath you have made. The **Keep Axis for Horizontal Regions** option is deactivated.

If this option is activated, the cutter will be positioned according to the defined Constant Angle with respect to the reference axis. The orientation of the cutter with respect to the reference axis is determined by the orientation of the cutter before reaching the horizontal zone, as illustrated below.



Keep Axis for Horizontal Regions

The toolpath illustrated above is the second of the **Planar Finishing** toolpath you have made. The **Keep Axis for Horizontal Regions** option is activated.

3.3 - No Change Strategy

Introduction

This strategy implies that **WorkNC** will attempt to maintain the same cutter axis as the previous point along the toolpath whenever possible. On convex areas it will follow the normal to surface position and on concave areas it will maintain the same axis as the previous point.

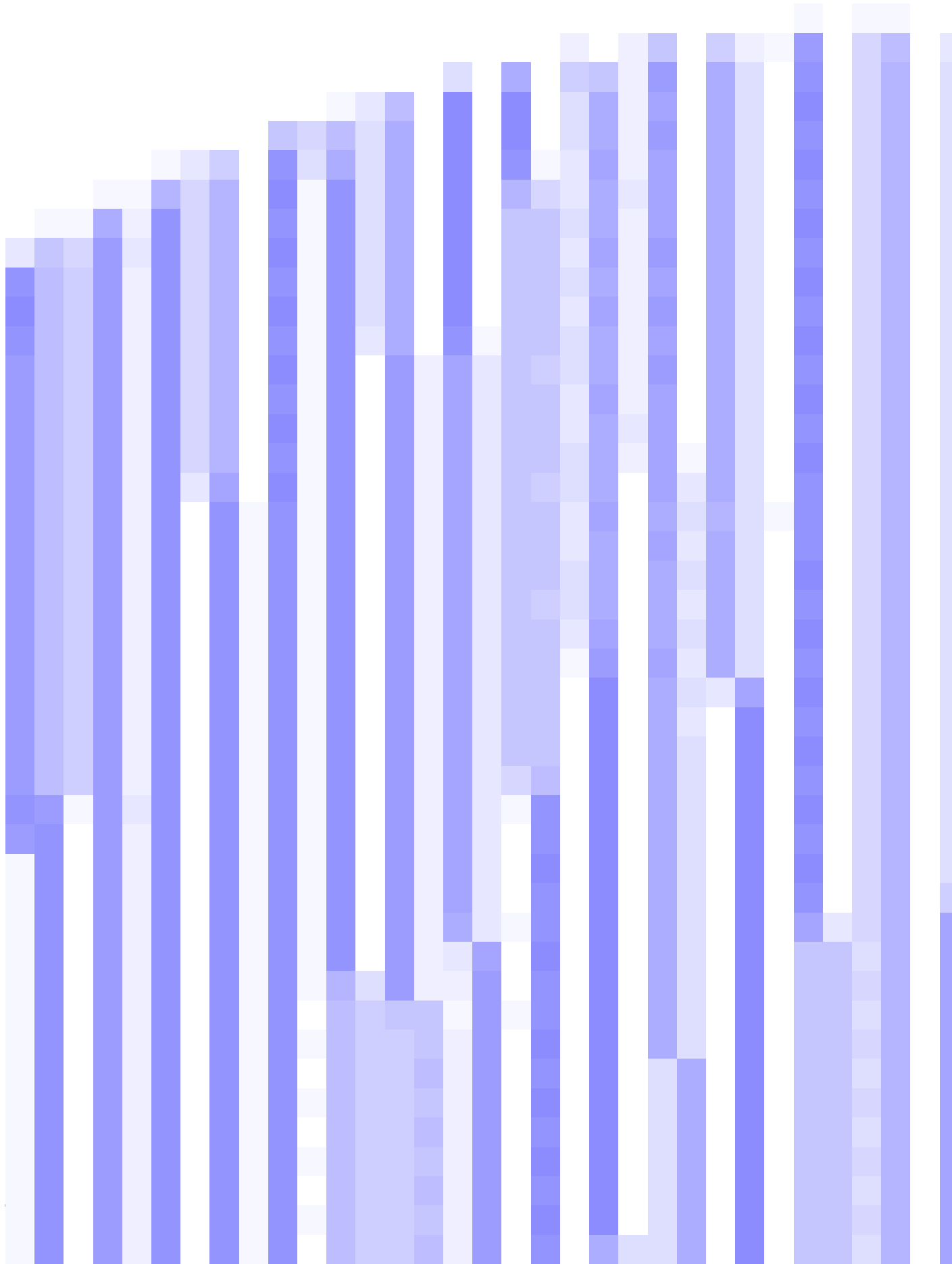
Use

This strategy is typically used for remachining parts and concave areas.

Example

1. Make a copy of the last **Planar Finishing** toolpath you calculated earlier in the *Auto5_initial*/workzone.
2. Start the **Auto5** module and click on the **Recalculate** button
3. Select the **No Change** strategy and start the conversion.

Some sections of your toolpath should be similar to the following:



1 Normal to Surface on convex area

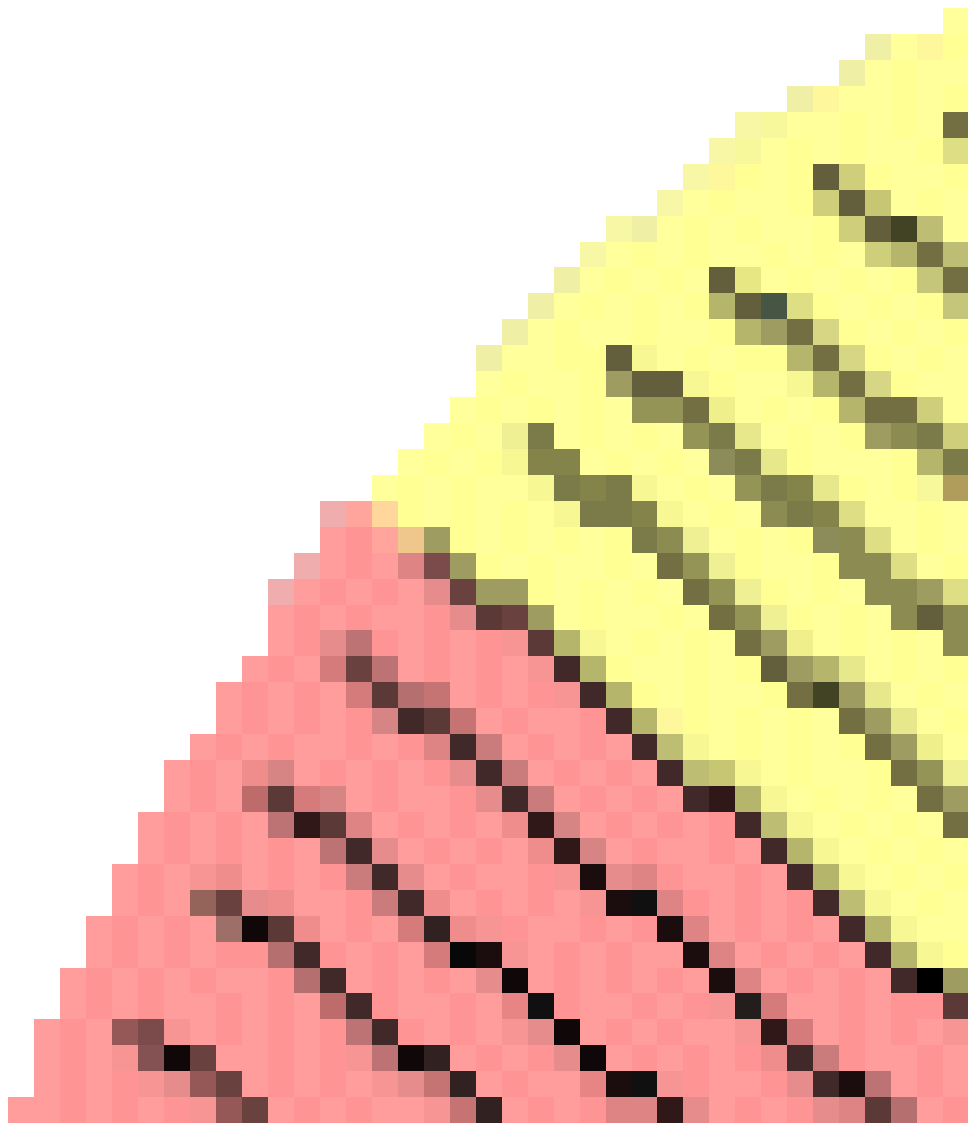
2 Same axis maintained on other areas

The **No Change** strategy may also be influenced by several parameters.

Maximal Vertical Angle

If the surface normal vector of a given toolpath point has an angle which is smaller than the maximum vertical angle V then the cutter axis is set to vertical.

The following two diagrams illustrate the use of this parameter.



Maximum Vertical Angle V of 10° and 30°

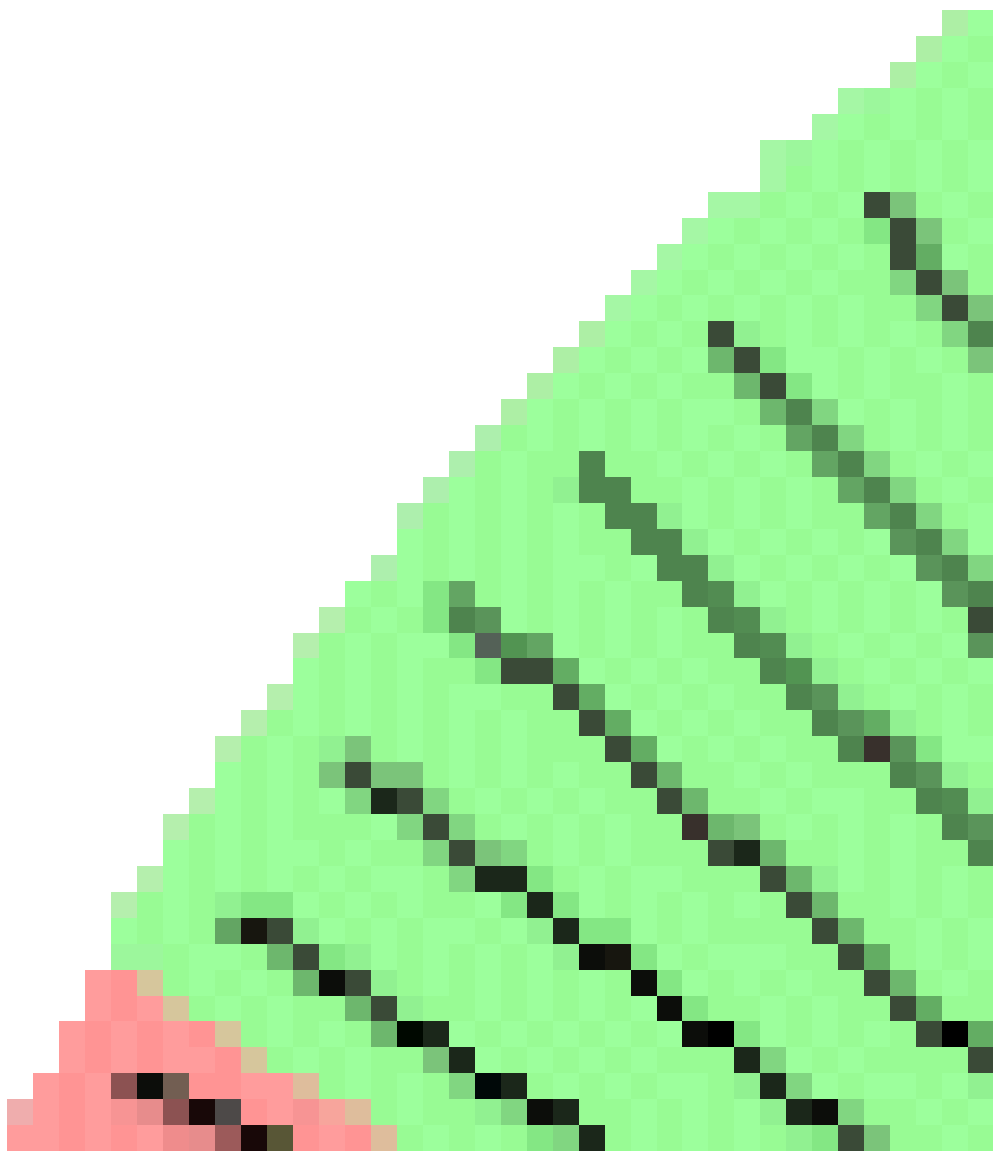
Minimum Normal Angle (M)

If the surface normal vector of a given toolpath point has an angle which is smaller than the Minimum Normal Angle **M** but bigger than the Maximum Vertical Angle **V** then the axis is set to **M**.

Maximum Normal Angle (N)

If the surface normal vector of a given toolpath point has an angle which is smaller than the Maximum Normal Angle **N** but bigger than the Minimum Normal Angle **M** then the axis is set to the normal vector. If the angle is bigger than **N** the axis is set to **N**.

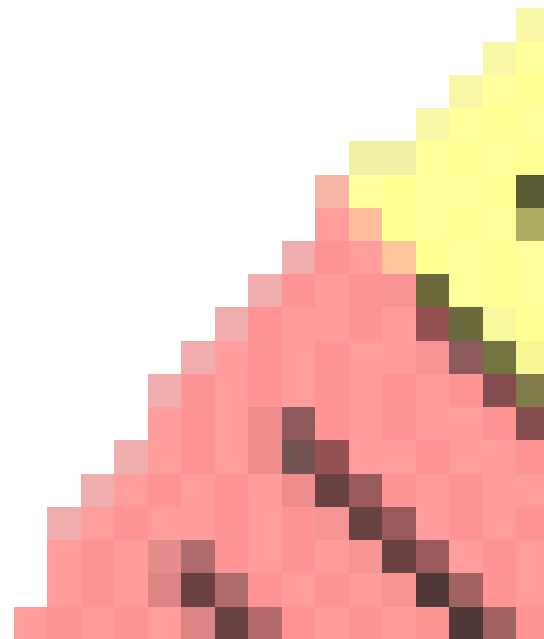
The following two diagrams illustrate the use of Minimum and Maximum Normal Angles (M + N).



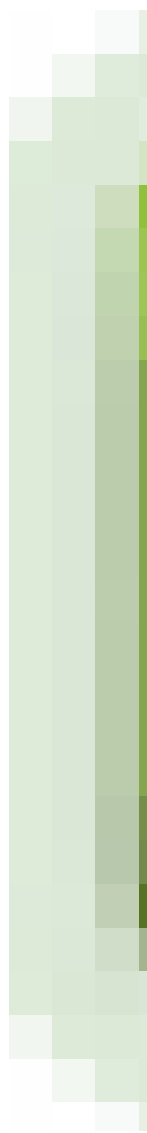
Minimum Normal Angle (M) of 20° and 40°. Maximum Normal Angle (N) has a value of 60° in both cases.

If both Minimum (M) and Maximum (N) Normal Angles are defined with the same values you can obtain a constant angle for all surface normals. The only exception is for plane regions having a vertical normal. Here the vertical axis is kept. This is exactly how the Constant to Axis strategy works.

The following diagram illustrates this point.



Minimum (M) and Maximum (N) Normal Angles both set to 45° to obtain a constant axis angle



Vertical Strategy

Setting $V = M = N = 0$ obtains the **Vertical** strategy.

- In the *Auto5_initial/workzone*, make a copy of the toolpath that you have just calculated and change the maximal vertical angle (e.g. 30°) as well as the minimum and maximum normal angles (e.g. 40° and 50°).

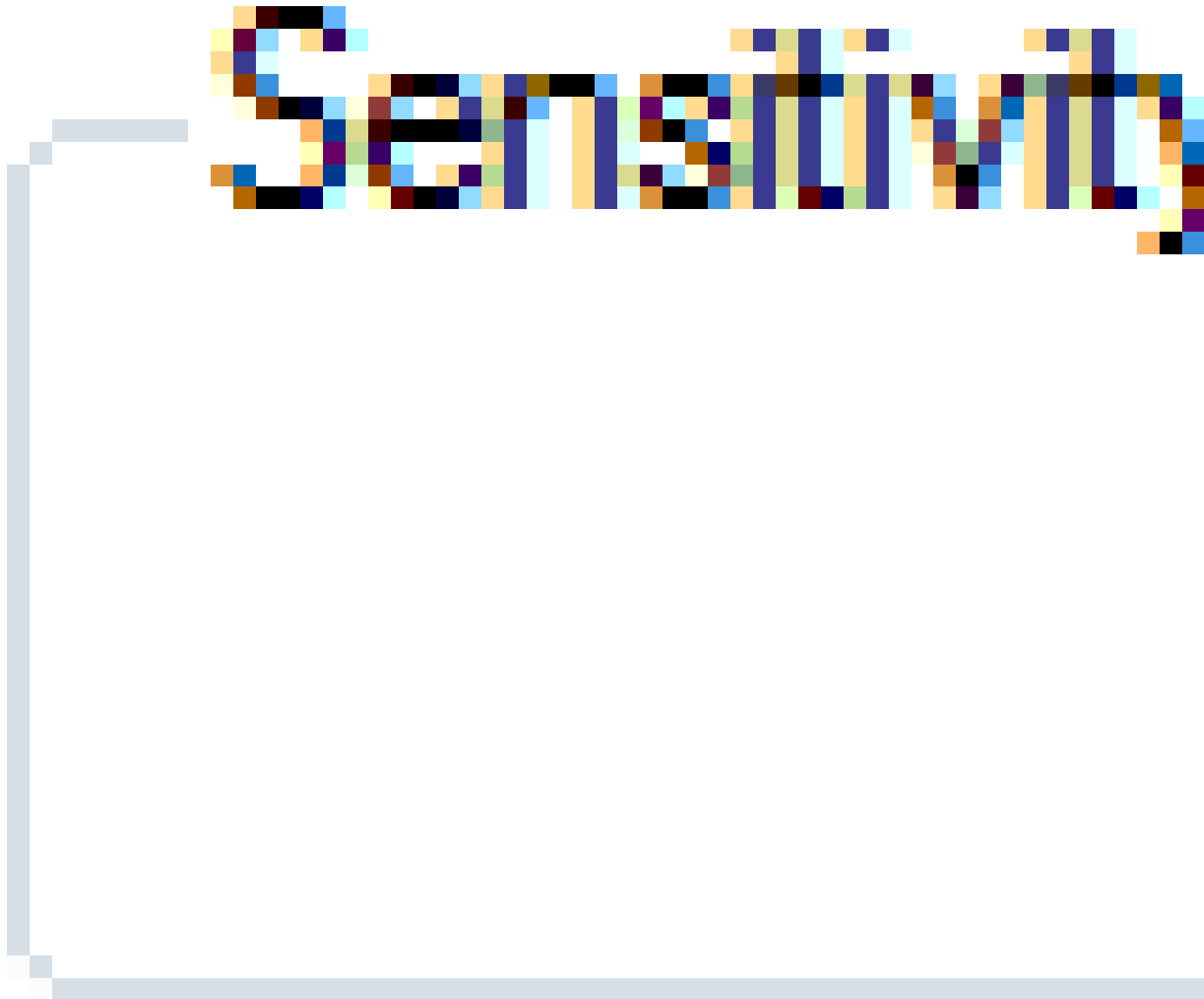
Data Entry

We recommend that you enter the **Maximum Normal Angle** first, followed by the **Minium Normal Angle** and finally the **Maximum Vertical Angle**. If you do not respect this sequence order, you have an error message due to inconsistent data.

- Compare results with the prior toolpath.

In the **Auto5** module, you can click on the **Additional** tab to access the **Sensitivity of convex movements** parameter.

Sensitivity of Convex Movements



The **No Change** strategy implies that the current toolpath point cutter axis orientation depends on the orientation of the cutter on the previous point.

This parameter is available in the **Additional** tab for the **No Change** strategy.

For longer convex movements it follows the normal of the surface and for concave movements the previous axis is chosen. How sensitive a convex movement is will be influenced by the **Sensitivity of Convex Movements** parameter. It ranges from **Low (lazy)** to **High (sensitive)**.

Example

The following table indicate the influence of this parameter on a **Z-Level Finishing** toolpath.

Depending on the Sensitivity value, we can evaluate the number of constant axis movements.

Sensitivity Value	Low	Medium	High
No. of Constant Axes	47.3%	41.0 %	31.5%

The following examples show a section of one level of a **Z-Level Finishing** toolpath with the different sensitivity values.



Example of High Convex Movement Sensitivity



Example of Medium Convex Movement Sensitivity



Example of Low Convex Movement Sensitivity3.4 - Normal to Surface Strategy

Introduction

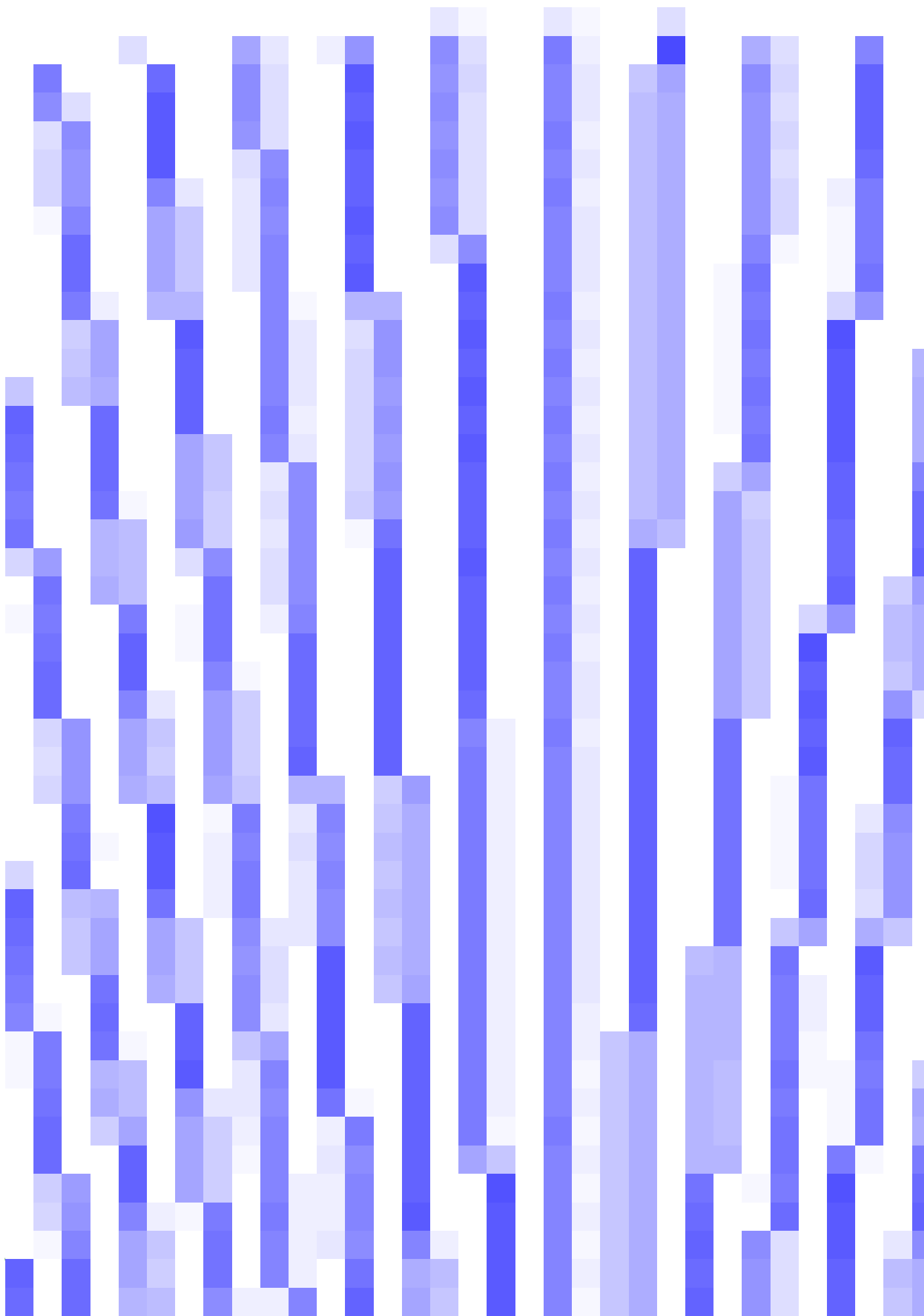
This strategy implies that the cutter is positioned along a vector which is normal to the surface at each point along the toolpath. The angle of the cutter can be varied from the normal to surface position thanks to a variety of parameters which makes this strategy very powerful and versatile.

Use

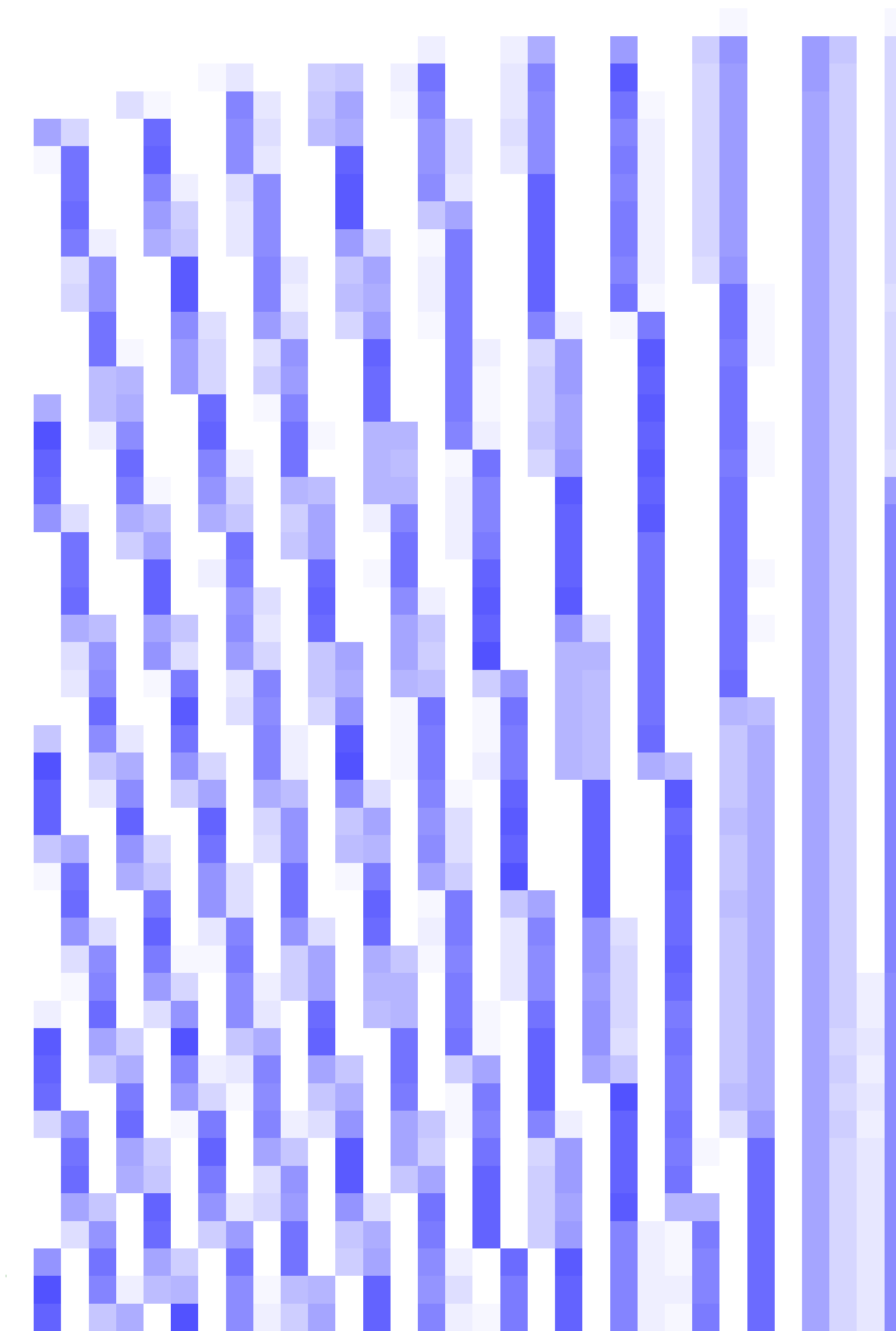
This strategy is typically used on parts with steep or vertical walls (e.g. **Z-Level Finishing**) by adapting the strategy so that the A/B angle remains as constant as possible. For toolpaths used on shallow convex or concave parts (e.g. **Planar Finishing**) the strategy can use pure normals.

Examples

The following example shows a toolpath with pure normals which, due to the shape of the part, results in many tool axis movements.

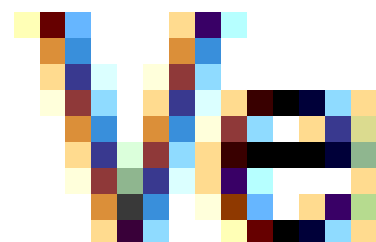
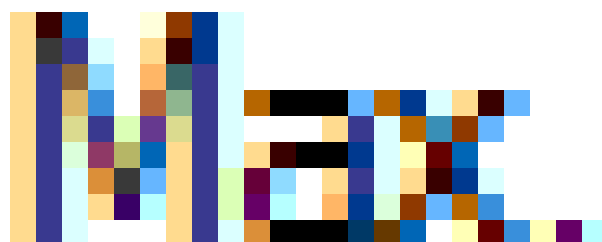
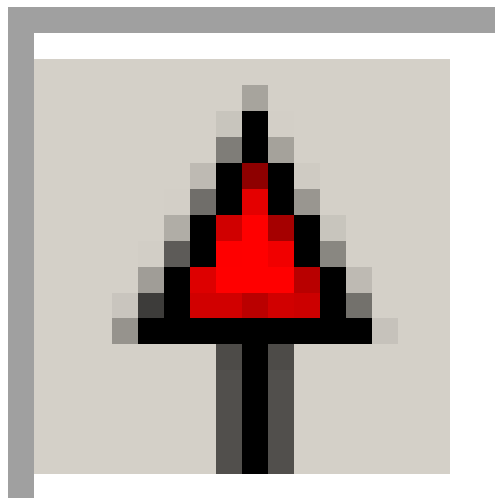
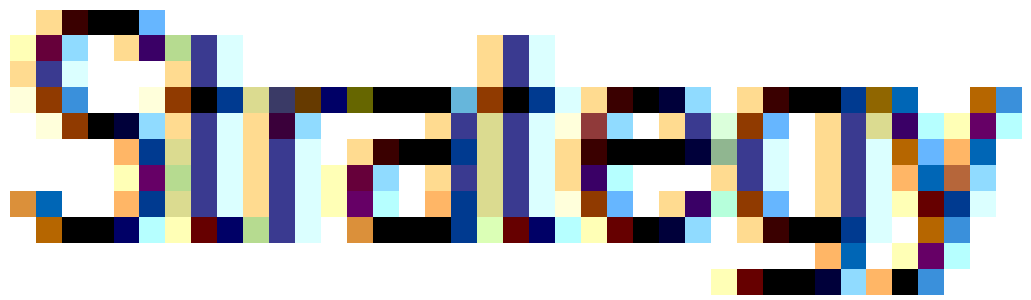


The following example shows the same part but this time the tool axis movement is restricted to a maximum angle of 45°.



Lateral Angle

The **No Change** and **Normal to Surface** strategies have an additional parameter, the **Lateral Angle**, which allows you to incline the tool to the right or the left against the machining direction.



Normal to Surface with Lateral Angle

In the example above, we have defined a 35-degrees lateral angle, which will incline the tool 35 degrees to the right against the machining direction:

- Positive values in the **Lateral Angle** field make the tool incline to the right.
- Negative values in the **Lateral Angle** field make the tool incline to the left.
- To illustrate this parameter, open the *Auto5_examples* workzone and compare toolpaths #3 and #4.

Both are copies of toolpath #2 (**On-Curve Engraving**) that have been transformed by **Auto5**. Toolpath #3 has no lateral angle whereas toolpath #4 has a 35-degrees lateral angle.

Normal to Surface

Normal to Surface with Lateral Angle (35°)

Note that if the application detects collisions with the lateral angle defined, this angle is automatically corrected to avoid any collision.

1. In the *Auto5_examples* workzone, make a copy of toolpath #4 and start the **Auto5** module.
2. Click on the **Recalculate** button and enter **-60** in the **Lateral Angle** field.
3. Calculate the toolpath and simulate it in the **Progressive Display** mode.

Lateral Angle Correction to Avoid Collisions

In the example above, the toolpath starts machining with a 60 degrees inclination to the left. The inclination is then reduced to avoid collisions with the boss on the left side of the toolpath.

3.5 - Constant (Attractive) Strategy

Introduction

This strategy implies that for any point on the toolpath an axis with a constant angle with respect to the Z-axis of the View (or the machine) is determined. The direction of the axis is defined either by the shortest distance to an attraction point or an attraction curve.

If more than one attraction point is chosen a mean direction considering all points is calculated.

The attraction curve does not need to be closed and can consist of several sub-curves. The Z-height of the point or curve is not taken into consideration.

The advantage of using attraction points/curves compared to **Normal to Surface** is that the axes are always directed towards the attraction element. As a result the axes have fewer variations along the toolpath which helps to ensure machine stabilization.

Use

This strategy is typically used for **Z-Level Finishing** on cavities with steep walls and varying geometry.

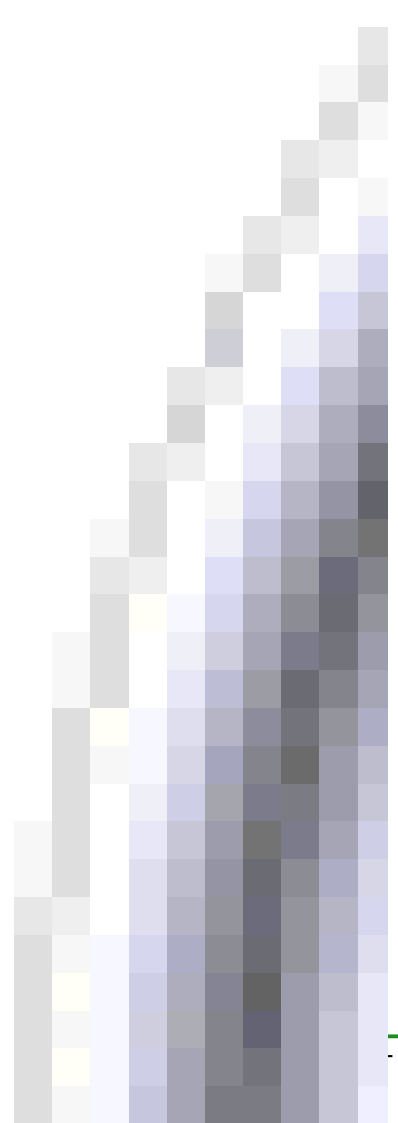
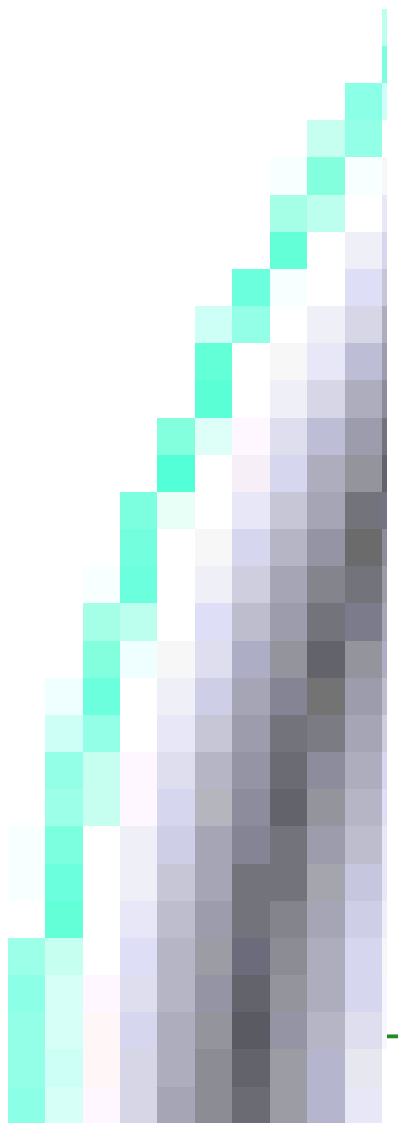
Examples

The two following examples shows hemispherical cavity machining with attraction points (1) in the center of the hemisphere and then to the left of center.

The following two examples shows cavity machining with an open and closed attraction curve (1) on cavities of a more complex shape.

You can also define more than one attraction point. This implies that for any toolpath point the closest attraction point is the most important one and has the biggest influence on its axis.

The following examples show a toolpath with three attraction points compared to a toolpath on the same area with an open curve.

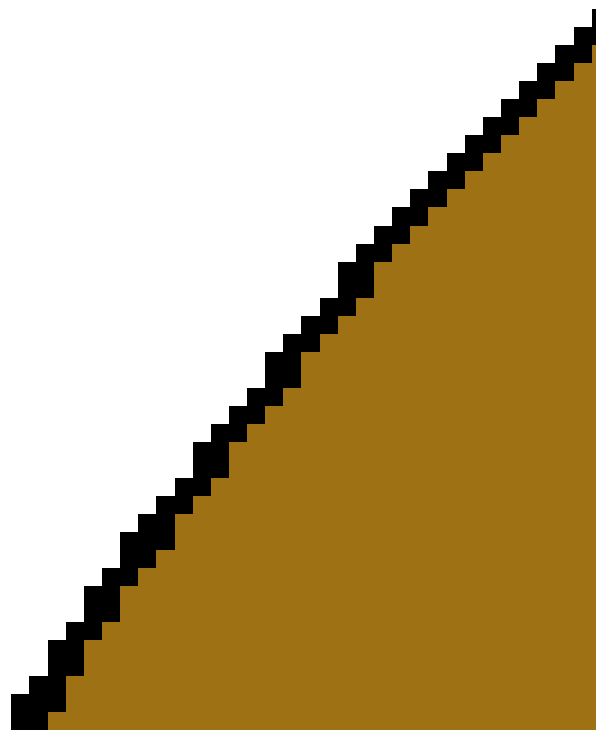


The open curve toolpath results in smaller cutter axis variations.

The following example shows the Constant (Attractive) strategy to machine the bottom of a bottle. A constant angle of 30° was defined with the axis of the attraction point in the center of the bottle.

Practice

1. Open the *Mold_Moule_Form* workzone and create two copies of the existing **Z-Level Finishing** toolpath.
2. Create a first point at the center of the part (1): we will use it for the **Constant Attractive** strategy. Create a second point outside the part (2), like in the picture below:



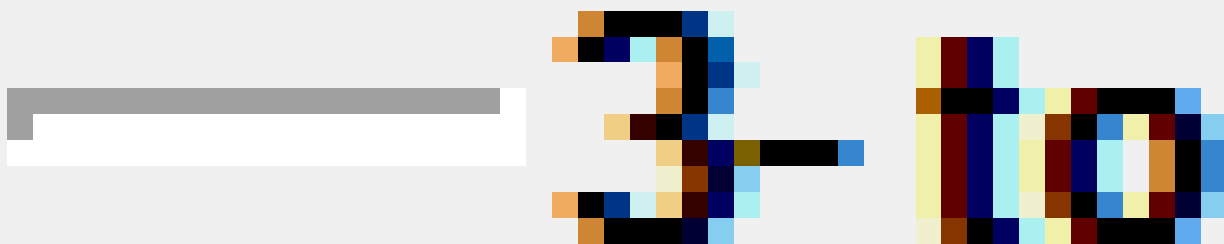
Auto5: Constant Attractive: Attraction Points

3. Select the first copy of the initial toolpath and click on the



icon.

4. Click on the **Recalculate** button.
5. Select the **Constant Attractive** strategy, define a 25° constant angle and select the center point as the attraction point:



Auto5: Constant Attractive: Center Point

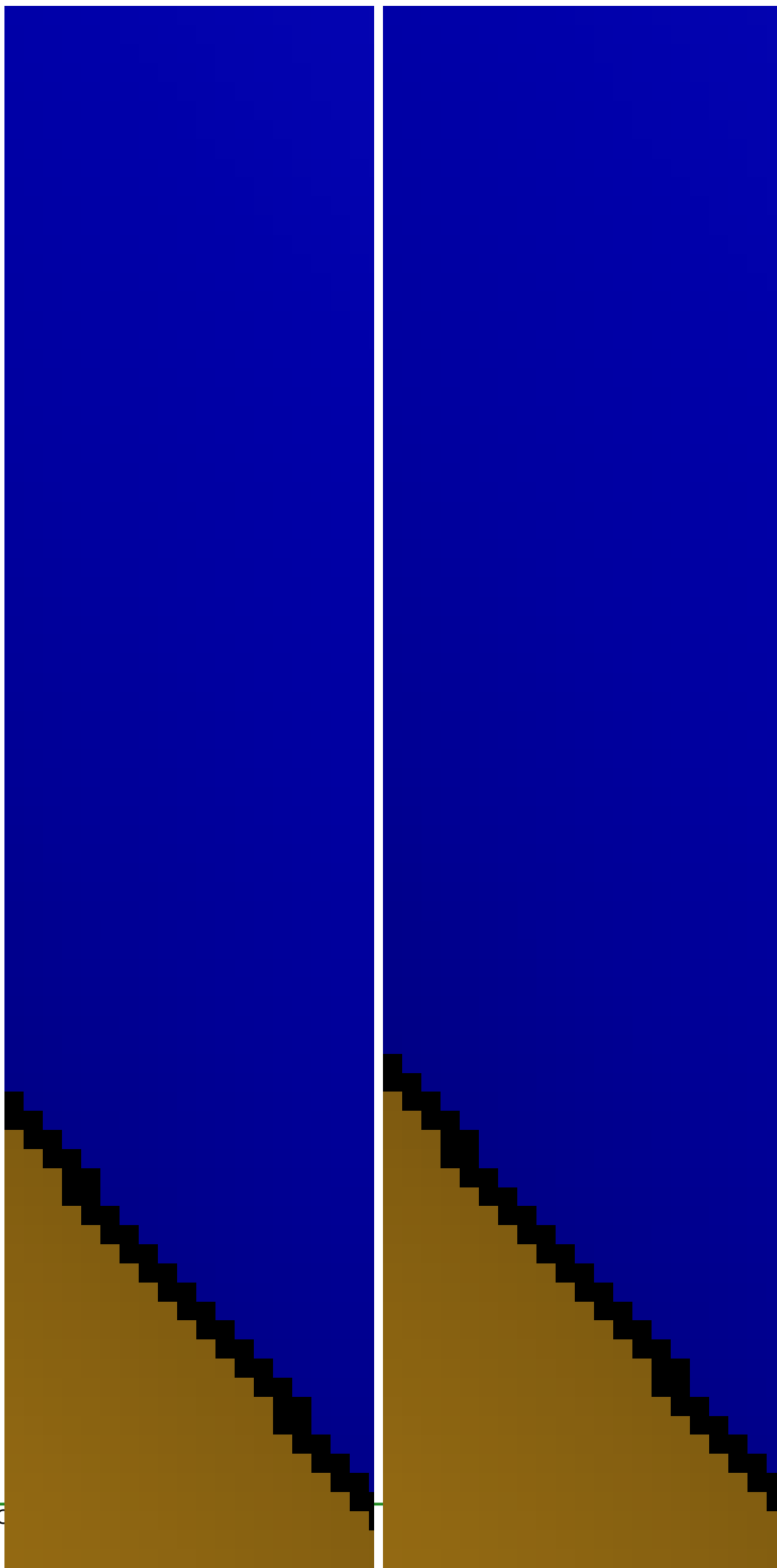
6. Validate the parameters and run the toolpath calculations.
7. Select the second copy of the initial toolpath and and click on the



icon.

8. Click on the **Recalculate** button.
9. Select the **Constant Attractive** strategy, define a 25° constant angle but this time, select the outer point as the attraction point.
10. Validate the parameters and run the toolpath calculations.

If you now compare both toolpaths you should obtain the following results:



The **Constant (Attractive)** strategy interface displays an additional parameter:

Force Constant Angle (4-Axis Machining)

If the constant condition cannot be maintained, **Auto5** tries to avoid the collision by keeping a constant A/B angle. If such a position cannot be found the point is marked as colliding. The resulting toolpath always maintains a constant A/B angle.

3.6 - Constant (Repulsive) Strategy

Introduction

This strategy works in the same way as **Constant (Attractive)** except that the opposite angle direction is used. Cutter axes are no longer attracted by a point or curve but are repulsed by them.

For any point of the toolpath, an axis with a constant angle to the Z-axis of the View (or the machine) is determined. The direction of the axis is defined either by the shortest distance to a repulsion point or a repulsion curve.

If more than one repulsion point is chosen, a mean direction considering all points is calculated.

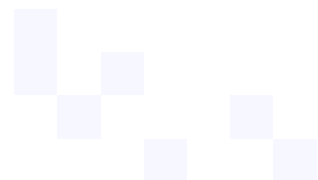
The repulsive curve does not need to be closed and can consist of several sub-curves. The Z-height of the point or curve is not taken into consideration.

Use

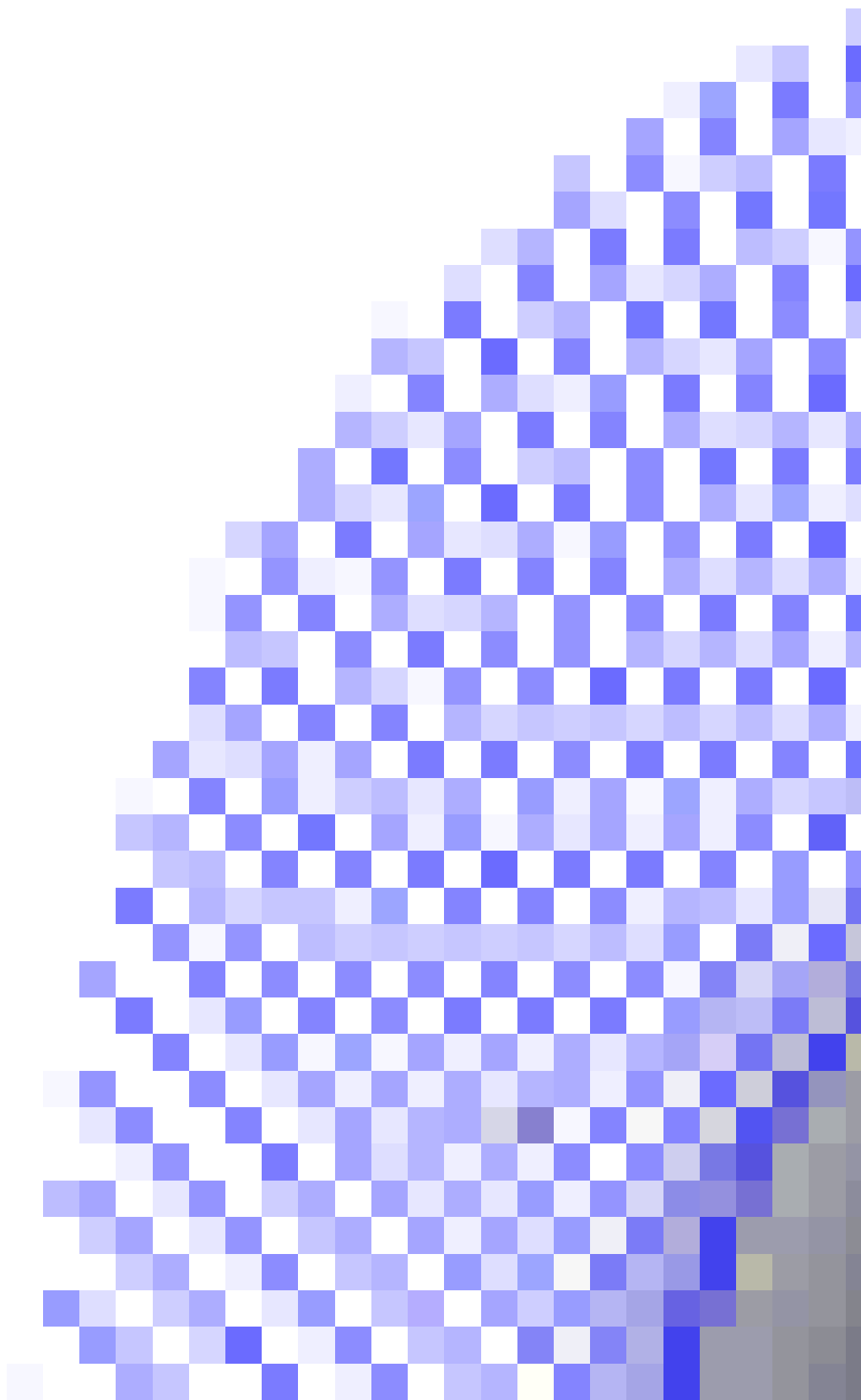
This strategy is typically used for **Z-Level Finishing** on cores and die parts with steep walls and varying geometry.

Examples

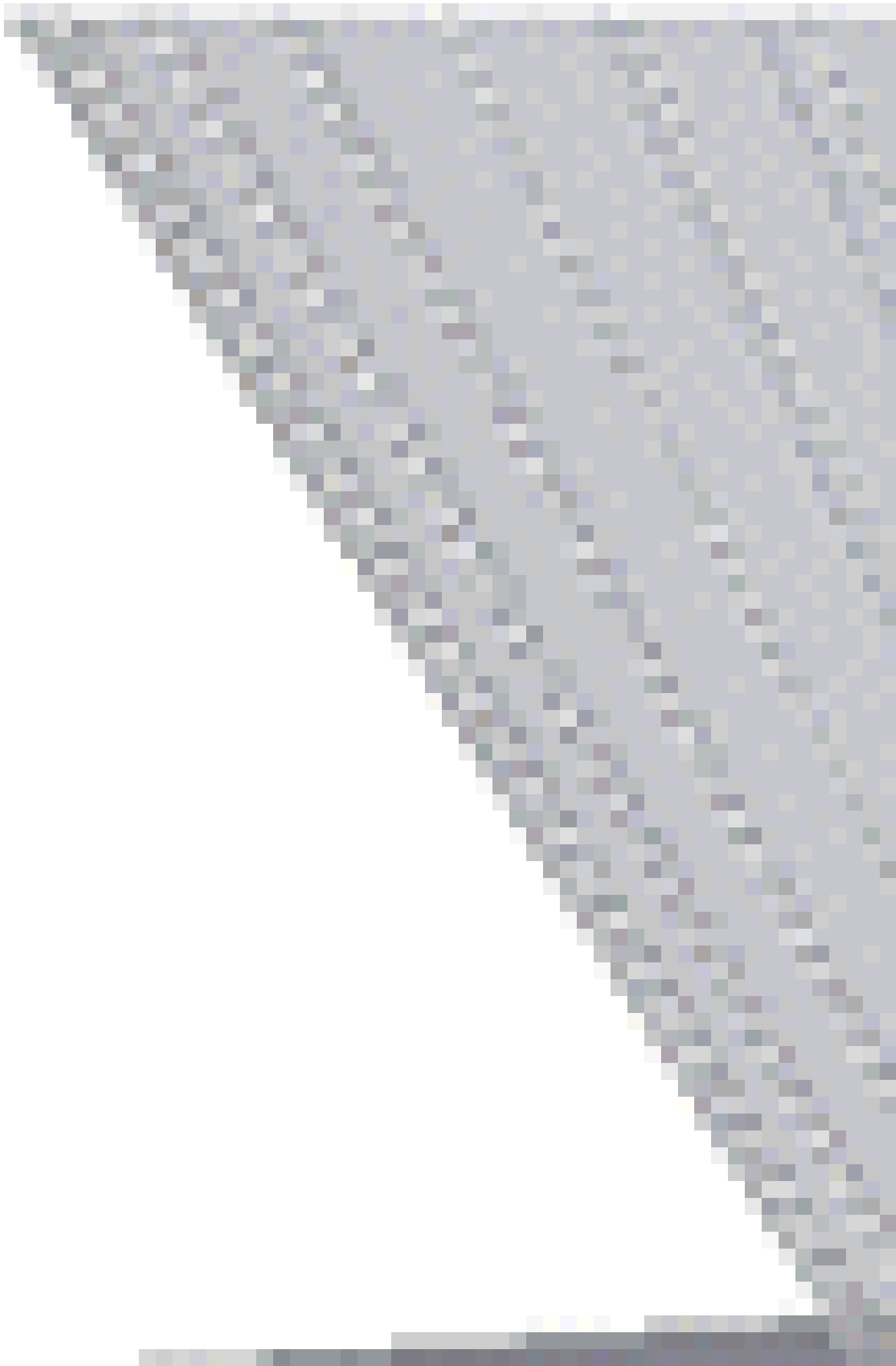
The two following examples shows hemi elliptical core machining with a repulsion point (1) in the center and a closed curve (1).



The next example shows the effect of a straight open repulsion curve (1) on a core area of greater complexity.



The following example shows the machining of a hemisphere by using a single repulsion point with a constant angle of 20°.



The next example shows the toolpath for an inclined cone on a plane. A single repulsion point at the top of the cone was chosen by using a constant angle of 20° . The reference axis is the Z-axis of the machine so that the A/B angle remains constant.

3.7 - Guide (Attractive) Strategy

Introduction

This strategy implies that for any point on the toolpath an axis in the direction of the guide point/curve is determined. The direction of the axis is defined by the shortest distance to an attraction point or an attraction curve.

The guide curve does not need to be closed and can consist of several sub-curves.

The attraction point or curve must be positioned at the correct distance from the part. If it is too close to the part, axis variations may be too important. If it is too far from the part the machining will appear more like a constant attractive strategy.

Use

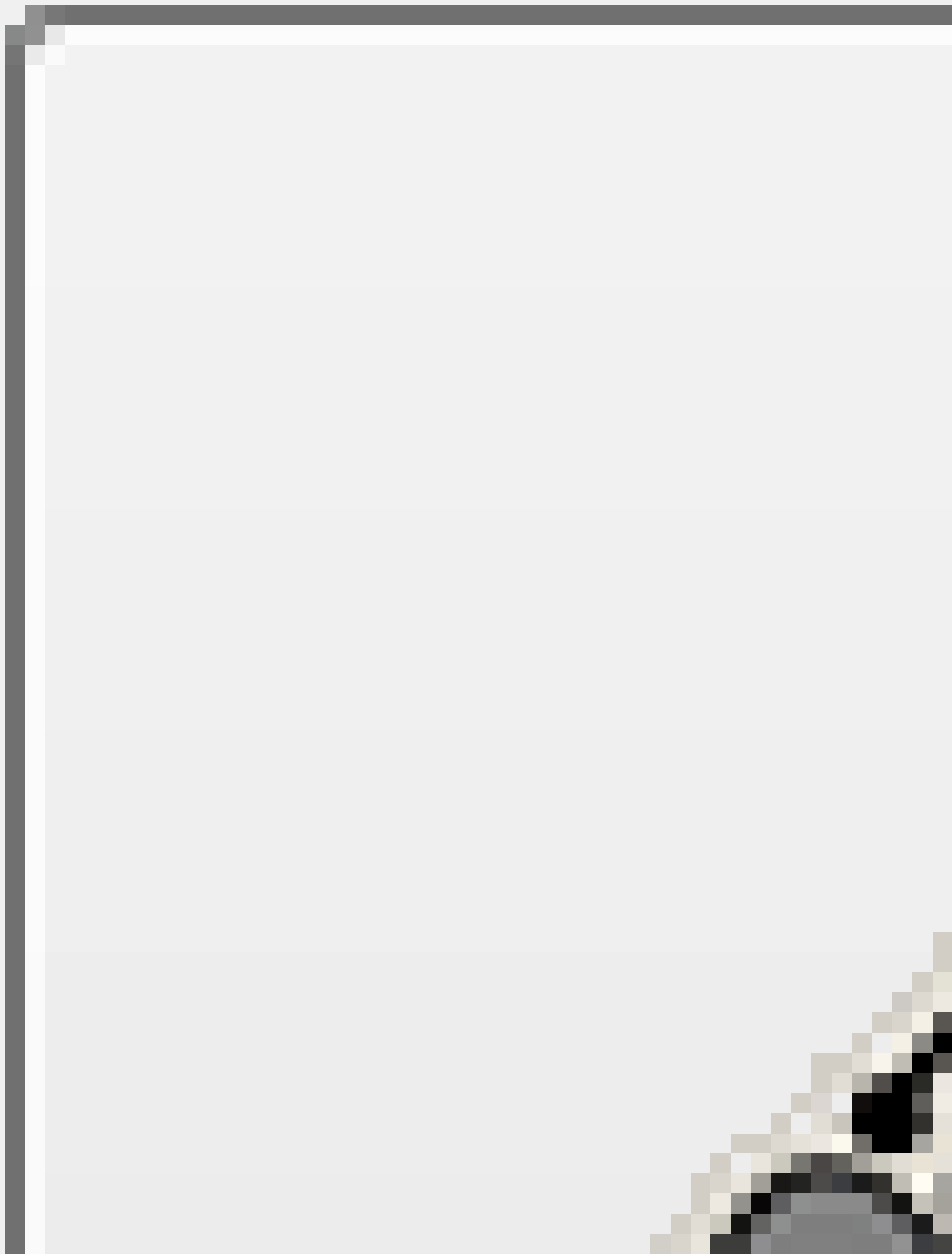
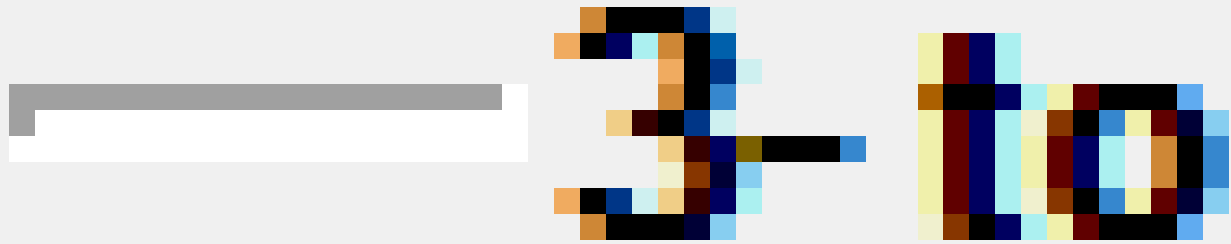
This strategy is used for special parts which require surface independent axis direction machining.

Example

1. Open the *Mold_moule_form* workzone and create two new copies of the **Z-Level Finishing** toolpath.

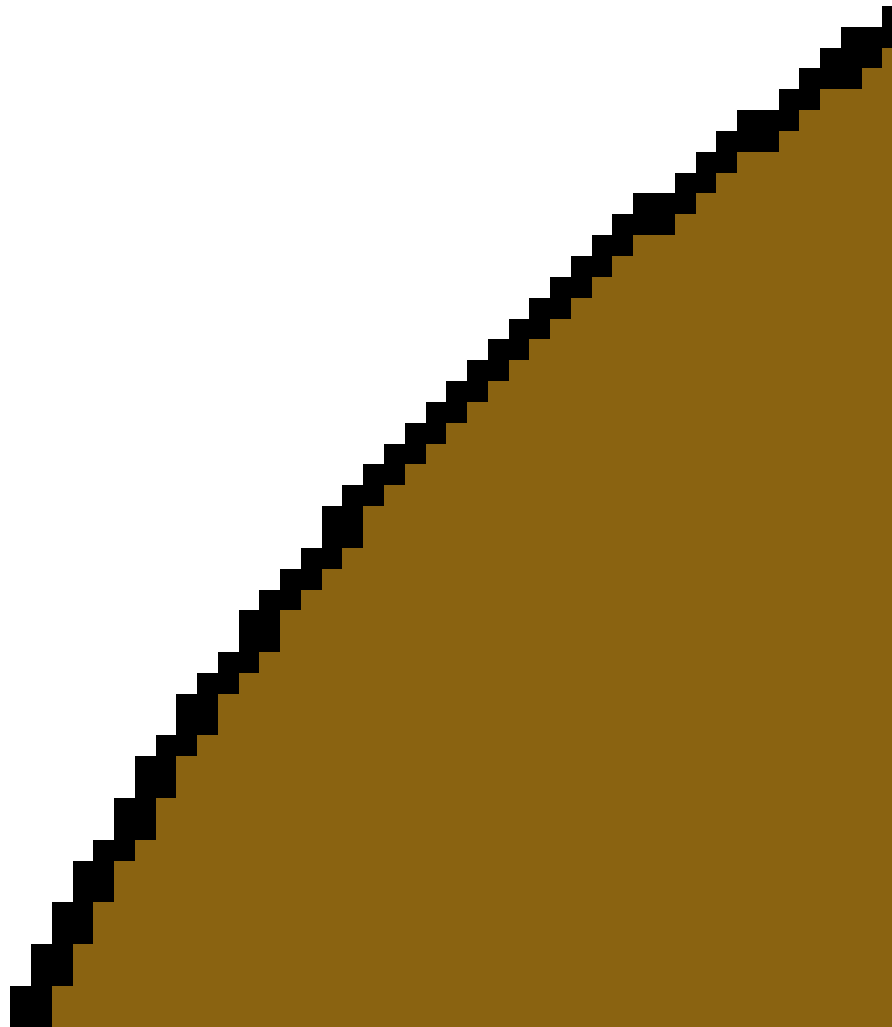
We are now going to convert those two new toolpaths using the **Guiding** strategies and the same points as we used for the **Constant** strategies.

2. Select the first copy of the toolpath and start the **Auto5** module
3. Click on the **Recalculate** button and select the **Guiding (Attractive)** strategy:



Guiding (Attracting) Strategy Interface

4. Select the center point that we defined earlier.
5. Validate the parameters and run the toolpath calculations.



Auto5: Guiding Attractive: Center Point

6. Select the second copy of the toolpath, start the **Auto5** module, click on the **Recalculate** button and select the **Guiding (Attracting)** strategy but this time select the outer point.
7. Validate the parameters and run the toolpath calculations.

Auto5: Guiding Attractive: Outer Point 1

8. Create a new point by offsetting the Z-value of the outer point.
9. Create a new copy of the last calculated toolpath and use this newly created point.

Note that the Z height of the point has an influence on the toolpath:

3.8 - Guide (Repulsive) Strategy

Introduction

This strategy implies that for any point on the toolpath an axis in the opposite direction of the guide point/curve is determined. The direction of the axis is defined by the shortest distance to an attraction point or an attraction curve.

The guide curve does not need to be closed and can consist of several sub-curves.

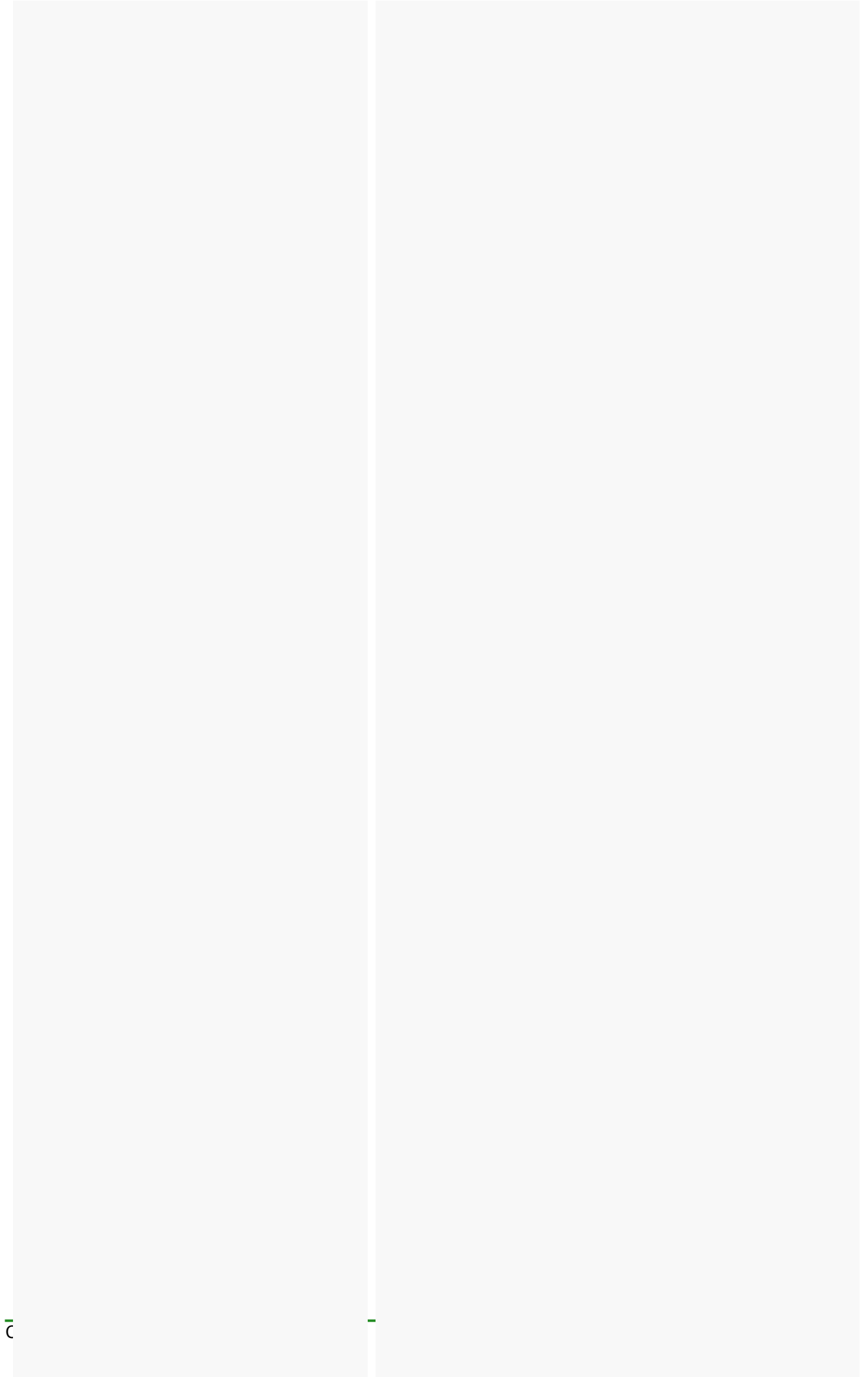
The repulsive point or curve must be positioned at the correct distance from the part. If it is too close to the part, axis variations may be too important. If it is too far from the part the machining will appear more like a constant repulsive strategy.

Use

This strategy is used for special parts which require surface independent axis direction machining.

Example

- Open the *Mold_moule_form* workzone and try the guide repulsive strategies with both center and outer points and compare results.



Auto5 Comparison: Guiding Attractive Center Point Auto5 Comparison: Guiding Repulsive Center Point

Auto5 Comparison: Guiding Attractive Outer Point Auto5 Comparison: Guiding Repulsive Outer Point

3.9 - Guide (Surface) Strategy

Introduction

This strategy consists in calculating a toolpath based on a guiding surface. This guiding surface is made either from one curve and a Z view or from a ruled surface defined with two curves.

The tool axis is oriented normal to the curve or the ruled surface created between two curves.

Use

An example of the use of this strategy is for machining complete automobile prototypes.

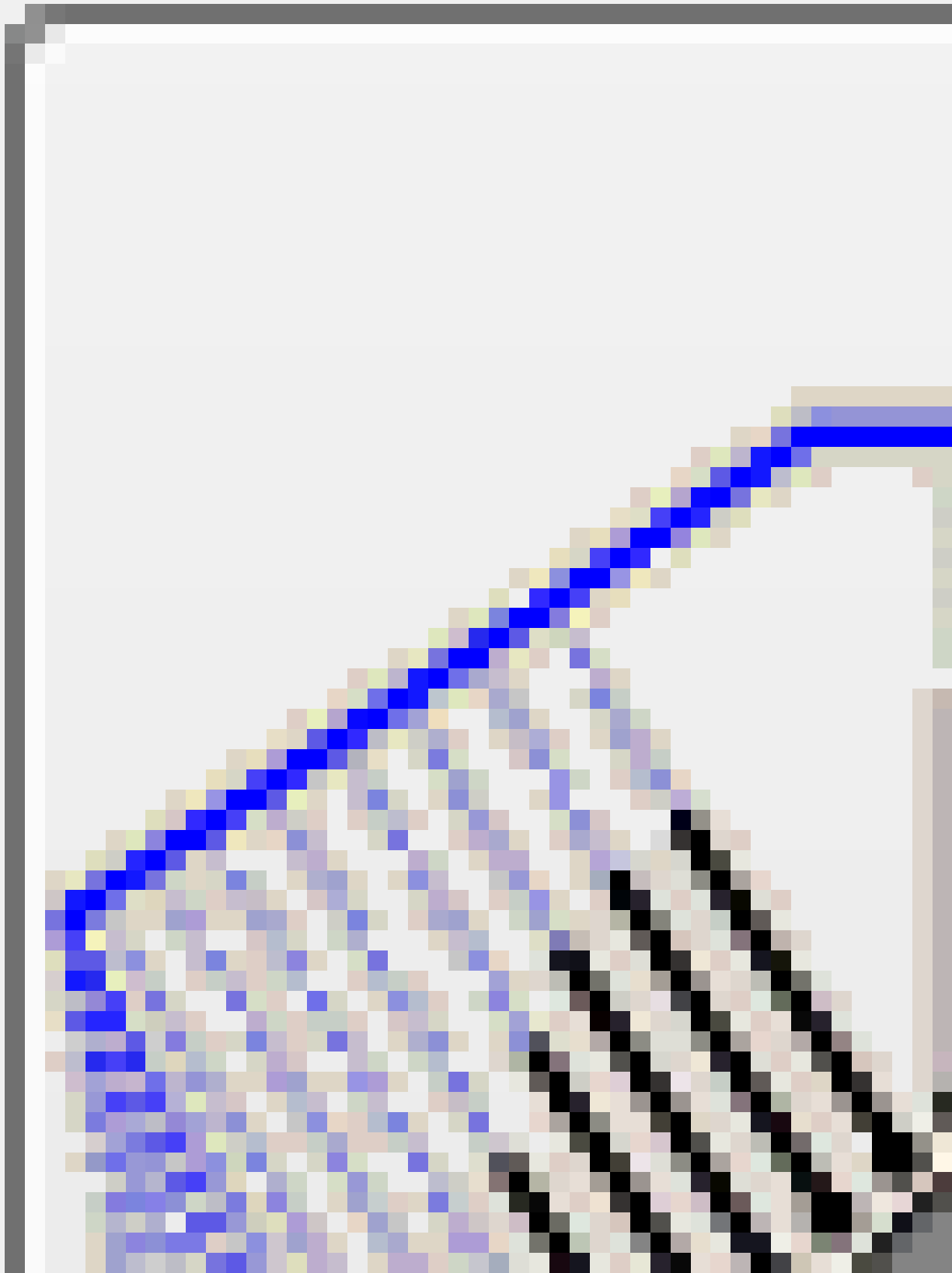
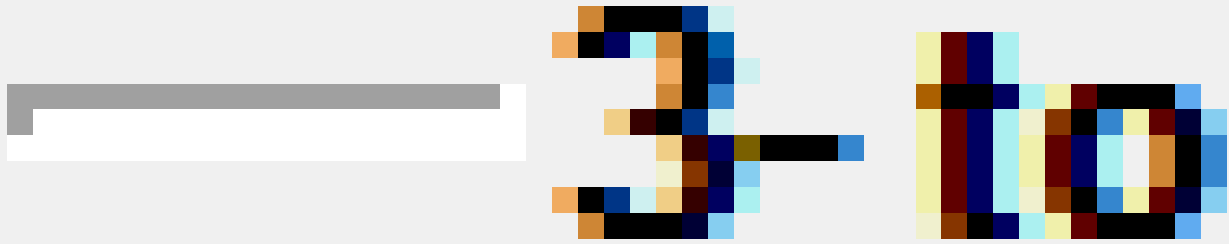
Example

1. Open the *Fender_auto5* workzone.

A **Planar Finishing** toolpath has already been calculated. We will use this toolpath to try the **Guiding (Surface)** strategy.

Planar Finishing Toolpath prior to the Auto 5 Conversion

2. Make two copies of the toolpath.
3. Select the first copy, start the **Auto5** module and select the **Guiding (Surface)** strategy.



Guide (Surface) Strategy Interface

First, we are going to convert the 3-axis toolpath by using the two predefined curves. In the first example, we are going to use the curve named *Curve_000* (A in the picture below) as the orientation curve and *Curve_001* (B) as the second curve:

Auto5: Curves for Guiding Surface

4. Select *Curve_000* in the **Orientation Curve** drop-down list.
5. Activate the **Second Curve** option and select *Curve_001* in the corresponding drop-down list.

For the 3 to 5 axis conversion, **WorkNC** creates a ruled surface based on both curves and orientates the tool according to the surface normals. If we had drawn the ruled surface in the CAD environment, we would have obtained the following:

Auto5: Ruled Surface

6. Validate the parameters and run the toolpath calculations.

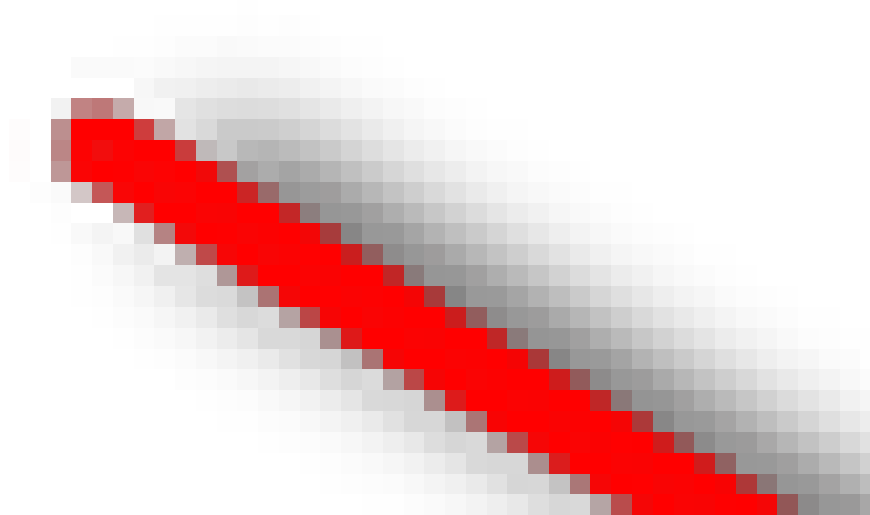
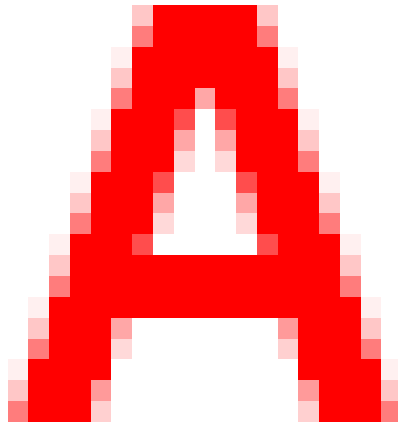
Auto5: Guiding Surface with two Curves

We are now going to use the second copy of the toolpath to try the other option, i.e. use a curve (*curve_000*) and a view (*view_000*):

Auto5: Guiding Surface with Z-Axis

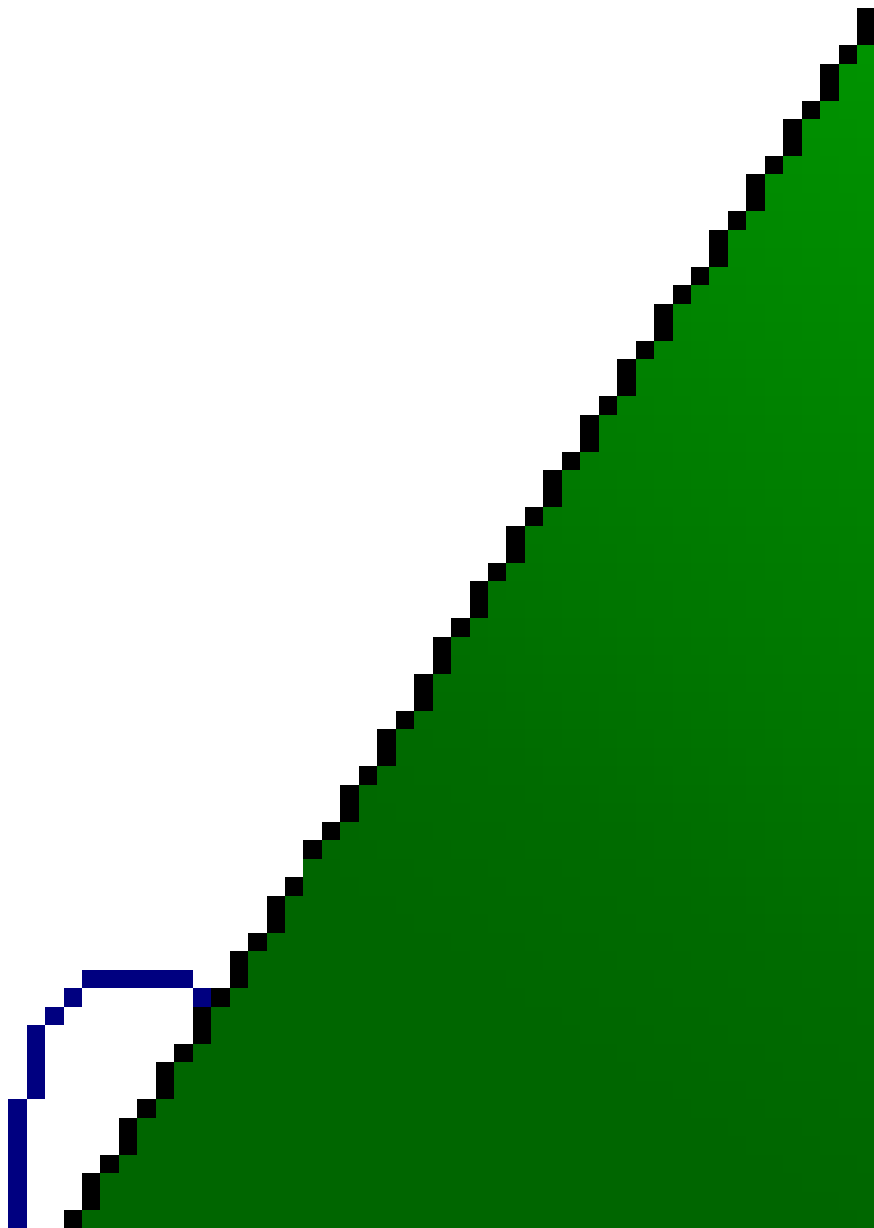
7. Select the second copy of the toolpath, start the **Auto5** module and select the **Guiding (Surface)** strategy.
8. Select *Curve_000* in the **Orientation Curve** drop-down list.
9. Select *View_000* in the **Z-Axis** drop-down list.

In this case, **WorkNC** creates a ruled surface based on the orientation curve (A in the picture below) and the Z axis of the selected view (B) and orientates the tool according to the surface normals. If we had drawn the ruled surface in the CAD environment, we would have obtained the following:



Auto5: Ruled Surface (Curve + View)

10. Validate the parameters and run the toolpath calculations.

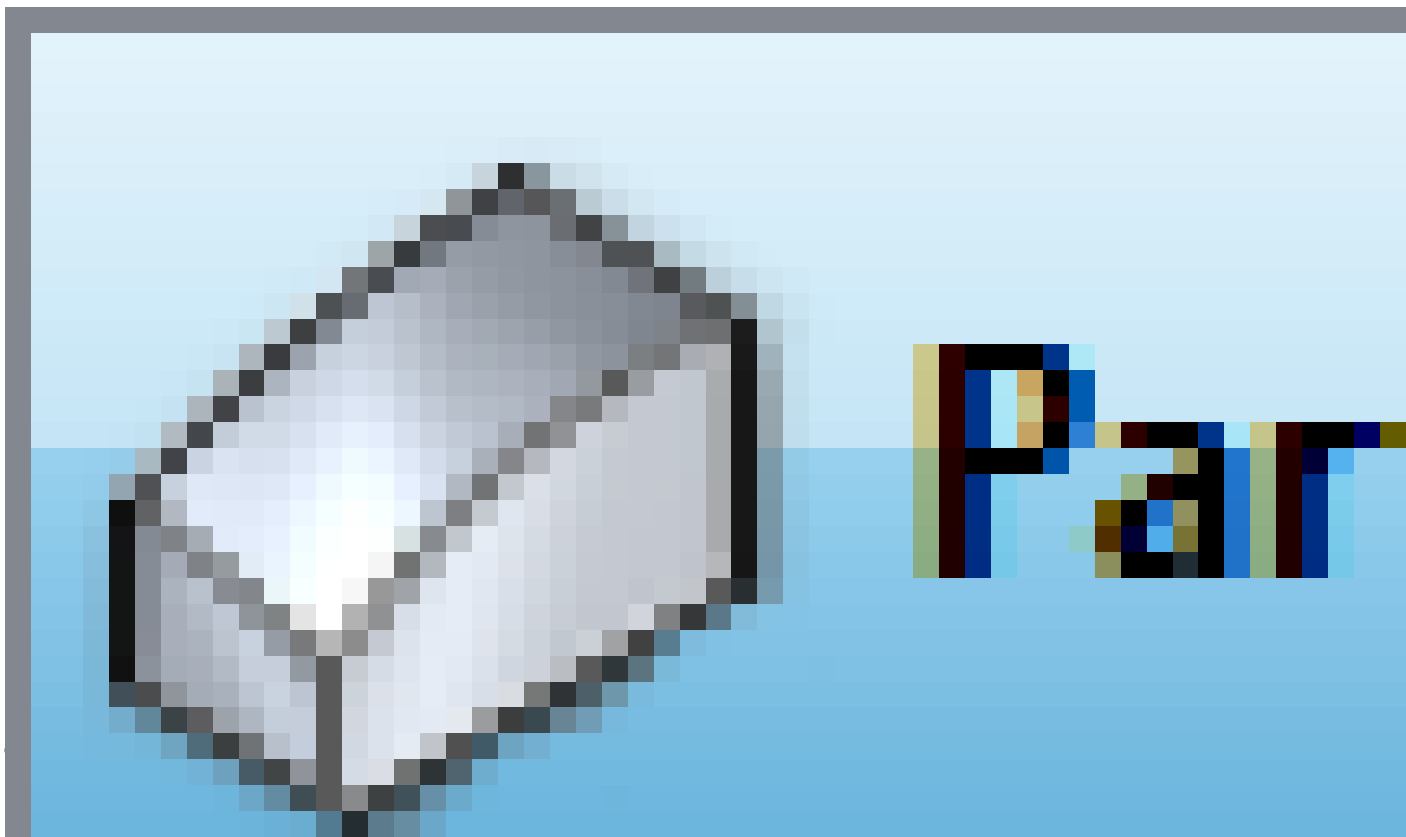
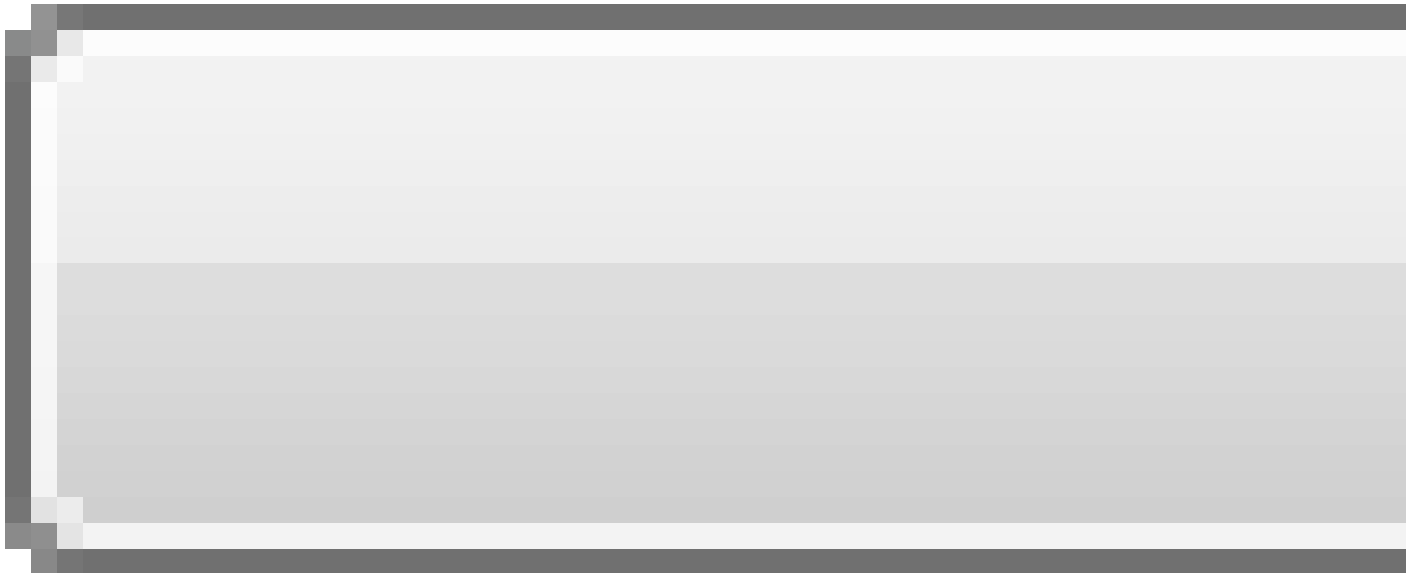
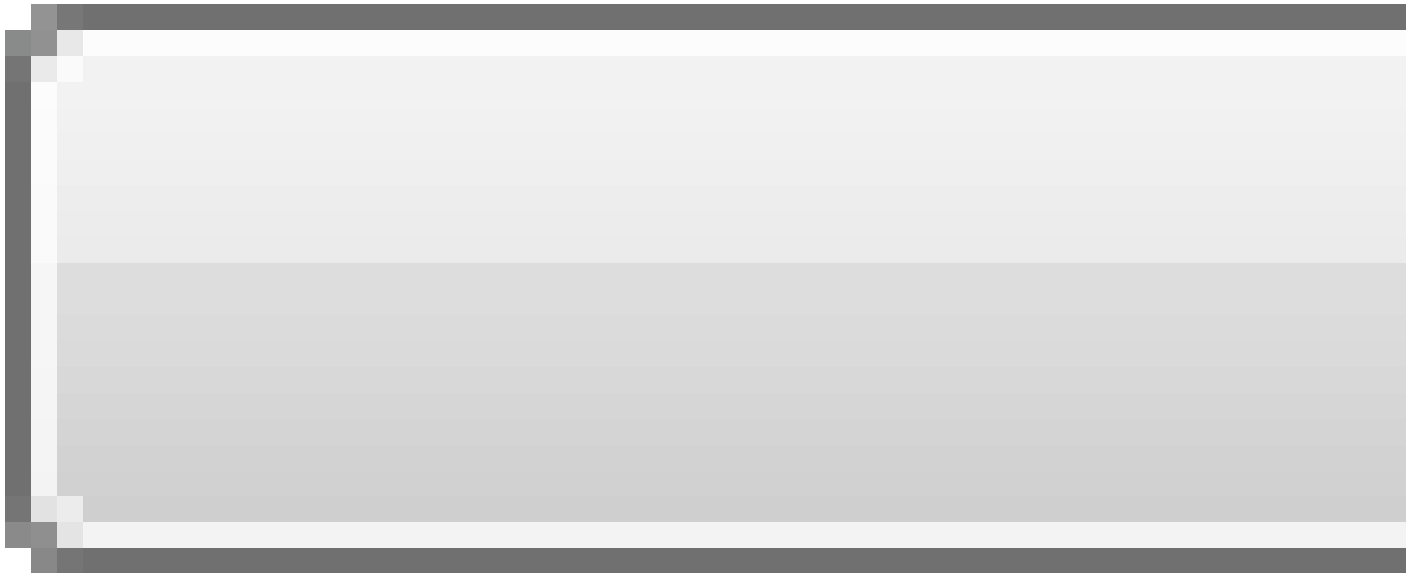


Auto5: Guiding Surface with a Curve and a View

3.10 - Vertical Rotation Strategy

When machining large parts on machines equipped with C-axis rotating tables, you can use the **Vertical Rotation** strategy to avoid Out-of-limit conditions that could otherwise be generated on a machine with a fixed table.

To illustrate this, we are going to use the *Out_of_limits* workzone.



Out of Limits

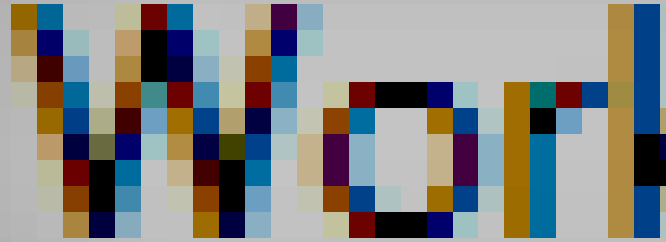
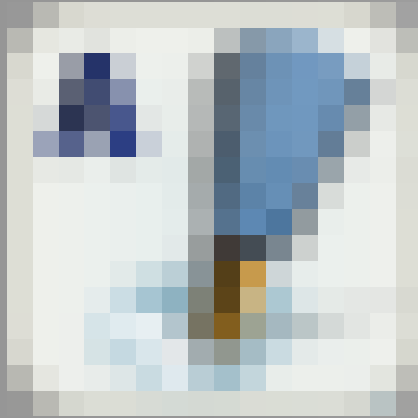
In this workzone, we have calculated a **Global Roughing** toolpath and used the **Machine Collision Detection** function. The rotational limits of the machine have been exceeded by the toolpath.

1. Make a copy of the toolpath
2. Click on the



icon on the left side of the user interface.

3. In the **Auto5** module, click on the **Select Strategy** button.
4. Select the **Vertical Rotation** strategy and validate.

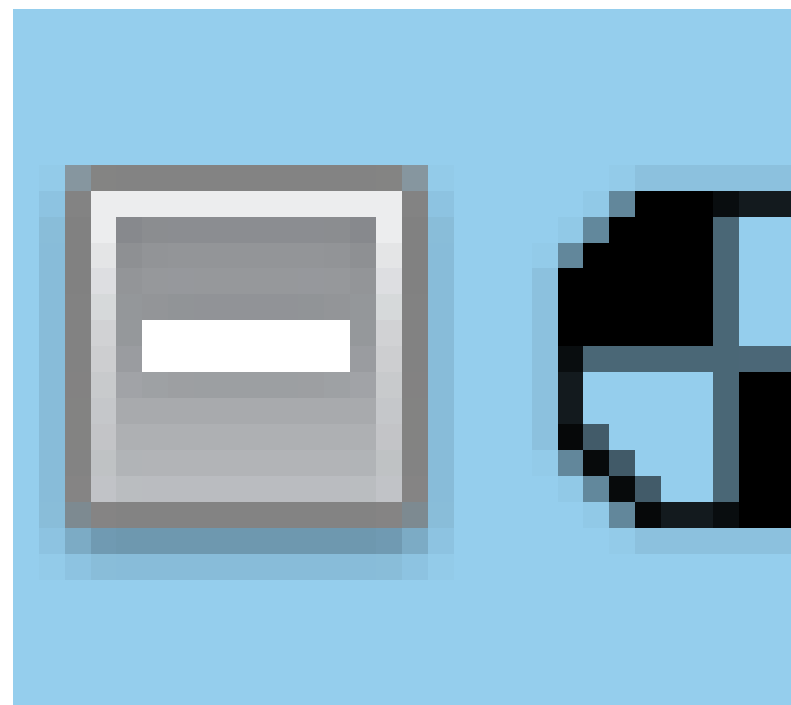
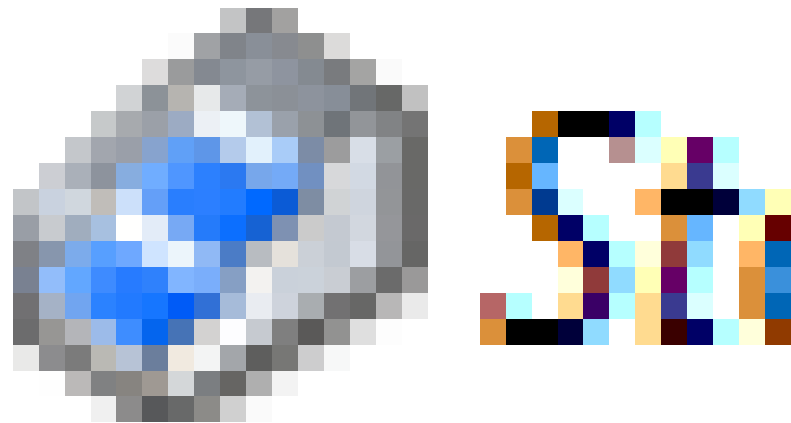
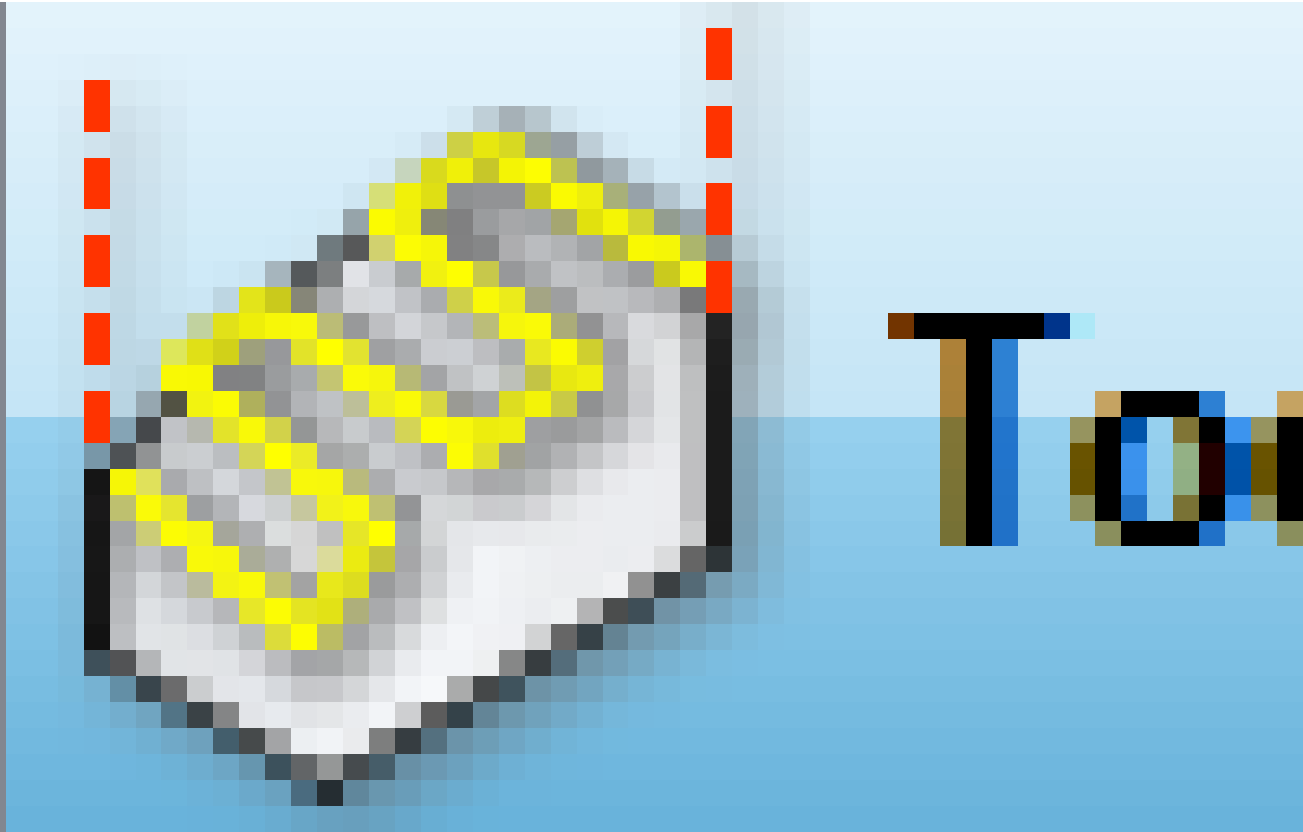


Auto5 Strategy Selection: Vertical Rotation

There are no specific parameters to define, you only have to activate the **Machine Limit Check** function in **Auto5**.

5. Click on the **Recalculate** button in the **Machine Limit Check** section.
6. Click **OK** to validate and run the toolpath calculations.

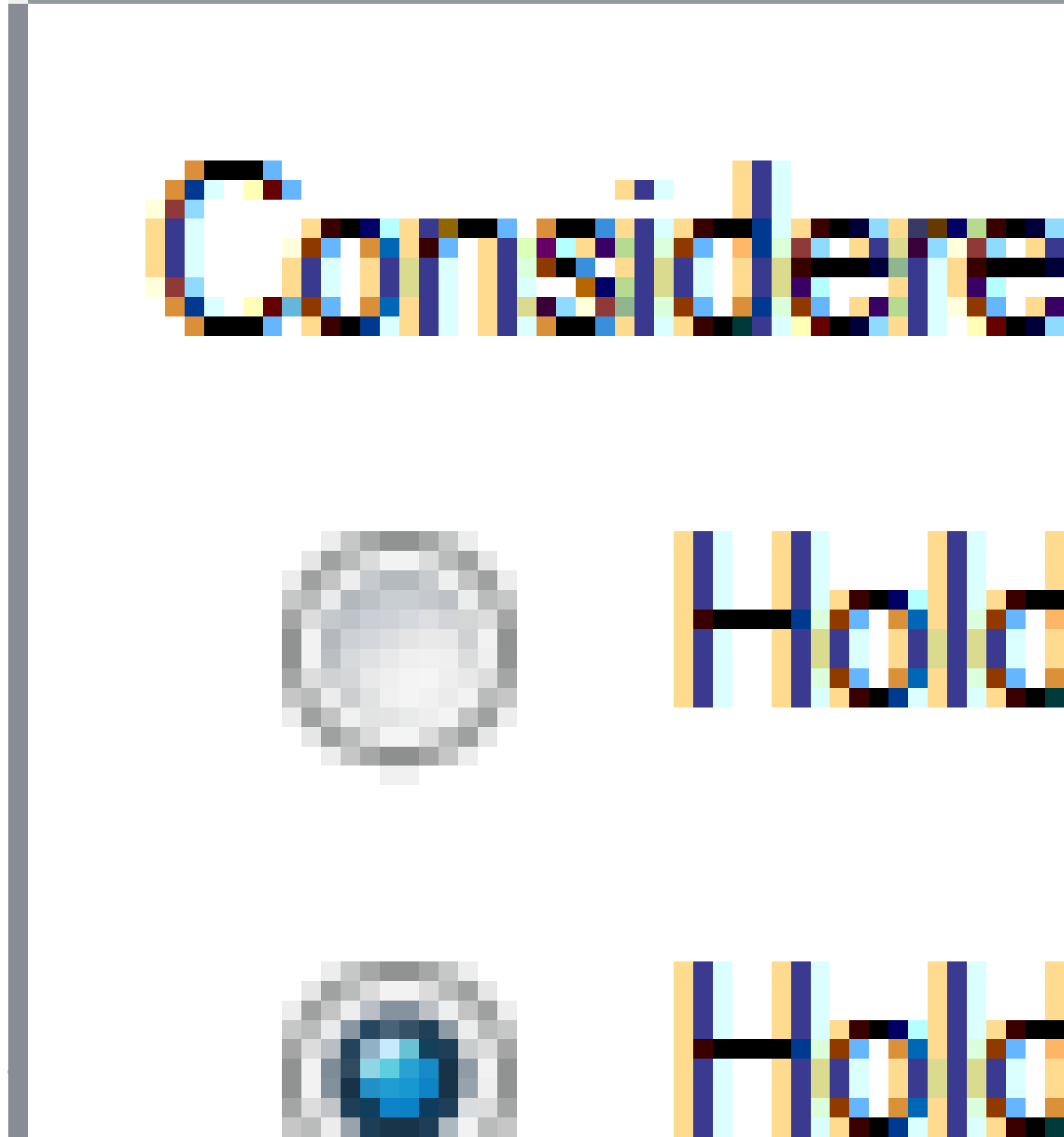
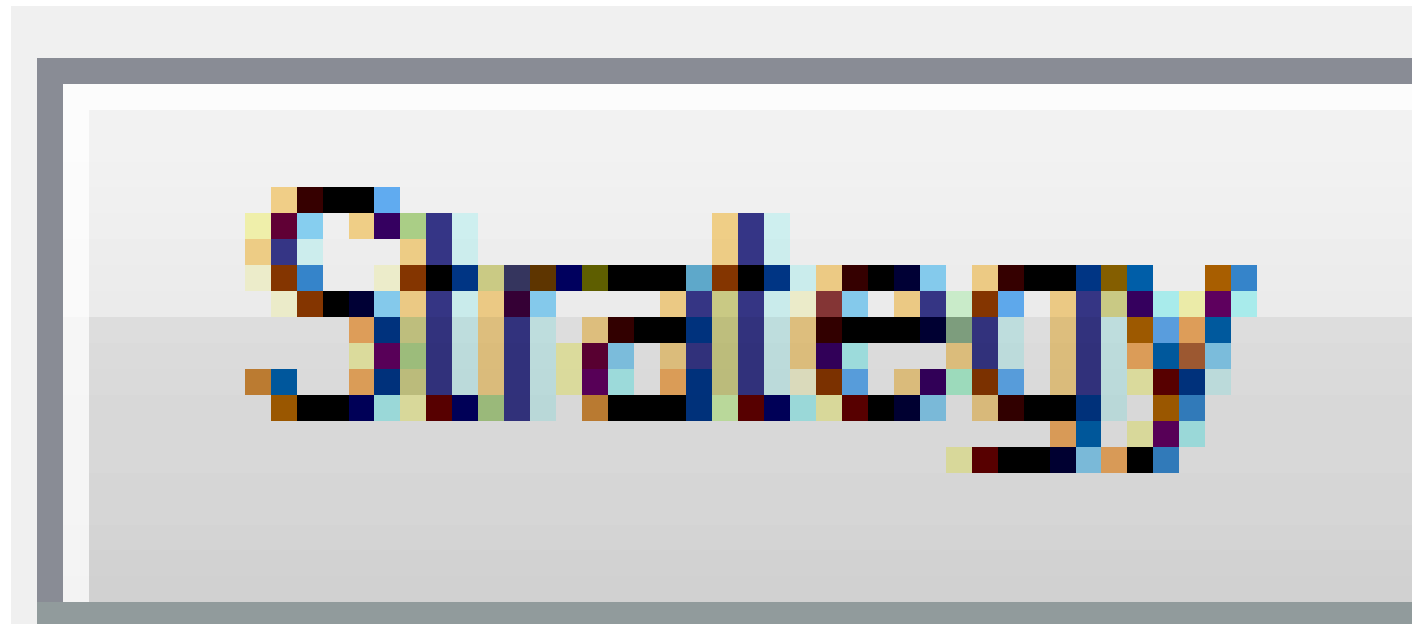
You should obtain the following result:



No Out of Limits Conditions

4 - 3 to 5 Machining Additional, Advanced and Holder Parameters

4.1 - Holder Parameters for Collision Avoidance



Holder for Collision Avoidance

Collision avoidance calculations are very complex and time consuming depending on the size of the holder. For this reason the user has the possibility to determine the exact areas of the spindle/tool holder assembly to be taken into consideration during calculations.

Holder and Spindle

The complete holder and spindle assembly is taken into consideration during calculations.

Holder

Only the holder section is taken into account even if a complete spindle/holder section has been defined for the toolpath.

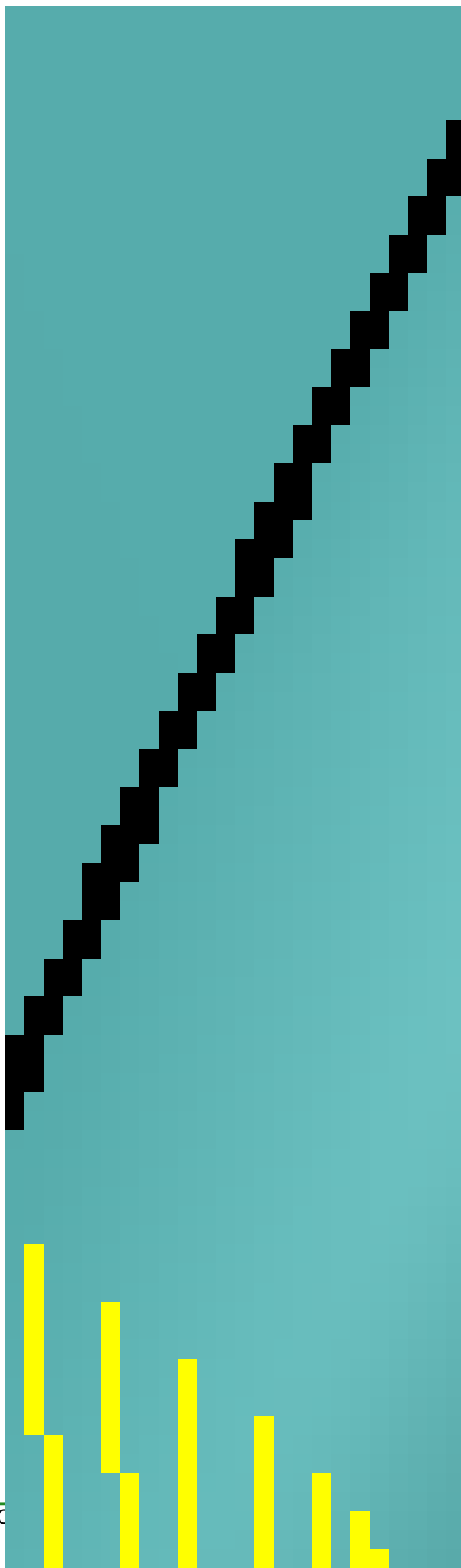
Shortened Holder

Only a user defined part of the holder (measured from the bottom of the holder itself) is taken into consideration during calculations.

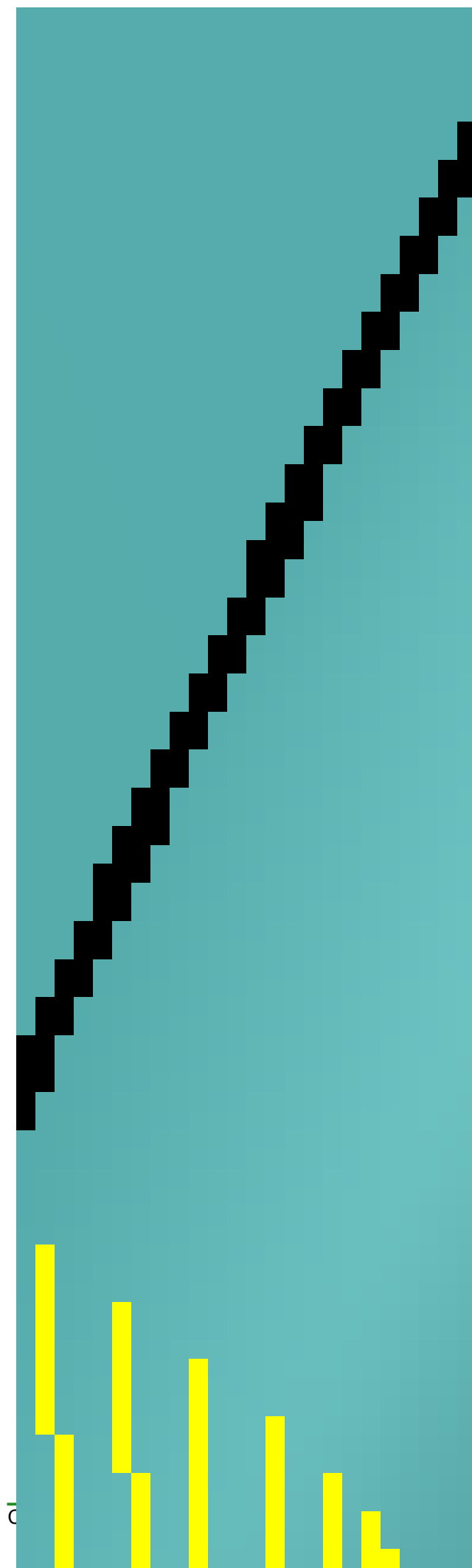
Shortened Holder

Using this option can be very dangerous and result in collisions because **Auto5** uses a potentially smaller holder than the strategy has defined. As a result the toolpath may be collision free for the shortened holder section but collide with a higher ignored section of the holder. It is the responsibility of the user to make sure that the toolpath under consideration cannot collide with parts above the shortened holder.

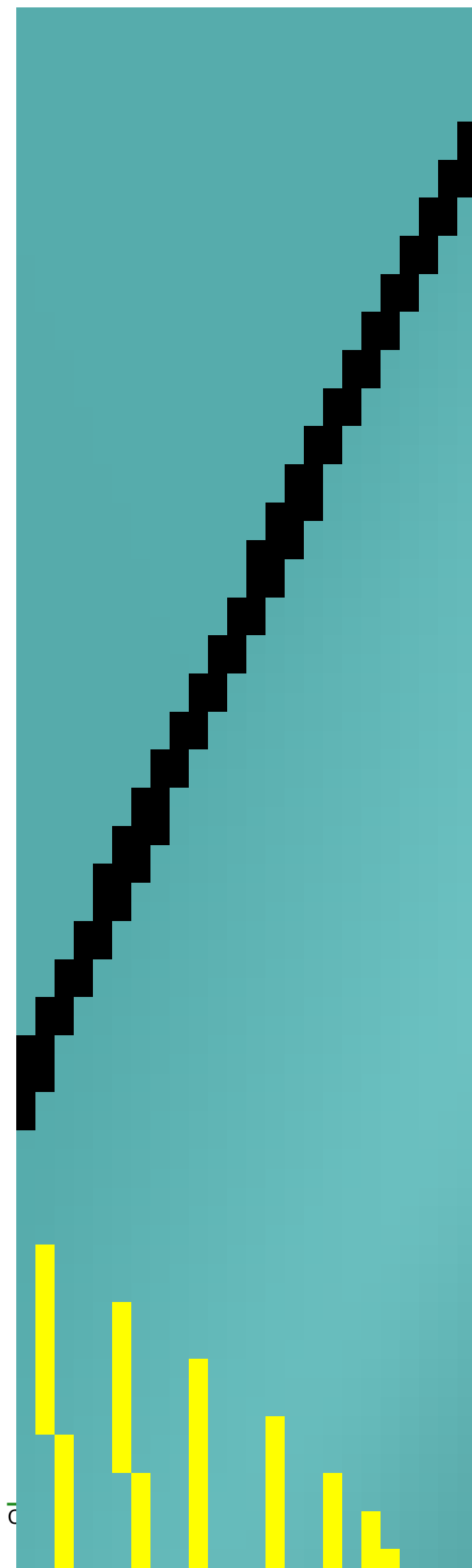
- Open the *Auto5_examples* workzone and compare toolpaths # 5, #6 and #7. The toolpath #6 is slightly different since it does not take account of the whole tool holder and spindle but only the holder. The toolpath #7 is different because it only takes into account a small part of the holder. You can check it in the **Auto5** module for each toolpath.



Auto5 Vertical: Holder and Spindle



Auto5 Vertical: Holder



Auto5 Vertical: Shortened Holder

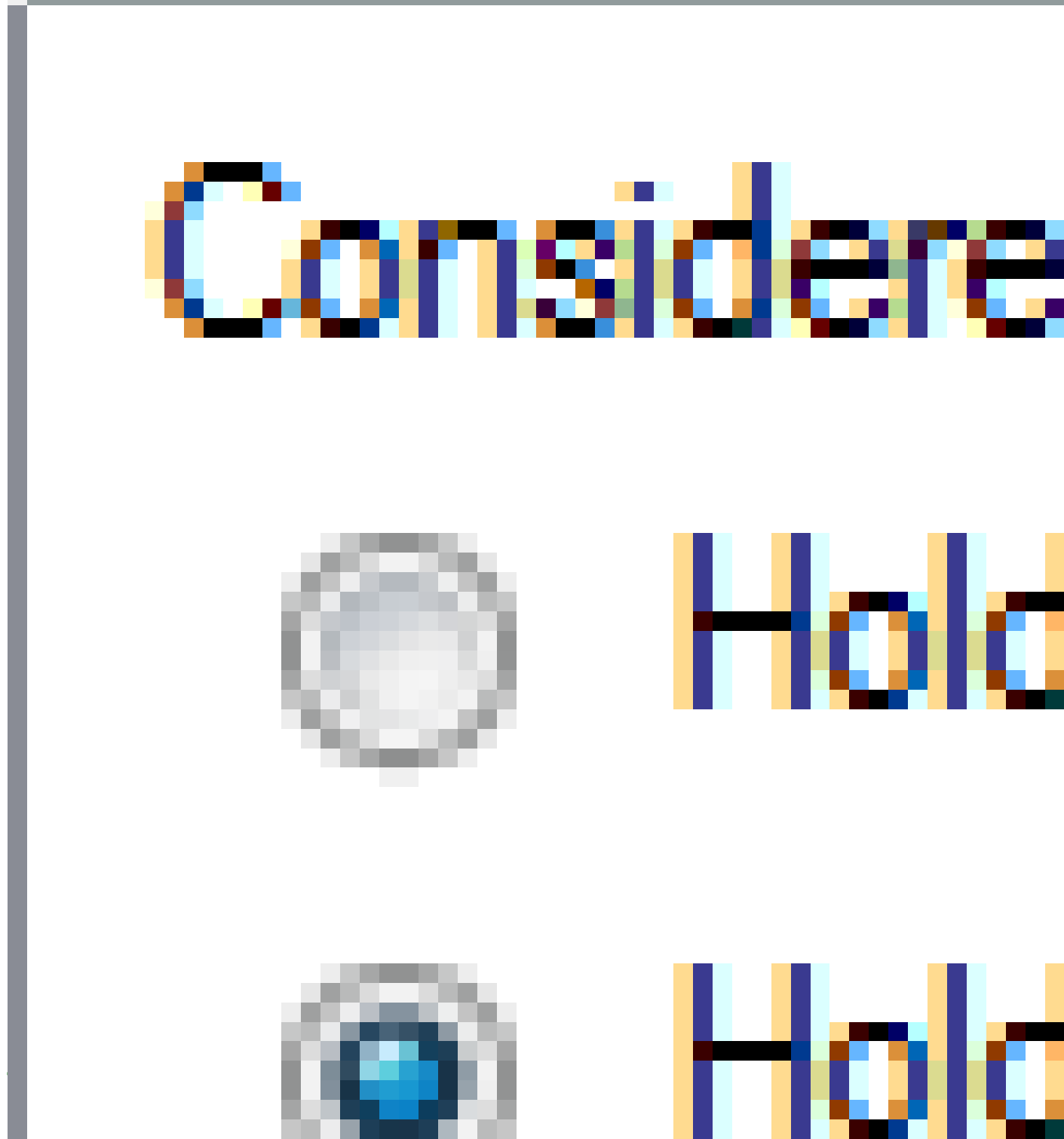
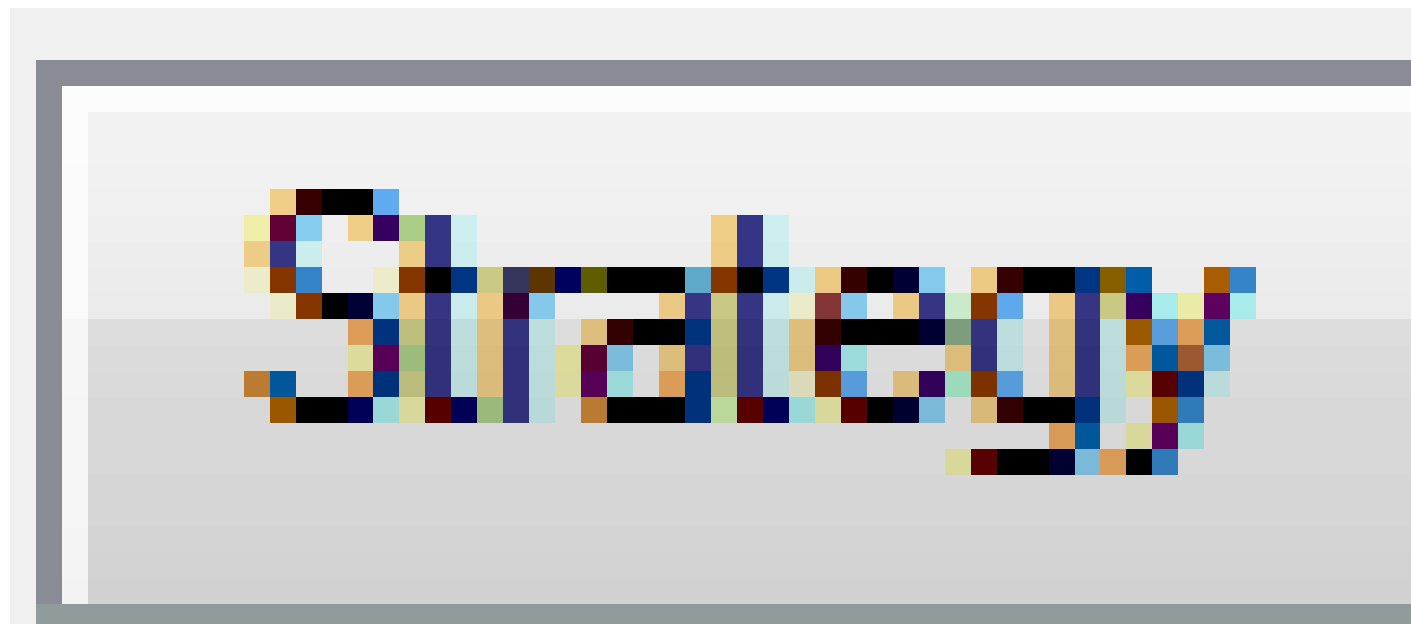
In this particular case, the tool inclination is more important with the shortened holder.

Consider Cutter for Collision Avoidance

This option allows you to deactivate collision detection for the cutter partially or completely.

- Activate the **Ignore Cutter over a Distance of** option to define the height of the cutter to be ignored for collision detection.

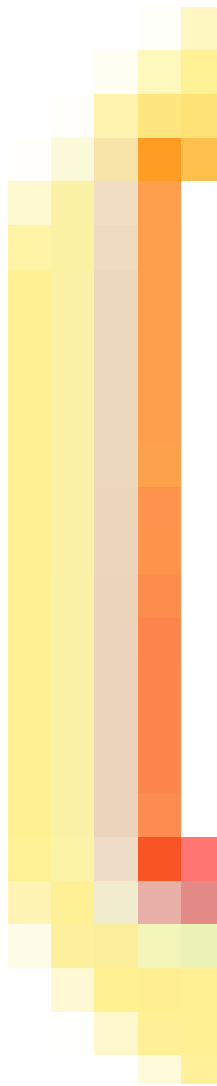
The example below shows that 10 mm of the tip are ignored:



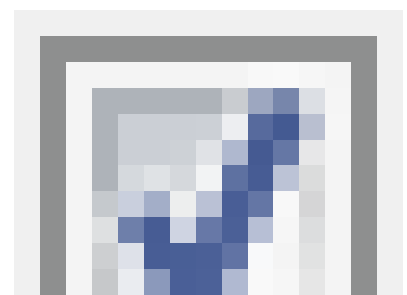
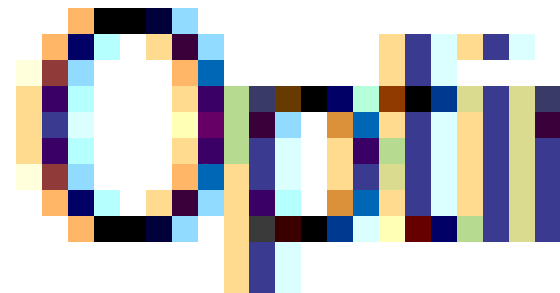
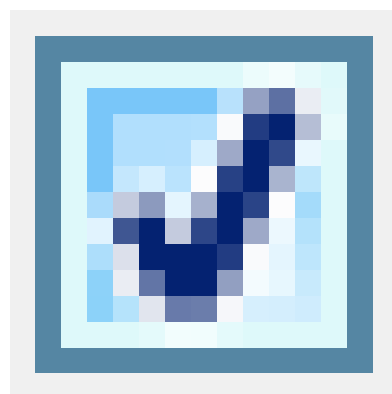
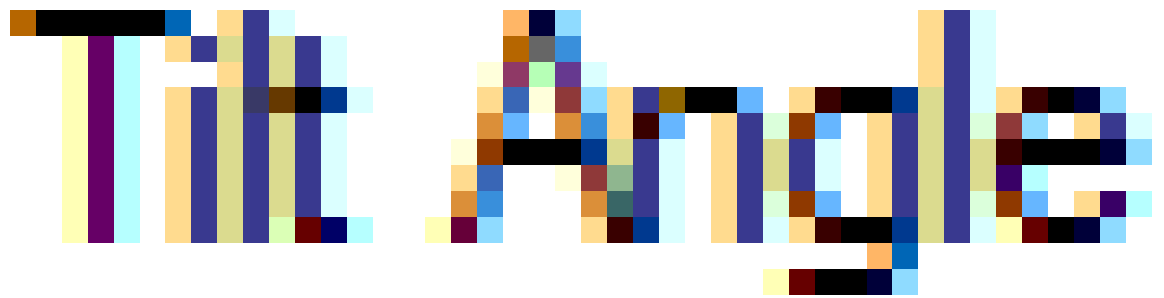
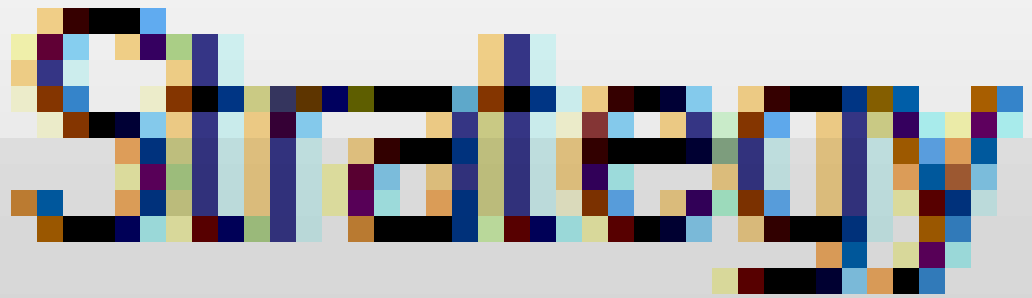
Holder Tab - Ignore 10 mm of the Cutter

Ignored Part of the Cutter

The height defined in this field is not considered for collision detection. The resulting toolpath may have collisions.



4.2 - Advanced Parameters



Auto5 Strategy Parameters - Advanced Tab

Tilt Angle Optimization

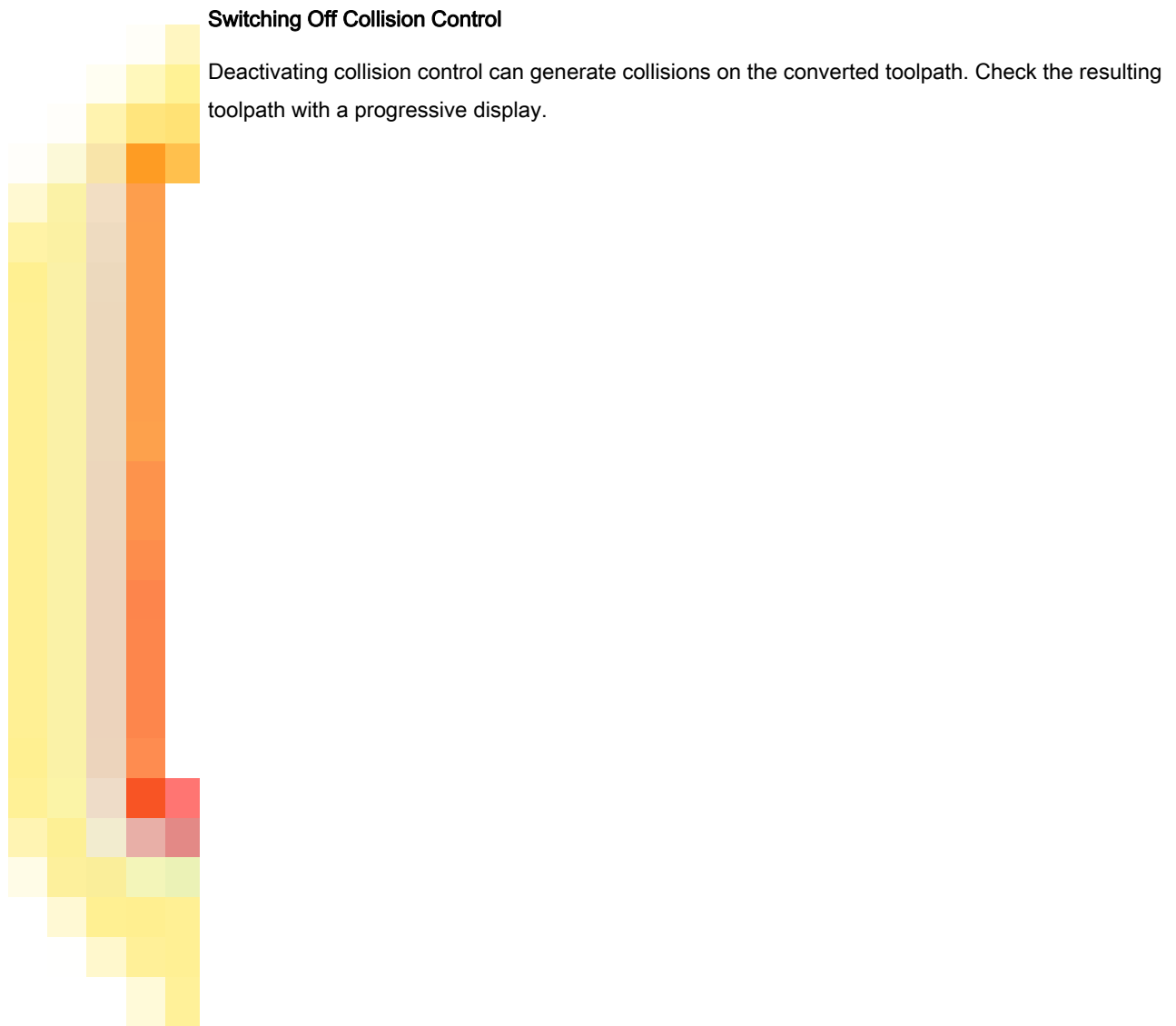
The **Optimize Tilt Angle** option is activated by default. It allows reducing large tilt angle variations within the machining area.

A link distance is automatically defined by the system.

- Deactivate the **Auto Link Distance** option if you want to specify the link distance manually.

Collision Control/Collision Avoidance

Activating this option allows to completely deactivate the collision control in the 3 to 5 axis conversion calculations.



Context Surfaces

This option is available only if a group of context surfaces has been defined in the Surface Selection option of the toolpath parameters.

If you activate the **Consider Ignored Surfaces** option, the ignored surfaces of the corresponding list group are taken into account in the 3 to 5 axis conversion calculations.

5-Axis Toolpath Creation

Activating this option allows you to program a strategy as if it was a 5-axis strategy. If the tool defined for the 3-axis toolpath is long enough to avoid collisions, the resulting toolpath is a 3-axis toolpath. Using this option may be useful when you use the same machining sequence for different parts.

4.2.1 - Adjusting Tilt Movements on Narrow Regions

You can indicate that the regions to be machined by the toolpath are narrow to obtain smaller tilt movements. This function is available for all 3 to 5-axis strategies of **Auto5**.

This is particularly useful on machines with inclinable tables (rotation around A axis) to avoid the tool to be heavily tilted and reduce the risk of tool breakage.

To illustrate this, we are going to use the *Auto5_slim_regions* workzone.

In this workzone, we have calculated a **Z-Level Finishing** toolpath on a specific level of the part (defined with a variable Z-Step) and added a cylindrical holder. We have also defined a Machining Context with a C600U machine with an inclinable table.

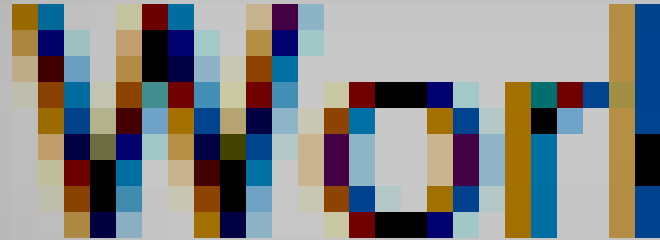
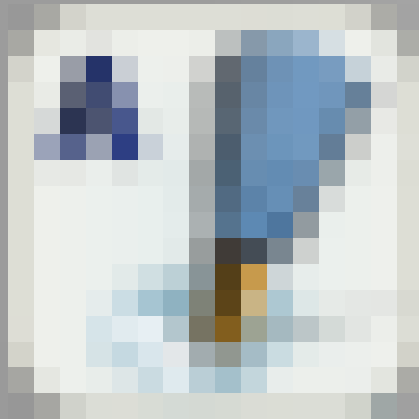
1. Make a copy of the toolpath.
2. Select the second toolpath and click on the



icon.

3. Select the **Vertical** strategy.
4. Click on the **Advanced** tab.

The **Geometric Regions to be Machined and Expected Tilting** section allows you to specify how narrow is the area to be machined and adjust the tilt movements accordingly.



No.

2

2

Geometric Regions to be Machined and Expected Tilting

5. For our example, leave the **Standard (Default)** option activated.
6. Click **OK** to validate and run calculations.

You should obtain the following result:



Default Tilting Parameters

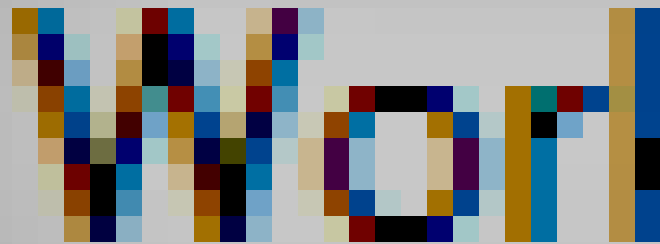
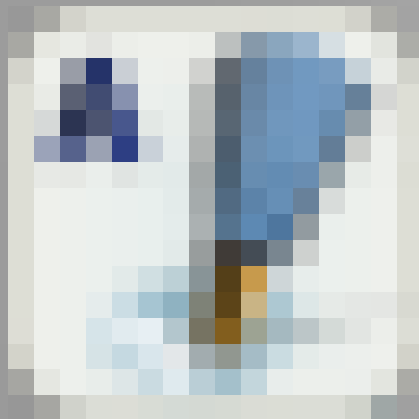
Now let's define very small tilt movements:

1. Make a copy of the second toolpath.
2. Select the new toolpath and click on the



icon.

3. Click on the **Recalculate** button.
4. Click on the **Advanced** tab.
5. Activate the **Extremely Narrow Regions** option.



No.

3

2

Extremely Narrow Regions Option Activated

6. Click **OK** to validate and run calculations.

You should obtain the following result:



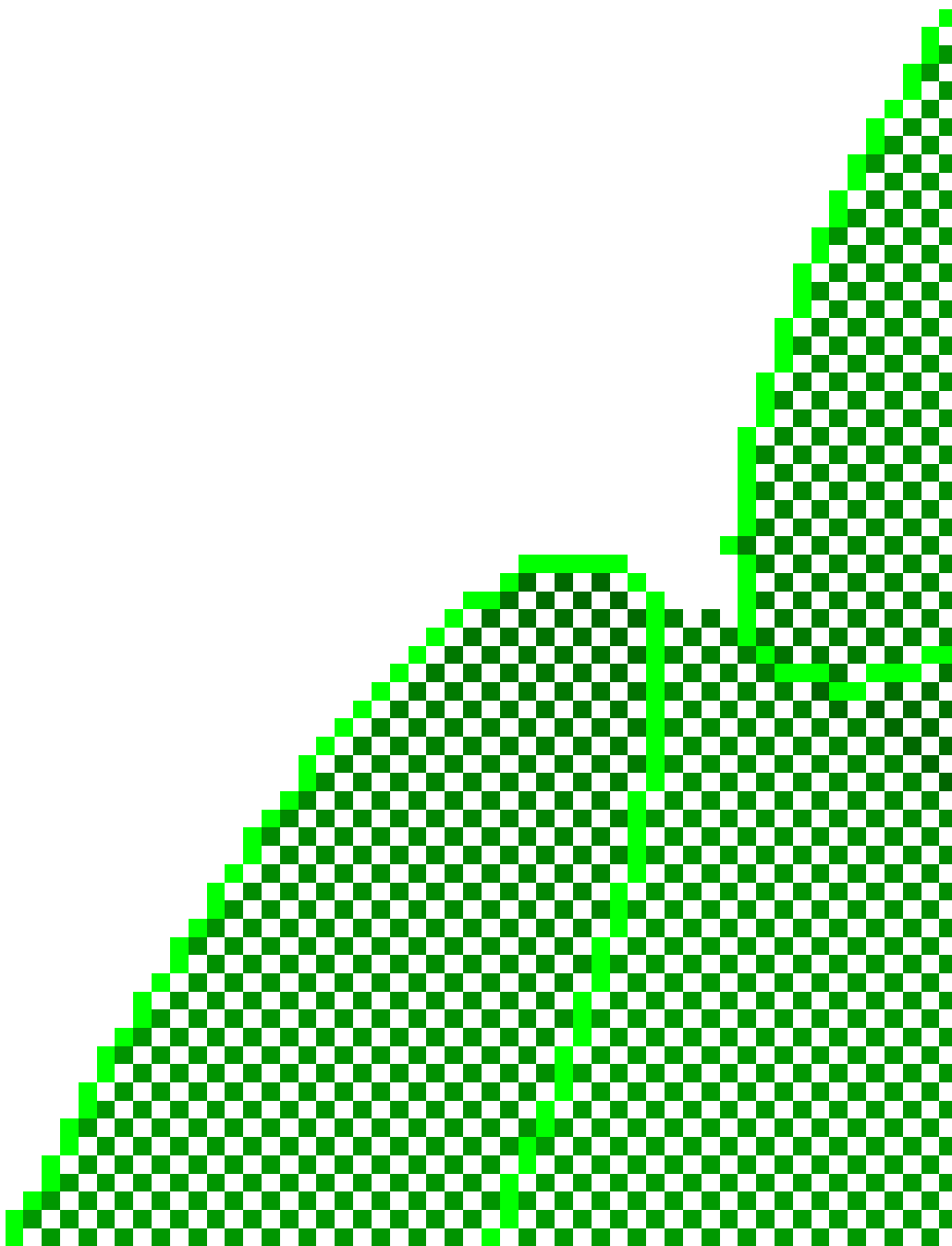
Small Tilt Movements

If you activate the Machining Context display and simulate both toolpaths, you should see the differences in the tilt movements (A angle of the machine table) between the toolpath with the **Standard** option activated



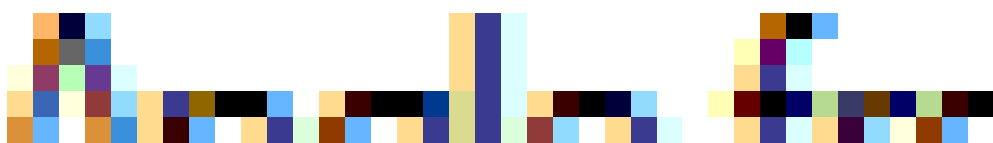
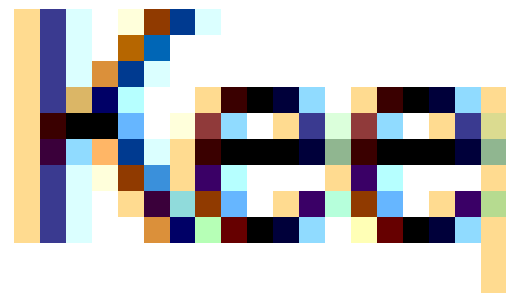
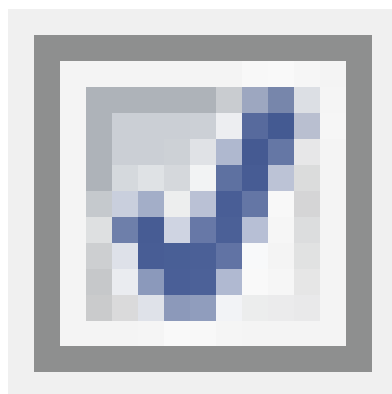
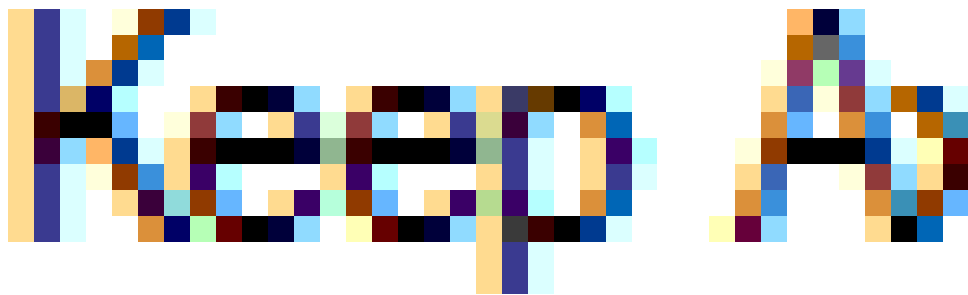
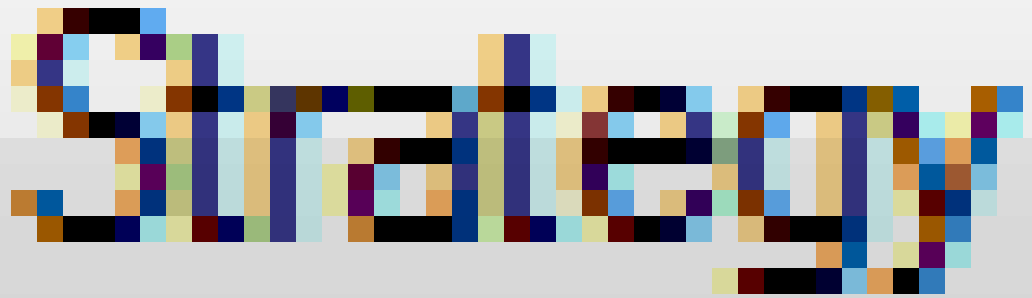
the **Extremely Narrow Regions** option activated





Toolpath Comparison

4.3 - Keep Axis Along Machining Direction



Keep Axis Along Machining Direction

All angles previously explained are angles with respect to a reference axis, especially for the Constant strategies, whose aim is to maintain the A/B axis in a constant position. It is possible to use an opposing strategy by keeping the C axis constant and varying A/B by selecting the **Keep Axis along Machining Direction** option. This option is applicable to **Constant to Axis**, **Normal to Surface**, **Constant (Attractive)** and **Constant (Repulsive)** strategies.

The following examples illustrate the difference for a finishing toolpath which follows the surface normal up to an angle of 40°.

The two following screenshots show the standard toolpath without the **Keep Axis along Machining Direction** option. The screenshot on the right is a head on view along the machining direction showing deviation from this direction.

Standard toolpath without the Keep Axis Along Machining Direction option

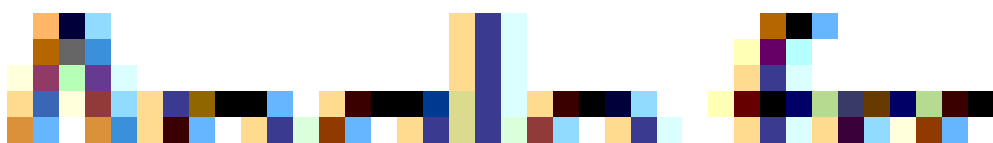
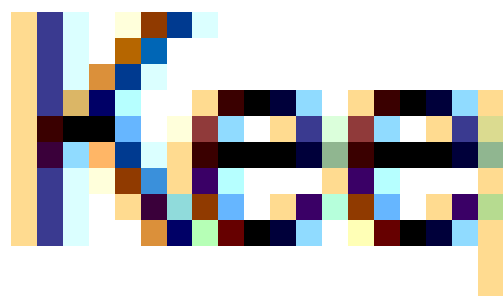
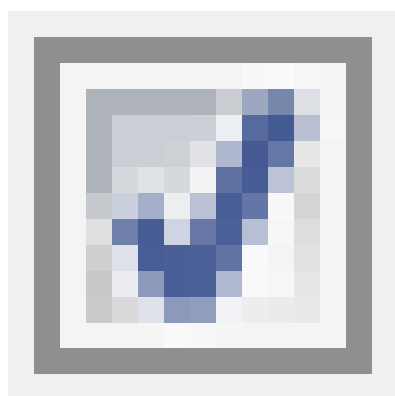
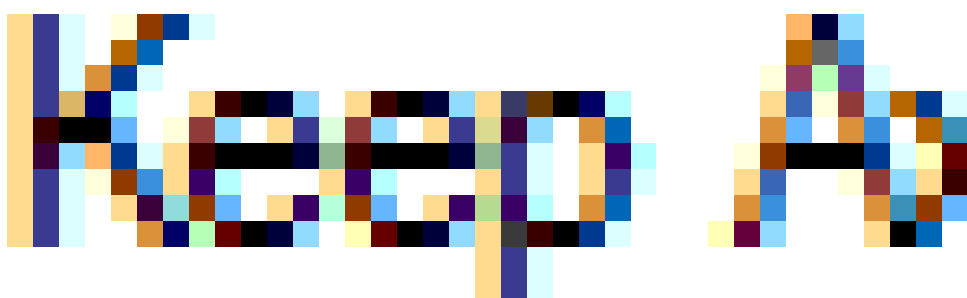
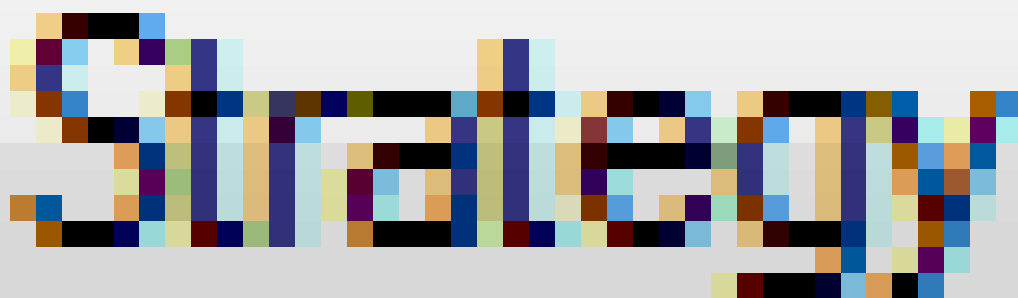
The next two screenshots show the effect of selecting the **Keep Axis along Machining Direction** option with an angle of 0°.
The screenshot on the right is a head on view along the machining direction showing that all axes are lying in the plane defined by the machining direction and the Z-axis of the View.

Standard toolpath with the Keep Axis Along Machining Direction option

It was necessary to define an angle of 0° because the toolpath uses an angled view and the machining direction is with reference to the view. For that reason it was also necessary to choose the **View Axis** as the reference axis.

- Open the *Auto5_Keep_Axis* workzone and compare the toolpaths # 2, 3 and 4 which were calculated with and without using the **Keep Axis** option.

4.4 - Smoothing Distance



For **Constant (Attracting)** and **Constant (Repulsive)** strategies it is possible to determine the behavior of the angle recommendation when the tool approaches the orientation entity.

The use of the parameter becomes apparent when we look at a toolpath as it moves from one side of an attraction curve to the other. Normally, the strategy would switch the angle orientation from one point to another (e.g. from -20° to 20°).

By setting the **Distance for Constant Angle** parameter, the user can influence the behavior within that distance. The closer the point is to the orientation entity the more it becomes vertical.

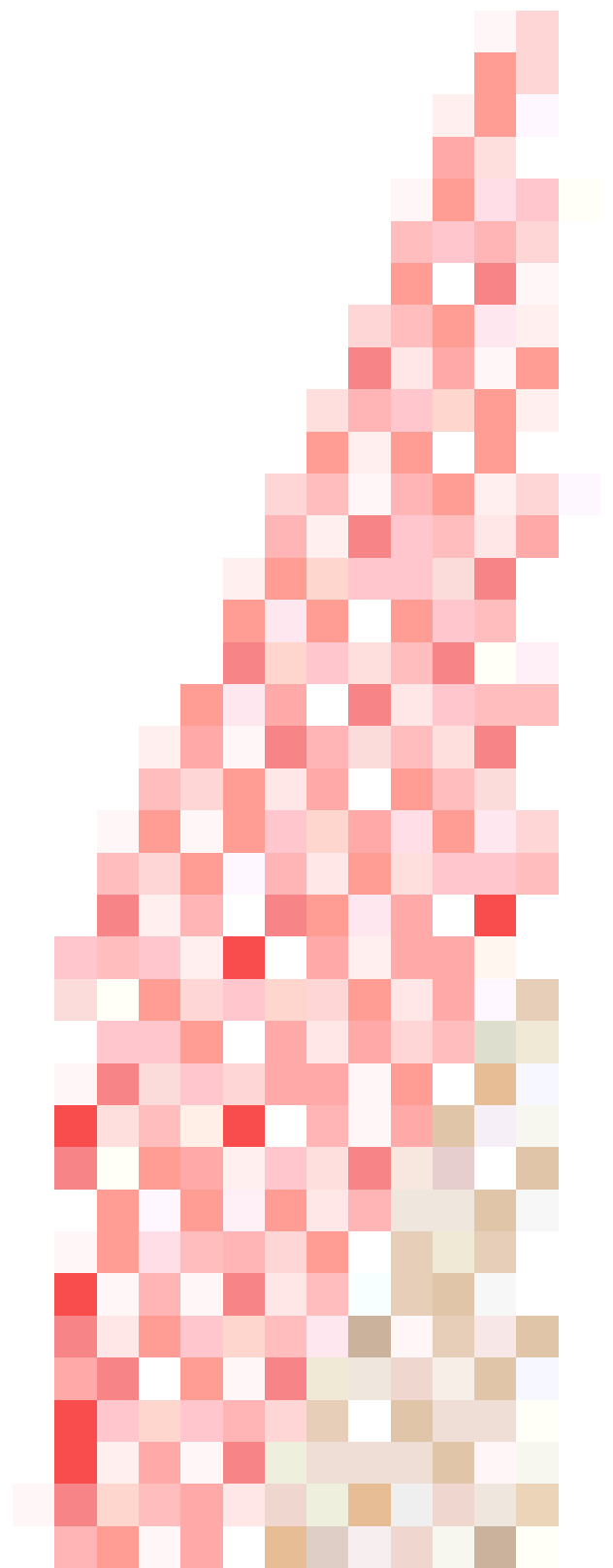
Example

The following example shows the difference for a constant strategy with an angle of 25° where the attraction line **T** is in the center of the U shaped profile.

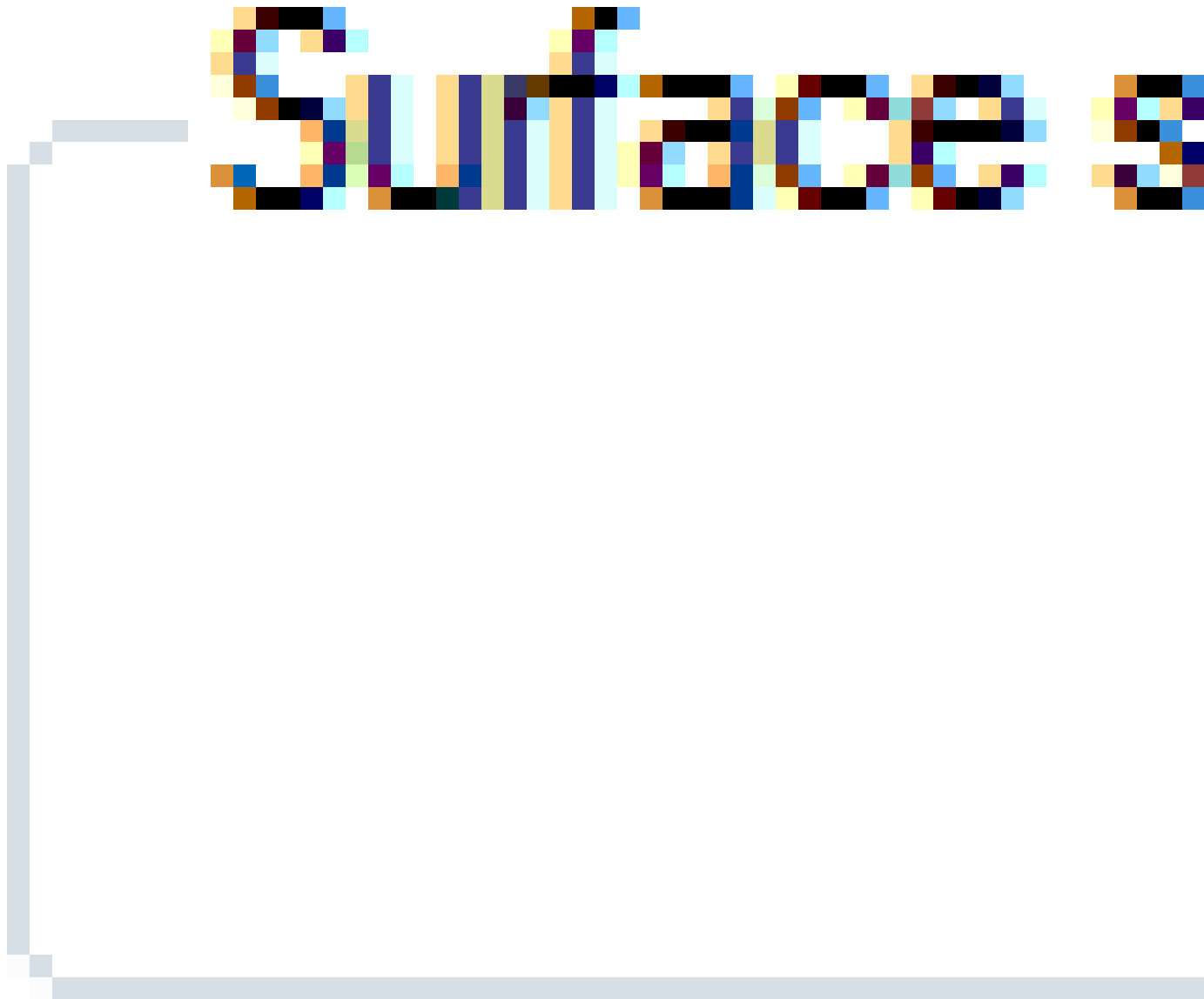
The first screenshot was calculated using 0.0 as minimum distance for the constant angle. In this case, the toolpath keeps the 25° angle as long as possible.

The transition from $+25^\circ$ to -25° is performed abruptly only when the tool is near the center of the profile.

In the second screenshot a distance of $R = 10$ mm has been defined. The transition begins as soon as the distance is less than 10.



4.5 - Surface Sensitivity



Surface Sensitivity Setting

This parameter is available for the **Constant to Axis** and **Normal to Surface** strategies. It allows you to determine the degree of influence of the part geometry on the tool axis at each point on the toolpath.

There are three possible settings : **Low**, **Medium** and **High**.

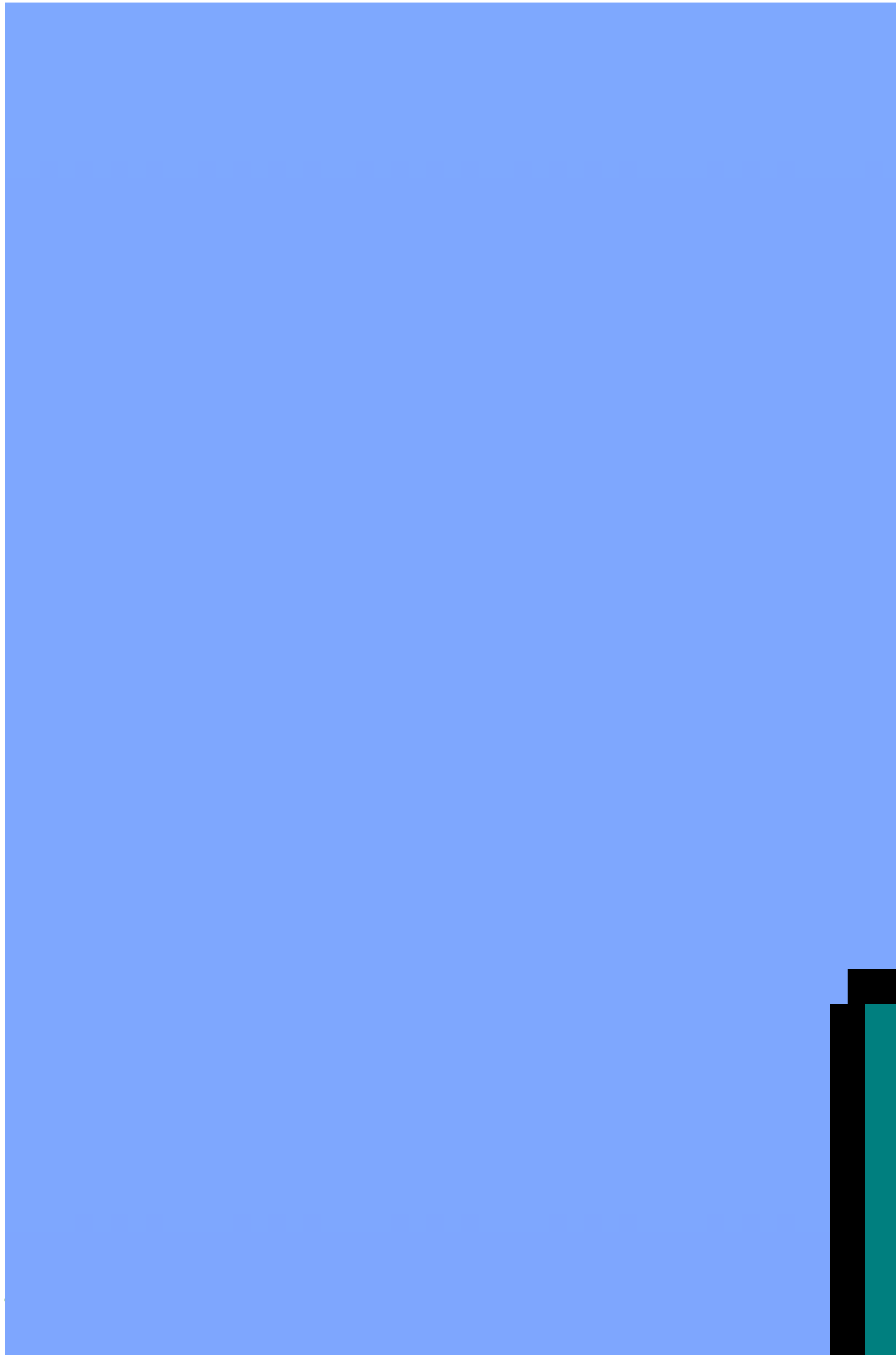
If the setting is set to **High**, the part geometry at a particular point will have a greater influence on the resulting tool axis.

If the setting is fixed as **Low**, the tool axes along the toolpath are smoothed and follow as near as possible a constant C axis position.

The following screenshots illustrate the influence of this parameter.



Example of High Surface Sensitivity



Example of Medium Surface Sensibility



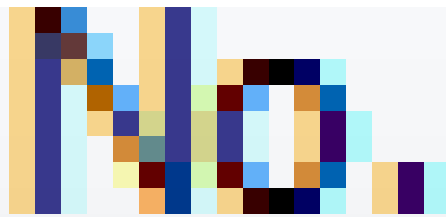
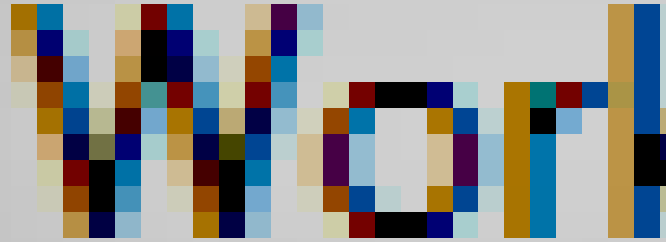
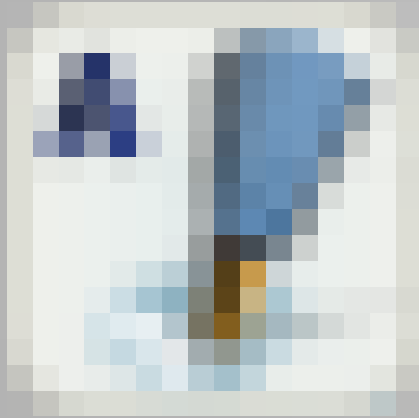
Example of Low Surface Sensibility

5 - Auto3+2 Strategy

The **Auto3+2** module requires a 5-axis license or - if you do not have one - a specific 3+2 license.

The **Auto5** dialog box looks different depending on whether you have an **Auto3+2** license only or a 5-axis license.

With an activated 5-Axis License



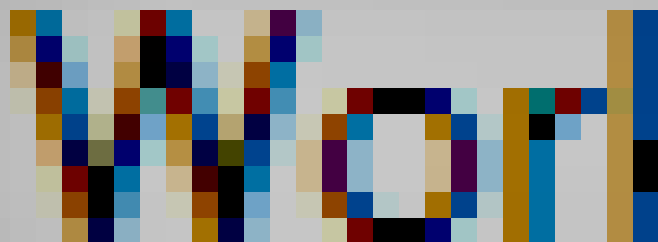
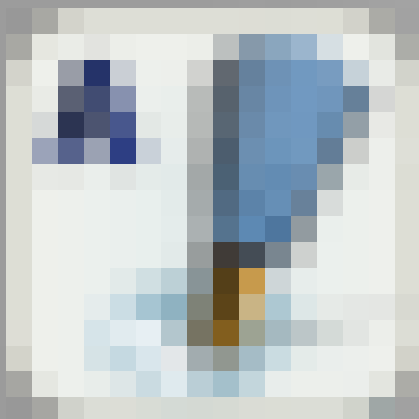
1

A

Auto5 Module with activated 5-Axis License

The above picture shows the **Auto5** module when you have a 5-axis license and when the workzone has been activated for 5-axis.

With an Auto3+2 License only or without any activated 5-Axis License



No.

1

2

Auto5 Module without any activated 5-Axis License

The above picture shows the **Auto5** module when you only have an **Auto3+2** license or when the workzone has not been activated for 5-axis.

This strategy consists in calculating a single toolpath according to several predefined views.

The views can be created automatically. In this case, you enter the number of views that you want to use for machining and their inclination by defining the Theta and Phi angle values. You can make the number of views match the increment value of the machine's rotating table. During calculations, the toolpath is divided into as many sub-toolpaths as there are views. An example of use of this strategy is for machining parts on big machines. Using views reduces the number of machine movements and facilitates the use of a rotating table.

Theta and Phi Angles

The opposite picture shows a view with a 12° **Theta** angle.

This theta inclination angle (**12.00** in our example) is calculated in accordance with the tangent of the tool and the holder.

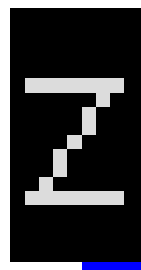
The **Phi 0** field is used to define the start angle from which views will be created. The **0.00** value means that the first view will be created starting at 3 o'clock, i.e. parallel to the X+ axis bar from a top view.

When the **Phi** value is set to 45°...

When the **Phi** value is set to 60°...

For example, if you create 3 views with a 75° **Theta** angle and a 0° **Phi** angle, the 3 views will be separated by 120°, starting from the **Phi** angle (360/3).

- A Auto5_0_75
- B Auto5_120_75
- C Auto5_120_75



5.1 - Programming an Auto3+2 Strategy

1. Create a workzone from the *multi-sided_part.xdw* file. For our example, do not activate the workzone for 5-axis toolpaths.
2. Create a **Z-Level Finishing** toolpath with the following parameters:
Expand Window by Radius + Stock: activated
Ball cutter, Body Radius: 2
Stock Allowance: 0
Tolerance: 0.01
Z-Step: 1
Vertical Lead-ins
Corner Smoothing Radius: 0
3. Validate the parameters, select the toolpath in the **Workzone Manager**, click on the



icon.

4. Select a tool holder (606373_07).
5. Activate the **Calculate Recommended Safe Tool Length** and **Create Collision Curve and Ranges** options.
6. Validate the parameters and run calculations.

You should obtain the following result:

The toolpath has several collision points. To reduce this, we are going to process the toolpath with the **Auto3+2** strategy in the **Auto5** module.

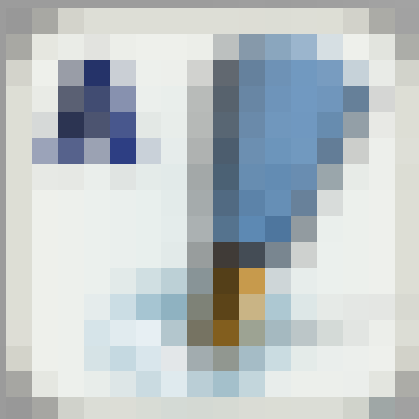
Running the Auto5 module

1. Make a copy of the toolpath.
2. Select the copy and click on the



icon.

The only available strategy is the **Auto3+2** strategy.



WORLD

No.

1

2

Auto5 Module without any activated 5-Axis License

3. Click on the **Auto3+2** button.

Defining Machining Views

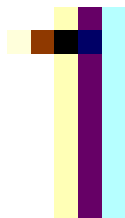
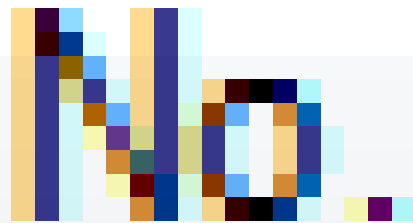
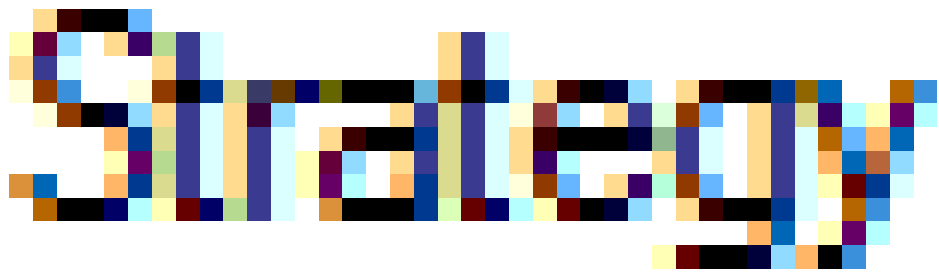
Machining View Limits

In the **Auto3+2** strategy, machining views are used to give a direction and have no influence on the machining area. No matter if the whole part is visible in the view or not.

Here, since we did not previously define machining views, they will be created automatically. Let's start by creating a vertical view.

1. Click on the **Vertical** button.

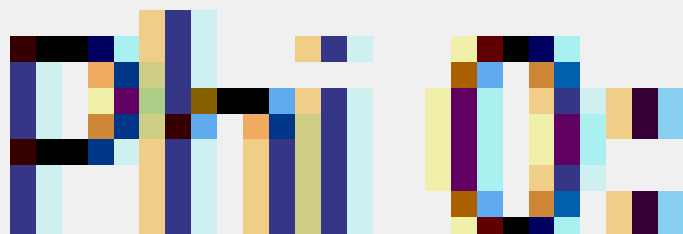
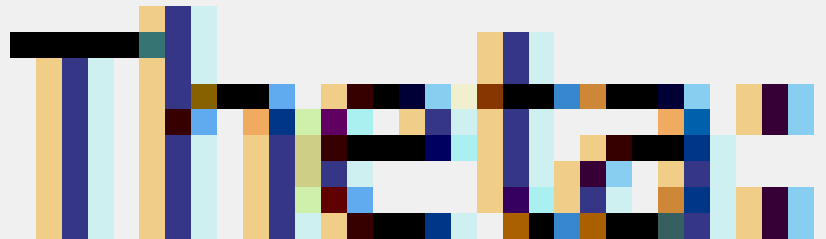
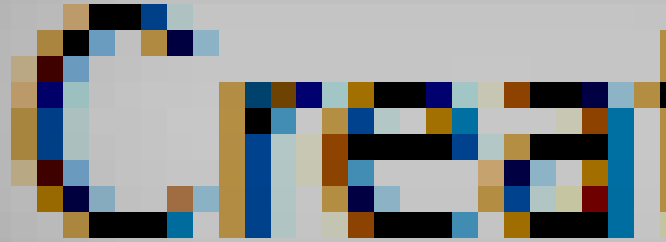
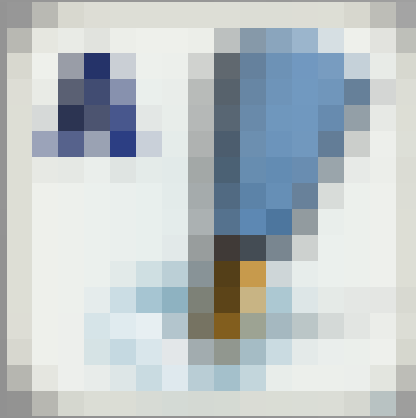
A vertical view is automatically added to the list of the views that will be used for machining.



Auto3+2 Strategy Interface - Vertical View

The vertical view will be created in accordance with the Z axis.

2. Click on the **Angle** button.



Auto3+2: Create Views

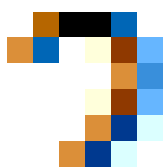
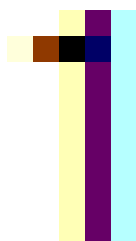
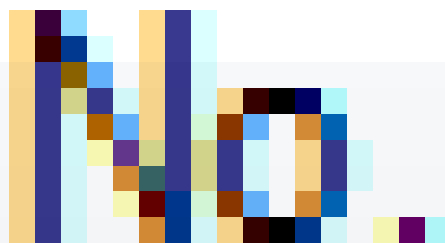
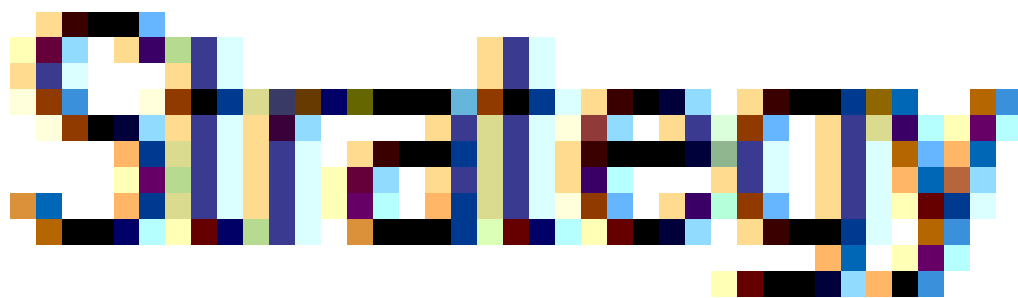
This dialog box allows you to configure the automatic view creation. For this exercise, we are going to use four additional views, each with a 12° inclination.

1. Enter **12** in the **Theta** field.
2. For our example, leave the 0.00 value in the **Phi** field.
3. Enter the number of views to be created in the **Number** field, 4 in our example.

This will create 4 views: when the **Phi** angle is set to 0, the first view is created along the X axis of the reference axis system and the second 90° further (360/4).

4. Click on the **Append** button.

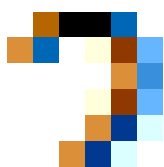
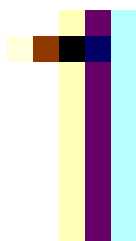
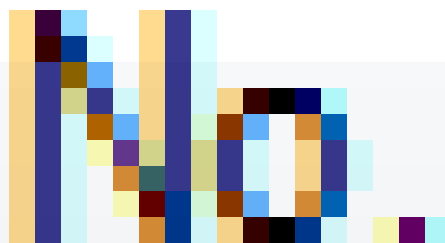
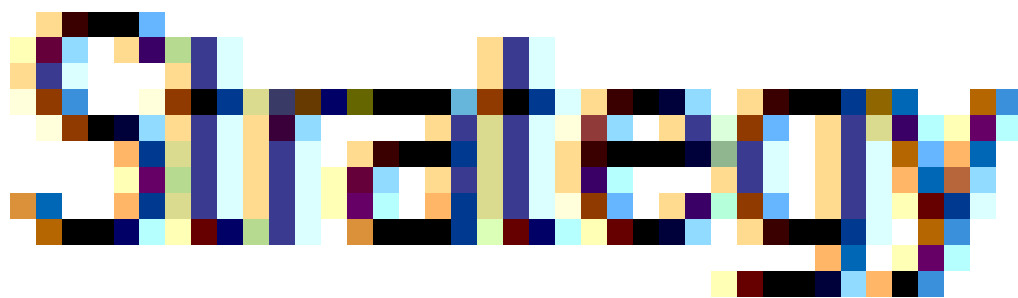
The **Auto3+2** user interface shows the list of each machining view that will be created. Note that views are named automatically. Their name always starts with **Auto5** and is followed by the **Phi** and **Theta** angle values.



Auto3+2: List of Views

5. Activate the **None** option in the **Priority of Views** section.

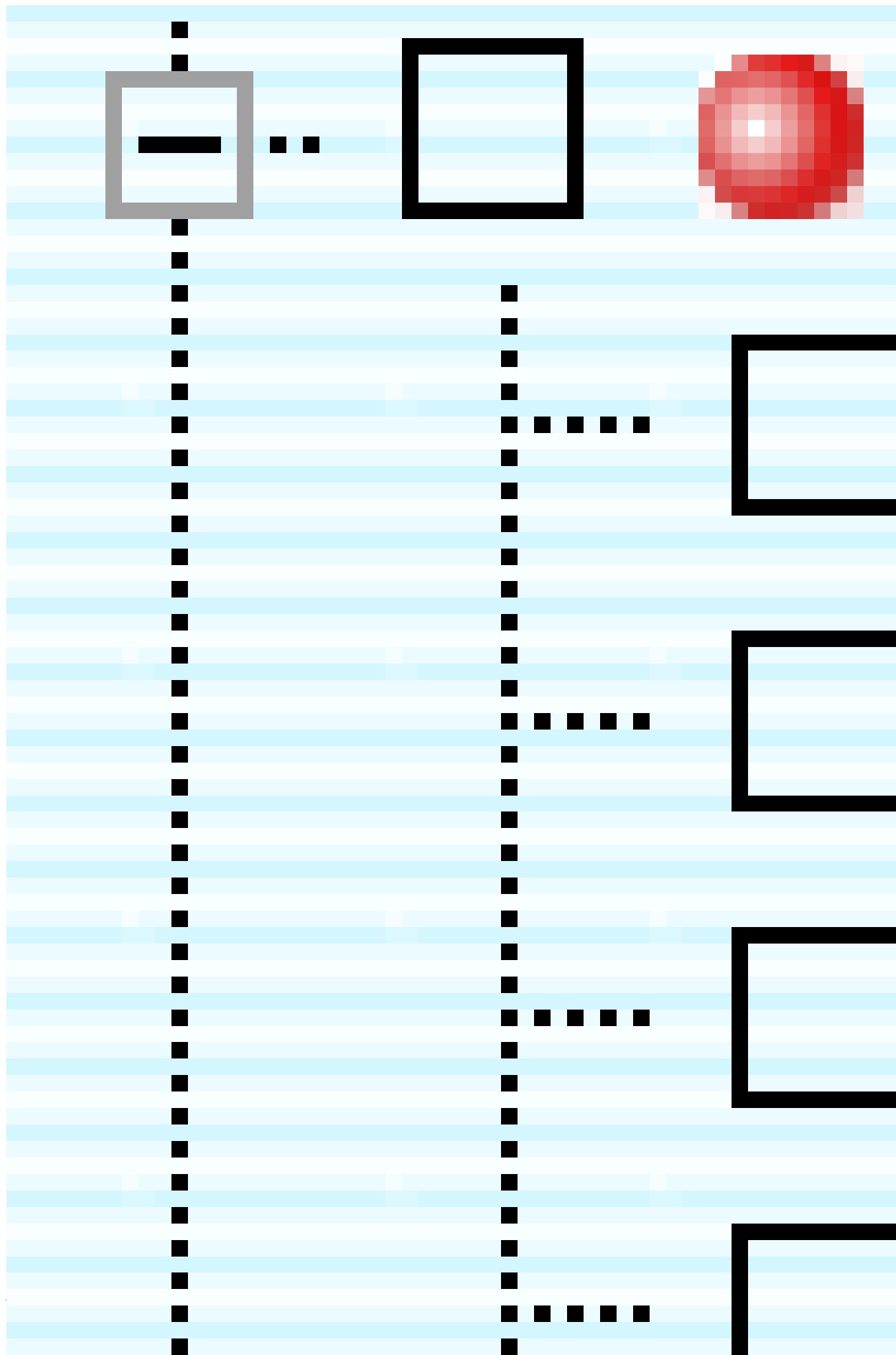
Views will be processed in their order of appearance in the list. You can change the processing order of the views in the list with the **Up** and **Down** buttons. You can also let **Auto3+2** determine the processing order by keeping the **Auto** option activated. You can also determine your own order by defining the number of each view in accordance with their priority order, starting from the number of the view which has top priority to the view that has the lowest priority. View numbers must be separated with a blank space, a semi-colon or a comma.



Auto3+2: Priority Order

6. Make sure that the **Append Colliding Regions for Toolpath Split** option is activated
7. Make sure that the **Split to 3+2 Sub Toolpaths** option is activated.
8. Enter 0 in the **Overlapping Distance** field. This will prevent the sub toolpaths from machining areas already machined by other sub toolpaths.
9. Click **OK** to validate and run toolpath calculations.

The resulting toolpath is divided into sub-toolpaths, each of them corresponding to a machining view. All the collision points have been inserted into an additional sub-toolpath at the end of the list:



Collision Sub-Toolpath

Adding Created Views to Auto3+2

Let's suppose to you want to add views that you have created to define the **Auto3+2** strategy.

1. Open the **View Creation** dialog box and activate the **View from Selection** function.
2. Click on the following planar surfaces as illustrated below:

VIEW C

20

1

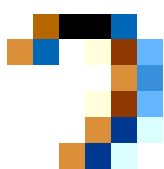
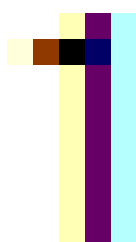
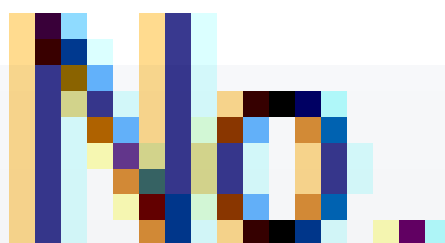
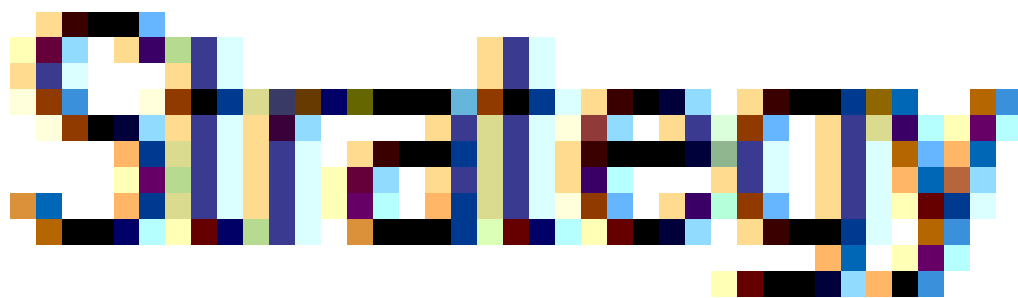
4 Views From Surface Selection

3. Make a copy of the initial toolpath and click on the



icon.

4. Select the **Auto3+2** strategy.
5. Click on the **Vertical** button.
6. Click on the **Select View** button.
7. In the **Select Views to Append** dialog box, select the views you have just created (TRN files). Use the [Ctrl] or [Shift] key for multi-selection and click on the **Open** button.



Added Views in Auto3+2

8. Make sure that the **Auto**, **Append Colliding Regions** and **Split to 3+2 Sub Toolpaths** options are activated.
9. Validate the strategy and run calculations.

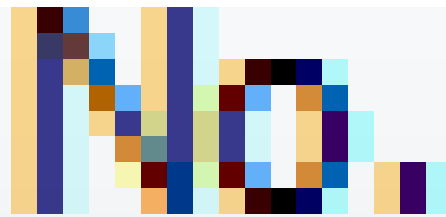
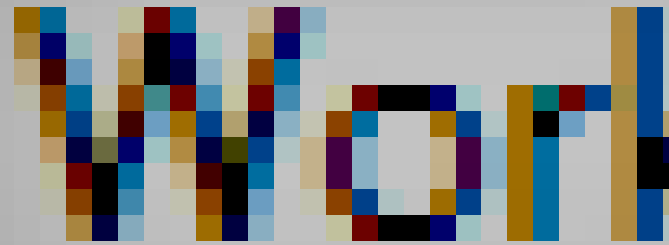
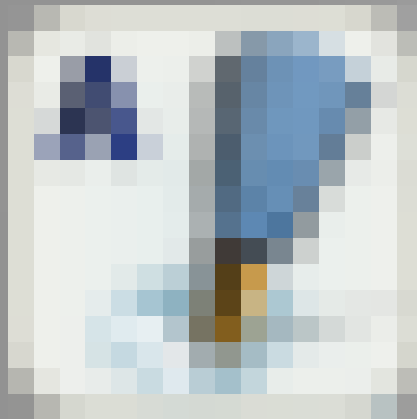
Auto3+2: Results with Created Views

There are no more colliding points in the toolpath. This is due to the orientation of the views that have been selected for the **Auto3+2** strategy.

Note that we would have obtained similar results by adding views (**Angle** button) with a higher **Theta** angle. Note also that one of the views has not been used by **Auto3+2**, as it was not necessary to complete the toolpath.

6 - Machine Limit Check

The lower part of the **Auto5** dialog box is dedicated to **Machine Limit Check (5 to Machine)** and allows defining several parameters.



4

A

Auto5: Machine Limit Check Section

It allows processing two types of 5-axis toolpaths:

- 3-axis toolpaths converted to 5-axis toolpaths by **Auto5**,
- Simultaneous 5-axis toolpaths.

In either case the output from this module ensures that all potential collisions are avoided by replacing these with retracts, approaches and lead-ins.

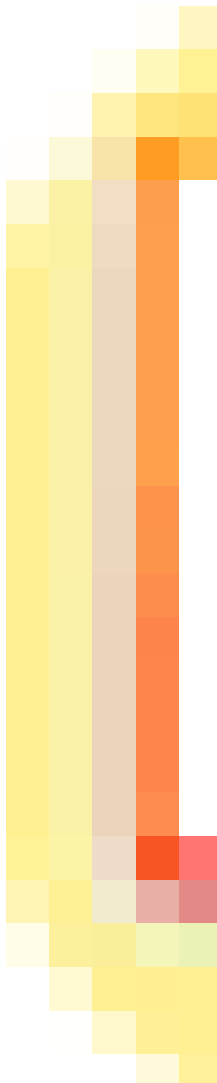
Press the [F1] key and search for the **Machine Limits Collision Check** section in the **Online Help**.

6.1 - Selecting a Machining Context

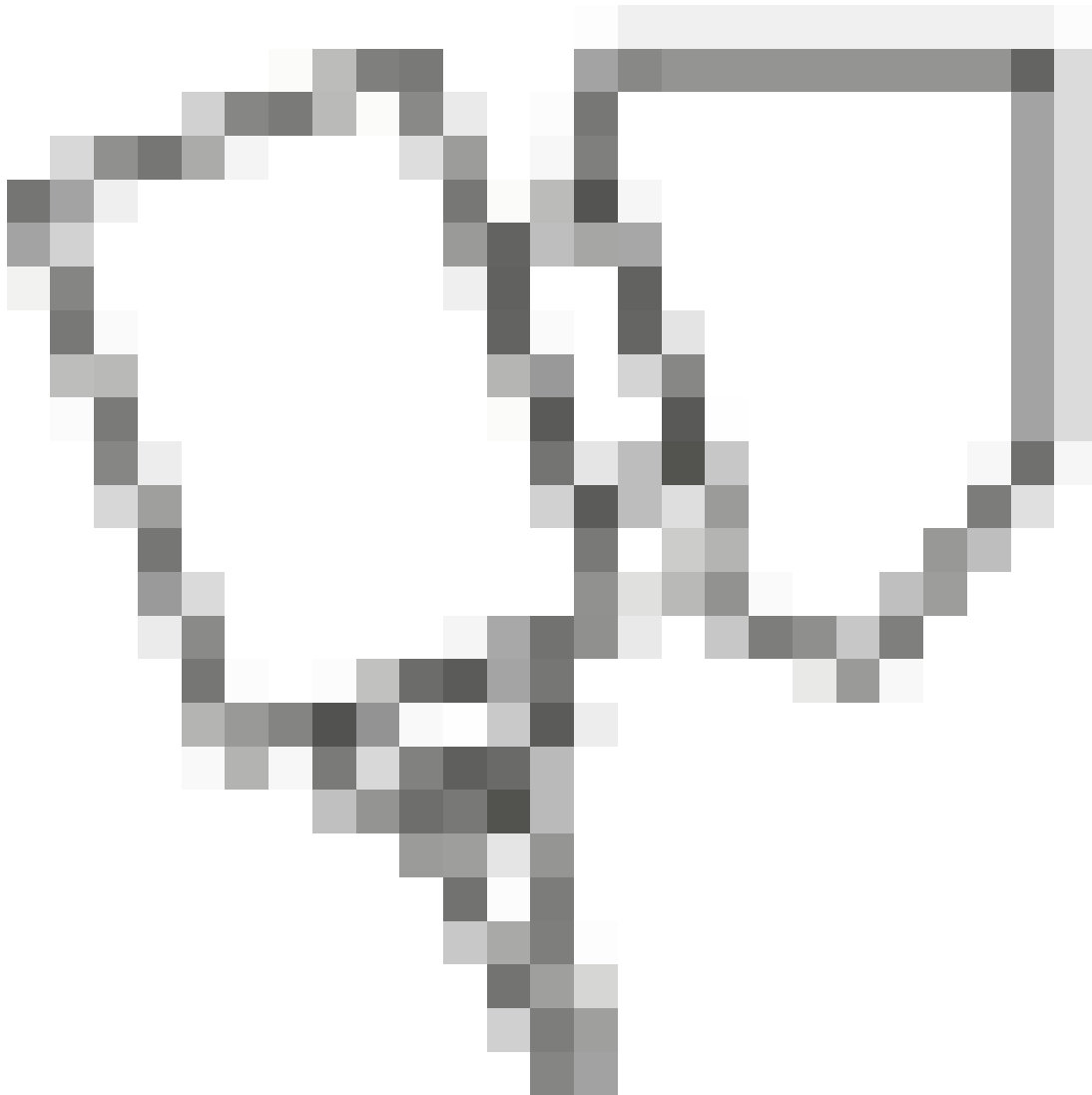
The **Machine Limit Check** section retrieves the machine configuration defined in the Machining Context of the selected toolpath. Press the [F1] key to open the **Online Help** and search for the **Machining Context – Introduction** section, or read the **Machining Contexts** training guide

Option Availability

If you already calculated a strategy (3 Axis to 5 Axis conversion), this section of the dialog box is no longer available unless you click on the **Recalculate** button.

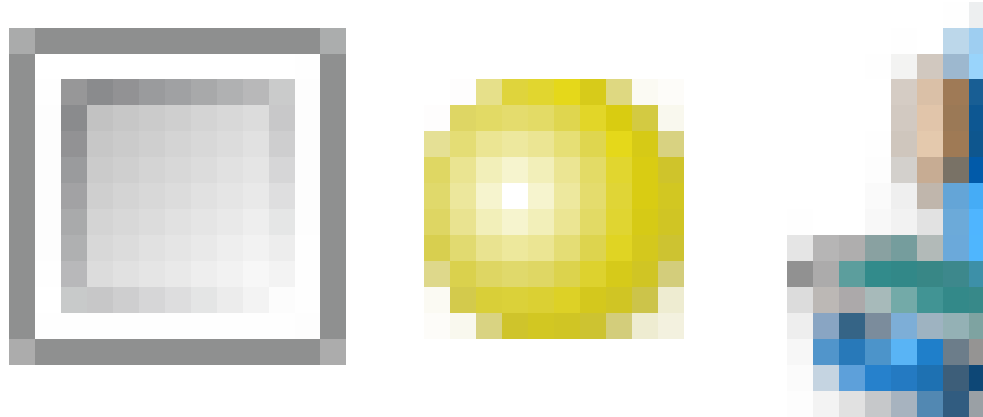
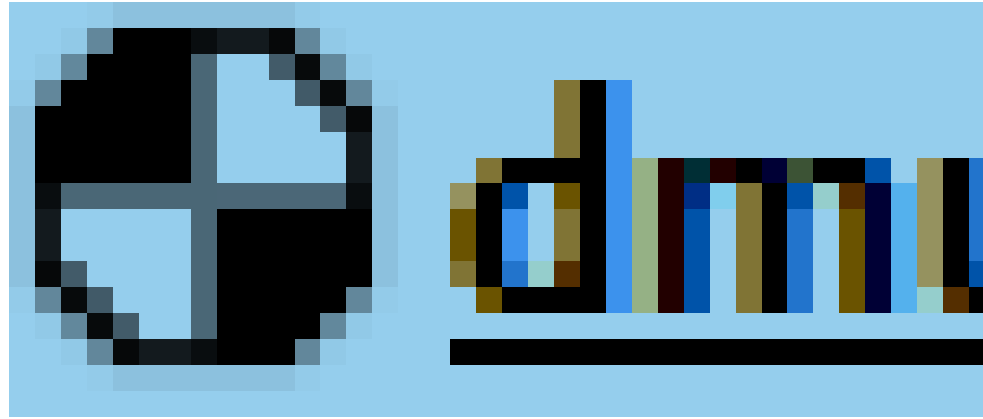


After having clicked on the **Recalculate** and on the **OK** buttons, a



icon is

displayed in the **Workzone Manager**, indicating that **Machine Limit Check** needs to be calculated:

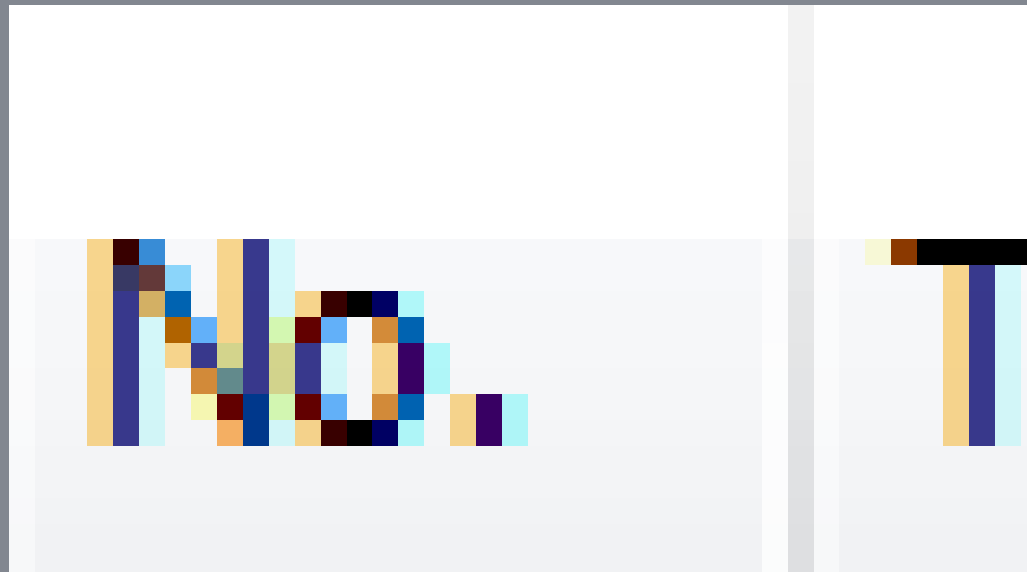
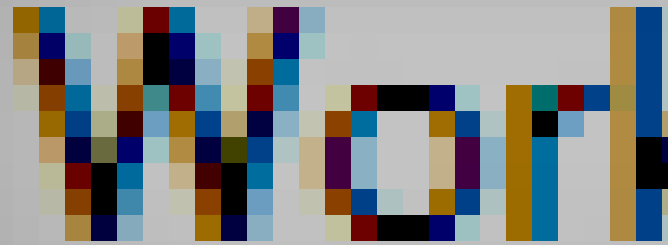


Workzone Manager: Machine Limit Check Ready to be Calculated

Percentage of Colliding Section Removed by 5 to Machine

Once you have calculated the **Machine Limit Check** function, you can open the **Auto5** module again and see the percentage of areas removed because of collisions.

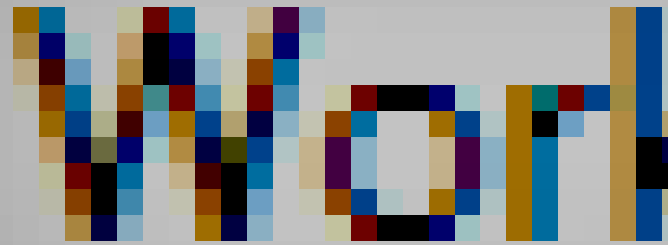
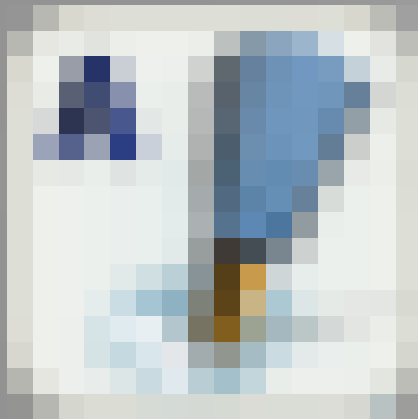
1. To illustrate this, open the *Auto5_examples* workzone.
2. Click on toolpath #8 and open the **Auto5** module.



No Machine Limit Check Calculation: Collision Percentage

This toolpath has not been calculated with **5 to Machine**. In this case, **Auto5** indicates the percentage of colliding sections of the toolpath, as illustrated above.

3. Click on toolpath #9 and open the **Auto5** module.



Percentage of Removed Section After Machine Limit Check Calculation

This toolpath is a copy of toolpath #8 that has been calculated with the **Machine Limit Check** function. In this case, **Auto5** indicates the percentage of removed colliding sections of the toolpath, as illustrated above.

6.2 - Defining the Main Settings

Once you have selected your machine, you can define the main setting to process a 3 to 5 axis toolpath or a simultaneous 5-axis toolpath taking into account the angle limits of the machine and other associated parameters.

Save Non Colliding Section only

This option allows you to keep only non colliding sections of the toolpath after machine limit check calculation.

Save both Sections

This option allows you to keep both the colliding and the non colliding sections of the toolpath after machine limit check calculation. If you choose to save both sections, the collideded section will be saved as a 3-axis toolpath.

Overlapping Distance

When you activate the **Save both Sections** option, you have the possibility to apply an overlapping distance to the colliding section. The default value is equal to the tool radius defined in the parameters of the selected toolpath.

- To illustrate this, open the *Auto5_examples* workzone and compare toolpaths #13 and #14.

Toolpath #13 has been calculated with a 5mm overlapping distance whereas toolpath #14 has been calculated with 1mm overlapping distance.

Clamping Axis System

This drop-down list allows you to select the reference axis system for **Machine Limit Check** calculations.

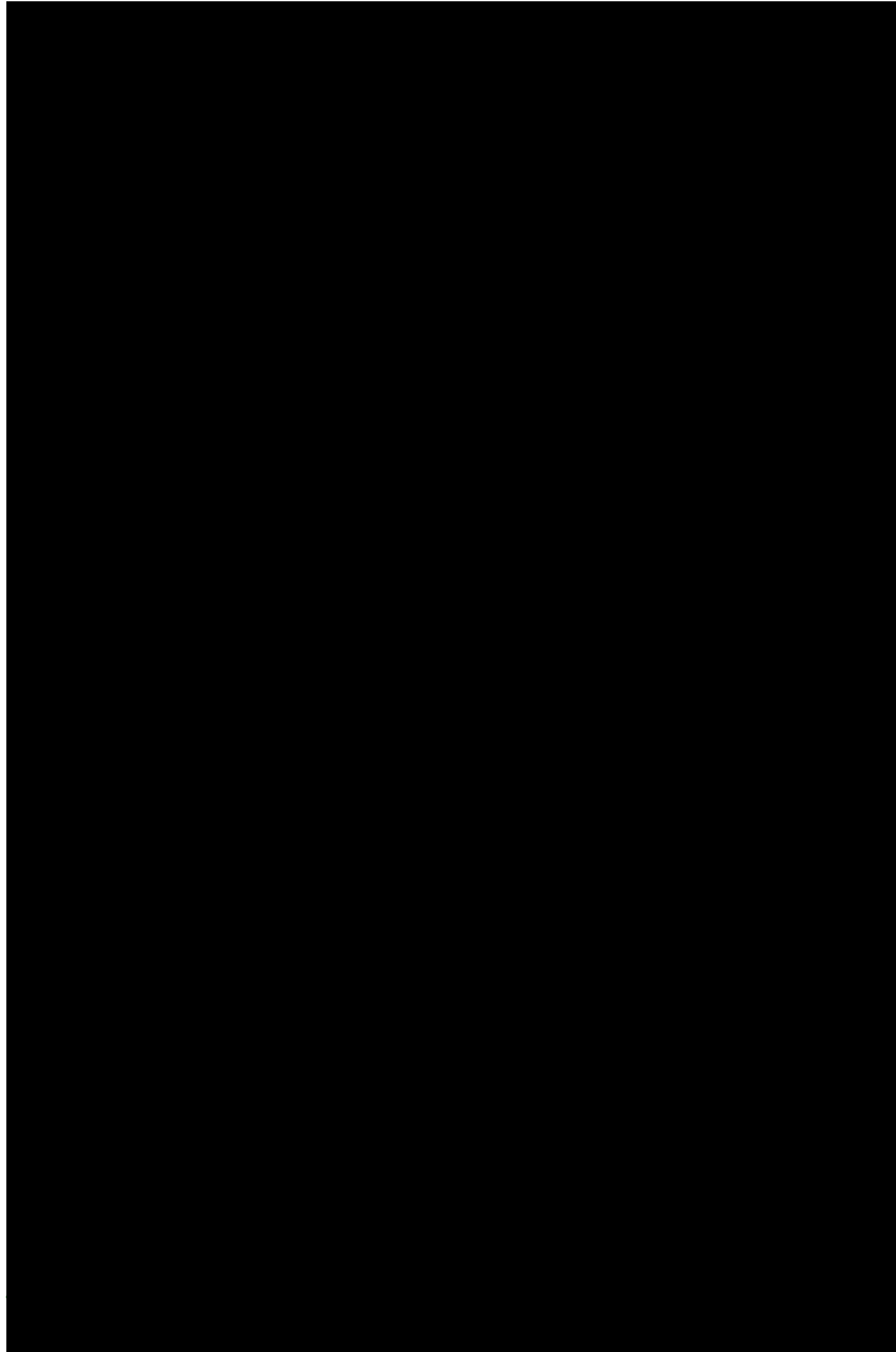
Remove Unnecessary Points

This option allows you to remove any consecutive toolpath points which have the same cutter axis vectors. This option is activated by default.

Retract Axis Movements in Retract End Position

This parameter is used to determine how the tool axis is moved to a new position during retract movements.

If this option is activated, the machine moves to the destination point of the next lead-in at the retract security level while maintaining the same tool axis as when the lead-out was performed. Once the destination point has been reached, the tool axis is moved to the new position at one single toolpath point before the lead-in is performed. The following screenshot illustrates this strategy.



Retract Axis in Retract End Position Option Activated

1+2 Cutter axis modification at retract destination point

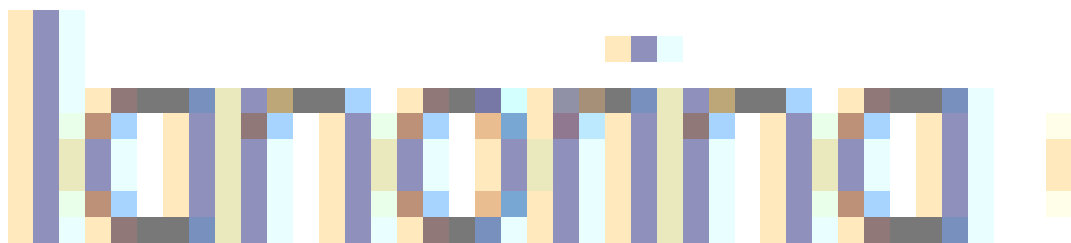
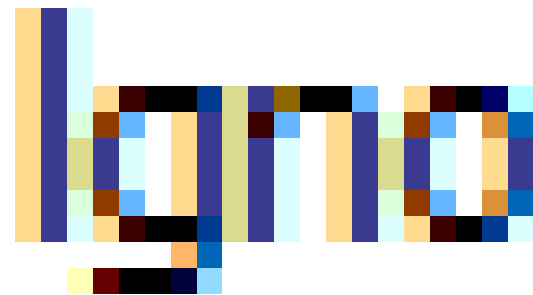
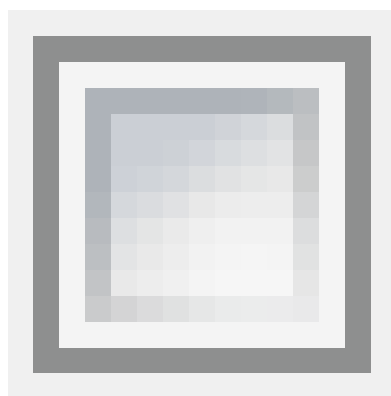
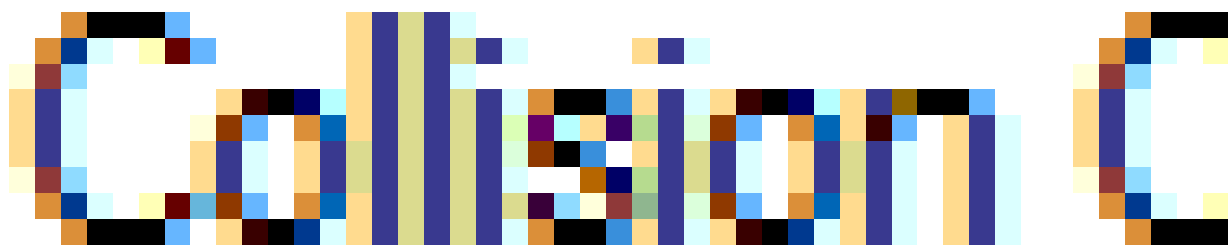
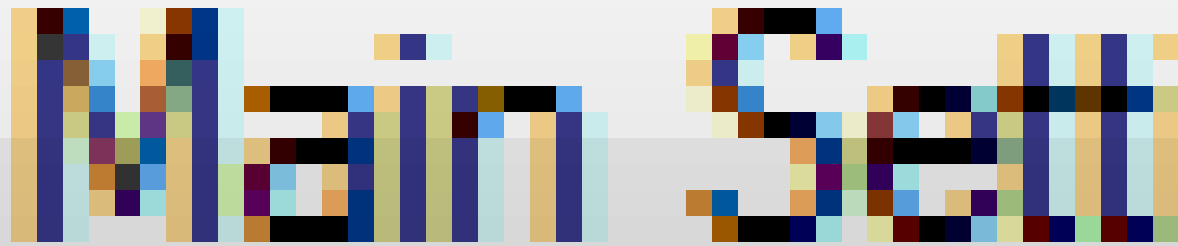
If the option is not selected the machine changes the tool axis progressively at the retract security level between the lead-out position and the following lead-in position as illustrated below.



Retract Axis in Retract End Position Option Deactivated

Using this option allows you to save time on machines with low speed rotational capacities.

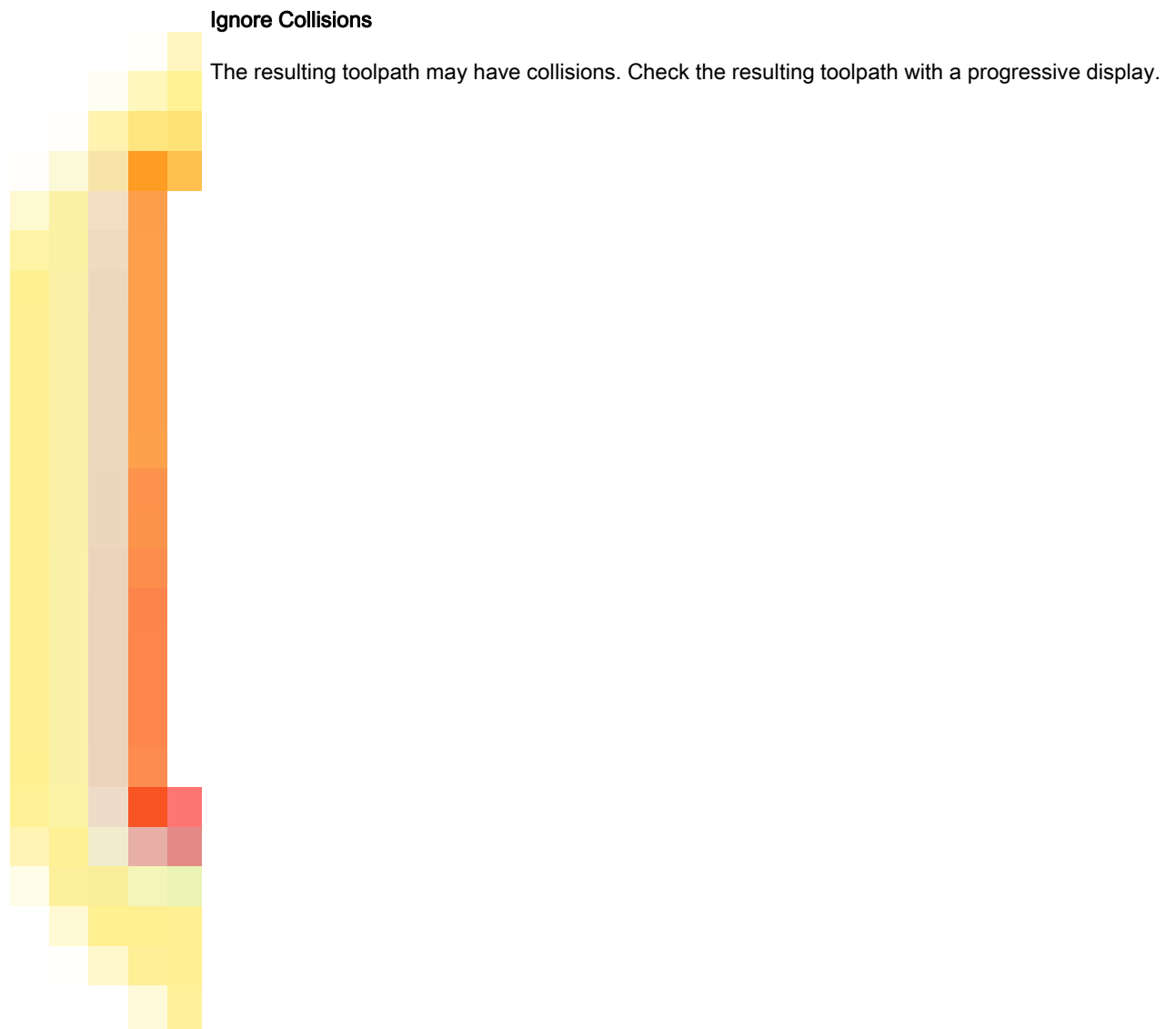
6.3 - Defining Additional Settings



Machine Limits Check Parameters - Additional Settings Tab

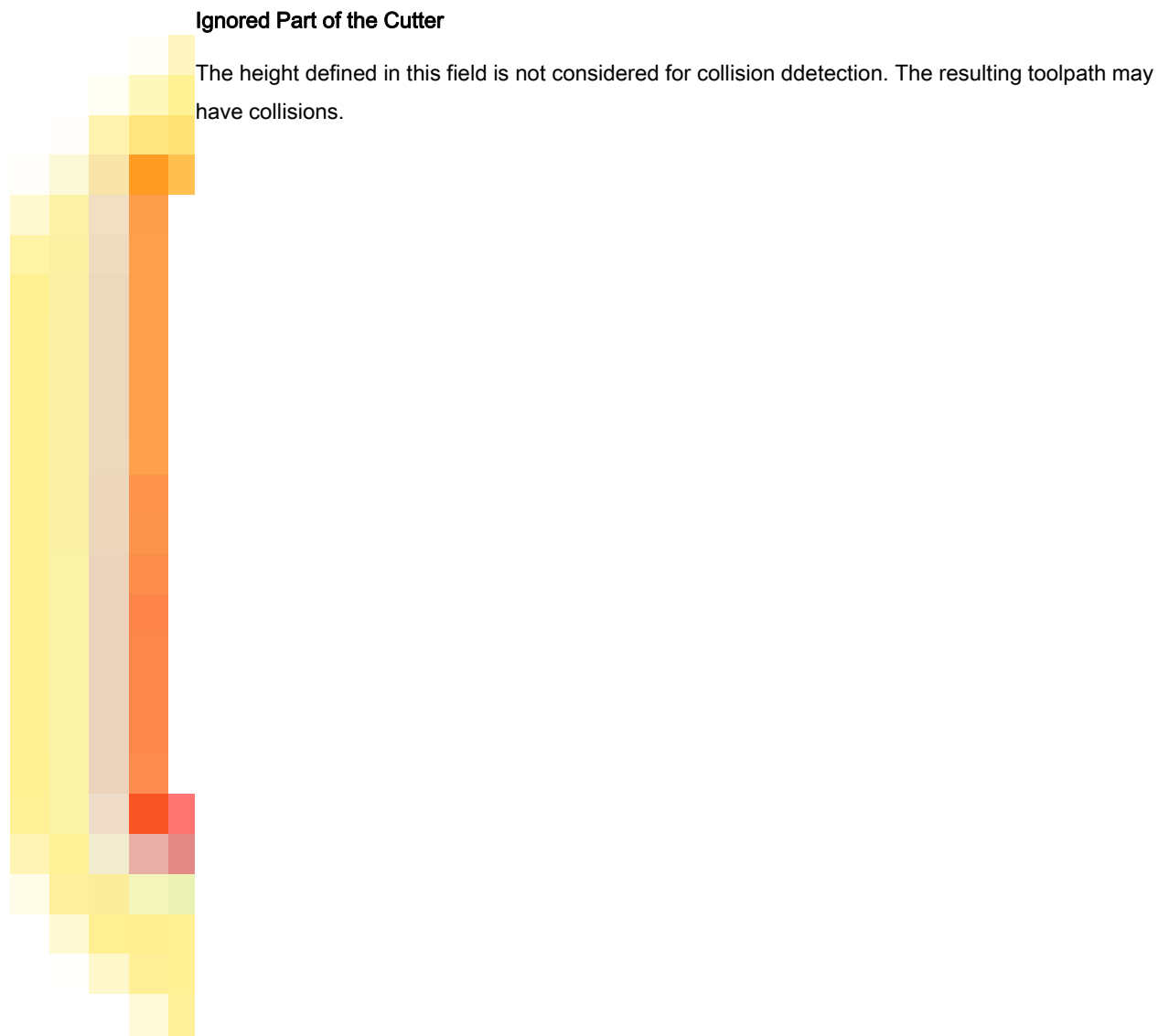
Collision Control/Avoidance

Activating the **Ignore Collisions** option allows you to ignore the collisions detected during the 3 to 5 conversion calculations.



Considered Cutter for Collision Avoidance

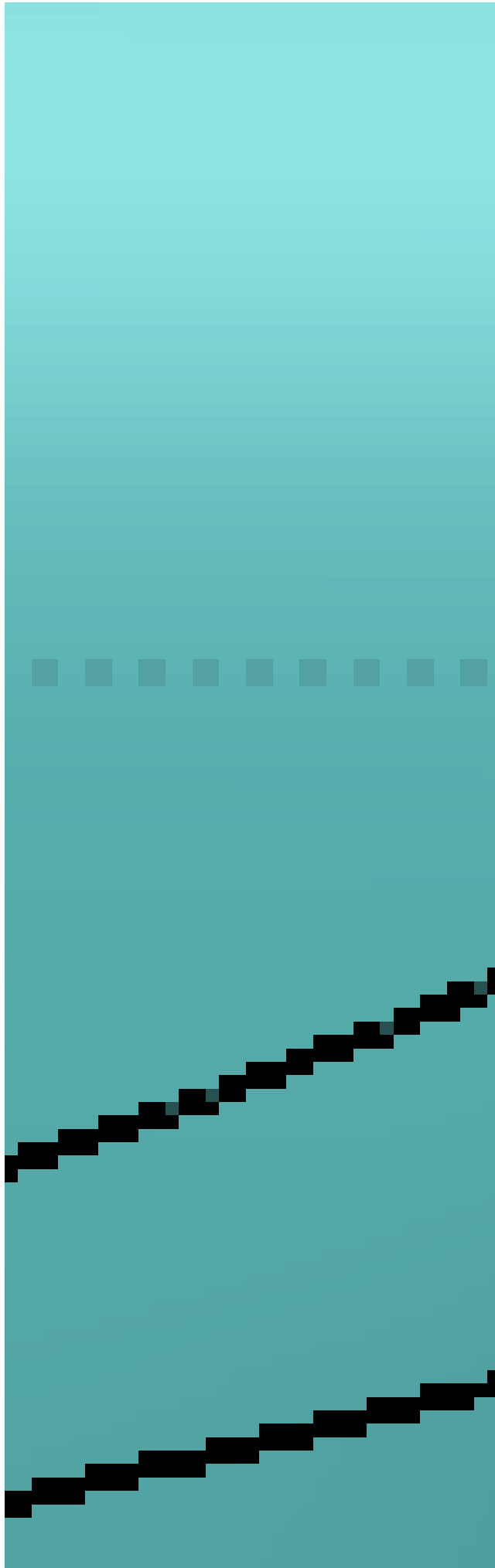
Activating the **Ignore Cutter below Height** option allows to define the height of the cutter to be ignored for collision detection.



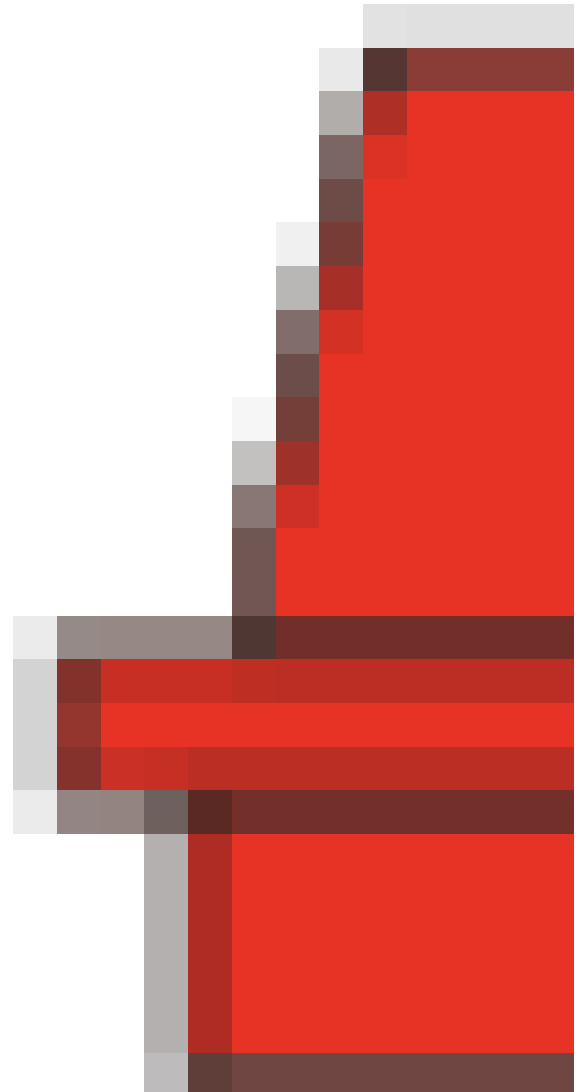
You can use this option with 5-axis toolpaths that have been calculated with a negative offset distance.

- To illustrate this, open the *Auto5_examples* workzone and compare toolpaths #11 and #12.

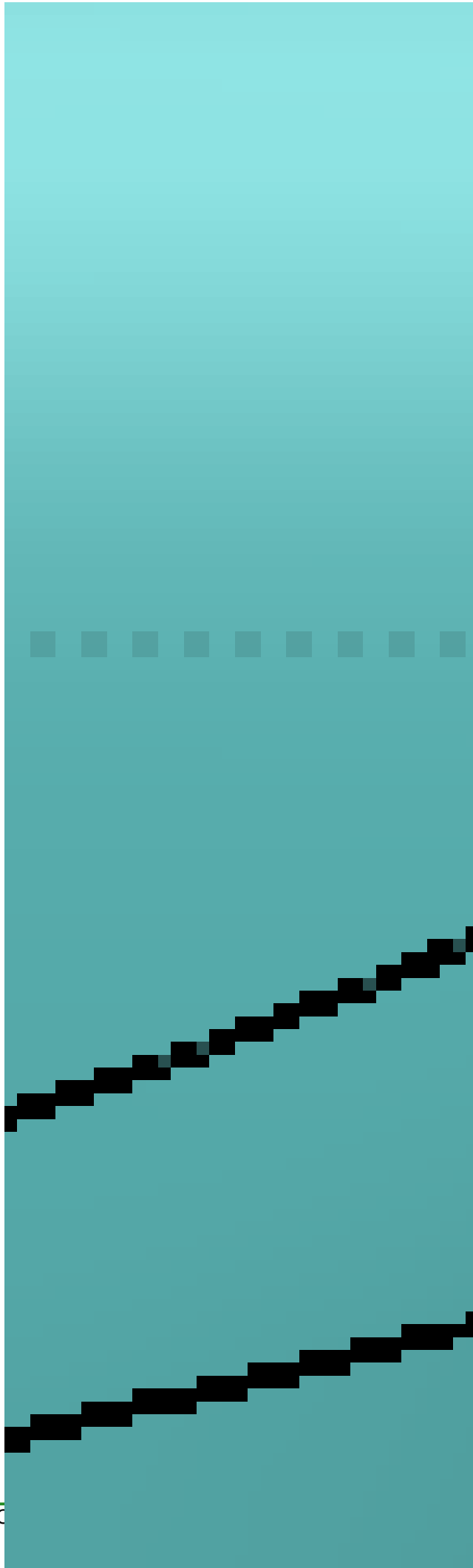
Both are copies of a **5-Axis – Curve Machining** toolpath (#10) that has been calculated with a negative **Tool Axis Offset** value.



The opposite example shows toolpath #11, where the **Ignore Cutter below Height** option is deactivated. Areas with the negative offset are considered as colliding section. You cannot generate the NC file with this toolpath. Besides, the



icon is displayed in the **Workzone Manager** for the corresponding toolpath.



The opposite example shows toolpath #12, where the **Ignore Cutter below Height** option is activated. Areas with the negative offset are ignored. You can therefore generate the NC file with this toolpath.

Task When Reaching Machine Rotation Limitation

This option allows you to determine the type of movement made by the machine when reaching its rotational limits.

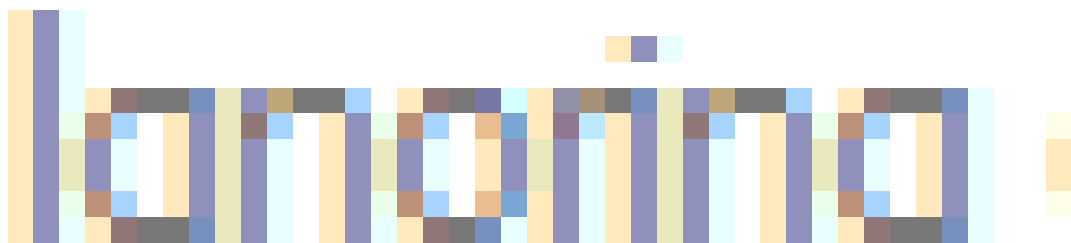
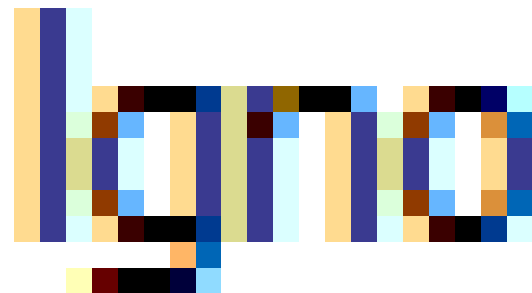
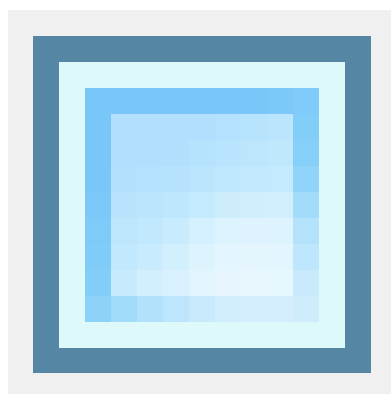
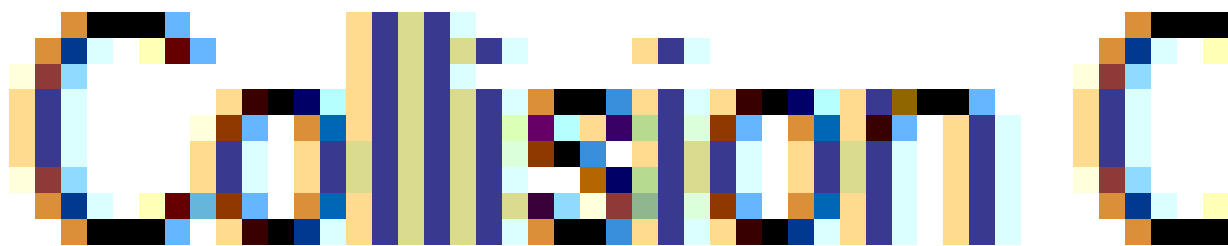
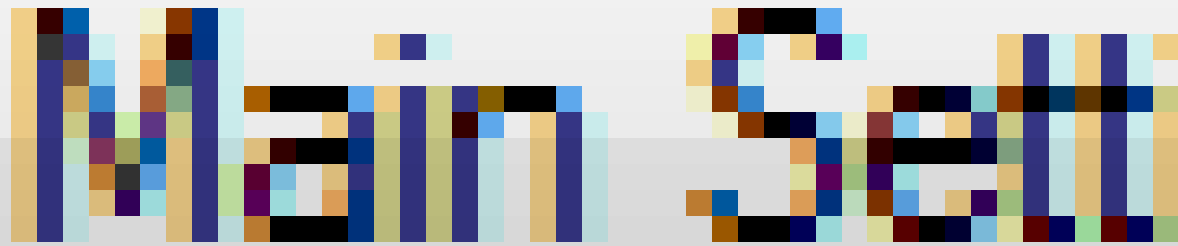
You can select the **Relaxing** option if you are using strictly limited machines or for Z-level finishing toolpaths.

You can select the **Flipping** option if the direction of the rotational axis is constantly changing, as is the case in planar finishing strategies.

You can select the **Automatic** option to let the system choose the best movement or if you use machines without rotational axis limitations.

Rotation Axis Adjustment if C Axis is Vertical

This option allows you to define a rotation movement when the machine limits are reached in the A or B axis. By default, it is deactivated (**Off**). You can choose between 3 rotation angles (90, 180 or 360 degrees). The **Link Distance** is the distance over which the rotation will be executed.



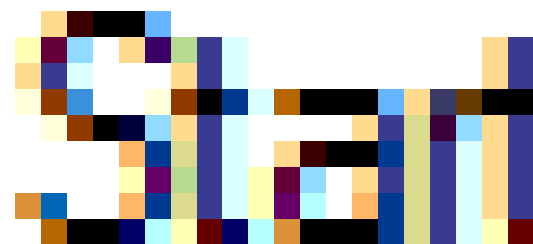
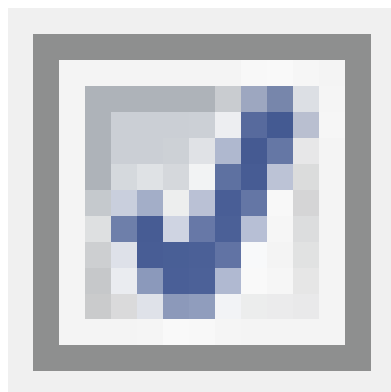
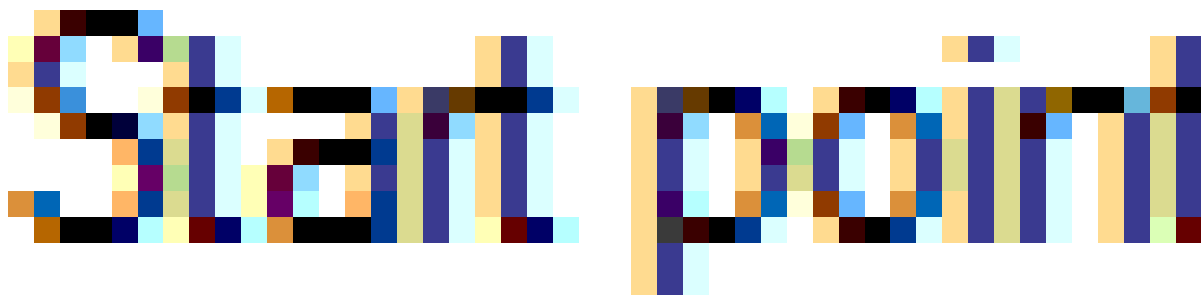
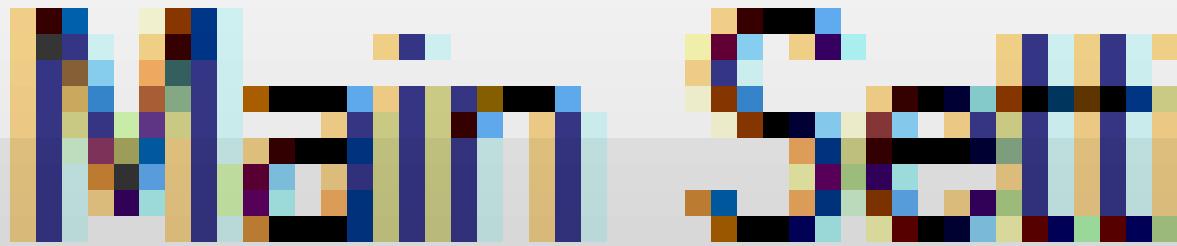
Rotation Axis Adjustment if C Axis is Vertical

6.4 - Defining Start and End Points

After defining the machine you want to use, you can define the start and end points of your tool. These points are actually added to the toolpath and can be viewed during the toolpath simulation.

Start Point

- Click on the **Start** tab.



Machine Limit Check Parameters - Start Tab

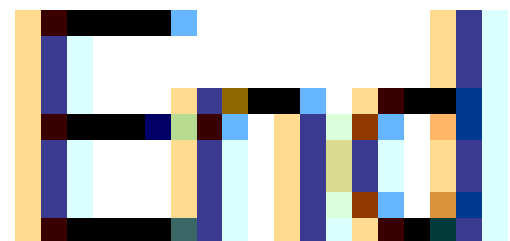
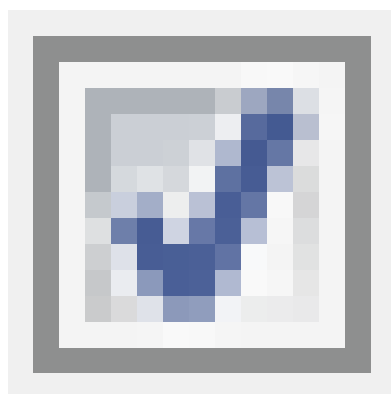
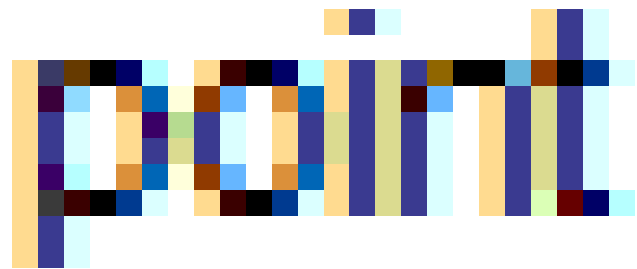
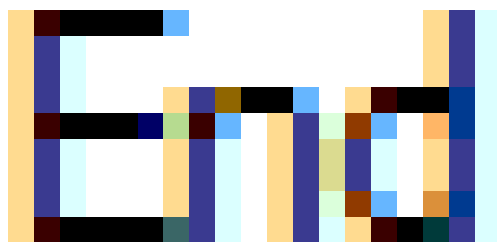
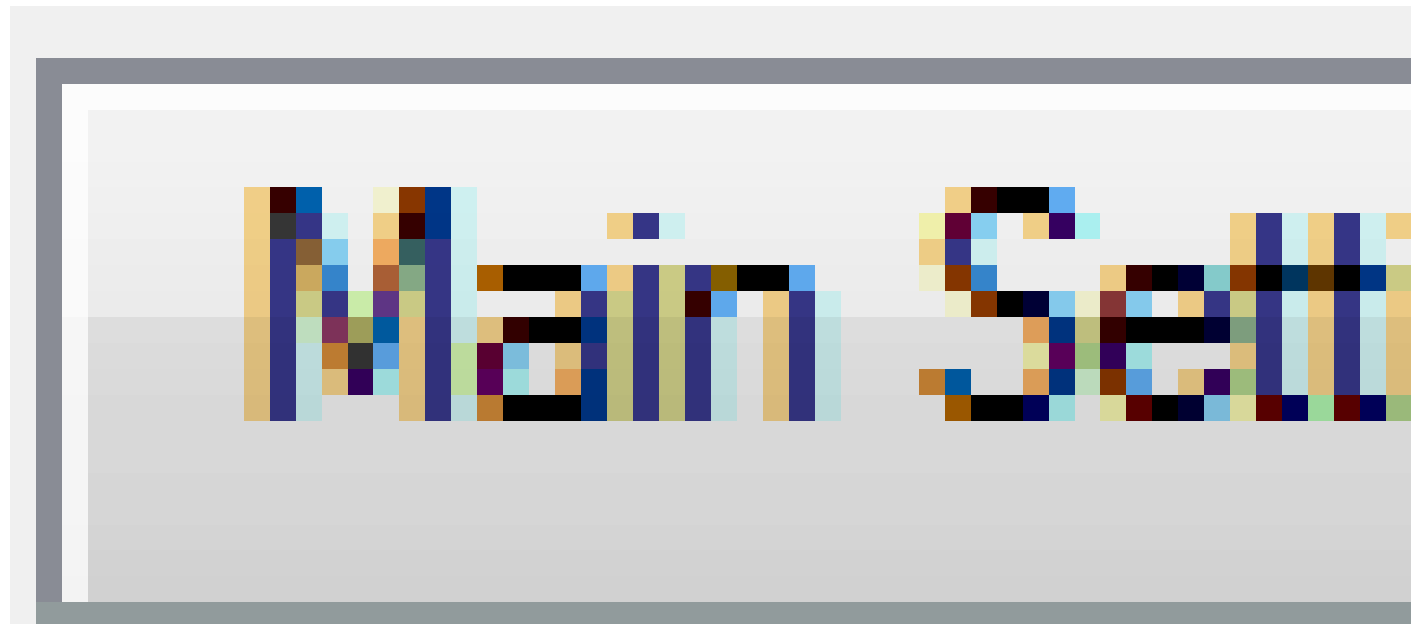
By default, the **Z over Part** option is activated, which means that the initial position of the cutter is set to the highest point of the part.

Activating the **Start Point** option allows you to determine the initial position of the cutting tool for each toolpath in the X, Y and Z axes with respect to the reference axis system.

Activating the **Absolute Z** option will force the input data to be an absolute value, if not activated the value will be relative with respect to the highest point of the part.

End Point

- Click on the **End** tab.

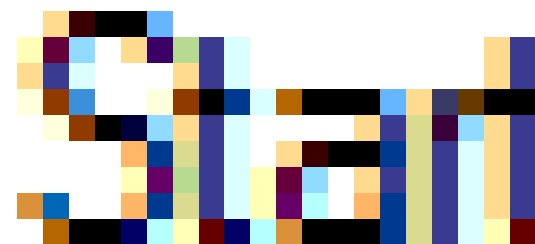
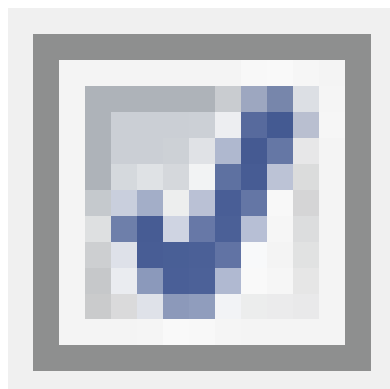
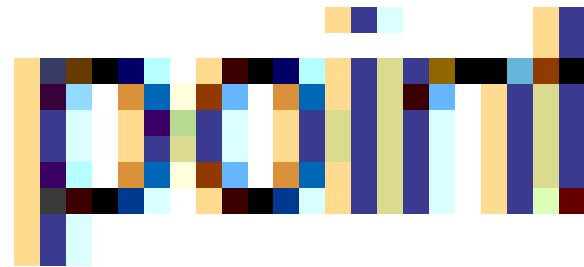
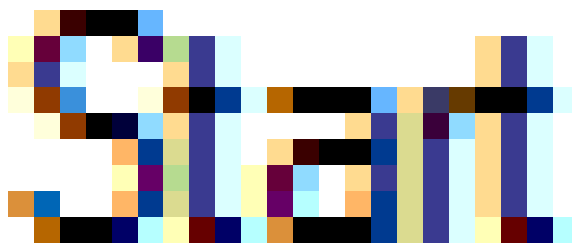
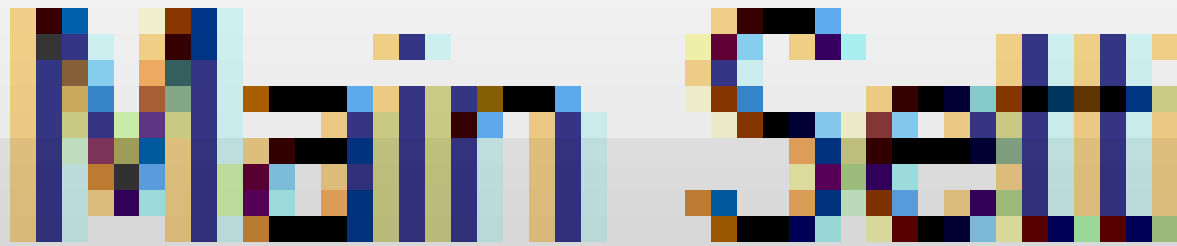


Machine Limit Check Parameters - End Tab

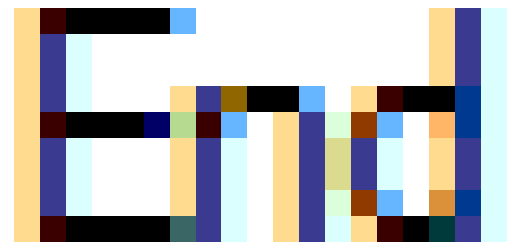
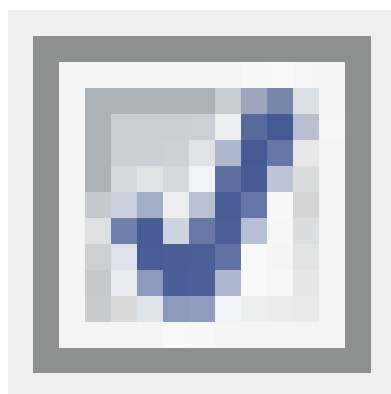
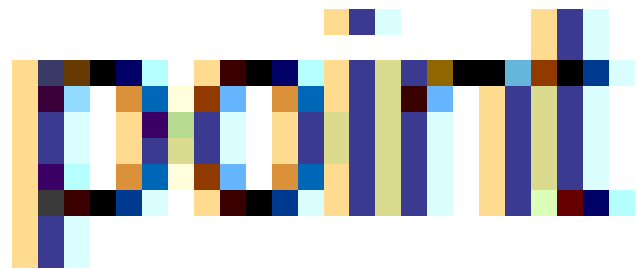
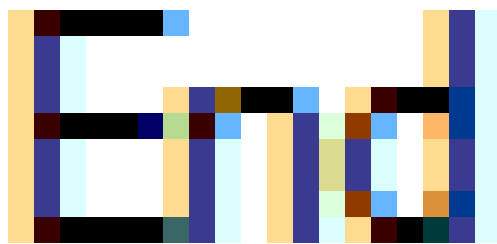
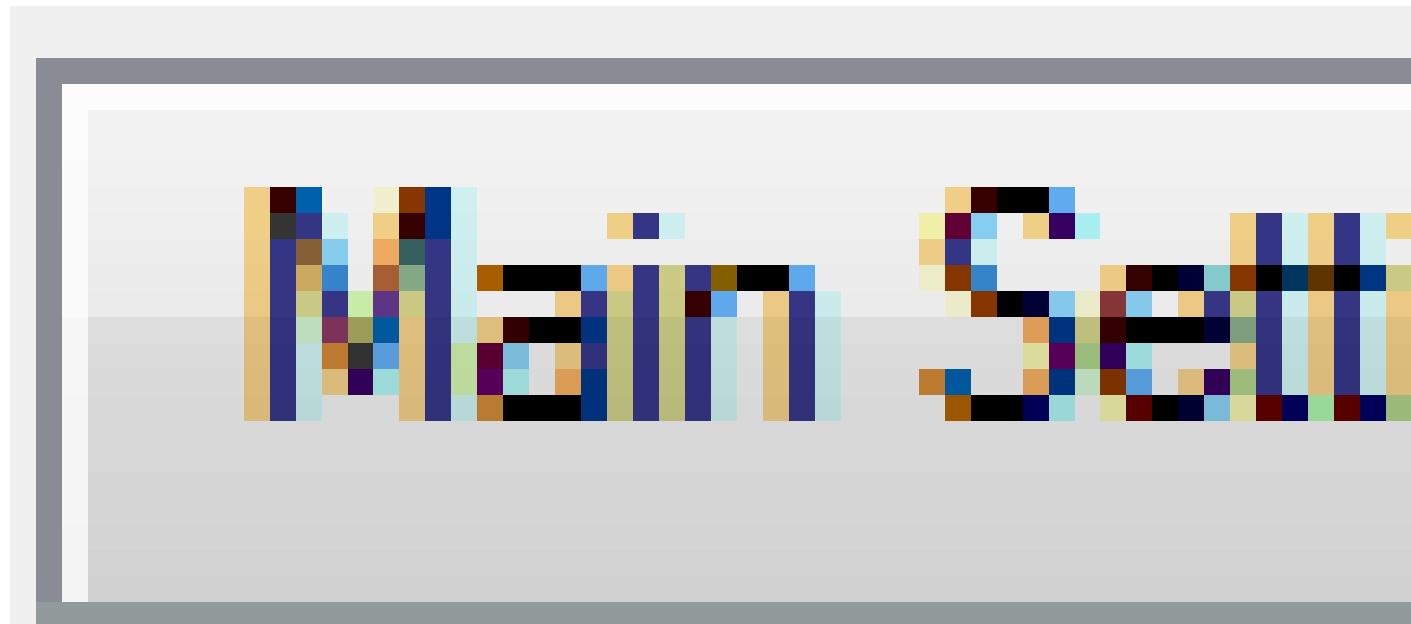
Activating the **End Point** option allows you to determine the final position of the cutting tool.

6.5 - Defining Start and End Angles

With the **Auto 5** module, you are free to define specific cutter orientation start and end positions on the machine.



Machine Limits Check Parameters - Start Tab



Machine Limits Check Parameters - End Tab

For your information, the relevant axes for the selected machine, along with their rotational limits, are displayed in the menu. Any axis which does not apply to the selected machine is grayed out and marked "****unused!***".

View Direction

The cutter orientation position is in the direction of the view that was used to calculate the toolpath.

User Defined

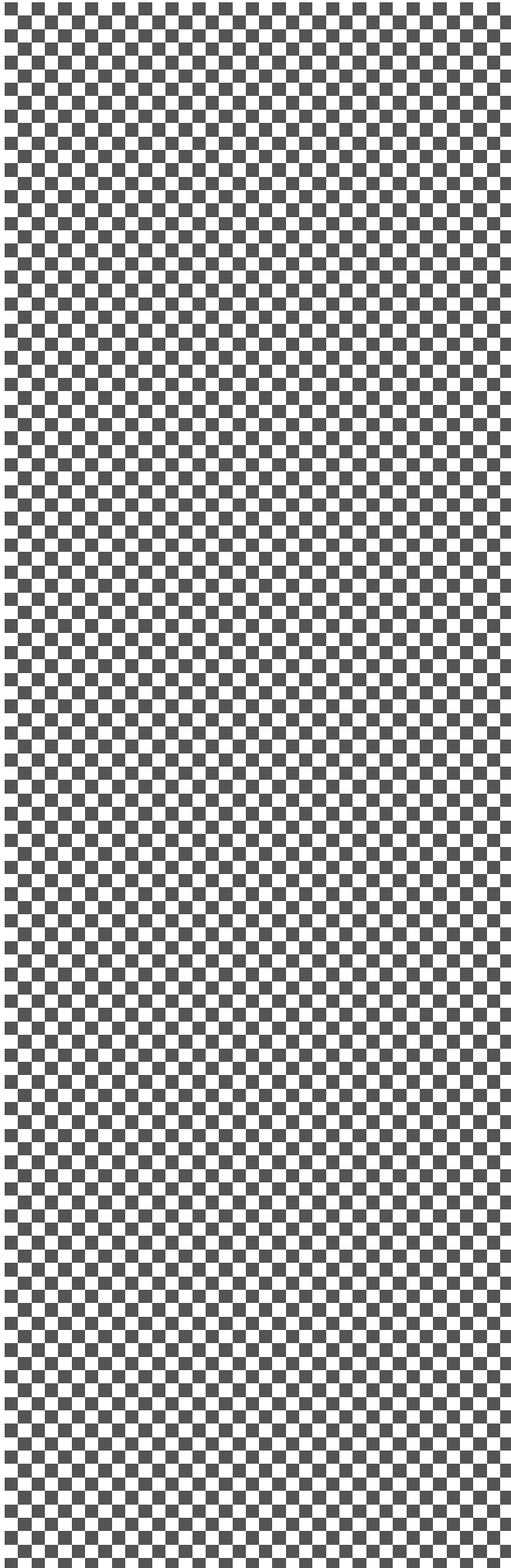
Cutter orientation is defined by the user (within the limits of machine axis orientation).

First/Last Axis of the Toolpath

Initial/Final cutter orientation is determined by the first axis of the toolpath.

Activating the **Use Start Angle** option positions cutter orientation at the end of the toolpath in the same position as the first point on the toolpath.

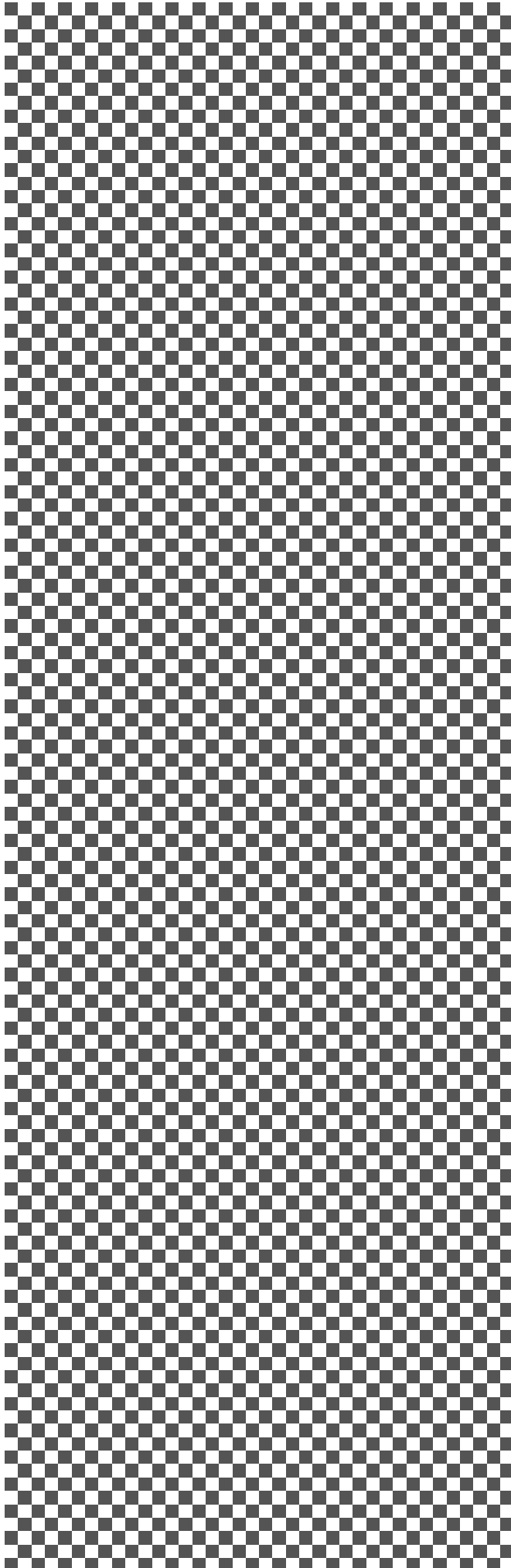
- Open the *Start-end_angles* workzone and compare the toolpaths # 3, 4 and 5. The toolpaths were calculated with different start and end angles and the machine selected was **C600U**.



Toolpath # 3

Start angle = First axis of the toolpath

End angle = Start angle



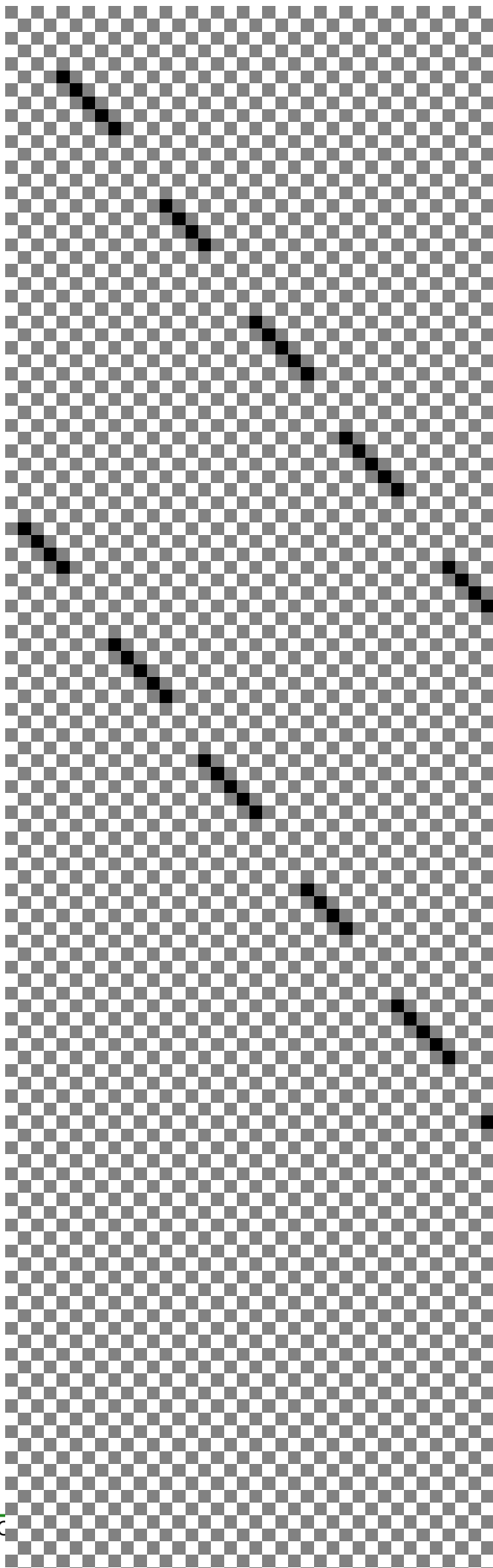
Toolpath # 4

Start angle = User defined

C = 0.0°

End angle = User defined

C = 90°



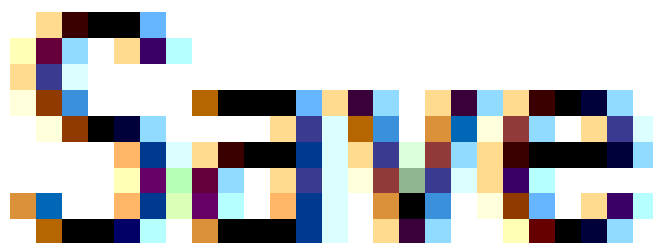
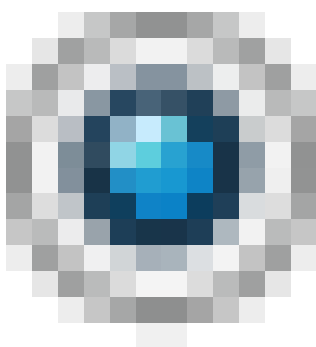
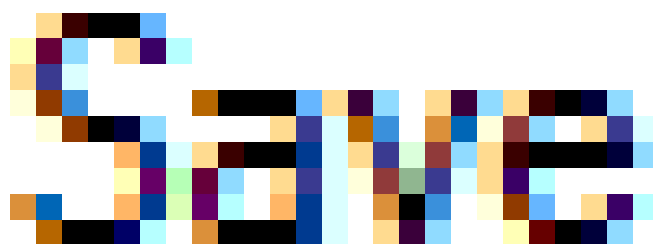
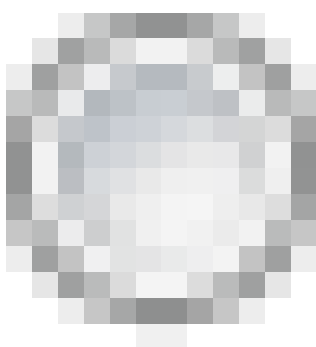
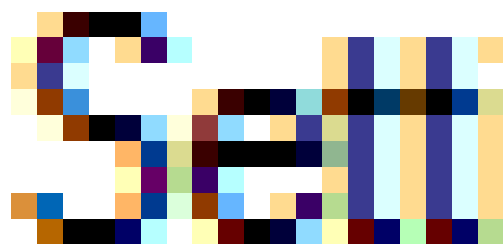
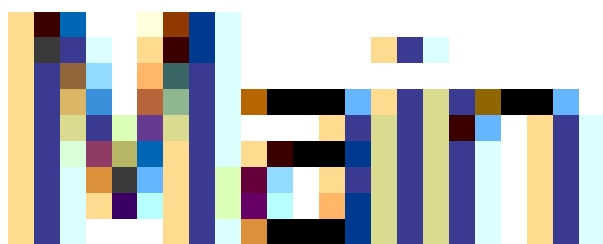
Toolpath # 5

Start and end angles = View Direction

6.6 - Saving Machine Settings

Once you have defined the main and additional settings, you have the possibility to save these settings for the corresponding Machining Context. These settings will be available for other toolpaths in any other workzone using the same Machining Context.

1. Define your **Machine Limit** settings and click on the **Save** button in the **Main Settings** tab.



Main Settings Tab

2. Activate the **Auto5** module for another toolpath with the same Machine Context (you can select it from another workzone).
3. Click on the **Recalculate** button in the **Machine Limit Check** section.
4. Click on the **Load** button in the **Machine Defaults** section.
5. Click on the **Yes** button in the pop up message to validate.

The settings that have been saved for the Machine Context are automatically defined and you just have to validate the strategy.

You can overwrite the settings you have defined:

1. Define new settings.
2. Click on the **Save** button.

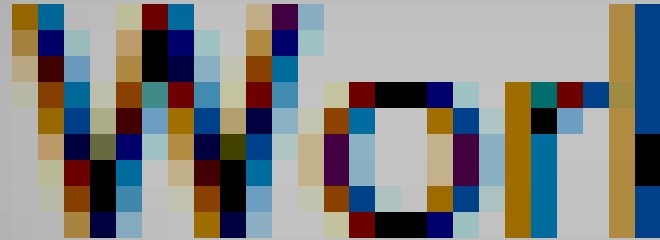
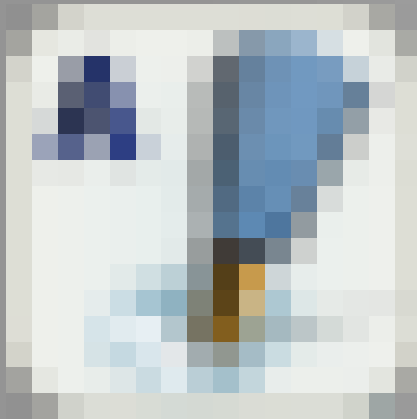
You can delete the settings you have defined:

1. Click on the **Delete** button.
2. Click on the **Yes** button in the pop up message to validate.

The settings are no longer available for the Machine Context for any toolpath in any workzone.

6.7 - Defining Advanced Parameters for the Machine Limit Check Function

In the **Machine Limit Check** section, the **Advanced** tab contains parameters which allows **Auto5** to recalculate the retracts.



No.

1

T

Machine Limit Check: Advanced Tab

To illustrate this, we are going to compare the various toolpaths in the *Auto5_retract* workzone.

Retract Modification

- Display and compare toolpaths #2 and #3. Both are copies of toolpath #1 which have been recalculated with the **Machine Limit Check** function.

Toolpath #3 uses the original retracts: the **Use Auto5 Retracts** option has been deactivated.

Toolpath 3: Original Retracts

For toolpath #2, the **Use Auto5 Retracts** option has been activated.



Toolpath 2: Auto5 Retracts

5-axis users prefer to avoid the machine being heavily tilted when the cutter is close to the part geometry. Some machines are also very slow if retract movements are continuously moved and tilted, therefore Auto5 retracts are rectangular.

Reference System for Retract Movements

These options allow you to define retracts movements according to a View or a Machining Context.

- Display and compare toolpaths #4 and #5.

The retracts of toolpath #4 follow the View direction:

Retracts Following View Direction

The retracts of toolpath #5 follow the Z axis of the Machining Context:

Retracts Along Machining Context Z-Axis Direction

Usually the main retract direction is parallel to the view. However, in some situations and because of specific kinematics, it is preferable to perform retracts differently.

Movements between 2 Positions

These options allow you to define how the angle transition between two tilts is performed. They are particularly useful to optimize retracts when having high spindle movements. Their effects highly depend on the type of machine you are using.

Spherical Movements

In the above example which uses the **Spherical Movement** option, the A angle is maintained and the C angle evolves. This is usually the preferred movement as C angle rotation is more rapid. It is therefore recommended for movements on machine tables.

In the **Direct Movement** example shown below, the first movement is made to a position which is parallel to the Z-axis of the reference system and the translation is then performed.

Direct Movement

To illustrate this, we are going to use the *Auto5_retracts2* workzone.

It is located in the **Auto5_retracts2** folder of the file package for this training session. This folder contains data related to the machines in this workzone.

1. Copy the machine data in the following directory:
*C:\WorkNC**\pospro\5axismachines.*
2. Open the workzone.
3. Check the parameters of the different toolpaths.
4. Activate the various Machining Contexts and simulate the toolpaths to compare the movements of the machine during retracts.

Index

3

3 to 5 Axis Machining Strategies, 8

3 to 5 Machining Additional, Advanced and Holder Parameters, 32

A

Adjusting Tilt Movements on Narrow Regions, 35

Advanced Parameters, 34

Auto3+2 Strategy, 42

Auto5, 3

C

Constant (Attractive) Strategy, 20

Constant (Repulsive) Strategy, 23

Constant to Axis Strategy, 12

Converting a 3-Axis Toolpath, 5, 11

D

Defining Additional Settings, 52

Defining Advanced Parameters for the Machine Limit Check Function, 57

Defining Start and End Angles, 55

Defining Start and End Points, 54

Defining the Main Settings, 50

G

Guide (Attractive) Strategy, 25

Guide (Repulsive) Strategy, 26

Guide (Surface) Strategy, 27

H

Holder Parameters for Collision Avoidance, 32

K

Keep Axis Along Machining Direction, 38

M

Machine Limit Check, 49

N

No Change Strategy, 14

Normal to Surface Strategy, 17

P

Programming an Auto3+2 Strategy, 44

S

Saving Machine Settings, 57

Selecting a Machining Context, 49

Smoothing Distance, 39

Surface Sensitivity, 40

V

Vertical Rotation Strategy, 30

Vertical Strategy, 9

获取 WorkNC 更多资讯：

1.关注 WorkNC 公众微信号：qh_worknc

扫二维码，即刻获取更多信息及参与 WorkNC 实时活动：



2. 访问 WorkNC 官网：www.qianghu.com

WorkNC 资料（产品手册、文档及视频教程、客户案例）下载：官网—>教程与案例—>下载中心

3、WorkNC 论坛：http://www.qianghu.com/bbs

WorkNC 技术问题咨询：官网—>论坛—> WorkNC 强互工程师解答专区

上海强互信息科技有限公司

上海市浦东新区新金桥路 1122 号方正大厦 1201、1202 室

Tel: 021-50304630 Fax : 021-50307213

www.qianghu.com

Email : qianghu@qianghu.com