TR473369

How Autodesk Can Streamline the Automotive Clay Studio Workflows

Samarth Gupta
Autodesk India Pvt Ltd

Dushyant Atara
Autodesk India Pvt Ltd

Learning Objectives

- Learn How Alias data can be seamlessly used to create cutter paths in Clay Milling Utility
- Learn how small design modifications in Alias gets quickly translated in Clay Milling Utility
- Learn how the scanned model gets aligned to the design coordinates
- Learn how PowerInspect can scan & digitize the clay and export the data to Alias

Description

This session will talk about how Autodesk solutions like Alias , VRED , PowerMill & PowerInspect can empower automotive design studio's with technology to push the boundaries of creativity and collaborate radically with the goal of making better products, faster .

Speaker(s)

Samarth Gupta
Samarth is a Technical Specialist for Industrial Design & Visualization at Autodesk . He has been spearheading the business efforts in India by introducing various design studios to latest technologies and toolkits & help them accelerate and streamline their workflows , reduce their time to market and help them give their customers a whole new brand experience using Autodesk tools.

Dushyant Atara
Dushyant Atara serves as a Technical manager for Autodesk India for digital manufacturing group MAKE Product. He works closely with the DMG sales team and Make partners to help new and existing customers meet their business objectives with Autodesk's Make portfolio.

Introduction

Autodesk® Clay Milling Utility has been created to satisfy the CNC milling requirements of the design sector. Models are milled from clay and foam to create a realistic 3D prototype which can then be refined by skilled modellers. This type of machining is often used in the automotive industry, as well as general mechanical engineering, art and restoration.

Autodesk Clay Milling Utility can be used to drive studio milling machines and CMMS or layout machines. The CMM or layout machine must be equipped with a milling head and the structure must be strong enough to be able to withstand the forces arising from milling.

Autodesk Clay Milling Utility can perform the entire milling process, from creating the cutting path to running the machining program on a CMM or layout machine.

The software comprises three areas:

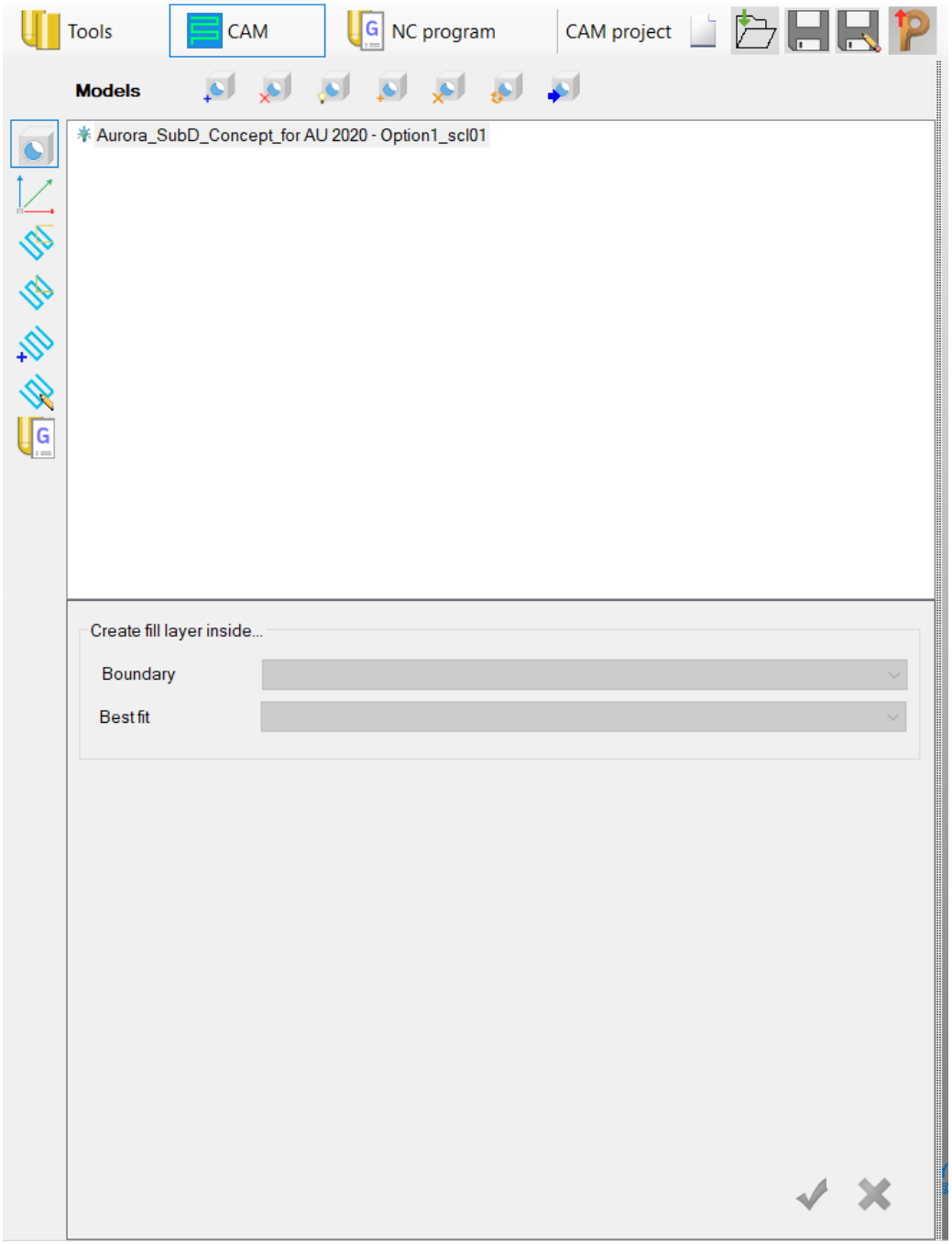
- **Tools area** – Here you manage milling heads, tools (cutters), assembly configurations and alignment tools.
- **CAM area** – Here you manage CAD data, workplanes (milling head adjustments), boundaries, engraving, toolpaths and NC program creation.
- **NC Program area** – Here you create the alignment (position of the part to be machined), and load and run the corresponding NC program

You can resize the Tools and CAM areas with the mouse. Reducing the size of these areas increases the size of the graphics window.

CAM tab

Use this tab to assign directories and set options for the milling heads. In the CAM and NC program modules, these values are used to create workplanes and define the orientation of the milling head.

Set up your file directories for saving your project and NC programs, CAD model and option file complete with file extension.



In the **Directories and data files** area, you can specify where to save ClayMilling projects, NC programs, tool data, alignment tools and model data. These settings are machine-specific, because each machine may have its own tools and alignment tools.

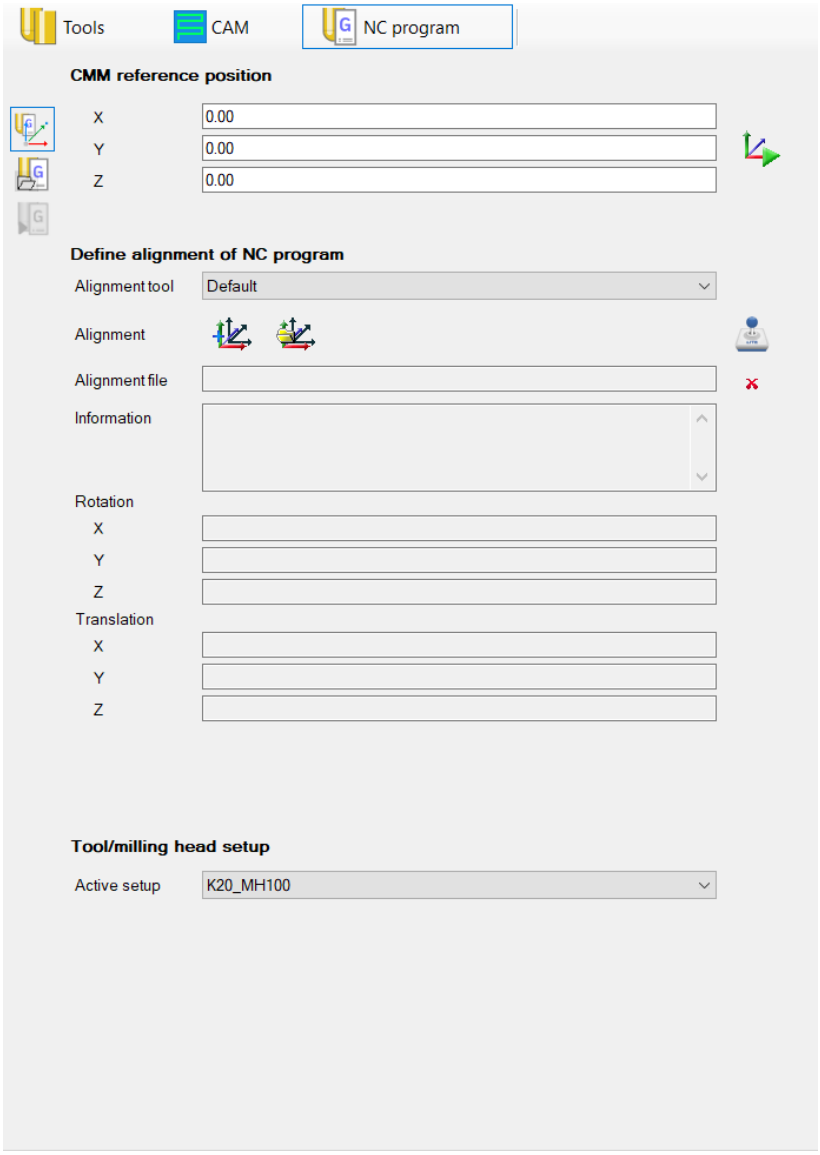
Milling heads are often shared between several machines, so the data may be stored on a server.

In the **NC program settings** area, you can specify where the post-processor file (kinematics of the CMM and milling head) is stored. The file extension and data type are defined automatically when the milling programs are created. The appropriate post-processor file is required to enable creation of a program suitable for the machine.

The attributes in the **Stock model settings** area affect the calculation of the Roughing strategy. They represent a virtual model, the **Thickness** is the allowance, and the **Stepdown factor** is the Z cut height as a factor of the tool diameter.

NC Program tab

Use this tab to specify the behaviour of the application in the NC program module.



In the **Alignment file directory** field, specify where to save the alignments created in

Tools

The other values in this dialog are related to the milling head and must be taken from the manufacturer's specifications.

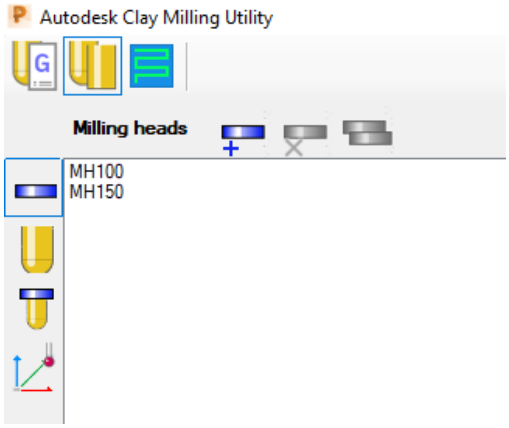
In the **Tool value** list, select **Tip** or **Centre**. For undercuts on edges, **Centre** should be selected.

The option **Milling head offset measuring active** allows an alternative calculation of the milling head position. By default, the coordinate offset of the milling head is determined by calculation, taking into account the data entered in the tool management and the respective current angle. As an alternative to calculation, use this option to re-measure the milling head offset after each angle adjustment.

The **Tracking rate** determines how often the position values of the CMM are updated in ClayMilling. The **CAN - Baudrate** is used to define the communication parameters when adjusting the angle of an automatic adjustable milling head (for example the Wenzel Excalibur 2 axis milling head.)

Select **Tools**  in the top-left corner to manage milling heads, tools (cutters), assembly configurations and alignment tools.

This area is split into 4 parts – **Head**, **Tool**, **Setup** and **Alignment tools**.



- The head is the milling head you are using. Ensure this information is accurate.
- The tools are milling tools or cutters.
- The setup is the combination of the milling head and tools. It is important that the active box is checked as this defines your active setup.

- The alignment tools are used for scribing and marking out and to define datums.

When you set or change these, you need to restart ClayMilling.

To access, select **Milling Head Administration-Milling Head New**, **Milling Head Delete** or **Milling Head Copy**.

Defining milling heads

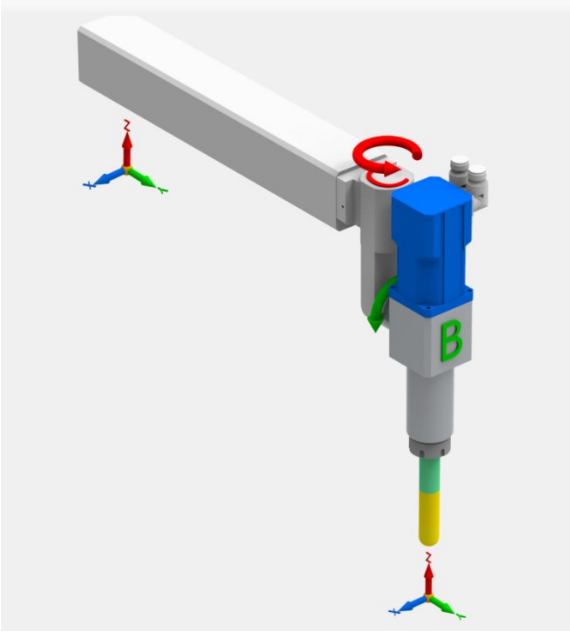
The milling head is defined by the following input matrix. The values are supplied by the manufacturer of the milling head or must be accurately measured.

The coordinate system at the front of the tool tip refers to the local milling head coordinate system. The coordinate system on the Y-transverse arm corresponds to the machine coordinate system. The definition of the milling heads is always specified from the direction of the tool tip to the flange point / mounting position. The first rotational axis is described as A, the second rotational axis is described as C. The min / max step size refers to the allowable rotation range of the respective rotation axis in degrees.

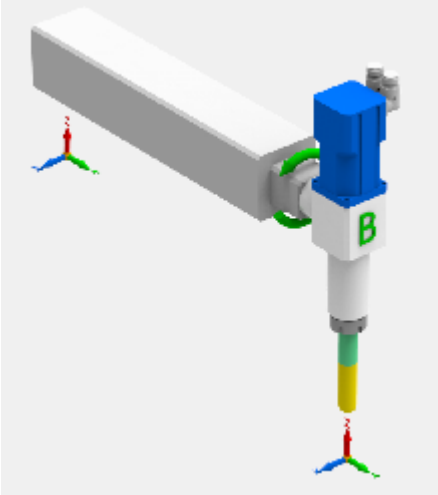
The vector and the corresponding distance define the respective direction from A axis to C axis or from C axis to the flange point.

Here are some examples of different milling heads which can be operated in conjunction with ClayMilling:

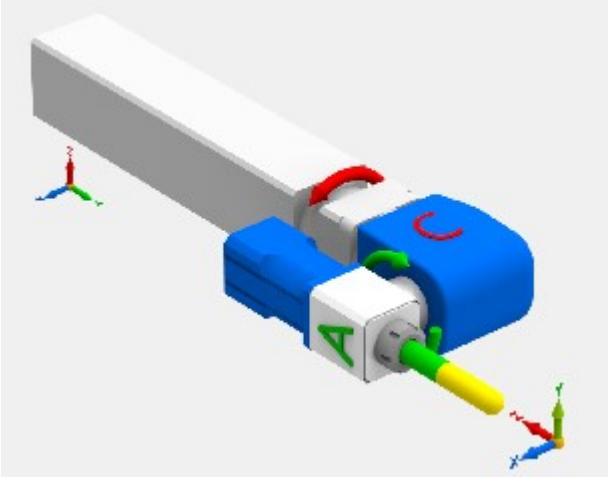
Milling head with manually adjustable B and C axis.



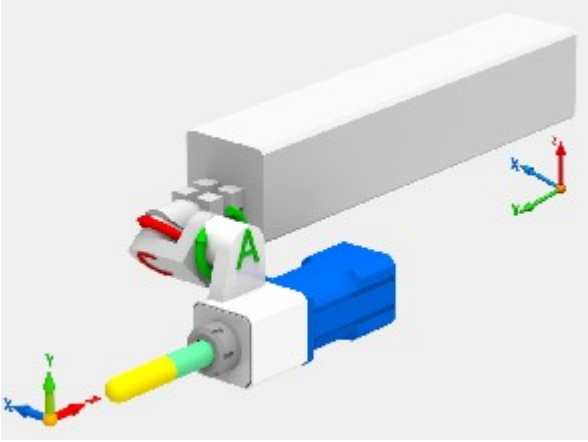
Milling head with single adjustable B axis.



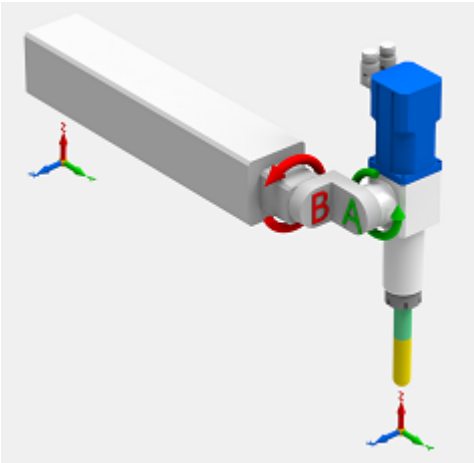
Milling head with adjustable A and C axes.

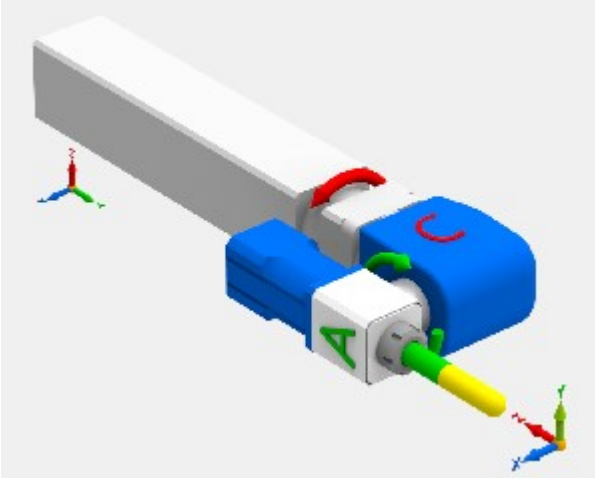


Milling head with manually adjustable A axis and 45 degree slanting C axis.



Milling head with motorized A and C axis.





The specifications for the distances 1 to 3 have an influence on the calculation of the milling head kinematics, and affect the graphical representation of the spindle and the tool in the CAM module. These values usually describe an enveloping cylinder around the milling head.

The corresponding type of the milling head is specified via the **Kinematics** field (swivel axes). The following types are supported:

bc:	manually adjustable b and c axis
b:	manually adjustable b axis
ac:	motorized adjustable a and c axis
ac45:	manually adjustable a and 45 degrees tilted c axis
ab:	manually adjustable a and b axis

In the remaining fields, the values supplied by the manufacturer (or measured values) must be entered.

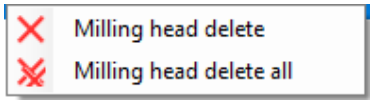
The fields 1 and 2 Rotation axis indicate the possible adjustable angles and the vectors of the mounting position a viewed from the milling tip (milling head coordinate system) to the flange point of the machine (machine coordinate system) as shown in the graphic representation.

The distances and vector progression from milling head coordinate system to the machine coordinate system are described in the fields **Vector** and **Offset 1** to **3**. As a rule, these must be captured by measurement, since there is usually a slight deviation in the rotation between the milling head coordinate system and the machine coordinate system. Failing to take this into account may lead to a misalignment of the milled surfaces when the milling head is re-oriented or when a tool is changed.

Any number of milling heads can be created and managed.

All milling heads are displayed in the ClayMilling software list. If they are not part of configuration they can be edited  , deleted  or copied  .

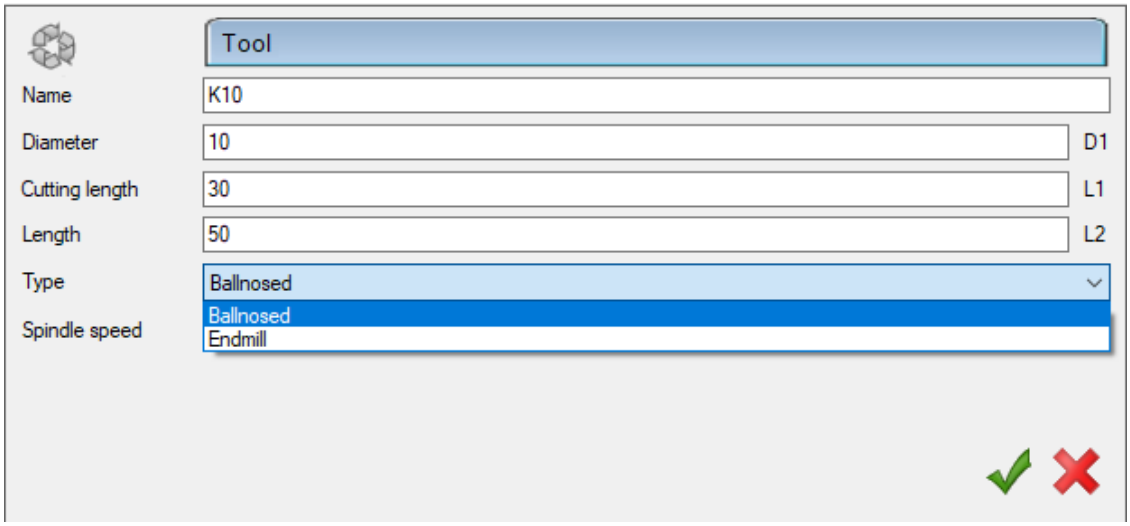
Right-click in the main menu under the displayed milling heads to display the context menu which you can use to delete the selected milling head or delete all defined milling heads.



Defining milling tools

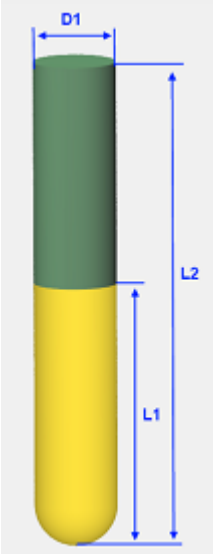
The milling tool comprises a shank and a cutter. This information is displayed graphically in the CAM module to help you program toolpaths.

You set all parameters for the tool using the following dialog:



ClayMilling supports ballnosed and endmill tools. The speed of the tool is written into the NC program and is set as the default when the NC program is run on the machine.

The dimensions of the tool are pure nominal values and serve as a graphic representation of the tool, with toolpaths being calculated based on the tool diameter.



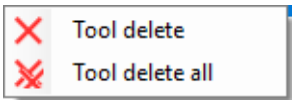
The real effective tool length is determined by calibration. Precise calibration is required for overall accuracy and to ensure smooth transitions between different tools or tool angles.

Tools can be created  , deleted  or copied  .

Existing tools can be edited by selecting  .

If a tool is part of a Setup configuration, it cannot be changed or deleted.

Right-click in the main menu under the displayed tools to display the context menu that enables you to delete the selected tools or to delete all defined tools.

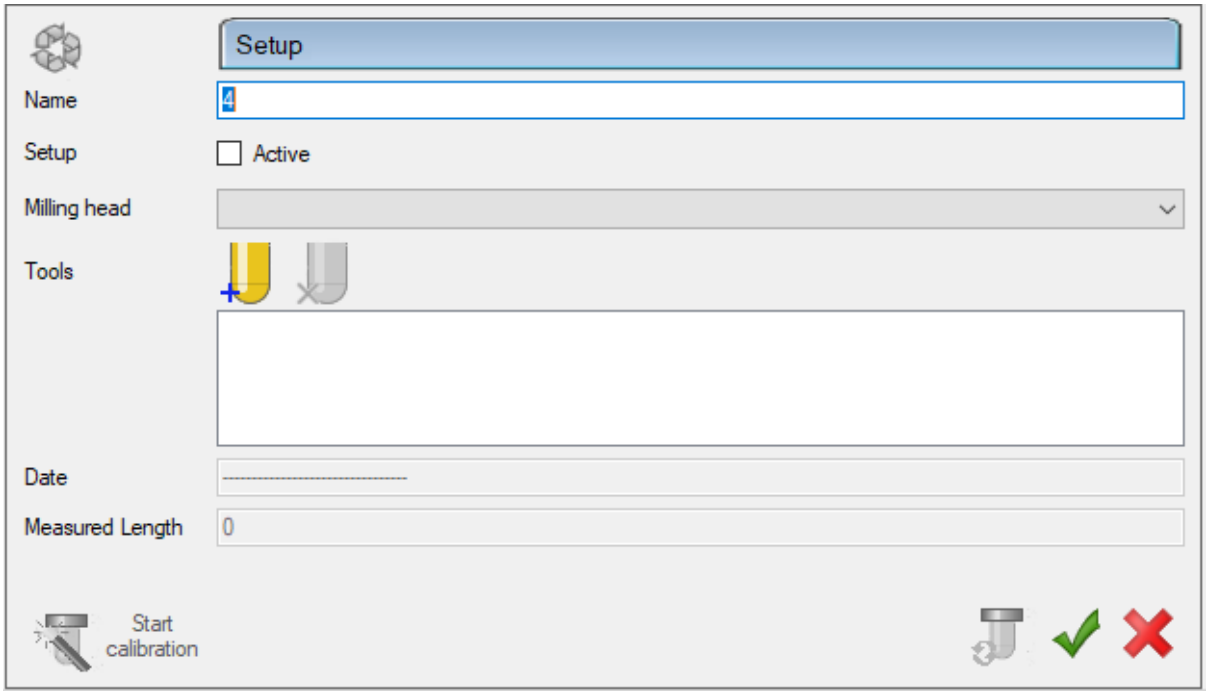


Creating a Setup

The Setup  is the combination of the milling head and the tool. The CAM module and the NC program module each access the active Setup and load the relevant information from the module according to the requirements.

You can have multiple Setups, but only one active Setup.

Click  to create a new configuration and open the following dialog:

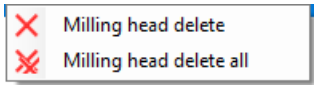


All data relevant to the configuration are entered here.

The **Milling head** list displays all milling heads you have created. You can add tools to the milling head. Multiple tools can be selected at the same time in the menu in which the tools are displayed.

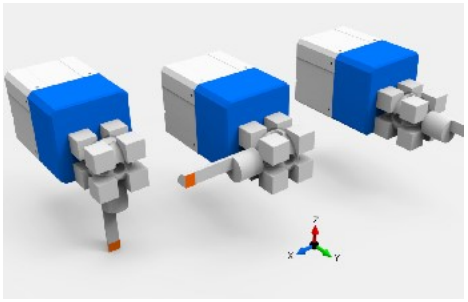
If you edit a Setup, for example by adding tools, the changes are not automatically available in the CAM module. Click  to send these changes to the CAM module. The current project is complemented by these tools. The **Edit**  button enables editing of setups to add or remove tools.

Right-click in the main menu under the displayed tools to display the context menu that enables you to delete the selected setup or to delete all setups.



Managing alignment tools

Use the **Alignment tool management** menu  to create and manage scribing and setting tools for aligning the clay model for machining (milling).



Any alignment tool can be created in the appropriate axis, taking the offsets into account.

Alignment tools can be created , deleted , copied or edited .

The name of the alignment tool should correspond to the mounting position in CMM coordinates. It is helpful to include the length of the scriber in the name.

The specified offset values/directions always refer to the global machine coordinate system.

The values must always be entered as if they had been recorded on the Master, even if they are later used on a Slave. The distinction between Master and Slave is not relevant here.

The clamping cube, which is used to hold the scriber, usually has dimensions of 50mm x 50mm x 50mm. The centre of the cube is the zero point for calculations. If a 50mm length scriber is used with a 50mm clamping cube, the result will be an offset in the corresponding axis direction of 75mm.

The zero point must always be the point at which the milling head is also mounted on the arm.

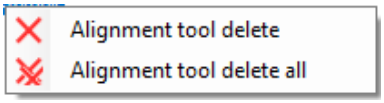
This applies to the Wenzel DMH 120 2 axis milling head. The DMH 120 is attached to the universal mounting head and thus the centre of the universal mounting head is the common zero point for scribing tools and milling.

This does not apply to the Stiefelmayer-FK. Here, the common zero point is the point on the Y-arm at which the clamping cube itself or the FK is attached. For the common zero point to be the same for all FK and alignment tools, the common zero point should not be placed in the centre of the clamping cube.

It is recommended that the zero point is positioned at the mounting position of the clamping cube on the arm.

For example, an alignment tool Y of 100mm comprises a 75mm needle and a 25mm clamping tube.

Right-click in the main menu under the displayed alignment tools to display the context menu that you can use to delete the selected alignment tools or to delete all defined alignment tools.



The alignment tool/setup data are stored and managed in one file. The file and location are defined in the **Options** dialog.

CAM

You can use the CAM to calculate and edit toolpaths, and to export them as NC programs.

The main toolbar in the CAM module  enables you to create new projects, open existing projects, save projects, or store them under a different name.

Creating a new project

You must create and save a project before working in ClayMilling. Click **New Project** to open the **Save As** dialog, and then select where to save the project, and enter a name for the project. The default location is the path defined in the **Options** dialog, but you can select a different location. The file extension of projects is .picmpri, which simplifies the exchange of projects from one machine to another.

When you create a project, all active tools are loaded and other functions are enabled.

Models

Models (CAD data) can be read in a variety of formats including IGES, VDA, STL or DDZ (Autodesk format) and CATIA.

Importing models

From the Model tab, you can import new CAD data models. You can import several models to create an assembly. Each CAD model will be displayed in the list. Models can use a mixed data, including surfaces and STL.

Displaying model data

Loaded CAD data models are displayed in the graphics window as wireframe or as a shaded model, depending on the function activated. On the right side of the graphics window are tools for manipulating the view.

Resize to fit

View along the Z axis

View along the -Y axis

View along the X axis

View along the -Z axis

View along the Y axis

View along the -X axis

Isometric view 1

Isometric view 2

Isometric view 3

Isometric view 4

View corresponding to the active workplane or global coordinate system

Undercut shading, model can be displayed shaded with toolpath

Plain shade

Wireframe

Hide/Show toolpath links

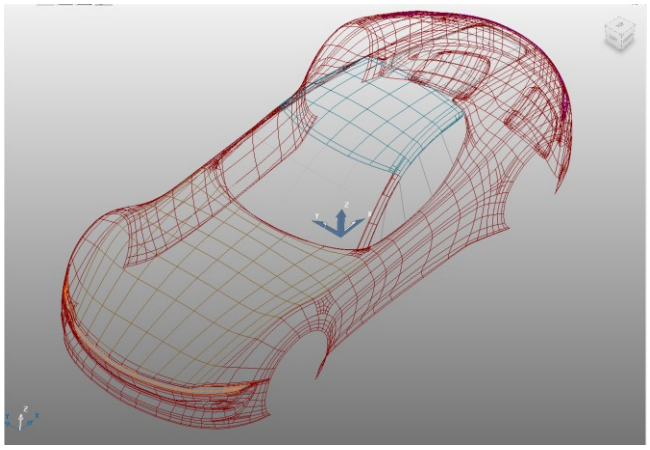
Hide/Show stock model

Tool hide

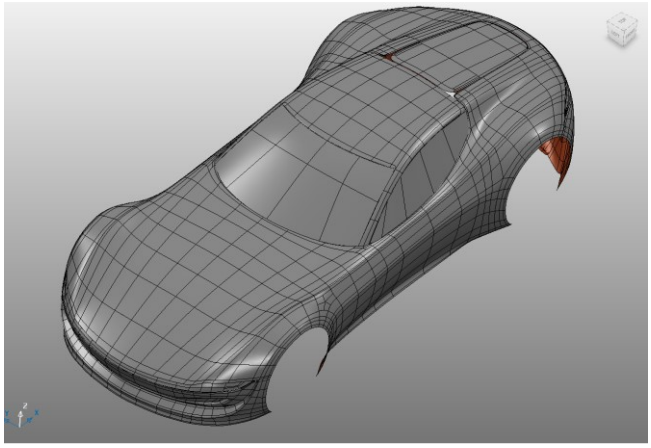
To select items, left-click them in the CAD area. To zoom, use the scroll wheel. To rotate the view, hold the middle mouse button and move the mouse. To pan the view, hold the Shift key and the middle mouse button and move the mouse.

If the model is not shown either as wireframe or shaded, it is hidden. If wireframe and shading are active, selected areas are shaded, and all other surfaces are shown as wireframe.

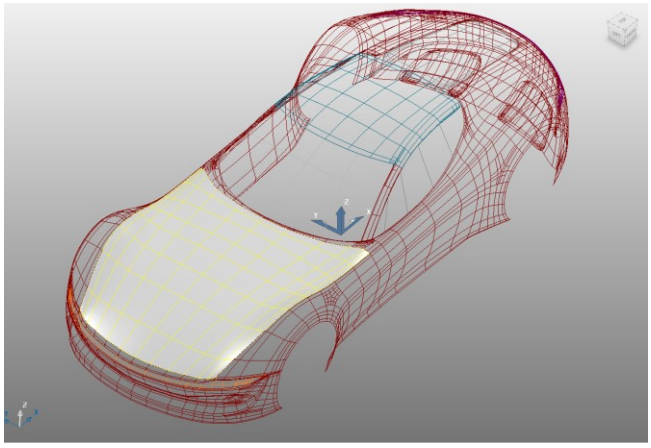
Wireframe:



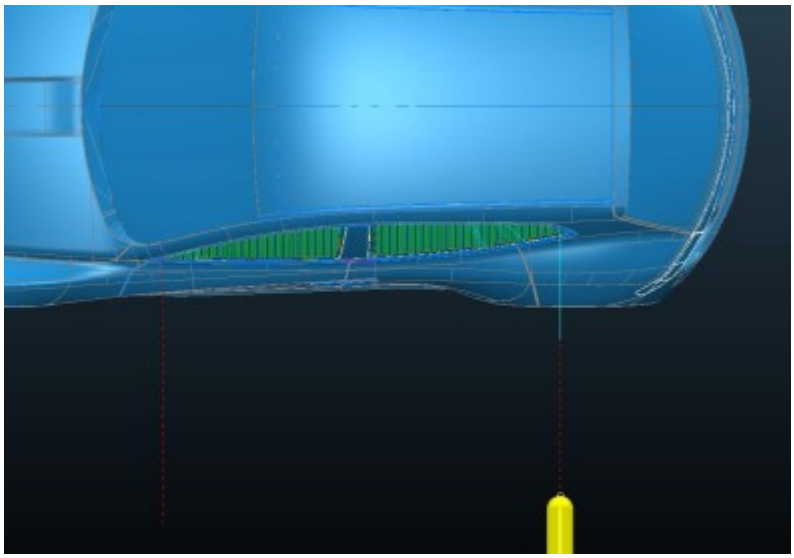
Shaded:



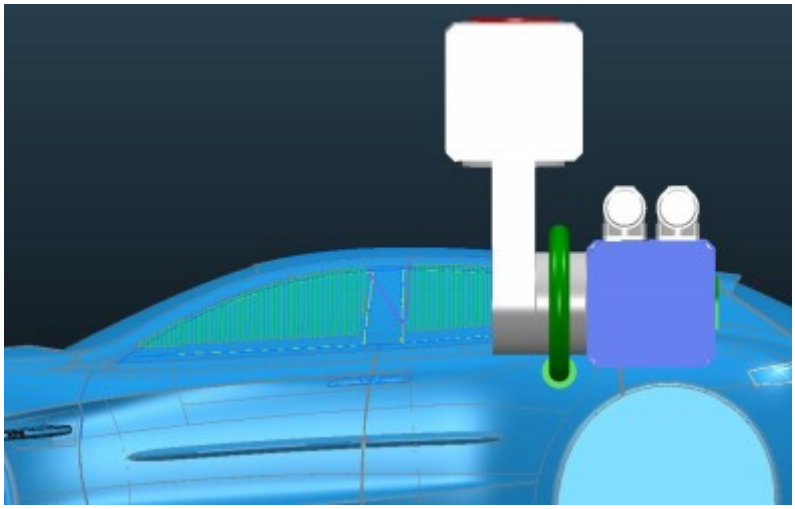
Selected surface shaded:



With the **View corresponding to active workplane button**  , you can display the view related to the active workplane:



Or the view related to the milling head setting:



Manipulating Model data

When a CAD model is loaded and selected in the list, The **Models** panel is available.

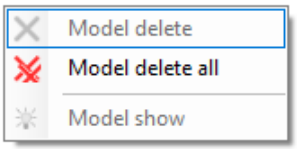
Autodesk Clay Milling Utility for PowerInspect allows you to delete  CAD models from the list as well as to show or hide  selected models.

Use the **Create plane of boundary** function  to create a flat surface from the selected boundary. This is covered in more detail in the **Boundaries** section.

You can delete  or invert  (rotate the bottom of the surface to the outer side or vice versa) selected surfaces.

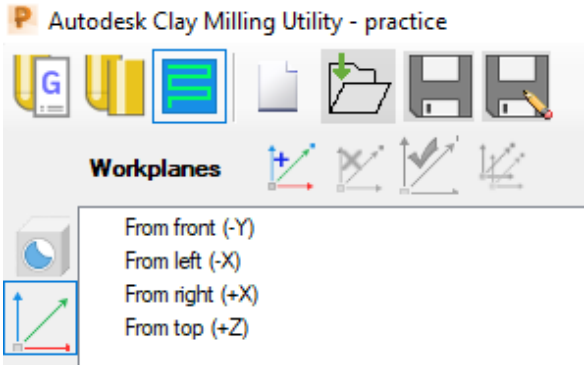
In the **definition of plane from boundary**, you can select corresponding boundaries. Select a **Best fit** of **upper**, **lower**, or **both** planes. The best fit from the selected planes defines a filling plane optimized for their deviations.

Right-click in the main menu under the displayed CAD models to display the context menu that you can use to delete the selected CAD models, delete all defined CAD models, or display all hidden CAD models.



Workplanes 

The workplane defines the positioning of the milling tool relative to the model surface. The world or vehicle coordinate system is the main workplane. All NC programs are exported according to this workplane.



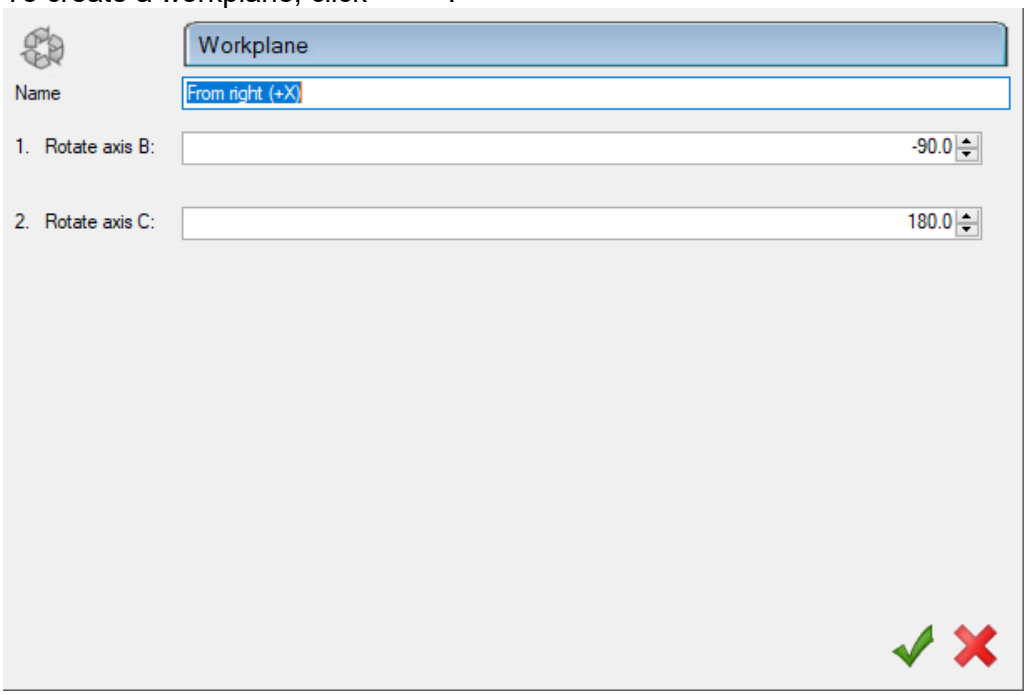
Depending on the shape and orientation of the surfaces, you may need to define different workplanes to complete the milling process successfully.

The Z axis of the workplane always corresponds to the vector of the milling tool. ClayMilling automatically creates four workplanes. To make a workplane active, double-click it.

It is not possible to delete the workplanes generated by ClayMilling.

Creating workplanes

To create a workplane, click  .



The view of the model can be positioned in the graphics window so that it corresponds the desired orientation of the tool to the model. When you select **New**, the newly created view has **angular** values within the limits of the **Min/Max** values and **grid size** defined in the Tools module. These values can be changed later.

Generally, you assign a name to the workplane and then move the workplane into position. First, the 1st axis is rotated and then the 2nd axis. Use the **Up**/**Down** buttons to change the values according to the specified step size. The view of the graphics window is adjusted after each value change.

It is useful to change the view between related to workplane and related to global machine coordinate system.

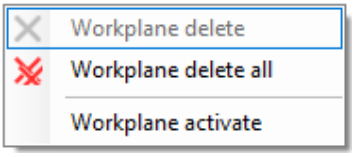
Editing workplanes

To change the properties of a workplane, select it and click  . You cannot change the properties of a workplane that is used in a toolpath.

Use the 

tab to delete  , activate  and copy  a workplane.

Right-click in the main menu under the displayed workplanes to display the context menu that you can use to delete the selected workplane, delete all defined workplanes, or activate the selected workplane.

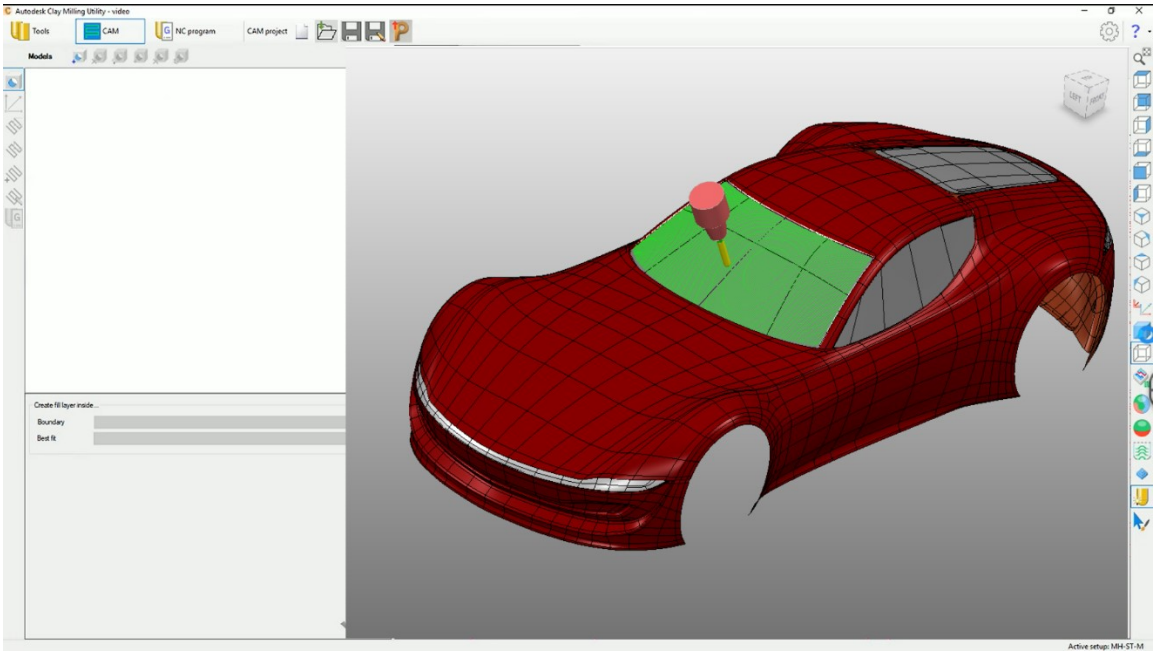


To activate no workplanes, right-click in the workplanes list without selecting a workplane (for example, under the last one in the blank area) and then select **Workplane activate; no workplane**.

Boundaries

You can use boundaries to calculate toolpaths within a defined area, and for creating fill surfaces. If no boundaries are defined, toolpaths are calculated over the entire model.

Toolpath inside boundary

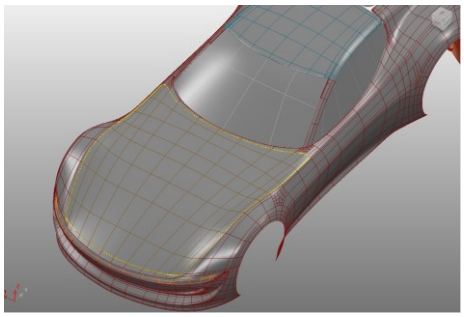


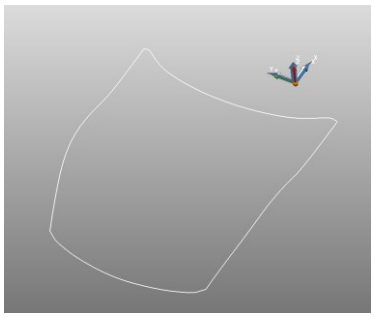
Defining boundaries

You can create boundaries from surfaces or by sketching a polygonal line on the CAD model.

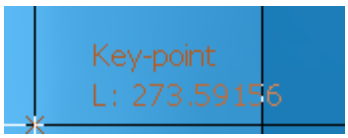
To calculate boundaries from surface edges or wireframe curves, select surfaces or curves in the graphics window and click  . In the **Boundary** dialog, enter the parameters, **name** and **tolerance**. Gaps on the edges of the surface are closed within the tolerance.

The boundary is shown as a curve in the graphics window.

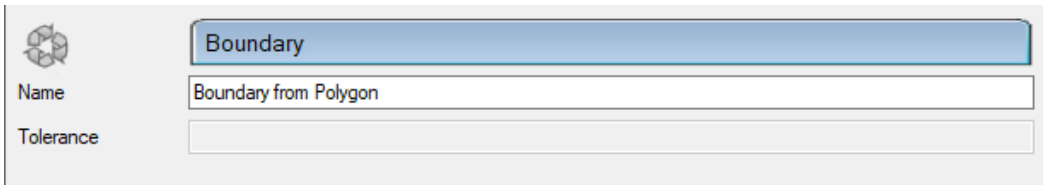




To create a polygon boundary, click  . The Z view of the model is displayed. Click in the graphics window to draw a polygon. The cursor snaps to important points such as model edges or polygon endpoints.

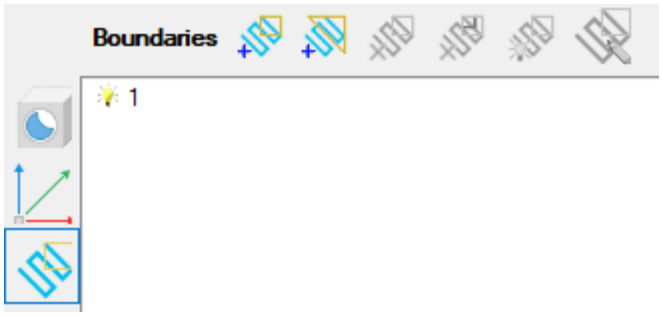


When you have finished drawing the polygon, click the green check mark to close the dialog.



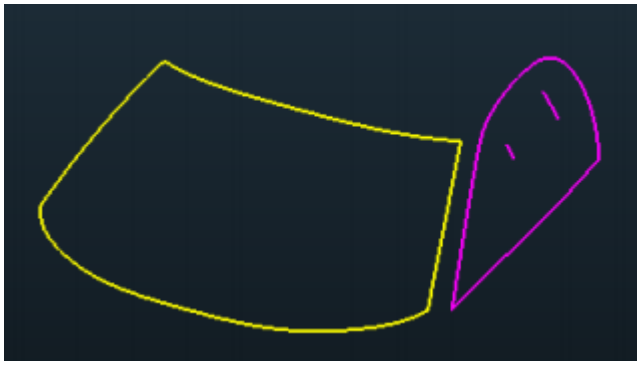
Clay Milling automatically closes the polygonal line, since a closed boundary is always required for the calculation of toolpaths.

Editing boundaries



To delete a boundary, click  . You cannot delete a boundary that is used in a setup.

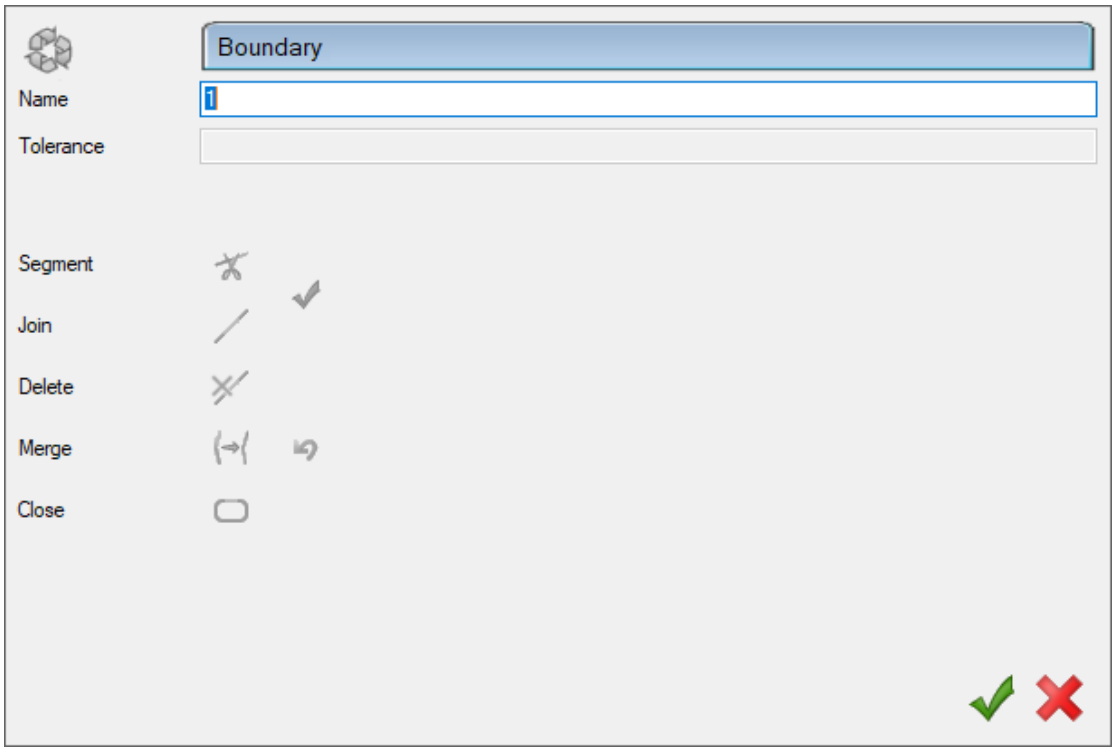
To delete curves from a boundary, select them in the graphics window and click  .



To hide or display a boundary, select it in the list and click  .

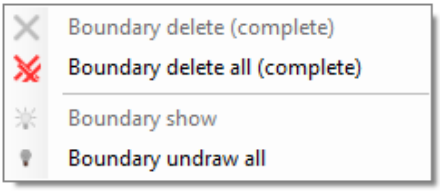
Use the **Editing boundaries** button  to break existing boundaries, chain elements to form a new boundary, and close boundaries. This functionality is useful when the boundary from surfaces does not have a closed boundary or if areas have been separated because the trimming of the surfaces have not been performed properly due to the CAD model quality.

You can also use this button  to edit boundaries created from curves.



Click **Segment** to split existing lines at a point on the curve. You can chain individual segments into a line, or click **Delete** to delete them. Use **Merge** to link individual segments into one, and click **Close** to merge open boundaries into a close curve.

Right-click in the main menu under the displayed Boundaries to display the context menu that you can use to to delete the selected boundary, delete all defined boundaries, display selected boundaries, or hide all boundaries.



Pattern

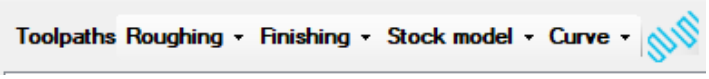
You can use patterns to program contours or engraving patterns. Patterns can be open or closed. A pattern does not have to be on the CAD model, but the resulting toolpath is projected on the CAD model. When calculating the toolpath, a projection along the milling axis takes place internally. There are two types of pattern: Pattern from surfaces/wireframe curves or from a polygonal line.

Toolpaths

ClayMilling calculates milling programs based on surfaces, volumes or STL data. Patterns can be calculated based on closed or open curves. Milling programs are always calculated in a coordinate system, with the Z axis determining the tool axis. A milling program can be calculated inside or outside a boundary. The block dimensions in X, Y and Z are the maximum calculation limits for the imported CAD model data. The milling programs are calculated based on a tool from the tool management. ClayMilling calculates the contact point of the tool cutter to the CAD model and presents it graphically.

Creating toolpaths

To create a toolpath, select a strategy in the menu.



Each of the main groups has drop-down menus, which are explained below.

In general, Autodesk Clay Milling Utility for PowerInspect uses the following terms/parameters.

Parameter	Description	Standard	By strategy
Toolpath name	User-definable name of the milling program	✓	
Workplane	The coordinate system for the calculation of the milling paths	✓	
Boundary	The boundary for the calculation of the milling paths; if no selection is made, the entire CAD model is processed	✓	

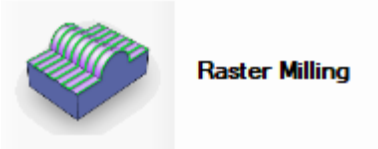
Trimming	Specifies whether the milling paths should be calculated inside or outside a boundary; if no boundary is selected, the selection is not taken into account		
Tool	Selection of tool for this milling path		
Angle	Angle for calculating the milling path		
Start corner	Specifies the start point of the milling path in the current workplane		
Cut direction	Unidirectional or bidirectional (Climb or Climb and Conventional combined)		
Tolerance	Accuracy for the calculation of the milling path on the CAD model		
Thickness	Oversize or undersize for the calculation of the milling path. In general, the allowance must not exceed the tool radius. (If 10 mm is used, an excess of material with a thickness of 10 mm remains)		

Stepover	Stepover distance for the milling path, laterally or in Z depending on the toolpath		
Reference tool	Previous tool of this strategy. Base on this, material is found and only there milling paths are calculated		
Drive curve	Limits a pocket to machine. The drive curve is closed and of type Boundary		
Curve side	Should be calculated inside or outside a curve		
Axial offset	Offset of the milling path along the tool axis. The value can be either positive or negative		
Curve thickness	Allowance over a curve in the XY plane to remove the tool from this curve		
Lower limit	Z machining height when milling pockets. The measure is considered absolutely in the model coordinate system		
Engraving curve	Reference line for engravings; the line can be open or closed. The curve can be 3D or a 2D sketch		

Raster Milling finishing strategy

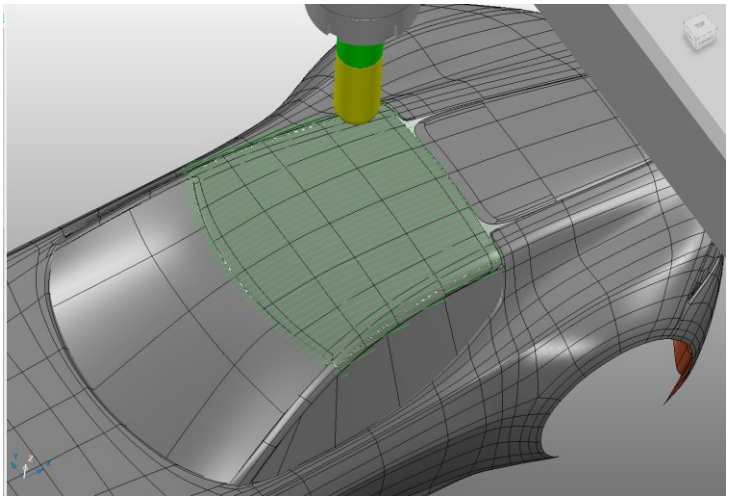
Use the Raster Finishing strategy to create a raster toolpath on the flat areas on a model. Raster flat toolpaths are useful for finishing flat areas, particularly on open, rectangular regions

The Raster milling strategy can be accessed from the Finishing drop-down menu.

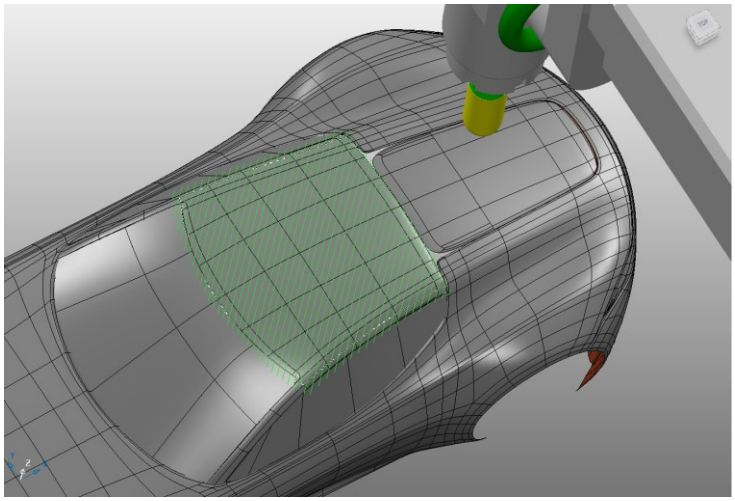


Raster milling allows the machining of the geometry along a user-defined grid along an angle in the coordinate system. An angle of 90 degrees would result in a calculation of the toolpaths along the Y axis. The stepover corresponds to a line on a plane which is then projected onto the model along the Z axis. This can lead to corresponding distortions in steep areas.

A raster under 90 degrees



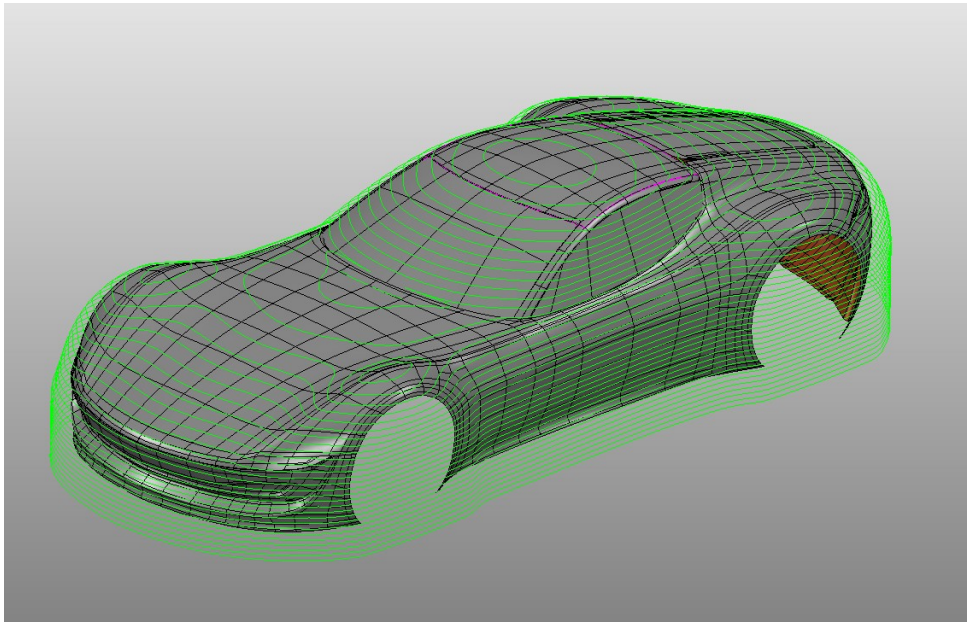
A raster under 45 degrees



Constant Z Milling strategy

This type of machining is used to for quick and efficient finishing of steep walls. Use the Constant Z finishing page to create a toolpath by slicing the model at specific Z Heights.

The Constant Z Milling strategy can be accessed from the Finishing drop-down menu.



Editing toolpaths

Editing is a very powerful function for manipulating existing toolpaths. Often the definition of the areas to be machined is complex. Toolpaths can be adapted to particular areas with the editing functionality. In editing mode, Autodesk Clay Milling Utility for PowerInspect provides various tools via the following toolbar.

You can limit a toolpath with a primary (X, Y or Z) plane or with a polygon.

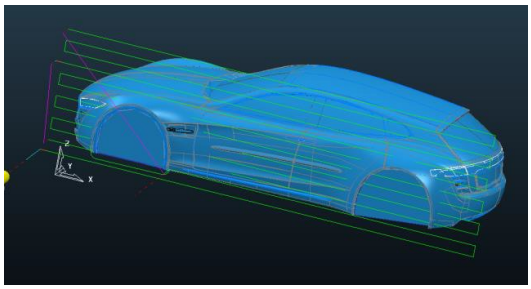
The marked toolpath can be deleted with the **Toolpath delete** button .

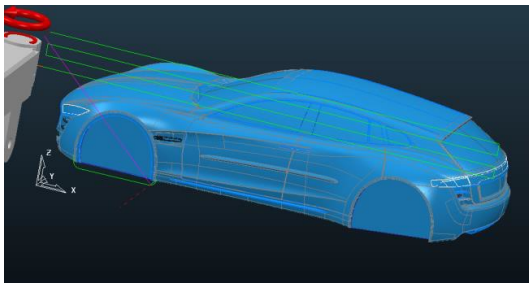
Deleting individual paths

By deleting individual paths, you can quickly remove areas from toolpaths that are not required for machining.

To delete an individual path, select in the list and click **Toolpath delete selected component** . Selected paths are displayed in yellow in the graphics window.

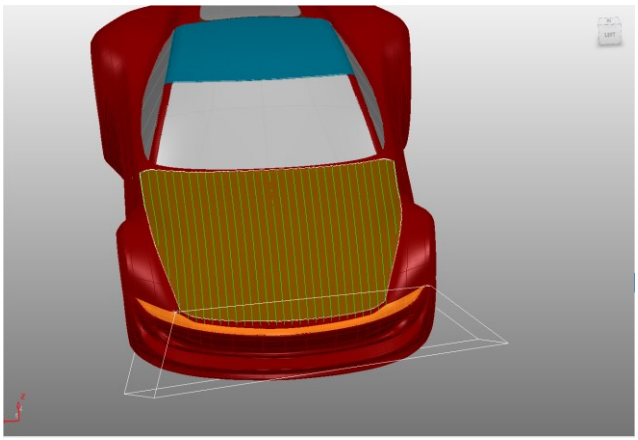
Before deleting individual paths: After editing:





Trimming toolpaths with a polygon

Trimming a toolpath with a polygon is a very fast way to remove areas from a sketch. You can keep toolpaths inside or outside the polygon.

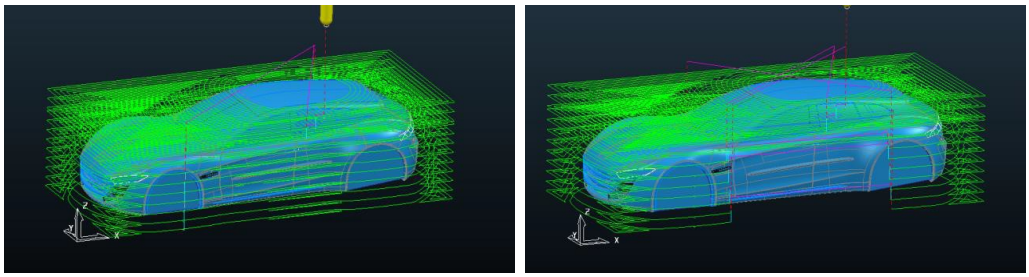


To do this, select the desired tool path in the list and click **Limit toolpath by boundary**  .

In the graphics window, the polygon is drawn; select the option **Keep inside** or **Keep outside** and complete the dialog. The polygon is a 2D sketch and is generated from the current view in the graphics window. All the milling paths found along this view are trimmed according to the selection.

Toolpath before:

Toolpath after:

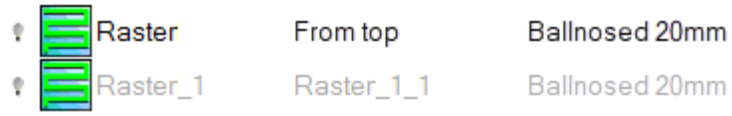


Mirroring toolpaths

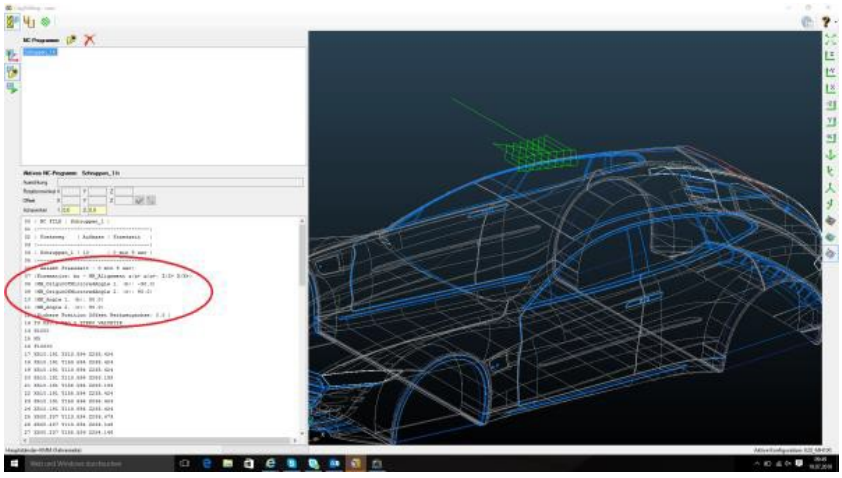
To mirror a toolpath, select it in the list and click **Mirror toolpath**  .

Toolpaths are always mirrored over the ZX plane, since in the automotive industry the vehicles are aligned axis-parallel to the machine axes.

Mirrored toolpaths are a copy of the original and cannot be recalculated. Mirrored toolpaths are displayed with a gray font.

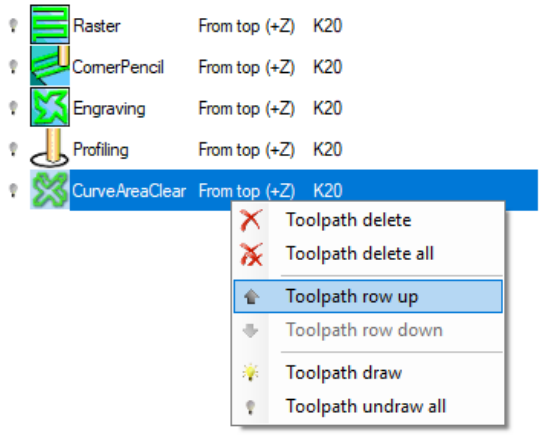


If mirrored toolpaths are to be exchanged from the Main CMM to the Slave CMM or by a rotated one, consider that the settings of the milling head might have to be changed. For this purpose, a message about the milling head position originally used is displayed when the corresponding milling program is started in the NC program module.



```
00 ( NC FILE : Schruppen_1 )
01 (-------------------------------------)
02 ( Freasweg    | Aufmass | Fraeszeit   )
03 (-------------------------------------)
04 ( Schruppen_1 | 10      | 0 min 9 sec )
05 (-------------------------------------)
06 ( Gesamt Fraeszeit : 0 min 9 sec)
07 (Kinematics: bc - MH_Alignment z/z+ x/x+: Z/Z+ X/X+)
08 (MH_OriginOfMirroredAngle 1. (b): -90.0)
09 (MH_OriginOfMirroredAngle 2. (c): 90.0)
10 (MH_Angle 1. (b): 90.0)
11 (MH_Angle 2. (c): 90.0)
12 (Sichere Position Offset Werkzeugachse: 0.0 )
13 T3 D20.0 H50.0 TYPE0 VALUETIP
14 S1000
```

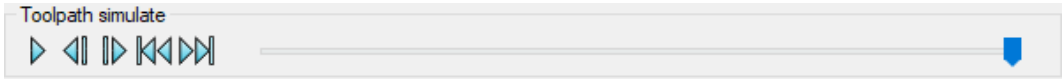
In edit mode, you can control the arrangement of the toolpaths using the right mouse button.



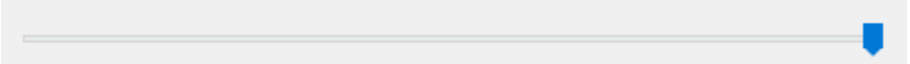
Simulating toolpaths

To simulate a toolpath, select it in the list and click **Toolpath simulate** .

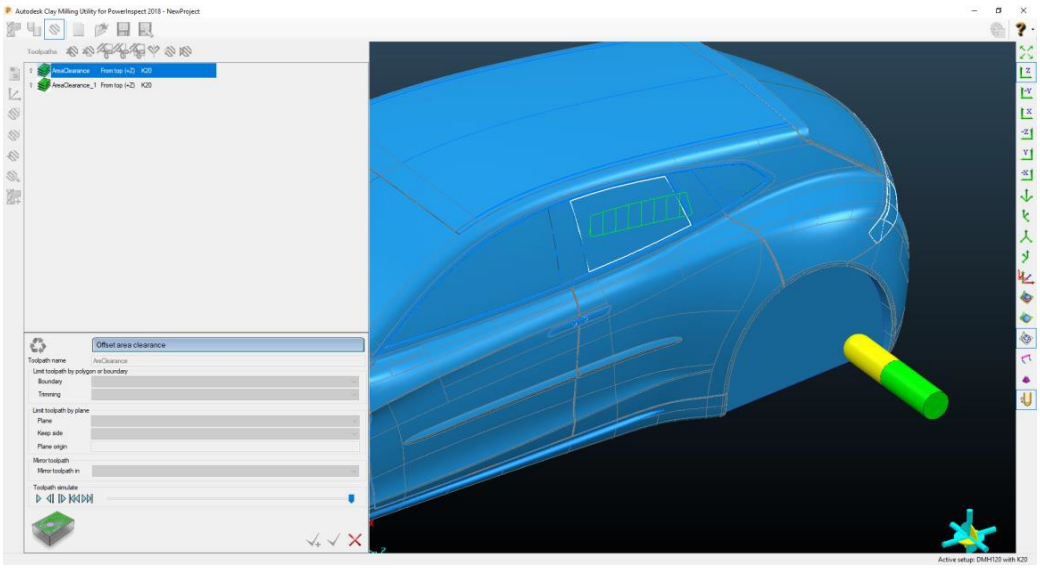
The simulation can be controlled using the following toolbar.



Press the button to start the simulation; press the button to return to the previous position. Press the button to move to the next position. Press the button to get to the beginning of the simulation, and the button to get to the end. The simulation speed can be selected via the slider.



To interrupt the simulation, double-click in the graphics window.



The current toolpath is always displayed in green. If a toolpath is displayed (lamp on) and a different toolpath is selected, the newly selected toolpath is displayed in green, and the previously selected toolpath will be displayed in gray. This enables you to see if toolpaths overlap.

NC programs

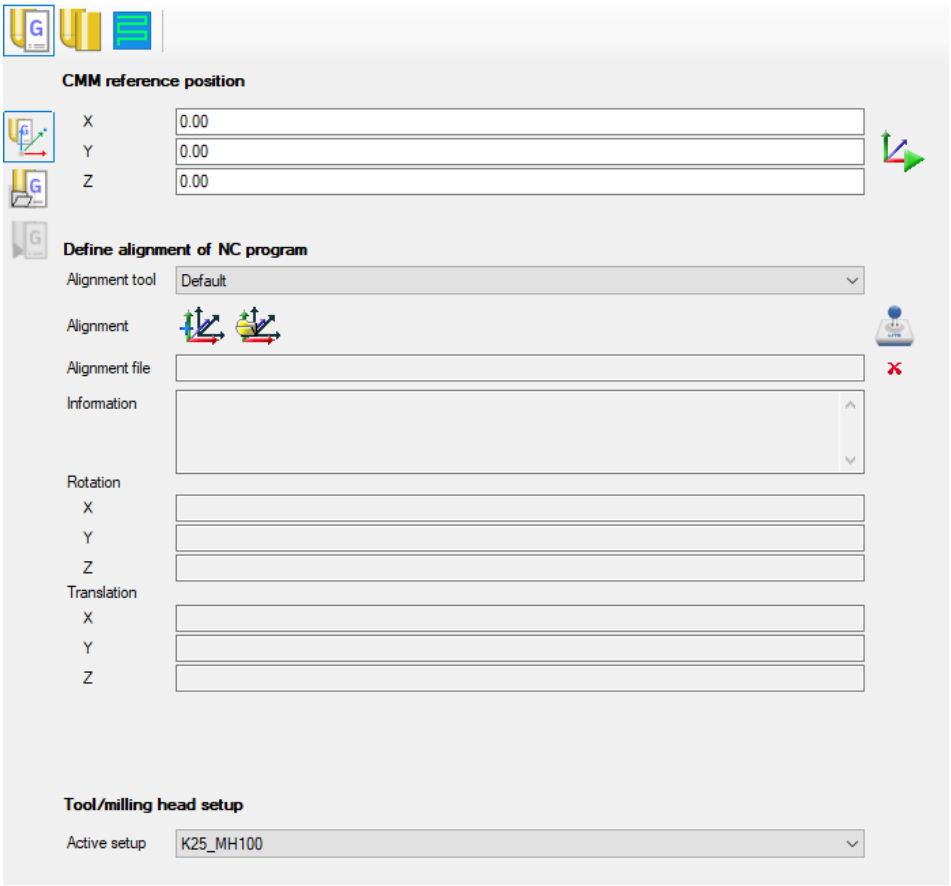
NC programs are required to execute the programs on the machine. The Autodesk post-processor translates the toolpaths programmed with ClayMilling into a data format readable by the control. You can export toolpaths to individual NC programs or combine them into a single NC program. Individual NC programs automatically take the name of the toolpath in ClayMilling. Concatenated toolpaths are given the project name as filename, and a consecutive numbering is appended. Example: ClayMillingProject_1.h

Toolpaths can only be concatenated if they have been programmed with the same tool and in the same coordinate system. This status can be seen in the ClayMilling toolpath list.

Raster	From top	Ballnosed 20mm
CornerPencil	From top	Ballnosed 10mm
Engraving	From top	Engraving
Profiling	From top	Ballnosed 20mm
CurveAreaClear	From top	Ballnosed 20mm

NC program

The NC program module provides all relevant functionalities for executing NC programs on a CMM or layout machine. You can run NC programs created with ClayMilling, or import NC programs created by other CAM systems, if they comply with the guidelines of ClayMilling.



On the CMM reference position area, the CMM can be moved to a position in the CMM coordinate system. Enter the desired coordinate in the X, Y and Z fields and press the button to execute.

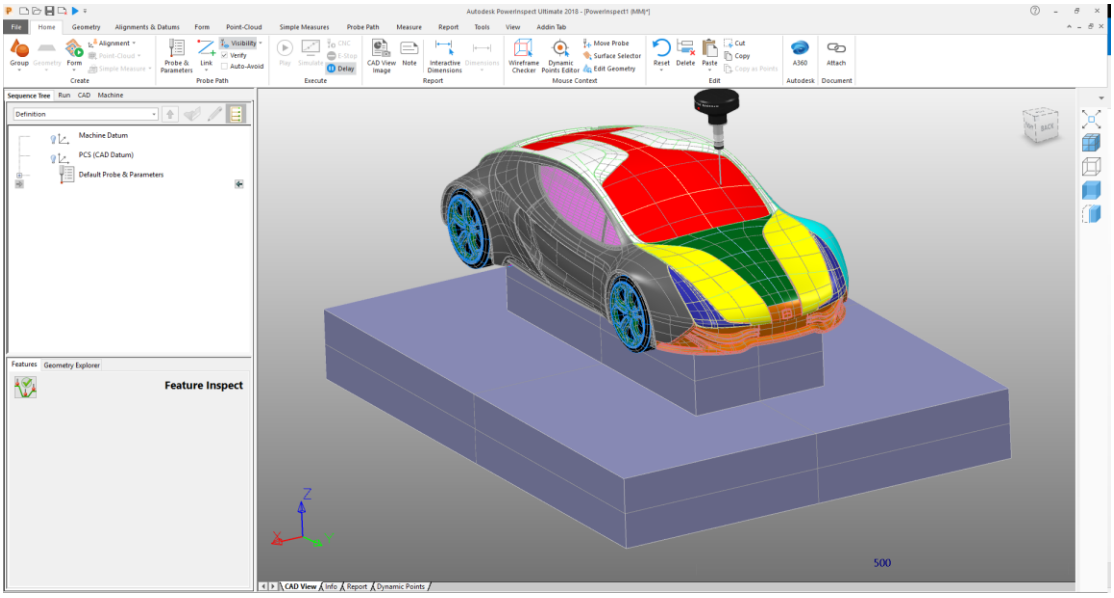
The CMM automatically moves to the specified position in CNC mode!

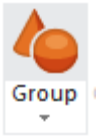
Creating an alignment in PowerInspect

Before using ClayMilling, we need to create an alignment in PowerInspect and export it.

Open PowerInspect.

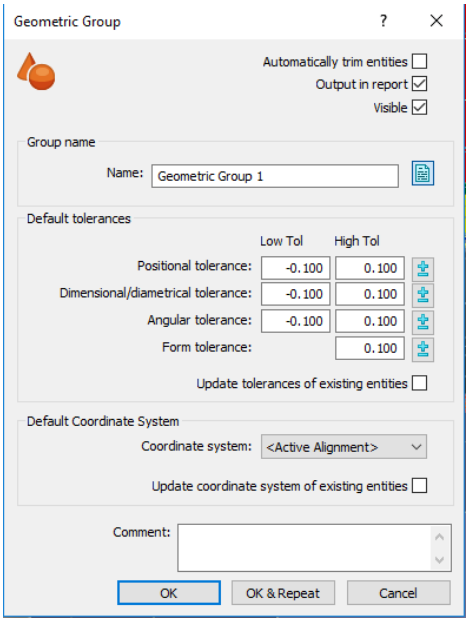
Load CAD model with alignment geometry present.



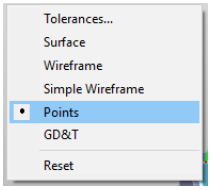
Create a Geometric group 

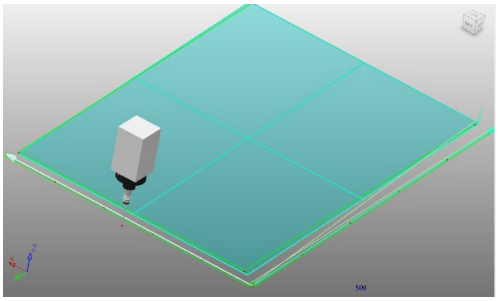
.

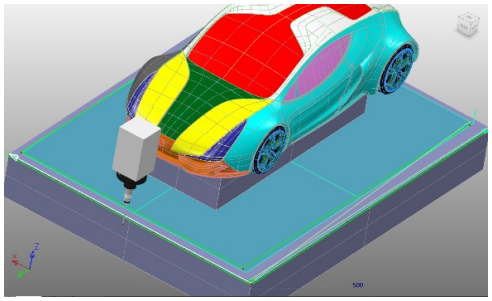
Click **OK** in the modify item dialog.



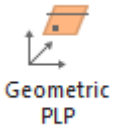
Use the **Wireframe Checker – Points** to create two probed lines.



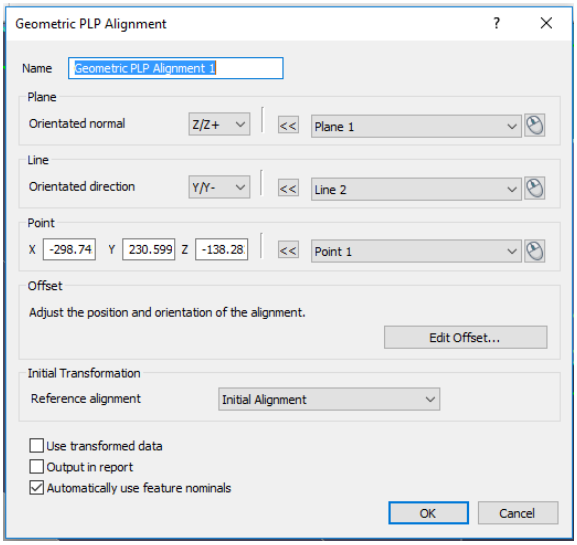




Create a point intersection between both created probed lines .

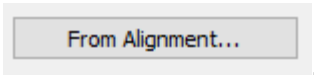


Create a GPLP alignment and populate it with created geometry .



Create user defined alignment 

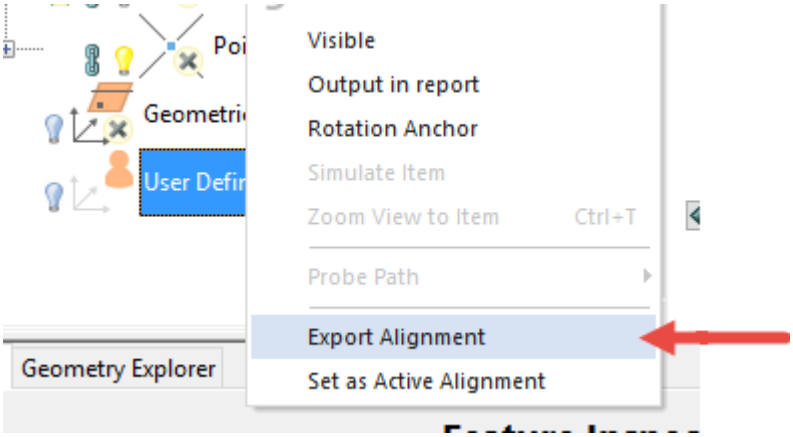
.

Click **From Alignment** 

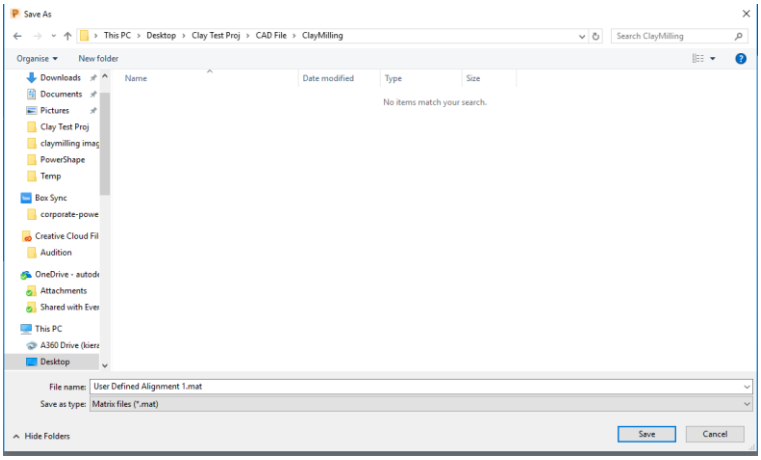
.

Select the GPLP alignment and click **OK**.

Right click the User Defined Alignment and select **Export Alignment**.



Save the alignment file in a user defined location.



Scanning and digitizing in PowerInspect

Open the group in which you want to create the Point-Cloud item.

Click Home tab > Create panel > Point-Cloud > Scanned Point-Cloud.
The **Point Cloud** dialog is displayed.

Edit the item's settings, and then click **OK** to close the dialog and create the item.

Play the item. PowerInspect switches to full-screen mode and displays the **Point Cloud Tool** dialog.

Use the buttons in the dialog to specify parameters and perform other actions.
Click:

 to change the acquisition parameters.

 to reduce the number of acquired points during scanning.

 to reject acquired points below the specified plane during scanning.

 to enable the sound made when the laser probe is near a point.
Note: The button is active only when PowerInspect is connected to a laser-scanner.

 to prevent the CAD view from changing as you create the point cloud.

 to create a new Point Cloud item each time you pause and resume scanning. The button changes to  .

Note: To select the button by default, select the **Commit point clouds on pausing** check box in the **Full Screen Measurement** page of the **Options** dialog.

1. Scan the part. Click:

 to delete all points.

 to delete points in the last point cloud.

 to close the dialog without saving the points.

When you have finished, click  to save the points and close the dialog. The point cloud is displayed in the CAD view.