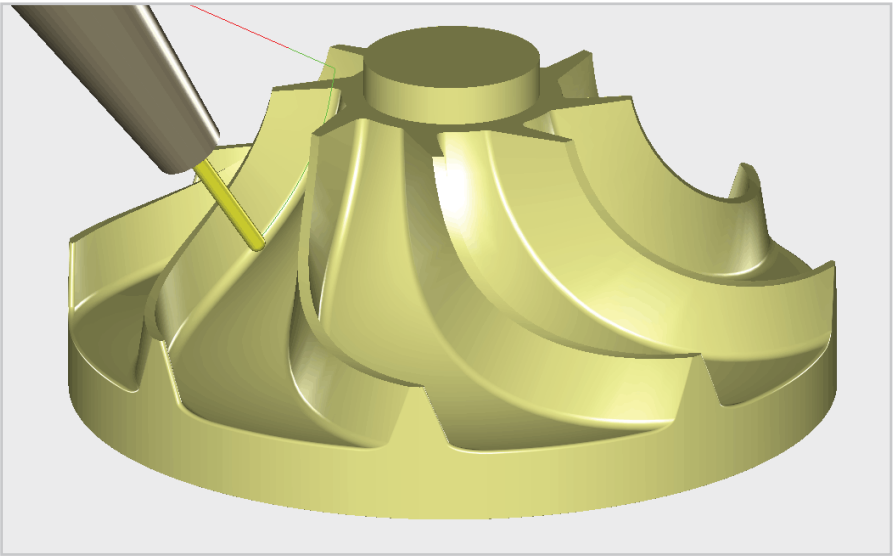
OneCNC Mill Multi-Axis



This manual covers the optional 4 Axis and 5 Axis machining modules which can be added to OneCNC Mill Professional and Mill Expert. These modules provide powerful yet easy to use modes which help you program parts quickly and reliably for multi-axis machining centers.

The purpose of this Manual

This manual is designed to get you started with OneCNC multi-axis toolpathing, so you can begin creating your own toolpaths as soon as possible.

Multi-axis machining is an advanced subject. You should have a working knowledge of 3 Axis toolpathing and construction planes in OneCNC before you start Multi-axis toolpathing.

The subjects covered are not machine specific and for operational methods and settings you should consult the instruction manual for your machine.

The modes made available by each OneCNC module are:

4 Axis Module
- 4 Axis Position
- 4 Axis Simultaneous
- 4 Axis Wrap

5 Axis Position + 4 all Module
- 5 Axis Position
- 4 Axis Position
- 4 Axis Simultaneous
- 4 Axis Wrap

5 Axis SImultaneous Module
- 5 axis Simultaneous

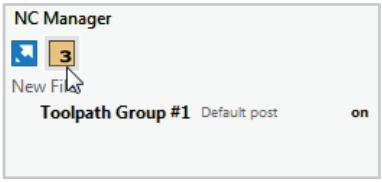
Note:

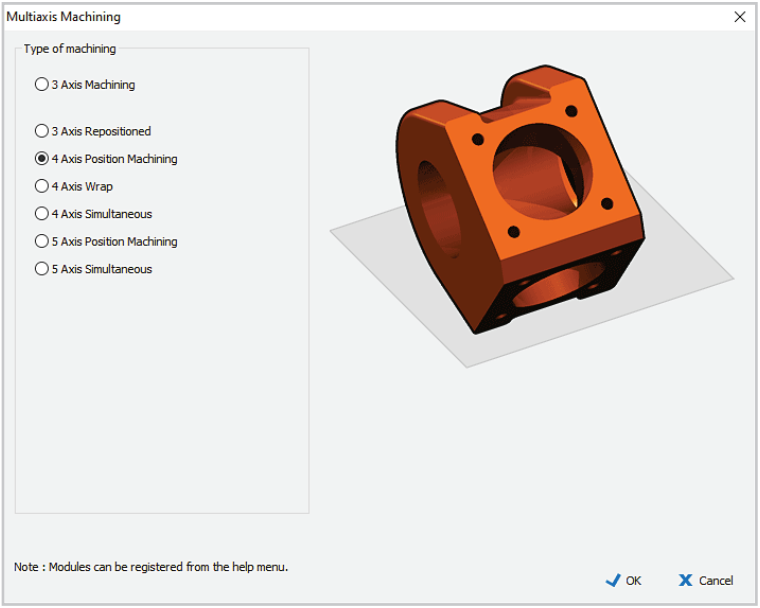
The 5th Axis Position + 4 all Module must be present before the 5th Axis Simultaneous module can be added.

Contents

Multi-axis machining mode selection

At the top of the NC Manager the icon next to the Process icon shows the current multi-axis mode. Clicking on this icon will open the Multi-axis selection dialog.





Select the multi-axis mode you want to use. The image in the Multi-axis selection dialog indicates the type of machining selected.

For the 4 Axis Wrap and Simultaneous modes, you can select the machine axis the 4th axis is aligned with.

For the 4 Axis Wrap mode, text boxes will appear for the input of values for Material Diameter and Length.

The multi-axis icon in the NC Manager will change to indicate which multi-axis mode is currently selected.

Note:
The modes available depend on the module you have licensed.

Before any modules are activated the only modes available will be:

3 Axis

Normal 3 axis environment. This is the default mode.

3 Axis Reposition

The 3 axis reposition mode simplifies the process of creating 3 axis toolpaths which are to be machined in different positions.

Adding the 4 axis module will activate these modes:

4 Axis Position

3 axis toolpaths on a stock with a rotation angle for the 4th axis. Defined on a plane at this angle.

4 Axis Full

A 3D model method which will mill a shape in incremental passes as it is rotated on the 4th axis.

4 Axis Wrap

Applies 3 axis stock or model toolpaths to the circumference of a 4 axis part.

Adding the 5 axis Position module will activate this mode:

5 Axis Position

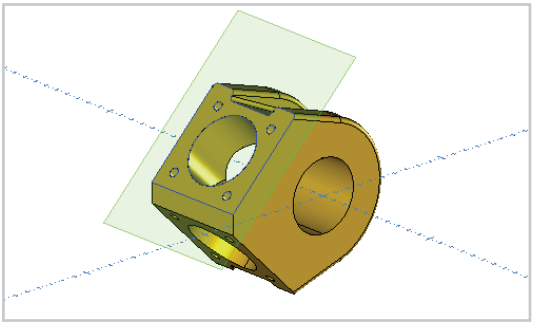
3 axis toolpaths defined on a plane with rotation angles for the 4th and 5th axes.

Adding the 5 axis Simultaneous module will activate this mode:

5 axis Simultaneous

Toolpaths which can move up to 5 machine axes at the same time.

For 4 and 5 Axis Position toolpaths, it is not necessary to rotate the part in the OneCNC model.



To simplify the toolpath definition, the Planes dialog is used to define a plane on the part, and the toolpath is defined relative to that plane.

When the part is machined it will rotate to the position the plane represents.

When editing a toolpath, the plane used when the toolpath was defined is automatically recalled so you do not have to change the plane you are working on when the edit is made.

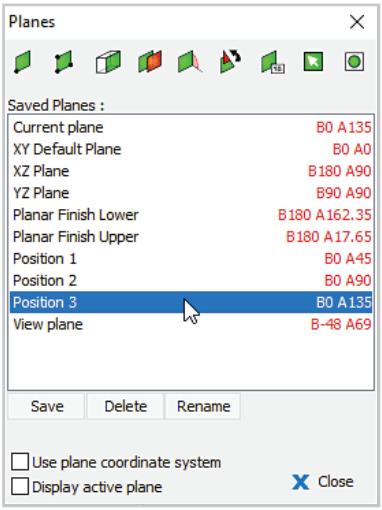
Note:
4 Axis Full and 4 Axis Wrap toolpaths do not use position planes. The Angular range of the toolpath is specified in the toolpath wizard settings.

The toolpath plane is fixed in space when the toolpath is defined, so once you have started programming a model it must not be moved, as the plane will not move with it.

For more information, see 'Precautions for multi axis toolpaths' in OneCNC Help > OneCNC Mill > Mill Multi Axis Machining.

Multi-axis parts can become complex, but there are some simple strategies you can follow to make the process easier.

As OneCNC uses planes for toolpath positions, you will need a plane defined for each position.

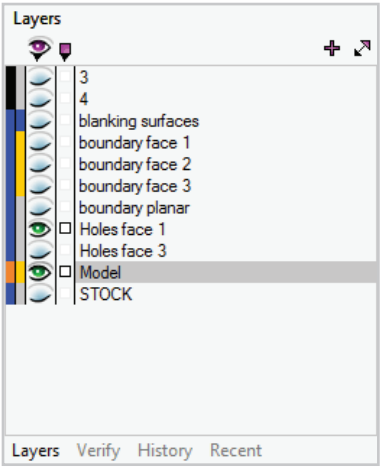


You will find your work is quicker and easier if you make a habit of defining and saving the planes you need at the outset. This way you can return to the exact plane used for a toolpath at any time.

Having a set of saved planes allows you to easily create the toolpaths for each tool in the actual order of machining.

Machine Layers

OneCNC can display a model of your machine in the toolpath preview. Layers 1 to 5 and the DMG_AXIS4 layers are used for machine preview, and should not be used for part modelling or geometry. Use Layer 3 for the body of the machine, Layer 4 for the 4th axis rotary or trunnion, and Layer 5 for the 5th axis table.



It is good practice to keep these layers turned off while defining toolpaths. As well as making the part model easier to see, having these layers off will reduce the processing time for Hole Feature Recognition and Model toolpaths.

STOCK Layer

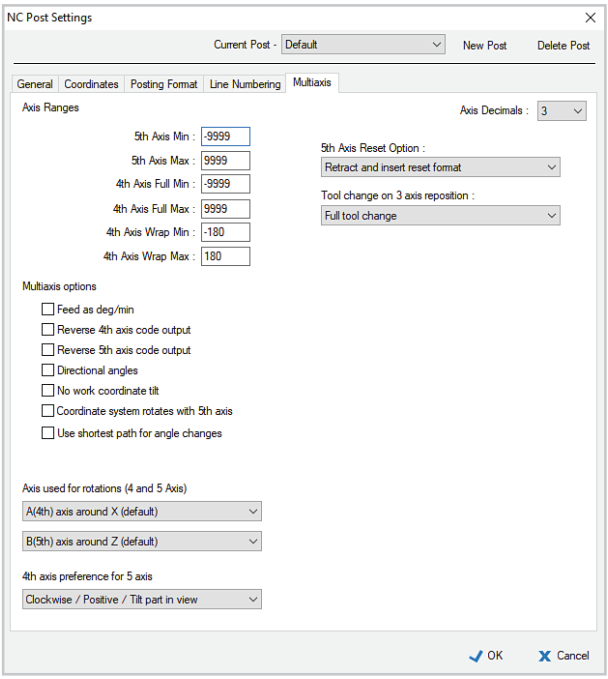
For an accurate simulation if you are machining a casting or pre-machined stock you can create a solid model for the simulation stock.

Create a layer named STOCK, then create one or more solids on the STOCK layer. As well as being useful for multi-axis operations, this means you can model clamps, fixtures and machine parts and they will be included in the simulation view.

You can program your part without the machine model in your part file, and then use the File Merge command to include the machine model file to check clearances.

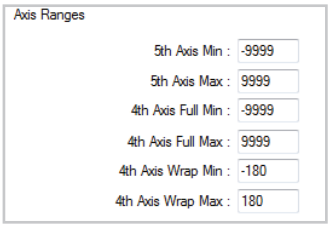
Multi-axis post settings

There is a Multi-axis tab in the NC Post Settings dialog.

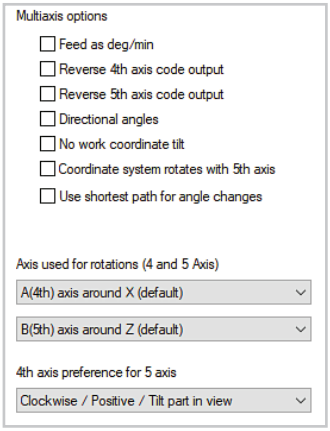


It is essential that the settings in this dialog are correct for your machine.

These settings directly affect the NC output and must not be changed without a sound knowledge of posting formats and the requirements for your machine.



The axis ranges determine how far the rotary axis can rotate in each direction.



The other settings are determined by the Post requirements for your machine.

If your machine has the 4th axis aligned with the X or Z axis, select the '4th Axis Around X' option.

If your machine has the 4th axis aligned with the Y axis, select the '4th Axis Around Y' option.

For details of Multi-axis Post settings, see the NC Post and Properties topic in the Mill NC Post and DNC Settings section of OneCNC Help.

The Multi-axis Post Settings topic in the Mill Multi-axis Machining section has additional information.

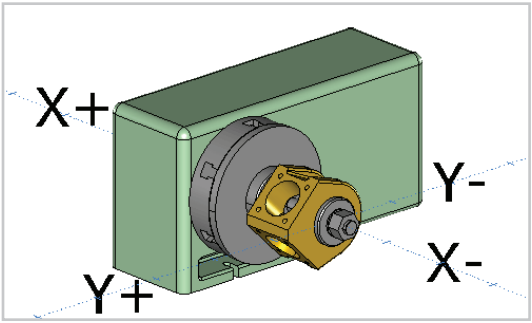
For 4th axis wrap programming the part must be on the X axis in OneCNC.

When you select the multi-axis machining mode, match the Axis of Rotation setting to the axis the 4th is aligned with on your machine.

This setting combined with the correct 4th Axis Around setting in the post will produce the correct output for your machine.

OneCNC 4 Axis Position Machining

This tutorial is intended for OneCNC Mill Expert or Mill Professional with 4 Axis or 4 & 5 Axis modules activated.

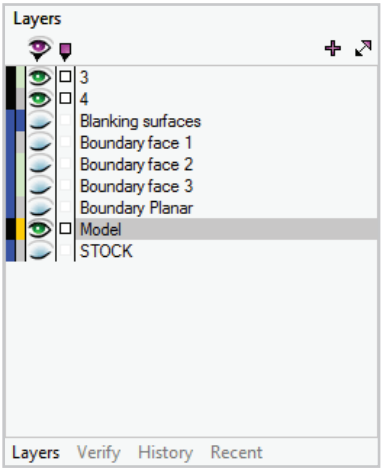


Open the sample file 'mill expert 4 axis.ONECNC', and save a copy as 'Tutorial 4 axis position.ONECNC' Note the position of the rotary axis which is on the X axis. The degrees of rotation for this axis will be written as the A axis degree values in the NC file.

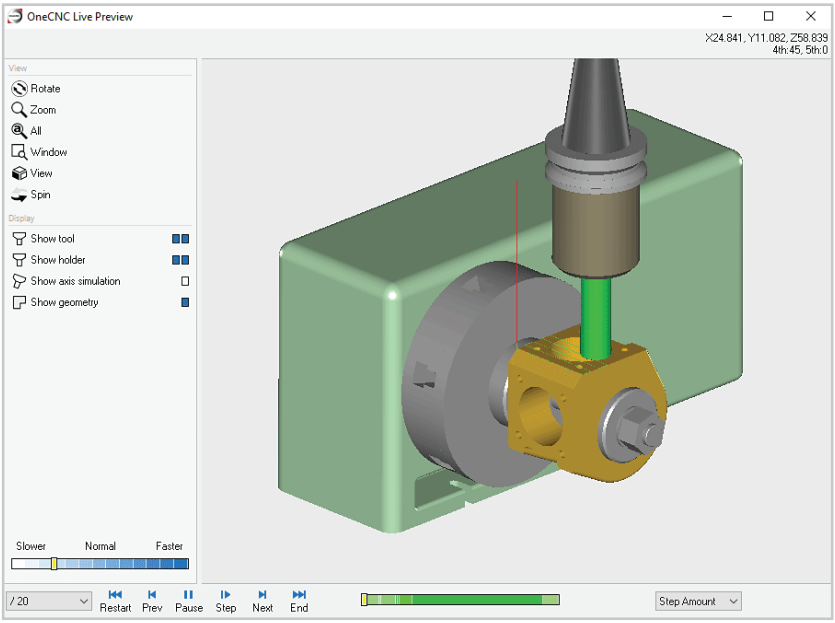
Open the layers tab to see how the part file is arranged.

The layer named 3 holds the machine body which does not move in the preview mode. The layer 4 holds the rotary table, and the part is on the model layer.

A model of the casting to be machined is on the STOCK layer, which Mill Expert uses to define the simulation stock. The remaining layers contain boundary geometry for the toolpath definitions.

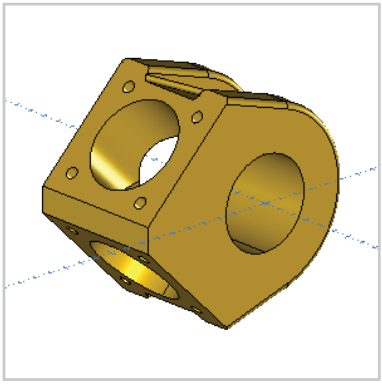


Preview the example toolpath group and observe the full machine simulation in action.



Select show axis simulation in the toolbox, and you will see the 4th axis rotates to each position as it is required.

Return to the part environment and we will begin recreating the operations you have just observed.



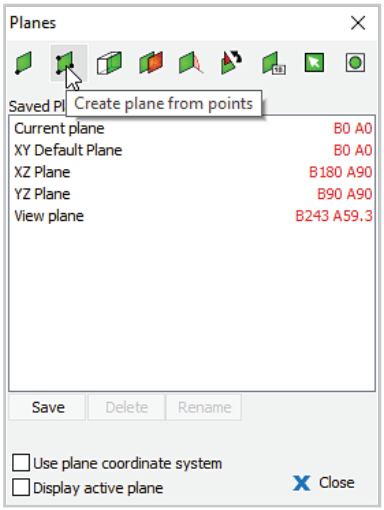
Before continuing, turn off layer 3 and layer 4, so the machine model is not visible.

Creating work planes

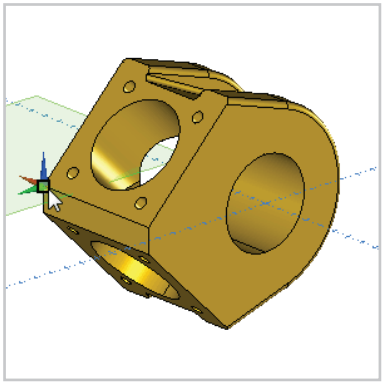
The part model does not have to be moved to create 4th axis position toolpaths. The required positions are defined by a construction plane at the required angle.

The plane used must be tangential to an imaginary cylinder centered on the X axis. The toolpaths created in this mode will then rotate the part to the orientation of the construction plane when it is machined. To simplify the toolpath clearance settings the plane is regarded as Z0 when the toolpath is created.

The first step for creating toolpaths in positions rotated on the X axis is to define the planes required. While working on a part, you may need to return to a plane a few times as you define various operations. It is good practice to save all the planes you need before you start defining toolpaths.

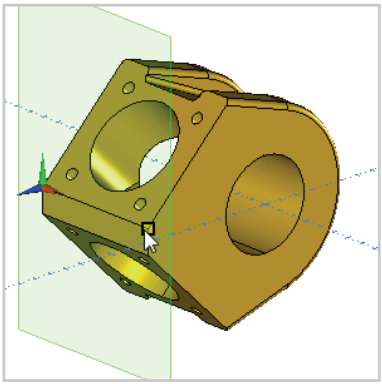


To create a plane for the first position, open the Planes dialog, and click on Create plane from points.

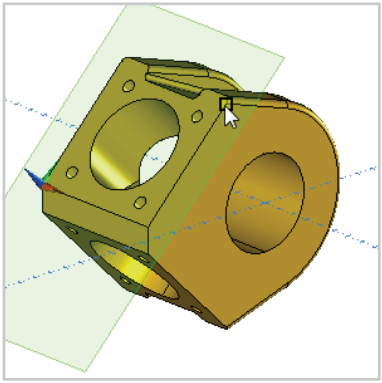


Click on the endpoint at the lower left of the face as shown.

This point will be the origin of the new plane.

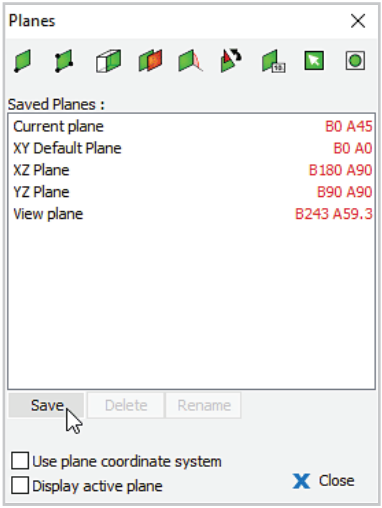


Click on the opposite corner to set the X direction of the plane.



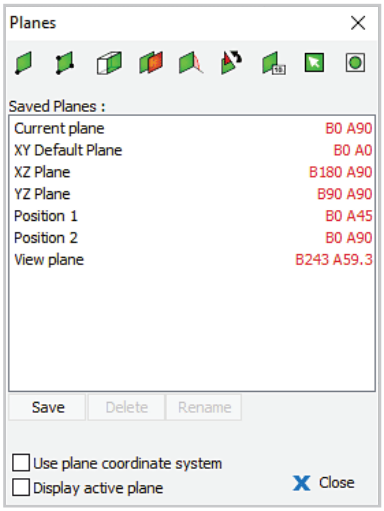
Click on the corner shown to set the Y direction of the plane.

The plane will now be created on the surface.

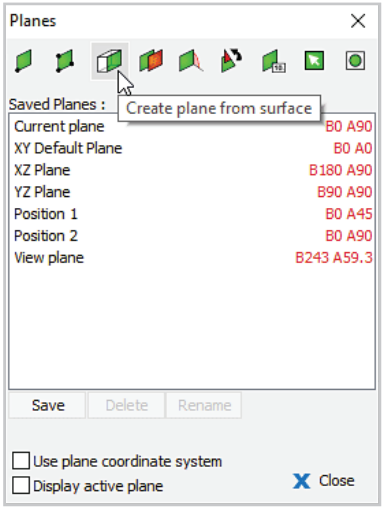


To save the plane, open the Planes dialog and click on the Save icon.

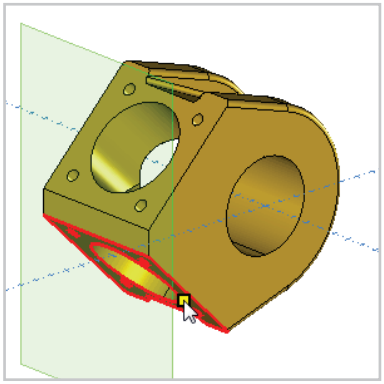
Enter Position 1 for the plane name, and click OK.



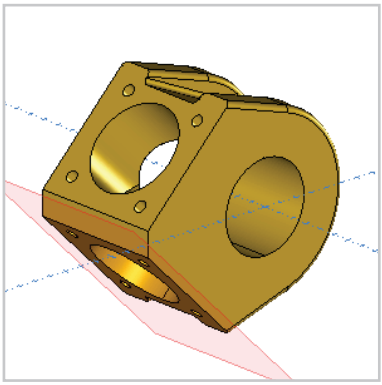
Working anticlockwise around the part, use the Plane from points command to create a plane on the adjoining small surface, and save it as Position 2.



To create the third plane, click on the Plane from surface icon in the Planes dialog.

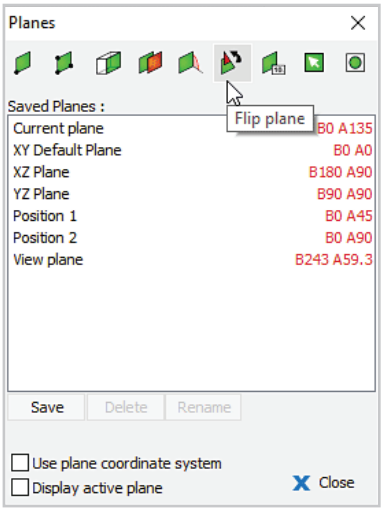


Click on the lower face to create the plane.



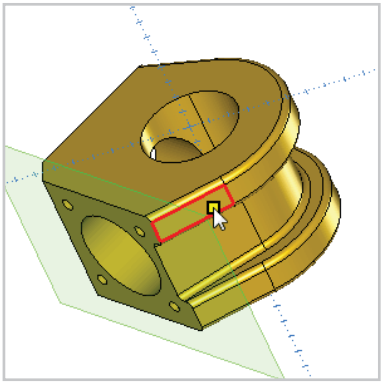
The plane is created but when using the Plane from surface method the orientation of the plane is dependent on how the surface was constructed.

In this case the positive side of the plane is facing the part.

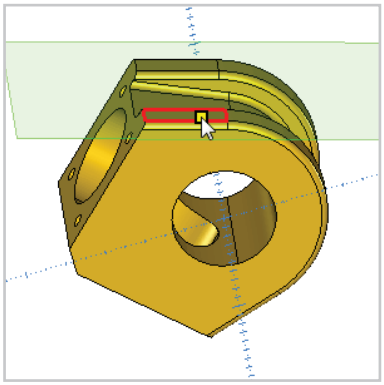


Open the Planes dialog and click on Flip Plane to correct the orientation of the plane.

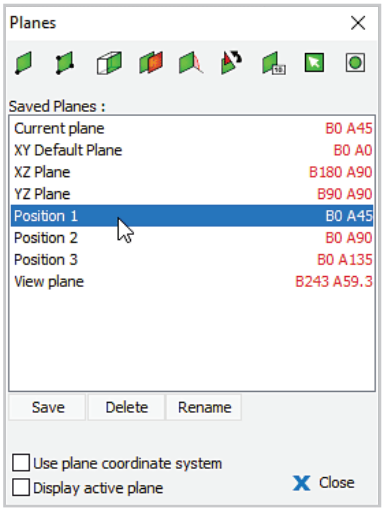
Save the plane as Position 3.



Create a plane on one of the flats at the bottom of the part and save it as Planar Finish Lower.



Create a plane on one of the flats at the top of the part, and save the plane as Planar Finish Upper.

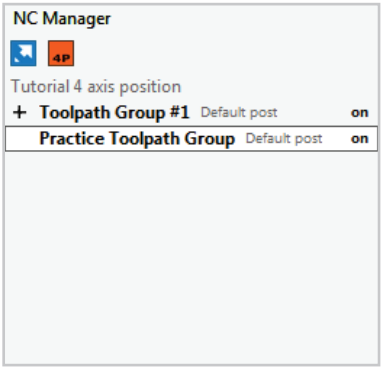


Select Position 1 as the current plane before proceeding.

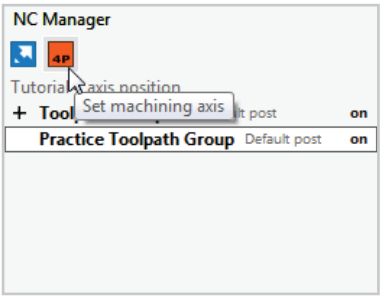
Preparing for machining operations

Create a new toolpath group to re-create the toolpath operations in, and rename the Description in the Process dialog to Practice Toolpath Group.

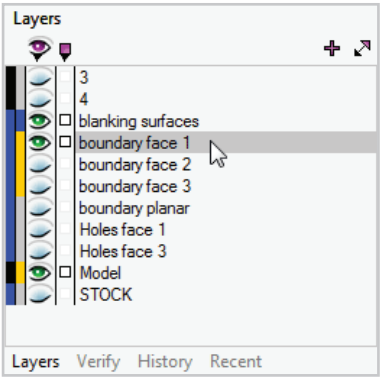
You can click on the icon next to the example Toolpath Group #1 to collapse it.

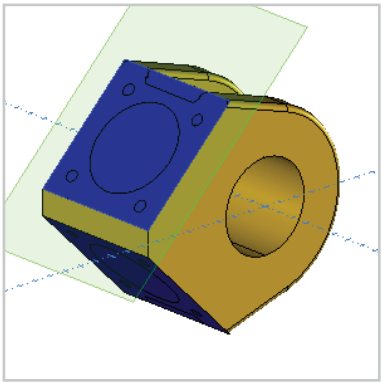


The axis mode icon should show that the 4 Axis Position machining mode is active.



Turn on the blanking surfaces and boundary face 1 layers.



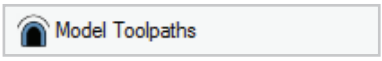


You will now have an unobstructed view of the part, ready for a toolpath definition with a boundary on the first face and blanking surfaces covering the holes.

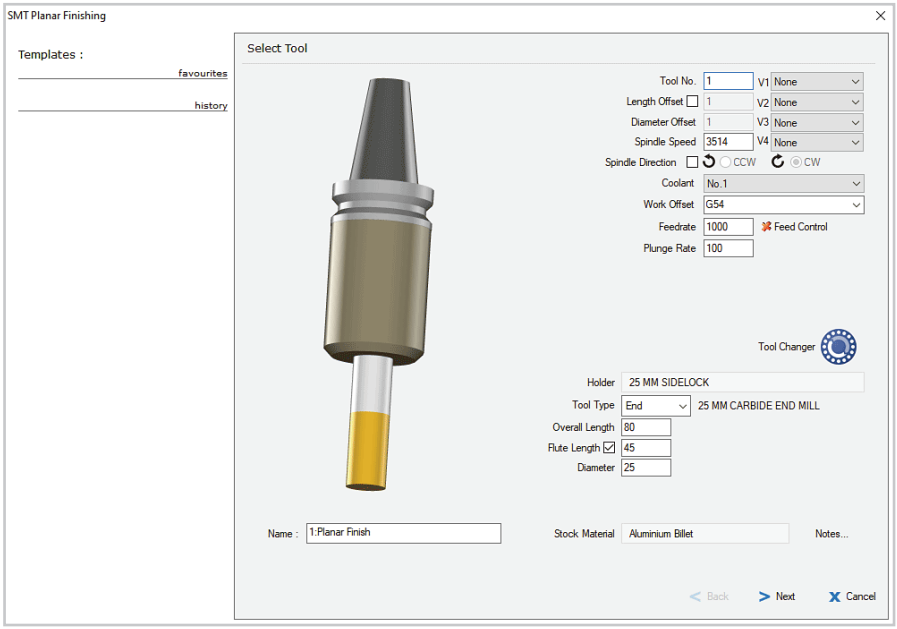
We will begin by showing how the Planar Finish model toolpath can be used with the blanking surfaces to create a facing operation.

Note:

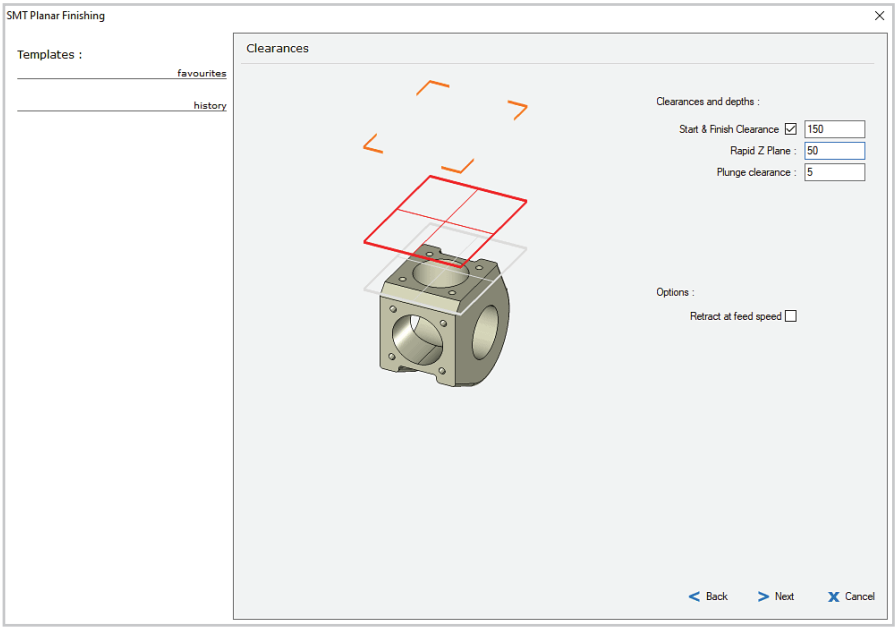
Because the blanking surface and model surface are coplanar, they may display with an interference pattern. This is an effect of screen rendering only, and will not affect the toolpathing process.



Click on the Model toolpaths icon, and select Planar Finish.



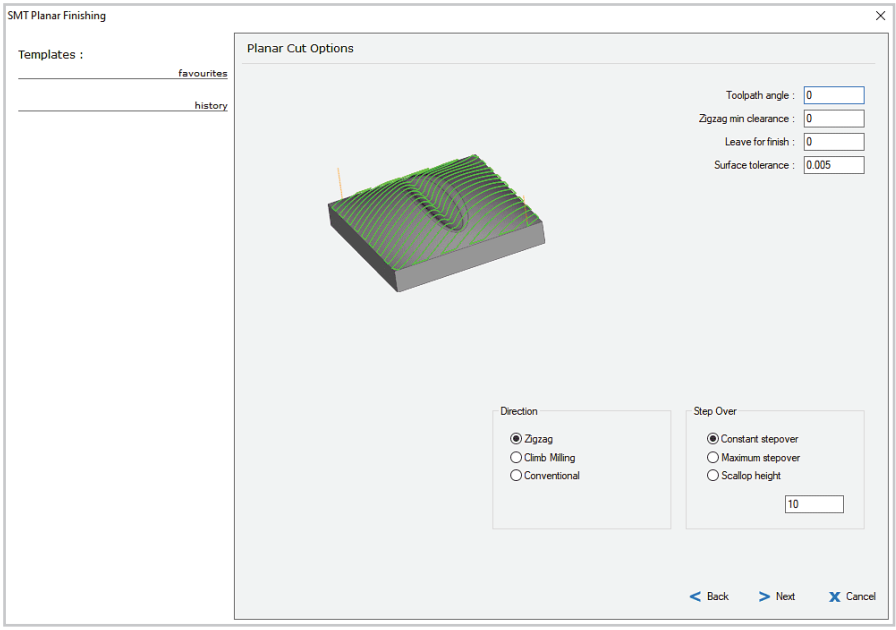
In the Select Tool dialog, select the 25MM CARBIDE END MILL tool, and set Feedrate manually to 1000, and Plunge Rate to 100.
Click Next to continue.



In the Clearances dialog, the Z levels for the toolpath are defined. These levels are all relative to the active plane.

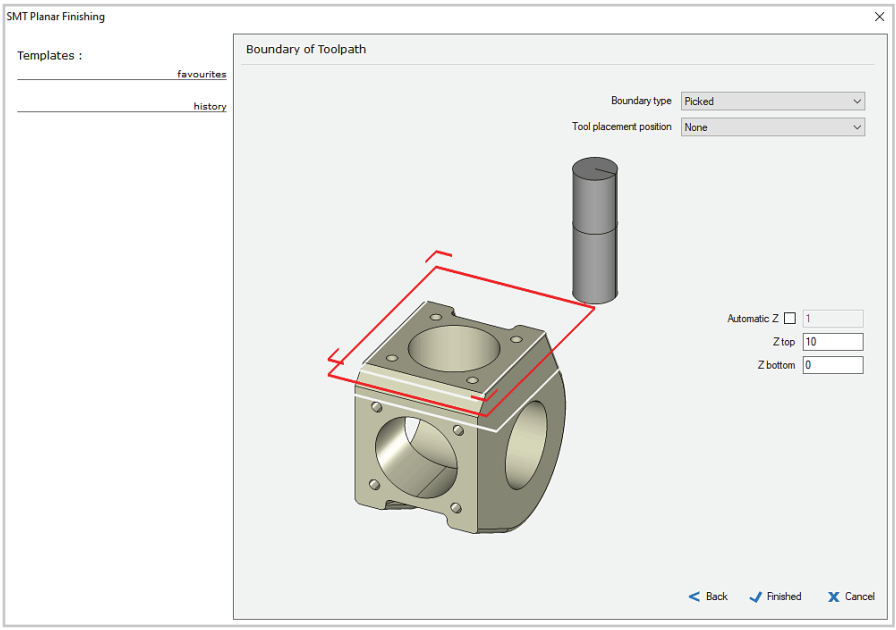
Select Start and Finish Clearance and set it to 150. This will ensure the tool is high enough to clear the part as it rotates while indexing.

Set the Rapid Z Plane to 50, and Plunge clearance to 5. Click Next to continue.



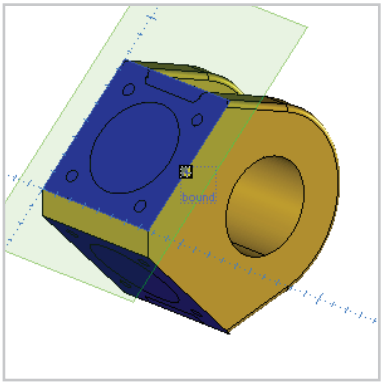
To make the Planar cut toolpath act as a Facing toolpath, enter 0 for Toolpath angle, Zigzag minimum clearance and Leave for finish.

Select the Zigzag option for direction, and select Constant stepover with a stepover value of 10.



For the Boundary type, select the Picked option.
Select None for tool placement, which allows the cutter centerline to travel up to the boundary.

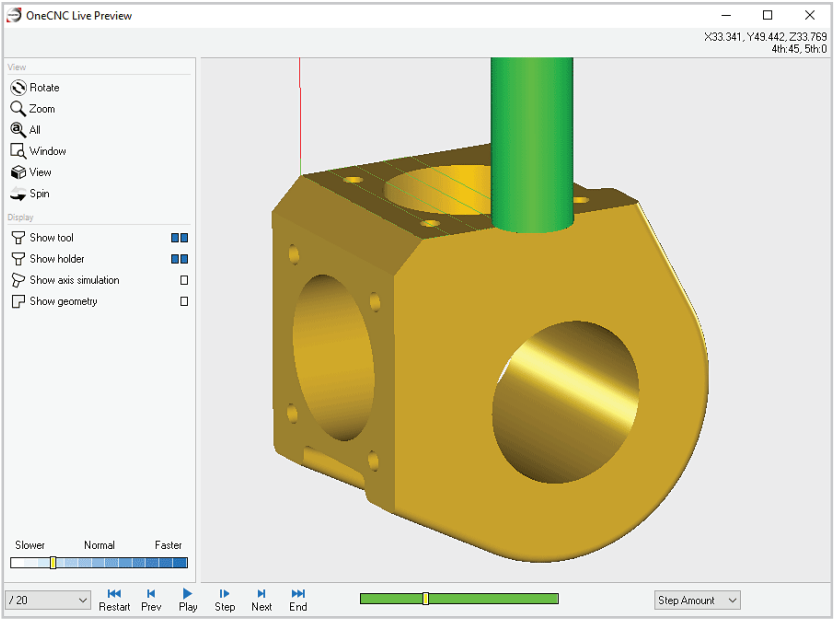
Clear the Automatic Z Offset checkbox. Set Z Top to 10, and Z Bottom to 0. Click Finished to end the toolpath input stage.



For the toolpath boundary, select the rectangle on the boundary face 1 layer, which is at the edge of the blanking surface.

Right click to end the boundary selection and the Planar Finish toolpath will be created.

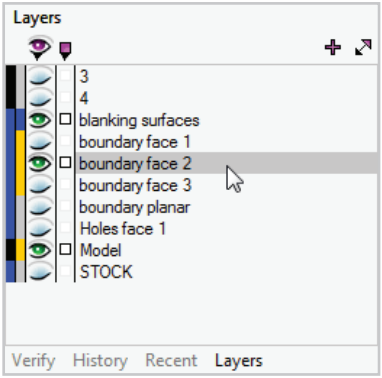
Right click on the new operation and select Preview toolpath.

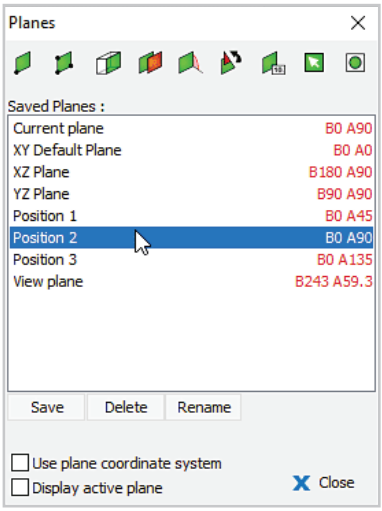


Click on the Axis simulation icon in the Live Preview toolbox to activate axis simulation. The model will rotate into the required position before the milling starts.

Close the Live Preview window when you are ready to continue.

To prepare for milling the next face, turn off the boundary face 1 layer, and turn on the boundary face 2 layer.

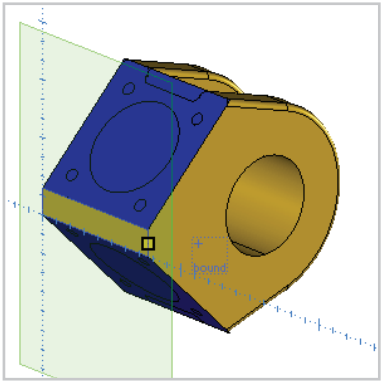




Select the Position 2 plane.

Select the Planar Finish toolpath again.

This toolpath will use the same settings as the previous toolpath, so you can click through the dialogs of the wizard without making any changes, and click Finished to close the toolpath wizard.

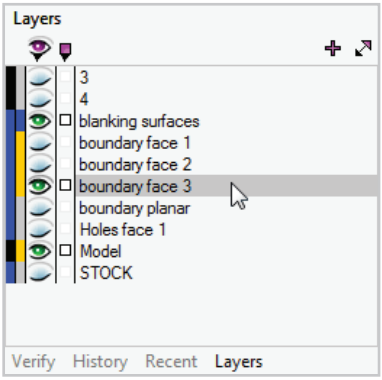


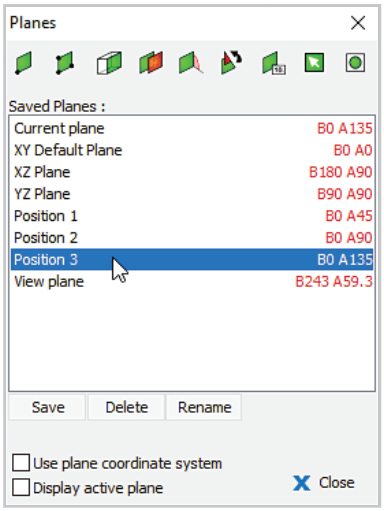
Select the boundary on the narrow face to define the new toolpath.

Right click to end the boundary selection and the Planar Finish toolpath will be created.

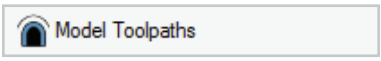
We will now define a Planar Finish toolpath for the third face in the same way as the first two were done.

To show the boundary for the third face, turn on the boundary face 3 layer, and turn off the boundary face 2 layer.



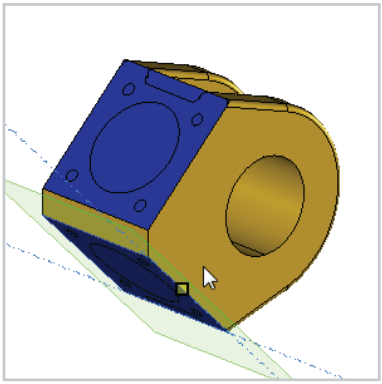


Select the Position 3 plane.



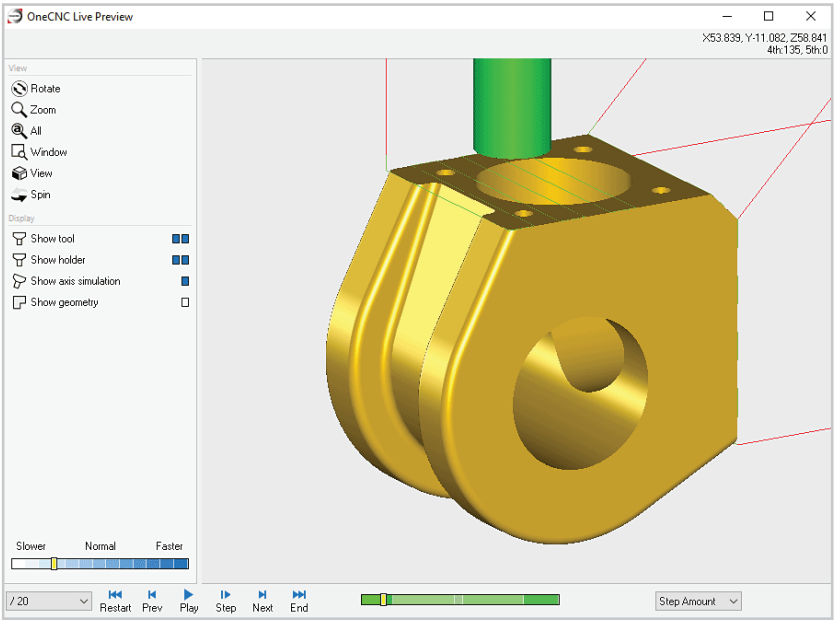
Select the Planar Finish toolpath again.

Click through the dialogs of the wizard without making any changes. Click Finished to close the toolpath wizard.



Select the boundary on the third face to define the new toolpath.

Right click to end the boundary selection and the Planar Finish toolpath will be created.



Preview the three facing toolpaths before continuing.

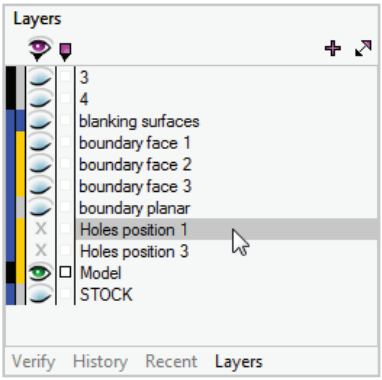
Hole operations

The next toolpaths to define are the hole operations.
Hole feature recognition can recognize circular holes in models or geometry, but the large circular bores will be filtered out as they do not have a circular base. We will use geometry for the Hole feature recognition.

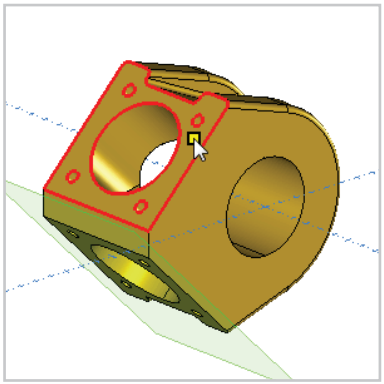
Turn off all layers except the model layer.

Create two new layers and name them Holes position 1, and Holes position 3.

Select Holes position 1 as the current layer.

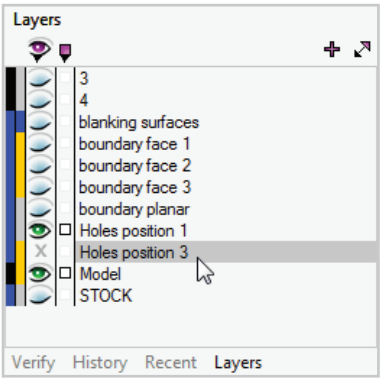


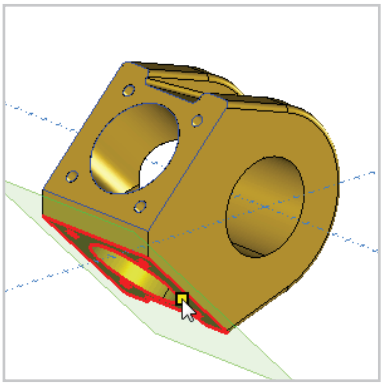
Click on the Model Tools icon in the toolbox and select Extract surface edges.



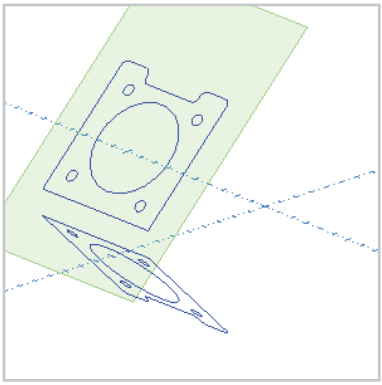
Click on the position 1 surface to extract an outline of all the edges to the Holes position 1 layer.

Change the active layer to Holes position 3.





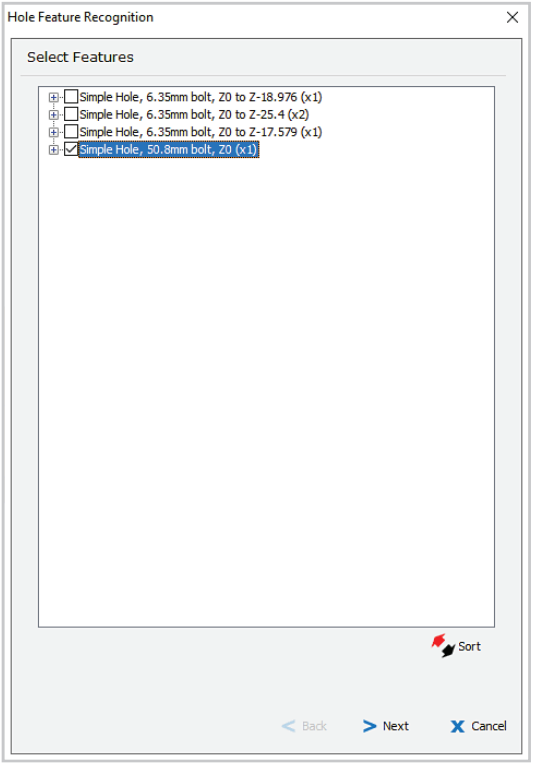
Click on the position 3 surface to extract an outline of all the edges to the Holes position 3 layer.



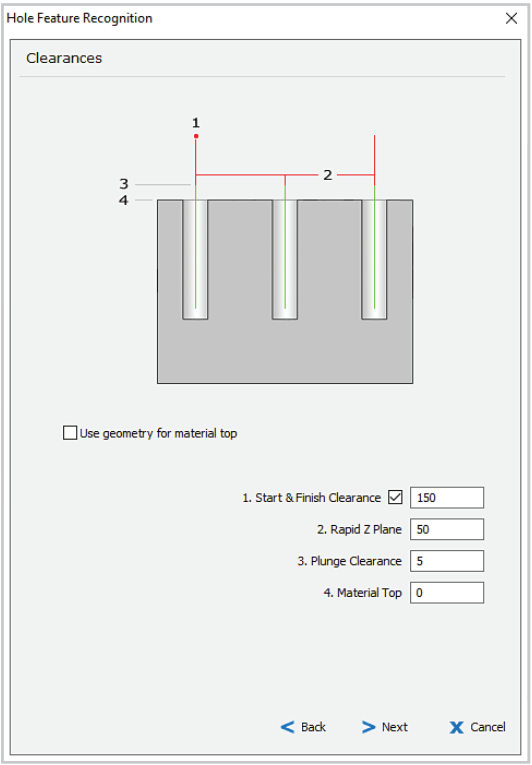
We will begin on the first face, so select the Position 1 plane.

Turn off the model layer so you can see the geometry clearly.

Click on the Drill Hole Wizard icon in the Stock toolpaths menu.



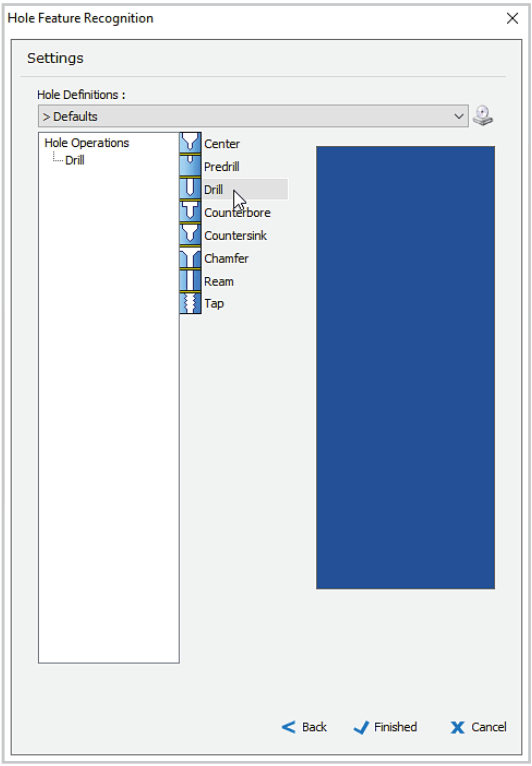
Select the 50.8mm hole for machining.



Select Start and Finish Clearance and set it to 150. This will ensure the tool is high enough to clear the part as it rotates while indexing.

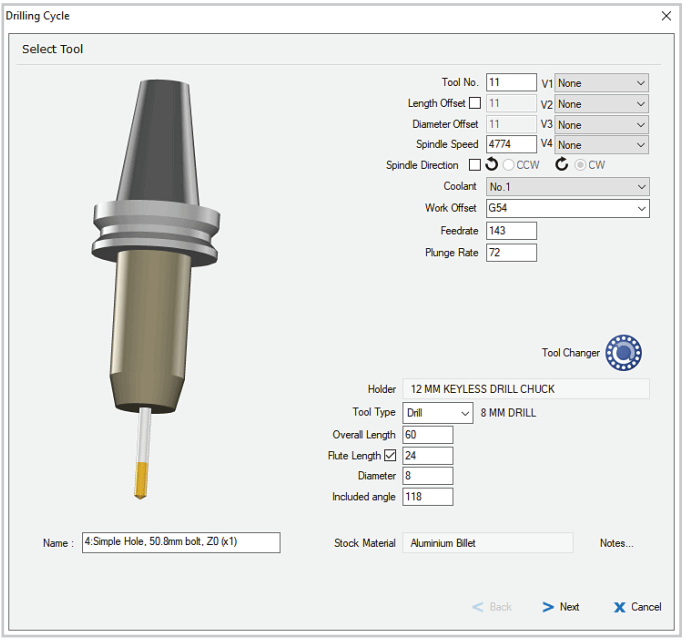
Set the Rapid Z Plane to 50, and Plunge clearance to 5. Click Next to continue.

The Material Top is 0, which is the Z0 of the position plane.



Click the Drill icon to add a drill operation to the Hole Operations list.

Double click on the Drill operation to enter the settings for it.

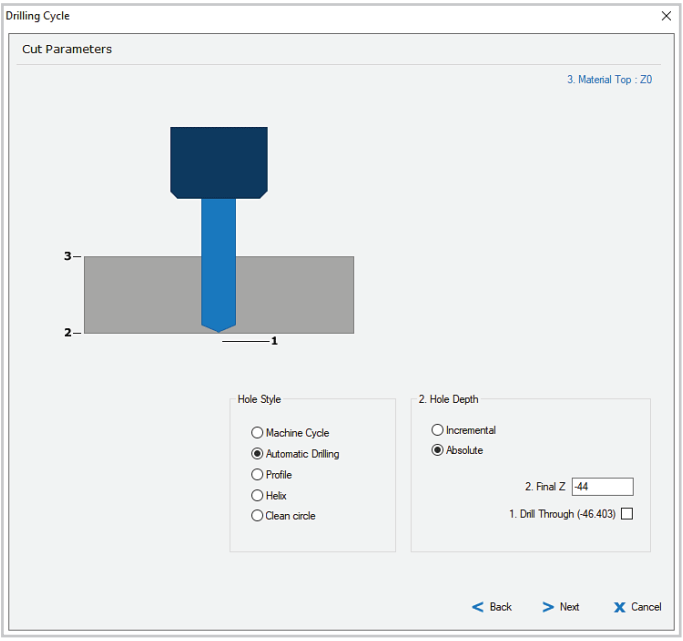


The first step in defining the Drill operation is to select the 8mm Drill from the Tool Library.

Enter Tool number 11, and set Coolant to No1 and Work Offset to G54.

Select Aluminium Billet from the Material List to set the feedrates and spindle speed.

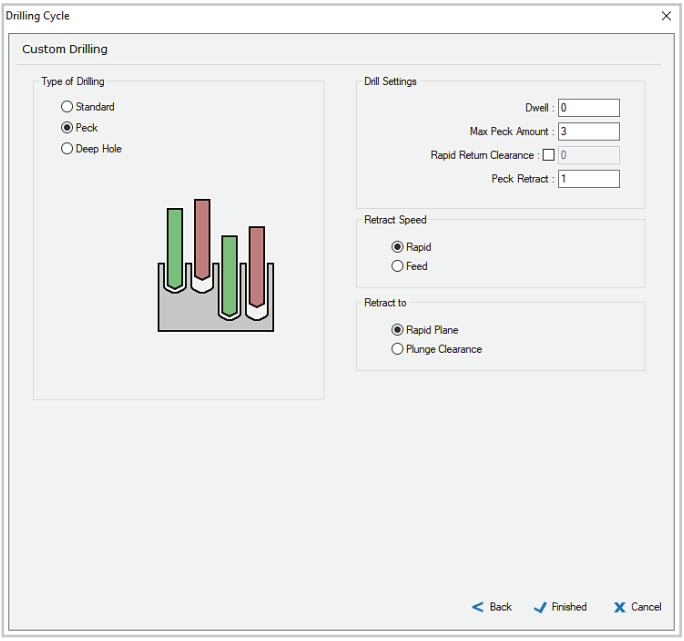
Click Next to continue.



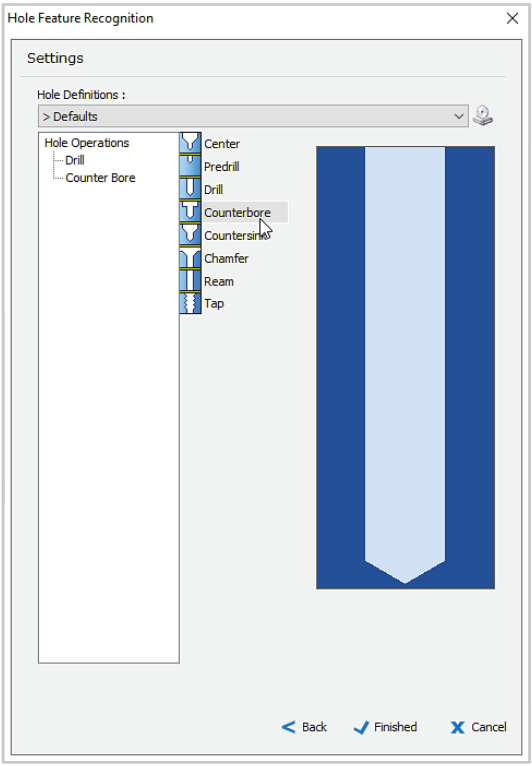
Select Automatic drilling and Absolute depth style.

Enter a Z depth for the hole of -44. This is the depth relative to the plane.

Click Next to continue.

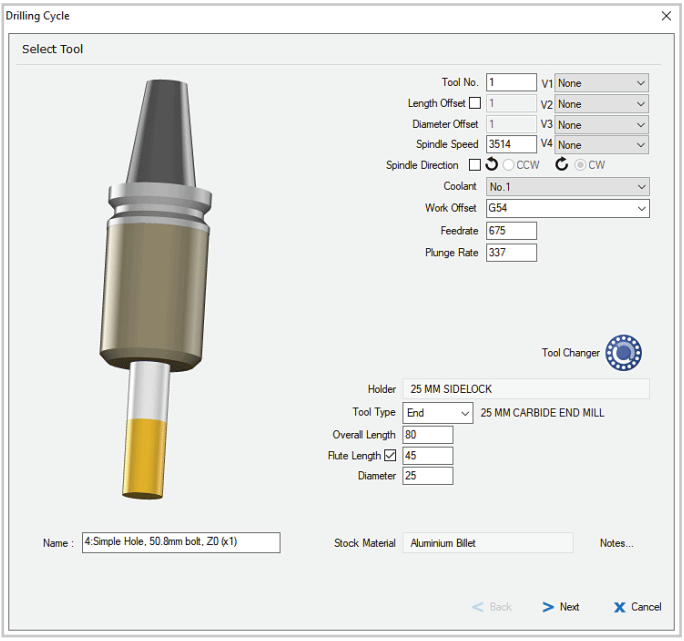


Select the Peck option of the Automatic drilling method, with a Max Peck Amount of 3. Click Finished to complete the Drill definition.



Add a counter bore operation to the Hole Feature definition list.

Double click on the counter bore operation to enter the settings for it.

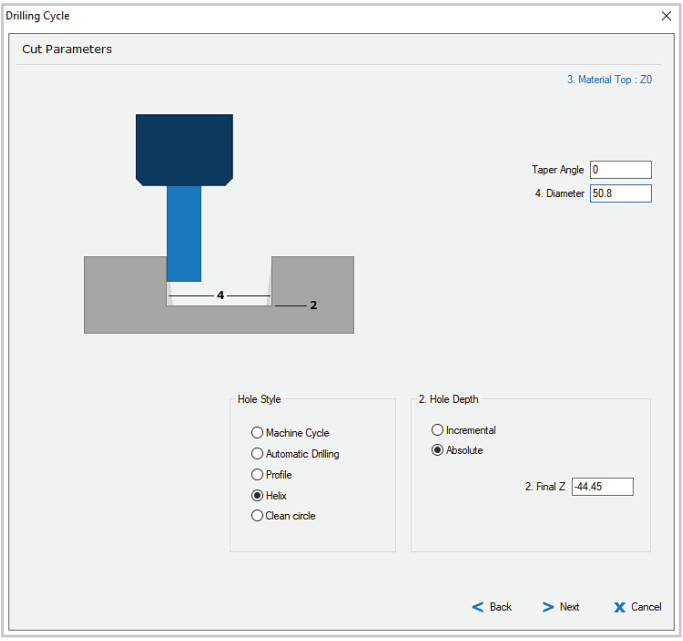


In the Select Tool dialog, select the 25MM CARBIDE END MILL tool.

Enter Tool number 1, and set Coolant to No1 and Work Offset to G54.

Select Aluminium Billet from the Material List to set the feedrates and spindle speed.

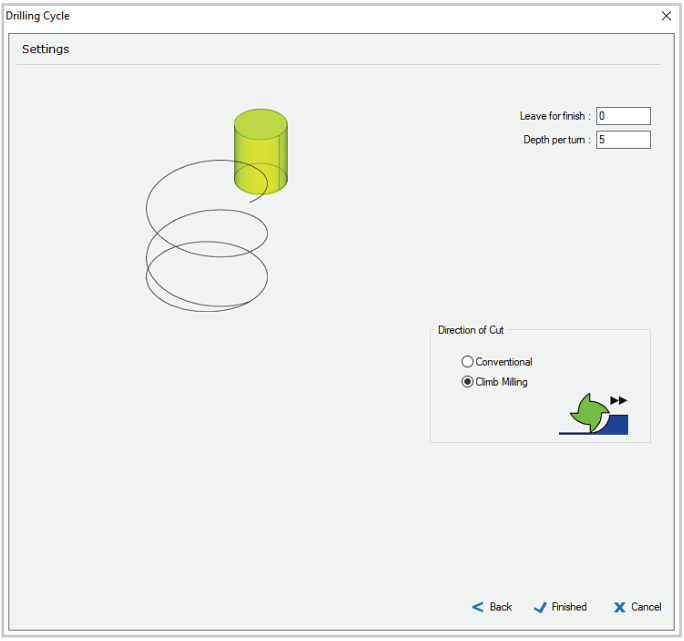
Click Next to continue.



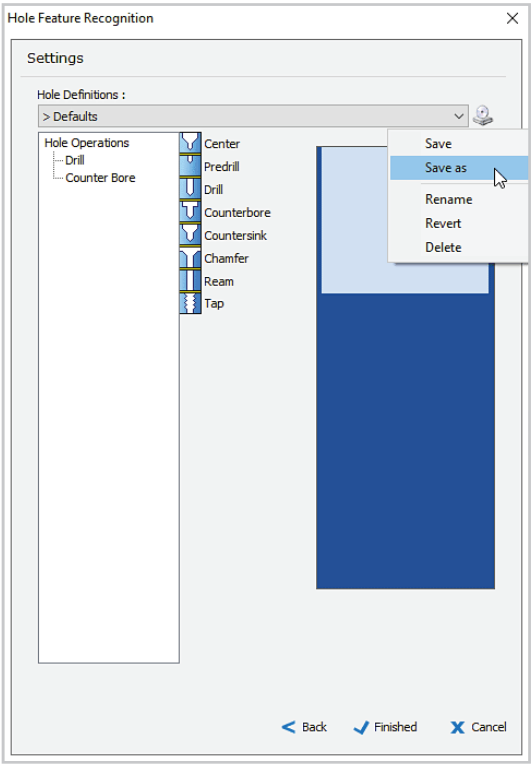
Select the Helix method and enter a diameter of 50.8.

Select Absolute depth style and enter a Z depth for the hole of -44.45.

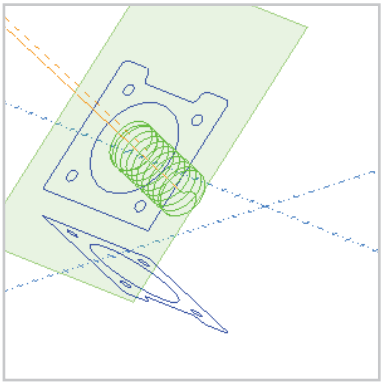
Click Next to continue.



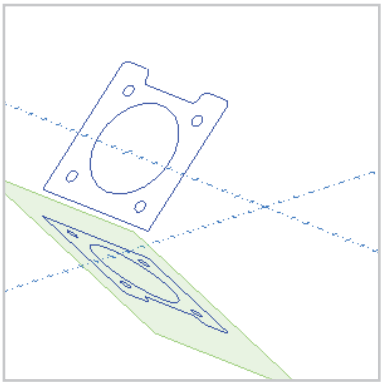
In the Helix settings dialog, set Leave for finish to 0, and Depth per turn to 5. Click Finished to return to the Hole Feature dialog.



Save the operations to complete the 50.8 hole as '50.8 predrill and bore' before clicking Finished.

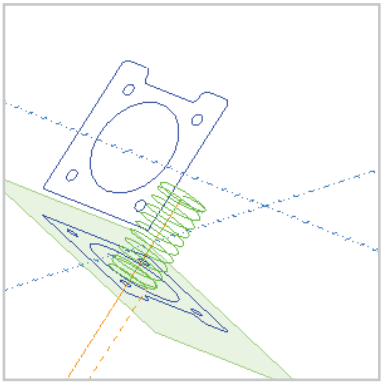


The drill and counterbore operation will be seen in the instant preview.

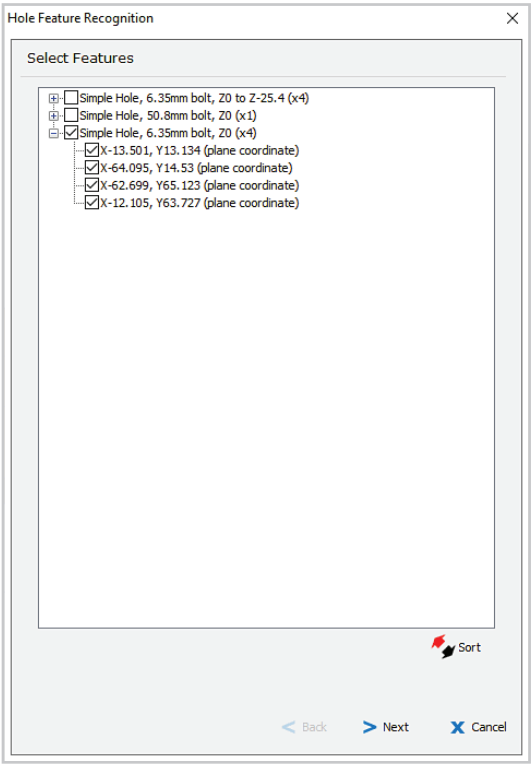


To bore the other 50.8 hole, select the Position 3 Plane.

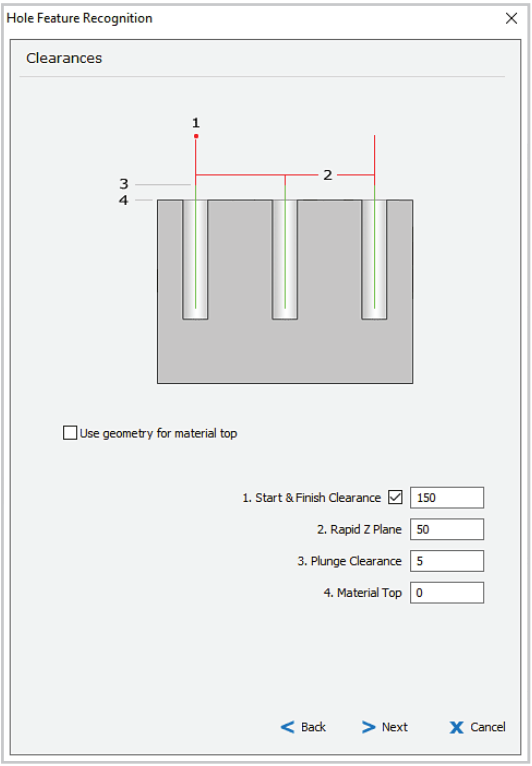
Start Hole Feature Recognition and select the 50.8 hole in the Select Features dialog.



Apply the '50.8 predrill and bore' definition to machine the bore using the same settings as the first hole.

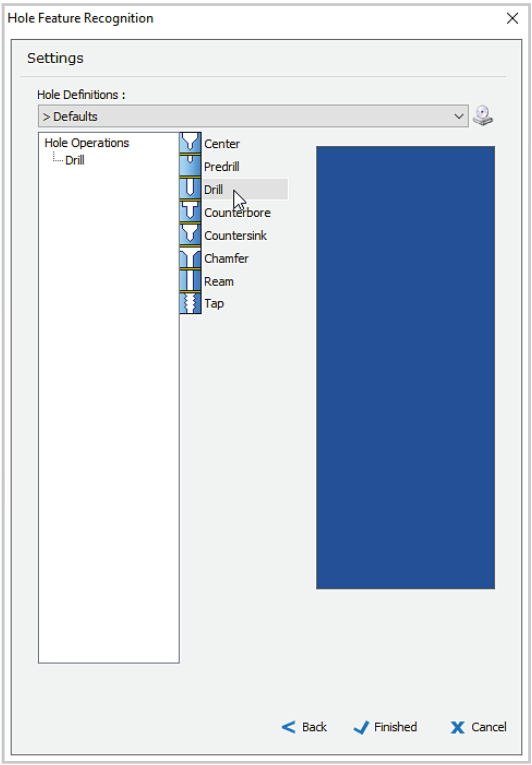


To drill the 6.35mm holes in position 3, start the Drill Hole Wizard again and select them in the Select Features dialog.



Use the same clearances as for the previous hole operation.

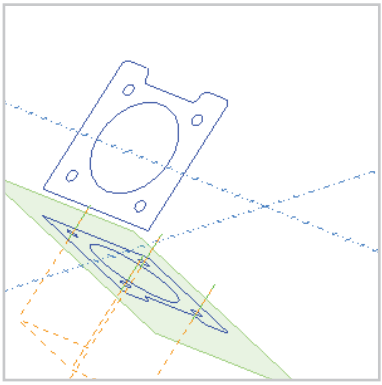
Click Next to continue.



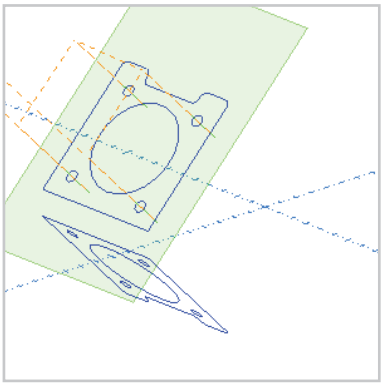
Add a drill operation and double click it to define the settings.

The settings will be similar to the 8mm drilling we defined earlier, but select the 6mm drill from the tool library and edit the diameter to 6.35. The depth will be -17.

Save the settings as '6.35 drill Z-17', and click Finished.

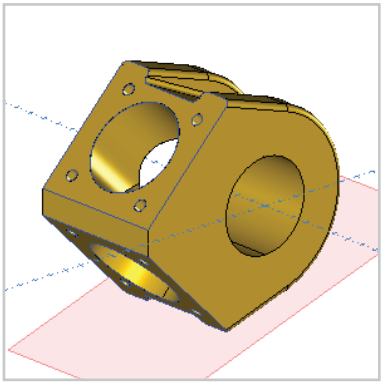


The drilling will be seen in the instant preview.



Change to the Position 1 plane, and start the Drill Hole Wizard again.

Apply the '6.35 drill Z-17' hole definition to the four holes in Face 1.



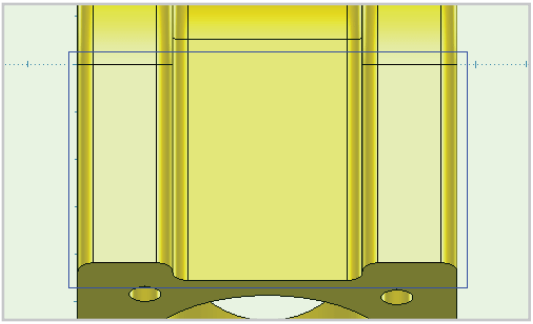
Next we will define Planar Finish toolpaths at the top and bottom of the outer surface of the part.

Before continuing, turn on the model layer and activate the plane on the bottom of the part which you saved as Planar Finish Lower.

Planar finishing

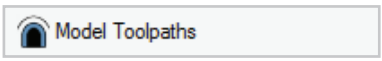
Select the Plane option from the view menu so you are looking directly at the plane.

Create a layer named boundary planar, and draw a rectangular boundary for the toolpath as shown.

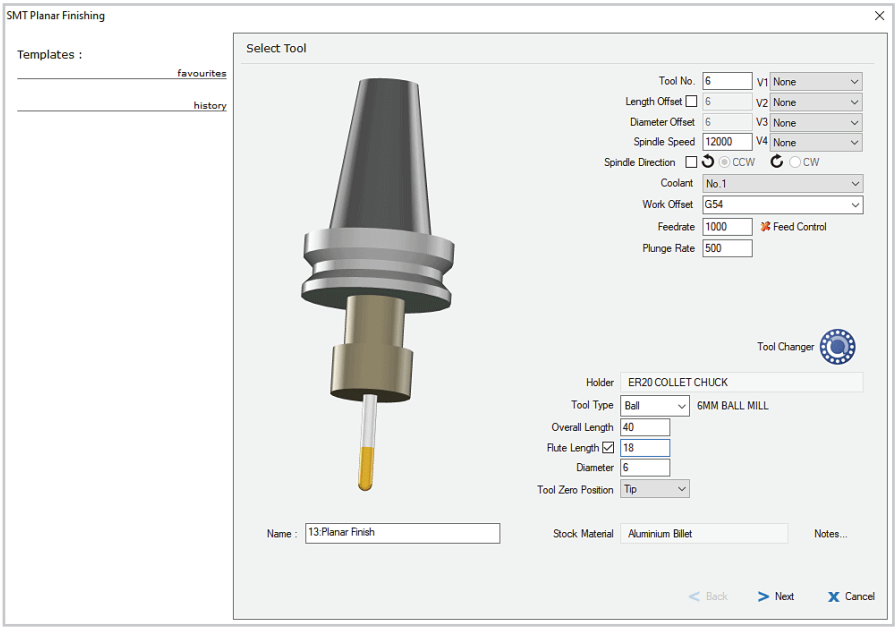


You can use grid snaps to place the corners of the rectangle near the fillet vertices.

Click on the Model Toolpaths icon and select Planar Finish.



For parts that require a precise boundary you can extract edges from the model and project them to the plane if necessary.

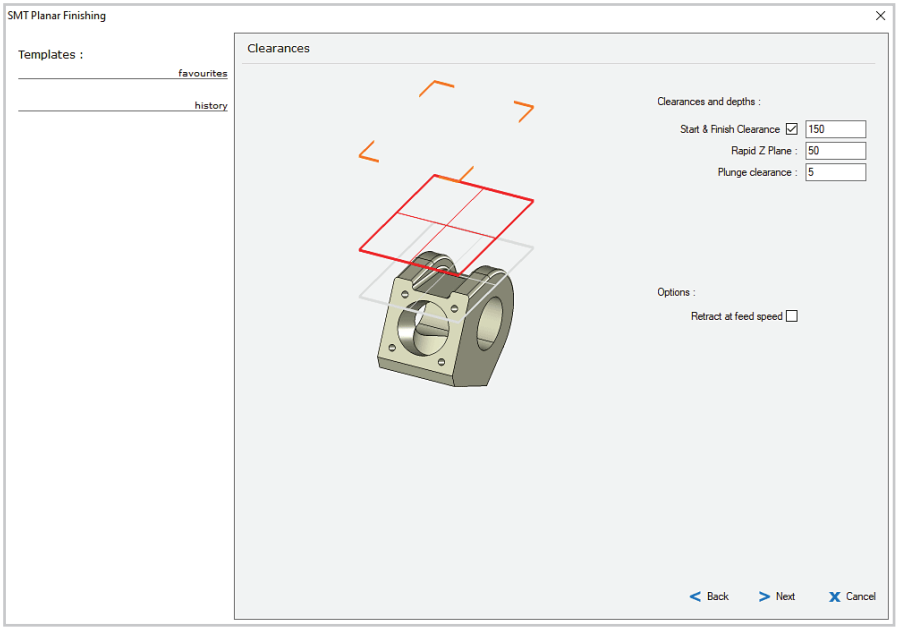


Select the 6 MM CARBIDE BALL MILL tool.

Enter a Spindle speed of 2000.

Set the Feedrate to 1000 and the Plunge feed to 500.

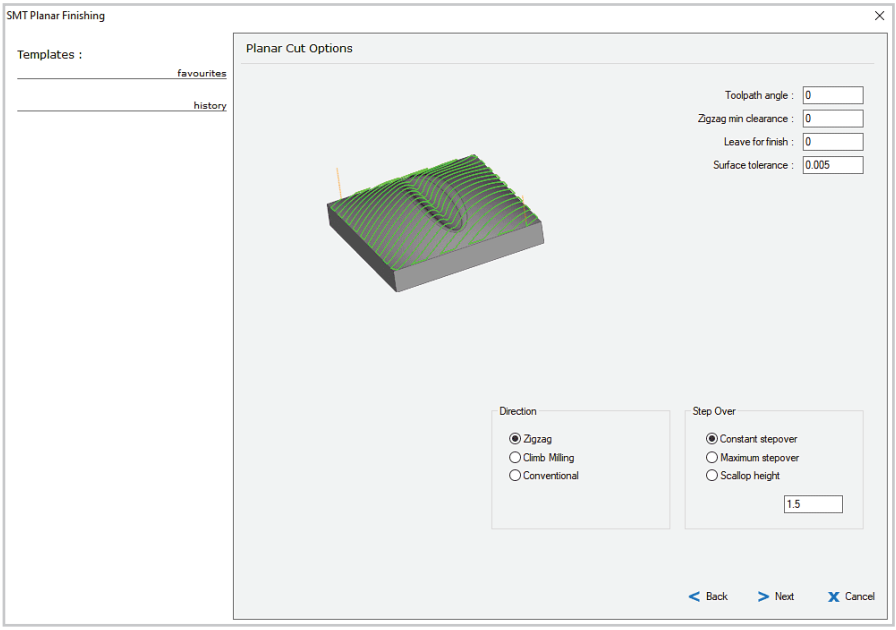
Click next to continue.



Select Start and Finish Clearance and set it to 150. This will ensure the tool is high enough to clear the part as it rotates while indexing.

Set the Rapid Z Plane to 50, and Plunge clearance to 5.

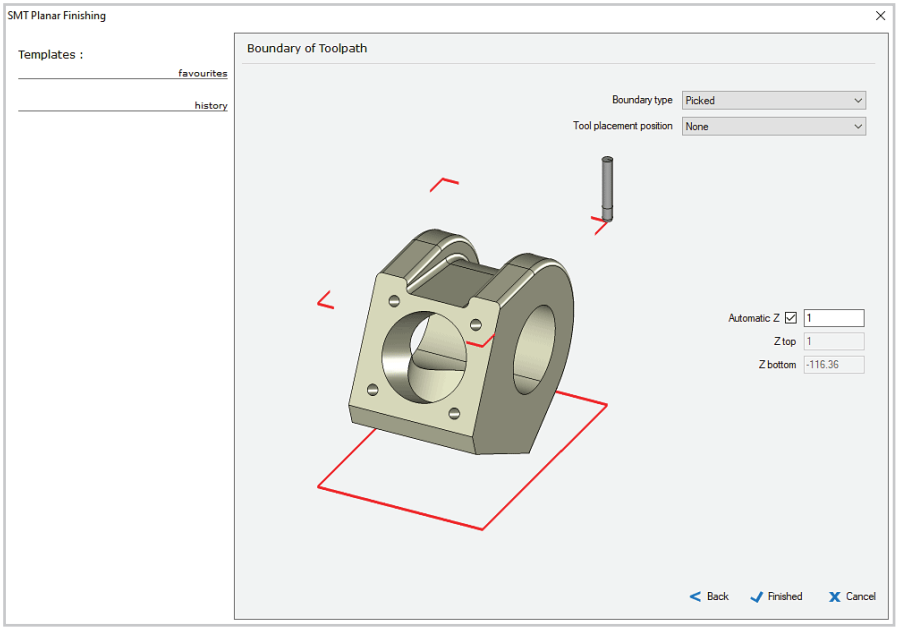
Click Next to continue.



Enter 0 for Toolpath angle, Zigzag minimum clearance and Leave for finish.

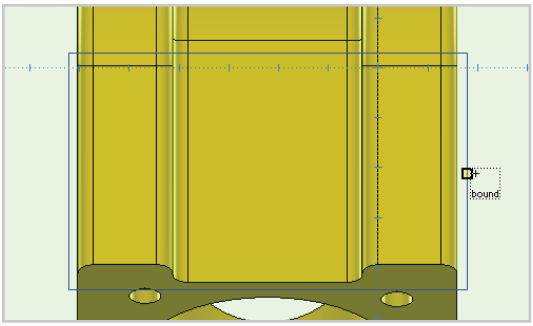
Select the Zigzag option for direction, and select Constant stepover with a stepover value of 1.5mm.

Click Next to continue.

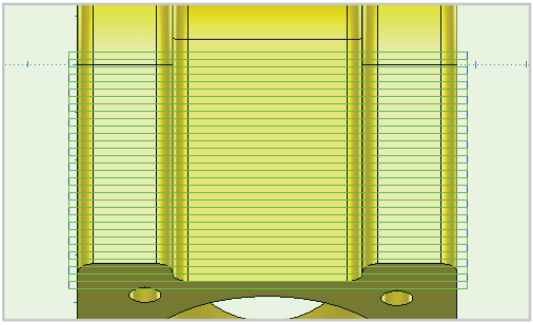


For the Boundary type, select the Picked option.
Select None for tool placement, which allows the cutter centerline to travel up to the boundary.

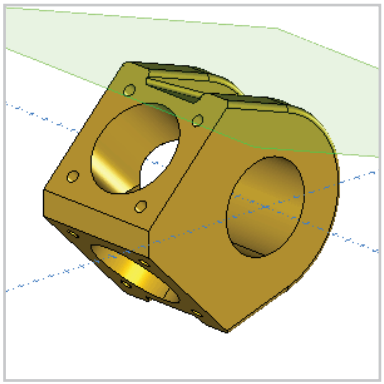
Set an Automatic Z offset of 1 and click Finished.



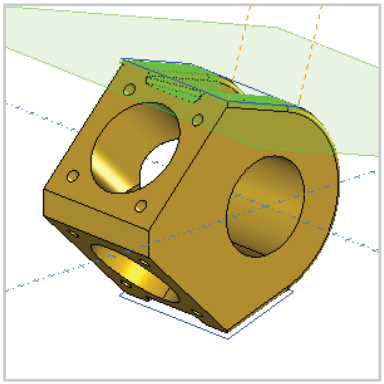
Select the rectangle as the boundary for the toolpath.



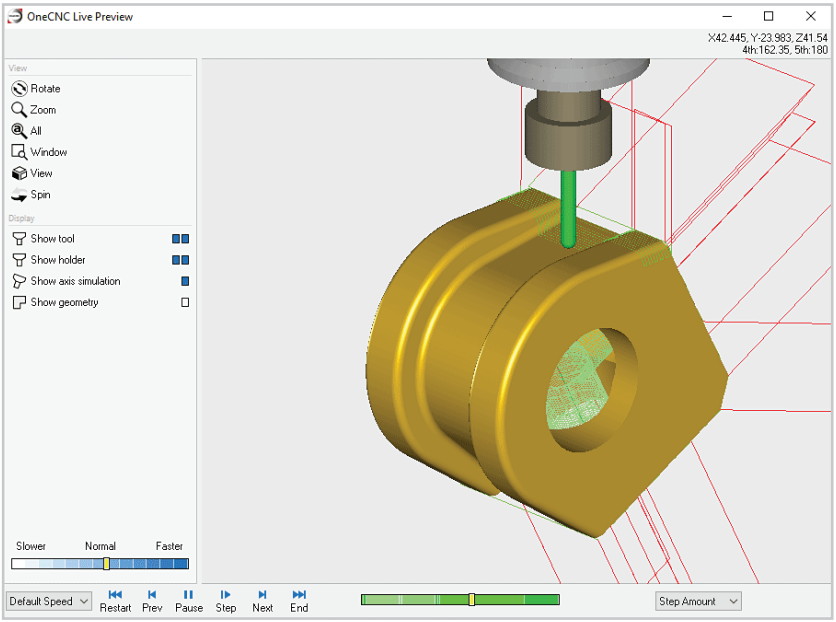
Right click to end the selection process and the toolpath for the lower section of the outer shape will be created.



Activate the plane on the top of the part, which you saved as Planar Finish Upper.



Change to the Plane view again and create a Planar Finish toolpath for the upper section in the same way as you did for the lower section.



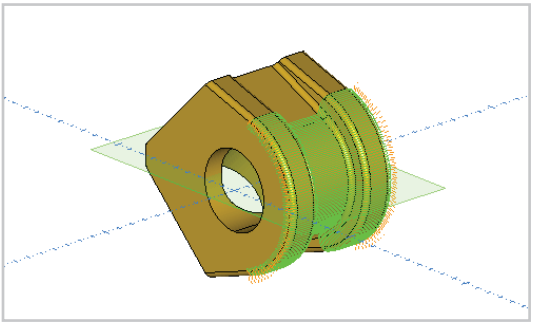
Preview the toolpath group to see the toolpaths you have defined.

In this position tutorial, we were able to complete most of the toolpaths for the part using 4 axis position toolpaths.

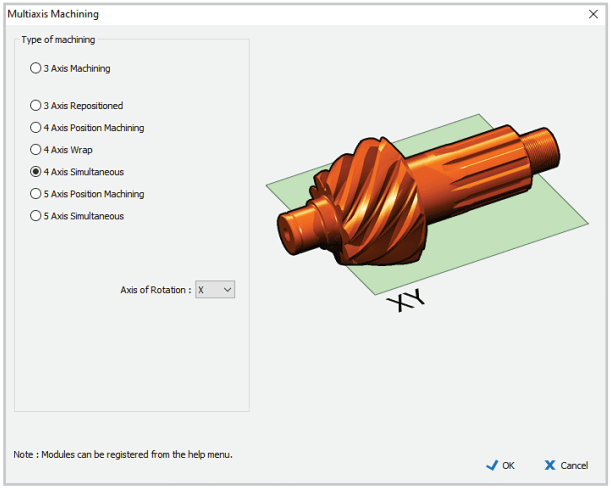
The remainder of the part requires the use of the 4 axis simultaneous method. This will be covered in the next tutorial.

OneCNC 4 Axis Simultaneous

This tutorial is intended for OneCNC Mill Expert or Mill Professional with 4 Axis or 5 Axis modules activated.



The remaining toolpath for this part is the 4 Axis simultaneous toolpath around the outside of the part. Open the file you used for the 4 axis position tutorial, and save a copy as 'Tutorial Multi 4 axis simultaneous.ONECNC'.



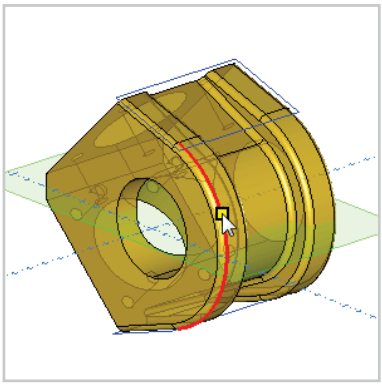
Click on the axis selection icon and select the 4 axis Simultaneous mode.

The part is aligned on the X axis, so select the X option for Axis of Rotation.

The 4 axis Simultaneous icon will be shown on the NC Manager toolbar.

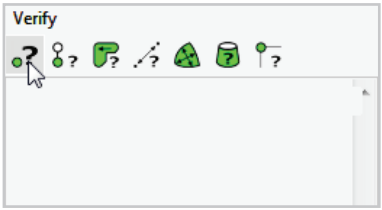
Select the default XY plane. The current plane does not affect 4 axis simultaneous toolpaths, but we need to know the angular range we want to apply it to. We can get the values from an extracted edge.

With the 'boundary planar' layer active, click on Model Tools icon and select Extract an edge.

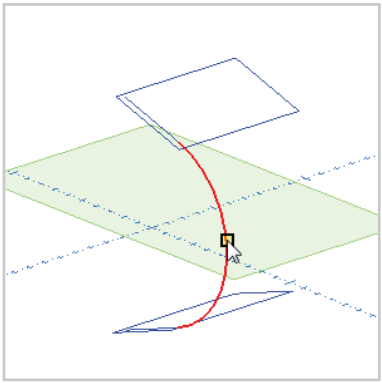


Click on an outer edge of the model as shown, and right click to end the command.

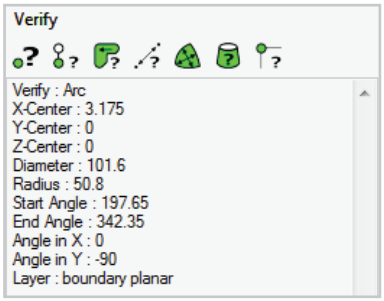
Turn off the model layer so you have a clear view of the extracted geometry.



Click on the Verify single entity icon in the Verify toolbar.



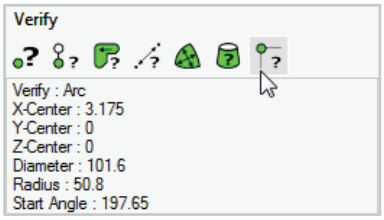
Click on the arc segment of the extracted edge.



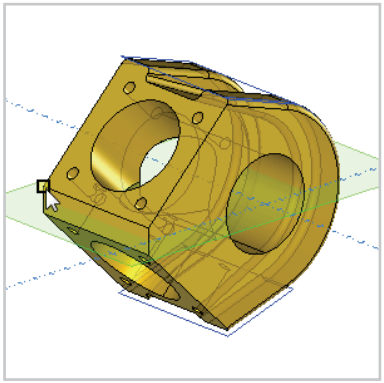
The Verify panel shows the Start Angle is 197.65 and the End Angle is 342.35.

It is important to remember the angle is measured anticlockwise from vertical.

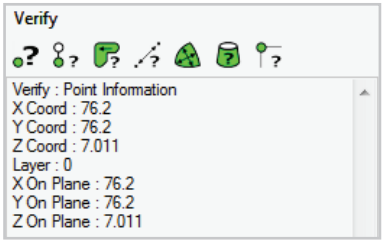
The external radius of the part is shown as 50.8.



To find the X axis range of the part, turn on the Model layer and click on the Verify endpoint icon in the Verify toolbar.



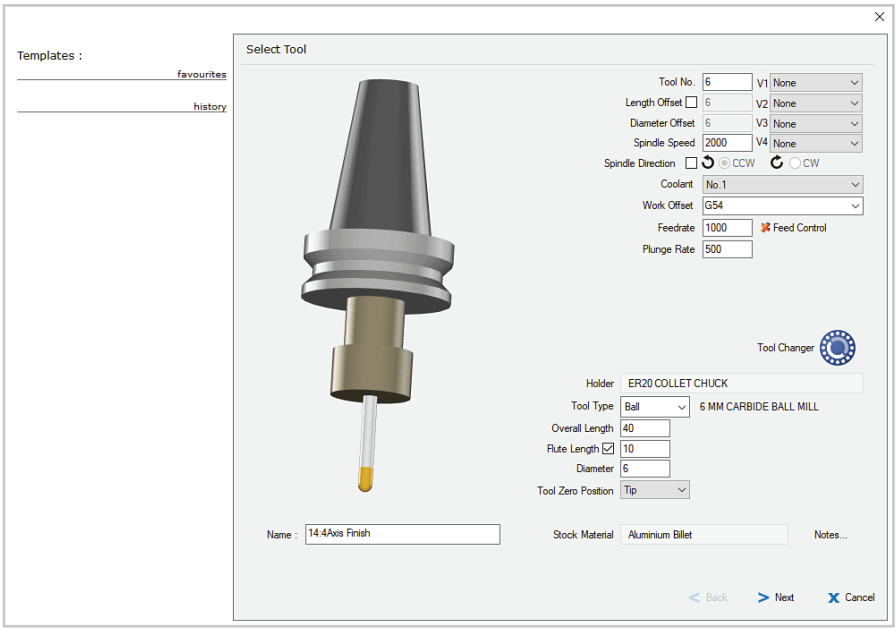
Click on a vertex at the X+ side of the part.



The Verify panel will show the side is at X76.2.

Now that we know the angular and X axis range for the toolpath, we can continue with the toolpath definition.

Click on the Simultaneous toolpaths icon and select 4 Axis Machining. In this mode other strategies are not applicable.

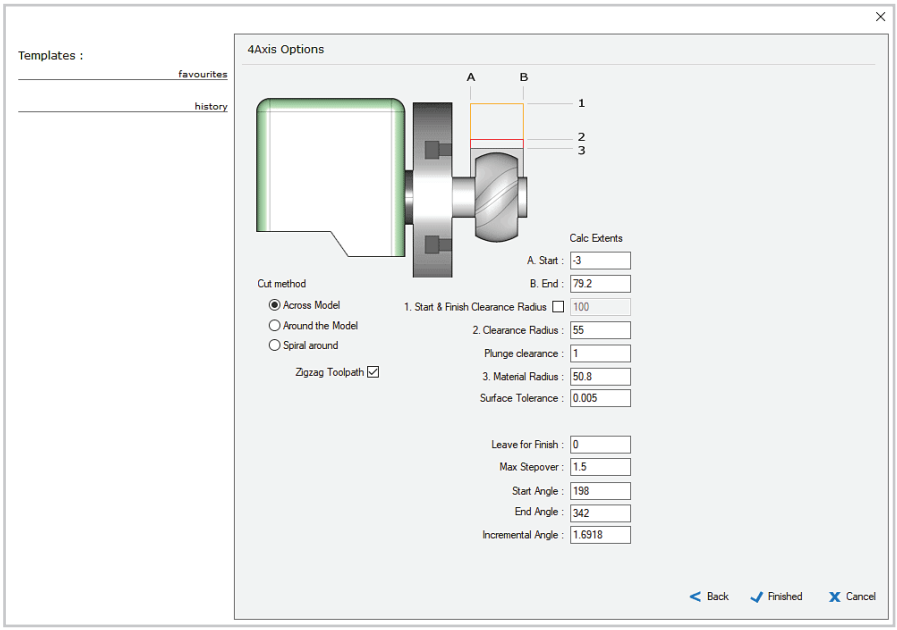


Select the 6mm Ball Mill used for the Planar Finishing.

Enter a Spindle speed of 2000.

Set the Feedrate to 1000 and the Plunge feed to 500.

Click next to continue.

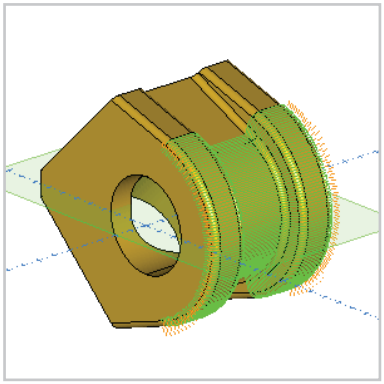


In the 4 axis Options dialog, select the Across Model cut method. Select the Zigzag Toolpath check box.

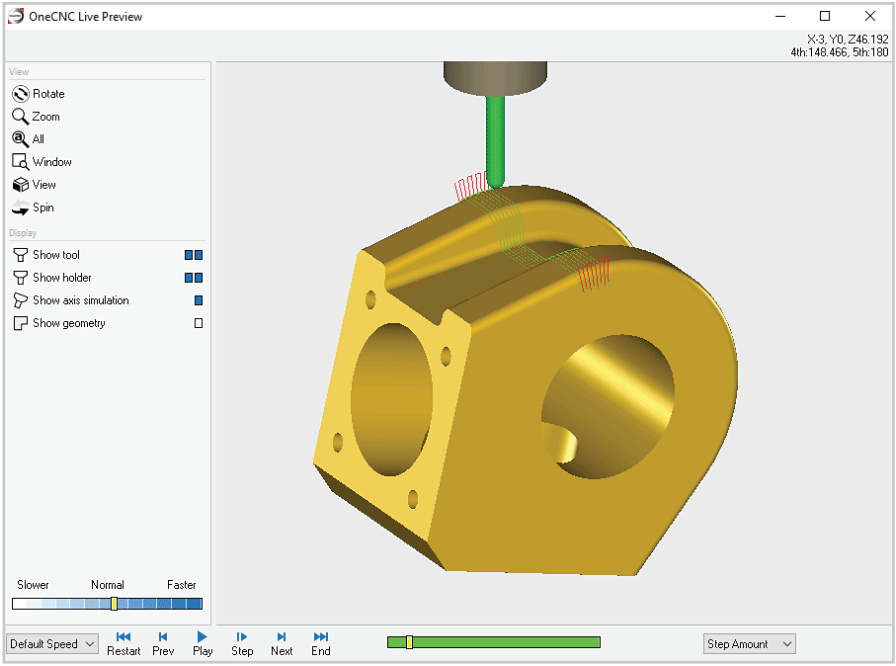
Set X Start to -3 and X End to 79.2, to allow the cutter to fully round the fillet. Enter a Clearance radius of 55, Material Radius of 50.8, and Surface Tolerance of 0.01. Set Leave for Finish at 0.

Enter a Maximum Stepover of 1.5, and the Incremental Angle will update automatically, as these two settings are related. The Start Angle can be rounded to 198, and the End Angle to 342.

Click Finished to create the toolpath.



The toolpath is applied to the angular range between the flats.



Preview the completed toolpath group to confirm the settings.
The programming of this part is now complete.

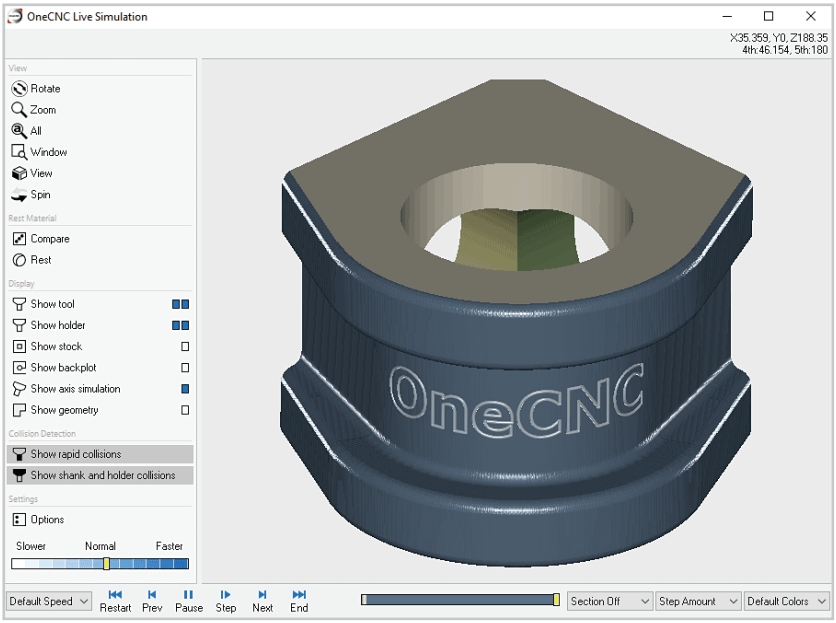
A tip for faster part simulation

For parts with a very fine surface finish, enter a coarser stepover than you actually require when you first define the toolpaths. Toolpath simulation or preview will run quicker.

When the toolpaths are complete, edit the stepover to the fine setting you actually require before the final check and output of the NC file.

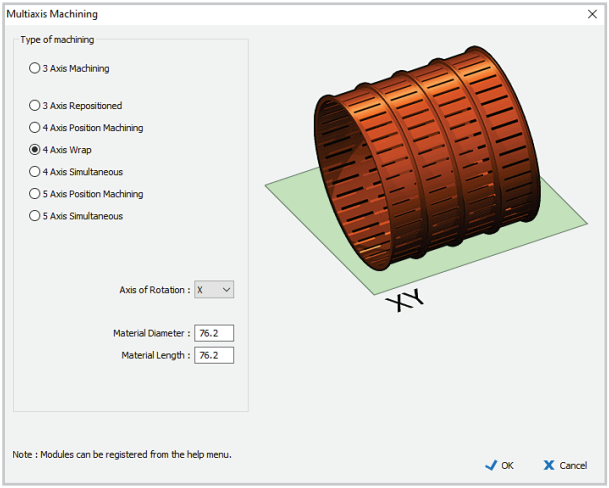
OneCNC 4 Axis Wrap

This tutorial is intended for OneCNC Mill Expert or Mill Professional with 4 Axis or 5 Axis modules activated.



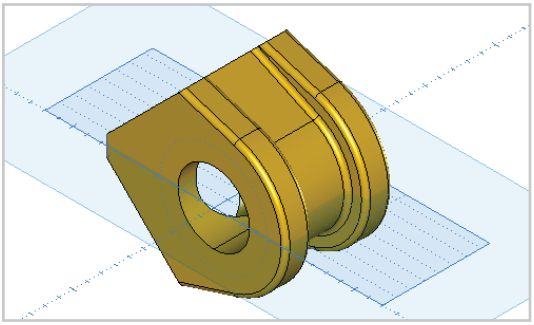
4th axis wrap machining applies toolpaths around a cylindrical surface on a part. In this tutorial we will use 4 axis wrap to engrave some text on the part used in the previous 4 axis tutorials.

Open the file, and save a copy as 'Tutorial Multi 4 axis wrap.ONECNC'.

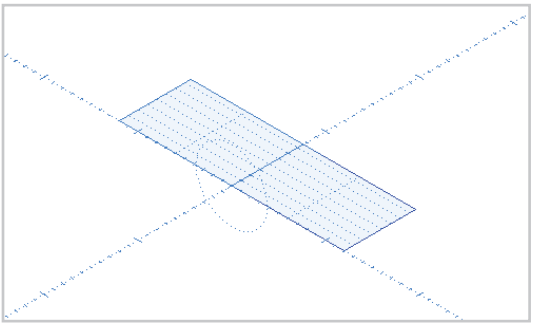


Click on the axis selection icon and select 4 axis Wrap mode. Enter 76.2 for Material Diameter and 76.2 for Material Length, and click OK.

The 4 axis Wrap icon will be shown on the NC Manager toolbar.

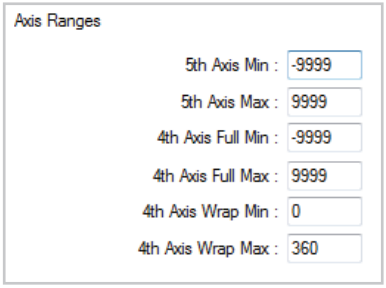


When you enter 4 axis wrap mode you will see a virtual grid on the XY plane which represents the unwrapped material cylinder. The X length of the grid is the Material Length. The Y length is the circumference of the Material Diameter. 2D geometry or 3D surfaces for wrap machining should be located within this grid.



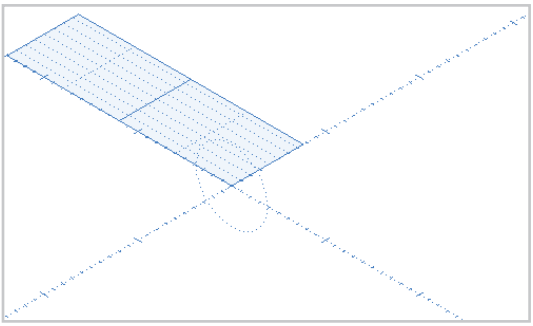
The grid position in Y is determined by the 4th Axis Wrap range set in the Multiaxis tab of the post settings. The default range of -180 to 180 will put the grid in a range from -Y to +Y.

We can accurately position our engraving by using geometry created by unwrapping a cylinder. The unwrap function creates geometry in the range of 0 to 360 so we will set the post to match that range.



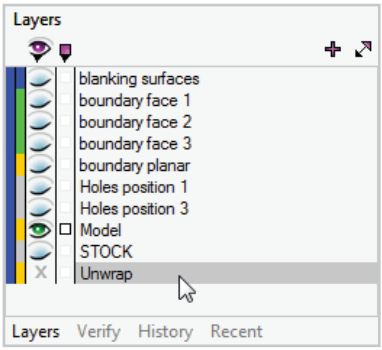
Open the NC Post Settings and select the Multiaxis tab.

Set 4th Axis Wrap Min to 0.
Set 4th Axis Wrap Max to 360.

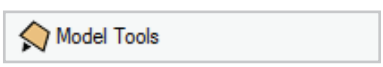


The grid will move to the unwrap cylinder range.

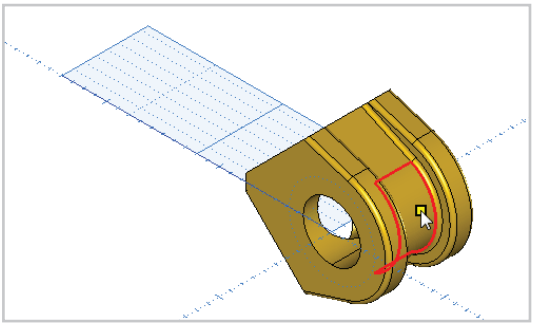
Create reference geometry



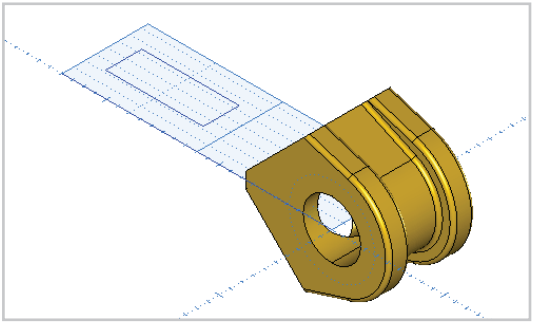
Create a layer named Unwrap.



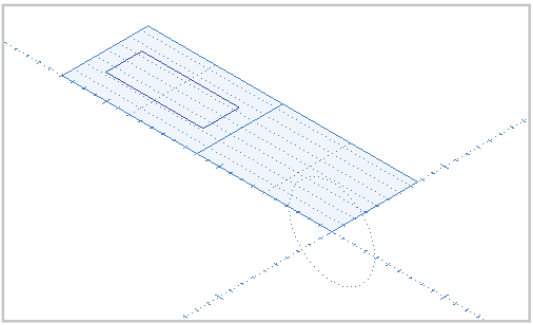
Click on the Model Tools icon and select Unwrap cylinder.



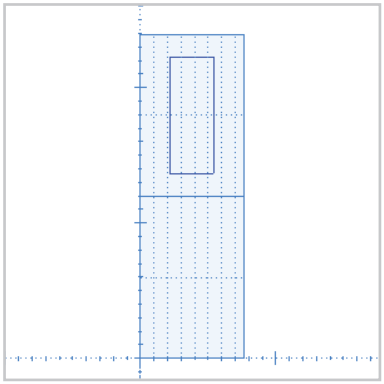
Click on the recessed cylindrical surface of the solid model.



A developed outline of the surface will be created.

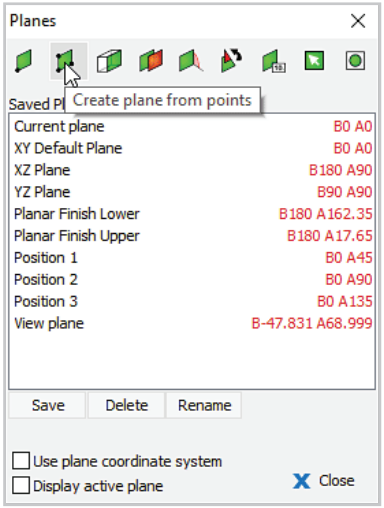


Turn off the model layer so you have a clear view of the geometry.

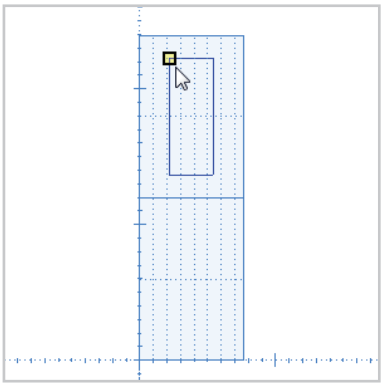


If you switch to CAD or Top View now, the view will be aligned with the X axis horizontal as shown.

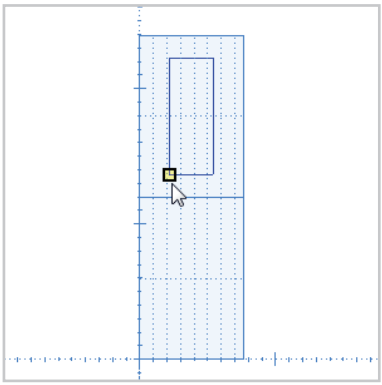
To make it easier to place the text we want to machine, we will create a plane to suit the orientation of the developed cylinder.



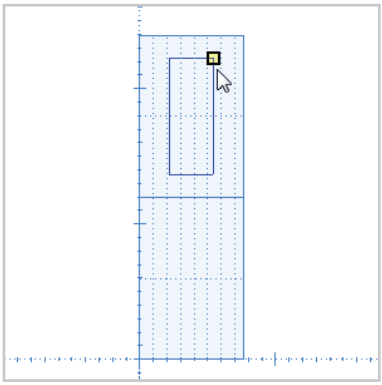
Click on the 'Create plane from points' icon in the Planes dialog.



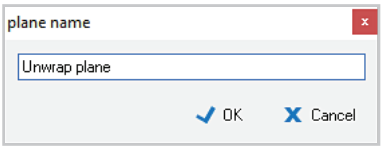
Pick the top left endpoint of the unwrap rectangle to define the origin of the plane.



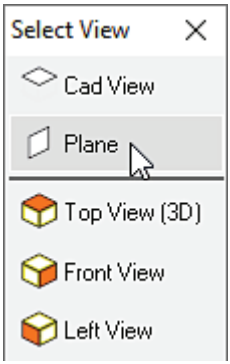
Pick the lower left endpoint of the rectangle to define the X direction of the plane.



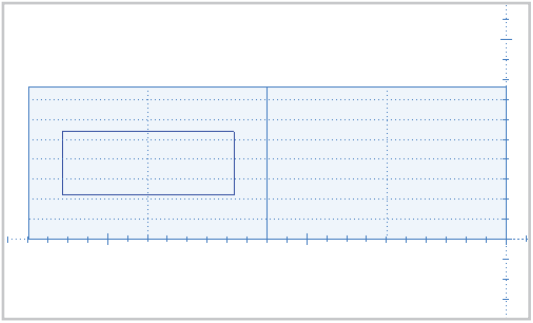
Pick the top right endpoint of the rectangle to define the Y direction of the plane.



Save the plane you have just defined as Unwrap plane.



Select the Plane option from the View Menu.



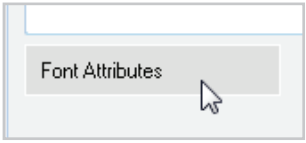
The view will rotate so the rectangle is horizontal.

Zoom in on the rectangle using the Zoom window command.

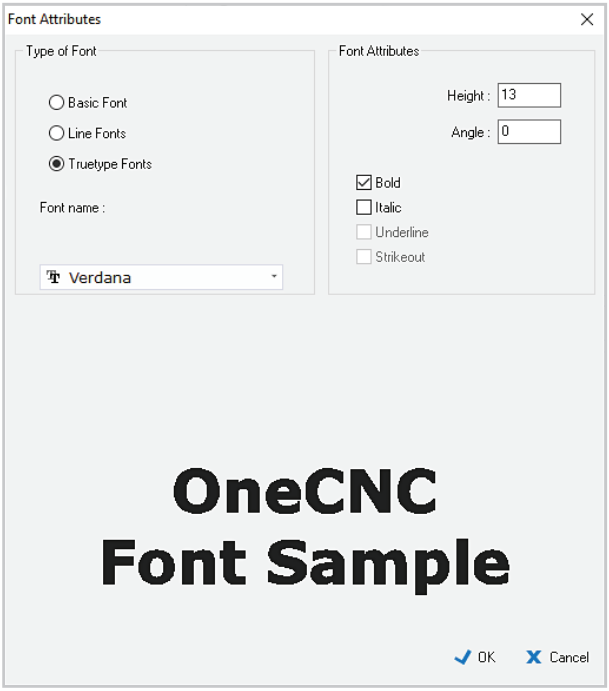
Create geometry to machine

We are now ready to place the text which we want engraved on the cylindrical face of the part. Create a new layer named Text and make it current before proceeding so the text outlines can be isolated for the Engrave All - Constant Z function.

Click on the Text icon in the toolbox and select the Text command.

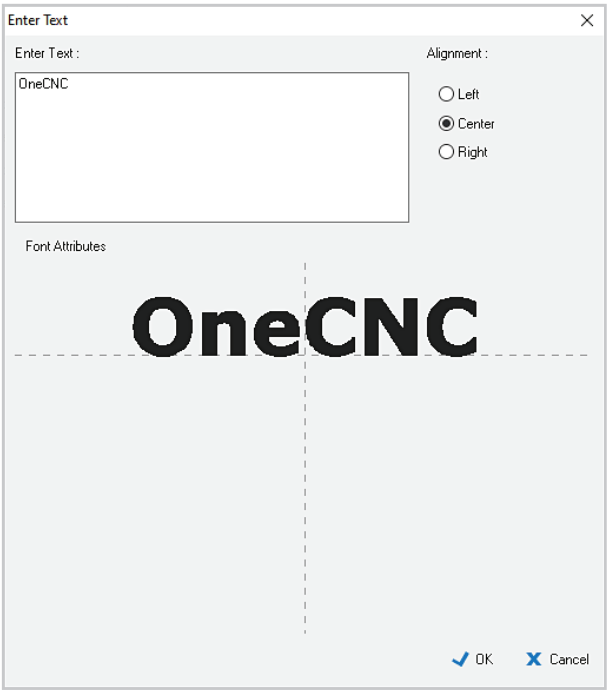


Click on the Font Attributes button in the Enter Text dialog.

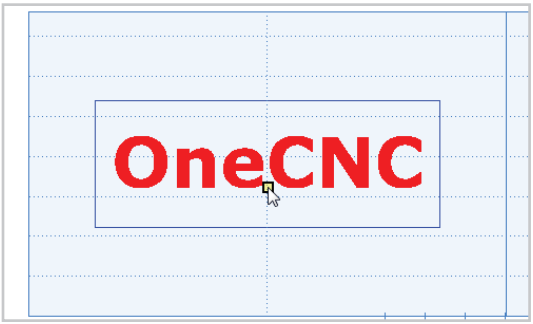


Select Truetype Fonts, pick Verdana from the Font dropdown list, and select the Bold check box. Set the Height to 13. As we are working on a plane aligned with the rectangle, we can leave Angle at 0.

Click OK to return to the Enter Text dialog.

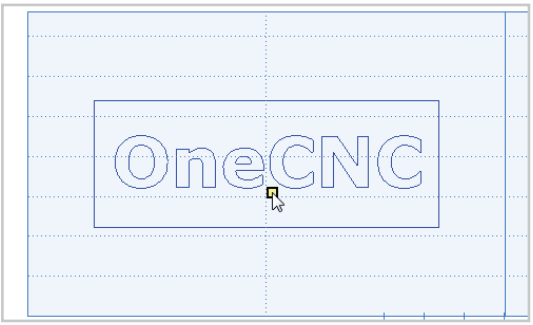


Select Center Alignment.
Type ‘OneCNC’ in the Enter Text dialog, and click OK.



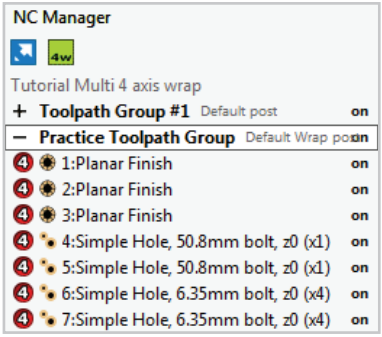
Place the text entity in the rectangle.

Click on the Text icon in the toolbox and select the Text to geometry command.

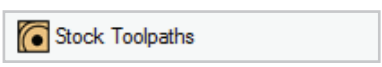


Click on the text entity to convert it to machinable geometry.
If you want to adjust the placement of the text outlines, use the selection box for the Text layer in the Layer Manager to select the geometry, and move it using the sketch position option.

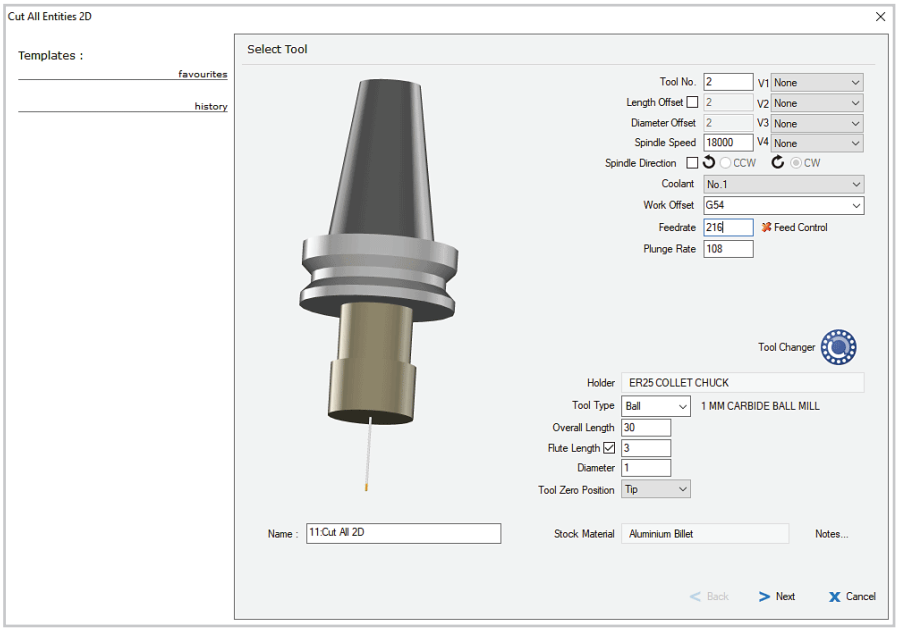
Turn off the Unwrap layer so only the vectorized text is visible.



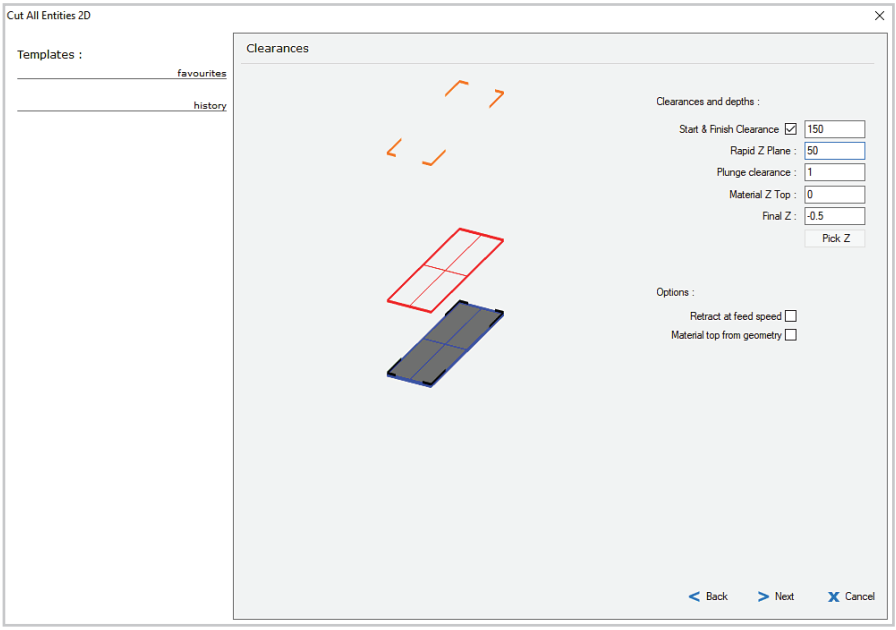
Before continuing, check that the Practice Toolpath Group is active, and that the 4 axis wrap icon is showing in the NC Manager toolbar.



Click on the Stock Toolpaths icon and select the Engrave all 2D toolpath.

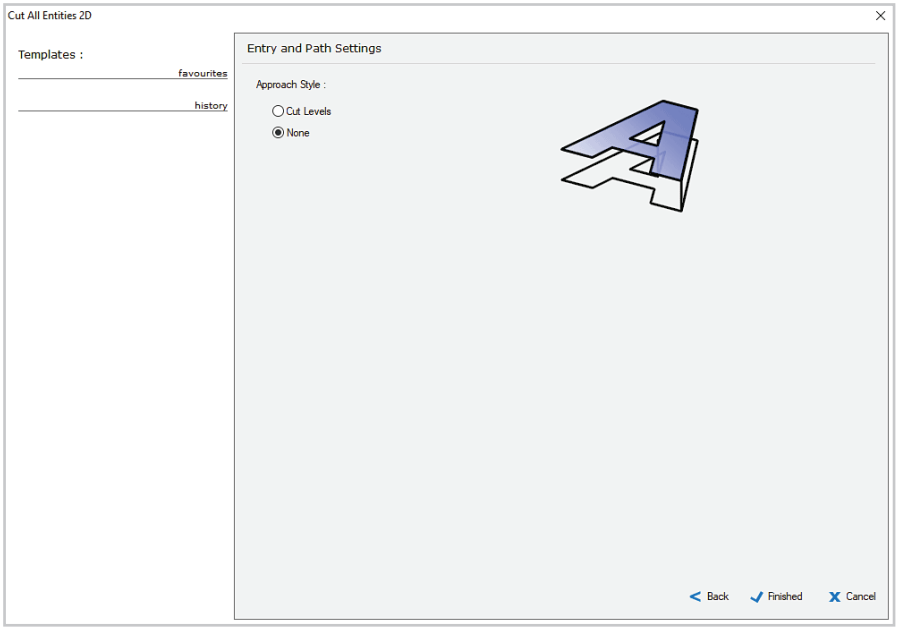


Click on the tool image in the Select Tool dialog, and select the 1mm Ball mill from the Tool List.



Set the Start and Finish Clearance to 150, the Rapid Z Plane at 50 and Plunge clearance to 1.

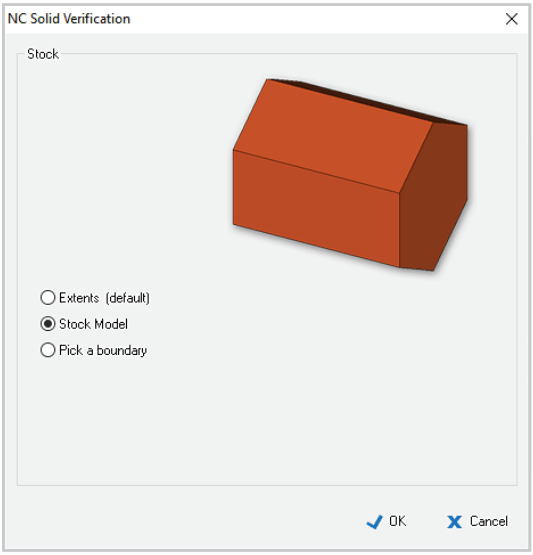
Enter a Final Z of -0.5.



The Path Creation settings allows you to mill geometry to the full depth in more than one pass.

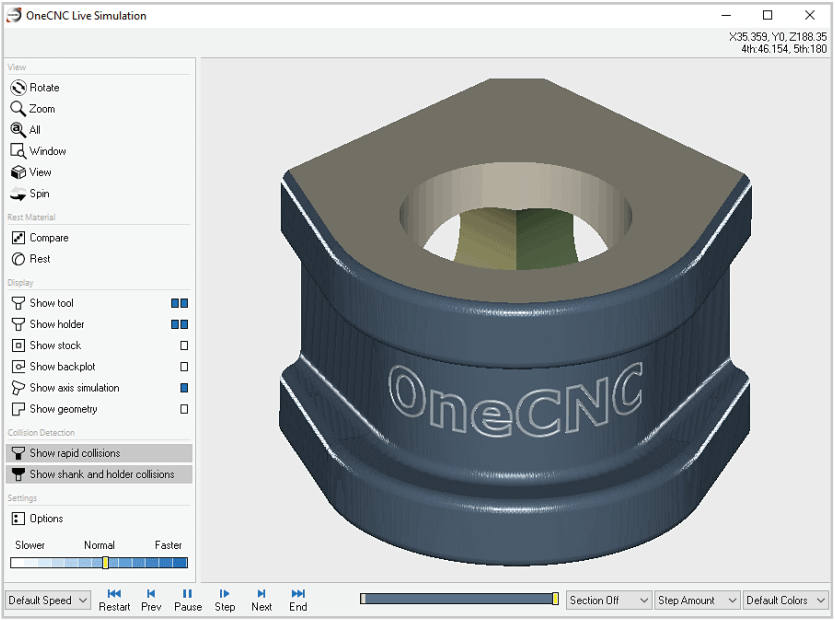
As we are only engraving in this example select None for the Toolpath style.

Click Finished and the toolpath will be created.



Right click on the toolpath group and select Simulate/Rest.

Select the Stock Model option.



After the main part machining, you will see the text geometry engraved around the cylindrical surface.

An engraving tip

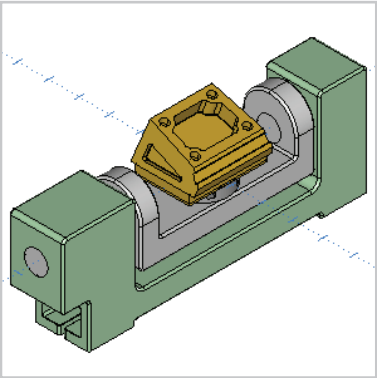
For small text, a single line shx font such as _romans will be clearer and take less time to cut.

When engraving larger text, a bold TrueType font will give a better result as the separation of the outlines is more distinct.

OneCNC 5 Axis Position Machining

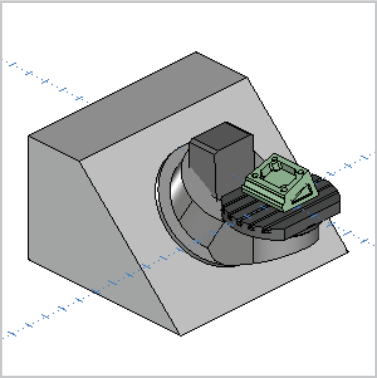
This tutorial is intended for OneCNC Mill Expert or Mill Professional with the 5 Axis position machining module activated.

There are two main types of machines OneCNC 5 axis position machining can be used to create NC programs for.



There are those with a trunnion operated by the fourth axis, which holds a rotary table controlled by the fifth axis.

This is the sample file '5 axis position trunnion.ONECNC'.

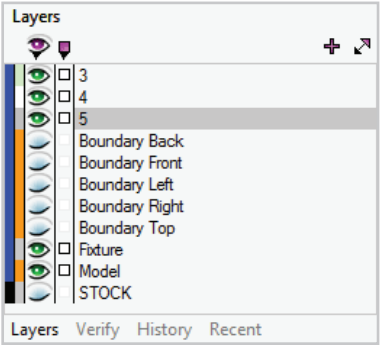


The other type has a fifth axis rotary table mounted on a swivel table rotated by the fourth axis, which is not necessarily horizontal.

This is the sample file '5 axis position dmg.ONECNC' which is for the same part on a machine with a swivel table.

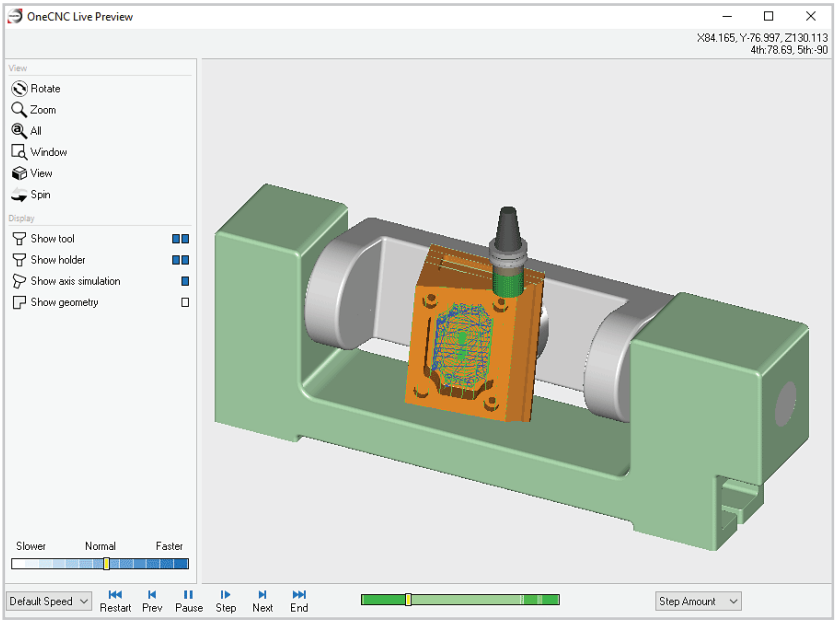
In this tutorial we will work with the trunnion style sample, but as the OneCNC 5 axis position programming method is the same, you can complete this tutorial working on the sample part most suited to your machine.

Select the sample file which is most like the machine you will be programming for, and save a copy as 'Tutorial 5 axis position. ONECNC'.



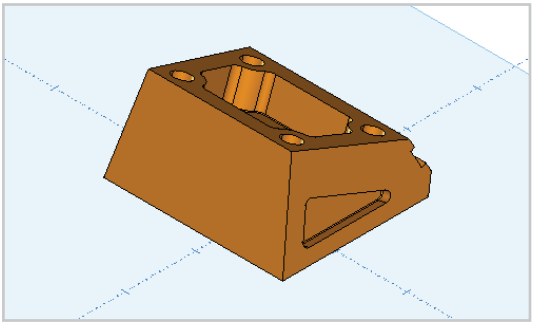
Open the layers tab to see how the part file is arranged. The layer named 3 holds the machine body. This is the base layer which does not move in the preview mode. The trunnion is on layer 4, and the rotary table is on layer 5.

For the swivel type of machine the layer named 'DMG_AXIS4' contains geometry which defines the swivel table axis.



Preview the example toolpath group and observe the full machine simulation in action. The rotary table and trunnion rotate to each position as required.

Define planes

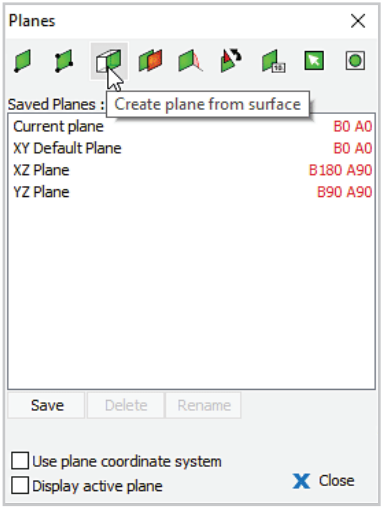


Turn off all layers except the one named Model, and select the Isometric view. We will now prepare to create 5 axis toolpaths on this part.

To create toolpaths in a 5 axis position, we define a plane for that position, and create toolpaths in 5 axis position mode. When the program is run the machine will rotate to the plane position.

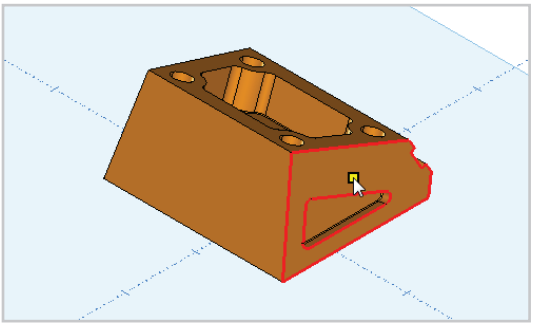
The plane can be in any position in OneCNC but it must be a position which can be reached by the machine.

While working on a part it is good practice to save all the planes you will need before you start, so you can quickly return to a plane.

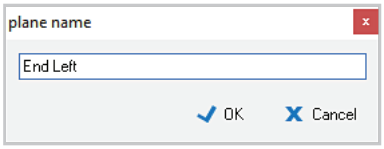


If you have suitable surfaces on your part, the Plane from surface command is convenient for creating the required planes.

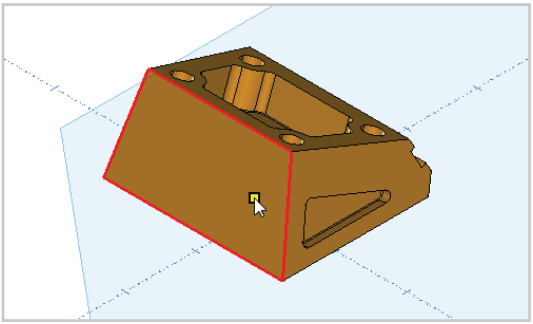
From the Planes dialog, select the Create plane from surface icon.



Click on the closest end face of the part to create a plane on it.

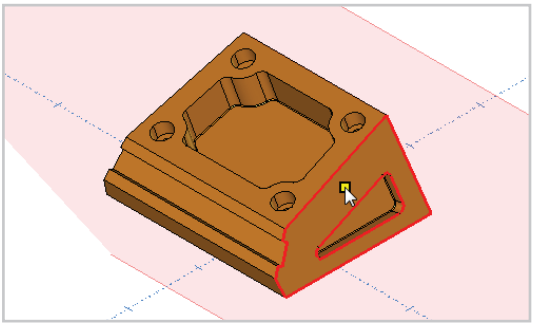


Open the Planes dialog again, and save the plane as End Left.

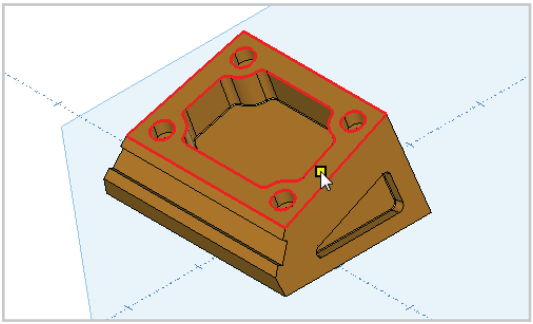


Create a plane on the trapezoidal face, and save it as Back.

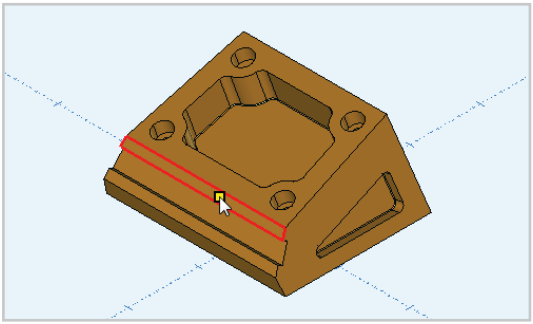
When using the Plane from surface command, the plane created may sometimes be red on top, indicating the negative side. If this happens, use the Flip plane command to reverse the orientation of the plane.



Rotate the view of the part so you are looking at it from the opposite direction. Define a plane on the right side of the part and name it End Right.

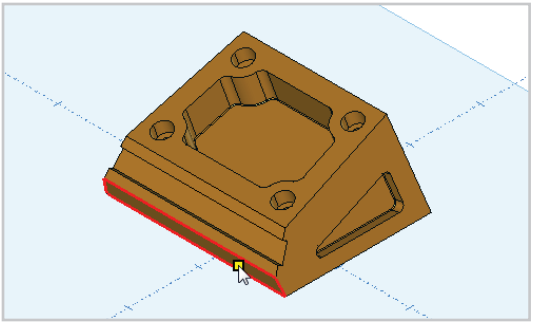


Save a plane for the pocketed and drilled face, naming it Top.

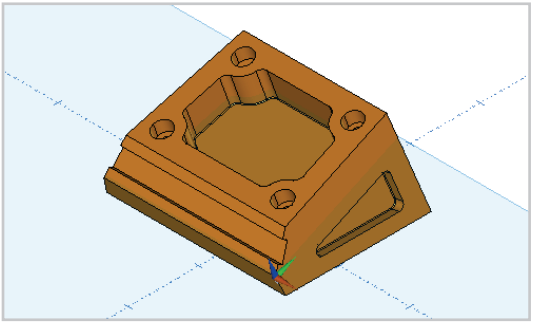


Create a plane to face and slot the upper portion of the front of the part. Save this plane as Front upper.

The last plane required is that required for facing the bevel between the upper front and the base of the part.



For a trunnion type machine this can be on the lower surface. Name this plane Front lower.



For a swivel table type machine, to mill the lower front face you will need a plane perpendicular to the face.

Create a plane on the surface and start the Rotate plane command. Rotate the plane with an X tilt of -90. Use the Move plane command to position the origin on a vertex of the upper edge. Name the plane Front lower.

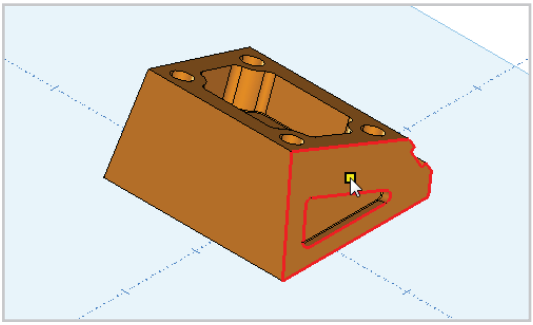
Prepare boundaries

For some toolpaths you will need geometry boundaries. Construction geometry may be used, but there will be times when you have to create the geometry.

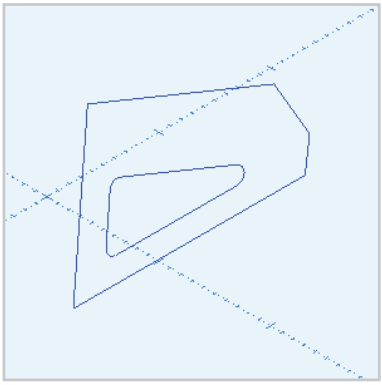
It is good practice to keep the geometry for each plane on a separate layer with a corresponding name. Create 6 new layers and name them 'Boundary Right', 'Boundary Left', 'Boundary Top', 'Boundary Front' and 'Boundary Back'.

In the plane dialog, select the End Left plane.
Select the Boundary Left layer as the active layer.

Click on the Model Tools icon and select Extract surface edges.

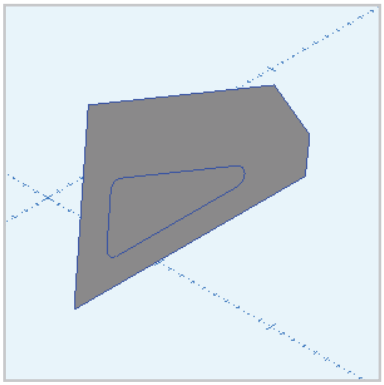


Click on the left end of the part to extract the edges as wireframe geometry.



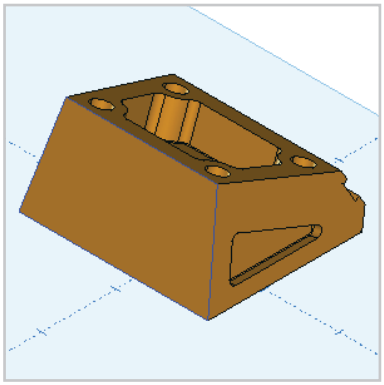
Turn off the Model layer so you can see the extracted geometry.

Delete the lines extracted at the end of the slot on the upper front face, and draw a new line across that corner to create a simple boundary for facing.



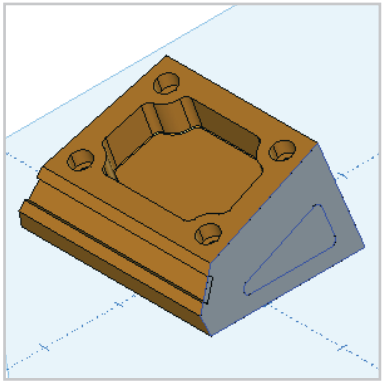
Use the surface from curves command to create a blanking surface to cover the pocket while facing.

The outer boundary and blanking surface will be used for a facing operation, and the internal boundary will be used for the pocketing operation.



Turn on the Model layer again. Select the Back plane and the Boundary Back layer.

Create a boundary for a facing operation. There is no need for a blanking surface.

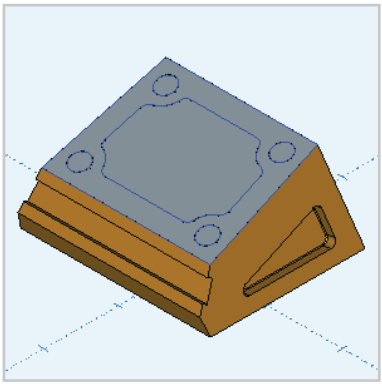


Select the End Right plane and the Boundary Right layer.

Create boundaries and a blanking surface for the right end of the part to match those for the left end.

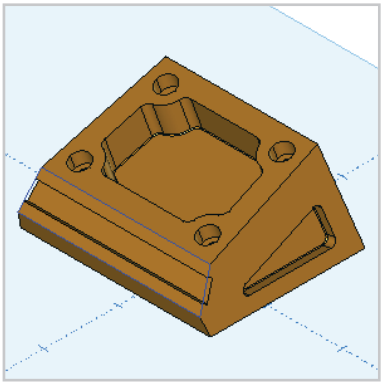
A note regarding extracted geometry

Due to the nature of computer modelling, extracted geometry may be segmented. This is especially true of arcs extracted from imported models. Extracted geometry will be in the correct position, but be prepared to trim or redraw as necessary.



Select the Top plane and the Boundary Top layer.

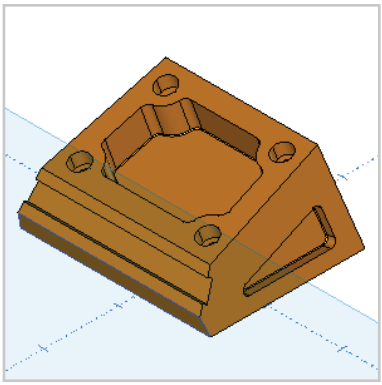
Create boundaries and a blanking surface for the top of the part.



Select the Front Upper plane and the Boundary Front Upper layer.

Use the Endpoint option of the Universal Line command to connect the corners of the face to form a boundary.

There is no need for a blanking surface.

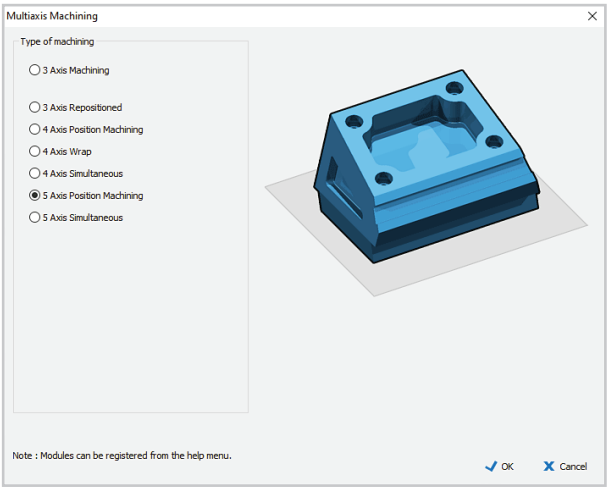


For trunnion machines, create a boundary on the lower surface for a facing operation. There is no need for a blanking surface.

For swivel type machines, you only need to extract the top edge of the front lower face, to use for a profile operation.

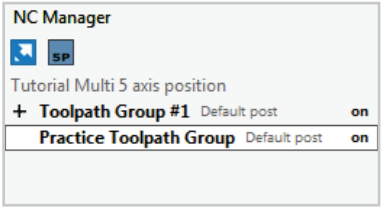
5 Axis Position Toolpathing

Once suitable planes and geometry have been defined, we can begin defining the actual toolpaths.

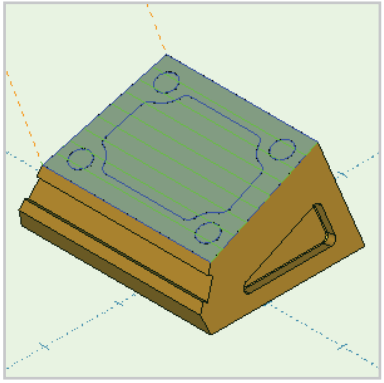


Before continuing, click on the axis mode selection icon in the NC Manager toolbar, and select 5 Axis Position Machining.

Create a new toolpath group in the NC Manager and name it Practice Toolpath Group.



With the 'Top' plane active, use the outer boundary and blanking surface created earlier to define a Planar Finish toolpath.



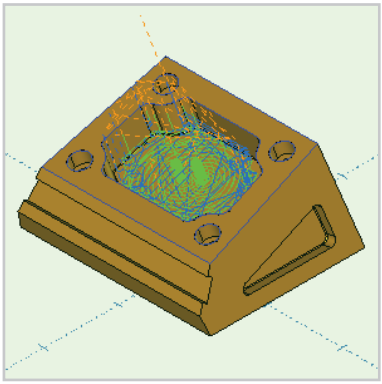
Use the 50 MM CARBIDE FACE MILL with a Rapid Plane clearance of 50.

The planar cut will have an angle of 0 and constant stepover of 20.

Use the Pick a boundary option, with Normal tool offset.

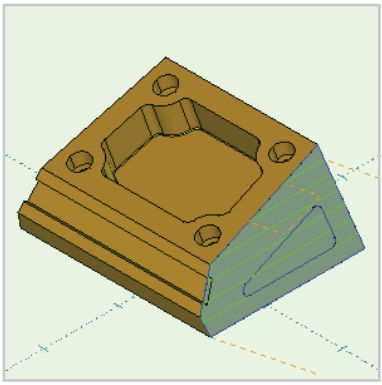
Clear the Automatic Z Offset check box, and set Z Top of Job to 1 and Z Bottom Of Job to 0. This will limit the toolpath to the level of the plane.

Select the blanking surface and click on the Blank icon to hide it.

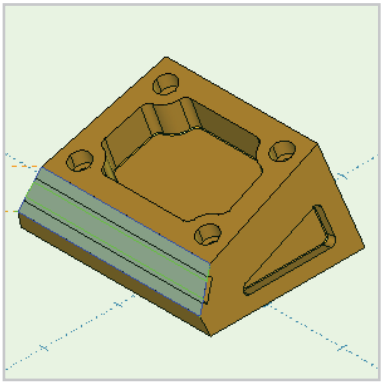


Create a Z Level roughing operation to machine the pocket. Use the 25 MM CARBIDE END MILL.

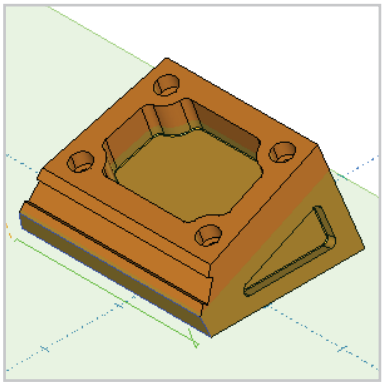
Select the Highspeed Closed machining style and Ramp Helix entry.
Enter a 25% stepover with 25mm rough depths.
Use the Picked boundary option, with no tool placement.
Set Z Top Of Job to 0 to force the toolpath below the plane.
Set Z Bottom Of Job to -45.
Select the internal boundary for the toolpath.



Change to the End Right plane and boundary layer, and create a Planar Finish facing toolpath in the same way as the top was done.

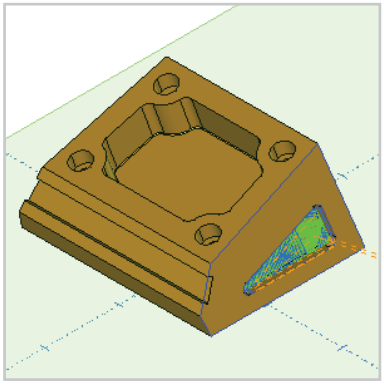


Continue creating facing toolpaths around the part, selecting planes and geometry as required for each face.

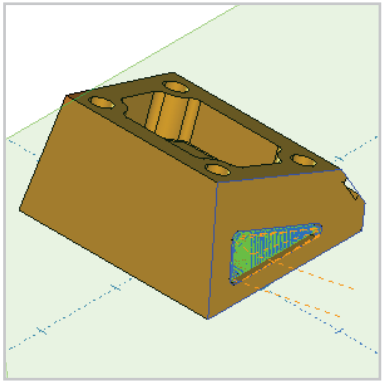


Note that for a swivel table type machine, you will have to mill the lower front face as a stock toolpath profile chain operation.

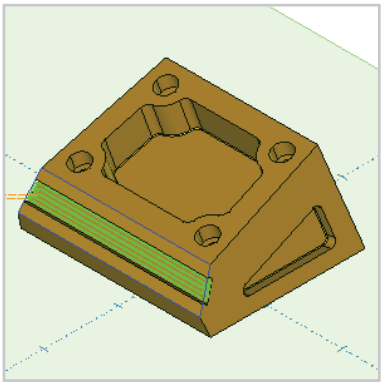
Activate the plane perpendicular to the face, and use an end mill to profile the edge at the top of the flat area.



With the 'End Right' plane active and the facing surface blanked, create a Z Level roughing operation to machine the pocket on the end of the part. Use the 10 MM 1R HSS BULLNOSE tool. The rest of the settings are similar to the pocketing done on the top of the part.



Change planes and use the same settings to create the pocket on the other end of the part.

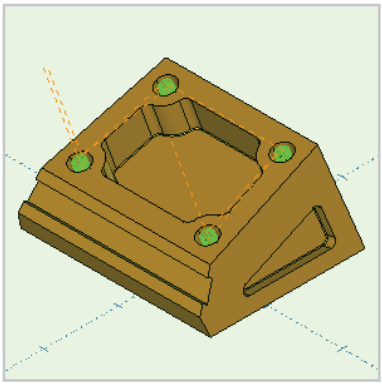


To cut the front slot, define a Z Level Rough toolpath with the same boundary used for the facing outline.

Use the 6mm carbide end mill and the Traditional machining style.

Select Find Flat Areas in the Custom Levels dialog to force the toolpath to the level at the base of the slot.

Clear the check box for Automatic Z Offset, and set a Z Top Of Job value of 0. The Z Bottom Of Job value can be any number that exceeds the depth of the slot, which is 6.375.



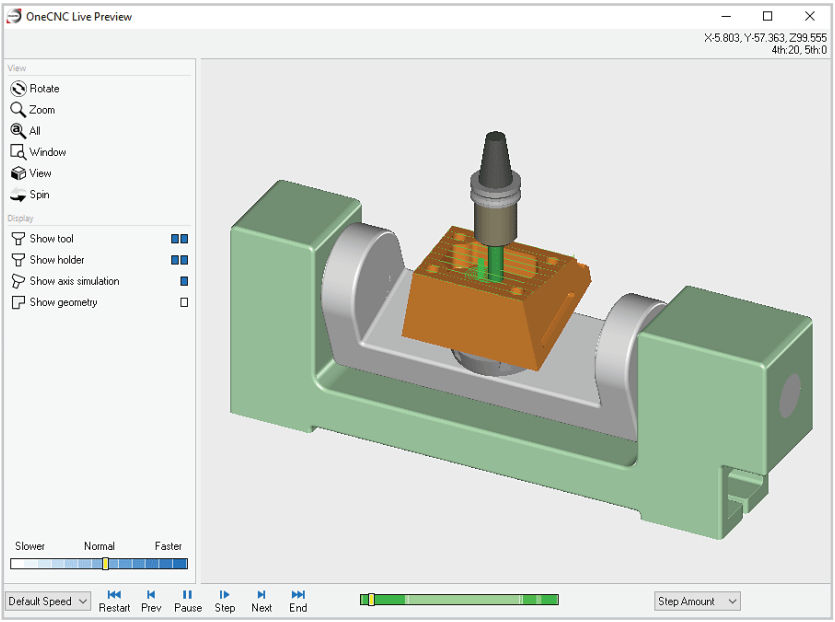
The remaining machining is the drilling and counterboring of the bolt holes.

Use Hole Feature Recognition to create the bolt hole machining. You can specify all the operations in one Hole Feature definition.

Center Drill the positions, then drill the stem holes 50mm deep with a 12.7mm diameter drill.

Define a helical Counter Bore operation with a 10mm End Mill. The counterbore is 19.05mm deep with a diameter of 25.4.

Turn the machine layers back on, and preview the toolpath group you have defined.

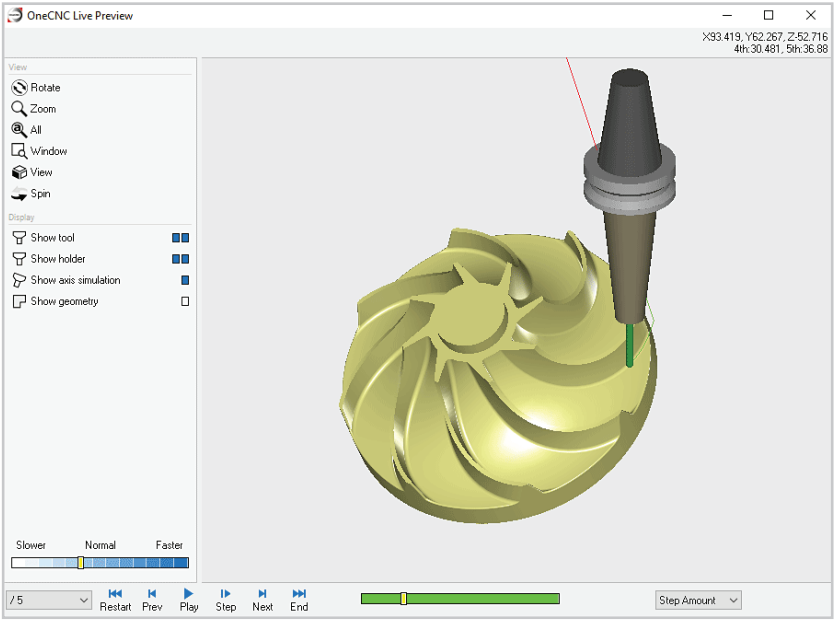


We have now covered the general methods of 5 axis position toolpathing, and you can now go on to programming your own parts.

Many 5 axis parts look complex at first, but by defining planes and boundaries and progressing logically through the machining strategy they will usually become a straightforward programming exercise.

OneCNC 5 Axis Simultaneous

This tutorial is intended for OneCNC Mill Expert with the 5 Axis simultaneous module activated.

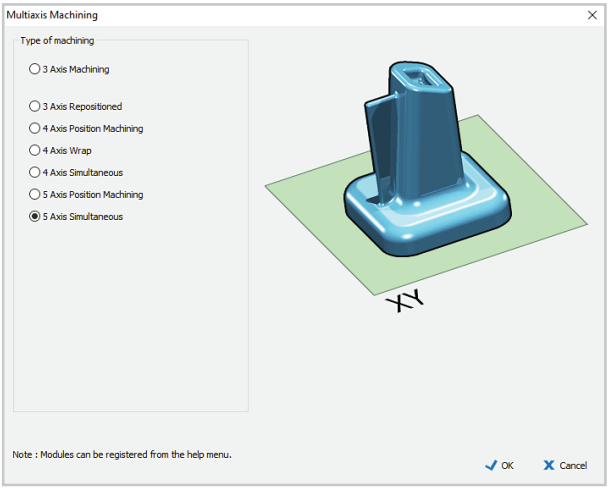


The 5 axis simultaneous mode creates model toolpaths which can move up to 5 machine axes at the same time. When the mode is active, the simultaneous toolpaths menu will show 5 axis simultaneous toolpaths, which are all finish type strategies.

5 axis simultaneous toolpaths are similar to their 3 axis counterparts, but the angle of the spindle relative to the Z axis can change when necessary to avoid collision of the tool holder with a surface. This enables the cutter to machine areas of the part which a 3 axis toolpath could not.

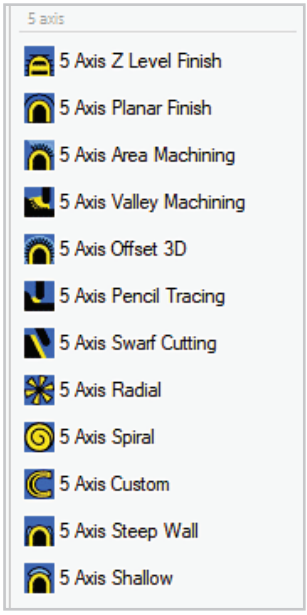
Position planes are not used in the 5 axis simultaneous mode, and the current plane should be set to the default XY plane before defining 5 axis simultaneous toolpaths.

Even though the underlying computation is very complex, defining 5 axis simultaneous toolpaths is similar to defining 3 axis model toolpaths.



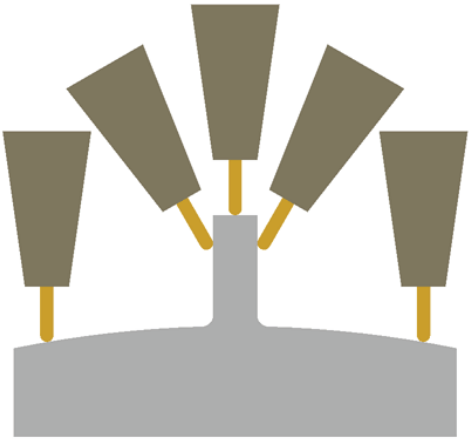
The 5 axis simultaneous mode is entered by selecting it in the Multiaxis dialog.

Clicking on the Simultaneous Toolpaths icon in the toolbox will open the Simultaneous Toolpaths menu.

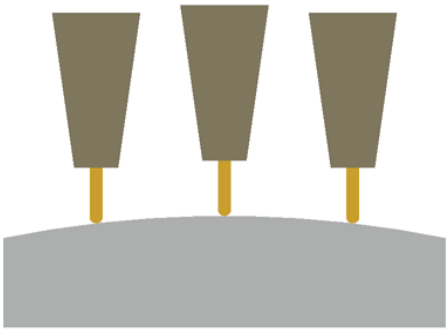


The available strategies are:

5 axis Z Level Finish
5 axis Planar Finish
5 Axis Area Machining
5 Axis Valley Machining
5 Axis Offset 3D
5 axis Pencil Tracing
5 Axis Swarf Cutting
5 axis Radial
5 axis Spiral
5 axis Custom
5 axis Steep Wall
5 axis Shallow



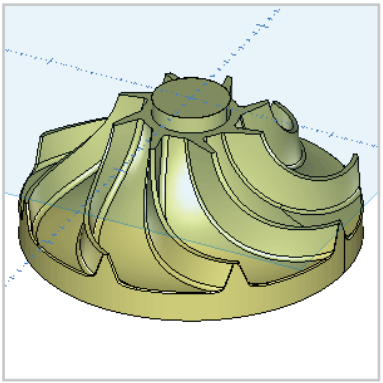
The minimum clearance between the holder and the part is maintained automatically. The Clearances dialog in each toolpath wizard gives you the option to add greater clearance if you want more for a particular toolpath.



If the surface being machined does not present any obstruction to the tool and holder, they will remain vertical.

Simulate a 5 axis simultaneous toolpath

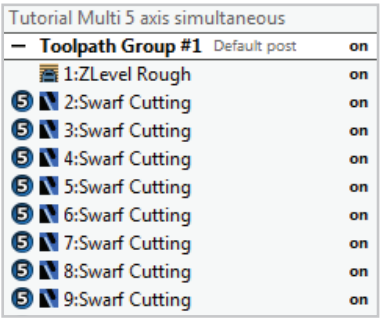
We will start by simulating a sample file, to see how a 5 axis simultaneous toolpath behaves.



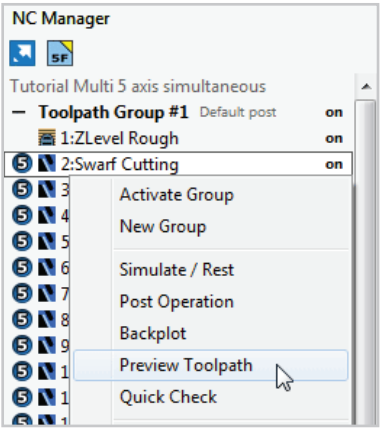
Open the sample file 'mill expert 5 axis impeller.ONECNC', and save a copy of the file to practice in.

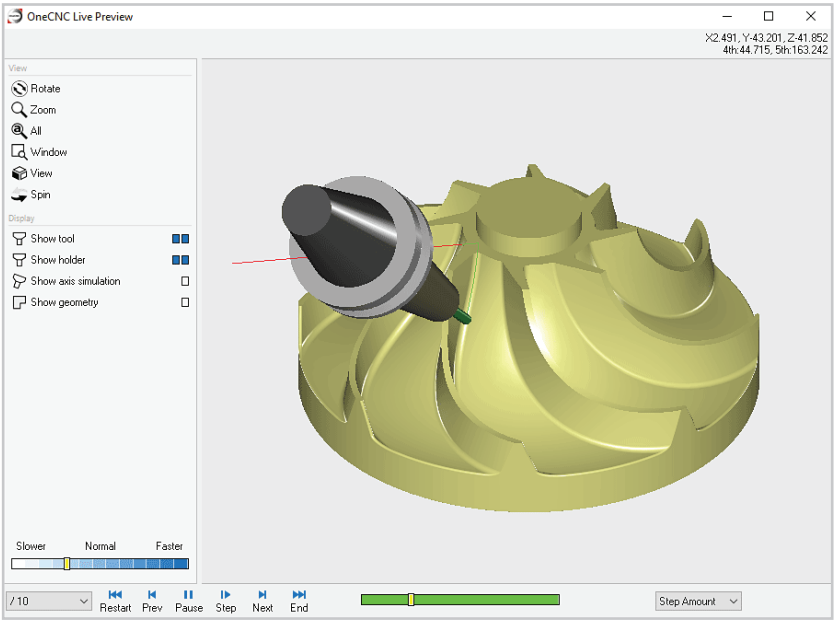
There is an example toolpath group in the NC manager with pre-defined operations to machine the part.

The first operation is a 3 axis Z Level roughing toolpath to rough out the shape.

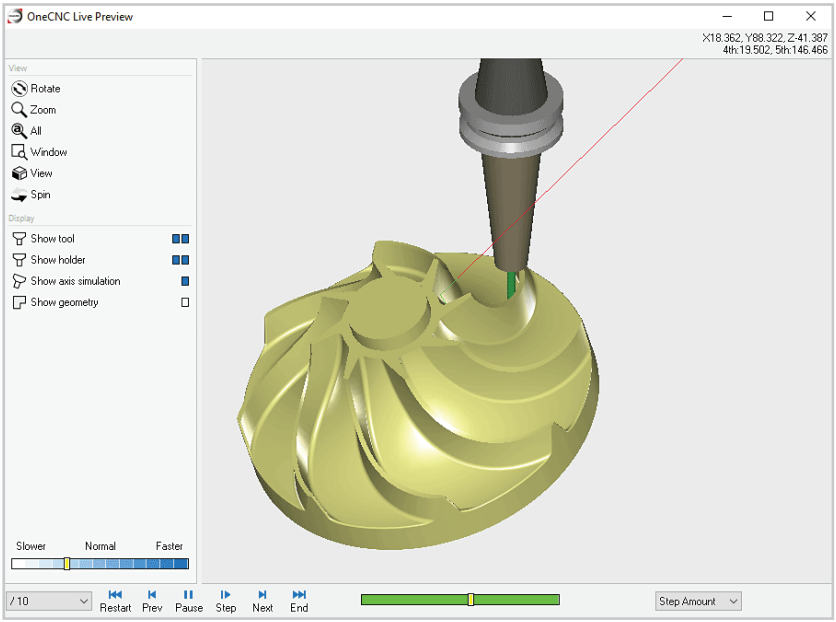


The rest of the example group uses various 5 axis simultaneous strategies. To see one of them in action, right click on the first Swarf Cutting operation and select Preview toolpath from the context menu.





As the toolpath machines down the shape at each Z level, you will see the holder tilting to keep clear of the part. The default display shows the part fixed in place, with the holder moving on all five axes. With this type of motion it is easier to visualize the cutting action.



If the 'Show axis simulation' icon is selected, the fourth and fifth axes will be displayed moving the part, indicating the motion you would actually see on a machine with rotary table axis movement.

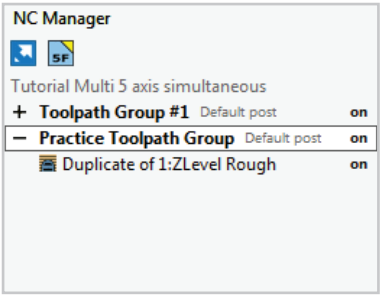
Close the preview window when you are ready to continue

Prepare for 5 axis simultaneous toolpaths

We will practice 5 axis simultaneous toolpathing by re-creating some of the existing operations in a new group. To save time we can copy the Z level roughing operation.

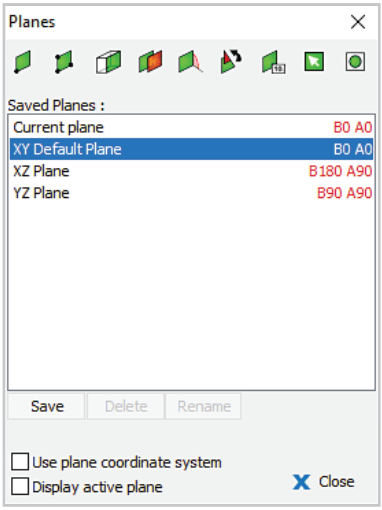
Create a new toolpath group and rename it 'Practice Toolpath Group'. Duplicate the Z Level roughing operation in the example toolpath group, and drag the copy into the practice group.

Your NC Manager should now look like this.



The multi-axis mode icon should show the 5 axis Simultaneous mode is current.

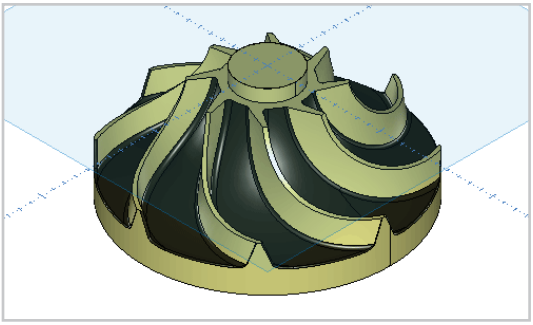
If it is not, click on the icon to open the selection dialog. Select the 5 axis Simultaneous mode, and click OK.



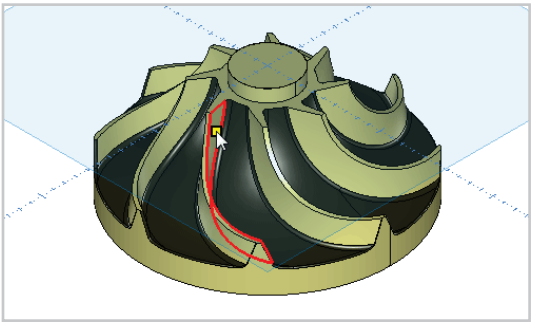
Before continuing, check that the XY default plane is current.

Click on the Simultaneous Toolpaths icon and select the 5-axis Swarf Cutting toolpath.

When you start a swarf cut operation, the first step is to select the surfaces to be machined. To be machined with the side of the tool, a surface must be flat, or only curved in one direction. During the surface selection stage, any surfaces with complex curvature will be colored black.

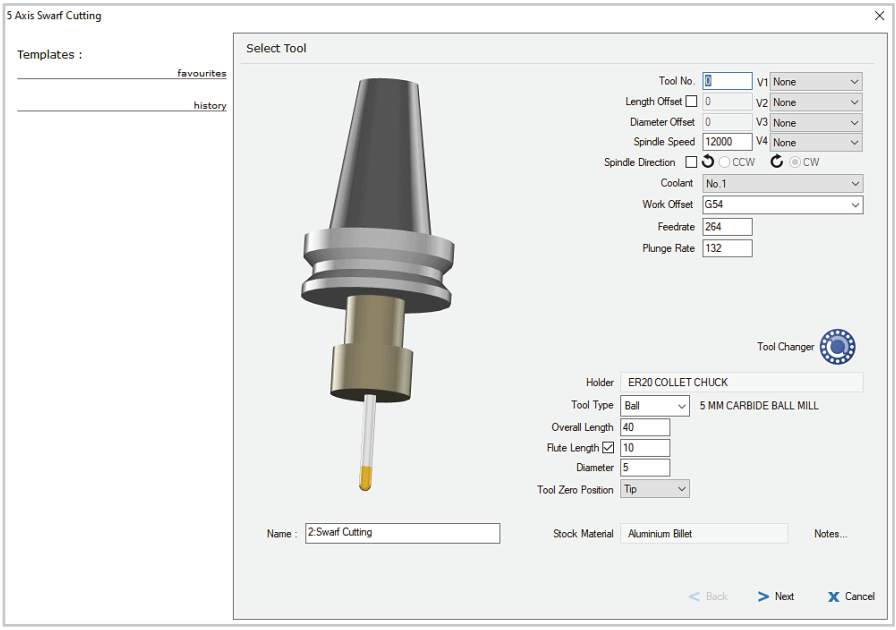


In this part the sides of the blades can be swarf cut, but the fillets and surfaces between the blades have complex curvature which makes them unsuitable for Swarf Cutting.



Select the right side of the nearest blade, and right click to end the selection stage.

Note:
For this operation we are only machining one surface. You can select more than one surface for a swarf cutting operation but the surfaces must be a single contiguous set.

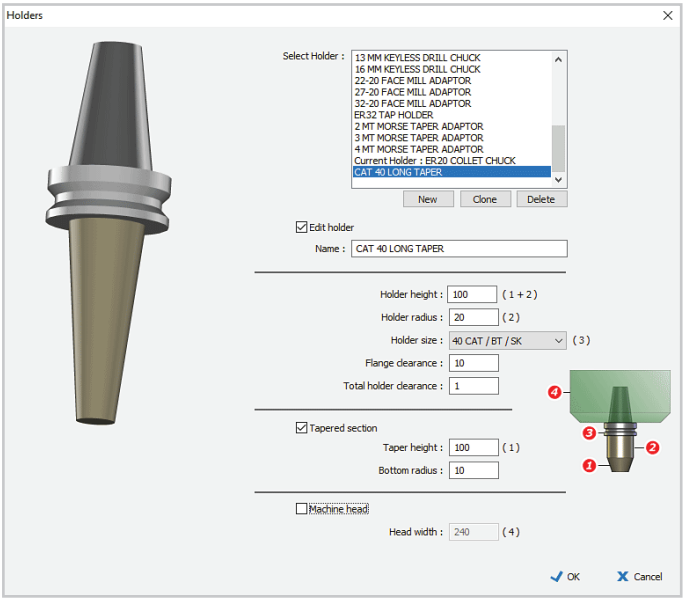


When the Select Tool dialog appears, select a 5mm carbide ball mill from the Tool Library.

The holder associated with this tool is not suitable for a 5 axis simultaneous toolpath.

The defined holder is used by OneCNC to determine the position and angle of the holder during the toolpath. The holder must be defined accurately, so that clearance between the holder and part is calculated correctly.

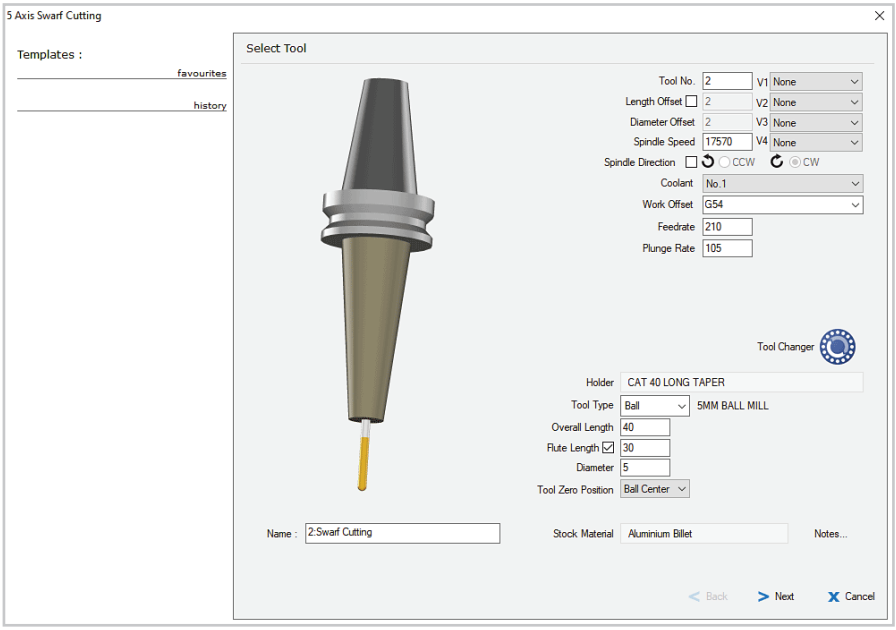
Click on the Holder name to open the Holder dialog.



Click on the ‘new’ button below the holder list, to add a new holder to the holder list. Select the new holder listing, and select the Edit holder check box to define the holder, naming it CAT 40 LONG TAPER.

Enter a holder height of 100 and radius of 20. Select the 40 CAT/BT/SK holder size. Enter 0 for Flange clearance and 5 for Total holder clearance. It is absolutely essential to have the holder clearance defined correctly.

Select the check box for Tapered section and enter a Taper height of 100 with Bottom radius of 10. Leave the Machine Head check box empty, and click OK to return to the Select Tool dialog.



Change the tool Flute Length to 30.

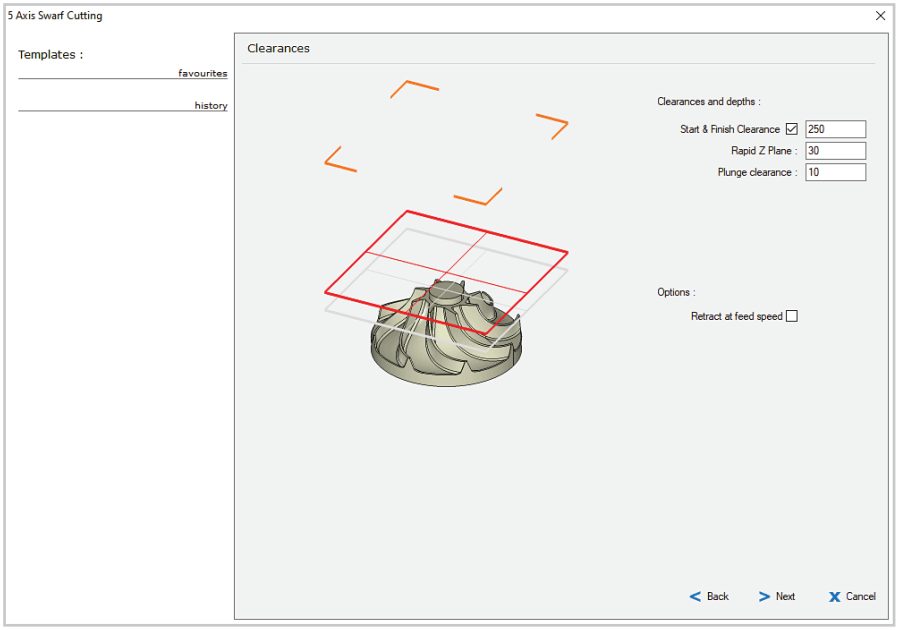
Enter Tool number 2, and set Coolant to No1 and Work Offset to G54.

Enter 210 for the feedrate and the plunge rate will update to 105.

Set the spindle speed at 17570.

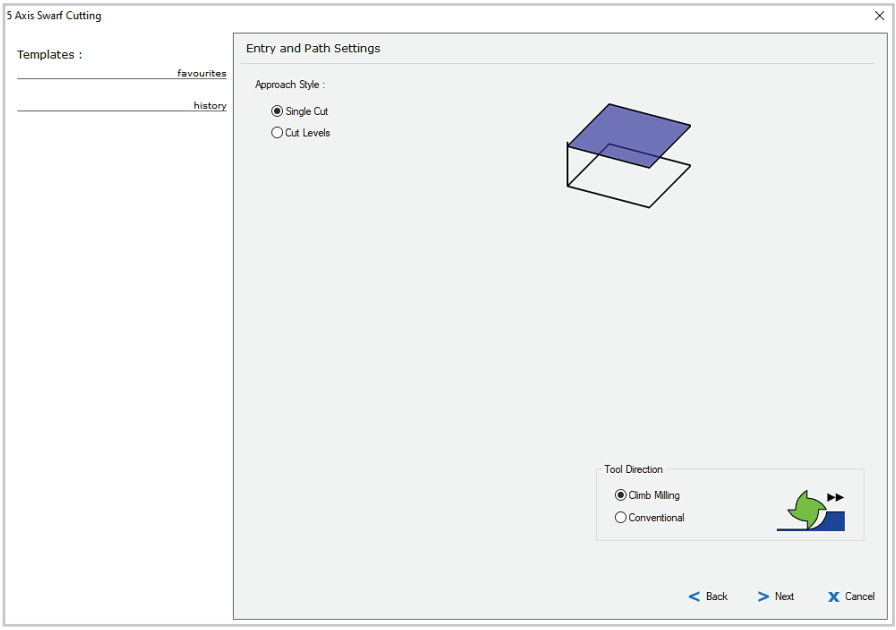
Select Ball Center for Tool Zero Position.

Click Next to continue.



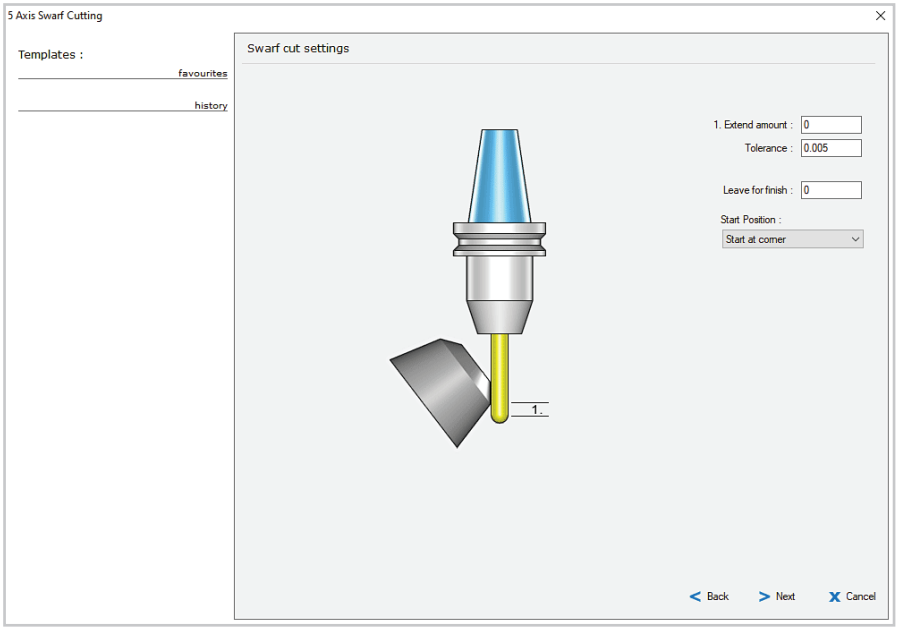
Set the Start and Finish clearance to 250 and Rapid Z plane at 30, with a Plunge clearance of 10.

Click Next to continue.



We do not need multiple passes to machine this surface, so we can select Single Cut in the Entry and Path Settings dialog.

Select Climb Milling and click Next to continue.



Enter 0 for Extend Amount as we do not want the tool to cut past the blade surface.

Enter 0.005 for Tolerance.

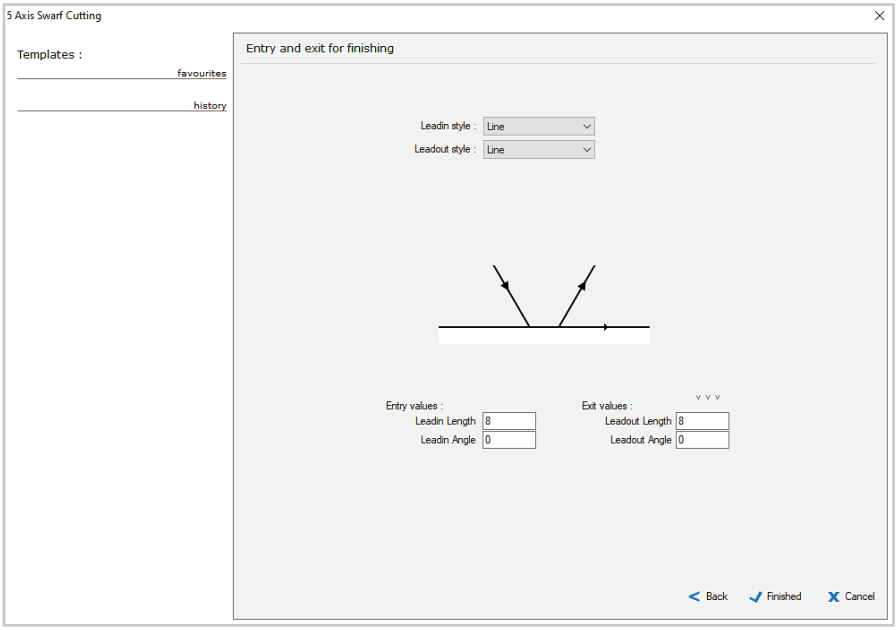
Enter 0 for Leave for finish.

Select Start at Corner for the Start Position.

Click Next to continue.

Note:

If you are Swarf Cutting multiple surfaces that form a loop you can select Start at mid to have the toolpath start midway along a surface.

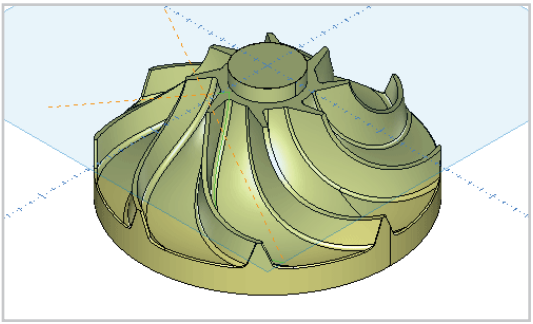


Select Line for Leadin style.

Set Leadin Length to 8 and Leadin Angle to 0.

Click the copy icon to copy the Leadin values to the Leadout settings.

Click Finished, and the toolpath will be created.

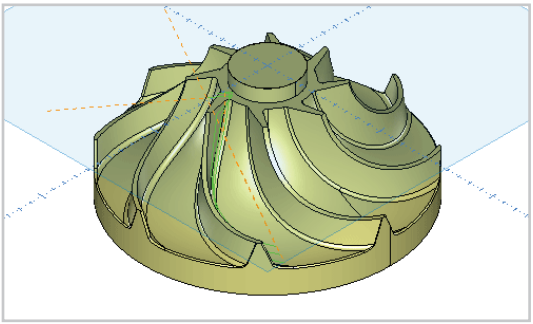


Backplot the toolpath to see the result. The angles of the plunge and retract lines in the backplot show that this is a 5 axis simultaneous operation.

5 axis Swarf Cutting - multiple passes

We have just defined a toolpath which machines the surface in a single pass. To see how it can be machined in multiple passes, create a duplicate of the toolpath and edit it.

Select Cut Levels and enter Depth of cut 10 in the Entry and Path Settings dialog.

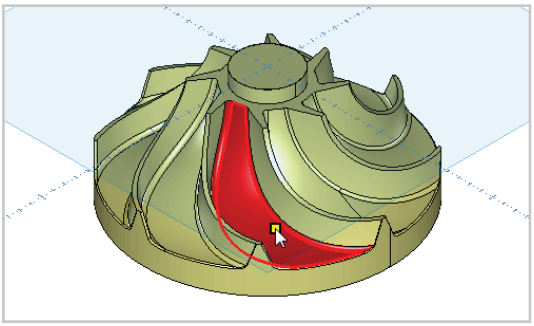


Backplot the edited toolpath and you will see the multiple passes at increasing increments across the surface.

5 axis simultaneous Area Machining

The next toolpath we will define is an Area Machining toolpath to finish the surface between the blades.

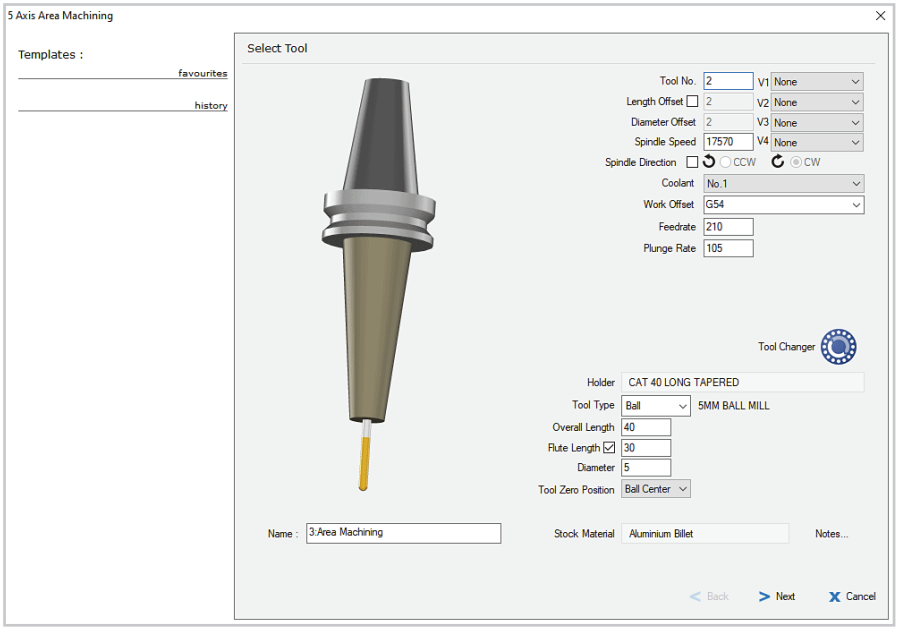
Click on the Simultaneous Toolpaths icon and select the 5-axis Area Machining strategy.



Select the surfaces between the blades, including the fillets.

Right click to end the selection stage.

The Select Tool dialog will open.



Select the 5mm ballnose cutter in the CAT 40 Long Taper holder which was used for the Swarf Cutting toolpath.

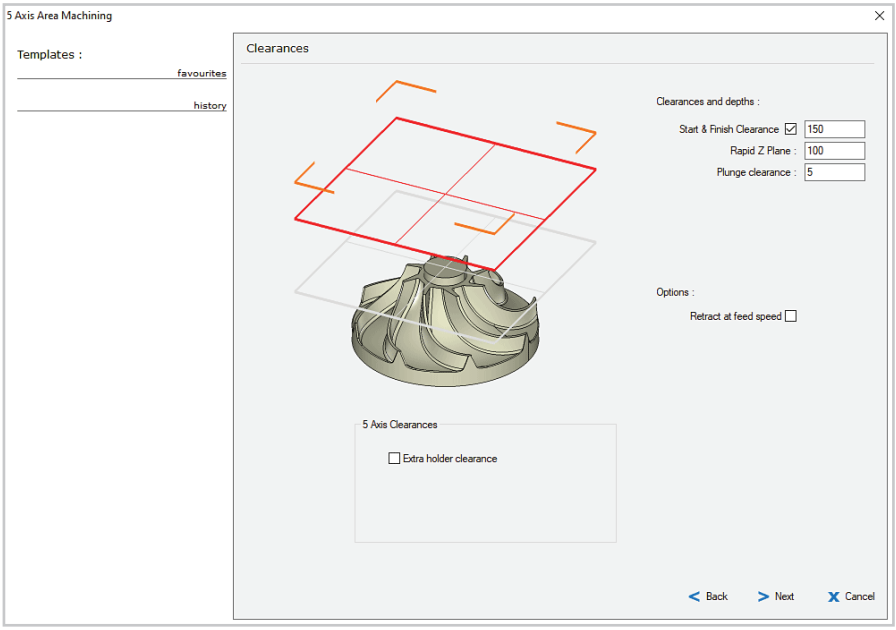
Enter Tool number 2, and set Coolant to No1 and Work Offset to G54.

Enter 210 for the feedrate and the plunge rate will update to 105.

Set the spindle speed at 17570.

Select Ball Center for Tool Zero Position.

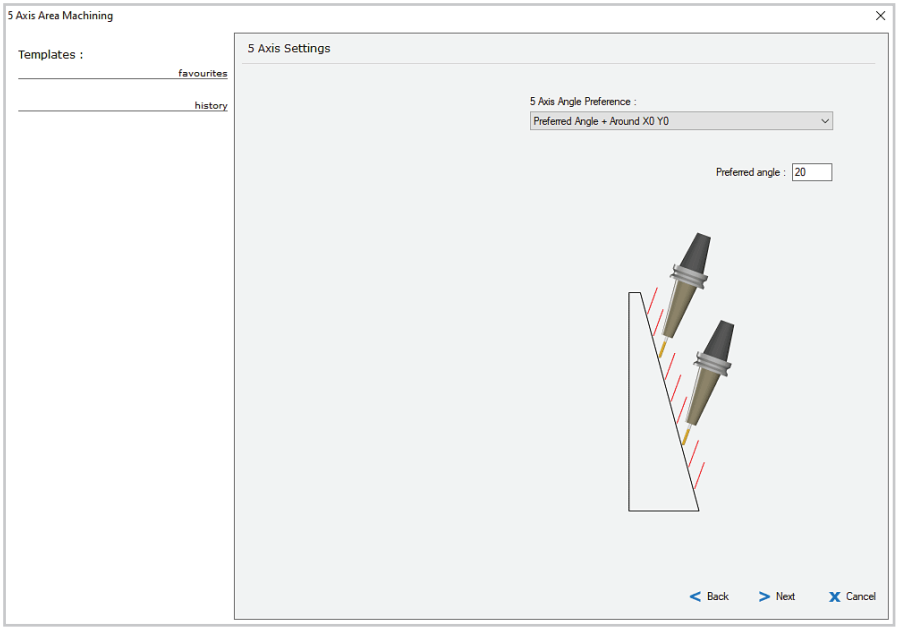
Click Next to continue.



Set the Start and Finish clearance to 150.
Set the Rapid Z plane at 100, with a Plunge clearance of 5.

Click Next to continue.

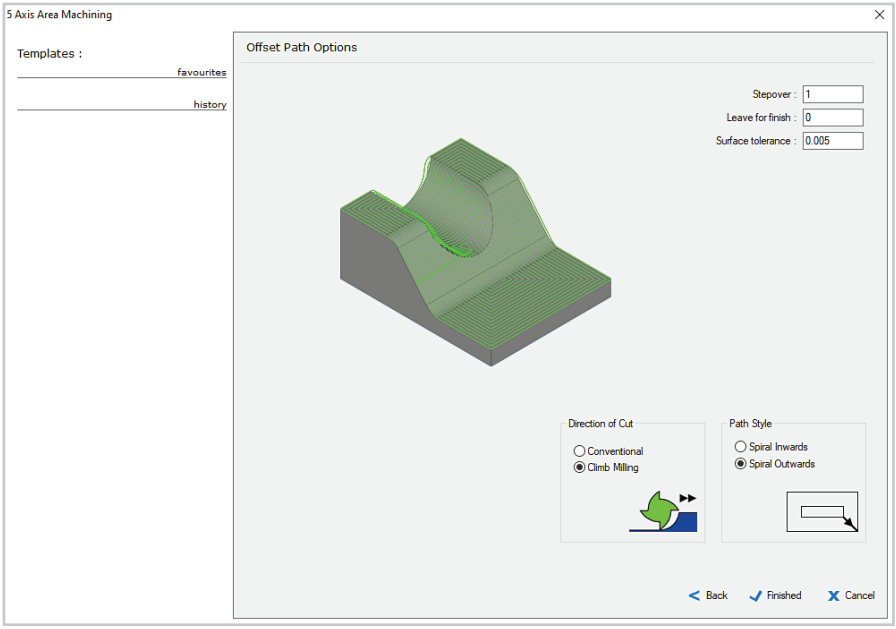
The next dialog controls how the tool is angled relative to the part as it travels along the toolpath. The Dynamic option tilts the tool automatically, but you can set a preferred angle that will be used if possible.



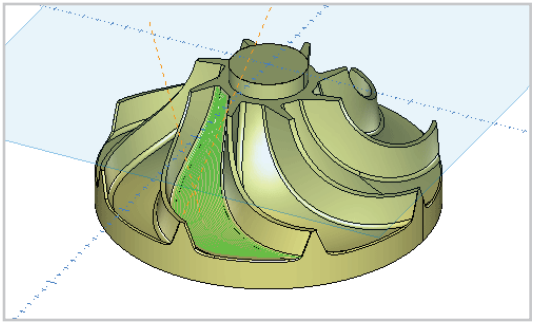
Select the Preferred Angle + Around X0 Y0 option, and enter a preferred angle of 20.

The preferred angle setting will maintain the orientation of the tool as close to 20° from vertical as possible.

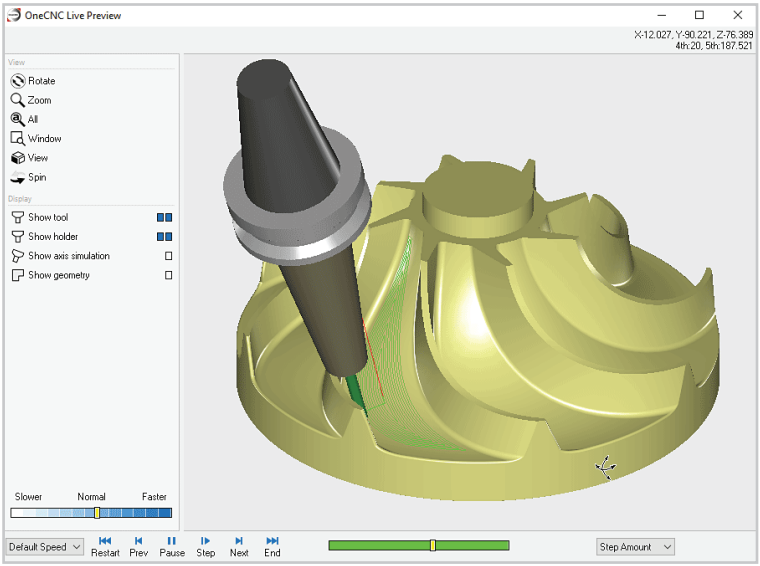
The around X0 Y0 option will keep the tool oriented towards X0 Y0 so it will always be in a radial position relative to the part.



Enter a Stepover value of 1, and 0 for Leave for finish.
Enter 0.005 for Surface Tolerance.
Select Climb Milling for Direction of Cut.
Select the Spiral Outwards Path Style.



Click Finished and the tooolpath will be created.



Preview the Area Machining toolpath and you will see that the preferred angle is maintained until the tool approaches the blade.

Once interference with the model is detected the tool will tilt to avoid the blade.

Practicing 5 axis simultaneous toolpaths

In this tutorial we covered 5 axis Swarf Cutting and 5 axis Area Machining.

The remaining 5 axis simultaneous toolpath strategies are all defined in a similar way to their 3 axis counterparts.

You can create simple models to practice these toolpaths so that you become familiar with their use.

By doing this you will be confident in their application when you need to use them.

More information about Multi axis toolpaths is available in the 'Mill Multi Axis Machining' section under 'OneCNC Mill' in OneCNC Help.

OneCNC Support

For assistance contact the OneCNC office for your location.

If you are emailing a support question, it will help us to help you if you zip and attach the file you are having a problem with, and include the following information:

Client Number
Click on Help > About OneCNC. Your client number is a five digit number with the prefix CN. The number is the same as your dongle number.

Version
Click on Help > About OneCNC to find the exact version you are running, e.g. 62.79

OneCNC product
The type and level of OneCNC such as Mill Expert.

Units
Please let us know if the file is drawn in metric or imperial units.

OneCNC Global Offices

Australia

OneCNC Australia

Phone	+61 (0)7 3286 2527
Fax	+61 (0)7 3286 4992
Email	support@onecnc.com.au
Internet	www.onecnc.com.au

Benelux

OneCNC Benelux

Phone	+31 (0) 40 22 66 212
Fax	+31 (0) 40 22 40 794
Email	support@onecnc.nl
Internet	www.onecnc.nl

Denmark

OneCNC Denmark

Phone	+ 45 20 40 02 68
Fax	+ 45 20 40 02 68
Email	support@onecnc.dk
Internet	www.onecnc.dk

France

OneCNC France

Phone	+ 33 (0)4 72 33 38 74
Fax	+ 33 (0)4 72 91 42 92
Email	support@onecnc.fr
Internet	www.onecnc.fr

Germany

OneCNC Germany

Phone	+49(0)5261 288940
Fax	+49(0)5261 2889449
Email	support@onecnc.de
Internet	www.onecnc.de

India

OneCNC India

Phone	+61 7 3286 2527
Email	support@onecnc.com.au
Internet	www.onecnc.in

Indonesia

OneCNC Indonesia

Phone	+62 31 8411187
Email	support@onecnc.id
Internet	www.onecnc.co.id

Ireland

OneCNC Ireland

Phone	+353 7196 33200
Fax	+353 7196 33500
Email	support@onecnc.co.uk
Internet	www.onecnc.co.uk

Italy

OneCNC Italy

Phone	+39 393 438 3373
Fax	+39 178 271 6215
Email	support@onecnc.it
Internet	www.onecnc.it

Japan

OneCNC Japan

Phone	+81 (0)72 760 3134
Fax	+81 (0)72 760 3135
Email	onecnc@onecnc.co.jp
Internet	www.onecnc.co.jp

Korea

OneCNC Korea

Phone	+ 82 31 695 7250
Fax	+ 82 31 695 7253
Email	support@onecnc.kr
Internet	www.onecnc.kr

Mexico

OneCNC Mexico

Phone	+52 (55) 85017429
Email	esanchez@onecnc.com.mx
Internet	www.onecnc.com.mx

Portugal

OneCNC Iberia

Phone	+34 936 473 117
Email	josep.soler@1cnc.es
Internet	www.onecnc.es

Poland

OneCNC Poland

Phone	+48(0)22 388 3460
Fax	+48(0)22 388 3461
Email	support@onecnc.pl
Internet	www.onecnc.pl

South Africa

OneCNC South Africa

Phone	+27 31 7014732
Fax	+27 31 7014736
Email	sredman@iafrica.com
Internet	www.onecnc.net

Spain

OneCNC Iberia

Phone	+34 936 473 117
Email	josep.soler@1cnc.es
Internet	www.onecnc.es

Sweden

OneCNC Sweden

Phone	+46 (0)35-7777030
Fax	+46 (0)35-7777031
Email	support@onecnc.se
Internet	www.onecnc.se

Taiwan

OneCNC Taiwan

Phone	+ 886 2 26665010
Fax	+ 886 2 26660917
Email	chou@wvision.asia
Internet	www.onecnc.com.tw

United Kingdom

OneCNC UK

Phone	+44 (0) 190 237 3054
Fax	+44 (0) 190 237 5593
Email	support@onecnc.co.uk
Internet	www.onecnc.co.uk

USA

OneCNC LLC

Toll Free	+1 877 626 1262
Phone	+1 813 874 2335
Fax	+1 813 874 2919
Email	support@onecnc.com
Internet	www.onecnc.com

USA California

OneCNC West

Phone	+1 909 931 7811
Fax	+1 909 985 6967
Email	support@onecncwest.com
Internet	www.onecncwest.com

Copyright Notice

Introduction to OneCNC Mill Multi-axis v62.80