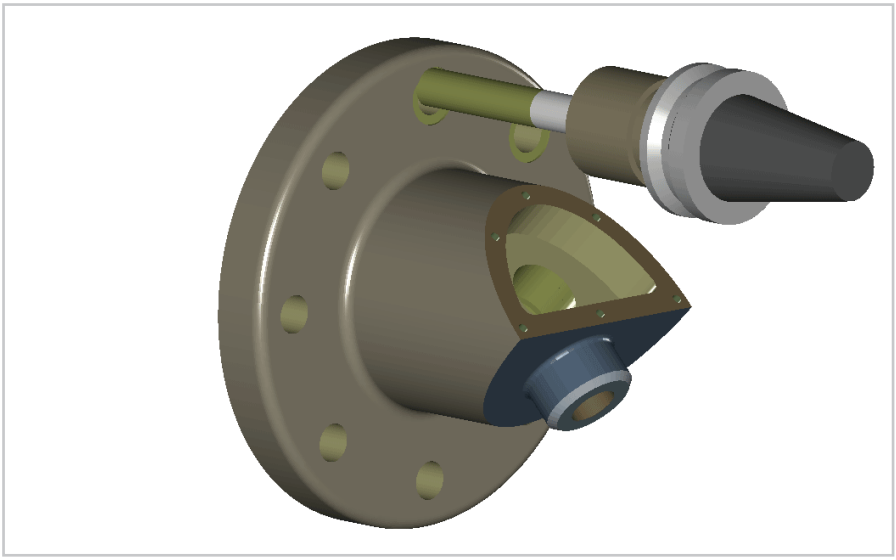
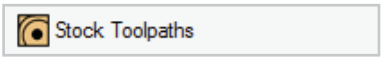
OneCNC Lathe Mill-Turn

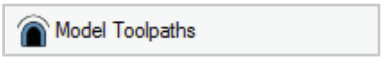


With the addition of optional multi-axis modules, OneCNC Lathe can be used for Multi-axis Mill-Turn operations.

With the activation of a Mill-Turn module you will have added icons in the toolbox:



Stock Toolpaths — Lathe Expert, Lathe Professional, Lathe Express:
The Stock Toolpaths icon opens the Stock Toolpaths dialog, which allows you to select from a range of toolpaths based on wireframe geometry such as lines and arcs.



Model Toolpaths — Lathe Expert and Lathe Professional only:
The Model Toolpaths icon opens the SMT Mold Toolpaths dialog, which has options for creating toolpaths based directly on your solid modeling.

The modes made available by each OneCNC module are:

C Axis Module
C Axis Face
C Axis Wrap

C + Y Module
C Axis Face
C Axis Wrap
C Axis Full
Y axis Position

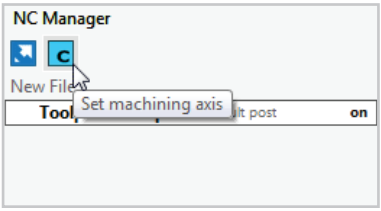
C + Y + B Module
C Axis Face
C Axis Wrap
C Axis Full
Y axis Position
B axis Position

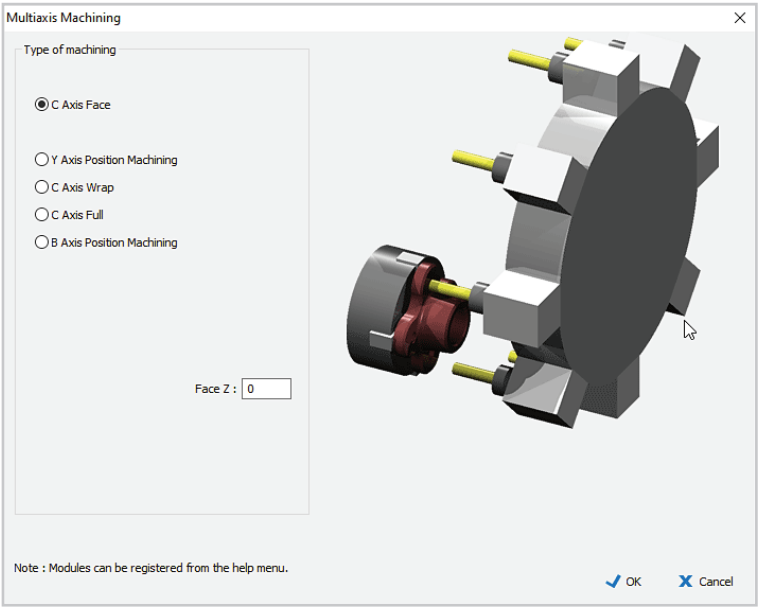
CONTENTS

OneCNC Lathe Mill-Turn 1
OneCNC Mill - Turn Modes 2
Multi-axis machining mode selection 4
Multi-axis post settings 6
OneCNC Lathe C axis module 7
C Axis Face machining 7
Highspeed Open Pocket 14
Chamfer milling 27
Feature recognition drilling 33
Saving Hole Feature settings 42
C Axis Wrap machining 43
Create geometry for Wrap machining 46
Define a C Axis Wrap Engraving toolpath 54
OneCNC Lathe C+Y axis module 62
C Axis Full machining 62
Define a C Axis Full toolpath 63
Y Axis Position machining 67
Set a Y axis machining plane 70
Define a Y axis toolpath 72
OneCNC Lathe C+Y+B axis module 80
B Axis Position machining 80
Set a B axis machining plane 84
Create a B axis toolpath 87
Recreate sample toolpaths in the first position 95
Change the B axis position 98
Recreate sample toolpaths in the second position 101
OneCNC Support 104
OneCNC Global Offices 105
Copyright Notice 109

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 mode you want to use, and the image in the dialog will show the type of machining selected.

For C Axis Face and C Axis Wrap, text boxes will appear for the input of values relevant to those methods. Enter any values if necessary, and click OK.

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

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

C module: Lathe Expert, Lathe Professional and Lathe Express

C Axis Face

C axis Face stock and model toolpaths are applied using geometry parallel to the World YZ plane.

C Axis Wrap

C axis Wrap applies milling toolpaths defined on flat geometry to a cylindrical surface centered on the world X axis.

The Wrap function is often used in conjunction with the model unwrap function which generates developed geometry from a cylindrical model. Unwrapped or manually drawn geometry can be used to engrave or pocket an external diameter.

C+Y module: Lathe Expert and Lathe Professional only

C Axis Full

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

Y Axis Position

Y axis position machining involves the application of milling tool-paths to geometry on a plane which has one axis parallel to the lathe axis. The plane can be defined manually. If radial holes are being drilled you can use the Plane From Hole function to generate the required plane automatically.

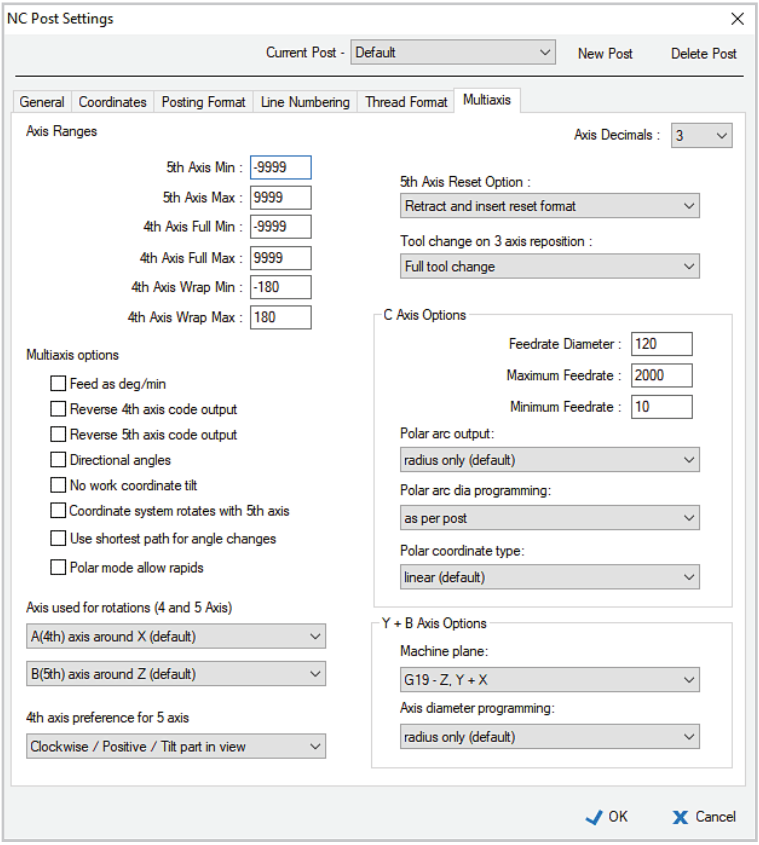
C+Y+B module: Lathe Expert and Lathe Professional only

B Axis Position

For B Axis Position toolpaths, 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, the part will rotate to the position specified by the plane.

Multi-axis post settings

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

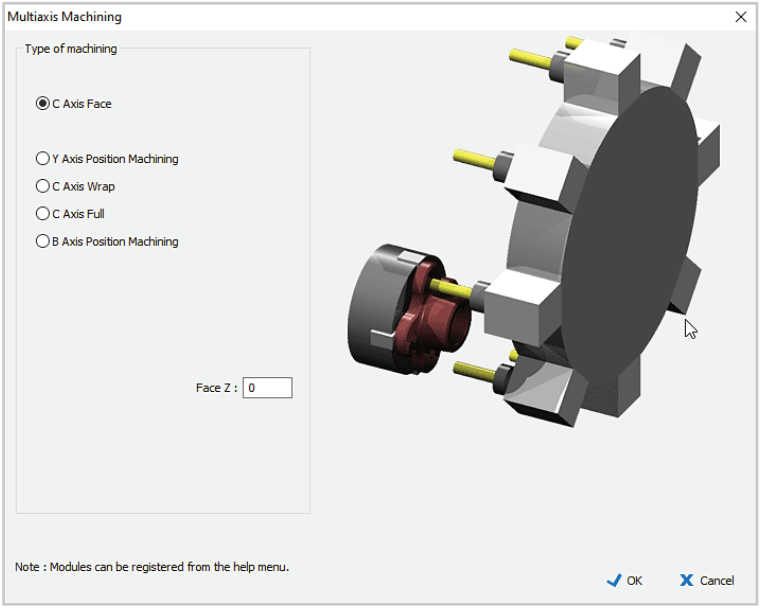


It is essential that the settings in this dialog are correct for your machine.

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.

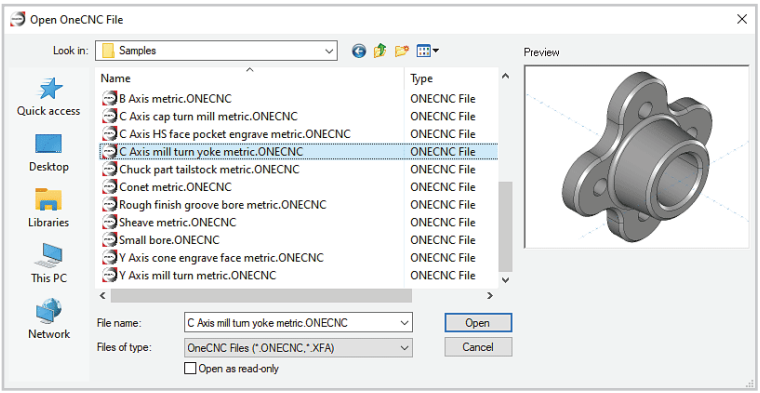
For more information on Post settings, see the NC Post and Properties topic in the Lathe NC Post and DNC Settings section of OneCNC Help.

OneCNC Lathe C axis module
C Axis Face machining

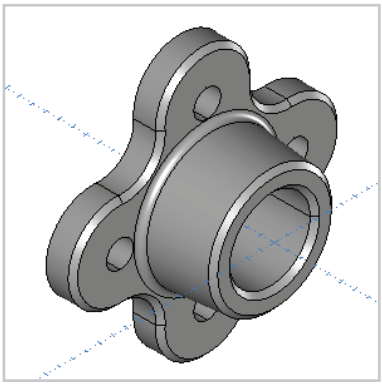


C axis Face stock and model toolpaths are milling operations using a tool whose axis of rotation is parallel to the lathe Z axis.

To see how they are created, we will copy a completed toolpath group in a sample file, then delete and recreate the C axis toolpaths in the duplicate group.



Open the sample file ‘C Axis mill turn yoke.ONECNC’.

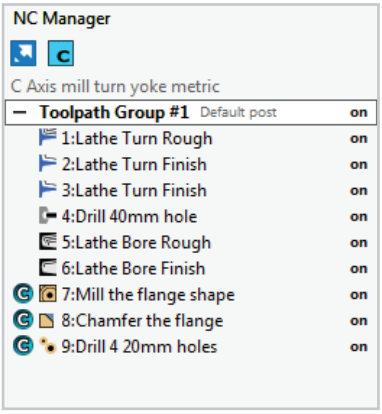


The file contains a part model and example toolpath operations.

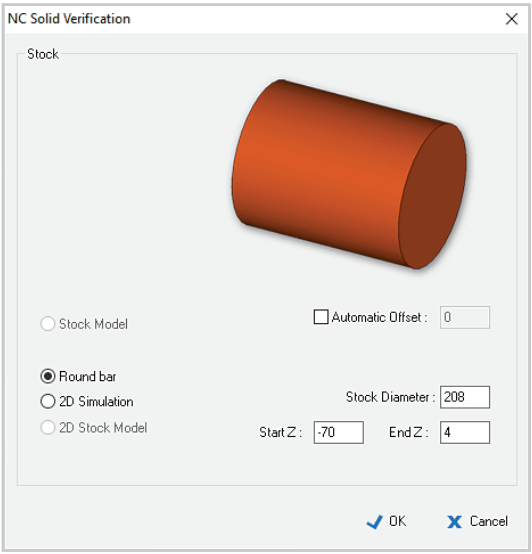
Use Save As to save a copy of the file as ‘Tutorial Lathe C axis face.ONECNC’.

In the NC manager operations list you will see the example toolpath group.

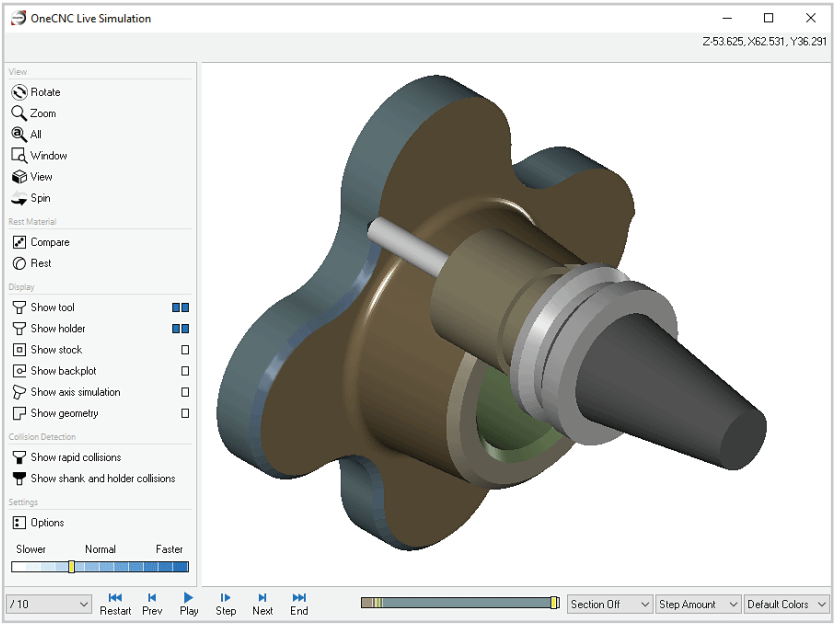
It has Lathe turn, drill, and bore operations, and C axis mill-turn operations marked with a C symbol.



Right click on the example toolpath group and click on Simulate/Rest in the NC Manager context menu.



Fill out the Stock settings as shown, and click OK.

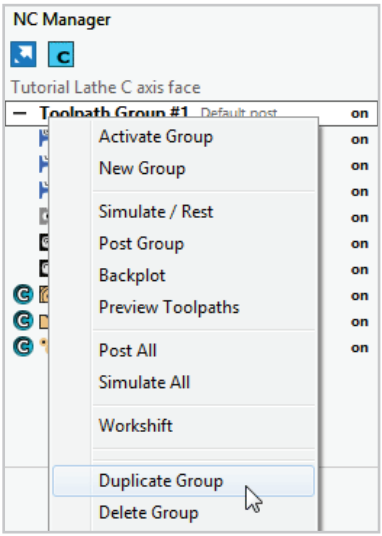


You will see the Lathe operations machine the basic shape of the part, followed by the C axis mill operations we will be working on.

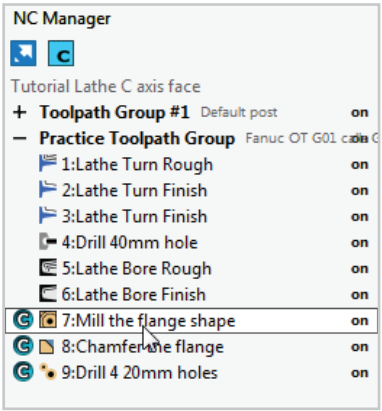
Close the simulation window when you are ready to proceed.

Right click on Toolpath Group #1 in the NC manager, and select Duplicate Group.

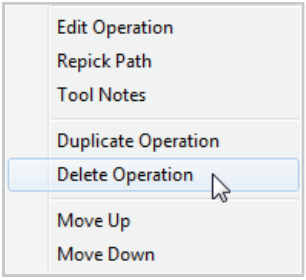
Double click on the duplicate group and rename it 'Practice Toolpath Group'. This group will now be the active group.



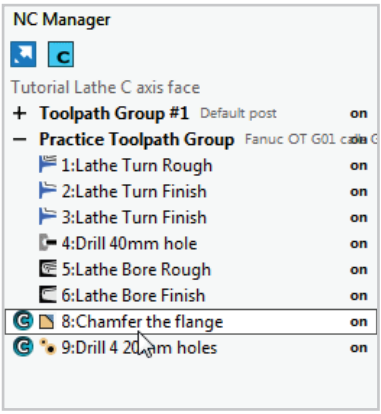
Right click on the first C axis operation in your practice group.



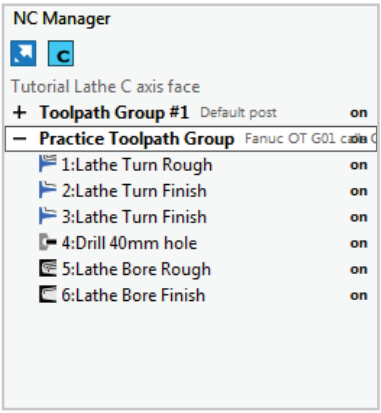
Select Delete Operation from the context menu. Click Yes when the confirmation dialog appears.

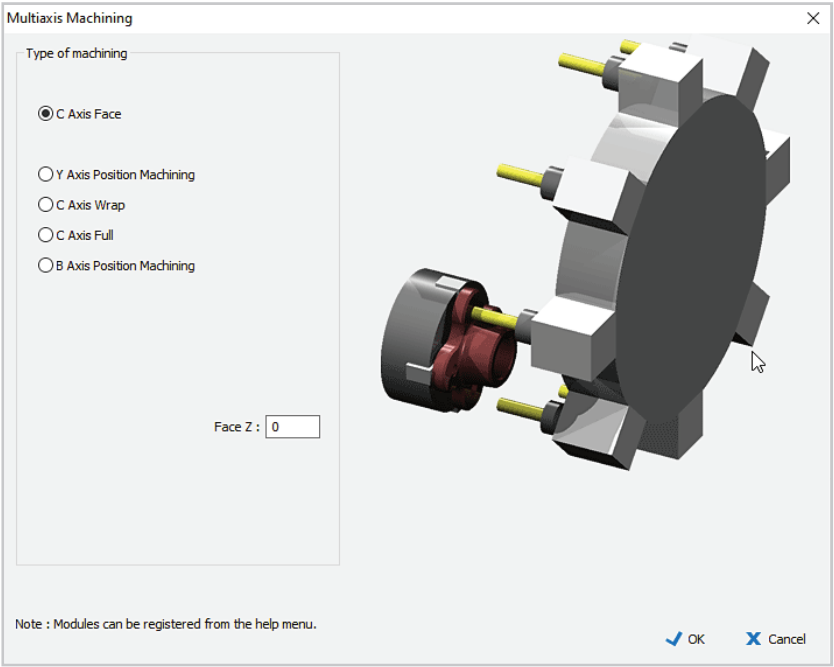


Delete the other C axis operations in the Practice group.



The operations list in the NC manager should now show the new Practice Toolpath Group as the active group, with only the original lathe operations.





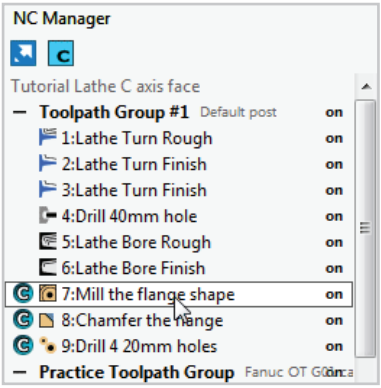
To prepare for C axis toolpathing, click on the machining axis mode icon and select the C Axis Face machining method.
Set Face Z at 0, and click OK.

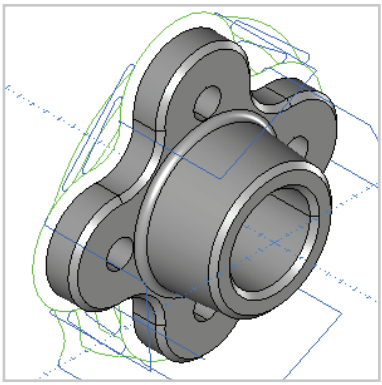
The icon will indicate the C axis Face mode is active.

C Axis Face toolpaths are created in the coordinate system of the C-axis Plane, which corresponds to the World YZ Plane. The current construction plane is not affected.

Our first C axis toolpath will mill the flange shape with an open pocket operation defined by an inner and outer boundary.

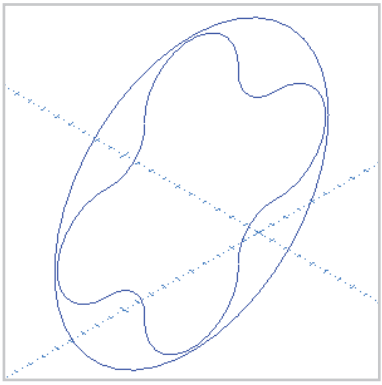
Click on the function named 'Mill the flange shape' in the original group.





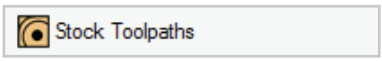
You will see the preview in the drawing window of the open pocket toolpath backplot.

Turn on the Flange layer, and turn off the Model and Construction layers.

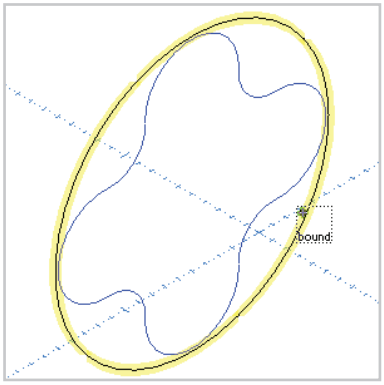


You will see the inner and outer boundary used for this toolpath.

Before continuing make sure the Practice Toolpath Group is active.

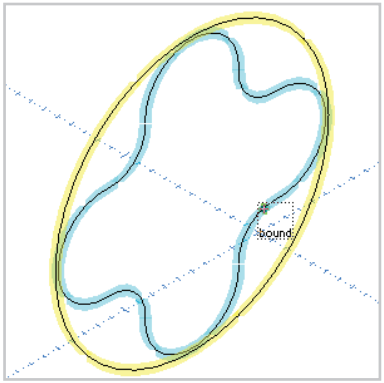


Click on the Stock Toolpaths icon and select the HS Pocketting toolpath.



Click on the outer circular boundary to select it as the main boundary for the toolpath.

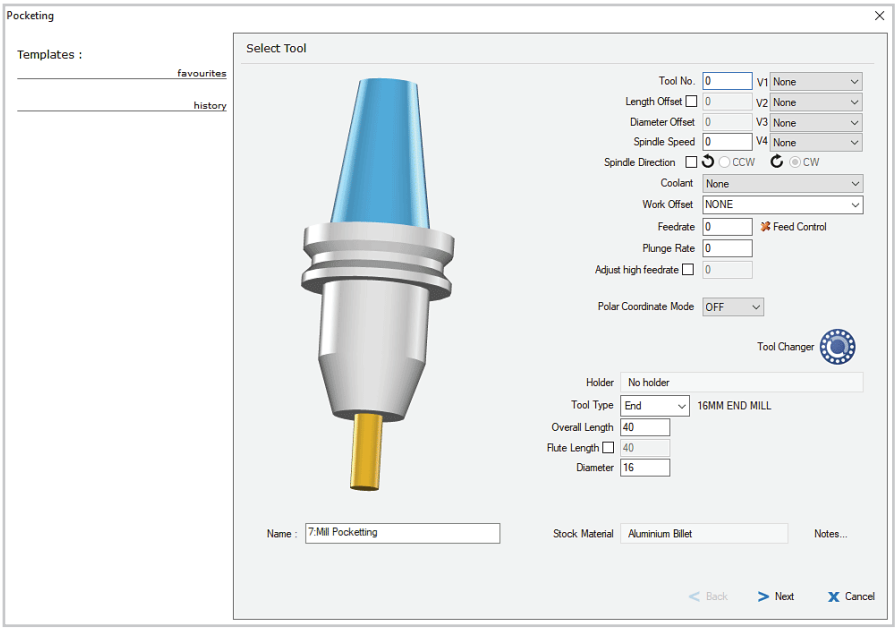
An Open Pocket toolpath is allowed to cross the main boundary.



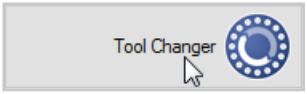
Click on the inner boundary to select it as a wall boundary for the toolpath.

An Open Pocket toolpath is not allowed to cross a wall boundary.

Right click to end the selection process.

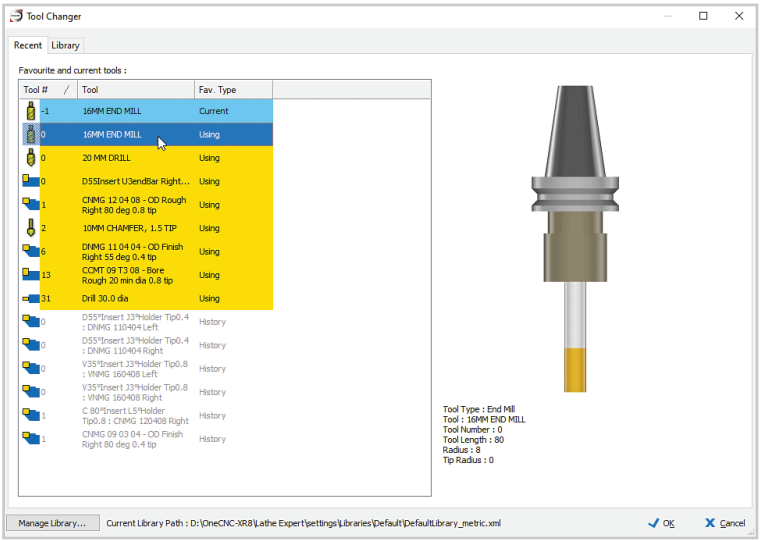


The Select Tool dialog appears so you can select the tool to use from the Tool Changer, and enter speeds and feeds along with other relevant tool settings.



Click on the Tool Changer icon to open the Tool Changer, the OneCNC mill tool management system in which you can store definitions for all the tools you use.

The Tool Changer dialog opens, with the Recent tab active.



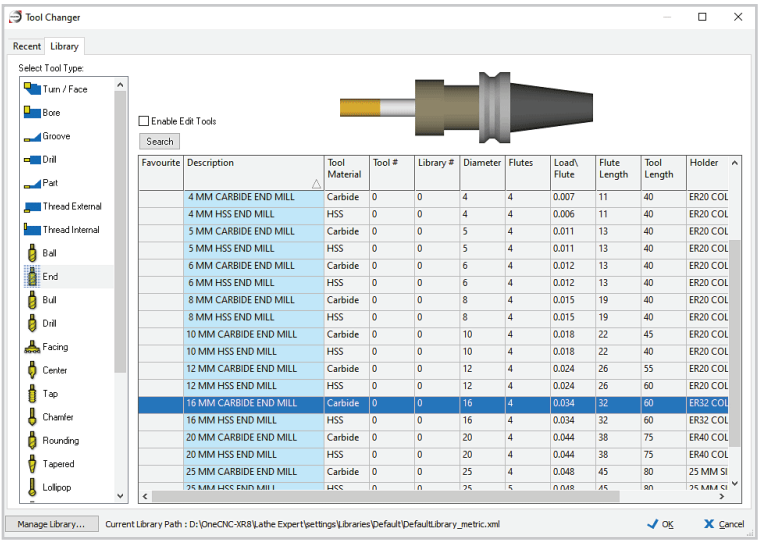
Tools you use regularly can be selected quickly from this tab.

The list shows tools you have tagged as favourites in the Tool Library, tools which are in use in the current file, and tools which have been used recently.

The tools already in use in the current file will be highlighted in yellow.

The selected tool will be displayed with its associated holder in the preview on the right of the dialog.

When the tool you want to use is not shown in the Favourites list, click on the Library tab to see a categorized listing of all available tools.



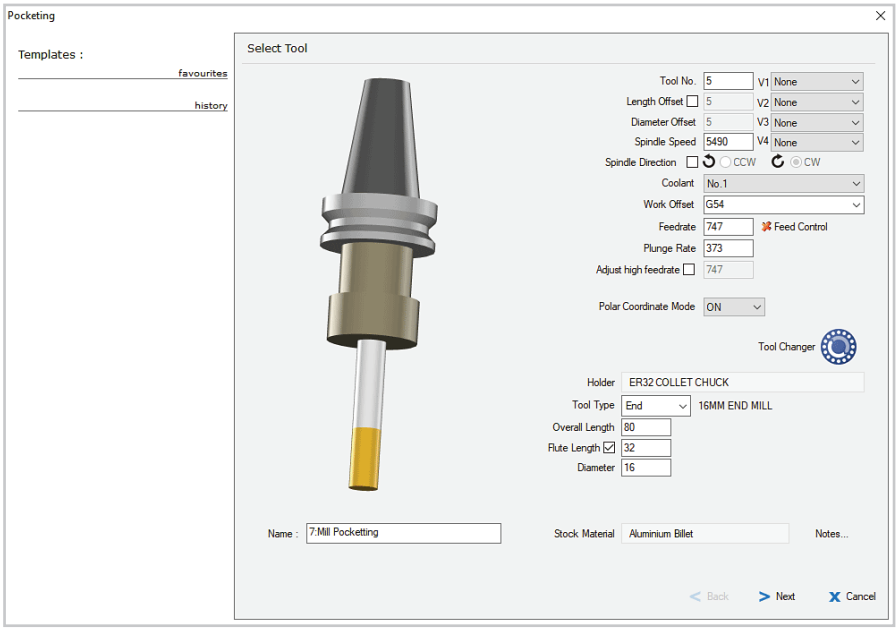
For this example you would click on the End Mill icon at the left of the Tool Library to select the End Mill list. You can then select the 16MM CARBIDE END MILL in the End Mill list, by clicking anywhere in the row.

For this example we need a long series tool, so return to the Recent tab and select the 16MM END MILL that is already in use.

Click OK to return to the Select Tool dialog.

For full details of milling tool and holder selection, open OneCNC Help and see the topic:

OneCNC Mill -> Mill Common CAM Settings ->Selecting a Tool

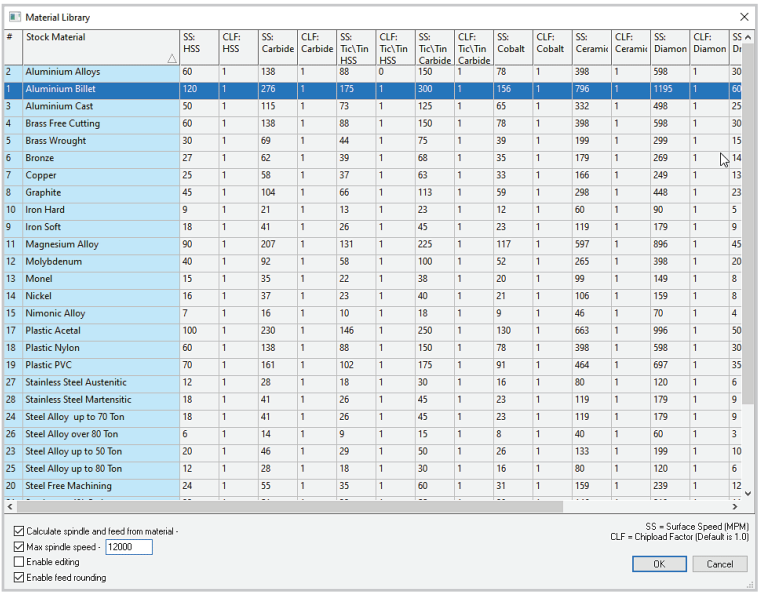


The Select Tool dialog now shows the selected tool and its associated holder.

Enter Tool No. 5, and select Work Offset G54 and Coolant No. 1. Set Polar Coordinate Mode to ON.

Feedrates and spindle speeds will be calculated automatically when we select our stock material.

Click on the Stock name in the lower area of the Select Tool dialog to open the Material List.



#	Stock Material	SS: HSS	CLF: HSS	SS: Carbide	CLF: Carbide	SS: Tic\Tin HSS	CLF: Tic\Tin HSS	SS: Tic\Tin Carbide	CLF: Tic\Tin Carbide	SS: Cobalt	CLF: Cobalt	SS: Ceramic	CLF: Ceramic	SS: Diamon	CLF: Diamon	SS Di
2	Aluminium Alloys	60	1	138	1	88	0	150	1	78	1	398	1	598	1	30
1	Aluminium Billet	120	1	276	1	175	1	300	1	156	1	796	1	1195	1	60
3	Aluminium Cast	50	1	115	1	73	1	125	1	65	1	332	1	498	1	25
4	Brass Free Cutting	60	1	138	1	88	1	150	1	78	1	398	1	598	1	30
5	Brass Wrought	30	1	69	1	44	1	75	1	39	1	199	1	299	1	15
6	Bronze	27	1	62	1	39	1	68	1	35	1	179	1	269	1	14
7	Copper	25	1	58	1	37	1	63	1	33	1	166	1	249	1	13
8	Graphite	45	1	104	1	66	1	113	1	59	1	298	1	448	1	23
10	Iron Hard	9	1	21	1	13	1	23	1	12	1	60	1	90	1	5
9	Iron Soft	18	1	41	1	26	1	45	1	23	1	119	1	179	1	9
11	Magnesium Alloy	90	1	207	1	131	1	225	1	117	1	597	1	896	1	45
12	Molybdenum	40	1	92	1	58	1	100	1	52	1	265	1	398	1	20
13	Monel	15	1	35	1	22	1	38	1	20	1	99	1	149	1	8
14	Nickel	16	1	37	1	23	1	40	1	21	1	106	1	159	1	8
15	Nimonic Alloy	7	1	16	1	10	1	18	1	9	1	46	1	70	1	4
17	Plastic Acetal	100	1	230	1	146	1	250	1	130	1	663	1	996	1	50
18	Plastic Nylon	60	1	138	1	88	1	150	1	78	1	398	1	598	1	30
19	Plastic PVC	70	1	161	1	102	1	175	1	91	1	464	1	697	1	35
27	Stainless Steel Austenitic	12	1	28	1	18	1	30	1	16	1	80	1	120	1	6
28	Stainless Steel Martensitic	18	1	41	1	26	1	45	1	23	1	119	1	179	1	9
24	Steel Alloy up to 70 Ton	18	1	41	1	26	1	45	1	23	1	119	1	179	1	9
26	Steel Alloy over 80 Ton	6	1	14	1	9	1	15	1	8	1	40	1	60	1	3
23	Steel Alloy up to 50 Ton	20	1	46	1	29	1	50	1	26	1	133	1	199	1	10
25	Steel Alloy up to 80 Ton	12	1	28	1	18	1	30	1	16	1	80	1	120	1	6
20	Steel Free Machining	24	1	55	1	35	1	60	1	31	1	159	1	239	1	12

The Material List is a database of surface speeds for machining materials with different tool types.

When the Calculate spindle and feed from material check box is selected, the surface speeds are combined with tool information from the Tool Library to calculate the speed and feed settings.

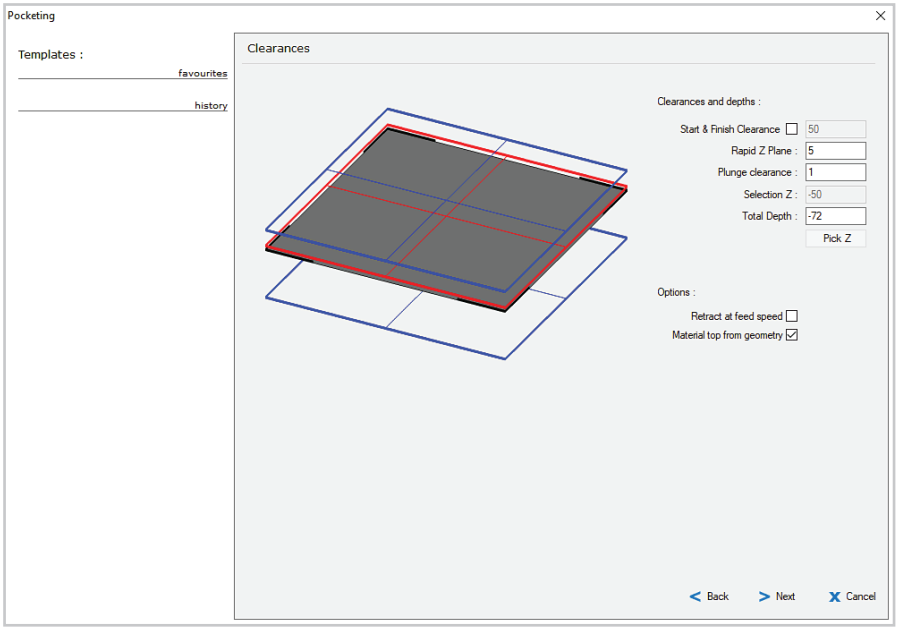
Select Aluminium Billet and click Accept. You will see the Spindle Speed, Feedrate and Plunge Rate update automatically.

Select the check box for Adjust high feedrate, and the Max box which appears. This is the speed for repositioning moves in the high speed pocketing toolpath which do not require the tool to retract.

Click Next to continue.

Plunge Rate

The Plunge Rate is calculated by default to 50% of the calculated Feedrate. You can change it to any percentage by typing the percentage you want in the Plunge Rate box and that will be the new default. For instance if you enter 40% in the box the Plunge Rate will update immediately, and 40% will be the new default.

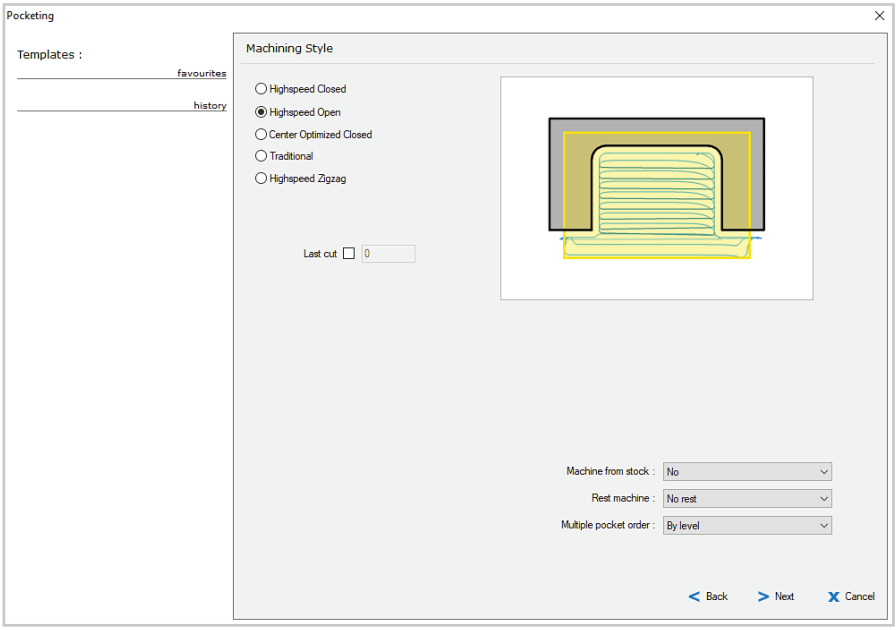


In the Clearances dialog, set the Rapid Z plane to 5.

Plunge Clearance is the incremental distance above the material at which the plunge move changes from rapid to feed speed, and must be a positive value. Enter a Plunge clearance of 1.

Select the check box for Material top from geometry, and set total depth to 22. The selected boundaries are at Z-50, and the pocketing will be 22 deep from that reference level.

Click Next to continue.

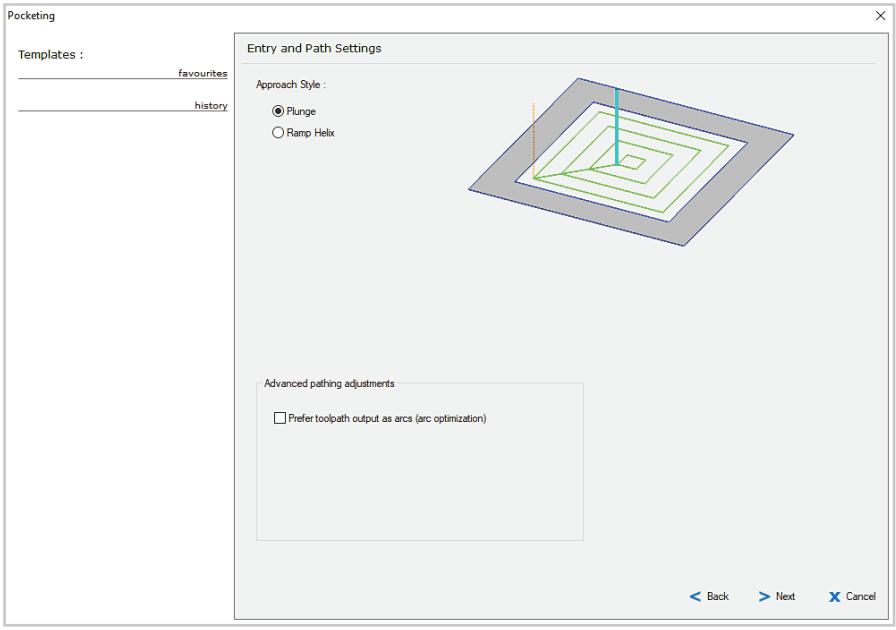


Select the Highspeed-Open Machining Style.

This toolpath will machine the pocket area without crossing the inner boundary, but is allowed to cross the outer boundary so there are no uncut areas left.

You can compare the toolpath methods by clicking on the option buttons and observing the preview.

Click Next when you are ready to continue.

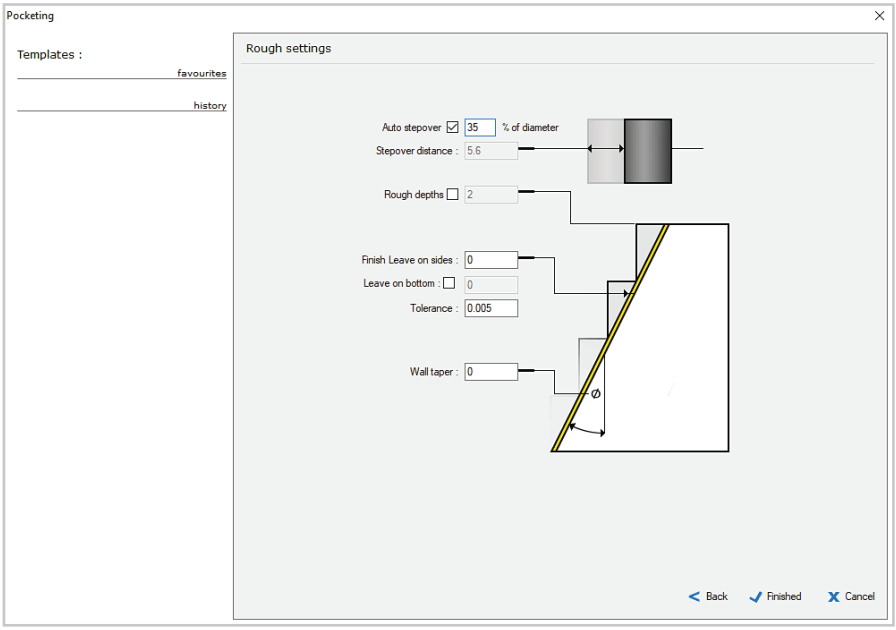


The settings in the Path Creation dialog determine how the tool will approach the job.

Plunge entry is suited for open pocketing where the tool will plunge off the job.

Select Plunge and click Next.

The Ramp Helix method is more suitable for closed pockets, as the tool will enter the job on a spiral path which allows chips to escape. You can use a Ramp Angle of about 3 for most operations.

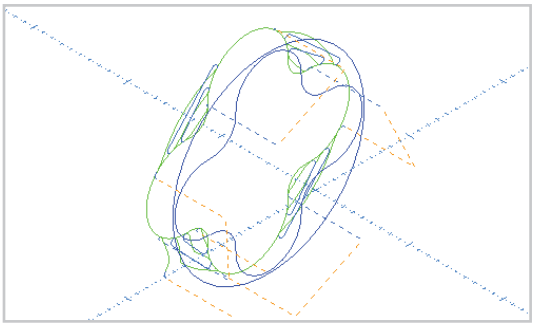


Rough settings allow you to cut the pocket in multiple passes.

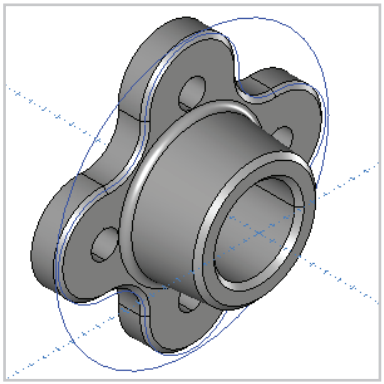
We do not need roughing passes for this job, so set Auto stepover of 75% and leave Rough Depths unchecked.

Set Leave on sides and Wall taper at 0.

Set Tolerance to 0.005.

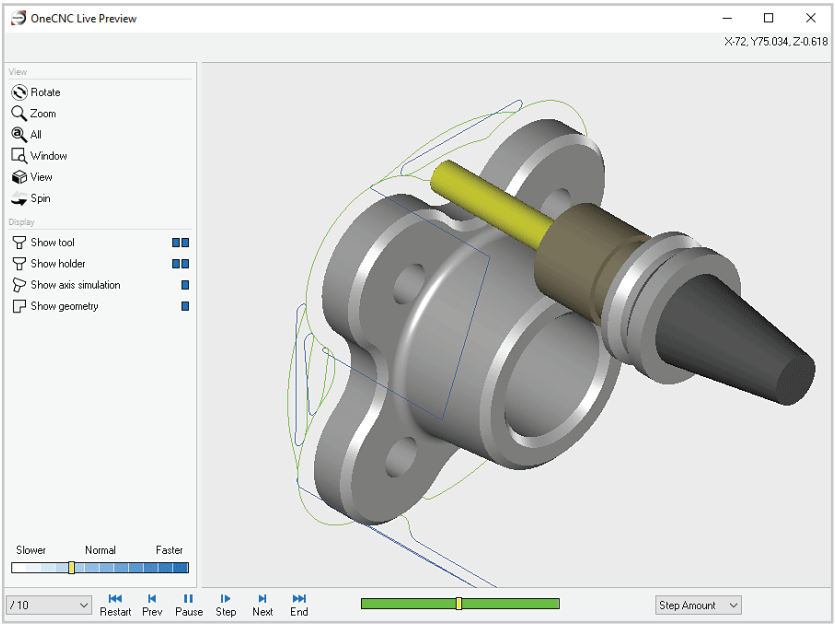


Click Finished and the toolpath will be created.



To preview the toolpath, turn on the Model layer and right click on the new toolpath in the NC manager.

Select Preview Toolpath from the NC Manager context menu.

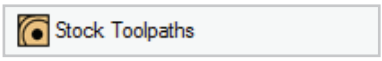


The preview will show the solid model and the tool movement.

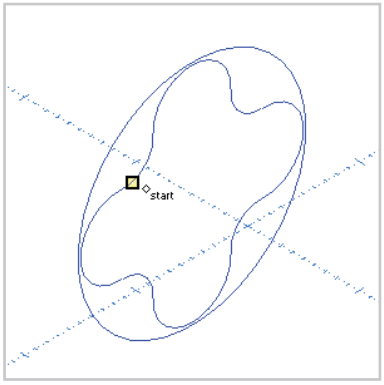
Click on Show axis simulation in the Live Preview toolbox to change the motion mode.

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

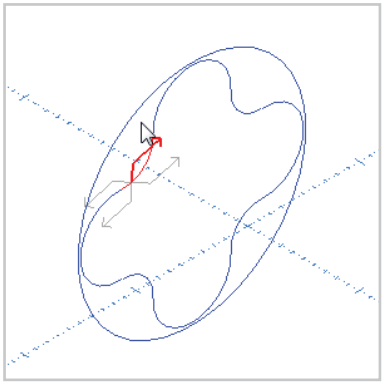
In the next step we will machine the chamfer on the flange. Turn off the Model layer again so you have a clear view of the flange boundary.



To machine the chamfer on the flange, click on the Stock Toolpaths icon in the toolbox, and select the Chamfer edge toolpath.



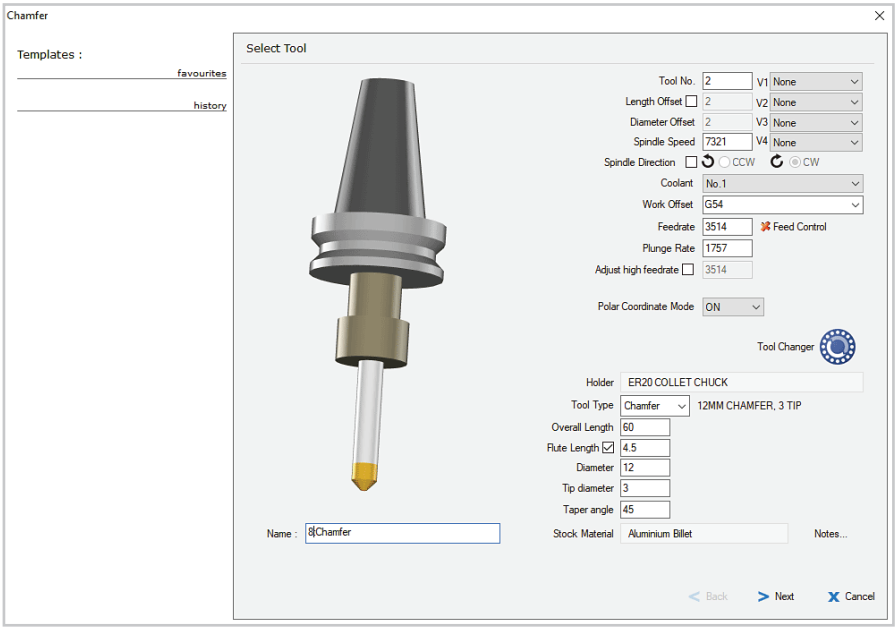
Click on the flange outline to set the start position of the operation as shown.



Hold the cursor outside the profile and click to set the offset side and direction for the toolpath.

Press F3 to automatically select the rest of the profile chain.

Right click to end the selection process.



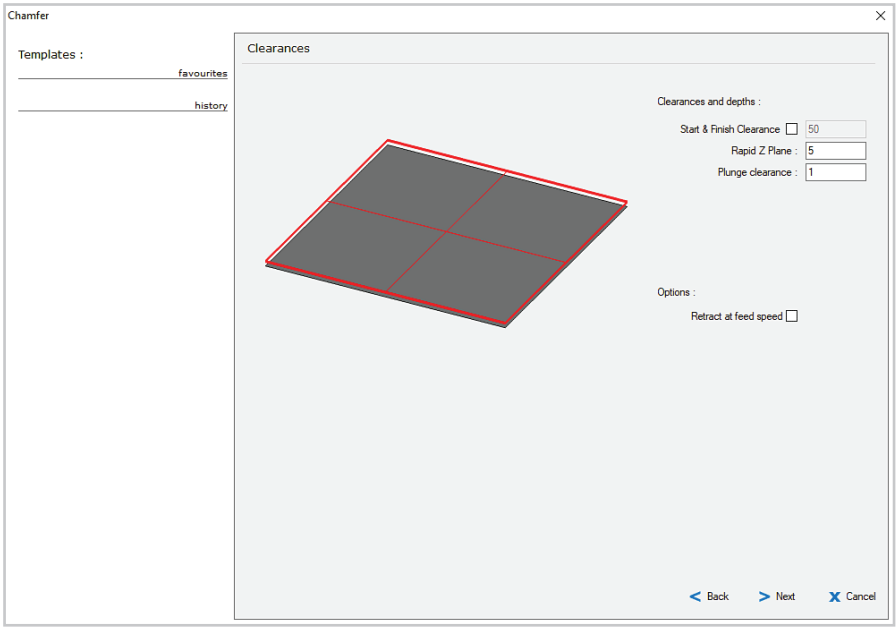
Select the 12mm chamfer tool and enter Tool No. 2.

Set Coolant to No 1 and Work Offset to G54.

Change the tool length to 60, so the tool holder will clear the part.

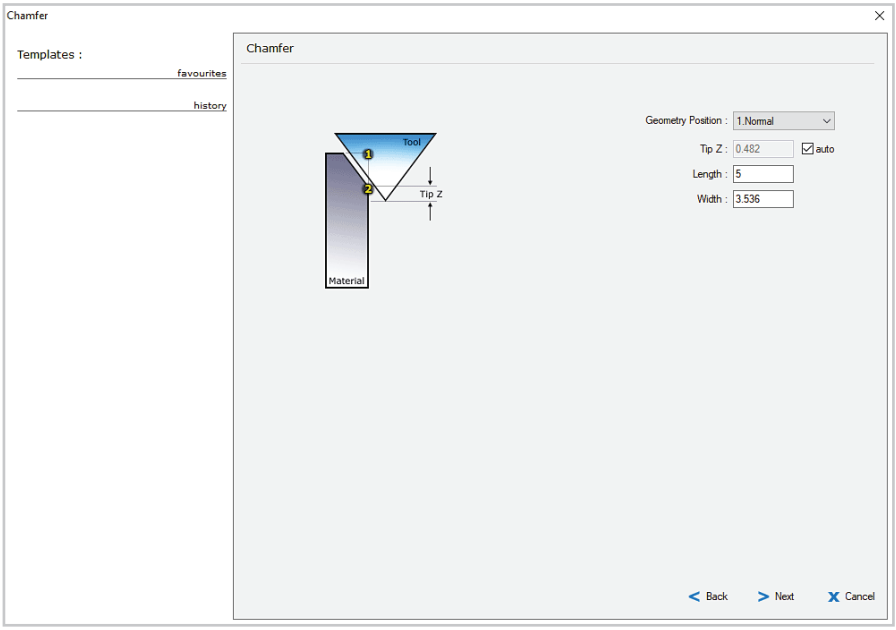
Select Aluminium Billet stock.

Clear the check box for Adjust high feedrate, set Polar Coordinate Mode to On, and click Next.



In the Clearances dialog, set the Rapid Z plane to 5, and plunge clearance to 1.

Click Next to continue.

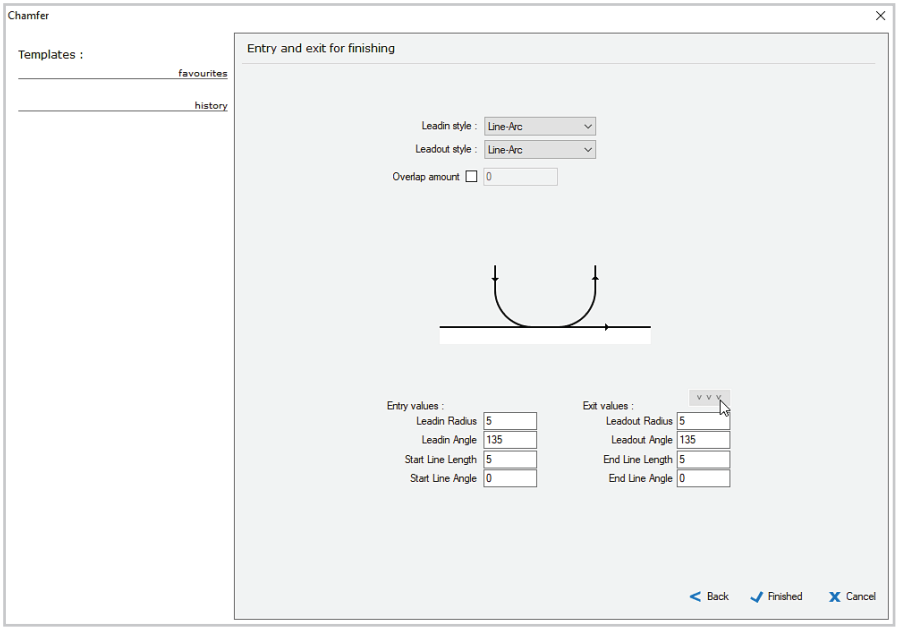


Select Normal for Geometry Position, as the defining boundary is at the intersection of the surfaces being chamfered.

Select the auto check box for Tip Z, which will calculate the Z depth so that the flute of the cutter is centered on the chamfer.

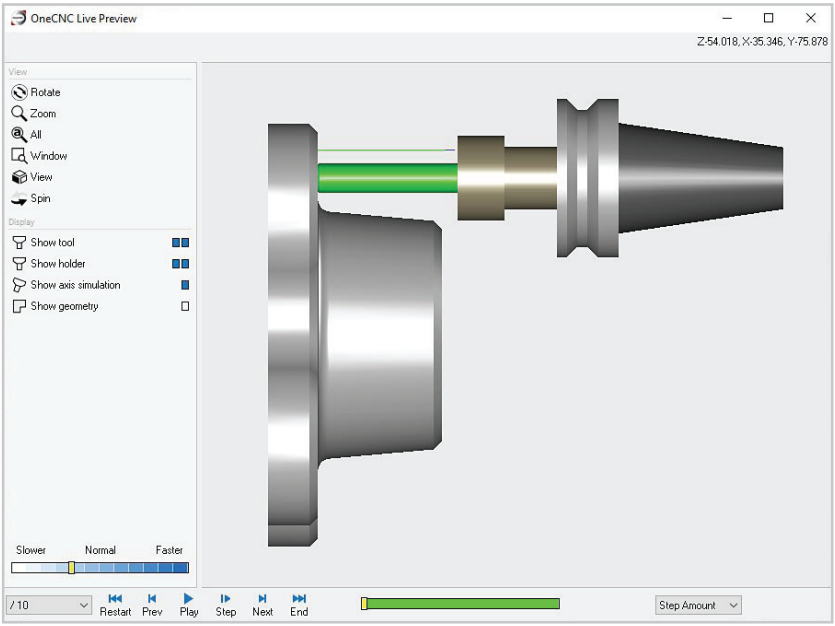
Enter a value of 5 for Length, which is the diagonal size of the chamfer. The Width, which is measured level with the face, will update to 3.536.

Entry/Exit conditions determine how the cutter approaches and leaves the profile path.



Select Line/Arc for Leadin and Leadout. Enter a Leadin Radius of 5, Leadin Angle of 135, a Start Line Length of 5 and a Start Line Angle of 0.

Click on the icon over the Exit values to copy the Leadin settings to the Leadout settings, and click Finished to create the toolpath.

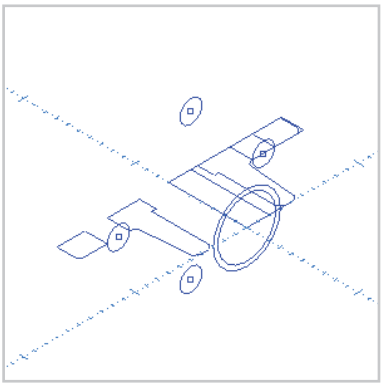


Preview the toolpath and select the Front View.

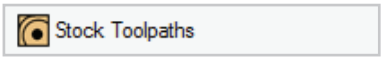
You will be able to visually confirm that the tool holder stays clear of the part.

Feature recognition drilling

To drill the four holes in the flange, we will use Hole Feature recognition. This function can find circular hole features in geometry or solid modelling, but to avoid confusion it is best to have only one type visible.

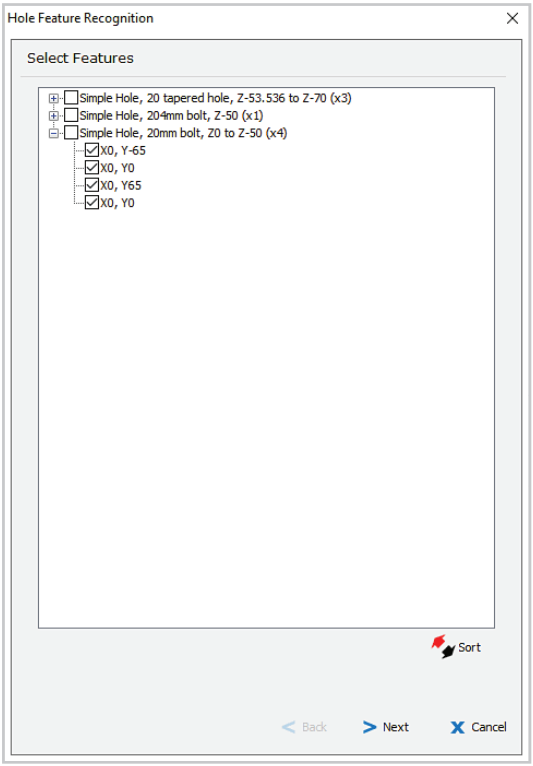


Turn on the Construction layer and turn off the Model and Flange layers.



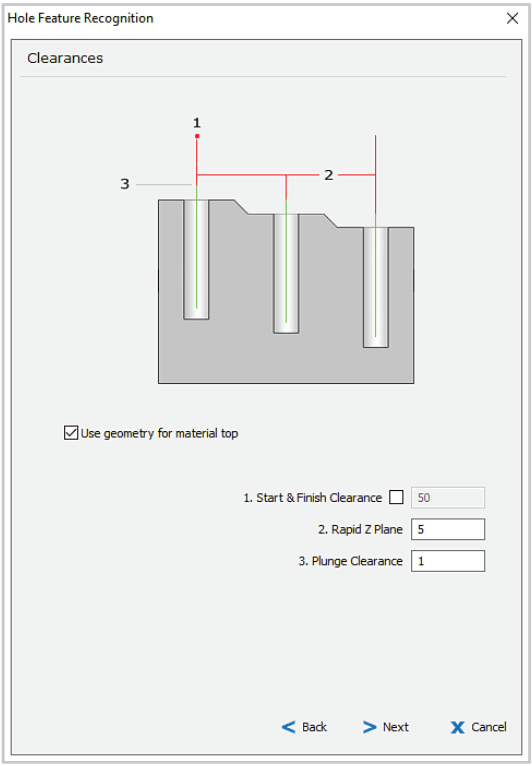
Click on the Stock Toolpaths icon and select the Drill hole wizard.

The holes found are listed in the selection dialog.



Select the check box for the ‘Simple Hole 20mm bolt, Z-50 (x4)’.

You will see the holes selected for drilling highlighted in the drawing window. Click Next to proceed.

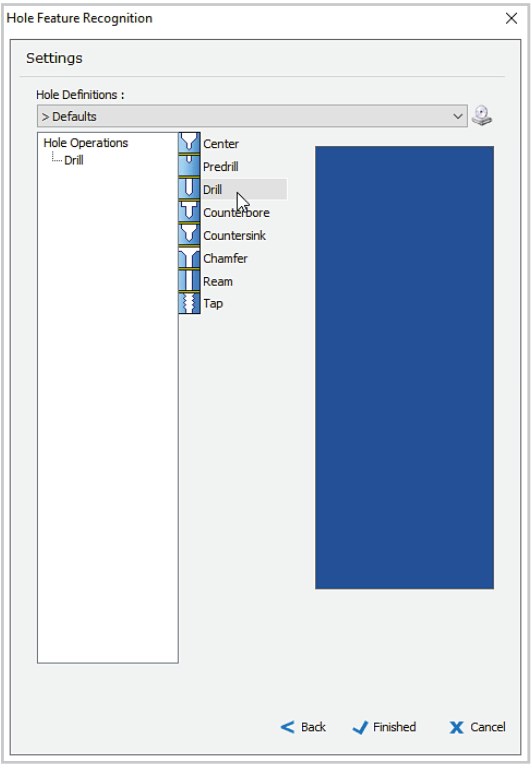


Enter 5 for the Rapid Z Plane clearance, and 1 for the Plunge Clearance.

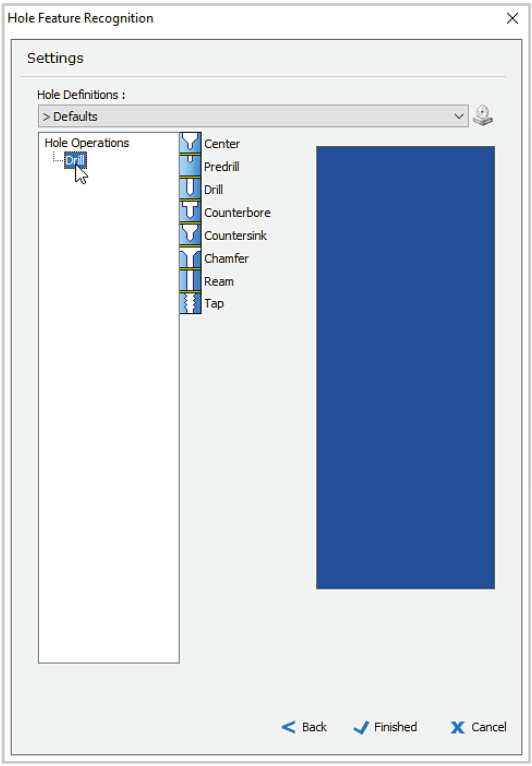
Select the check box to use geometry for material top.

This will set the Material Top for the drilling operation to the selected geometry which is at Z-50.

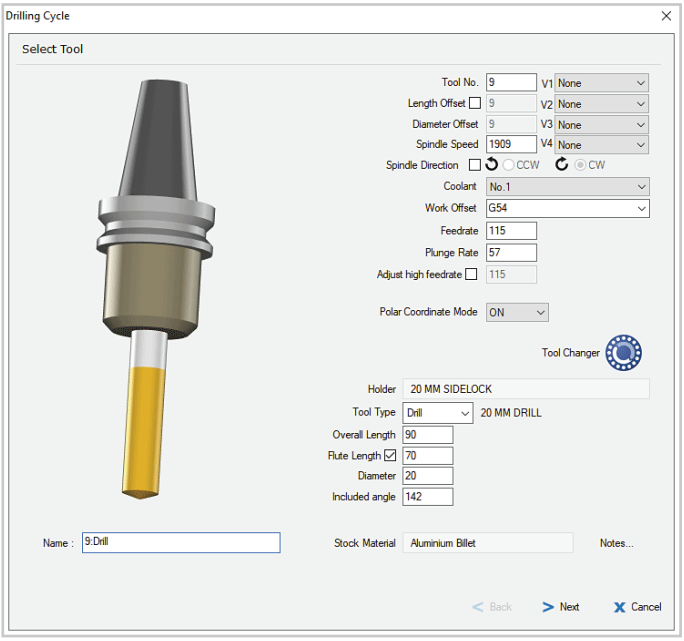
Now we have specified the positions and clearance values, we can apply one or more hole operations in the settings dialog.



Click on Drill to add a standard drill operation. The operation appears in the hole operations list but is not defined, so there is no preview in the blue pane on the right of the dialog yet.



Double click on the new drill operation in the hole operations list to define the drilling settings.

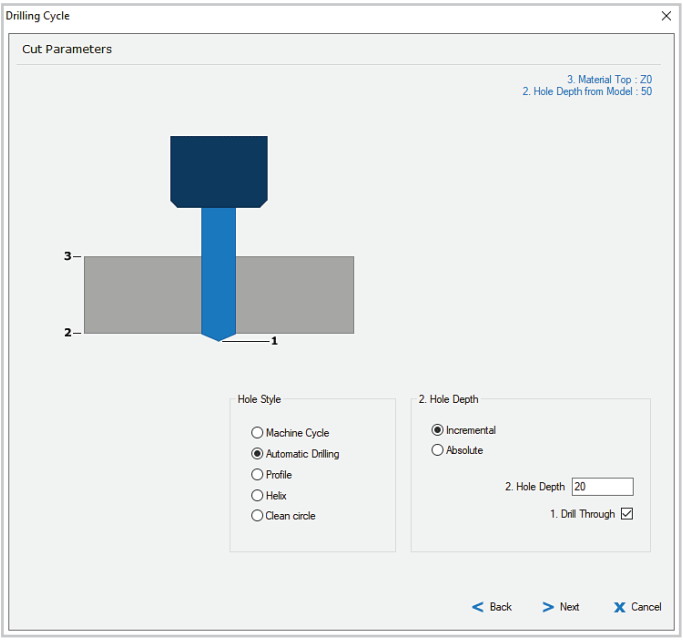


Click on the tool image in the Select Tool dialog, and choose the 20mm Drill from the Recent tab.

Click Accept in the Tool List to return to the tool dialog.

Click on the Stock icon and choose Aluminium Billet from the Material List.

Select Coolant No1, and Work Offset G54. Leave Adjust high feedrate and Polar Coordinate Mode off.

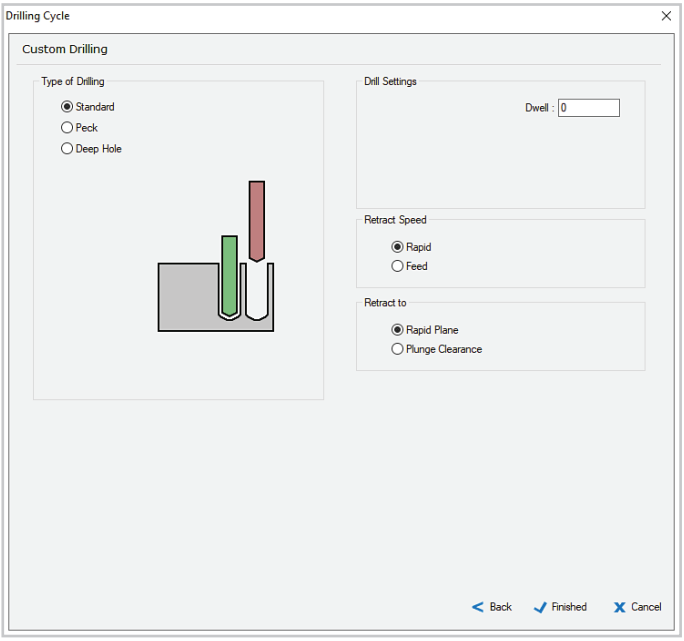


Select Automatic Drilling for the Hole Style.

The face of the flange is at Z-50 and the flange is 20 thick.

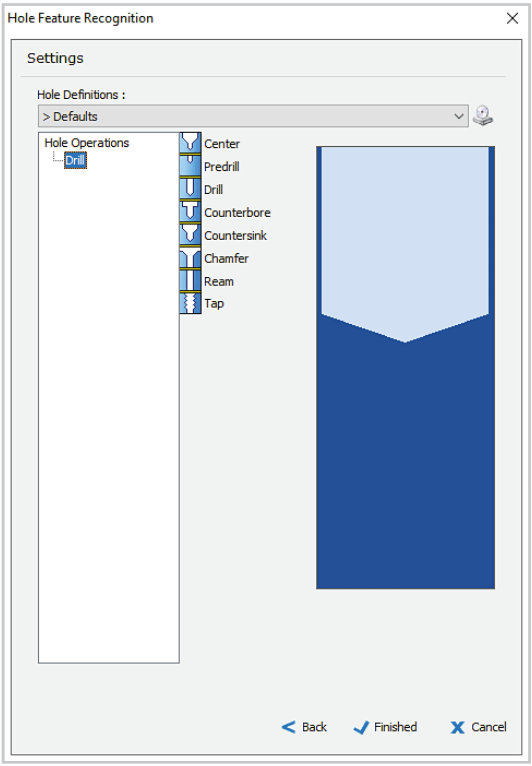
Select Incremental for Depth Style and enter a Hole Depth of 20.

Select the Drill Through check box. This will increase the Z depth of the drill point so that the outside of the drill passes the base of the hole.



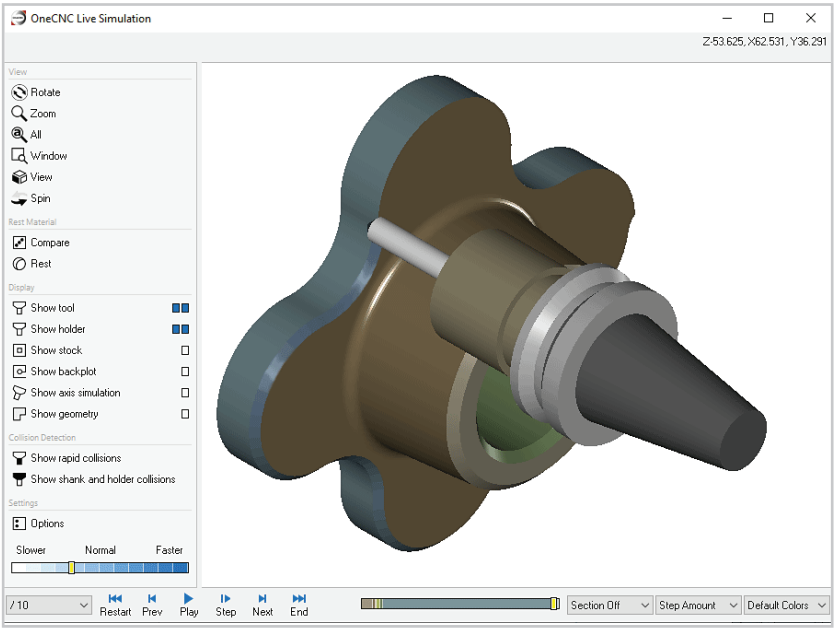
Set the Drilling Type to Standard, with Retract to Rapid Plane and Rapid Retract Speed.

Click Finished to end the Drill definition, and you will be returned to the Hole Feature Recognition Settings dialog.



A cross section preview of the hole operation is displayed on the right of the dialog.

We do not have other operations to add, so click Finished and the drilling toolpath will be created.



Simulate the toolpath group to confirm the settings you have made. The C-axis operations are now complete.

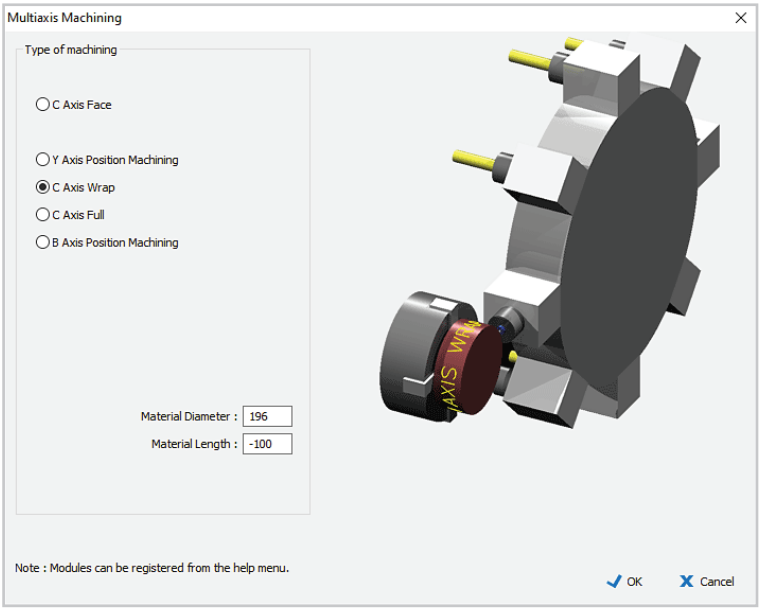
In this tutorial we have only covered stock toolpaths, but in Lathe Professional and Lathe Expert solid model toolpaths are also available in the C axis Face environment. To learn about these toolpaths, go to the SMT 3D Toolpaths section, under OneCNC Mill in the OneCNC Help files.

Saving Hole Feature settings

If you define hole operations you want to re-use, you can save the settings by clicking on the icon at the right of the Hole Definitions selector, and selecting Save As.

Once you have saved a definition, it can be selected from the list and used as it is, or edited and saved as a new definition.

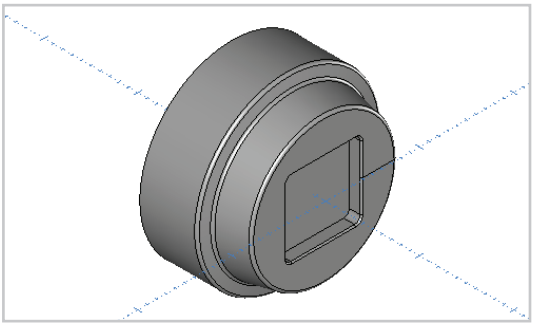
C Axis Wrap machining



A C axis Wrap toolpath is designed to apply a machining operation to a cylindrical surface.

To simplify the creation of wrap toolpaths they are defined using flat geometry, located using the unwrapped cylinder as a reference.

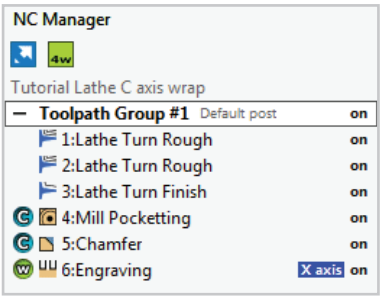
To see how the toolpaths are created, we will open a sample file, then re-create an existing C axis wrap toolpath.



Open the sample file ‘C Axis HS face pocket engrave.ONECNC’
Use Save As to save a copy of the file as
‘Tutorial Lathe C axis wrap.ONECNC’

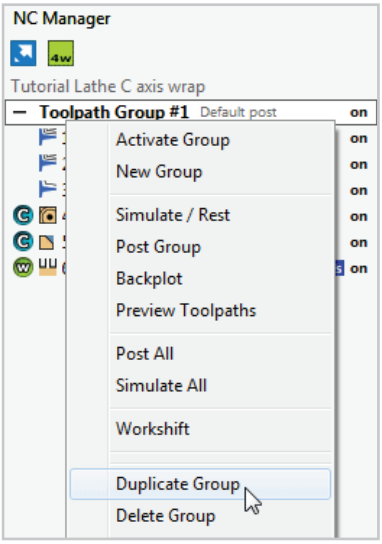
In the existing toolpath group you will see the part has C axis Face pocket and chamfer operations which have a blue C icon.

The text Mill-Turn on the XY plane is then wrap machined around the part in the Wrap operation named Engraving, which has a green W icon.



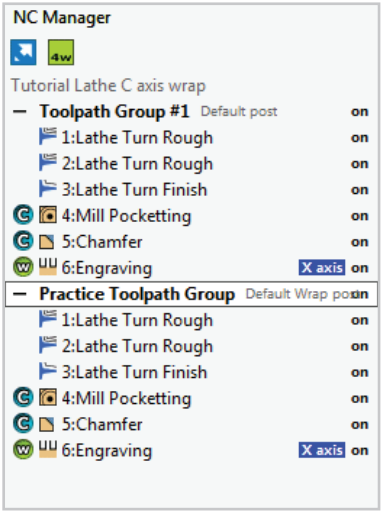
The wrap toolpath has been created using the ‘Engrave All - Constant Z’ stock toolpath. We will create a new Wrap engraving toolpath in a copy of the original Toolpath Group #1.

Right click on Toolpath Group #1 in the NC manager, and select Duplicate Group.

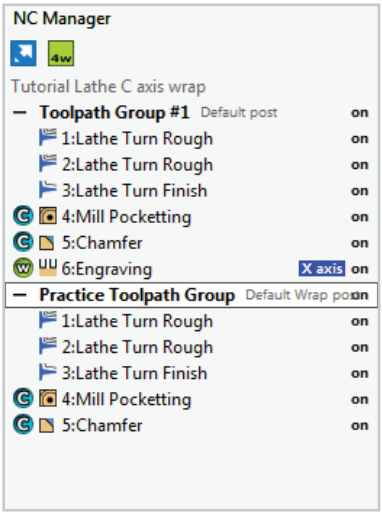


Double click on the duplicate group and rename it ‘Practice Toolpath Group’.

This group will now be the active group.



Delete the Wrap operation in the Practice group, which is the one named Engraving with a green W icon.

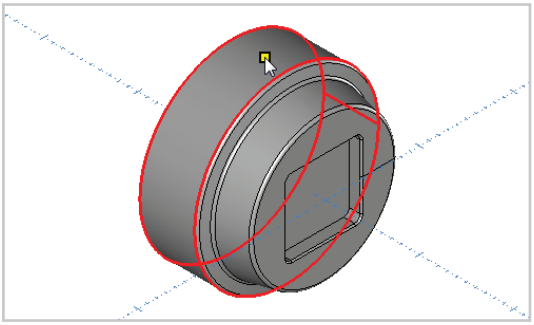


Now that we have a toolpath group for the wrap operation, we will prepare the geometry it will be based on.

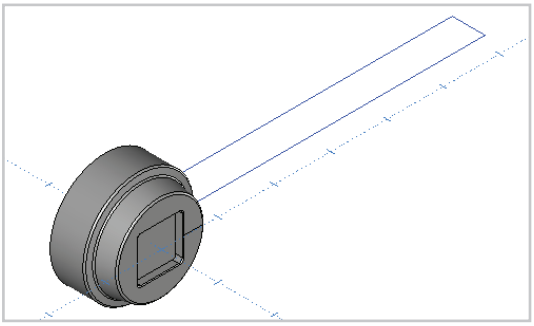
Create geometry for Wrap machining

You can use an unwrapped cylinder as a reference to place a feature at a certain position on the perimeter of a round part. We will use this method to engrave new text at the top of the part.

Click on the Model Tools icon and select the Unwrap cylinder function.



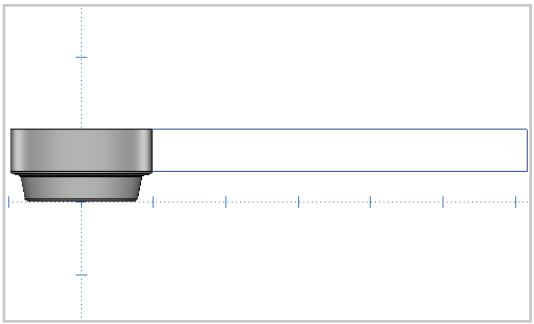
Click on the cylindrical surface which forms the outside of the part.



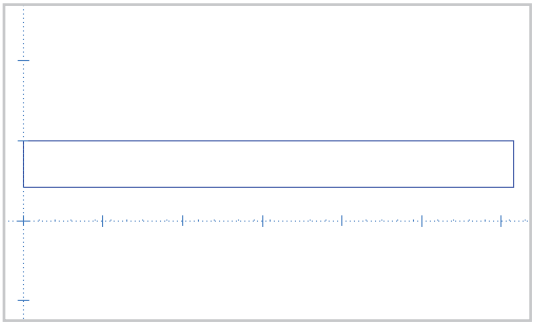
A rectangle which corresponds to the unwrapped cylinder will be drawn. The right end of the rectangle corresponds to 0 degrees and the left end corresponds to 360 degrees on the C axis.

Note:
In 4th axis wrap mode a virtual grid is shown to indicate the wrap range.

C For clarity we have selected C axis mode while preparing the geometry in this exercise.



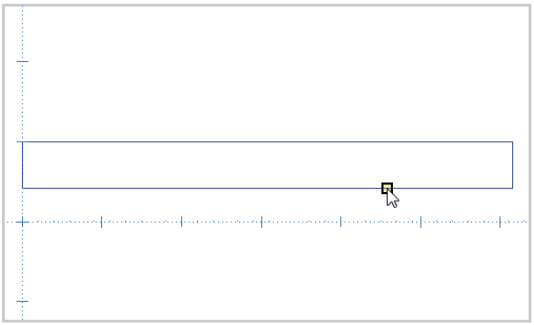
Rotate the view so you are looking down on the rectangle, with the Y axis horizontal on the screen.



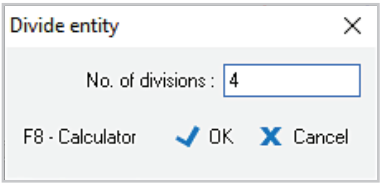
Turn off the Model layer and you will have a clear view of the rectangle.

We will now use the Divide entity function to split the circumference line into quadrant lengths.

Click on the Trim/Break icon and select the Divide entity function.

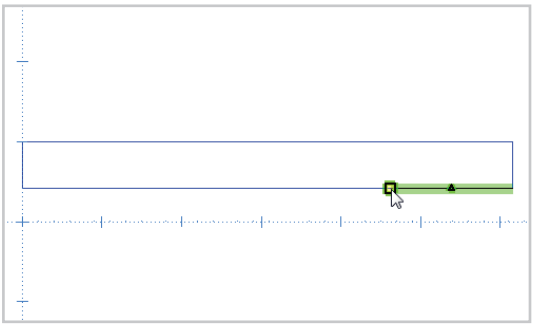


Click on one of the long lines that correspond to the circumference of the circle.

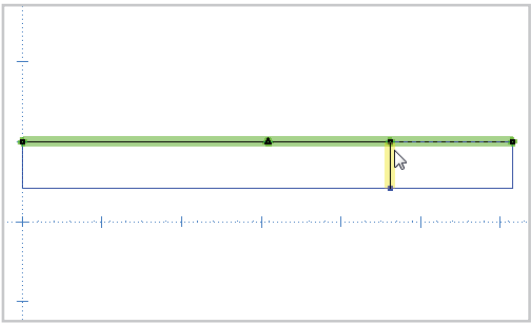


Enter 4 for the number of divisions, click OK, and right click to end the Divide function. The line will be divided into 4 equal lengths.

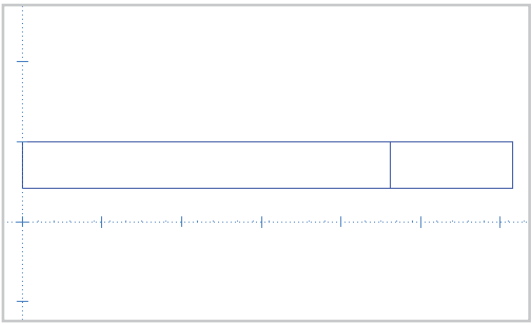
Click on the Line icon. The Line function starts automatically.



To start the line, click on the endpoint of the division on the right. This corresponds to the 90° position on the circumference.

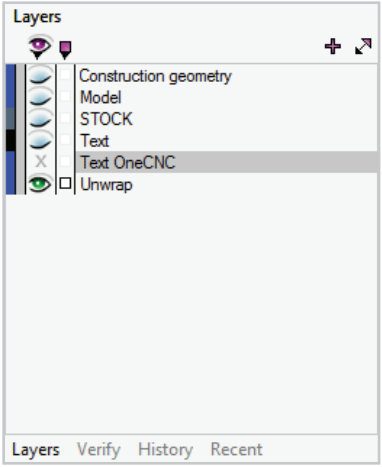


Click on the opposite line to end the new line.

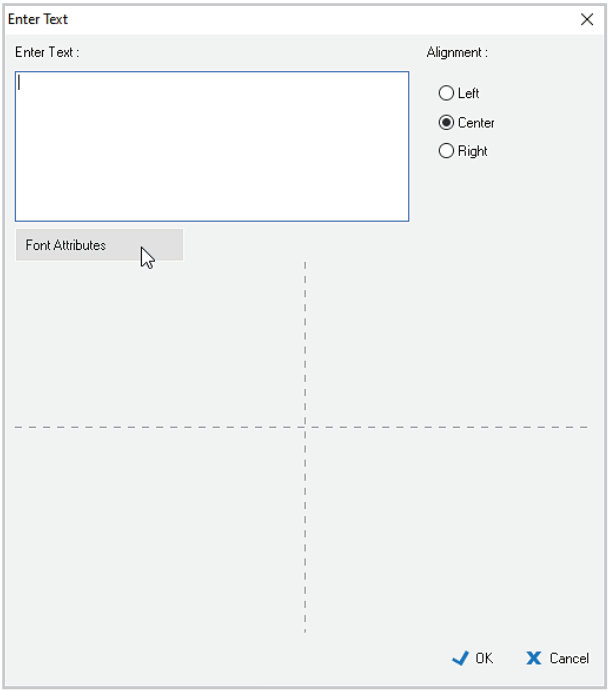


You now have a line that corresponds to 90° on the C axis.
We will use this line to position text which we will then vectorise for engraving.

Create a new layer for the text, and name it Text OneCNC.

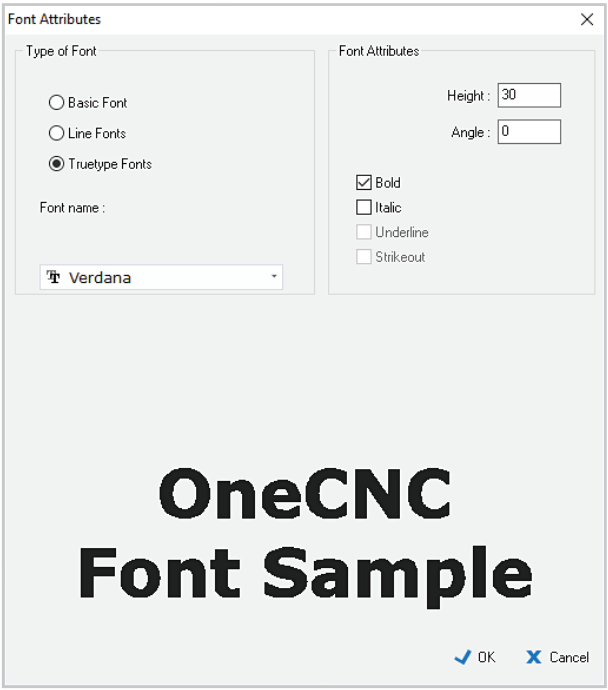


Click on the Text icon and select the Text entity function.



When the Enter Text dialog opens click on the Font Attributes button.

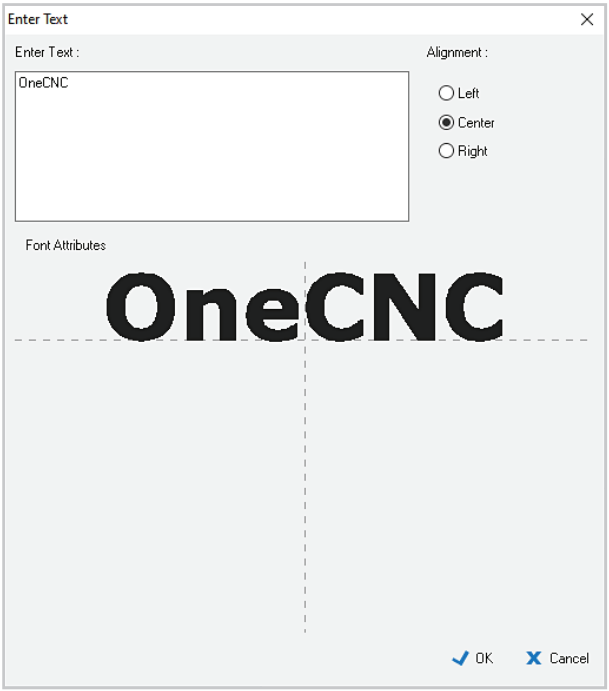
The Font Attribute dialog will open with the text settings that have been set in the OneCNC Properties dialog.



Change the selected font to the Truetype font Verdana, and select the Bold option.

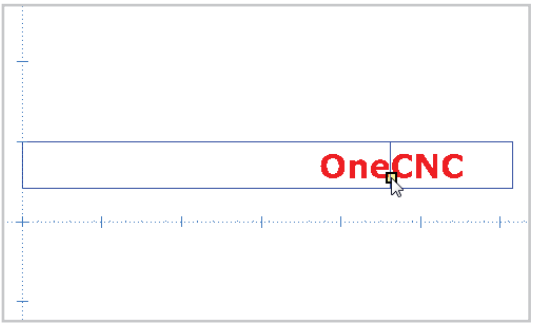
Enter a font height of 30, and change the angle to 90 so the text will be rotated to align with the Y axis.

Click OK to return to the Enter Text dialog.



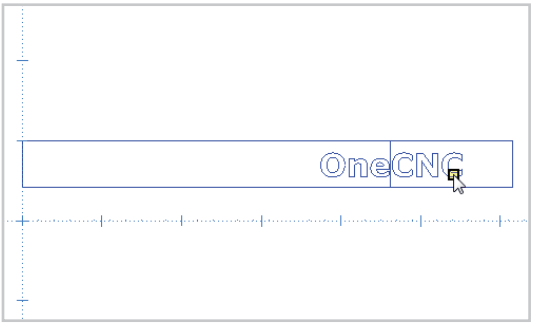
Enter the text “OneCNC” and select Center alignment.

Click OK to close the Enter text dialog.

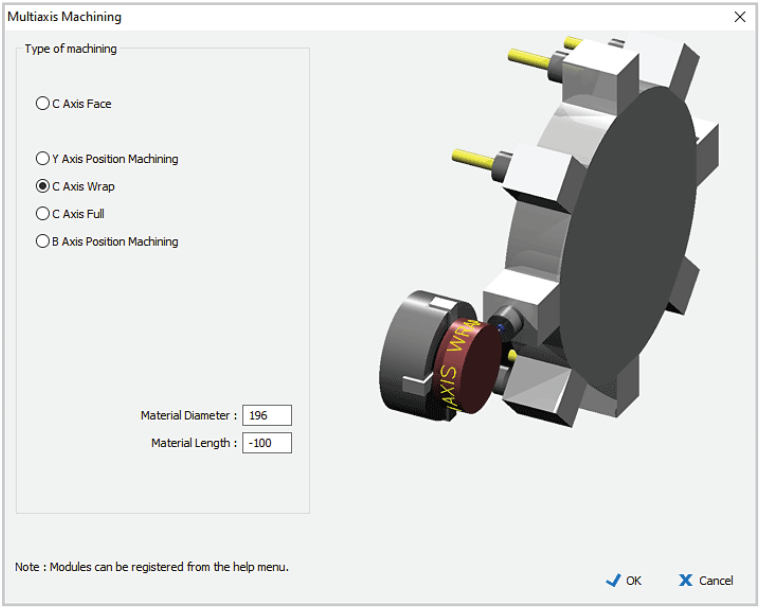


Use the quadrant line as a guide to placing the text.

Click on the Text to geometry icon in the Text tools menu.



Click on the text to convert it to geometry which can be machined.



If it is not currently active, click on the axis mode selection icon and select the C Axis Wrap option.

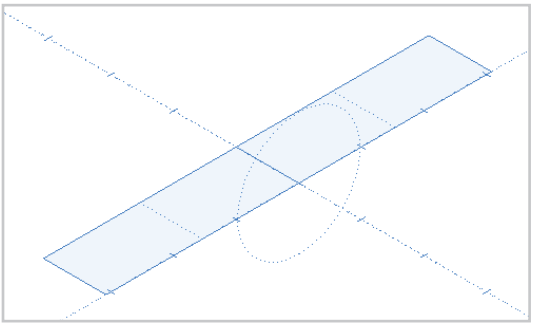
Enter 196 for material diameter, and -100 for material length.

Click OK and the axis mode selection icon will show that C Axis Wrap mode is active.

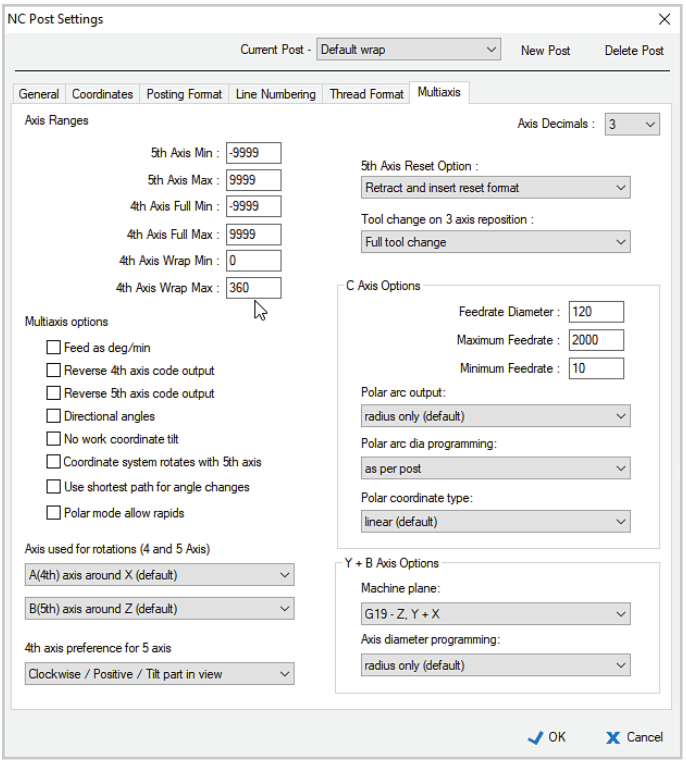
Turn off all the layers in the file, and you will see a dotted outline of the cylinder diameter. You will also see a rectangle representing the developed cylinder.

Note:

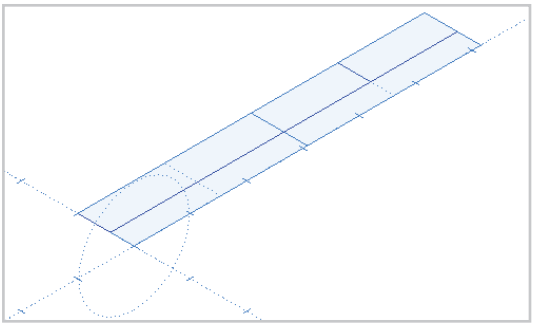
The axis of the cylinder must be aligned on the World X axis in OneCNC. If your machine has the 4th axis aligned with the Y axis, program your part on the X axis and select 4th axis around Y in the Multiaxis tab of the NC Post Settings dialog.



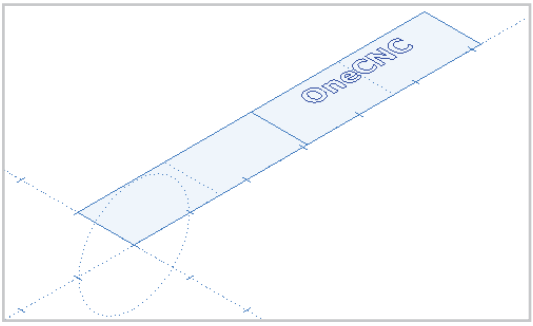
The geometry for C Axis wrap toolpaths should be kept within this rectangle, which is positioned according to the wrap range settings in the post. The default wrap range is from -180 to 180 which will put the wrap spaced evenly across the Y axis.



The unwrap cylinder function unwraps from 0 to 360, so to align the post with the unwrapped cylinder open the post settings dialog and change 4th Axis Wrap Min to 0 and 4th Axis Wrap Max to 360.

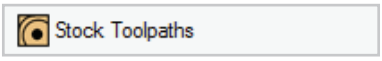


The wrap range rectangle now corresponds to our unwrapped cylinder.



Turn off all layers except for the Text OneCNC layer and you will see the vectorized text geometry is within the wrap range rectangle.

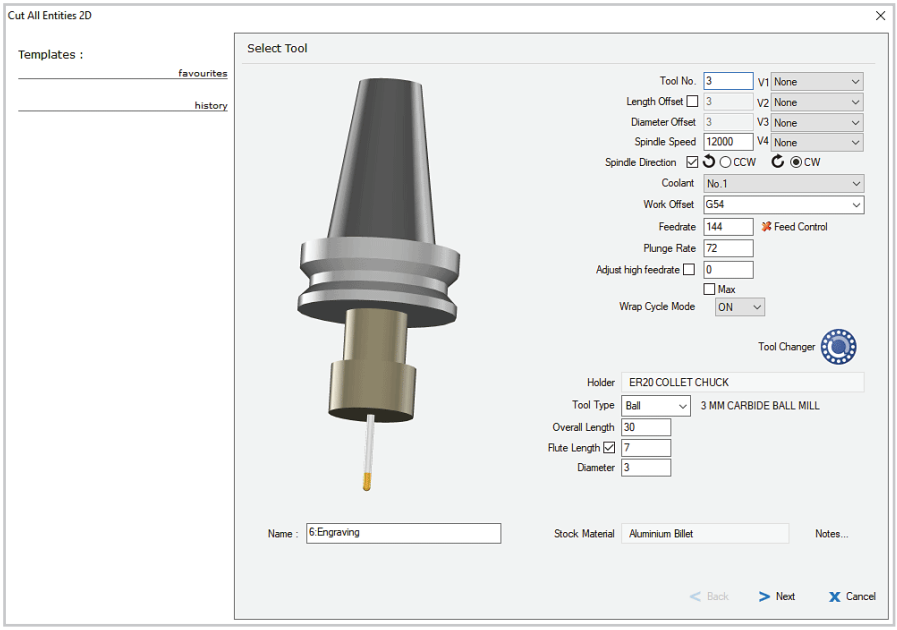
We are now ready to create a Wrap toolpath.



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

This toolpath automatically engraves all visible entities, which is why we made sure any layers with geometry not to be engraved were turned off.

All visible entities are automatically selected and the Select Tool dialog will open.



Click on the tool image in the Select Tool dialog, and select the 3mm Ball mill from the Tool List.

Click on the Stock icon and set the stock to Aluminium Billet.

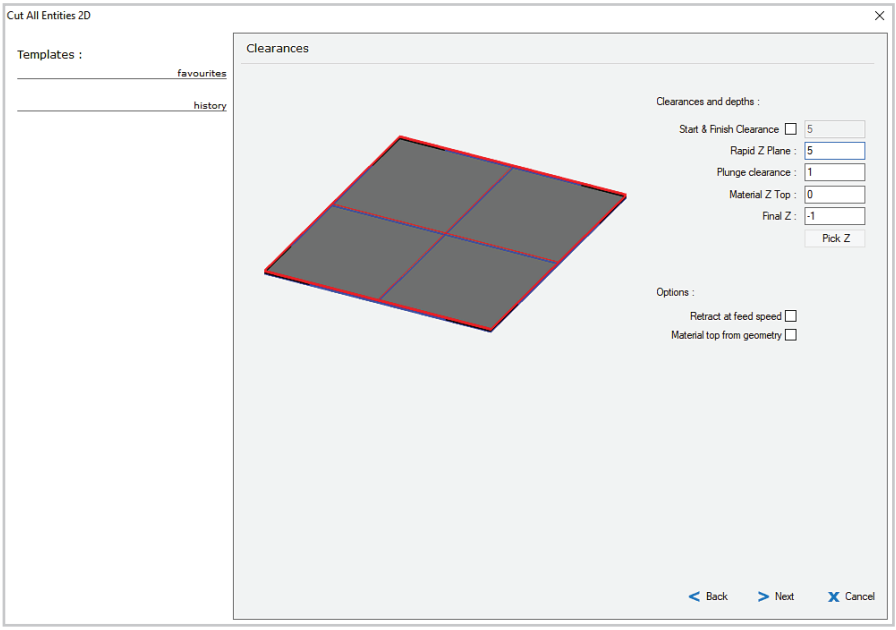
Set coolant to No1 and Work Offset to G54.

Set Wrap Cycle Mode to ON and click Next.

Note:
The wrap cycle mode selection affects how the toolpath will be posted.

When wrap cycle mode is set to ON, the toolpath will be posted using the WRAP CYCLE FORMAT.

When wrap cycle mode is set to OFF, the toolpath will be posted using the WRAP FORMAT.



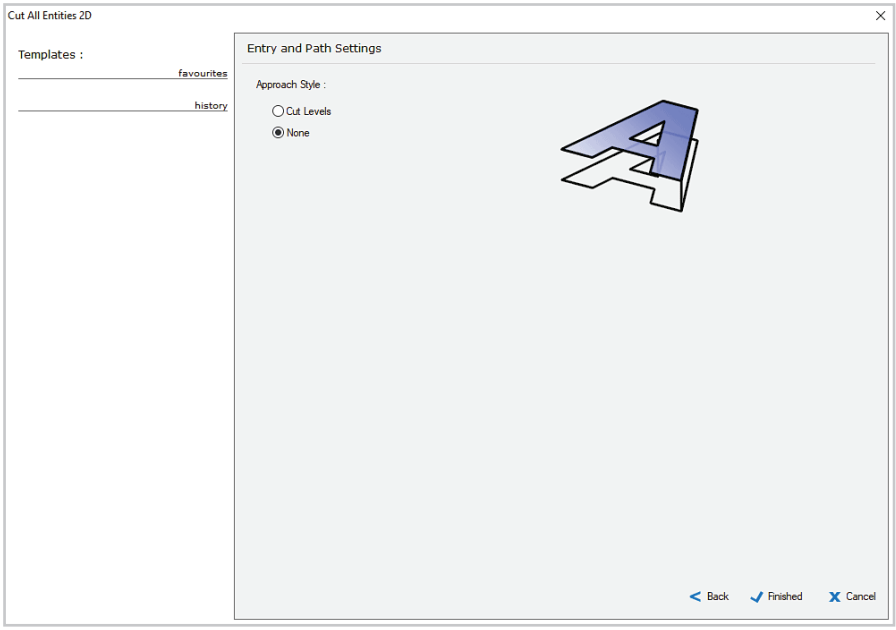
The Clearances dialog will open.

Set Rapid Z Plane to 5, and Plunge clearance to 1.

Set Material Z top to 0 and Final Z to -1. This means the cutter will cut 1mm into the part.

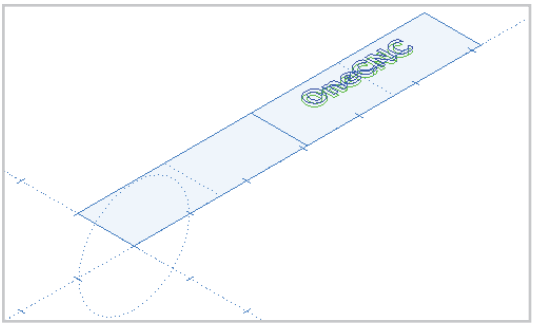
When defining a wrap toolpath, Z0 is the World Z0 on the XY plane.

When the wrap toolpath is generated, the XY plane is wrapped to the cylinder defined in the multiaxis dialog, so Z0 in the dialog will be at the surface of the cylinder when the operation is posted.



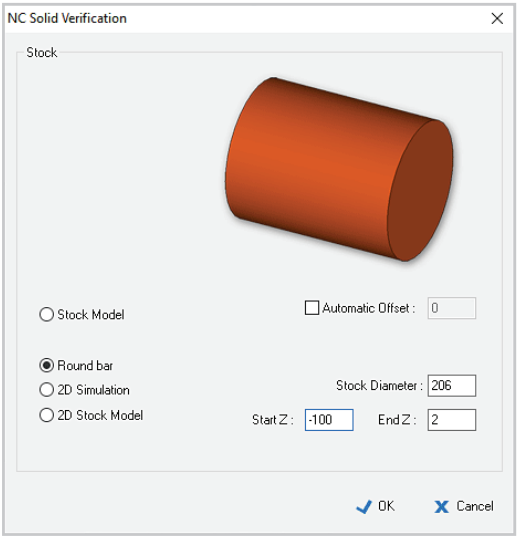
The Entry and Path Settings dialog gives you the option of cutting the geometry in more than one pass.

For this operation, select None and click Finished to create the wrap toolpath.



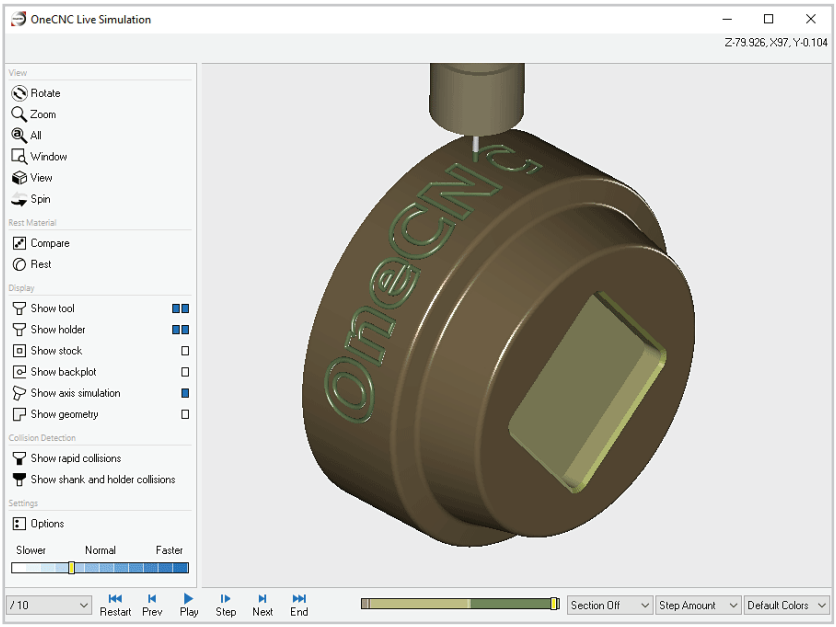
The backplot of the toolpath is drawn in the XY plane, relative to the geometry it has been created from.

To see how it is wrapped to the part when machined, we will simulate the toolpath.



Right click on the Practice Toolpath Group and select Simulate / Rest.

Define a round bar stock with a diameter of 200, Start Z of -100 and End Z of 2.



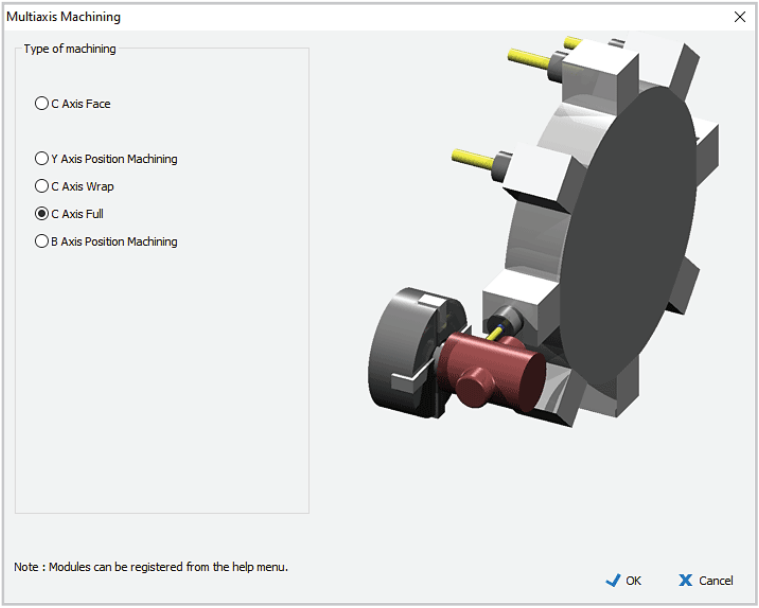
You will see how the toolpath has wrapped the vectorized text geometry around the cylinder.

The Engraving is centred on the 90° position at the top of the part.

Note:

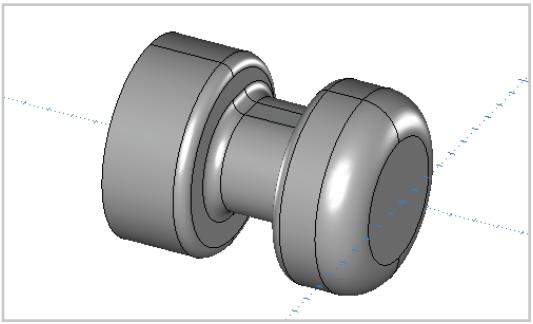
In Lathe Expert and Professional it is also possible to create model toolpaths on surfaces in the unwrap position and machine them around the circumference of the part.

OneCNC Lathe C+Y axis module
C Axis Full machining

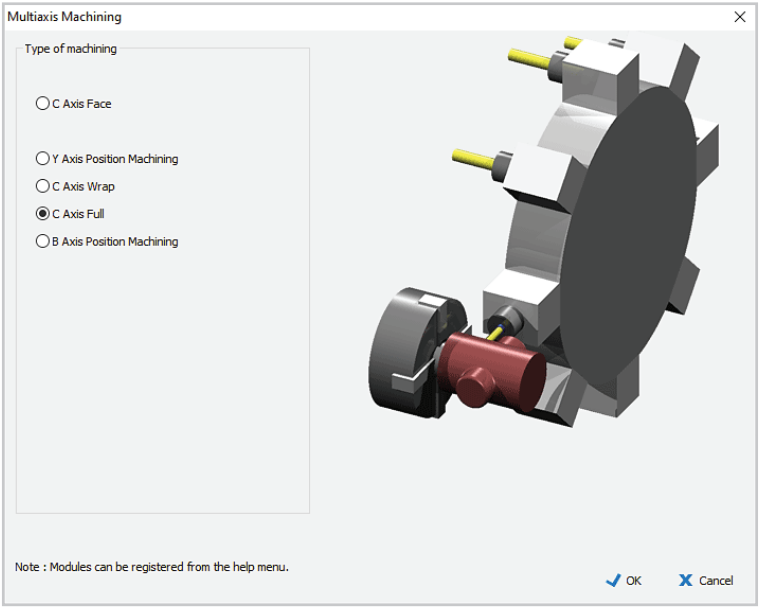


C Axis Full machining is the application of solid model toolpathing in the C axis environment.

We will define a 4 axis toolpath to create the offset section in the middle of the cam roller created in the 'Modelling - Extrusions' tutorial in the OneCNC Help Tutorials.



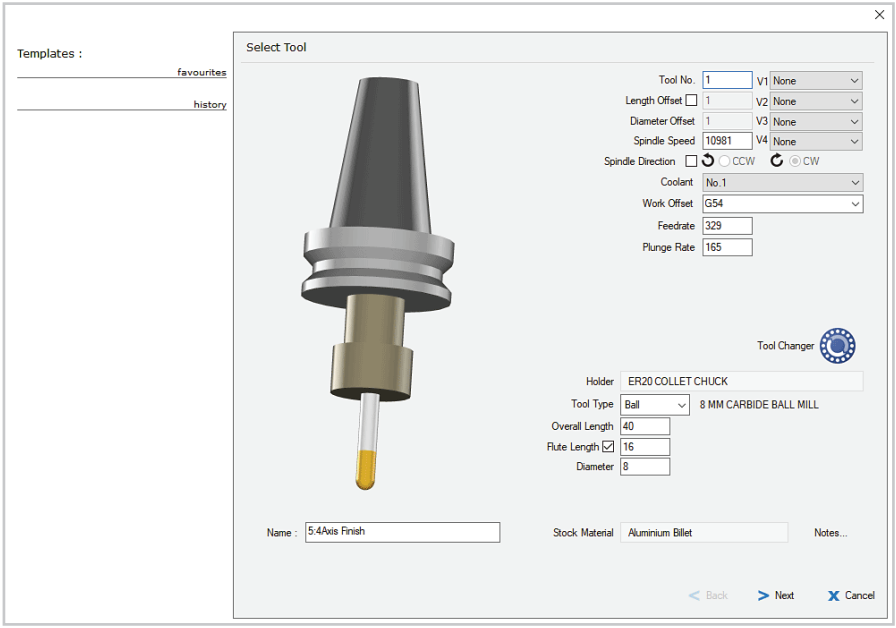
Complete the modeling by extrusion tutorial, and use Save As to save a copy of the file as ‘Lathe Tutorial C axis full.ONECNC’.



Click on the axis mode selection icon and select the C Axis Full mode.

Click on the Simultaneous Toolpaths icon and select 4 Axis Machining.

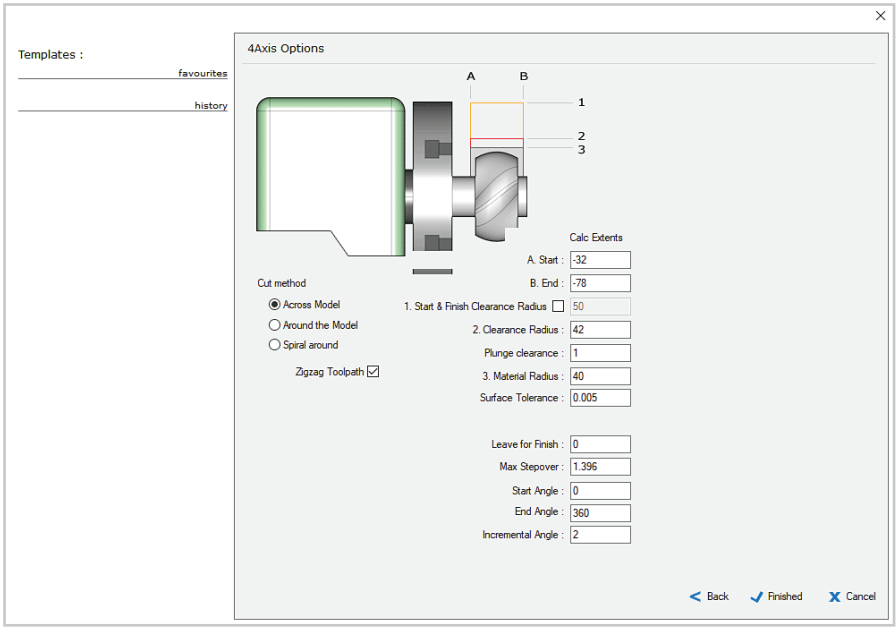
Select the 8mm carbide ball mill tool.



Enter 1 for the Tool No., set Coolant to No 1, and Work Offset to G54.

Click on the Stock icon and select Aluminium Billet from the Material List.

Click Next to continue, and you will see the 4 Axis Options dialog where all the settings which define the toolpath are made.



In the 4 axis Options dialog, select the Across Model cut method.

Select the Zigzag Toolpath check box.

Set X Start to -32 and X End to -78, to allow the cutter to fully round the fillets.

Enter a Clearance radius of 42, Material Radius of 40, and Surface Tolerance of 0.005.

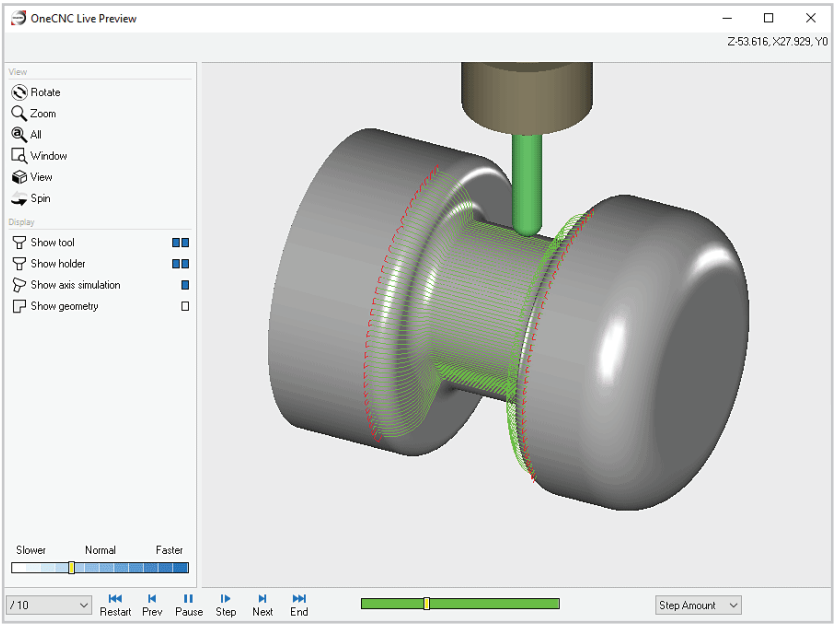
Set Leave for Finish at 0.

Enter 0 for Start Angle and 360 for End Angle.

Enter an Incremental Angle of 2, and the Maximum Stepover will update automatically, as these two settings are related.

Click Finished to create the toolpath.

Right click on the new operation and select Preview Toolpath to see the new toolpath in action.



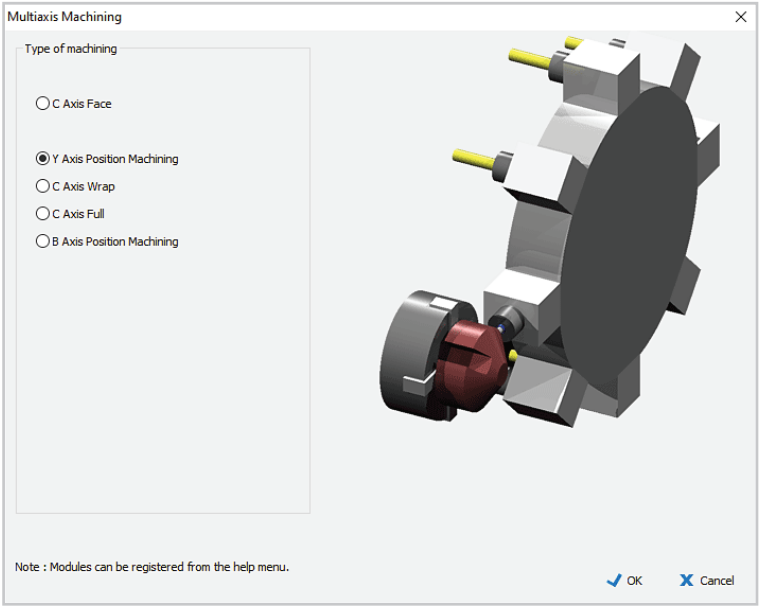
Because we selected Zigzag Toolpath the tool runs backward and forward over the part.

If we had not selected the Zigzag check box, the cut would be in the same direction each time and the tool would retract after each pass and rapid back to the next start position.

Note:

The C Axis Full toolpath is only designed for finishing. There is no roughing equivalent as this can be done more efficiently with Z Level or Planar Roughing in Y Axis Positions.

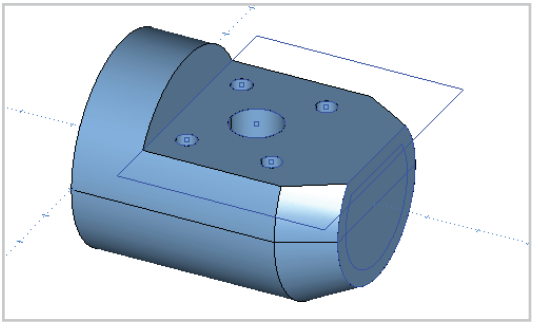
Y Axis Position machining



4P Y axis position machining applies milling toolpaths at a position defined by a plane which has one axis parallel to the lathe axis.

A part can have more than one plane position, but for parts with machining on only one plane, it is simplest to use a plane parallel to the default XY plane.

To see how a Y axis position toolpath is created, we will open a sample file, then re-create an existing Y axis toolpath.

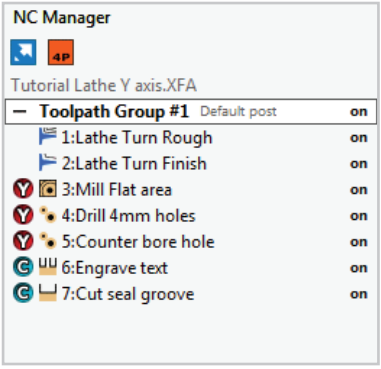


Open the sample file 'Y Axis Mill Turn.ONECNC'.
Use Save As to save a copy of the file as 'Tutorial Lathe Y axis.ONECNC'.

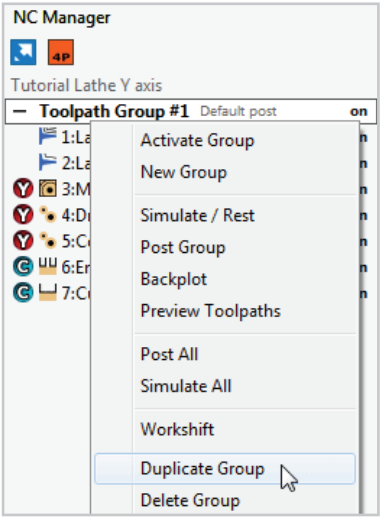
Set view to Trimetric, and turn on all layers so you can see the construction geometry.

In the NC manager operations list you will see an example toolpath group, which has three Y axis position toolpaths marked with a red Y symbol.

We will recreate a Y axis toolpath in a copy of the example group.

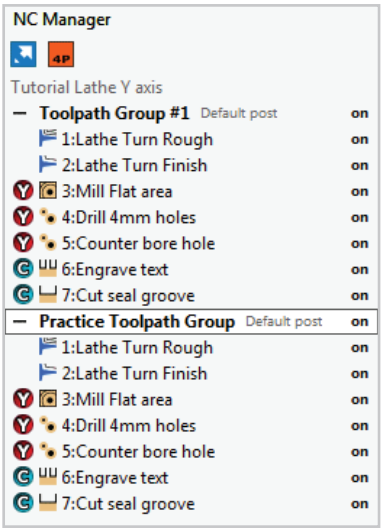


Right click on the Toolpath Group #1 heading in the NC manager, and select Duplicate Group.



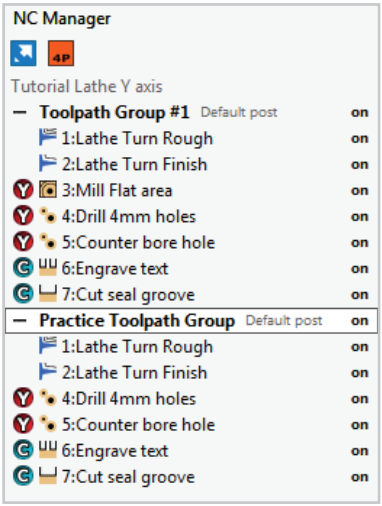
Double click on the duplicate group and rename it ‘Practice Toolpath Group’.

This group will now be the active group.

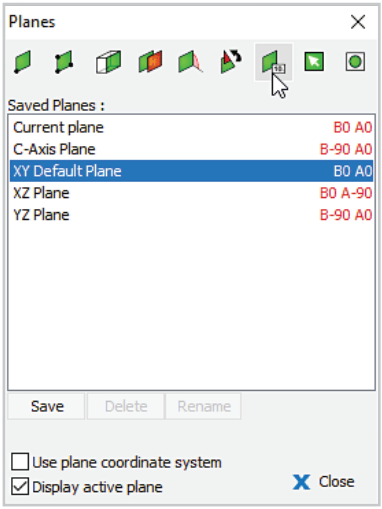


Delete the Y axis operation ‘3: Mill flat area’ in your practice group.

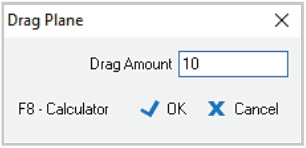
Your NC Manager tab should now appear as shown.



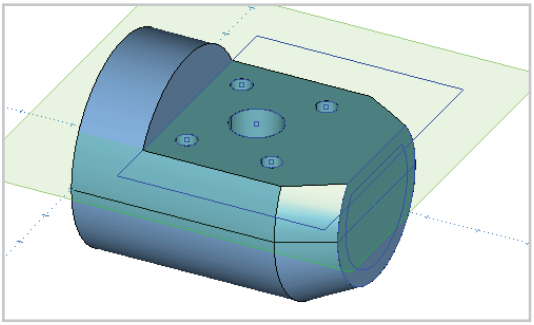
Click on the Plane icon to open the Planes dialog, and select the check box for Display active plane.



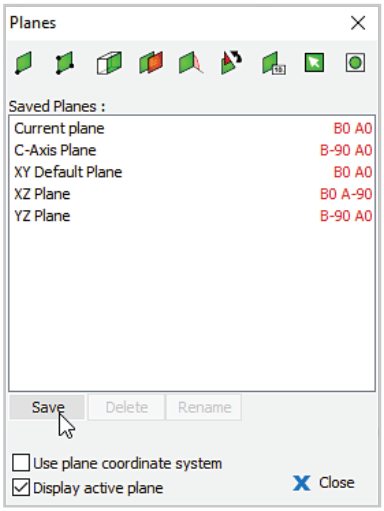
Make sure the XY default plane is selected, and select the Push or Pull Plane command.



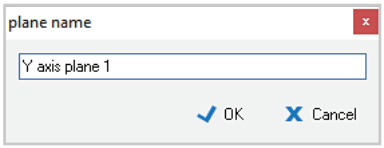
Enter a value of 10 to move the plane 10mm in Z.



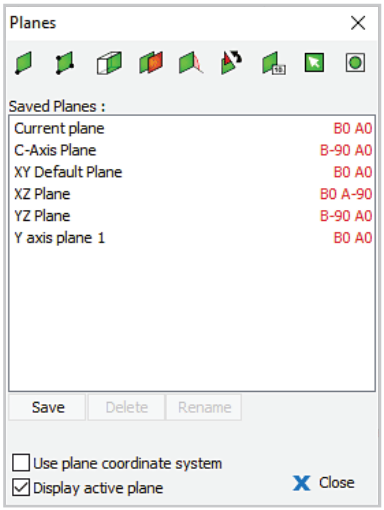
The plane will be moved to the level of the flat on the part.



Open the Planes dialog again and click Save to save the plane for future reference.



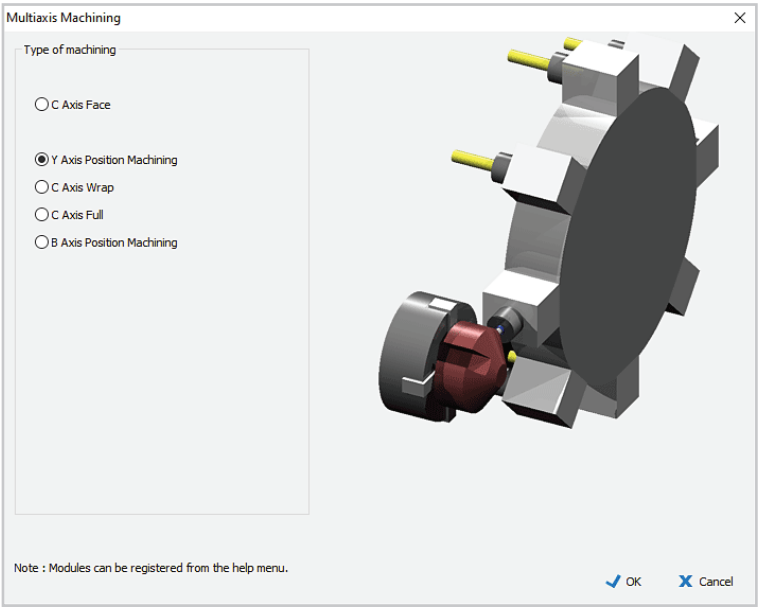
Enter a name for the plane and click OK.



In this example we are only using one plane but if a part requires multiple Y axis positions it is good practice to save all the planes you need at the outset.

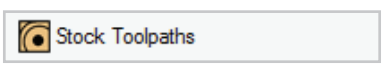
B0 A0 to the right of the plane name refers to the planes orientation. The A angle defines rotation about the world X (lathe Z) axis. The B angle defines rotation about the world Y (lathe X) axis. For Y axis position machining, the plane can have any A angle, but must have a B angle of 0.

Define a Y axis toolpath

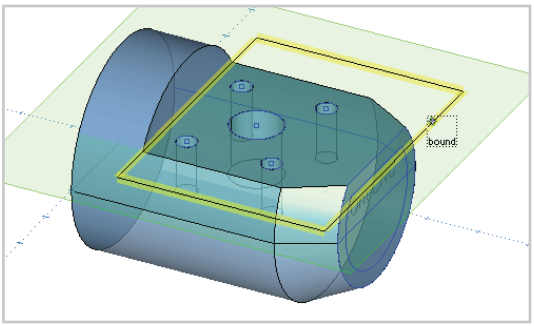


Click on the multi-axis selection icon and select Y axis position machining.

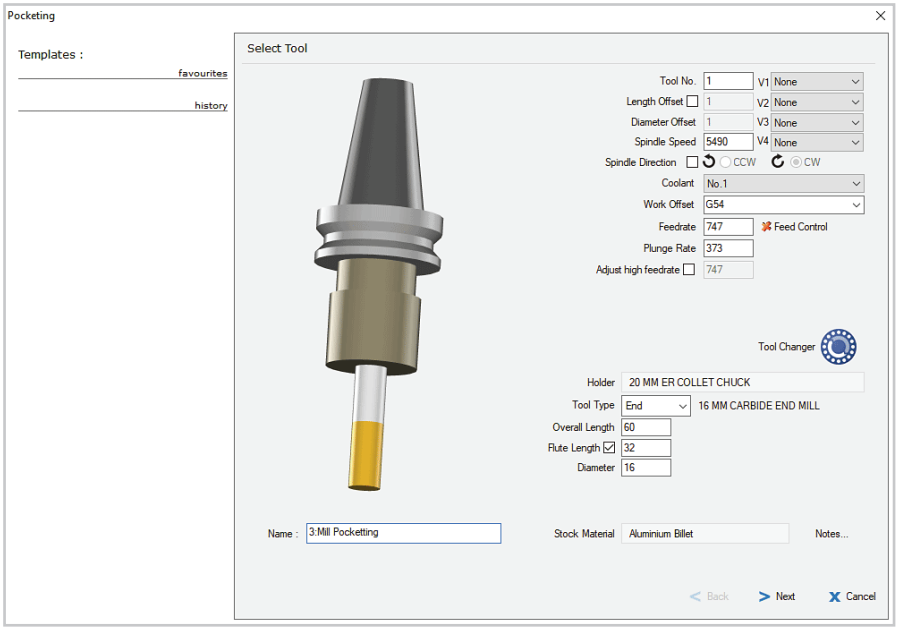
The Y axis position mode icon will be displayed in the NC Manager.



Click on the Stock Toolpaths icon in the toolbox, and select HS Pocketing.



Pick the rectangular boundary as shown. Right click to end the selection process.



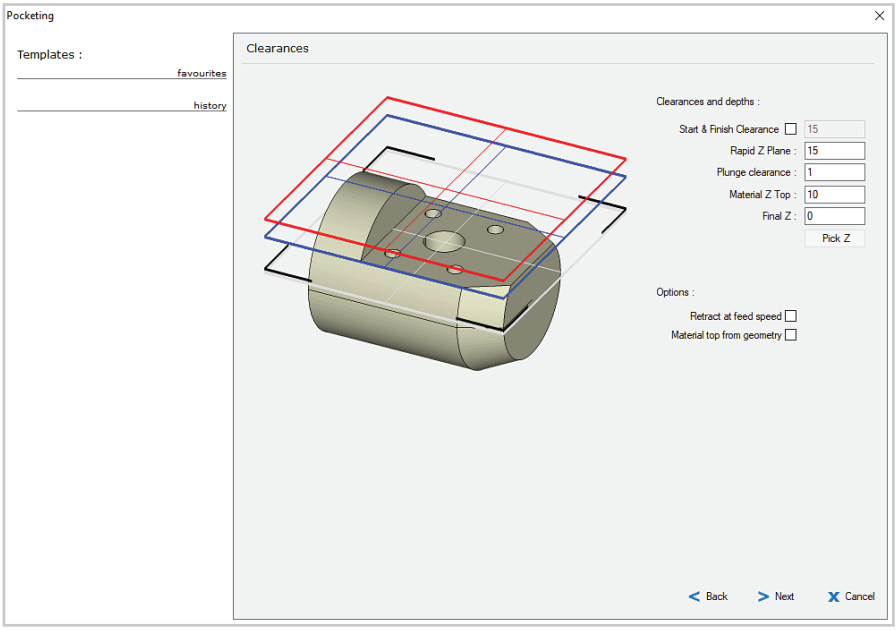
The Mill Select Tool dialog appears.

Open the Tool Changer and select the 16mm carbide end mill. Click Accept to return to the toolpath wizard.

Click on the Stock name in the lower right corner and select Aluminium Billet from the Material List.

Set Coolant to No 1 and Work Offset to G54.

Clear the box for Adjust high feedrate, then click Next to continue.

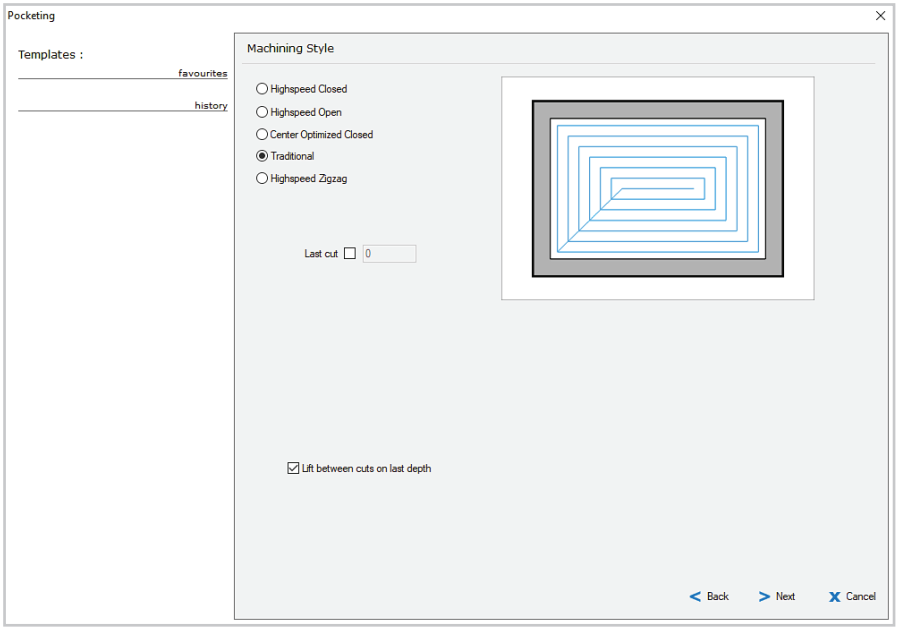


Clearances are defined from the current plane.

Set Rapid Z plane to 15 and Plunge clearance to 1

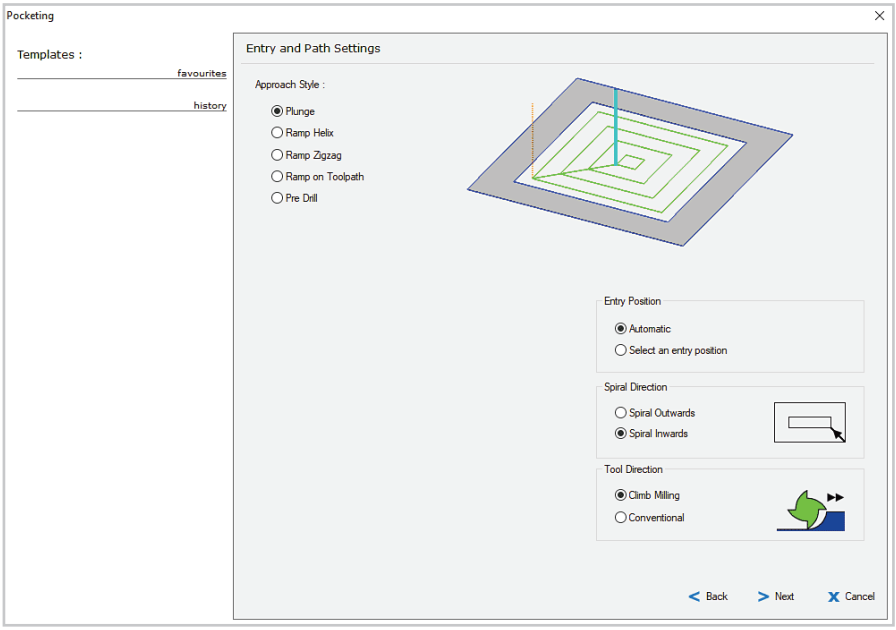
Set Material Z top to 10.

Set Final Z to 0, and click Next to continue.



Set the Machining Style to Traditional, which is a simple closed pocket method.

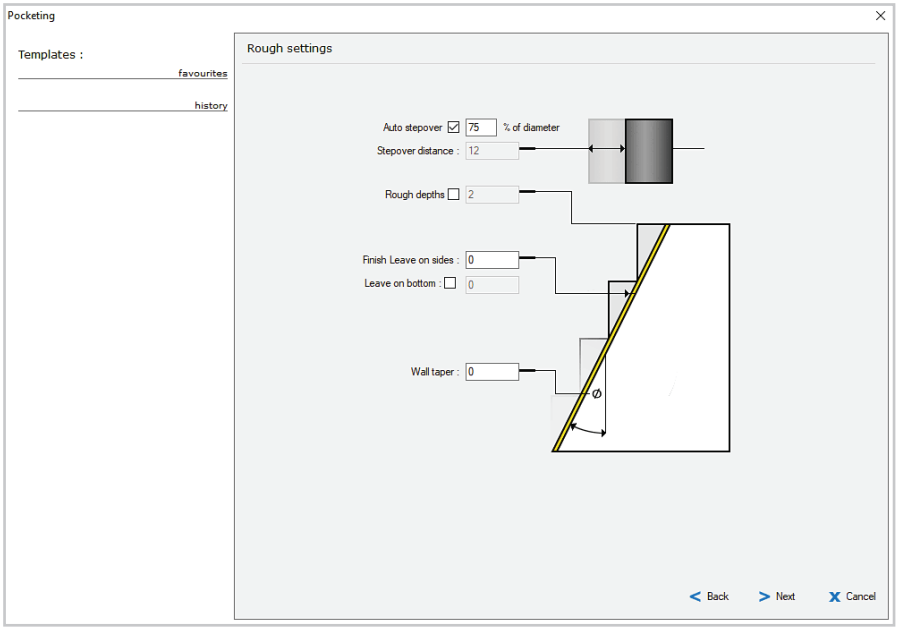
Click Next to continue.



For a Traditional pocketing toolpath you have to set more options than for high speed machining.

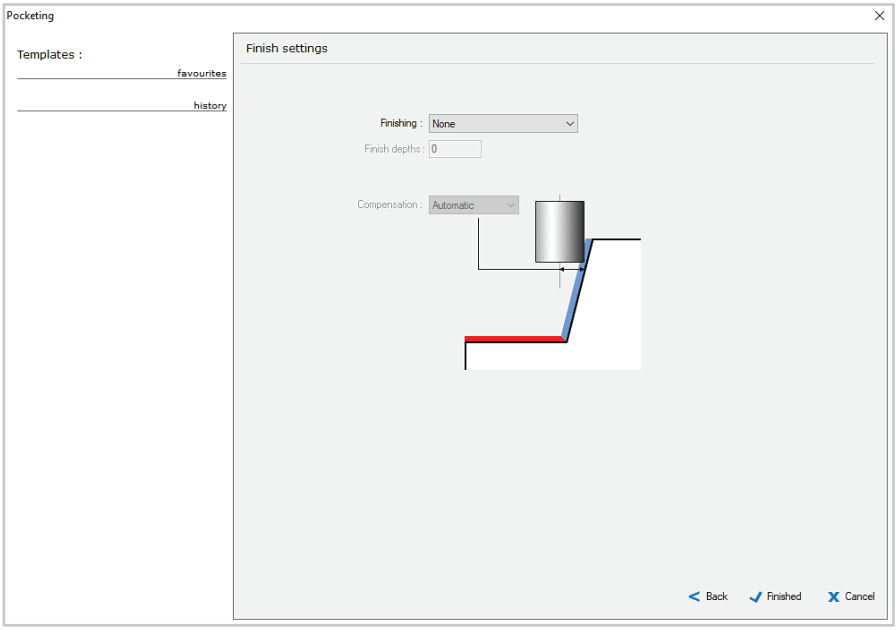
Select Plunge entry, Automatic entry position, Spiral Inwards, and Climb milling.

Click Next when you are ready to continue.



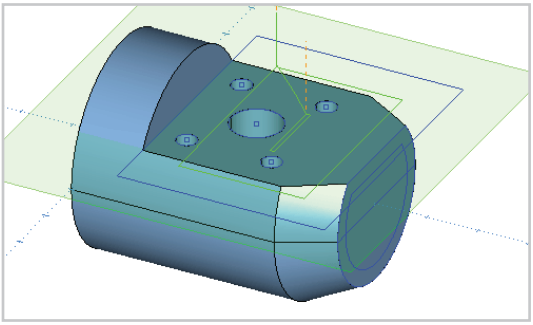
Rough settings allow you to cut the pocket in multiple passes.

For this operation set Auto stepover at 75%, clear the Rough Depths check box, and set Leave on sides and Wall taper at 0.



Finish settings give you the option of creating final finish passes.

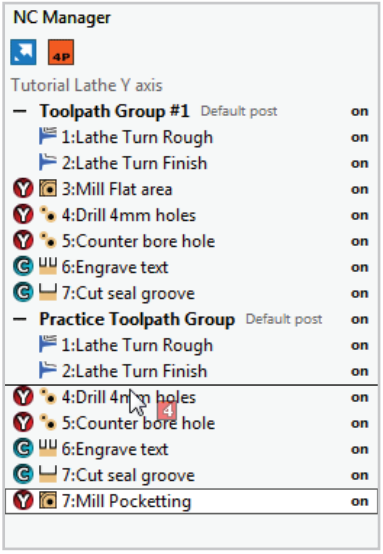
For this operation leave Finishing set to None, and click Finished.



The toolpath is calculated, and you will see a preview of the toolpath in the drawing window.

In the NC manager operations list, the pocketing toolpath will show as a new listing at the end of the group.

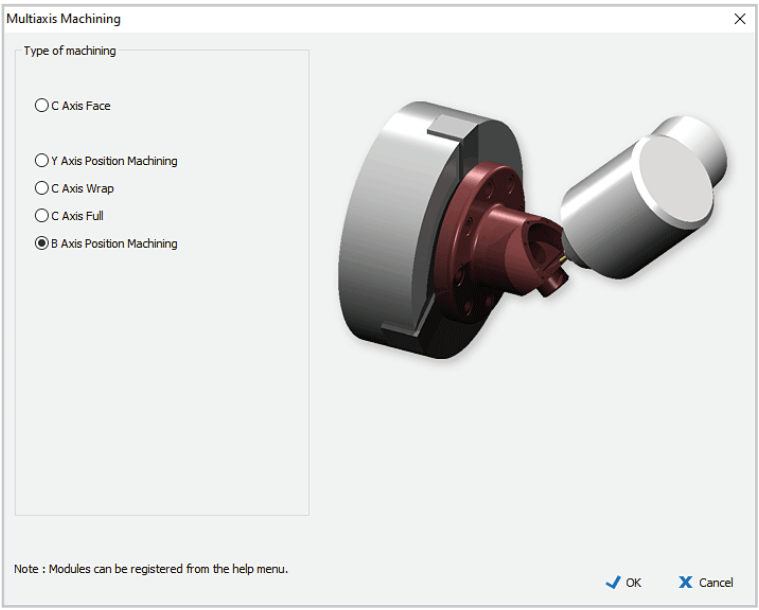
Hold the mouse pointer down on the toolpath and drag it to its correct position.



The Practice Toolpath Group is now complete.

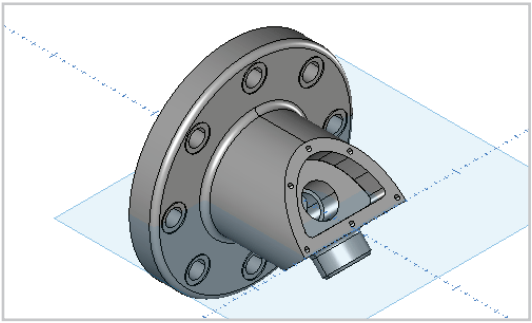
OneCNC Lathe C+Y+B axis module

B Axis Position machining



5P B axis position machining applies milling toolpaths to geometry on a plane rotated about the machines Y axis.

As the part can be rotated on the lathe axis, the plane can be at any orientation in OneCNC. Complete 'CAD Tutorial 4 - Working with planes' before commencing this tutorial.

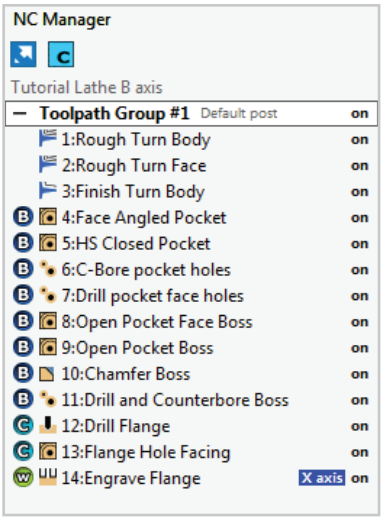


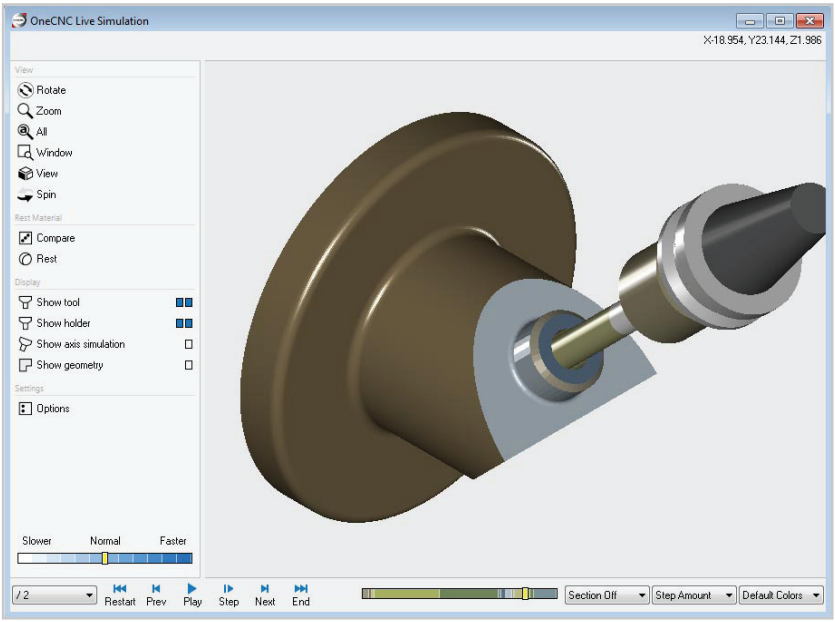
We will work with a copy of this sample part file in this tutorial. Open the sample file 'B_AXIS.ONECNC', and use Save As to save a copy as 'Tutorial Lathe B axis.ONECNC'.

In the NC manager there is an example toolpath group.

The B axis operations are those marked with a dark blue B symbol.

This part also has C axis face and wrap operations, which are explained in their relevant sections.



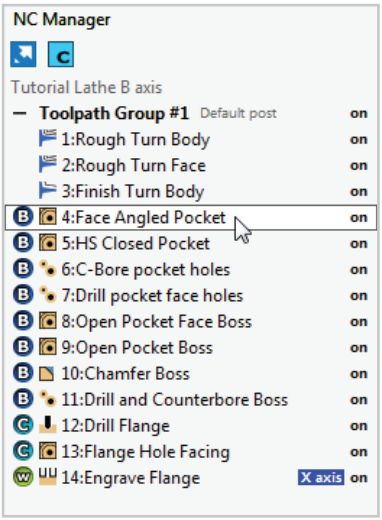


Simulate Toolpath Group #1 to see how the toolpaths are carried out, with the part rotating between plane positions.

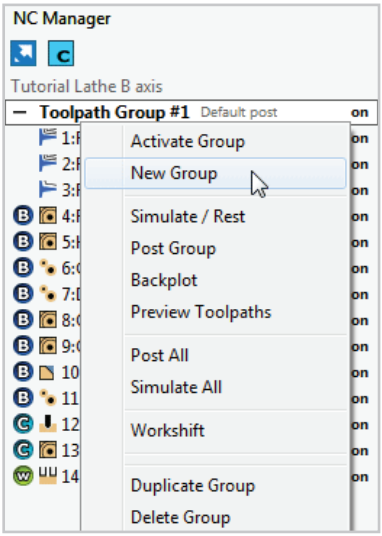
Close the simulation when you are ready to continue.

The first B axis operation in the example toolpath group is a high speed pocketing operation which creates the angled face.

We will recreate this B axis toolpath in a new toolpath group.



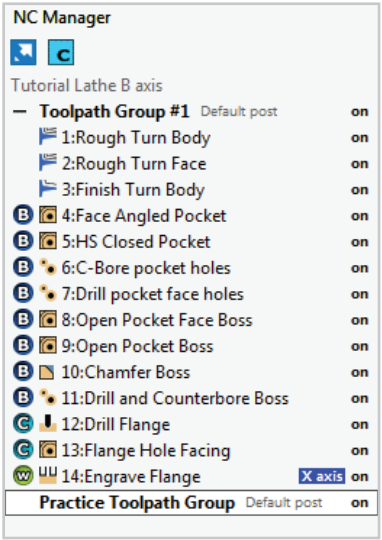
Right click in the NC manager, and select New Group.

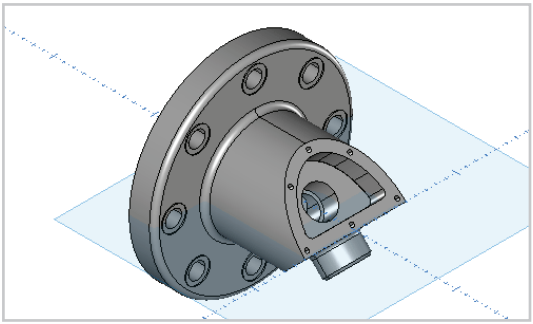


Double click on the new group and rename it 'Practice Toolpath Group'.

This group will now be the active group.

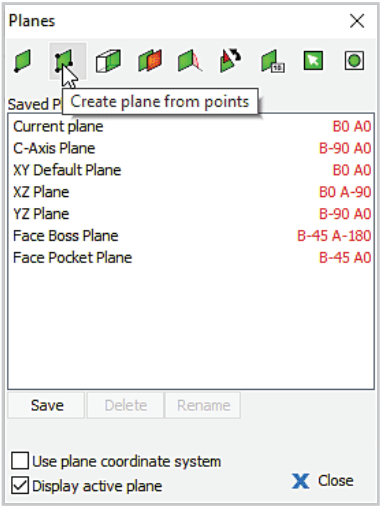
Your NC Manager tab should now appear as shown.



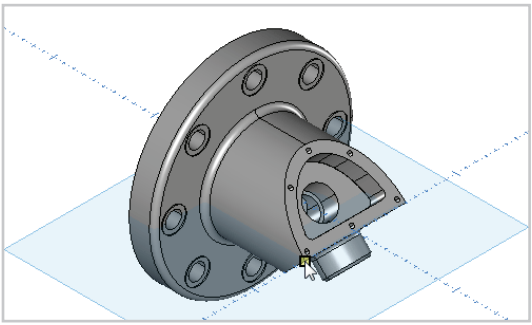


Set the view to Isometric, so that the angled pocket is visible.

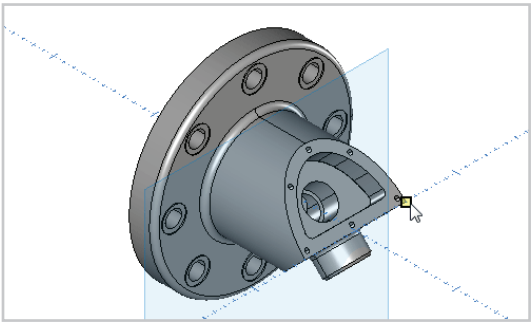
We will create and save a plane on the semi-elliptical surface at the top of the pocket.



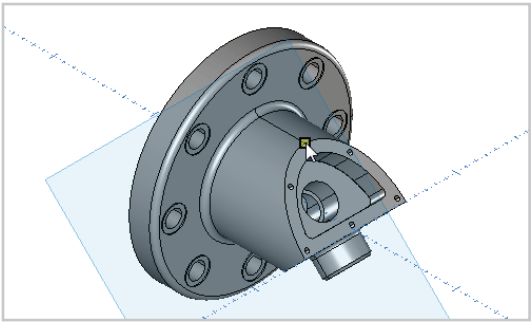
Open the Planes dialog and click on the Plane from points icon.



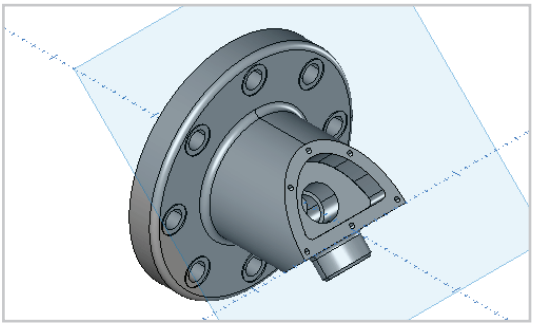
Click on the lower left corner of the angled face to select it as the origin of the new plane.



Click on the lower right corner of the angled face to select it as the X axis direction in the new plane.

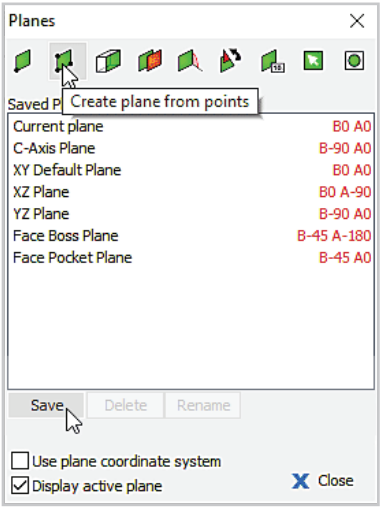


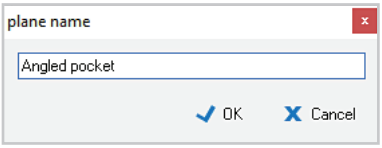
Click on the endpoint at the top of the angled face to define the Y axis direction in the new plane.



The plane has now been defined.

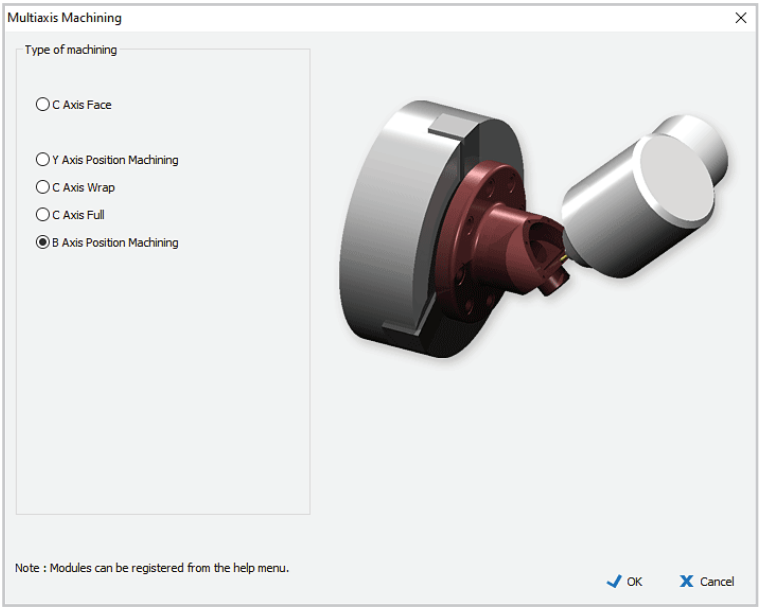
Click Save in the Plane dialog.





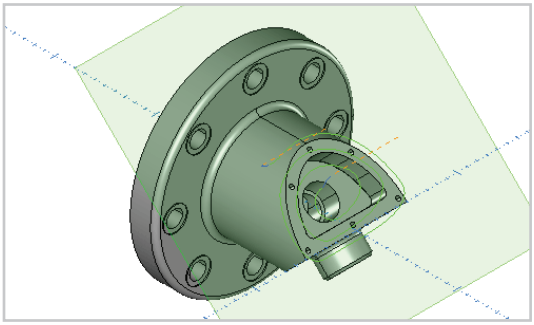
Name the plane Angled pocket.

We can now return to this plane at any time by selecting it in the Planes dialog.



Click on the multi-axis selection icon and select B axis position machining.

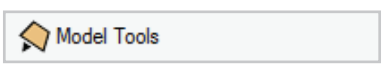
The B axis position mode icon will be displayed in the NC Manager, and the plane will change colour.



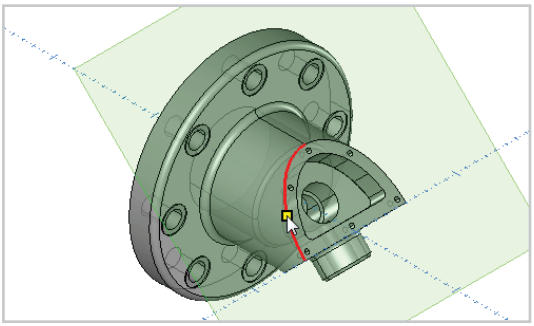
Right click on the operation named 4: Face Angled Pocket, and select Backplot. You should now see the toolpath as it appears here.

Click Undo to clear the backplot from the screen.

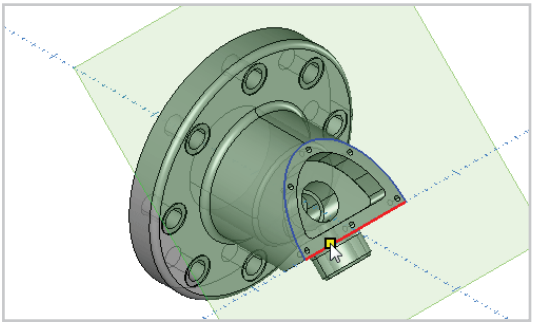
The operation is a high speed open pocket toolpath based on the outer boundary of the flat surface it mills to.



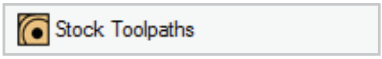
Open the Model Tools menu in the Toolbox and select Extract an edge.



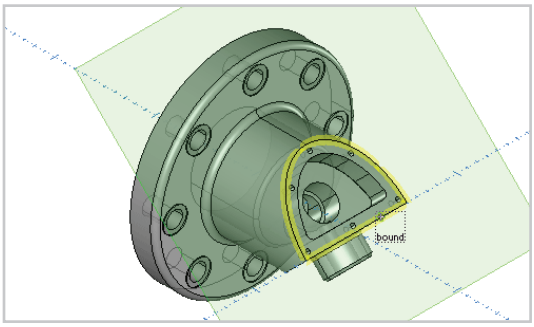
Click on the curved edge of the surface we want to machine to create the curved part of the boundary.



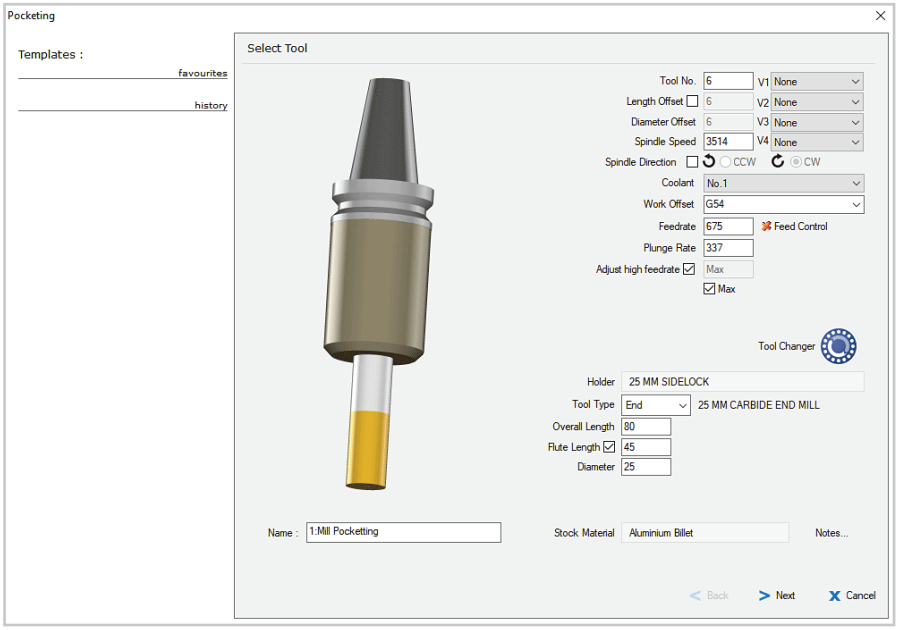
Click on the straight edge of the surface to complete the boundary. You can rotate the view to be sure of selecting the straight edge and not the surface below it.



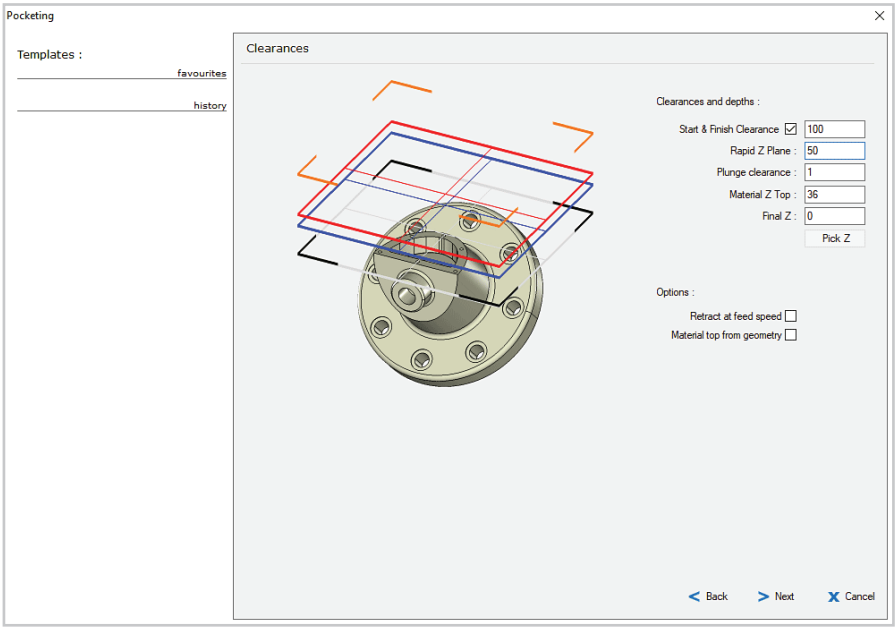
Open the Toolbox Stock Toolpaths menu and select HS Pocketing.



Select the new boundary, and right click to end the selection stage.



Select the 25mm carbide end mill from the Tool Library, and enter the settings for it as shown.



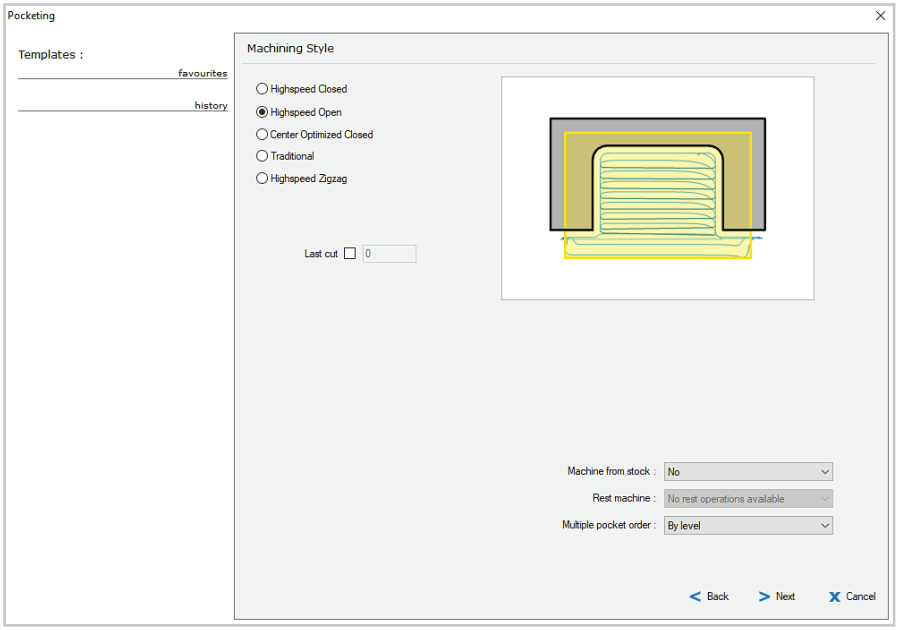
Clearances are defined from the current plane.

Select the Start and Finish Clearance check box and enter a value of 100. This will keep the tool away from the part while it is indexing.

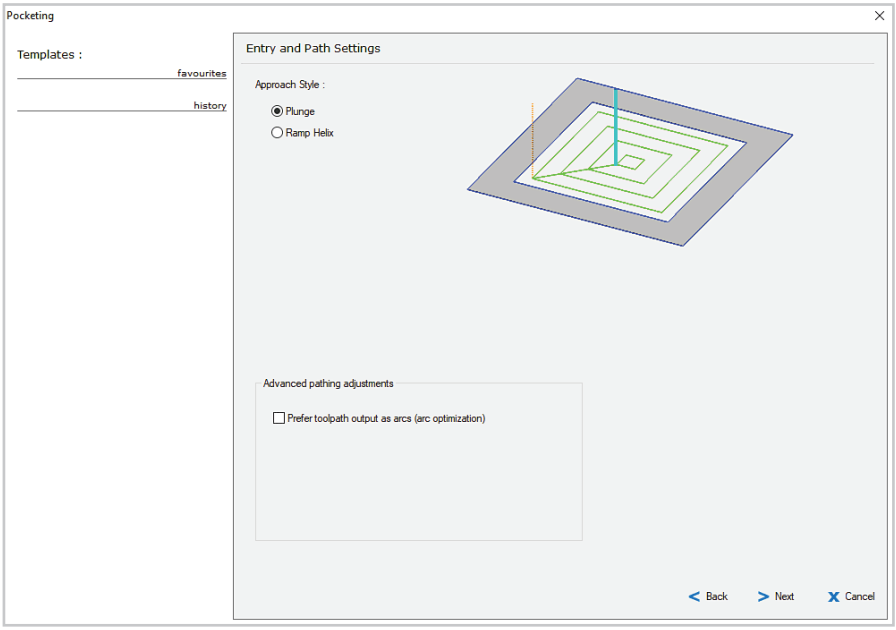
Set Rapid Z plane to 50 and Plunge clearance to 1.

Set Material Z top to 36.

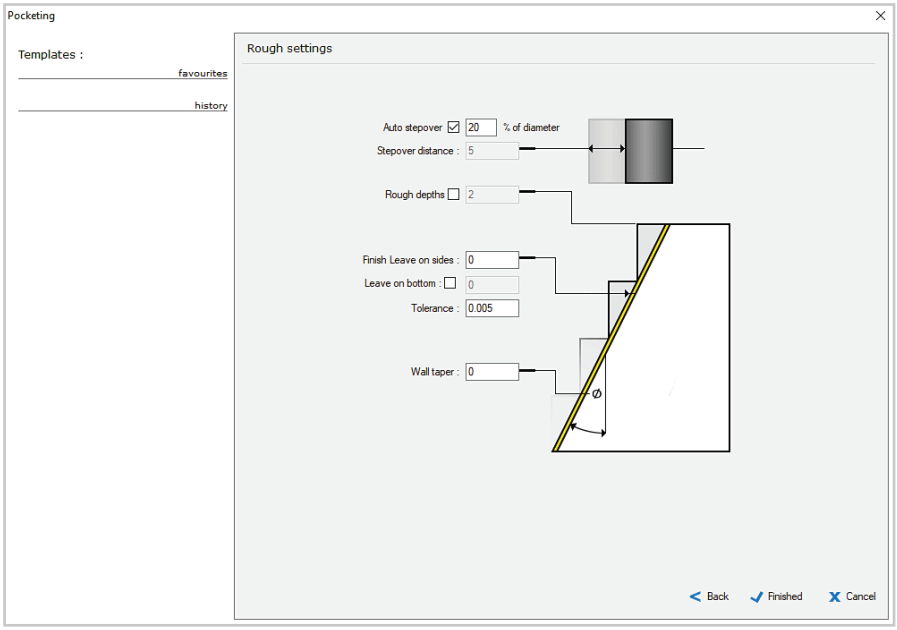
Set Final Z to 0, and click Next to continue.



Select the Highspeed Open strategy.



In the Enry and Path Settngs dialog select Plunge entry.

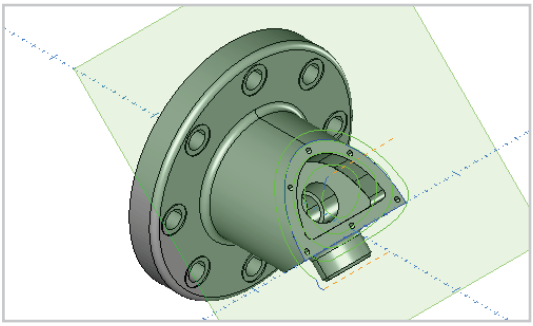


The Rough Settings dialog controls the action of the toolpath.

Highspeed machining is designed to use a greater depth of cut and reduced stepover. Stepover of up to 30% of the tool diameter with 2 x tool diameter depth of cut will create an efficient toolpath which uses more of the flank of the tool.

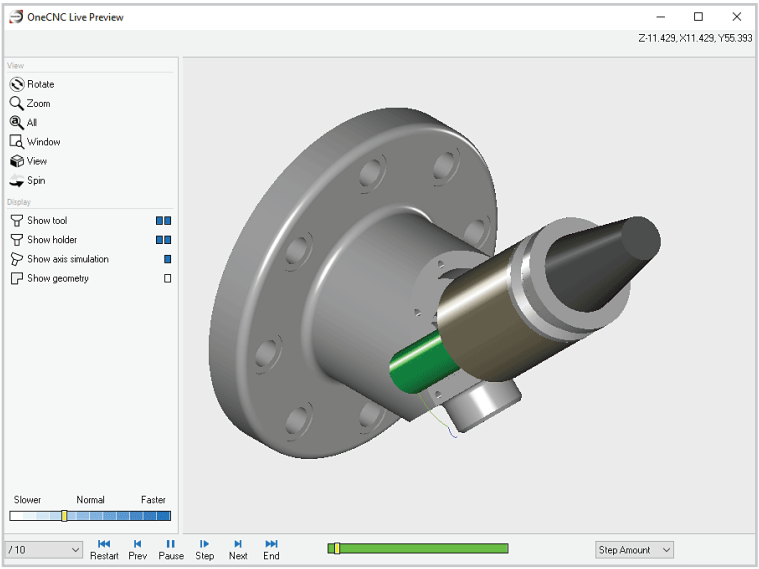
For this toolpath, change the Auto stepover to 20%, and clear the Rough depths check box. Enter 0 for Leave on sides.

Click Finished to create the toolpath.



Backplot the new operation to see the path the tool will take to machine the flat.

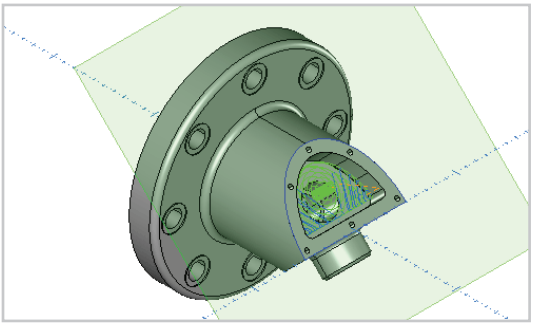
Click Undo to clear the backplot from the screen.



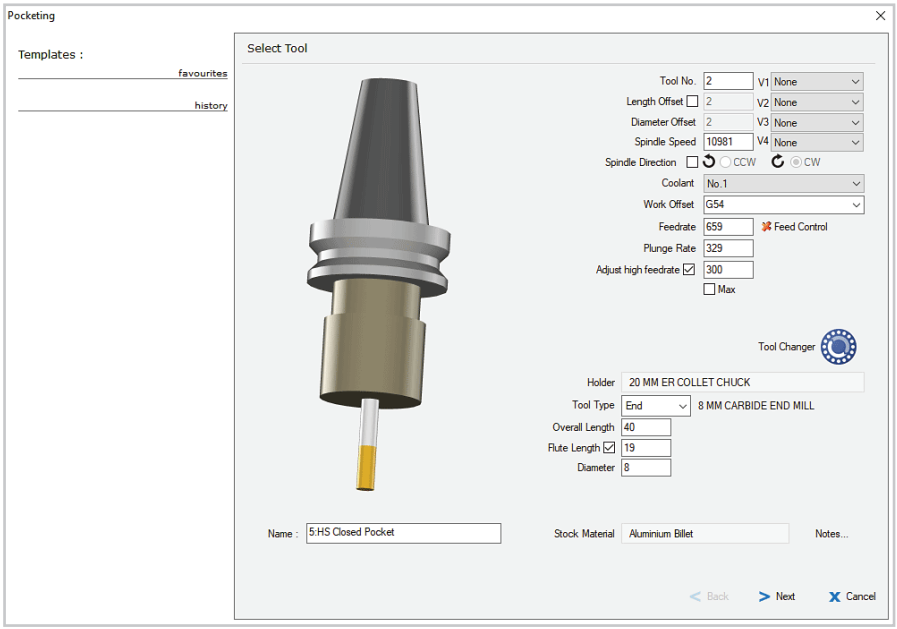
A preview of the toolpath shows it is safe to use plunge entry in this case because the open high speed pocketing strategy always plunges the tool outside the pocket boundary.

Recreate sample toolpaths in the first position

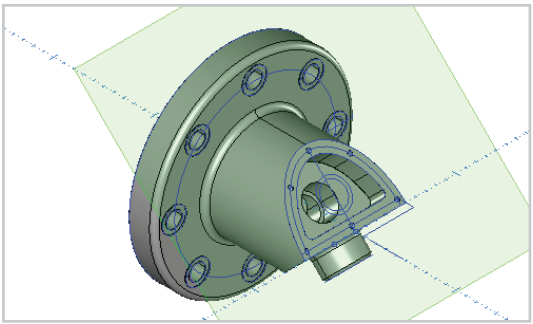
To complete the B Axis operations for this plane position, re-create the example toolpaths in the original toolpath group.



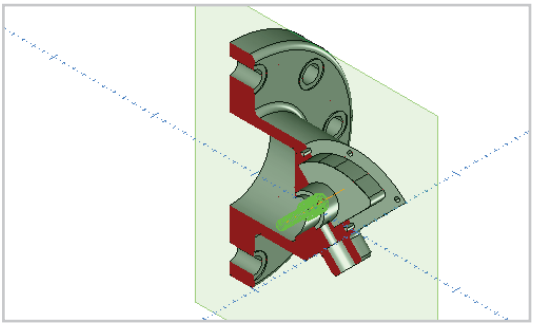
Click on the operation named 5:HS Closed Pocket in the original group and observe the preview in the drawing window. This toolpath is a Closed highspeed pocket operation based on wireframe geometry.



The toolpath can be reproduced easily by opening the original toolpath for editing and saving the settings as a template.

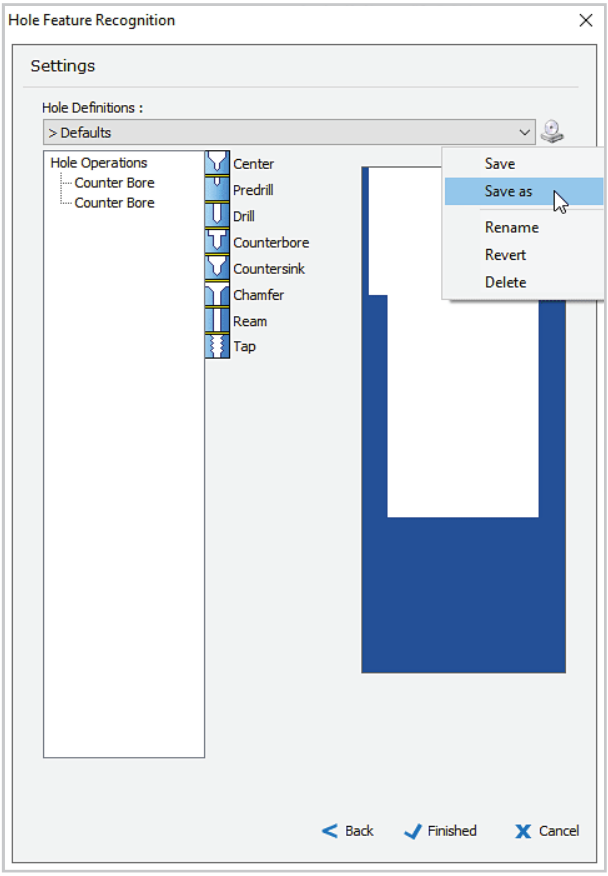


Turn on the Construction layer to see the boundary of the pocket. You can now re-create the HS pocket operation using the boundary and template you have saved.

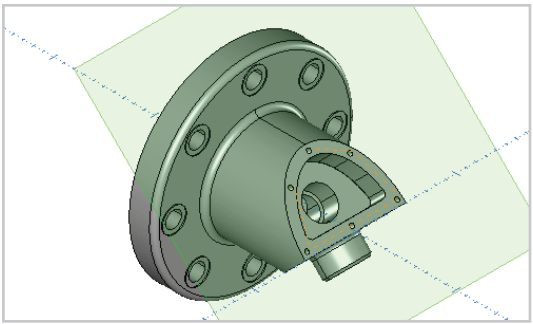


This image shows a section of the model on the XZ plane, with a backplot of the next operation which is named 6:C-Bore pocket holes.

This operation has been created using the OneCNC Hole Feature recognition function, and is made up of two helical counterbore toolpaths.



You can re-create this toolpath and the next easily by saving hole definitions in the last stage of the Hole Feature recognition dialog.



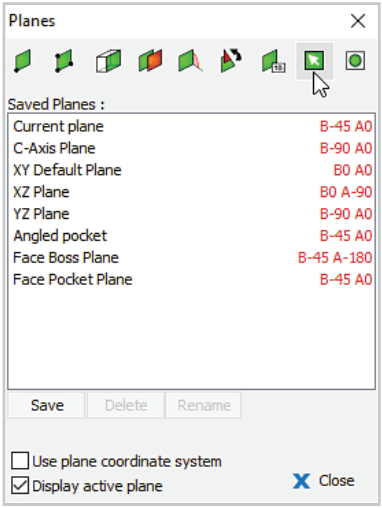
The next operation, named 7:Drill pocket face holes, drills the 4mm diameter holes in the face around the pocket.

When you have re-created the toolpaths at this plane position, you will be ready to proceed to the next B Axis position.

The next four operations create the angled boss opposite the pocket, using a plane based on the flat surface around the boss.

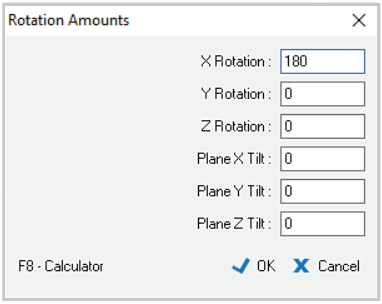
Change the B axis position

To get a clear view of the boss, we will rotate the isometric view by 180°, using a temporary view plane.



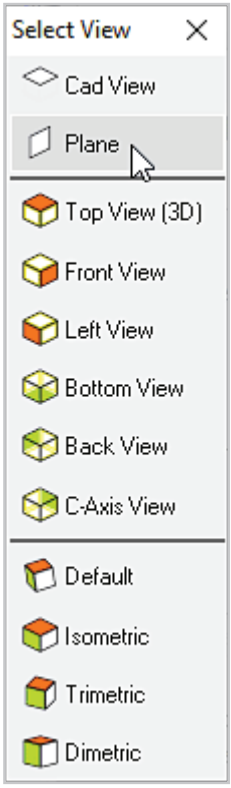
With the view set to Isometric, open the Planes dialog and click on the Plane from Screen icon.

The current plane will now correspond to the Isometric view.

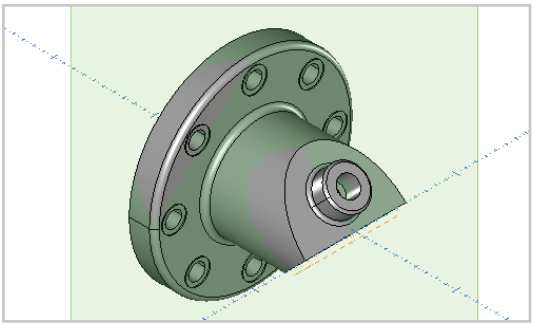


Click on the Rotate Plane icon, and rotate the plane by 180° about the X axis.

Save the new plane as Boss view plane, so you can return to this view easily.

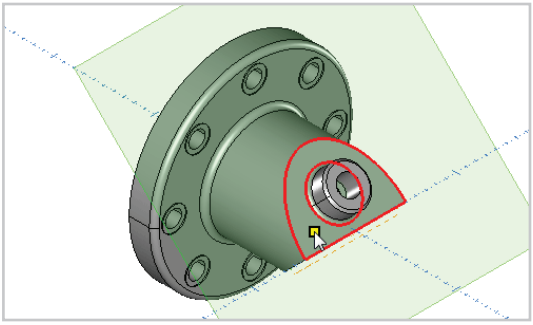


Open the View selector, and click on the Plane icon.



The view will rotate so you have a clear view of the boss feature.

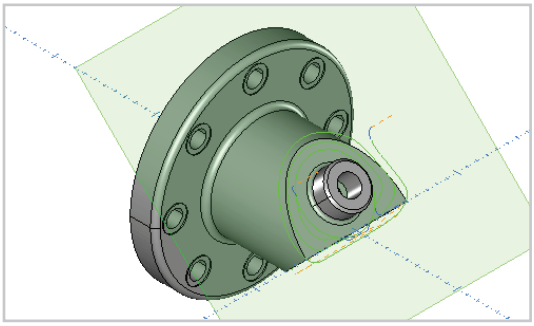
This plane was useful to set the view but it isn't the plane we want to machine to.



Use the Plane from surface command to define a plane on the flat around the boss as shown. Save the plane as Angled boss.

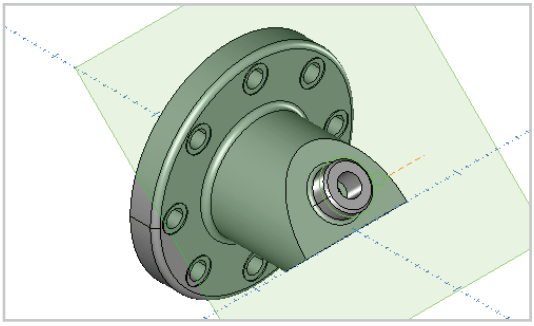
Recreate sample toolpaths in the second position

To complete the B Axis operations for this position, edit the toolpaths in the original toolpath group to create templates, and re-create them in your new group.

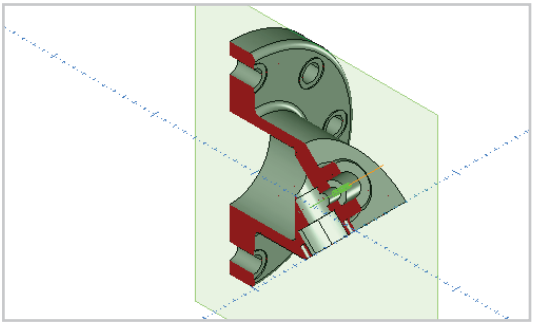


Backplot the operations 8:Open Pocket Face Boss and 9:Open Pocket Boss and you will see how the boss has been created.

Note that the 9:Open Pocket Boss operation is defined with an internal circular boundary to preserve the boss itself, and uses a bullnose cutter to create the fillet at the base of the boss.

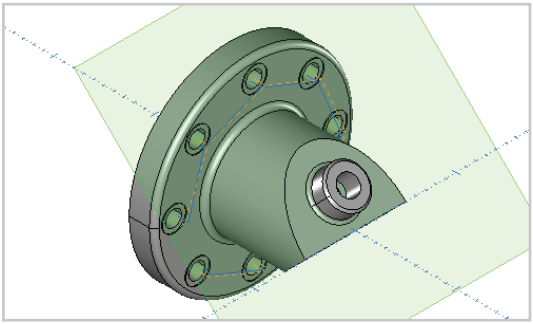


The next operation is 10:Chamfer Boss which is similar to that described in the C axis Face section.

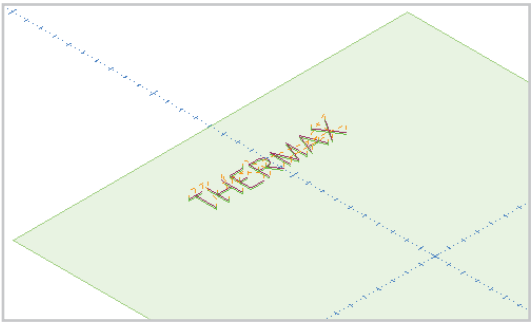


The 11:Drill and Counterbore Boss is a Hole Recognition operation like that for the pocket side of the part.

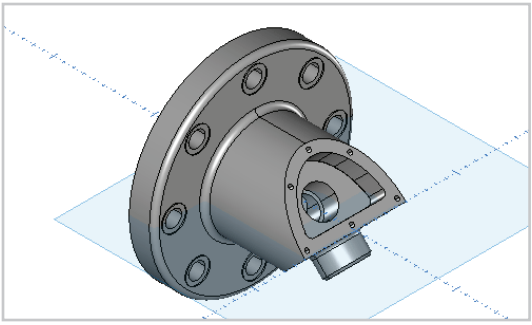
The remaining operations in the toolpath group are C axis face and wrap operations to complete the part.



C Axis Face operations are used to drill and spot face the flange.



The text is engraved on the rim using a C Axis Wrap operation defined with vectorized text on the XY plane.



You have now seen how this part is created.

What may have looked like a complex part at first is actually a straightforward programming exercise if you progress logically through the machining strategy one step at a time.

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

QARM Pty Ltd. MAKES NO WARRANTY, EITHER EXPRESSED OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE, REGARDING THESE MATERIALS AND MAKES SUCH MATERIALS AVAILABLE SOLELY ON AN "AS-IS" BASIS. IN NO EVENT SHALL QARM Pty Ltd BE LIABLE TO ANYONE FOR SPECIAL, COLLATERAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES IN CONNECTION WITH OR ARISING OUT OF PURCHASE OR USE OF THESE MATERIALS. THE SOLE AND EXCLUSIVE LIABILITY QARM Pty Ltd., REGARDLESS OF THE FORM OF ACTION, SHALL NOT EXCEED THE PURCHASE PRICE OF THE MATERIALS DESCRIBED HEREIN.

QARM Pty Ltd reserves the right to revise and improve its products as it sees fit. This publication describes the state of this product at the time of its publication, and may not reflect the product at all times in the future.

Introduction to OneCNC Lathe Mill-Turn v62.80