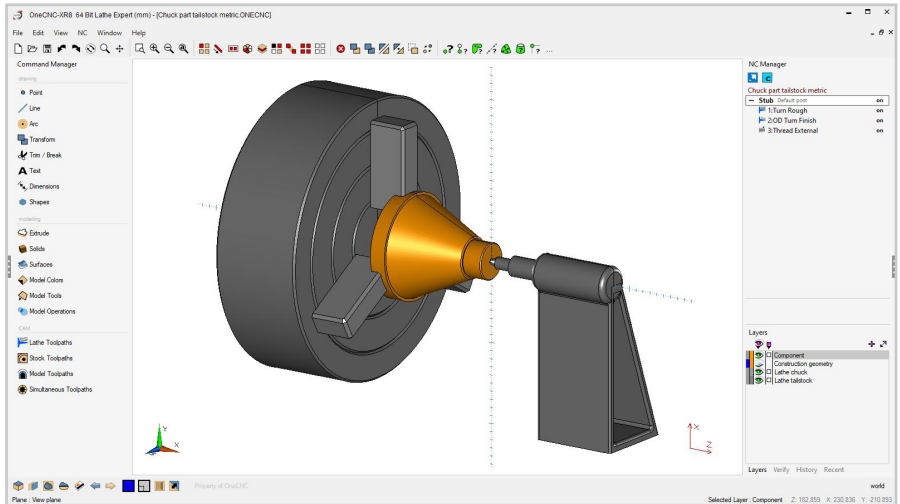


OneCNC Lathe



OneCNC Lathe is a hybrid CAD-CAM system with a user-friendly interface. OneCNC Lathe simplifies the process of designing and machining parts using CNC lathes and multi-axis turning centers.

OneCNC integrates the Microsoft Windows interface with a powerful CAD engine. Parts can be designed and modeled in OneCNC, or imported from 2D or 3D CAD files. The part model and geometry data is directly available to the inbuilt CAM functions.

The purpose of this Manual

This manual covers OneCNC Lathe Expert, Professional, and Express. The manual provides information on how to get started and use OneCNC, it is not designed as a machining instruction manual.

Much of the information in this manual is also contained in the OneCNC Help file, which also contains tutorials and reference information about available procedures and commands.

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OneCNC Environment

OneCNC has been designed as an integrated manufacturing environment, with purpose built interaction between the CAD and CAM functions.

New users familiar with Windows-based software will be at home with the well organized pull-down menus, toolbox and shortcut keys together with meaningful graphics included throughout the software interface.

The interface is similar to other common Windows-based drawing software and provides compact access to frequently used functions. Users experienced with other mechanical CAD software will appreciate the ability to customize the functionality in a way they are used to using.

CAD functions

The CAD functions are used to draw and create parts.

Line, arc, and spline geometry can be combined with NURBS surfaces and solids in a hybrid modeling process. As well as the complete CAD functions for constructing parts from scratch, robust translators are provided for importing parts in other formats. 2D drawing and 3D modeling methods have been designed for ease of use.

CAM functions

The CAM functions are used to create the required NC program toolpaths for CNC machining of your parts. Toolpaths can be edited and reviewed in the NC Manager, which also has functions for backplot, real time preview, and metal removal simulation.

Multiple Views

OneCNC can show single or multiple views of the same object or model. Any view can be dynamically panned, rotated or zoomed. Multiple views make it easier for you to create complex models. While in a command it is possible to select a point in one view and then select the next point from another view.

Multiple Document

You can have more than one part drawing open in OneCNC. This makes it easy to cut and paste from one drawing to another and toggle between drawings. The Recent File tab shows previews of files to help you quickly find the drawing you want. Document management is simple in OneCNC due to the fact that the CAM data is stored in the drawing file.

OneCNC Installation

Minimum System Requirements

Operating System

Microsoft Windows 10, 8, 7, Vista or Windows XP (32 or 64bit)
Internet Explorer 7 or later
NTFS File System is recommended

Hardware

Minimum 1 GB of quality memory for Windows XP, or 3 GB of quality memory for Windows 8, 7, or Vista, especially if working with complex parts.

Pentium 4 single, dual, or quad core PC. AMD Athlon can be used only if later than 2004 with specs equal to the Pentium 4.

NVIDIA GeForce or similar OpenGL and DirectX graphics card, minimum 1Gb. It is essential you download and install the latest graphics driver for your card, as drivers supplied on install discs may be outdated due to production lead times.

A mouse with 3 or more buttons and a scroll wheel.

One available USB 1.1 or 2.0 port or a suitable parallel port depending on which licensing dongle you have.

CD-ROM drive for installation of standalone version if required.

Broadband Internet access is recommended for updating the software.

OneCNC Installation

You will need:

OneCNC installer disk or a downloaded update setup file

OneCNC dongle

CD Key and serial number

There are three steps to installing OneCNC on a single computer:

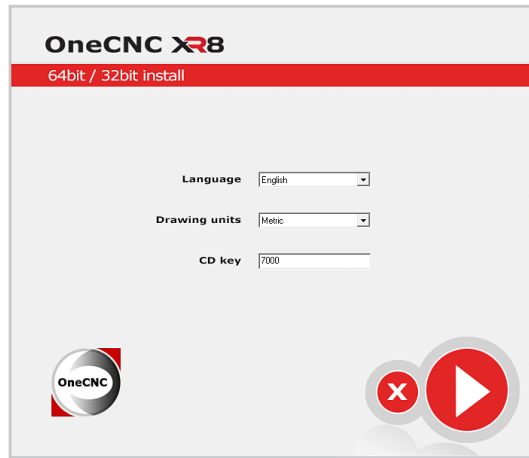
Install the software from disk or an update setup file

Plug in the dongle

Run OneCNC and enter the serial number to activate it

For network installations, network server software must be installed as well. That process is outlined in the next chapter.

All OneCNC products can be installed from the one installer package.



Insert the OneCNC CD or plug in the OneCNC flash drive.
Display the drive contents in Windows Explorer and double click
'onecnc_setup.exe'

Language Selection

Select your language from the drop down list.

Units Selection

Select the units you work in, Metric or Imperial (inches). This will set the default drawing units and post-processor configuration units. The units setting can be changed after installation using the Properties dialog on the File menu.

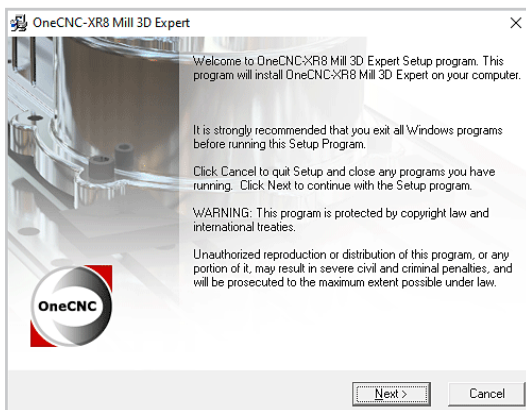
One CNC will open a file drawn in the alternate units at the correct real world size, but the NC toolpaths will be invalid, so you should save the files to separate locations.

CD Key

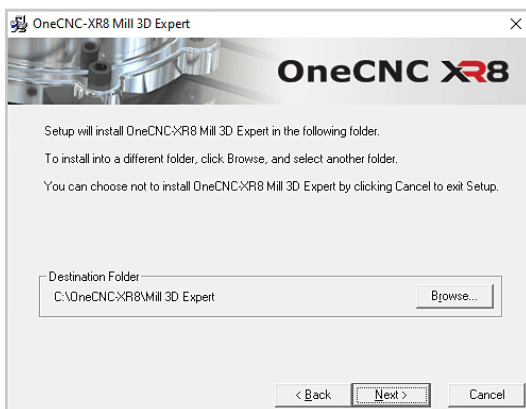
OneCNC installation requires a CD Key, which is usually found on the inside rear cover of this manual with your serial number, but is sometimes shipped separately from the software for security. Enter the CD Key and click the large 'play' button to start the installation.



The serial number will be requested when you first run the software. Keep a copy of your CD Key and Serial in a safe place as you will need them if you upgrade your computer and need to re-install.



The next dialog advises you to have all other programs closed, which is standard software installation procedure.



You then have the choice of where to install OneCNC. You can accept the default location or click the Browse button if you want to select your own location. Click Next when you are ready to continue.

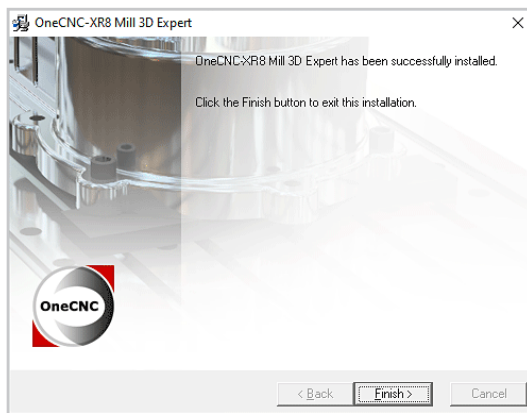
In the confirmation dialog you can click Back to review your settings, or click Next to install OneCNC.



If you are upgrading from a previous version of OneCNC, the new installation will be in a new location.

You can still use your existing tool and material libraries, and custom Posts. To do this, copy the Settings folder from your previous installation to your new installation Settings location.

It is good practice to keep a backup copy of your Settings folder.



When all the files have been installed, click Finished to complete the installation. There will be a pause while the dongle driver for Series 1 dongles is installed.

When the installer closes, you are ready to insert the dongle and start OneCNC.

OneCNC Updates



Registered users can download updates of the setup file for their version of OneCNC at no extra charge.

Go to <http://onecncsupport.com/> and click on OneCNC Updates when you have read the update details. Click on Check For Updates.

Enter your dongle and serial numbers to enter the site. Click on the Register button on the next page to appear. Fill in your registration details, and click Save. Click on the Download button to go to the actual download page.

Select the check box to confirm you have read the update details, and click on the Download button. A dialog will appear asking if you want to Run or Save the file. Select Save and choose the folder on your computer to save the file in. Double click the downloaded file to install the update.

The update can also be accessed by clicking on OneCNC Updates in the Help menu, or the OneCNC Software Updates, Drivers and Posts link in the OneCNC Users Forum.

Dongle Installation

OneCNC requires a dongle for the licensed program to run. A dongle is a small hardware device which verifies your software license.

Series 2 Dongle Installation - dongle numbers from 10000

All OneCNC dongles with a number higher than 10000 are Series 2 dongles.



Series 2 dongles are USB HID devices which look like this. These dongles do not require drivers for a single computer installation.

If you have a network license you will need to install a network dongle driver, as described in the next chapter.

To install a Series 2 dongle

Make sure OneCNC is not loaded or attempting to load. To install the dongle simply plug it into a USB port on your computer. Basically, if the dongle fits, it's the correct port. Do not try to force the hardware or you may damage your computer or dongle.

If you have a timed licence the Windows date and time must be correct before connecting the dongle. It is recommended that the USB port a dongle is plugged into should not be used for other devices.

Dongle Security



Because it is the dongle which verifies your purchase of OneCNC, if a dongle is accidentally damaged or broken the damaged dongle must be returned to OneCNC before a replacement dongle can be issued.

If a dongle is lost or stolen, the replacement price is the full price for the software. Take steps to secure your dongle, and insure it for the replacement cost of the software.

Series 1 Dongles - dongle numbers 1 to 9999

If you have upgraded from an earlier version of OneCNC, you may have a Series 1 dongle with a number less than 10000.



These dongles may be USB or Parallel port dongles, and are now obsolete.

There is no 64 bit driver available for these older dongles, so if you have a series 1 dongle please see your OneCNC dealer for a replacement series 2 dongle.

A series 1 dongle will be replaced at no extra cost with any upgrade to the current version of OneCNC.

Note:



A series 1 dongle will not run the 64bit version of OneCNC.

Until you get your replacement dongle you can install the 32bit version of OneCNC on 64bit Windows by changing the last two digits of the CD Key to 32. For example the CD Key 6432 will install 32bit Mill Express which can be run with a series 1 dongle.

When you change to a new series dongle re-install OneCNC with the standard CD Key.

Start OneCNC and enter the Serial Key

Once installation is complete and the dongle is installed, OneCNC can be started. Double click on the OneCNC shortcut on your desktop, or select the OneCNC shortcut in the OneCNC folder in the Start menu.



The first time OneCNC is started the Licence Manager will look for the dongle. Select 'Single user licence' if you have a standalone installation, and click Continue.


Network licence - Series 2 dongle



To run OneCNC on a client PC connected to a network using a Series 2 dongle, you must select 'Network licence'.

To run OneCNC on a computer which is also the network server, the server installation needs to be set to 'Single user licence', as in this case the dongle is on the local system and not on the network.

The Licence Manager will open so you can enter the licensed user name and serial number.



The Licence Manager dialog box is titled "Licence Manager" and contains the following fields and options:

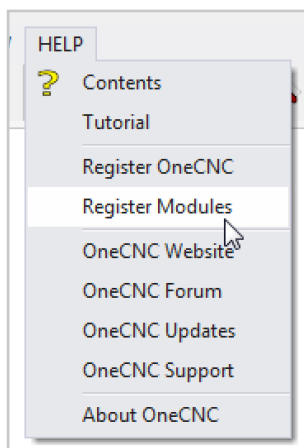
- Product: OneCNC×R8 64 Bit Mill 3D Expert - Version 62.4
- Client Number: CN12496
- Company Name:
- Email:
- Serial Number: - - -
- Licence Status: [Enter your serial number](#)

At the bottom right, there are two buttons: "OK" (with a checkmark icon) and "Cancel" (with an 'X' icon). To the right of the input fields is an image of a blue USB drive.

The serial is made up of digits 0 to 9 and letters from A to F. The code must be entered exactly as it appears.

When the serial key has been entered the Licence Manager will indicate that the licence is acceptable, and the software can then be started.

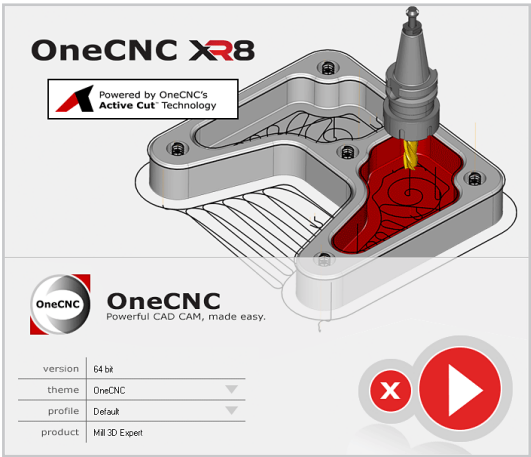
Entering serials for optional modules



If you have a serial for an optional module, start OneCNC and click on Register Modules in the OneCNC Help menu.

Enter the serial in the dialog which opens, and the module will be activated.

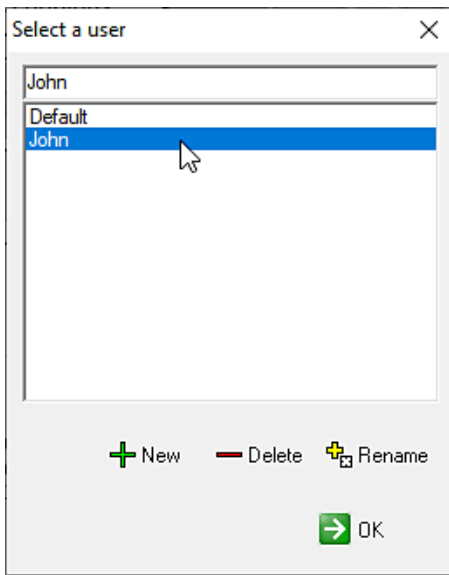
User profile selection



OneCNC starts with this screen, which gives you the option of selecting the user profile you want to use. The user profile stores all your settings such as toolbar layout, interface color settings and your recent files list.

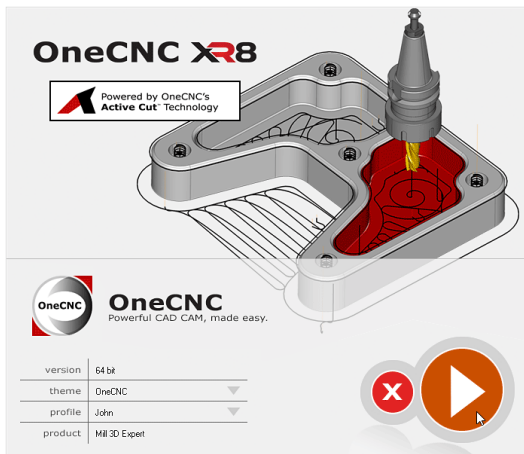
version	64 bit
theme	OneCNC
profile	Default
product	Mill 3D Expert

Clicking on the small arrow opens the user selection dialog, which allows you to select or create settings in the names of specific users for customizing.



Select the user profile you want to use from the list, and click OK.

You can create, delete, and rename user profiles using the buttons below the drop down list.



Click here to start OneCNC with the selected user profile.



If your user profile or other settings are not being saved, open Windows Explorer and right click on your Settings folder. Select Properties, and clear the "Read-only" check box. Click Apply and click OK to apply the change to all subfolders and files.

OneCNC Network Installation

This step is only necessary for network installations.

For network installations, network server software must be installed on the server or computer the dongle is plugged into.



Series 2 dongles use the OneCNC Dongle Server which is included in the OneCNC installer, and is accessed by a CD Key similar to installing the OneCNC software itself.

To install OneCNC Dongle Server on a computer or server with a Series 2 dongle, you will need:

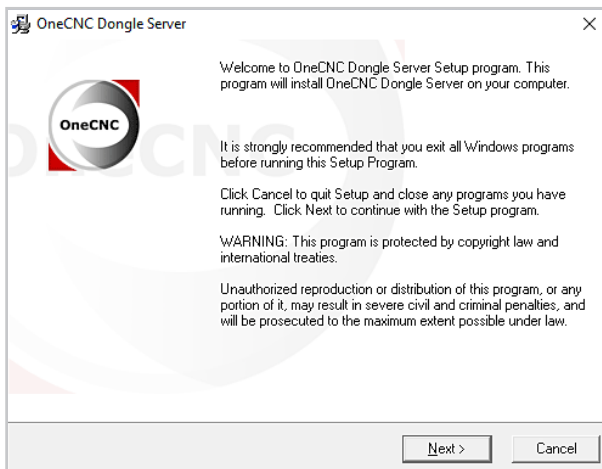
- OneCNC installer disk or a downloaded update setup file
- OneCNC dongle
- Network CD Key and serial number



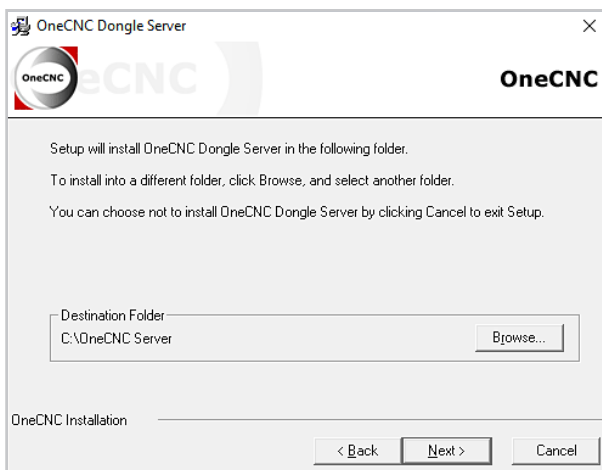
Run the installer on the server computer. To install the network dongle server software enter the CD Key 60200.



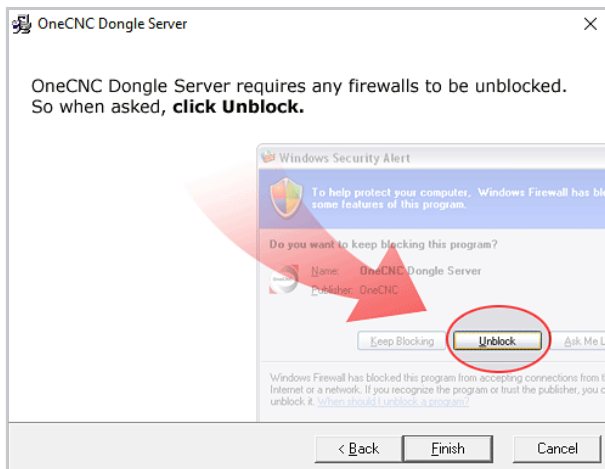
When the next dialog appears, enter the server software serial. The serial is made up of digits 0 to 9 and letters from A to F. This is not the same as the software serials which are entered later when installing the particular versions of OneCNC such as Mill and Lathe.



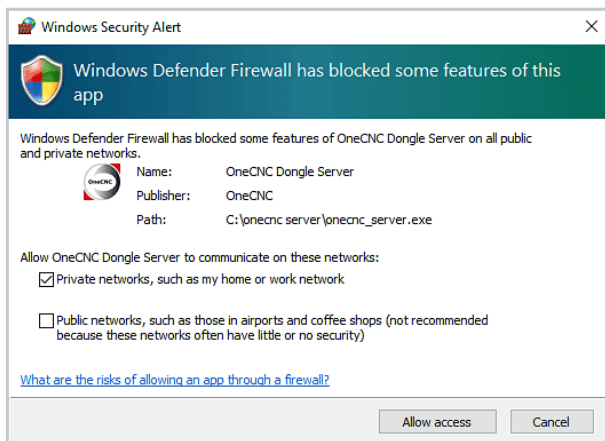
The next dialog advises you to have all other programs closed, which is standard software installation procedure.



You then have the choice of where to install the server software, the default installation is as shown.



The next dialog indicates you will need to unblock the firewall for this application only. Click Finish to complete the installation.



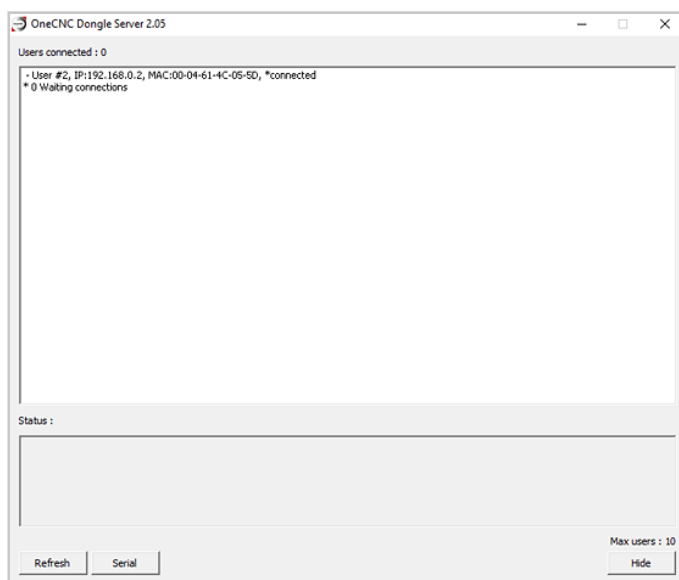
Here is the Windows Firewall dialog where you have to click Unblock for the dongle server software to work. This has no effect on network or internet security, it is only allowing any PC on the local network to access the OneCNC Dongle Server.

The server software is now installed and working.



There will now be an icon in the Taskbar at the bottom of your screen, indicating the dongle server software is running.

You can click the icon to open the Dongle Server interface, but it is not necessary to have the Dongle Server interface open for client computers to access the dongle.



The interface allows you to see how many clients are connected to the Dongle Server. You will not see a connection as shown here, until the OneCNC programs are installed on the client computers.

To install OneCNC on the client computers, start the OneCNC installer on each computer, and load the OneCNC application using the CD key for OneCNC.

Running OneCNC Dongle Server automatically

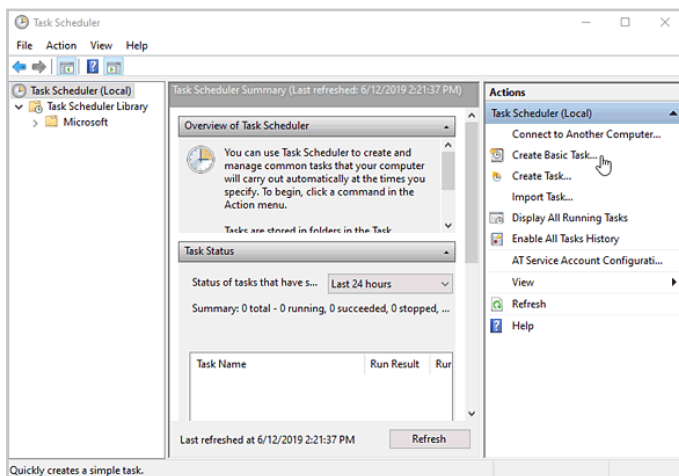
You can run OneCNC Dongle Server automatically so you do not have to start it if the server has been shut down.

Setting up a scheduled task will install OneCNC Dongle Server as a service which will automatically restart when the network server is started.

Firstly, install OneCNC Dongle Server as shown in the previous section. You can then use the Windows task scheduler to set up an action to start the OneCNC Dongle Server when the computer is started. You must be logged on as an administrator to perform these steps.

To open the Windows Task Scheduler, open Control Panel, click on System and Security, and click on Administrative Tools (if Control Panel opens in Classic view you can just double click on Administrative Tools). Double click on Task Scheduler.

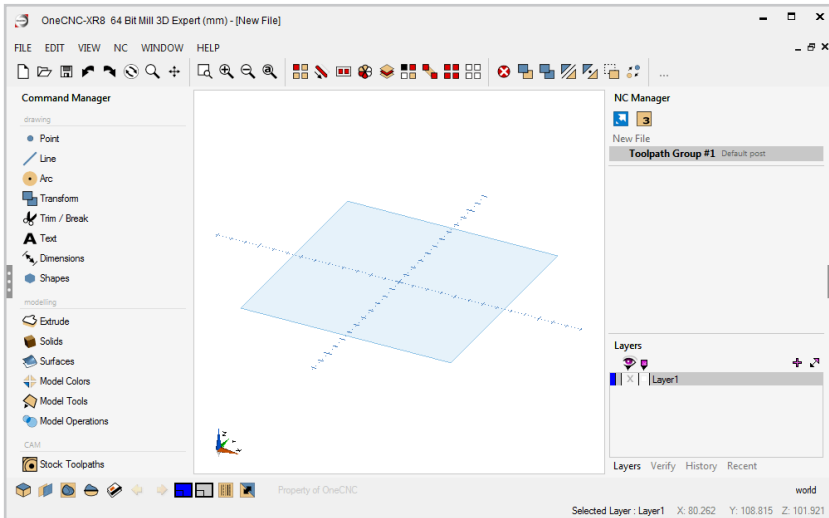
In Windows Server, open the Server Manager and navigate to Server Manager > Configuration > Task Scheduler .



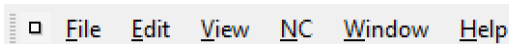
Click Create Basic Task. Enter the name and description for the task, and then click Next. Select When the computer starts, and click Next. Select Start a program, and click Next. Click Browse to find the OneCNC server program, which will typically be at C:\OneCNC Server\onecnc_server.exe Click Next when you have selected the program. Review the settings you have made and click Finish.

Restart the server to apply the changes you have made.

OneCNC User Interface



The OneCNC user interface has been carefully designed for ease of use.

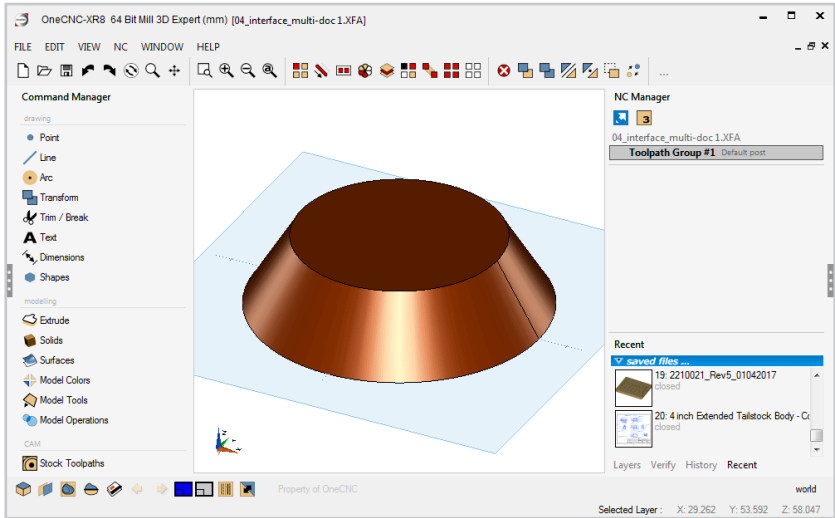


Windows menus give you access to many features in OneCNC. Menu items with a pointer on the right side automatically open sub-menus which you can slide the mouse across to select from.

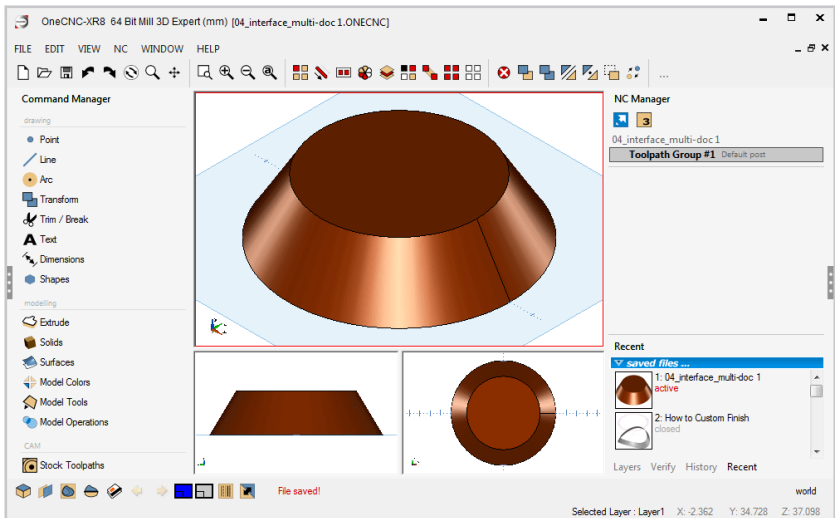


The Mill Expert interface is shown in this chapter. The available modeling and CAM functions you see will vary for different types or levels of OneCNC. OneCNC Solid Creator is intended for design only and has no CAM functions.

Drawing Window

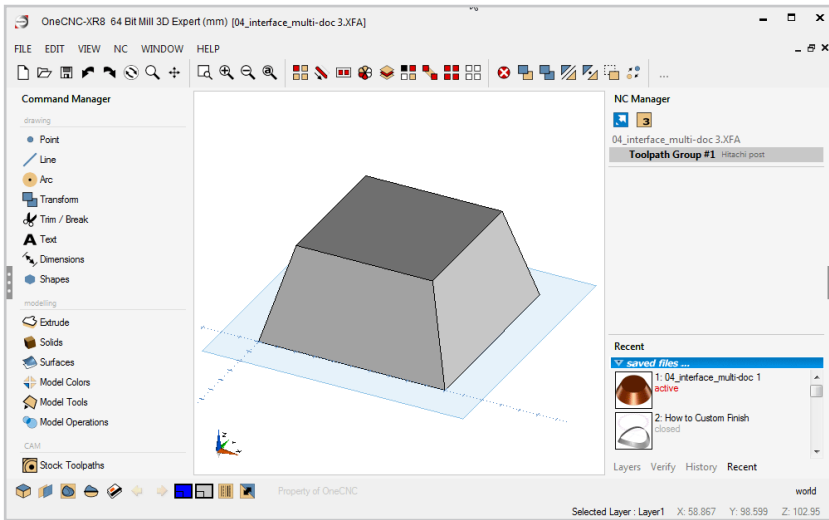


The file you are working on is displayed in the drawing window, which is a real-time OpenGL model view. This allows you to view your object by means of dynamic rotation, panning and zooming.

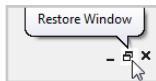


Multiple Views of the same file can be shown. When parts become complex it can be useful to view and edit one object from various angles simultaneously. This function can be accessed from Window > Window Layout.

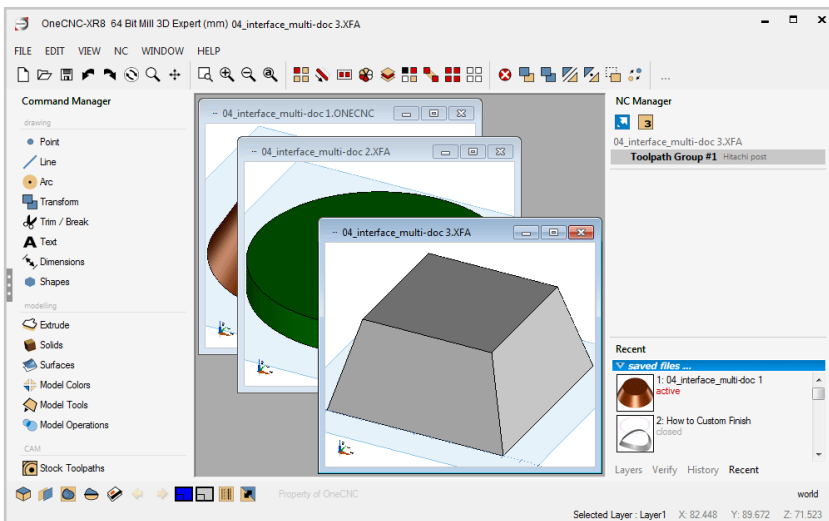
Multi Document



You can have more than one file open in OneCNC. When the file windows are maximized you will only see one file at a time. You can select the file to display from the Window menu.



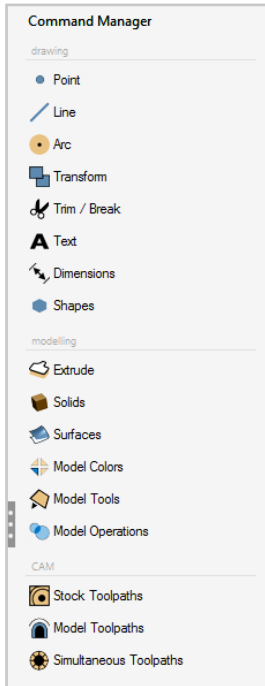
Clicking the Restore Window button will display each file in its own window.



The file windows can be controlled by dragging, or using the Cascade and Tile commands on the Window menu.

Toolbox

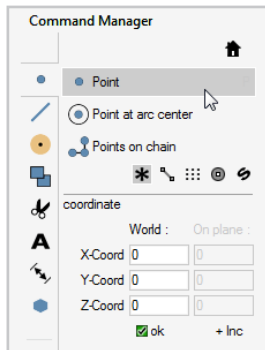
The Toolbox is a menu system which is shown in the left sidebar.



When you start OneCNC you will see the toolbox Home menu.

The Home menu icons are grouped into commands for drawing, modeling, and CAM functions.

The toolpath functions available will vary with the version and level of OneCNC you are using.



Each icon in the toolbox Home menu will open a sub-menu from which you can select a specific command.

For many commands entry boxes will appear where you can input required values.

Click on the Home icon at the top of the Toolbox to return to the Home menu.

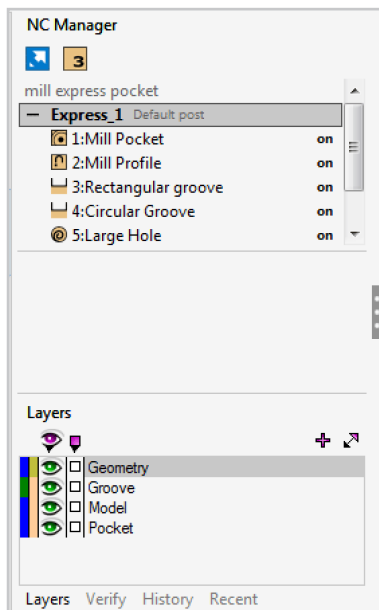
Clicking on the dotted button at the left of the toolbox will hide it.

Click on the button again to bring the toolbox back into view.

Right Sidebar

On the right of the screen the right sidebar has panels for:

- NC manager
- Layers Manager
- CAD Verify
- History
- Recent file list



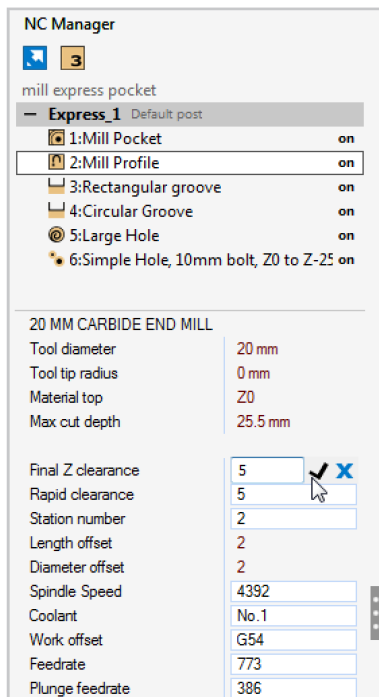
Clicking on the dotted button at the right of the sidebar will hide it.

Click on the button again to bring the sidebar back into view.

NC Manager

The NC Manager displays the toolpaths you have defined.

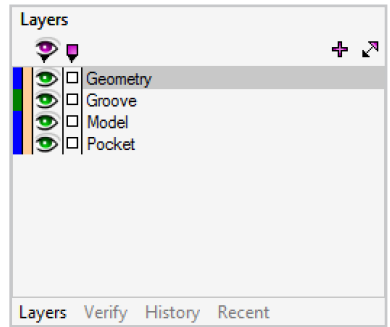
Machining operations can be grouped, and operations can be renamed, edited, rearranged or duplicated.



Layer Manager

When you draw an entity it's created on the currently selected layer. The Layer Manager is used to create, name or delete drawing layers.

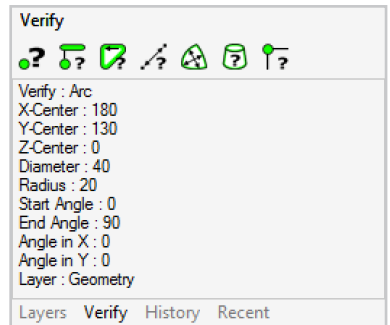
The panel shows a list of layers in a drawing. You can turn off a layer by closing the eye for that layer in the Layer Manager. Entities on the layer are still in the drawing but will not be visible.



Verify

The Verify panel holds tools to read data from geometry.

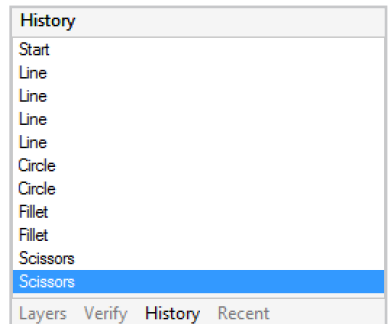
You can find exact values for position length or area for 2D entities, and volume of 3D solids. You can also find the angle and distance between entities.



History

As you construct your part, a record of commands used is created in the History tab.

You can roll construction back or forward like a multiple undo and redo. If you roll-back your history and start new construction, the rolled back construction is replaced with the new construction.

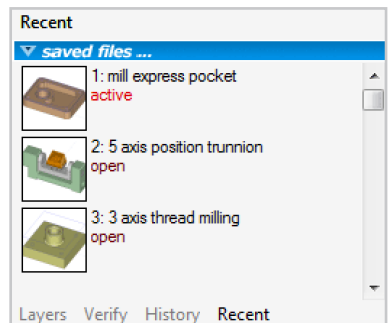


Recent Files list

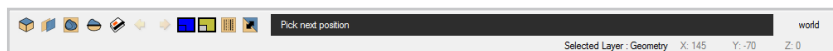
When you save a file, a thumbnail image of it is added to the Recent file list.

The list shows if a file is open, active or closed.

You can open a closed file by clicking on the thumbnail image.



Status Bar



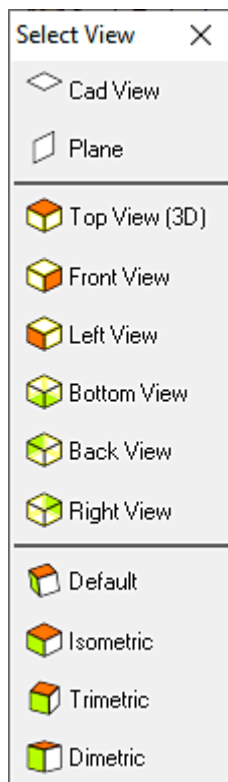
The Status Bar at the bottom of the OneCNC window provides you with functions related to the appearance of objects, and a message panel which shows prompts to help you use the program.

Preset Views

OneCNC has a set of preset views so you can select the direction you want to look at your model from.



Click on the View icon to open the preset view selector.



Click on the icon for the view you would like to change to.

For 3D modeling the Isometric or Trimetric Views are the most commonly used.

The CAD View is a special view optimized for 2D drawing, so it is not suitable for 3D commands. Use the Top View if you want to use 3D commands in a view similar to the CAD view.

Construction Plane

When you draw or model in OneCNC you can use a coordinate system which is independent of the world XYZ coordinate system. OneCNC displays the Construction Plane which is a visual reference for the XY plane of the current construction coordinate system.



The Construction Plane is controlled using the Plane dialog, which is opened by the Plane icon in the Status Bar.

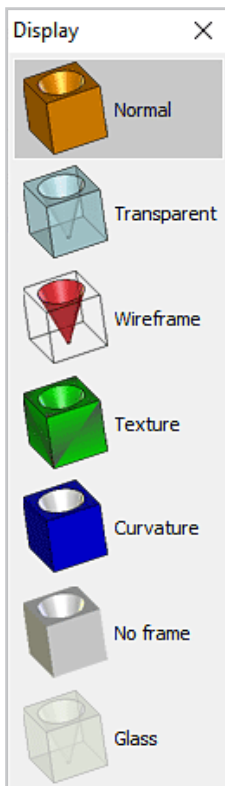
The Working With Planes tutorial in this manual covers the use of Construction Planes.

Display mode

Display controls the appearance of 3D surfaces and solids.



Click on the Display icon to open the Display mode selector.



Click on the icon for the display mode you would like to change to.

If you are extracting 2D profiles from 3D objects it can be useful to use the Transparent or Wireframe modes to access obscured parts of the model.

If you set the display mode to Wireframe, be sure to return it to another mode before attempting any modeling commands as you will not see any surfaces or solids.

Section Tool



The Section tool is used to view a cross-section of your work parallel to the current working plane. It can also be used to create section geometry.

3D to 2D Automated drawing system



In Expert versions of OneCNC and OneCNC Solid Creator you will see the Pages icon which opens the automated 3D model to 2D CAD drawing system. This can be added as an optional module to OneCNC Professional versions.

Drawing Color and Style selection



OneCNC has a 2 color drawing system. Color selection boxes in the Status Bar set the color for new entities. The color selection box on the left is for geometry and that on the right is for surfaces and models.

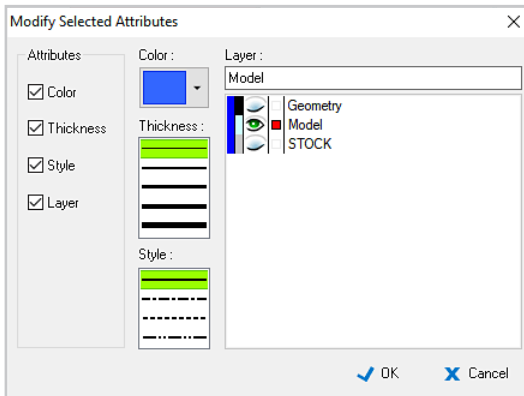


To select the line style and line thickness for new geometry entities click the Styles icon.

Modifying Entities



Click Modify to change the color, line thickness, line style or layer of existing entities.



Select the check boxes for the attributes you want to change and select the new values.

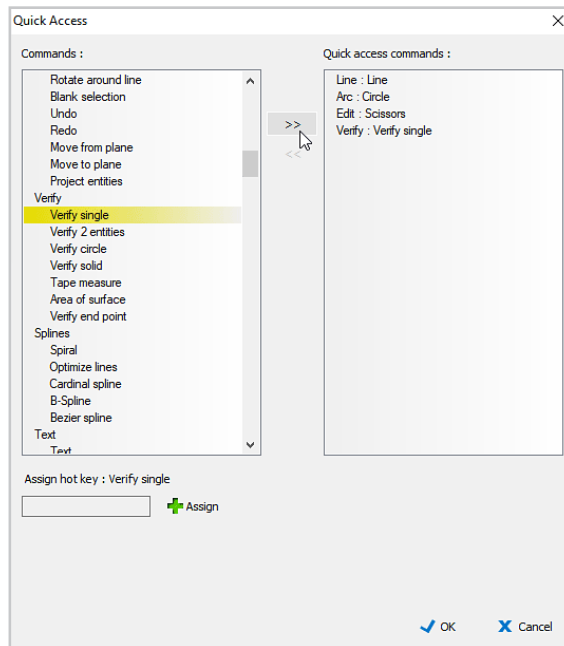
Quick Access Toolbar



The Quick Access toolbar is a custom toolbar you can use to display icons for commands you use often.



When you install OneCNC the Quick Access toolbar will be empty except for the Quick Access icon. Click on the Quick Access icon to open the Quick Access customization dialog.



Select the command you want to add in the Commands list on the left of the dialog. A keyboard shortcut can be assigned to the command in the 'Assign hot key' entry box.

You can right click on an icon you use regularly to add it to the Quick Access toolbar.



Keyboard Shortcuts

OneCNC provides a default set of keyboard shortcuts, which are a powerful way of accessing functions quickly by pressing a key or combination of keys. You can add your own keyboard shortcuts in the 'Assign hot key' entry box of the Quick Access dialog.

The following lists give the default shortcut key settings in OneCNC.

View shortcuts

Arrow Keys	Rotate model horizontally and vertically
Arrow Keys + Shift	90 Deg steps rotate right left up down
Ctrl + Arrow Keys	Pan
Space Bar	Preset View selection dialog
Space Bar double press	CAD View
Delete	Delete selected entities
F	Zoom to fit
Z + Shift	Zoom in
Z	Zoom out
W	Zoom to a window
Ctrl + 1	Top View
Ctrl + 2	Bottom View
Ctrl + 3	Left View
Ctrl + 4	Right View
Ctrl + 5	Front View
Ctrl + 6	Back View
Ctrl + 7	Isometric View
Ctrl + 8	Trimetric View
Ctrl + 9	Dimetric View
Ctrl + 0	Default View

Drawing shortcuts

P	Point
L	Line
R	Rectangle
O	Offset
H	Hatch
C	Circle with given radius
Ctrl + F	Fillet Arc

File shortcuts

Ctrl + N	New file
Ctrl + O	Open file
Ctrl + S	Save file
Ctrl + Tab	View next open file
Ctrl + P	Print

Edit and modify shortcuts

S	Select Single
L	Select nearest line when in Single Select
mode	
A	Select nearest arc when in Single Select
mode	
Ctrl + A	Select All
Ctrl + Z	Undo
Ctrl + Y	Redo
Ctrl + X	Cut to clipboard
Ctrl + C	Copy to clipboard
Ctrl + V	Paste from clipboard
Ctrl + D	Delete
M	Move
B	Blank selection
T	Trim One
Ctrl + T	Trim Two



Mouse Actions

Left Button	Enter / Select
Right Button	Escape / End Function
Mouse Wheel	Zoom to/from cursor
Middle Button + Ctrl	Pan
Middle Button	Rotate view
Middle Button + Shift	Zoom

OneCNC Overview

The purpose of OneCNC is to create NC code programs for CNC machines. The workflows for the various types of OneCNC are as follows:

OneCNC Mill or Mill with Multi-axis module

1. Draw or import part geometry or model
2. Define toolpaths to cut the part
3. Preview or simulate to verify toolpaths
4. Output NC program to send to the machine

OneCNC Lathe

1. Draw or import part geometry or model
2. Define toolpaths to turn the part
3. Preview or simulate to verify toolpaths
4. Output NC program to send to the machine

OneCNC Lathe with Mill-Turn module

1. Draw or import part geometry or model
2. Define turning and live tooling toolpaths
3. Preview or simulate to verify toolpaths
4. Output NC program to send to the machine

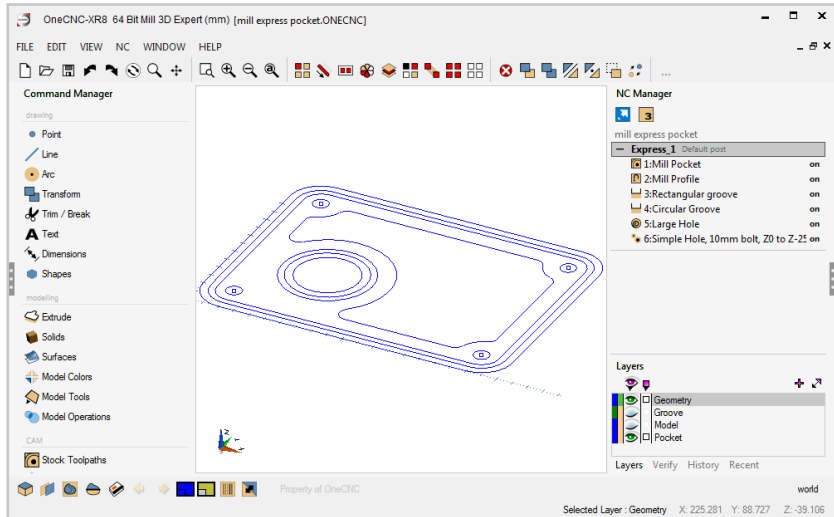
OneCNC WireEDM

1. Draw or import part geometry or model
2. Define wirepaths to cut the part
3. Preview to verify wirepaths
4. Output NC program to send to the machine

OneCNC Profiler (Nesting for plasma, laser, waterjet or router)

1. Draw or import part geometry
2. Add part shapes to the Parts list
3. Nest parts to sheets
4. Define drill, engrave and cutting operations for the nested sheets
5. Preview to verify toolpaths
6. Output NC programs to send to the machine

2D and 3D Geometry



Geometry drawing

OneCNC parts are constructed using pure geometry. Boundaries can consist of lines, arcs or splines constructed in 3D space.

Layers

The Layer Manager sidebar panel is used to create, select, open and close layers which are used to organize your drawing.

Dimensions and Text

All 2D and 3D geometry can be dimensioned and annotated. Text labels can be added to parts and converted to geometry for engraving.

Drawing Layout Patterns

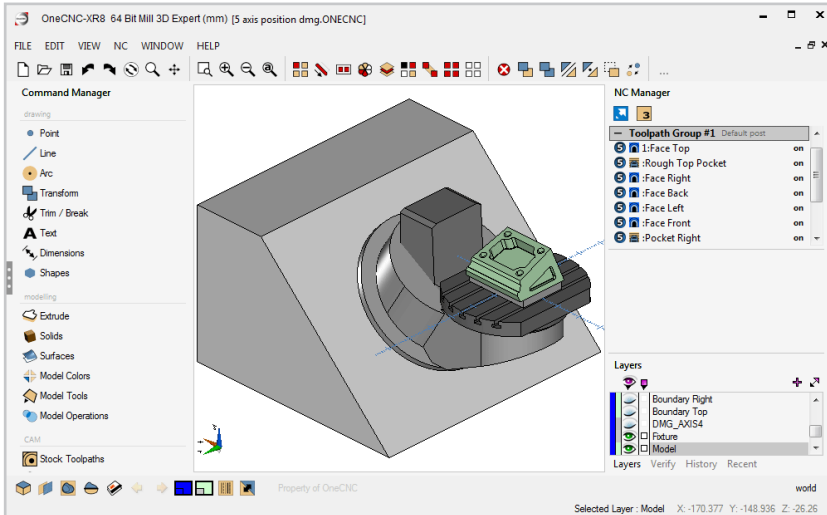
Borders, title and revision blocks, and bill of material lists can be saved as patterns and loaded into drawings as required.

3D to 2D Technical Drawing System

Expert versions of OneCNC include the 3D to 2D drawing system which automates the preparation of 2D drawings for printing. 2D technical drawings are created as page layouts in your ONECNC file, using title block and border templates. 2D views of the 3D part model can be created automatically and added to a technical drawing. The system can be added to Professional versions of OneCNC as an optional module.

3D Modeling

Available in OneCNC Expert and Professional versions



Surfaces Creation

Planar and curved surfaces can be constructed using the many available strategies.

Solids Creation

Solids creation from Primitives allows you to create basic shapes such as Cubes, Spheres, and Cylinders. Solids can also be created by revolving or extruding geometry.

Hybrid Solids Creation

In the hybrid modeling technique a closed set of surfaces can be unioned to become a new solid. Existing solids can be merged with a closed set of surfaces to become a new solid.

Model Operations

New solids can be created by the cutting, uniting or intersection of existing solids. Solids can be split by a plane, or broken into separate surfaces.

Expert Tools

Expert versions of OneCNC have additional advanced modeling tools.

Toolpathing

In all versions of OneCNC except Solid Creator, toolpaths can be defined by clicking an icon in a Toolpaths menu in the Toolbox and completing any selection or settings entries that the toolpath requires.

Toolpath management

When you define a toolpath it will be added to the active toolpath group in the NC Manager.

An NC file can be output from a single operation, all the operations in a single group, or all operations in all groups in the NC Manager.

Right clicking on a group or toolpath will open a context menu with commands for frequently used functions such as preview and backplotting.

In Mill or Lathe the lower section of the NC Manager displays settings for the currently selected toolpath which can be edited.

NC Manager

mill express pocket

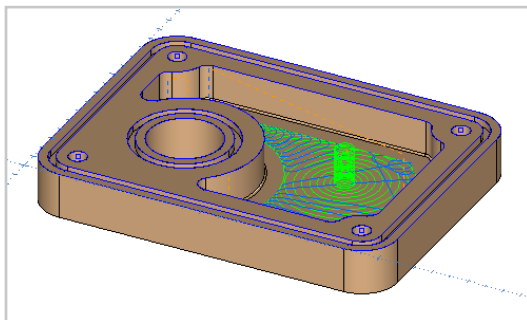
— **Express_1** Default post

- 1: Mill Pocket on
- 2: Mill Profile on
- 3: Rectangular groove on
- 4: Circular Groove on
- 5: Large Hole on
- 6: Simple Hole, 10mm bolt, Z0 to Z-25 on

20 MM CARBIDE END MILL

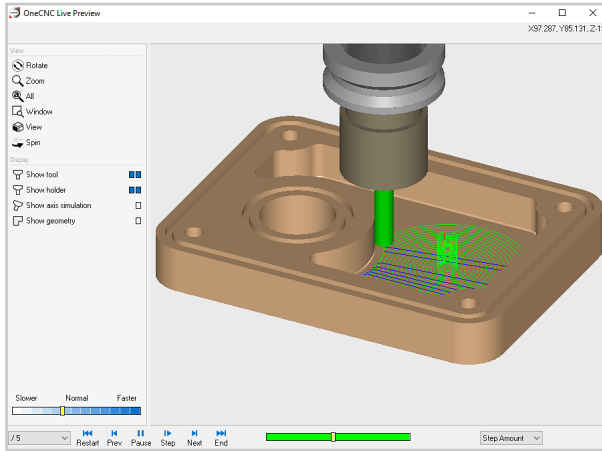
Tool diameter	20 mm
Tool tip radius	0 mm
Material top	Z0
Max cut depth	25.5 mm
Final Z clearance	5
Rapid clearance	5
Station number	2
Length offset	2
Diameter offset	2
Spindle Speed	4392
Coolant	No.1
Work offset	G54
Feedrate	773
Plunge feedrate	386

Toolpath Backplot



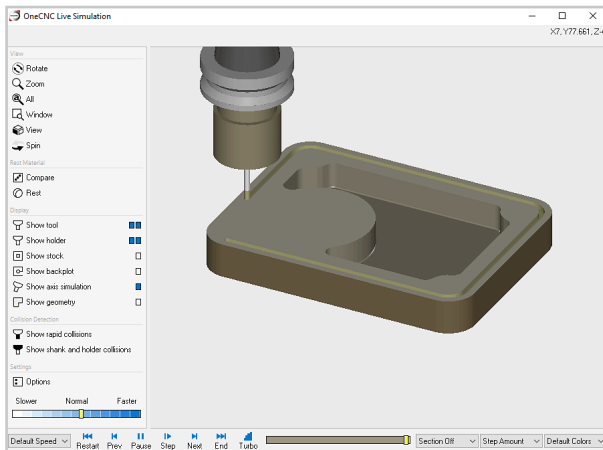
Toolpaths can be backplotted, which means they are drawn as geometry on a special layer named Backplot. Feed moves are drawn in green, and rapid moves in orange. Reposition moves in highspeed machining are drawn in blue.

Toolpath Preview



The Live Preview window shows an animated preview of the toolpath, backplotting as it goes. In Mill or Lathe the cutting tool will be shown traveling along the toolpath.

Metal Removal Simulation (OneCNC Mill and OneCNC Lathe only)



In all levels of Mill and Lathe, OneCNC provides real-time toolpath metal removal simulation. By simulating the feed and speed of the process, you can check the machining operation prior to sending to the CNC machine.

OneCNC Mill also provides Rest Material Verification, which highlights the thickness of material left on the part until it is fully machined.



NC Processing

Process Toolpath Group Multiple Parts

Simulate / Rest

Post

Job Sheet

Backplot

Preview

Quick Check

Post: Default Setup

Status: ☒ On ☐ Off

Description: Express_1

Program Number: 4532

Part Number: P000034

Author: OneCNC

Workshift: X0.000, Y0.000, Z0.000 Set

Table Load Move: None Set

Notes: Express Demonstration Part

Renumber operations

OK Cancel

Clicking the Process icon in the NC Manager opens the NC Processing dialog. You can enter notes or a program number before the file is output. OneCNC formats the NC file using a machine specific definition called a 'Post'. OneCNC is supplied with Posts for all popular machines, and it is easy to create a custom Post by copying and editing an existing format.

mill express pocket.NC - OneCNC Link

File Edit View Tools Bookmarks Help

O4532

N10 (PART - P000034)

N20 (NOTES - EXPRESS DEMONSTRATION PART)

N30 G00 G17 G40 G49 G80 G90

N40 (10 MM 1.0R CARBIDE BULLNOSE)

N50 T1 M06

N60 G00 G90 G54 X140.155 Y70.039

N70 S8785 M03

N80 G43 H1 Z5.

N90 M08

N100 G00 X140.155 Y70.039 Z5.

N110 Z1.

N120 G01 X141.369 Y70.349 Z0.956 F316.0

N130 X142.467 Y70.951 Z0.912

Configuration: default

Line 1 Col 1

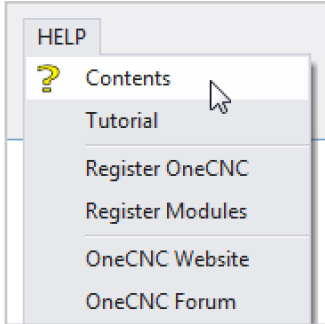
Editing and sending the created program

When you output the NC file for your job it will open in OneCNC Link, the complete editor and DNC transfer program, where you can see your NC code before the final stage of sending the completed NC program to the CNC machine.

OneCNC Integrated Help

At OneCNC we put a lot of effort into making OneCNC the world's easiest CAD/CAM package to use. OneCNC has several features within the program to help new users quickly adapt to its powerful but easy to use features.

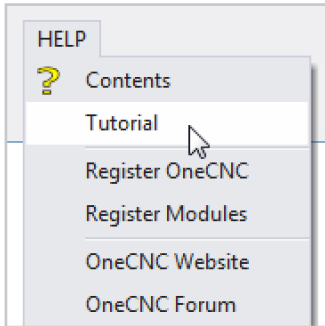
Help



If there is a specific part of the program you want to learn about, click on Contents in the Help menu.

The help files will open in a new window, with navigation controls at the top. The Back or Forward buttons allow navigation between help files you have visited, like an internet browser. The Home button will return you to the first page shown on the screen.

Tutorials



Clicking on Tutorial in the Help menu will take you straight to the OneCNC Tutorials folder in the Help files.

There are step by step tutorials for 2D CAD, 3D modeling, and how to define machining toolpaths for each version of OneCNC.

Context Sensitive Help

OneCNC uses Context Sensitive help. This means if you are using a function and need help on it, press F1 and OneCNC Help will open showing the relevant topic.

OneCNC CAD Tutorial 1

Introduction To Cad Drawing

This Tutorial is designed for all OneCNC products.

This tutorial will quickly take you through the essential functions for drawing and modifying geometry in OneCNC, to get you up to speed in the shortest possible time. In the exercises in this chapter, exact positioning is not important. The aim is to get an overall view of the process of drawing in OneCNC.

Start a new drawing

When you start OneCNC you will have a new file ready to draw in.



You can also start a new file by clicking on the New File icon in the Standard toolbar.

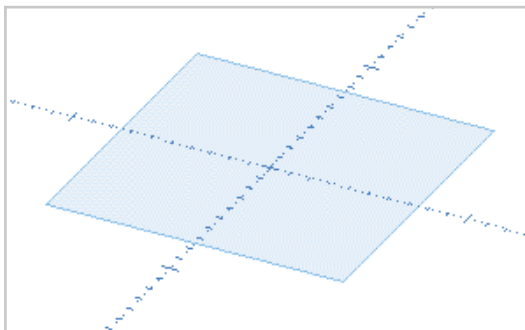


The first thing you should do when starting work in a new file is to click Save and save the .ONECNC file. If you save this tutorial you will be able to open it and resume where you left off if you are interrupted.



It is good practice to save your file regularly as you work. When working on complex parts, you can use the Save As command on the File menu to make incremental copies of your work, saving as "file_01.ONECNC", "file_02.ONECNC", and so on.

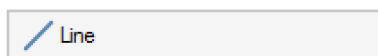
It is strongly recommended that you save a backup of all your work in another location once a day, and in an offsite location every week.



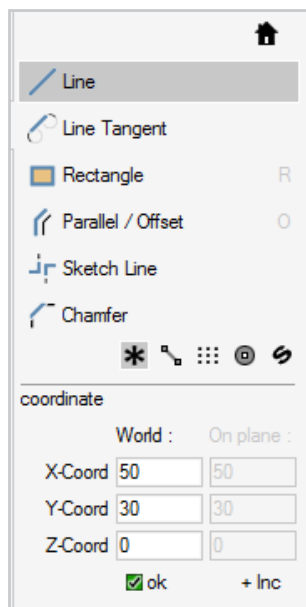
The part will open in the default 3D view. The darker rectangle is the construction Plane, which indicates the XY plane of the coordinate system you will be drawing in. Later you will see how you can move and rotate the Plane to define a new coordinate system but for now we will leave it in the default XY position.

Lines

We will start with lines, the most common drawing entity.



Click on the Line icon to open the Line menu in the toolbox.

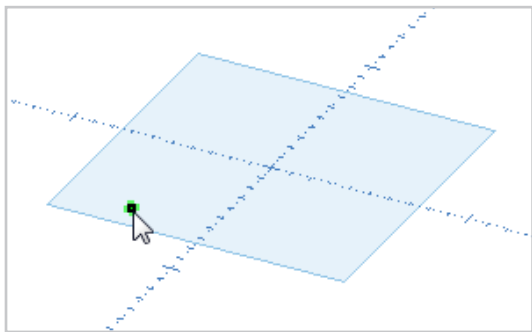


The Line icon in the Line menu is highlighted, showing that the Line command is started automatically.

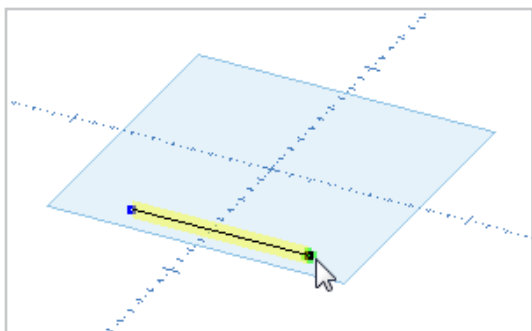
When the Line command is active, coordinate entry boxes will be available in the toolbox. Coordinate entry can be used for the first or second point of any line segment.

You can enter coordinates for a position in the World coordinate system, or the coordinate system of the current plane.

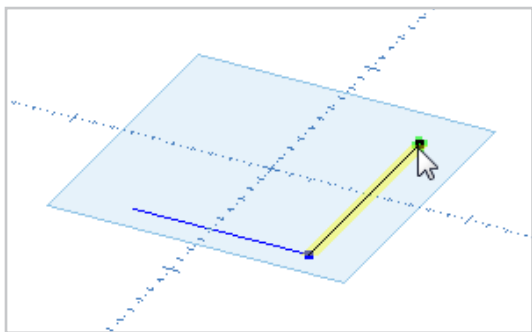
The current mouse pointer position is displayed in the toolbox, so you can also use it to check the position for a grid snap.



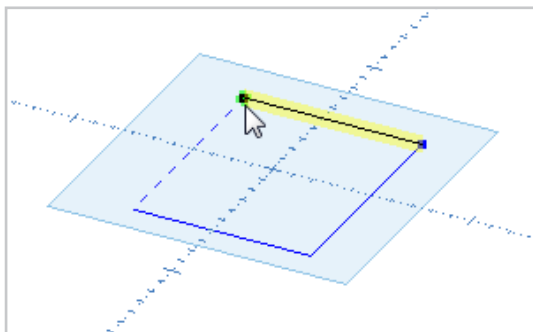
The mouse pointer will snap to grid positions in the drawing plane. Click to select a start point for the new line.



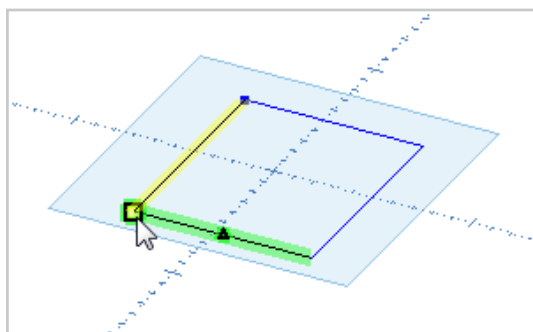
The mouse pointer will have a preview of the line to be drawn attached to it. Click on a new grid point to draw a horizontal line.



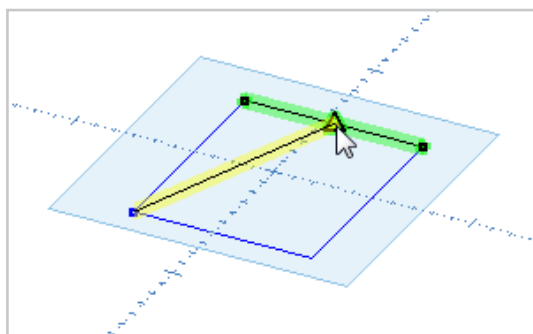
The Line command draws consecutive lines. Click again to draw a vertical line from the last point.



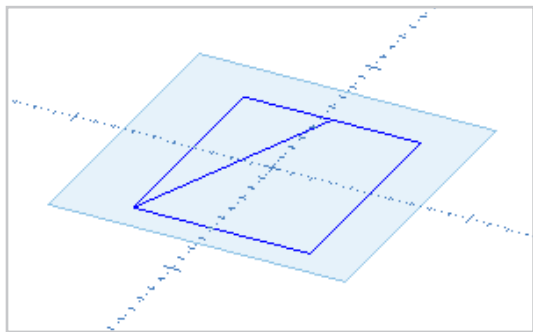
Move the mouse pointer horizontally again and you will see an indicator line when the new line endpoint is aligned with the start position. Click to end the line on the vertical alignment.



Hold the mouse pointer near the start of the chain of lines and you will see the Endpoint indicator. Click to end the new line on the first line endpoint.

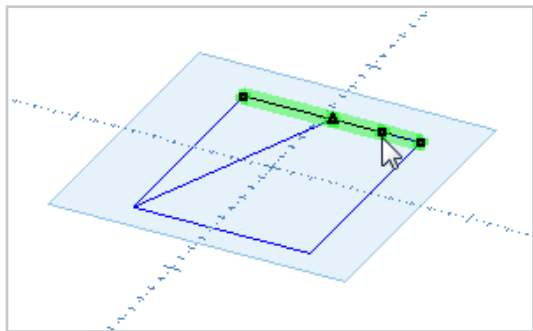


Hold the mouse pointer near the previous line and you will see the Midpoint indicator. Click to end the new line on the previous line midpoint.



Right click to end the chain of lines.

The Line command will still be active. You can start a line at a new position and continue as before.



Click near a line you have already drawn. The start point will be on the existing line.

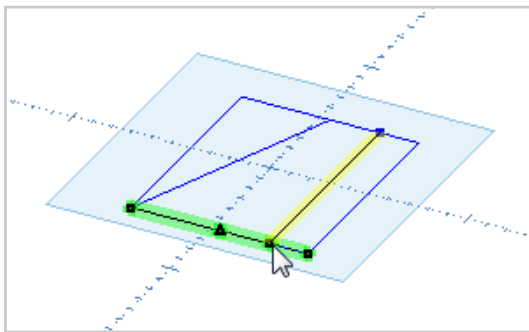
coordinate	
World :	On plane :
X-Coord <input type="text" value="10"/>	<input type="text" value="10"/>
Y-Coord <input type="text" value="20"/>	<input type="text" value="20"/>
Z-Coord <input type="text" value="0"/>	<input type="text" value="0"/>
<input checked="" type="checkbox"/> ok	+ Inc
parameters	
Line Length <input type="text" value="0"/>	
Line Angle <input type="text" value="0"/>	
adjust along position	
Distance along <input type="text" value="15"/>	
Ratio along <input type="text" value="0.3"/>	

The toolbox will display the distance the new start point is along the existing line.

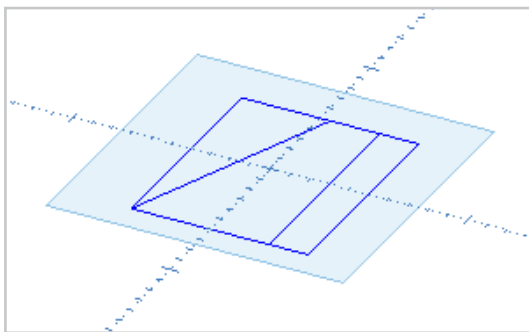
Change the value in the Distance along entry box to move the start point to a new position along the base line.

If you prefer you can enter the position on the base line as a ratio of the base line length.

The Distance along and Ratio along values are interactive.



Move the mouse pointer near another line and click to connect to it.



Right click twice to end the Line command.



Click on the Select All icon to select the lines you have just drawn.



Click the delete icon to clear the drawing area.

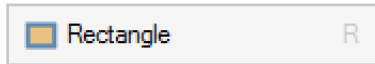


Start the Line command again and practice drawing lines. You can enter an exact position for a line endpoint by entering coordinates in the toolbox. You can also define a line by entering a value for line length, or angle from the last line segment.

Delete the lines when you are ready to continue.

Rectangle

The Rectangle command creates a rectangle with the option of a given radius fillet on all the corners.



Click on the Rectangle icon in the Line menu to start the Rectangle command.

coordinate

World : On plane :

X-Coord 0 0

Y-Coord 0 0

Z-Coord 0 0

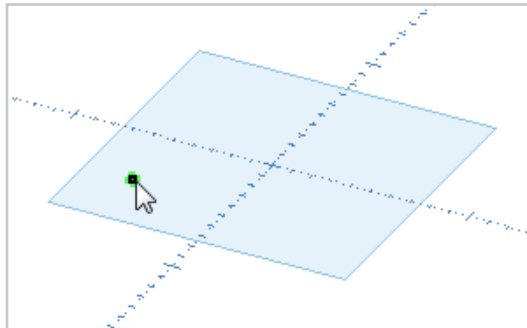
☒ ok

parameters

Corner Radius 5

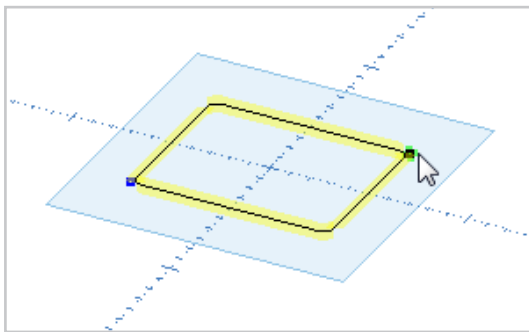
When the command is started, you have the option of entering a corner radius value.

Enter a Corner Radius of 5.

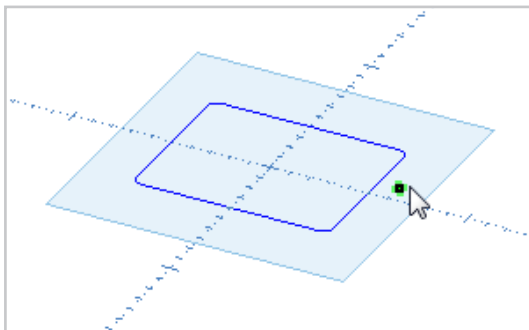


To define the first corner of the rectangle you can click on an entity or grid snap, or enter coordinates in the Toolbox.

This will be the point where the sides of the rectangle intersect before they are filleted.



A preview of the rectangle is shown, with the second corner attached to the mouse pointer. Click on or enter a position for the second corner. The second corner can also be set by entering an X length and Y width for the rectangle.



The mouse pointer is still active, ready to select a start corner for a new rectangle.

Draw some more rectangles, using different options such as a zero Corner Radius, or using values for Length or Width.



Right click when you are ready to end the command.



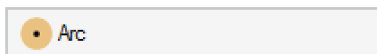
Click on the Select All icon to select the rectangles you have just drawn.



Click the Delete icon to clear the drawing area.

Circles

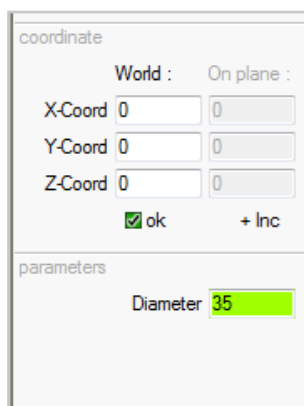
The Circle command creates a Circle with a given diameter.



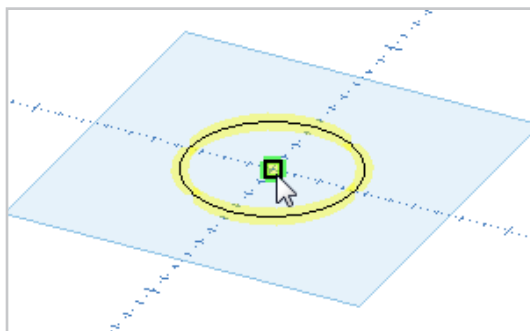
Click on the Arc icon to open the Arc menu in the toolbox.



The Circle icon in the Arc menu is highlighted, showing that the Circle command is started automatically.

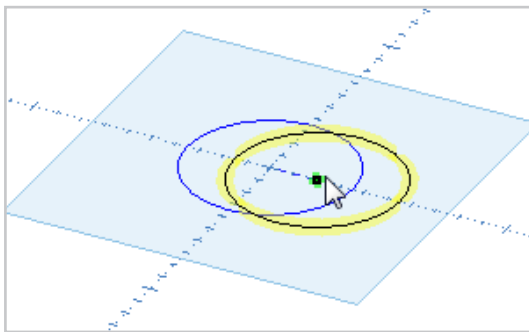
A vertical panel with a light gray background. The top section is titled "coordinate" in a small font. Below it are two columns of input fields: "World :" and "On plane :". Each column has three fields for "X-Coord", "Y-Coord", and "Z-Coord", all containing the value "0". Below these fields are two buttons: a green "ok" button with a checkmark icon and a "+ Inc" button. The bottom section is titled "parameters" in a small font. Below it is a single input field labeled "Diameter" containing the value "35".

Enter a Diameter for the circle in the Toolbox.

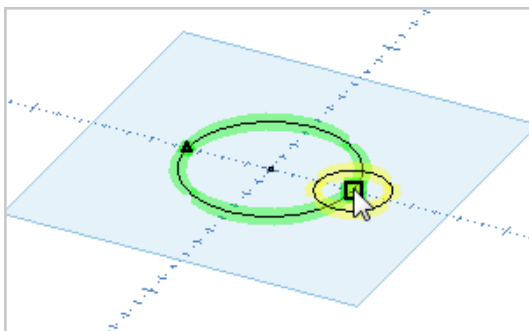


A preview of the circle is attached to the mouse pointer.

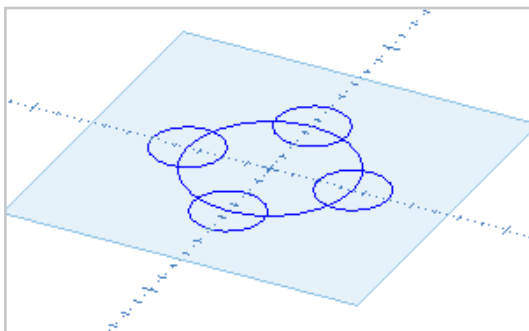
Click on or enter a position for the center of the circle.



The Circle command continues and you can place another circle by its center point.



You can change the Diameter in the Toolbox before placing the next circle.



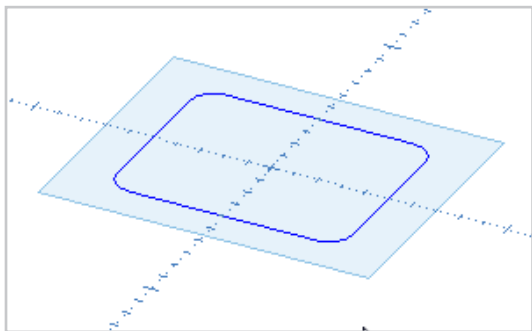
Right click when you want to end the Circle command.



Click Select All and then click the Delete icon to clear the drawing area.

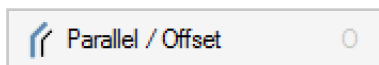
Offset entities and boundaries

An offset entity or boundary is often needed when drawing a part. In OneCNC this is done using the Parallel/Offset command.

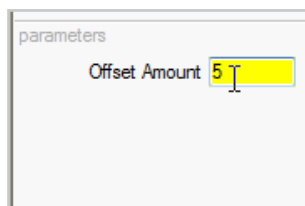


To practice the Parallel/Offset command, draw a rectangle with filleted corners.

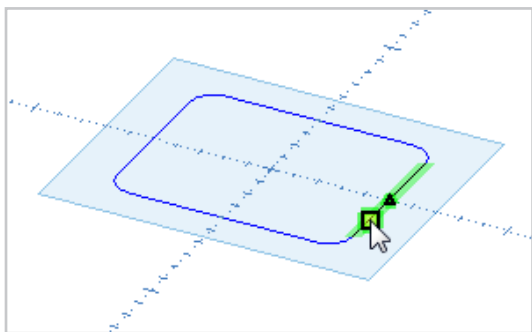
Offsetting single entities



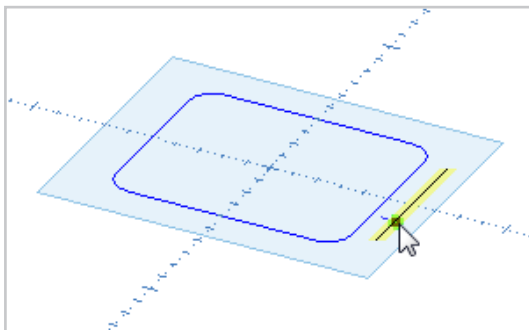
Click on the Parallel/Offset icon in the Line menu to start the command.



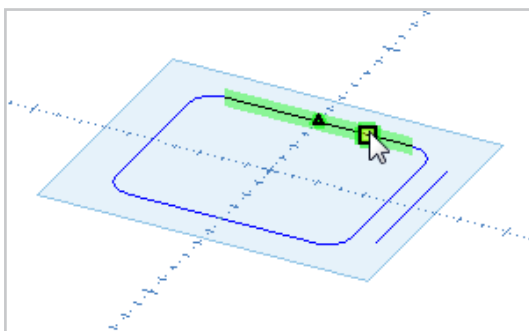
When the command is started, enter the distance you want to offset in the Offset Amount entry field in the Toolbox.



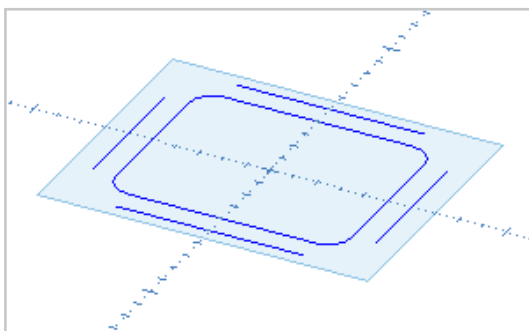
Select a side of the rectangle.



Move the mouse pointer around, and you will see a preview of the selected entity being offset. When the preview is where you want the offset, click to create the offset entity.

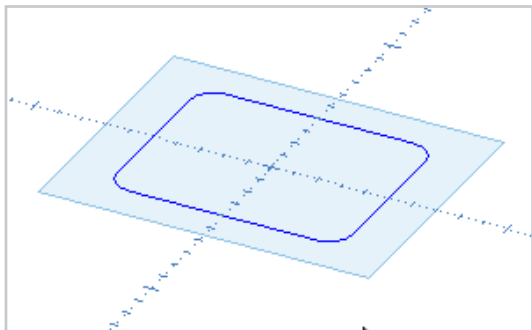


You can now select another single entity, and create an offset by clicking when the preview is where you want the offset.

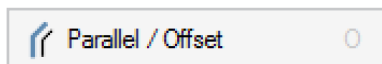


Right click to end the command when you have finished making single offsets.

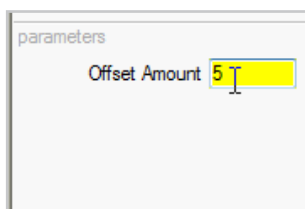
Offsetting a boundary



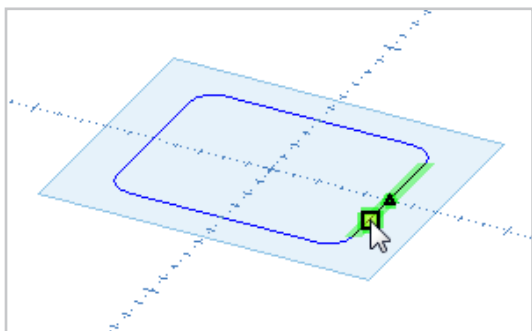
Delete the single offsets you made, or click on Undo till you return to the rectangle with filleted corners.



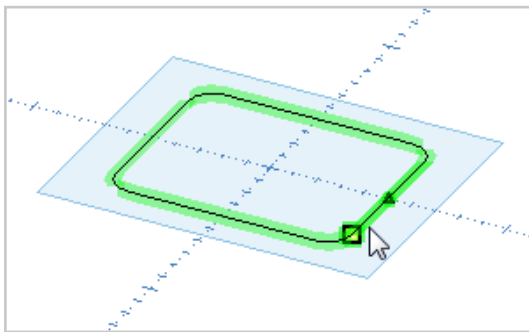
Click on the Parallel/Offset icon in the Line menu to start the command.



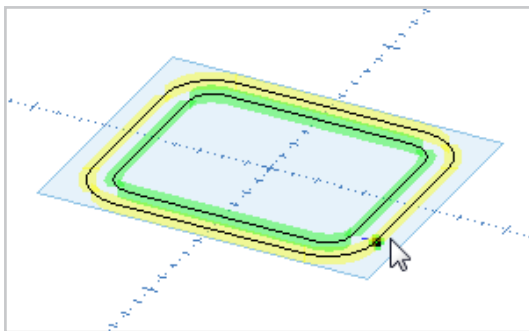
Enter the distance you want to offset in the Offset Amount entry field in the Toolbox.



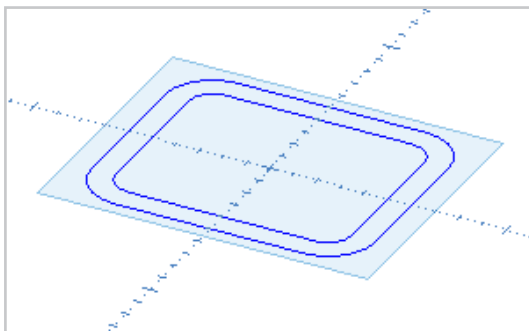
Select the same side of the rectangle.
The single offset preview will appear, but do not click to place it.



Move the mouse pointer near the end of the selected line, and the connected boundary will be highlighted. Click to select the boundary for offsetting.



Now the preview will show a preview of the entire boundary being offset. Click to place the offset. The command will recur.



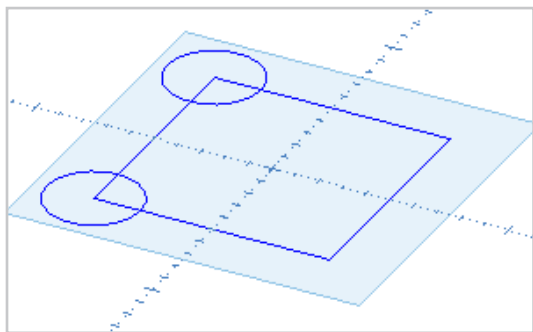
Right click when you have finished creating offsets.



Select All and Delete before continuing.

Moving Entities

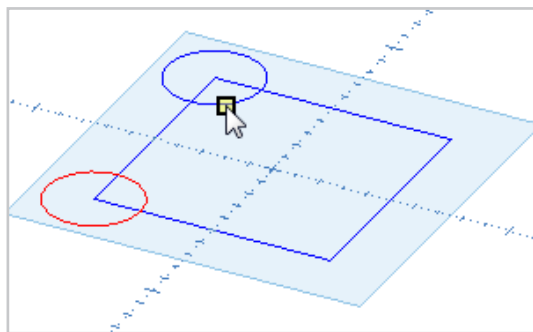
Entities can be moved, copied, scaled, rotated and mirrored. These Transform functions can be accessed by clicking on the Transform icon in the Toolbox to open the Transform menu.



We will start with the Move command. Use grid snaps to draw a square evenly spaced around the world origin, and draw two circles on the corners at the left of the square.



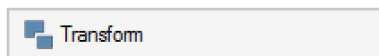
Before starting a Transform command, entities must be selected. Click on the Select icon to select entities one at a time.



Click on each of the circles to select them. Selected entities change color to red when they are selected. If you select an entity by mistake, click on it again and it will be removed from the selection set.



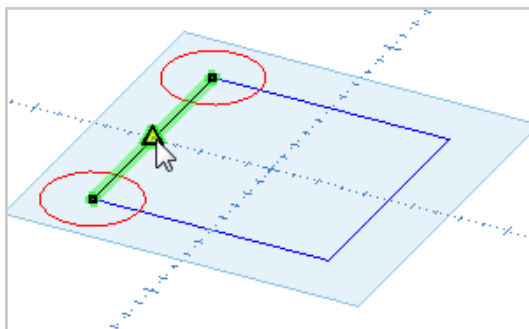
Right click to finish selecting.



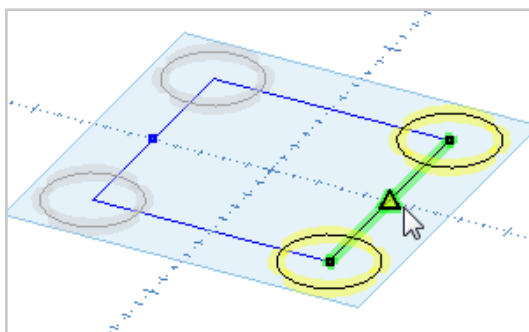
Click on the Transform icon to open the Transform menu in the toolbox.



The Move icon in the Transform menu is highlighted, showing that the Move command is started automatically.



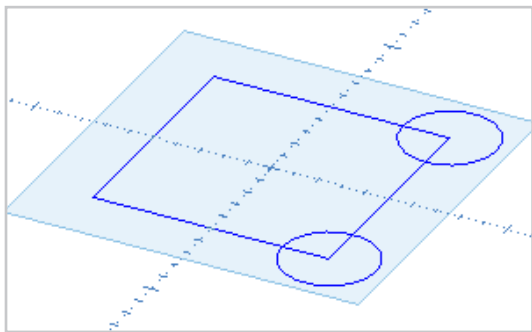
You can select or enter a position as the start point for a move. Click on the midpoint of the line between the circles.



A preview of the entities being moved will be attached to the mouse pointer. Shadows indicate the original position of the entities. Click on the midpoint of the opposite line to move the circles.

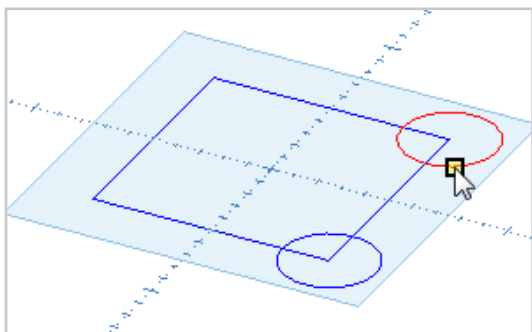


Right click to end the Move command. The moved entities are still selected, ready for another command.



Click Deselect All to leave the circles unselected.

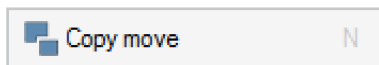
Copy Entities



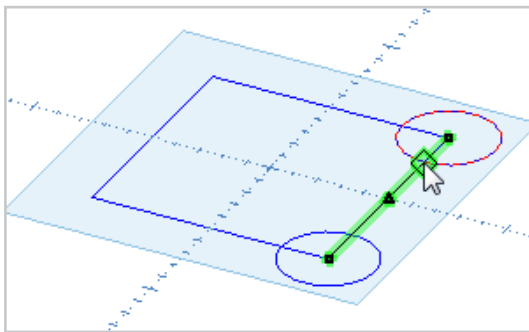
Click on the Select icon, and select the circle at the top of the square.



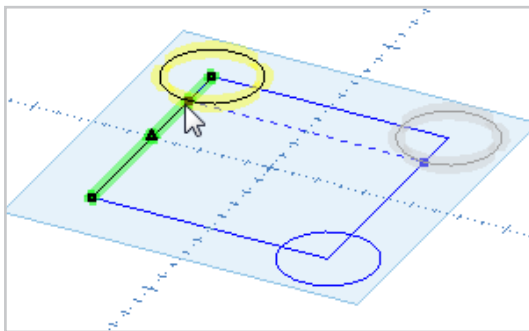
Right click to end the selection.



Click on the Copy Move icon in the Transform menu of the toolbox to start the Copy command.



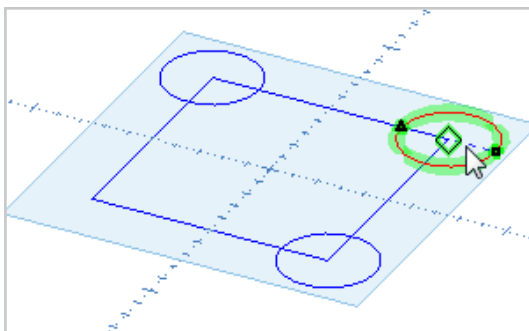
Select the intersection of the circle and the right side of the square as the start point of the move.



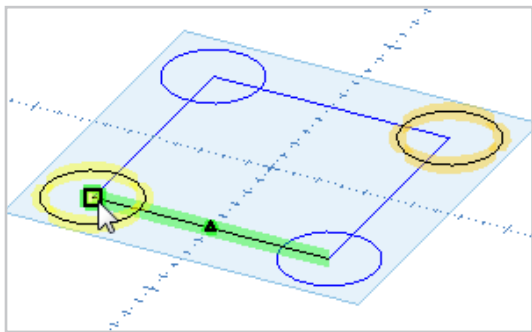
Move the mouse pointer to the left side of the square. Click on the intersection of the left side of the square and the dashed horizontal indicator to set the finish point of the move.



Right click to end copying from the current start point.



The Copy move command is still active. Select the center of the circle as the start point for another copy.



Click on the diagonally opposite corner of the square to create a copy there.



Right click to end the current copy.



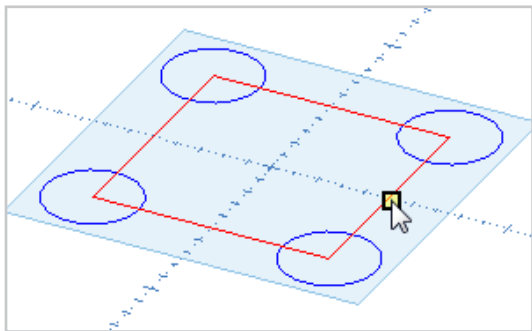
Right click again to end the Copy move command.



Click Deselect All to leave the original circle unselected.

Rotate

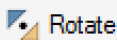
Entities can be rotated about a selected center of rotation. The axis of rotation can be parallel to the X, Y, or Z axis of the World or Plane coordinate system.



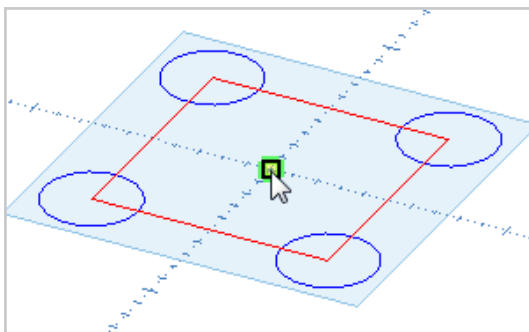
Select all the sides of the square.



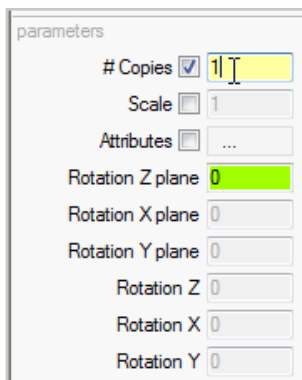
Right click to end the selection.



Click on the Rotate icon in the Transform menu.



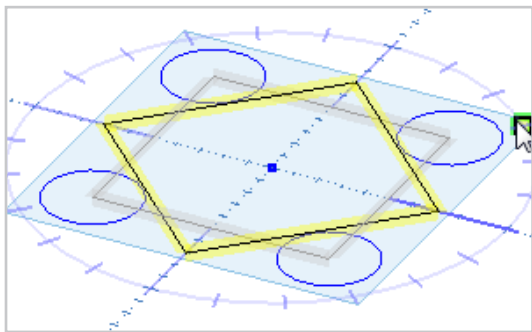
To select the center of rotation, move the mouse pointer to the intersection of the X axis and Y axis. Click when the square World origin indicator appears.



To create the rotated square as a copy of the original, enter 1 for the #Copies value in the Toolbox.

The toolbox now shows six options for the axis of rotation. The default rotation mode is Rotation Z plane, which rotates entities in the current drawing plane.

You can enter the exact degrees of rotation in any of these fields, but for common rotation amounts a rotation protractor with markers at 15° intervals is shown in the drawing window.



Move the mouse pointer around the protractor, and click to rotate the square by 45°.



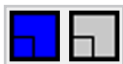
Right click twice to end the rotation command.



Click Deselect All to leave the original square unselected.

Setting Current Drawing Properties

OneCNC has a 2 color drawing system, controlled by color selection boxes in the Status Bar.



The color selector on the left is for geometry and that on the right is for surfaces and models. To set the current color, click the selector and select the color you want to use. To switch back to the default color just click the small square in the corner of the selector.



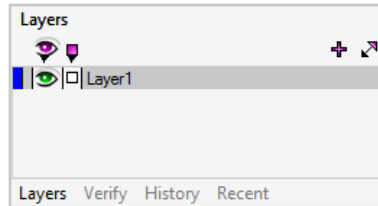
To select the line style and line thickness you would like all new entities to be drawn with, click the Styles icon.

OneCNC uses red to indicate an object is selected, so don't use red for drawing.

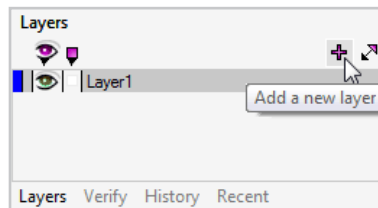
Using Layers

Every entity created in OneCNC is said to be 'on' a Layer. Layers give you control over entities on each layer as a group. For example, if you draw dimensions on a layer named 'Dimensions', you can then turn off display of the layer to hide all dimensions while working on the drawing geometry.

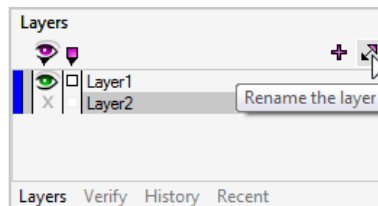
Click on the Layer tab at the lower right of the screen to open the Layer Manager.



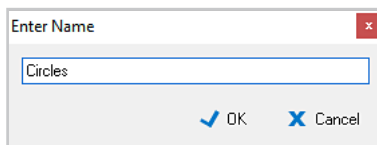
Up till now you have been working in a new file, on the default layer, Layer 1. We will now create a new layer and move some of the geometry to the new layer.



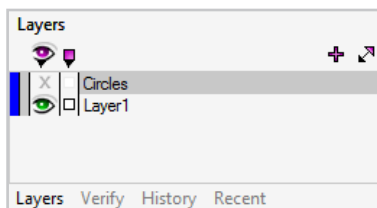
To add a new layer, click on the + icon at the top of the Layer Manager.



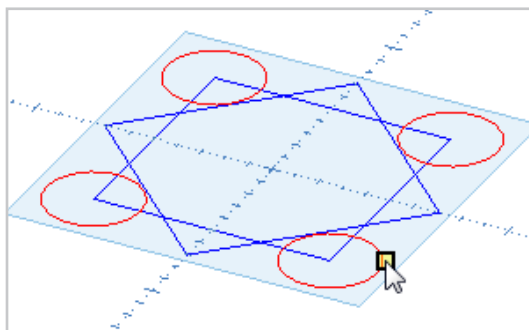
The new layer is created with the default name Layer 2. Click on the icon next to the + icon to rename the layer.



Enter 'Circles' as the name for the new layer, and click OK.



The Layer list is sorted alphabetically, so the Circles layer is now above Layer 1. It is highlighted in blue, indicating it is the active layer new entities will be created on, but the X in the visibility column shows it is an empty layer.



Select all the circles in your practice drawing.

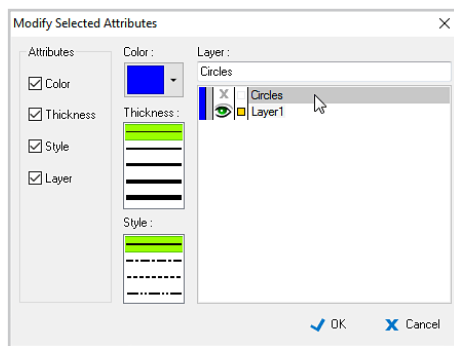


Click on the Modify icon in the Status Bar.

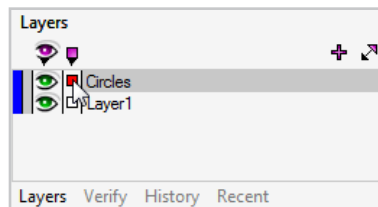
Default Layers



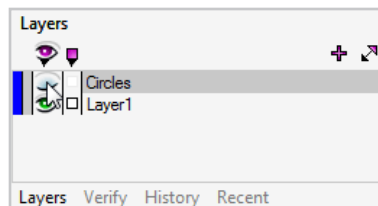
You can set default layers to be contained in each new drawing from the File menu > Properties dialog. Select the Layers tab, click New, give the layer a name, and set a default color, line type, and width.



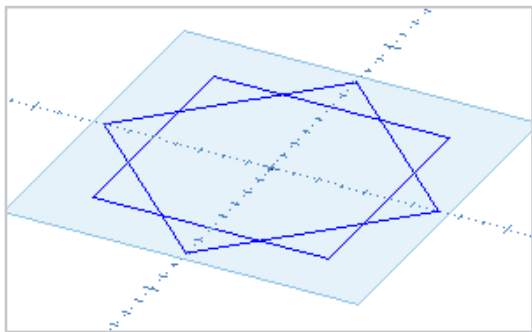
In the Modify Selected Attributes dialog, you can change the color, line type, thickness, or layer of entities. Select the Circles layer in the Layer list to move the selected entities to that layer.



There is now a green eye icon in the Circles layer visibility column, indicating all entities on the layer are visible. The red selection square indicates all entities on the layer are selected. Click on the square to deselect the circles.



Click on the open eye icon for the Circles layer and it will close. This will turn off display of all entities on the layer.



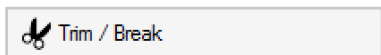
Now you will be able to see the squares clearly.

Trimming Entities

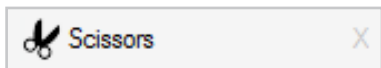
Trim functions are used to shorten or extend existing entities.

Scissors

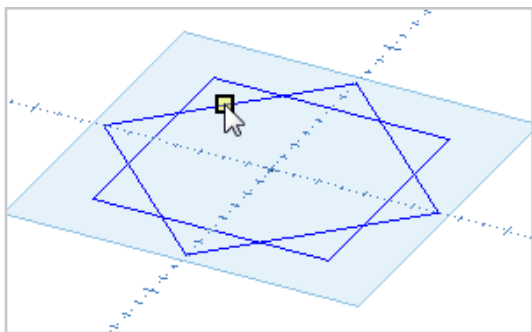
The quickest way to trim away unwanted parts of geometry is to use the Scissors tool.



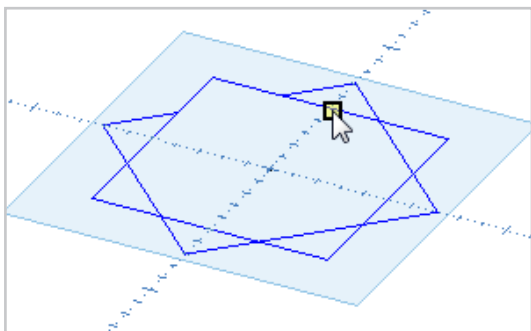
Click on the Trim/Break icon in the Toolbox to open the Trim/Break menu.



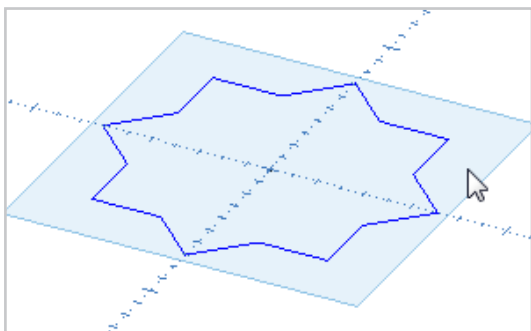
Click on the Scissors icon to start the Scissors command.



Click on an unwanted part of a line or arc.



The geometry will be trimmed back to the next intersection with another line or arc, and the command continues. You can continue trimming by clicking on unwanted parts of geometry.



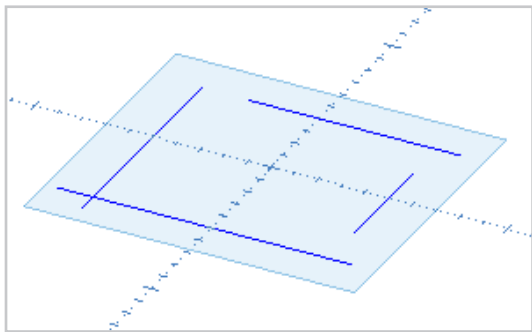
Right click to end the command when you have finished trimming entities.



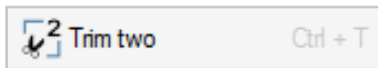
The Scissors command cuts entities back to the next intersection or endpoint in each direction from the point where you clicked on the entity. If you click on an entity which is not crossed by another line or arc, it will be deleted.

Trim Two

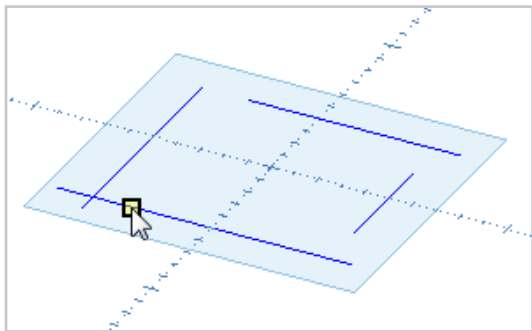
This function will shorten or extend two arcs or lines to their common intersection. It is an important function as it can be used to make sure a chain of entities are properly connected at their endpoints.



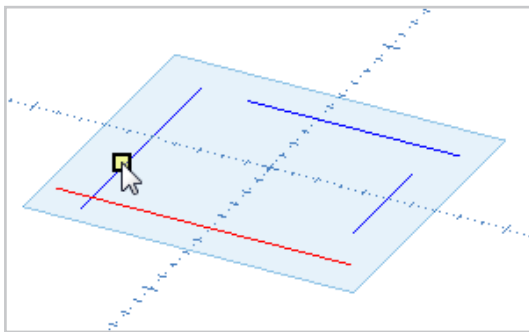
Open a new file to practice using the Trim Two command. Draw some lines that cross, and some unconnected lines.



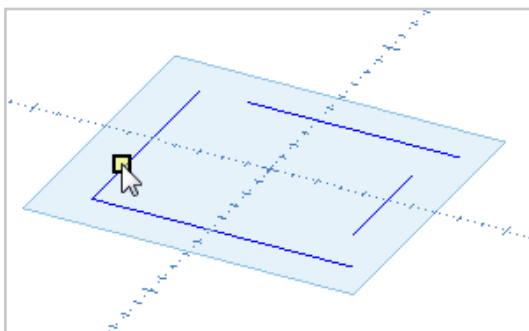
Start the Trim Two command by clicking the Trim Two icon in the Trim menu of the Toolbox.



Click on the first line you want to trim, on the side of the intersection that you want to keep.

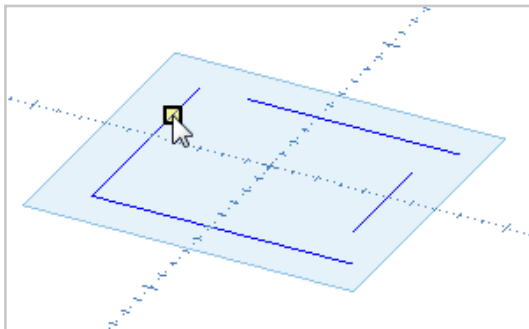


Click on the second line you want to trim, on the side of the intersection that you want to keep.

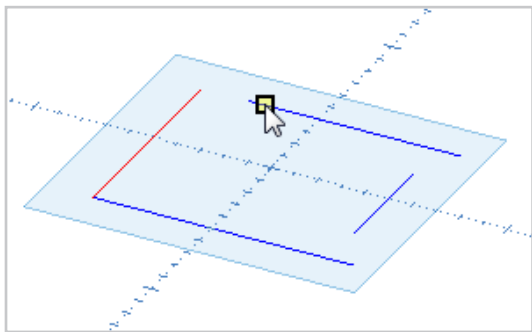


The lines will be trimmed to their intersection. In this example both lines were shortened.

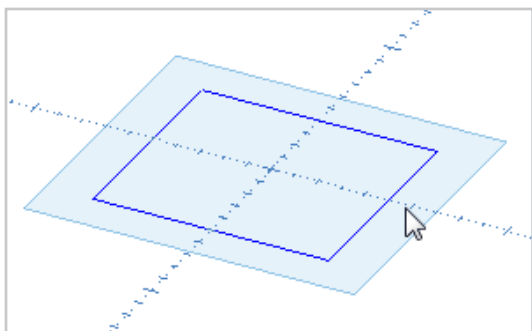
The command continues, and you can click on the next line you want to



trim.



Click on the next line to trim. In this example one line will be shortened and one will be extended to meet at their intersection.



Right click to end the command when you have finished trimming. The four lines in this example have now been trimmed to form a closed boundary.



Click Select All and then click the Delete icon to clear the drawing area.

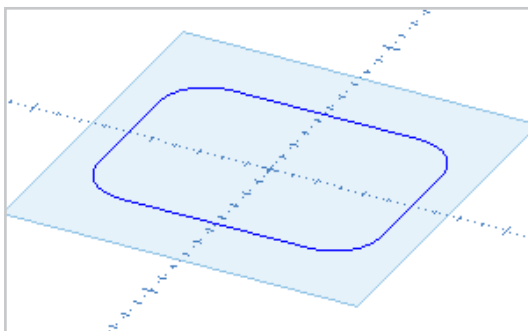


Some CAM operations require selection of consecutive entities as a profile chain.

Trim Two ensures that the junction between two entities is properly trimmed for chain selection, and can be used to repair a chain that will not select properly.

Chain Selection

Chain selection is an important function that selects consecutive entities to form a profile chain. A profile chain can be open or closed. For many CAM functions you will need to select a profile chain of consecutive entities as the first step of the toolpath definition.

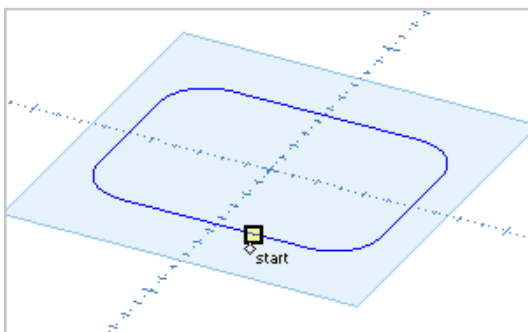


To practice chain selection, draw a filleted rectangle.

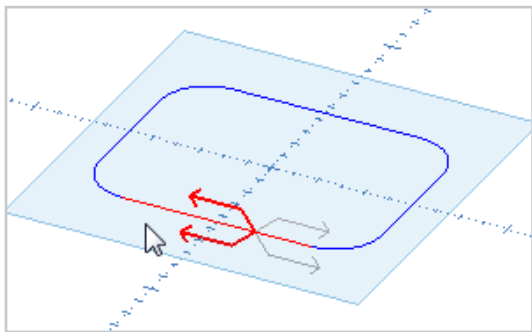
Selecting a complete profile as a chain



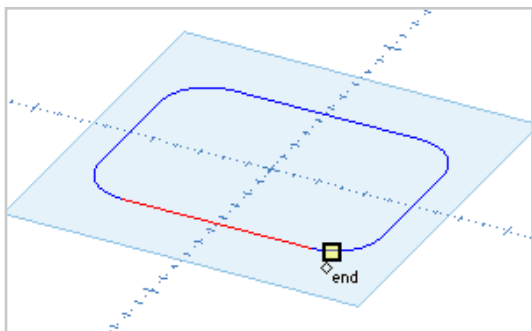
Click on the Chain Selection icon to enter chain selection mode.



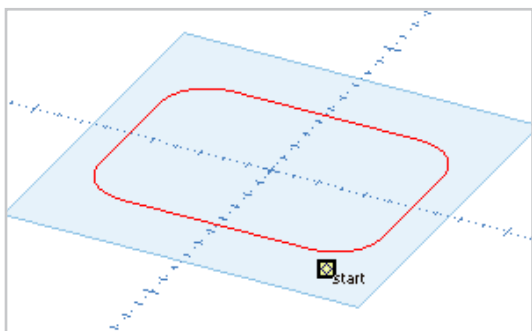
The mouse pointer changes to the word 'start'. Click on the entity that will be the first in the chain.



Move the mouse pointer around till the red arrows are pointing in the direction that you want to chain select. Click to set the direction.



The mouse pointer changes to the word 'end'. Click on the last entity of the chain you want to select, or press the F3 key to select all consecutive entities automatically.



The chain will be selected. You can now start to select another chain, or right click to end chain selection.

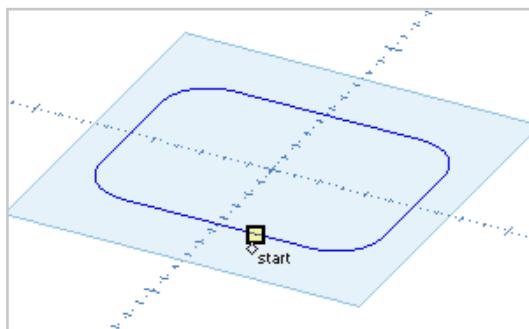
Selecting part of a profile as a chain



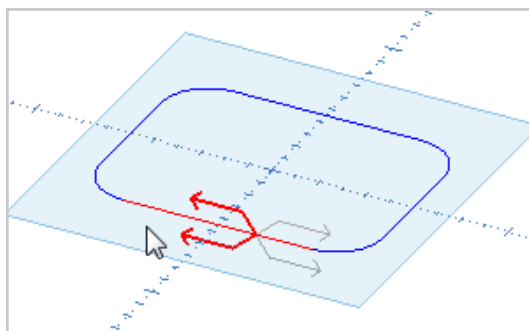
Click on the Deselect All icon to clear the previous selection.



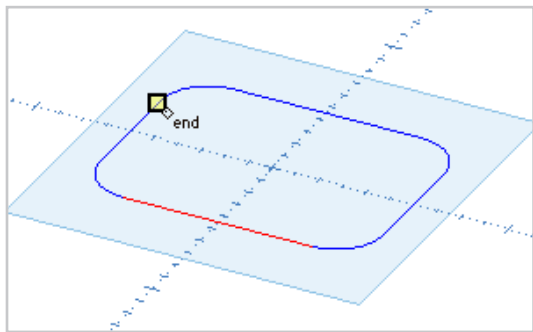
Click on the Chain Selection icon to enter chain selection mode.



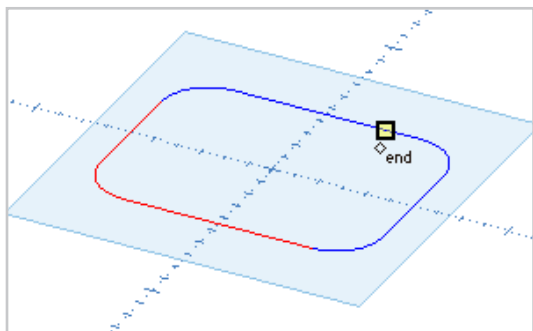
The mouse pointer changes to the word 'start'.
Click on the entity that will be the first in the chain.



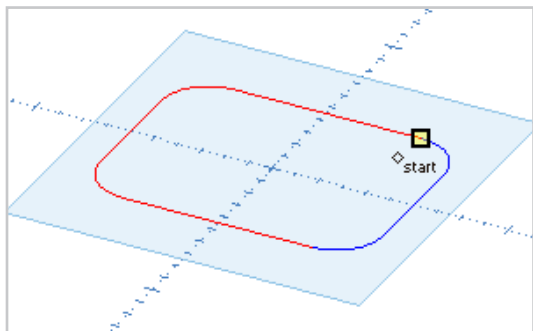
Move the mouse pointer around till the red arrows are pointing in the direction that you want to chain select. Click to set the direction.



The mouse pointer changes to the word 'end'. Click on an entity to end the chain.



The mouse pointer is still seeking a chain end. You can click further along the chain to add one or more entities to the chain selection.

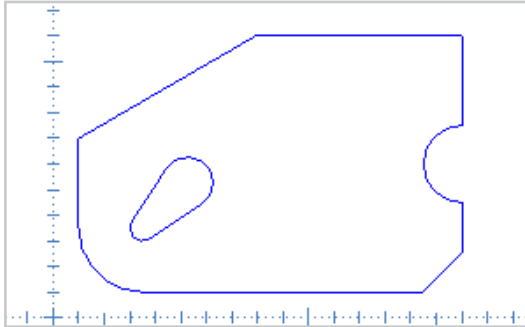


Right click when you are ready to finish selecting the chain. You can now start to select another chain, or right click again to end chain selection.

OneCNC CAD Tutorial 2

2D Drawing

This Tutorial is designed for all OneCNC products.



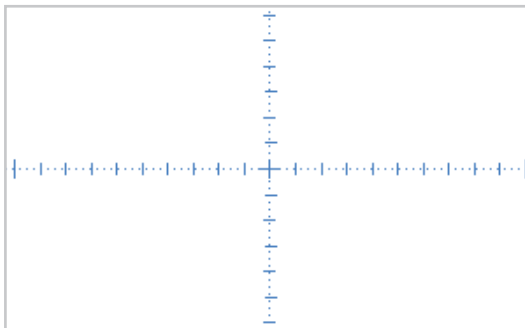
In this tutorial we will be using some of the time saving features of OneCNC CAD drawing to show how easily a part can be constructed. We will be drawing a part with fillets and chamfers, and a slot at an angle of 75 degrees.

Start a new file, and save it as 'Cad Tutorial 2'.



Click on the View icon in the Status Bar to open the View menu, and select CAD View.

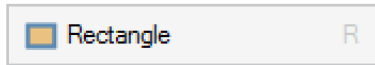
CAD view is a special top down view, specifically for 2D drawing in the



XY plane.

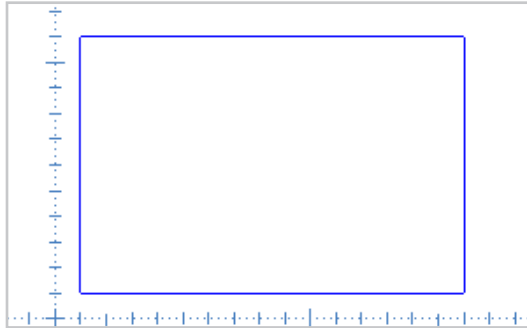
While in CAD view all drawing is in World coordinates.

Step 1: Draw outer profile



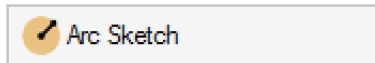
Click on the Rectangle icon in the Line menu to start the Rectangle command.

Enter 0 for Corner Radius, to draw the rectangle with sharp corners.



Enter coordinates in the toolbox or use a grid snap to start the rectangle at X10 Y10. You can then enter 150 for Length and 100 for Height to make the rectangle 150 wide in X and 100 high in Y.

To create the semi-circle on the right side of the part, start the Arc



Sketch command on the Arc menu.

Click on the midpoint of the vertical line at the right side of the part to



locate the center of the arc.

The Arc Sketch function now requires the arc Radius, Start Angle, and End Angle. Each of these can be defined by entering a value in the toolbox, or clicking on an entity or grid snap.

coordinate	
World :	On plane :
X-Coord	175
Y-Coord	60
Z-Coord	0
<input checked="" type="checkbox"/> ok	

parameters	
Radius	15
Start Angle	0
End Angle	0

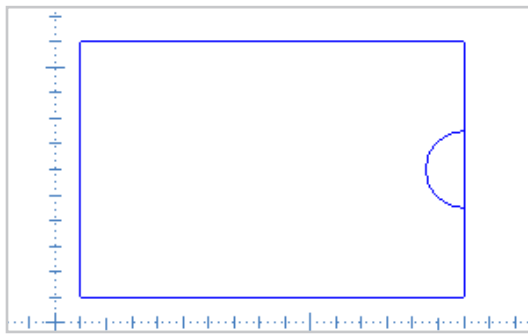
Enter 15 in the toolbox for the Radius of the arc.



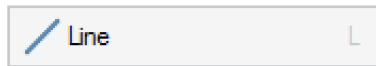
Click on the endpoint above the arc center to set the Start Angle to 90°.



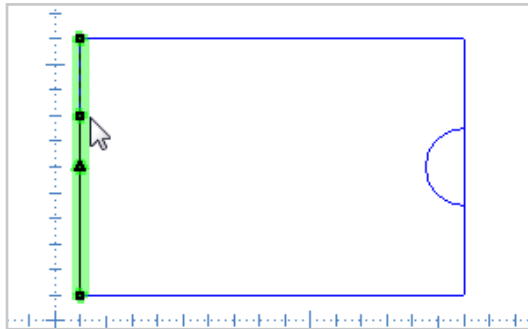
Click on the endpoint below the arc center to set the End Angle to 270°.



Right click to end the Arc Sketch command.



To draw the angled line at the top left of the part, select the Line menu to start the Line command.



Click on the vertical line at the left of the part, between the midpoint and upper end.

parameters	
Line Length	0
Line Angle	-90
adjust along position	
Distance along	40
Ratio along	0.4

To fix the start point of the line at 40mm from the end of the selected line, enter 40 as the Distance along value in the toolbox.

parameters

Line Length

Line Angle

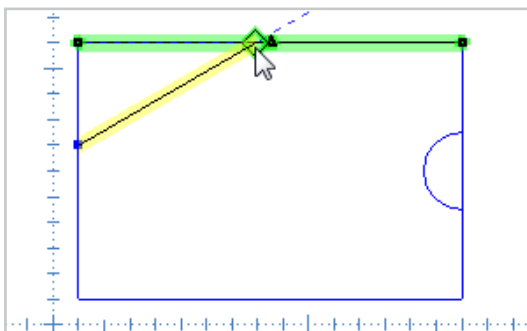
adjust along position

Distance along

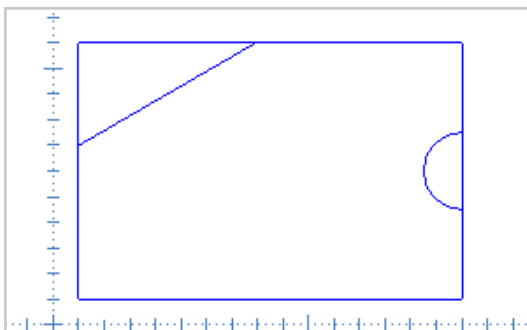
Ratio along

Enter -60 for the Line Angle.

This will fix the angle of the new line at 60° from the direction of the anchor line.

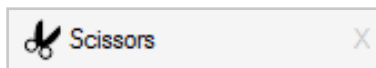


Move the mouse pointer along the new line preview until it locks onto the horizontal line at the top of the part, and click to draw the line.

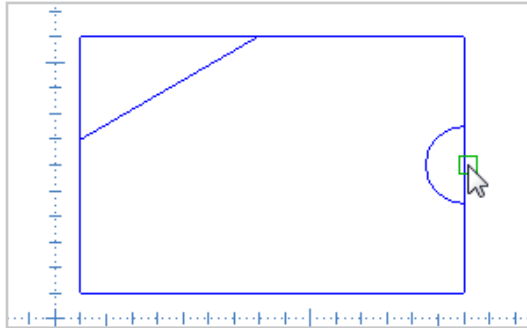


Right click to end the Line command.

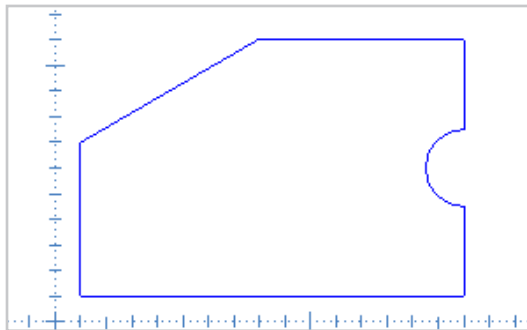
We can now trim away the unwanted line segments.



Start the Scissors command on the Trim/Break menu.



Click on the segment within the semicircle to remove it.

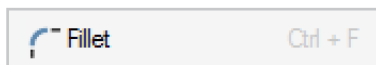


Click on the segments at the top left of the part to remove them.

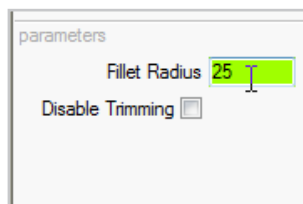


Right click to end the Scissors command.

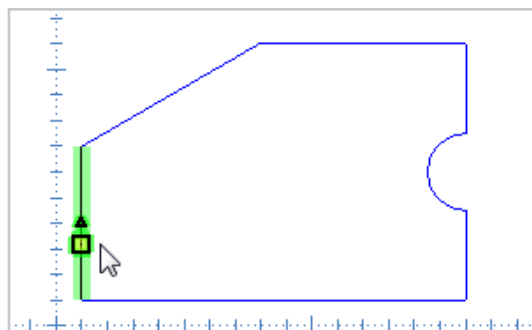
Next we will create the fillet on the lower left corner of the part.



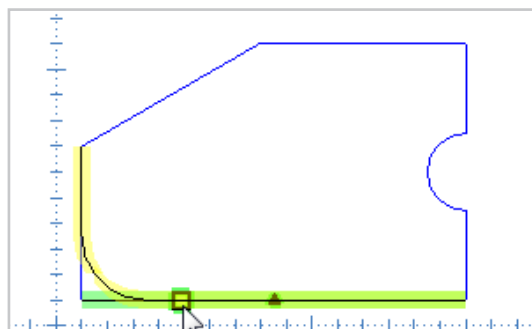
Start the Fillet command on the Arc menu.



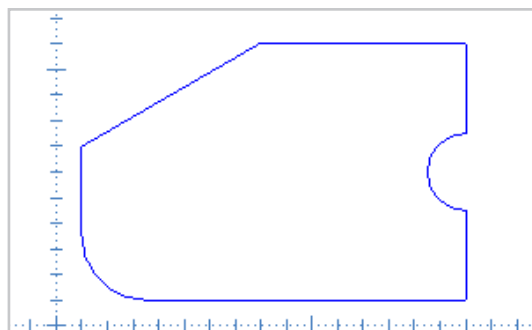
Enter a Fillet Radius of 25 in the toolbox.



Click on the left side of the part near the lower left corner.



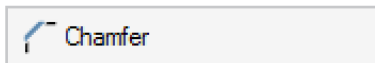
Click on the lower side of the part near the lower left corner.



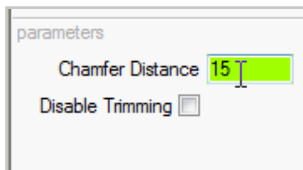
The fillet is created, and the corner is trimmed automatically.

Right click to end the Fillet command.

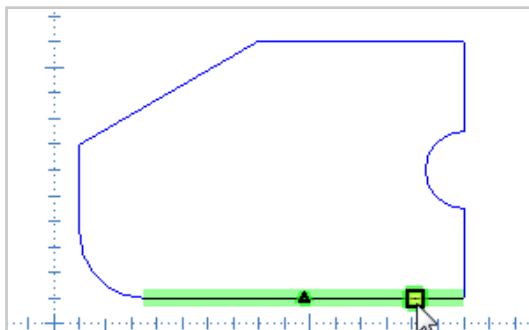
Next we will create the chamfer on the lower right corner of the part.



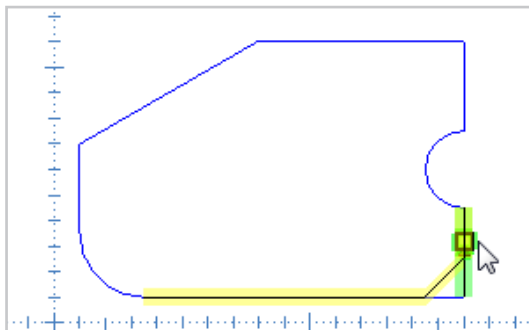
Start the Chamfer command on the Line menu.



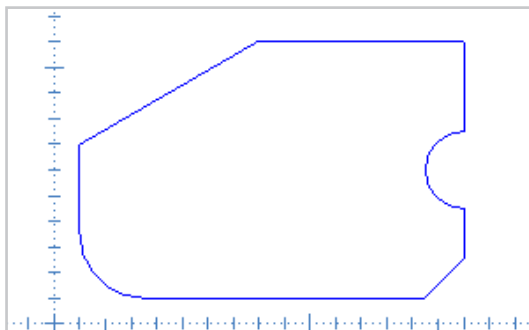
Enter a Chamfer Distance of 15 in the toolbox.



Click on the lower side of the part near the lower right corner.



Click on the right side of the part near the lower right corner.

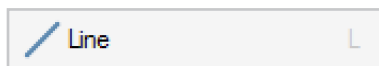


The chamfer is created, and the corner is trimmed automatically.

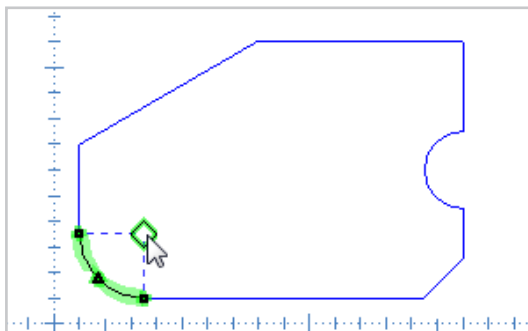
Right click to end the Chamfer command.

Step 2: Draw the angled slot

The first step in construction of the slot will be to draw the centerline.



Select the Line menu to start the Line command.



Move the mouse pointer near the center of the fillet arc, and click to place the start point of the line there.

parameters	
Line Length	25
Line Angle	0
adjust along position	
Distance along	0
Ratio along	0

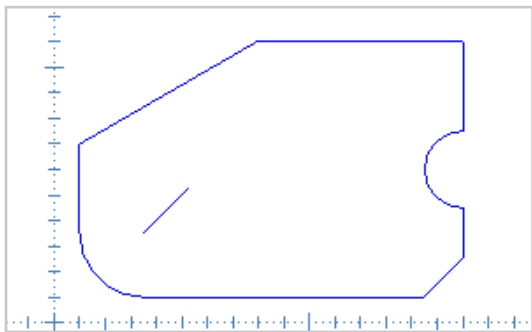
Type 25 in the Line Length box, and press Enter.

parameters	
Line Length	25
Line Angle	45
adjust along position	
Distance along	0
Ratio along	0

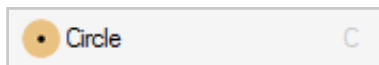
Type 45 in the Line Angle box and press Enter again.

The line will be drawn.

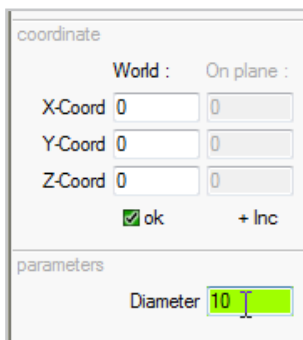
Right click to end the current Line sequence, and right click again to end the Line command.



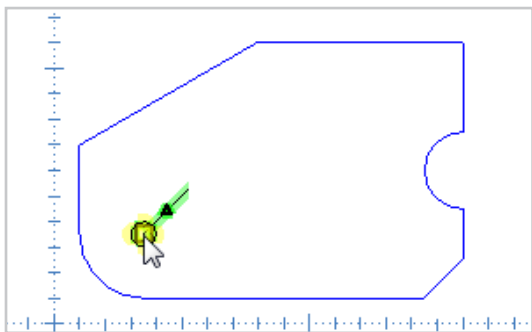
The part drawing should now look like this.



Start the Circle command to draw circles which will become the arcs at each end of the slot.



Type 10 in the Diameter entry box.



Click on the lower endpoint of the centerline to draw the first end arc of the slot.

coordinate

World : On plane :

X-Coord 35 35

Y-Coord 35 35

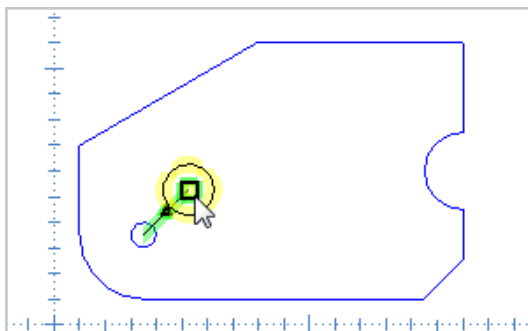
Z-Coord 0 0

☒ ok + Inc

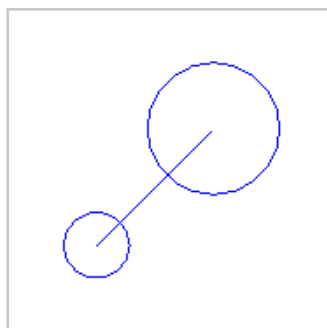
parameters

Diameter 20

Type 20 in the Diameter entry box.

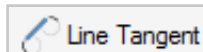


Click on the upper endpoint of the centerline to draw the second end arc of the slot.

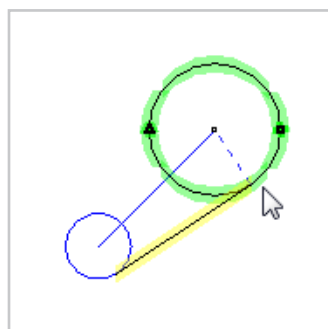
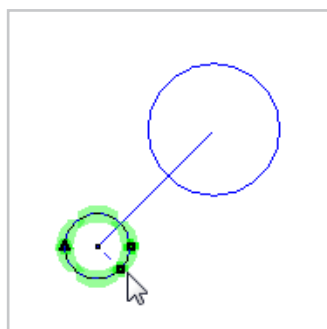


Right click to end the Arc command. You should now have two circles of unequal diameter, one at each end of the centerline.

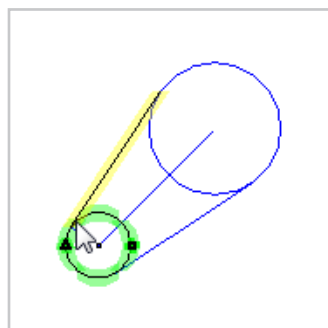
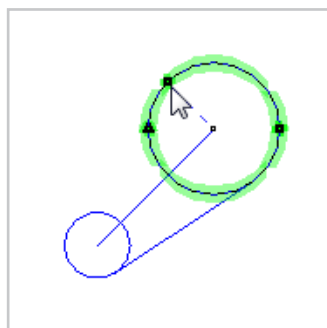
Next we will join the circles with tangent lines.



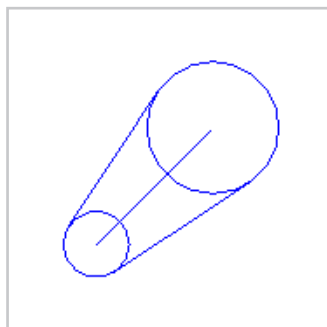
Start the Line Tangent command in the Line menu of the Toolbox.



Draw the first tangent line by clicking near where the tangents will be.



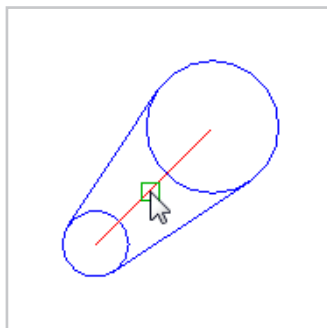
Draw the second tangent line by clicking near where the tangents will be.



Right click to end the Line Tangent command and you will have two tangential lines as shown here.



Begin the Single Selection command.



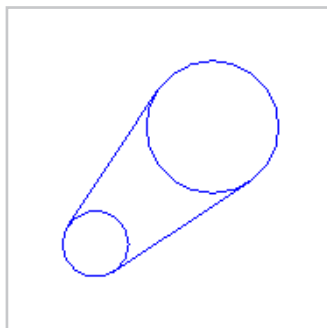
Select the 45 degree centerline.



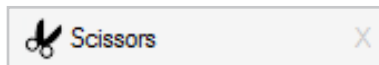
Right click to end the selection process.



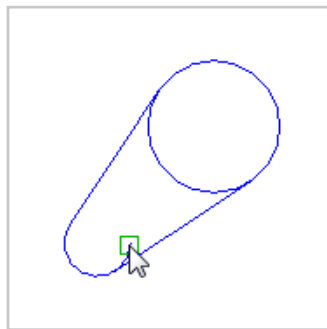
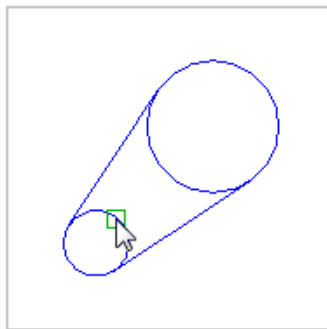
Click the Delete icon or press the Delete key to delete the centerline.



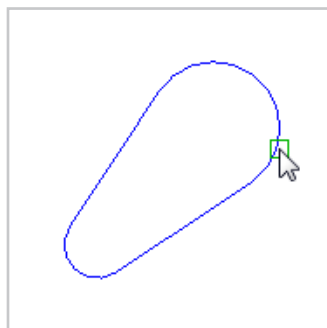
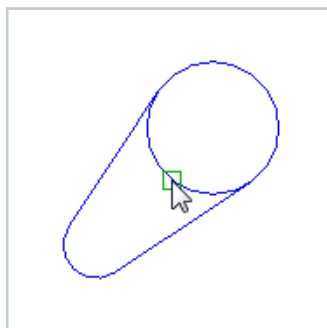
All we have to do now is trim the circles to the tangent lines to finish the slot.



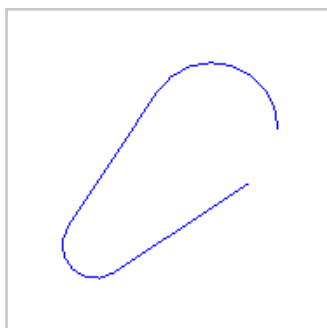
Start the Scissors command on the Trim menu.



Click on the unwanted segment of the lower circle to trim it away. Circles have an endpoint at 0° so you must click twice to remove both ends of it.



Click on the unwanted segment of the upper circle to trim it away. The remaining arc will be composed of two arcs because of the circle endpoint at 0° . It is good practice to trim this to one arc. Click below the 0° endpoint to remove the smaller arc.

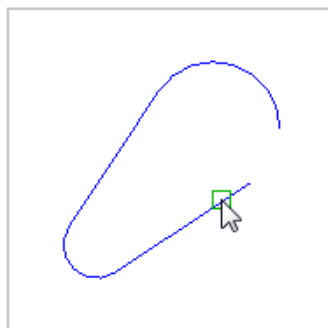


Because the smaller arc is not crossed by any other entity, the Scissors command removes it completely.

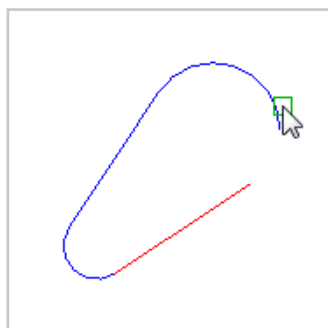


Right click to end the Scissors command.

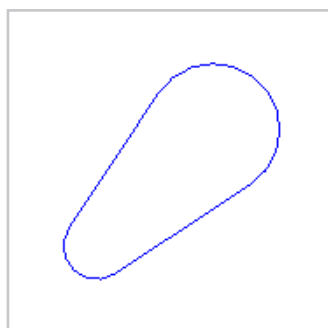
Start the Trim Two command.



Click on the tangent line to select it as the first entity to trim. Click between the endpoint and the midpoint of the line.



Click on the arc to select it as the second entity to trim.



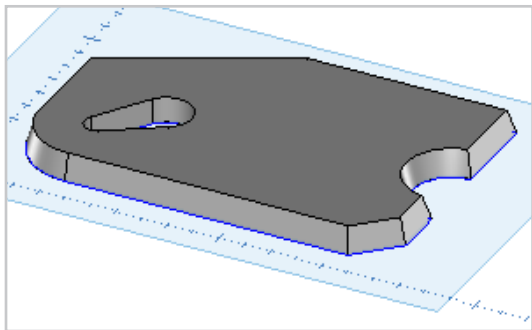
The two entities will now be joined perfectly.

Your drawing should now look like the picture at the beginning of this tutorial, which is now complete.

OneCNC CAD Tutorial 3

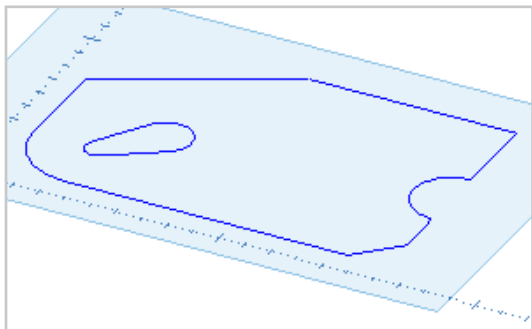
Solid Modeling by Extrusion

This tutorial is intended for Expert and Professional versions of OneCNC.



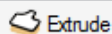
In this tutorial we will use OneCNC solid extrude commands to construct a 3D solid part. You will need the outlines drawn in CAD Tutorial 2 before attempting this tutorial. We will extrude the part with a taper angle and cut the through slot with vertical walls.

Step 1: Complete CAD Tutorial 2 and save a copy.

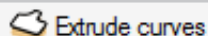


Complete 'CAD Tutorial 2' and save the file. Use Save As on the File menu to save a copy as 'CAD Tutorial 3'. Press the Space bar and set the view to Trimetric. Your part should appear as shown in this image.

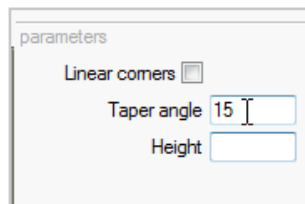
Step 2: Create a solid by extrusion from curves.



Click on the Extrude icon in the Toolbox to open the Extrude menu.

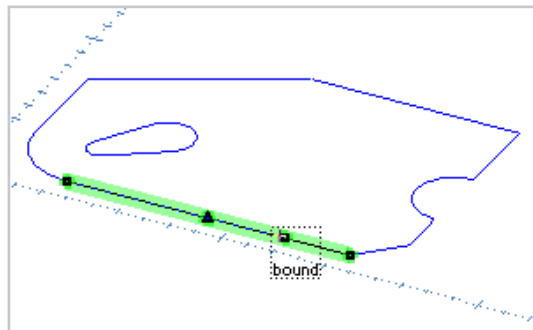


Click on the Extrude Curves icon.

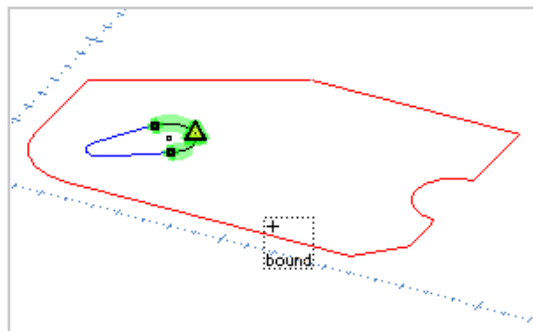


Enter a Taper Angle of 15 in the parameters section of the Toolbox. This will tilt the walls of the extrusion inwards.

Leave the Linear Corners check box unselected.



OneCNC now seeks a boundary to create an extrusion from. Hover the mouse anywhere near the external boundary and click to select it as the outline of the part. The entire outer boundary will be selected with one click.

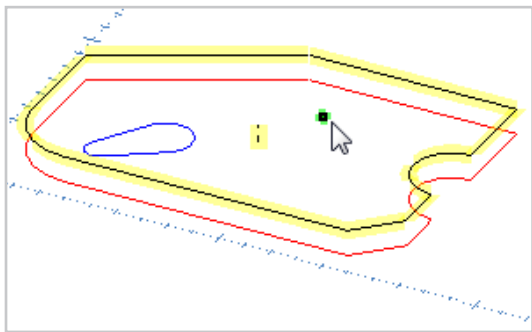


You can extrude a shape with one or more openings, so the cursor will continue to seek boundaries.

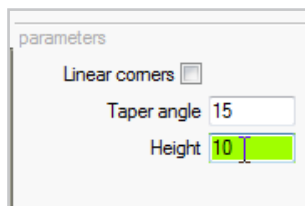
Do not select the inner boundary.



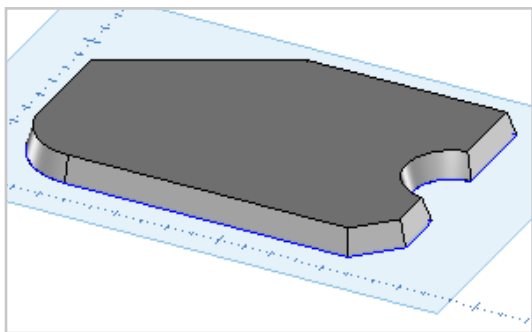
Right click to end the selection stage of the command because in this case we want to create the slot using different settings.



The height of the extrusion can be set by a mouse click or by entering the height in the toolbox. As you move the mouse, you will see a preview of the extrusion, and the current height will be displayed in the toolbox height entry.



Enter a Height of 10 in the parameters section of the Toolbox, and press the Enter key.



Click in the drawing window and the extruded solid is created with an external taper of 15 degrees.



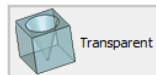
Right click to end the command.

Step 3: Create the slot with an extrude cut

The slot boundary we want to use is obscured by the solid we have made. We could change the view direction but there is no need to do this, as we can change the model display setting to Transparent.



Click on the Display icon to open the Display menu.

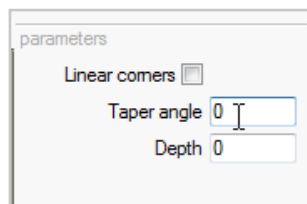


Select Transparent. The slot geometry will now be visible through the part.

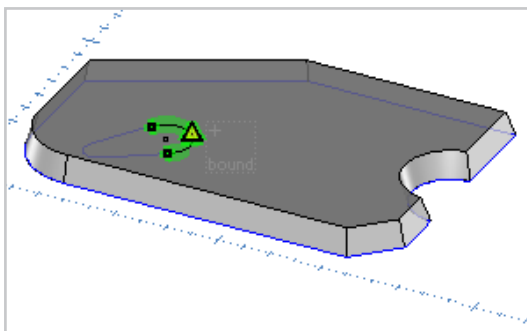
Click on the Extrude cut icon in the Extrude menu.



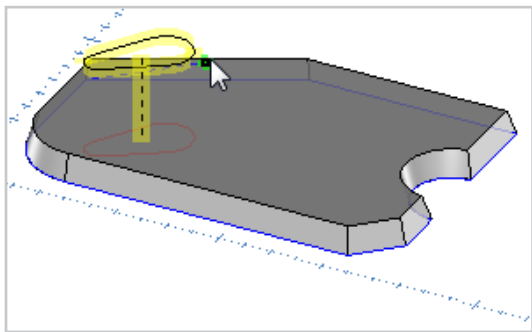
Extrude cut



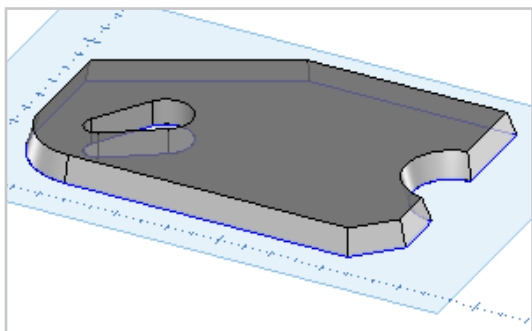
Change the Taper Angle to 0 so the slot will have vertical walls.



Select the internal boundary to create the cut extrusion. Only one boundary can be used for an Extrude cut so you do not have to right click to end the boundary selection.



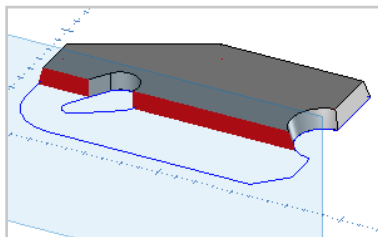
Because we want a through cut the exact extrusion distance is not important. Move the mouse pointer up until the extrusion preview is above the part, and click to make the extrude cut.



Right click to end the Extrude Cut command.



Click on the Display icon and change the display mode back to Normal.



To see how you can create a section view of your model, open the Section Tool topic in OneCNC Help.

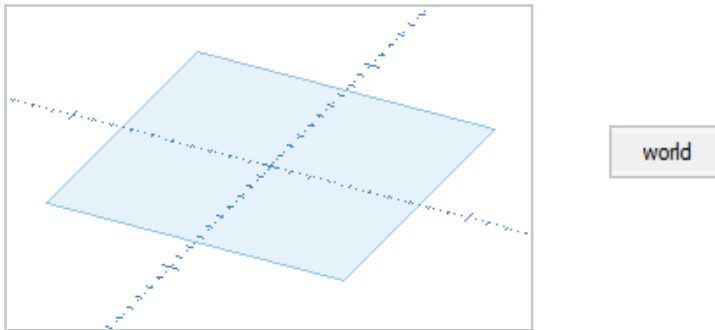
You can find the topic in Help Contents -> CAD Drawing and Modeling -> Modeling Tools -> Section Tool.

OneCNC CAD Tutorial 4

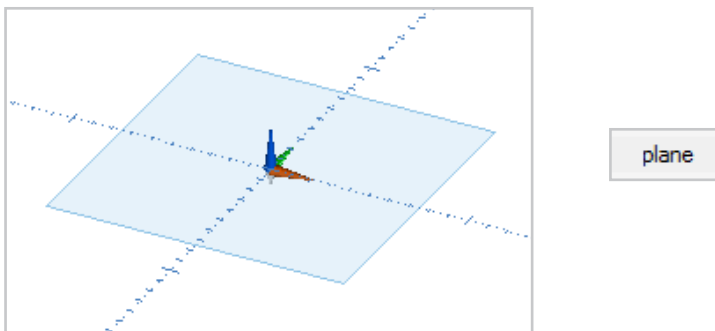
Working With Planes

All new line and arc geometry, and surface and solid models, are created in the current construction coordinate system, which is independent of the world coordinate system.

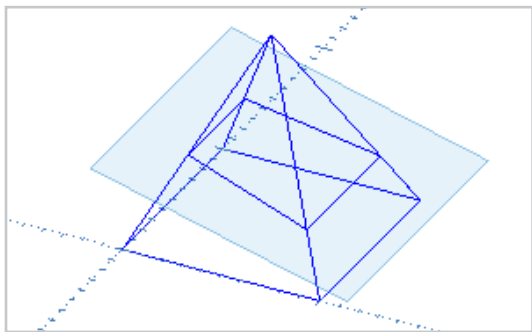
When you open a new file in OneCNC, the current construction plane is the XY plane. The X and Y axes of this plane coincide with the X and Y axes of the world coordinate system.



A transparent rectangle indicates the position and orientation of the XY plane of the current construction coordinate system. The Origin selector at the right of the Status bar shows World coordinate input is active.



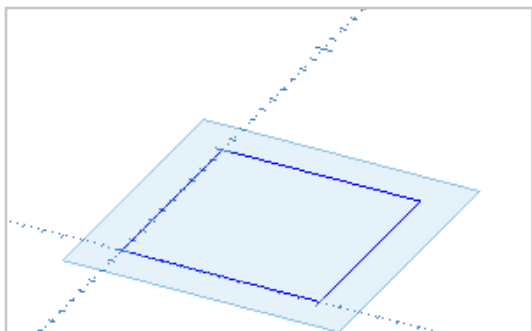
Clicking on the Origin selector will activate Plane coordinate input. A plane origin indicator will appear.



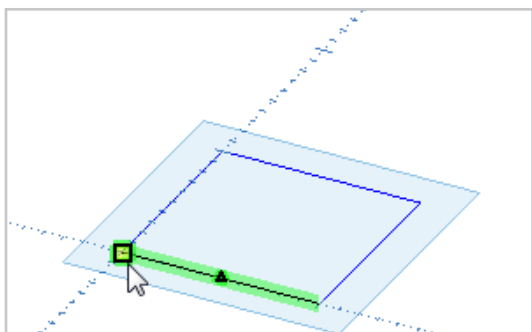
As an introduction to Plane functions we will position a Plane as an aid to constructing a truncated wireframe pyramid.

Step 1: Draw a pyramid wireframe

Start a new file, and save it as 'Cad Tutorial 4'.



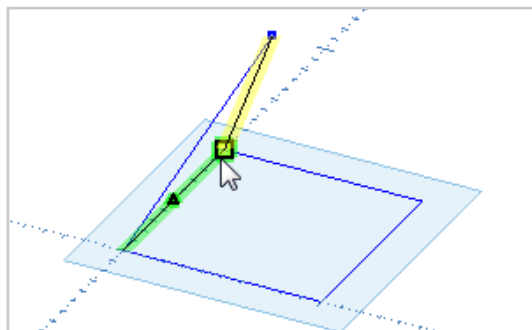
Select World coordinate input, and use the Rectangle command to draw a square from X0 Y0 to X100 Y100.



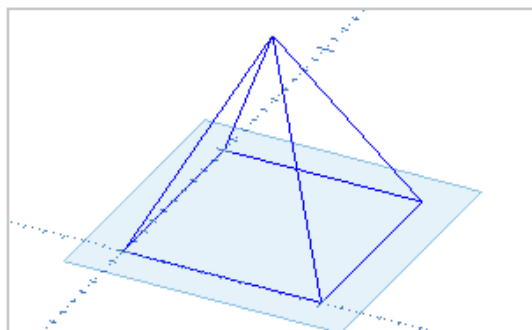
Start the Line command, and click on the corner of the square to start the first edge of the pyramid.

coordinate		
	World :	On plane :
X-Coord	<input type="text" value="50"/>	<input type="text" value="50"/>
Y-Coord	<input type="text" value="50"/>	<input type="text" value="50"/>
Z-Coord	<input type="text" value="100"/>	<input type="text" value="100"/>
<input checked="" type="checkbox"/> ok		+ Inc

Enter World coordinates as shown to end the edge at the vertex of the pyramid.



Click on an adjacent corner of the square to finish the second edge of the pyramid.

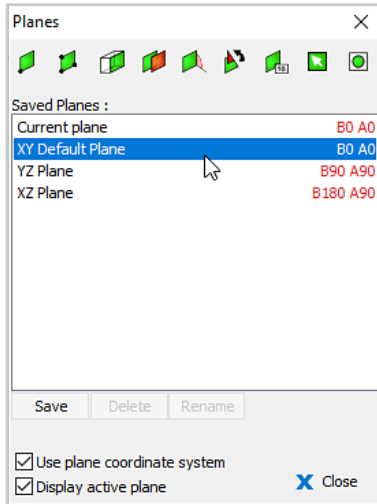


Right click to end the current Line sequence, and construct the other two edges by clicking on corner, vertex, and corner. Right click twice to end the Line command.

Step 2: Position the plane



Open the Planes dialog by clicking on the Plane icon in the Status Bar.



In the Planes dialog you will see the XY Default Plane is selected in the Saved Planes list.

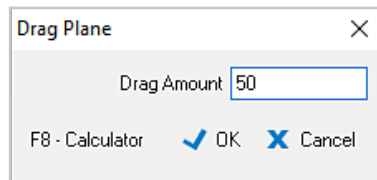
You will also see the pre-defined YZ and XZ planes.

The Display active plane check box must be selected for the plane to be visible.



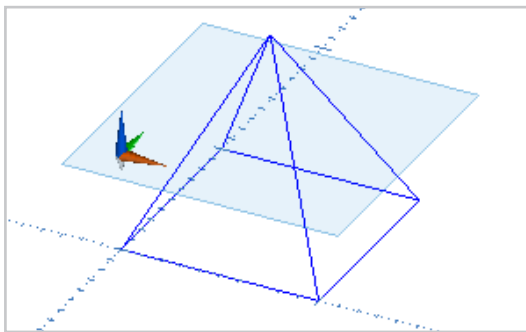
Click on the Push or Pull Plane icon in the Planes dialog.

This command will move the current plane up or down along its Z axis.



Enter a drag amount of 50, and click OK.

The plane will be moved vertically by 50mm.



plane

Click on the Origin selector to activate plane coordinates. The Plane origin indicator shows the origin of the moved plane.

We will now tilt the plane on its own Y axis to create the angle for truncation of the pyramid



Click on the Plane icon again to open the Planes dialog.



Click on the Rotate Plane icon.

Rotation Amounts

✕

X Rotation : 0

Y Rotation : 0

Z Rotation : 0

Plane X Tilt : 0

Plane Y Tilt : 15

Plane Z Tilt : 0

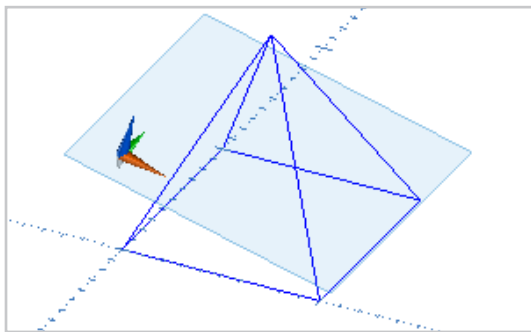
F8 - Calculator

✓ OK

✕ Cancel

The plane can be rotated around the World axes or its own axes.

To rotate the plane around its own Y axis, enter 15 in the Plane Y tilt entry box and click OK.



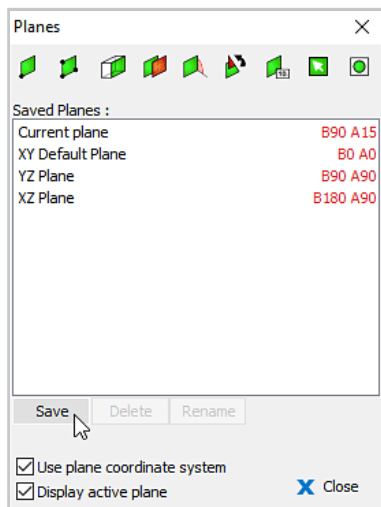
The plane is tilted on its own Y axis. The plane origin will still be directly above the World origin.

We will save this plane position so we can return to it easily.

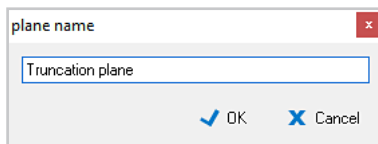
Step 3: Save the plane position



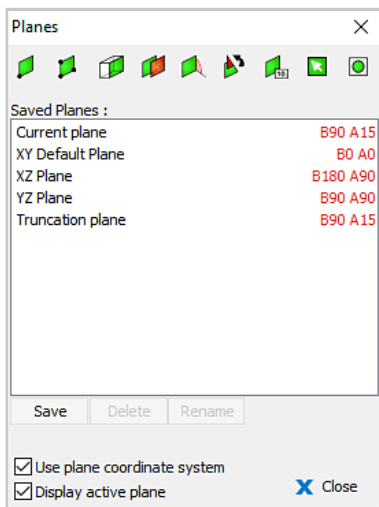
Click on the Plane icon again to open the Planes dialog.



Click on the Save icon.

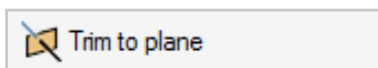


Name the plane Truncation plane and click OK.

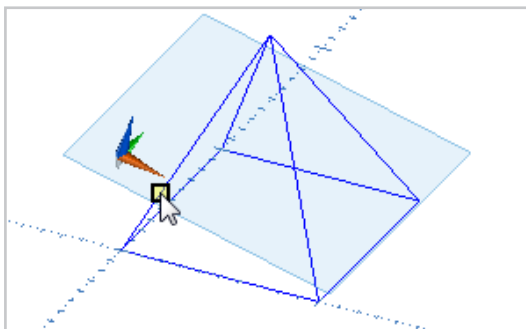


The plane now appears in the Saved Planes list.

Step 4: Create the truncated pyramid

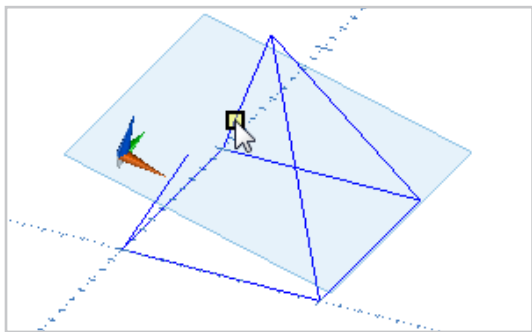


To truncate the pyramid edges, open the Trim menu in the Toolbox, and click on the Trim to plane icon. This command trims lines or arcs which pass through the plane to the plane intersection.

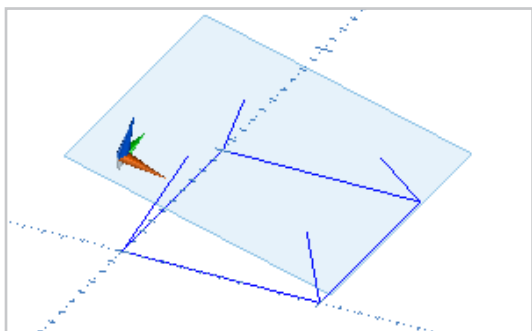


Click on an edge of the pyramid, between the base and the plane.

The edge will be cut off where it meets the plane. Click on the next edge to trim it.

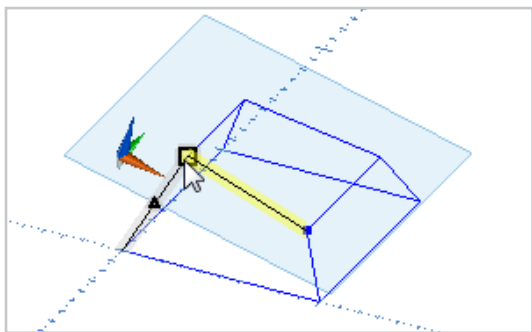


The edge will be cut off where it meets the plane. Click on the next edge to trim it.



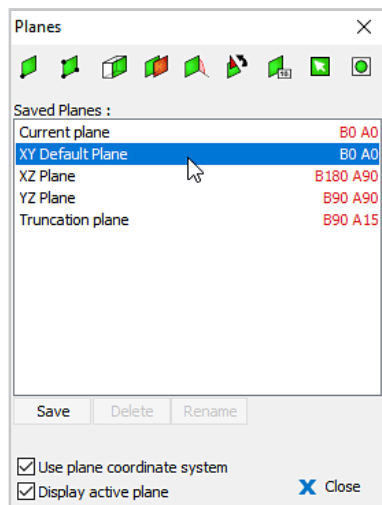
Continue until all four lines are trimmed to the plane.

When you have trimmed all four side edges of the pyramid, use the



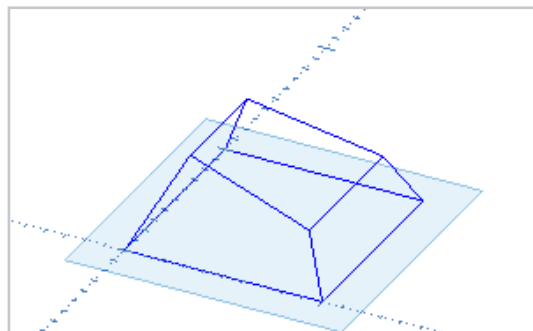
Line command to join them. These lines form the boundary of the truncation.

Step 5: Return to default XY plane

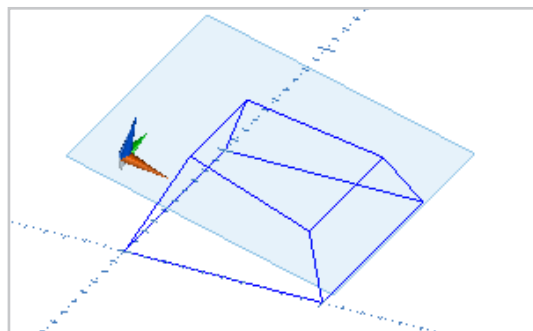


To return to the XY plane, open the Planes dialog and select the XY plane in the Saved Planes List.

To return to World coordinate input, clear the Use plane coordinate system check box before clicking OK.



You will see the XY plane is current again.



You can return to the truncation plane by selecting it in the Saved Planes list.

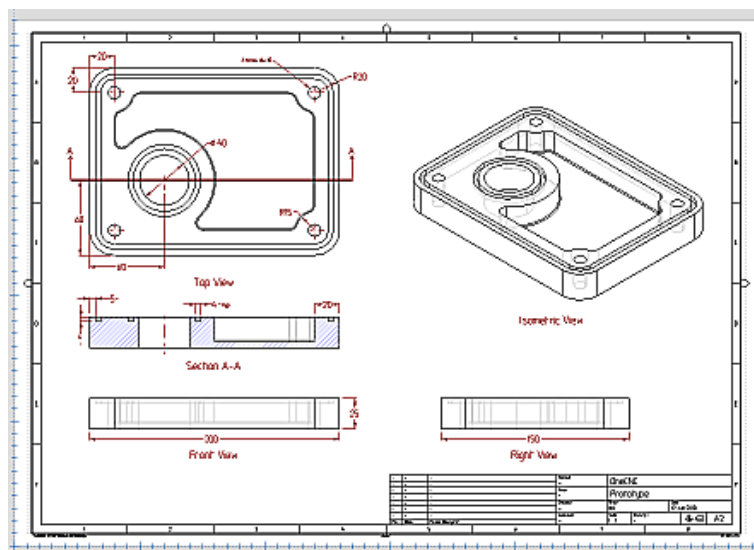
To learn more about plane commands in OneCNC, go to 'Construction Planes' in the CAD section of OneCNC Help.

OneCNC Technical Drawing

Automated 3D to 2D Drawing Layouts

This tutorial is intended for Expert versions of OneCNC, and Solid Creator.

Automated 3D model to 2D CAD drawing is a standard feature of OneCNC Solid Creator and OneCNC Expert versions, and can be added as an optional module to OneCNC Mill Professional.



Traditionally, a technical drawing was the first step in production of a part. Now, you can design the part as a 3D model, and create one or more technical drawings automatically from the part.

The paperspace drawings are associated to the model, so if the design changes, you can change the part model and then update the drawing views automatically to show the revised part.

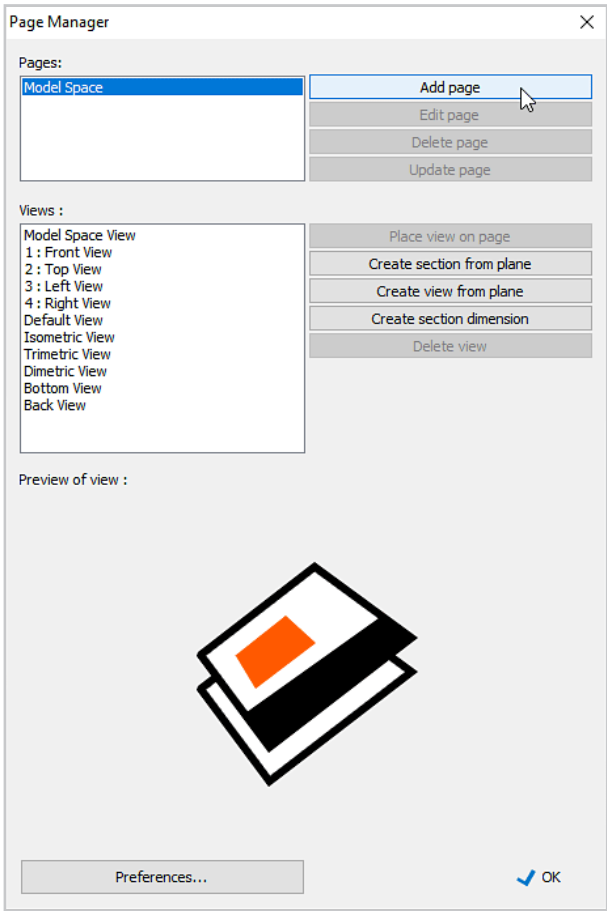
There are two stages to creating a technical drawing in OneCNC. The first step is to create a drawing page layout with title block and border. You can then create and manage views of your model in the layout.

Add a drawing page with border and title block

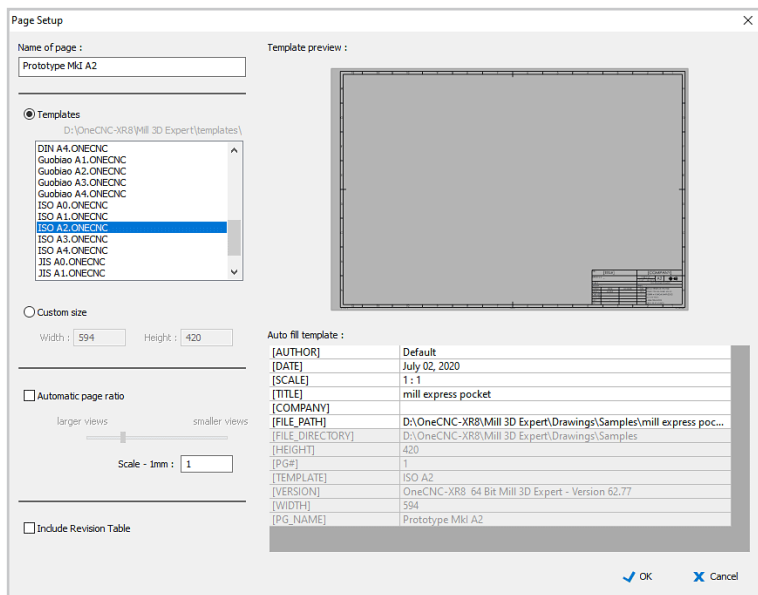


To add a new page, click on the Pages icon in the Status Bar.

The Page Manager dialog will open.

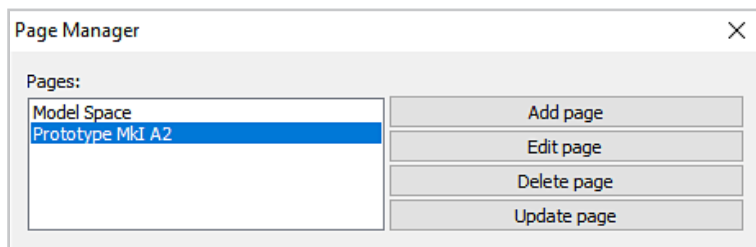


Click on the Add page icon to open the Page Setup dialog.



Templates are supplied for standard paper sizes with pre-drawn borders and title blocks. Select a template for the paper size you want from the Templates list, and enter a name for the page.

The Auto fill template section contains entries which can be automatically added to your title block. Fill these out as necessary, and click OK.



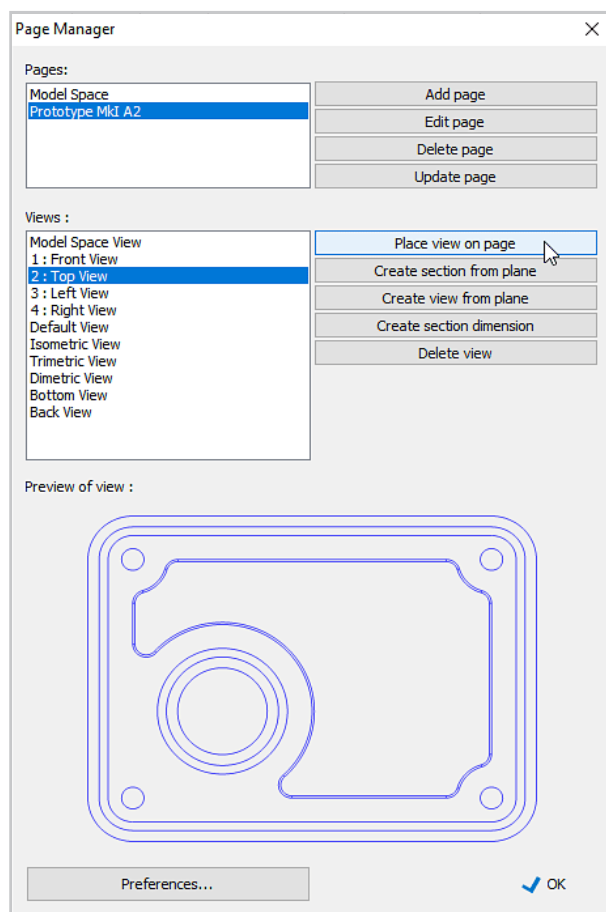
The new page now appears in the Pages list in the Page Manager.

The selected page will be seen in the main drawing window, with the title block showing the details input in the Page Setup dialog. You can now place views from the Views list into the new page.

To return to the main drawing environment, select Model Space in the Pages dialog.

Add views of the model to a drawing page layout

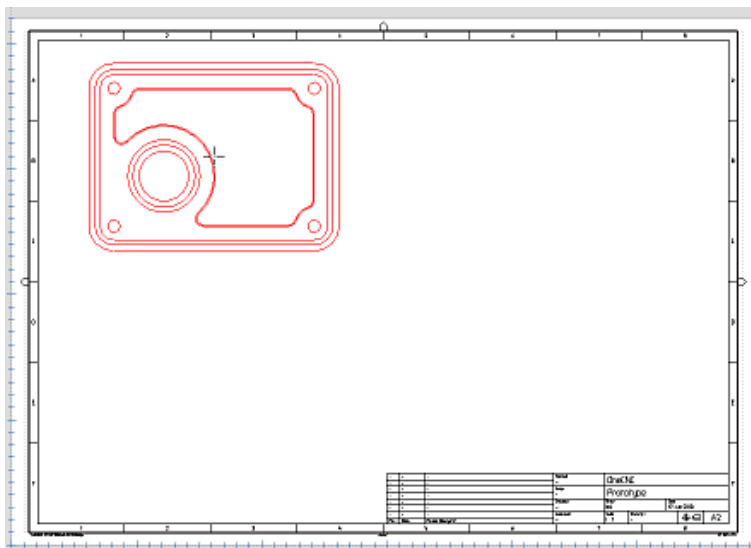
You can add a view of the model to an existing drawing page layout at any stage of the design process.



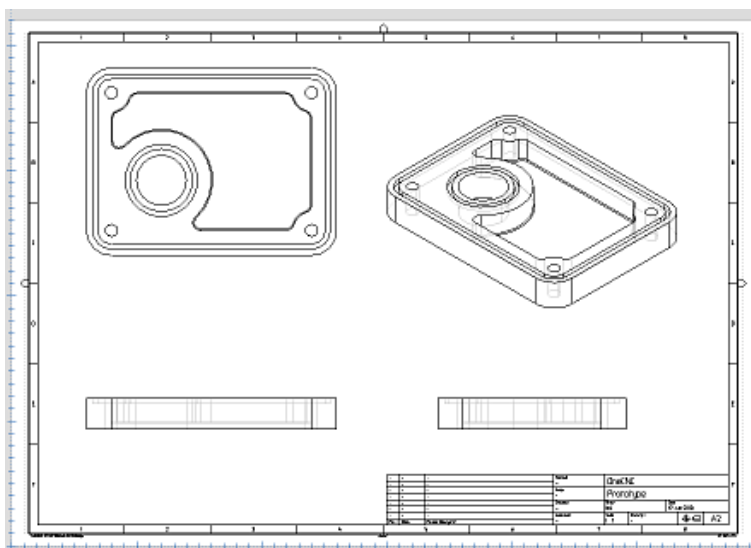
In the Pages list of the Page Manager, select the page layout you want to add the view to.

Select the view in the Views list, and a preview of the view will appear in the lower section of the Page Manager dialog.

Click the Place view on page icon to add the view to the page.

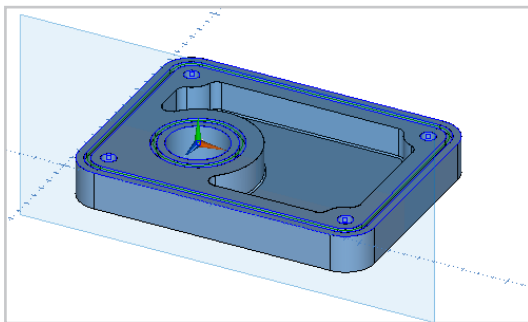


The view will be attached to the cursor. Click to place the view when it is in the position you want.



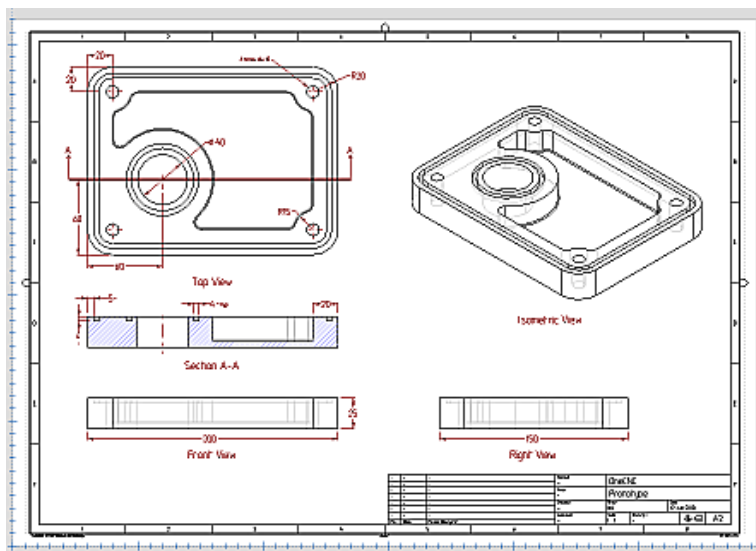
After placing the view, you are returned to the Pages Manager again, and you can continue placing views as necessary.

Views are not restricted to orthogonal representations. An Isometric view gives a good indication of what the part actually looks like.



You can create a section view in the Views list using a plane positioned where you want the section.

You can also use a plane to define a view from a specific direction.



Views and section views can be dimensioned as necessary to complete the drawing.

Using the 3D to 2D Technical Drawing module you can quickly create professional drawings to recognized standards. For more detail on using the module see the Automated 3D to 2D drawing topic in the CAD section of OneCNC Help.

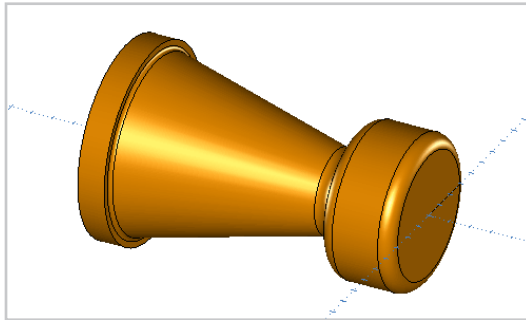
OneCNC Lathe Tutorial 1

View A Sample Part

This tutorial is intended for all levels of OneCNC Lathe.

In this tutorial we will open a sample file with turning already defined, and copy the example turning in a new toolpath group.

Open the sample file and save a copy.



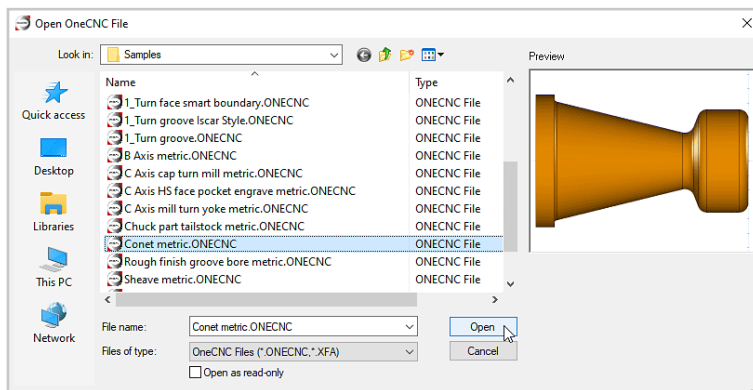
Sample files are supplied with OneCNC which provide examples of parts and toolpaths. Click on the Open icon and go to the Samples folder, which is located in the Drawings folder where you installed OneCNC.

The World and Lathe Coordinate systems



OneCNC Lathe can be used with the World XYZ Coordinate system, or set so that the World X and Y axes becomes the lathe Z and X axes. In the lower corners of the screen you will see the World axis origin indicator on the left, and a red ZX reminder indicator on the right.

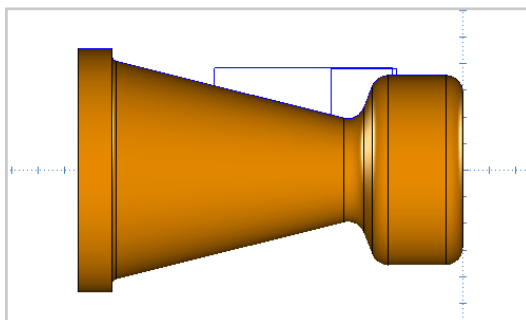
To change the Coordinate system, open the OneCNC Properties dialog from the File menu. On the General tab you will see the coordinate options in the Interface section.



Select the file 'Conet metric.ONECNC' and click Open.

Select Save As from the File menu, and save a copy of the file as Tutorial Lathe 1.

Press the Spacebar and select CAD View from the View menu.



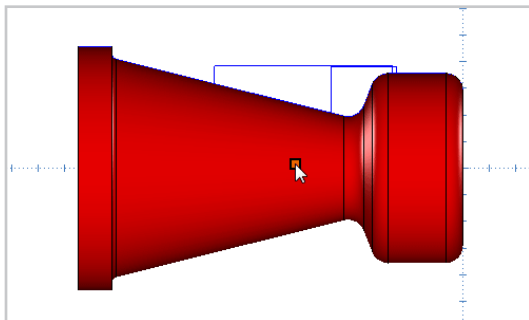
This solid model has been created in Lathe Professional from the 2D profile drawing. Lathe Express can open solid models but does not have the tools to create them.

All levels of OneCNC Lathe can display a visualisation of the turned profile called a lathe model. The lathe model is created by selecting a profile chain which is then revolved about the lathe axis. It is a 3D entity and can be viewed from any direction but is a visualisation only and is not selectable in the way a solid model is.

Display a lathe model



Click on the Select Single icon.

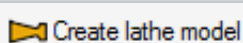
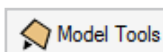


Click on the solid model to select it.

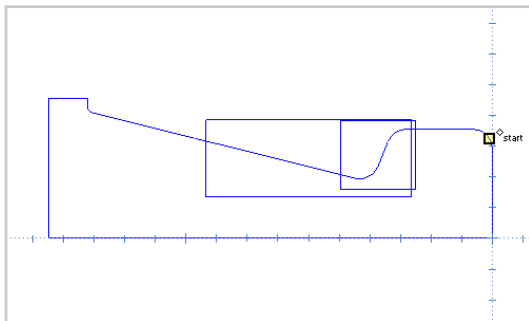


Click on Blank in the Modify toolbar.

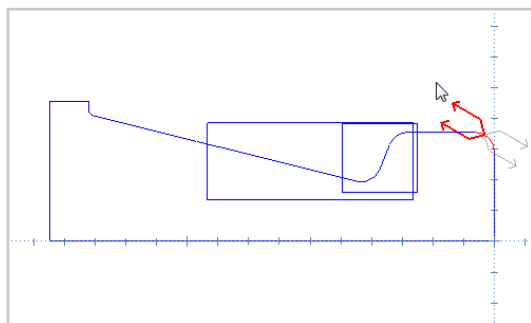
The solid model will be hidden and you will see the profile used to create it. You will also see two rectangles which are boundaries used to define the back turning operations.



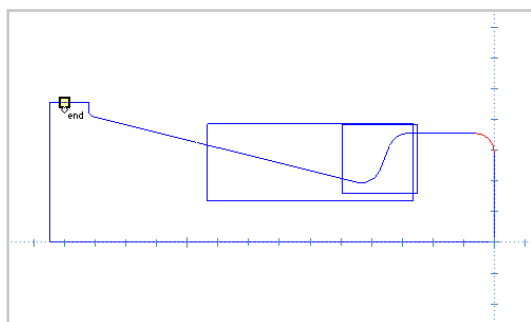
Click on the Model Tools icon and select Create Lathe Model. You will be prompted to select a profile chain.



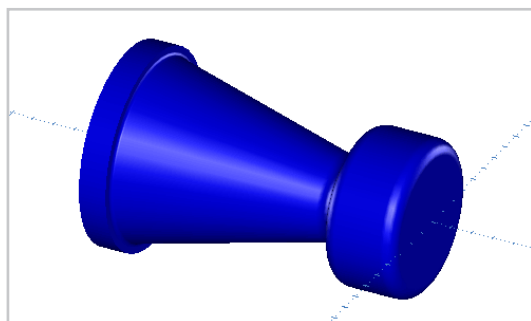
Click on the arc on the right to select it as the start of the profile.



Click while holding the mouse above the profile to set the direction the chain will be selected in.



Select the horizontal line at the left of the profile for the end of the profile chain. Right click to end the selection.



The lathe model is created by revolving the selected profile about the lathe axis, and can be viewed from any direction. It is a visualisation only and will be the default geometry color as it is not a solid model.

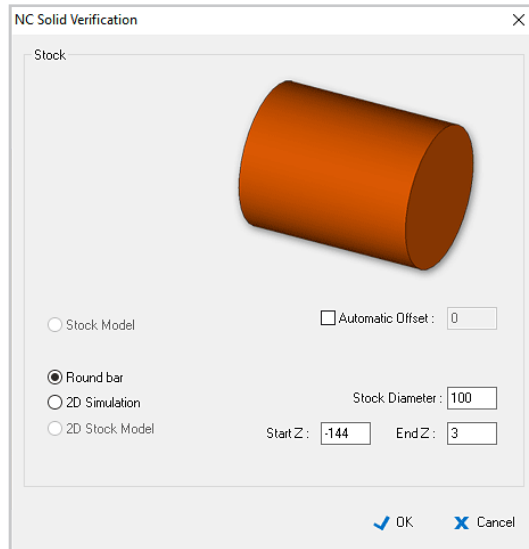
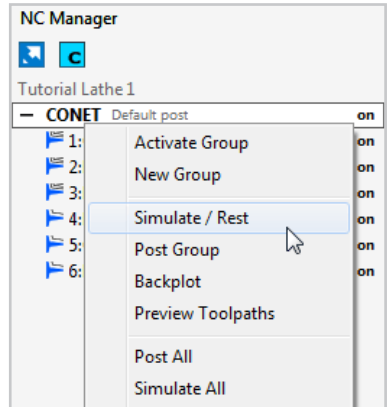
When you are ready to continue, click on the Clear Model icon in the



Model Tools menu to remove the visualisation from the screen.

Simulate the existing toolpath group

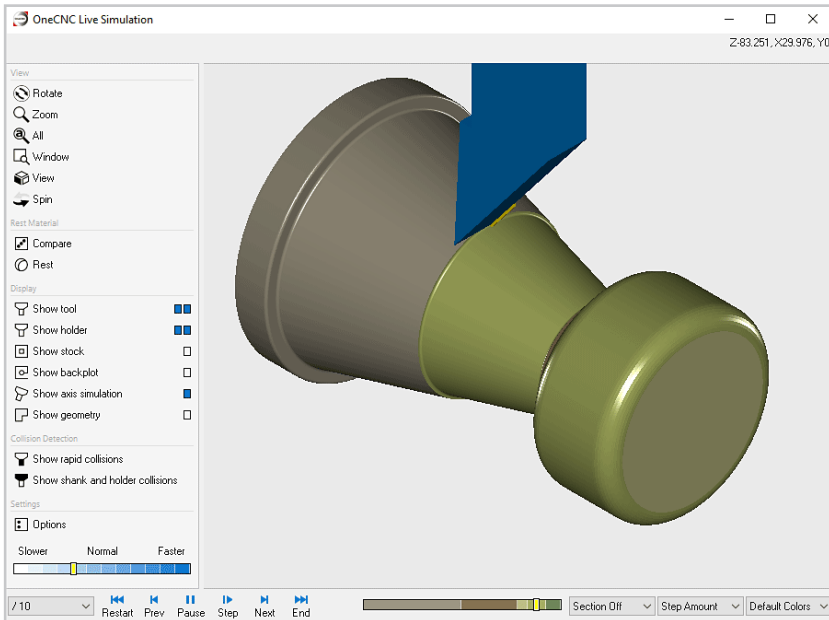
The sample file has a toolpath group called CONET with turning operations already defined. Right click on the group name CONET in the NC manager, and select Simulate / Rest.



The NC Solid Verification dialog appears.

Select the Round bar stock type. Clear the automatic offset check box, and enter a Stock Diameter of 100. Enter -144 for Start Z and 3 for End Z, and click OK.

The Live Simulation window opens with the same view as the drawing window, and simulates the existing toolpath group operations.



The current coordinates of the tool are displayed in the top right corner.

Use the video controls to play, pause or rewind the operation.

Observe the simulation, and take note of the OD, Face and Back Turn operations for roughing and finishing, which we will recreate in a new toolpath group.

You will notice the tool color changes to red while in a rapid move.

The Live Simulation window has simple playback controls.



Go back to start of simulation.



Go back to the start of the previous tool.



Pause. This changes to Play when paused, which will resume the playback.



Enter simulation Step Mode.



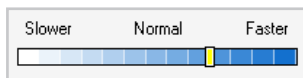
Skip forward to the start of the next tool.



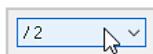
Go to end of simulation.



Turbo mode enables rapid simulation at lower resolution.

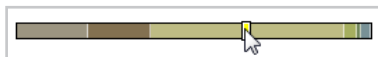


You can adjust the playback speed of the simulation by dragging the indicator in the sliding speed control.



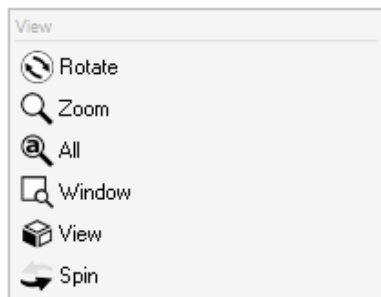
You can select a specific playback speed using the drop-down selector below the sliding speed control.

For example, if you select x2 the playback will be twice as fast. If you select /2 the playback will be at half the speed.

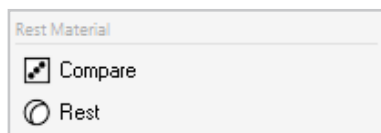


During simulation, the progress indicator in the progress bar moves from left to right. The background color of the progress bar changes to match the current tool. You can change the playback position by dragging the progress indicator in the progress bar.

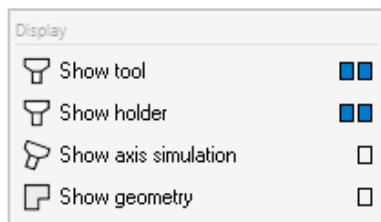
The toolbox on the left of the simulation window has view and display controls. It also holds useful functions for checking your toolpaths.



The view tools allow you to change your viewpoint and zoom magnification.

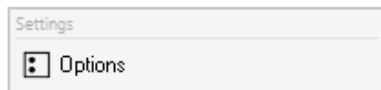


The Rest material functions are used to check for material which has not been removed yet.



The Display options can be used to show or hide the tool or geometry.

If you are simulating a multi-axis toolpath you can also change the axis simulation mode.



The Options icon re-opens the stock settings dialog.

When the simulation ends the simulation window remains open. Close the simulation window when you are ready to return to OneCNC.

OneCNC Lathe Tutorial 2

Turning A Sample Part

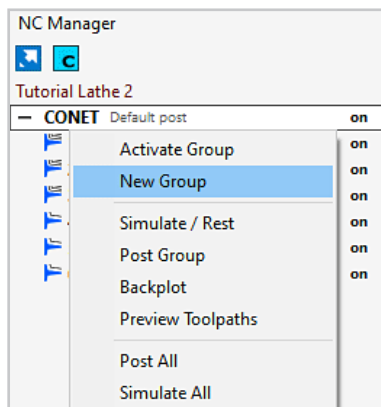
This tutorial is intended for all levels of OneCNC Lathe.

In this tutorial we will create a new group of toolpaths for the sample part viewed in the previous tutorial.

Step 1: OD Turn Rough toolpath

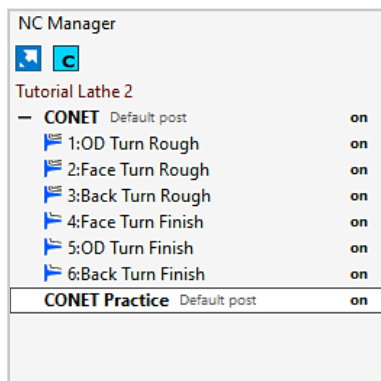
Right click in the NC manager toolpath list area, and select New Group.

A new group is created with the default name 'Toolpath Group #2'

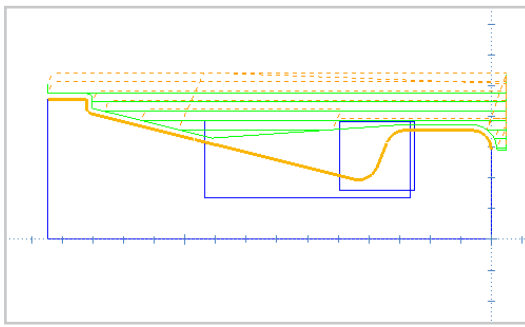


Click on the Process icon in the NC Manager tab to open the Process operation dialog.

Change the group name in the Description box to CONET Practice, and click OK.

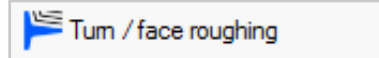
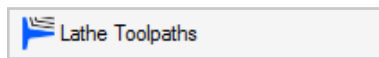


Press the Spacebar twice to select the CAD View.



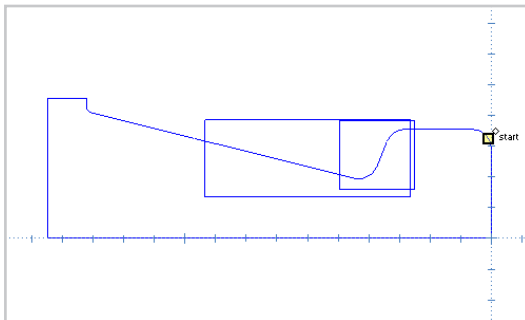
Click on the existing 1:ODTurn Rough operation in the first group, and you will see an instant preview for that toolpath in the drawing window. The new group remains the active group new toolpaths will be created in.

For clearer images, we have turned off display of the active plane, but it is not necessary to do so to follow this tutorial.

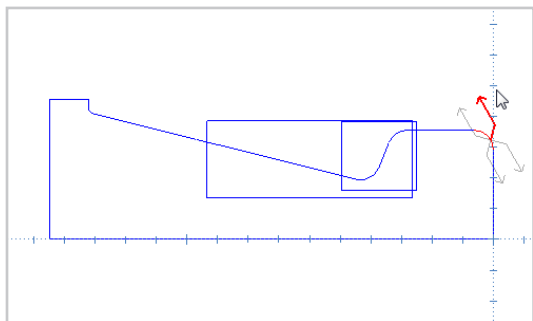


To re-create the OD Turn Rough lathe operation in the practice group, click on the Lathe toolpaths icon and select Turn / face roughing.

It is now time to select the chain of entities we want the toolpath to follow, and specify the offset side and direction of travel.

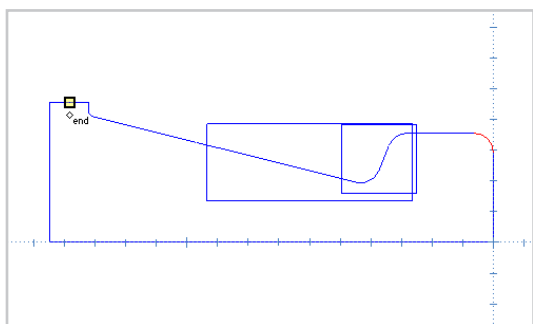


The cursor will change to the word 'start', and the selection box will attach itself to the nearest entity. Hover the cursor near the arc at the right side of the part, and click to select this as the start of the chain.



Four direction and side indicator arrows appear.

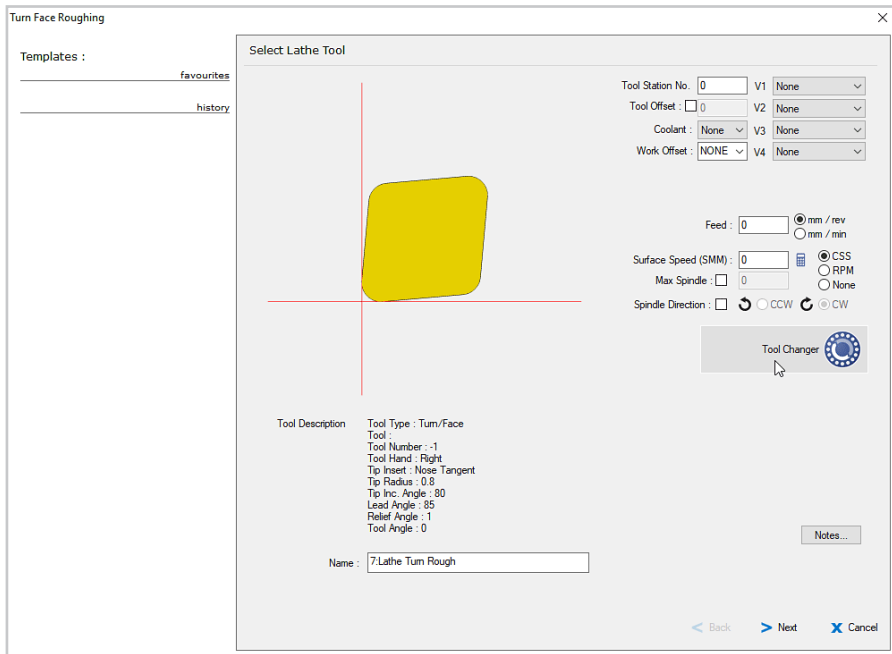
Hold the mouse so that the red arrow is above the arc, and pointing to the left, and click to confirm this side and direction.



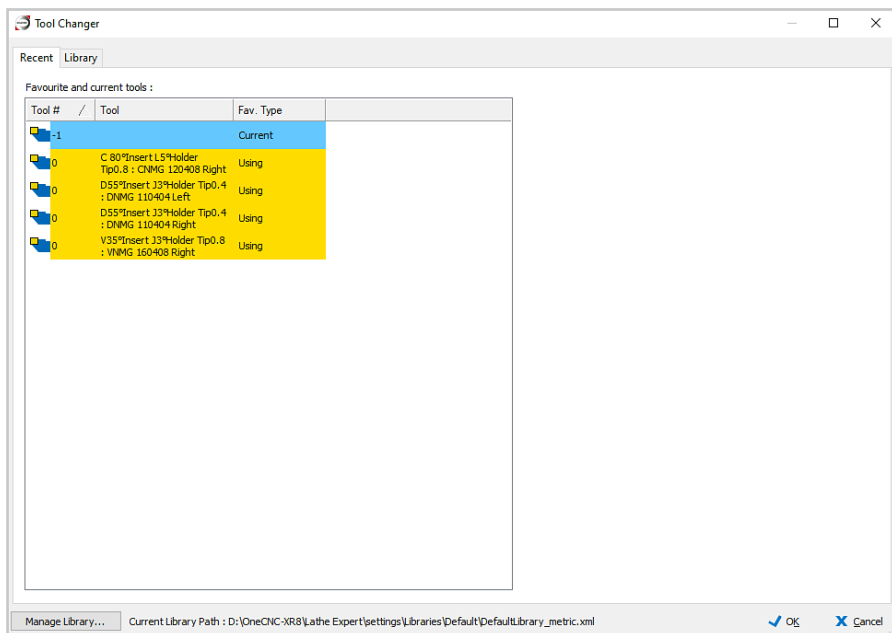
The cursor will change to the word 'end'. Click on the horizontal line at the left of the part to select it for the end of the profile chain.



Right click to end the selection process.

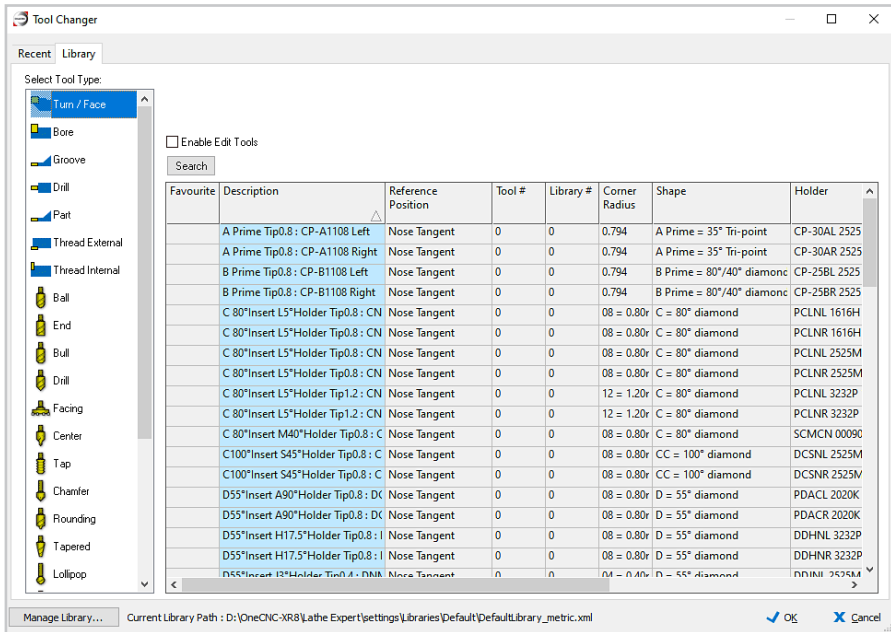


The first stage of the toolpath wizard is the Select Tool dialog. Click on the Tool Changer icon to open the Tool Library.



The Tool Changer dialog opens, with the Recent tab active.
Tools you use regularly can be selected quickly from this tab.

To select a tool which is not shown in the Favourites list, click on the Library tab to see a categorized display listing all available tools.



The Tool Library is composed of separate lists for each category of tool. Each row in a list stores information for a single tool.

When the Turn / Face tool icon at the left of the dialog is selected, the displayed list will show available tools for OD turning.

The tool image will update with a preview of the selected tool.

Turn Face Roughing

Templates :

favourites

history

Select Lath Tool

Tool Station No. 1 V1 None

Tool Offset : 1 V2 None

Coolant : No.1 V3 None

Work Offset : G54 V4 None

Feed : 0.8 ☒ mm / rev ☐ mm / min

Surface Speed (SMM) : 60 ☒ CSS ☐ RPM ☐ None

Max Spindle : ☒ 2500

Spindle Direction : ☐ ☒ CCW ☐ CW

Tool Changer

Tool Description

Tool Type : Turn/Face
Tool : C 80°Insert L5°Holder Tip0.8 : CNMG 120408
Right
Tool Number : 1
Tool Hand : Right
Tip Insert : Nose Tangent
Tip Radius : 0.8
Tip Inc. Angle : 80
Lead Angle : 5
Relief Angle : 1
Tool Angle : 0

Name : 1.Lathe Turn Rough

Notes...

< Back > Next X Cancel

Enter 1 for the Station number, and the Tool Offset will be entered automatically.

Select Coolant No. 1 and Work Offset G54.

Enter a Feedrate of 0.8 to match the tip radius of the tool, and select the mm/rev option.

Enter a Surface Speed of 60 and select the CSS Constant Surface Speed option.

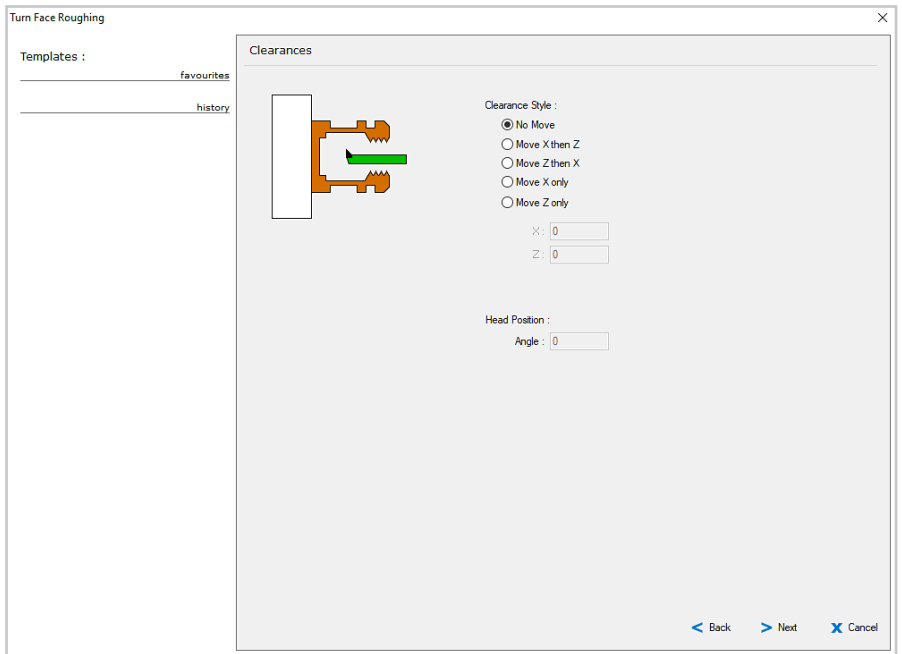
Select the Max spindle speed check box and enter a value of 2500.

Click Next to proceed.

Note:



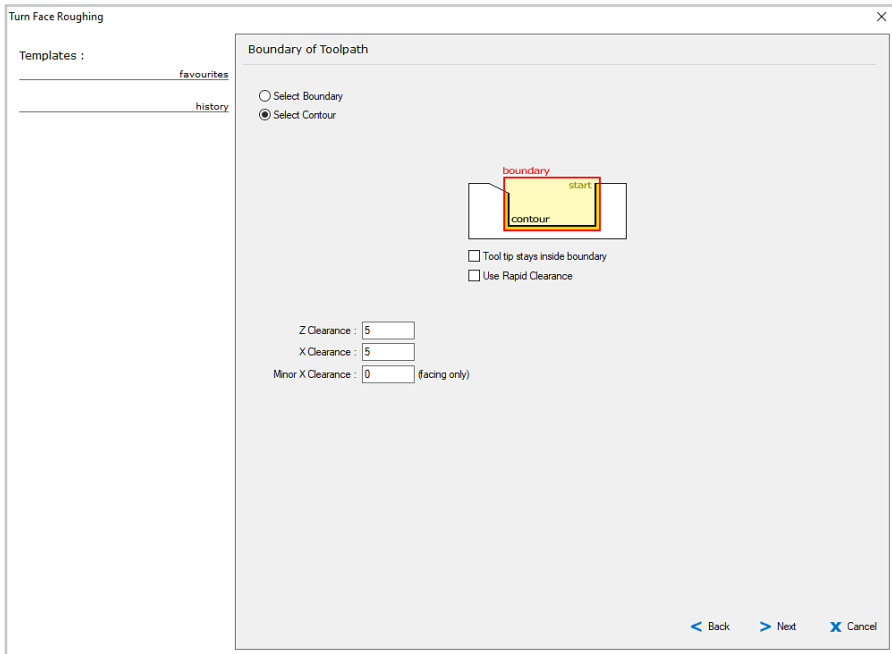
If the tool has a Station number in the Tool Library, the Station number and Tool Offset will be set when the tool is selected.



The next dialog gives you options for clearance moves, which are only necessary if avoidance is required.

Select No Move and click Next.

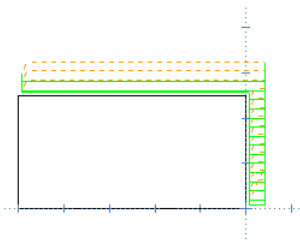
In the Boundary of Toolpath dialog pick the Select Contour option, set Z and X clearance to 5, and click Next.



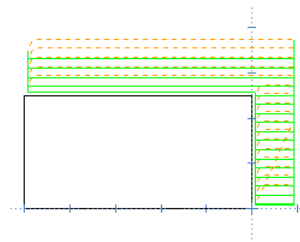
The importance of setting correct Clearance values



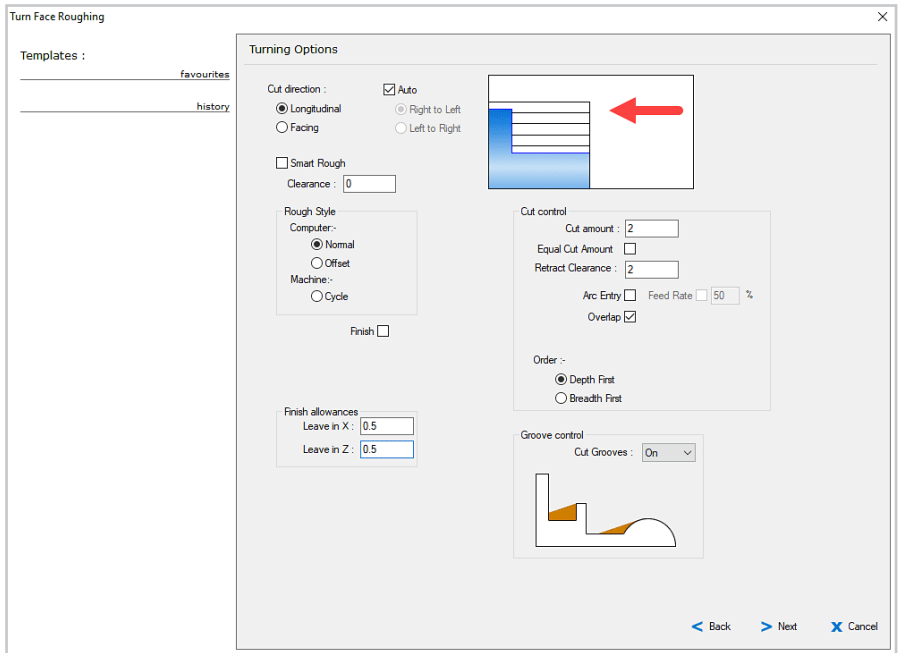
The clearance values in Z and X define where the stock surface is in relation to the part. This is where cutting starts, and must be set accurately or the tool could contact metal while still in a rapid positioning move. Compare the toolpath result for a small part with a radius of 25.



Z and X clearance 5



Z and X clearance 10



In the Turning Options dialog, select Longitudinal and Auto for cut direction, and Normal roughing style.

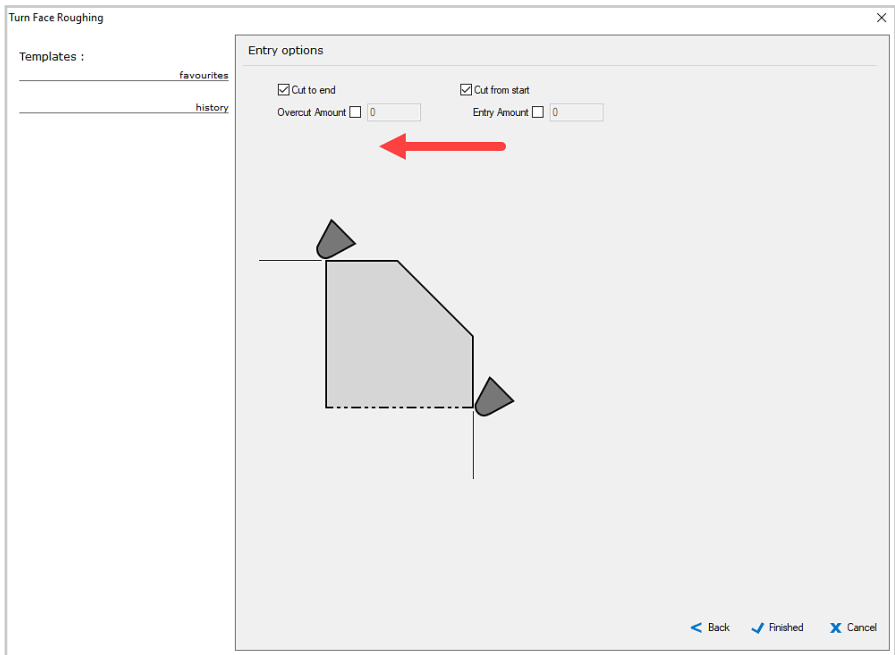
Set Finish allowances to 0.5 in X and Z.

Set the Cut amount for the cut depth of each pass to 2, the Retract Clearance between cuts will update to match this setting. Select Overlap to create a small move at the end of each pass that will clean up material that would otherwise be left behind.

Set Cut Grooves to on and click Next to continue.

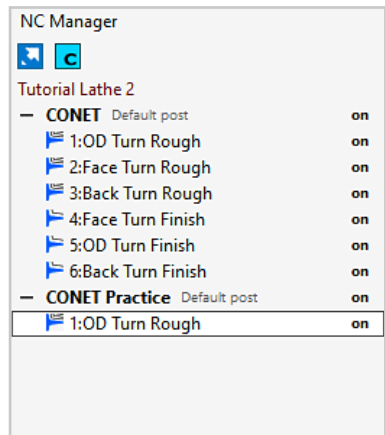
Select the check boxes for Cut from start and Cut to end.

These settings move the tool control point past the geometry by the tip radius, ensuring all of the geometry is cut. You can also add extra distance to the entry or overcut.



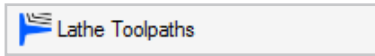
Click Finished to create the toolpath.

Double click on the new toolpath and rename it 1:ODTurn Rough in the NC Processing dialog.

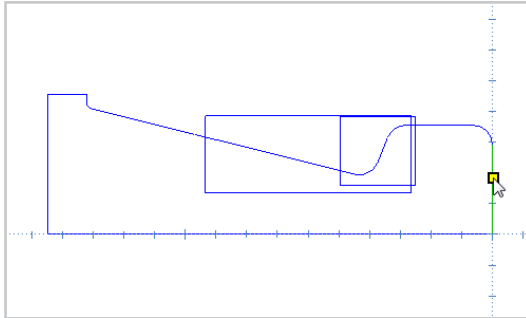


Step 2: Facing toolpath

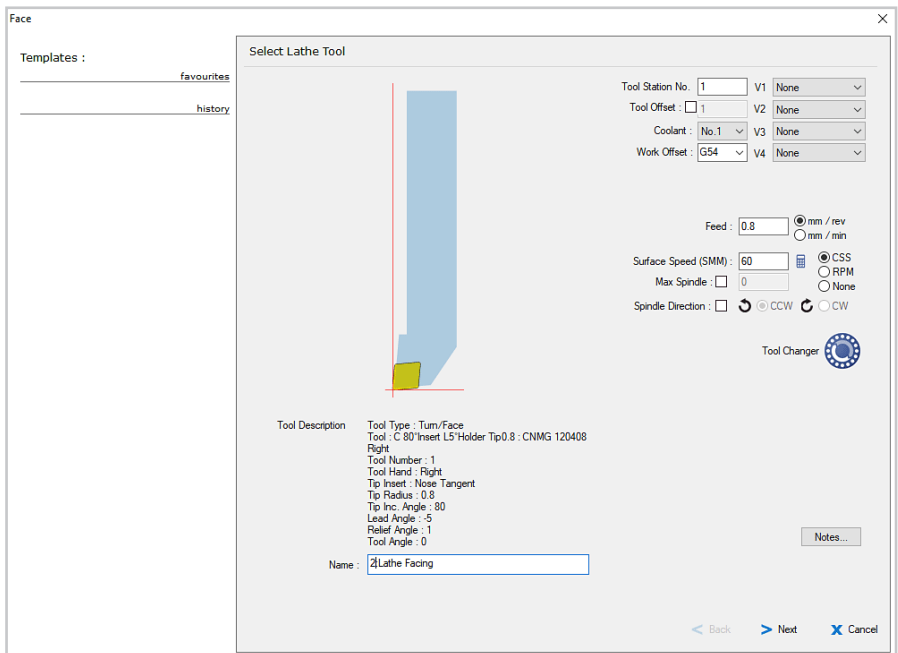
The next step is facing. This can be done using a Turn/Face Rough toolpath, but we will use the Facing toolpath.



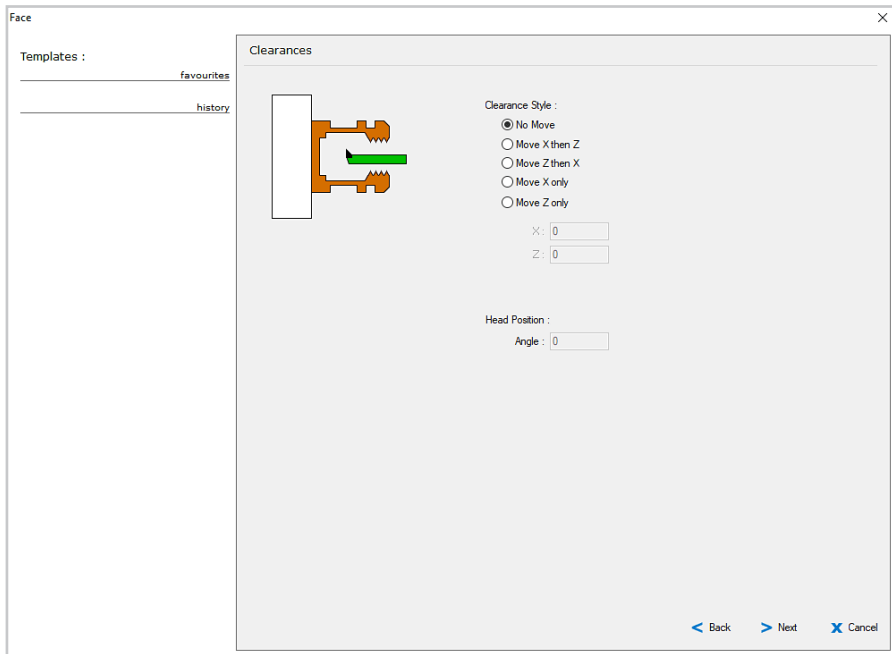
Click on the Face icon in the Lathe Toolpaths menu.



Select the vertical line at the right of the profile for the facing operation. The Select Tool dialog opens immediately.



Click on the Tool Changer and select the tool used for the previous operation from the Recent tab. Enter the same feed and speed settings.



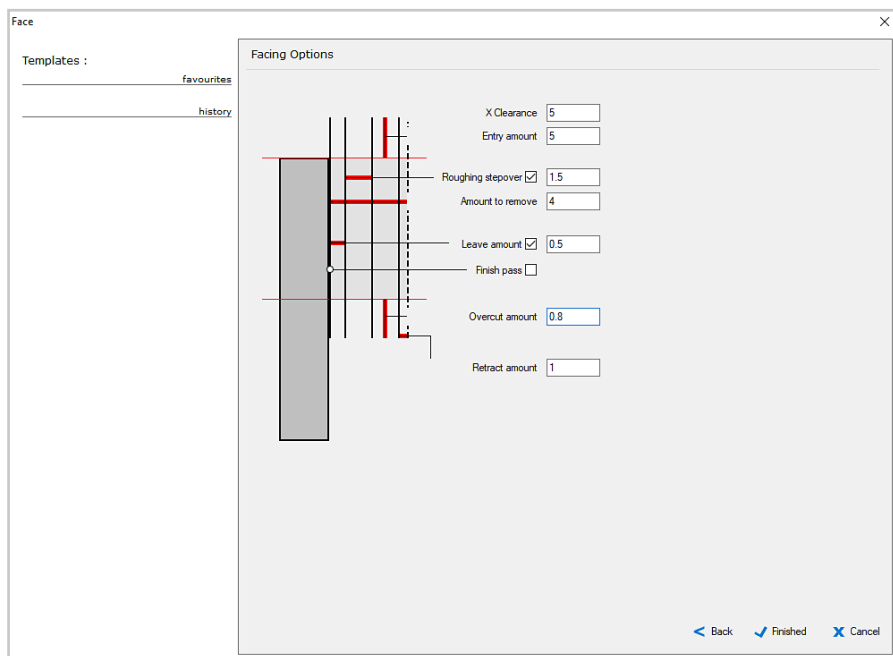
The next dialog gives you options for clearance moves, which are only necessary if avoidance is required.

Select No Move and click Next.

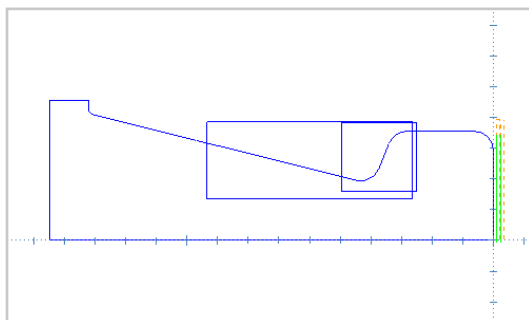


For details of the Clearance settings and how they affect the start and end of a toolpath, see the Clearance Settings topic in the Lathe Toolpath Settings section of OneCNC Help.

The Facing Options dialog will appear.



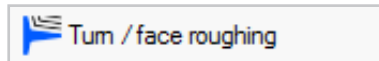
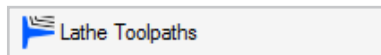
Enter 5 for X Clearance and Entry amount to start the facing before the tool reaches the geometry. Select Roughing stepover and enter a value of 1.5. Enter 4 for Amount to remove. Select Leave amount and enter 0.5. Leave Finish pass unchecked as we are only roughing at this stage. Enter 0.8 for Overcut amount and 1 for Retract amount. Click Finished to create toolpath.



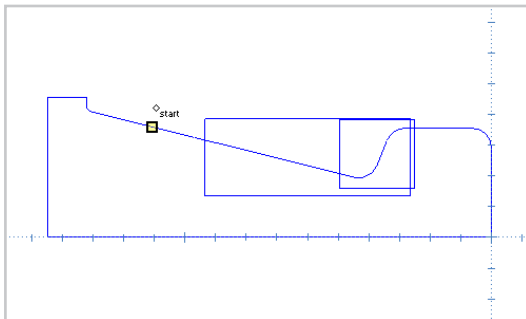
The new toolpath is generated, and will be previewed in the drawing window when it is selected in the NC Manager.

Step 3: Back Turn Rough toolpath

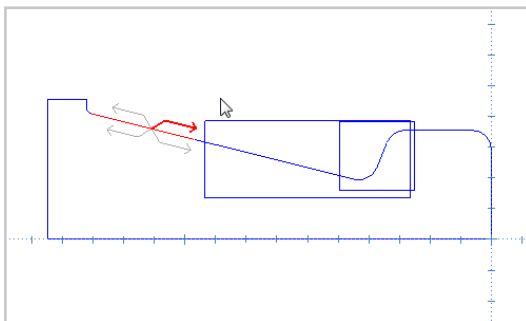
You will have noticed that the tool used for the OD Turn roughing is unable to reach the lower portion of the profile, so a back turn operation is required.



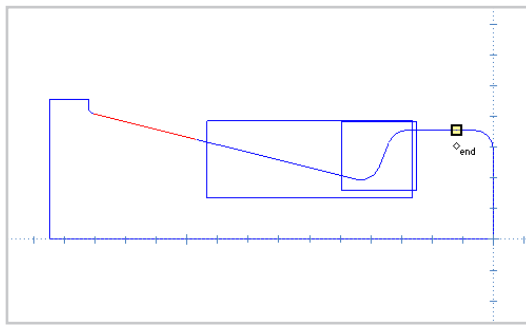
Select Turn / Face roughing again.



We will use the larger of the pre-drawn boundaries to limit the range of the toolpath. Click on the line to the left of the boundary to start the chain selection.



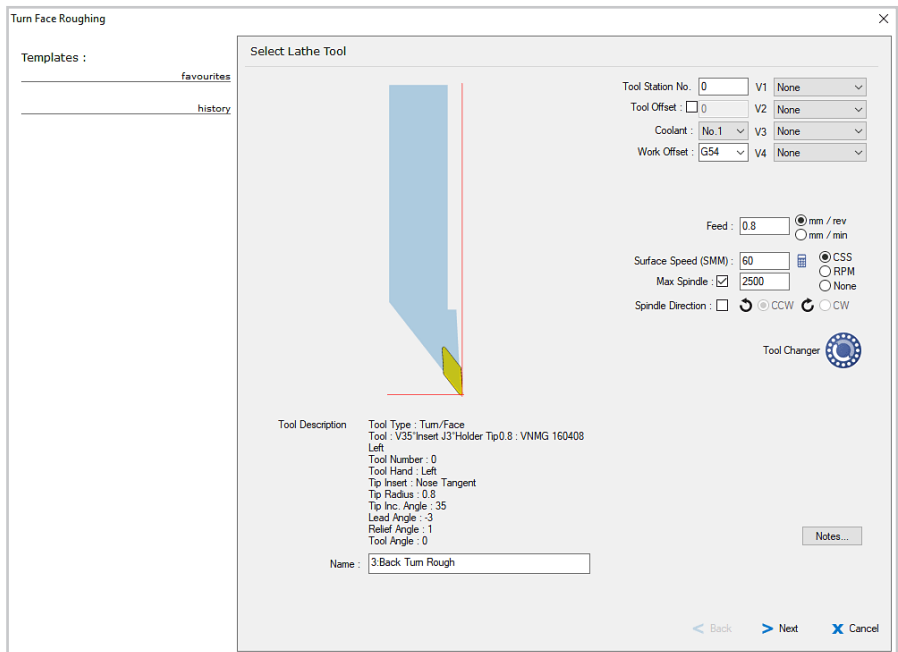
Click above and to the right of the start position to set the side and direction for the toolpath.



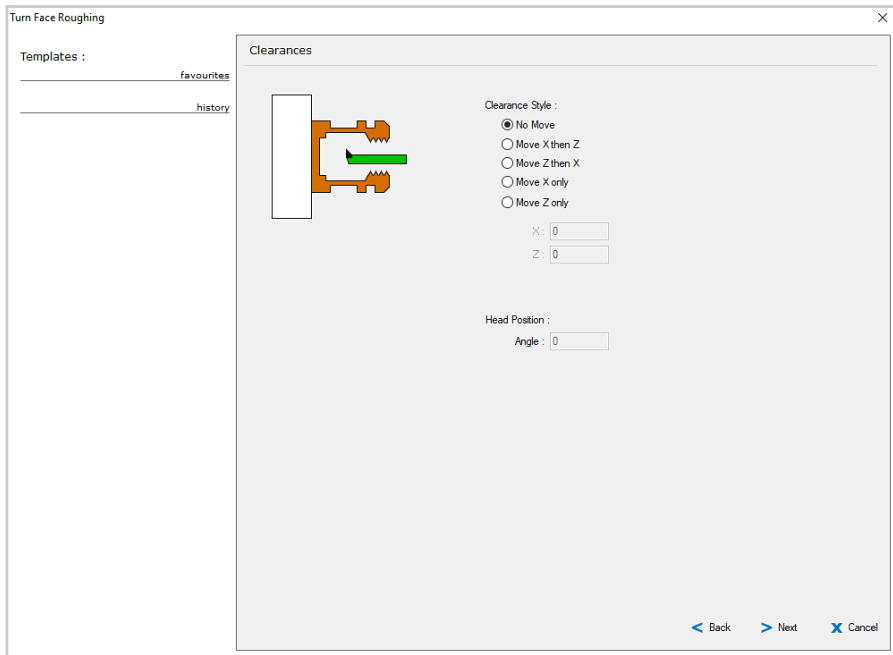
Click on the line to the right of the boundary to end the chain.



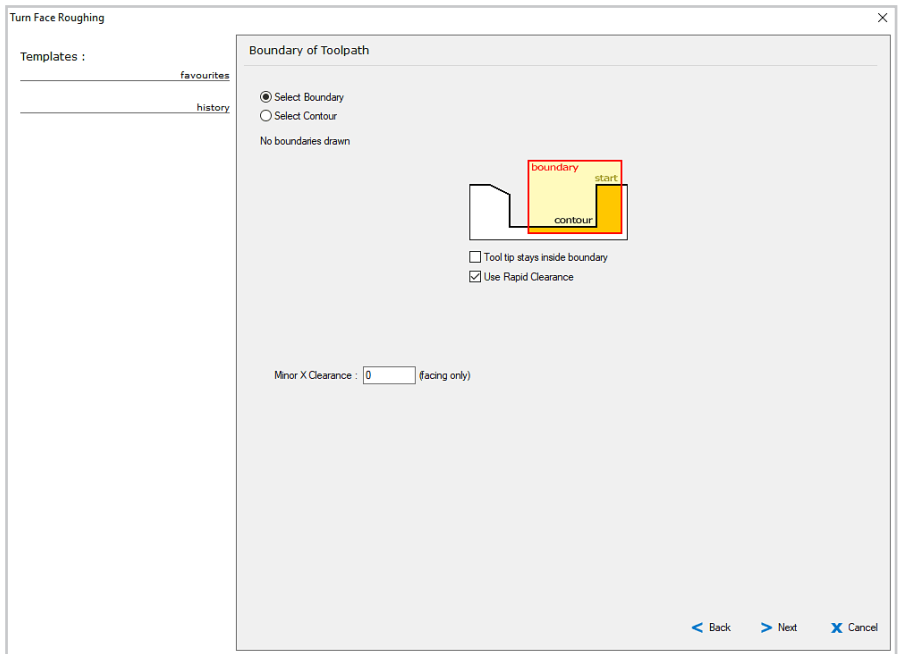
Right click to end the selection process.



Click on the Tool Changer icon to open the Tool Library, and select the VNMG 160408 insert in a left hand J type holder. Change the feed setting to 0.8 mm/rev, and click Next.



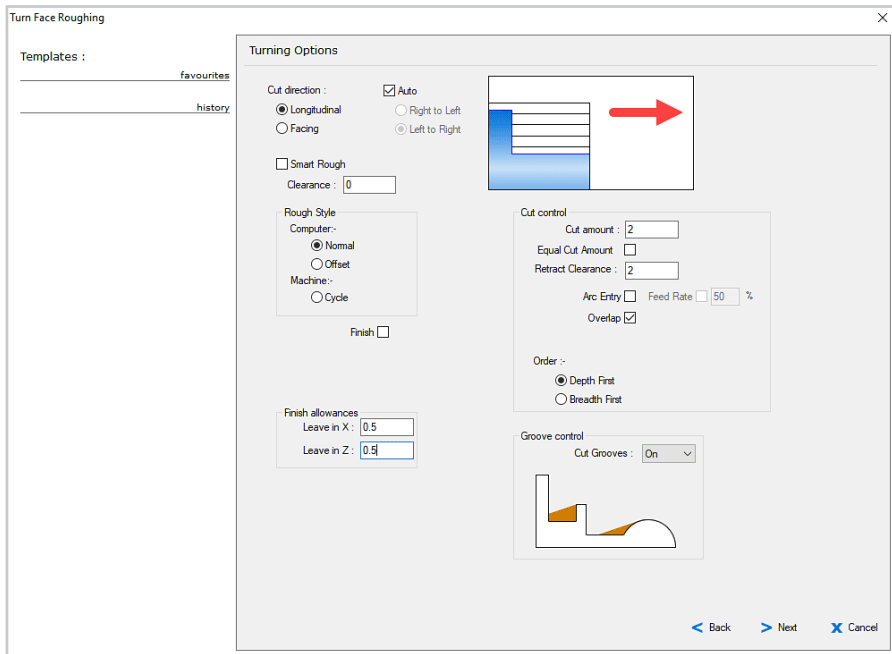
Select No Move in the Clearances dialog and click Next.



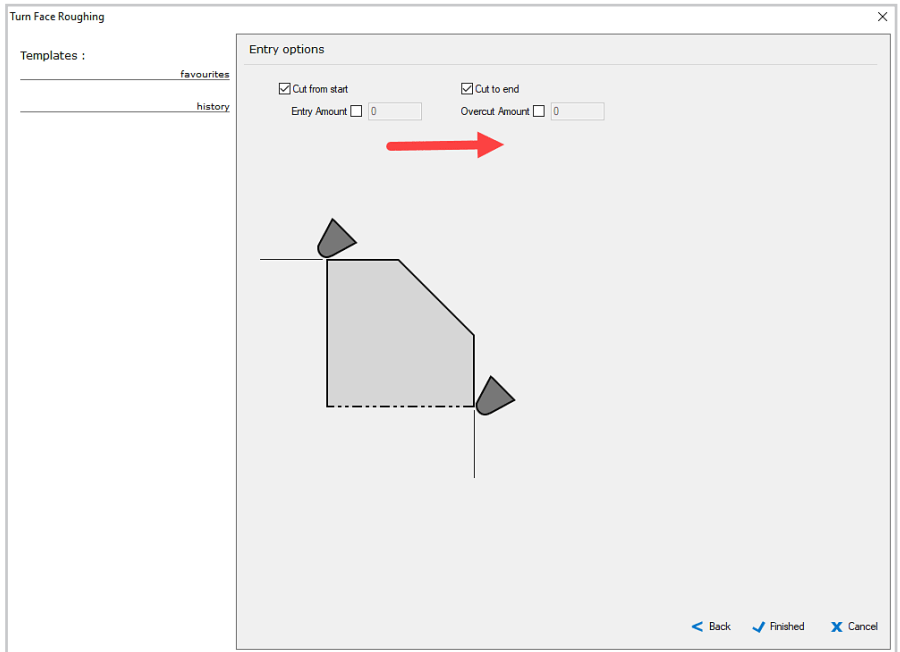
In the Boundary of Toolpath dialog, choose the Select Boundary option and click Next.



For details of the Boundary settings and how they affect a toolpath, see the Boundary Settings topic in the Lathe Toolpath Settings section of OneCNC Help.

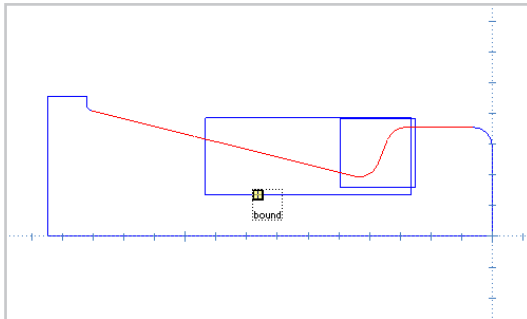


In the Turning Options dialog, select Longitudinal for Cut Direction, and click Next.

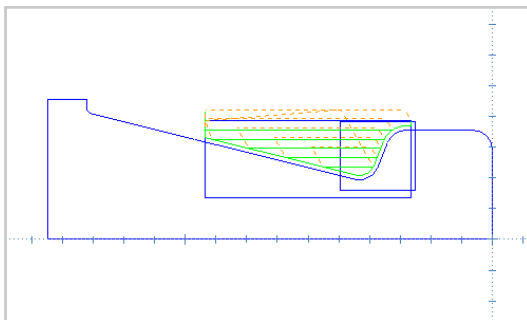


Select the check boxes for Cut from start and Cut to end.

Click Finished to create the toolpath.



The message pane in the Status bar prompts for a boundary. Click anywhere on the larger rectangle.

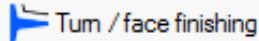
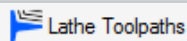


The instant preview is displayed, and you can see that the toolpath is limited to that part of the selected profile within the boundary rectangle.

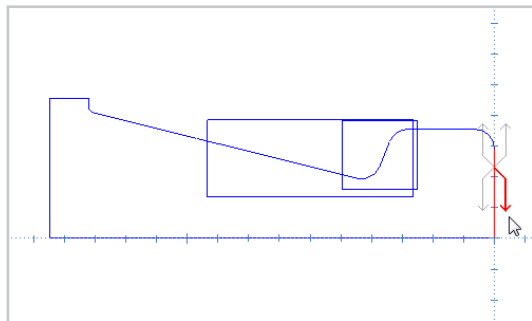
Double click on the new toolpath and rename it

3:Back Turn Rough.

Step 4: Face Turn Finish toolpath

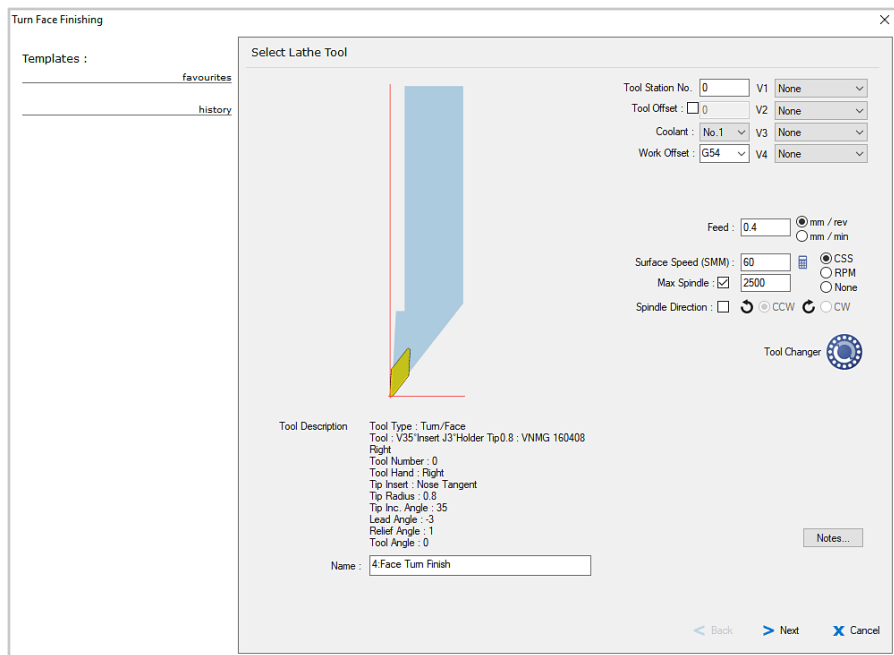


Click on the Lathe Toolpaths icon and select Turn / face finishing.

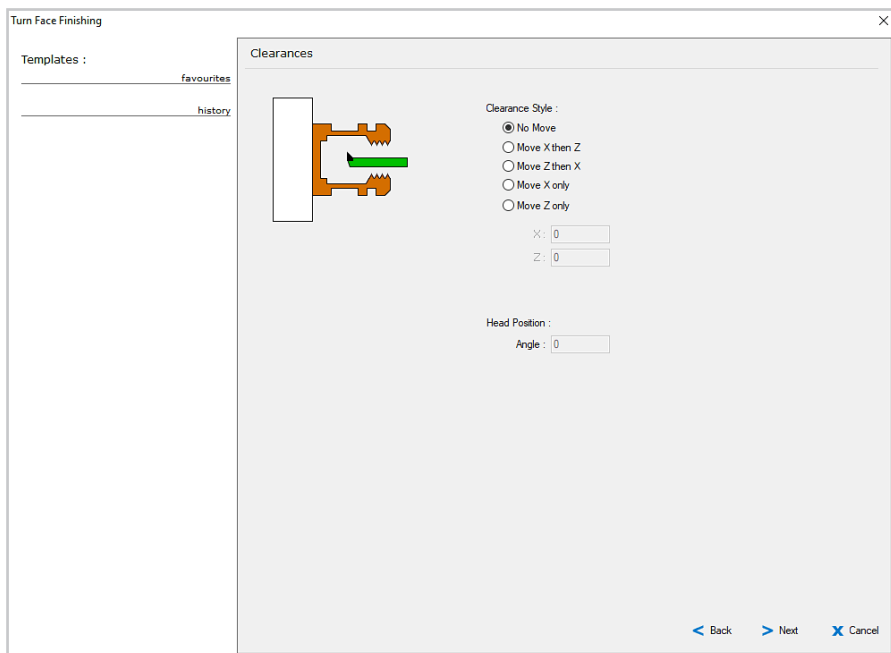


Click near the upper end of the line at the right of the part, and click to set the side and direction of the toolpath as shown.

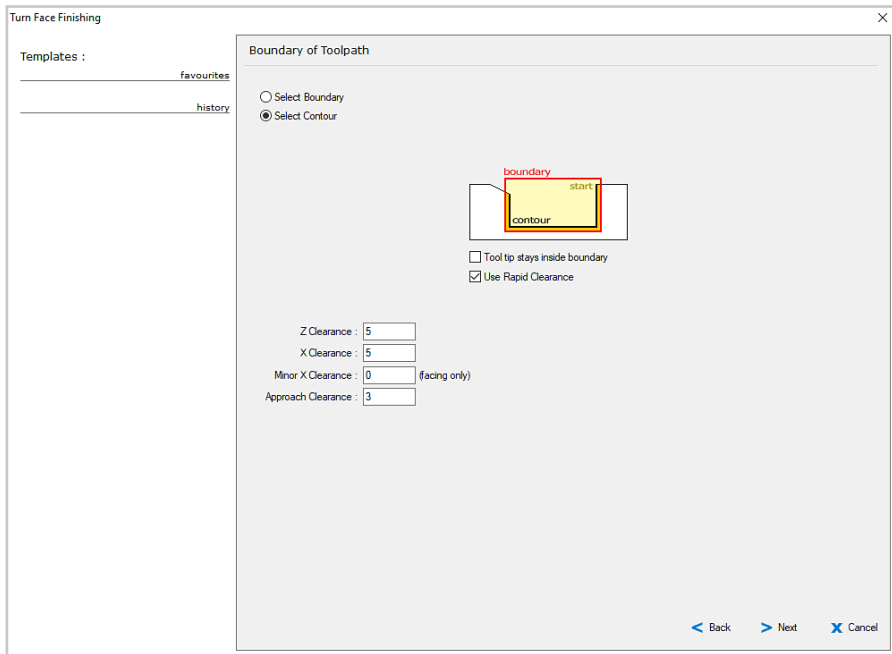
Right click to end the selection of the single line.



Click on the Tool Changer icon to open the Tool Library, and select the VNMG 160408 insert in a right hand J type holder. Enter the tool settings as shown, with a feed setting of 0.4 mm/rev, and click Next.



Select No Move in the Clearances dialog and click Next.

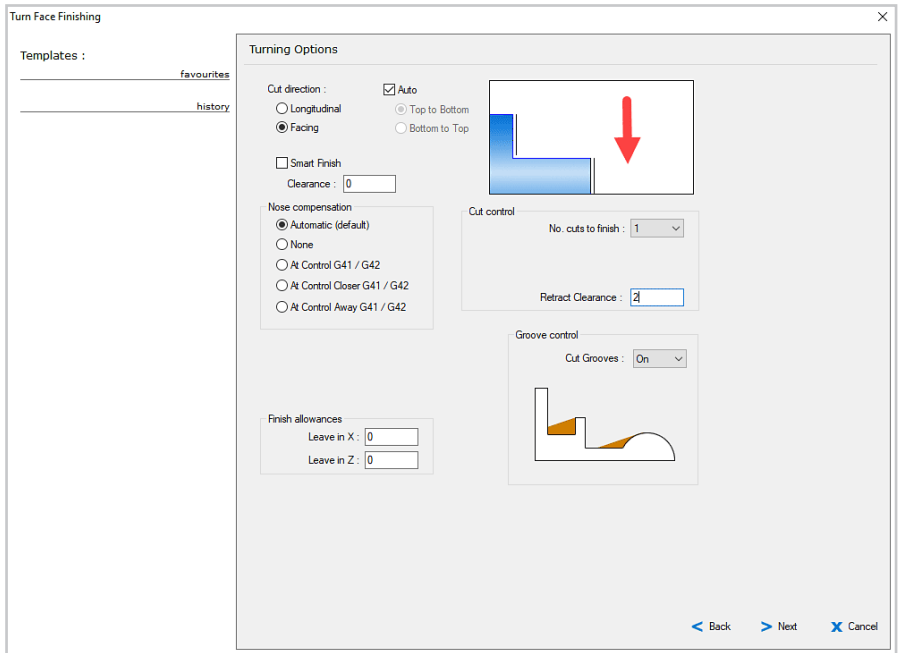


In the Boundary of Toolpath dialog, choose the Select Contour option.

Enter Z and X clearance of 5.

Minor X Clearance can be used to add clearance to the inner end of the toolpath, but we don't need it for this operation so leave this set to 0.

Enter an approach clearance of 3 and click Next.

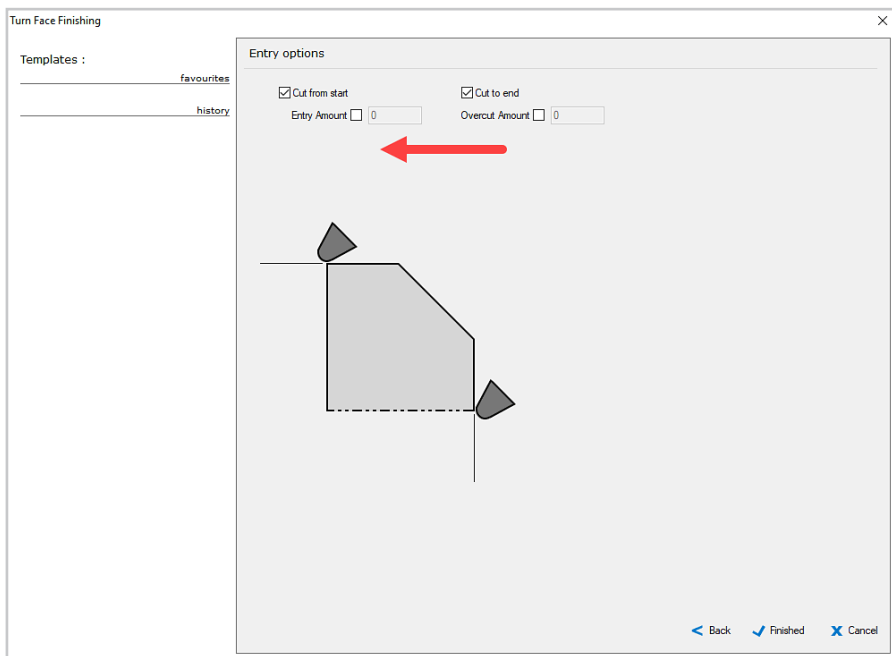


In the Turning Options dialog, select Facing for Cut Direction.

Enter the other settings as shown, and click Next.



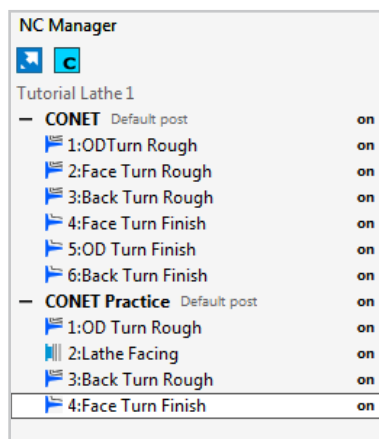
For details of the Turning Options settings and how they affect a toolpath, see the Turn and Face Cut Settings topic in the Lathe Toolpath Settings section of OneCNC Help.



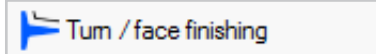
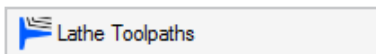
Select the check boxes for Cut from start and Cut to end.

Click Finished to create the toolpath.

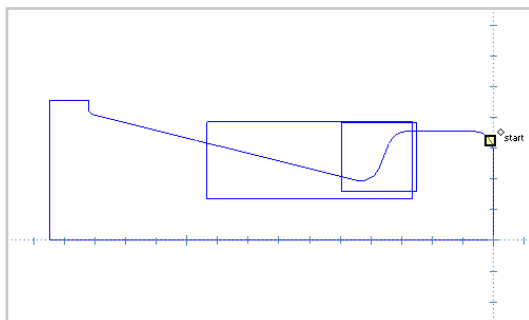
Rename the new toolpath 4:Face Turn Finish.



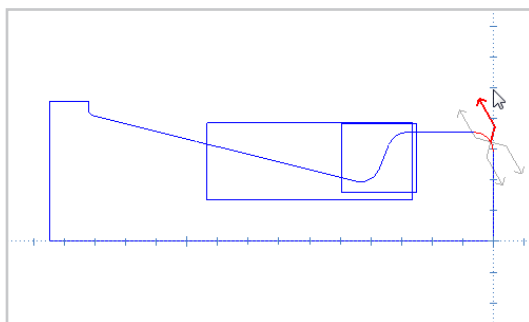
Step 5: OD Turn Finish toolpath



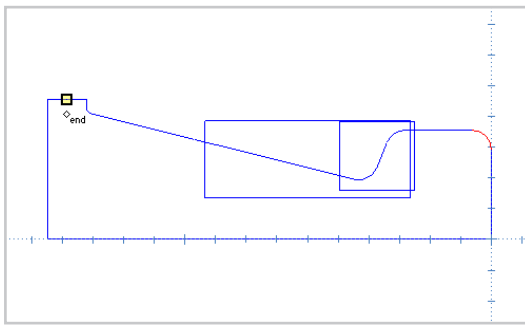
Click on the Lathe toolpaths icon and select Turn / face finishing.



The cursor will change to the word 'start', and the selection box will attach itself to the nearest entity. Hover the cursor near the arc at the right side of the part, and click to select this as the start of the chain.



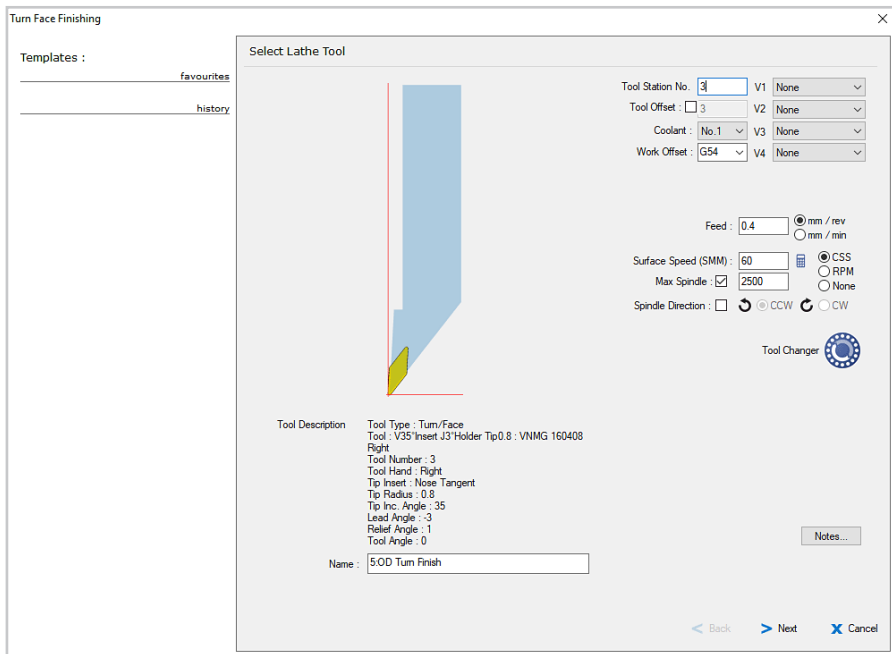
Hold the mouse so that the red arrow is above the arc, and pointing to the left, and click to confirm this side and direction.



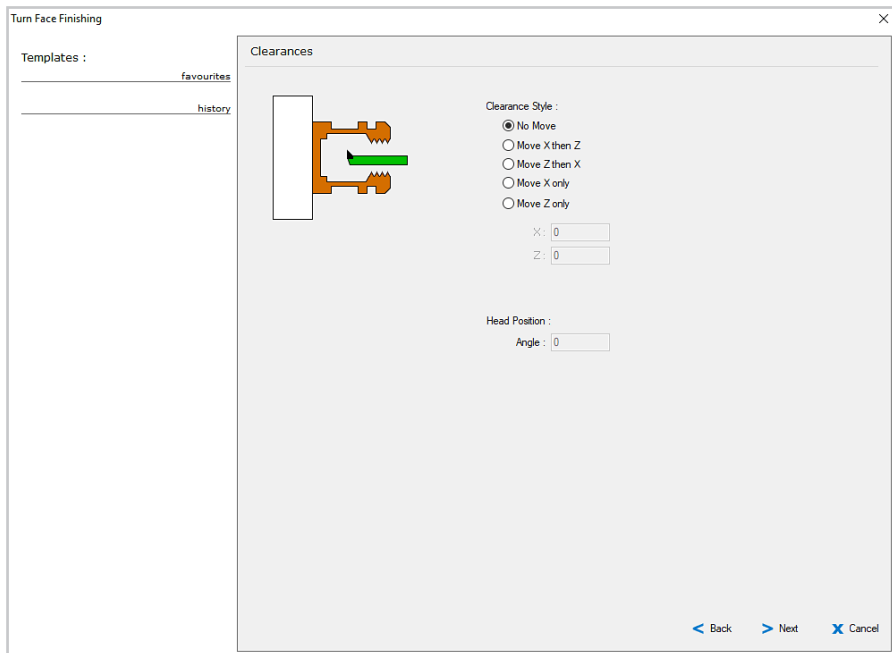
The cursor will change to the word 'end'. Click on the horizontal line at the left of the part to select it for the end of the profile chain.



Right click to end the selection process.



Click on the Tool Changer icon to open the Tool Library, and select the VNMG 160408 insert in a right hand J type holder. Enter a feed setting of 0.4 mm/rev, and click Next.



Select No Move in the Clearances dialog and click Next.

Turn Face Finishing

Templates :

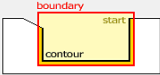
favourites

history

Boundary of Toolpath

☐ Select Boundary

☒ Select Contour



☐ Tool tip stays inside boundary

☒ Use Rapid Clearance

Z Clearance :

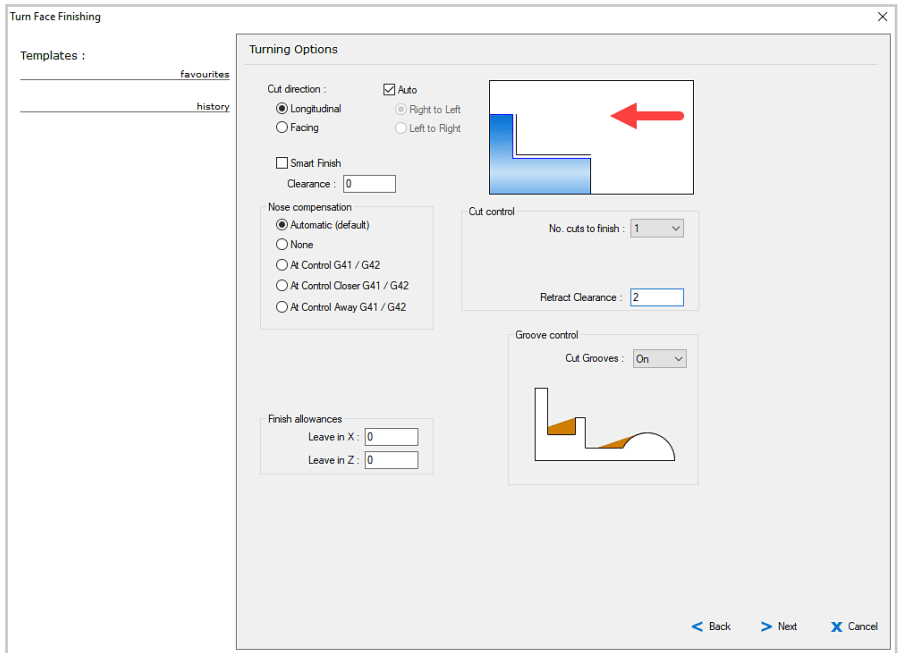
X Clearance :

Minor X Clearance : (facing only)

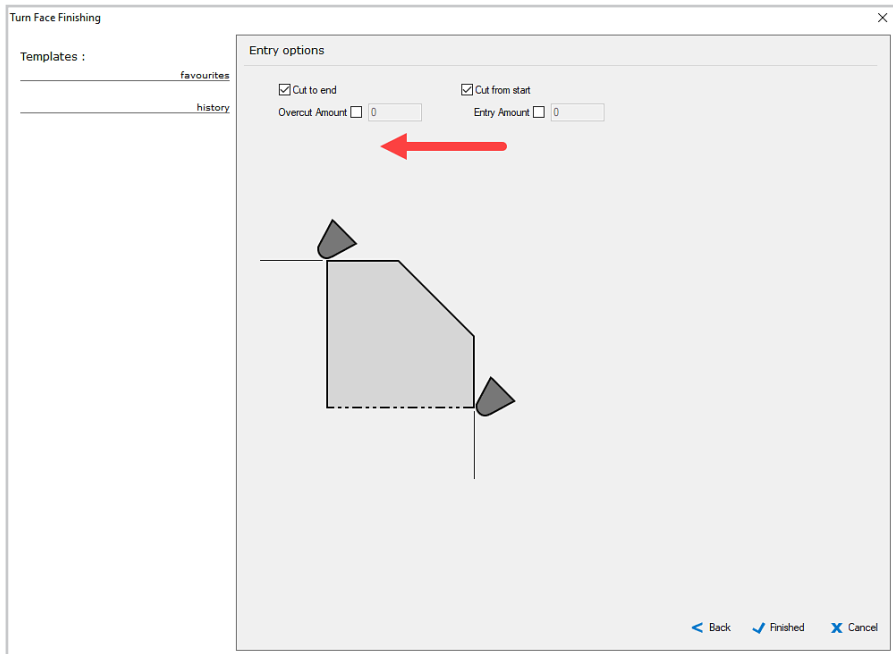
Approach Clearance :

< Back > Next X Cancel

In the Boundary of Toolpath dialog, choose the Select Contour option and click Next.



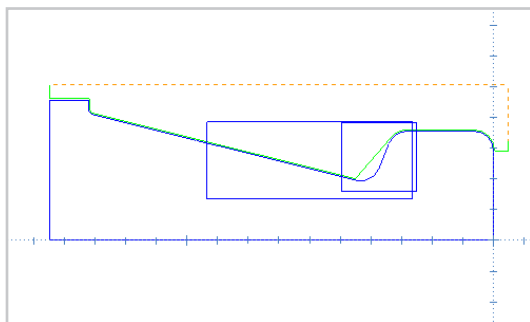
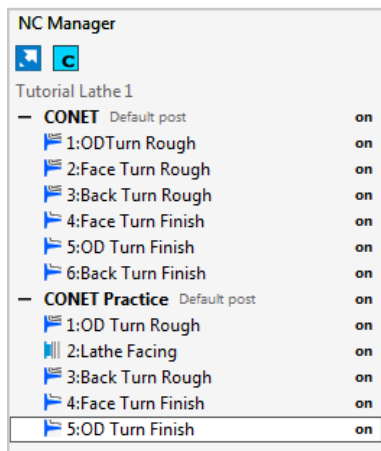
In the Turn and Face Options dialog, select Longitudinal for Cut Direction, and click Next.



Select the check boxes for Cut from start and Cut to end.

Click Finished to create the toolpath.

Rename the new toolpath 5:OD Turn Finish.



Looking at the preview of this toolpath, you will see the region the 35 degree tool could not reach is bounded by the smaller rectangle.

We will finish this region with a back turn operation.

Step 6: Back Turn Finish toolpath

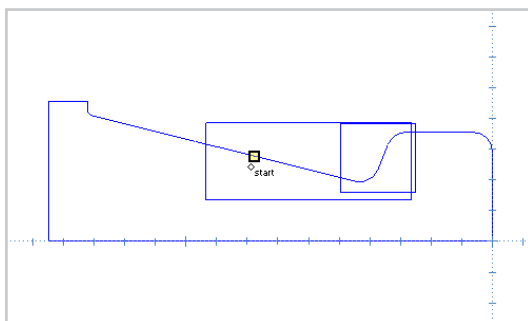


Lathe Toolpaths

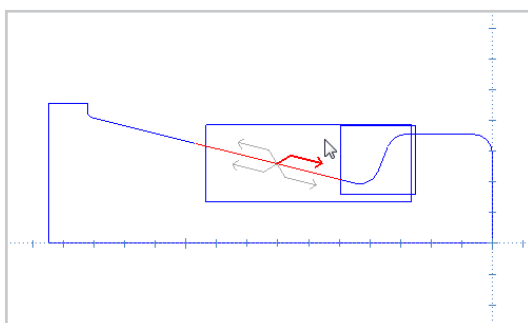


Turn / face finishing

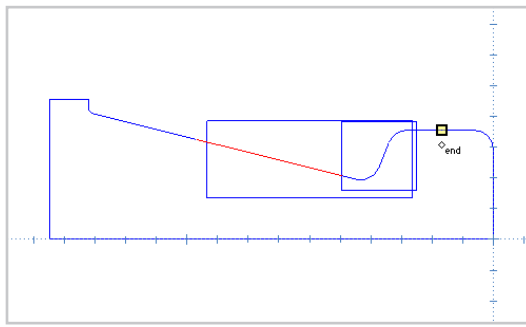
Click on the Lathe toolpaths icon and select Turn / face finishing.



Click on the part outline to start the chain selection, outside and to the left of the small rectangle.



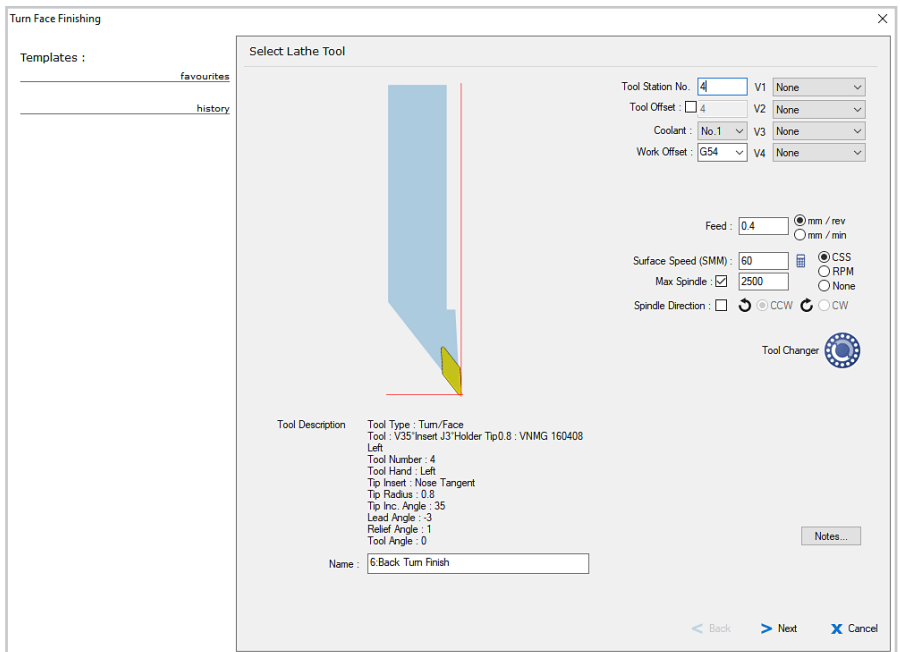
Hold the mouse pointer as shown and click to set the side and direction of the toolpath.



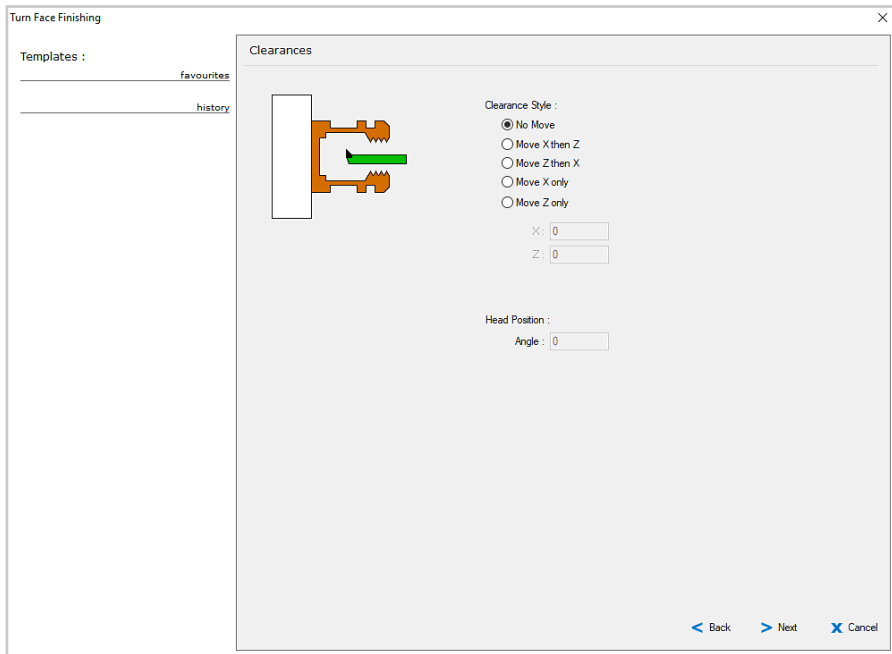
Click the horizontal line outside and to the right of the small rectangle to end the chain.



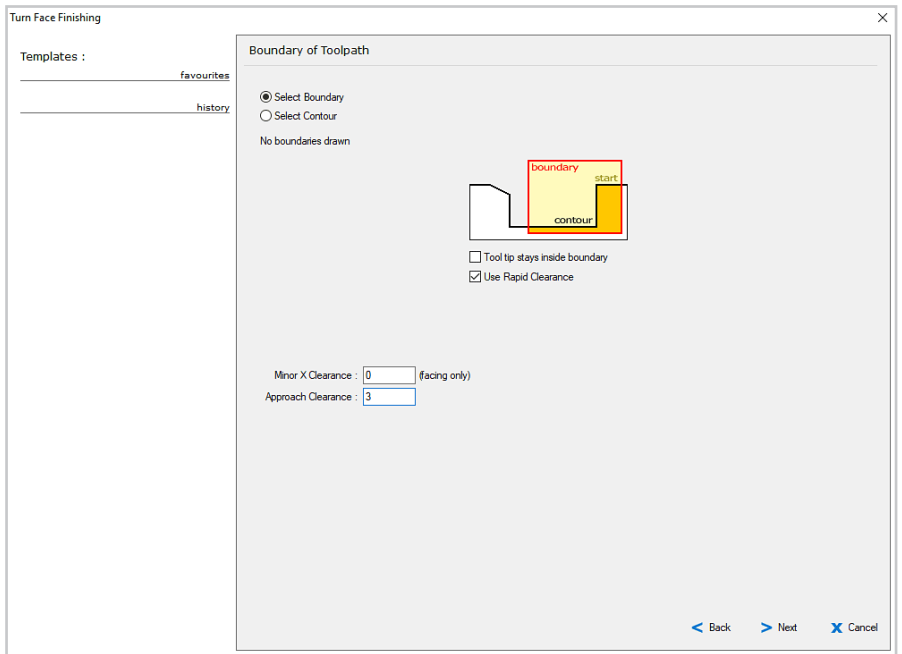
Right click to end the selection process.



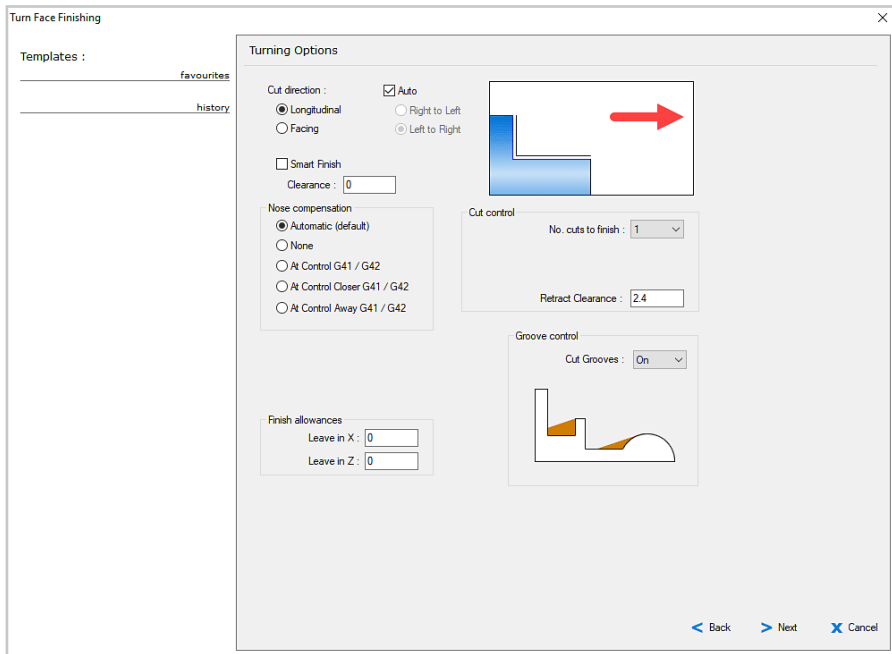
Click on the Tool Changer icon to open the Tool Library, and select the VNMG 160408 insert in a left hand J type holder. Enter a feed setting of 0.4 mm/rev, and click Next.



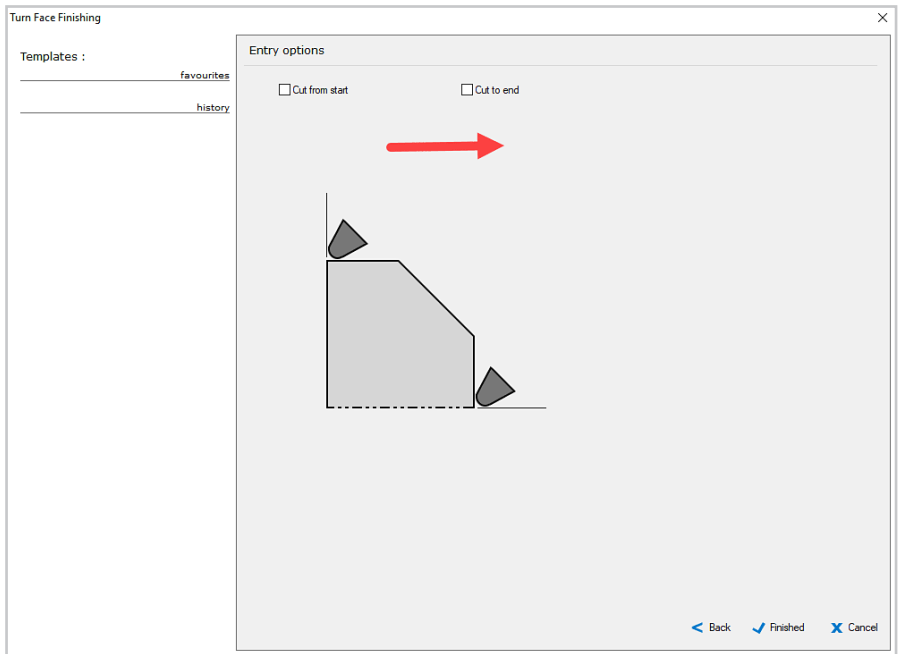
Select No Move in the Clearances dialog and click Next.



In the Boundary of Toolpath dialog, choose the Select Boundary option and click Next.



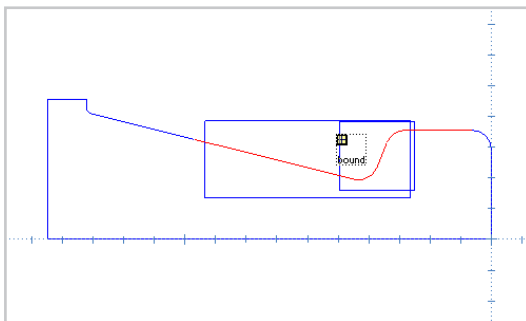
In the Turning Options dialog, select Longitudinal for Cut Direction, and click Next.



Because the limits of this toolpath will be defined by a boundary, the Cut from start and Cut to end settings can be left unselected.

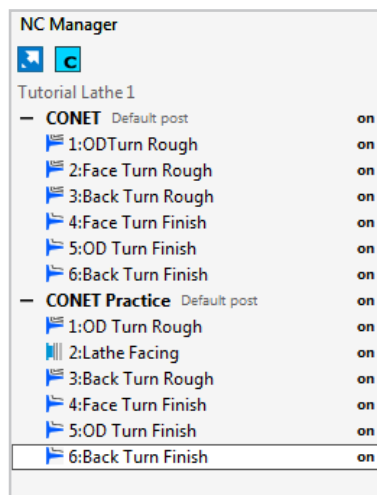
Click Finished to create the toolpath.

The message pane in the Status bar prompts for a boundary.



Click anywhere on the smaller rectangle to select it.

Rename the new toolpath to 6:Back Turn Finish.



You will now have the complete toolpaths for the part in your new group.

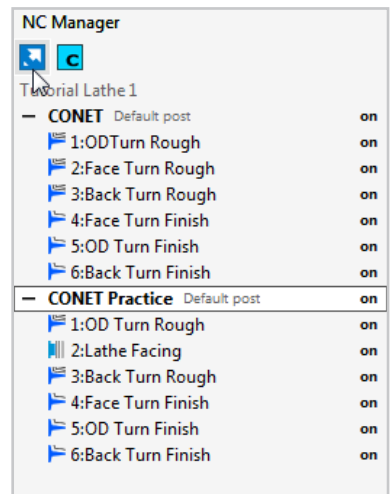
Step 7: Posting NC Code.

Now that the toolpaths are complete we can output the NC program to send to the machine.

Most CNC machines use NC programs in which the common G and M codes follow the ISO standard, but each maker uses special codes in different ways so it is necessary for OneCNC to format the file as it is output.

The formatting definition is called a 'Post', which is short for post-processor. OneCNC is supplied with prepared Posts for most machines.

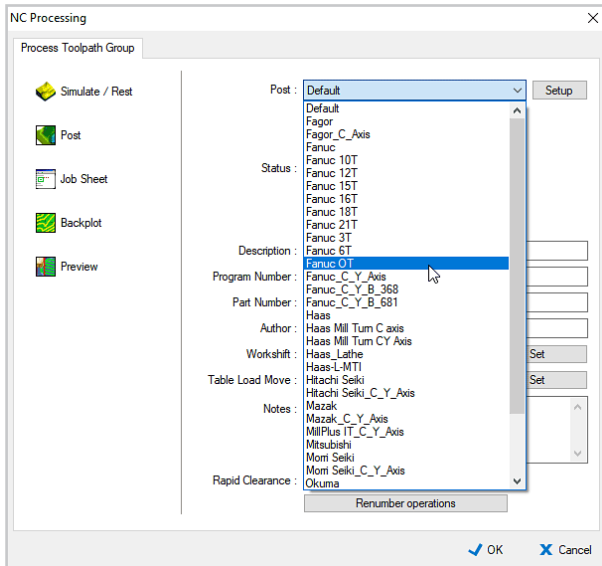
Select the heading of the CONET Practice group, and click on the Process icon at the top of the NC Manager.



Note:



Customizing a Post is covered in detail in the Help files in 'NC Post and DNC Settings', found in the OneCNC Lathe section.



This opens the NC Processing dialog which has controls and tools for post-processing your program.

Click on the drop down menu to select the post-processor for your machine from the list.

NC Processing

Process Toolpath Group

Simulate / Rest

Post

Job Sheet

Backplot

Preview

Post : Default Setup

Status : ☒ On ☐ Off

Description : CONET Practice

Program Number : 0000

Part Number :

Author : Default

Workshift : X0.000, Y0.000, Z0.000 Set

Table Load Move : None Set

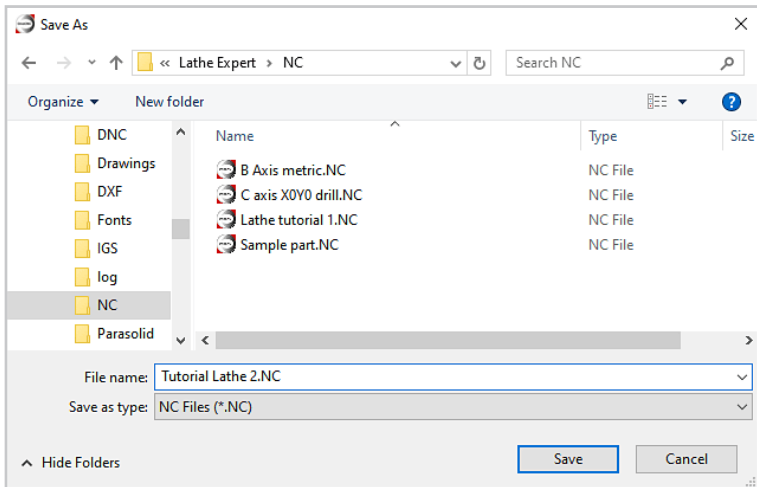
Notes : Aluminium 100 dia

Rapid Clearance : 0

Renumber operations

OK Cancel

Enter any additional information in the text fields of the NC Processing dialog and click on the Post icon to output the NC program.



The file Save As dialog appears. Choose the folder to save the file to, and enter a name for the file.

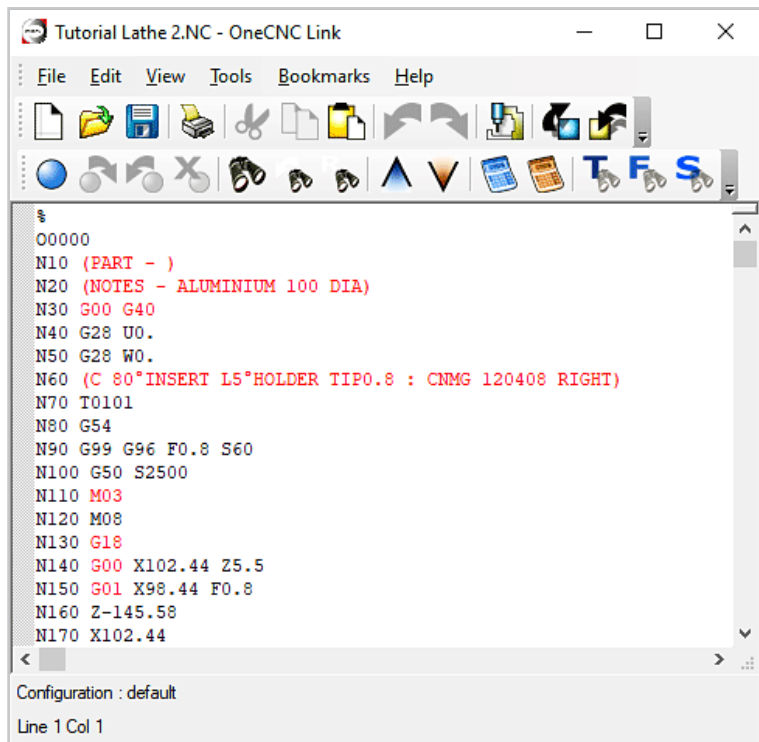
Select the file type from the drop down menu on the Save As dialog.

Click Save to create the file.

Note:



If your machine requires a file extension that is not listed, type the filename with dot and file extension in the File Name field and set 'Save as type:' to 'Text or Ascii files'. That extension will become the default extension, and will be added to the 'Save as type:' list.



Your NC file opens in OneCNC Link, which is an independent text editor and RS232 communications program.



When you first start using OneCNC you should check the output is formatted correctly for your machine. Once your format is proven you can post files with confidence.

Click on the Send icon on the OneCNC Link Standard toolbar to open the communication dialog for your machine.

Note:



Comprehensive information on how to use the OneCNC Link NC Editor, and setup communications, is available in the OneCNC Link Help files.

OneCNC Lathe Tutorial 3

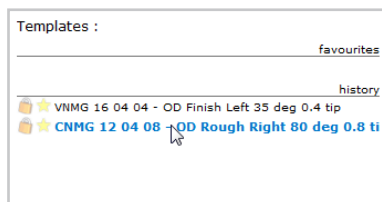
Using Toolpath Templates

This tutorial is intended for all levels of OneCNC Lathe.

As you have worked through these tutorials, you will have noticed the Templates area on the left of the toolpath wizard dialogs. Toolpath settings are saved automatically by OneCNC in the history section of the Templates panel so you can re-use them easily. The settings are available in any part you are working on, so you can confidently apply tested settings to new parts.

Using saved toolpath template settings

Every time you create a toolpath, template settings are automatically saved with default names based on the tool used.



To use a template, double click on the template name in the template panel, and the settings that were used for that toolpath will be reloaded in all the toolpath wizard dialogs.

You can then step through the toolpath wizard to check the settings, and click Finished to create the toolpath.

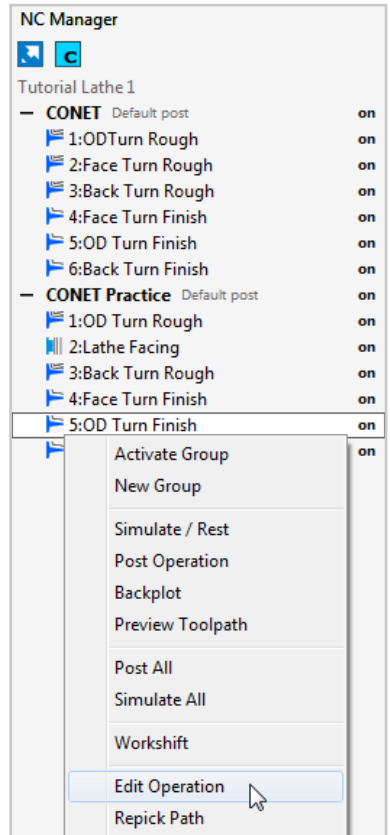
The settings can still be changed as you check them, which makes it easy to create a toolpath which is similar to but varies from the original.

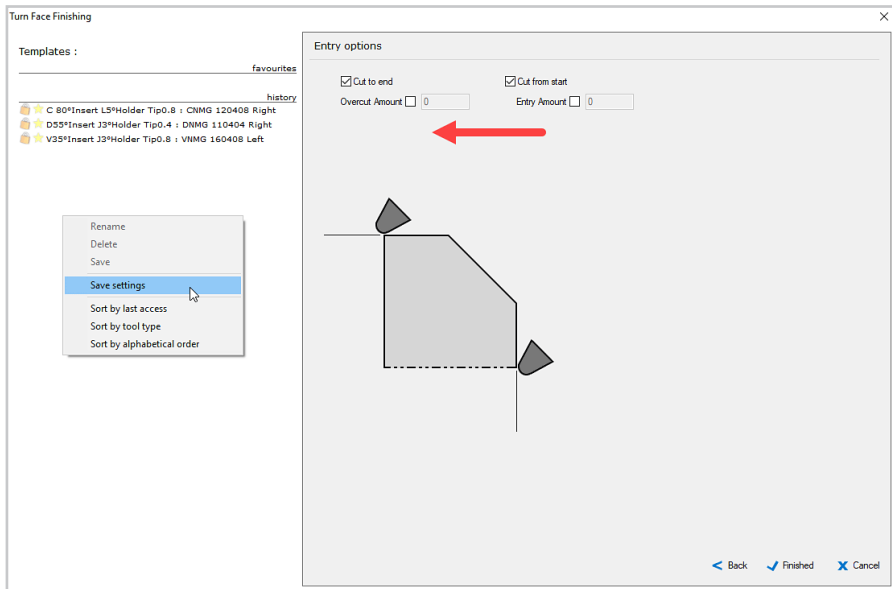
Saving a template as a favourite

While doing the finishing on our part, we created an OD Turn finishing operation.

This is an example of the type of toolpath which is a good candidate for saving as a favourite.

To access the settings, right click on the finishing operation, and select Edit Operation from the context menu.

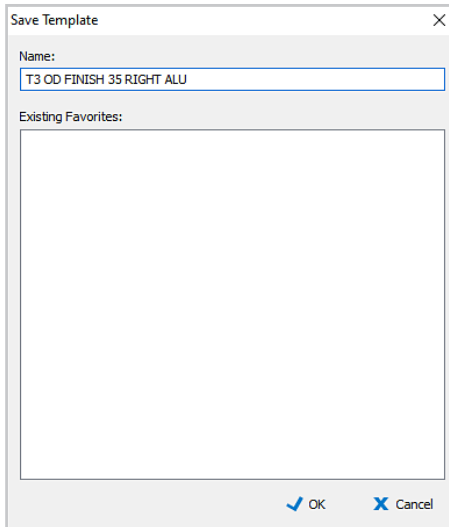




Before saving a history template as a favourite, step through the toolpath wizard, checking the tool and settings as you go, until you reach the last dialog of the wizard.

Right click in the area at the left of the screen, and select Save settings from the context menu.

The Save Template dialog opens. Type a name for your operation in the Name: box.



This operation has been named 'T3 OD FINISH 35 RIGHT ALU'

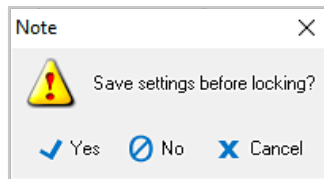
It is best to establish a naming system that records the most important identifying details of the template. This is very important in a team environment, but even if you are the only user of the system a good naming convention will help you find the template when you need it.



Once saved, the template appears in the favourites list of the Templates panel.



To guard against accidental editing, lock the template by clicking on the padlock icon next to the star.



Click Yes to save the settings as they are before locking.



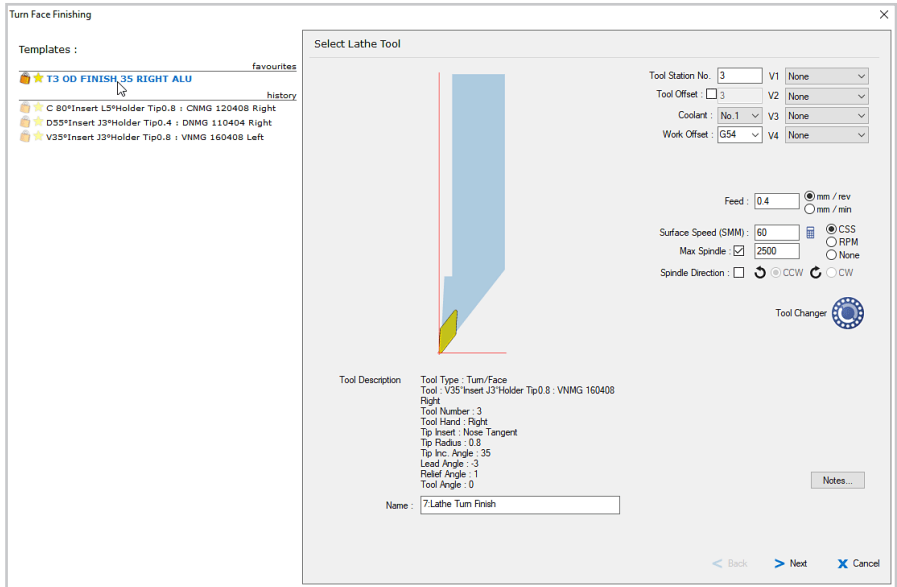
The template is now locked, and can be re-used with confidence.

It is important to understand that a template is simply a method of pre-filling the wizard settings, which can still be changed after the template has been selected.

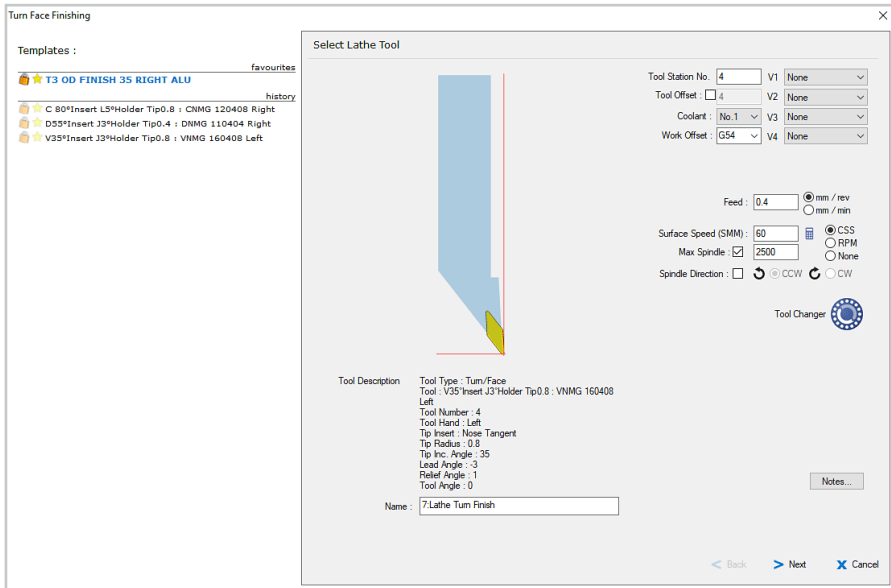
Creating templates from templates

One time-saving way to build your template set is to use an existing template as a basis for a new one.

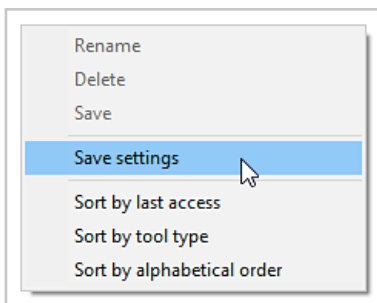
In this example we have saved a template using the 'OD Finish Right 35 deg .4 tip' tool. We can easily create a template for the equivalent left tool.



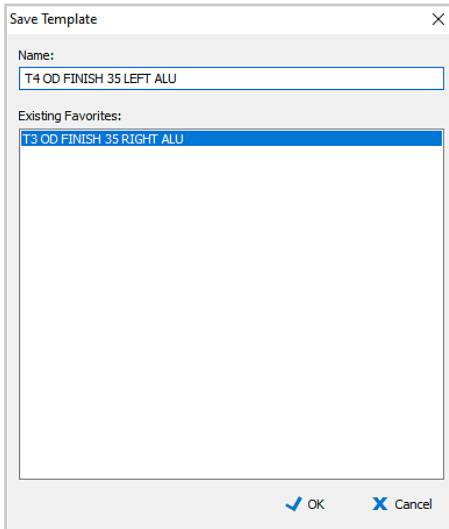
Start a new Turn Face Finishing operation, and load the T3 OD FINISH 35 RIGHT ALU template.



Click on the Tool Changer icon to open the Tool Library, and select the VNMG 160408 insert in a left hand J type holder.



Right click in the Templates panel, and select Save settings.



Save the current settings as 'T4 OD FINISH 35 LEFT ALU'.



You now have two templates with the same settings except for the tool being used.

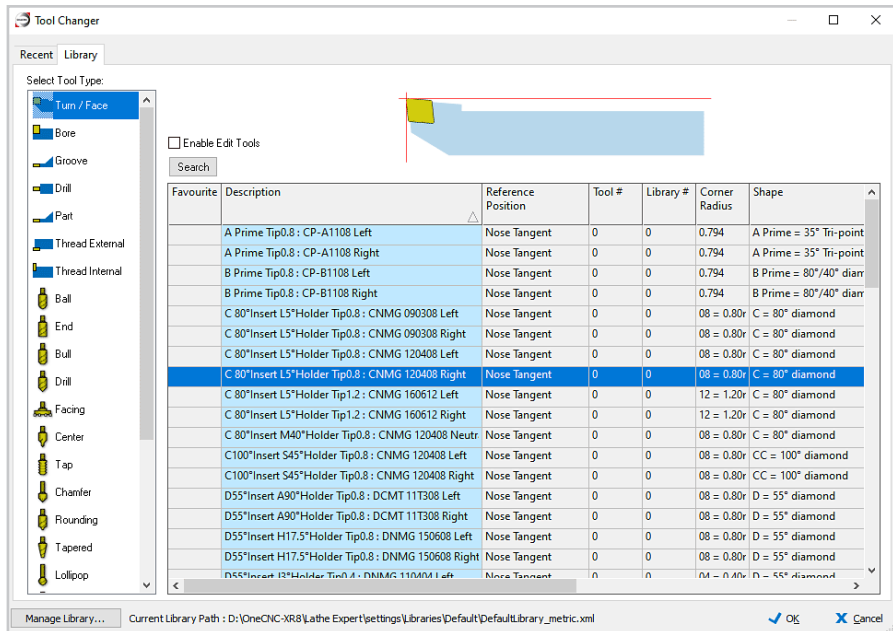
The original template will be unaffected.

OneCNC Lathe Tutorial 4

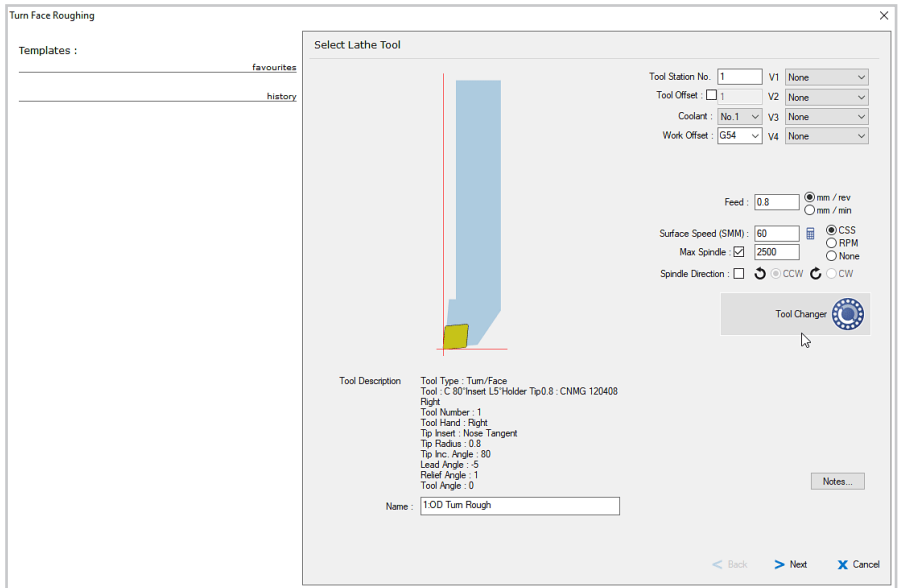
The Lathe Tool Library

The Lathe Tool Library

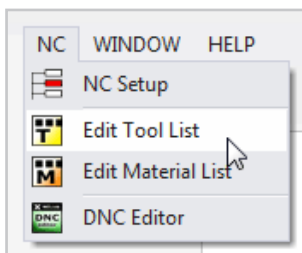
This tutorial is intended for all levels of OneCNC Lathe.



This tutorial will explain how to use and manage the Lathe Tool Library, which is a built-in database in which you can store definitions for all the tools you use.

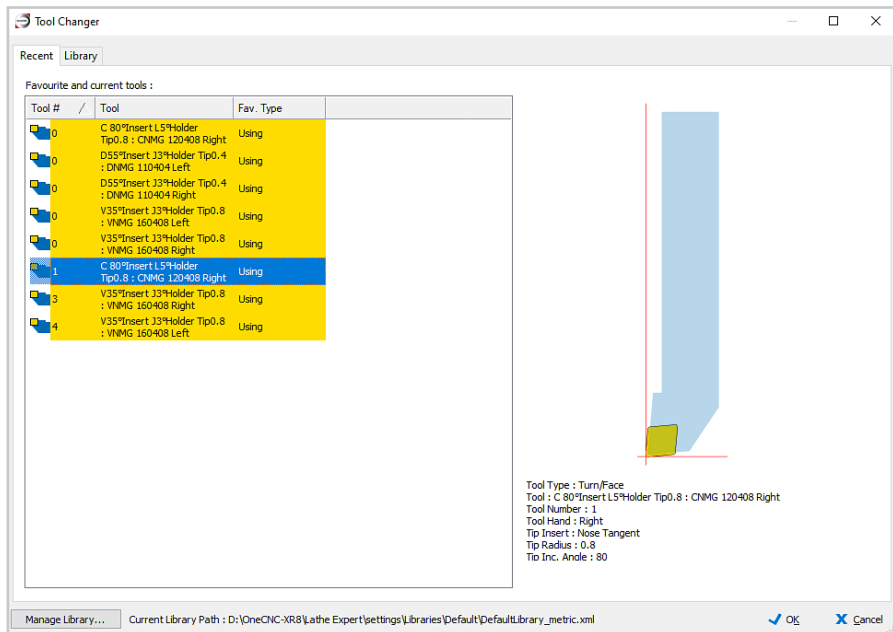


You will usually access the lathe Tool Library from the Select Tool dialog in a toolpath wizard, by clicking on the tool image or the Tool Changer icon.



When you are not in a toolpath wizard, it is also accessible by clicking on Edit Tool List in the NC menu.

There are two tabs in the Tool Library dialog.

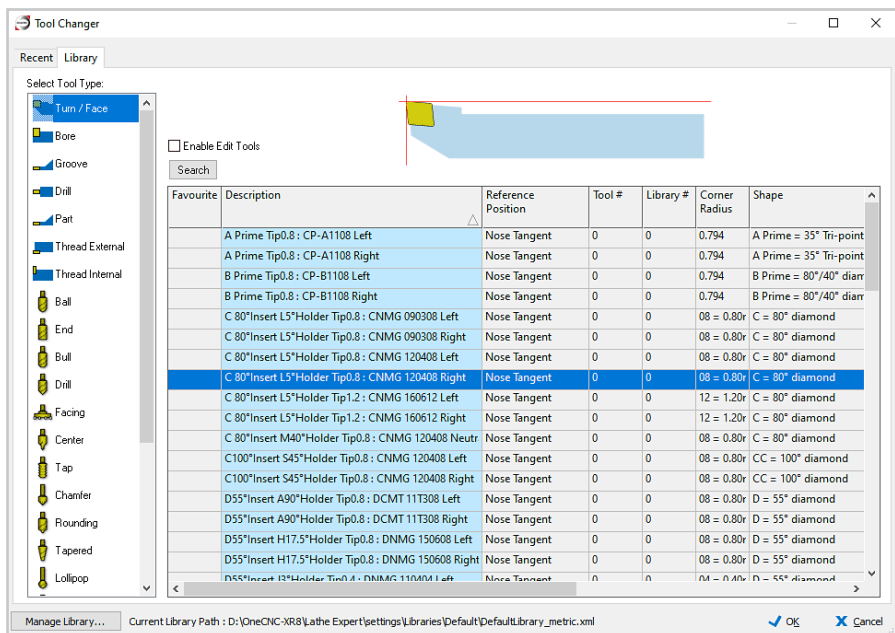


The Recent tab tool list makes it easy to select tools that you use often.

The list shows tools that are already in use in the current file.

The list also shows tools tagged as favourites in the Tool Library and tools that have been used in recent files.

Any of these tools can be selected for the current toolpath. A preview of the selected tool is shown beside the list.



The Library tab contains your complete tool library, with a separate list for each tool category.

The tool categories are shown at the left of the tab.

To select a category, click on the tool type icon and the list for that category will be shown.

Each row in the list stores information for a single tool. You can select a tool by clicking anywhere in the row. At the top of the Tool Library tab you will see a preview of the selected tool.

Note:



The tool library lists mill tools as well as lathe tools.

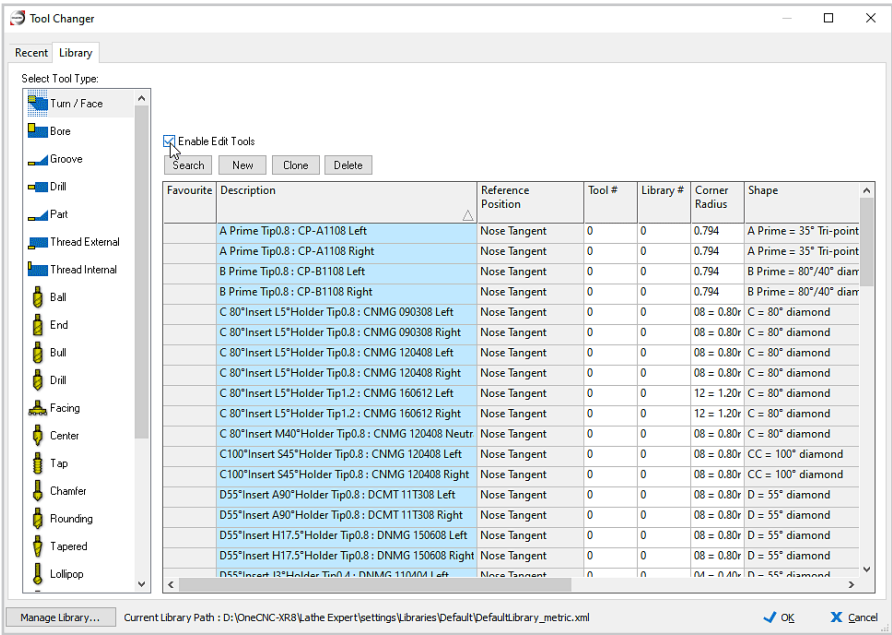
You can only select lathe tools for lathe toolpaths.

If you have an optional Mill-Turn module you will be able to select mill tools are for use with Mill-Turn toolpaths.

Adding a tool defined as an insert only

A tool can be defined as an insert only, or an insert and holder combination. An insert definition is all that is needed for the calculation of toolpaths.

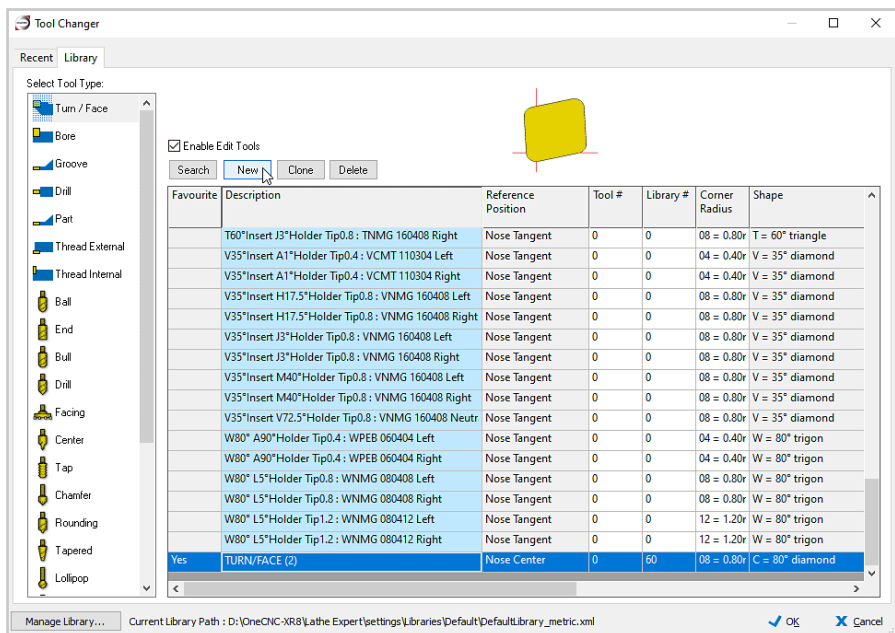
In this section we will go through the process of adding a tool defined as an insert only, and explain the various settings as we go.



The Library normally opens as a read only list to prevent accidental modification.

Select the Enable Edit Tools check box.

The Search, New, Clone, and Delete buttons will appear, and the background color of the list will change to remind you the list is in edit mode.



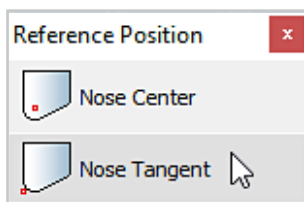
Click on the New button.

A new tool definition row will be added to the Tool Library.

Click on YES in the Favourite column to remove it from the Favourites list.

Description:

The default description in this category is TURN/FACE (1). Click on the description to select the cell and click again to open it for editing. Name the new tool CNMG 09 03 04 - OD Finish Right 80 deg 0.4 tip



Reference position:

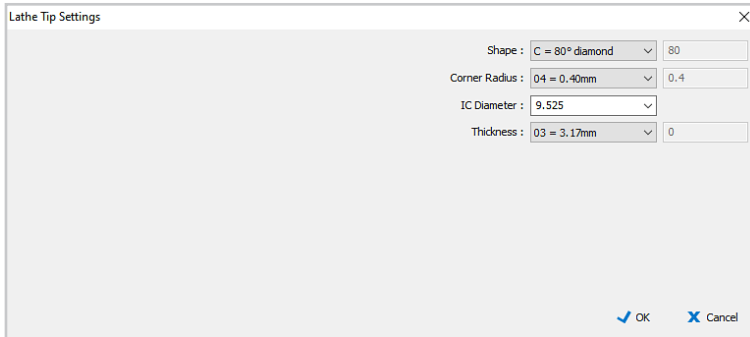
This is the zero/reference point of the tool used for calibrating the tool on the machine. Click in the entry box and the selection dialog will open. Click on Nose Tangent to select it.

Tool#:

Enter 1 for the station number the tool will be held in on the machine.

Library#:

This is a catalog number for identifying the tool, which should be unique in your system, as the Search function will return the first instance of the tool number it finds. The new tool is assigned the first available number automatically.

A screenshot of the 'Lathe Tip Settings' dialog box. The dialog has a title bar with 'Lathe Tip Settings' and a close button (X). Inside, there are four rows of settings, each with a label, a dropdown menu, and a text input field. The settings are: Shape (C = 80° diamond, 80), Corner Radius (04 = 0.40mm, 0.4), IC Diameter (9.525), and Thickness (03 = 3.17mm, 0). At the bottom right, there are 'OK' and 'Cancel' buttons with checkmark and X icons respectively.

Shape :	C = 80° diamond	80
Corner Radius :	04 = 0.40mm	0.4
IC Diameter :	9.525	
Thickness :	03 = 3.17mm	0

Click in the Corner Radius entry box and the Lathe Tip Settings dialog will open. Enter the tip settings as shown.

Shape:

The insert shape is defined by the first letter of the insert code. Select C = 80° diamond.

Corner radius:

The corner radius is defined by the last number of the insert code. Select 04 = 0.40mm

IC Diameter:

The first number of the insert code denotes the inscribed circle diameter that defines the size of the insert. Enter 9.525.

Thickness:

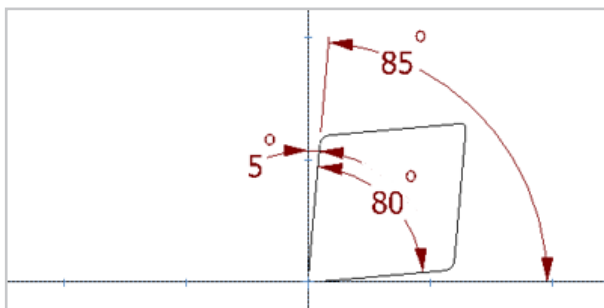
The insert thickness is defined by the second number of the insert code. Select 03 = 3.17mm

Click OK to return to the tool list.

Lead angle:

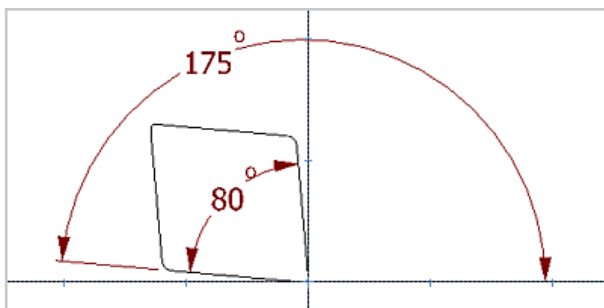
If you select a holder, you do not need to enter the lead angle, it will be set automatically by the holder style.

If you do not select a holder, this is the angle between the leading edge of the insert and the positive direction of the lathe Z axis.



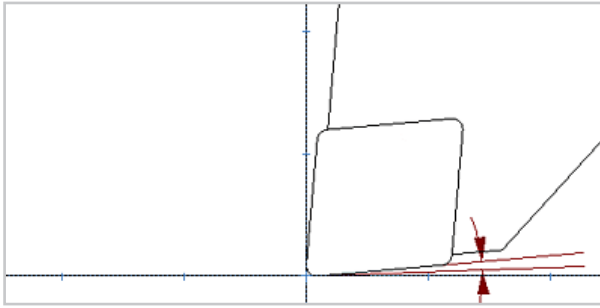
To find the angle for a right hand tool, subtract the angle between the tip and perpendicular from 90. For an 80° tip it is $90-5=85$, so enter 85 for the Lead Angle.

Note:



For a left hand tool, add 90 to the angle between the tip and perpendicular, and then add the tip angle. If we were adding a left hand tool the Lead Angle would be $90+5+80=175$.

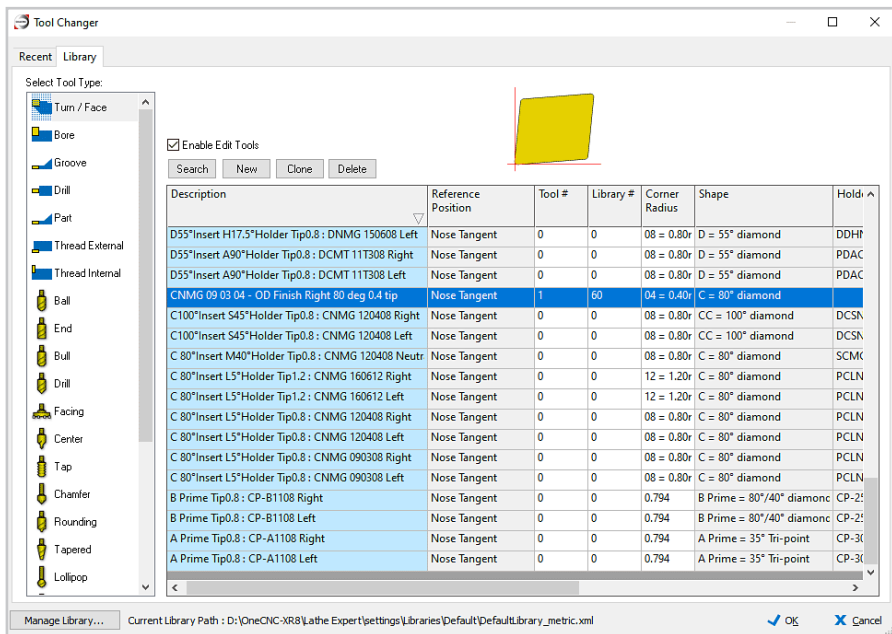
Relief angle:



This is a clearance angle from the side of the tool opposite the leading edge, which is used by OneCNC to determine the approach angle to the part. Set this to the default value of 1.

Color:

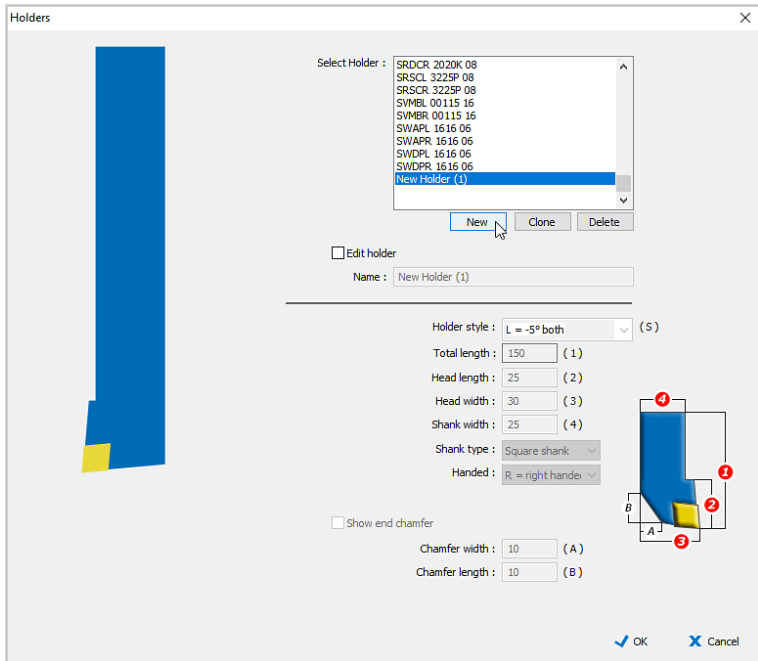
Click in the color box to pick a color to associate with the tool.



The insert tool definition is now complete and a preview of the insert will be shown. An insert definition is all that is needed for the calculation of toolpaths. The tool could be used as it is, but we will create a holder for the new tool in the next section of this tutorial.

Adding a tool holder

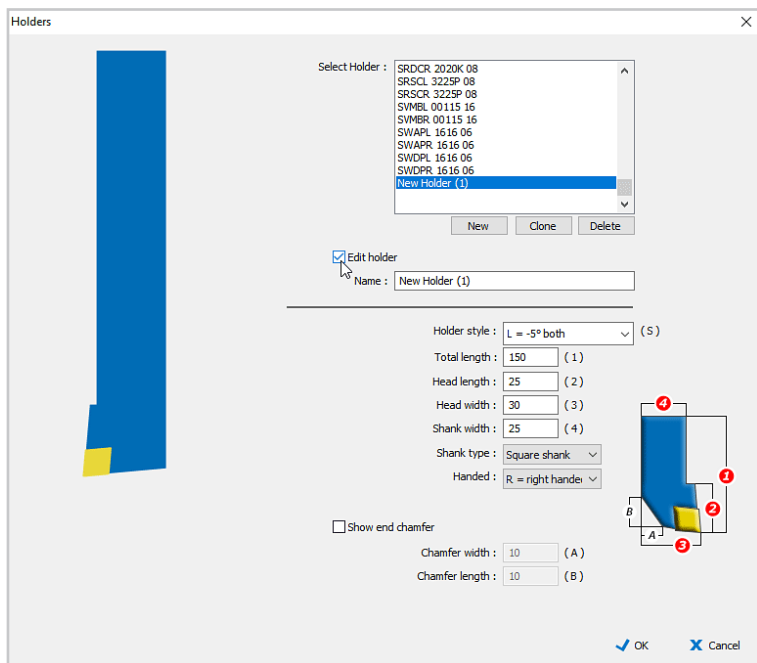
If you add a holder it will be displayed when the toolpath is previewed or simulated, providing a visual clearance check when simulating toolpaths.



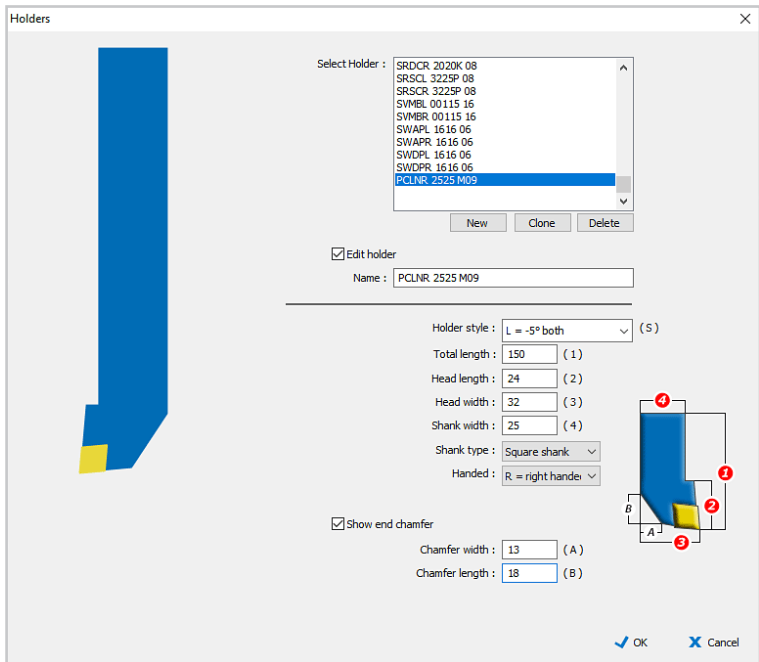
Click in the Holder box in the tool list and the Holders dialog will open.

The dialog displays a list of holders that have already been defined. If the insert you are using fits an existing holder you can select the existing holder and click OK.

We need a new holder for the insert we have just added. Click the New button to create a new holder.



Select the Edit holder check box to activate edit mode.



Enter the holder name PCLNR 2525 M09.

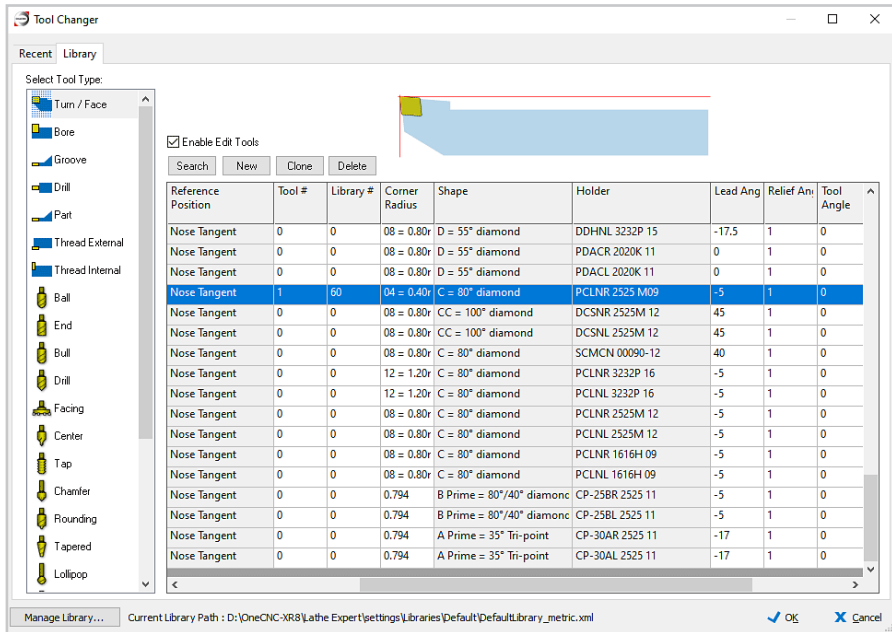
The third letter of the holder code is the style, so select L = -5 deg side (both).

Enter total length 150, head length 24, head width 32, and shank width 25.

Select the square shank type and R = right handed.

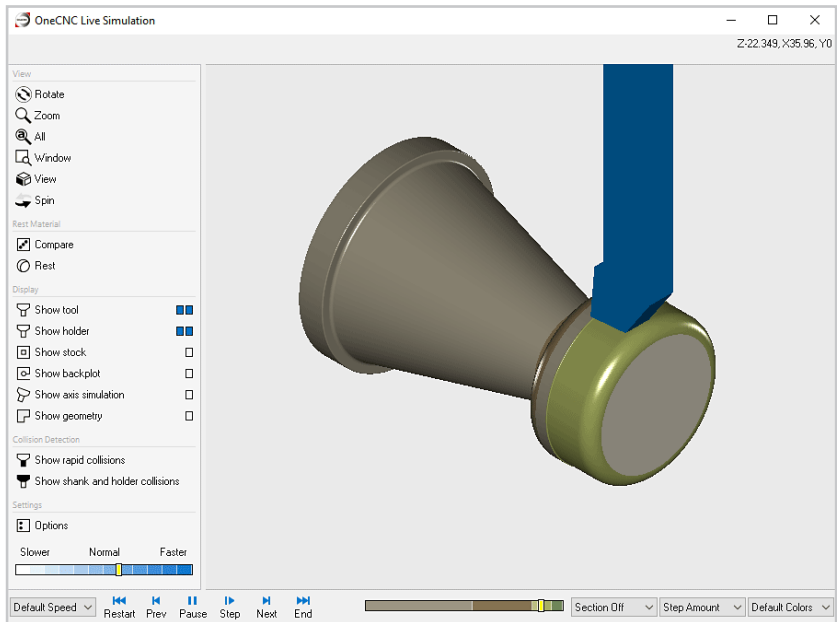
Select the check box for Show end chamfer, and enter chamfer width 13 and chamfer length 18.

Click OK to accept the new holder for the current tool.



The tool preview now shows the insert we created in the previous section in the new holder.

Note that the lead angle has been set automatically by the holder style. The holder style lead angle is measured from vertical so an insert lead angle of 85 from horizontal is the same as a holder style lead angle of -5.



Create a toolpath using the new tool and simulate it.

You will see the tool simulation using the automatically created holder and insert models.



The tutorials in this manual cover the basic elements of OneCNC Lathe toolpaths.

For more in-depth tutorials, see the OneCNC Lathe Tutorials in the OneCNC Help files.

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.

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