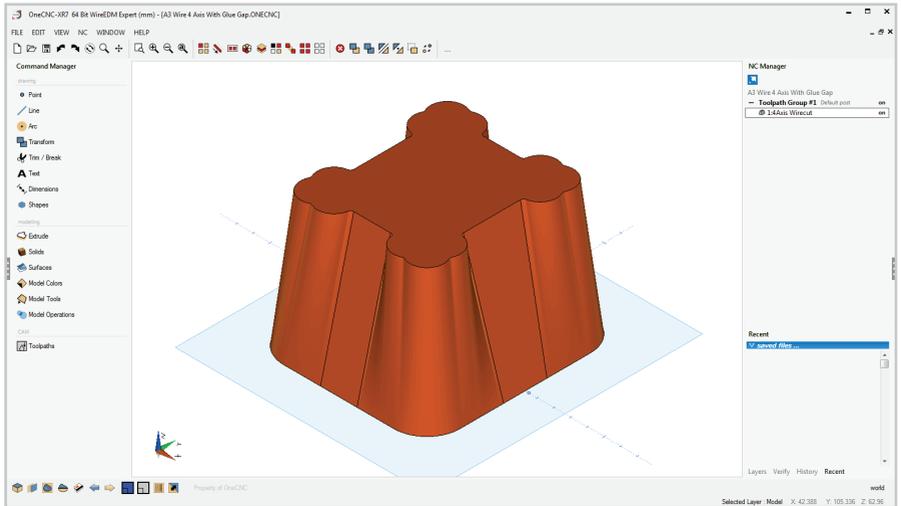


OneCNC WireEDM



OneCNC Wire EDM is a hybrid CAD-CAM system with a user-friendly interface for designing and wire-cutting parts using 2 or 4 axis CNC machines.

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 WireEDM 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.

CONTENTS

OneCNC WireEDM	1
OneCNC Environment	5
OneCNC Installation	6
Minimum System Requirements	6
OneCNC Installation	6
OneCNC Updates.....	9
Series 2 Dongle Installation - dongle numbers from 10000	10
Dongle Security	10
Series 1 Dongles - dongle numbers 1 to 9999	11
Start OneCNC and enter the Serial Key	12
Entering serials for optional modules.....	13
User profile selection	14
OneCNC Network Installation.....	16
Running OneCNC Dongle Server as a service.....	21
OneCNC User Interface	22
Toolbox.....	25
Right Sidebar.....	26
NC Manager.....	26
Layer Manager	27
Verify	27
History.....	27
Recent Files list.....	27
Status Bar.....	28
Preset Views.....	28
Construction Plane	29
Display mode.....	29
Section Tool.....	30
3D to 2D Automated drawing system	30
Drawing Color and Style selection.....	30
Modifying Entities.....	30
Quick Access Toolbar	31
Keyboard Shortcuts	32
Mouse Actions.....	33
OneCNC Overview	34
2D and 3D Geometry	35
3D Modeling	36
Toolpathing	37

Toolpath management	37
Toolpath Backplot	37
Metal Removal Simulation (OneCNC Mill and OneCNC Lathe only) ..	38
Outputting NC Files	39
OneCNC Integrated Help	40
Help	40
Tutorials.....	40
OneCNC CAD Tutorial 1.....	41
<i>Introduction To Cad Drawing</i>	41
Start a new drawing	41
Lines	42
Rectangle.....	47
Circles	49
Offset entities and boundaries.....	51
Moving Entities	55
Copy Entities	57
Rotate	59
Setting Current Drawing Properties.....	61
Trimming Entities	65
Chain Selection	70
OneCNC CAD Tutorial 2.....	74
<i>2D Drawing.....</i>	74
Step 1: Draw outer profile	75
Step 2: Draw the angled slot	82
OneCNC CAD Tutorial 3.....	89
<i>Solid Modeling by Extrusion</i>	89
Step 1: Complete CAD Tutorial 2 and save a copy.	89
Step 2: Create a solid by extrusion from curves.....	90
Step 3: Create the slot with an extrude cut.....	92
OneCNC CAD Tutorial 4.....	94
<i>Working With Planes.....</i>	94
Step 1: Draw a pyramid wireframe	95
Step 2: Position the plane.....	97
Step 4: Create the truncated pyramid	100
Step 5: Return to default XY plane.....	102
OneCNC Technical Drawing	103
<i>Automated 3D to 2D Drawing Layouts</i>	103
Add a drawing page with border and title block	104
Add views of the model to a drawing page layout.....	106

OneCNC WireEDM Tutorial 1	109
<i>2 axis closed profile</i>	109
Create a new part file and draw a filleted square.	109
Create a Wire start point.	110
Creating a Closed Profile Wirepath.	111
Previewing the wire path.	116
Posting NC Code.....	118
OneCNC WireEDM Tutorial 2	122
<i>2 axis open profile</i>	122
Prepare the drawing	122
Creating an Open Profile Wire toolpath.	124
OneCNC WireEDM Tutorial 3	130
<i>4 axis solid model with glue gap</i>	130
Open the sample file and save a copy to practice in.	130
Creating a 4 axis Wire toolpath with glue gap.....	131
Post the wire toolpath.	135
OneCNC WireEDM Tutorial 4	136
<i>4 axis from wireframe drawing</i>	136
Open the sample file and save a copy to practice in.	136
Creating a 4 axis Wire toolpath.	137
Backplot the wire toolpath.	140
Setting Interpolation Tolerance.....	141
Editing the start point.	144
Correcting profile geometry	147
OneCNC WireEDM Tutorial 5	151
<i>4 axis open shape</i>	151
Make a copy of an existing drawing.....	151
Prepare the drawing.	152
Create a 4 axis open shape wire toolpath.	156
OneCNC Support.....	161
OneCNC Global Offices.....	162
Copyright Notice.....	166

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:

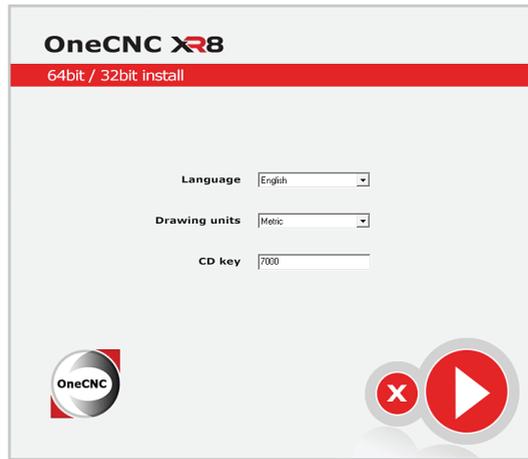
Install the software from disk or an update setup file

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://onecn-support.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.

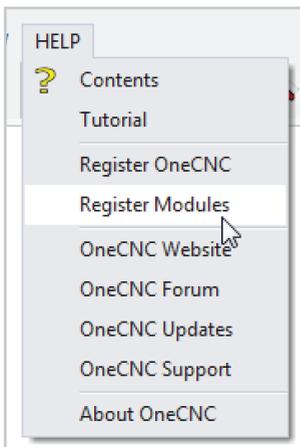
The Licence Manager will open so you can enter the licensed user name and serial number.



The 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.

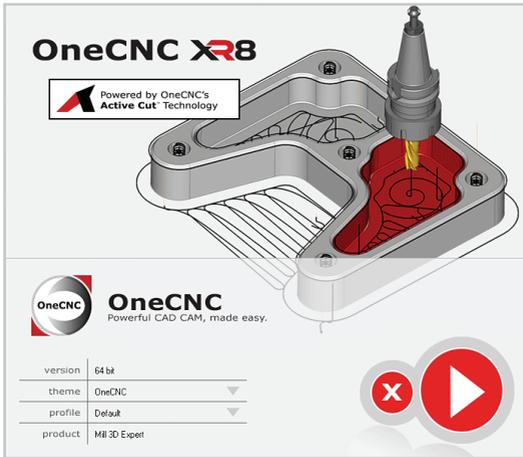
Entering serials for optional modules



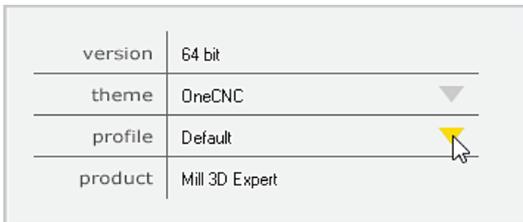
If you have a serial for an optional module, start OneCNC and click on Register Modules in the OneCNC Help menu.

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

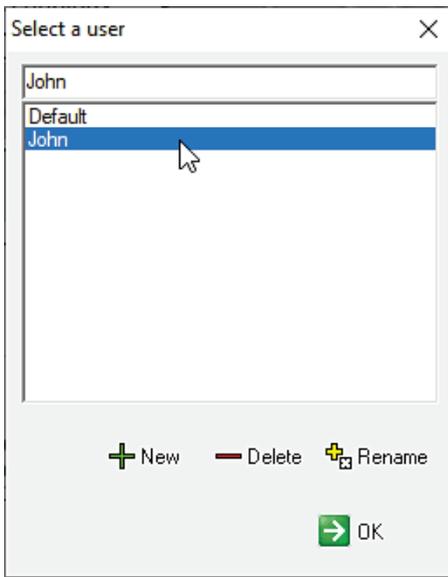
User profile selection



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.

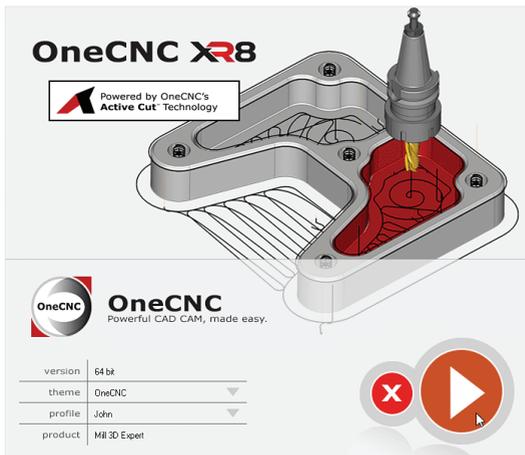


Clicking on the small arrow opens the user selection dialog, which allows you to select or create settings in the names of specific users for customizing.



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:

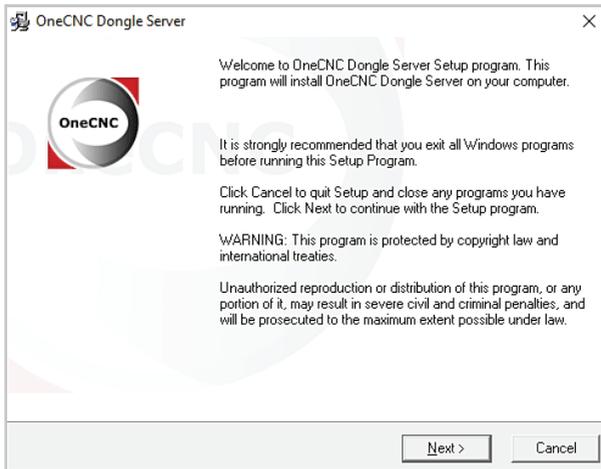
- OneCNC installer disk or a downloaded update setup file
- OneCNC dongle
- Network CD Key and serial number



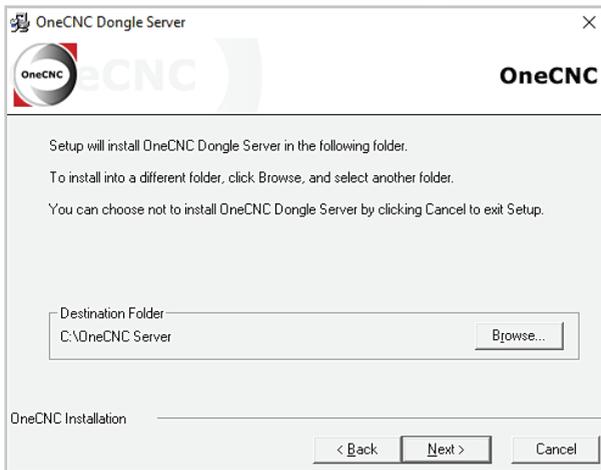
Run the installer on the server computer. To install the network dongle server software enter the CD Key 60200.



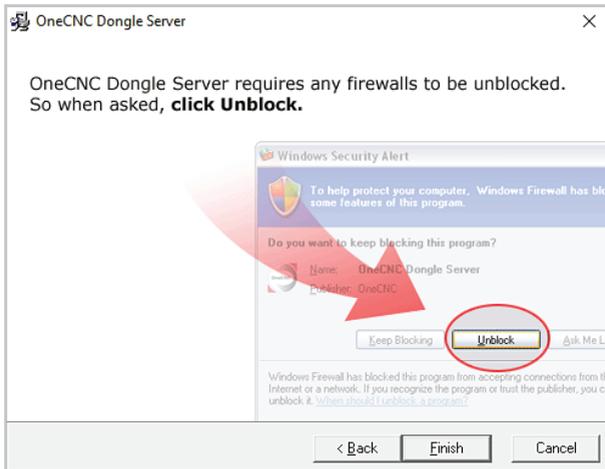
When the next dialog appears, enter the server software serial. The serial is made up of digits 0 to 9 and letters from A to F. This is not the same as the software serials which are entered later when installing the particular versions of OneCNC such as Mill and Lathe.



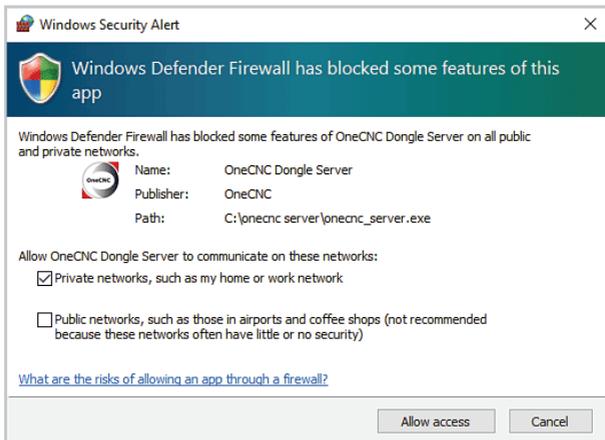
The next dialog advises you to have all other programs closed, which is standard software installation procedure.



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.



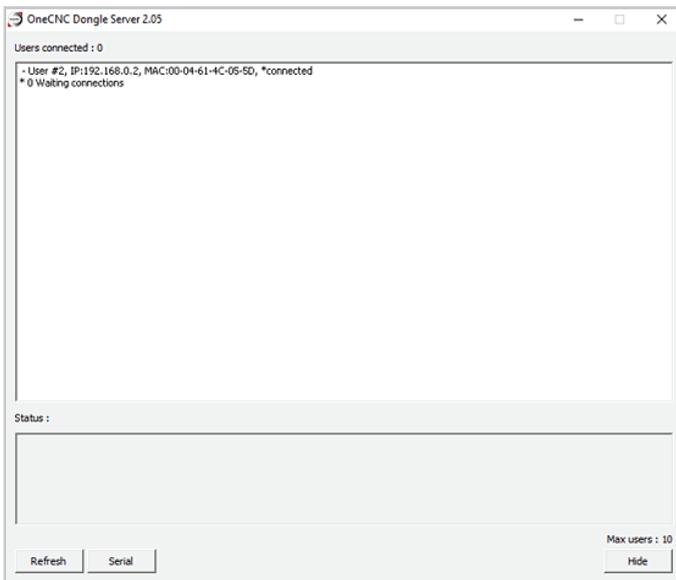
Here is the Windows Firewall dialog where you have to click Unblock for the dongle server software to work. This has no effect on network or internet security, it is only allowing any PC on the local network to access the OneCNC Dongle Server.

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

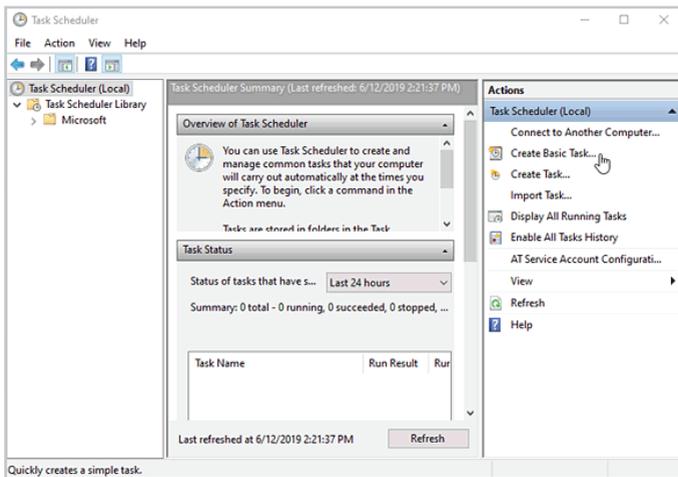
You can run OneCNC Dongle Server automatically so you do not have to start it if the server has been shut down.

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.

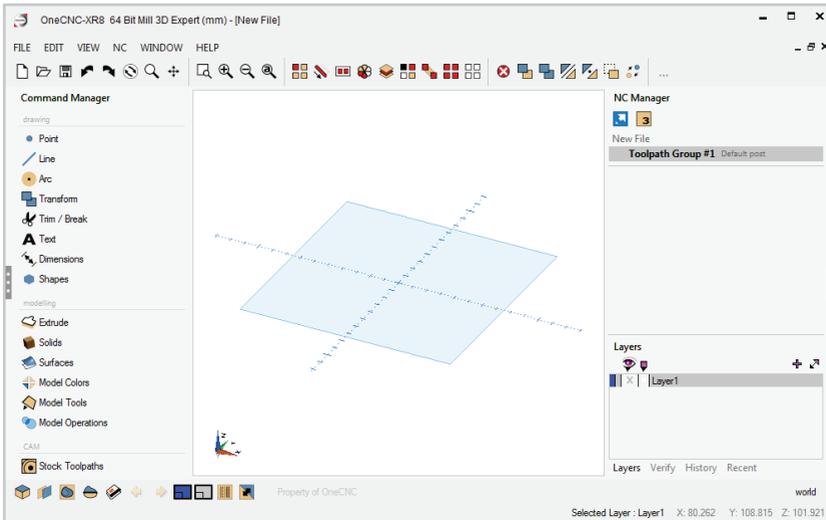
In Windows Server, open the Server Manager and navigate to Server Manager > Configuration > Task Scheduler .



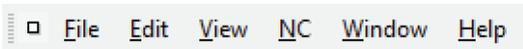
Click Create Basic Task. Enter the name and description for the task, and then click Next. Select When the computer starts, and click Next. Select Start a program, and click Next. Click Browse to find the OneCNC server program, which will typically be at C:\OneCNC Server\onecnc_server.exe Click Next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

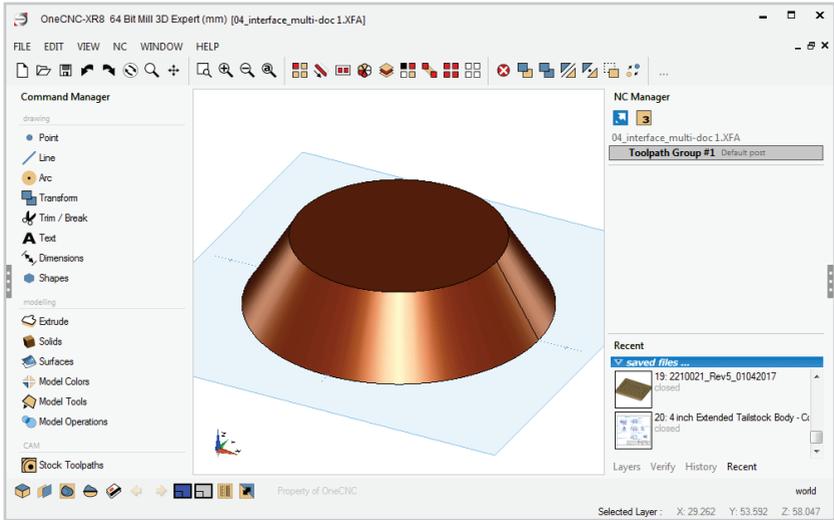


Windows menus give you access to many features in OneCNC. Menu items with a pointer on the right side automatically open sub-menus which you can slide the mouse across to select from.

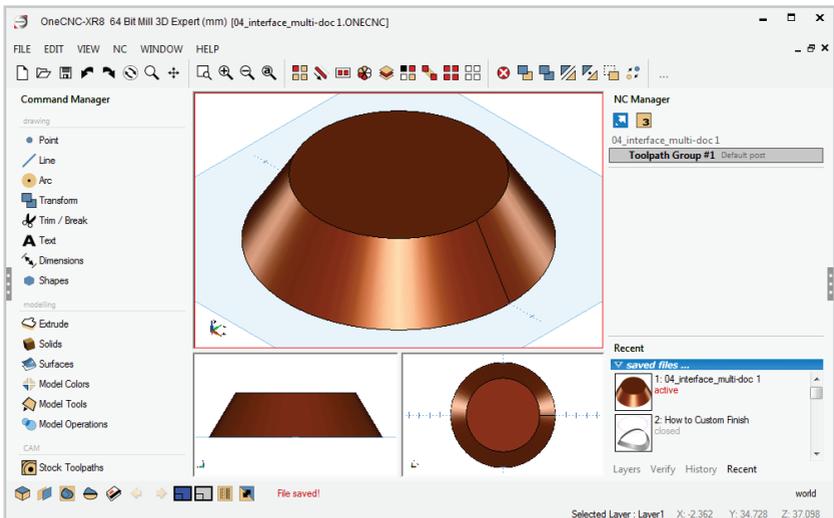


The Mill Expert interface is shown in this chapter. The available modeling and CAM functions you see will vary for different types or levels of OneCNC. OneCNC Solid Creator is intended for design only and has no CAM functions.

Drawing Window

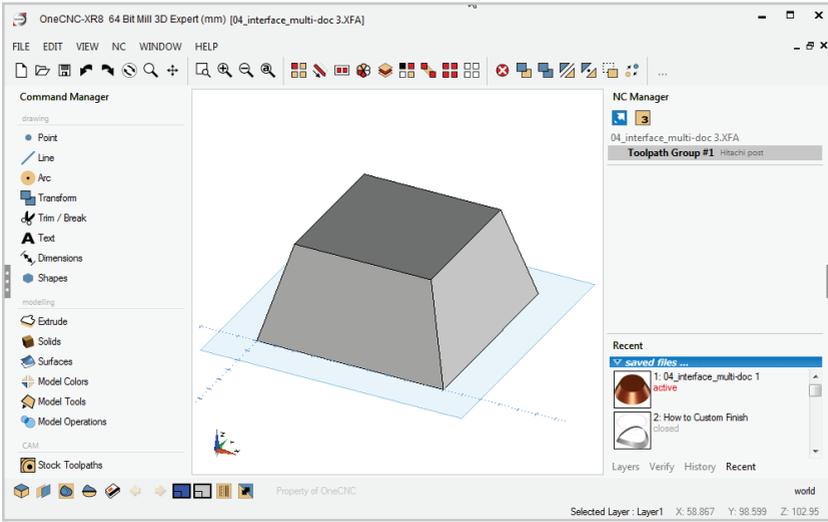


The file you are working on is displayed in the drawing window, which is a real-time OpenGL model view. This allows you to view your object by means of dynamic rotation, panning and zooming.



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.

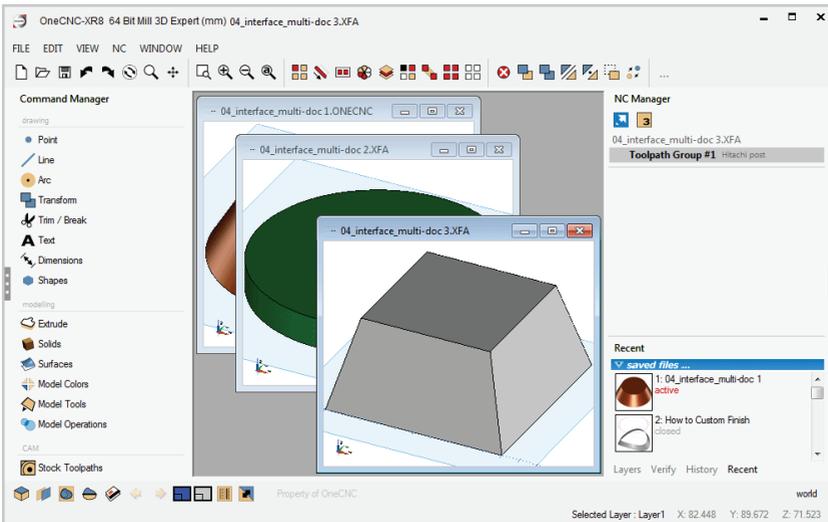
Multi Document



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.



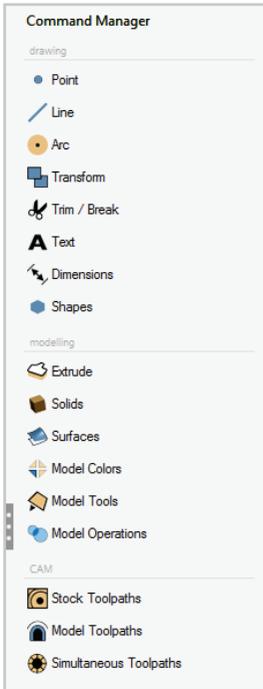
Clicking the Restore Window button will display each file in its own window.



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

Toolbox

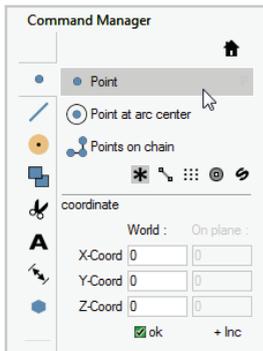
The Toolbox is a menu system which is shown in the left sidebar.



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.

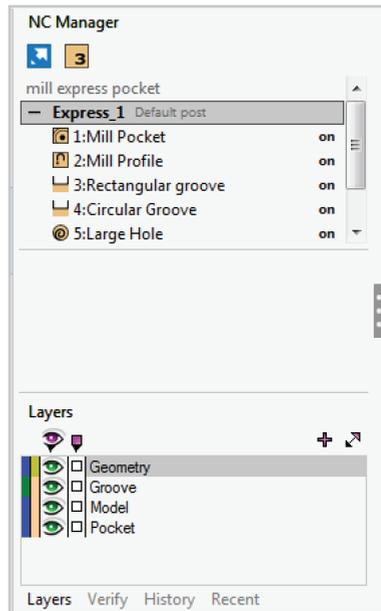
Clicking on the dotted button at the left of the toolbox will hide it.

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



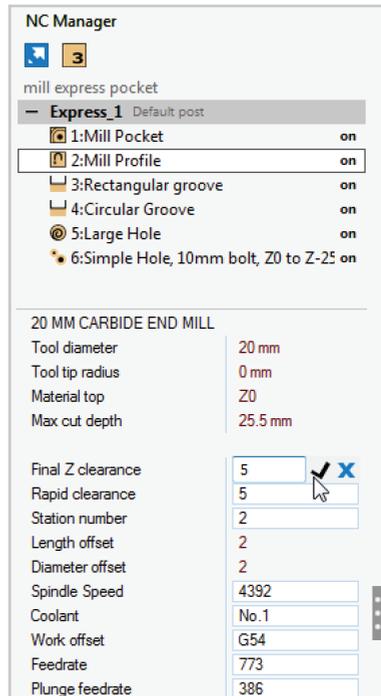
Clicking on the dotted button at the right of the sidebar will hide it.

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

NC Manager

The NC Manager displays the toolpaths you have defined.

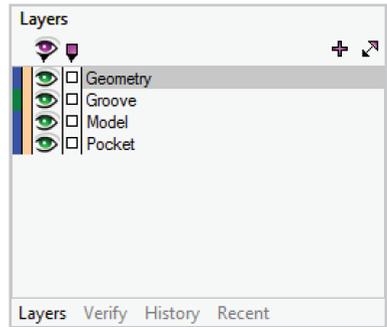
Machining operations can be grouped, and operations can be renamed, edited, rearranged or duplicated.



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.

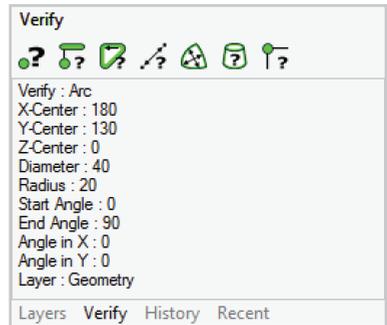
The panel shows a list of layers in a drawing. You can turn off a layer by closing the eye for that layer in the Layer Manager. Entities on the layer are still in the drawing but will not be visible.



Verify

The Verify panel holds tools to read data from geometry.

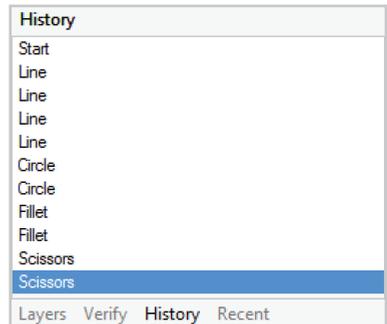
You can find exact values for position length or area for 2D entities, and volume of 3D solids. You can also find the angle and distance between entities.



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.

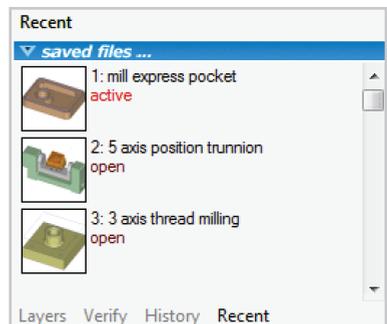


Recent Files list

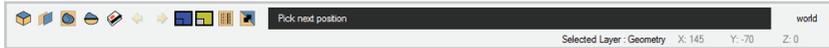
When you save a file, a thumbnail image of it is added to the Recent file list.

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.

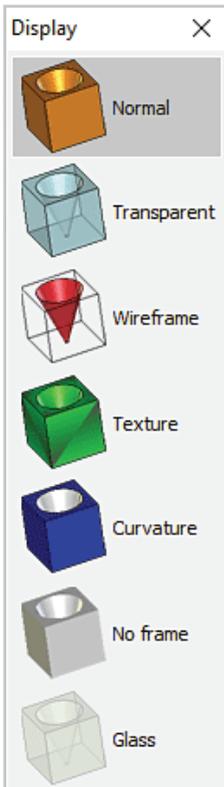
The Working With Planes tutorial in this manual covers the use of Construction Planes.

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.

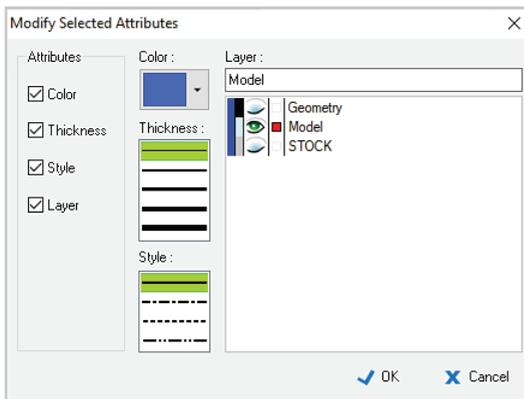


To select the line style and line thickness for new geometry entities click the Styles icon.

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.

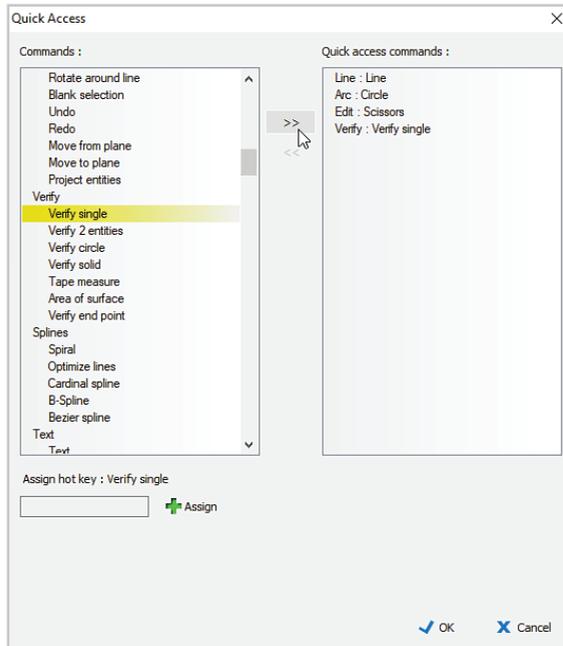
Quick Access Toolbar



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

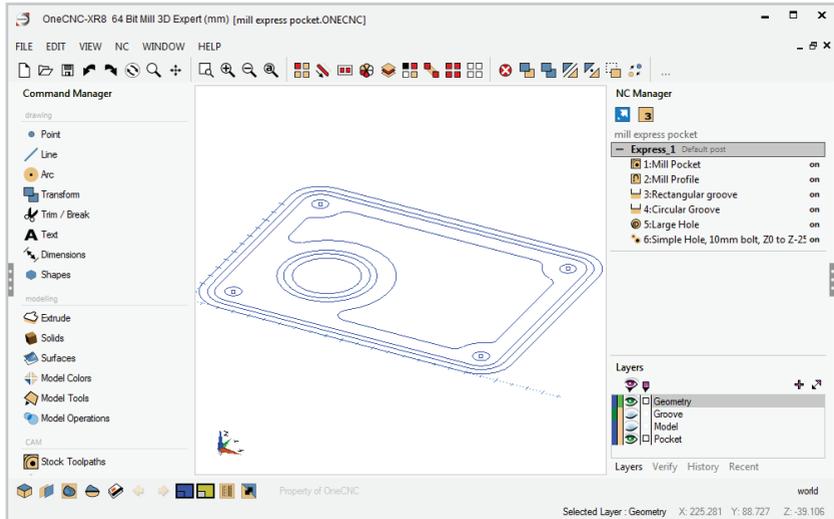
OneCNC WireEDM

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

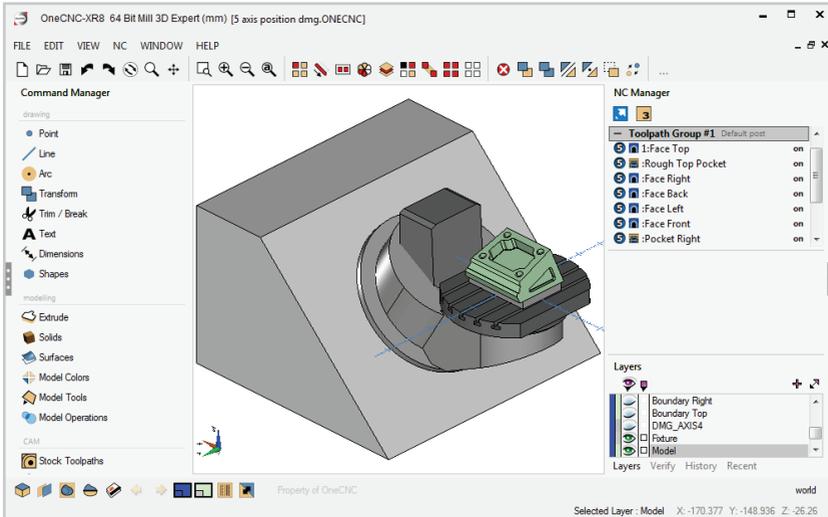
Borders, title and revision blocks, and bill of material lists can be saved as patterns and loaded into drawings as required.

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

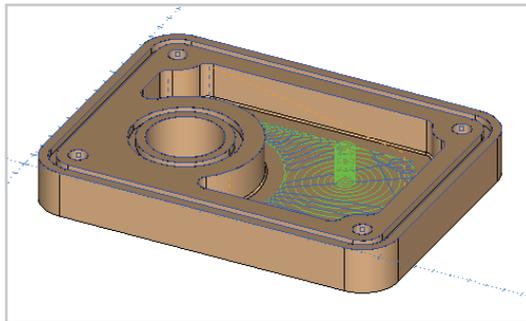
- 1: Mill Pocket on
- 2: Mill Profile on
- 3: Rectangular groove on
- 4: Circular Groove on
- 5: Large Hole on
- 6: Simple Hole, 10mm bolt, Z0 to Z-25 on

20 MM CARBIDE END MILL

Tool diameter	20 mm
Tool tip radius	0 mm
Material top	Z0
Max cut depth	25.5 mm

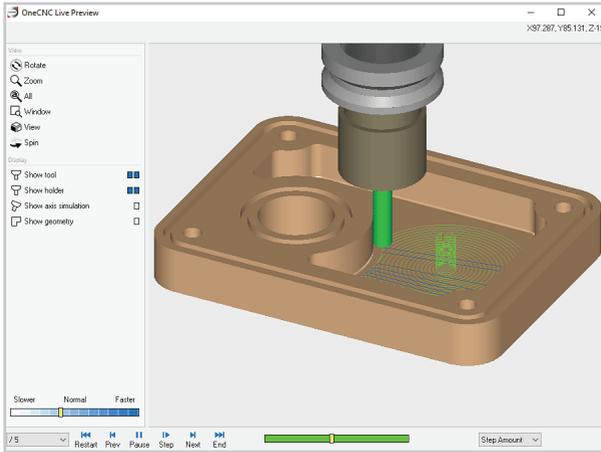
Final Z clearance	5	<input type="checkbox"/>
Rapid clearance	5	<input type="checkbox"/>
Station number	2	
Length offset	2	
Diameter offset	2	
Spindle Speed	4392	
Coolant	No.1	
Work offset	G54	
Feedrate	773	
Plunge feedrate	386	

Toolpath Backplot



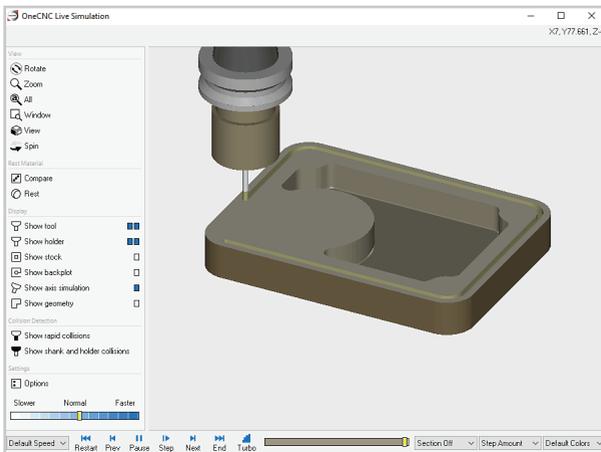
Toolpaths can be backplotted, which means they are drawn as geometry on a special layer named Backplot. Feed moves are drawn in green, and rapid moves in orange. Repositioning moves in highspeed machining are drawn in blue.

Toolpath Preview



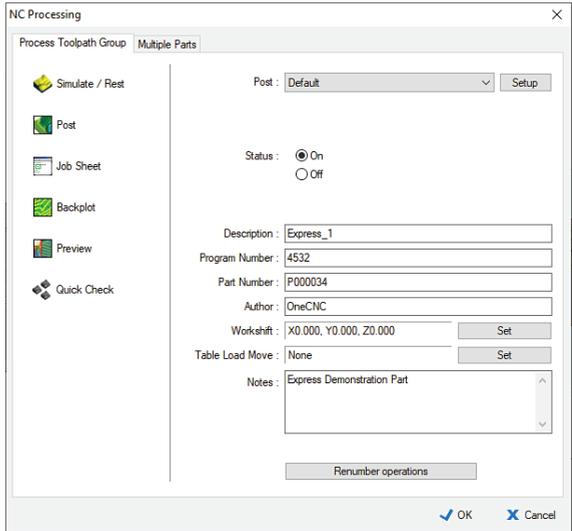
The Live Preview window shows an animated preview of the toolpath, backplotting as it goes. In Mill or Lathe the cutting tool will be shown traveling along the toolpath.

Metal Removal Simulation (OneCNC Mill and OneCNC Lathe only)

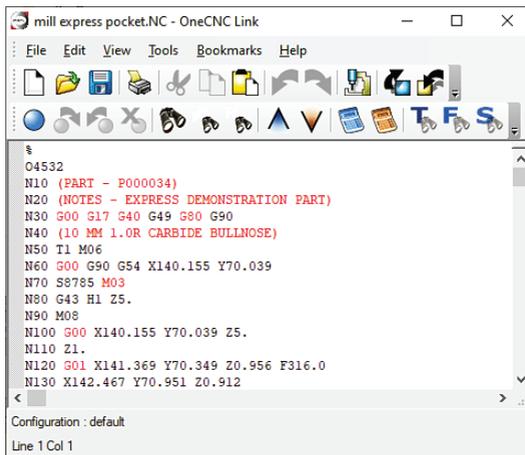


In all levels of Mill and Lathe, OneCNC provides real-time toolpath metal removal simulation. By simulating the feed and speed of the process, you can check the machining operation prior to sending to the CNC machine.

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



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.



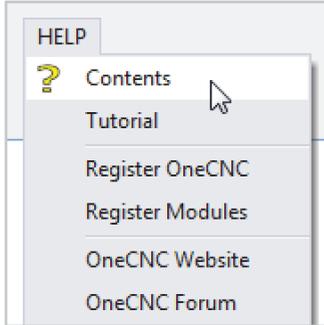
Editing and sending the created program

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.

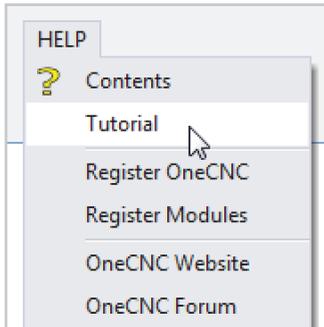
Help



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.

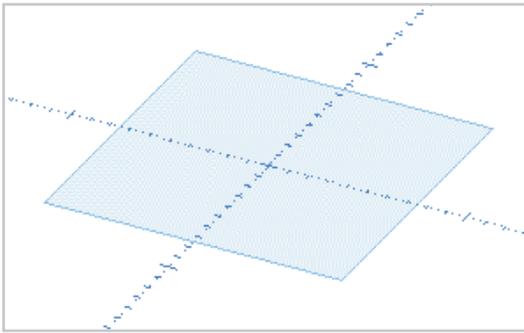


The first thing you should do when starting work in a new file is to click Save and save the .ONECNC file. If you save this tutorial you will be able to open it and resume where you left off if you are interrupted.



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.



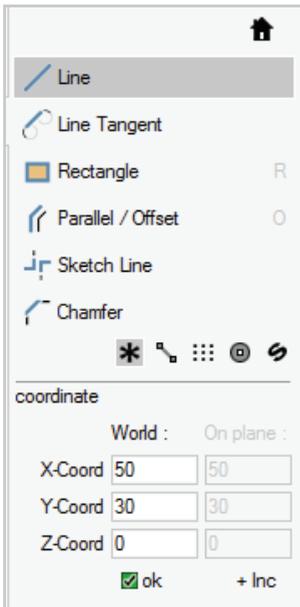
The part will open in the default 3D view. The darker rectangle is the construction Plane, which indicates the XY plane of the coordinate system you will be drawing in. Later you will see how you can move and rotate the Plane to define a new coordinate system but for now we will leave it in the default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

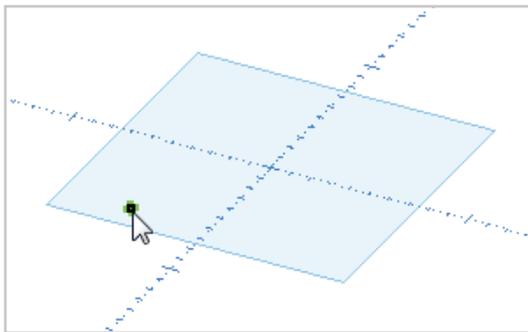


The Line icon in the Line menu is highlighted, showing that the Line command is started automatically.

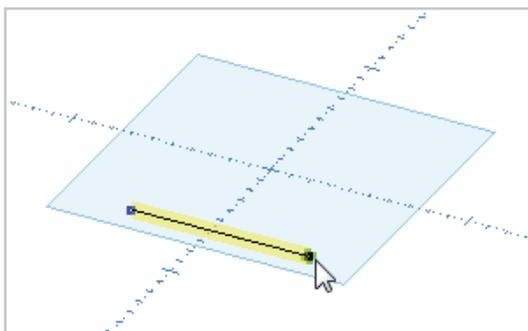
When the Line command is active, coordinate entry boxes will be available in the toolbox. Coordinate entry can be used for the first or second point of any line segment.

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

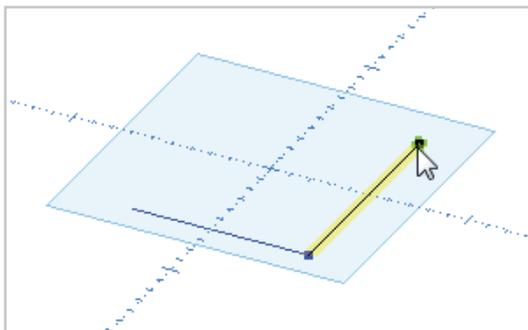
The current mouse pointer position is displayed in the toolbox, so you can also use it to check the position for a grid snap.



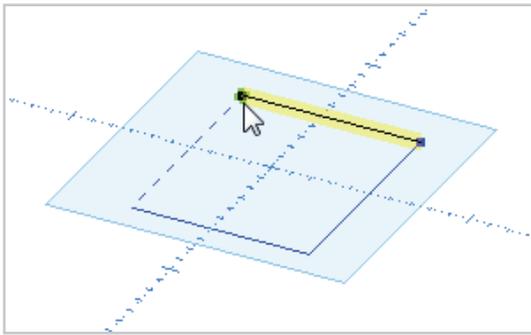
The mouse pointer will snap to grid positions in the drawing plane. Click to select a start point for the new line.



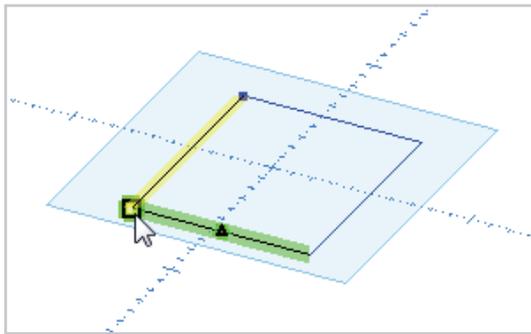
The mouse pointer will have a preview of the line to be drawn attached to it. Click on a new grid point to draw a horizontal line.



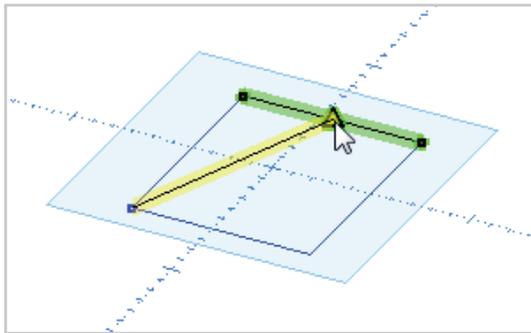
The Line command draws consecutive lines. Click again to draw a vertical line from the last point.



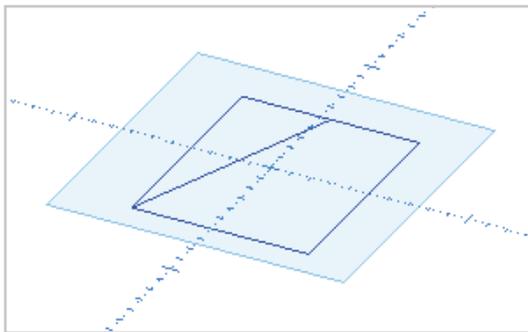
Move the mouse pointer horizontally again and you will see an indicator line when the new line endpoint is aligned with the start position. Click to end the line on the vertical alignment.



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.

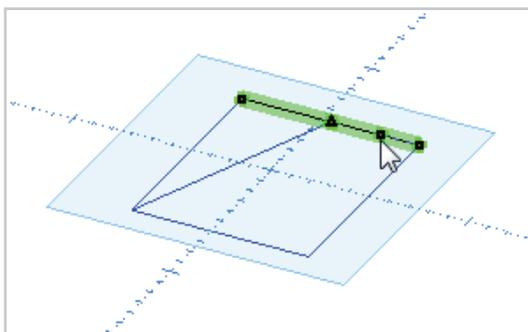


Hold the mouse pointer near the previous line and you will see the Midpoint indicator. Click to end the new line on the previous line midpoint.



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.

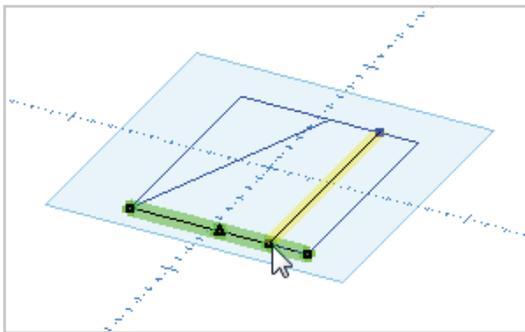
coordinate	
World :	On plane :
X-Coord <input type="text" value="10"/>	<input type="text" value="10"/>
Y-Coord <input type="text" value="20"/>	<input type="text" value="20"/>
Z-Coord <input type="text" value="0"/>	<input type="text" value="0"/>
<input checked="" type="checkbox"/> ok	+ Inc
parameters	
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.

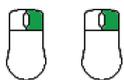
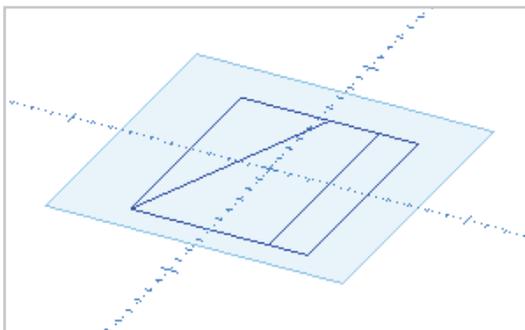
Change the value in the Distance along entry box to move the start point to a new position along the base line.

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.

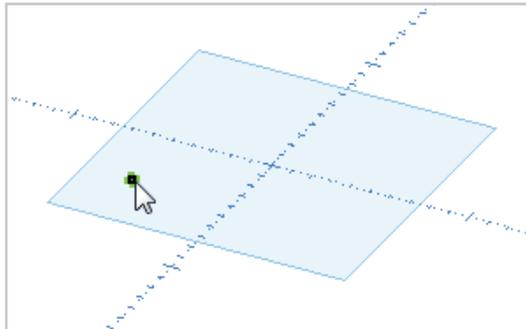


Click on the Rectangle icon in the Line menu to start the Rectangle command.

coordinate	
World :	On plane :
X-Coord <input type="text" value="0"/>	<input type="text" value="0"/>
Y-Coord <input type="text" value="0"/>	<input type="text" value="0"/>
Z-Coord <input type="text" value="0"/>	<input type="text" value="0"/>
<input checked="" type="checkbox"/> ok	
parameters	
Corner Radius	<input type="text" value="5"/>

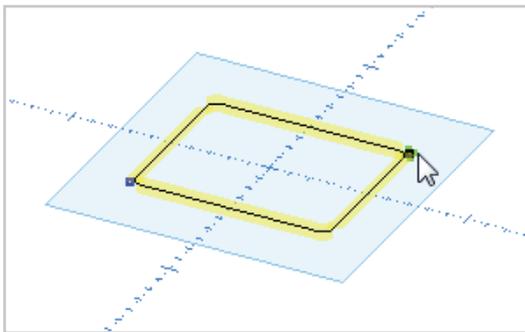
When the command is started, you have the option of entering a corner radius value.

Enter a Corner Radius of 5.

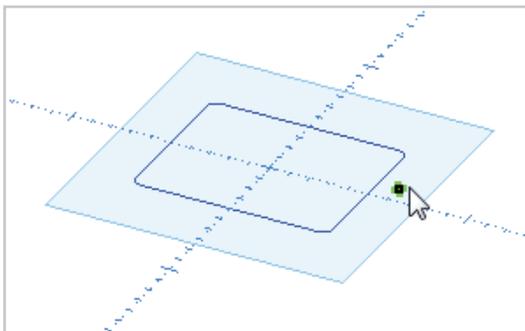


To define the first corner of the rectangle you can click on an entity or grid snap, or enter coordinates in the Toolbox.

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

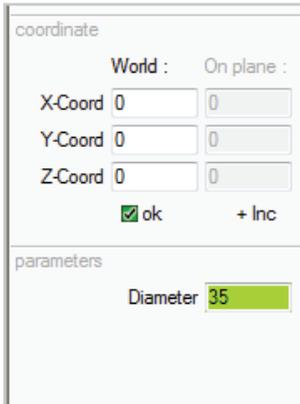
The Circle command creates a Circle with a given diameter.



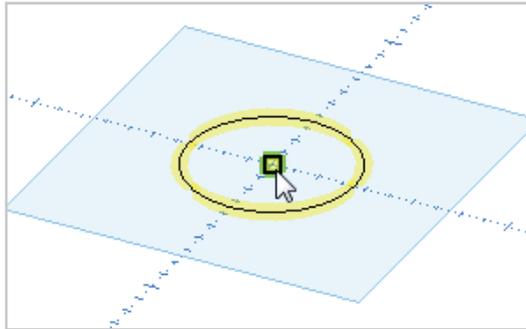
Click on the Arc icon to open the Arc menu in the toolbox.



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

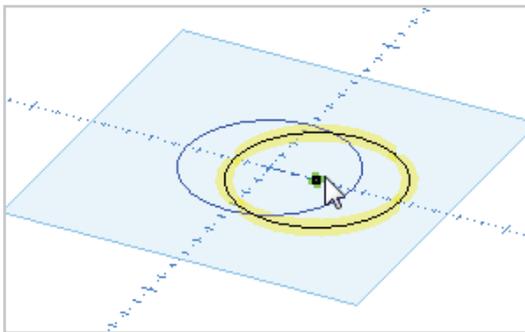
A screenshot of the software's input panels. The top panel, labeled 'coordinate', has two columns: 'World' and 'On plane'. Each column contains input fields for 'X-Coord', 'Y-Coord', and 'Z-Coord', all of which are currently set to '0'. Below these fields are two buttons: 'ok' (with a green checkmark) and '+ Inc'. The bottom panel, labeled 'parameters', has a single input field for 'Diameter' which is currently set to '35'.

Enter a Diameter for the circle in the Toolbox.

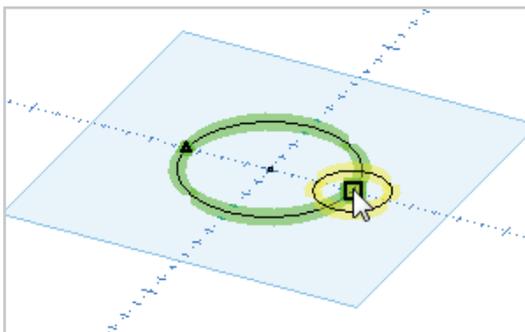


A preview of the circle is attached to the mouse pointer.

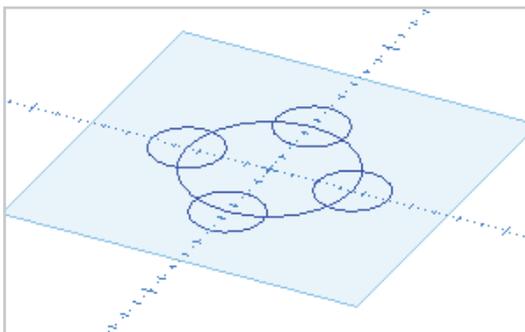
Click on or enter a position for the center of the circle.



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.



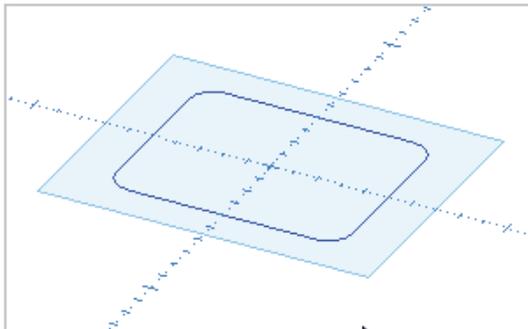
Right click when you want to end the Circle command.



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

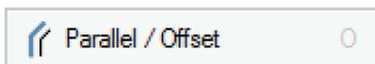
Offset entities and boundaries

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

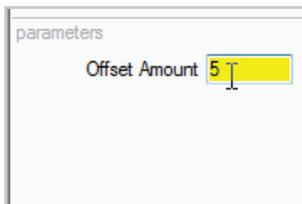


To practice the Parallel/Offset command, draw a rectangle with filleted corners.

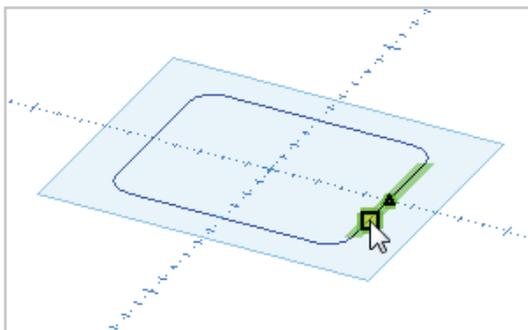
Offsetting single entities



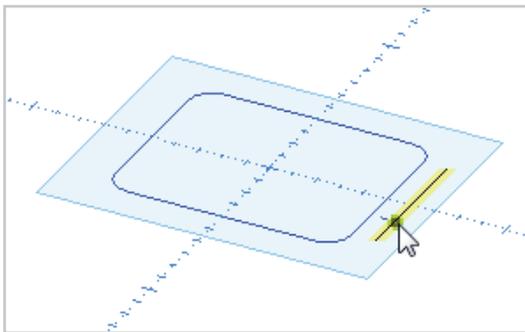
Click on the Parallel/Offset icon in the Line menu to start the command.



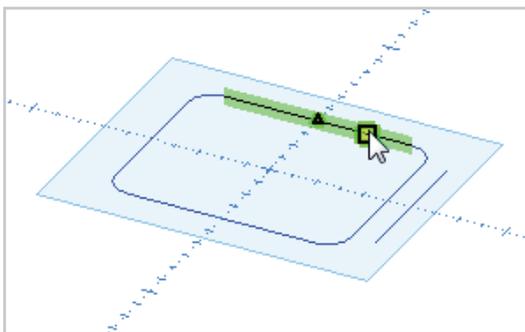
When the command is started, enter the distance you want to offset in the Offset Amount entry field in the Toolbox.



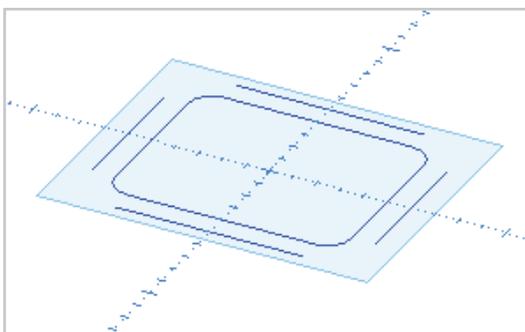
Select a side of the rectangle.



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.

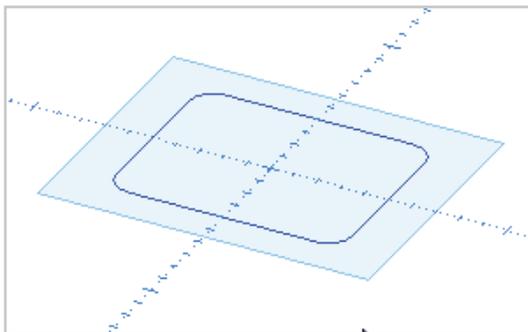


You can now select another single entity, and create an offset by clicking when the preview is where you want the offset.

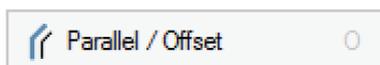


Right click to end the command when you have finished making single offsets.

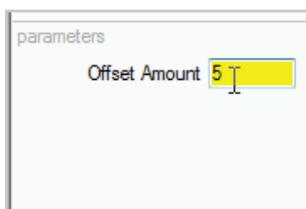
Offsetting a boundary



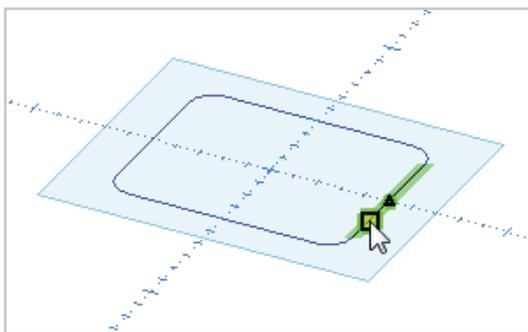
Delete the single offsets you made, or click on Undo till you return to the rectangle with filleted corners.



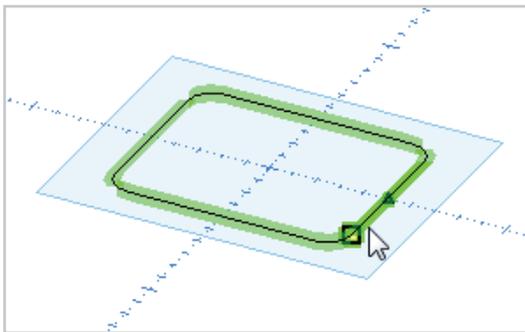
Click on the Parallel/Offset icon in the Line menu to start the command.



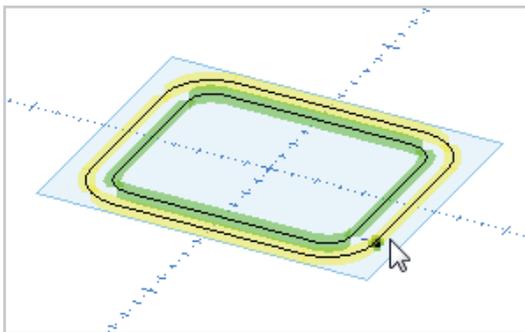
Enter the distance you want to offset in the Offset Amount entry field in the Toolbox.



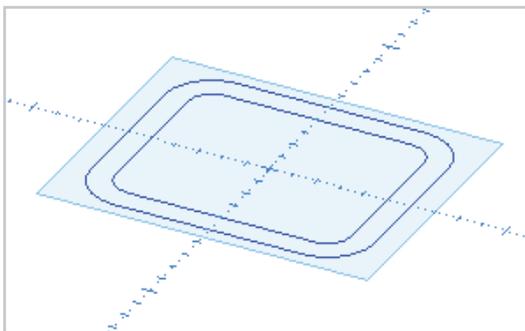
Select the same side of the rectangle.
The single offset preview will appear, but do not click to place it.



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.



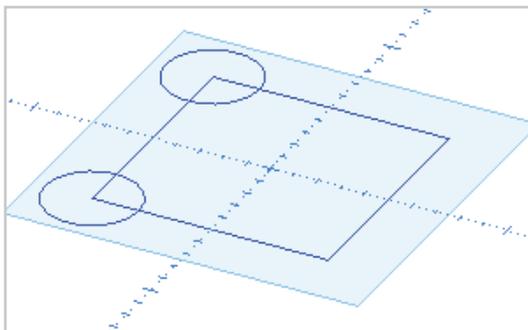
Right click when you have finished creating offsets.



Select All and Delete before continuing.

Moving Entities

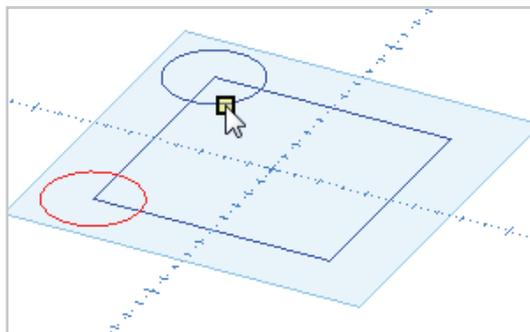
Entities can be moved, copied, scaled, rotated and mirrored. These Transform functions can be accessed by clicking on the Transform icon in the Toolbox to open the Transform menu.



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.



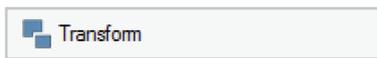
Before starting a Transform command, entities must be selected. Click on the Select icon to select entities one at a time.



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.



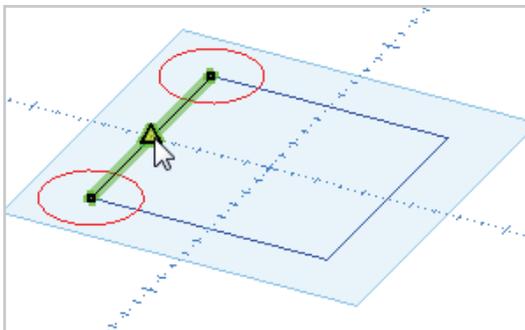
Right click to finish selecting.



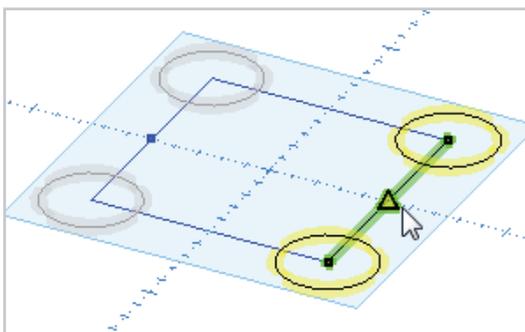
Click on the Transform icon to open the Transform menu in the toolbox.



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



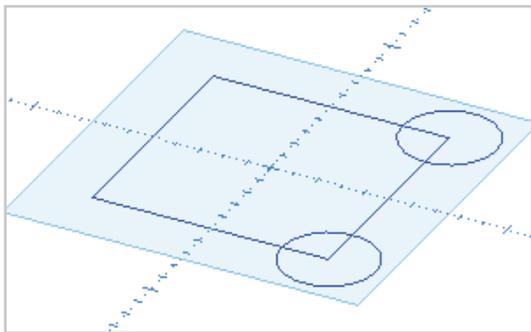
You can select or enter a position as the start point for a move. Click on the midpoint of the line between the circles.



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.

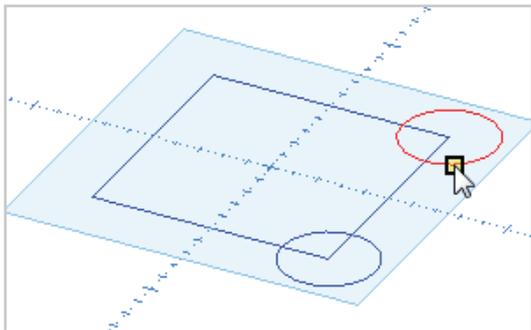


Right click to end the Move command. The moved entities are still selected, ready for another command.



Click Deselect All to leave the circles unselected.

Copy Entities



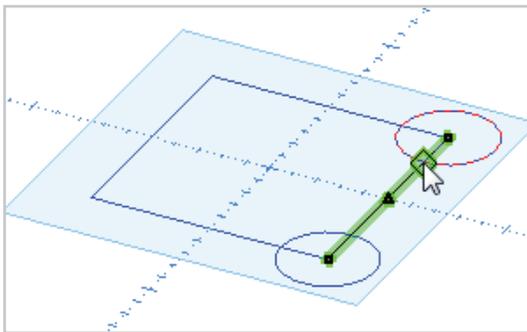
Click on the Select icon, and select the circle at the top of the square.



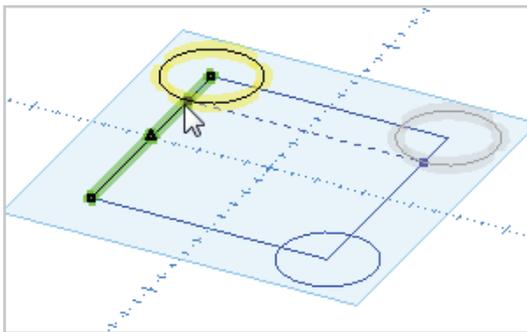
Right click to end the selection.



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



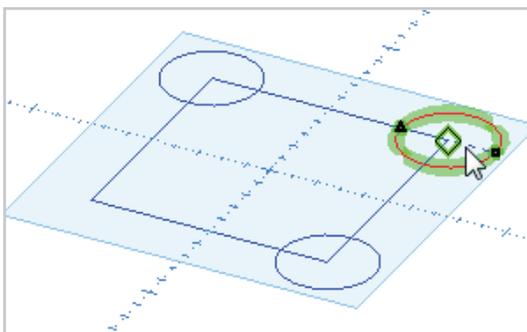
Select the intersection of the circle and the right side of the square as the start point of the move.



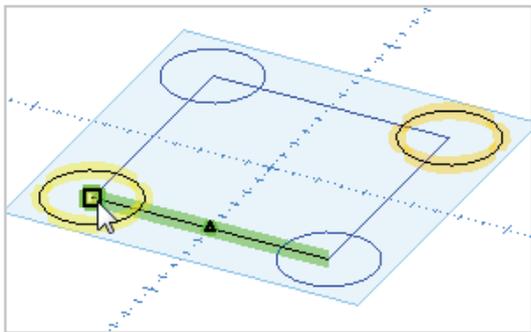
Move the mouse pointer to the left side of the square. Click on the intersection of the left side of the square and the dashed horizontal indicator to set the finish point of the move.



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.



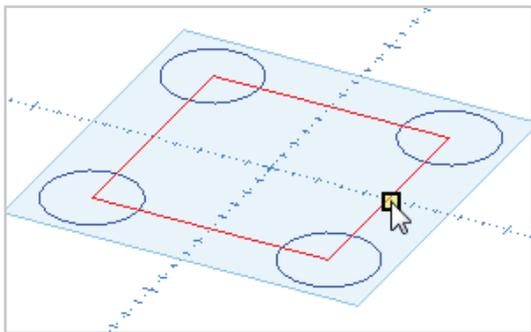
Right click again to end the Copy move command.



Click Deselect All to leave the original circle unselected.

Rotate

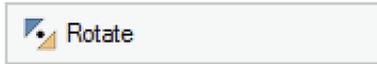
Entities can be rotated about a selected center of rotation. The axis of rotation can be parallel to the X, Y, or Z axis of the World or Plane coordinate system.



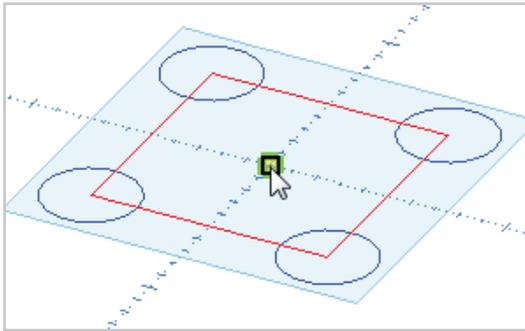
Select all the sides of the square.



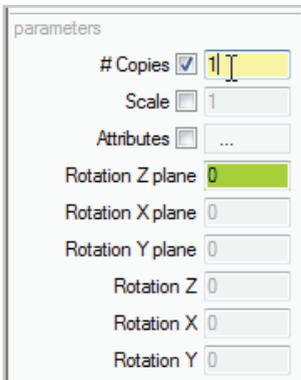
Right click to end the selection.



Click on the Rotate icon in the Transform menu.



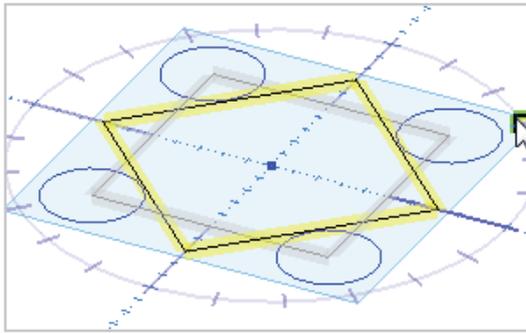
To select the center of rotation, move the mouse pointer to the intersection of the X axis and Y axis. Click when the square World origin indicator appears.



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.

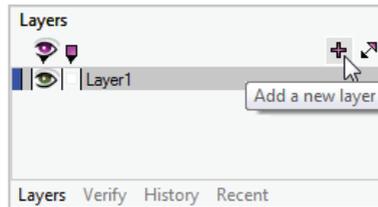
Using Layers

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.

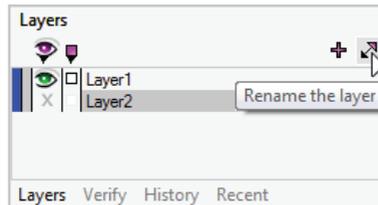
Click on the Layer tab at the lower right of the screen to open the Layer Manager.



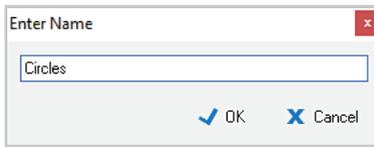
Up till now you have been working in a new file, on the default layer, Layer 1. We will now create a new layer and move some of the geometry to the new layer.



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



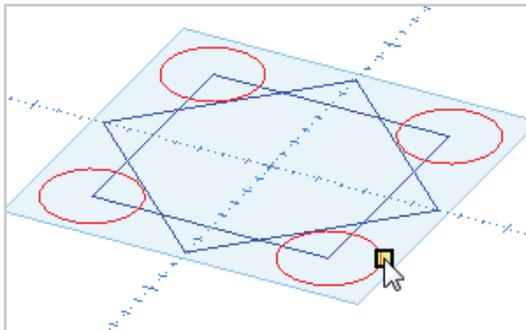
The new layer is created with the default name Layer 2. Click on the icon next to the + icon to rename the layer.



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.

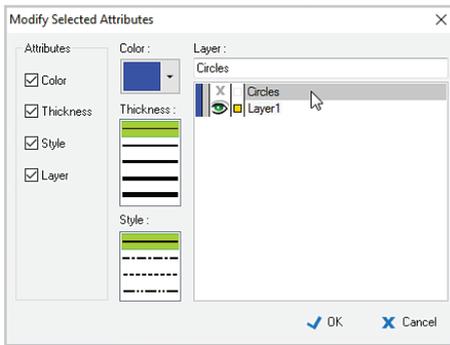


Click on the Modify icon in the Status Bar.

Default Layers



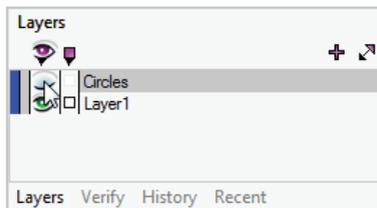
You can set default layers to be contained in each new drawing from the File menu > Properties dialog. Select the Layers tab, click New, give the layer a name, and set a default color, line type, and width.



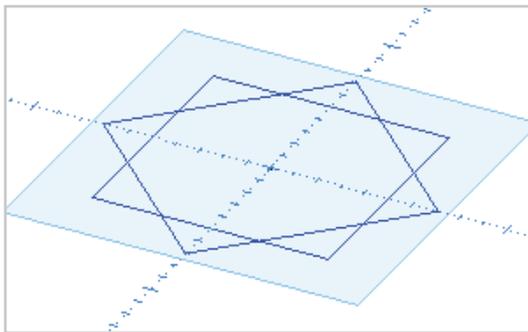
In the Modify Selected Attributes dialog, you can change the color, line type, thickness, or layer of entities. Select the Circles layer in the Layer list to move the selected entities to that layer.



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.



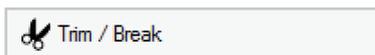
Now you will be able to see the squares clearly.

Trimming Entities

Trim functions are used to shorten or extend existing entities.

Scissors

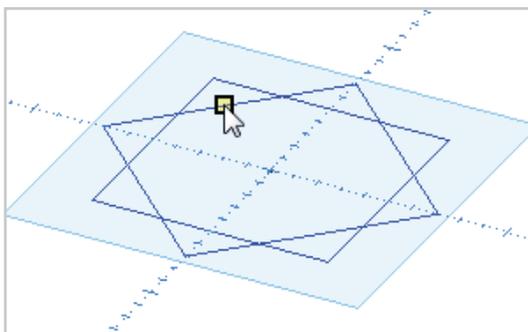
The quickest way to trim away unwanted parts of geometry is to use the Scissors tool.



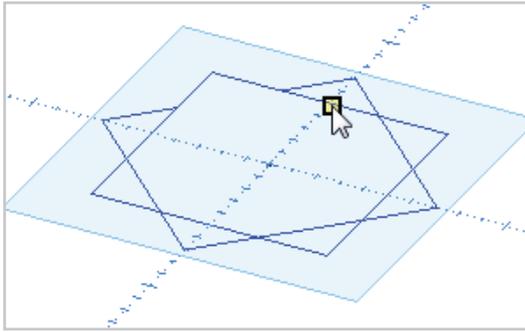
Click on the Trim/Break icon in the Toolbox to open the Trim/Break menu.



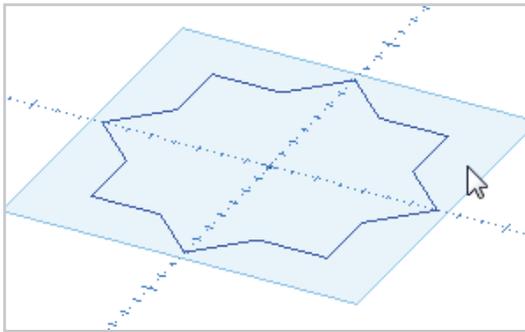
Click on the Scissors icon to start the Scissors command.



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.



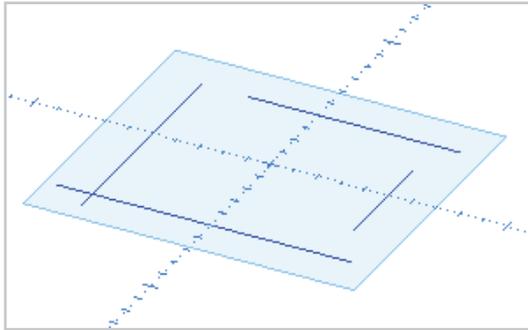
Right click to end the command when you have finished trimming entities.



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

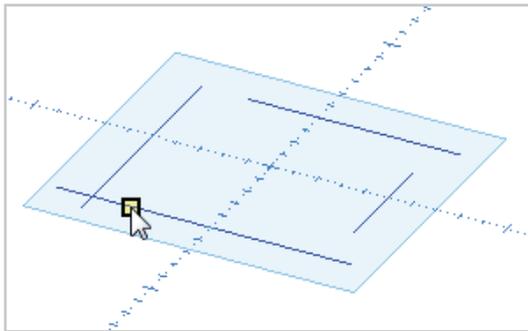
This function will shorten or extend two arcs or lines to their common intersection. It is an important function as it can be used to make sure a chain of entities are properly connected at their endpoints.



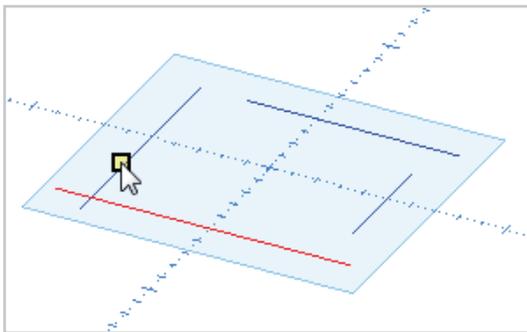
Open a new file to practice using the Trim Two command. Draw some lines that cross, and some unconnected lines.



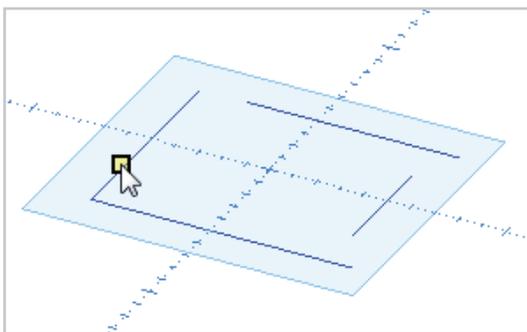
Start the Trim Two command by clicking the Trim Two icon in the Trim menu of the Toolbox.



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

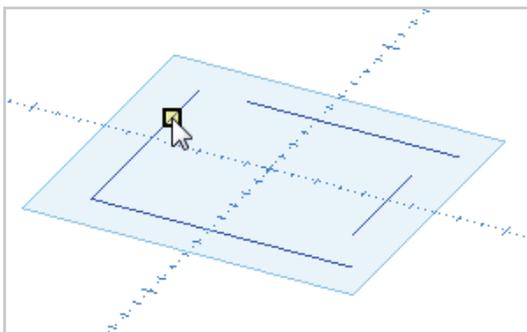


Click on the second line you want to trim, on the side of the intersection that you want to keep.

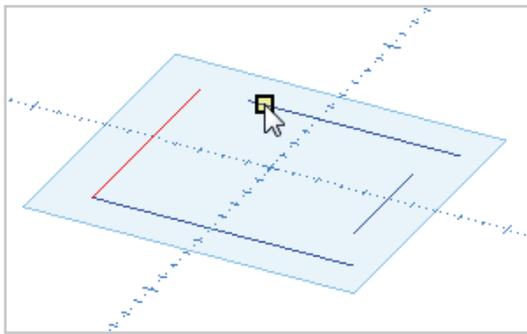


The lines will be trimmed to their intersection. In this example both lines were shortened.

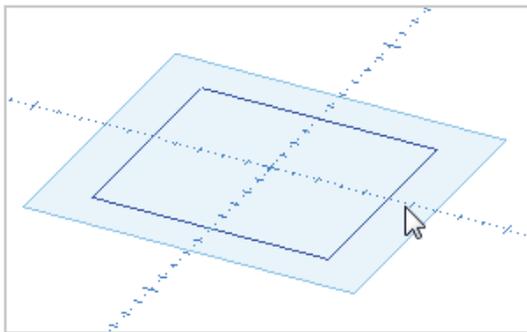
The command continues, and you can click on the next line you want to



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.

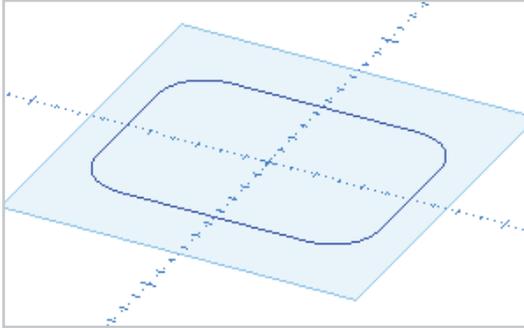


Some CAM operations require selection of consecutive entities as a profile chain.

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.

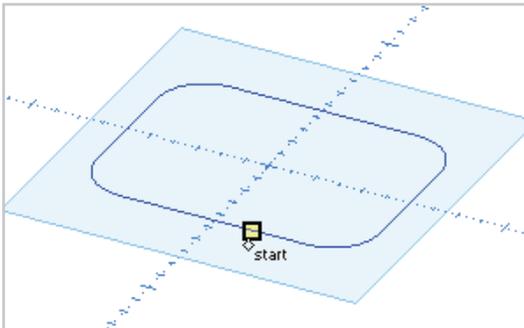


To practice chain selection, draw a filleted rectangle.

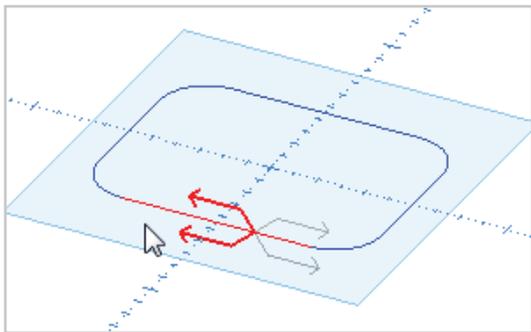
Selecting a complete profile as a chain



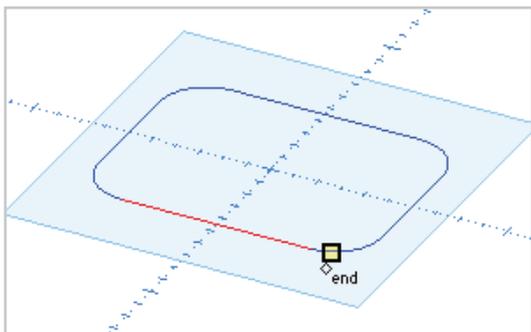
Click on the Chain Selection icon to enter chain selection mode.



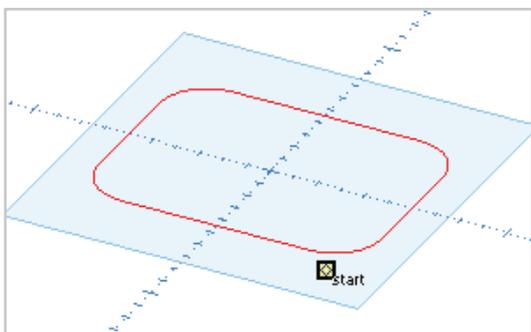
The mouse pointer changes to the word 'start'. Click on the entity that will be the first in the chain.



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.

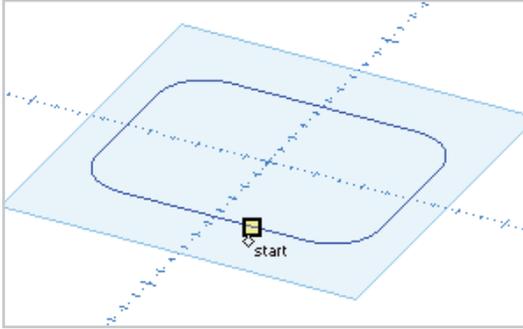
Selecting part of a profile as a chain



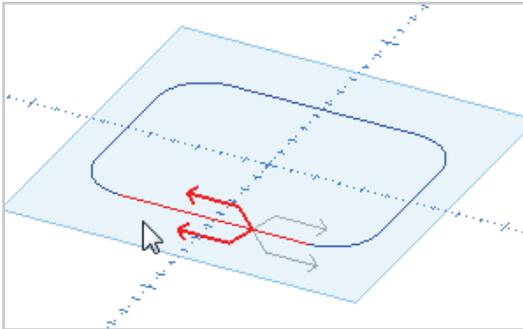
Click on the Deselect All icon to clear the previous selection.



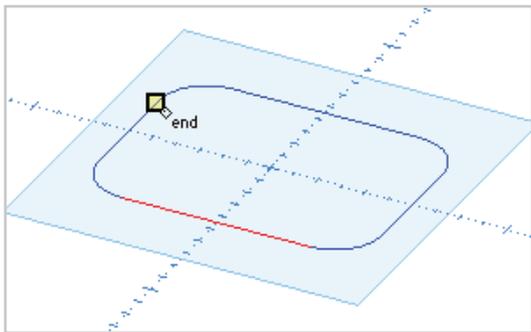
Click on the Chain Selection icon to enter chain selection mode.



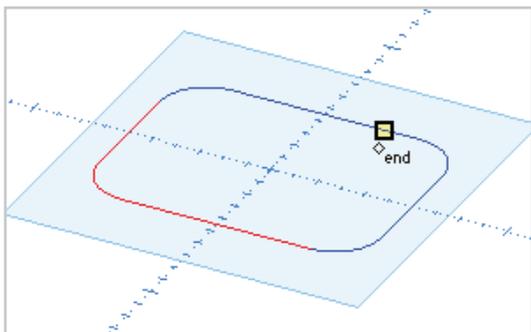
The mouse pointer changes to the word 'start'.
Click on the entity that will be the first in the chain.



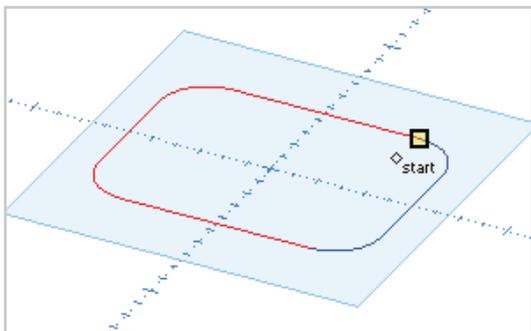
Move the mouse pointer around till the red arrows are pointing in the direction that you want to chain select. Click to set the direction.



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.

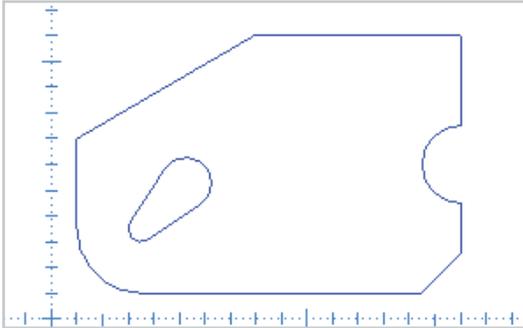


Right click when you are ready to finish selecting the chain. You can now start to select another chain, or right click again to end chain selection.

OneCNC CAD Tutorial 2

2D Drawing

This Tutorial is designed for all OneCNC products.



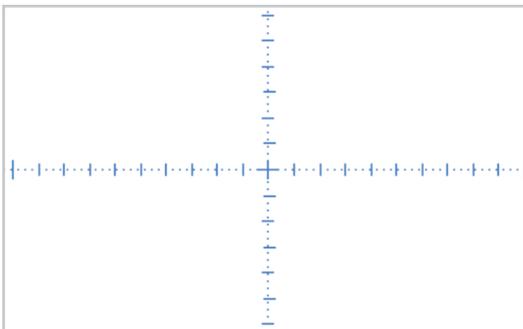
In this tutorial we will be using some of the time saving features of OneCNC CAD drawing to show how easily a part can be constructed. We will be drawing a part with fillets and chamfers, and a slot at an angle of 75 degrees.

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

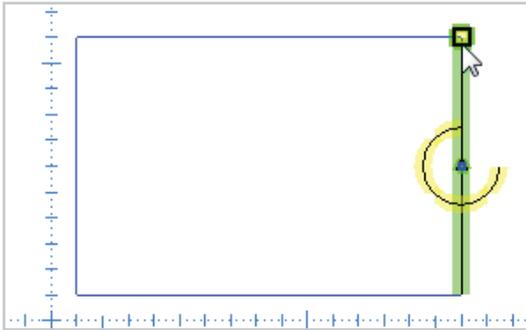


locate the center of the arc.

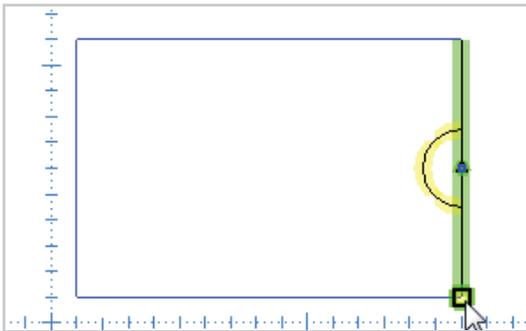
The Arc Sketch function now requires the arc Radius, Start Angle, and End Angle. Each of these can be defined by entering a value in the toolbox, or clicking on an entity or grid snap.

coordinate	
World :	On plane :
X-Coord 175	175
Y-Coord 60	60
Z-Coord 0	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°.



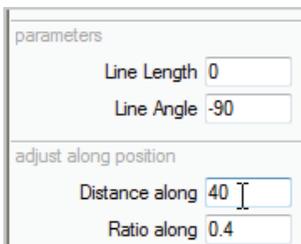
Right click to end the Arc Sketch command.



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.



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

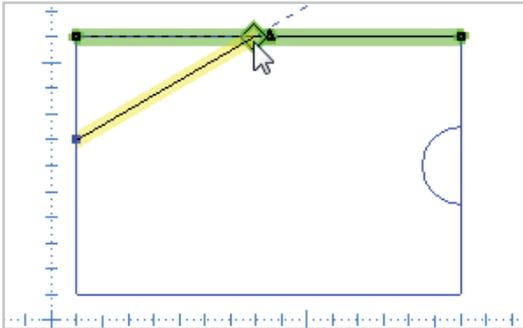
adjust along position

Distance along

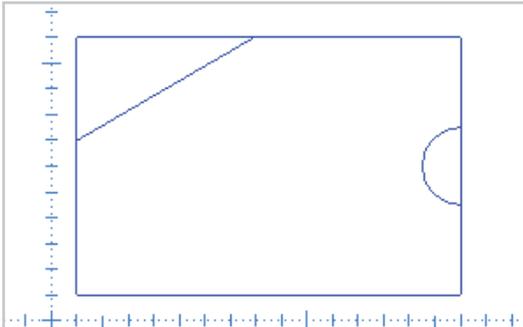
Ratio along

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.

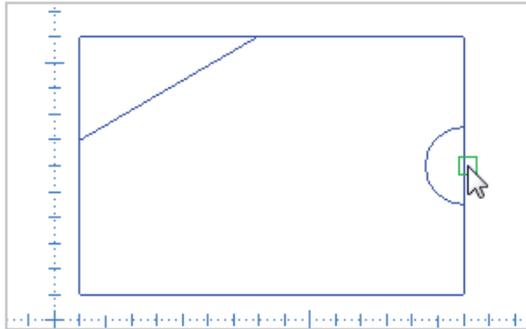


Right click to end the Line command.

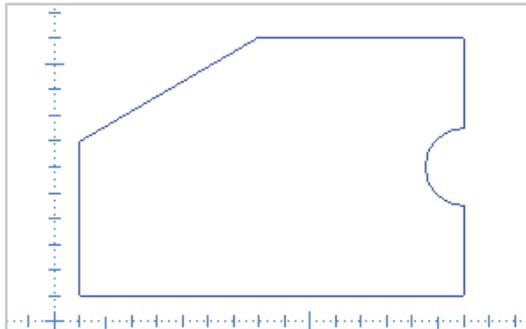
We can now trim away the unwanted line segments.



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.

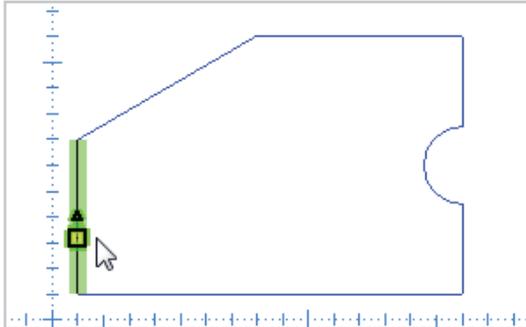
Next we will create the fillet on the lower left corner of the part.



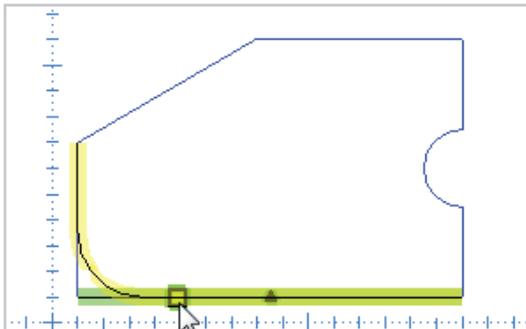
Start the Fillet command on the Arc menu.



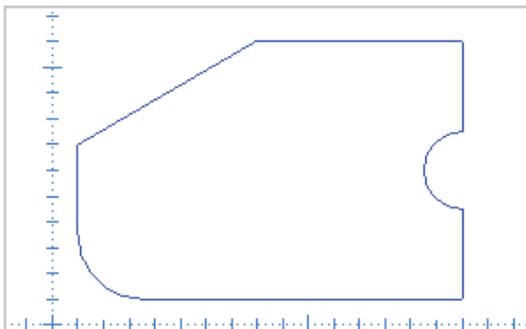
Enter a Fillet Radius of 25 in the toolbox.



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.

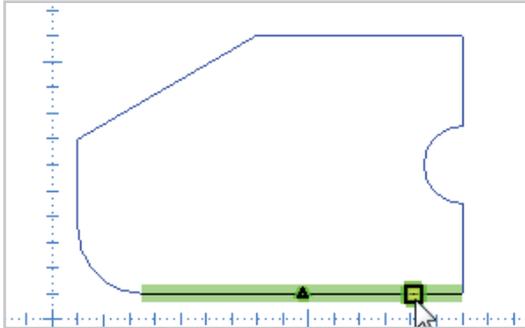
Next we will create the chamfer on the lower right corner of the part.



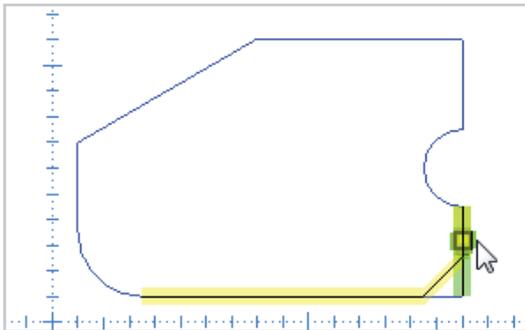
Start the Chamfer command on the Line menu.



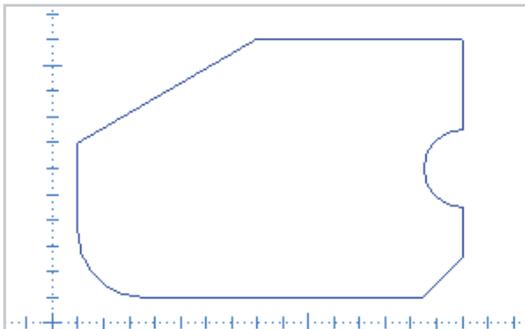
Enter a Chamfer Distance of 15 in the toolbox.



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.

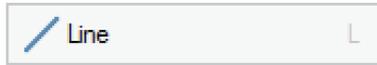


The chamfer is created, and the corner is trimmed automatically.

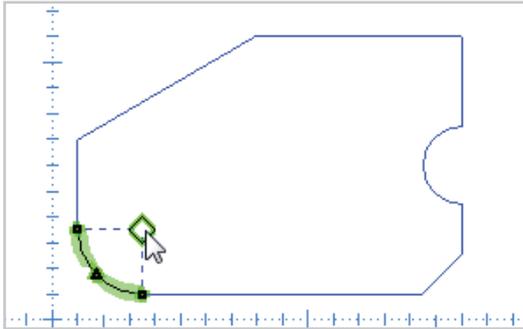
Right click to end the Chamfer command.

Step 2: Draw the angled slot

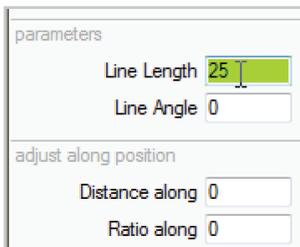
The first step in construction of the slot will be to draw the centerline.



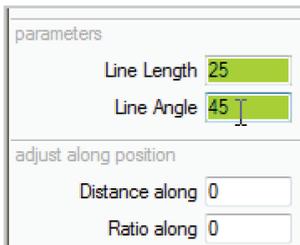
Select the Line menu to start the Line command.



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



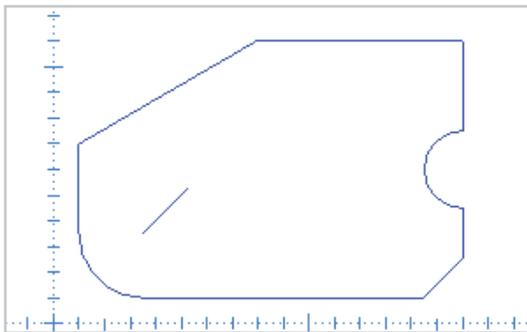
Type 25 in the Line Length box, and press Enter.



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

The line will be drawn.

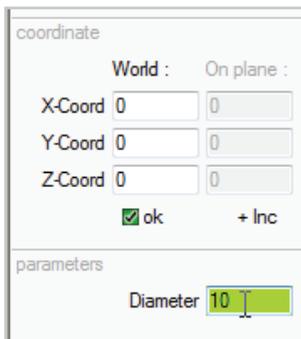
Right click to end the current Line sequence, and right click again to end the Line command.



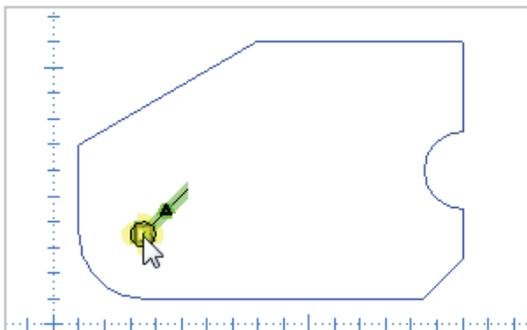
The part drawing should now look like this.



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



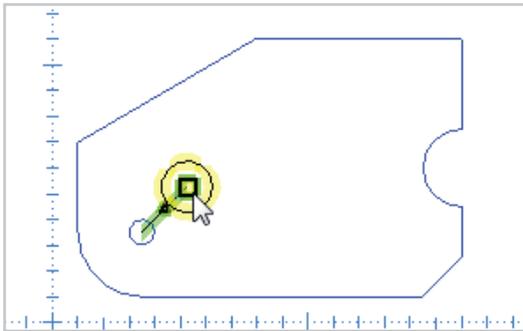
Type 10 in the Diameter entry box.



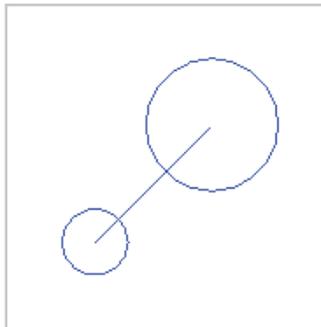
Click on the lower endpoint of the centerline to draw the first end arc of the slot.

coordinate	
World :	On plane :
X-Coord 35	35
Y-Coord 35	35
Z-Coord 0	0
<input checked="" type="checkbox"/> ok	+ Inc
parameters	
Diameter	20

Type 20 in the Diameter entry box.

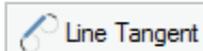


Click on the upper endpoint of the centerline to draw the second end arc of the slot.

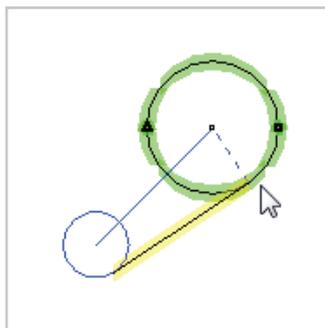
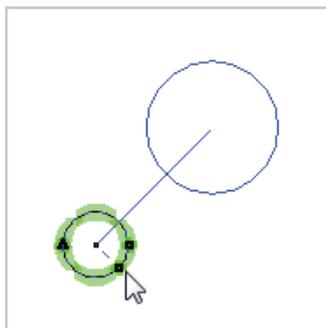


Right click to end the Arc command. You should now have two circles of unequal diameter, one at each end of the centerline.

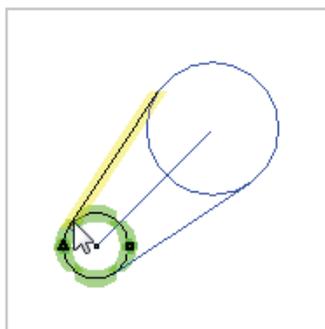
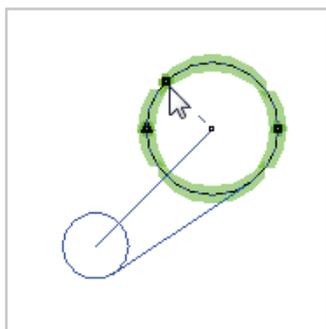
Next we will join the circles with tangent lines.



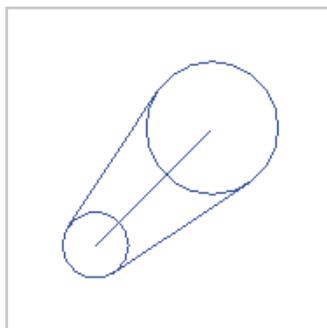
Start the Line Tangent command in the Line menu of the Toolbox.



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



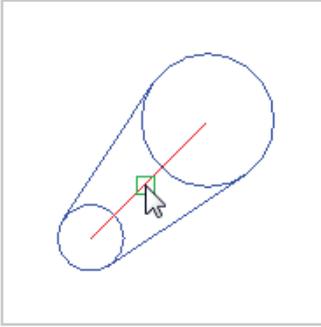
Draw the second tangent line by clicking near where the tangents will be.



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



Begin the Single Selection command.



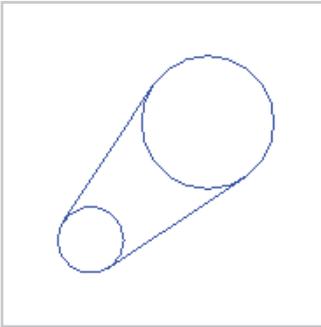
Select the 45 degree centerline.



Right click to end the selection process.



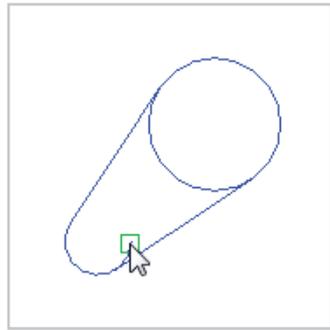
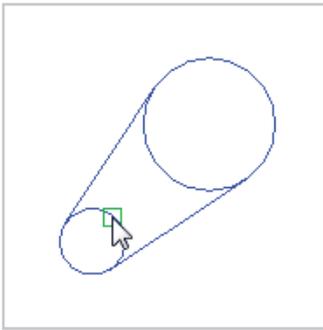
Click the Delete icon or press the Delete key to delete the centerline.



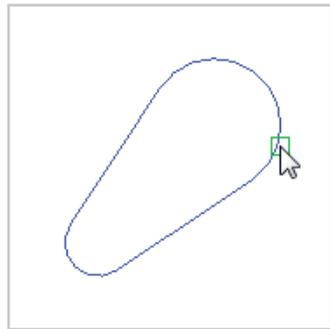
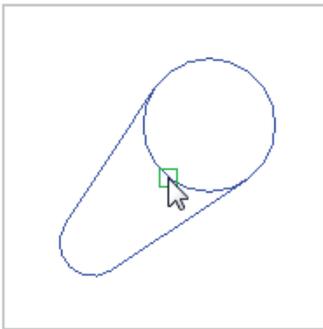
All we have to do now is trim the circles to the tangent lines to finish the slot.



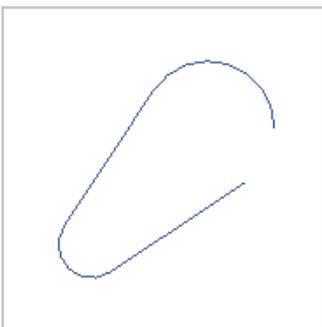
Start the Scissors command on the Trim menu.



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.

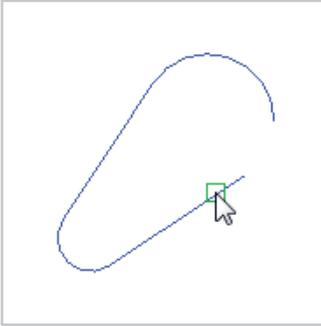


Because the smaller arc is not crossed by any other entity, the Scissors command removes it completely.

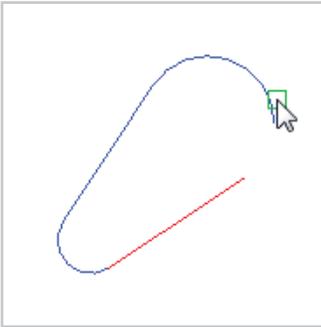


Right click to end the Scissors command.

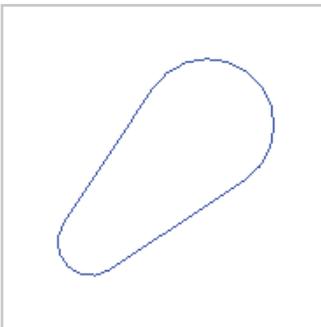
Start the Trim Two command.



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.



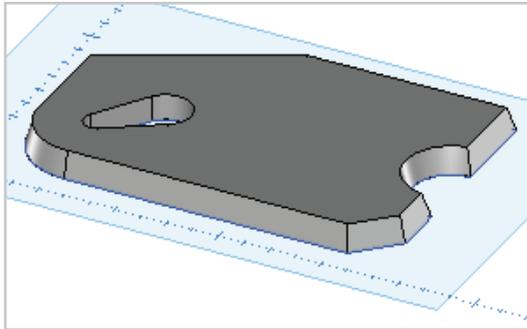
The two entities will now be joined perfectly.

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

OneCNC CAD Tutorial 3

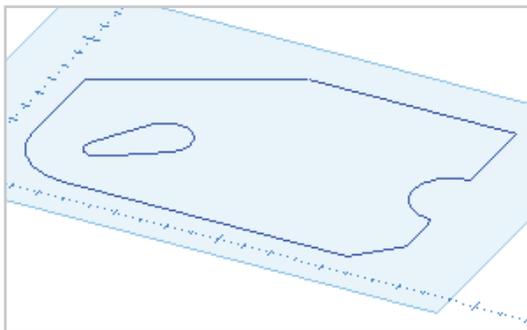
Solid Modeling by Extrusion

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.

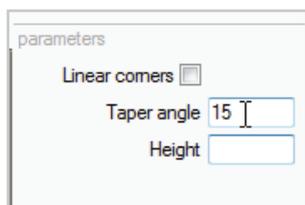
Step 2: Create a solid by extrusion from curves.



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

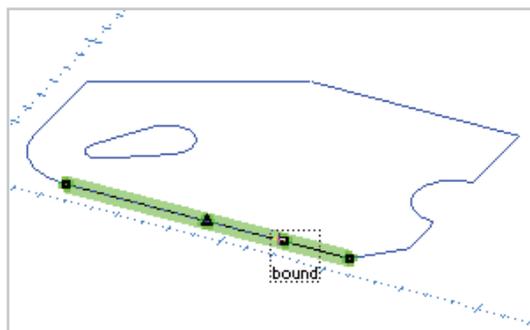


Click on the Extrude Curves icon.

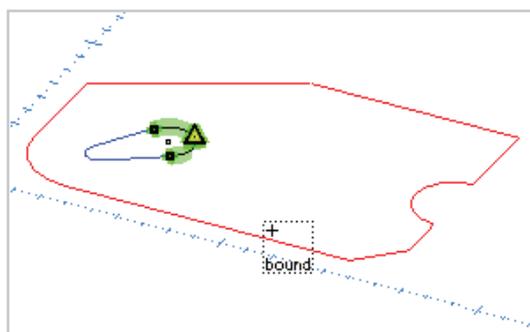


Enter a Taper Angle of 15 in the parameters section of the Toolbox. This will tilt the walls of the extrusion inwards.

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.

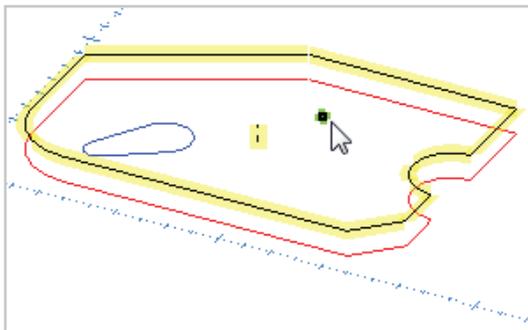


You can extrude a shape with one or more openings, so the cursor will continue to seek boundaries.

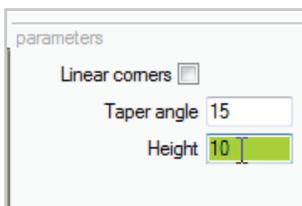
Do not select the inner boundary.



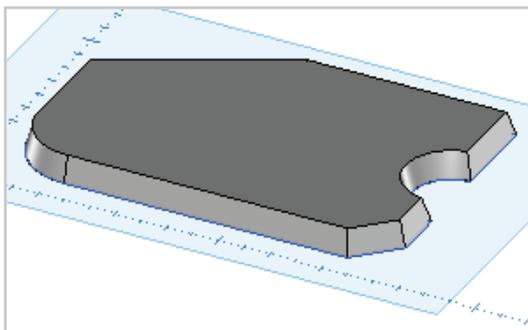
Right click to end the selection stage of the command because in this case we want to create the slot using different settings.



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.

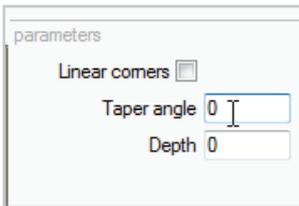


Click on the Display icon to open the Display menu.

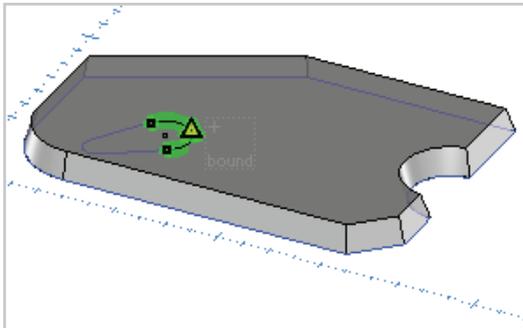


Select Transparent. The slot geometry will now be visible through the part.

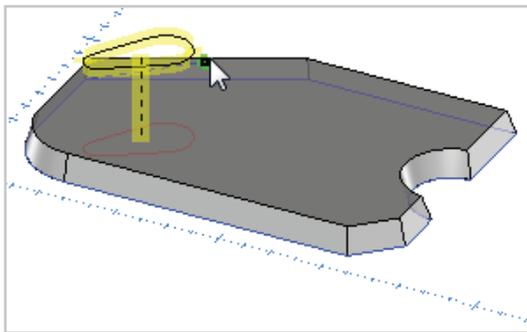
Click on the Extrude cut icon in the Extrude menu.



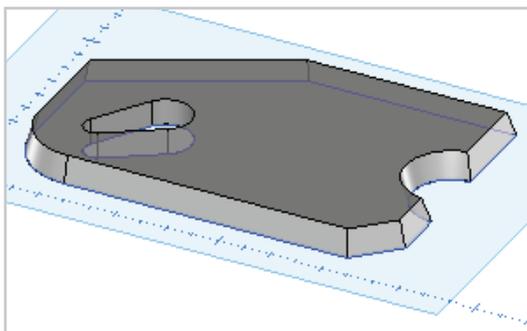
Change the Taper Angle to 0 so the slot will have vertical walls.



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.



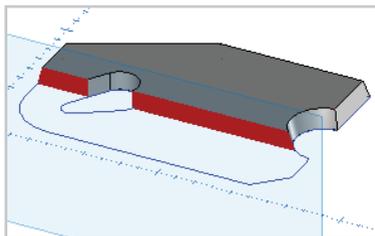
Because we want a through cut the exact extrusion distance is not important. Move the mouse pointer up until the extrusion preview is above the part, and click to make the extrude cut.



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.

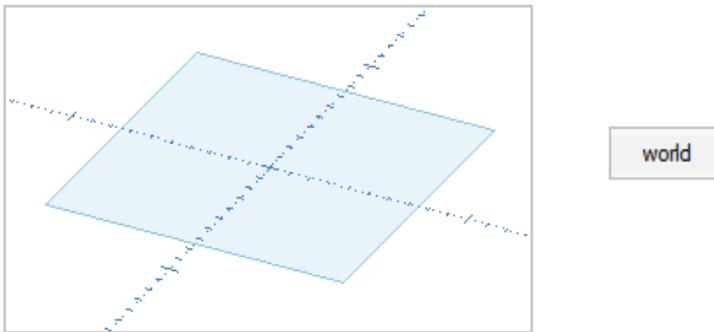
You can find the topic in Help Contents -> CAD Drawing and Modeling -> Modeling Tools -> Section Tool.

OneCNC CAD Tutorial 4

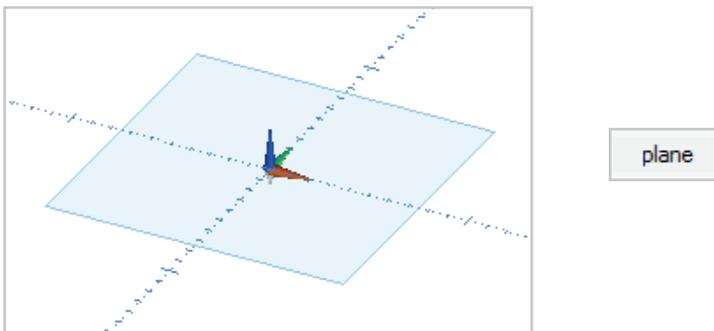
Working With Planes

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.

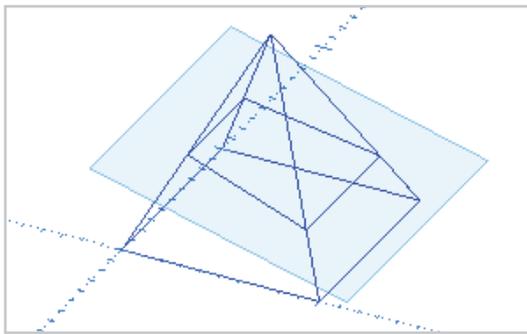
When you open a new file in OneCNC, the current construction plane is the XY plane. The X and Y axes of this plane coincide with the X and Y axes of the world coordinate system.



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.



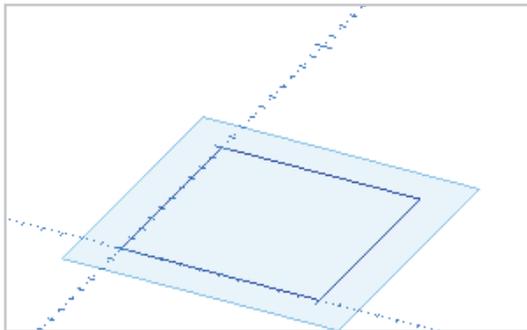
Clicking on the Origin selector will activate Plane coordinate input. A plane origin indicator will appear.



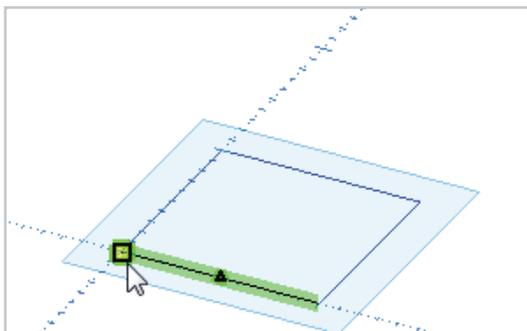
As an introduction to Plane functions we will position a Plane as an aid to constructing a truncated wireframe pyramid.

Step 1: Draw a pyramid wireframe

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



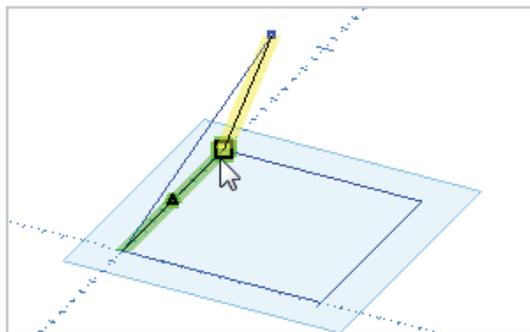
Select World coordinate input, and use the Rectangle command to draw a square from X0 Y0 to X100 Y100.



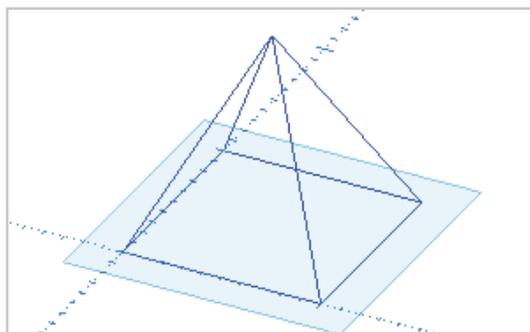
Start the Line command, and click on the corner of the square to start the first edge of the pyramid.

coordinate		
World :	On plane :	
X-Coord	50	50
Y-Coord	50	50
Z-Coord	100	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.

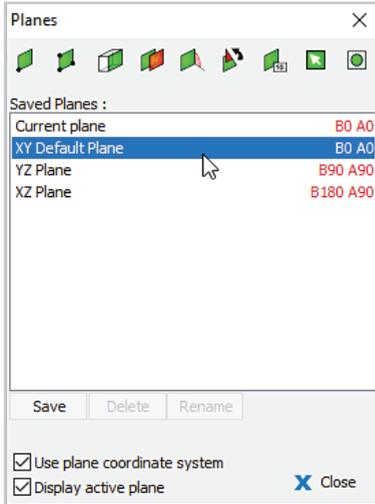


Right click to end the current Line sequence, and construct the other two edges by clicking on corner, vertex, and corner. Right click twice to end the Line command.

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.

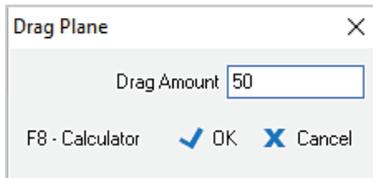
You will also see the pre-defined YZ and XZ planes.

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



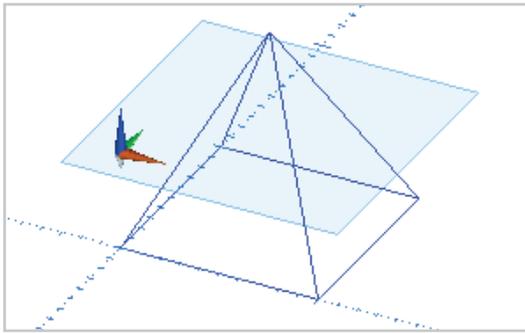
Click on the Push or Pull Plane icon in the Planes dialog.

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.



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

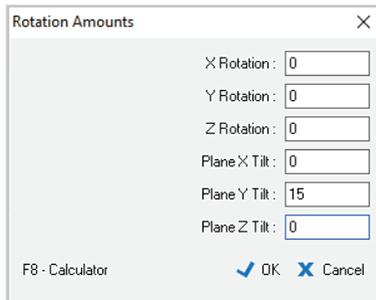
We will now tilt the plane on its own Y axis to create the angle for truncation of the pyramid



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

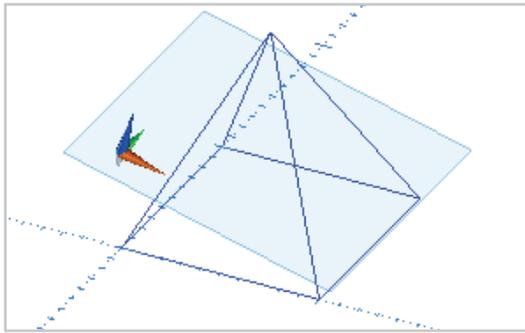


Click on the Rotate Plane icon.



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.



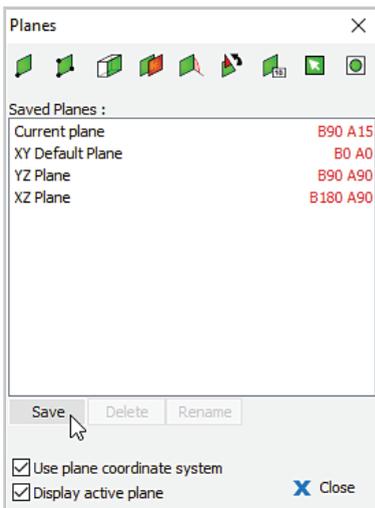
The plane is tilted on its own Y axis. The plane origin will still be directly above the World origin.

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

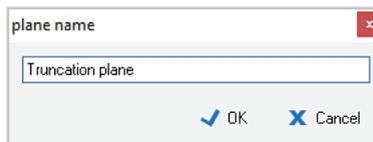
Step 3: Save the plane position



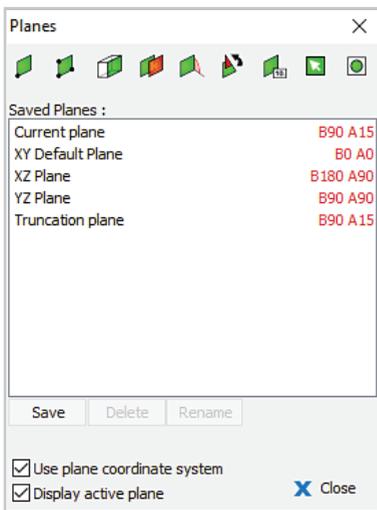
Click on the Plane icon again to open the Planes dialog.



Click on the Save icon.



Name the plane Truncation plane and click OK.

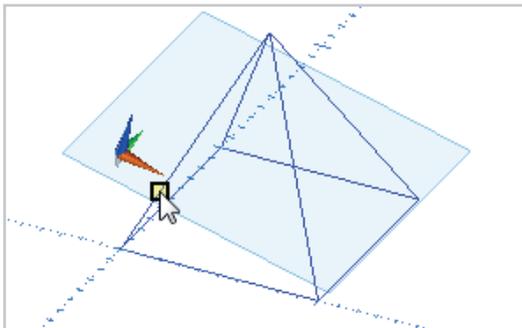


The plane now appears in the Saved Planes list.

Step 4: Create the truncated pyramid

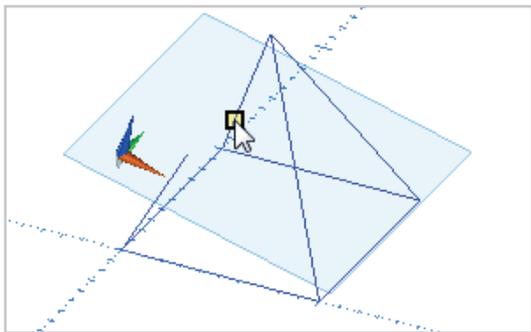


To truncate the pyramid edges, open the Trim menu in the Toolbox, and click on the Trim to plane icon. This command trims lines or arcs which pass through the plane to the plane intersection.

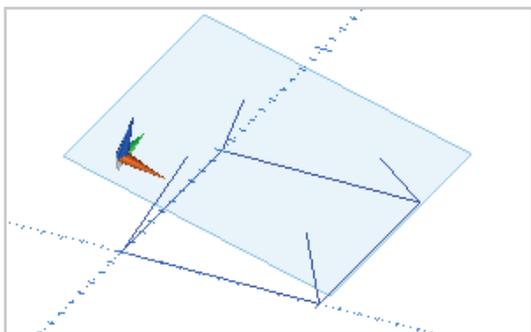


Click on an edge of the pyramid, between the base and the plane.

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

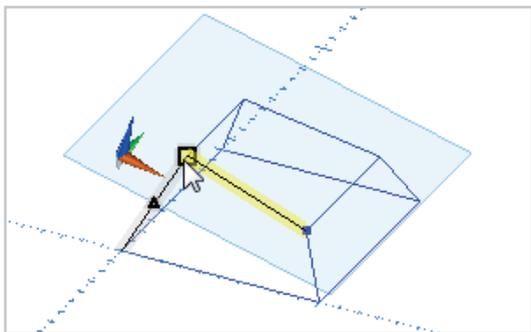


The edge will be cut off where it meets the plane. Click on the next edge to trim it.



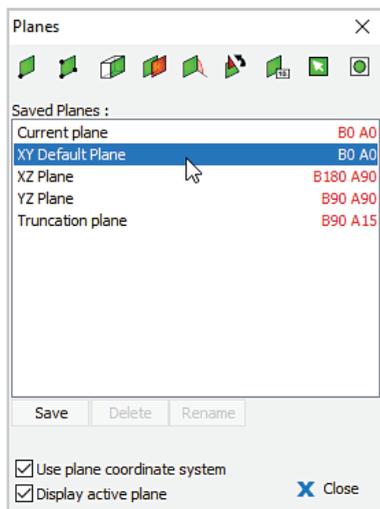
Continue until all four lines are trimmed to the plane.

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



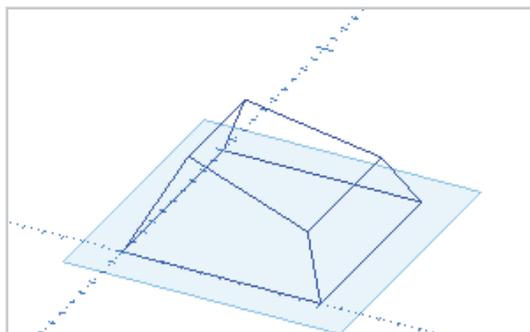
Line command to join them. These lines form the boundary of the truncation.

Step 5: Return to default XY plane

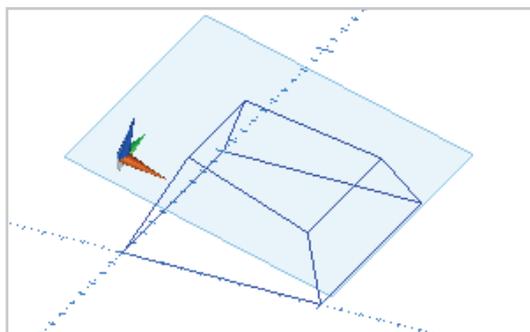


To return to the XY plane, open the Planes dialog and select the XY plane in the Saved Planes List.

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.

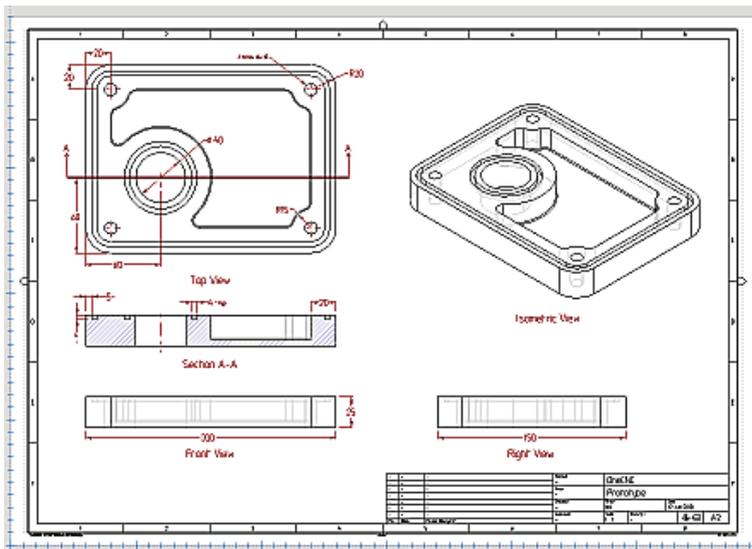
To learn more about plane commands in OneCNC, go to 'Construction Planes' in the CAD section of OneCNC Help.

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.

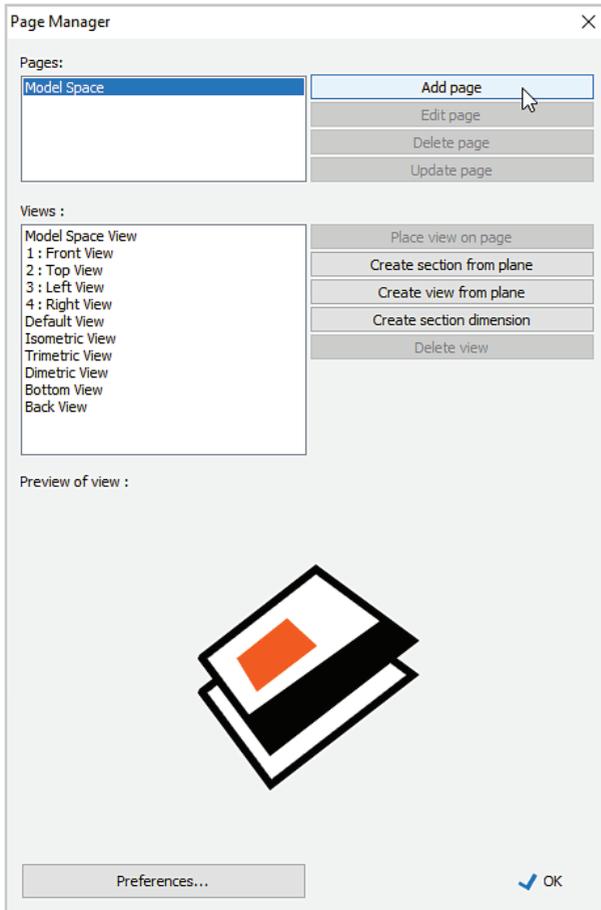
There are two stages to creating a technical drawing in OneCNC. The first step is to create a drawing page layout with title block and border. You can then create and manage views of your model in the layout.

Add a drawing page with border and title block

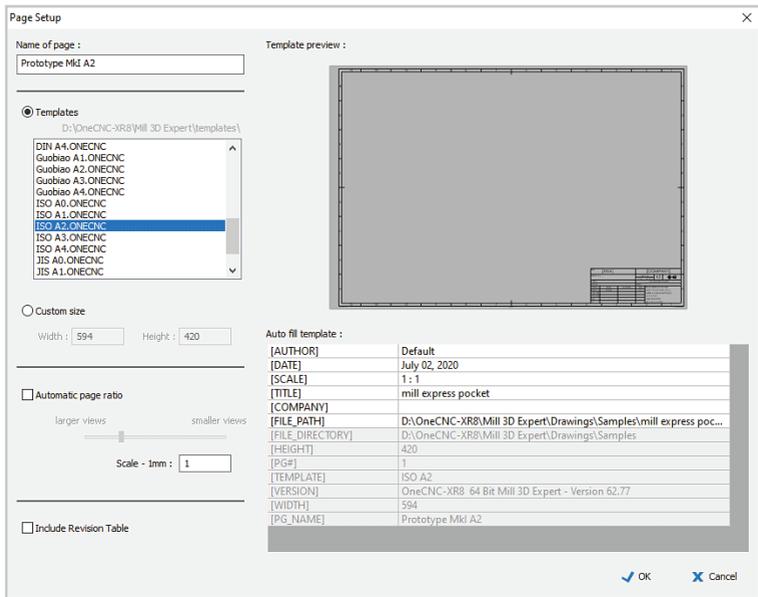


To add a new page, click on the Pages icon in the Status Bar.

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.



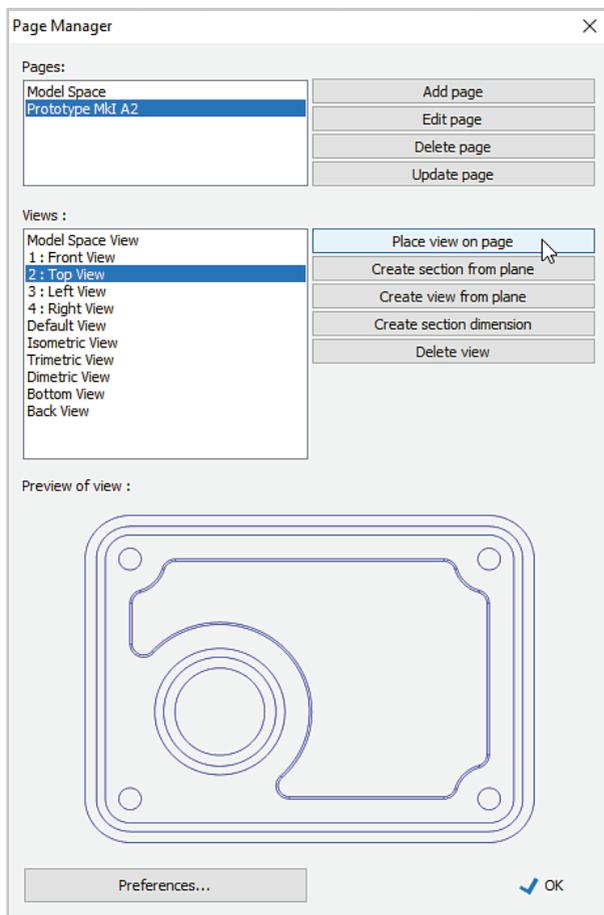
The new page now appears in the Pages list in the Page Manager.

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

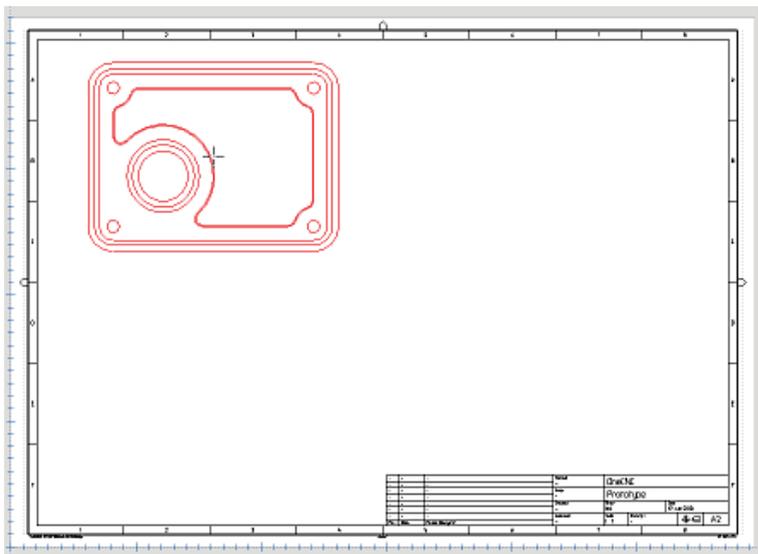
You can add a view of the model to an existing drawing page layout at any stage of the design process.



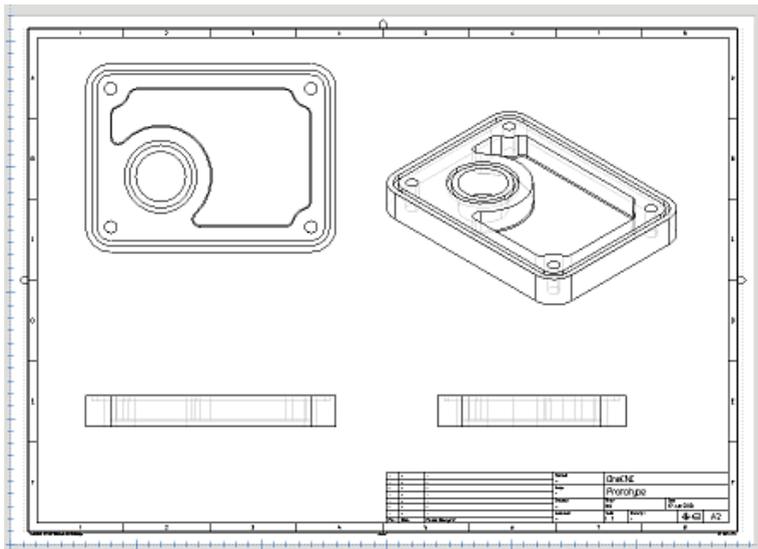
In the Pages list of the Page Manager, select the page layout you want to add the view to.

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.

OneCNC WireEDM Tutorial 1

2 axis closed profile

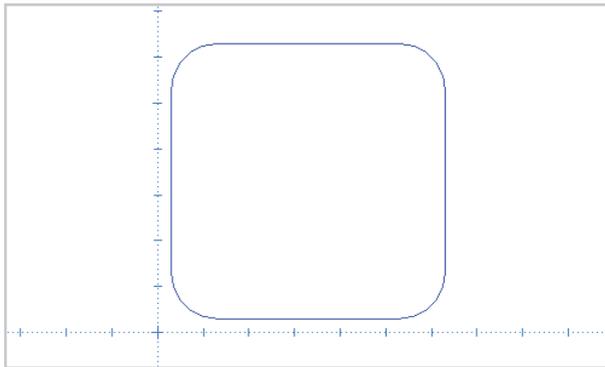
These tutorials describe how to use OneCNC Wire EDM to efficiently generate wire-cutting operations for EDM machines.

The workflow for wire path generation from geometry requires a defined start point and a profile which can be closed or open. For open profiles you will also need an end point.

Complete CAD Tutorial 1 before attempting this tutorial, in which we will generate a wire path for a closed profile, and output the NC code to cut the profile.

Create a new part file and draw a filleted square.

Start a new file and save it as 'Tutorial Wire 1.ONECNC'.
Set the drawing view to the CAD view.



Using the Rectangle tool, draw a square with a corner radius of 10. Start the square at X3 Y3 and make it 60 wide and 60 high.

Create a Wire start point.

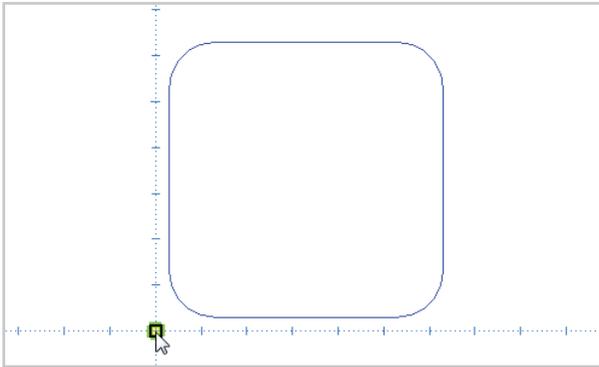
We will create a wire start point at the XY origin.



Click on the Point icon in the Toolbox Home menu.



The Point command will be started automatically.

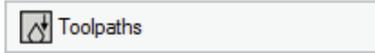


Click on the grid snap at the part origin to create a wire start point at X0 Y0.



Right click twice to end the Point command.

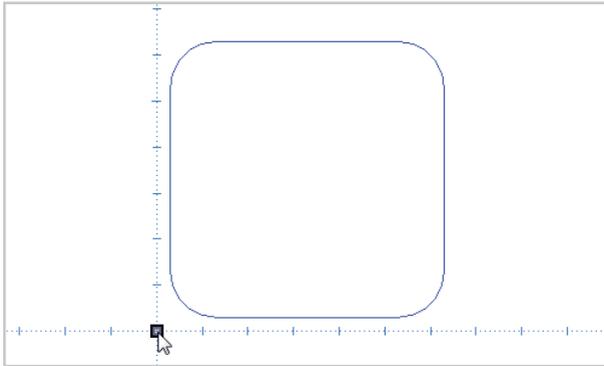
Creating a Closed Profile Wirepath.



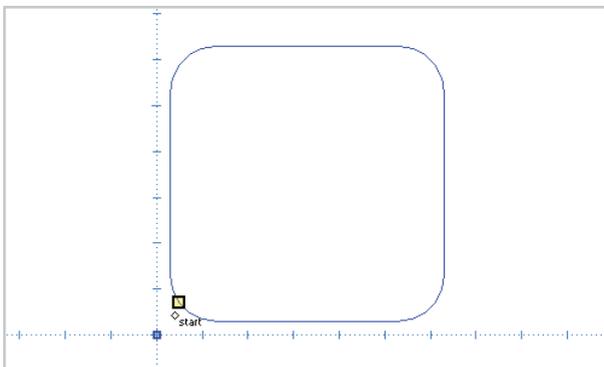
Click on the Toolpaths icon in the Toolbox.



Click on the Cut closed profile icon in the Toolpaths menu.

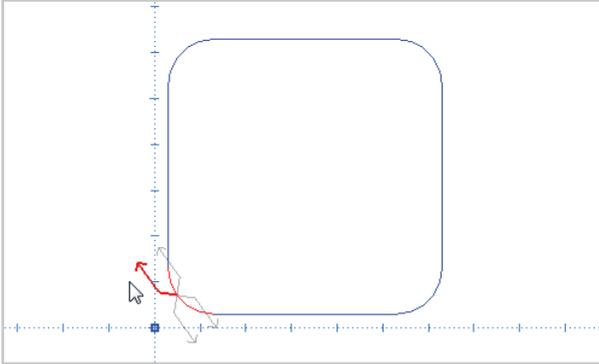


Click on the point at X0 Y0 to select it as the start of the operation.



Click the fillet radius closest to the wire start point. This will be where the wire begins to cut the profile.

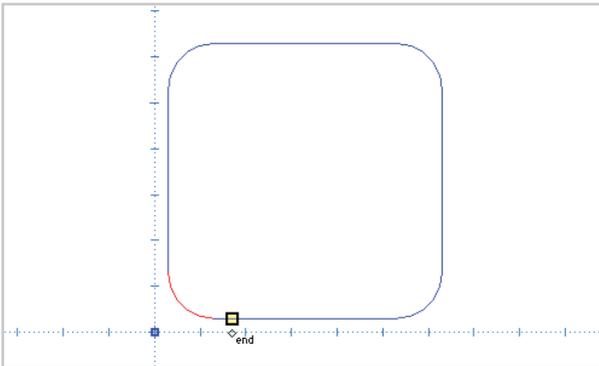
The radius will be selected and the cursor will change to direction arrows.



Hold the cursor outside the profile and above the profile start point so the arrow points in a clockwise direction around the profile.

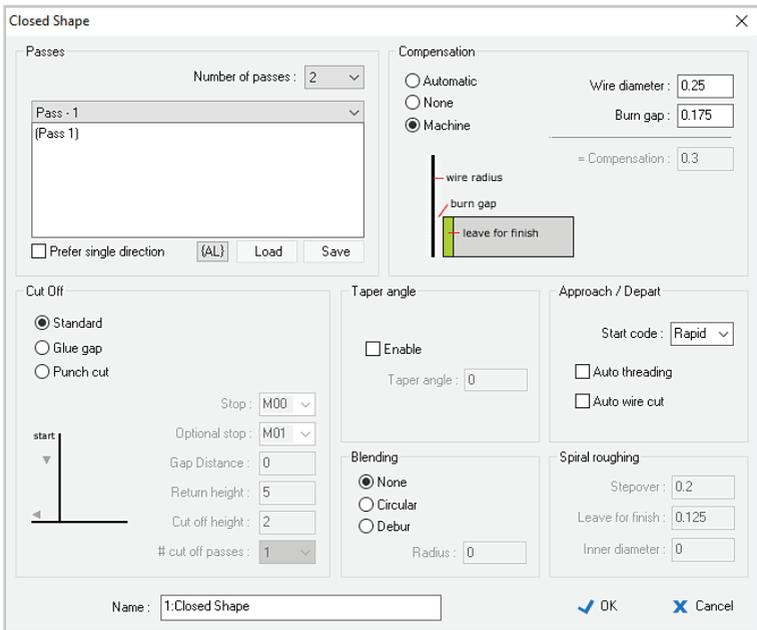
Click to set the cut direction.

The cursor now changes to the End cursor.



Click on the last segment of the profile to be cut.

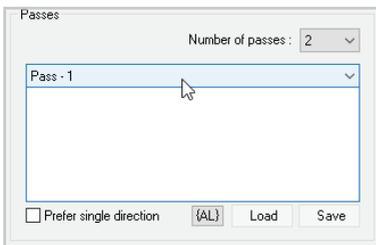
Instead of clicking on the last segment, you can press F3 on your keyboard to select the consecutive chain automatically.



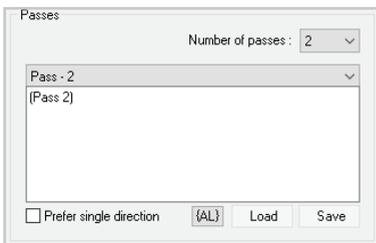
The closed shape dialog appears. The wire path settings are categorized into sections. Fill out the settings as follows:

Passes:

This section controls the settings for multiple passes. For this example set Number of Passes to 2. In the white text box you can enter code to set machining parameters for each pass.



The pass the code is for is selected by clicking on the bar at the top of the text box.



For this example enter a bracketed comment (Pass 1) for Pass 1 and (Pass 2) for Pass 2 so you will see where the code occurs in your output NC program. Leave the Prefer single direction box unchecked.

Cut Off:

Cut Off

Standard
 Glue gap
 Punch cut

Stop: M00
Optional stop: M01
Gap Distance: 0
Return height: 5
Cut off height: 2
cut off passes: 1



Select the Standard option for Cut Off.

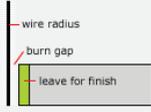
This is the simplest setting, which will complete the profile and return the wire to the start point.

Compensation:

Compensation

Automatic
 None
 Machine

Wire diameter: 0.25
Burn gap: 0.175
= Compensation: 0.3



Set this to Machine compensation.

In this mode the compensation G41 or G42 codes will be written in the NC program.

This allows changes to the wire diameter to be made at the control without regenerating the NC code in OneCNC.

Taper Angle:

Taper angle

Enable
Taper angle: 0

For machines with Taper angle programming you can enter the taper in degrees when you want taper in the NC program.

Leave this option unchecked for this example.

Approach/Depart:

Approach / Depart

Start code: Rapid
 Auto threading
 Auto wire cut

This section allows insertion of special codes at the beginning of a program. For this example leave start code at Rapid and the Auto threading and wire cut boxes unchecked.

Blending:



Blending

None
 Circular
 Debur

Radius : 0

Blending options are available for the Standard and Glue Gap Cut-off functions. Set blending to None for this exercise, so that the wire contacts the profile without any adjustment.

Spiral Roughing:



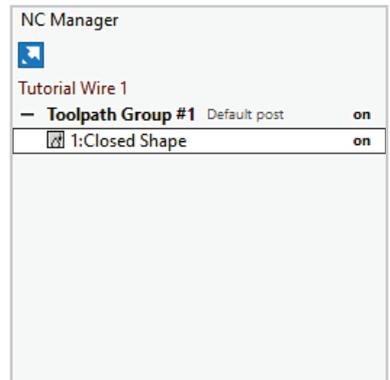
Spiral roughing

Stepover : 0.2
Leave for finish : 0.125
Inner diameter : 0

This section of the dialog is grayed out as it is only applicable when cutting a hole.

Once all the settings are entered click OK.

The path for the closed shape will be created, and appear in Toolpath Group #1 in the NC manager.

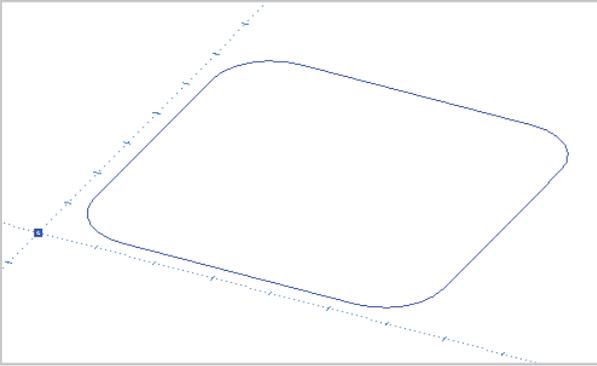


Previewing the wire path.

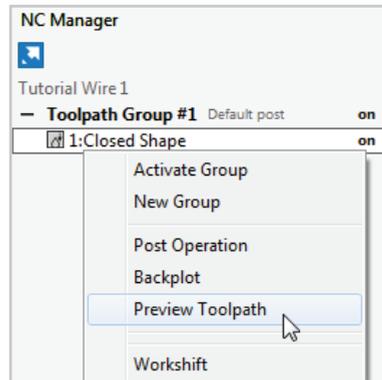
So far we have stayed in CAD view, the default view for a new file.

The Preview window opens with the same view as the drawing window, so we can preset the appearance of the preview by changing the drawing view.

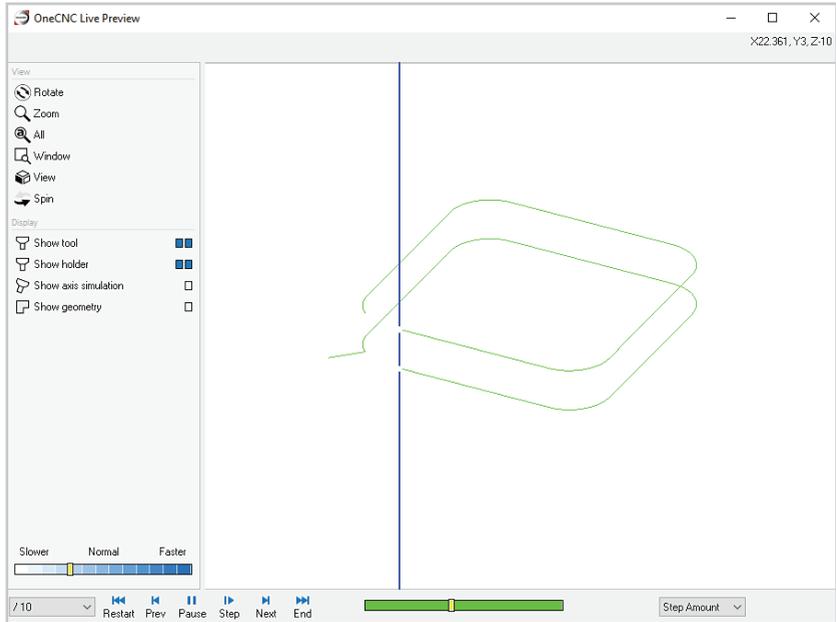
Press the Spacebar and select Trimetric from the View menu.



Right click the closed shape operation in the NC Manager, and Select Preview Toolpath from the drop down menu.



The Live Preview window appears, and simulates the wire toolpath.



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

The controls at the top of the toolbox allow you to change the view and zoom magnification.

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

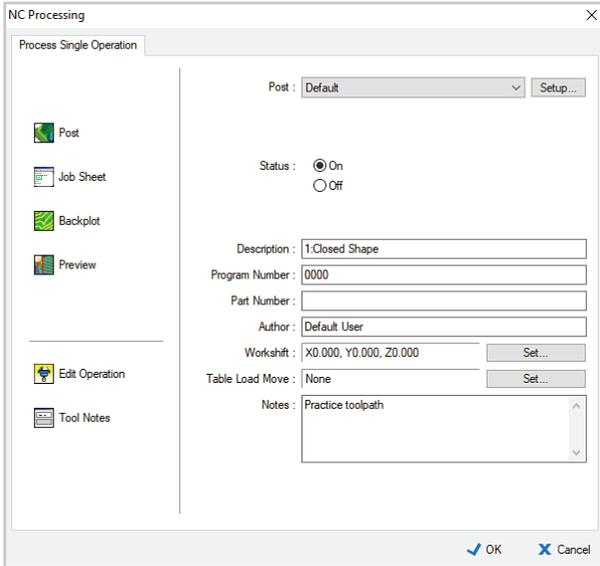
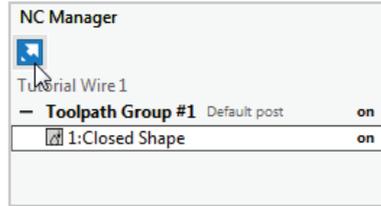
The simulation speed control is in the lower left corner of the preview window.

Close the Live Preview when you are ready to continue.

Posting NC Code.

You can output an NC program for a single wire toolpath operation, or for a toolpath group.

Select the Closed Shape toolpath and click on the Process icon at the top of the NC Manager.

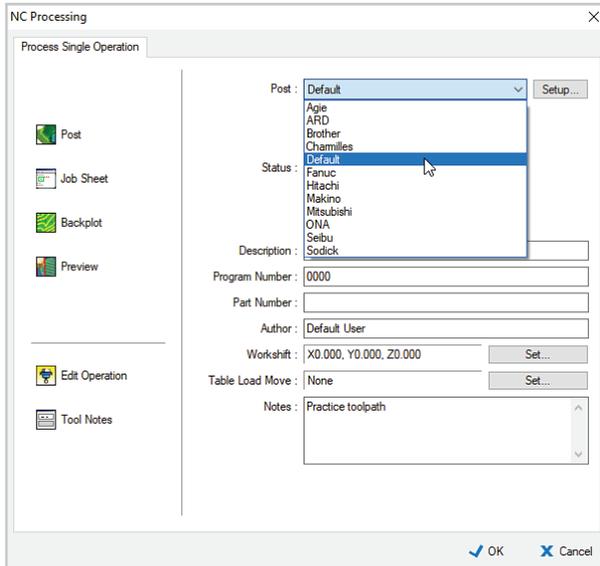


This opens the Process dialog which has controls and tools for Post-processing your program.

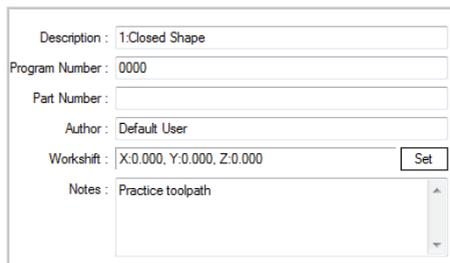


For most CNC machines the basic NC program G and M codes follow the ISO standard. Other codes vary between controllers so OneCNC must format the code when the NC file is output. The format setting is called a 'Post', which is short for post-processor. OneCNC is supplied with prepared Posts for most machines.

To edit a Post, click on Setup to open the NC Post Settings dialog. See 'Mill NC Post and DNC Settings', in the OneCNC Mill CAM section of the Help files for more information.



Click on the drop down list to select the Post for your machine.

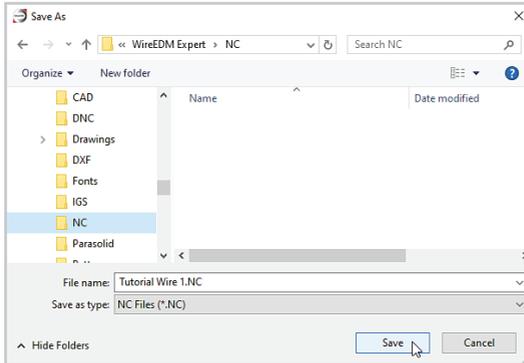


The text fields allow input of information such as the name of the operation, program and part numbers, and notes.

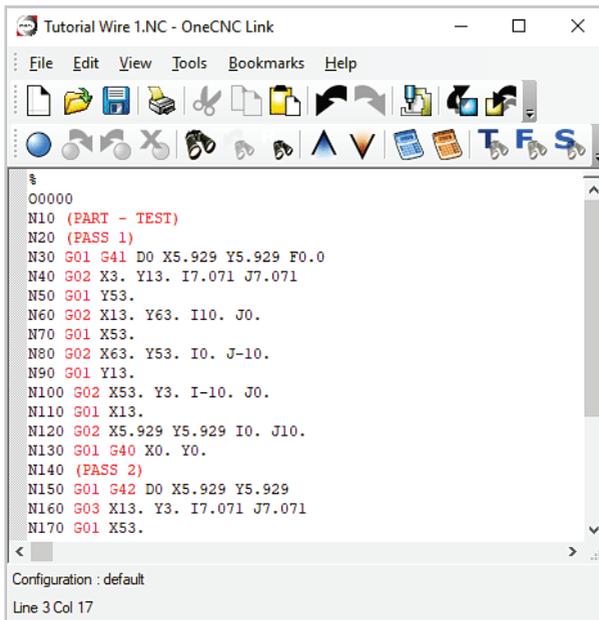
If the part origin on your machine is not the same as your part origin in OneCNC, you can use the Workshift function to relocate the program to suit. For example if the corner of your part is at X20 Y50 on the machine enter a workshift of X-20 Y-50 to correct the position.



Click on the Post icon to output the NC program.

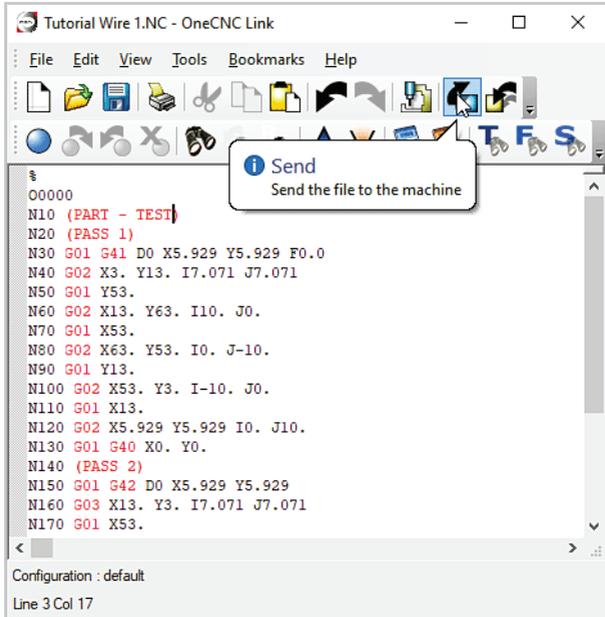


The Save As dialog appears. Choose the directory to save the file to, and enter a name for the file. Select the file type from the drop down menu on the Save As dialog.



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

You will see the (Pass 1) and (Pass 2) comments from the Passes input box where each pass begins.



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

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

Note:



When you first start using OneCNC you should check through the output to make sure it is formatted the way you want for your machine. Once your format is tested on the machine you can post and send files with confidence.

OneCNC WireEDM Tutorial 2

2 axis open profile

In this tutorial we will use part of the closed profile as an open profile.

Prepare the drawing

The wire returns to the start point when cutting a closed profile. For an open profile we need a separate end point for the wire to finish at.



Click on the Point icon in the Toolbox Home menu.
The Point command will be started automatically.

We will create the wire end point at an exact coordinate, using the coordinate entry panel in the toolbox.

coordinate

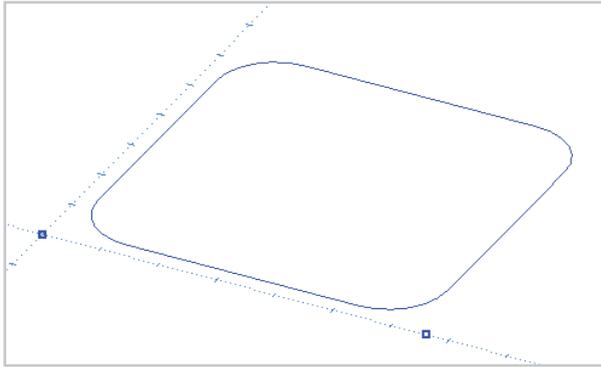
World :	On plane :	
X-Coord	66	66
Y-Coord	0	0
Z-Coord	0	0

ok + Inc

Type 66 for the World X Coord value, and press Enter.

Leave the World Y Coord value at 0, and press Enter.

Leave the World Z Coord value at 0, and press Enter.



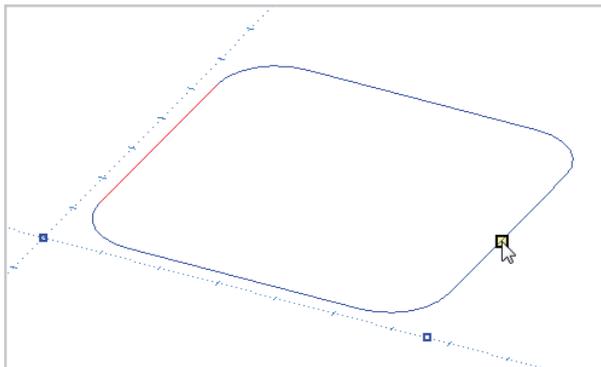
A wire end point is created at X66 Y0.

Right click to end the Point command.

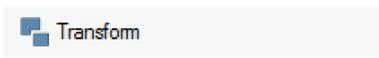
We could create the open profile operation by only selecting part of the closed profile, but for clarity we will blank the adjacent sides.



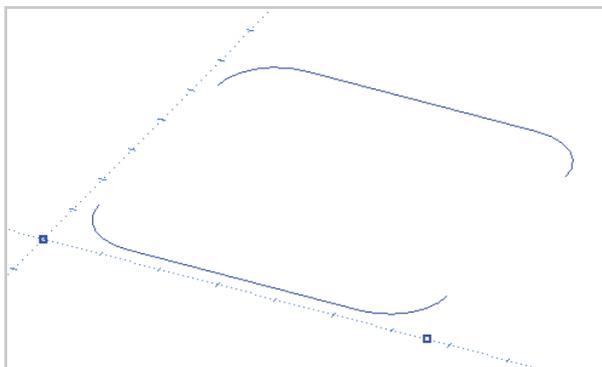
Start the Single selection command.



Select the vertical lines at either side of the square.

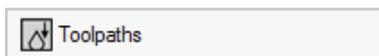


Click on the Blank icon in the Transform menu.

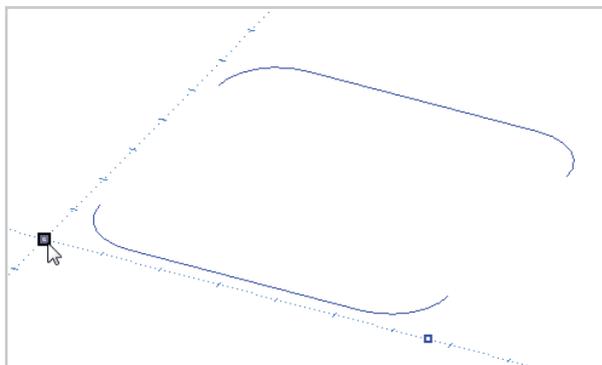


The sides are hidden so we can work on the lower part of the outline as a separate profile.

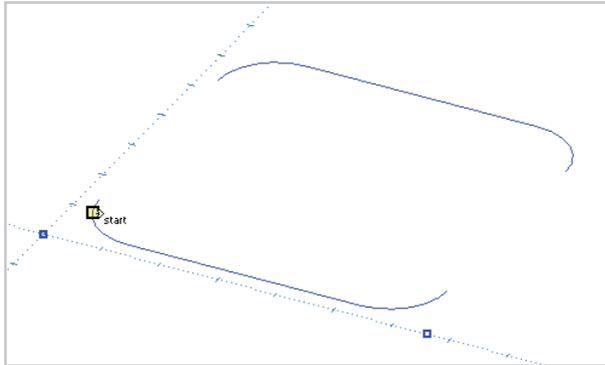
Creating an Open Profile Wire toolpath.



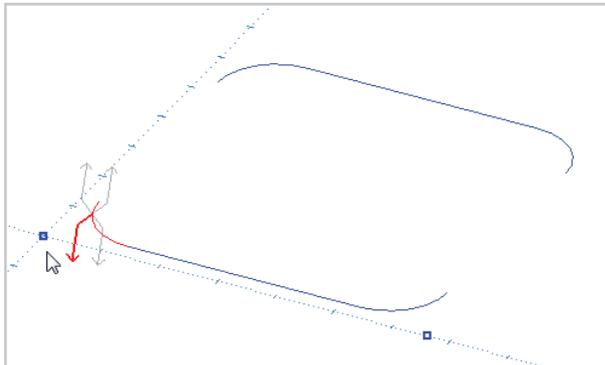
Click on the Toolpaths icon in the Toolbox, and click on the Cut open profile icon in the Toolpaths menu.



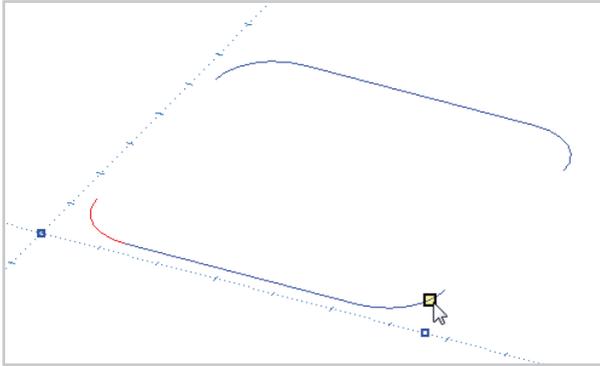
Click on the point at X0 Y0 to select it as the start of the operation.



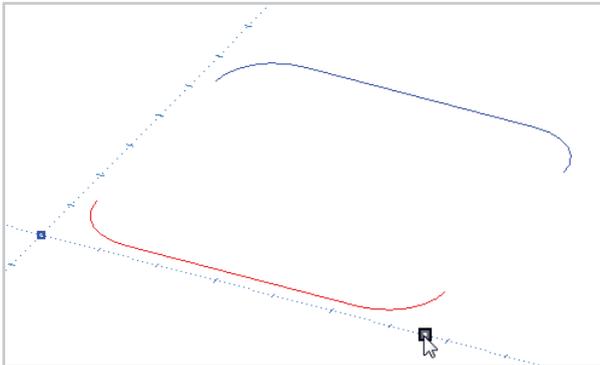
Click on the arc on the left to specify the start of the profile chain.



Hold the pointer to the left to pick the outside of the profile for the wire offset.

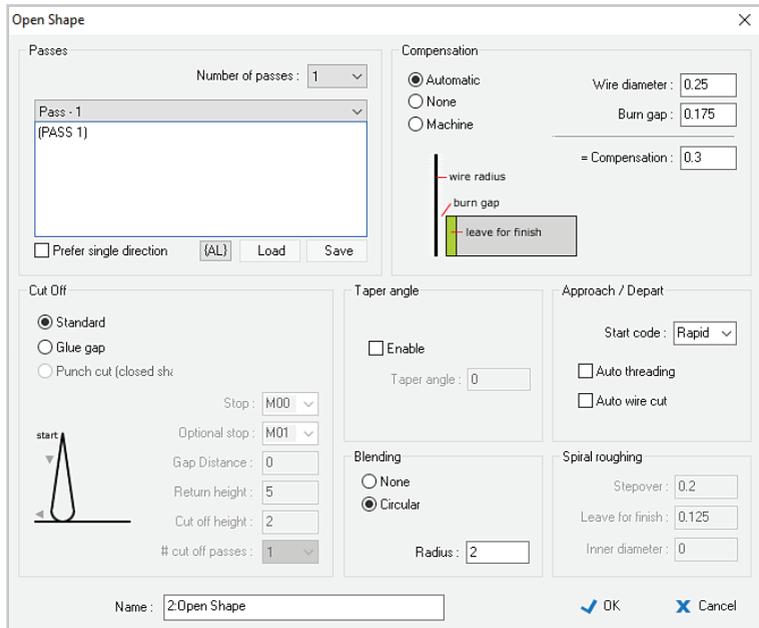


Click on the arc on the right to specify the end of the profile chain.



Right click to end the profile selection stage, and pick the point at X66 Y0 for the end position of the operation.

The Open Shape dialog will open.

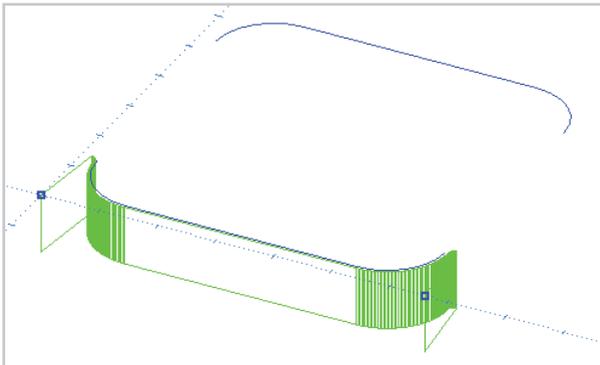
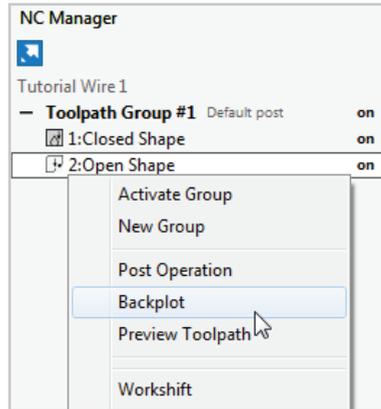


Select 1 for Number of Passes and Automatic compensation.

Select Standard Cutoff and Circular Blending. Enter a radius of 2 for the blending radius.

Click OK and the wire toolpath will be created.

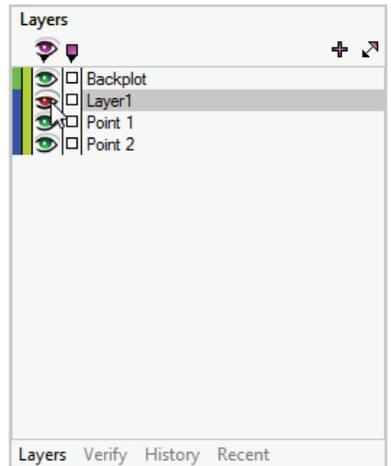
Right click on the Open Shape toolpath in the NC Manager, and select Backplot.



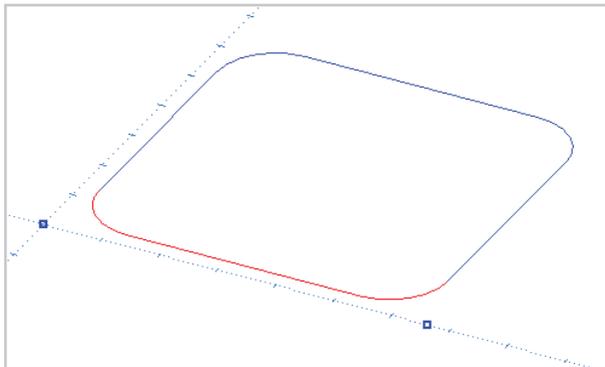
Here is the wire toolpath as it appears with the Standard Cutoff and Circular Blending radius of 2.

The wire toolpath can now be previewed and posted in the same way as the closed profile wire toolpath.

To make the blanked lines visible again, click on the eye icon for the layer in the Layers Manager and all objects on that layer will be visible again.



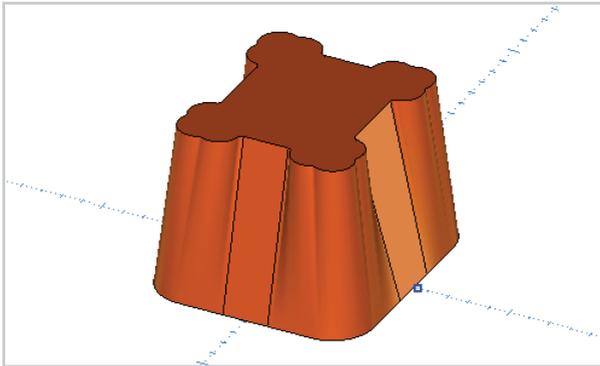
In this example we blanked the adjacent segments of the profile so it was clear what we were selecting.



You can select an open profile which is part of a closed profile by only selecting the start and end entities you want to use for the toolpath.

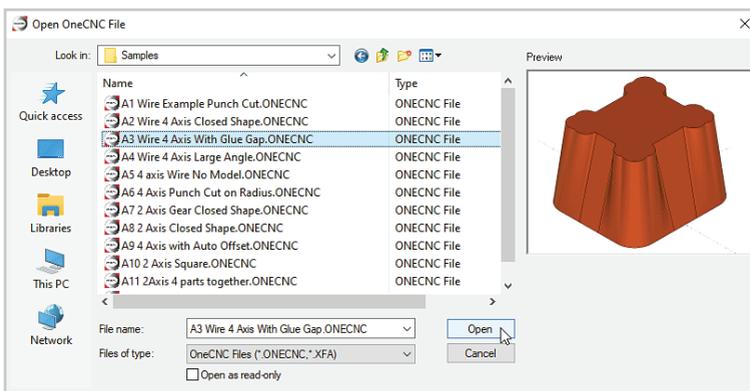
OneCNC WireEDM Tutorial 3

4 axis solid model with glue gap



In this exercise we are going to generate a 4 axis wire toolpath on a solid model and include a glue gap.

Open the sample file and save a copy to practice in.



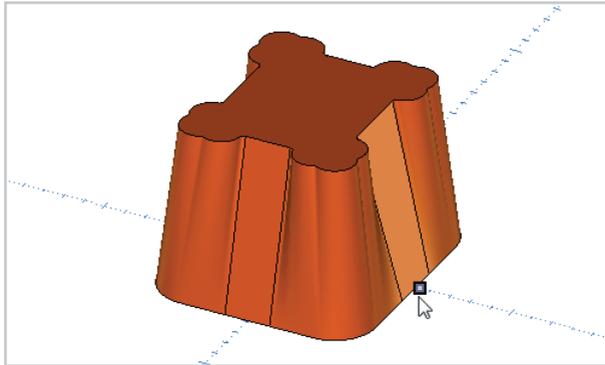
There is a Samples folder in the Drawings folder where you installed OneCNC. Open the sample file 'A3 Wire 4 Axis With Glue Gap.ONECNC', and save a copy of the file as Tutorial Wire 3.

Delete the existing wire toolpath in the NC Manager.

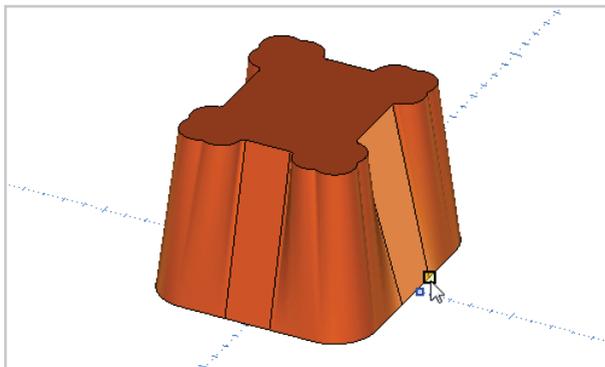
Creating a 4 axis Wire toolpath with glue gap.



Click on the Toolpaths icon in the Toolbox, and click on the 4 axis cut icon in the Toolpaths menu.



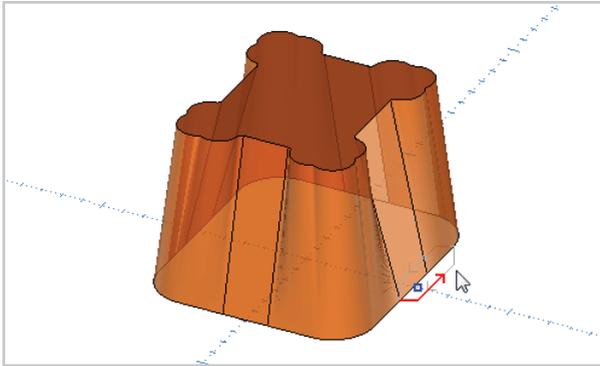
Like a closed or open profile wire toolpath, the 4 axis wire toolpath requires a start position. Click on the point entity to the right of the model.



You are prompted to pick a position on the model. This point will define the first connection, or join from the bottom profile to the top profile. It is not necessarily where the cut will start.

Click the surface endpoint at the base of the model as shown in the image.

The wire toolpath side and direction indicators appear.



Hold the cursor outside the model and on the far side of the start point, and click to set the side and direction.

The 4 axis wire cut dialog opens, as there are no other selections to make for a 4 axis toolpath.



In the Cut Off section of the 4 axis wire cut dialog, select the Glue gap option.

This will add machine stops to the last stage of the program.

Enter a Gap distance of 2.

4 Axis wire cut

Number of passes : 1

Pass - 1

Prefer single direction (AL) Load Save

Compensation

Automatic
 None
 Machine

Wire diameter : 0.25
Burn gap : 0.175
= Compensation : 0.3

wire radius
burn gap
leave for finish

Cut Off

Standard
 Glue gap
 Punch cut

Stop : M00
Optional stop : M01
Gap Distance : 2
Return height : 5
Cut off height : 2
cut off passes : 1

start
optional stop
stop
Gap

Guides

Guide control -
Part programming (default)

Top guide : 100
Bottom guide : 0

Blending

None
 Circular

Radius : 1

Approach / Depart

Start code : Rapid

Auto threading
 Auto wire cut

Spiral roughing

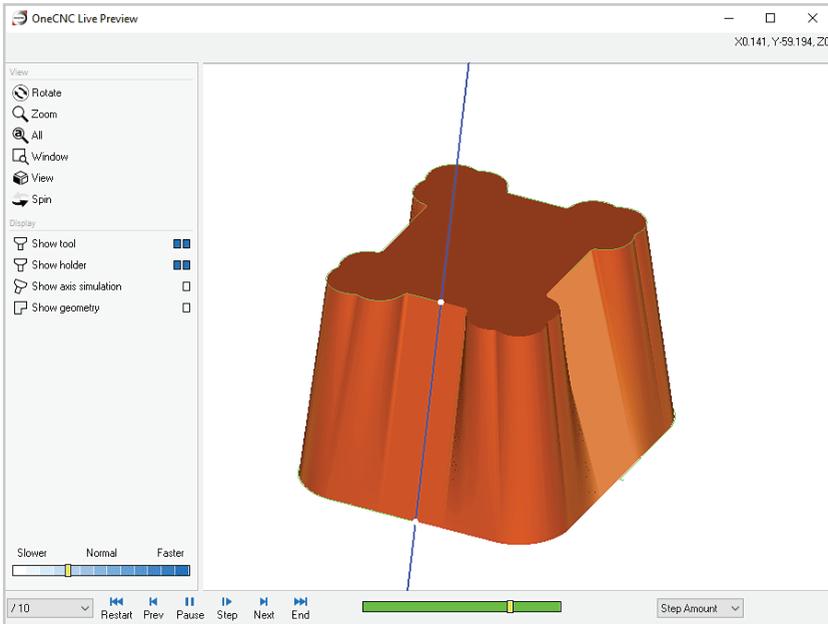
Stepover : 0.2
Leave for finish : 0.125
Inner diameter : 0

Name : 2:4Axis Wirecut

OK Cancel

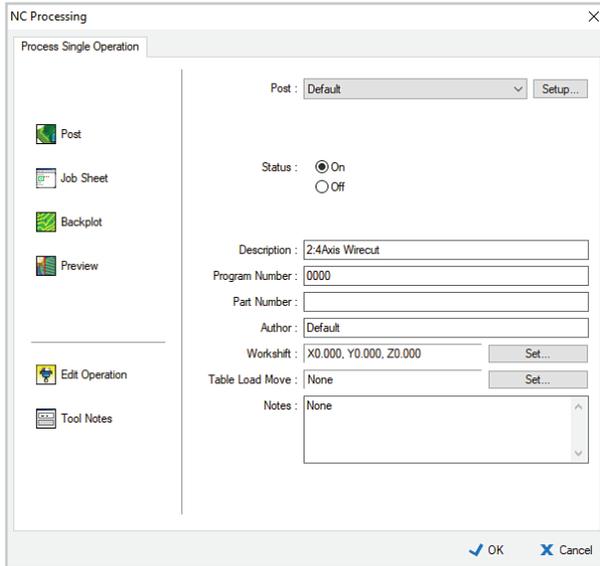
Select 1 for the Number of passes, and Automatic compensation. Select None for Blending.

Click OK and the machining operation will be created.

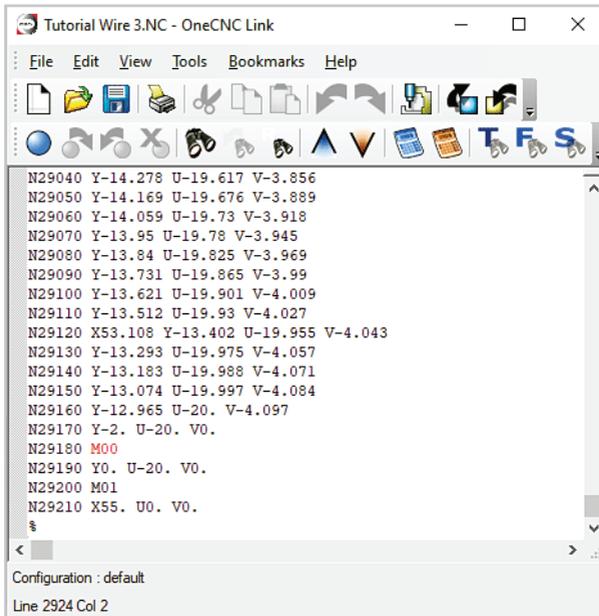


Right click on the new wire toolpath and select Preview Toolpath. You will see the wire move directly to the model, travel to the first connection, and then continue around the perimeter, maintaining continuous contact with the 3D surface as it does so.

Post the wire toolpath.



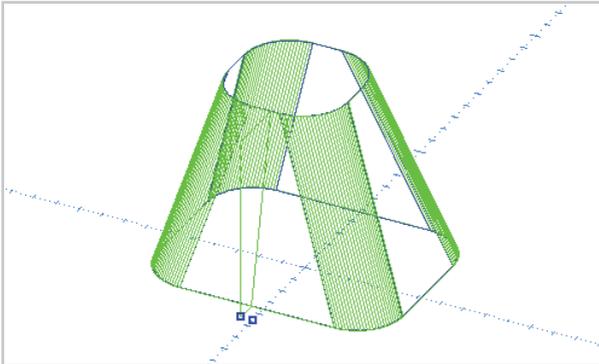
Post the NC file using your preferred post settings.
Press Ctrl and End simultaneously to jump to the end of the file. You



will see an M00 program stop at Y-2 with an optional M01 stop at Y0.

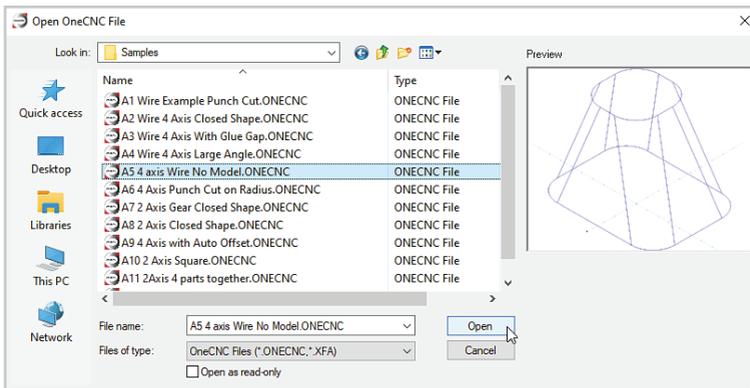
OneCNC WireEDM Tutorial 4

4 axis from wireframe drawing

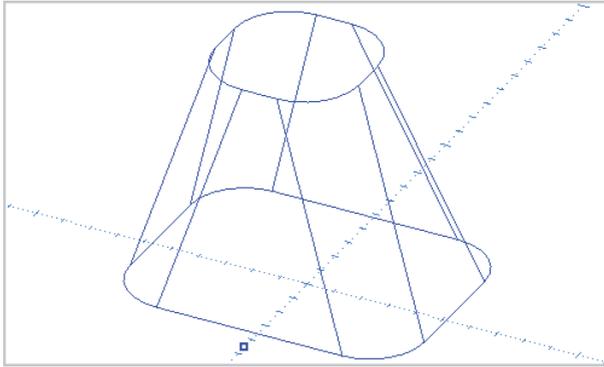


In this exercise we demonstrate OneCNC's ability to create a 4 axis program using only wireframe geometry without a solid model.

Open the sample file and save a copy to practice in.

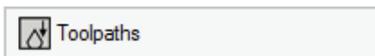


Open the 'A5 4 axis Wire No Model.ONECNC' sample file and save a copy as Tutorial Wire 4. Delete the existing wire toolpath in the NC Manager, and select the Trimetric View.

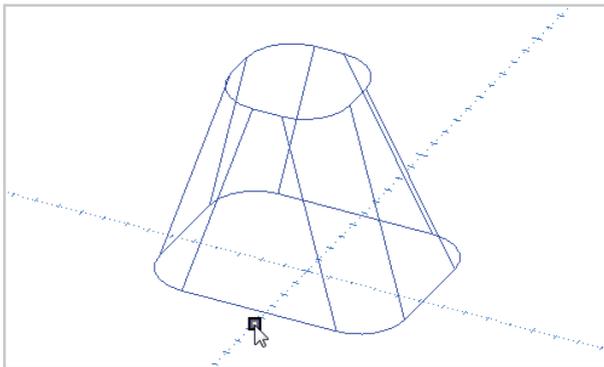


Note that a start point has been sketched near the Y axis on the Y-side of the wireframe model.

Creating a 4 axis Wire toolpath.

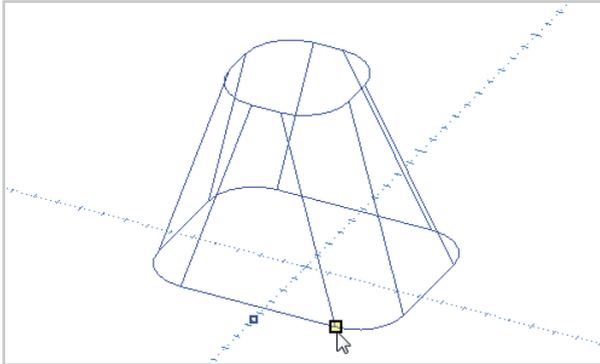


Click on the Toolpaths icon in the Toolbox.
Click on the 4 axis cut icon in the Toolpaths menu.

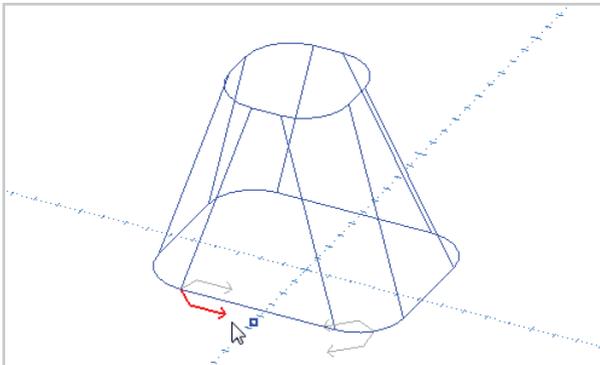


The Status bar prompts you to select the start position. Click on the point entity located below the wireframe as shown.

The Status bar now prompts you to pick a position on the model or wireframe.

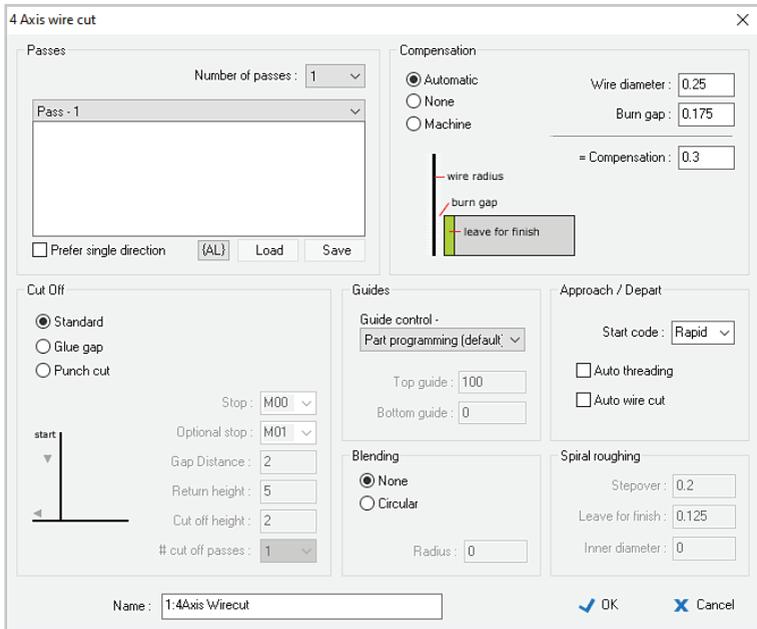


Click on the intersection to the right of the start point. This point will define the first connection, or join from the bottom profile to the top profile.



The direction and side indicator arrows appear. Position the pointer so the direction arrow is outside the wireframe model and pointing to the right, and click to set the side and direction.

The 4 Axis wire cut dialog appears.



Select 1 for the Number of passes, and Automatic compensation. Select None for Blending.

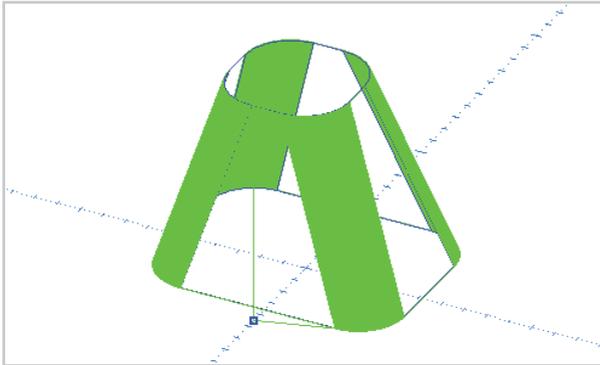
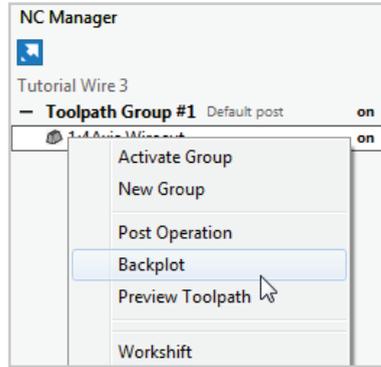
Select the Standard option for Cut Off.

Click OK and the wire toolpath will be created.

Backplot the wire toolpath.

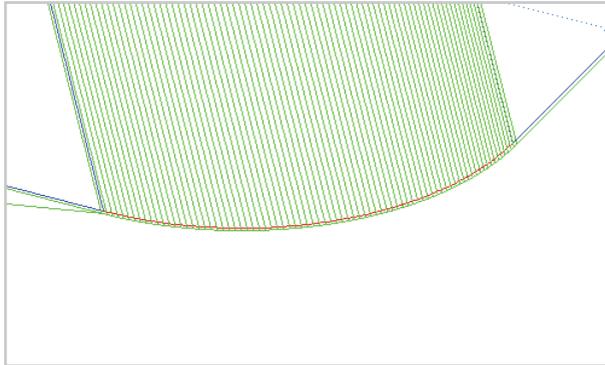
The wire toolpath operation is now listed in the NC manager. The operation will be displayed in the main drawing window while the operation is selected.

To draw the wire toolpath as geometry, right click the '1:4Axis Wirecut' operation in the NC manager and select Backplot.



The wire toolpath geometry is created on an automatically generated Layer named Backplot.

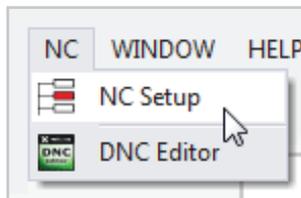
To create the correct shape as the wire moves around the corner arcs, the toolpath must be converted to line movements.



The wire toolpath appear solid on the corners, because the default tolerance setting creates a smooth curve by keeping the size of the line moves very small.

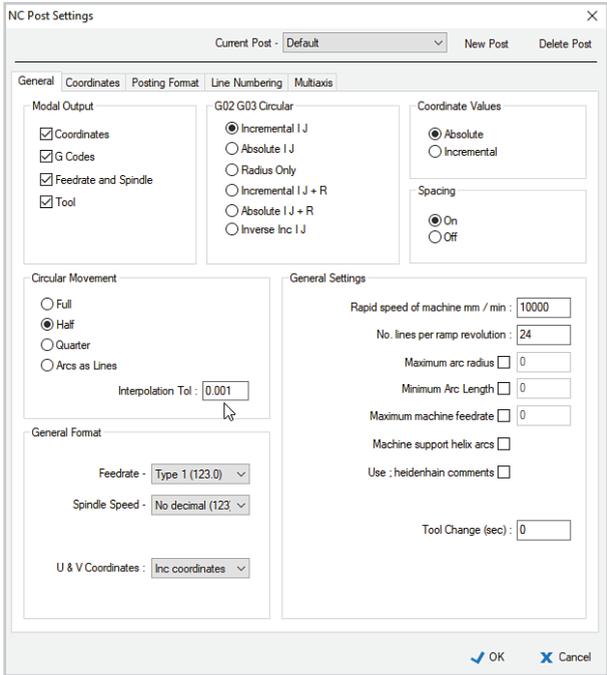
Setting Interpolation Tolerance

The resolution of wire toolpath movement on arc entities is controlled by the Interpolation Tolerance setting in the NC Post Settings dialog.

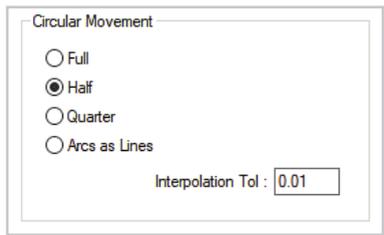


To change the setting, click NC Setup in the NC menu to open the NC Post Settings dialog.

On the left side of the General tab you will find the Circular Movement section.

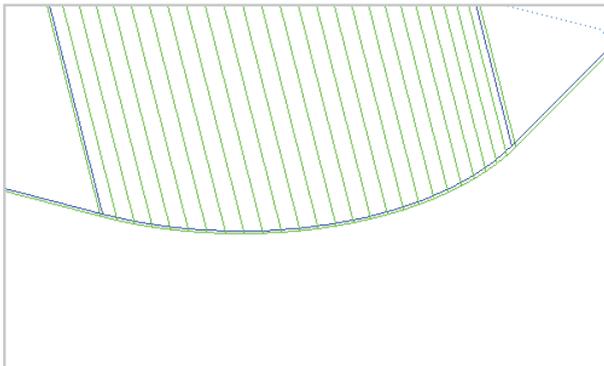
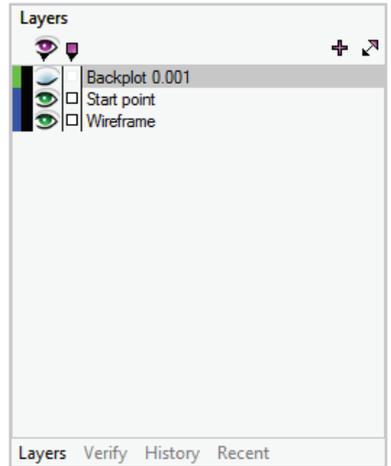


The default setting for Interpolation Tol is 0.001, which will give you a smooth wire toolpath.



To see the effect of this setting, change the Interpolation Tol value to 0.01, and click OK to close the NC Post Settings dialog.

Rename the existing Backplot layer to Backplot 0.001, and turn it off.

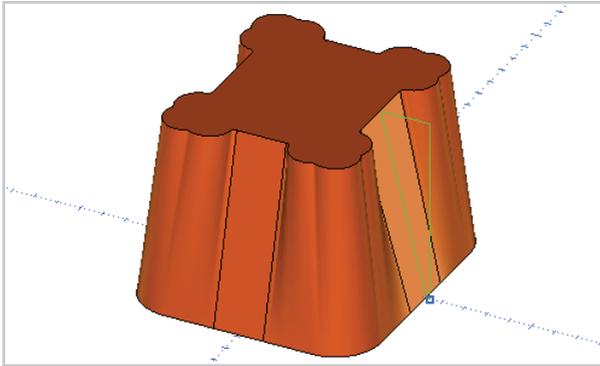


Backplot the '1:4Axis Wirecut' operation in the NC manager and you will see the wire toolpath resolution has been reduced.

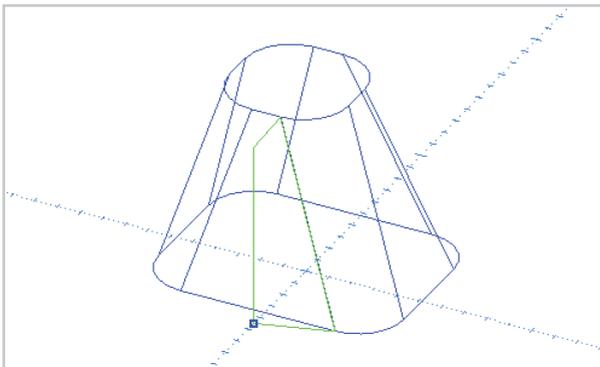
You can turn on the original backplot layer to compare the results.

Reset the Interpolation Tol value to the default tolerance setting of 0.001 before continuing.

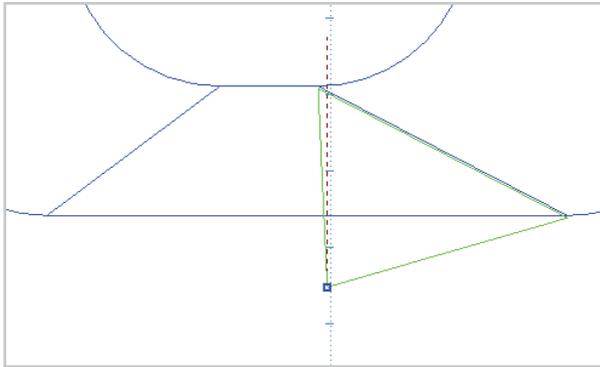
Editing the start point.



The 4 axis wire toolpath we created on the solid model in the previous tutorial started with a perpendicular move from the start point directly to the flat surface.

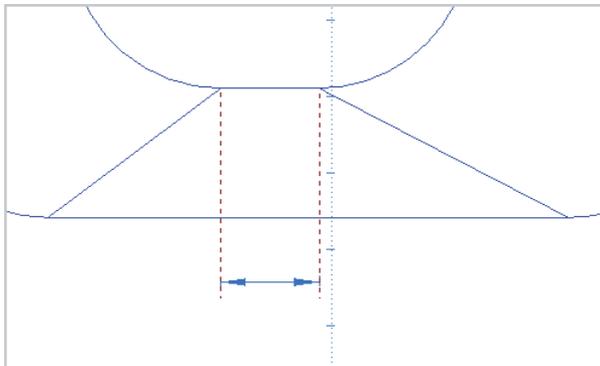


The wire toolpath we created in this tutorial starts with a move to the line connecting the top and bottom profiles.

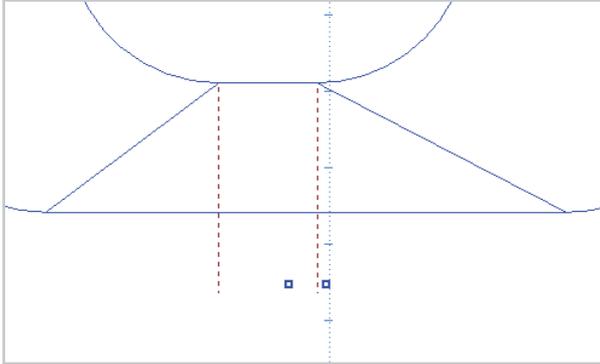


In the top view you can see that it is not possible to draw a line from the existing start point entity perpendicular to the horizontal line of the upper profile.

Because the start point lies across the connection, the wire toolpath will start on the connection itself.



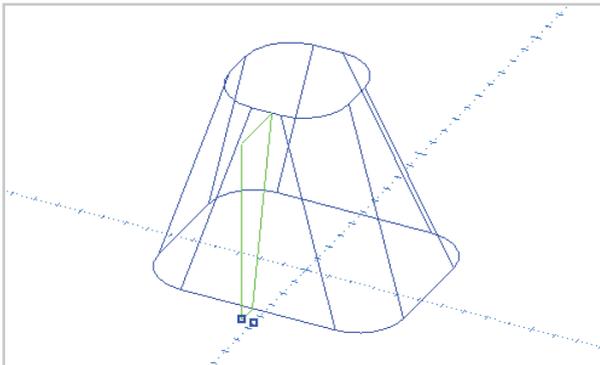
For the wire to approach the job directly we must use a start point perpendicular to both segments between a pair of connections.



Create a new start point using the Copy move command to copy the start point at an increment of X-5 from the original start point.

Right-click the '1:4Axis Wirecut' operation in the NC manager and select the Repick Path option.

Pick the new start point, then select the lower profile exactly as before.



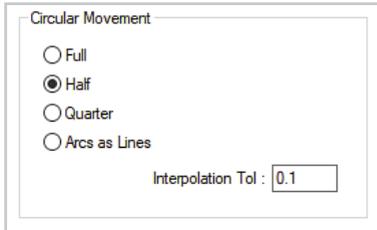
You will see the wire toolpath update with the approach moving directly to the flat side of the job.

Correcting profile geometry

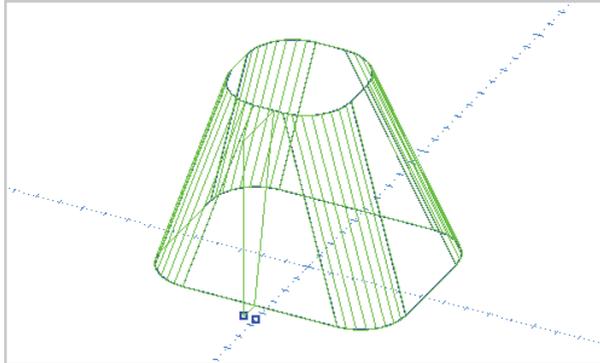
The wire toolpath consists of movements from connection to connection along geometric entities in an upper and lower profile.

If the number of entities between connections in each profile differs, the wire toolpath must compensate by creating excess interpolation points.

We will demonstrate this effect by dividing an arc in the lower profile into two segments.



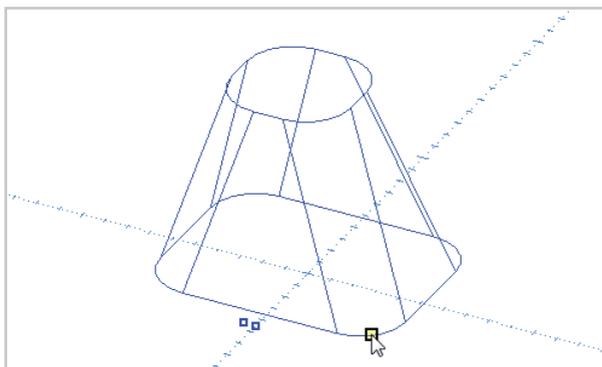
To make the effect obvious, change the interpolation tolerance to 0.1.



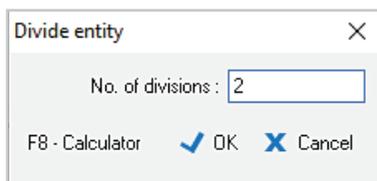
Select the toolpath and the instant preview will show the coarse interpolation setting.



Click on the Divide entity icon in the Trim/Break menu.



Click on the lower arc as shown.

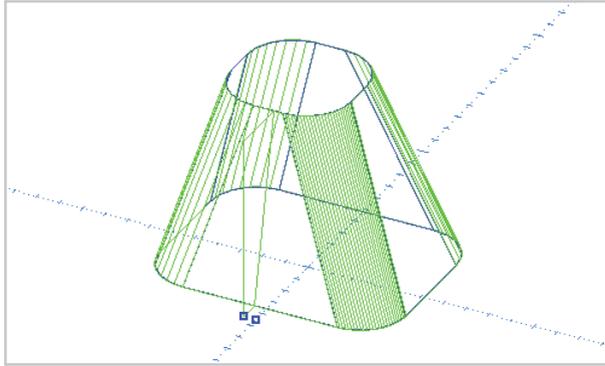


Enter 2 to divide the selected arc into 2 consecutive arcs of the same radius.

Click OK, and then right click to end the Divide command.

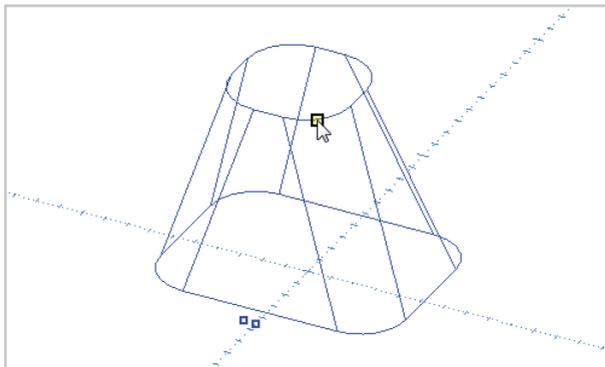
This corner will now have one arc in the top profile and two arcs in the bottom profile.

To update the wire toolpath with the changed geometry, right-click the '1:4Axis Wirecut' operation in the NC manager and select the Repick Path option. Pick the new start point, and select the lower profile exactly as before.



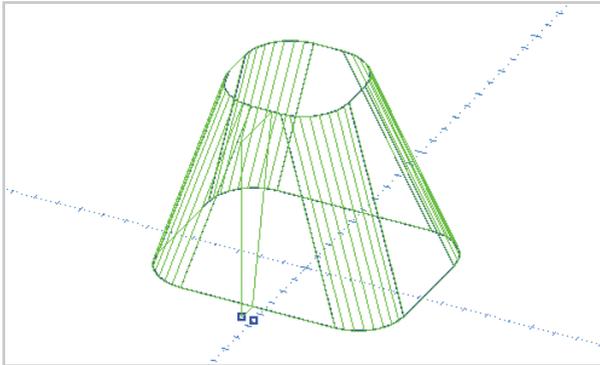
The backplot on the corner with unequal entities between the connections now appears solid as it is made up of many more lines.

The NC output for this wire toolpath will produce the correct shape, but will contain many more lines of G-code.

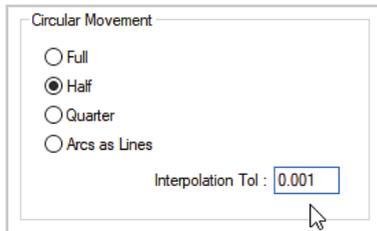


Use the Divide command to divide the upper arc into two segments.

Use Repick Path to update the wire toolpath with the changed geometry again.



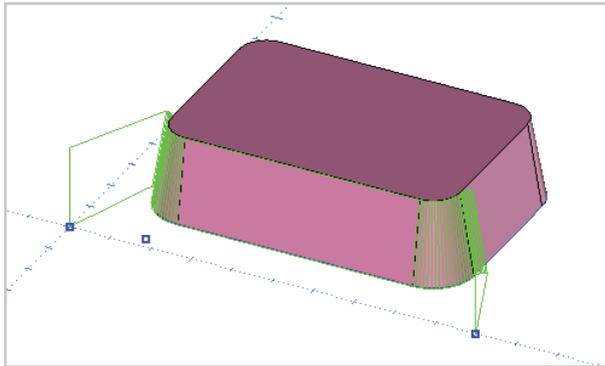
Now that the entity count for each profile matches again, the wire toolpath result will be normal.



Reset the interpolation tolerance to 0.001 before continuing.

OneCNC WireEDM Tutorial 5

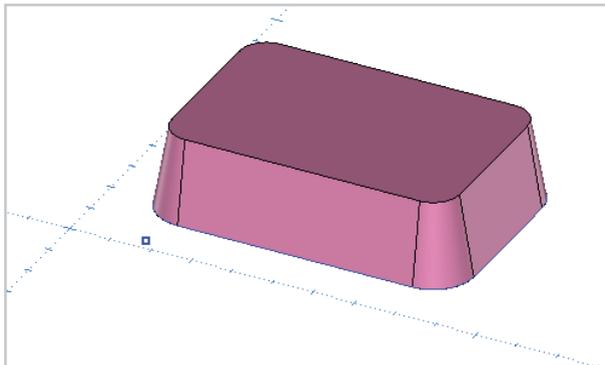
4 axis open shape



In this exercise we will create an open shape 4 axis program. The open shape will cut part of a solid model, using wireframe geometry extracted from the model.

Make a copy of an existing drawing

Open the sample file 'A9 4 Axis with Auto Offset.ONECNC', and save a copy of the file as Tutorial Wire 5.

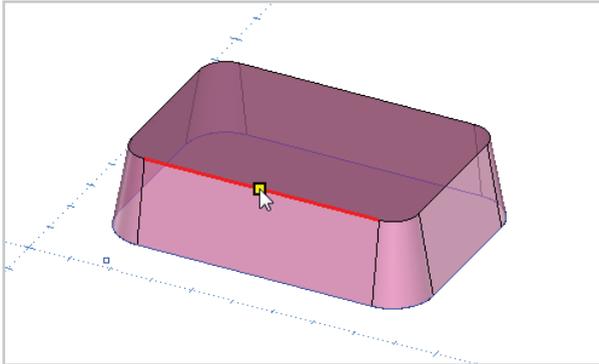


Press the Spacebar and select Trimetric View from the View menu. This file has a solid model and construction geometry on separate layers.

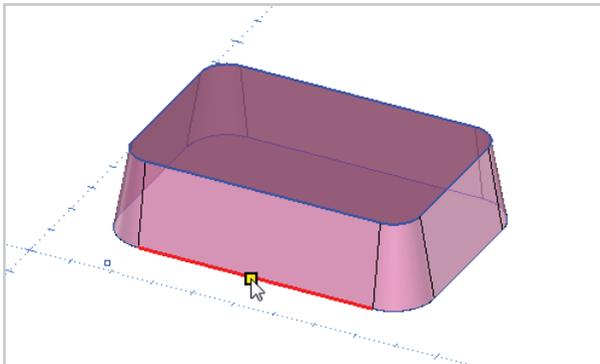
Prepare the drawing.



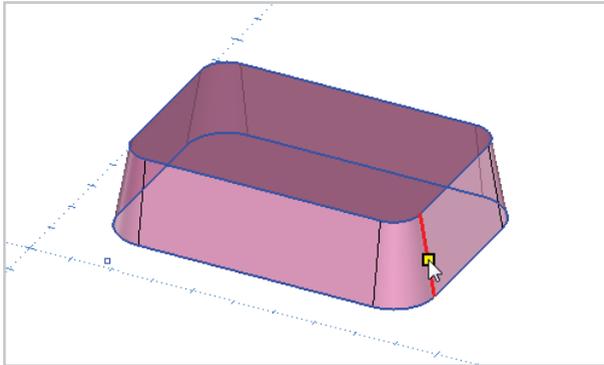
Click on the Extract an edge icon in the Model Tools menu.



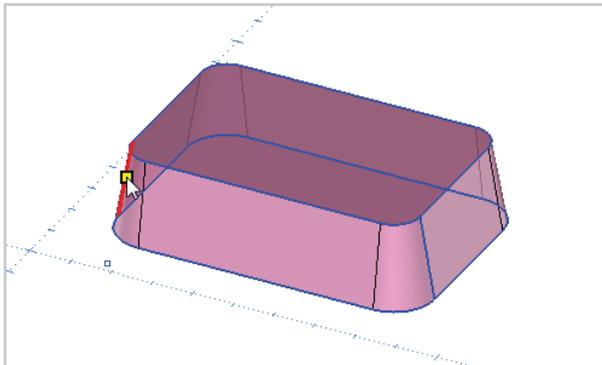
Click on an edge where the top of the model meets a side. The edge around the top will be extracted.



Click on an edge where the bottom of the model meets a side. The edge around the bottom will be extracted.



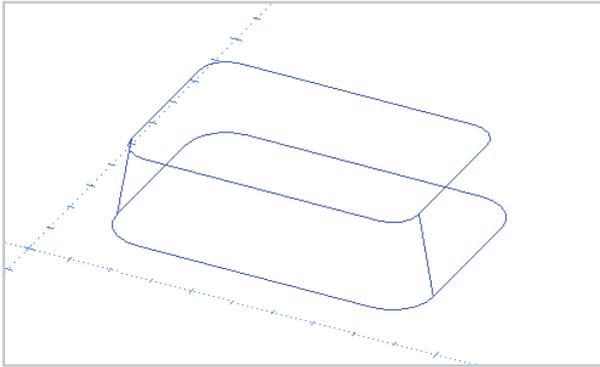
Click on the seam at the right of the model as shown.
An edge will be extracted for the seam.



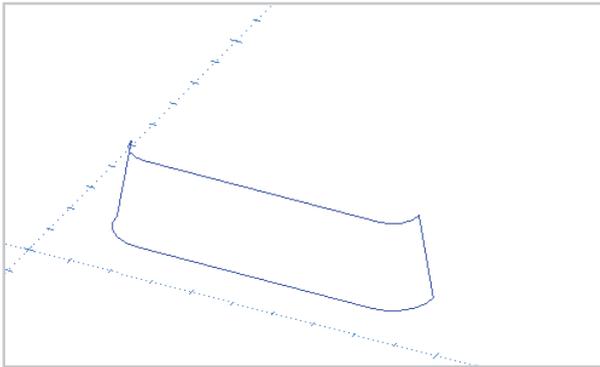
Click on the matching seam at the left of the model to extract the edge.



Right click to end the command.



Turn off display of the Geometry and Model layers and you will see the geometry which has been created on the Extracted layer.



Delete the segments of the upper and lower profiles on the far side of the connecting lines. We now have an open wireframe shape, but we still need to draw a start and end point.



Click on the Point icon in the Toolbox Home menu.
The Point command will be started automatically.

We will enter the coordinate position using the coordinate entry panel in the toolbox.

coordinate

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

Type 0 for the World X Coord value, and press Enter.

Leave the World Y Coord value at 0, and press Enter.

Leave the World Z Coord value at 0, and press Enter.

A wire start point is created at X0 Y0.

coordinate

World :	On plane :
X-Coord 100	100
Y-Coord 0	0
Z-Coord 0	0
<input checked="" type="checkbox"/> ok	+ Inc

Type 100 for the World X Coord value, and press Enter.

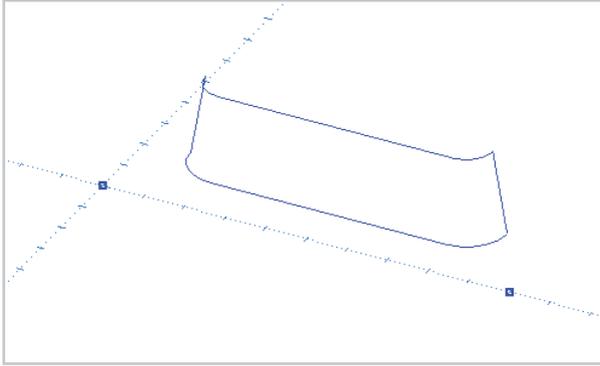
Leave the World Y Coord value at 0, and press Enter.

Leave the World Z Coord value at 0, and press Enter.

A wire end point is created at X100 Y0.



Right click to end the Point command.

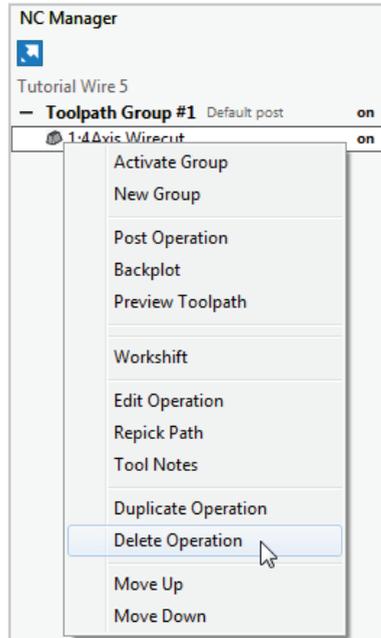


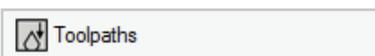
We now have a start and end point and we are ready to create the open wire toolpath.

Create a 4 axis open shape wire toolpath.

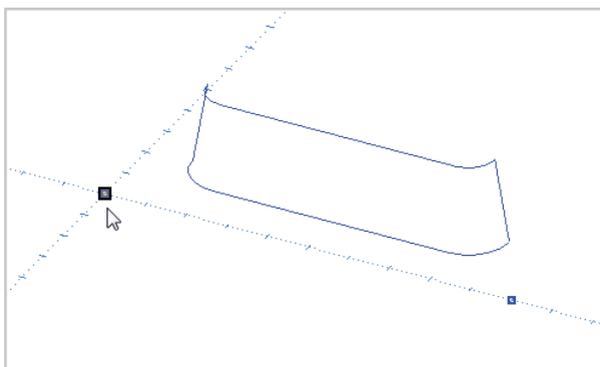
Right click the existing operation in the NC manager and select Delete Operation.

We will create our new wire toolpath in Toolpath Group #1, which is now empty.

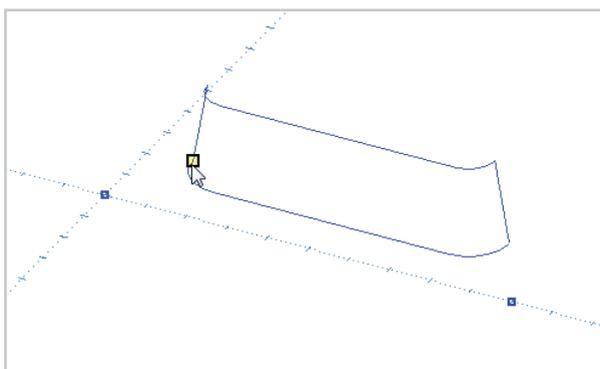




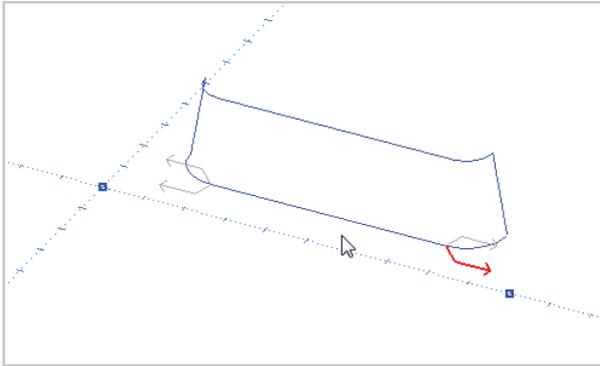
Click on the Toolpaths icon in the Toolbox.
Click on the 4 axis cut icon in the Toolpaths menu.



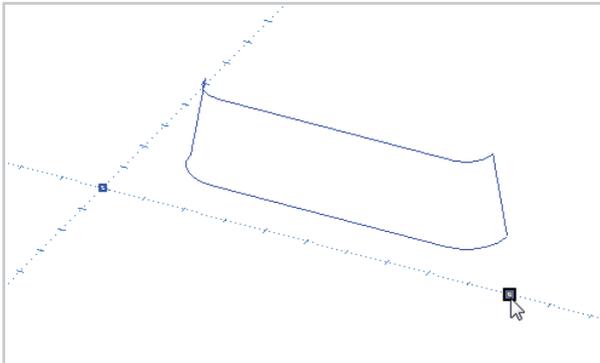
The Status Bar prompts you to pick a start position. Click on the point at X0 Y0.



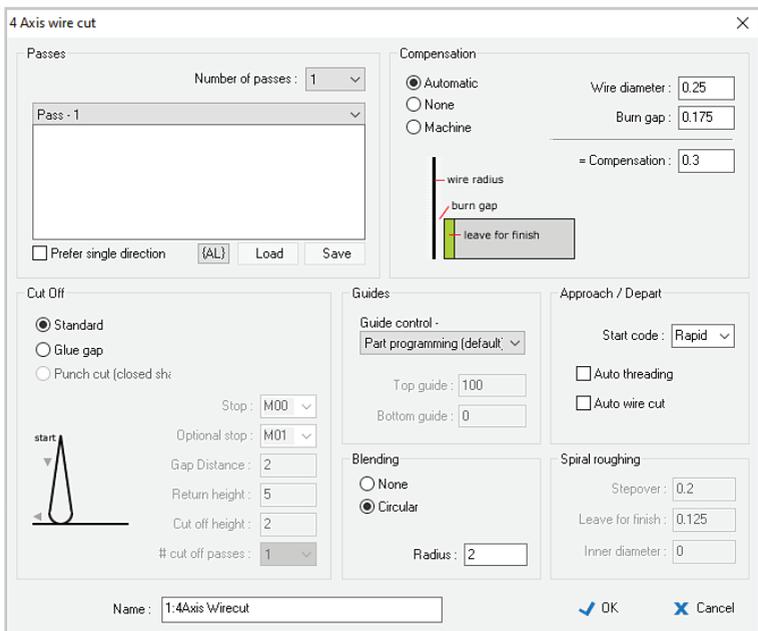
Click on the left end of the lower profile to select it as the start of the cut profile.



Hover the cursor to the left of the profile to indicate the direction and offset side for the wire toolpath, and click to confirm the selection.

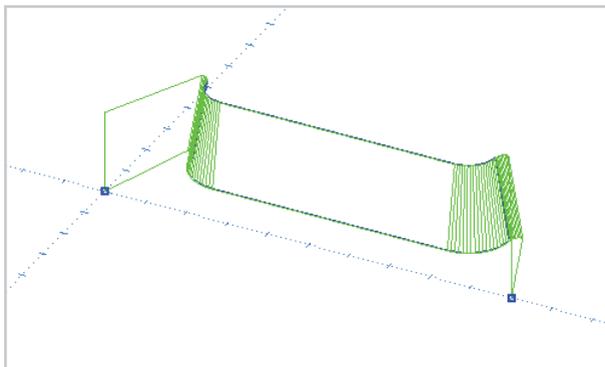


Click the point at X100 Y0 to select the end position.



The 4 Axis wire cut dialog appears. Select 1 for the Number of passes, and Automatic compensation.

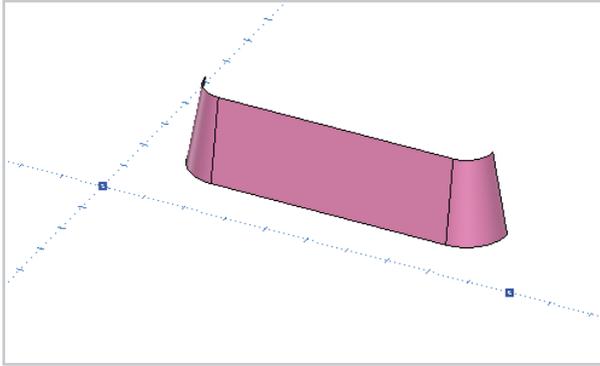
Select the Standard option for Cut Off. Select Circular Blending and enter a radius value of 2.



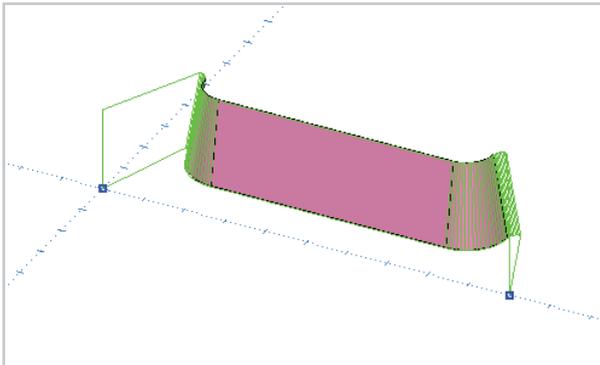
Click OK and the wire toolpath will be created.

An alternative method.

We have demonstrated OneCNC's ability to create an open operation using the wireframe method, but it is also possible to create 4-axis operations on open surface entities.



Another way of achieving the same result would be to create a copy of the model on a new layer, and use the Break to Surfaces model operation to separate the solid into surfaces. The surfaces we did not want to cut could then be blanked or deleted.



The 4 axis wire toolpath could then be defined on the surface shapes in the same way as it was for the wireframe.

OneCNC Support

For assistance contact the OneCNC office for your location.

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

Client Number

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

Version

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

OneCNC product

The type and level of OneCNC such as Mill Expert.

Units

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

OneCNC Global Offices

Australia

OneCNC Australia

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

Benelux

OneCNC Benelux

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

Denmark

OneCNC Denmark

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

France

OneCNC France

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

Germany

OneCNC Germany

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

India

OneCNC India

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

Indonesia

OneCNC Indonesia

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

Ireland

OneCNC Ireland

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

Italy

OneCNC Italy

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

Japan

OneCNC Japan

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

Korea

OneCNC Korea

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

Mexico

OneCNC Mexico

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

Portugal

OneCNC Iberia

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

Poland

OneCNC Poland

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

South Africa

OneCNC South Africa

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

Spain

OneCNC Iberia

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

Sweden

OneCNC Sweden

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

Taiwan

OneCNC Taiwan

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

United Kingdom

OneCNC UK

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

USA

OneCNC LLC

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

USA California

OneCNC West

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

Copyright Notice

Copyright Notice

Copyright © 2020 All Rights Reserved

This publication or parts thereof, may not be reproduced in any form, by any method, for any purpose.

Trademarks

The following are registered trademarks of QARM Pty Ltd, in Australia and/or other countries: OneCNC XR8, OneCNC XR7, OneCNC XR6, OneCNC XR5, OneCNC XR4, OneCNC XR3, OneCNC XR2, OneCNC XR, OneCNC XP, OneCNC 2003, OneCNC 2000, OneCNC, AusWire, AusMill, AusCAD, AusCADCAM, NCSentry, NCLink.

Third-Party Trademarks

All other brand names, product names or trademarks belong to their respective holders.

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

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



Introduction to OneCNC WireEDM v62.80