

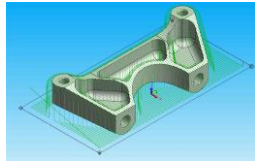


Dolphin 3DCAM Help

Table of Contents

Introduction	3
Getting Started	4
The Ribbon Toolbar	5
File	6
Geom	9
Solids	24
View	25
Milling	26
Screen Layout	69
Creating and using Sketches	70
Program Tree Objects	74
Imported File	74
Program Number	76
Tools	77
Patterns	80
Surfaces	82
Operations	84
Main Graphics Area	85
System Options	88
Properties	89
Messages	92

Introduction

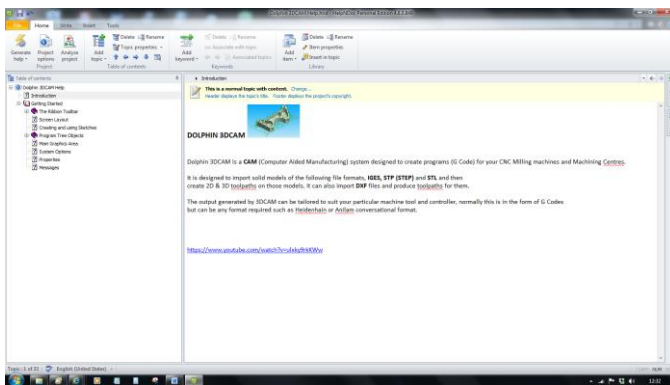


DOLPHIN 3DCAM

Dolphin 3DCAM is a **CAM** (Computer Aided Manufacturing) system designed to create programs (G Code) for your CNC Milling machines and Machining Centres.

It is designed to import solid models of the following file formats, **IGES**, **STP (STEP)** and **STL** and then create 2D & 3D toolpaths on those models. It can also import **DXF** files and produce toolpaths for them.

The output generated by 3DCAM can be tailored to suit your particular machine tool and controller, normally this is in the form of G Codes but can be any format required such as Heidenhain or Anilam conversational format.



Getting Started

Getting Started

Below is the main 3DCAM screen.

The screen is split into various areas, they are....

The [Ribbon toolbars](#) control all functions within the software. To display different [Ribbons](#) (topics) - please click on the appropriate ribbon item - such as File, Geom, Solids etc.

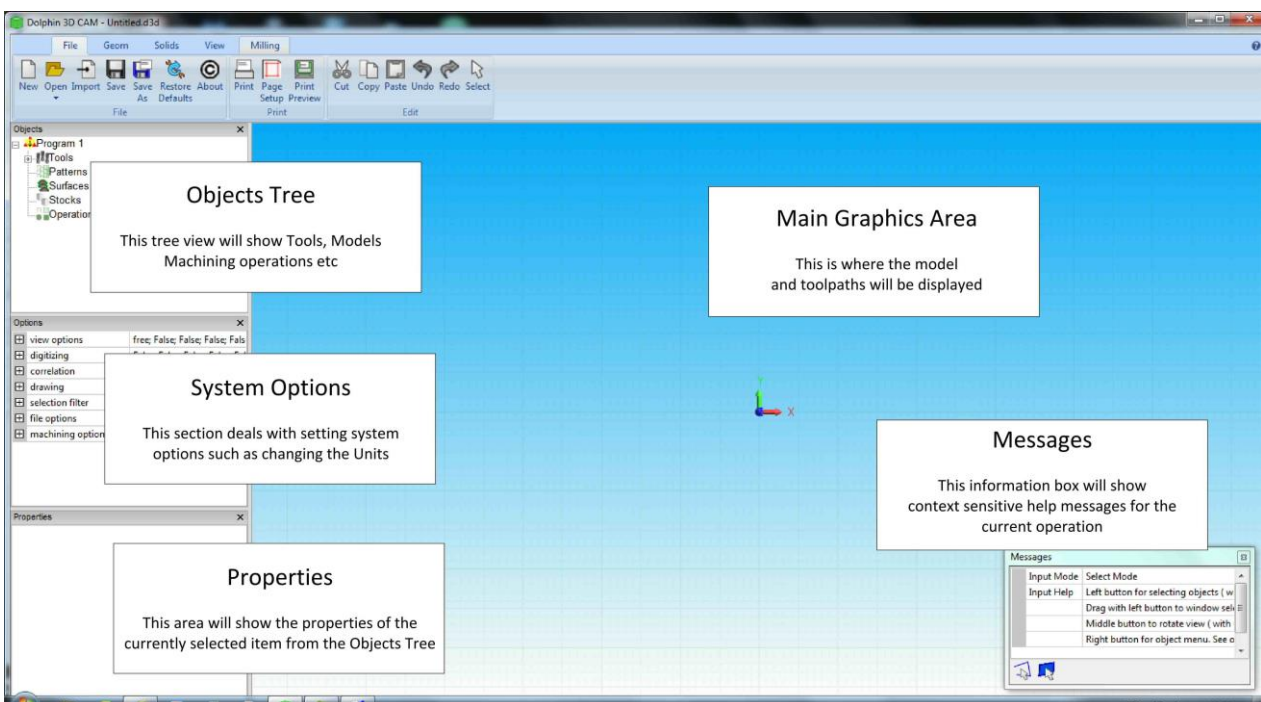
[Main Graphics Area](#)

[Objects Tree](#)

[System Options](#)

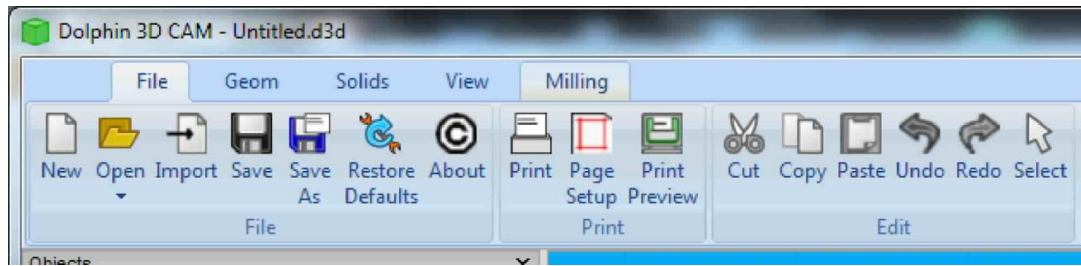
[Properties](#)

[Messages](#)



The Ribbon Toolbar

The Ribbon toolbars control all of the functions and options within 3DCAM



To change ribbons, click on the appropriate menu item

[File](#)

[Geom](#)

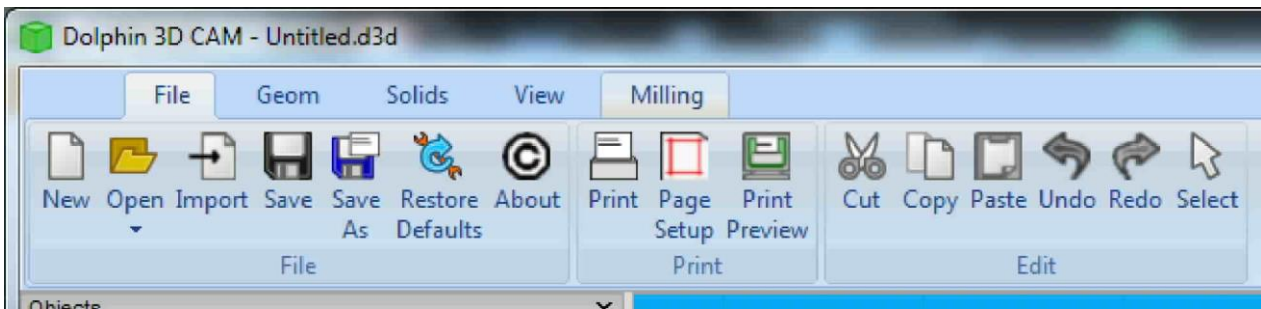
[Solids](#)

[View](#)

[Milling](#)

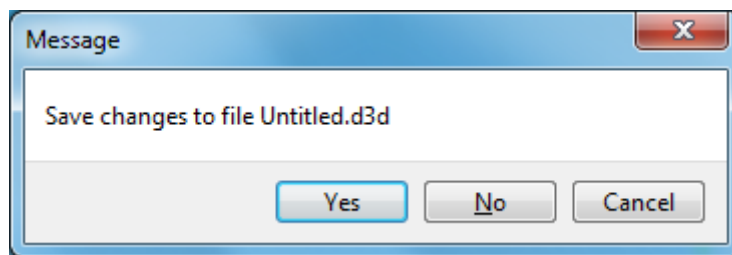
File

This is the File Ribbon Toolbar, it is split into 3 sections, File, Print and Edit - the options are listed below



THE FILE SECTION

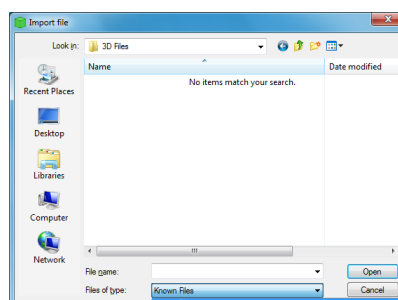
File New - this will create a new, empty workspace. If you haven't Saved your project you will see this dialogue box where you can save or discard any operations.



File Open - This will allow you to Open an existing 3DCAM project, you will be shown the **File Open** dialogue box where you can choose the project you wish to Open or browse to a different folder to choose your project.

Clicking on the **Down Arrow** - will show the **Recent File list**. This will list the last 12 files you worked on.

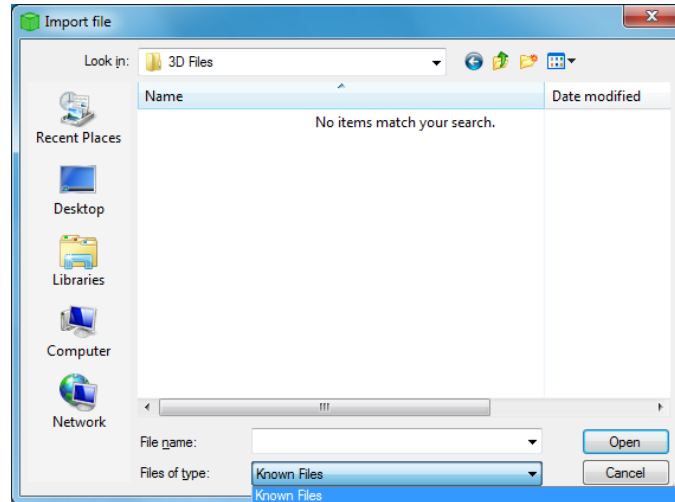
Files created by 3DCAM will have the extension **.d3d** for instance **MYFIRSTPART.d3d**. If you need to contact our technical support department with questions concerning a part you are working on, they will need to have the .d3d file you are working on as well as the original geometry file that you imported. Please attach these files to any emails you send.



The File Open dialogue box

File Import - Use this option to import a geometry file that you can add machining operations onto. The Import Dialogue box will be shown. The file types that can be imported are:

IGES files - 3D Solid and Surface models, **STEP (STP)** 3D Solid and Surface models, **STL** - 3D models made up of 3D triangles, **DXF** - 2D geometry files.



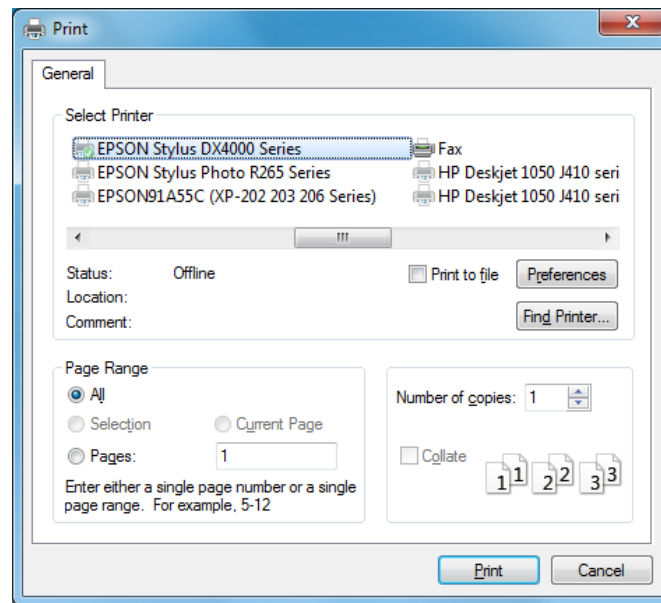
File Save - This option will save the current project under the name previously given with Save As... if a name has not been assigned to the project, the File > Save As dialogue box will be displayed. If a filename has been given, the current status of the project will be saved.

Restore Defaults - This will restore all system defaults, such as units, snap options etc to their factory settings. Please note; this action cannot be "undone"

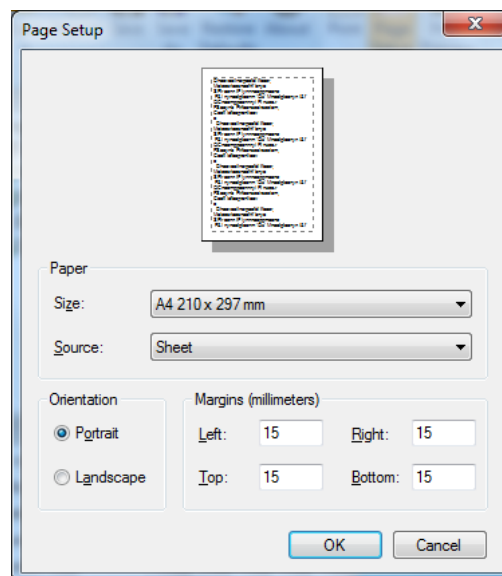
About - Will display the version and build number of the software any copyright notices.

THE PRINT SECTION

Print - This option will print the contents of the graphics window to your printer, the print dialogue box will be shown that allows you to select the printer to use



Page Setup - This option will you set the size and orientation of the paper to use for printing



Print Preview - Will show the view to be printed.

THE EDIT SECTION

Cut - This command will Cut (Delete) the current selection from the Objects Tree view

Copy - This command will Copy the current selection from the Objects Tree view to the Clipboard so that it can be used again

Paste - This command will Paste (Add) the current contents of the Clipboard into the Objects Tree view

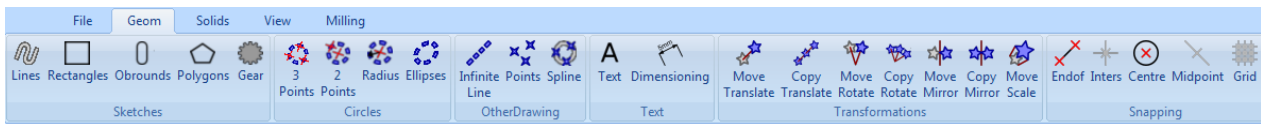
Undo - Will undo the previous operation, this could be changing a machining operation, moving the model, modifying a Sketch etc.

Redo - Will reverse the actions of the preceding Undo operation.

Select - This option will allow you to select items in the [Main Graphics area](#)

Geom

This is the Geometry Ribbon Toolbar, it is split into 6 sections, **Sketches**, **Circles**, **Other Drawing**, **Text**, **Transformations** & **Snapping**- the options are listed below



[Sketches](#)

[Circles](#)

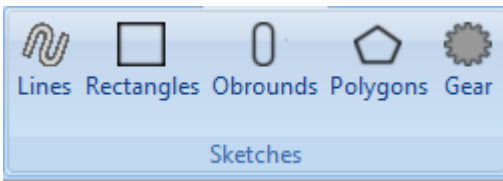
[Other Drawing](#)

[Text](#)

[Transformations](#)

[Snapping](#)

Sketches

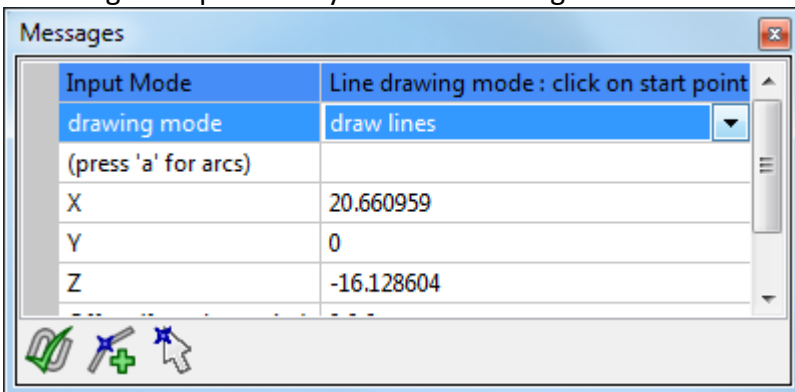


Sketches

When using any of these options, please ensure that your current Graphics Plane view is what you require, eg XY, YZ or XZ. Use the [View](#) toolbar to set your required View.

Lines

Selecting this option will you to draw a single line or a series of connected lines or arcs.



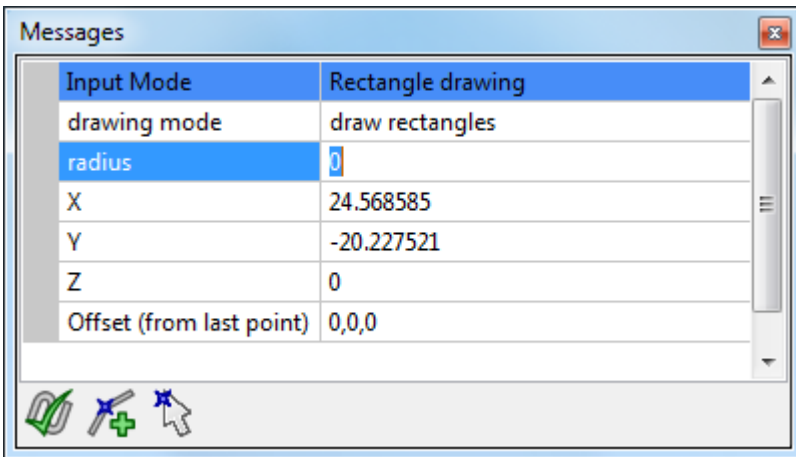
The Message box will display the current option, use the down arrow to change options. Use the mouse to position the cursor in the graphics area to draw the lines/arcs.

End points can be entered by clicking the X, Y or Z inputs.

Rectangles

This will draw Rectangles and create them as Sketches that can used for the Profile or Pocket machining commands

Use the Messages box to set a corner radius if required.

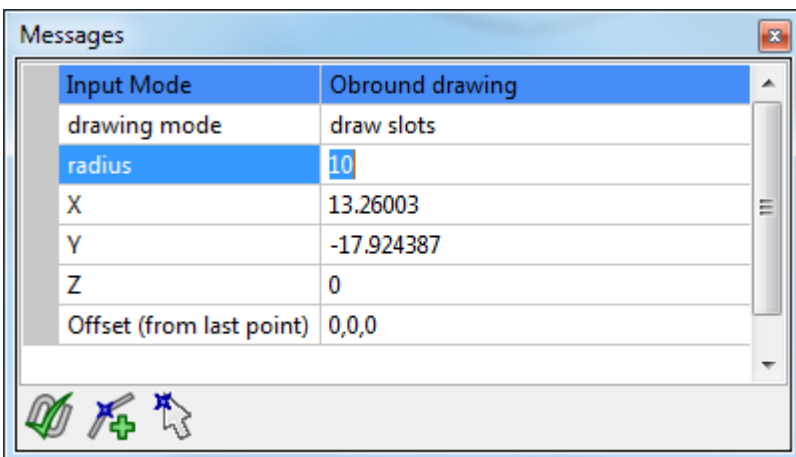


Use the mouse to position the cursor in the graphics area to draw the. The opposing corners of the Rectangle can be entered by clicking the X, Y or Z inputs.

Obrounds

This will draw Slots and create them as Sketches that can be used for the Profile or Pocket machining commands

Use the Messages box to set the radius (width) of the Slot.

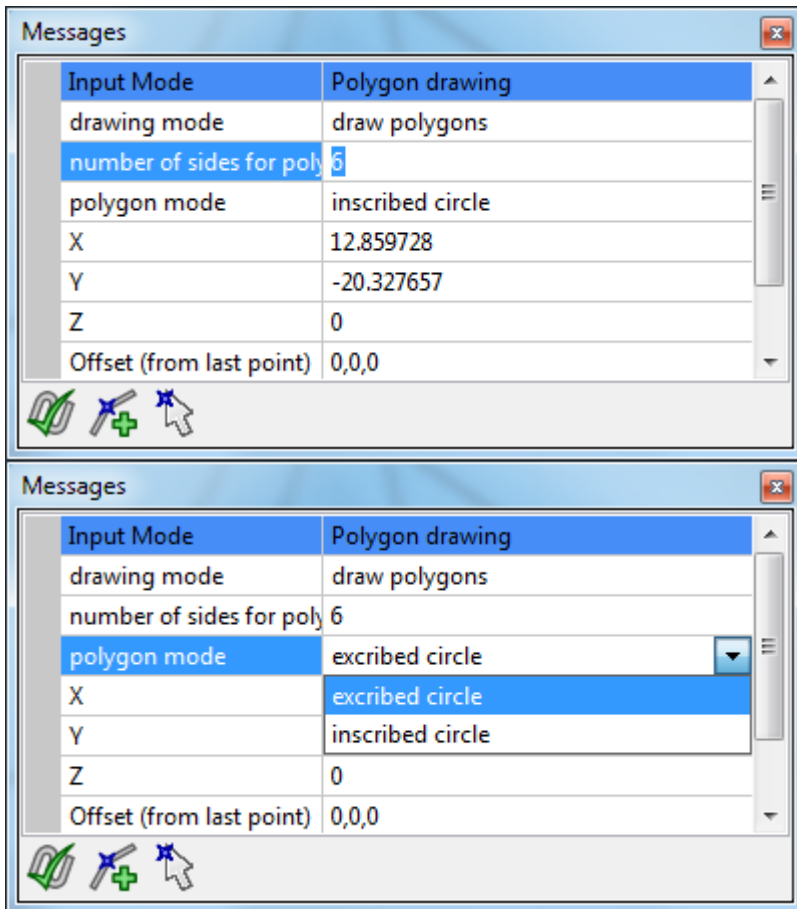


Use the mouse to position the cursor in the graphics area to draw the Slot. The Slot is created by selecting the centre points at the ends of the Slot and can be entered by clicking the X, Y or Z inputs.

Polygons

This will draw Polygons and create them as Sketches that can be used for the Profile or Pocket machining commands

Use the Messages box to set the number of sides, and also whether the Polygon should be created by using an excribed or inscribed mode.

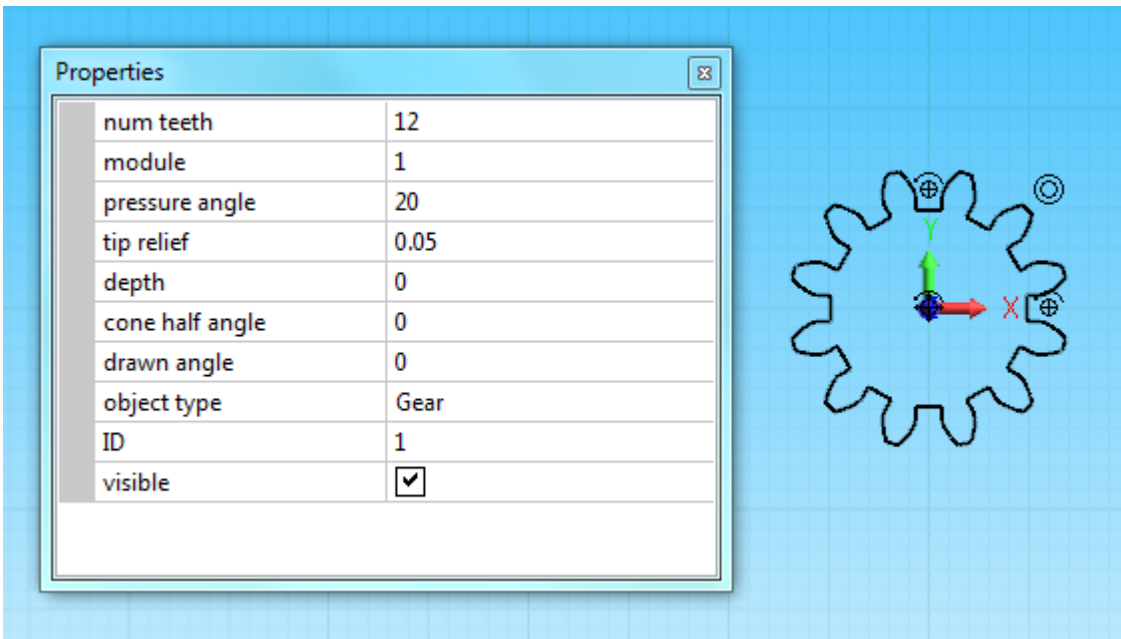


Use the mouse to position the cursor in the graphics area to draw the Polygon. The Polygon is created by selecting the centre point and the radius of the circle, the Polygon can be entered by clicking the X, Y or Z inputs.

Gear

This will produce an Involute gear form that can be used to create a Sketch for the Profile and Pocketing commands.

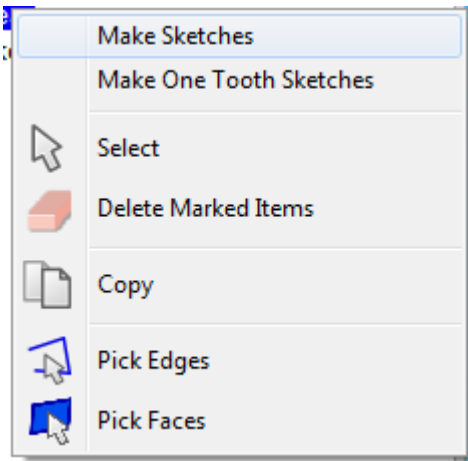
Use the Properties window to enter the parameters for the Gear



It is only possible to produce gears when the module is known, if you need to produce gears given the diametral pitch etc. Use this table or consult an expert.

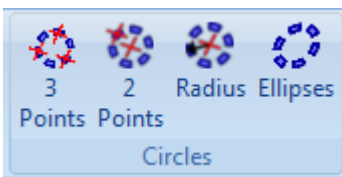
TO FIND:-	MODULE	IMPERIAL
PCD	= No. of TEETH x MOD (mm)	= $\frac{\text{No. of TEETH}}{\text{DP}}$ (ins)
ØD	= (No. of TEETH + 2) x MOD (mm)	= $\frac{\text{No. of TEETH} + 2}{\text{DP}}$ (ins)
DP	= $\frac{25.4}{\text{MODULE}}$	= $\frac{\pi}{\text{CP}}$
MODULE	= $\frac{\text{mmCP}}{\pi}$	= $\frac{24.5}{\text{DP}}$
No. of TEETH	= $\text{PCD (m)} \div \text{MODULE}$	= $\text{PCD} \times \text{DP}$
CP	= $\text{MODULE} \times \pi$ (mm)	= $\frac{\pi}{\text{DP}}$
APPENDUM	= MODULE	= $\frac{1}{\text{DP}}$
DEPENDUM	= 1.4 x MOD (0.25-1 MOD)	= $\frac{1.4 (100-24 \text{ DP})}{\text{DP}}$
	= 1.25 x MOD (1.25-4 MOD)	= $\frac{1.25 (20-6 \text{ DP})}{\text{DP}}$

To create a Sketch from a Gear, right click the Gear entry in the Object Tree window and choose either of the Make options.



NOTE - if you modify the Gear parameters, the Sketch that was created from it will not be updated, you will need to create a new Sketch using the above method.

Circles



Circles

Circles can drawn by the following methods.

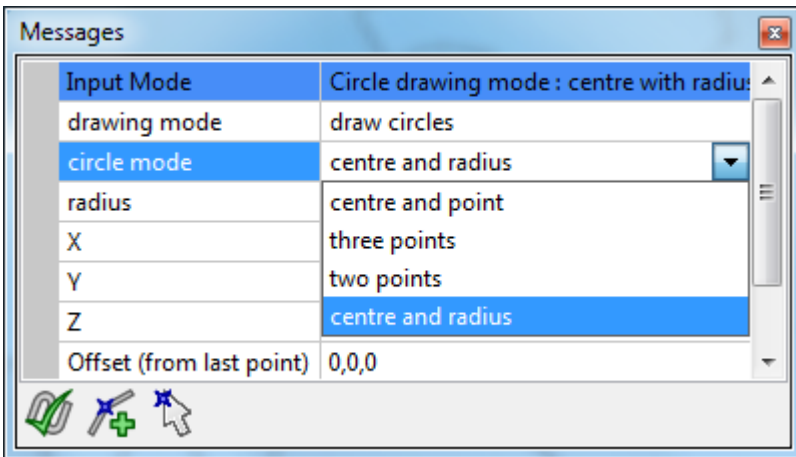
3 points on the circumference

2 points on the circumference which measured across the diameter

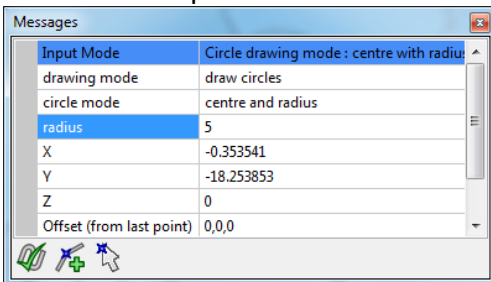
Centre point and radius

Centre point with radius entered.

To choose any of the methods, select the option from the Messages dialogue box

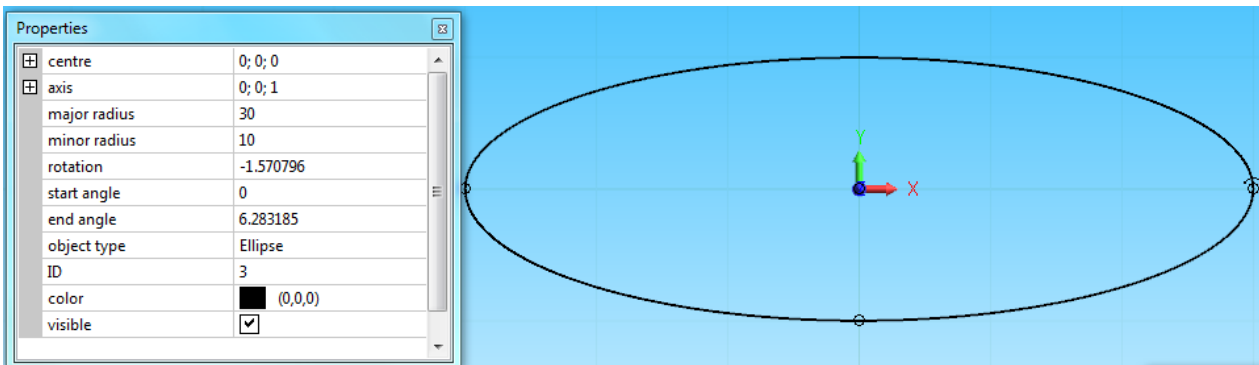


To use Centre point with radius entered method, use the Radius input window



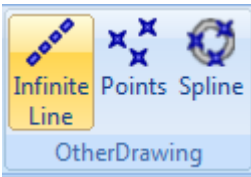
Ellipses

An ellipse is created by indicating the centre point, minor axis and major axis.



Draw the ellipse and then use the Properties box to adjust the parameters.

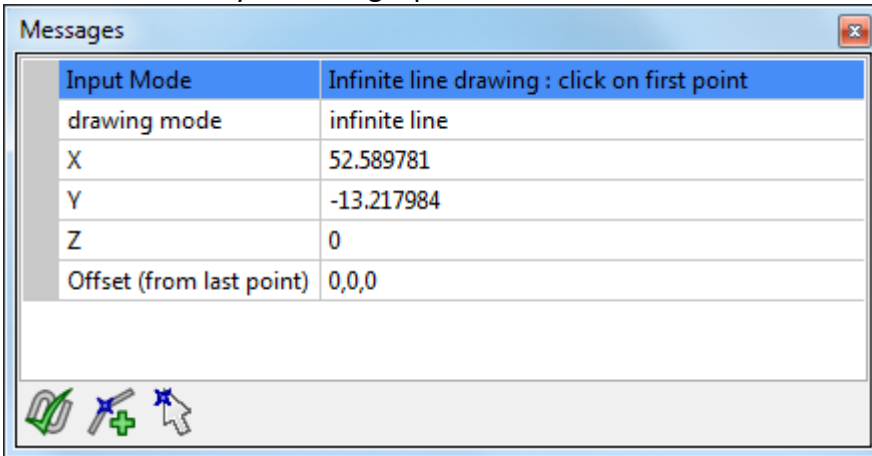
Other Drawing



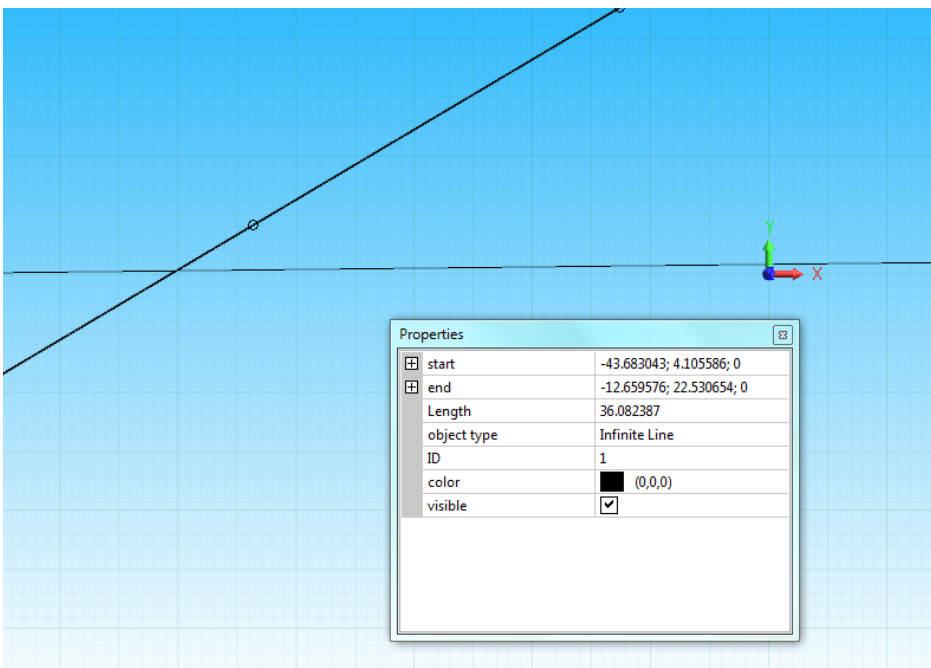
Other Drawing

Infinite Line

A line is created by indicating 2 points which the line will be drawn through.

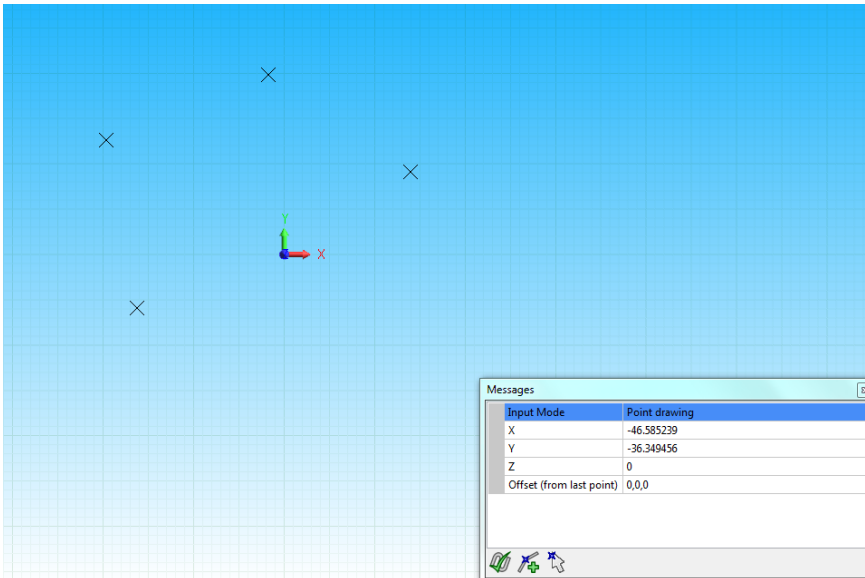


Selecting the Line will display the Properties box where the start and end points can be modified.

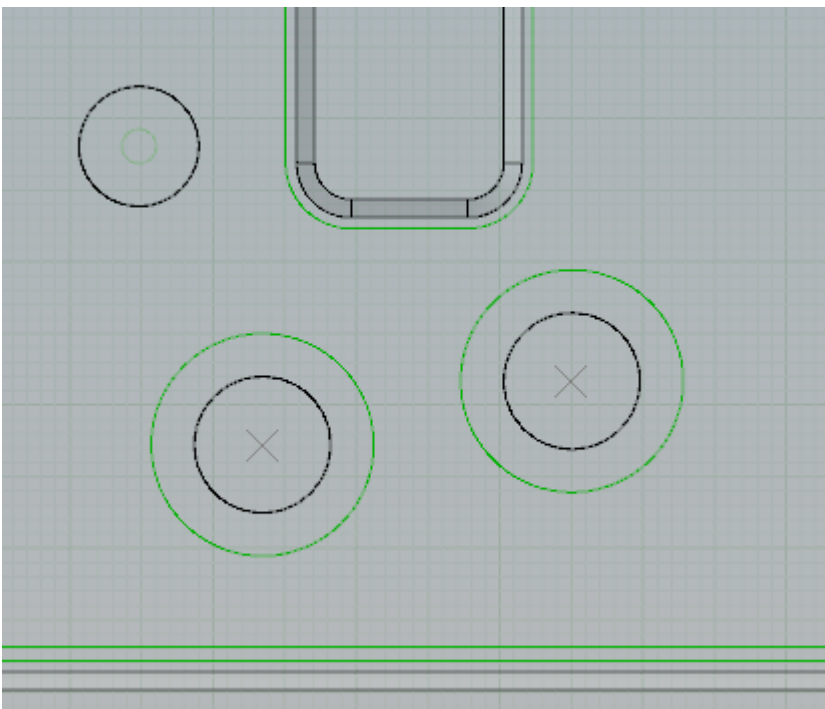


Points

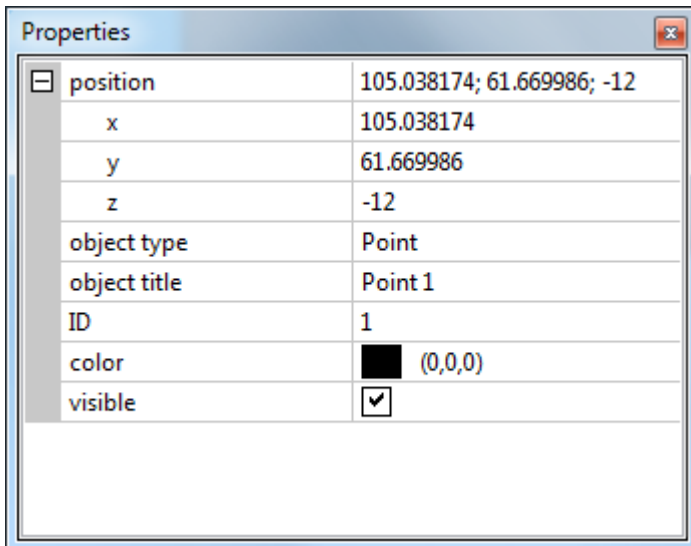
This will create a series of Points that can be used for [Drilling](#) or to fit a Spline curve through. Points do not have to be co-planar, they can have different Z depths. See Spline below.



Use the [Circle Snap](#) mode to create Points at the centre of circles for Drilling.



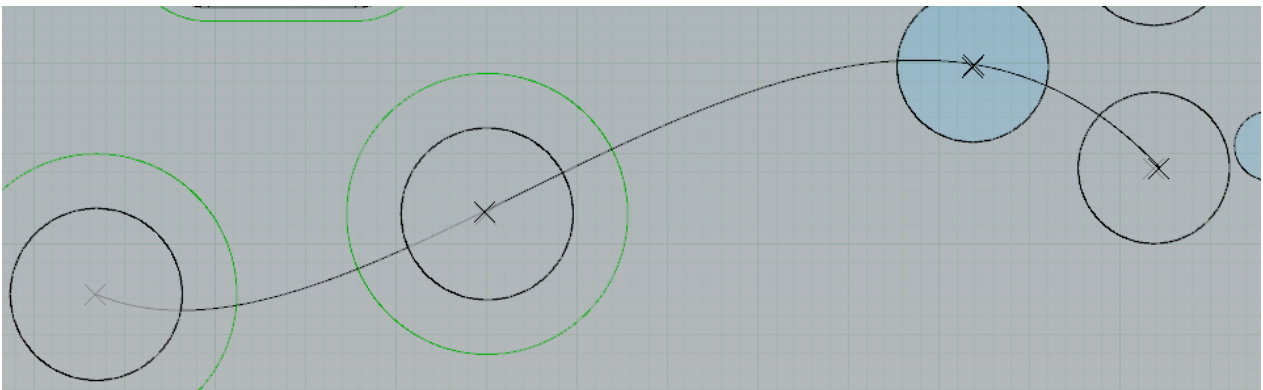
Select a Point from the [Objects Tree](#) window and then use the [Properties](#) box to modify the coordinates of a Point.



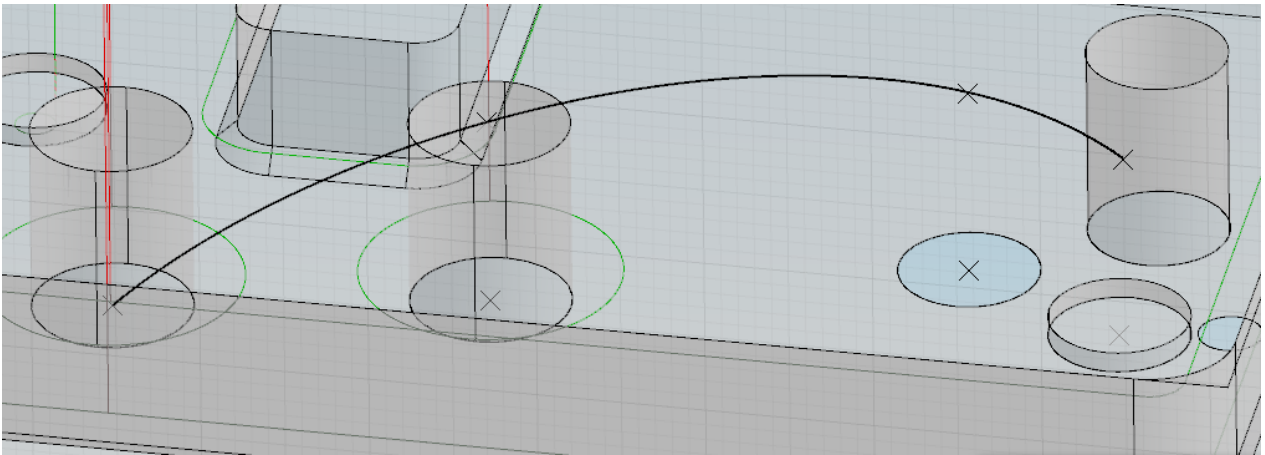
Spline

Will create a smooth series of tangential arcs through a previously created series of at least 3 Points.

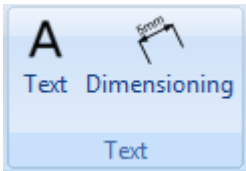
- Select the points first and then choose the Spline command.



- This is the Spline.
- As can be seen in this view, if the Points are not co-plane, a 3 dimensional Spline is created.

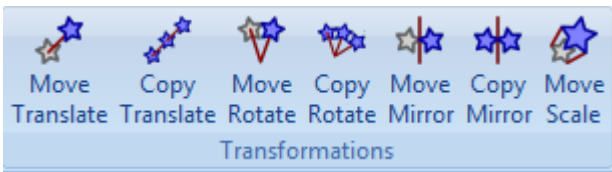


Text



Text

Transformations



Transformations

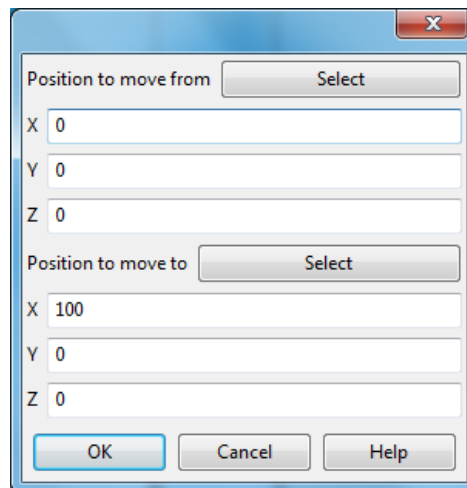
These Transformations will Translate (re-locate), Rotate, Mirror or Scale the selected elements. To **Select Items**, you must either choose them from the Graphics area by clicking the mouse over an item or by dragging a window around your selection, or by selecting them from the Objects window.

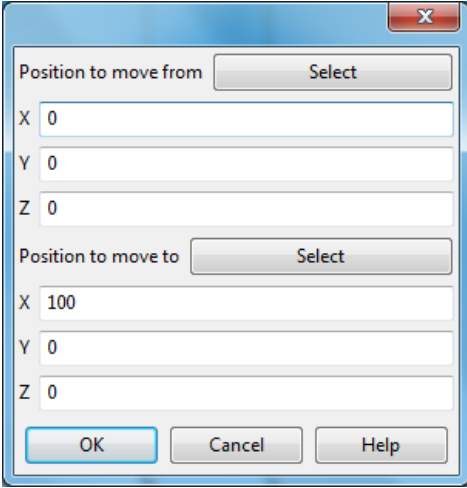
To **select more than one object**, hold down the Ctrl key and select further items.

Having selected the required items, press the Enter key to complete the selection process.

Move Translate - this will move the current selection from it's current location to a new one.

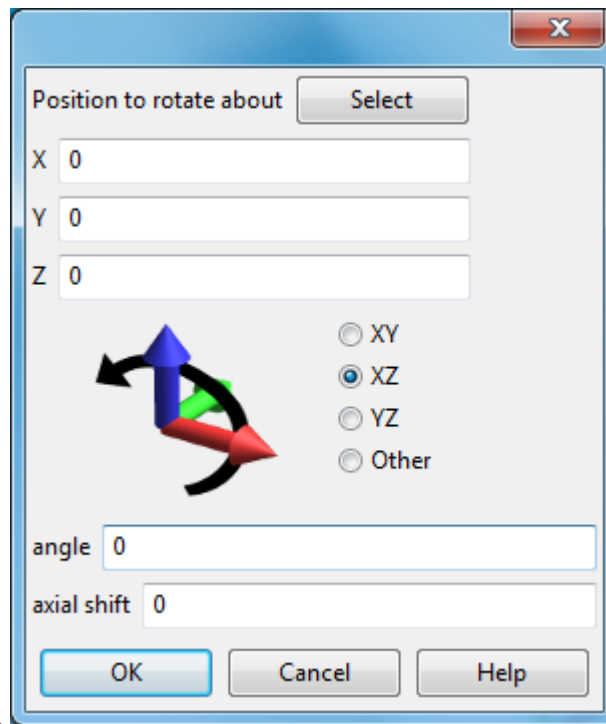
Copy Translate - this will create a copy of the item(s) and move the current selection to a new location and keep a copy in the original location.



This is the Translate dialogue box.  to move from the current location, leave the top XYZ inputs as zero and enter the new location in the **Move to** section. If you wish to Select the screen positions to move from and to - use the Select buttons.

Move Rotate - this will rotate the current selection from it's current location to a new one.

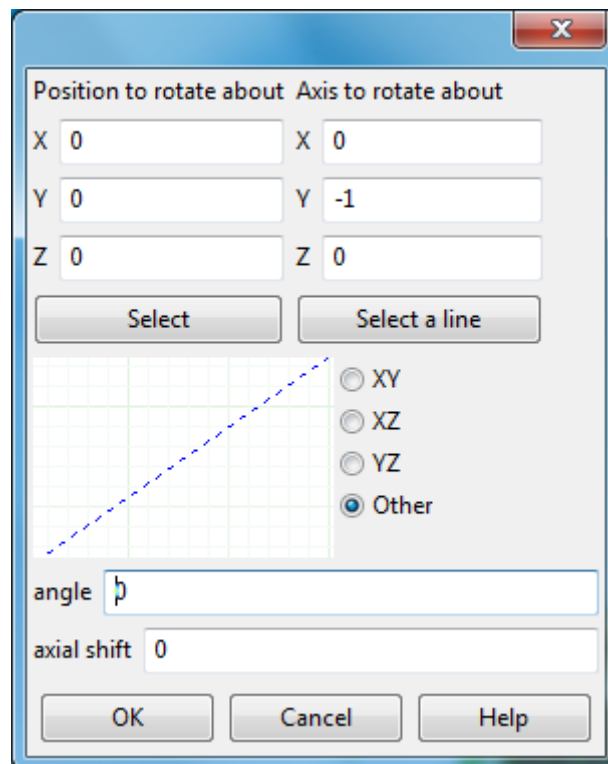
Copy Rotate - this will create a copy of the item(s) and rotate the current selection to a new location and keep a copy in the original location.



This is the Rotate dialog box. To Rotate about the current location, leave the top XYZ inputs as zero.

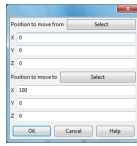
Choose the axes about which to rotate, XY, XZ or YZ. Note how the Triad graphic changes to show the axes chosen, enter the new angle in the **Angle** window.

If you wish to select an arbitrary line about to rotate, click on the **Other** radio button, the following dialogue will be shown that allows you to specify a different rotational plane.



Move Mirror - this will mirror image the current selection from it's current location to a new one.

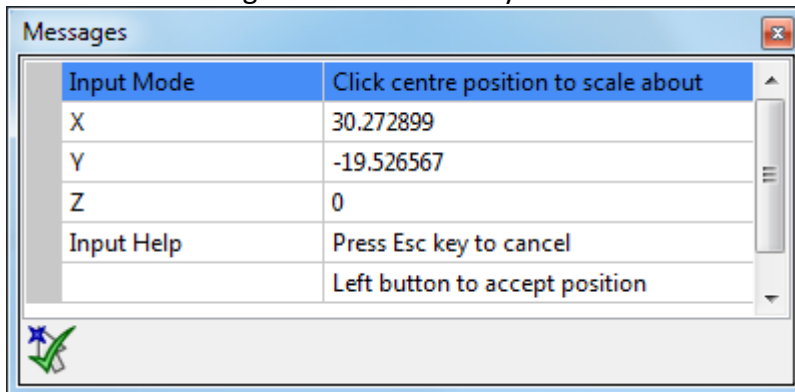
Copy Mirror - this will create a copy of the item(s) and mirror the current selection to a new location and keep a copy in the original location.



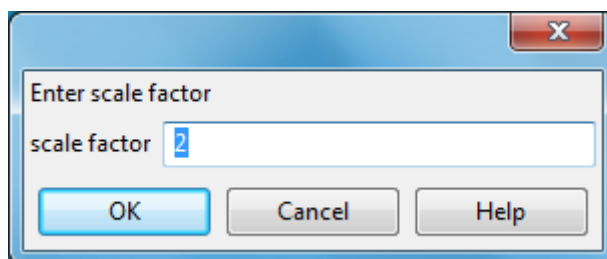
This is the Mirror dialogue box. To move from the current location, leave the top XYZ inputs as zero and enter the new location in the **Move to** section. If you wish to Select the screen positions to move from and to - use the Select buttons.

Move Scale

1. Select the object you want to Scale ([see section on Selecting Objects](#))
2. Look at the Messages box for this entry



and select the centre position to scale about




3. You will see this dialogue box. Enter the required scale factor, use an integer such as 3, 5, 10 to enlarge the scale and decimal such as 0.1 or 0.25 to reduce the scale.
4. The newly scaled elements will be shown in the graphics area, if you have used a very large scaling factor you may need to zoom out of the graphics window to see the objects.


Snapping




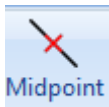
Snapping


Snap modes are used to control how a newly created piece of Geometry or Dimension will "snap" or lock onto an existing item. Clicking these icons will either switch on or off the appropriate snap mode. Multiple snap modes can be in force at any time.

Endof -  this is the end point of a Line or Arc

Inters -  this is the Intersection between 2 items

Centre -  this is the centre of an arc or circle

Midpoint -  this can the mid point of a line or arc

Grid -  this will snap to a Grid position, use the down arrow to set the Grid size

Solids

This is the Solid Ribbon Toolbar, it is split into 4 sections, Primitives, Make, Booleans & Chamfers - the options are listed below



Primitives

Make

Booleans

Chamfers

View

This is the **View** Ribbon Toolbar, it is split into 5 sections, Magnify, General Specific Views, View Dragging and Windows - the options are listed below



Magnify

Mag Extents - this will Magnify the model to it's maximum and reset the view to the nearest standard view XY, XZ or YZ.

Mag No Rotation - this will Magnify the model to it's maximum and leave the view rotation as is.

General

View Back - will return the to it's previous state after you have used the mouse wheel to zoom and/or pan the view

FullScreen - this will display just the model with no toolbars or other windows displayed. Use the Esc key to return to the normal view mode

Redraw - this will re-paint the screen

Specific Views

Choosing any of these options will rotate the model to that view

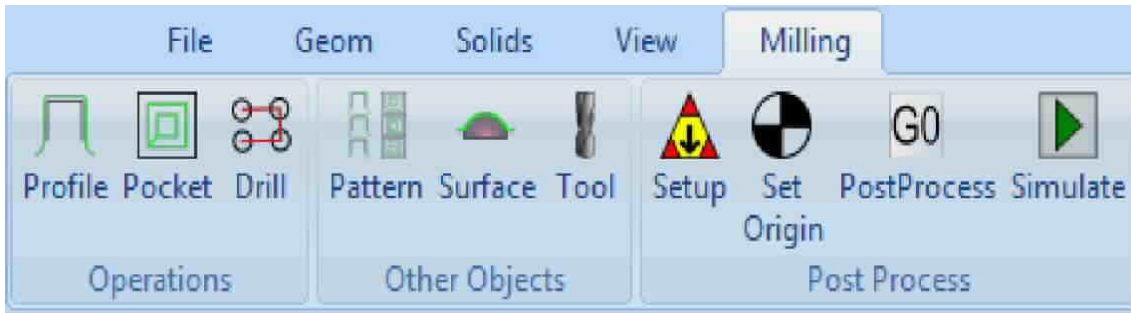
View Dragging

If you don't have mouse with a centre wheel or you are using a laptop with just a touch pad, then use these icons to control the mouse's left hand button actions.

Windows

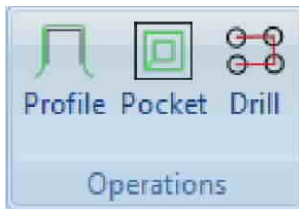
Using these options will hide or display the chosen window

Milling



This is the Milling Ribbon Toolbar, it is split into 3 sections - Operations, Other objects and Post Processing - the options are listed below

OPERATIONS

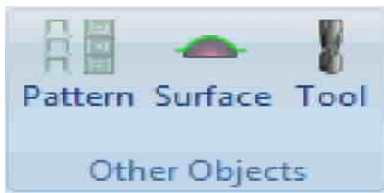


[Profile](#)

[Pocket](#)

[Drill](#)

OTHER OBJECTS



[Pattern](#)

[Surface](#)

[Tool](#)

POST PROCESS



[Setup](#)

[Set Origin](#)

[Post Process](#)

[Simulate](#)



Profile - This operation will create a 2D toolpath around a previously created Sketch, or a 3D toolpath if a [Surface](#) is also specified.

The Profile operation will be added to the [Objects Window](#)

The **Sketch** that is used can be one that has been:

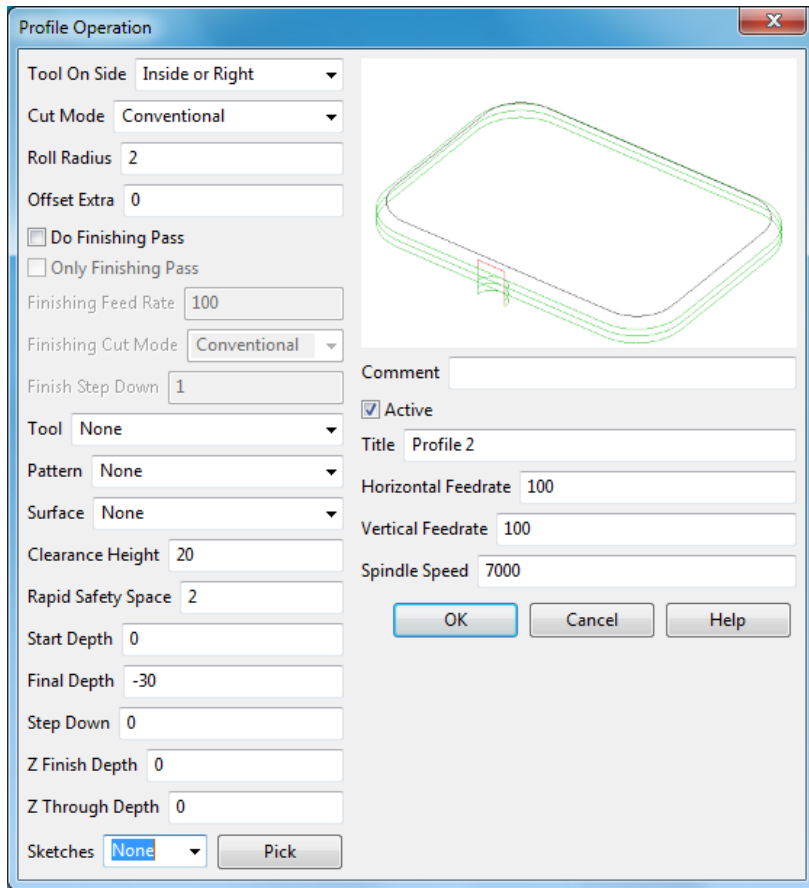
Extracted from a Solid Model

Drawn within 3DCAM using the Geometry operations

Imported as a DXF file

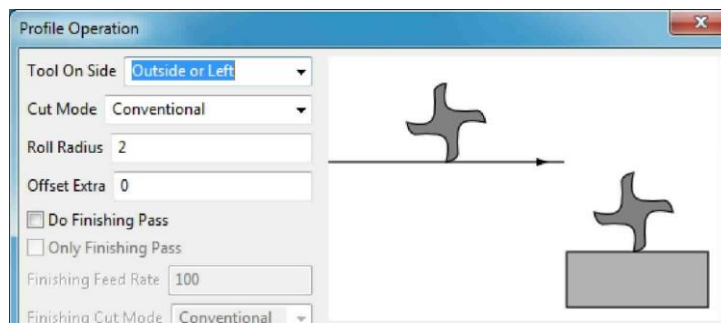
See the [Sketch](#) page for further information

This is the Profile dialogue box, the input windows are explained below.

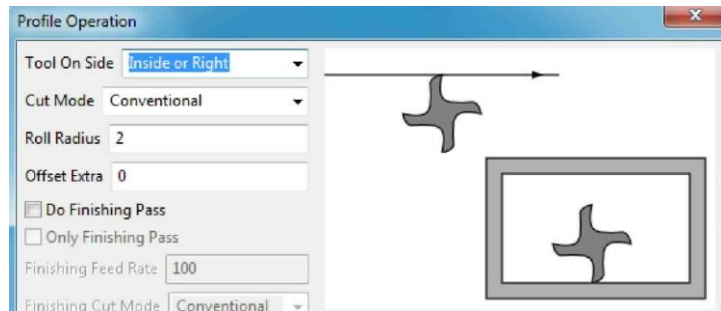


Tool On Side - this will set which side of the Profile the tool will cut on, the options are: Outside or Left, Inside or Right, On. The option chosen can be modified depending on the setting of the next option - Cut Mode - Conventional or Climb, please see below.

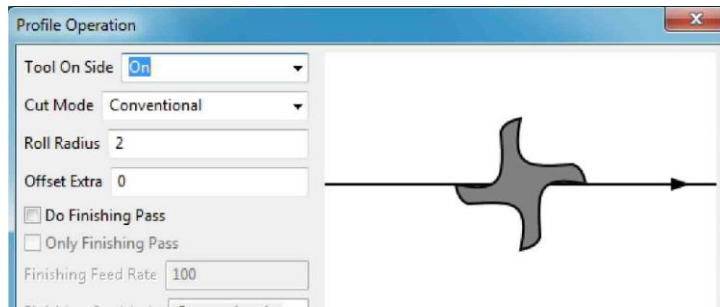
Outside or Left



Inside or Right

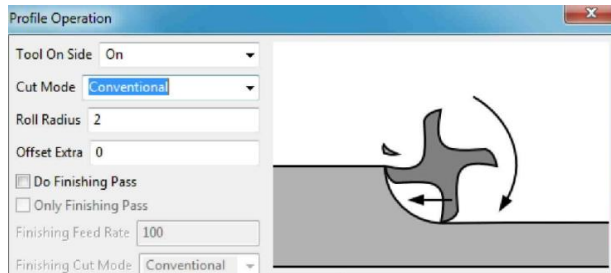


On



Cut Mode

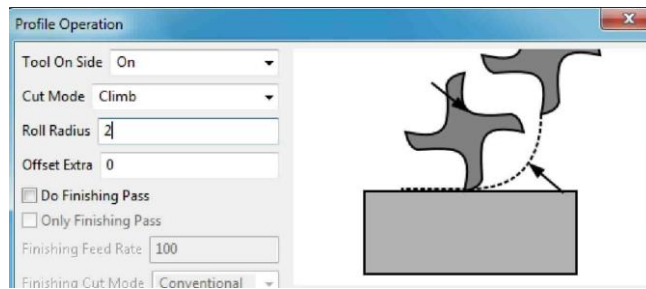
Conventional



Climb



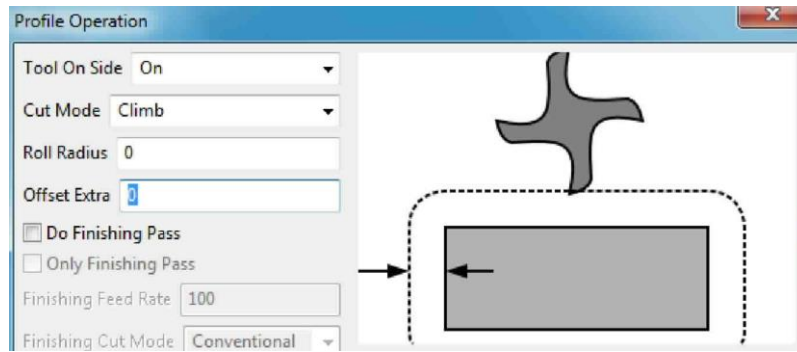
Roll Radius



This option will set the value of the radius that is applied as the tool approaches and runs off the **Profile**. This does not apply if the **Tool On Side** option is set to **On**

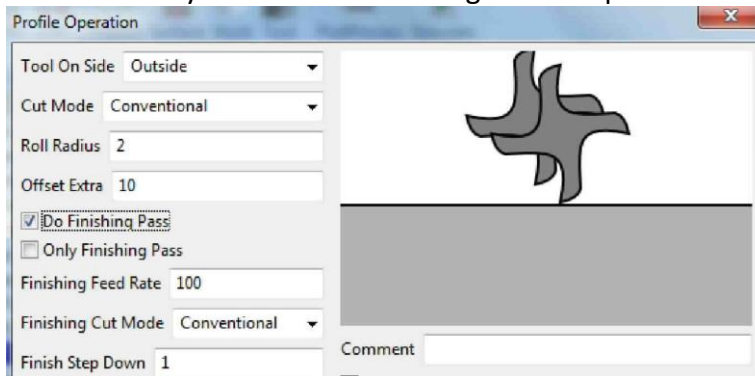
If not required, the radius should be set to zero.

Offset Extra



This option will add an allowance to either; Outside / Left or Inside / Right. If you have chosen **On** to the option **Tool On Side**, this value will be ignored.

Having set this value you have the following further options available



Do Finishing Pass - this will produce 2 toolpaths in the XY plane, the first will be offset from the Profile by the Offset Extra value, the second will be the actual Profile itself.

Only Finishing Pass - this will produce a single XY toolpath offset by the amount specified

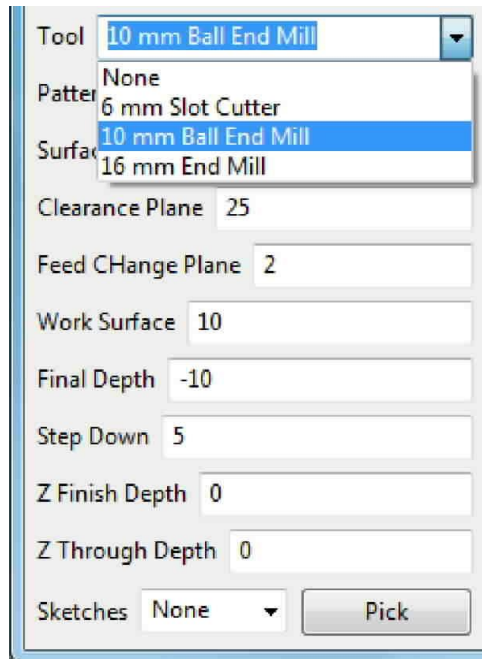
Finishing Feed rate - this will set the XY feedrate to be used on the finishing pass, as opposed the Horizontal feedrate set.

Finishing Cut Mode - This can be Conventional or Climb - please see **Cut Mode** above for more details

Finishing Step Down - The **Step Down** that has been set specified further down in this dialogue box can be overridden here. It might be that you have a Step Down set to 1mm on a total depth of 10mm - producing 10 cuts in the Z axis, but on the Finishing pass you need just a single cut of 10mm - in this case enter 10 here.

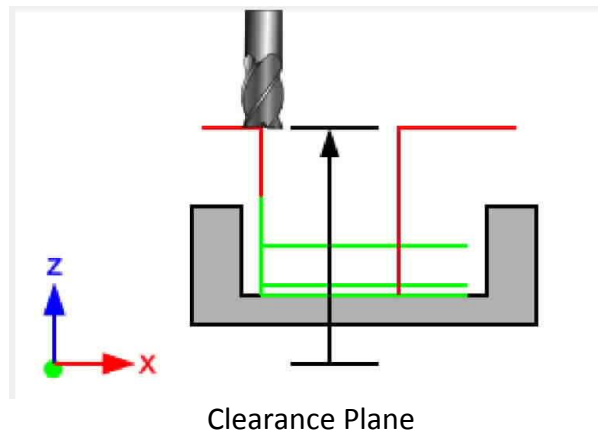
Tool - This will select the tool for this operation, use the Down Arrow to select a tool from those previously defined.

Use the [Tool icon from the main Milling ribbon](#) to define new tools

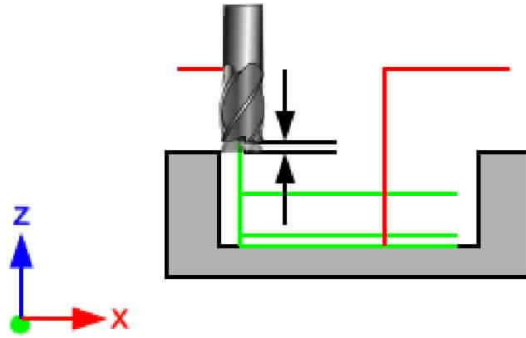


Pattern - This will use a previously created [Pattern](#) - use the [Pattern](#) icon from the Milling ribbon to create a new Pattern to machine multiple instances of this operation

Surface - Using this option will force the toolpath to be controlled by the shape of the [Surface](#). If you use just a 2D Sketch without specifying a Surface you will create a 2D toolpath. Using a Surface will "drape" the 2D Sketch over the [Surface creating a 3D toolpath](#)

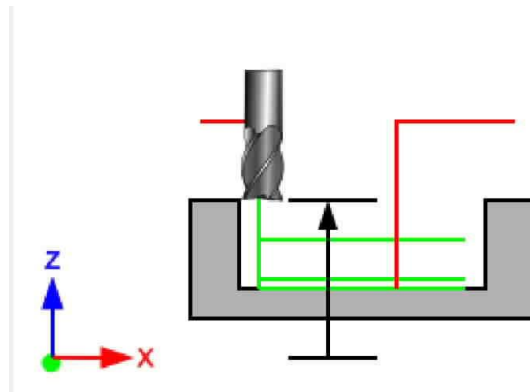


Clearance Plane - This is a Z value that must be clear of all obstructions and fixtures. It is the plane used to Rapid from one XY feature or position to the next XY position. This is an absolute value measured from Z zero - please see diagram above.



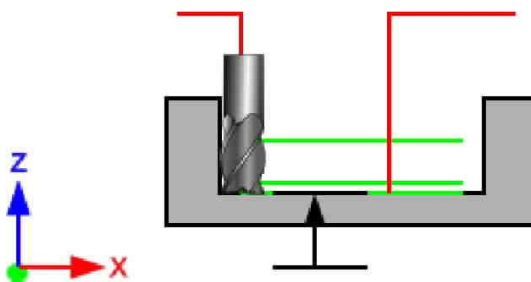
Feed Change Plane

Feed Change Plane - This is the Z plane where the tool will change from Rapid to the specified Feedrate while descending to the top of the workpiece. The distance is an incremental value based on the **Worksurface** of the part - please see diagram above.



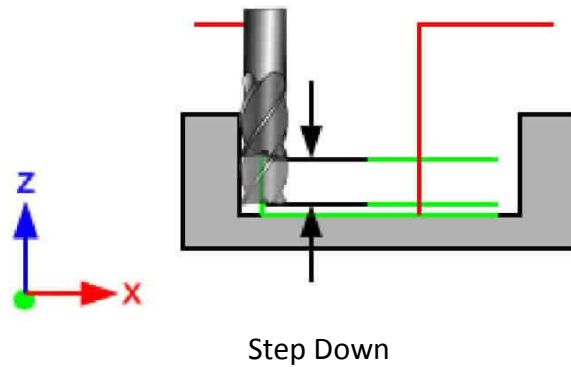
Work Surface

Work Surface - This specifies the topmost part of the workpiece. It is an absolute value measured from Z zero - please see diagram above. If Z zero is set to the top of the part this value will be zero. See [Set Origin](#)

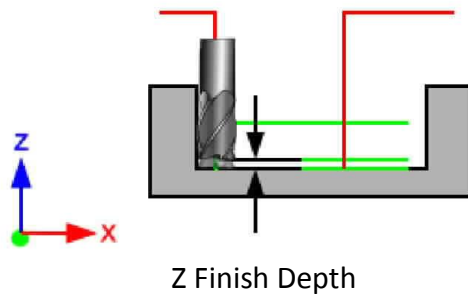


Final Depth

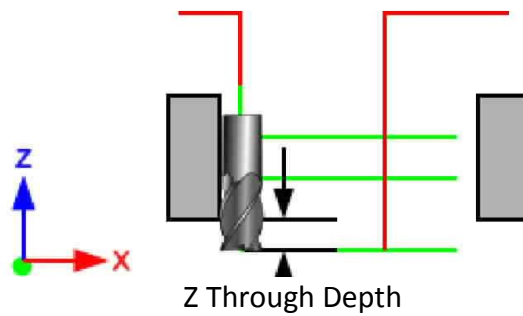
Final Depth - This is the maximum depth that you wish to cut in the Z axis, it is an absolute value measured from Z zero and will be a negative value if Z zero is set to the top of the part. See [Set Origin](#)



Step Down - This controls the Z distance of each pass in the Z axis. The size entered is the maximum amount of each cut per pass. This value maybe adjusted (reduced) if the value entered is not divisible into the total depth of cut. If you don't need to use this feature set the value to zero.

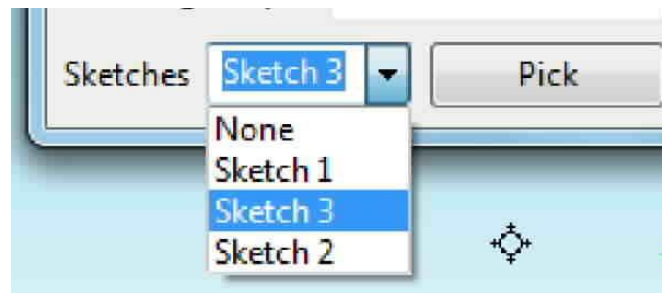


Z Finish Depth - This value is used to create an extra finishing pass, it will be machined with the current tool and with all the current feedrates in force. It will also be used after any passes created by use of the Step Down value entered above.



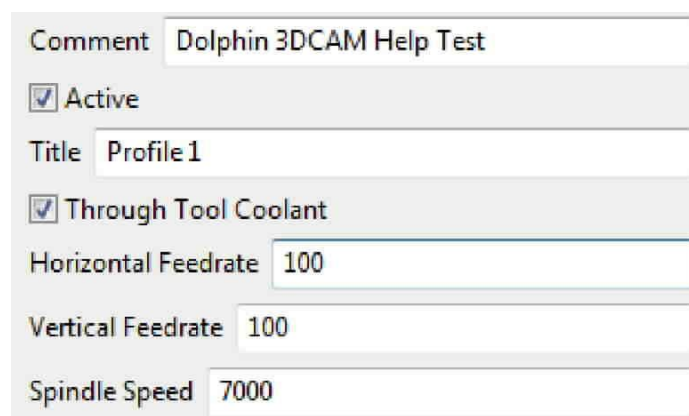
Z Through Depth - This will create an extra pass in the Z axis which will be added after the last cut set by the **Final Depth**.

The **Z Finish Depth** will be applied to **this** dimension, not the **Final Depth**.



Choosing a Sketch

Sketch - The Profile command must be based on a previously created **Sketch**. You can choose a **Sketch** from the pull down list or select a **Sketch** from the [Objects Tree](#) window. Please also see the page on creating [Sketches](#)



Setting the Spindle Speed and Feedrates

Comment - This will output a comment to the G code file to allow you to identify different parts of the program

Active - Selecting this box will make the current operation active, it will be calculated and will produce G codes

Title - This will appear in the Objects Tree window to identify this operation

Though Tool Coolant - Will switch on the appropriate command within the Post processor and output the required G or M code

Horizontal Feedrate - This will apply to all XY feed moves

Vertical Feedrate - This will control the feedrate in the Z axis when the tool is plunging into the workpiece.

Spindle Speed - This will set the spindle RPM speed for the current tool, it will output the required instructions to the post processor to switch on the spindle (normally a M6 command) and set the spindle RPM, typically an " S " word - S1234 for instance

Pocket



Pocket - This operation will create a 2D Area Clearance toolpath around a previously created Sketch, or a 3D toolpath if a [Surface](#) is also specified
The Pocket operation will be added to the [Objects Window](#)

The **Sketch** that is used can be one that has been:

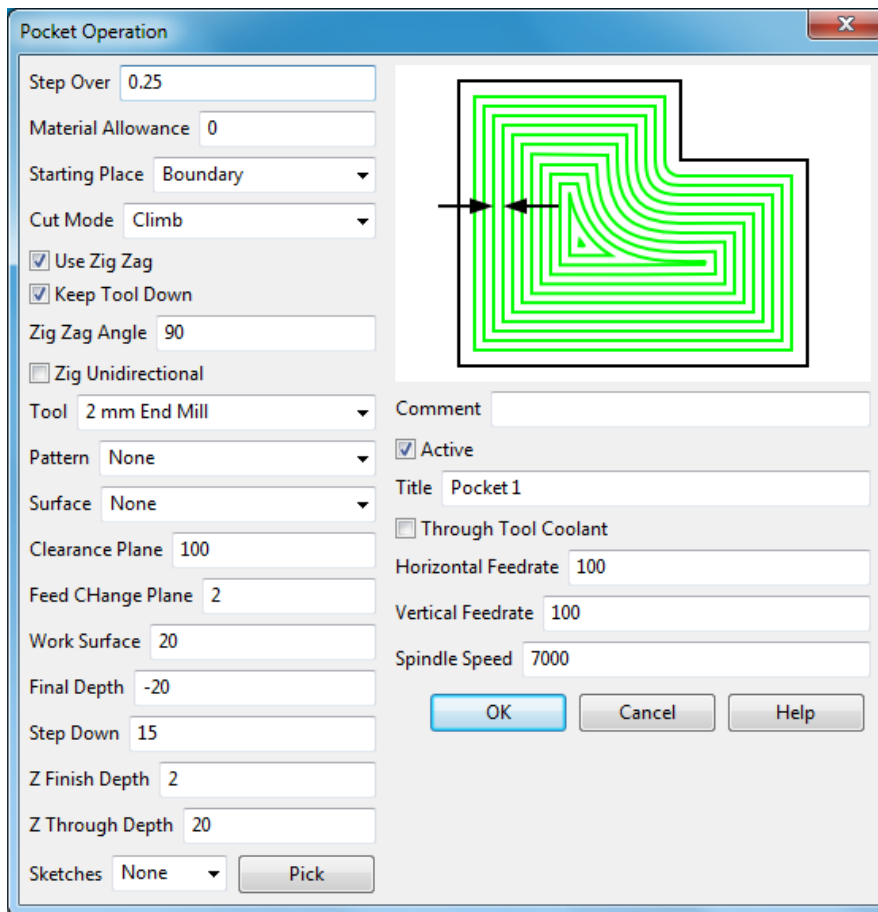
Extracted from a Solid Model

Drawn within 3DCAM using the Geometry operations

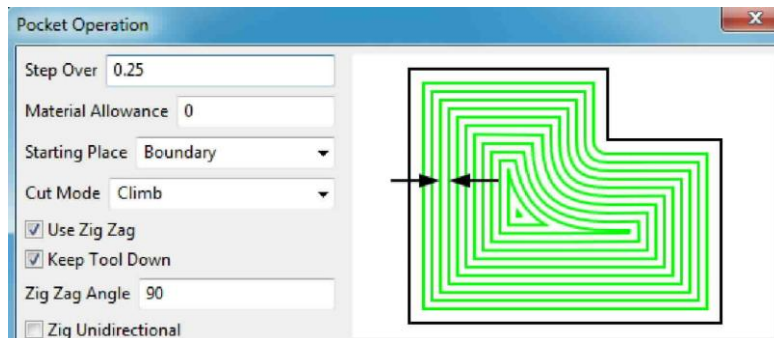
Imported as a DXF file

See the [Sketch](#) page for further information

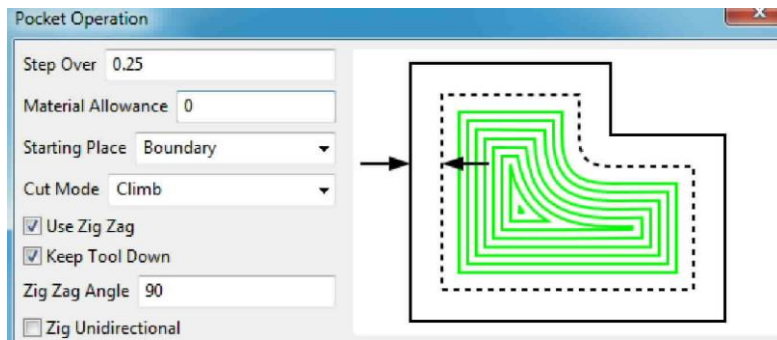
This is the Pocket dialogue box, the input windows are explained below.



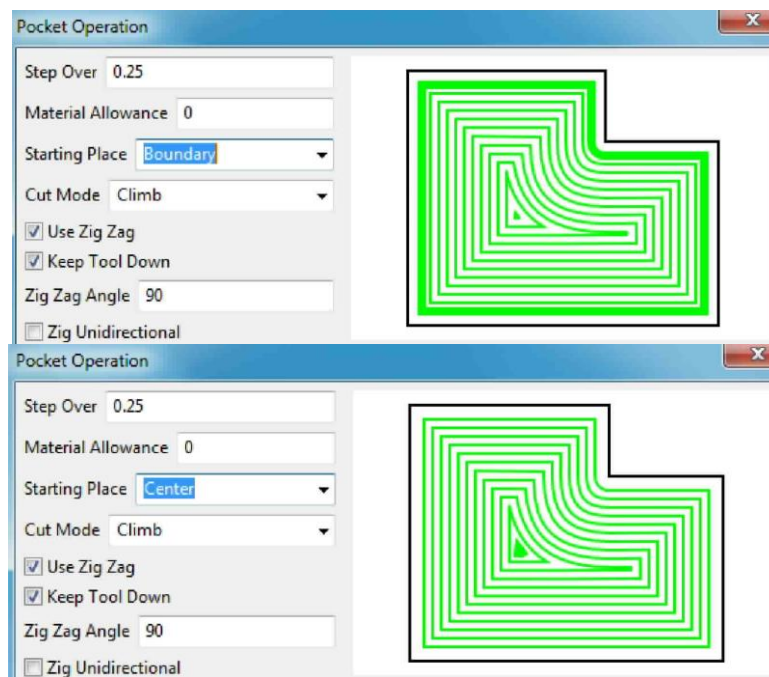
Step Over - This value controls the distance between each pass in the XY plane - it is expressed as a absolute value in the current units



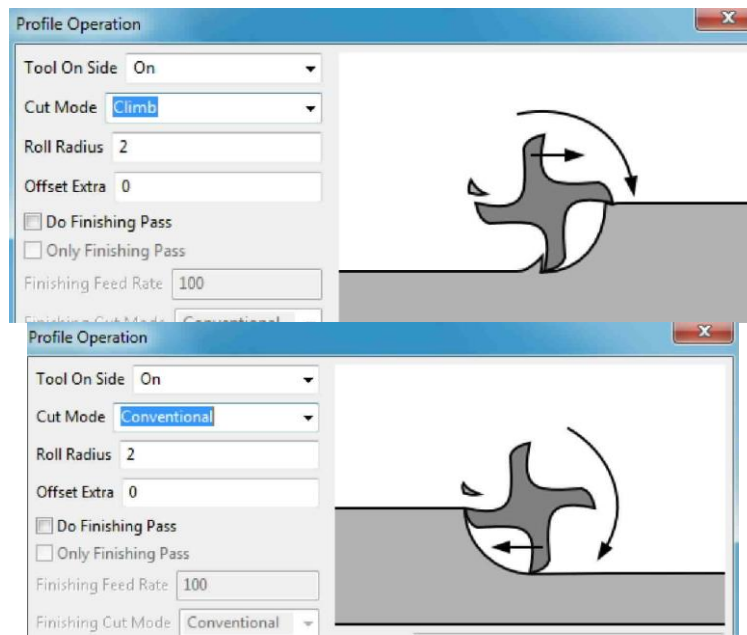
Material Allowance - This value will leave material around the inside of the Sketch specified - see Sketch below. This material can then be removed by use of the [Profile](#) command



Starting place - Use the pull down arrow to choose either Boundary or Centre for the start of machining



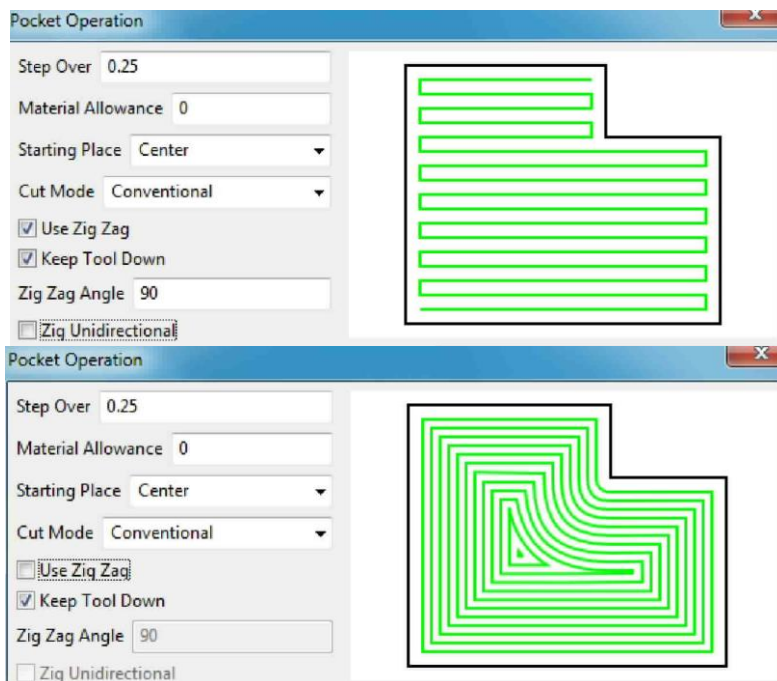
Cut Mode - Use the pull down arrow to select either Climb or Conventional machining. In most cases Climb milling is preferred.



Climb

Conventional

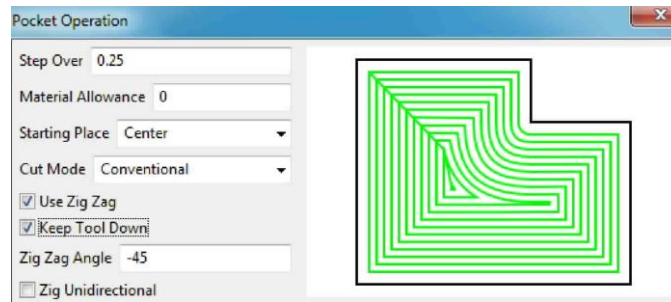
Use Zig Zag - Selecting this option will create a **ZigZag** toolpath with the tool moving backwards and forwards over the shape, un-checking this box will create a concentric toolpath.



Zig Zag selected

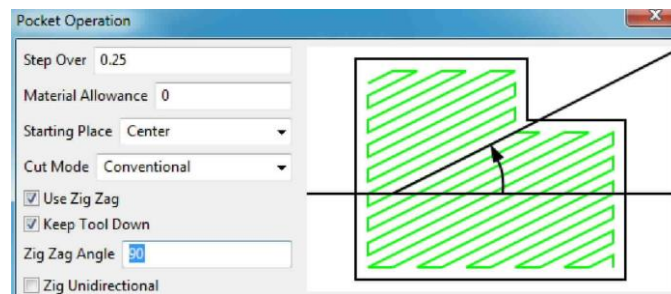
Zig Zag not selected

Keep Tool Down - Using this check box will force the tool to stay at the current Z position as it moves in XY between one cutting pass and the next. If not checked the tool will retract at rapid to the Feed Change Plane in Z, rapid move to the next XY position and then descend to the next Z cutting position at the current feedrate.

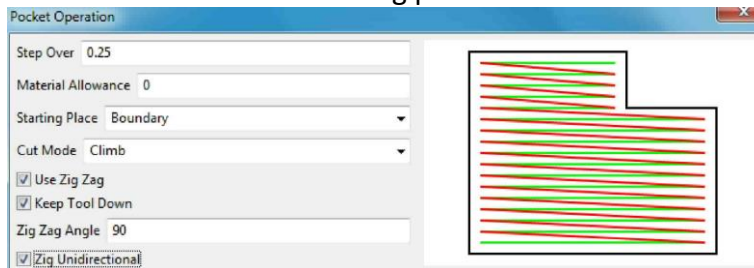


Zig Zag Angle - Use this to set the angle of the Zig Zag toolpath. The angle is measured from zero (3.0 o'clock) and can be positive or negative.

If the Zig Zag option is not selected, this input window will not be active.

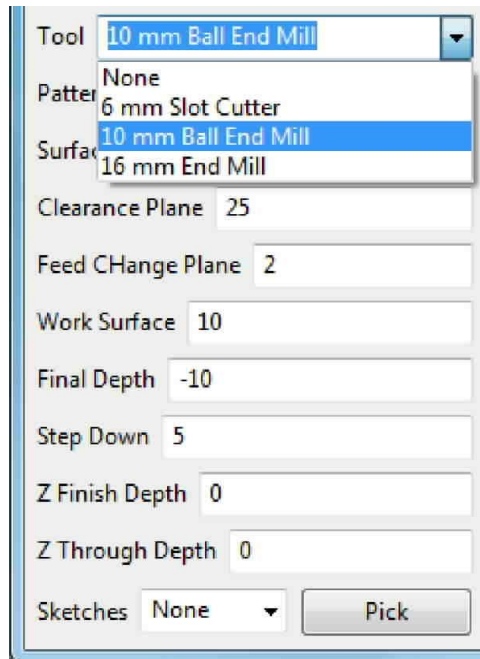


Zig Zag Unidirectional - Checking this box will force machining to cut in single direction set by the Zig Zag angle. The tool will retract at rapid to the Feed Change Plane in Z, rapid move to the next XY position and then descend to the next Z cutting position at the current feedrate.



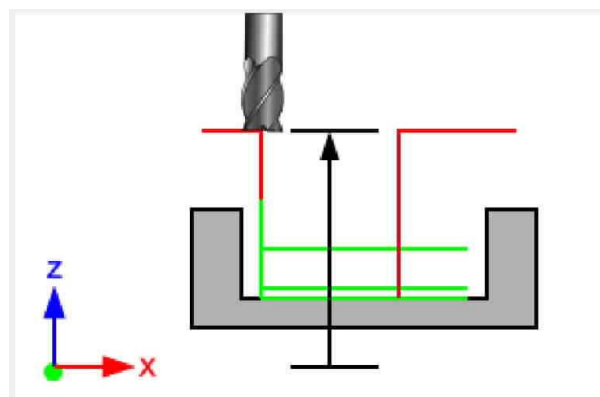
Tool - This will select the tool for this operation, use the Down Arrow to select a tool from those previously defined.

Use the [Tool icon from the main Milling ribbon](#) to define new tools



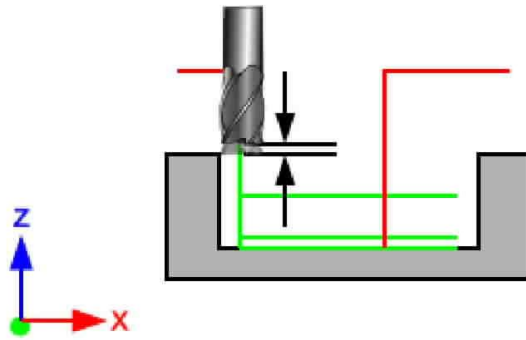
Pattern - This will use a previously created [Pattern](#) - use the [Pattern](#) icon from the Milling ribbon to create a new Pattern to machine multiple instances of this operation

Surface - Using this option will force the toolpath to be controlled by the shape of the [Surface](#). If you use just a 2D Sketch without specifying a Surface you will create a 2D toolpath. Using a Surface will "drape" the 2D Sketch over the [Surface creating a 3D toolpath](#)



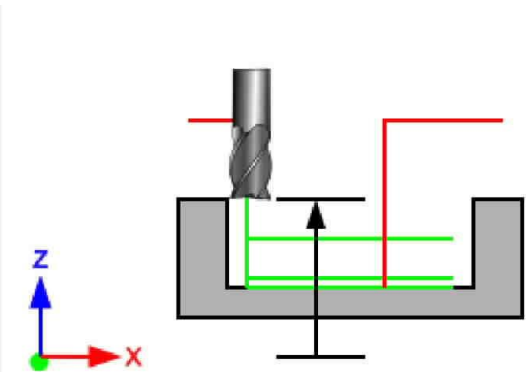
Clearance Plane

Clearance Plane - This is a Z value that must be clear of all obstructions and fixtures. It is the plane used to Rapid from one XY feature or position to the next XY position. This is an absolute value measured from Z zero - please see diagram above.



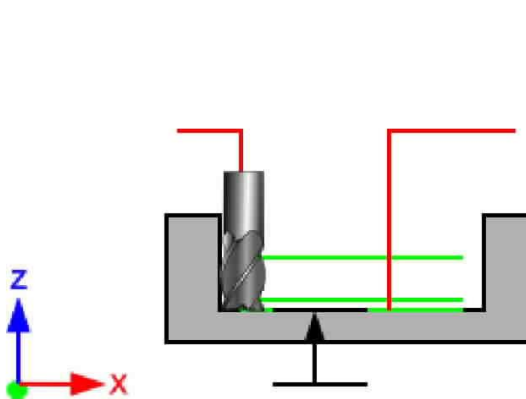
Feed Change Plane

Feed Change Plane - This is the Z plane where the tool will change from Rapid to the specified Feedrate while descending to the top of the workpiece. The distance is an incremental value based on the **Worksurface** of the part - please see diagram above.



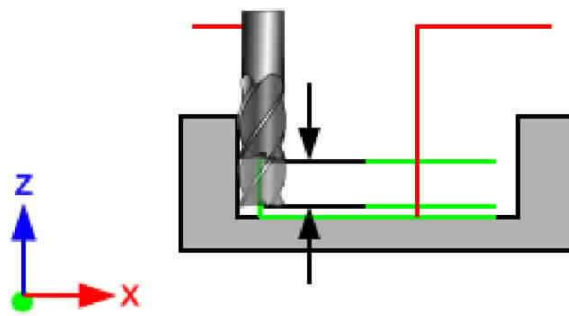
Work Surface

Work Surface - This specifies the topmost part of the workpiece. It is an absolute value measured from Z zero - please see diagram above. If Z zero is set to the top of the part this value will be zero. See [Set Origin](#)



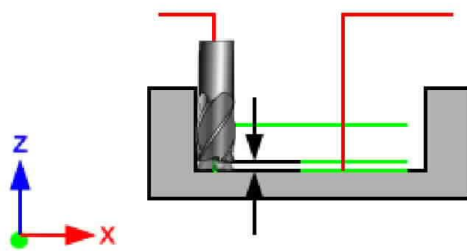
Final Depth

Final Depth - This is the maximum depth that you wish to cut in the Z axis, it is an absolute value measured from Z zero and will be a negative value if Z zero is set to the top of the part. See [Set Origin](#)



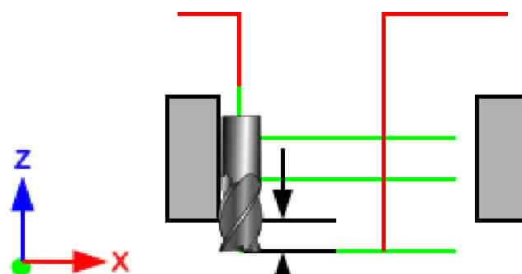
Step Down

Step Down - This controls the Z distance of each pass in the Z axis. The size entered is the maximum amount of each cut per pass. This value may be adjusted (reduced) if the value entered is not divisible into the total depth of cut. If you don't need to use this feature set the value to zero.



Z Finish Depth

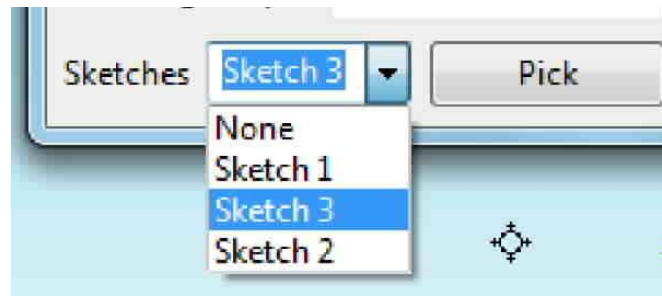
Z Finish Depth - This value is used to create an extra finishing pass, it will be machined with the current tool and with all the current feedrates in force. It will also be used after any passes created by use of the Step Down value entered above.



Z Through Depth

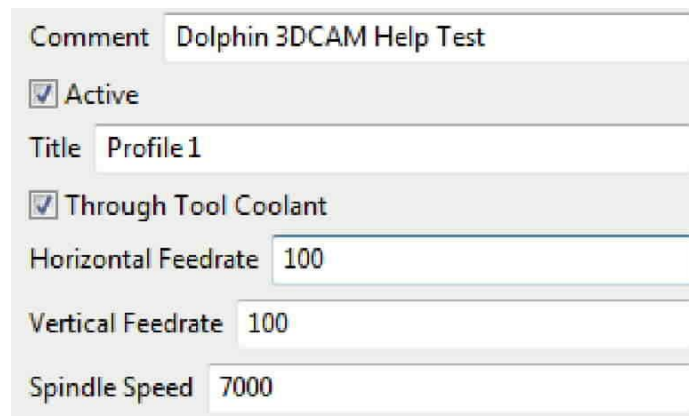
Z Through Depth - This will create an extra pass in the Z axis which will be added after the last cut set by the **Final Depth**.

The **Z Finish Depth** will be applied to **this** dimension, not the **Final Depth**.



Choosing a Sketch

Sketch - The Profile command must be based on a previously created **Sketch**. You can choose a **Sketch** from the pull down list or select a **Sketch** from the [Objects Tree](#) window. Please also see the page on creating [Sketches](#)



Setting the Spindle Speed and Feedrates

Comment - This will output a comment to the G code file to allow you to identify different parts of the program

Active - Selecting this box will make the current operation active, it will be calculated and will produce G codes

Title - This will appear in the Objects Tree window to identify this operation

Though Tool Coolant - Will switch on the appropriate command within the Post processor and output the required G or M code

Horizontal Feedrate - This will apply to all XY feed moves

Vertical Feedrate - This will control the feedrate in the Z axis when the tool is plunging into the workpiece.

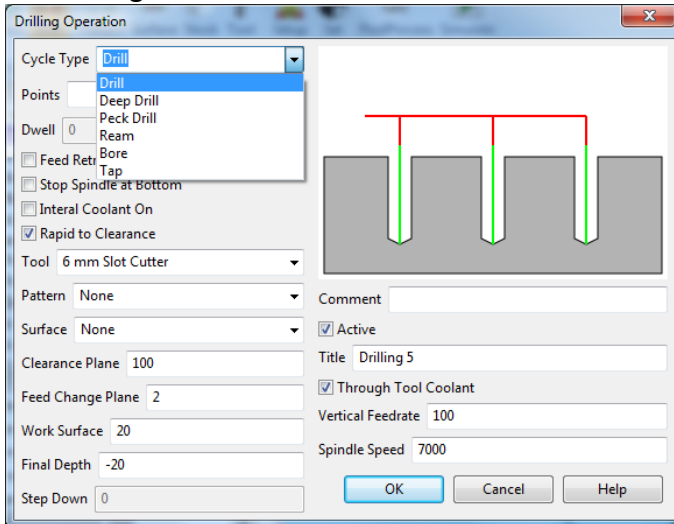
Spindle Speed - This will set the spindle RPM speed for the current tool, it will output the required instructions to the post processor to switch on the spindle (normally a M6 command) and set the spindle RPM, typically an " S " word - S1234 for instance

Drill



DRILL

This command will drill holes at previously created XY locations using a Drill type tool. This is the Drill dialogue box.



The available cycles are:-

Drill - Feed to depth, retract at rapid. - Can be used with tools defined as either MILL or DRILL.

The normal drill cycle moves the tool in XY to the drill position the tool then sinks at rapid feedrate to the **Feed Change Plane**. It then sinks at feedrate to the programmed depth of the hole and then retracts at rapid feedrate to either the **Feed Change Plane** or the **Clearance Plane** depending upon which option is chosen. Optionally you can program a **Dwell** at the bottom of the cycle.

Deep Drill - Drill hole in a series of pecks, retracting tool completely between each peck.

This cycle is normally used when drilling deep holes and is intended to provide swarf clearance as the drilling progresses. The tool is sunk in a series of pecks, the peck depth being calculated by PartMaster by dividing the total depth by the number of pecks, retracting completely clear of the job after each peck. The deep drill cycle moves the tool in XY to the drill position the tool then sinks at rapid feedrate to the **Feed Change Plane**. It then sinks at feedrate to the first peck depth. It then retracts at rapid feedrate to the **Feed Change Plane** before sinking at feedrate to the next peck depth. Optionally the feed down to the next peck can be made at **Rapid speed**, in which case it will switch back to feedrate a short distance before the previous peck depth is reached. After it has reached the final depth it then retracts at rapid feedrate to either the **Feed Change Plane** or the **Clearance Plane** depending upon which option is chosen.

Peck Drill - Drill hole in a series of pecks, retracting tool slightly between each peck. The Step Down is the Peck depth.

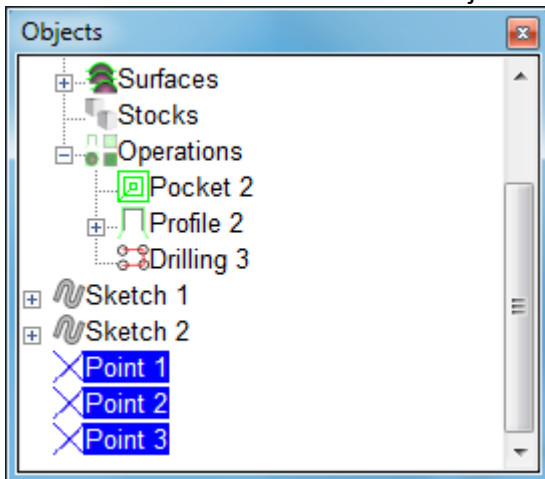
This cycle is a halfway house between a normal drill cycle and a deep drill cycle. it is normally used when drilling deep holes and is intended to provide a degree of swarf clearance as the drilling progresses. The tool is sunk in a series of pecks, the peck depth being calculated by PartMaster by dividing the total depth by the number of pecks. The tool is retracted after each peck, but by a standard amount rather than retracting completely clear of the job. The deep drill cycle moves the tool in XY to the drill position the tool then sinks at rapid feedrate to the **Feed Change Plane**. It then sinks at feedrate to the first peck depth. It then retracts at rapid feedrate by a standard amount before sinking at feedrate to the next peck depth. After it has reached the final depth it then retracts at rapid feedrate to either the **Feed Change Plane** or the **Clearance Plane** depending upon which option is chosen.

Ream - Feed to depth, retract at feedrate. Can only be used with REAMer

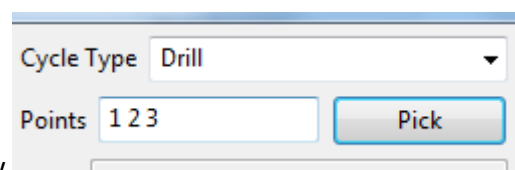
Bore - Feed to depth, spindle stop, retract at feedrate, spindle start. Can only be used with BORE tools

Tap - Feed to depth, reverse spindle, retract at feedrate, reverse spindle. Can only be used with TAPs

Points - Use this entry to select which previously defined points you wish to drill. Use the **PICK button** to select Points from the Objects window



Select a single point or multiple points by holding down the Ctrl key. When finished press the **Enter** key.



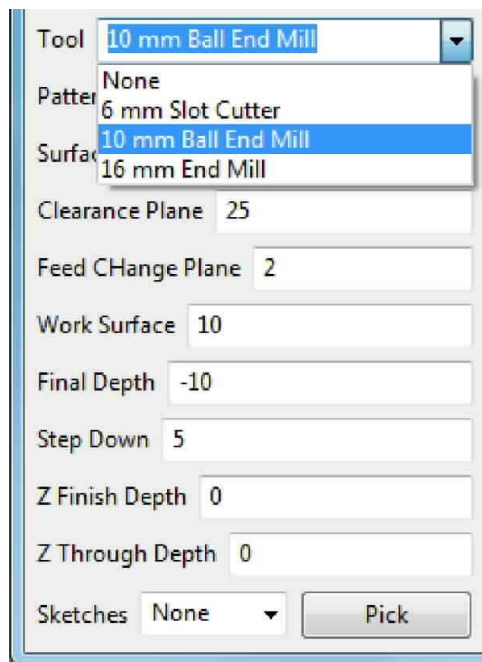
The points selected will be added to the **Points** window

To create points - please refer to the [Geometry > Points](#) page

Dwell -

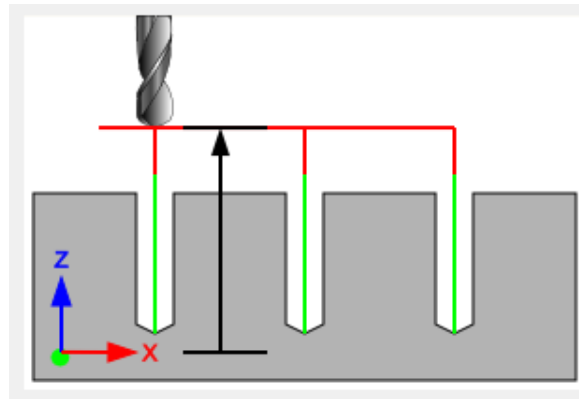
Tool - This will select the tool for this operation, use the Down Arrow to select a tool from those previously defined.

Use the [Tool icon from the main Milling ribbon](#) to define new tools



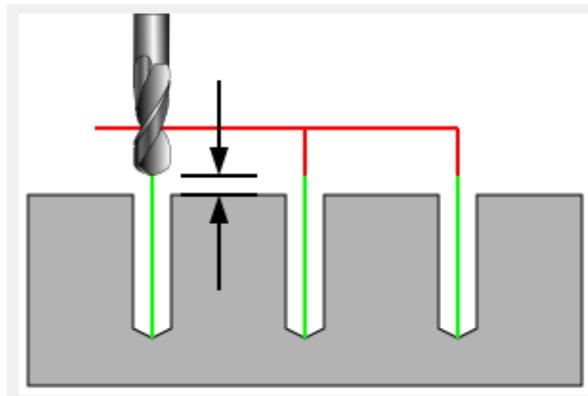
Pattern - This will use a previously created [Pattern](#) - use the [Pattern](#) icon from the Milling ribbon to create a new Pattern to machine multiple instances of this operation

Surface - Using this option will force the toolpath to be controlled by the shape of the [Surface](#). If you use just a 2D Sketch without specifying a Surface you will create a 2D toolpath. Using a Surface will "drape" the 2D Sketch over the [Surface creating a 3D toolpath](#)



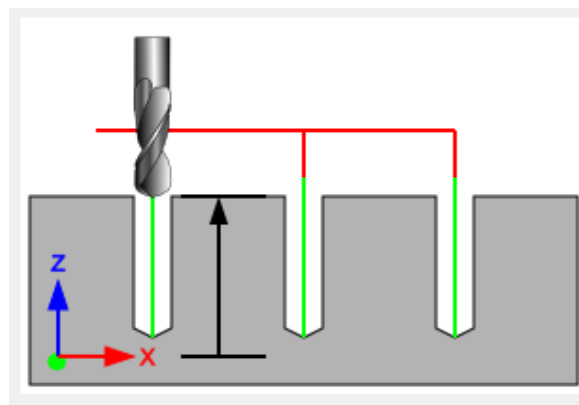
Clearance Plane

Clearance Plane - This is a Z value that must be clear of all obstructions and fixtures. It is the plane used to Rapid from one XY feature or position to the next XY position. This is an absolute value measured from Z zero - please see diagram above.



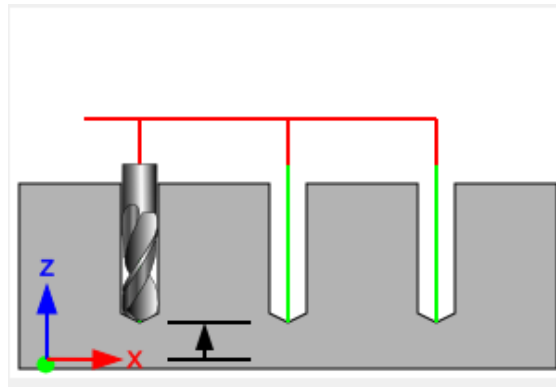
Feed Change Plane

Feed Change Plane - This is the Z plane where the tool will change from Rapid to the specified Feedrate while descending to the top of the workpiece. The distance is an incremental value based on the **Worksurface** of the part - please see diagram above.



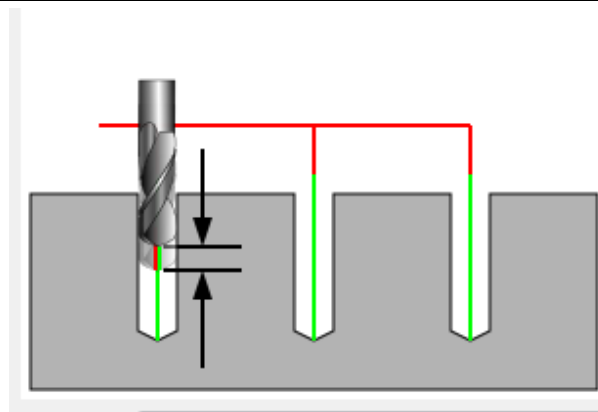
Work Surface

Work Surface - This specifies the topmost part of the workpiece. It is an absolute value measured from Z zero - please see diagram above. If Z zero is set to the top of the part this value will be zero. See [Set Origin](#)



Final Depth

Final Depth - This is the maximum depth that you wish to cut in the Z axis, it is an absolute value measured from Z zero and will be a negative value if Z zero is set to the top of the part. See [Set Origin](#)



Step Down

Step Down - This controls the Z Peck distance. This option is only available if the current cycle is Deep Drill or Peck Drill.

Comment

Active

Title

Through Tool Coolant

Vertical Feedrate

Spindle Speed

Setting the Spindle Speed and Feedrate

Comment - This will output a comment to the G code file to allow you to identify different parts of the program

Active - Selecting this box will make the current operation active, it will be calculated and will produce G codes

Title - This will appear in the Objects Tree window to identify this operation

Though Tool Coolant - Will switch on the appropriate command within the Post processor and output the required M code

Vertical Feedrate - This will control the feedrate in the Z axis when the tool is plunging into the workpiece.

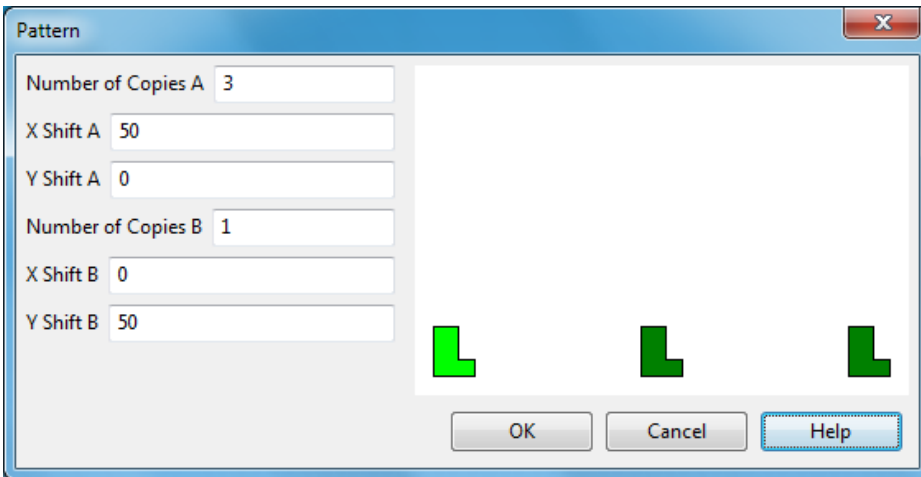
Spindle Speed - This will set the spindle RPM speed for the current tool, it will output the required instructions to the post processor to switch on the spindle (normally a M6 command) and set the spindle RPM, typically an " S " word - S1234 for instance

Pattern

Pattern

A Pattern will setup up repeat machining on subsequent machining operations, to activate the Pattern, you must specify the Pattern number in the [Profile](#) or [Pocket](#) machining operations

This is the Pattern Dialogue box



Number of Copies A

This is the number of machining instances, in the above example there are 3 instances separated by 50 mm in the X axis

X Shift A

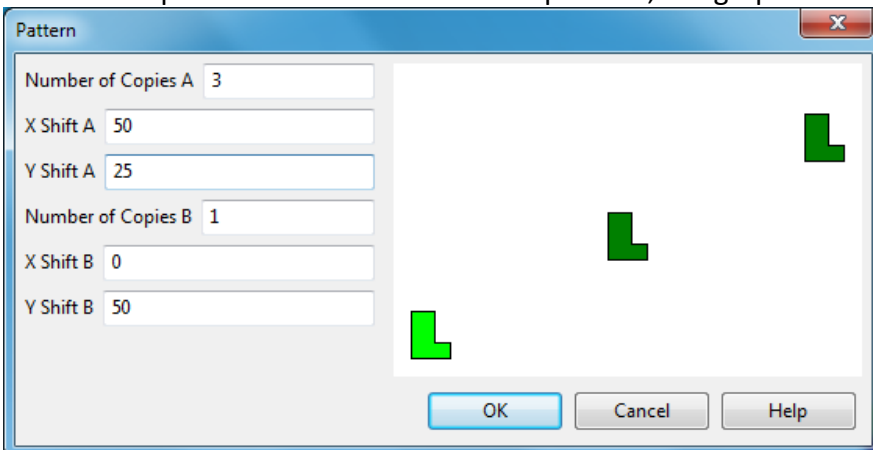
This is the amount each instance will be separated from the others in the X axis

Y Shift A

This is the amount each instance will be separated from the others in the Y axis

For the Pattern command to work correctly, you must specify a distance in either or both the X/Y Shift boxes

In this example the Y Shift has also been specified, the graphics show the results



Number of copies B

This is the number of machining instances to be combined with **Number of Copies A**

X Shift B

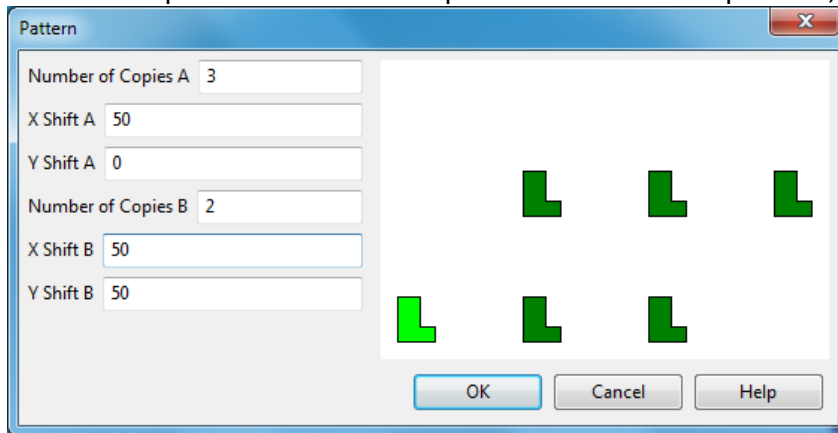
This is the amount each instance will be separated from the others in the X axis

Y Shift B

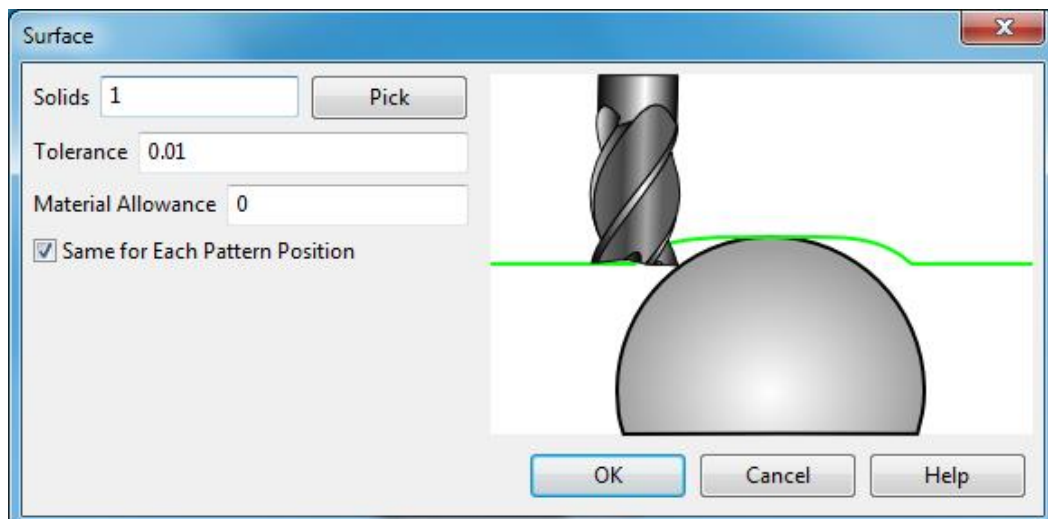
This is the amount each instance will be separated from the others in the Y axis

For the Pattern command to work correctly, you must specify a distance in either or both the X/Y Shift boxes

In this example the Number of Copies B has also been specified, the graphics show the results

**Surface****Surface**

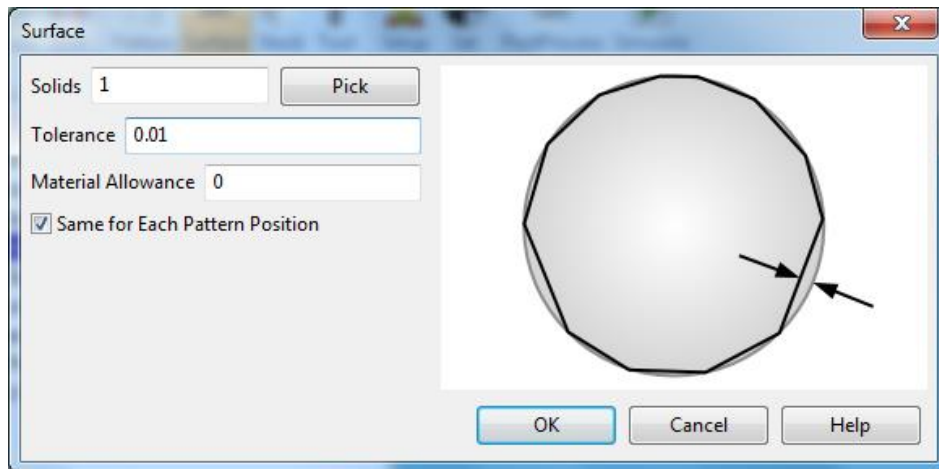
This command will create a Surface from an existing solid model, the surface can then be used in the [Profile](#) and [Pocket](#) commands to create a 3D toolpath



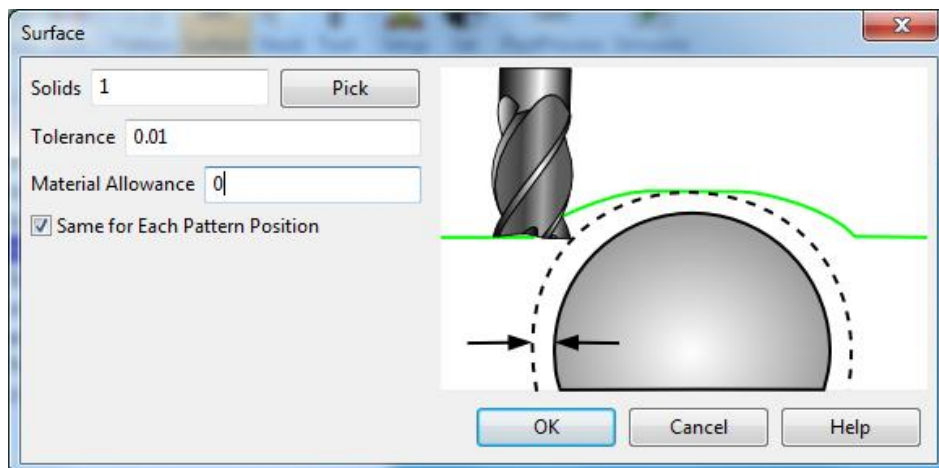
If you are working with a single solid model then you only need to create the surface once, the default name for the surface will be Surface 1, but you can set the number in the first input window. If you are working on multiple models you will need to select the model using the Pick button.

Tolerance - This will set the accuracy of the surface being created from the model. The value entered here will depend on your current units MM or INCH.

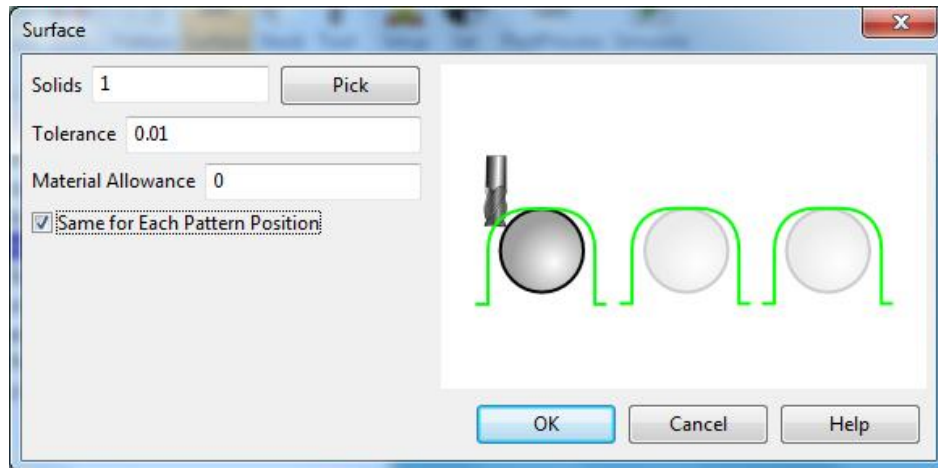
Setting this value to a smaller number will create a more accurate surface but will increase the time taken to calculate the toolpath and the size of the output file (G code) will be considerable larger. Use the largest value you can to give you the accuracy you need. You will need to experiment with this.



Material Allowance - Setting this value will produce a surface that is offset from the original model



Same for each Pattern Position - Checking this box will use this surface for each instance of machining if you have setup a Pattern to produce multiple toolpaths.



Created with the Personal Edition of HelpNDoc: [Produce Kindle eBooks easily](#)

Tool

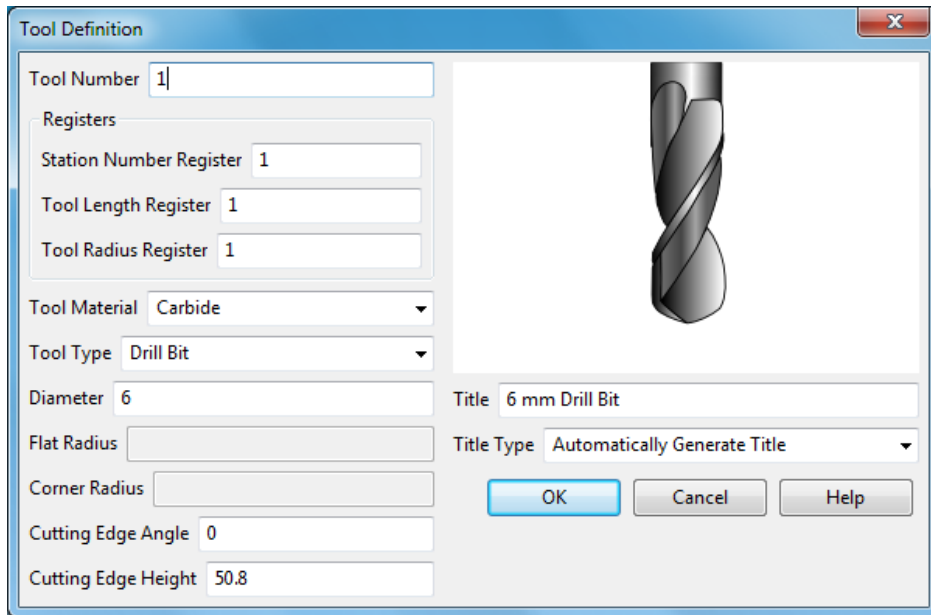


Tool

This option will allow you to define tools that can be used in the current program and also saved into a toolfile for use in any program.

Please also see the Tool entry in the [Program Objects tree](#) on the main screen.

This is the Tool dialogue box that is displayed.



Tool Number - this will set the number of the tool that you are defining

Registers - these parameters can be used to call up registers in the machine tool controller that store data associated with this tool

Station Number Register - this can be used to assign a tool station number on an automatic tool changer for this tool. This is often referred to as the "pot" number

Tool Length Register - this is the register number on the controller that holds the value of the tool's length

Tool Radius Register - this is the register number on the controller that holds the value of the tool's radius

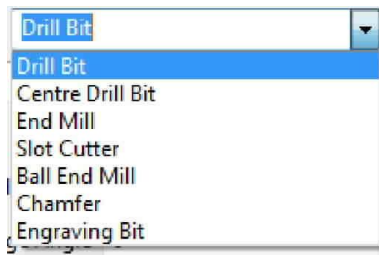
The register above must not contain the actual tool length or radius, they are merely used to call up a register (memory location) in the controller that holds this data.

As an example, on older Fanuc controls it was necessary to have the tool length value stored in registers 1 to 20 and tool radius in values register 20 to 40

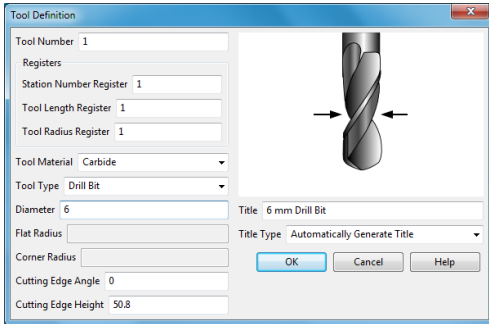
On most modern controllers these registers are not needed as the tool's length and radius information is stored in a tool table that is accessed when the tool selected by the use of a tool change sequence such as T1 M6

Tool Material - sets the tool material - may be used to select feeds and speeds.

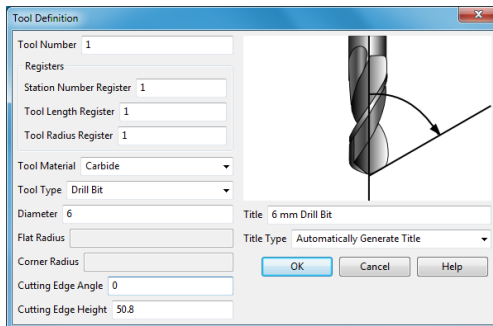
Tool Types - defines the type of tool you are defining. Available types are -



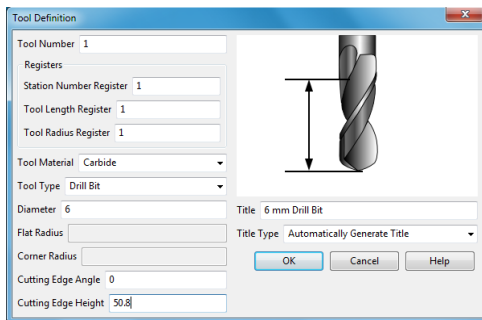
Drill bit - parameters



Diameter

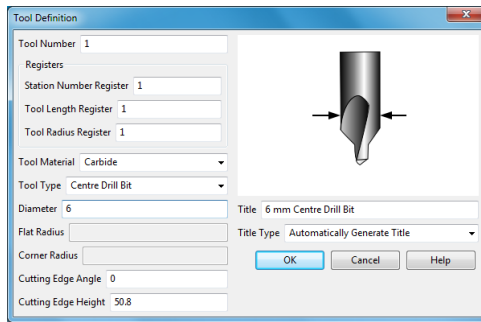


Cutting Edge Angle

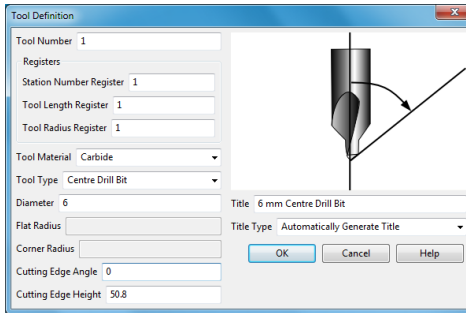


Cutter Edge Height

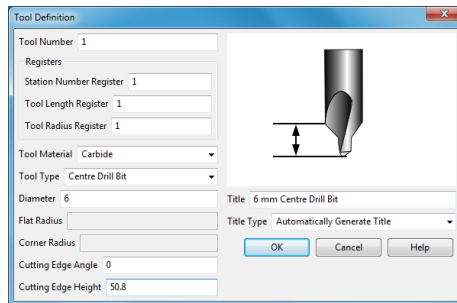
Center Drill Bit - parameters



Diameter

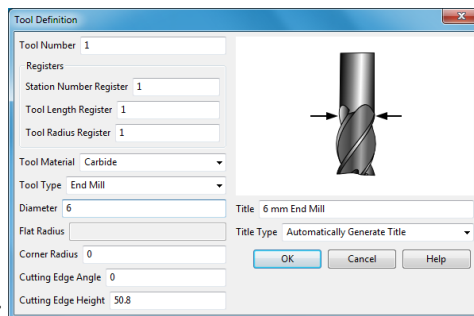


Cutting Edge Angle



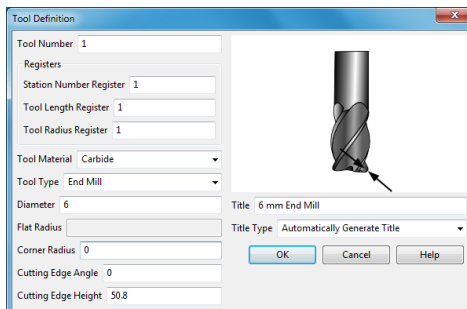
Cutter Edge Height

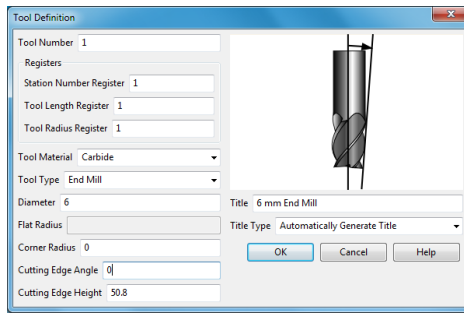
End Mill - parameters



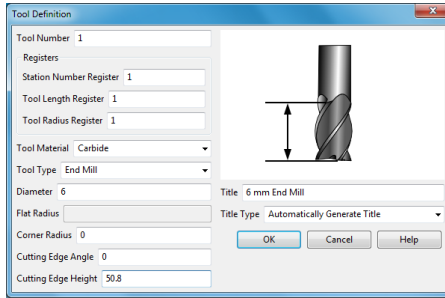
Diameter

Corner Radius



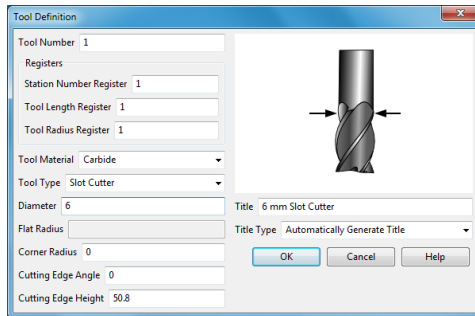


Cutting Edge Angle



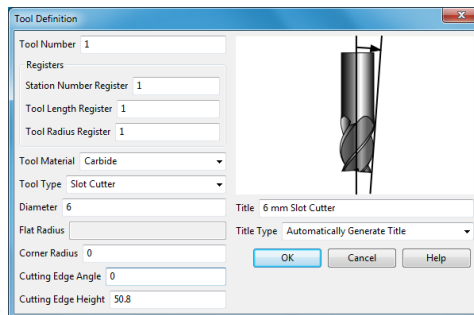
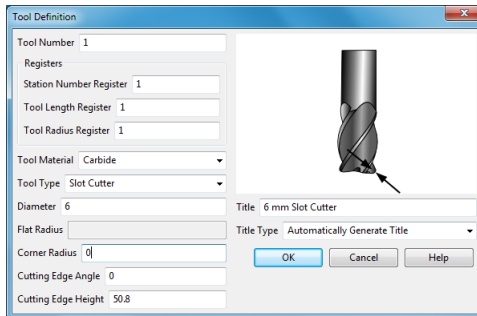
Cutting edge height

Slot Cutter - parameters

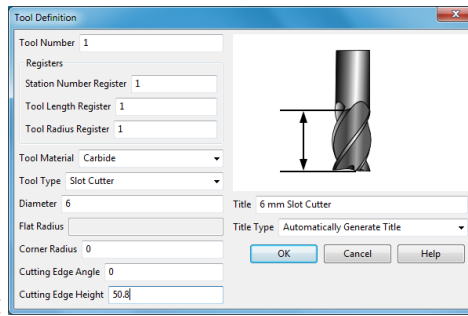


Diameter

Corner Radius

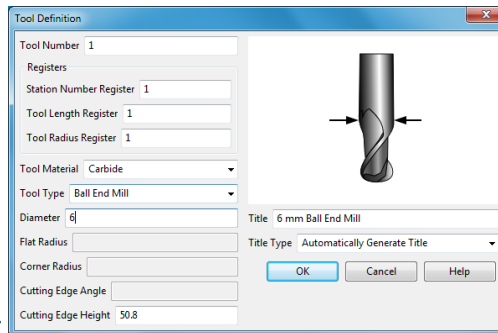


Cutting Edge Angle

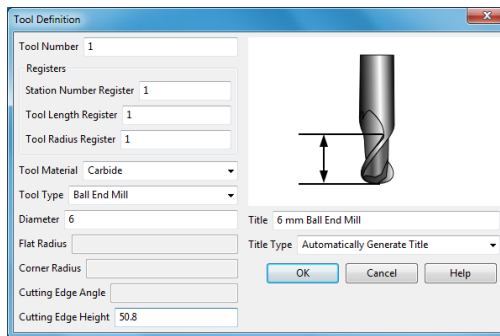


Cutting edge height

Ball End Mill - parameters

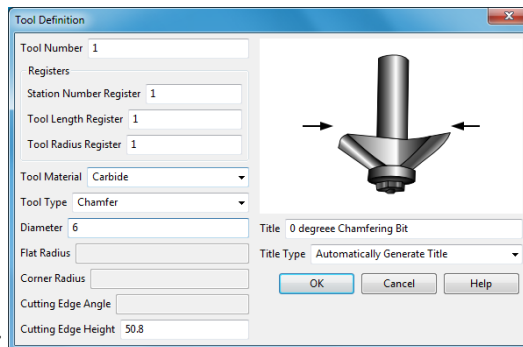


Diameter

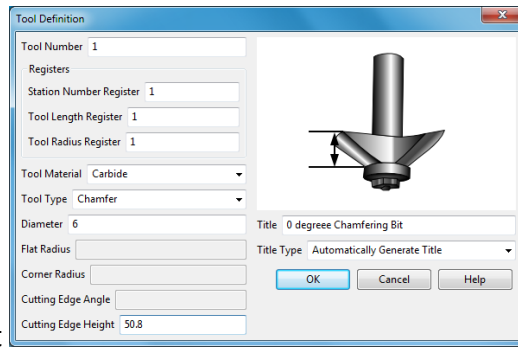


Cutting Edge Height

Chamfer - parameters



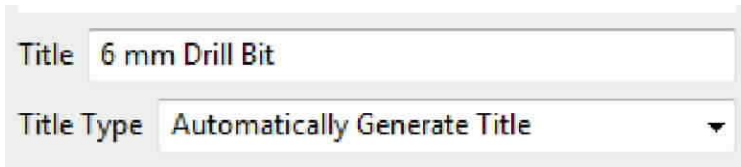
Diameter



Cutting Edge Height

Engraving Bit - parameters ----- still to come

Title and Title Type



Setting the Title type to "automatic" will create an entry based on the diameter and tool type. Setting the Title type to "leave manual" will allow you to enter your own Tool Title.

Setup



Setup

This section deals with the Tool Change position and default Post Processor to be used, the dialogue is shown here.

The screenshot shows the 'Program' dialog box with the following fields and options:

- Tool change position:** X 0, Y 0, Z 200. Buttons: OK, Cancel, Help.
- Post processor:** The Post-Processor will create an NC program which can be downloaded to the target machine. Path: C:\ProgramData\DolphinCadCam\PartMaster\PostProcessors. Button: Browse.
- Post processor name:** [Dropdown menu]. Check box: The Post processor is in the same folder as this CNC program.
- NC program file name:** test. **File extension:** tap.
- Output in inches:**
- Utility:** A utility program can be associated with the post-processor. This program will modify the post-processed output to a form that cannot be produced by the post-processor directly. e.g. The utility cvtiso.exe converts the post-processed output into even parity. **Utility program:** [Text box]. **New file extension:** [Text box].
- Debug:** Include Debug information; useful when developing new post-processors. or when modifying existing designs.

Tool Change Position

Enter the required Tool change position in X, Y and Z. These are absolute coordinates relative to the XYZ datum position of this part.

Post Processor

This window will show the default folder where all post processors are stored, unless you have specifically placed the post processors into a different folder, leave this as the default which is C:\ProgramData\DolphinCadCam\PartMaster\PostProcessors. Use the Browse button to choose a different folder if you have stored posts in a different folder.

The second window choose the post processor you actually want to use, choose the correct post processor for your machine tool / controller.

NC Program file name

This window shows you the name of the output file created by the post processor, the name will be the same as that which you have used to create the program.

File extension

3DCAM uses a different file extensions for the various files that are created when a part program is made. The default for the post processed file (normally a G code file) is "tap" - the file created will be a simple text file. It may be that you need to change the file extension for various reasons, if so, enter the new file extension here, for example txt.

Output in inches

Use this check box if you wish to output a Metric program in INCH units. If your current units are set to INCH you don't need to check this box as the output will be in INCHs

Utility Program and Debug

These facilities are supplied for advanced users only, for more information please contact your software supplier.

Set Origin

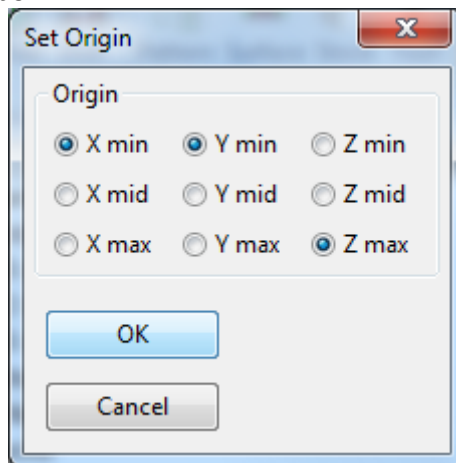


Set Origin

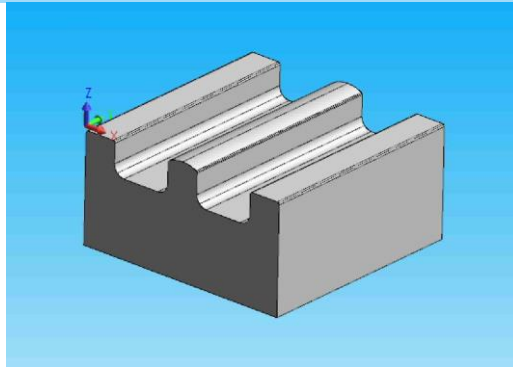
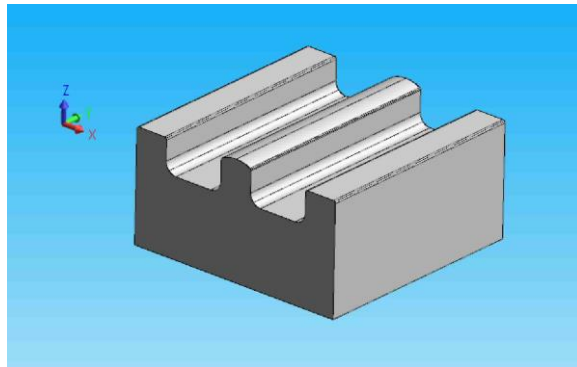
This option allows you to set the Origin (XYZ Zero position) on the imported model and therefore the XYZ Zero position for the part program you are creating.

To use this feature, you must have already imported a model.

This is the Set Origin dialogue box



In this example, the imported model has its origin away from the actual component, to set the XYZ Zero to the minimum dimension in XY and the maximum dimension in the Z axis. Choose X min, Y min, Z max



The model will be moved so that the bottom left corner of the part is set to XY zero and the top surface set to Z zero

Post Process



Post Process

When you create a program using 3DCAM, the data is stored in format know as CLD - Cutter Location Data - this data is a neutral format that needs to be converted into language that your machine tool's controller can understand.

This is accomplished by the use of the Post Processor module, the output from the Post Processor is normally a G Code type of file, similar to this:

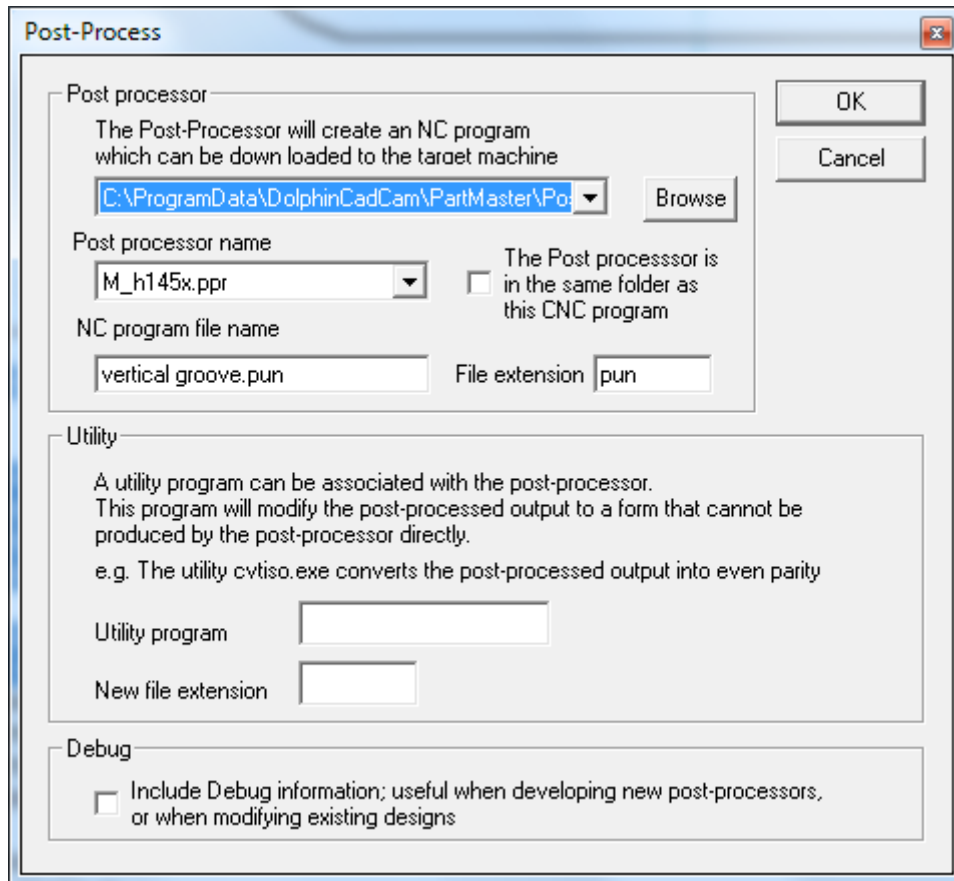
```
N5(Stepped Shaft.cnc)
N10 G92 S2500
N15 G21
N20 M09
N25 G00 G40 G97 Z100.0 X150.0 T0000
N30 T0101
N35 G92 S3500
N40 G94 G96 F150.0 S250 M03
N45 M08
N50 X124.486 Z4.127
N55 G01 Z2.064
```

But is could also be in this type of format:

```
1 L X0, Y0, R0 F9999 M
2 TOOL DEF 1 L+100, R+0,0
3 TOOL CALL 1 Z S1000
4 L X+60, Y+27, R0 F9999 M03
5 L Z+3, R0 F9999 M
6 L Z-3, R0 F100 M
7 L X-70, Y+27, RR F200 M
8 CC X-70, Y+60,
9 C X-103, Y+60, DR- RR F200 M
10 L Z+50, R0 F9999 M
```

In fact almost any output format is possible.

This is the Post Processor dialogue box.



Post Processor

This window will show the default folder where all post processors are stored, unless you have specifically placed the post processors into a different folder, leave this as the default which is C:\ProgramData\DolphinCadCam\PartMaster\PostProcessors. Use the Browse button to choose a different folder if you have stored posts in a different folder.

The second window choose the post processor you actually want to use, choose the correct post processor for your machine tool / controller.

NC Program file name

This window shows you the name of the output file created by the post processor, the name will be the same as that which you have used to create the program.

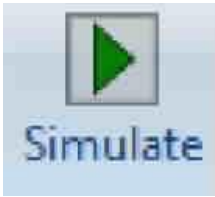
File extension

3DCAM uses a different file extensions for the various files that are created when a part program is made. The default for the post processed file (normally a G code file) is "tap" - the file created will be a simple text file. It may be that you need to change the file extension for various reasons, if so, enter the new file extension here, for example txt.

Utility Program and Debug

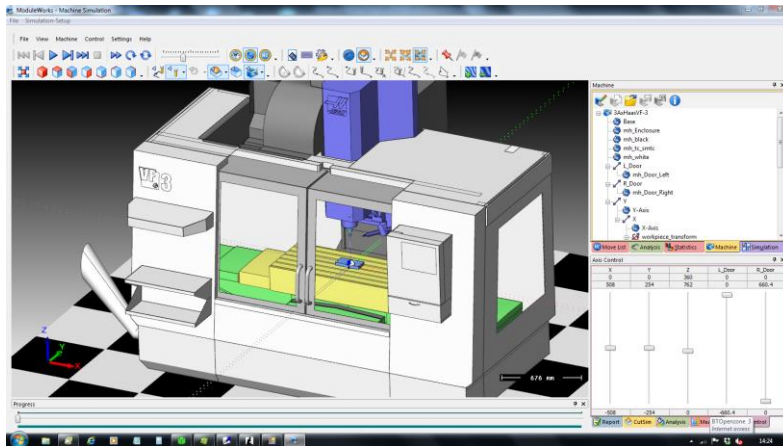
These facilities are supplied for advanced users only, for more information please contact your software supplier.

Simulate

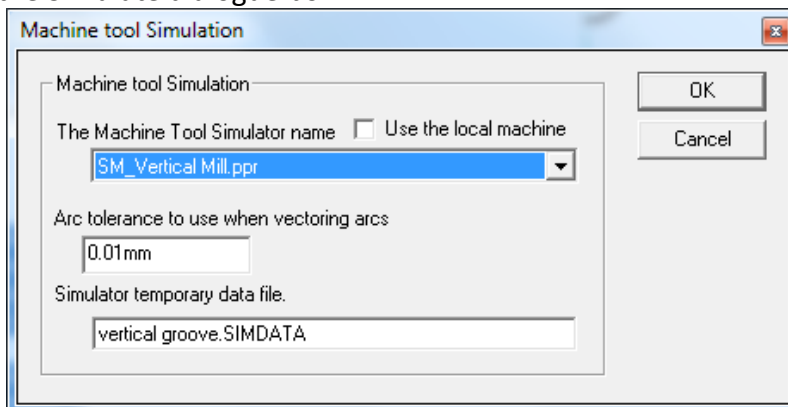


Simulate

This will run the Machine Tool Simulation module



This is the Simulate dialogue box



The available options are:

SM_Vertical Mill - a standard 3 axis machining centre.

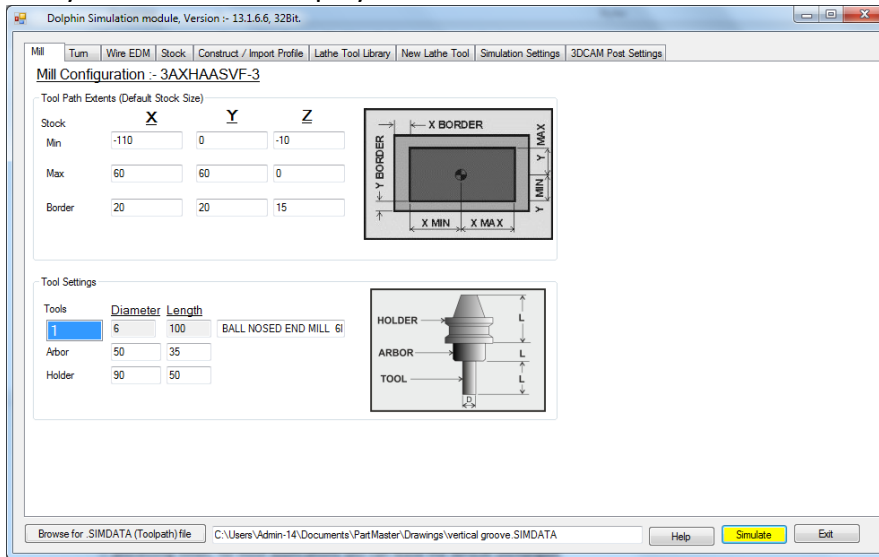
SM_4AXHorizontal Mill - a horizontal mill

SM_GantryMill - a vertical Gantry style mill

Arc tolerance

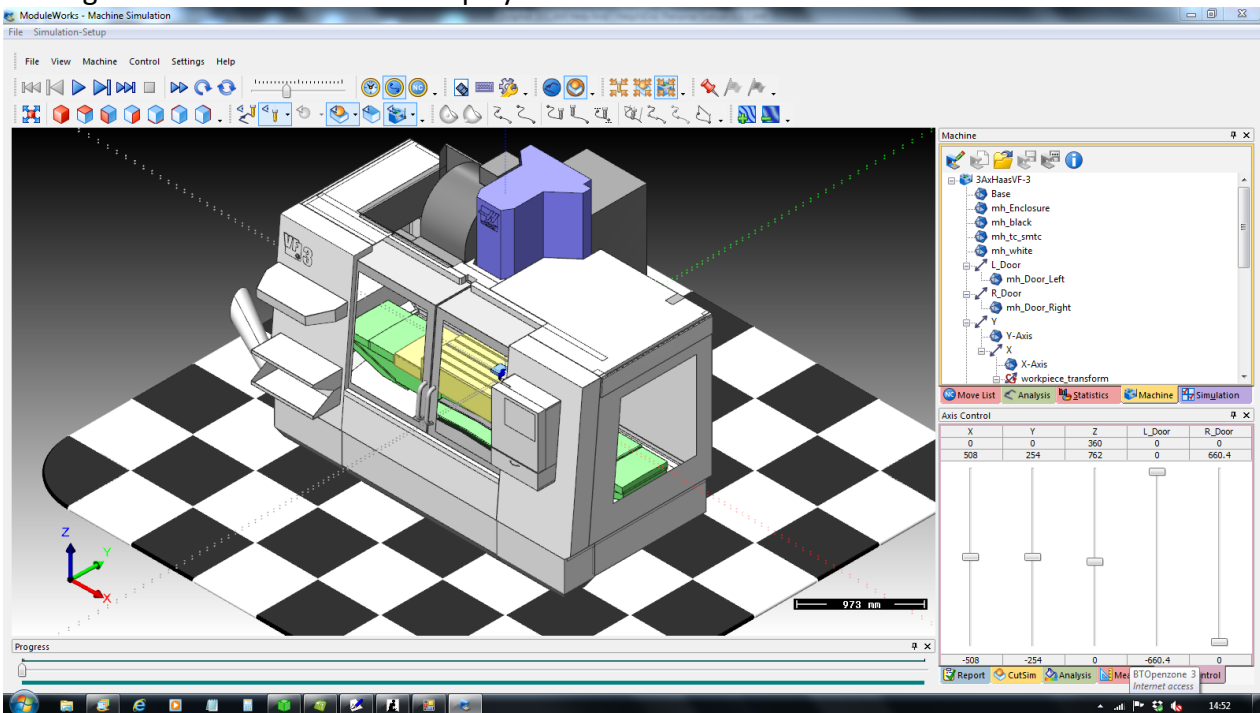
This is used as a coarseness factor when converting arcs into lines, setting this to a very small value will give more accurate results but will result in greatly increased processing times, for most applications you can leave the default unchanged.

When you click OK you will see this display:



This first screen shows the default stock sizes that have been extracted from the machining path. To accept these defaults, click OK to run the Simulation. If you wish to modify the Stock values, click the **Stock** tab.

Clicking the Simulate button will display the Simulator



With the cursor in the main graphics window, scroll the mouse wheel in/out to zoom in/out
To Rotate the graphics, press and hold the left hand mouse button whilst at the same time moving the mouse

To Pan the graphics, press and hold the right hand mouse button whilst at the same time moving the mouse

The video controls can be used to Run, Pause, Fast Forward and Reset the display





The **Focus** controls can be used to set the display's Focus

The 3 options are

- 1 = Focus on the Stock with the tool stationary
- 2 = Focus on the Stock with the tool moving
- 3 = Focus on the entire machine tool

For more help about the Machine Tool Simulator, click the **Help** button

Screen Layout

Screen Layout

Below is the main 3DCAM screen.

The screen is split into various areas, they are

The [Ribbon toolbars](#) control all functions within the software. To display different [Ribbons](#) (topics) - please click on the appropriate ribbon item - such as File, Geom, Solids etc.

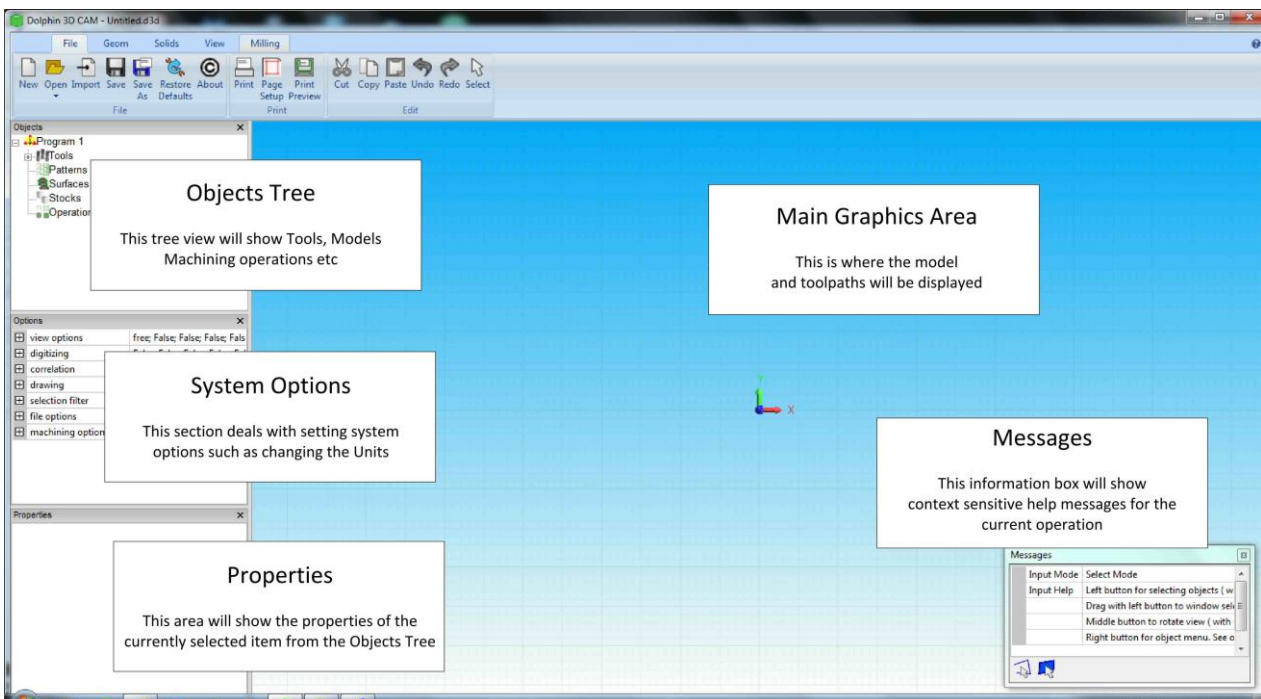
Main Graphics Area

[Objects Tree](#)

[System Options](#)

[Properties](#)

[Messages](#)



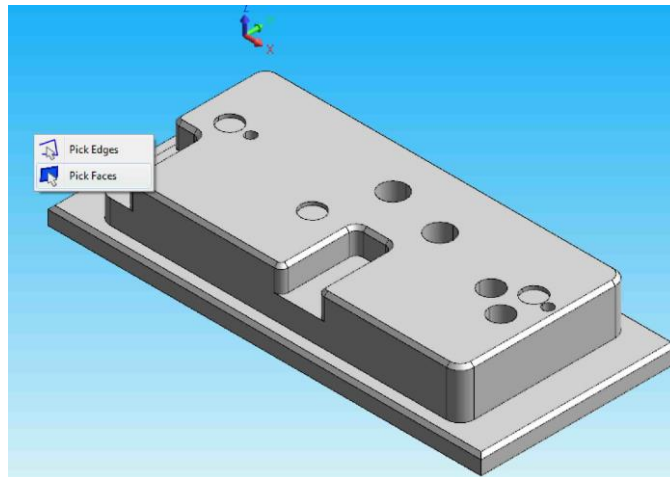
Creating and using Sketches

Creating and using Sketches

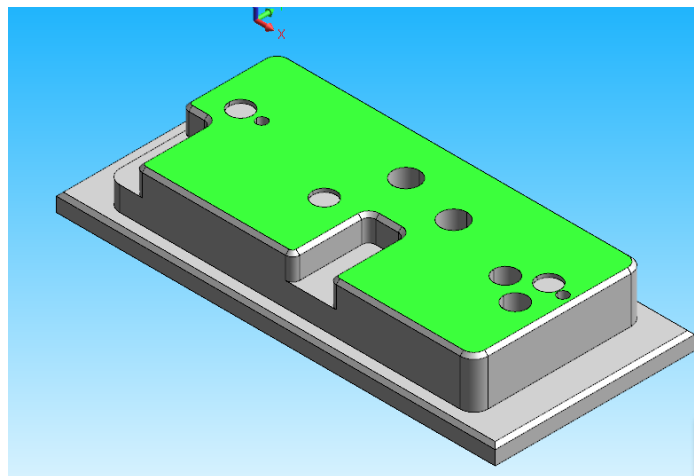
- 1) Extracting a Sketch from a model
- 2) Importing a DXF file
- 3) Using the Geometry construction options

1) Extracting a Sketch from a model

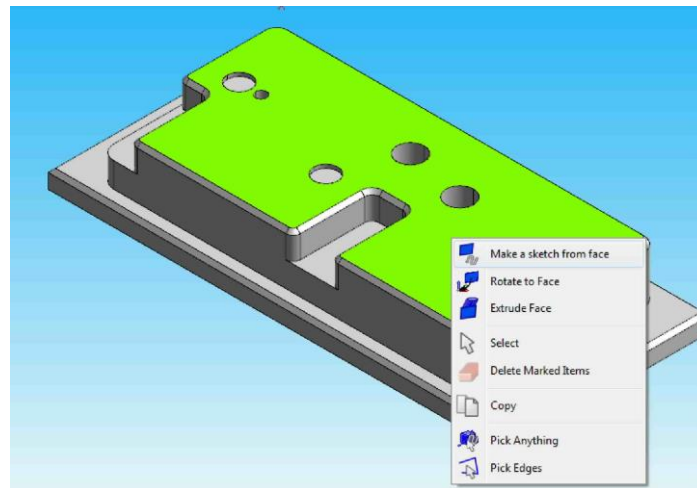
Firstly import the model, then right click the mouse in the Graphics area and choose Pick Faces.



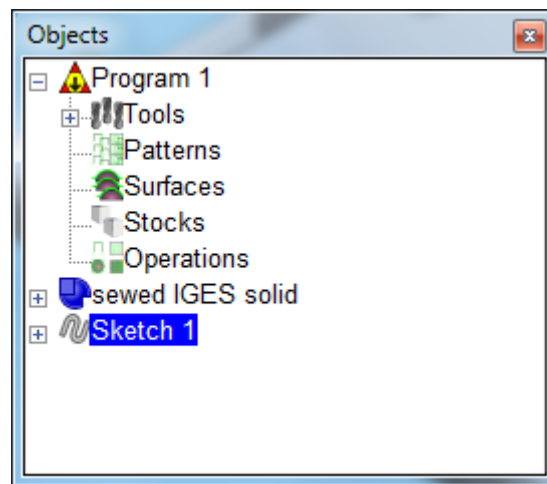
Move the mouse over the Face required - it will be highlighted



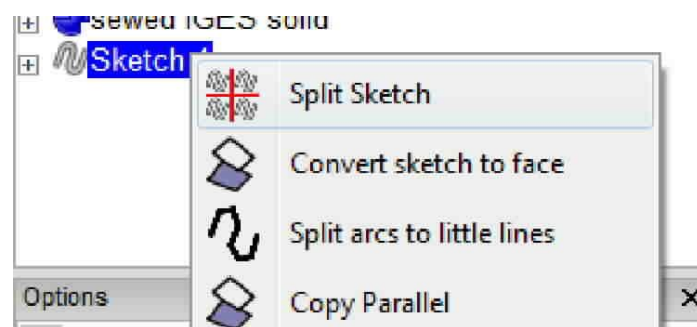
Next, right click and choose **Make Sketch from Face**



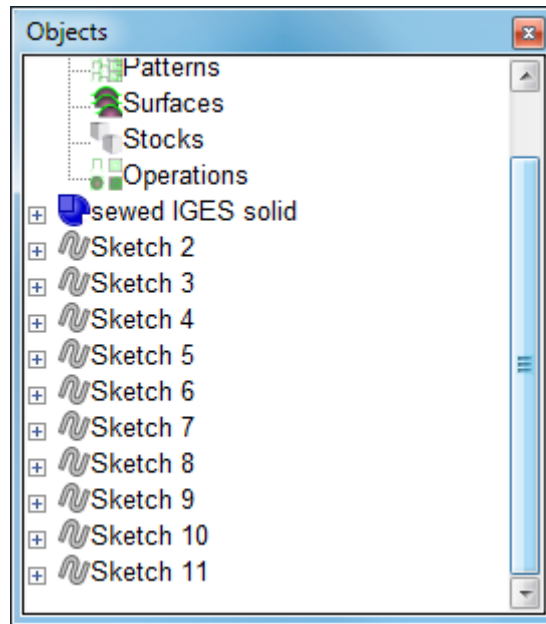
A new Sketch entry will appear in the Objects Tree



This Sketch will have more than shape, the outside shape and circles. Right click the on the Sketch and choose "**Split Sketch**"

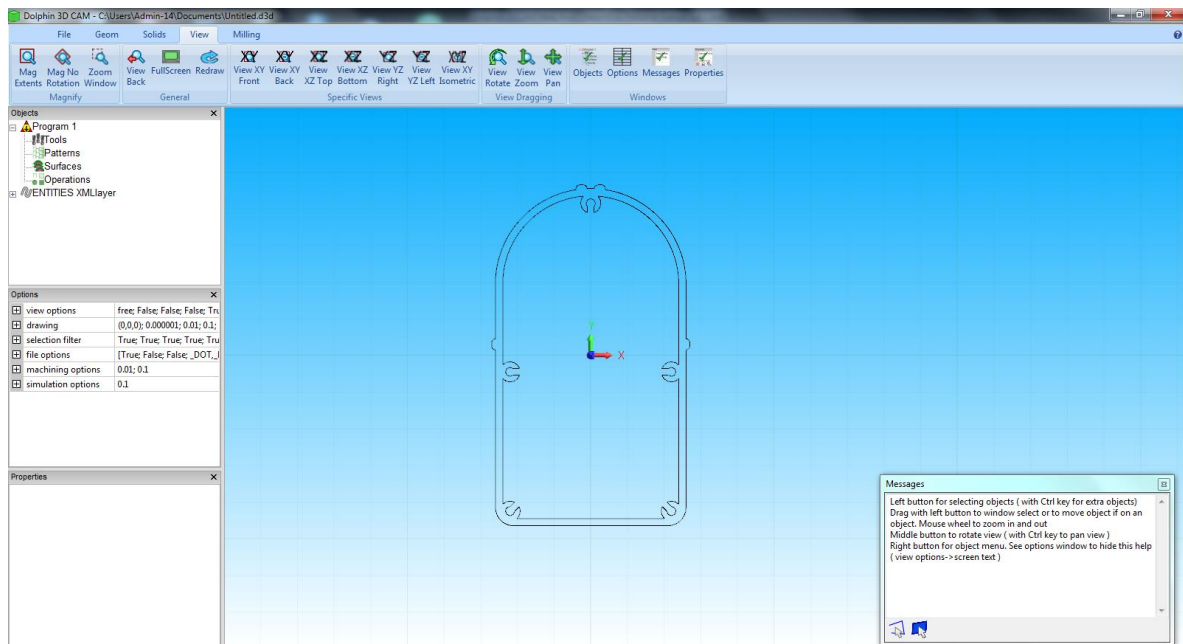


This will create a number of Sketches that can be used in [Profile](#) and [Pocket](#) operations.

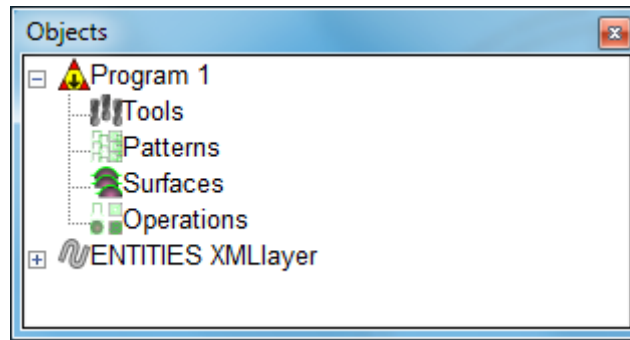


2) Importing a DXF file

Use [File > Import](#) to import the DXF file.

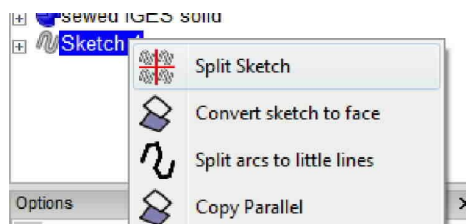


In this example, a entry has been added to the Objects tree called ENTITIES XMLayer (this will change depending on the information in the DXF file)

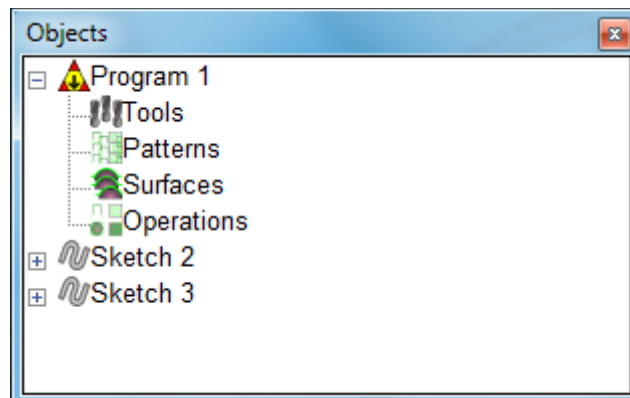


As you can see, the imported DXF file is made up from an outer and an inner shape, to separate these shapes, move the mouse over the entry and click the right hand mouse button.

From the displayed options choose Split Sketch



The previous entry will be replaced by a list of the Sketches



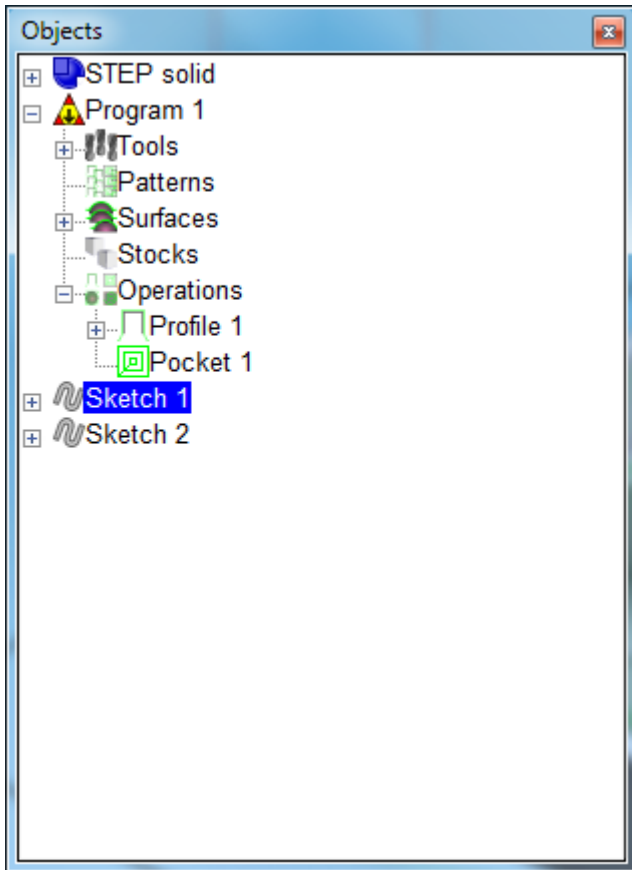
These Sketches can then be used in the [Profile](#) or [Pocket](#) operations

3) Using the Geometry construction options

Program Tree Objects

Program Objects Tree

This window will display the Objects that may form part of the current job. An Object can be a solid model, a tool, a milling operation or sketch for example.

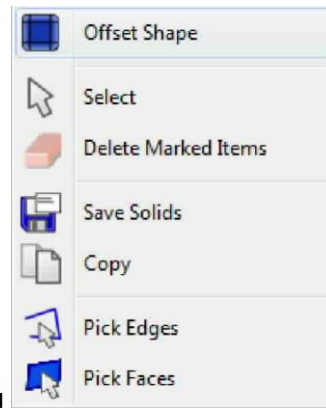


Objects can be Expanded or Collapsed using the + or - buttons next to each entry, only shown if there are further entries/parameters available.

Imported File

The type of imported file - **IGES, STEP or STL**





Right click the mouse over the entry to display this sub-menu

Offset shape - Will allow you to create a new solid that is offset by a value, either + or - from the original

Select - Will select the model

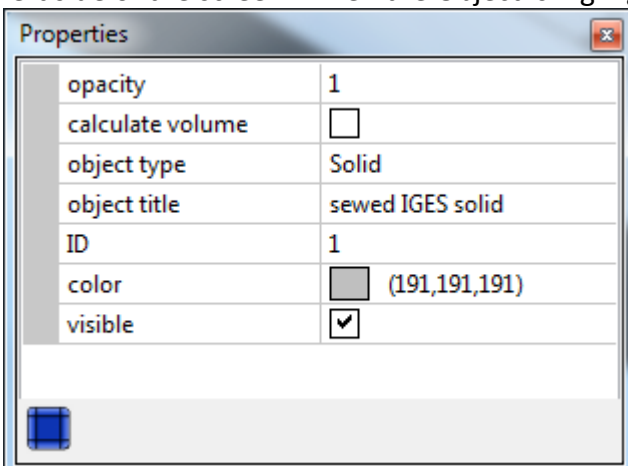
Delete Marked Items - This will immediately delete the model, to undo this command go to File > Undo

Save Solids - This will allow you to save the current model. This may be because you wish to keep the original model before modifying it, or so that it may be used in another job. You may save the model in the same, or a different file format from the original model.

Copy - Will create a copy of the original model. You can then use the right hand mouse when the cursor is in the Program Objects Tree window and "Paste" the copy into the tree.

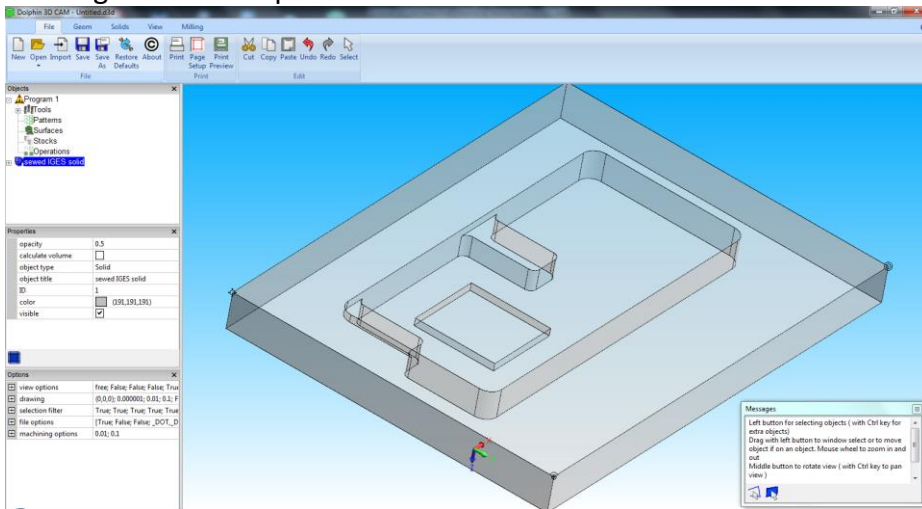
Pick Edges or **Pick Faces** - Will allow you to select the type of entities required, for instance when creating a Sketch of certain features on the model. See also [Creating and Using Sketches](#)

The **Properties** for this entry will be shown in the Properties window that is displayed at the bottom left side of the screen. When the Object is highlighted in the Program Objects Tree



This is the Properties of the imported IGES file.

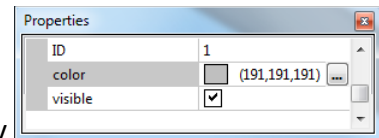
Opacity - This controls the view of the model, 1 = fully solid, 0 = invisible. Setting this to 0.5 would give this transparent view of the model.



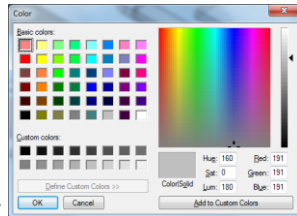
Calculate Volume - Will give the volume and centre of gravity of the model

Object type - Will show the type of file imported, sewed IGES solid, STEP solid or STL object

ID - is simply there to identify this model - you may import more than 1 model



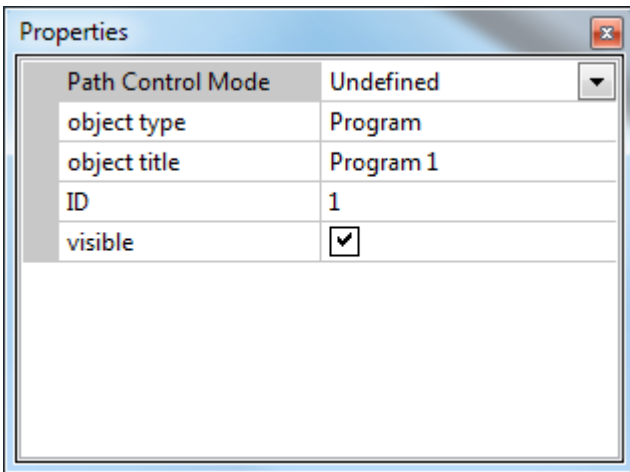
Colour - This can used to set the colour of the model, click the entry



to see the colour selector

Program Number

The **Program number** - default = 1



To change any of the parameters, click the entry

Path Control Mode - Options are:

Undefined = Not set

Exact Path Mode

Exact Stop Mode

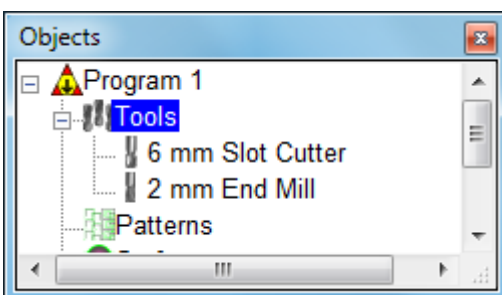
Best Possible Speed

Visible - uncheck to disable view of all toolpaths

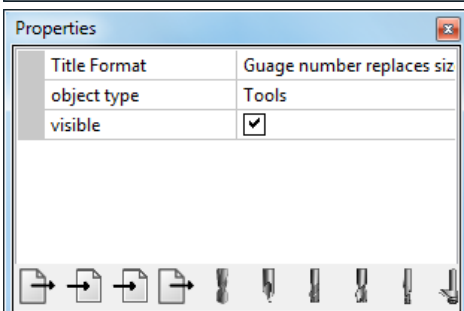
Tools

Tools

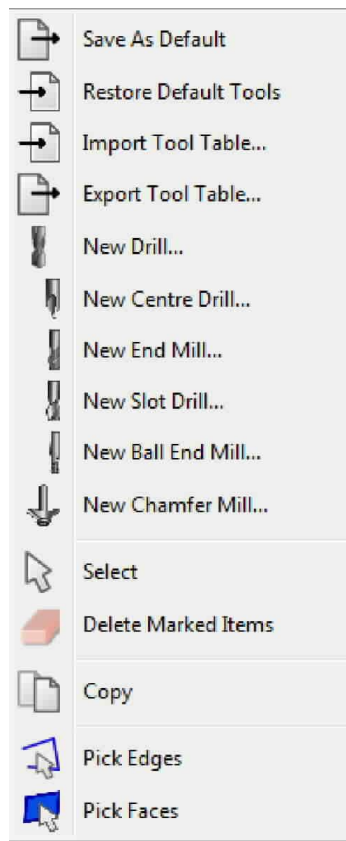
This will show the tools available. Click the + button to expand the tree and view the tools. See also the [Tool](#) option on the [Milling toolbar](#). Clicking on an individual tool will show the tool in the graphics window.



The Tool Properties window will show these options



Right click when the mouse is over the Tools entry to see this sub-menu of options.



Save As Default - This will save all the currently defined into the default tool table that is loaded automatically when a new job is started.

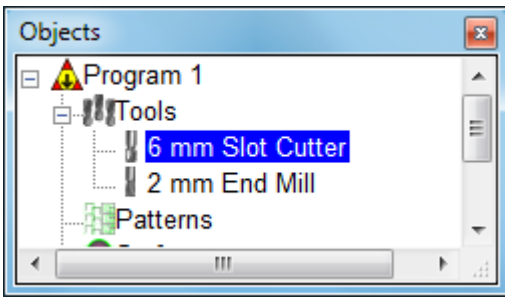
Restore Default Tools - Will replace all current tools with those in the default tool table.

Import Tool Table - Will allow you to open a previously exported (saved) tool table.

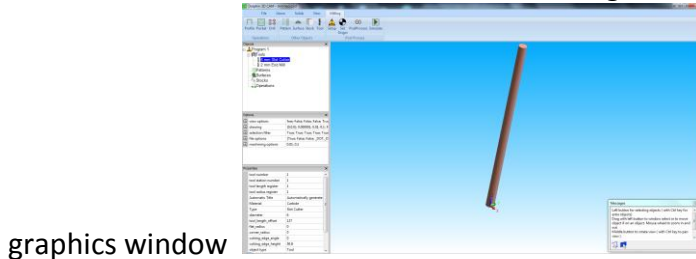
Export Tool Table - This will allow you to save all the currently defined tools into a new tool table, you may have any number of tool tables. For instance you may have a different tool table for each machine tool type you have.

The next section will allow you to define a new tool, See also the [Tool](#) option on the [Milling toolbar](#) for more details about how to define a tool

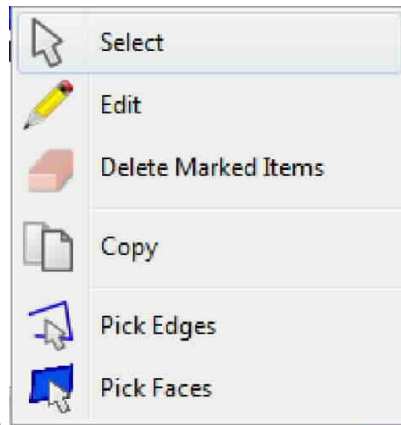
Copy, Pick Edges and Pick Faces - not functional in this context



Clicking on an individual tool will show the tool in the



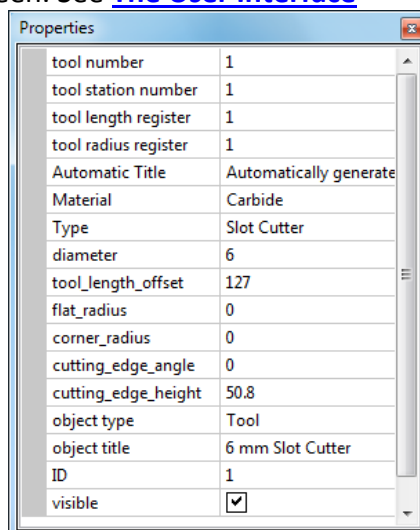
graphics window



Right click over the tool to show these options

Select - Will show the tool in the graphics window

Edit - Will display the dialogue box that was used to create the tool originally so that you may edit the contents. This is also possible by using the **Tool Properties** window that is displayed at the bottom left on the main screen. See [The User Interface](#)



Delete Marked Items - this will delete the current tool entry. To undo this go to File > Undo

Copy - will place a copy of the entry on the clipboard, you can then right click and Paste the tool to create a new tool.

Pick Edges and Pick Faces - not functional in this context

Patterns

Patterns

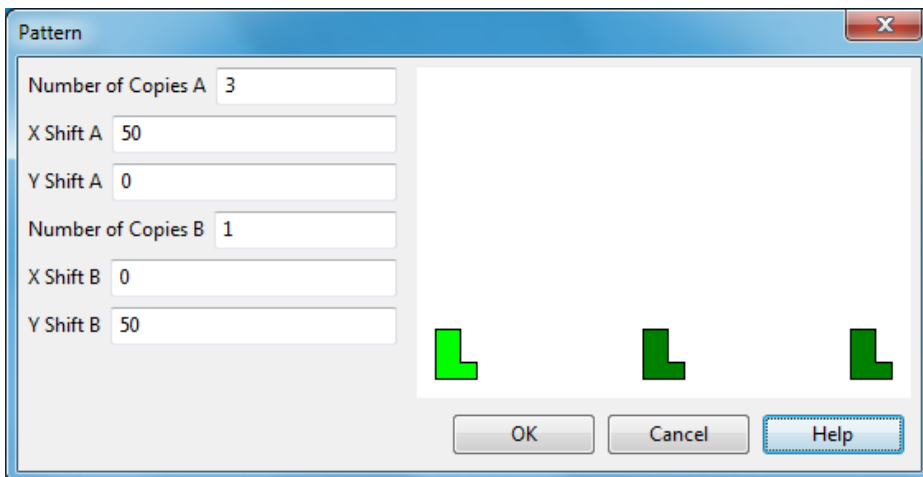
This will show any previously created Patterns using the [Pattern](#) command found in the [Milling Toolbar](#)

This is the entry from the Pattern command

Pattern

A Pattern will setup up repeat machining on subsequent machining operations, to activate the Pattern, you must specify the Pattern number in the [Profile](#) or [Pocket](#) machining operations

This is the Pattern Dialogue box



Number of Copies A

This is the number of machining instances, in the above example there are 3 instances separated by 50 mm in the X axis

X Shift A

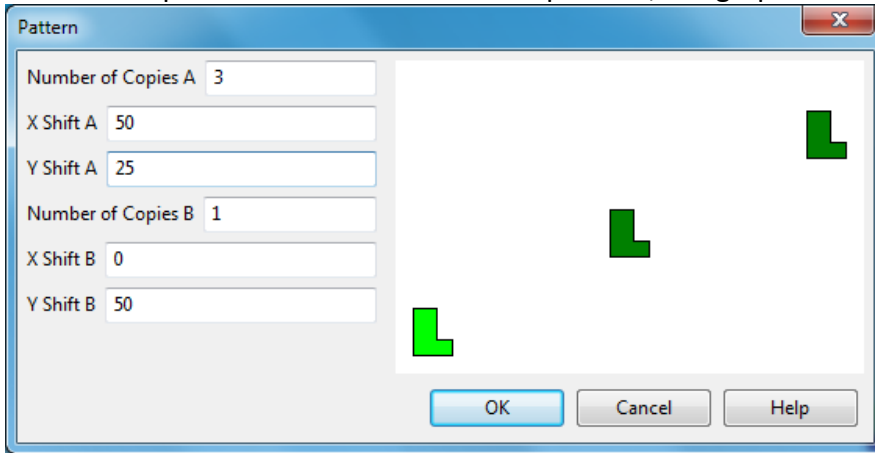
This is the amount each instance will be separated from the others in the X axis

Y Shift A

This is the amount each instance will be separated from the others in the Y axis

For the Pattern command to work correctly, you must specify a distance in either or both the X/Y Shift boxes

In this example the Y Shift has also been specified, the graphics show the results



Number of copies B

This is the number of machining instances to be combined with **Number of Copies A**

X Shift B

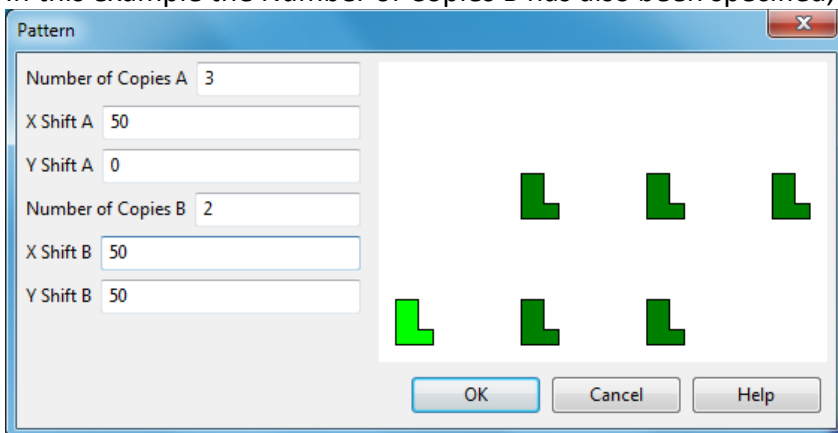
This is the amount each instance will be separated from the others in the X axis

Y Shift B

This is the amount each instance will be separated from the others in the Y axis

For the Pattern command to work correctly, you must specify a distance in either or both the X/Y Shift boxes

In this example the Number of Copies B has also been specified, the graphics show the results



Surfaces

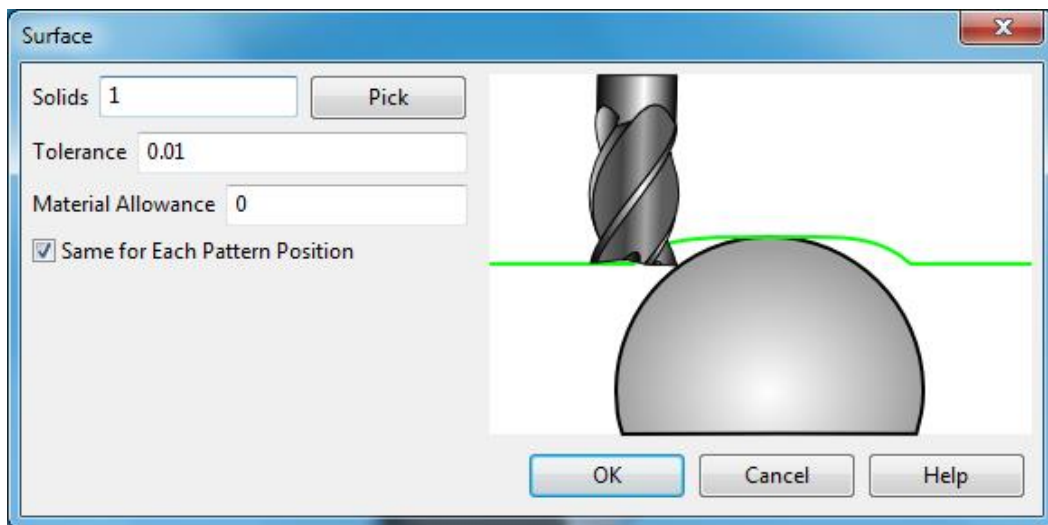
Surfaces

This will show any previously created Surfaces using the [Surface](#) command found in the [Milling Toolbar](#)

This is the entry from the Surface command

Surface

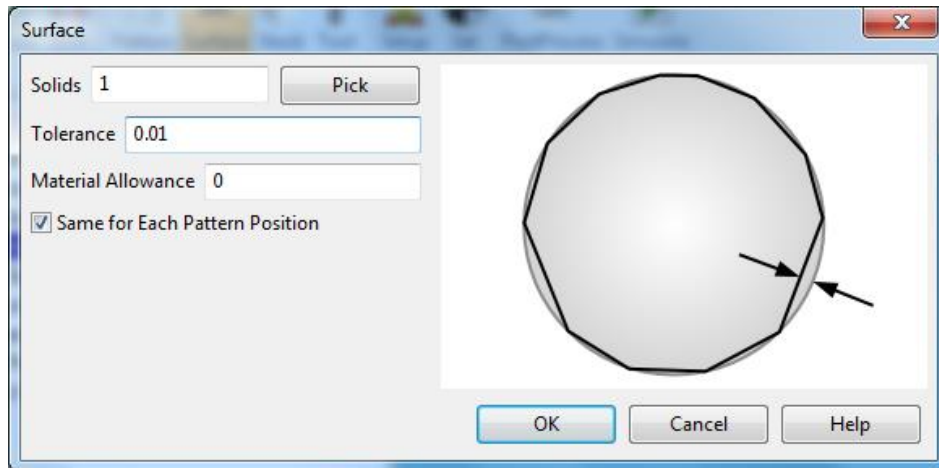
This command will create a Surface from an existing solid model, the surface can then be used in the [Profile](#) and [Pocket](#) commands to create a 3D toolpath



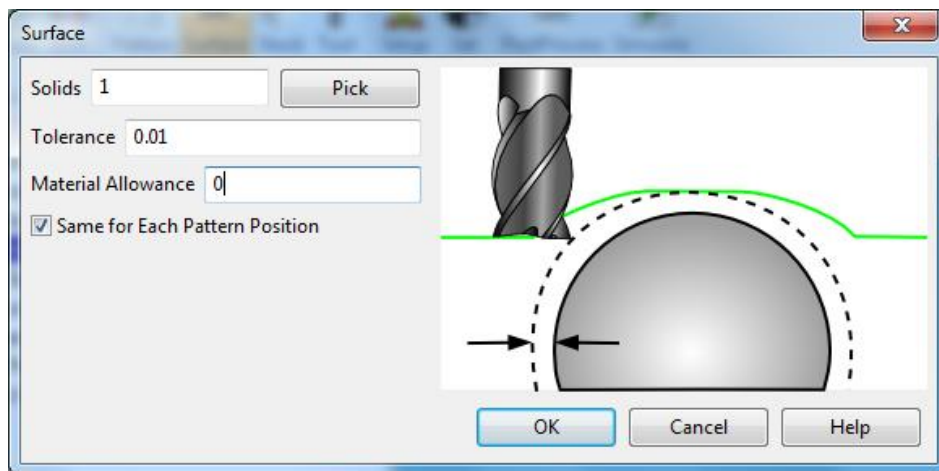
If you are working with a single solid model then you only need to create the surface once, the default name for the surface will be Surface 1, but you can set the number in the first input window. If you are working on multiple models you will need to select the model using the Pick button.

Tolerance - This will set the accuracy of the surface being created from the model. The value entered here will depend on your current units MM or INCH.

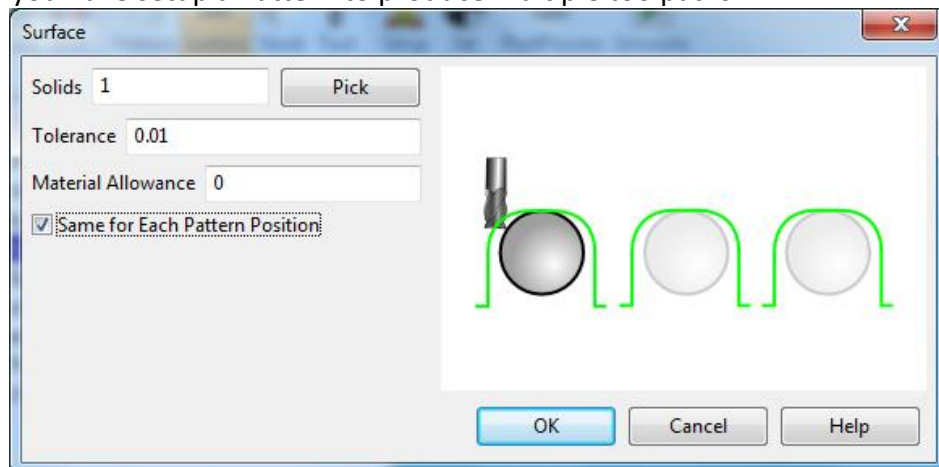
Setting this value to a smaller number will create a more accurate surface but will the time taken to calculate the toolpath and the size of the output file (G code) will considerable larger. Use the largest value you can to give you the accuracy you need. You will need to experiment with this.



Material Allowance - Setting this value will produce a surface that is offset from the original model

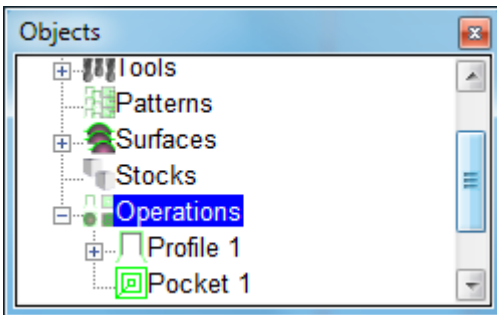


Same for each Pattern Position - Checking this box will use this surface for each instance of machining if you have setup a Pattern to produce multiple toolpaths.



Operations

Operations



This section deals with the actual machining operations. Machining Operations are created by choosing the appropriate option from the [Milling Toolbar](#)

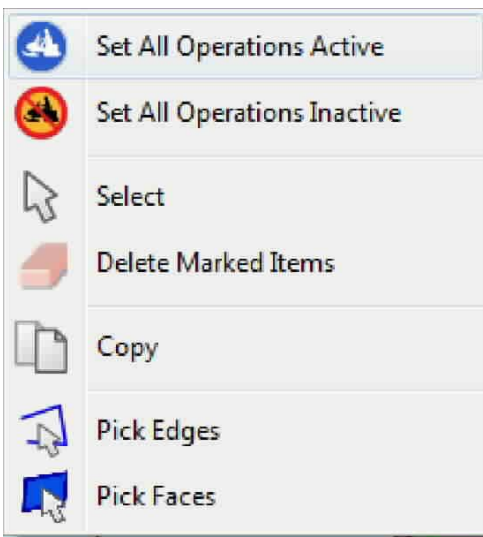
The Operations are:-

[Profile](#) - for creating 2D or 3D Profiling Toolpaths around an existing Sketch.

[Pocket](#) - for creating a 2D or 3D Pocketing Toolpaths on an existing Sketch or Sketches.

[Drill](#) - for creating drilling operations on a previously created series of point positions that represent the centre point of the drilling operation.

Right click the mouse over the Operations item to reveal this sub-menu

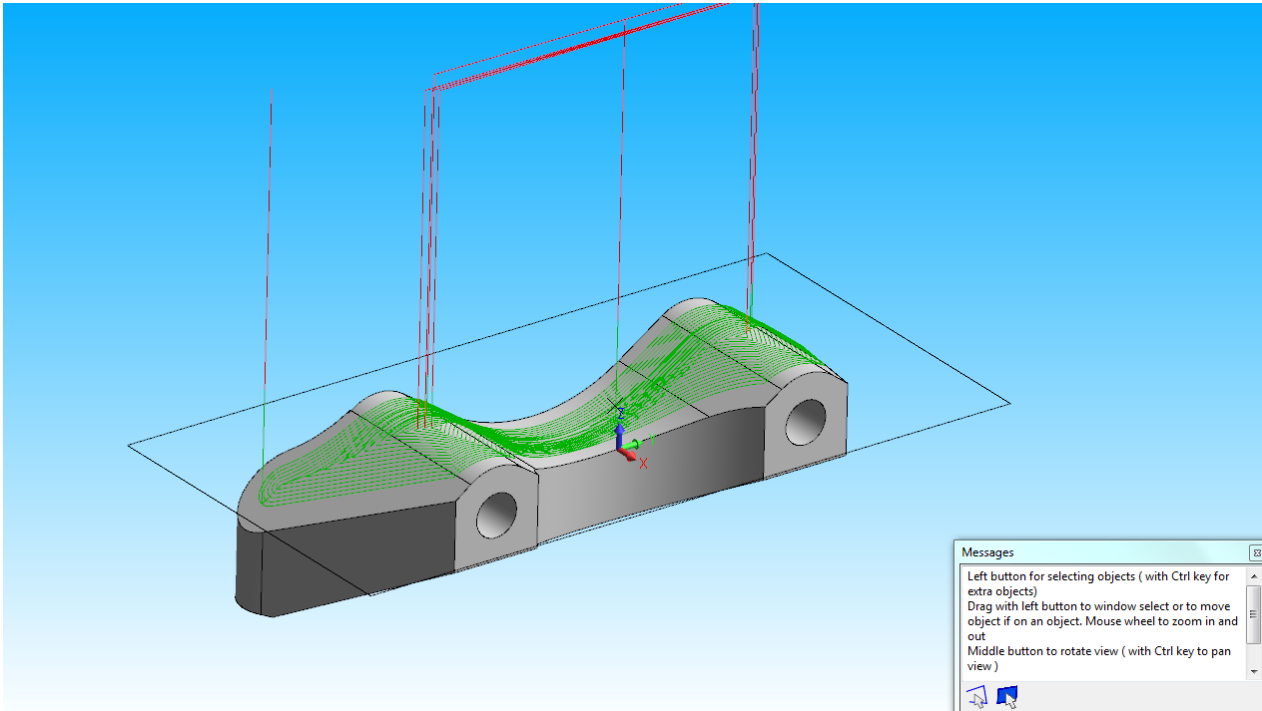


Only the first 2 options have any function in this context.

Main Graphics Area

Main Graphics Area

This area of the screen will display the imported model, sketches and toolpaths, as shown here.



To manipulate the view use:-

Mouse Wheel rotated forward and backward to **zoom in/out** of the view

Press Mouse Wheel down and move mouse to **Rotate the view**

Press Mouse Wheel + Ctrl key and move mouse to **Pan** the view

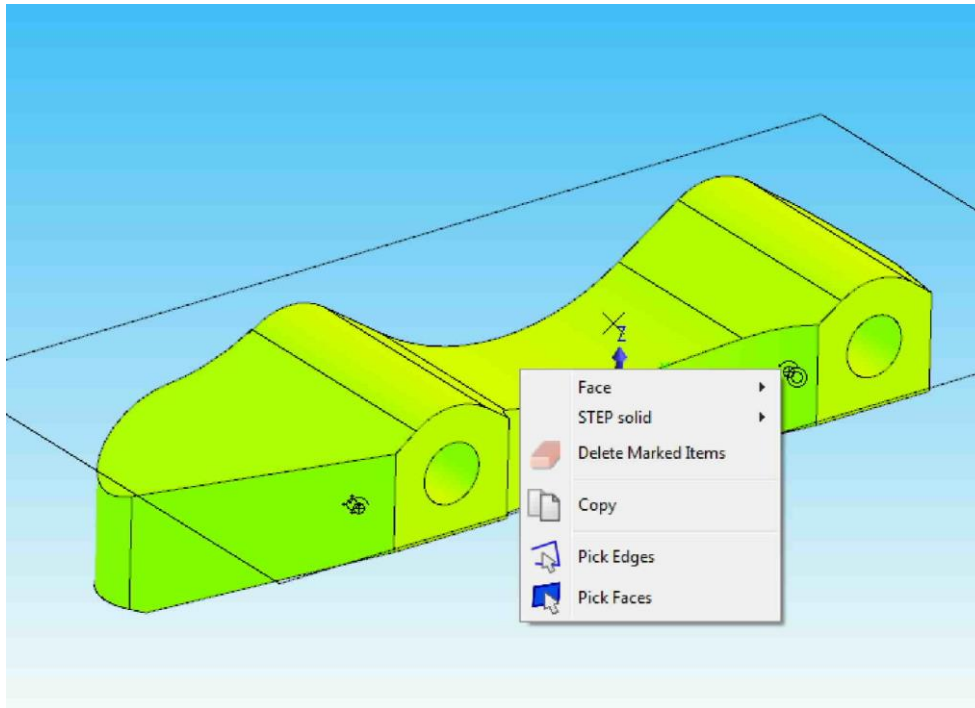
To **Select** an item use the Left button. To select multiple items hold down the **Ctrl** key and click the items.

Selecting Objects

To select on object such as a model, sketch or other object, move the mouse over the object. The object under the cursor will be highlighted (see System Options > View Options > highlight items under cursor) Having selected an object, the item will be also be shown in the Objects Tree.

To view a **Context Sensitive Menu** - click the right hand mouse button, depending on what object you have selected the menu will display the options relevant to that object.

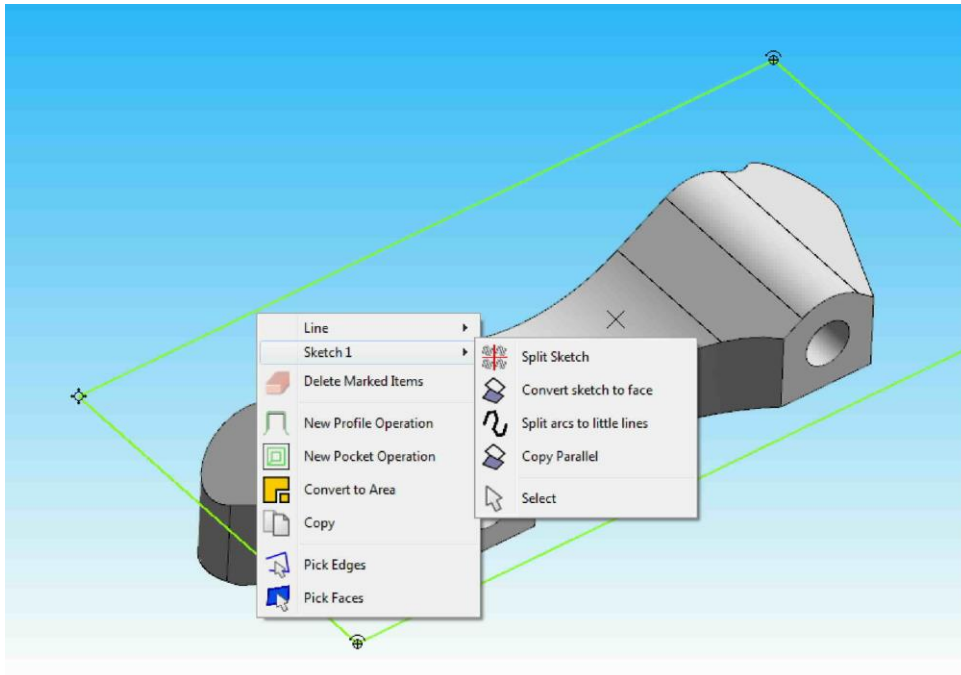
Selecting a model and right clicking will display this menu, choose the option or sub option required. If you delete the model by mistake, go to File > Undo



Pick Edges - will allow you create a Sketch from edge elements, hold down the Ctrl key to select multiple edges. When you have selected all the required edges, right click the mouse again and choose **Make to Sketch**. A new sketch object will be shown in the Objects Tree window

Pick Face - will allow you select a face, select the face and then right click again and choose **Make Sketch from Face** - this will create a Sketch from the Face that can be used in the [Profile](#) or [Pocket](#) operations

Selecting a Sketch and right clicking will display this menu along with the various sub options available

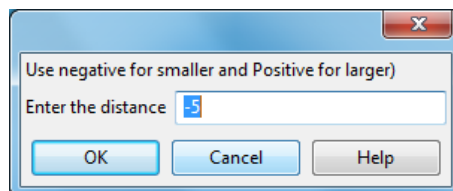


Split Sketch - will convert the sketch into lines and arcs.

Convert Sketch to Face - will convert the sketch into a Face that can be extruded for instance

Split arcs to little lines - will convert arcs to line vectors

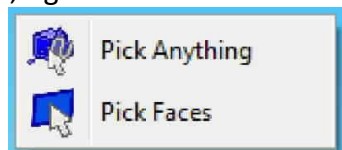
Copy Parallel - will create a new Sketch that is Offset from the original, either smaller or larger using this dialogue box



New Profile Operation - will create a new machining operation based on the selected Sketch

New Pocket Operation - will create a new machining operation based on the selected Sketch

Re-selecting objects - if you have used the option Pick Edges or Pick Faces and wish to cancel this filter and be able to pick any object, right click the mouse in an empty area of the graphics



window. This box will be displayed the previously used selection filter.

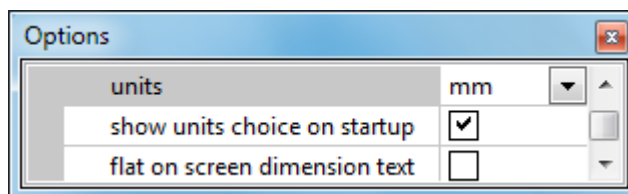
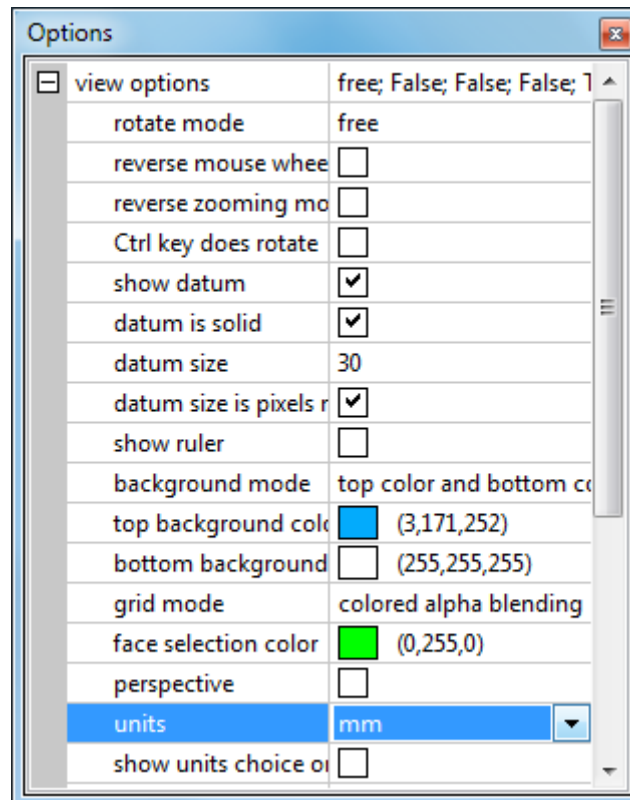
Choose **Pick Anything** to remove

System Options

System Options

This window will show settings and parameters.

To set the system units use View Options > Units



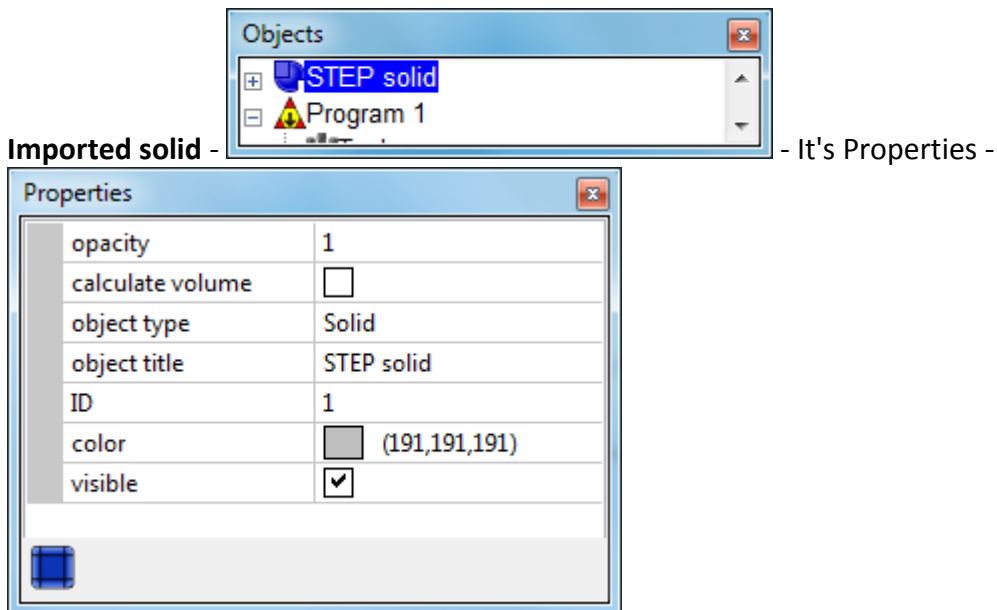
Use this option to choose the units each time you start 3DCAM.

Properties

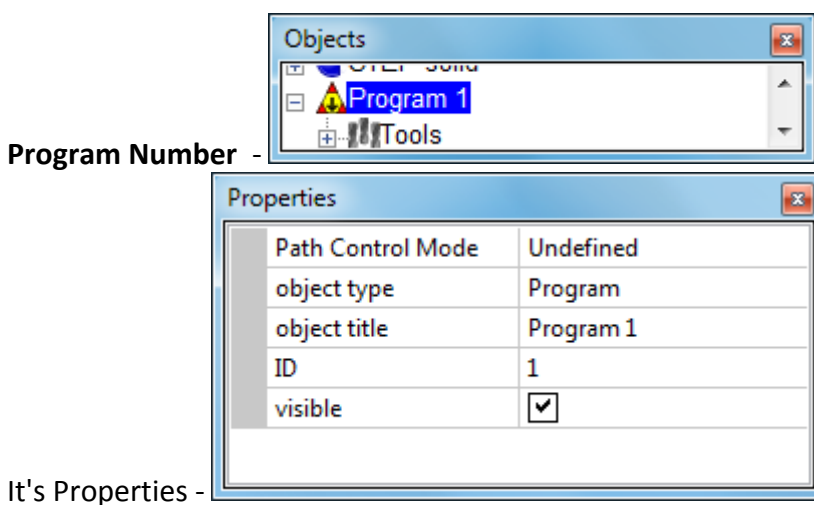
Properties

This window will display the Properties of an object chosen from the [Objects Tree Window](#).

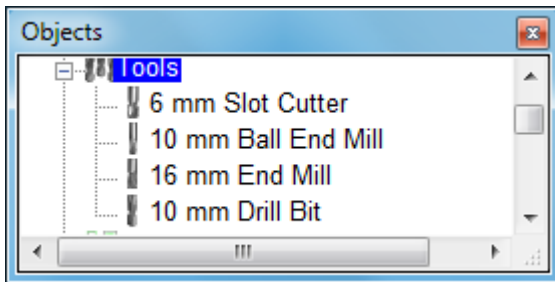
To modify or edit the Properties of an Object double click the entry in the Objects Tree Window, or use the Properties window where you can click on an individual entry and change its parameters.



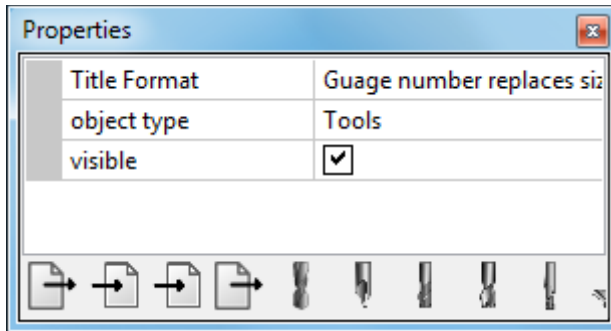
[Click here for details](#)



[Click here for details](#)

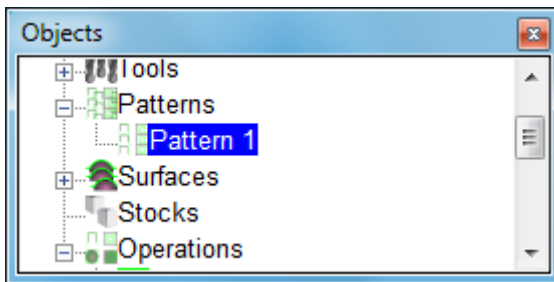


Tools -

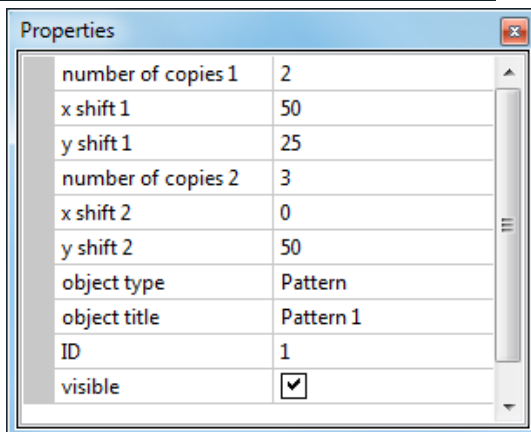


It's Properties -

[Click here for details](#)

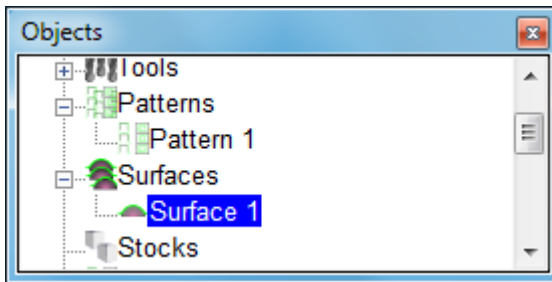


Patterns -



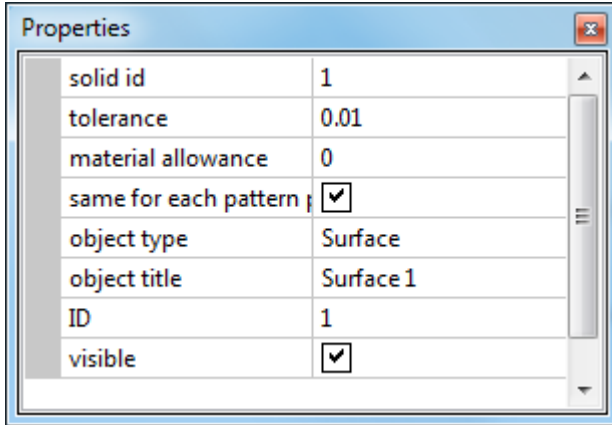
It's Properties -

[Click here for details](#)



Surfaces -

- It's Properties -



Properties	
solid id	1
tolerance	0.01
material allowance	0
same for each pattern	<input checked="" type="checkbox"/>
object type	Surface
object title	Surface 1
ID	1
visible	<input checked="" type="checkbox"/>

[Click here for details](#)

Stocks - not yet implemented

Operations - Please see the individual machining operations for details.

[Profile](#)

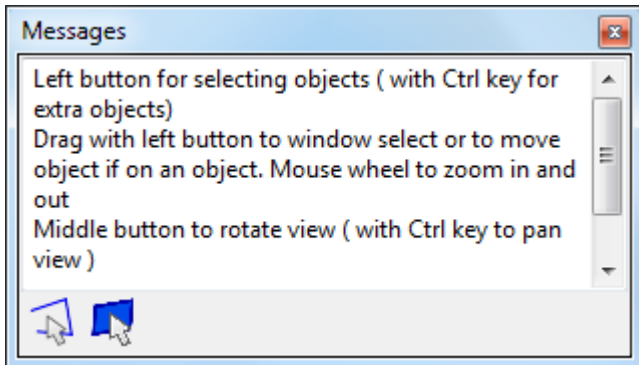
[Pocket](#)

[Drill](#)

Messages

Messages

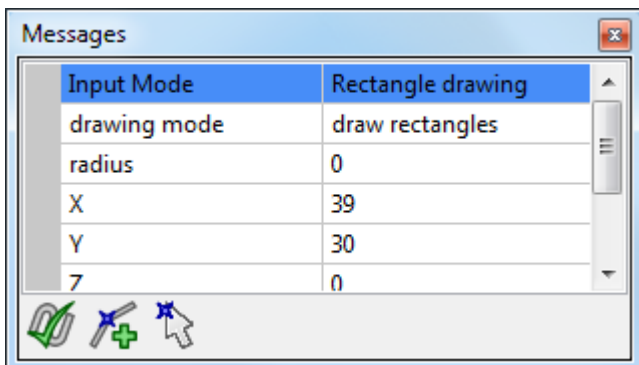
This area will display the context sensitive **help message** when performing a particular operation.



The information box can be resized by moving the mouse over any corner, click and drag to a new size. The box can also be re-positioned by click and drag on the top part of the box

The Icons shown in the above example are for picking Edges or Faces from a model.

This message box will appear when using the Rectangle draw mode from the Geometry toolbar



The 3 Icons are 1 = Accept selection. 2 = Snap to near point. 3 = Snap to cursor