



Version 14 : September 2020

Plug-Ins Tutorials



Proprietary Notice

This document contains proprietary information of Cambrio Acquisition, LLC (“CAMBRIO”) and is to be used only pursuant to and in conjunction with the license granted to the licensee with respect to the accompanying licensed software from CAMBRIO. Except as expressly permitted in the license, no part of this document may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any language or computer language, in any form or by any means, electronic, magnetic, optical, chemical, manual or otherwise, without the prior expressed written permission from CAMBRIO or a duly authorized representative thereof.

It is strongly advised that users carefully review the license in order to understand the rights and obligations related to this licensed software and the accompanying documentation.

Use of the computer software and the user documentation has been provided pursuant to a CAMBRIO licensing agreement.

Copyright © 2021 CAMBRIO. All rights reserved. The Gibbs and GibbsCAM logos, GibbsCAM, Gibbs, Virtual Gibbs, and “Powerfully Simple. Simply Powerful.” are either trademark(s) or registered trademark(s) of CAMBRIO in the United States and/or other countries. All other trademark(s) belong to their respective owners.

Portions of this software and related documentation are copyrighted by and are the property of Siemens Digital Industries Software.

Microsoft, Windows, and the Windows logo are trademarks, or registered trademarks of Microsoft Corporation in the United States and/or other countries.

Contains PTC Creo GRANITE® Interoperability Kernel by PTC Inc. All PTC logos are used under license from PTC Inc., Boston, MA, USA. CAMBRIO is an Independent Software Provider.

Portions of this software © 1994-2021 Dassault Systèmes / Spatial Corp.

Portions of this software © 2001-2021 Geometric Software Solutions Co. Ltd.

Contains Autodesk® RealDWG™ kernel by Autodesk, Inc., © 1998-2021 Autodesk, Inc. All rights reserved.

DMG MORI Models provided in conjunction with GibbsCAM © 2007-2021 DMG Mori Seiki Co., Ltd.

Contains VoluMill™ and VoluTurn™ software by Celeritive Technologies, Inc. © 2007-2021 Celeritive Technologies, Inc. All rights reserved.

This Product includes software developed by the OpenSSL Project for use in the OpenSSL Toolkit (<http://www.openssl.org/>). This Product includes cryptographic software written by Eric Young (eay@cryptsoft.com).

Portions of this software © MachineWorks Ltd.

Portions of this software and related documentation are copyrighted by and are the property of Electronic Data Systems Corporation.

Other portions of GibbsCAM are licensed from GibbsCAM licensors, which may not be listed here.

PLUG-IN TUTORIALS

This section provides an introduction to plug-in usage. There exists a separate tutorial for each plug-in that should take only a few minutes to complete. These tutorials assume user familiarity with part creation, machining, and posting parts; please review the Geometry Creation and Mill tutorials if you have not done so already.



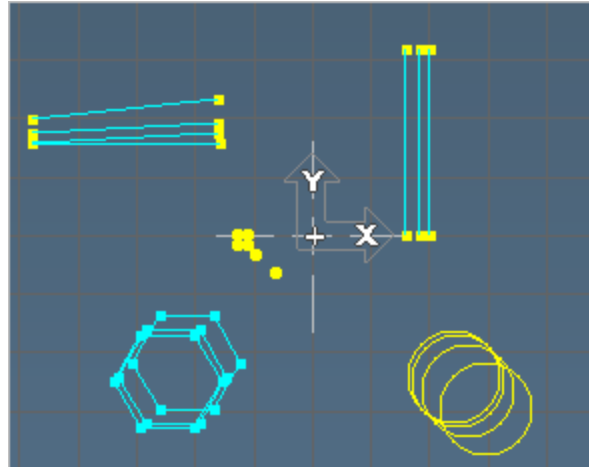
CLEANUP

This feature will delete duplicate points, circles, and lines within a given tolerance. CleanUp will also delete duplicate shapes (including splines) given that the shape is terminated at both ends (if it is not open). The user simply selects which feature type (or types) to delete, sets the tolerance and clicks the OK button. The system will keep the feature with the lowest number.

This exercise will introduce you to the use of the CleanUp Plug-in.

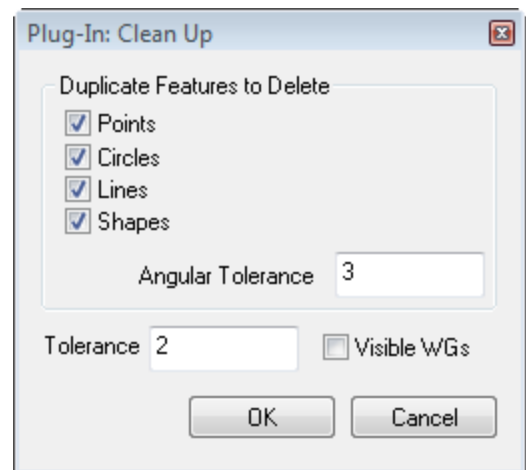
1. Open the part CleanUp.vnc.

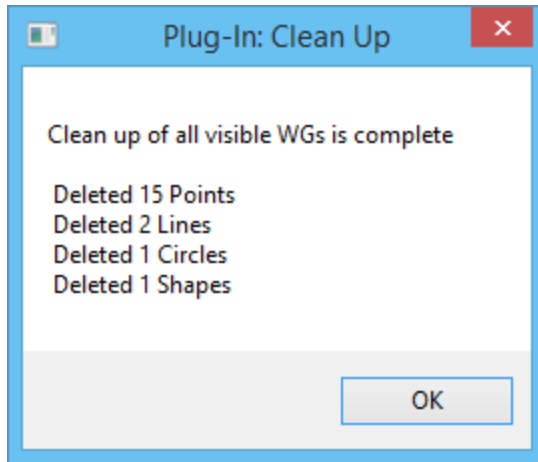
Your screen will look like the image on the right. There are 28 pieces of geometry, 7 lines, 4 circles, 3 shapes and 14 unconnected points (some of which are coincident).



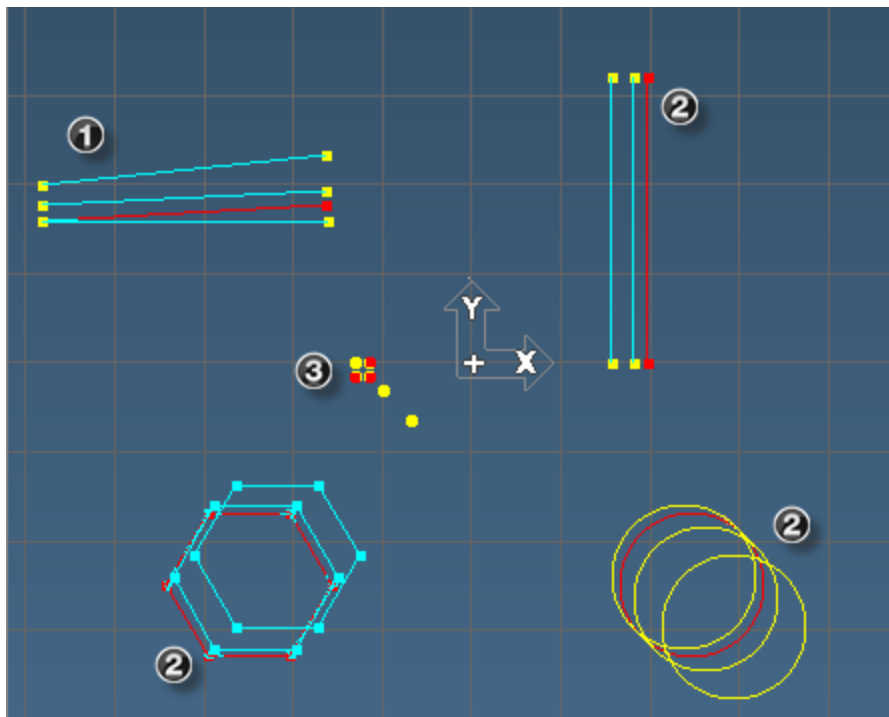
2. Choose  CleanUp from the Plug-Ins>Geometry menu.

3. Enter the values shown and Click OK.






This message will be displayed, giving details of what has been deleted. (Shown below in red.)



What was cleaned up and why?

1. The deleted (angled) line was within two degrees of another line.
2. Only one circle was within the **Tolerance** of another circle, as was the vertical line and hexagon.
3. There were coincident points as well as points within the general **Tolerance**.

CleanUp also deletes points that are used to terminate a deleted feature.

Therefore, the system returned the message that more features were deleted than originally thought. Use **Edit>**  **Undo** to go back and experiment with tolerances and features.

D CREATE D-HOLE

This feature creates single- or double-sided DHole geometry. The geometry will be created at the coordinates specified in the dialog at a depth of zero. The Create DHole plug-in does not require any existing geometry--only an open part. All parameters are entered in a single dialog.

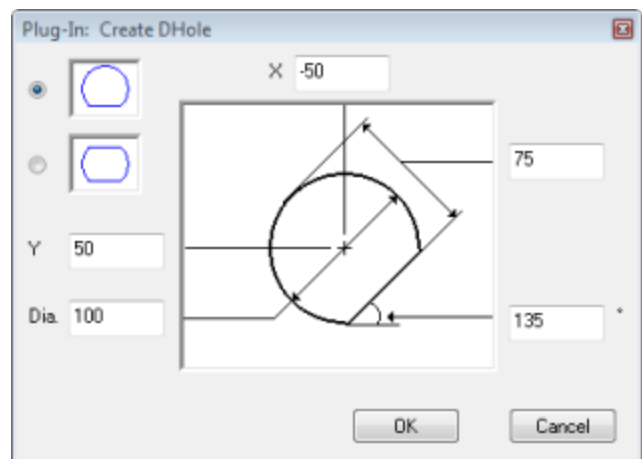
This exercise will introduce you to the use of the **Create DHole** Plug-in.

1. Create a new metric part **D-Hole.vnc** with the following dimensions: **+X125, -X-125; +Y125, -Y-125; +Z0, -Z-125**.

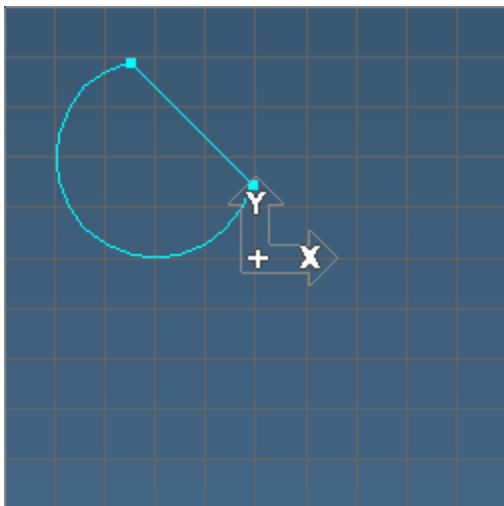
2. Select **Geometry > D Create DHole** from the **Plug-Ins** menu.

We will first create the geometry for a single-sided DHole with a 100mm diameter and a 75mm height centered at X-50, Y50. The flat will be 135°.

3. Enter the values shown.

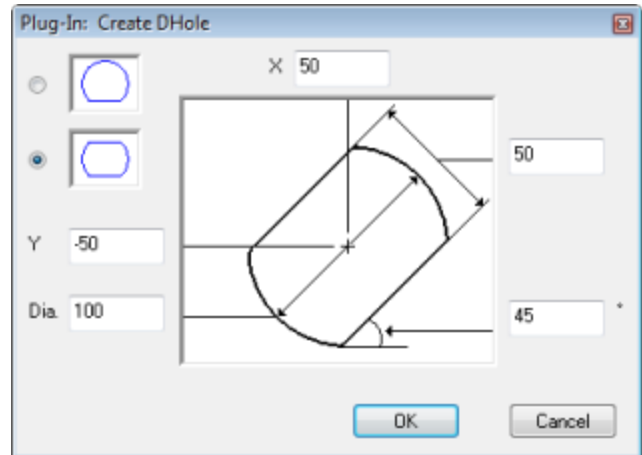


4. Click **OK** to create the D Hole.

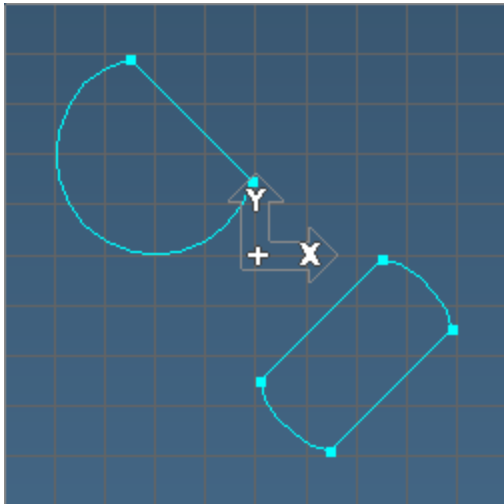


We will now create the geometry for a double-sided DHole. The hole will have a 100mm diameter with a height of 50mm. The hole will be located at X50, Y-50 and be rotated 45°.

5. Select the double-sided DHole shape and enter the values shown.



6. Click OK to create the DHole.




Once a DHole has been generated, fillets or chamfers can be added to the corners. Remember to save your part.

CREATE SPIRAL

This feature creates spiral geometry which may then be used to generate spiral cuts. The geometry will be created at the coordinate specified in the dialog at a depth of zero. The Create Spiral plug-in does not require any existing geometry—only an open part. All parameters are entered in a single dialog.

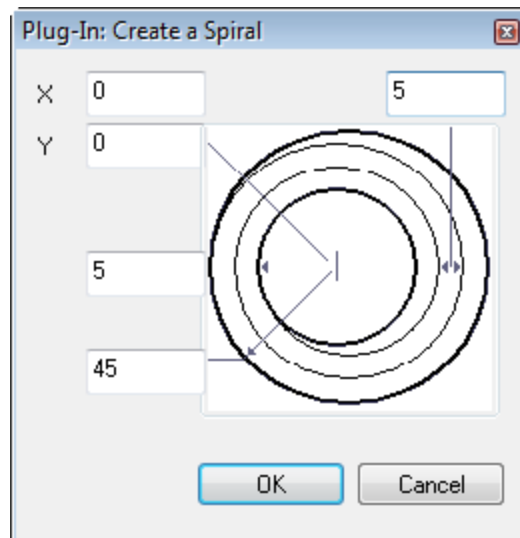
This exercise will introduce you to the use of the **Spiral** Plug-in.

1. Create a new metric part named Spiral.vnc with the dimensions **+X50, -X-50; +Y50, -Y-50; +Z0, -Z-25**.

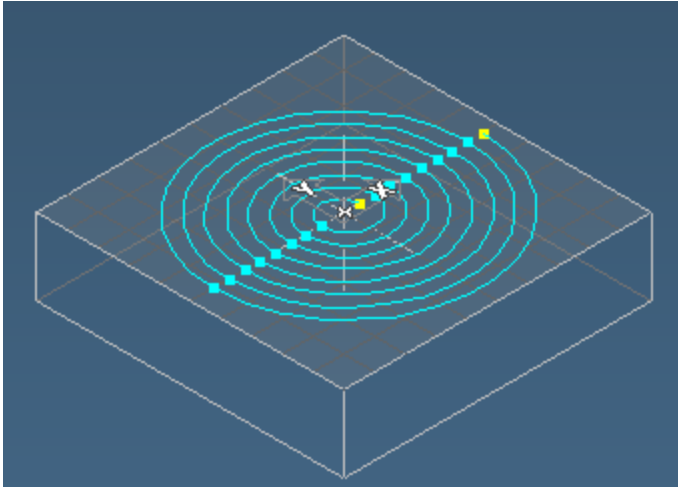
2. Select **Geometry>**  **Spiral** from the **Plug-Ins** menu.

The spiral will be centered around the part's origin. The spiral's Outer Radius is **45mm**, the Inner Radius is **5mm** and the offset between revolutions is **5mm**.

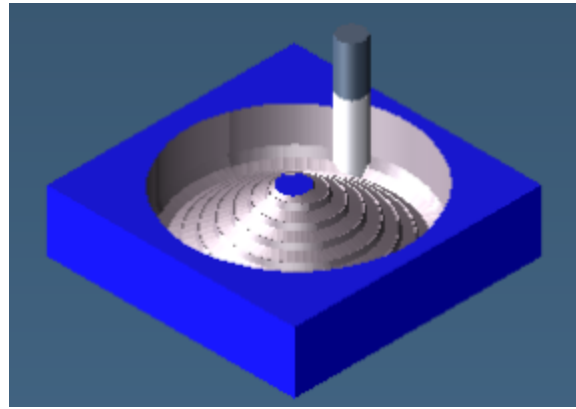
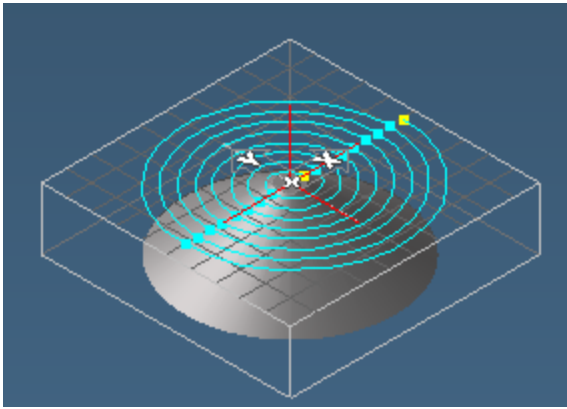
3. Enter the parameters as shown and **click** **OK**.



When viewed from the isometric view, your screen will look like the following image. Note that the geometry is centered around Z0.






This spiral is now ready to be machined. If you were to apply a Contouring process to the geometry, specify the depth you've got a spiral cut. If you have SolidSurfacer, the toolpath may be projected down onto the part.



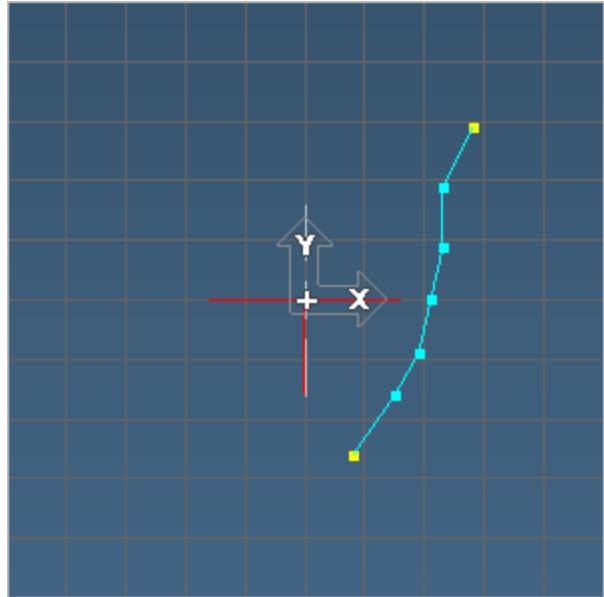
We will now create a spiral with existing geometry.

4. Delete the geometry.

5. Select Mouse Line  from the  Line sub-palette in the  Geometry Creation palette.

6. Switch to Home view.

7. Create geometry similar as shown.

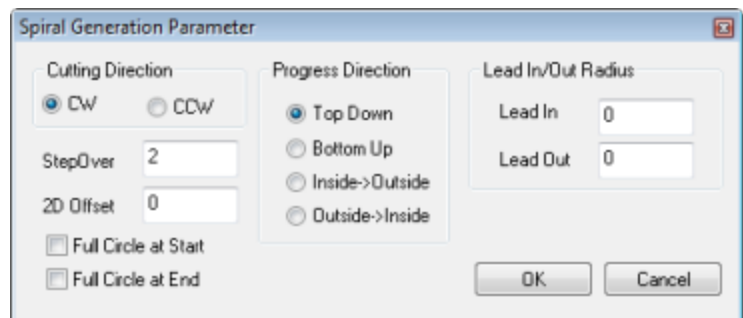


Be sure the geometry does not cross X0 (or the Y axis) as the shape will be revolved.

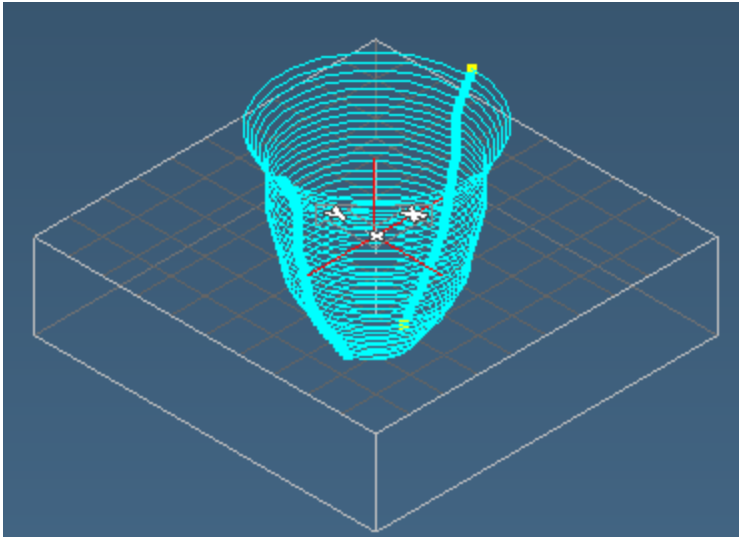
8. Select the geometry and choose  Spiral from the Plug-ins Menu.

This will open the Spiral Generation Parameter dialog, which has more options than the Create a Spiral dialog we used earlier.

9. Enter the values shown and click OK.



10. Delete the line segment for a better view of the spiral.



A spiral shaped with the profile of the selected geometry.



TAPER THREAD

This feature creates the geometry for a tapered thread. The geometry will be created at the coordinates specified in the dialog. Prior to selecting Tapered Thread, one or more points and/or circles must be selected as the centerpoint of each tapered thread. The tapered thread will be projected downward from the Z depth of the selected feature(s).

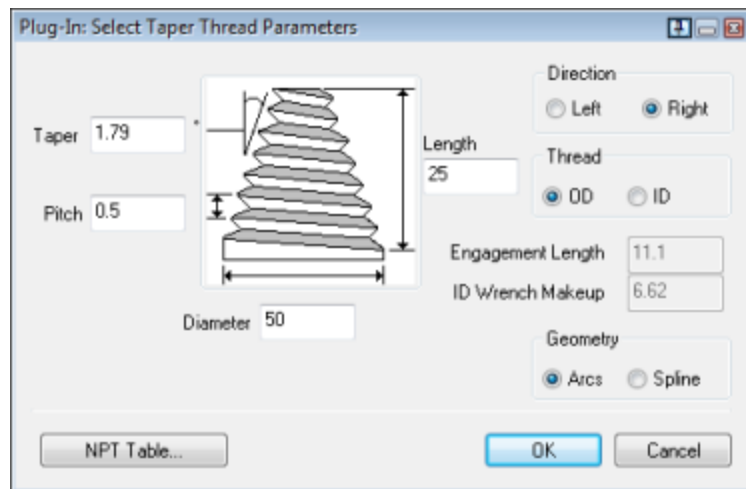
The Tapered Thread plug-in requires existing geometry. The part must contain point(s) or circle(s) that define the location of the thread(s).



This exercise will introduce you to the use of the **Taper Thread Plug-in**.

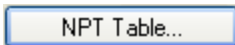
1. Create a new metric part named **Taper Thread.vnc** with the dimensions **+X50, -X-50; +Y50, -Y-50; +Z0, -Z-200** using the 3-Axis Horizontal Mill MDD.
2. Create a single point at the part's origin (**X0 Y0, Z0**) to function as the thread location.
3. Select the point you just created.

4. Select **Geometry>Taper Thread** from the **Plug-Ins** menu.



The thread will be generated largely from the information in the National Pipe Thread library that is built into the Tapered Thread feature.

- Click the NPT Table... button

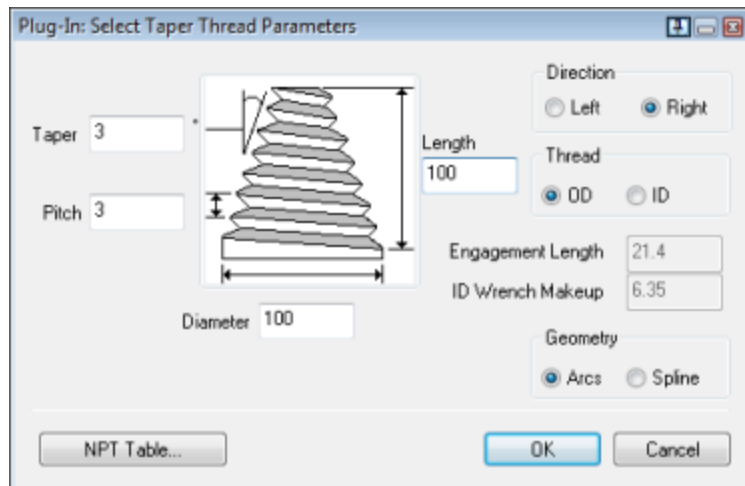


and select the 101.6mm Nominal Pipe Size as shown below and click OK.

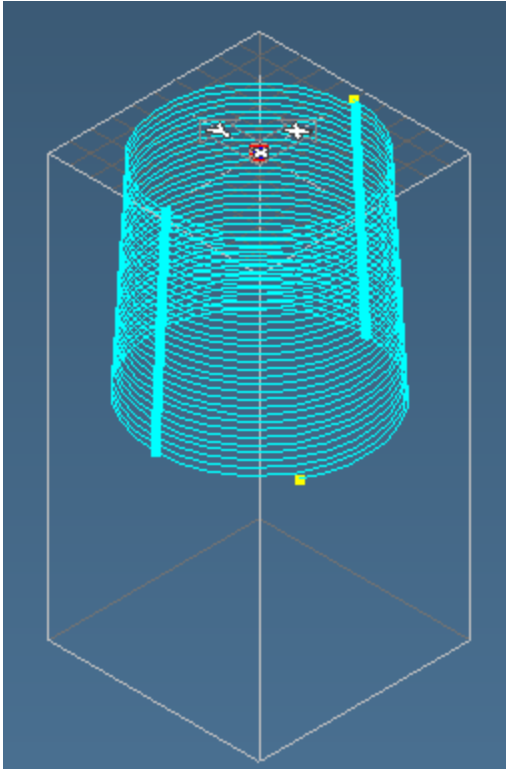
Nominal Pipe Size	Outer Pipe Diameter	Pitch	Thread Length	Taper Angle
6.4	13.700000	1.411000	15.100000	1.790000
9.5	17.100000	1.411000	15.300000	1.790000
12.7	21.300000	1.814000	19.900000	1.790000
19.1	26.700000	1.814000	20.200000	1.790000
25.4	29.200000	2.210000	25.000000	1.790000
31.8	42.200000	2.210000	25.600000	1.790000
38.1	48.300000	2.210000	26.000000	1.790000
50.8	60.300000	2.210000	26.900000	1.790000
63.5	73.000000	3.175000	39.900000	1.790000
76.2	88.900000	3.175000	41.500000	1.790000
88.9	101.600000	3.175000	42.800000	1.790000
101.6	114.300000	3.175000	44.000000	1.790000
127.0	141.300000	3.175000	46.700000	1.790000
152.4	168.200000	3.175000	49.100000	1.790000

We will modify the settings of the thread in order to get a better look at the tapered geometry.

- Enter/modify the thread's parameters as shown in the following image and click OK.



The geometry is now ready to be machined. Be sure to save the part.

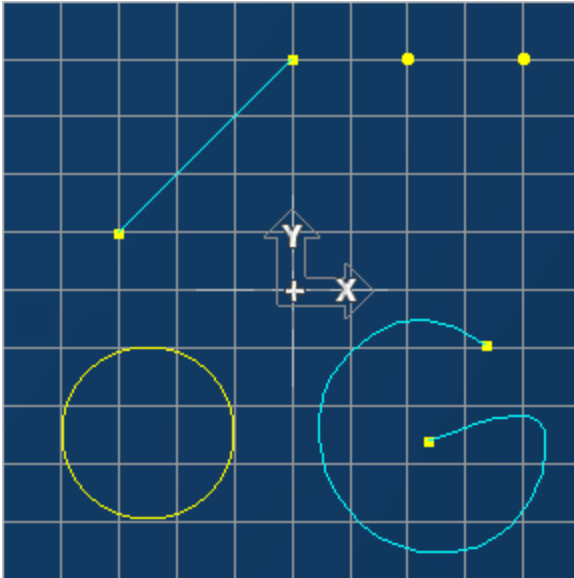


GEOMETRY PLUG-INS

The Geometry Plug-ins consist of a subset of functions for working with geometry, most of which have to do with breaking geometry into smaller components.

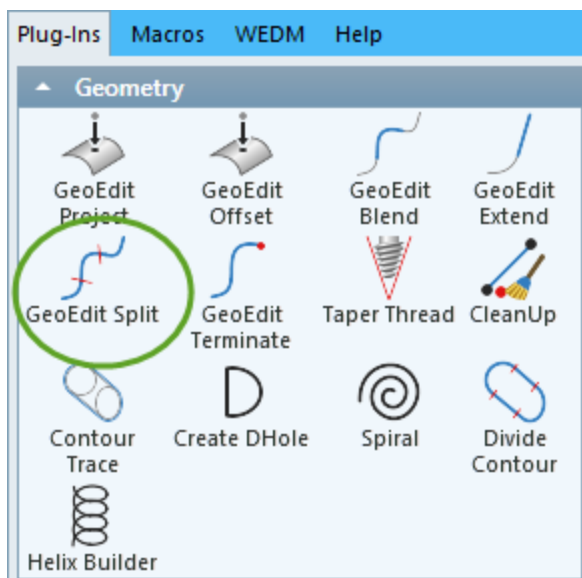
This exercise will introduce you to the use of the **Geometry Plug-in**.

1. Open part **GeoTools.vnc** from the sample part files.



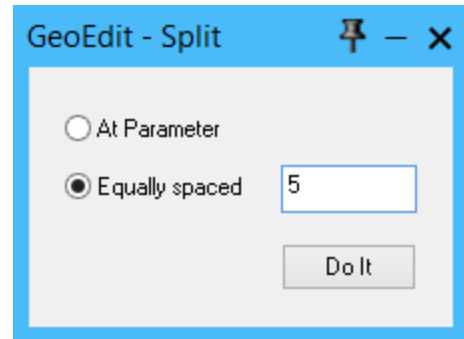
We will now use the Geometry plug-ins to manipulate the geometry.

2. **Click** the line segment to select it and then choose **Geoedit Split** from the Geometry Plug-ins menu.

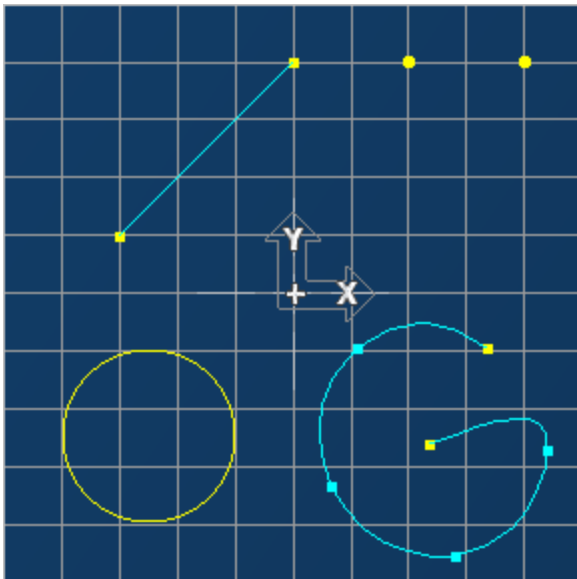


You can use this dialog in two ways, **At Parameter** and **Equally spaced**. Equally spaced will evenly divide the selected geometry into the number of segments specified.

- This command will also work on arcs and circles. Select the spline and choose the **Equally spaced** option.

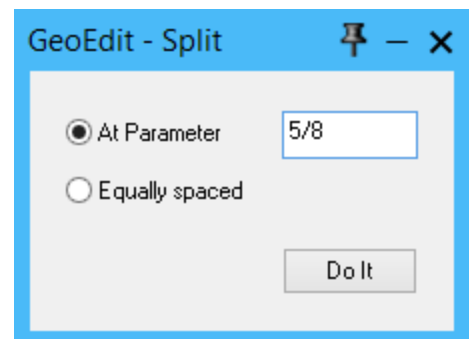


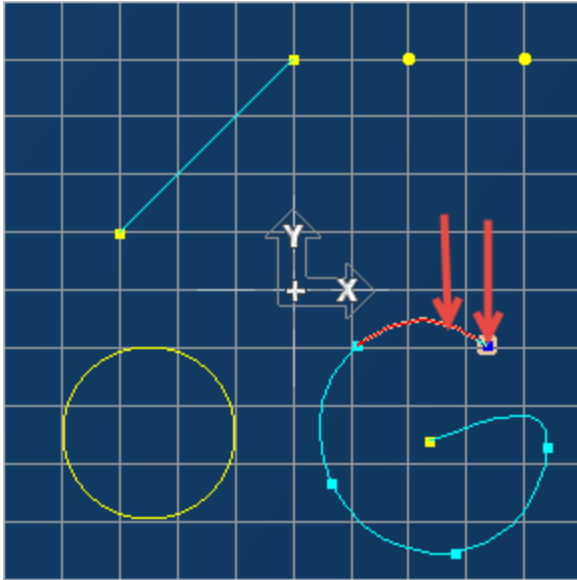
The geometry is now split into five equal segments.



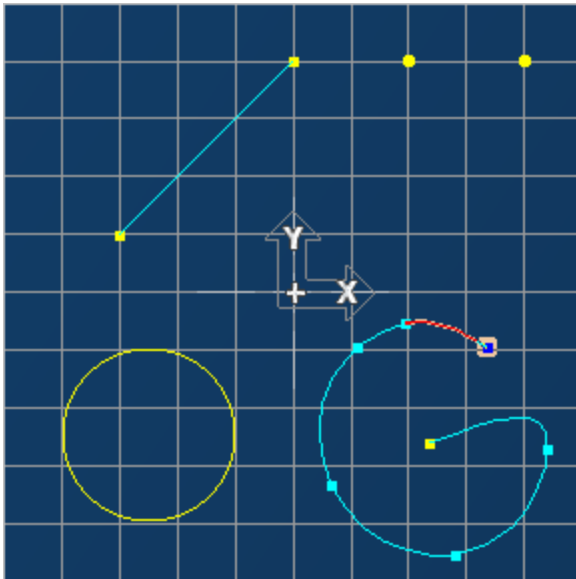
We will now split geometry at a specific parametric value. For this you must select an arc or line and a starting point, and then enter either a decimal value or a fraction. The value must be between 0.0 and 1, and to begin with you must select the end corresponding to the 0.0 value for the parameter.


- Ctrl-click first the end point, and then the arc as shown. Enter the following:

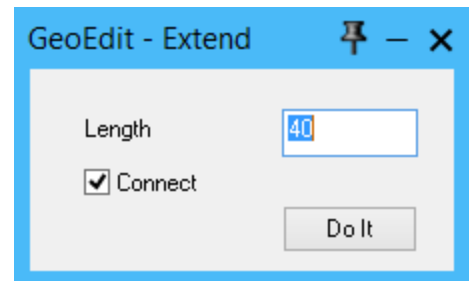




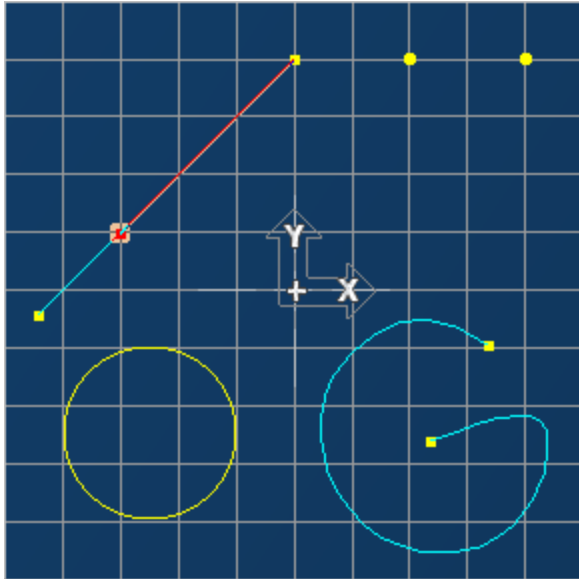
There is now a new point at 5/8 along the length of the selected arc.



5. Now Ctrl-click the line and the lower end point. From the geometry menu choose  Geoedit Extend. Enter the following:



The line is extended by 40mm and because we checked the **Connect** box, the geometry is automatically connected (blue connector point). This function will also work with curves.





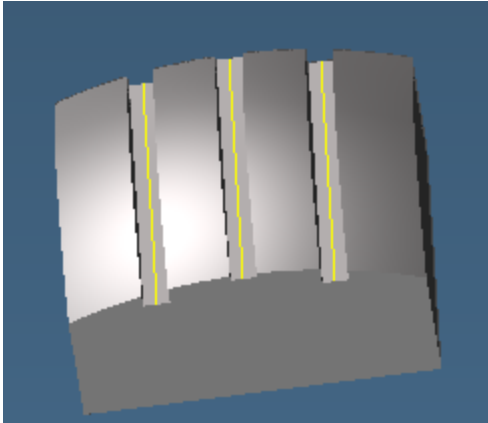


HSM PLUG-IN

The High Speed Machining (HSM) Plug-in will extend the toolpath and can put loops on the lace cut toolpath, necessary for high speed machining. This exercise will introduce you to the use of the HSM Plug-in.

1. Open part **HSM Tutorial.vnc** from the sample part files.

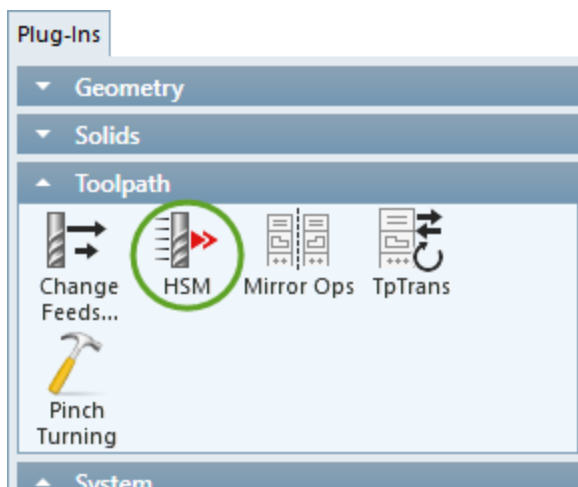
2. Click on the  Tool List and the  Machining buttons.



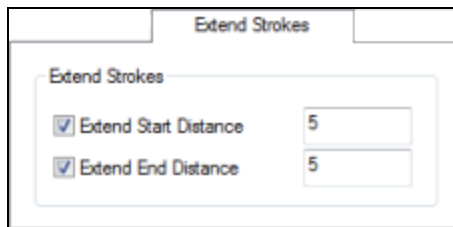
The part file has the solid as shown on the left and one tool, a 15mm ball endmill. There are four pre-defined operations that we will modify with various HSM actions. In our first operation we will extend the strokes of our toolpath.

3. Double-click Operation #1 to load the toolpath. Click the process to highlight it.

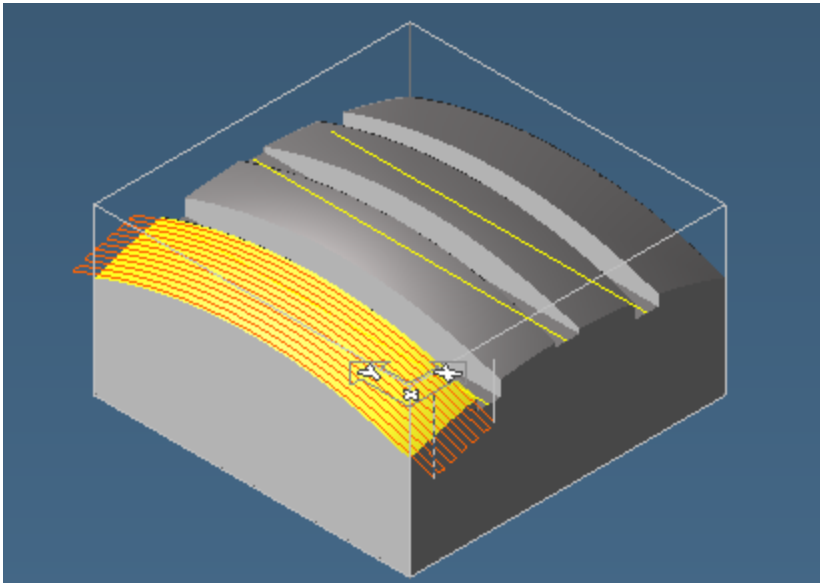
4. Select Toolpath>  HSM from the Plug-Ins menu.



5. In the Settings tab, Check Extend Strokes. We will extend the start and end distances of our current toolpath by 5mm.
6. Go to the Extend Strokes tab. Check and enter 5mm in the Extend Start and Extend End Distances as shown here.

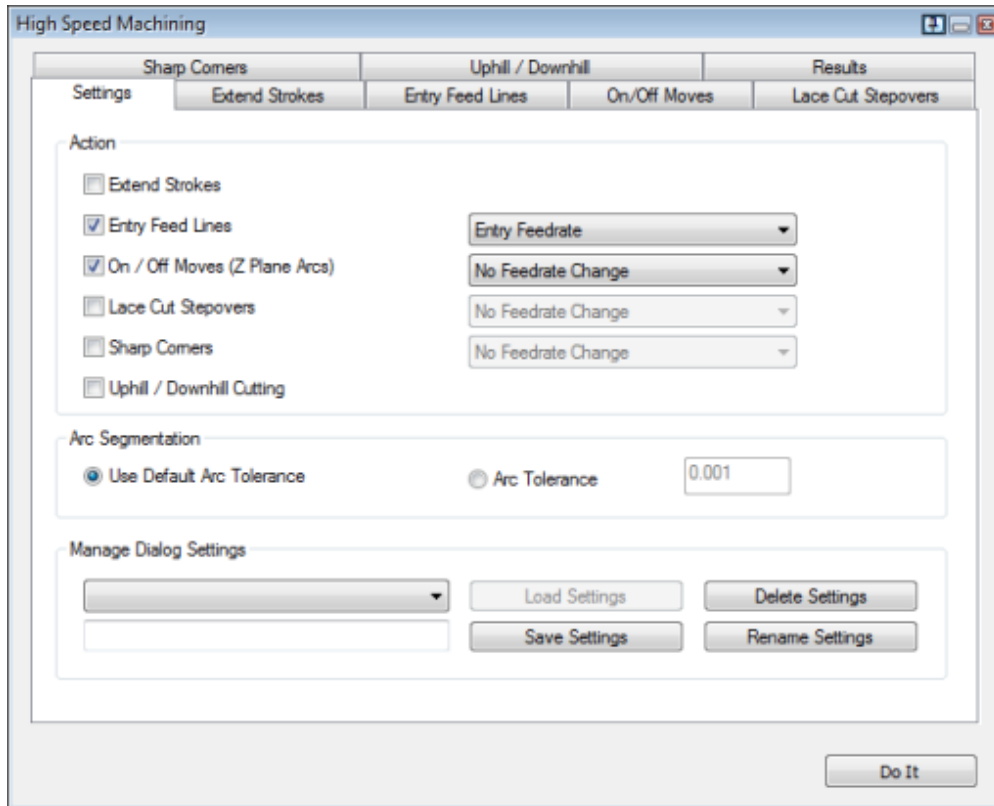


7. Click the Do It button at the bottom of the dialog. Your screen will resemble the following isometric view.

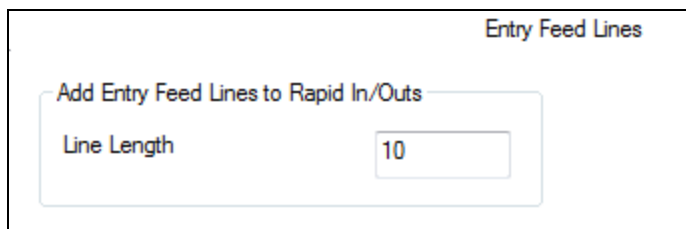


We now will change a rapid move into a feed move upon entering the part and add Z plane arcs to the entry and exit moves on our toolpath.

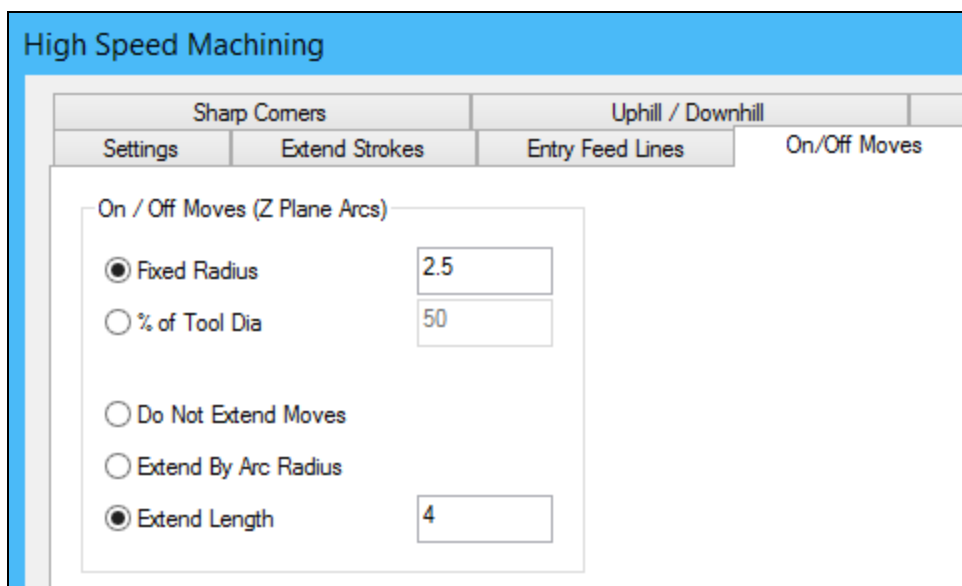
8. Go back to the Settings tab. Uncheck Extend strokes and check Entry Feed Lines . Choose Entry Feedrate from the pull down menu. Now check On / Off Moves (Z Plane Arcs), and leave the default No Feedrate Change on the pull-down menu.



9. Go to the **Entry Feed Lines** Tab and enter **10mm** in the Line Length Text box as shown here.

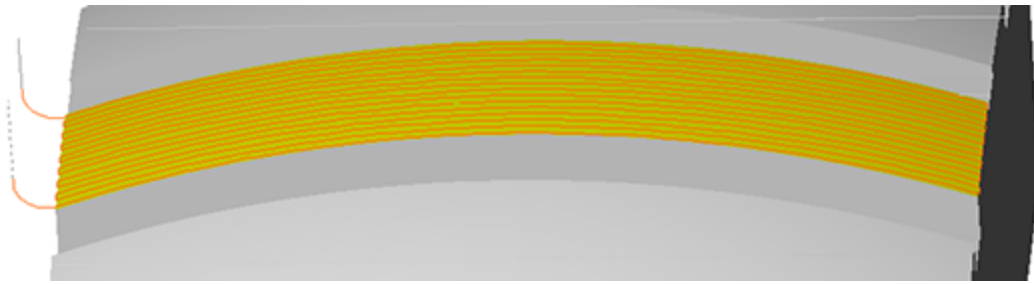


10. Now go to the **On/Off Moves** Tab. Enter the parameters shown in the following image.



11. Click Do It.

Your screen will resemble the image shown below.



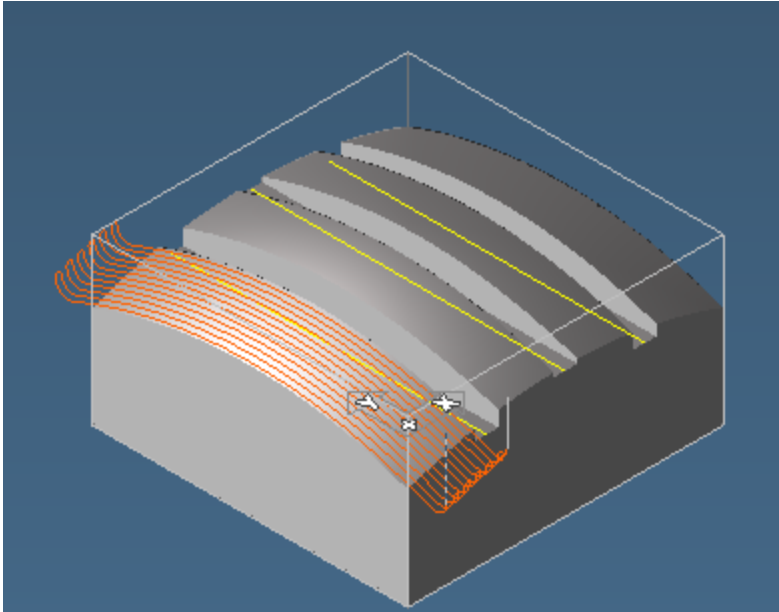
12. Return to the Settings tab. Deselect Entry Feedlines and On / Off Moves. Check Lace Cut Step Overs and Entry Feedrate from the pull-down menu.

13. Go to the Lace Cut Steppers Tab. Enter the parameters shown in the image shown here.

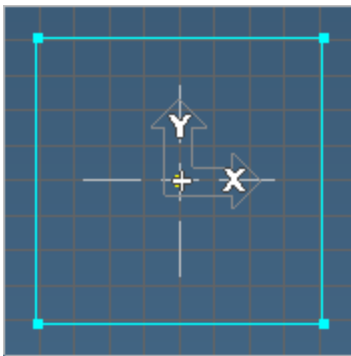
Lace Cut Steppers	
Entry / Exit Radius	
<input type="radio"/> Fixed Radius	0.1
<input checked="" type="radio"/> % of Tool Dia	50
<input type="radio"/> % of Stepper	50
Maximum Stepper Loop Radius	
<input type="radio"/> Fixed Radius	0.1
<input checked="" type="radio"/> % of Tool Dia	50
<input type="radio"/> % of Stepper	50
<input checked="" type="checkbox"/> Extend Toolpath	

14. Click Do It.

Your screen will resemble the image shown here.

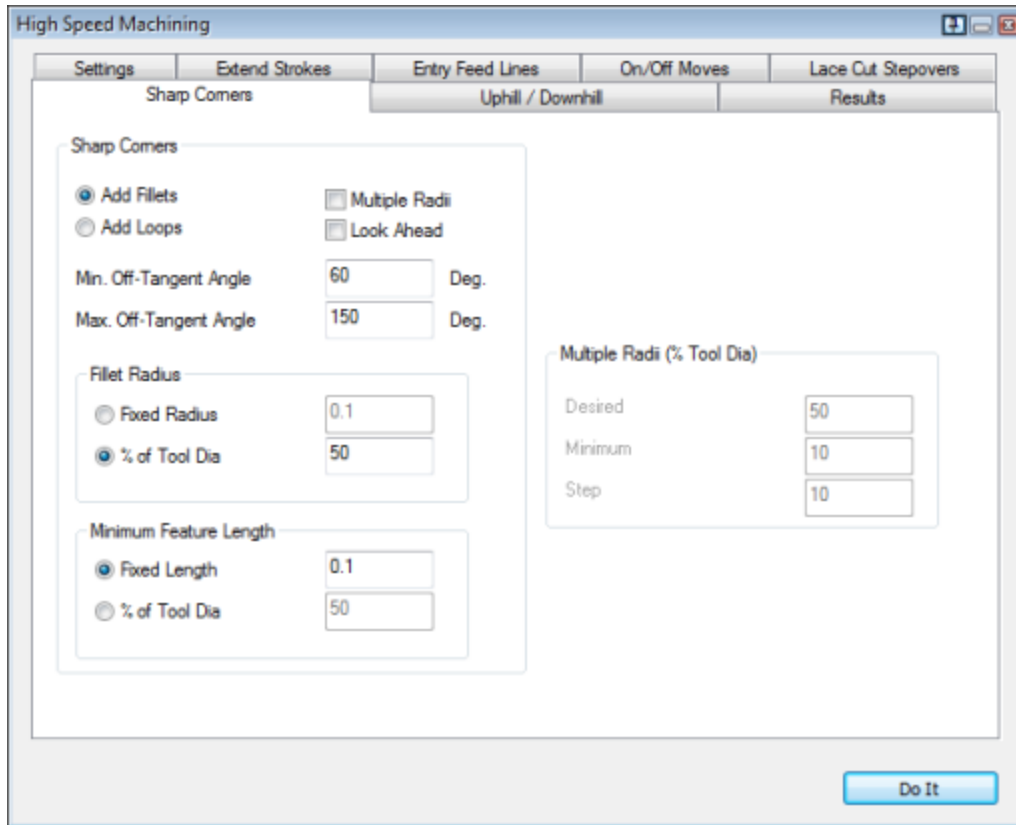


15. Open the part `Sharpcorners.vnc` from the sample part files.



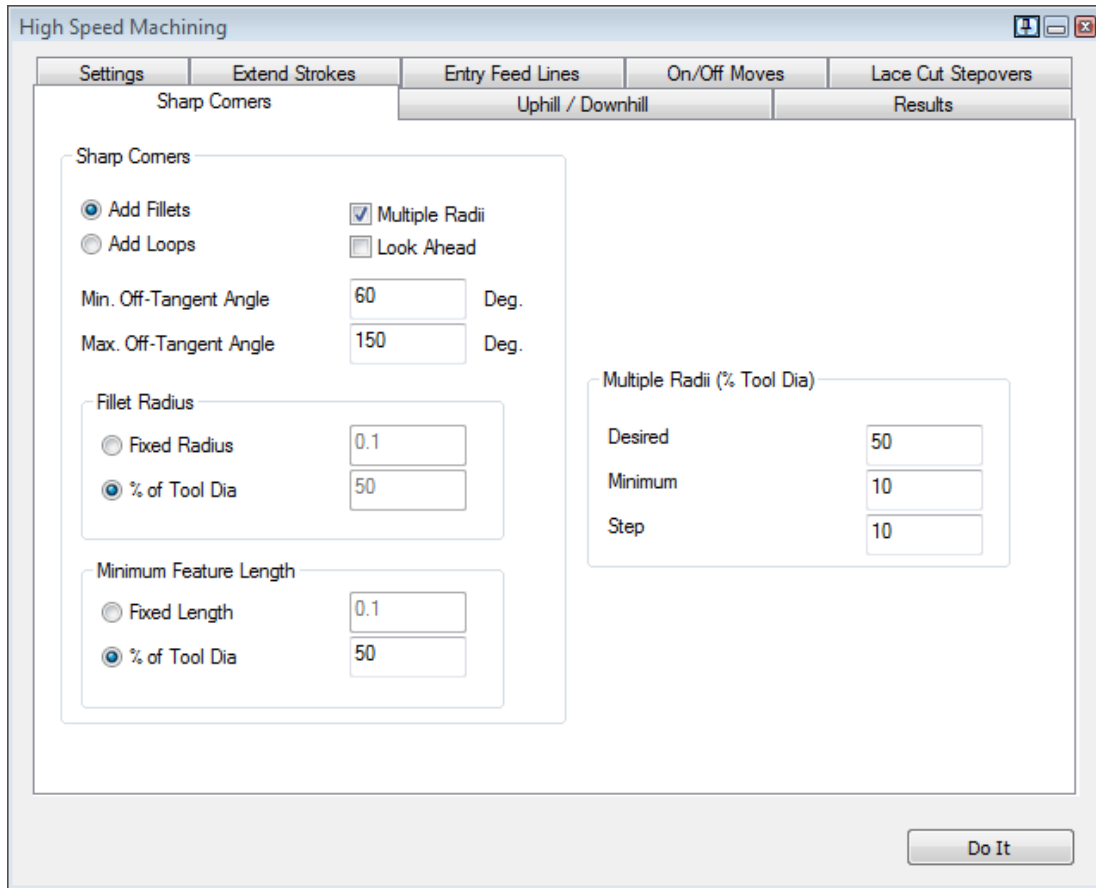
This part has geometry, a 20mm rough endmill, and three pre-defined operations that we will modify with Sharp Corners in the HSM Plug-in.

16. **Double-click** the first operation.
17. Go to the Settings tab in the High Speed Machining dialog. **Check** Sharp Corners and **%age** Contour Feedrate from the pull down menu. We will use **75%** of our Contour feedrate.
18. Go to the Sharp Corners Tab. Enter the parameters shown in the image shown below. **Click Do It** when you are ready.

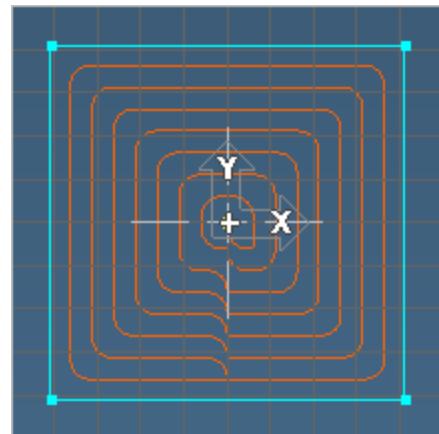


Now we will add Fillets with Multiple Radii to the sharp corners in our toolpath.

19. Enter the parameters shown in the image shown here.



Your screen will resemble the image shown on the right. A warning will appear that not all actions could be completed. A look at the results tab will show sharp corner warnings.



MIRROR OPS

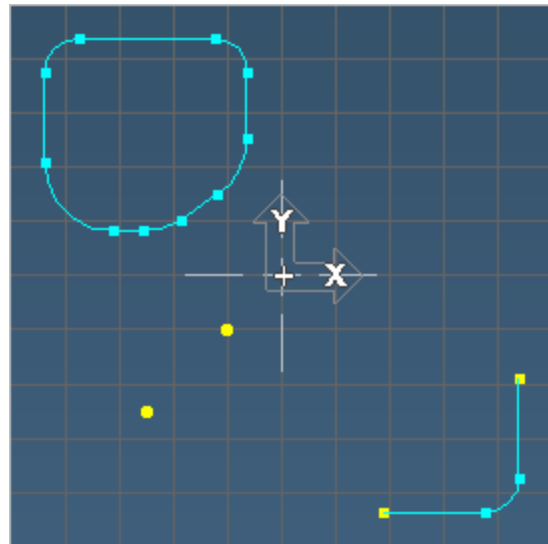
This feature mirrors selected geometry and toolpath. Select one or more ops and the Plug-in will make copies of these with the toolpath mirrored in X or Y.

This exercise will introduce you to the use of the Mirror Ops Plug-in.

1. Open part `MirrorOpsTutorial.vnc` from the sample part files.

2. Click on the  Tools and  CAM Palette buttons within Toolbar Commands.

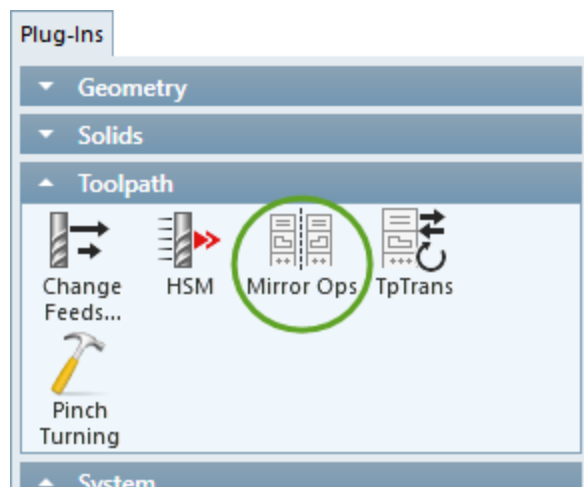
We have three separate operations to illustrate this Plug-in, one holes and two contour operations. The exercise contains three tools: a 10mm and a 12mm rough endmill and a 12mm drill. We have a closed shape, an open shape and two points for drilling.



First, we will mirror the closed shape.

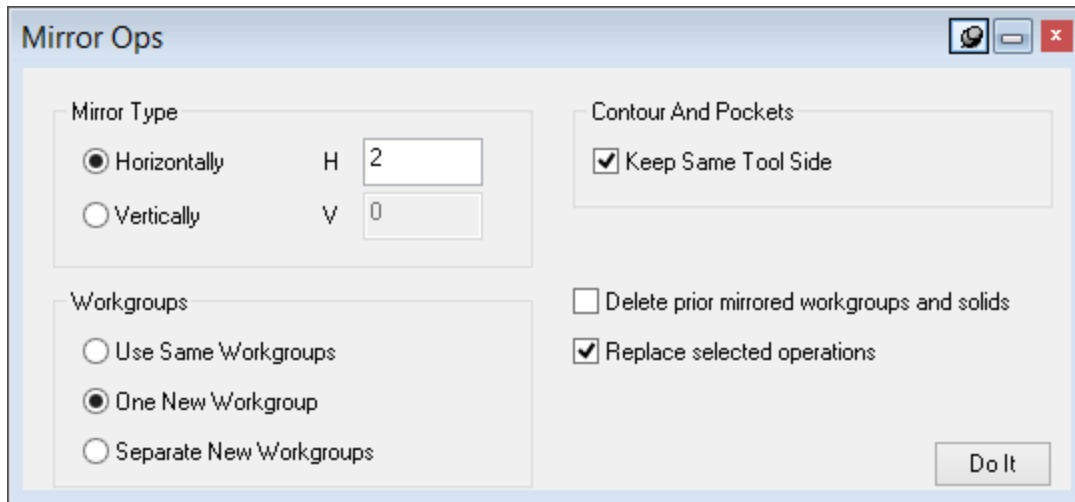
3. Double-click Operation #1, then select

Plug-Ins >  Mirror Ops.



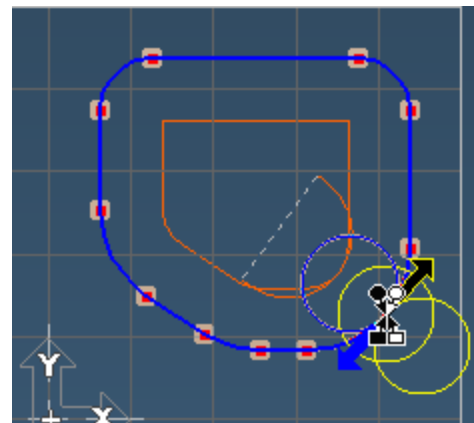
For this operation we will mirror the closed shape Horizontally. Checking One New Workgroup, will create a new workgroup called "Mirror X." We will replace the operation with our mirrored operation by checking Replace selected operations. In this operation we will keep the tool on the

same side after our mirroring operation so **check** Keep Same Tool side. Your dialog will look as below.



4. **Click** Do It.

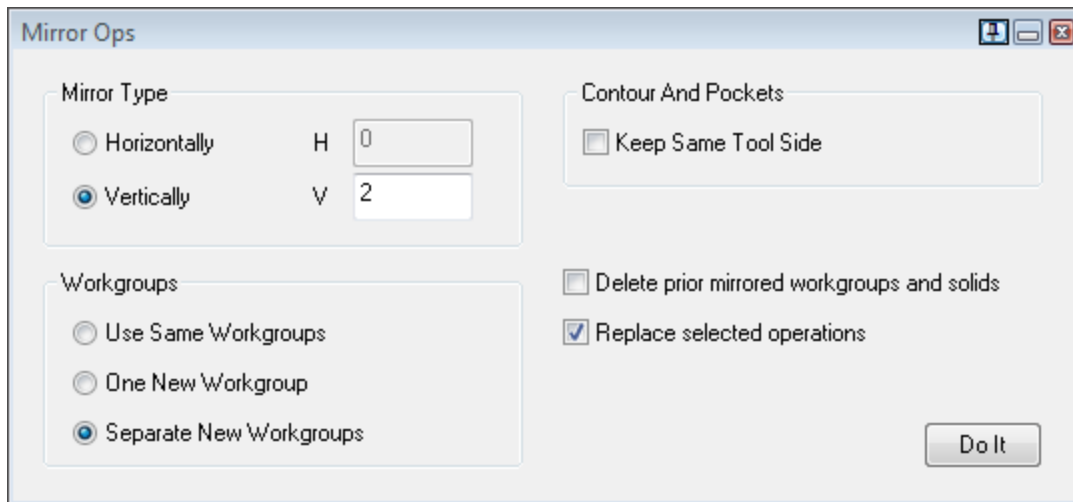
The closed shape on your will look like the image shown.



Now we will mirror the two drill points.

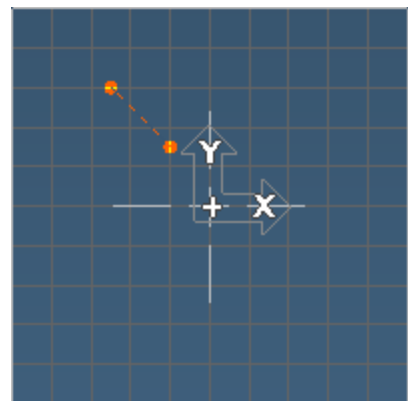
5. **Double-click** Operation #2, then select Plug-Ins > Mirror Ops.

We will mirror the two drill points **Vertically**. **Checking** Separate New Workgroups, creates a new workgroup for each workgroup used by the original geometry and will add a new workgroup Mirror Y. **Check** Replace selected operations. Your dialog should look as below.



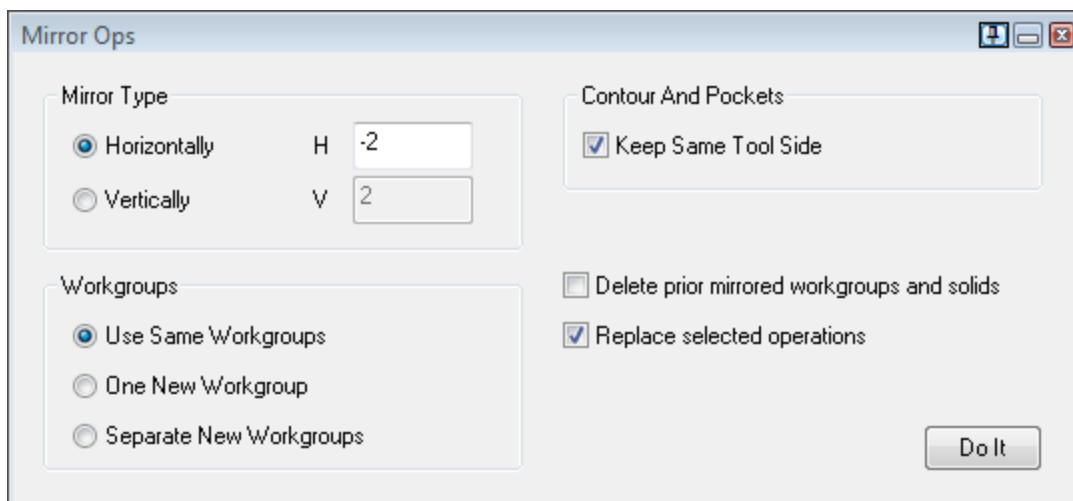
6. Click Do It.

The drill points on the screen will look like the image shown.

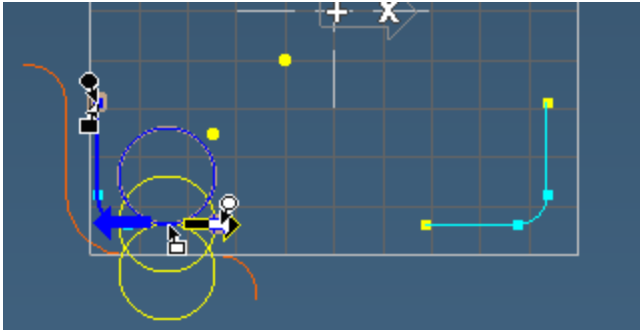


For our next exercise we will mirror our open shape Horizontally.

7. Double-click Operation #3, then select Plug-Ins > Mirror Ops. Checking Use Same Workgroups, will use our current workgroups. Check Replace selected operations and enter the value shown below. Click Do It.



The resulting toolpath will look like the image below.







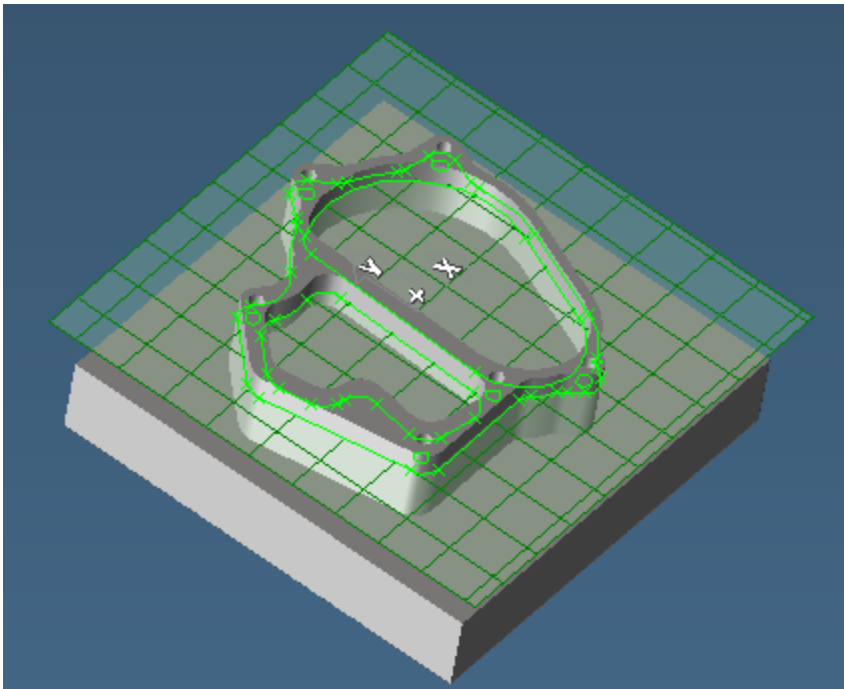
MODEL ASSOCIATIVITY

The Model Associativity Plug-in allows you to associate toolpath with a changed solid model. This exercise will introduce you to the use of the Model Associativity Plug-in.

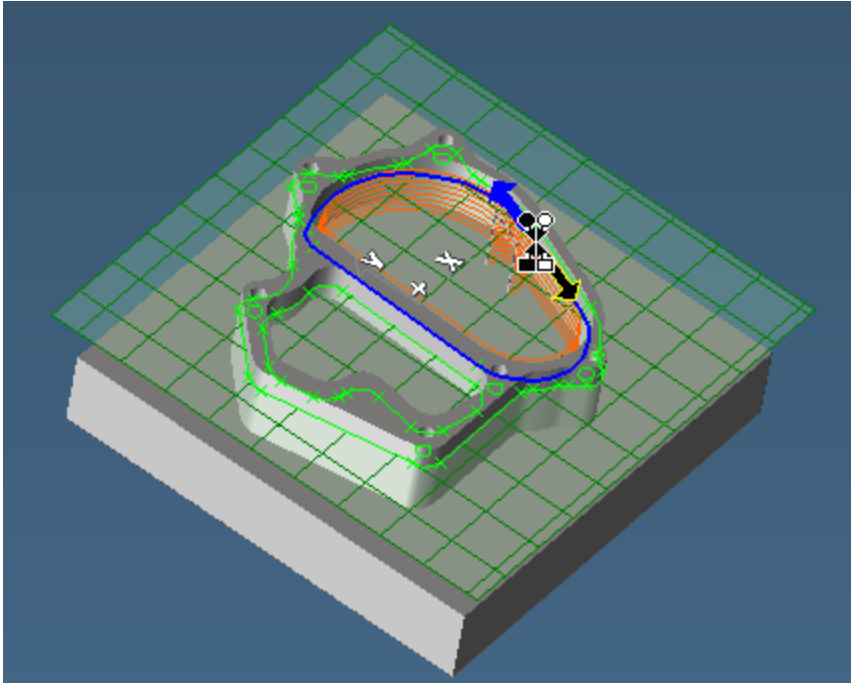
1. Open part Model AssociativityTutorial.vnc from the sample part files.

2. Click on the  Tools and the  Machining buttons.

We have two solids (one in the bodybag), one tool and one operation to illustrate this Plug-in. Our original solid is shown below.



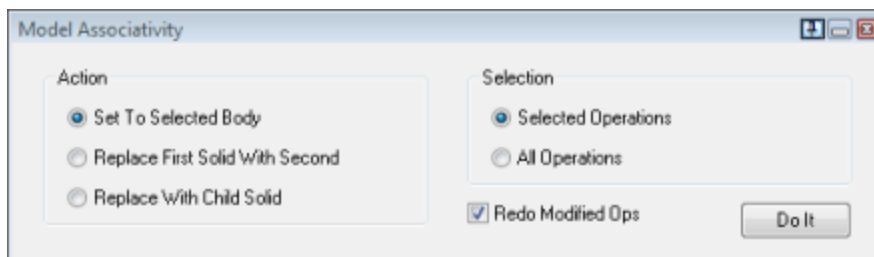
3. Double-click Operation #1 to load your toolpath.



- Now select the Modified Solid out of the Body Bag.

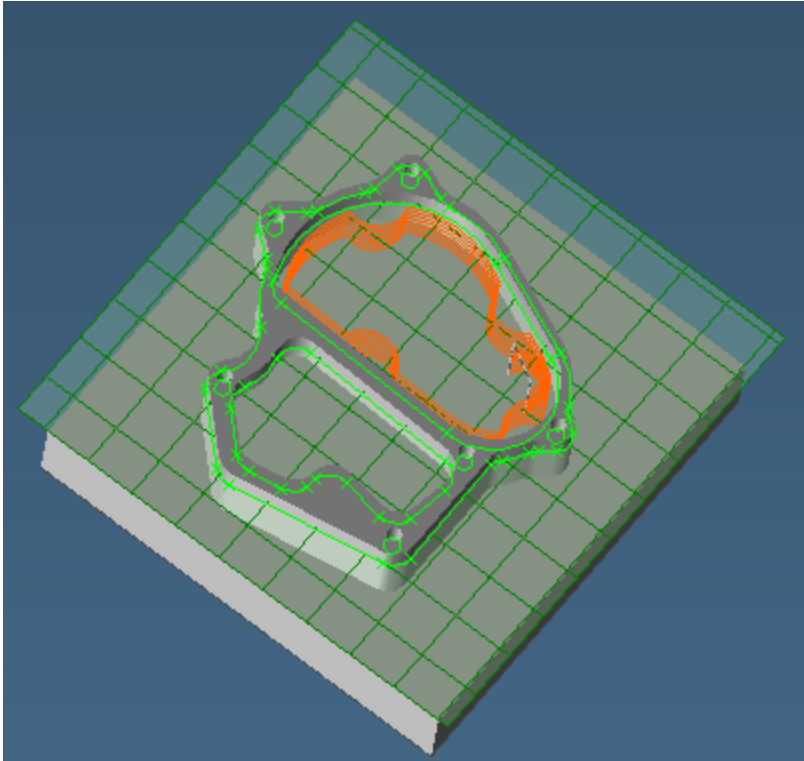


- Launch the Solids > Model Associativity Plug-in.
- Enter the parameters as shown in the image below.



- Click Do It.

You now have an updated toolpath that resembles the image shown here.

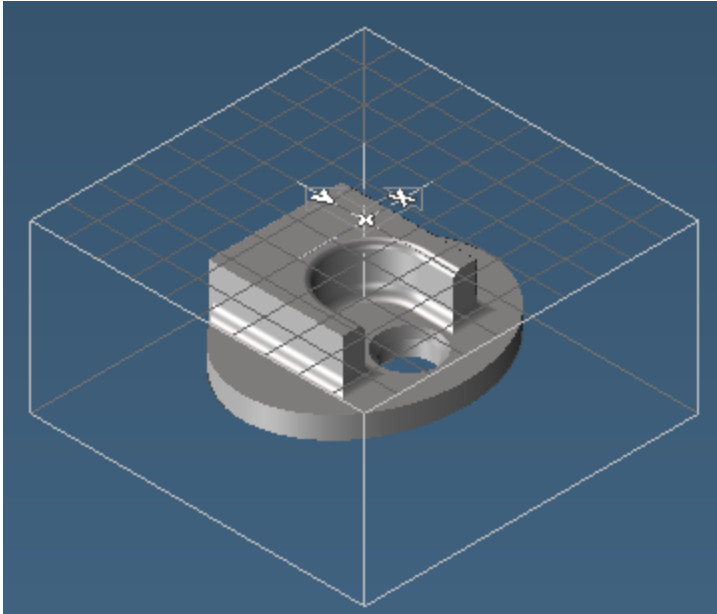


Project Geometry

The Project Geometry Plug-in will project points, lines, and contour shapes onto bodies, modifying their depth and/or shape as needed.

This exercise will introduce you to the use of the Project Geometry Plug-in.




Open the metric part named Project.vnc.

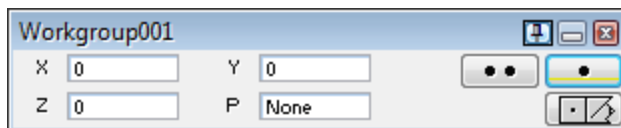


Your screen will look like the image on the left. As you can see, the solid lies beneath the plane at $Z=0$.

Project a point

We will now create a point at a depth of zero and then project that point down onto the solid.

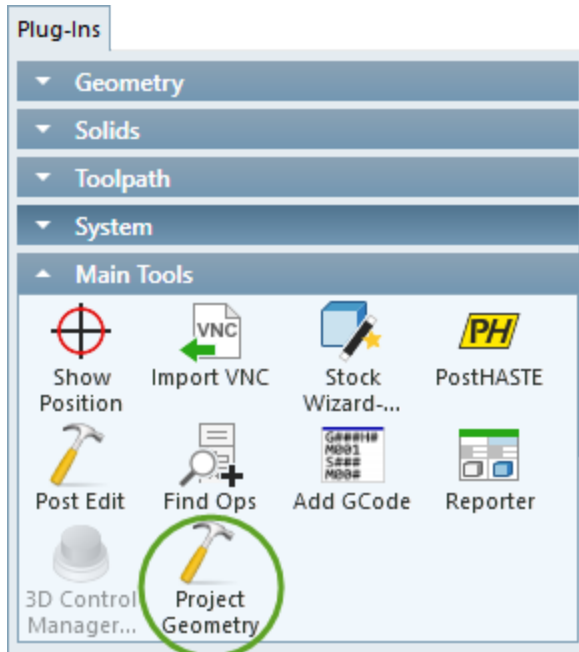
1. Create a point at the origin ( -  - ).



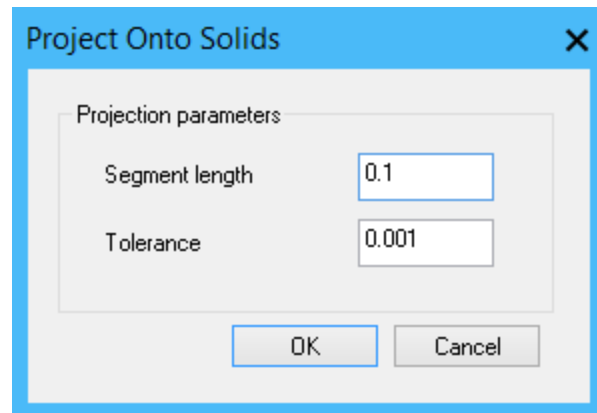
2. With the point and the solid selected, activate the plug-in by selecting **Plug-Ins > Main Tools >**

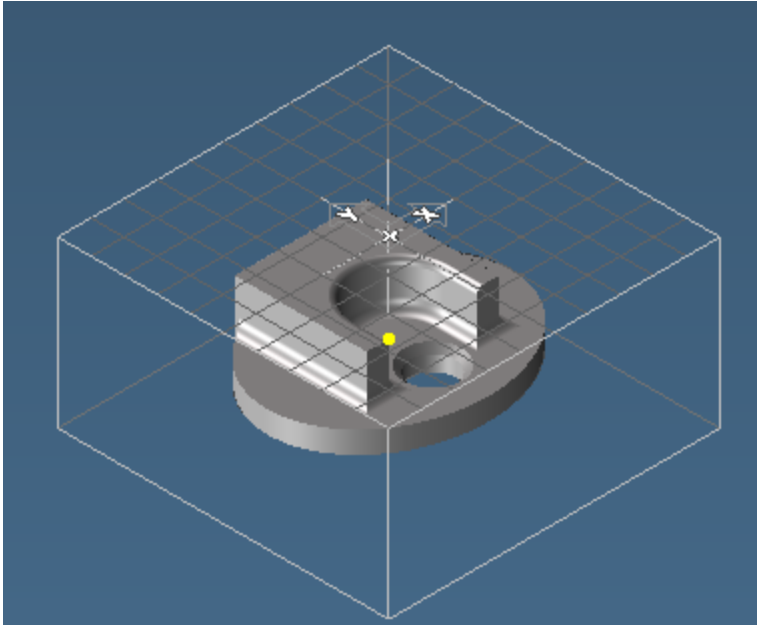


Project Geometry from the **Plug-Ins** menu.





Accept the defaults on the parameters dialog

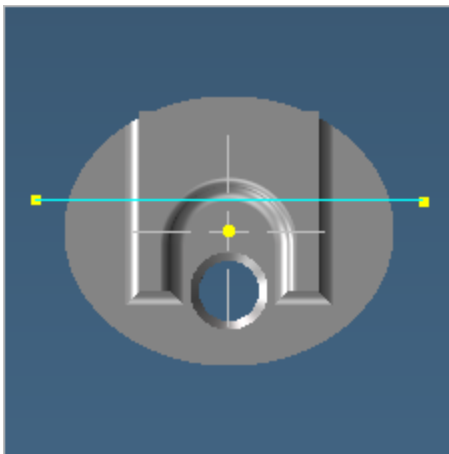





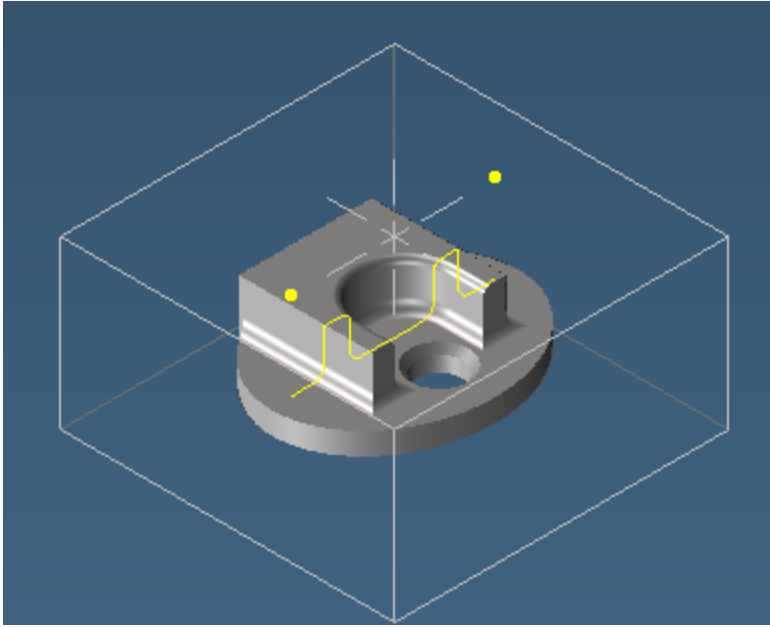
The point now lies on the solid.

Project a line

1. Delete the projected point. Change the view to Top view and using the mouse line tool ( - ) , create a line segment like the one shown below at a depth of zero.



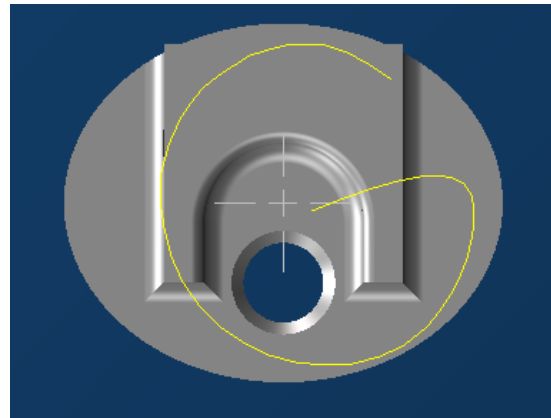
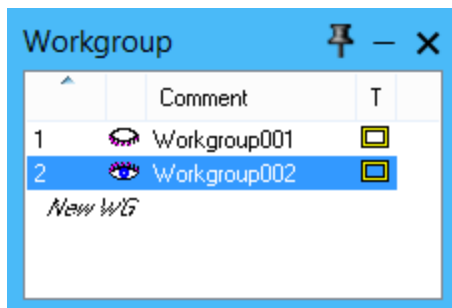
2. With both the line segment and the solid selected, activate the plug-in by selecting **Main Tools > Project Geometry**  from the **Plug-Ins** menu. Please note this could take some time - check the Progress on the progress bar at the bottom of the screen.



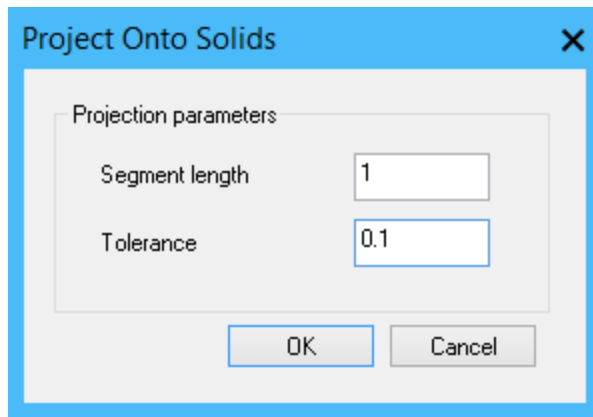
Note that only the line becomes projected while the endpoints remain where they were.

Project a curve

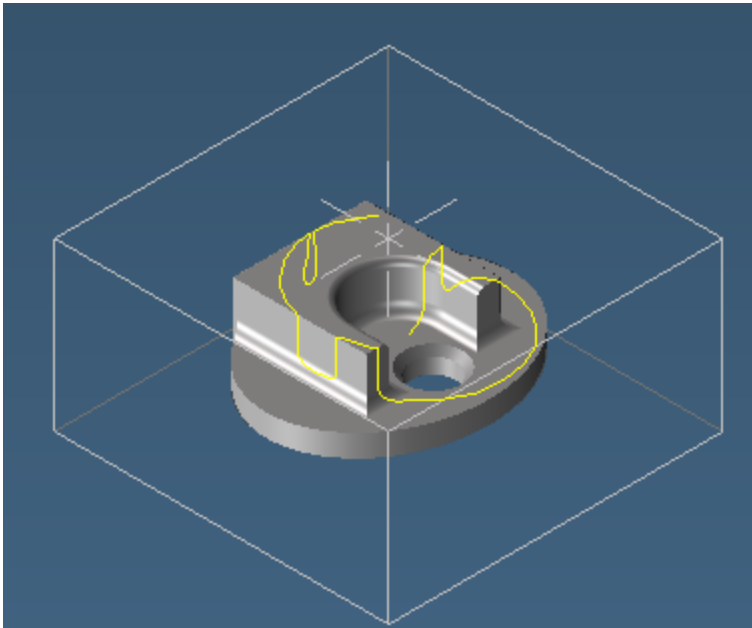
1. Delete the projected line and extraneous endpoints. Switch to **Workgroup002**, which contains an open contour.



2. With both the contour and the solid selected, activate the plug-in by selecting **Project Geometry** from the **Plug-Ins** menu.
3. Change the **Settings Segment length** to as shown below:



Your screen will resemble the image below.





Reports that are customized to the needs of your business can be very helpful to your records and operations. This exercise provides an introduction to making custom reports. Please note that to add Reporter templates to the GibbsCAM folder you must have Administrator access to the computer.

The report we will make is relatively simple, and is a combination of the existing Part and Operation Reports. The report will not be as detailed as either report, but will provide a more complete overview of the part.

This exercise will consist of customizing an Excel file (the Model file) and generating the text with the desired commands for the report (the Template file). For example files that match this exercise, see [Plug-Ins/Tutorial Files - Required/Production/MillSampleMaster.xls](#) and [Plug-Ins/Tutorial Files - Required/Production/MillSampleMaster.txt](#).

There are four steps to making a custom report:

1. Determining what functions you need in the report.
2. Making the Excel Model file (the report's layout).
3. Making the Template text file which contains the commands for the necessary functions.
4. Testing the report.

Step 1

Included in the report will be:

the user's name	the current date	the name of the part
the units of measurement	the stock material	the machine type
the post processor	the NCF file name	the min and max values
the overall part size	the operation number	the operation type
the cut type	the depth of cut	the tool numbers
the tool's diameter	the type of tool	the tool's CRC number
a graphic of the start and end condition of each operation		

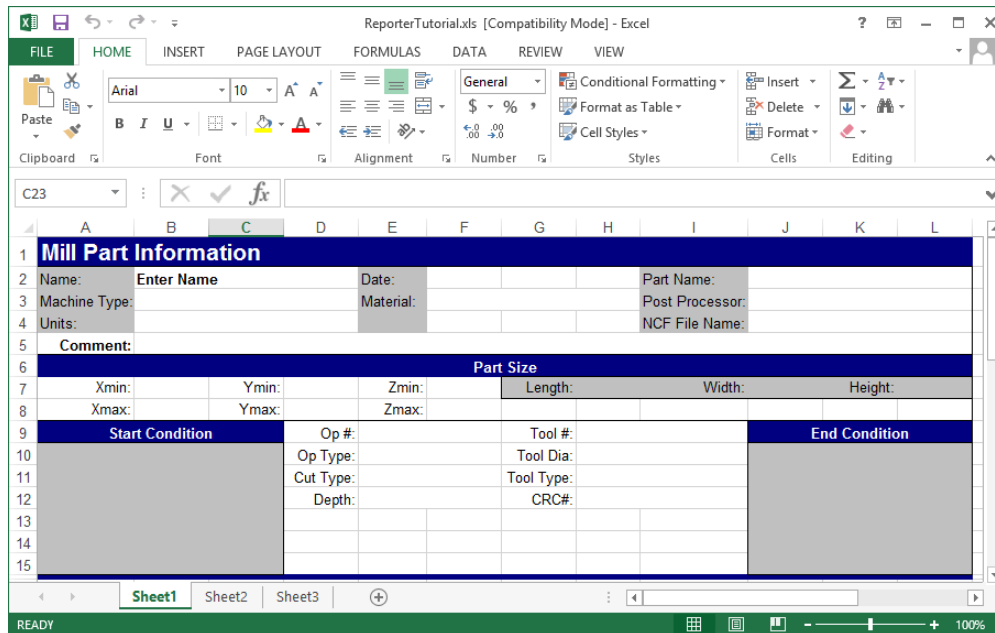
Step 2

We have a partially completed Model file that you will need to complete. The file consists of all the labels for the items to be output in the report and their position. You will learn how to enter certain items and format the Excel document to make it more presentable.

If you need more information on working with Excel, try the program's online help feature or look through the following websites for tips:

- <http://www.microsoft.com:80/office/excel/using/default.asp>
 - <http://www.j-walk.com/ss/excel/index.htm>
1. Open ReporterTutorial.xls (located in the Parts folder).

Most of the document has been formatted; however, there is still some work to do. The first thing we will do is change the user name to your name.



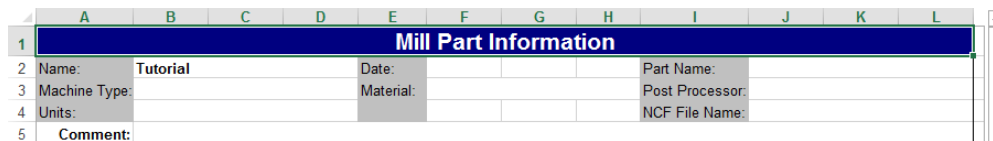
2. Select cell B2 replace Enter Name with your name.

Simply select the cell and type. There may be some instances where you will need to enter information in the Formula Bar. The Formula Bar is located directly below the toolbar(s) and will display any text entered in a cell.

The next thing we will do is center the title over the width of the spreadsheet.

3. Select cells A through L on the first row. Click the Merge & Center button  in the Home/Alignment tab.

The result will look like the following image.




We will now enter calculations the report will perform. This will include automatically inserting the current date and determining the overall stock size from the Min and Max values in a part's Document Control dialog. We will start with the current date.


4. Select cell F2 and in the Formula Bar enter `=Now()` and click .

The report will automatically return the current date and time. To enable this to be displayed correctly, select cells f2 and g2 and choose Home>Cells>Format>Format Cells and in the Alignment Tab, check the Merge cells option.


Next we will calculate the part's stock Length.

5. Select cell H7 and enter $=(B8-B7)$ and click .

Now we will calculate the part's stock Width.

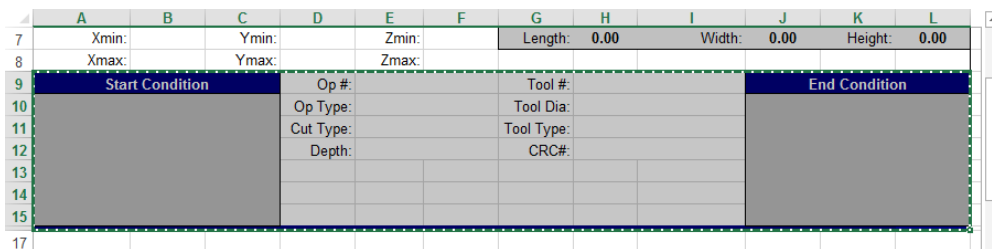
6. Select cell J7 and enter $=(D8-D7)$ and click .

Next is calculating the part's stock Height.

7. Select cell L7 and enter $=(F8-F7)$ and click .

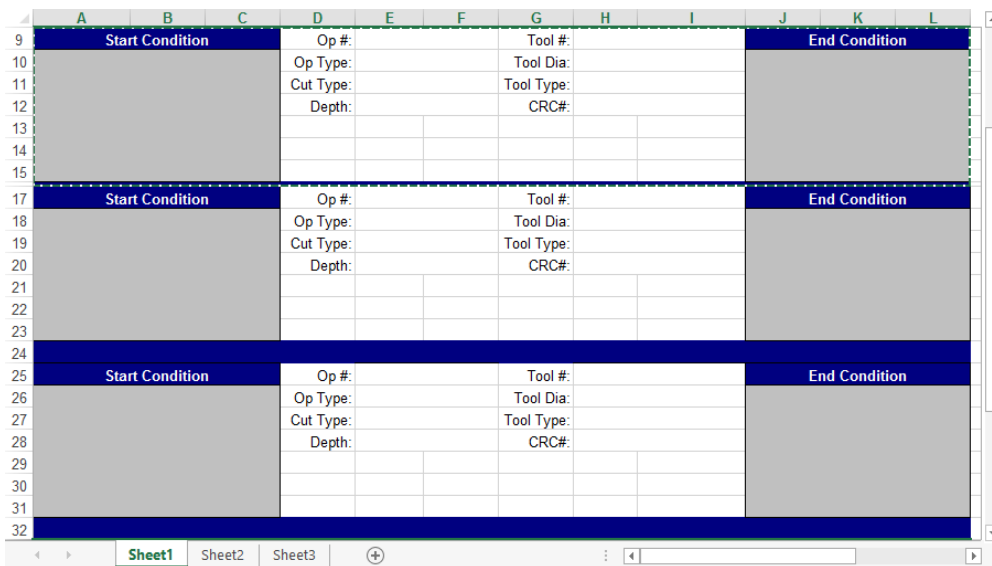
The next step will be to duplicate the operation data section of the report. The report will need a complete duplicated section for each operation in a part. For the purposes of this tutorial, duplicating this section twice will be sufficient.

8. Select cells A9 through L16. Copy this data by using **Ctrl+C** or choose Home>Clipboard>Copy from the menu. The cells will now be highlighted, as shown.



A	B	C	D	E	F	G	H	I	J	K	L
7	Xmin:	Ymin:	Zmin:			Length:	0.00	Width:	0.00	Height:	0.00
8	Xmax:	Ymax:	Zmax:								
9	Start Condition		Op #:	Tool #:		End Condition					
10			Op Type:	Tool Dia:							
11			Cut Type:	Tool Type:							
12			Depth:	CRC#:							
13											
14											
15											

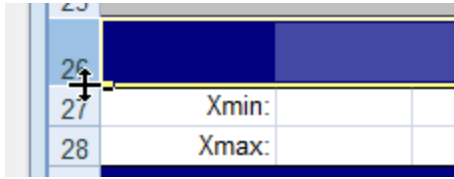
9. Select cell A17 and paste the selected data by using **Ctrl+V** or choose Clipboard>Paste from the menu. Once the first duplicate section has been pasted, select A25 and paste the data again.



A	B	C	D	E	F	G	H	I	J	K	L
9	Start Condition		Op #:	Tool #:		End Condition					
10			Op Type:	Tool Dia:							
11			Cut Type:	Tool Type:							
12			Depth:	CRC#:							
13											
14											
15											
17	Start Condition		Op #:	Tool #:		End Condition					
18			Op Type:	Tool Dia:							
19			Cut Type:	Tool Type:							
20			Depth:	CRC#:							
21											
22											
23											
24											
25	Start Condition		Op #:	Tool #:		End Condition					
26			Op Type:	Tool Dia:							
27			Cut Type:	Tool Type:							
28			Depth:	CRC#:							
29											
30											
31											
32											

Note that rows 24 and 32 are much taller than row 16. We need to make these rows thinner. The width of rows and columns may be adjusted by clicking on the title line and while holding the mouse button down, dragging the mouse until the desired size is shown.

10. Click the number 25 on the left title line and drag the mouse slightly upward until the right arrow becomes a crosshair (as shown below) then click and drag this crosshair upwards until you have resized its height to 3.00. Repeat this process on row 33.



A shortcut to modifying the cell size is to select entire cell rows instead of cell ranges. Thus, if rows 17 through 24 were selected and pasted, the cell size formatting would be automatically adjusted.

The Model file is now complete.

11. Save this file as an Excel workbook ReporterTutorial.xlsx in the following location: C:\Program Files\CAMBRIO\GibbsCAM\ <version> \Plugins\Data\Report\ReporterTutorial.xlsx. The Model (xlsx) and Template (txt) files must have the same name and be saved in the correct place.

Leave the Excel file open; we will be referencing back to the Model file for the rest of this tutorial.

Step 3

We will now create the Template text file. We will use Notepad to generate the file in this exercise, but any text editor or word processor may be used.

1. Launch Notepad. Notepad can be opened from the windows Start menu.

We will start with several commands that are essential to all Template files.

2. Type UseNewImageOffsets and click . This command makes it much easier to place any pictures you wish to add to reports.
3. Type SetPage 1 and click .

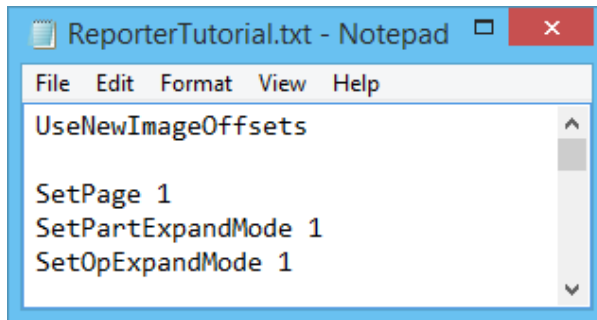
This designates that the following data will be output on Worksheet 1 of the Excel file. In the default Plug-In reports, Mill data is output on a separate Worksheet from Lathe data to organize information more efficiently.

4. Type SetPartExpandMode 1 then click .

This designates that the template will be outputting Mill part commands.



5. Type `SetOpExpandMode 1` and click .

This designates that from this point on, the template will output Mill operations only. The `SetPartExpandMode` command may be used more than once in a template. You would enter a `2` for Lathe operations only or `0` for all operation types.



The above commands are common to all templates. Their exact order in the file is not important.

We will now enter commands to output the name of the part and the name of the MDD used in the part. We will be referencing the Model file and specifying where to display the results.



6. Type `PartName 2 10` in the Template file then click . Be sure to put a space between the two and the ten.
7. Type `PartMddName 3 2` in the Template file then click . Be sure to put a space between the three and the two.

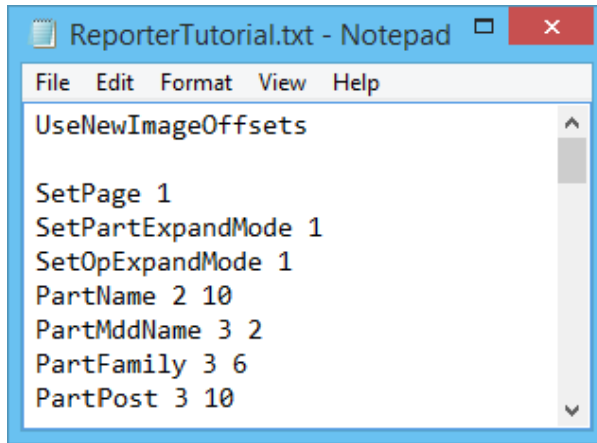
We have specified that the output for the part name should be placed two rows down and ten columns over. The output for the MDD Name is one row down from that, which is three rows down and two columns over. The Excel file cells would be `J2` for the PartName output and `B3` for the PartMDDName output.

The image below shows an Excel spreadsheet. The two numbers in each cell represent the numbers used in the commands of the Template file to place data in that particular cell. Note that merging cells does not change their numbering.

	A	B	C	D	E	F	G	H	I	J
1	11				15	16	17	18	19	110
2	21	22	23	24	25	26	27	28	29	210
3	31	32	33	34	35	36	37	38	39	310

We will now enter two more lines of data. These commands will specify the stock alloy and the post processor to be used for the part.

8. Type `PartFamily 3 6` in the Template file then click . Be sure to put a space between the three and six.
9. Type `PartPost 3 10` in the Template file and then click . Be sure to put a space between the three and ten.



```
ReporterTutorial.txt - Notepad
File Edit Format View Help
UseNewImageOffsets

SetPage 1
SetPartExpandMode 1
SetOpExpandMode 1
PartName 2 10
PartMddName 3 2
PartFamily 3 6
PartPost 3 10
```

We have specified that the output for "material" should be placed three rows down and six columns over. The output for the "post processor" is four columns over from the stock alloy, which is three rows down and ten columns over.

Next we will enter the commands for the units of measurement, the name of the NCF file generated from the part, and any comment in the Document Control dialog.

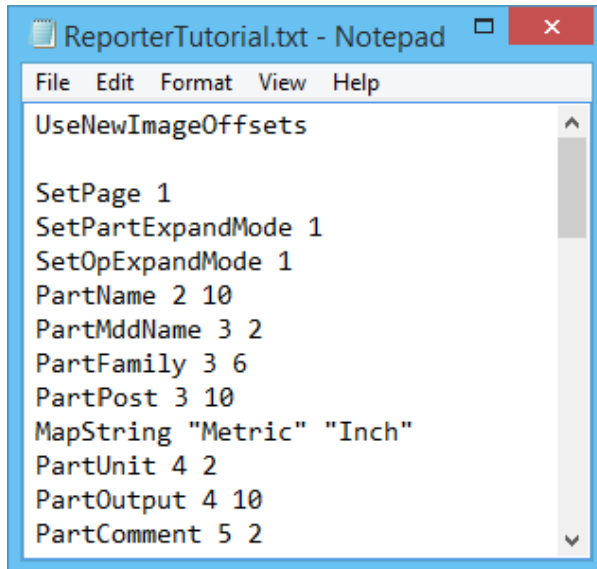
The units command is not generated by text input from the user, but is handled internally by the system. These types of commands will output a number (for example, 1 for metric and 2 for inches). Since numbers rarely have meaning to the user, we use the MapString command to tell the system to replace the numbers with some meaningful text. The Mapstring command is always entered just before the command it relates to. In this case we are defining the output for the PartUnit command.

10. Type `MapString "Metric" "Inch"` in the Template file then click . Be sure to put a space between `Metric` and `Inch`.

When the units of measurement for the part are output, the system will replace a 1 with the word Metric and the number 2 with Inch.

Terms must be separated by spaces. If a term consists of more than one word, it must be surrounded by quotation marks eg. (`MapString "Rough Endmill"`).

11. Type `PartUnit 4 2` in the Template file and then click .
12. Type `PartOutput 4 10` then click .
13. Type `PartComment 5 2` then click .




```
ReporterTutorial.txt - Notepad
File Edit Format View Help
UseNewImageOffsets

SetPage 1
SetPartExpandMode 1
SetOpExpandMode 1
PartName 2 10
PartMddName 3 2
PartFamily 3 6
PartPost 3 10
MapString "Metric" "Inch"
PartUnit 4 2
PartOutput 4 10
PartComment 5 2
```

We have specified that the units of measurement will be placed four rows down and two columns over. The PartOutput command will place the name of the NCF file generated by this part four rows down and ten columns over. The PartComment command will place text from the Comments section directly below the units of measurement (five rows down and two columns over).

We will now enter commands for the custom report to output the stock dimensions as defined in the Document Control dialog. We need to specify the min and max values for X, Y and Z. The Model file will take this data and calculate an overall size.

14. Type the following commands in the Template file. Be sure to **click**  between entries so that they are on separate lines.

```
PartMinX 7 2↵PartMinY 7 4↵PartMinZ 7 6↵PartMaxX 8 2↵PartMaxY 8 4↵
PartMaxZ 8 6
```

```

ReporterTutorial.txt - Notepad
File Edit Format View Help
UseNewImageOffsets

SetPage 1
SetPartExpandMode 1
SetOpExpandMode 1
PartName 2 10
PartMddName 3 2
PartFamily 3 6
PartPost 3 10
MapString "Metric" "Inch"
PartUnit 4 2
PartOutput 4 10
PartComment 5 2
PartMinX 7 2
PartMinY 7 4
PartMinZ 7 6
PartMaxX 8 2
PartMaxY 8 4
PartMaxZ 8 6

```

The PartMinX command will be output seven rows down and two columns over. The Xmin value goes in B7.

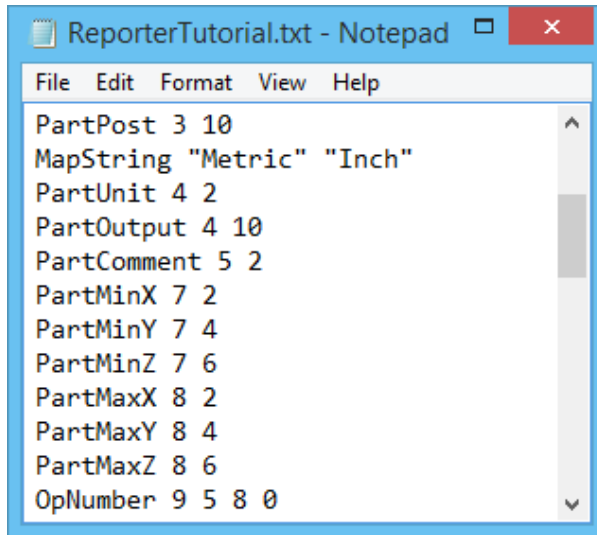
	A	B	C	D	E	F
7	Xmin:		Ymin:		Zmin:	
8	Xmax:		Ymax:		Zmax:	

You will recall that earlier in this exercise when we set up the spreadsheet, we set up cells H7, J7 and L7 to calculate the part size from the inputs in cells B7, B8, D7, D8, F7 and F8.

	G	H	I	J	K	L
7	Length:	0.00	Width:	0.00	Height:	0.00
8						

We will now enter the commands to output information from each individual operation. We begin with the operation number then move on to the type of operation, the type of cut used in the operation and the cut depth.

- Type OpNumber 9 5 8 0 in the Template file then click .




```

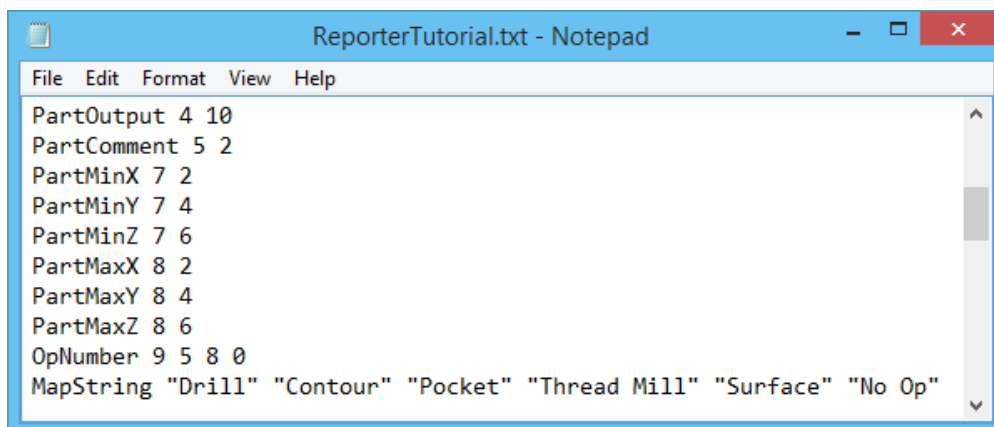
ReporterTutorial.txt - Notepad
File Edit Format View Help
PartPost 3 10
MapString "Metric" "Inch"
PartUnit 4 2
PartOutput 4 10
PartComment 5 2
PartMinX 7 2
PartMinY 7 4
PartMinZ 7 6
PartMaxX 8 2
PartMaxY 8 4
PartMaxZ 8 6
OpNumber 9 5 8 0

```

This is the first command we have come across that has four numbers for the position. The last two numbers are for multiple results from the same command, in this case, the Operation Number. For all successive operations the Operation Number will be output eight rows down and zero columns over. You can confirm that this is the correct amount by which to offset additional operations by counting the number of rows in the Model file that are given to each operation. Start at row 9, which contains Start Condition, Op # and Tool #. Count the number of rows between this row and the next time these items show up in the Model file.

We will now enter the command to output the type of operation used for a given operation number. We will use the MapString command to output text instead of a number.

16. Type `MapString "Drill" "Contour" "Pocket" "Thread Mill" "Surface" "No Op"` then click .



```

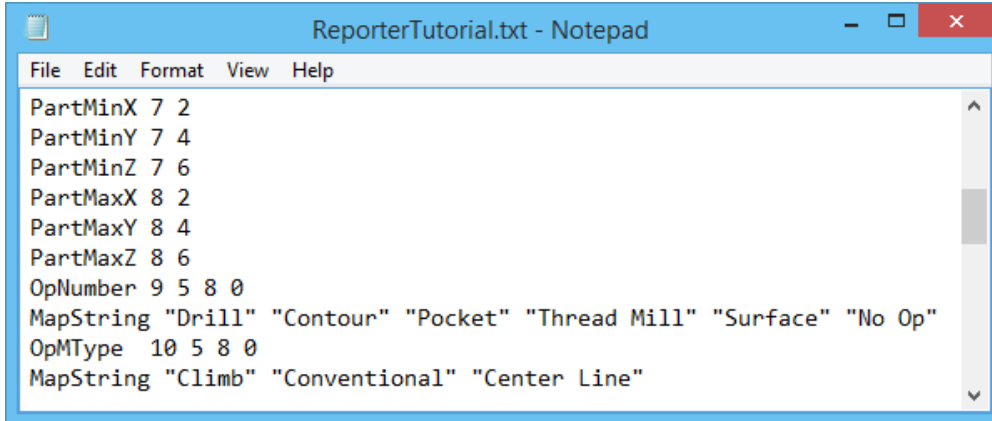
ReporterTutorial.txt - Notepad
File Edit Format View Help
PartOutput 4 10
PartComment 5 2
PartMinX 7 2
PartMinY 7 4
PartMinZ 7 6
PartMaxX 8 2
PartMaxY 8 4
PartMaxZ 8 6
OpNumber 9 5 8 0
MapString "Drill" "Contour" "Pocket" "Thread Mill" "Surface" "No Op"

```



17. Type `OpMType 10 5 8 0` then click .

We will now finish entering the operation data. Included is a MapString command for the type of cut (Climb Conventional or Center Line) and the final cut depth of this operation.




18. Type `MapString "Climb" "Conventional" "Center Line"` then click .



```
ReporterTutorial.txt - Notepad
File Edit Format View Help
PartMinX 7 2
PartMinY 7 4
PartMinZ 7 6
PartMaxX 8 2
PartMaxY 8 4
PartMaxZ 8 6
OpNumber 9 5 8 0
MapString "Drill" "Contour" "Pocket" "Thread Mill" "Surface" "No Op"
OpMType 10 5 8 0
MapString "Climb" "Conventional" "Center Line"
```

19. Type `OpCutType 11 5 8 0` and then .
20. Type `OpDepth 12 5 8 0` and then .

We will now enter the commands to output information on the tools used in the operation. This includes the tool number, the diameter of the tool, the type of tool and the tool's CRC#. Please note that these are a special class of commands combining Tool Commands with Operation Commands.

21. Type `OpToolNumber 9 8 8 0` .
22. Type `OpToolDiameter 10 8 8 0` .
23. Type the following text and press  when you are done. (NB if you are using a screen-based version of the tutorial, you can just copy the list below and paste it straight into the text file.) There are no carriage returns in the list.

```
MapString "REM" "FEM" "BEM" "Shell Mill" "Face Mill" "Key Cutter" "Drill"
"Center Drill" "Spot Drill" "Bore" "Tap" "Countersink" "Reamer" "Spot
Face" "Fly Cutter" "Single Point Thread Mill" "Back Bore" "Rigid Tap"
"Roundover" "2D Form" "3D Form" "Full Profile Thread Mill" "Lollipop"
"Convex Tip Mill" "Barrel Mill" "Dovetail" "Inserted Mill" "Custom Mill"
"80 Deg Diamond" "55 Deg Diamond" "35 Deg Diamond" "Round" "Square"
"Triangle" "Trigon" "Pentagon" "Parallelogram" "Rectangle" "Groove"
"Cutoff" "Thread N" "Laydown Thread" "Profile VN" "2D Form" "3D Form"
"Utility Tool" "Custom Lathe" "Unsupported"
```

Typing Shortcut

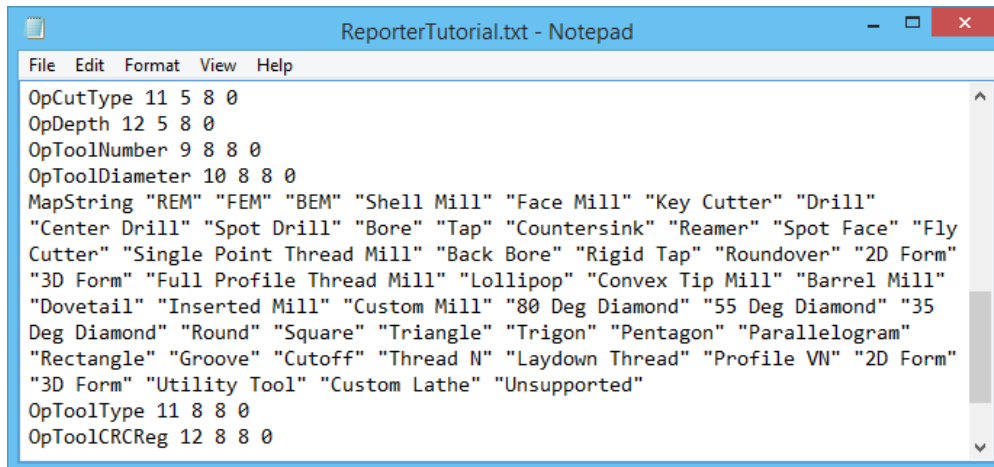


You may find it helpful for any future Reporter work to open the online Help in GibbsCAM and go to the Plug-ins>Advanced Reporter Use> Commands pages. There, you will find lists of commands available to reporter with a brief description of what they do. At the bottom of each of the pages there are various lists that can be

used in conjunction with Mapstring commands. Simply copy the required list and paste it straight into the text file you are editing.

24. Type `OpToolType 11 8 8 0` .

25. Type `OpToolCRCReg 12 8 8 0` .



```
ReporterTutorial.txt - Notepad
File Edit Format View Help
OpCutType 11 5 8 0
OpDepth 12 5 8 0
OpToolNumber 9 8 8 0
OpToolDiameter 10 8 8 0
MapString "REM" "FEM" "BEM" "Shell Mill" "Face Mill" "Key Cutter" "Drill"
"Center Drill" "Spot Drill" "Bore" "Tap" "Countersink" "Reamer" "Spot Face" "Fly
Cutter" "Single Point Thread Mill" "Back Bore" "Rigid Tap" "Roundover" "2D Form"
"3D Form" "Full Profile Thread Mill" "Lollipop" "Convex Tip Mill" "Barrel Mill"
"Dovetail" "Inserted Mill" "Custom Mill" "80 Deg Diamond" "55 Deg Diamond" "35
Deg Diamond" "Round" "Square" "Triangle" "Trigon" "Pentagon" "Parallelogram"
"Rectangle" "Groove" "Cutoff" "Thread N" "Laydown Thread" "Profile VN" "2D Form"
"3D Form" "Utility Tool" "Custom Lathe" "Unsupported"
OpToolType 11 8 8 0
OpToolCRCReg 12 8 8 0
```

We will now enter the last set of commands which will output two images. The `UseNewImageOffsets` command that we inserted in step 1 of the tutorial, enables us to position an image using the four positioning specifiers that we have already been using to insert data, ie. First, row and column number of the first occurrence, then number of rows and columns to increment between subsequent data.

For graphics we can also specify vertical and horizontal offsets from the top and left of the cell. If nothing is entered, or the value entered is too great, a value of 0 is used, placing the image at the cells top left margin.

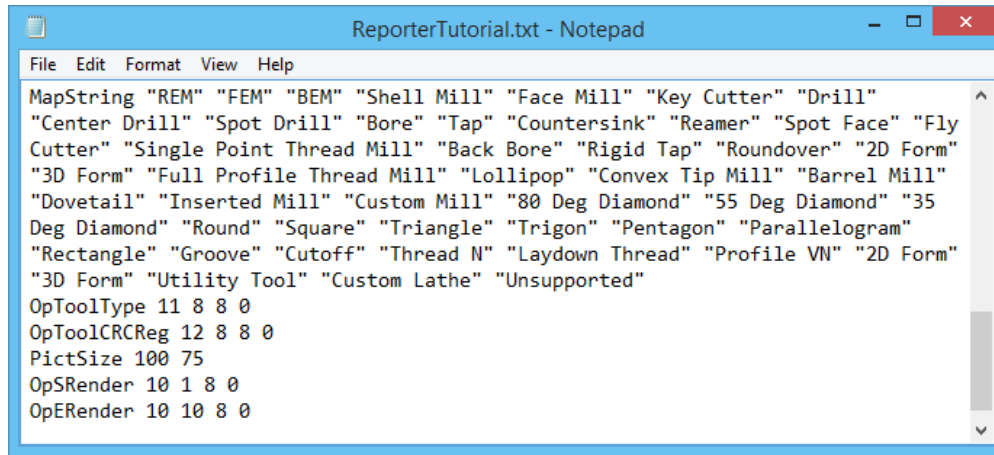
26. Type `PictSize 100 75` .

This command specifies the width and height of a graphic in pixels. To prevent pictures from being distorted, keep the numbers in a 4:3 ratio, such as 120x90 or 100x75.

27. Type `OpSRender 10 1 8 0` .

28. Type `OpERender 10 10 8 0` .

These commands specify where to place the start and end conditions of the stock for each operation. Notice we have not specified offsets as these are not necessary in this particular case.




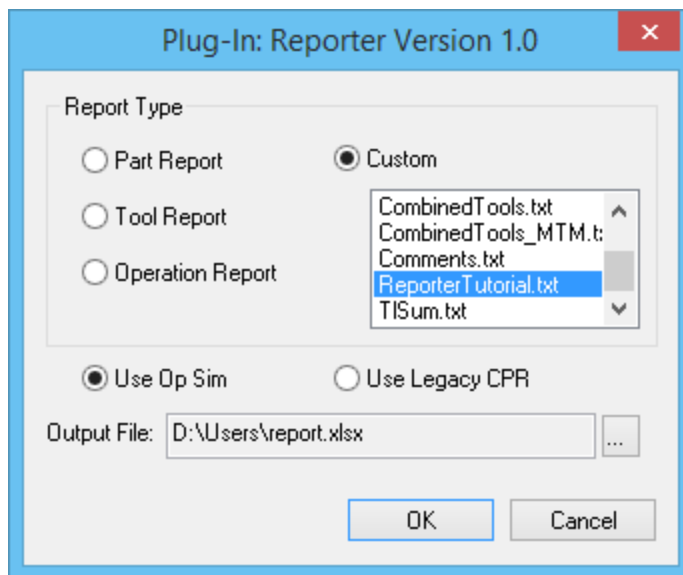
29. Save this file as ReporterTutorial.txt in the directory `C:\Program Files\CAMBRI0\GibbsCAM\<version>\Plugins\Data\Report\ReporterTutorial.txt`

Step 4

The Template file is complete. We will now test it.



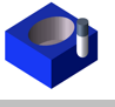
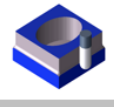
1. Open the part named ReporterTutorial.vnc. This was installed with the Tutorial Parts.

2. From the Plug-Ins menu select  Reporter. In the Report type dialog, check Custom and select ReporterTutorial.txt. The default output file is D:\Users\report.xlsx this is OK for the tutorial purposes. Now click OK.

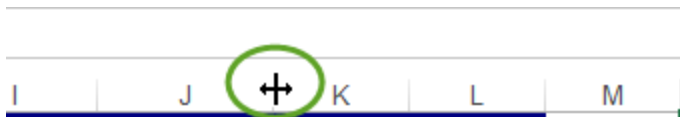


The system will automatically activate cut part rendering for the part. When the rendering is complete, the system will launch Excel and generate the report.

If you look at the report, you will see that we have a problem with the graphics - the graphic could be larger and is positioned too far over to the top left of the cells, and also the right hand graphic seems to occupy more space than the left.

	Op #: 1	Tool #: 1	
	Op Type: Pocket	Tool Dia: 1.5	
	Cut Type: Climb	Tool Type: REM	
	Depth: -2	CRC#: 51	
	Op #: 2	Tool #: 2	
	Op Type: Contour	Tool Dia: 0.9	
	Cut Type: Climb	Tool Type: BEM	
	Depth: -1.5	CRC#: 52	

- Open the ReporterTutorial.xlsx Excel file. You will notice that the column width of Column J is too narrow - change the width to 11.86 by clicking into the divider between column title J and K and drag the divider to the correct width.



- Open the ReporterTutorial.txt file. Change the PictSize entry to PictSize 175 100.

```

ReporterTutorial.txt - Notepad
File Edit Format View Help
MapString "REM" "FEM" "BEM" "Shell Mill" "Face Mill" "Key Cutter" "Drill"
"Center Drill" "Spot Drill" "Bore" "Tap" "Countersink" "Reamer" "Spot Face" "Fly
Cutter" "Single Point Thread Mill" "Back Bore" "Rigid Tap" "Roundover" "2D Form"
"3D Form" "Full Profile Thread Mill" "Lollipop" "Convex Tip Mill" "Barrel Mill"
"Dovetail" "Inserted Mill" "Custom Mill" "80 Deg Diamond" "55 Deg Diamond" "35
Deg Diamond" "Round" "Square" "Triangle" "Trigon" "Pentagon" "Parallelogram"
"Rectangle" "Groove" "Cutoff" "Thread N" "Laydown Thread" "Profile VN" "2D Form"
"3D Form" "Utility Tool" "Custom Lathe" "Unsupported"
OpToolType 11 8 8 0
OpToolCRCReg 12 8 8 0
PictSize 100 75
OpSRender 10 1 8 0
OpERender 10 10 8 0
    
```

We also need to make sure to specify the horizontal and vertical position.

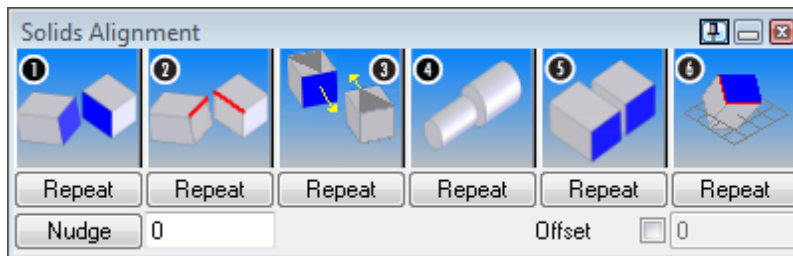
- Change the OpSRender entry to OpSRender 10 1 8 0 10 25.
- Change the OpERender entry to OpERender 10 10 8 0 10 25.
- Be sure to save the text file.
- Rerun the report. You may get the error message The process cannot access the file because it is being used by another process. This is because you still have the report.xlsx file open. Close it and try again.

The graphics will now look more centered.

The Excel master file can hold up to 100 operations (the operation data fields in the Excel file must be duplicated 100 times). You might find this report helpful for most parts.

SOLIDS ALIGNMENT

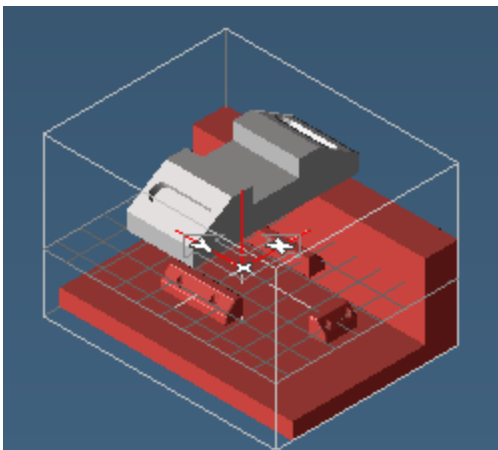
The Solids Alignment Plug-in allows you to manipulate the positions of solid models. The positioning is done by rotating/translating the first of the two selected models. The solid that was selected first becomes aligned to a second model. Cylindrical bodies can be aligned by making them concentric. Further functions are provided to align bodies using two edges or two points. Once two bodies have been aligned, additional models may be selected and aligned using the same transformation, enabling multiple models to be aligned.




1. Align Two Faces
2. Align Two Edges
3. Flip About A Face
4. Move/Rotate
5. Face-To-Face Mate
6. Align With CS

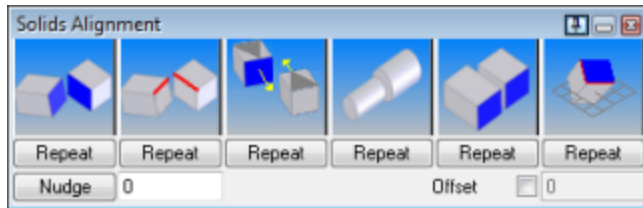
This exercise will introduce the user to manipulating the position of solids using the Solids Alignment plug-in. In this case, we will use the plug-in to correct the position of a part in relation to its fixtures.

1. Open part `SolidAlignment.vnc` from the sample part files.



The slot block and fixtures are out of place due to errors during importing. We will now use the Solids Alignment function to correct this.

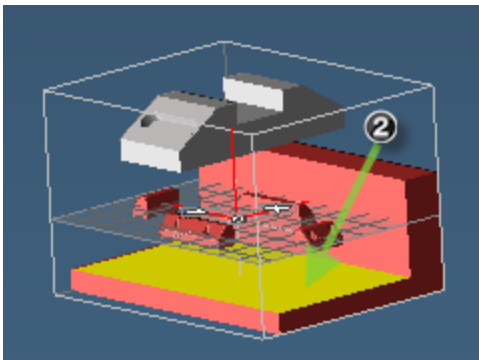
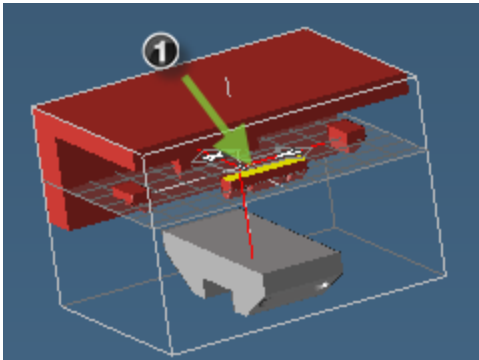
2. Open Solids >  Solids Alignment from the Plug-Ins menu.



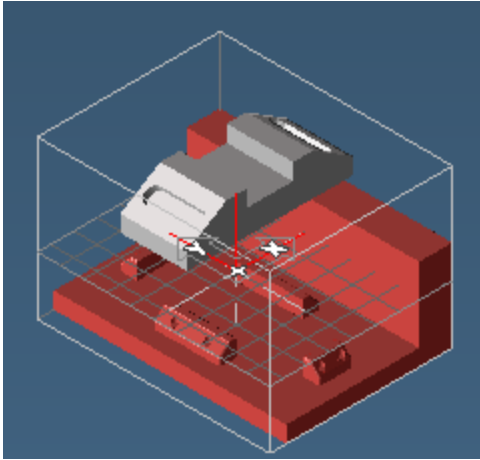
The file `AlignModeless.dll` must be in the Program Files\CAMBRIO\GibbsCAM*<yourversion>*\Plugins\Solids folder.

The first thing we will do is match up the fixture bodies.

3. Select the two faces shown and **click** the Face-to-Face Mate button .



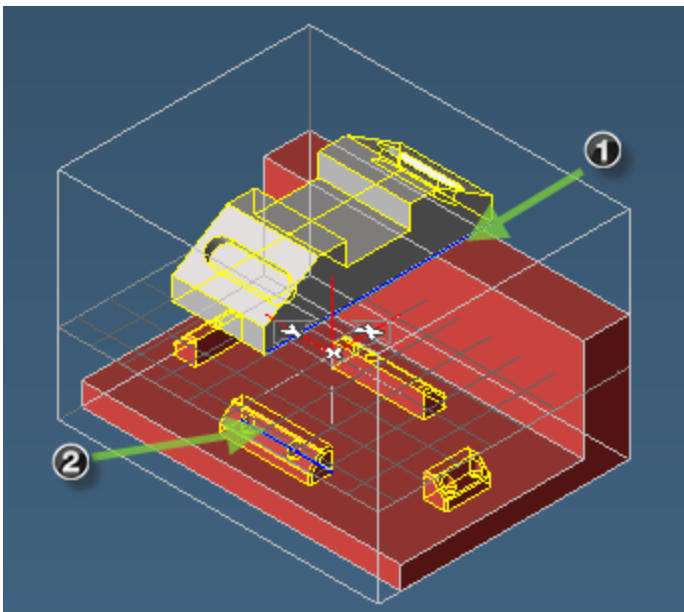
Make sure to select the faces in the order indicated since the first body is modified to accommodate the second.




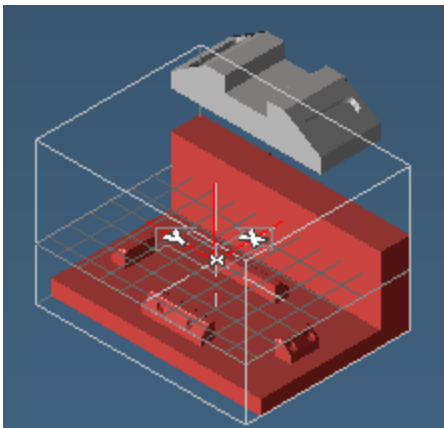
This aligns the fixtures.

We will now rotate the slot block body to align with the fixtures.

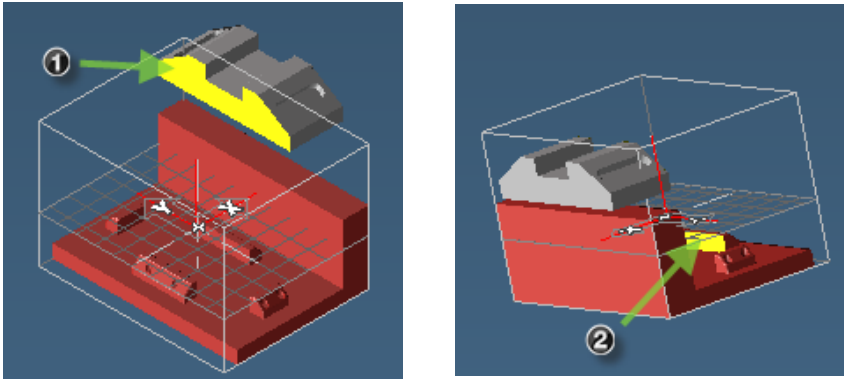
4. While in Edge Selection mode, select the two edges shown in the order indicated.



5. Click the Align 2 Edges button  .

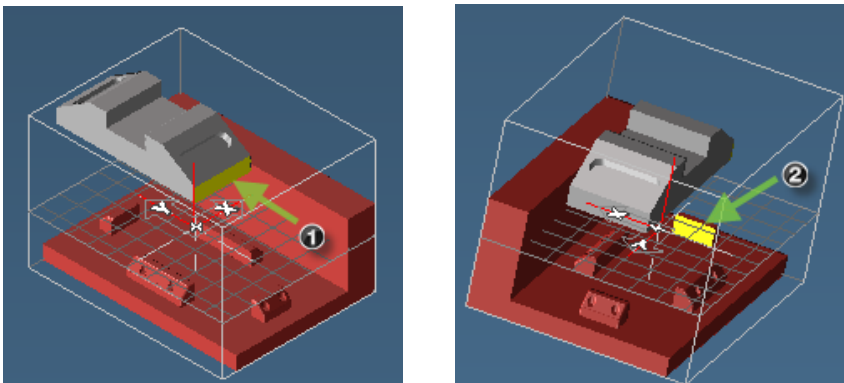


6. Select the two faces shown and click the Face-to-Face Mate  button.



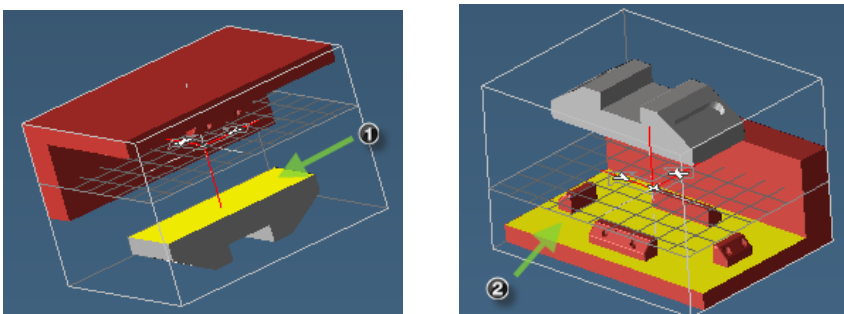
The two faces we are aligning are the front face of the slot block and the back face of the foremost fixture. This will slide the part in the -X direction to align with the fixture.

7. Select the two faces shown and click the Face-to-Face Mate  button.



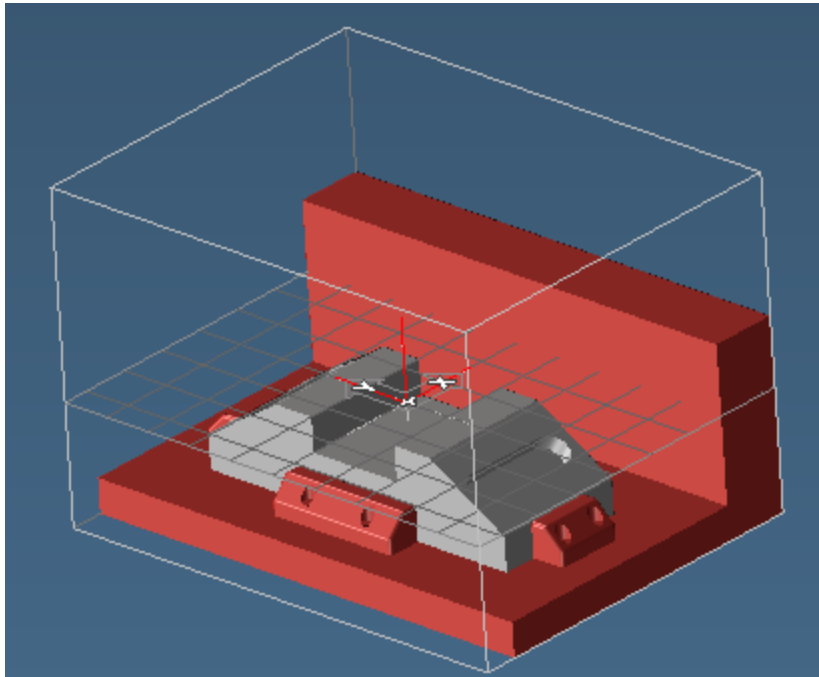
The two faces we are aligning are the right face of the slot block and the left face of the rightmost fixture. This action will slide the part in the -Y direction.

8. Select the two faces shown and click the Face-To-Face Mate  button. Select Shrink Wrap from the View menu to fit the stock boundaries to the part.



The two faces we are aligning are the bottom face of the slot block and the top face of the fixture plate. This will move the part down along the Z axis to lie on the fixture plate.

The slot block part is now in place in relation to the fixtures.





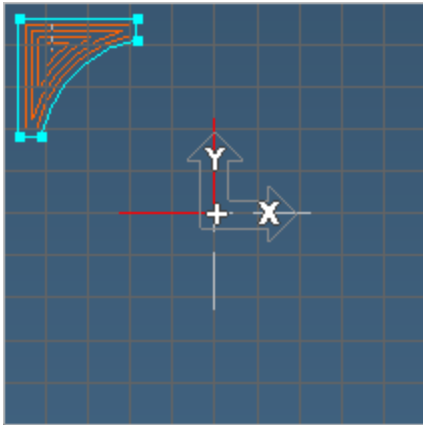
TP TRANS



This Plug-in allows you to modify the location, orientation, and size of selected toolpath. Operations may be moved in X, Y and Z, rotated, mirrored, and scaled. The toolpath is duplicated when it is transformed and an operation is created for each instance of the toolpath.

This exercise will introduce you to the use of the TP Trans Plug-in.

We will start with one simple pocket and translate, rotate, mirror, and scale the toolpath.


1. Open part Transform.vnc from the sample part files.

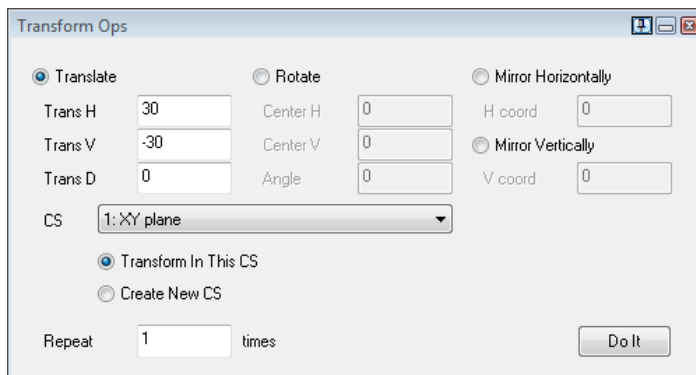


1. Click on the  Tools and the  Machining buttons.

This part has one existing operation and one geometric shape that is pocketed by this operation.

2. Double-click the operation to load the toolpath.

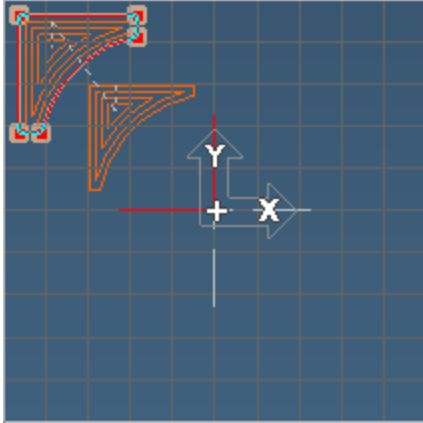
3. Select  TpTrans from the Plug-Ins menu. The Transform Ops dialog appears.
4. Select Translate and enter the values shown in the following image.



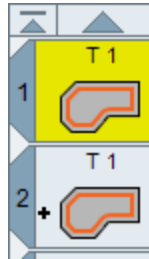
We will be translating (and duplicating) the toolpath down and to the right of the existing pocket so that the additional pocket's upper left-hand corner is near the arc of the first pocket.

5. Click Do it.

Your screen will look like the image below. Keep the Transform Ops dialog open. You will notice that the operations are all highlighted.

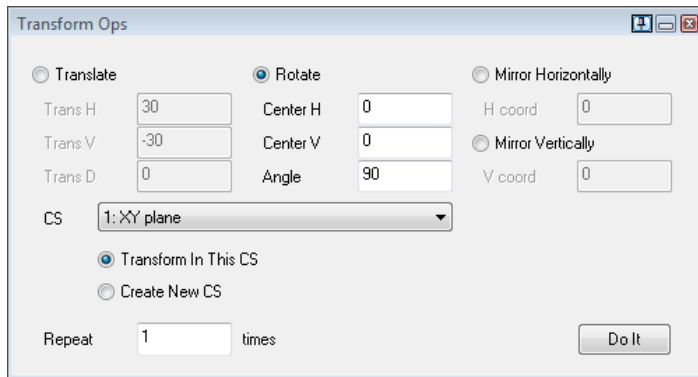


If you look at Operations in the operation list, you will see that the two operations are grouped, indicated by a plus sign.

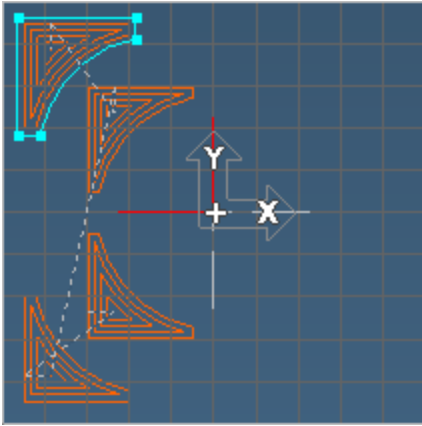


With the operations selected, we will now duplicate and rotate the existing toolpath.

- In the Transform Ops dialog, select **Rotate** and enter the values shown in the image. **Click Do it.**

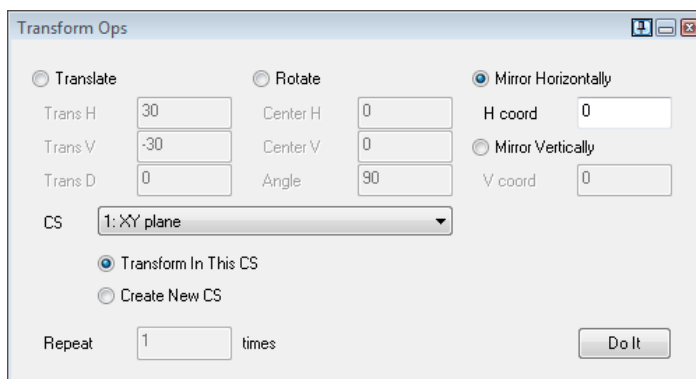


Your screen will look like the following image.

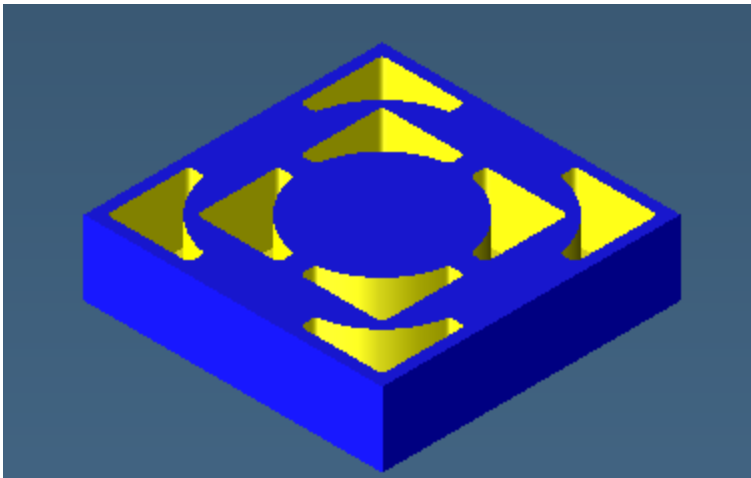


We will now duplicate and mirror the toolpath to complete this exercise.

7. Select **Mirror Horizontally** and enter **0** for the H coord. **Click Do it.**



We now have eight grouped operations in the Operation List and the toolpath has been duplicated symmetrically around the part. When rendered, your part will look similar to the following image.



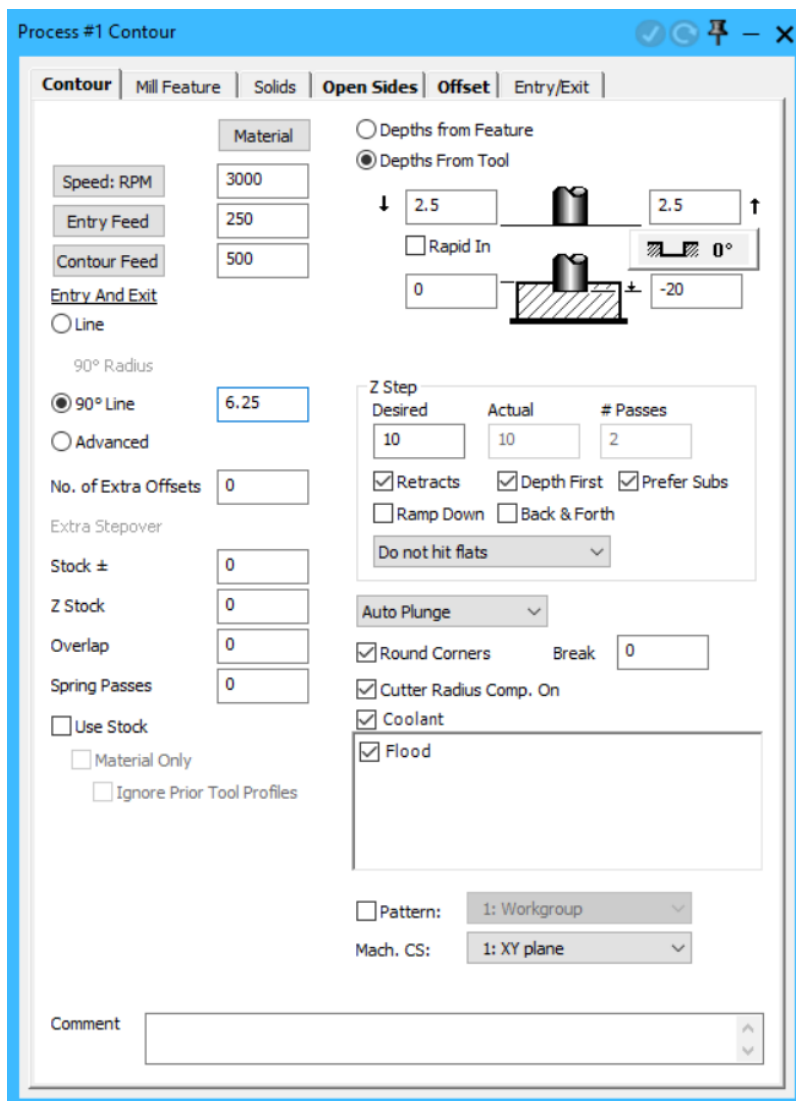
CONVERT TO TROCHOIDAL TOOLPATH (OP MODIFIER)

This exercise will introduce you to creating Trochoidal Toolpath.

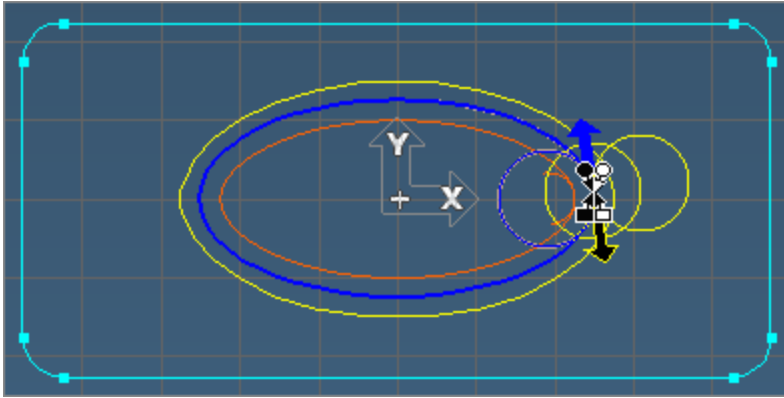
1. Open part Trochoidal.vnc from the sample part files.

This part currently contains two tools, three geometric shapes and a stock body defined through geometry. First, we will create a Contouring operation to go around the inside pocket of the stock. This will be the first set-up operation.

2. Create this Contour process with Tool #1.



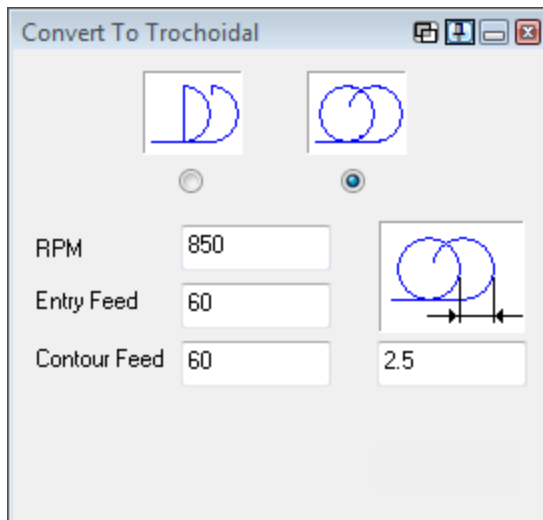
3. Set the machining markers as shown on the smaller ellipse.



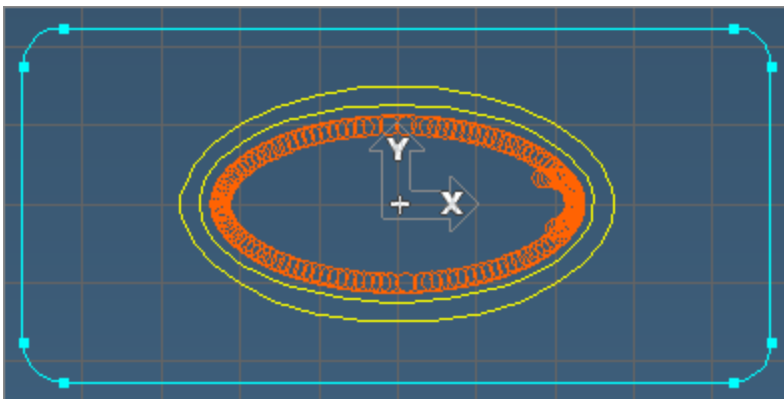
4. Create the toolpath.

We will now create Trochoidal Toolpath from the operation.

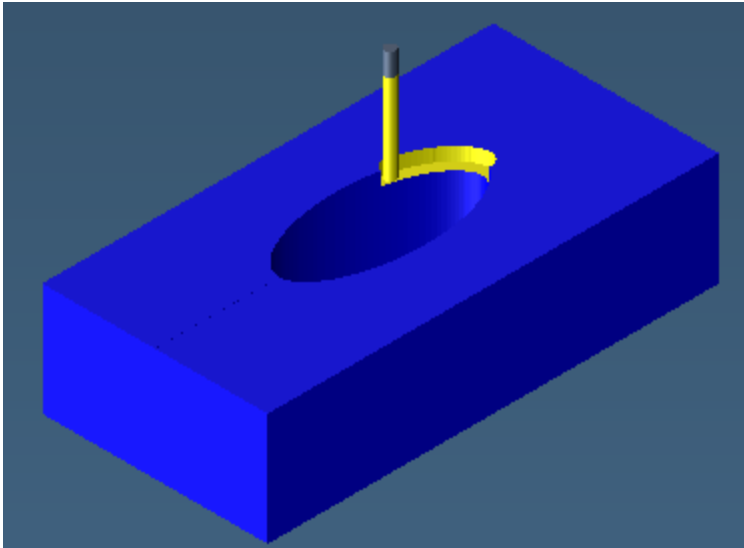
5. Select the smaller tool #2, and the Contour operation we created, **Right-click** the operation and choose **Operation Modifiers** from the menu, then choose **Convert to Trochoidal**.
6. Select the **Full-Circle** option and enter an offset of 2.5mm. Click **Do it** when you are ready.



The Contour operation is modified to the trochoidal parameters.

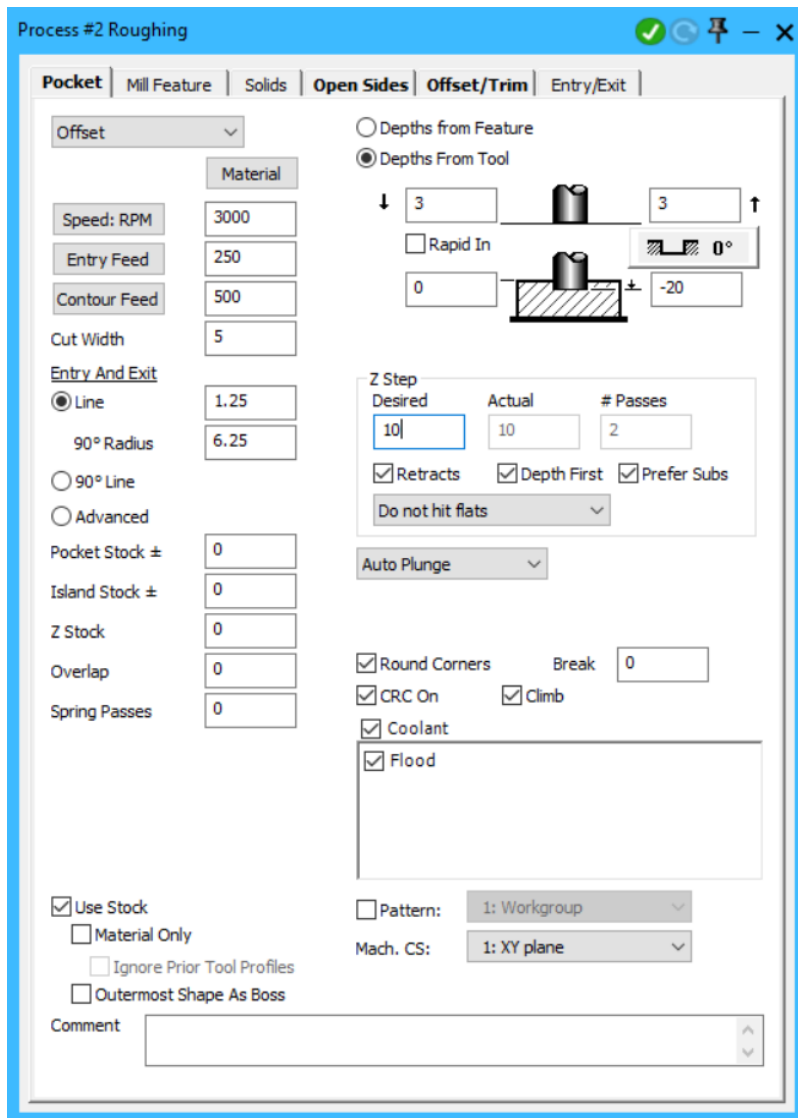


During rendering, you will notice how the Trochoidal Toolpath is effectively scooping out the inner shape from the stock. Trochoidal Toolpath is able to simulate large cuts by performing a series of small, fast cuts.

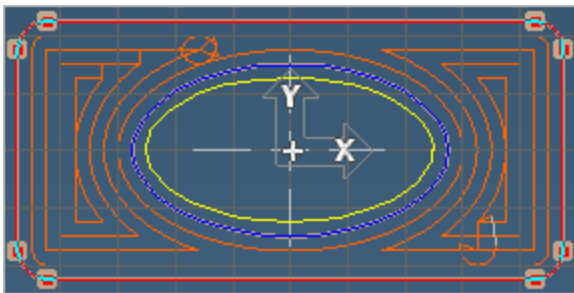


We will now create a Roughing operation to pocket out the rest of the part.

7. Create this Rough process with tool #2.

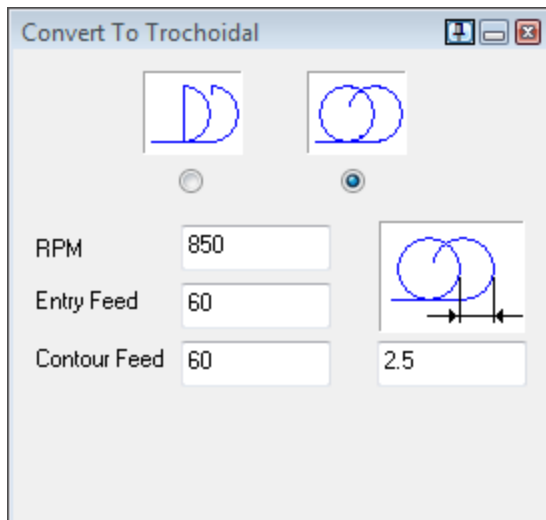


8. Select the larger ellipse and the outer geometry.
9. Create the toolpath.

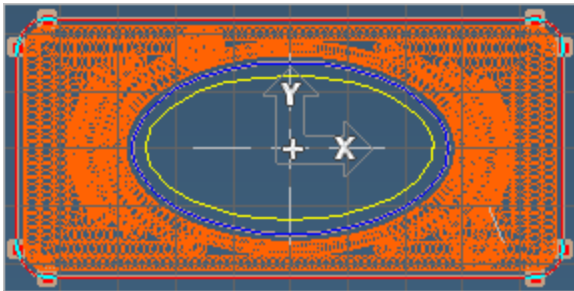


We will now modify this toolpath to a Trochoidal toolpath.

10. Select tool #2 and the Roughing operation we created, the operation and choose **Operation Modifiers** from the menu, then choose **Convert to Trochoidal**.
11. Again we will use the Full-Circle option with a 2.5mm offset. Click **Do** it when you are ready.

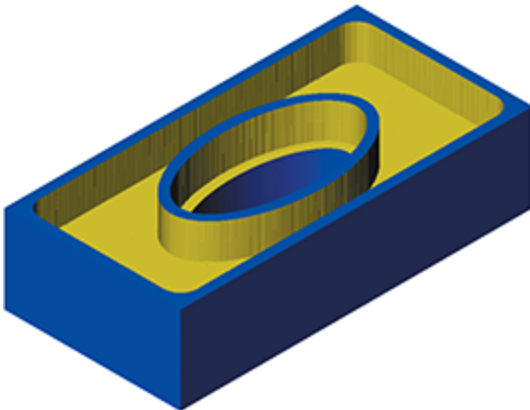


A new Roughing operation is created. Double-click the new operation; your screen should look like the image below.



When rendered, you can see just how wide a groove this type of operation is able to make. While a 10mm tool would cut a groove that is just as wide, the smaller tool will provide for more accurate corners.

When fully rendered, the part should look like the image.



Once the operations are complete, you may delete the setup operations. However, it is recommended that you not do so until you are sure you have the desired Trochoidal Toolpath. Remember to save your part.



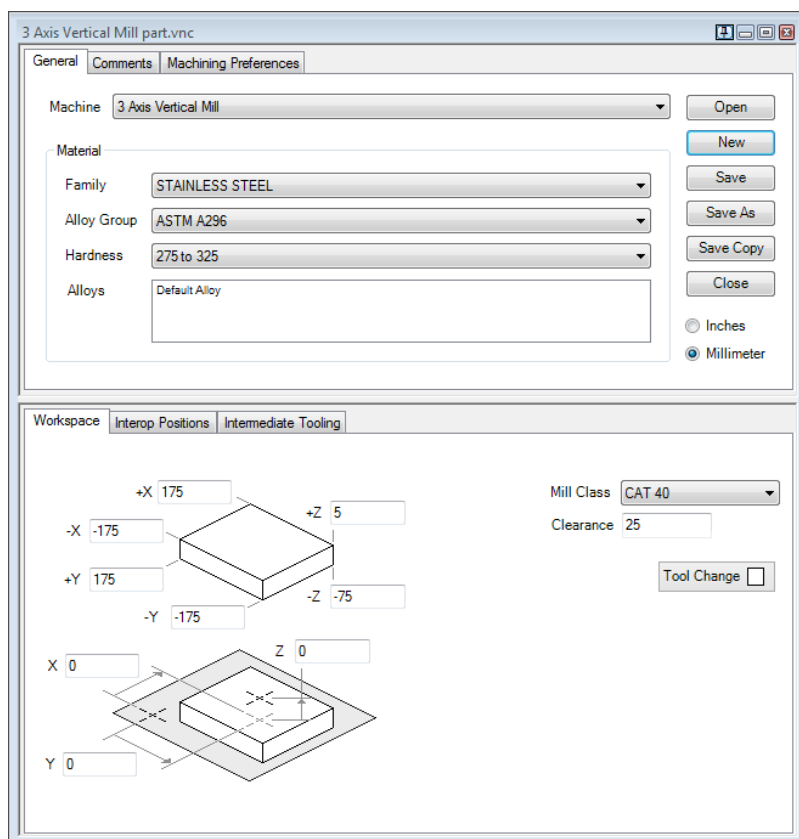
The Stock Wizard helps to create a rectangular or cylindrical stock definition with or without a single hole, rather than manually creating it. The hole may be a rectangular or circular shape. The hole in the stock may be a blind hole (a pocket) or a through hole

This tutorial is intended to illustrate the operation of the Stock Wizard and familiarize you with its use. In this exercise we will make a cylindrical stock with a rectangular blind hole but will start out creating a cuboid stock to illustrate changing the stock.

Please Note: The Stock Wizard was not designed to work with spinning Part Stations.


Defining Stock

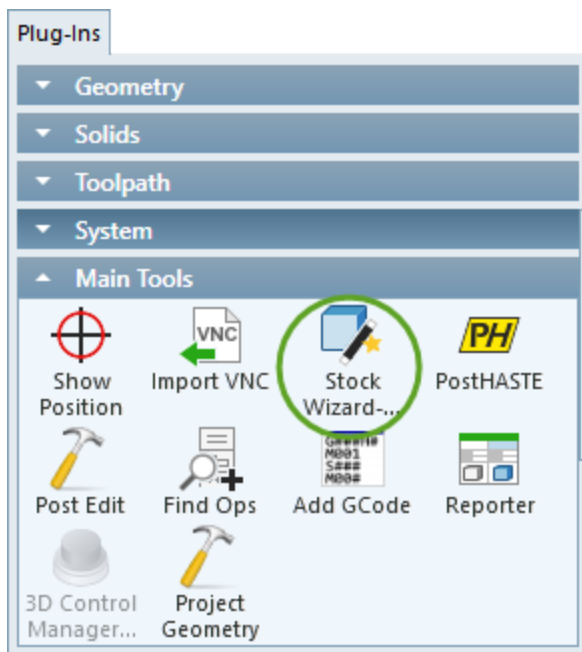
1. Create a metric 3 Axis Vertical Mill part with the following workspace stock size:
 $+X: 175 -X: -175, +Y: 175 -Y: -175, +Z: 0 -Z: -75$



This will provide a workspace large enough to visualize the new stock.

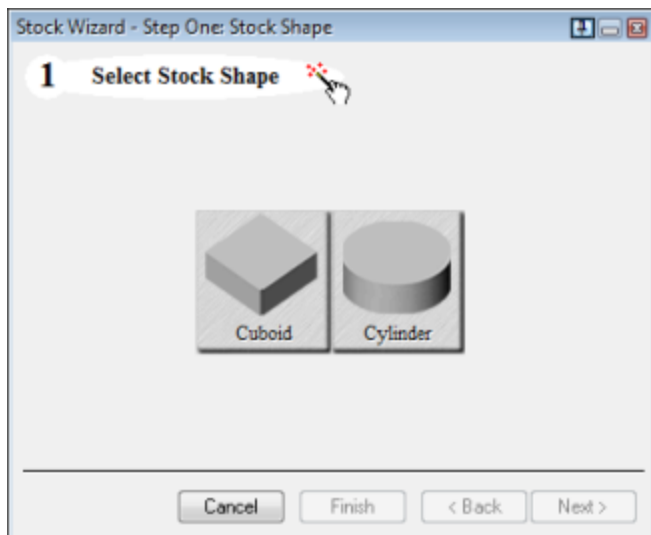
The Z values in this dialog are critical to the Stock Wizard. The depth value of the stock will be taken directly from this dialog. All other settings for the stock size may be adjusted in the Stock Wizard dialog but the stock depth is defined in and controlled by the Document Control dialog.

2. Choose Main Tools >  Stock Wizard from the Plug-in Menu.



Cuboid

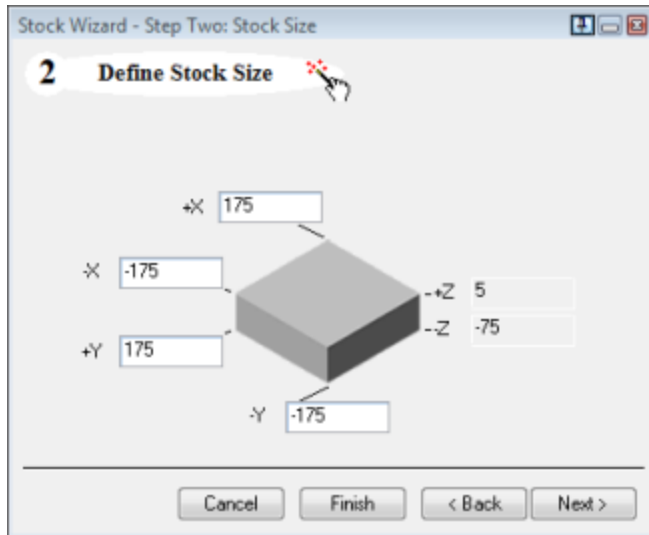
When the Stock Wizard is first opened you can choose to make a Cuboid or Cylinder shaped stock.



1. Click the Cuboid button.

This will make a cuboid shaped stock and progress to the next step where we will define the size of the stock.

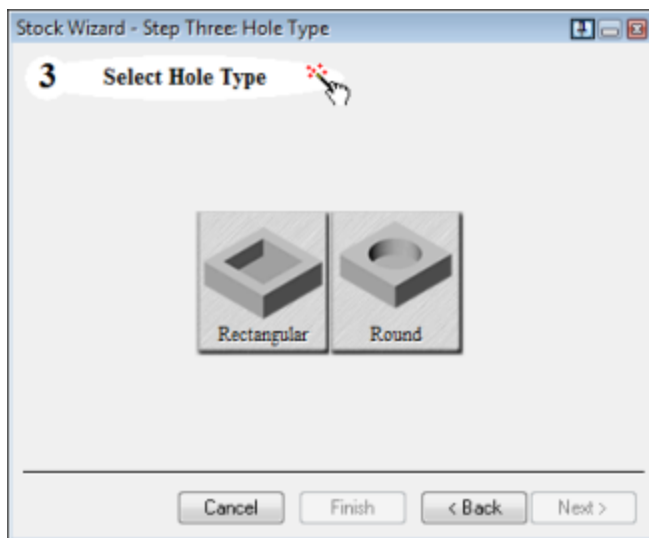
2. Enter the values shown and click the Next button.



The stock will always be equal to or less than the workspace defined in the Document Control dialog and the depth of the stock is always equal to the Z values in the Document Control dialog. The Stock Wizard definition will always override the Document Control dialog.

In step 3 we will define the shape of a rectangular hole in the stock.

3. Select the Rectangular hole type.

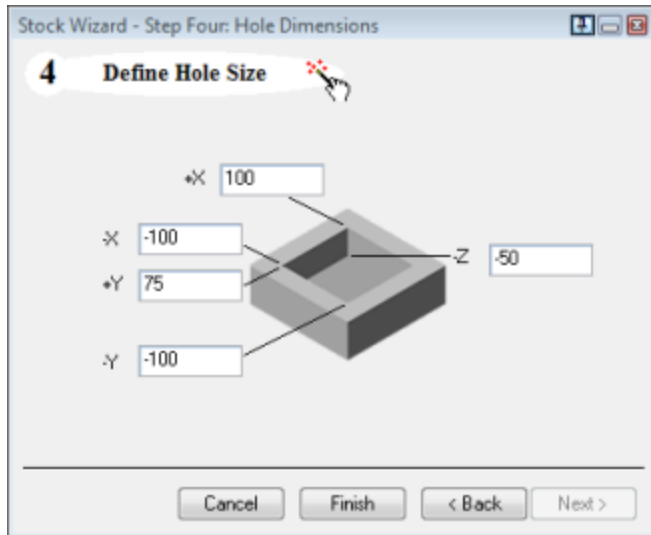


Blind Rectangular Hole

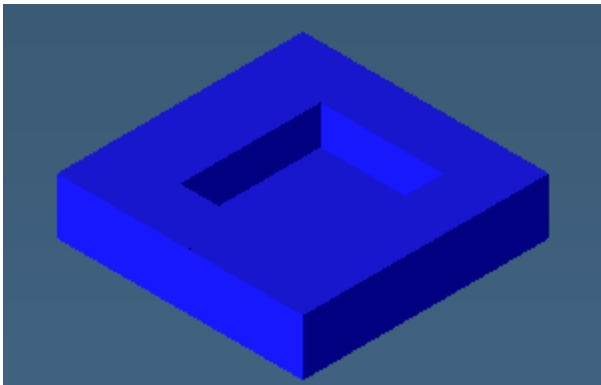
We will create a blind hole in the stock. Holes must always be contained within the X and Y bounds of the stock. If the hole overlaps the stock no hole will be produced.

1. Enter the values shown to define the size of the blind hole and **click** the **Finish** button.

Rectangular holes can be placed anywhere within the stock boundary.



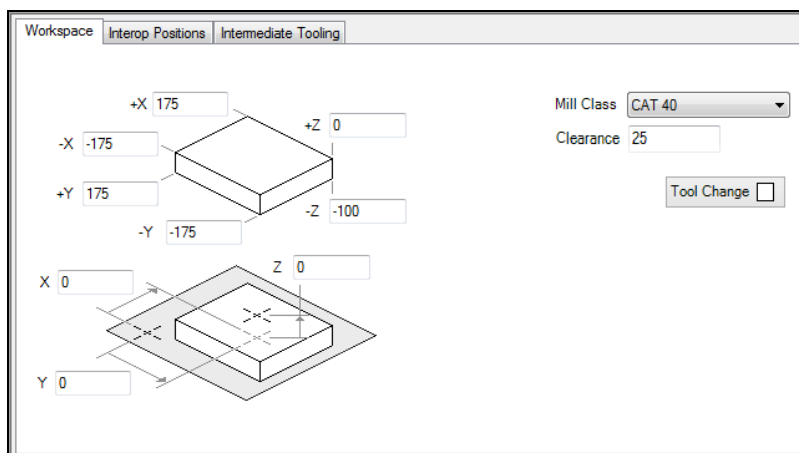
2. View the stock in rendering mode (F6).



Cylindrical

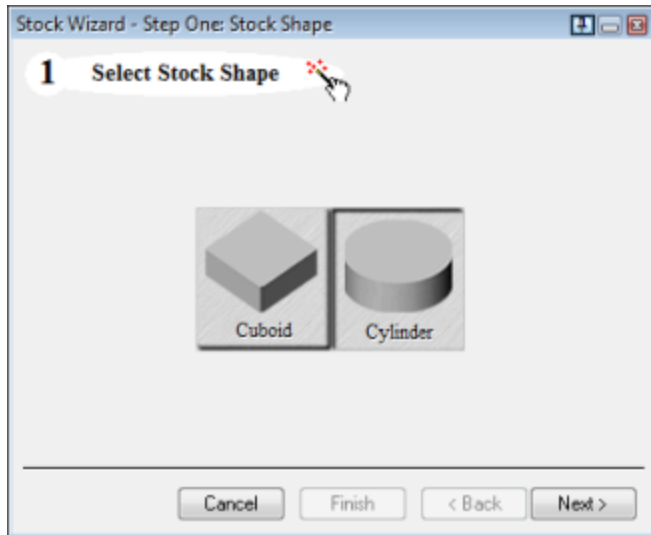
We will now recreate the stock to be a cylinder with a depth of 100mm. Doing so will illustrate how the workspace size will change.

1. Modify the workspace stock size as shown.

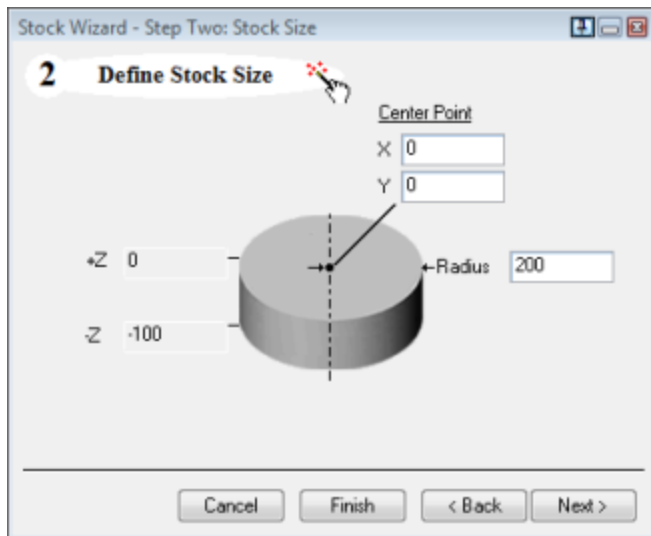


To change the depth of the stock we must modify it from the Document Control dialog.

2. Open the Stock Wizard and select **Cylinder** for the stock shape.

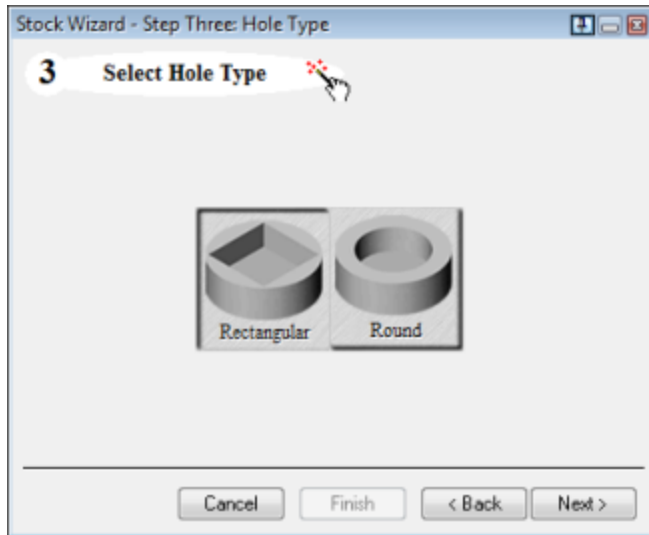


3. Define the cylindrical stock as shown and click the Next button.

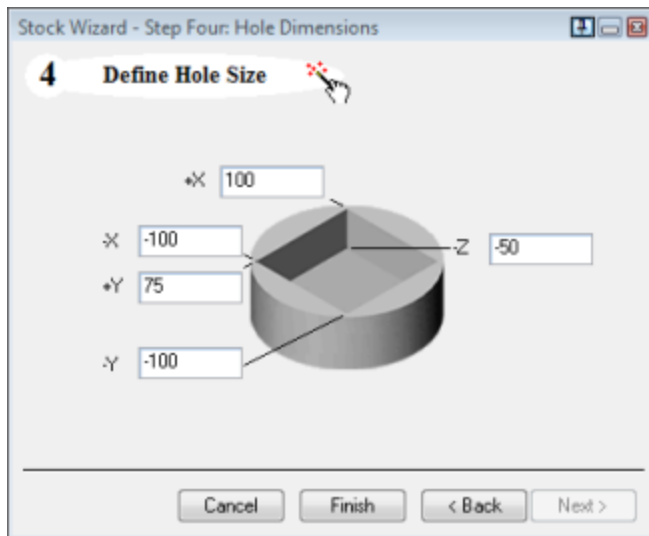


Note that the radius of the stock is greater than the workspace.

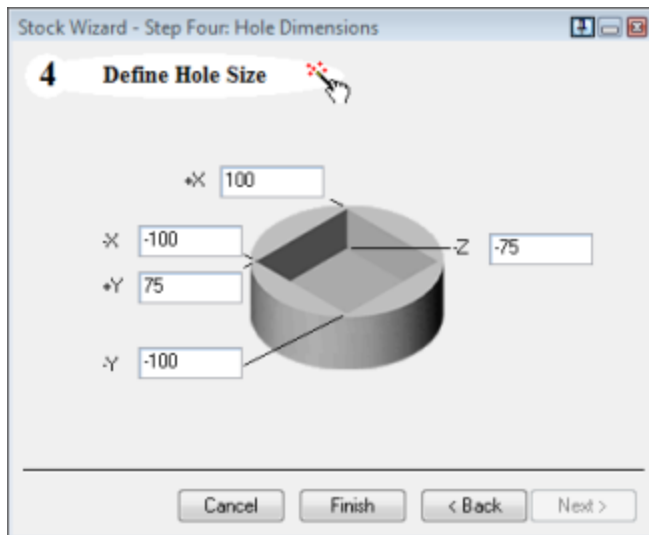
4. Select a rectangular hole shape.



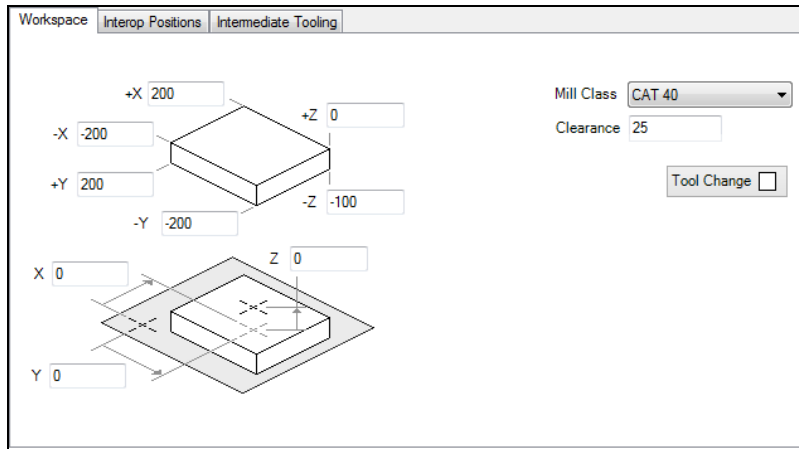
The Stock Wizard has kept the hole size from the previous stock definition.



5. Change the depth of the hole to **-75** and **click** the **Finish** button.

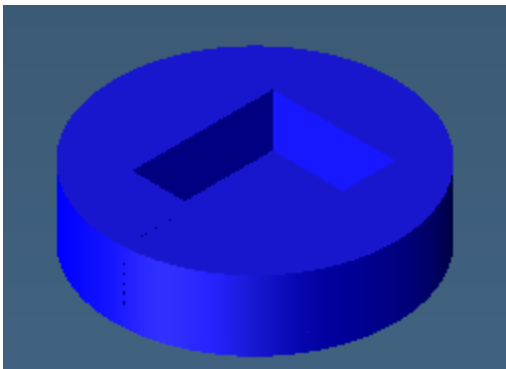


6. Open the Document Control dialog.



Note that the X and Y values have been modified to accommodate the stock.

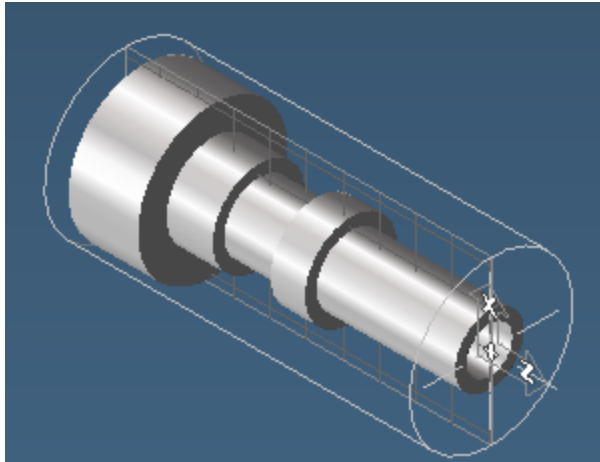
7. View the stock in rendering mode (F6).



DISTRIBUTION ONLY TUTORIALS

Distribution Only plug-ins are items that were developed to fit a particular need and may be of use elsewhere. These items often do not adhere to the GibbsCAM look and feel and are not considered a normal part of the system.

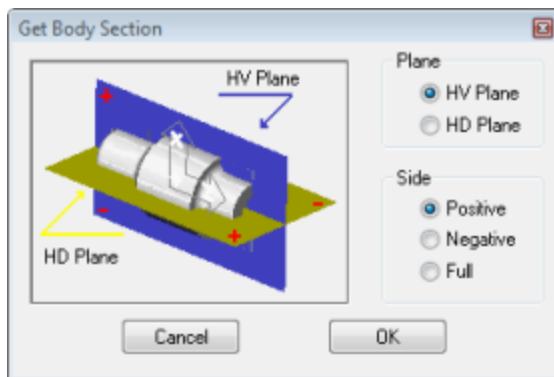
Get Section



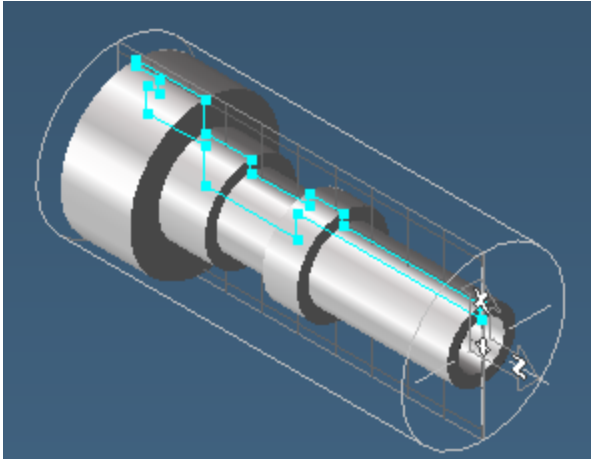
The Get Section Plug-in is generally used with solid Lathe parts to extract a flat profile of geometry for machining.

This exercise will introduce you to extracting geometry with the Get Section plug-in.

1. Open part `Get Section.vnc` from the sample part files.
2. Select the solid and choose `Solids > Get Section` from the `Plug-Ins` menu.
3. Select `HV Plane` and the `Positive` side and click `OK`.



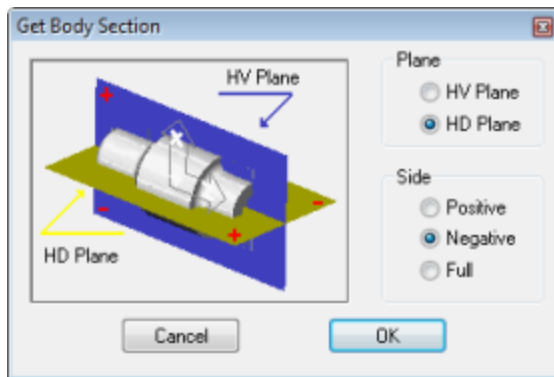
The system has extracted geometry from the intersection of the solid and the HV Plane (the current CS), which in this case is the XZ Plane. Because the `Positive` option is selected, the system will only extract geometry from the positive side of the V (or X) Axis.



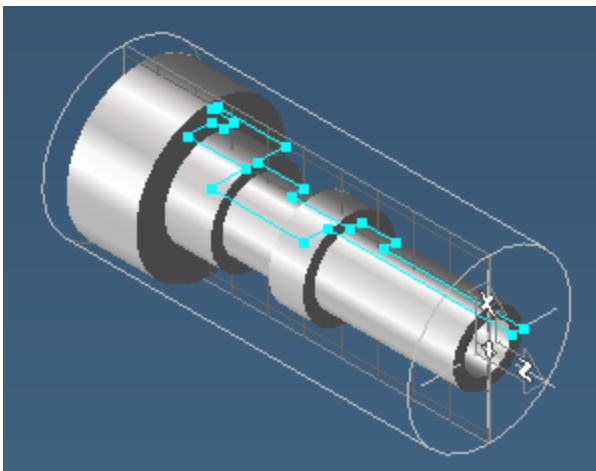
4. Delete the extracted geometry.

We will now extract geometry from the HD Plane.

5. Select the solid and choose **Get Section** from the **Plug-Ins** menu.



6. Select **HD Plane** and the **Negative** side and click **OK**.



The system has extracted geometry on the negative side of the D axis from the intersection of the solid and the **HD Plane** (plane perpendicular to the current CS).

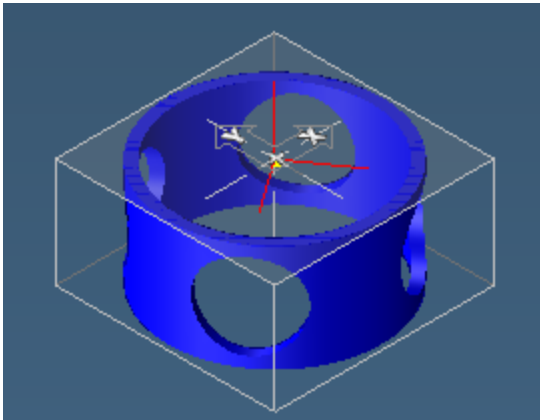
Deburring Process

The Plug-in cleans up selected edges of parts. This exercise will introduce you to the use of the Deburring Process Plug-in.



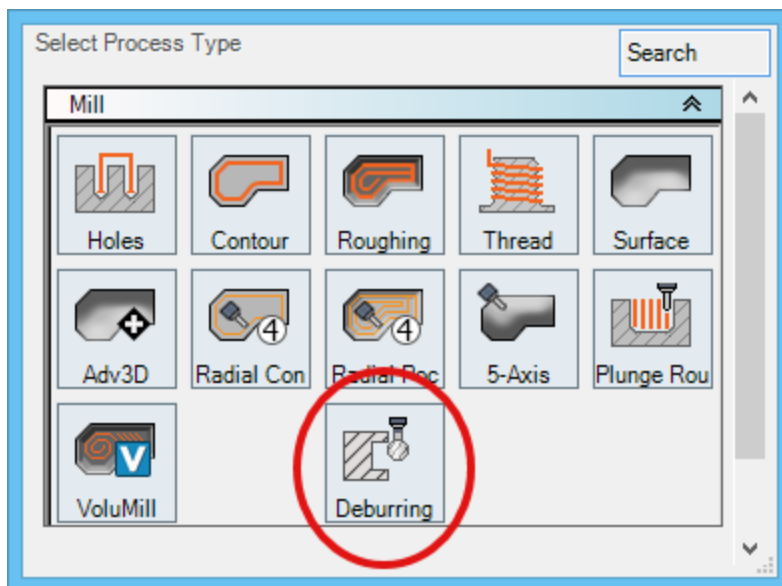
First, ensure that “Deburring Process” is enabled within the Plug-In Manager.

1. Open part `Deburr.vnc`.



We will use the Deburring Process to clean up the inside and outside edges of the four circular holes on the side of the part.

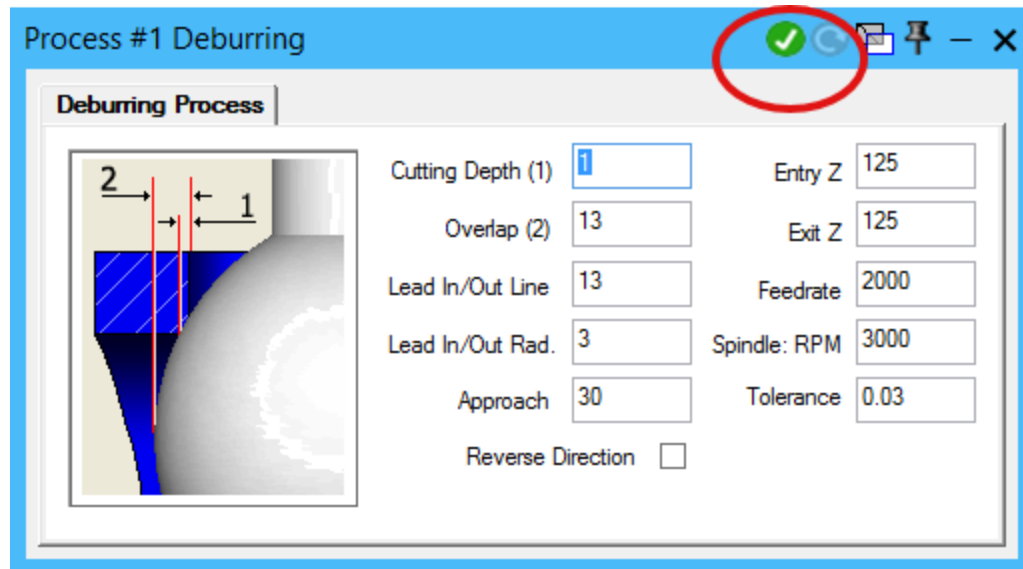
2. Select and drag the form tool provided  onto Process #1 and select Deburring . (You can also use a lollipop tool for this process .)





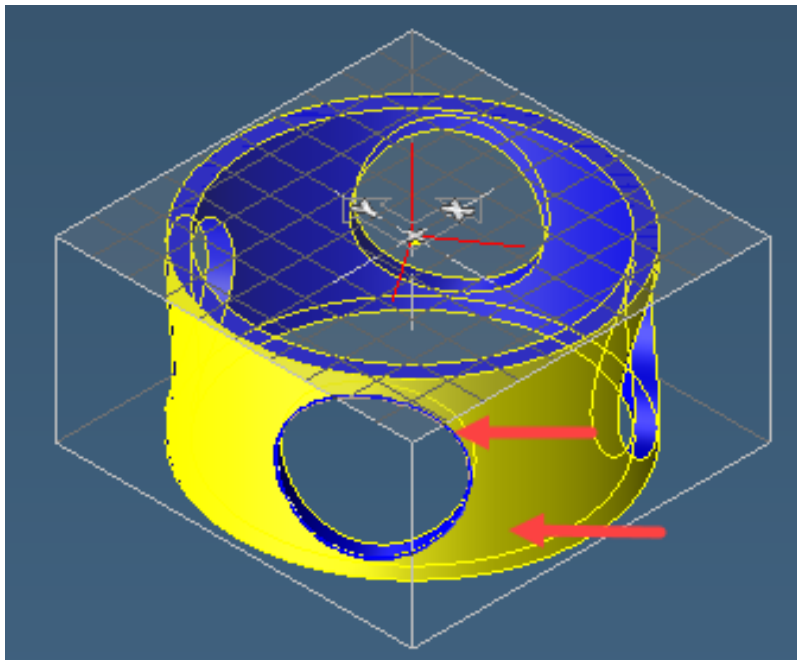
This opens the machining palette and creates a process tile



3. The Process dialog opens. Enter the values shown below.

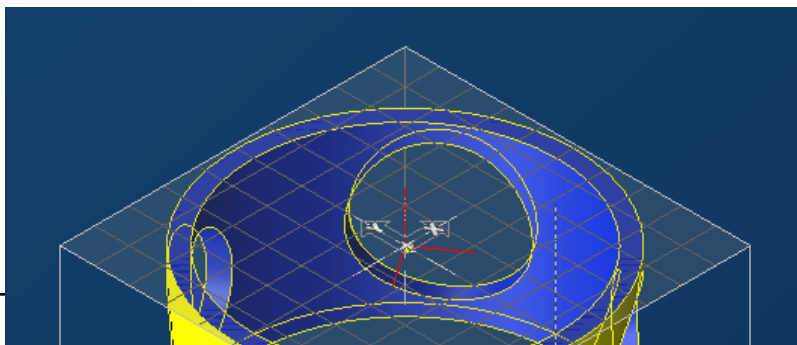



4.  Ensure Face and  Edge selection are enabled and select the outside face and one of the outside edges of the circular holes as shown.



5. You will notice that the Do-it icon is now available on the Process dialog. Click this to create the toolpath.

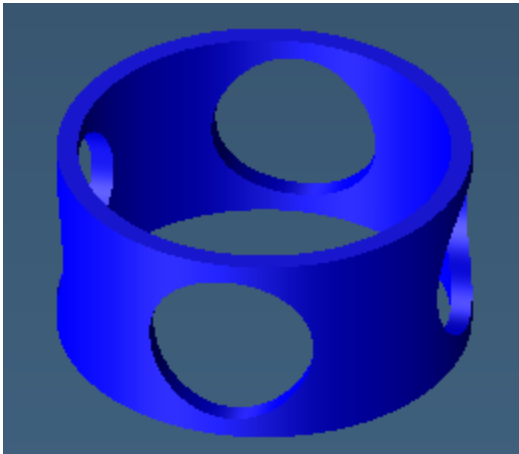
The toolpath will look like this.



6. Deselect the operation in the operation list. Select the outside face and edge of the next hole.
Click  Do it.
7. Using the same selection method, create a deburring operation for the other two outside edges of the circular holes.
8. Use the same selection method to deburr the inside edge by selecting the inside face and one of the inside edges of the circular holes.
9. Repeat the procedure for each of the three remaining inside edges.

Remember to deselect the operation tile immediately after you create it so the next operation doesn't overwrite the previous one. You will now have a total of eight separate deburring operations.

10. Render the operations.



11. Save the part.