



Version 14 : September 2020

Turning 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.

ABOUT THE TURNING TUTORIALS

The Turning module is the most basic of the functions supported by the system. This tutorial provides basic instruction on how to create turning operations. You will be guided through tool setup, process definition, operation creation, custom stock definition, cut part rendering and form tool use.


For creation and using a form tool, see [“Exercise #2: 2D Form Tool”](#) on page 37.

EXERCISE #1: TURNING TUTORIAL

The following tutorial shows how a simple lathe part is set up, tooled and output. Each step is bulleted and additional information and tips are also provided. It is recommended that you read all the information contained in this tutorial in order to gain the most thorough understanding.

The part was created in the Geometry Creation tutorial and a copy is provided in the sample parts file. Dimensions for the part can be found in the part prints section at the end of the tutorial, see “[Lathe Part Blueprint](#)” on page 45.


Refer to the Geometry Creation tutorial for details on constructing the geometry for this part. You

may convert between metric and imperial units (mm and inch) with the **Modify >**  **Scale** command if necessary.

This tutorial explains how to:

1. Setup the Part - How to set up stock, clearance and tool change settings.
2. Specify custom stock.
3. Create a tool list with standard and custom tool settings.
4. Generate different types of operations that rough, contour, drill and thread a part.
5. Render the machining operations.
6. Create posted code for the part.

Part Setup

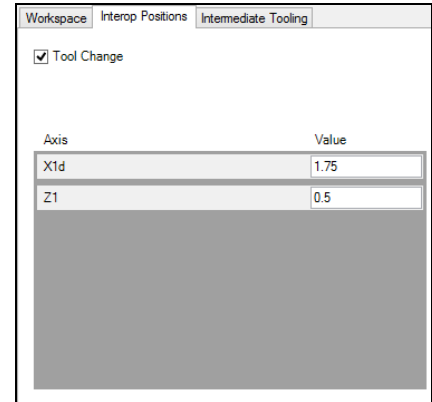
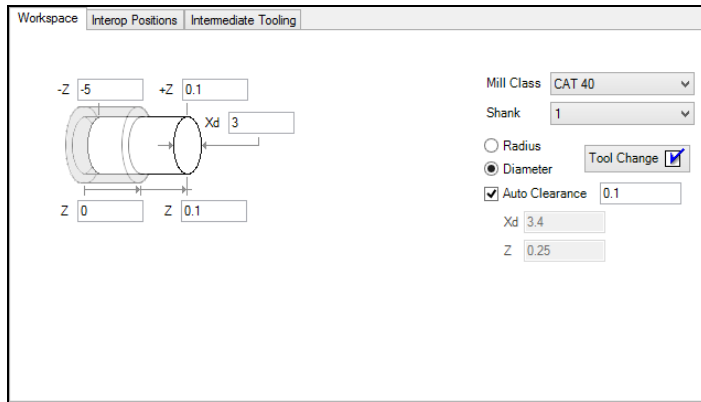
1. Open the Document control dialog found in the top left corner of your screen. You can also use **(Ctrl+1)** or **(F1)**. 

The Document control dialog contains information about the Machine Type, Material, measurement system, stock, Tool Change, Clearance and a part Comment. It also has the file management tools to open, save, close, etc.

2. Select **Open** in the Documents dialog.
3. Locate the Turning tutorial part that you created using the Geometry Creation tutorial.

If you do not have the part, one is provided in the sample parts file (**Sample Parts/Production/Tutorial Parts - Required/Lathe Tutorial.vnc**).

4. Enter the stock, **Auto Clearance** and **Interop Positions** shown.



Interop Positions will send the tool to the specified location for the tool change. If Tool Change is unchecked, it is assumed that fixture offsets are being used, or the operator will manually input the tool change moves.

Auto Clearance sets the default clearance values of each process unless overridden by unchecking Use Auto Clearance in the Process dialogs.


The Auto Clearance option will calculate part clearances (in both Z and Xd/Xr) that are used to position the tool between operations. These values are dynamically calculated as the stock condition changes as material is removed. The clearance positions calculated will also take into account where the tool needs to move to begin the next operation's toolpath. Additionally, the Auto Clearance option may add entry and exit moves as needed to safely maneuver around the part and complete each toolpath.

5. Close the Document control dialog (Ctrl+1), (F1), (Ctrl+W).



6. Open the View palette (Ctrl+5), (F5).



7. Choose Help >  Balloons from the Main menu.

Balloons provide a reference to the interface. The icon will be highlighted in blue, indicating that it is on. Move the cursor over objects to see the balloon help information.

8. Select Unzoom (Ctrl+U).

The part may already be unzoomed if the part was saved.

9. Select the Isometric view (Ctrl+I).

The Isometric view is a 45° perspective of the part. This view gives the best overall perspective of the current status of a part from the top, right and front view of the part.


The Top or Home view is recommended for creating geometry and toolpaths, while the Isometric view is an excellent choice for rendering and viewing the cut part.


10. Choose Help > Balloons or **Ctrl+B** to deactivate them.

11. Switch back to the Top view. (**Ctrl+E** or (**Ctrl+H**).

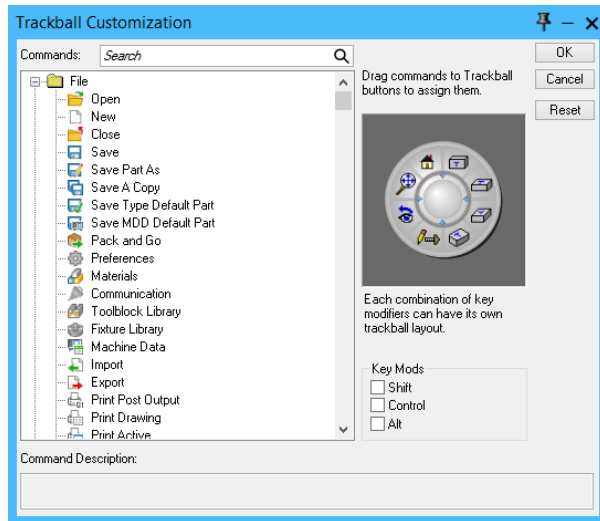
12. Click the View button again to close the View palette, (**Ctrl+5**) or (**F5**).



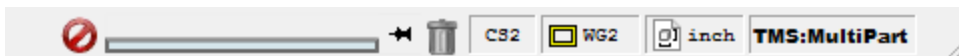
Now look at the bottom left of the screen. You will see an axis marker.  Hover over this marker and you will see a Trackball appear. This is very useful to enable quick view changes to made. You

can change the position of this marker to any corner of the screen by editing the File > Preferences > Display > Edit Appearance Settings > Background location (or turning it off). 


Now try right-clicking on the center of the trackball or view palette. A dropdown menu appears with a Customize option. This option enables you to add common commands to your trackball, simply by dragging them onto the required buttons. Extra commands can be added in combination with the Shift, Ctrl and Alt keys. Use the Reset option to delete the customization.

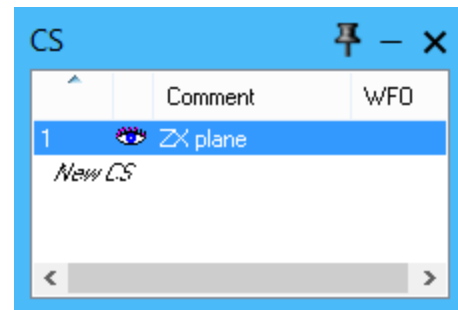


Look at the bottom right border of the screen. This area gives basic information about the part (Inch or metric). Current Coordinate System and Workgroup are shown (click them to open the dialogs).




The Trashcan and Progress bar are also situated here, where they are easily accessible. If your part is to be duplicated, this will also be indicated.

13. Double-click the CS button. The CS dialog opens. Use the  to close the dialog.



Custom Stock

The stock specifications entered in the Document Control Dialog  simply creates a cylindrical stock shape using the X and Z coordinates entered. It is possible to define a custom stock shape by drawing the stock shape in a separate workgroup designated for part stock.

This is very useful because the stock shape affects the machining of the part in that the stock conditions are used to calculate the clearance positioning moves when using Auto Clearance. The custom stock shape also appears when viewing the rendered image of the part.

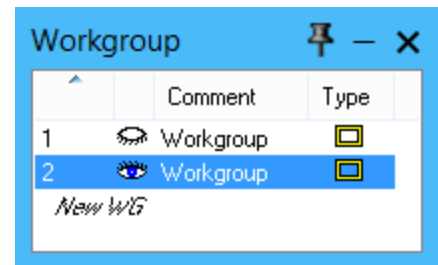
1. Open the Geometry palette in the Command palette (**Ctrl+2**), (**F2**).



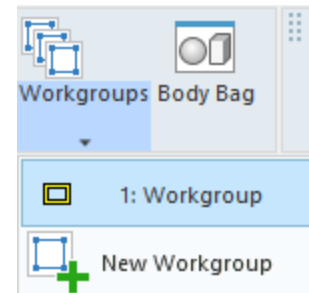
2. **Click** the Workgroup list button.



The Workgroup dialog opens.




This button also accesses a workgroup selection menu if you hover over it with the mouse. The menu will be discussed later in this tutorial.



Or you can simply double-click the WG button on the bottom bar to open the dialog.

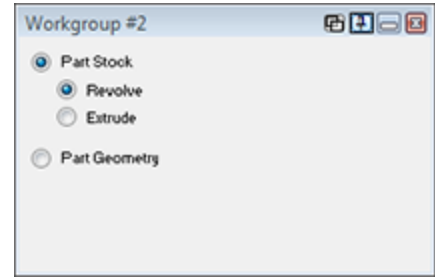
3. **Click** *New WG* and double-click the name and delete it. Rename it "Stock".

A second workgroup is created and is now the current workgroup, indicated by the open eye.

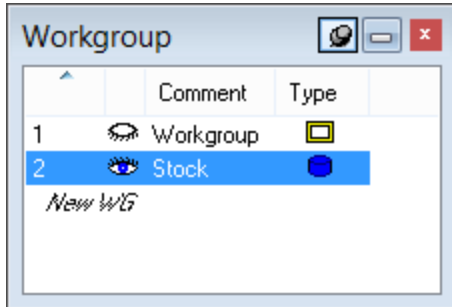
4. **Right-click** the "Stock" workgroup name, from the menu, choose  **WG Info**.

The Workgroup #2 dialog defines whether the workgroup will contain part geometry or a stock outline.

- Select the Part Stock Revolve option.



- Close the Workgroup #2 Info dialog.



You will notice that the icon in the workgroup dialog Type column has changed to a blue solid cylinder to indicate revolved part stock.

In order to create the geometry for the stock outline, it will be necessary to view the part geometry that we have already created as a machining workgroup. The entry highlighted in the Workgroup Selection dialog is the active workgroup.

Other workgroups can be viewed by opening the eye next to their name in the Workgroup Selection dialog. Geometry in inactive workgroups are displayed in gray and cannot be edited while inactive.

- Double-click the closed eye next to Workgroup #1.



Both eyes are now open. Workgroup 2: Stock is highlighted to indicate that it is the active workgroup. Workgroup #1 is visible but inactive (colored grey).

- Close the Workgroup list.



We will use the Mouse-line tool in the Geometry Creation palette to draw a rough outline of the stock. Any geometry creation method can be used to create a stock outline.

- Click the view command  and choose home view.

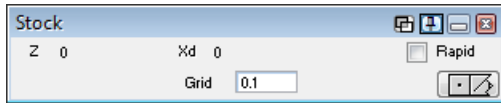


10. **Click** the Line button in the geometry creation palette.

Note that the active Workgroup name is displayed at the top of the palette.



11. **Click** the Mouse-Line button.



The Mouse-Line dialog contains **Z** and **Xd** values determined by the location of the cursor. The **Grid** value specifies constraint spacing the lines will snap to. For a review on using the Mouse-Line tool, refer to Exercise #5: Doodle in the Geometry Expert Section of the Geometry Creation manual.

The initial stock is a cylinder with two different diameters for our stock outline. The stock does not need to be perfect, but should not cross the finish geometry defined. The only requirement is that the bottom line must be drawn at X0 to have a correctly revolved part.

12. Draw a stock shape around the part as shown.

This will be the stock condition before cutting is performed.

13. Close the Geometry Creation palette.

14. Hover over the Workgroup Selection button until the selection menu appears.

15. Select Workgroup 1 by moving the mouse to the option and then clicking.

Tool List

1. Click the Tool list button in the Commands palette.

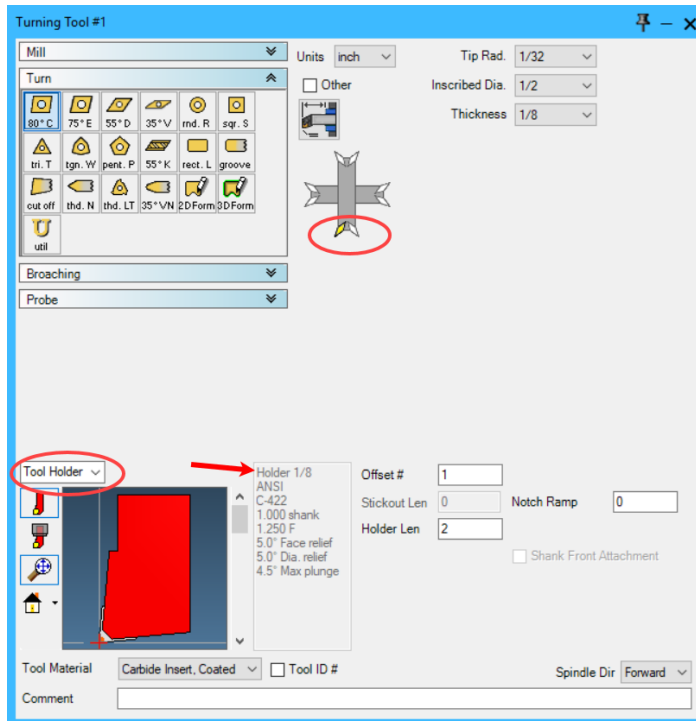


2. **Double-click** tile position #1 in the empty tool list.

The Tool Creation dialog contains all tool specifications necessary for the system to calculate accurate toolpath and rendering. Tool information can be edited at any time by clicking on tool tiles and editing the information contained in the Tool dialog. For more information on Tools, refer to the [Turning user guide](#), "Tools" chapter.

3. Select the **80° C** insert with the following specifications.





Values for insert specifications, such as Tip Radius, IC (Inscribed Circle), Thickness, etc. can either be selected from the list of standard inserts in the menus or manually entered by selecting Other.

Holders can be chosen by using the scroll bar next to the insert diagram. The text to the right of the scroll bar displays the holder specifications which provide detailed information about the insert angle and holder type. For our part, the first position provides the desired selection. The red plus sign at the end of the insert indicates the touch-off point of the tool.

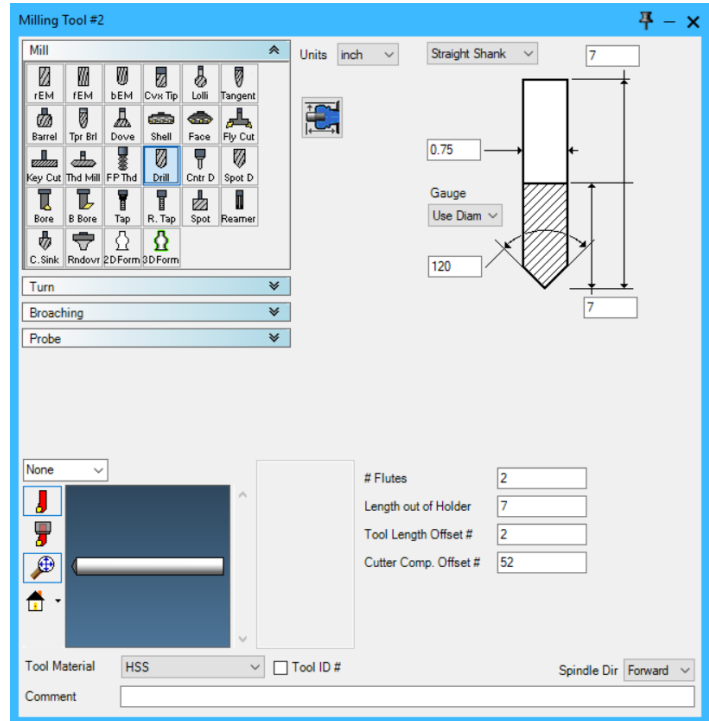
The selection of insert and tool holder also affects the toolpaths created using this tool. The system uses the selections made here when constructing a toolpath to prevent tool interference. Therefore, selections made here directly affect the machining of the part.

4. **Double-click** the empty tool tile at position #2.

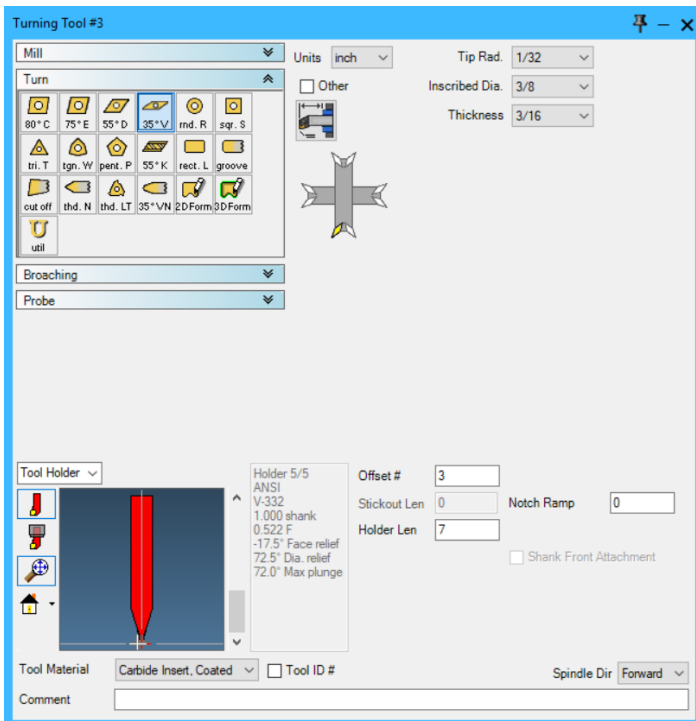
This closes the Turning Tool #1 dialog and opens the Turning Tool #2 dialog. A tool tile displaying the insert type and tip radius you created appears in position #1 of the tool list.

This next tool is a drill, so use the scroll bar to scroll up to the Mill tools.

5. Select the Drill tool with the following specifications.



6. Double-click tile position #3 in the Tool List.
7. From the Lathe Tools, select the 35° V diamond insert.

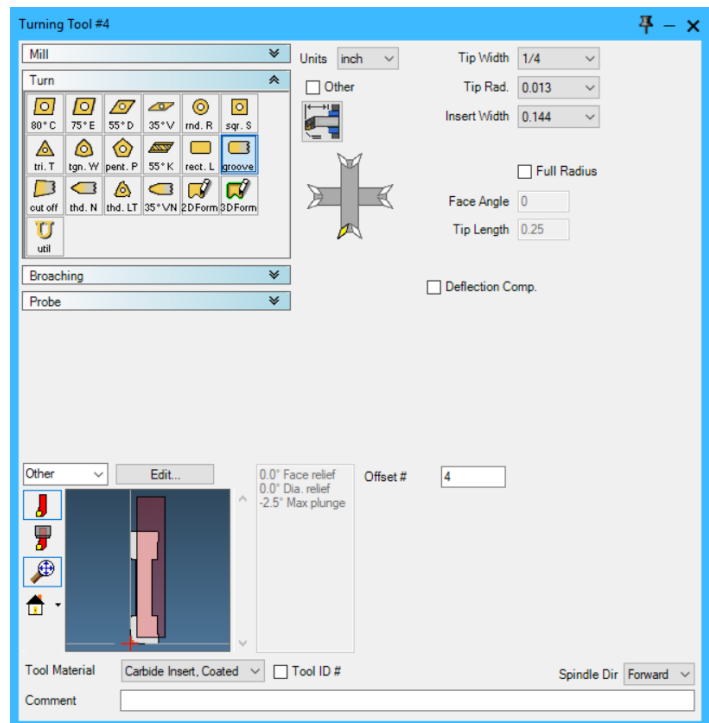


8. Enter the tool information shown.

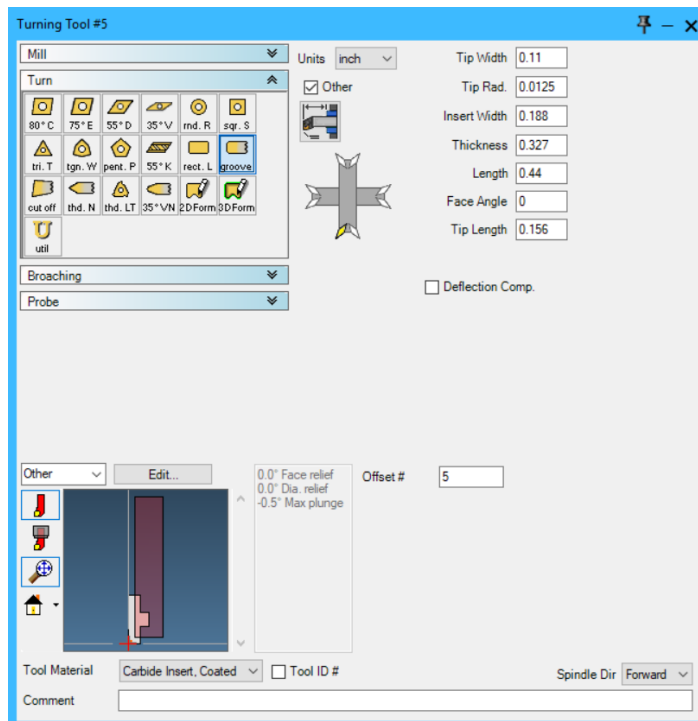
The position on the insert orientation diagram remains the same as the previous insert selection, so no adjustment is necessary.

The insert angle and holder type selection is the last one in the group.

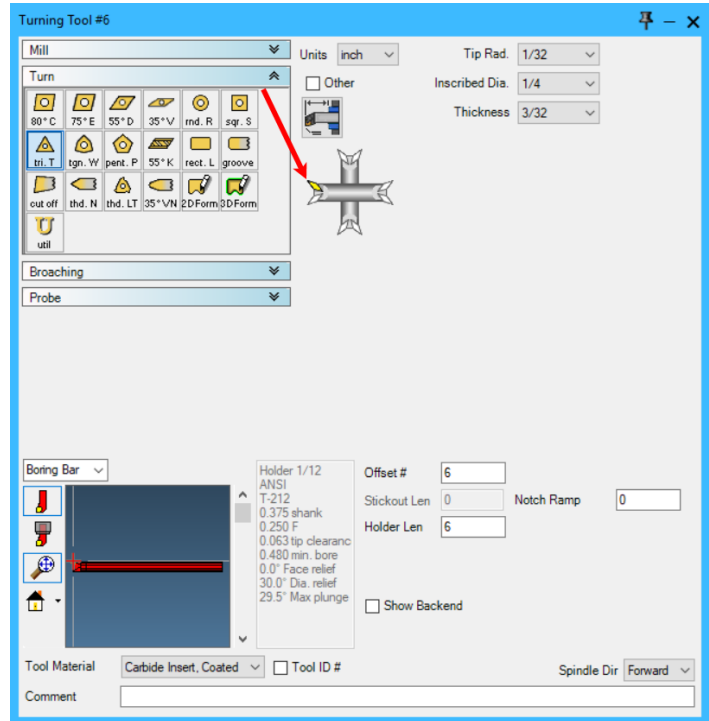
9. Create this insert as tool #4.



10. Create this insert as tool #5.



11. Create this insert as tool #6.



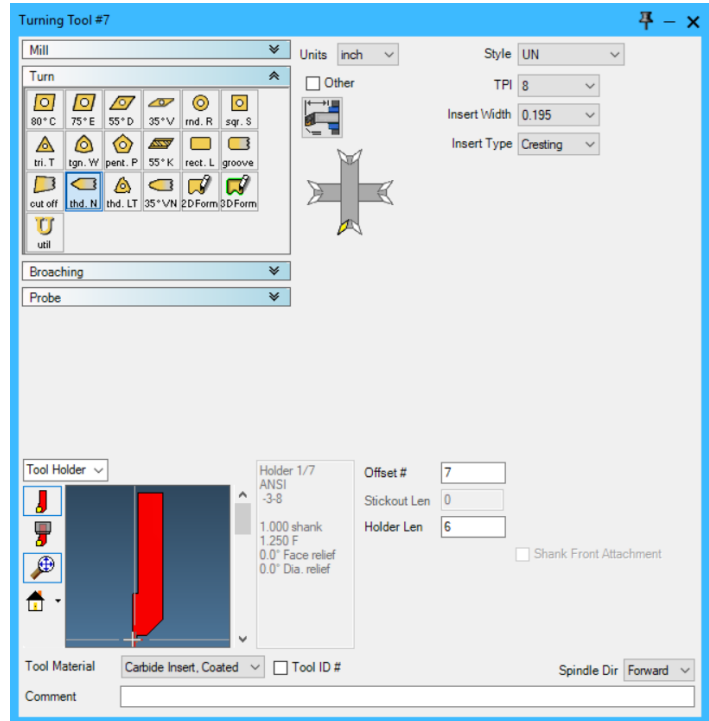
The Insert Orientation Diagram changes in appearance when the Boring Bar option is selected.

12. You can make the tool tile list longer or shorter as you require by positioning the cursor on the bottom of the list.

When the cursor changes to "up and down" arrows, **click** and drag the list up or down.

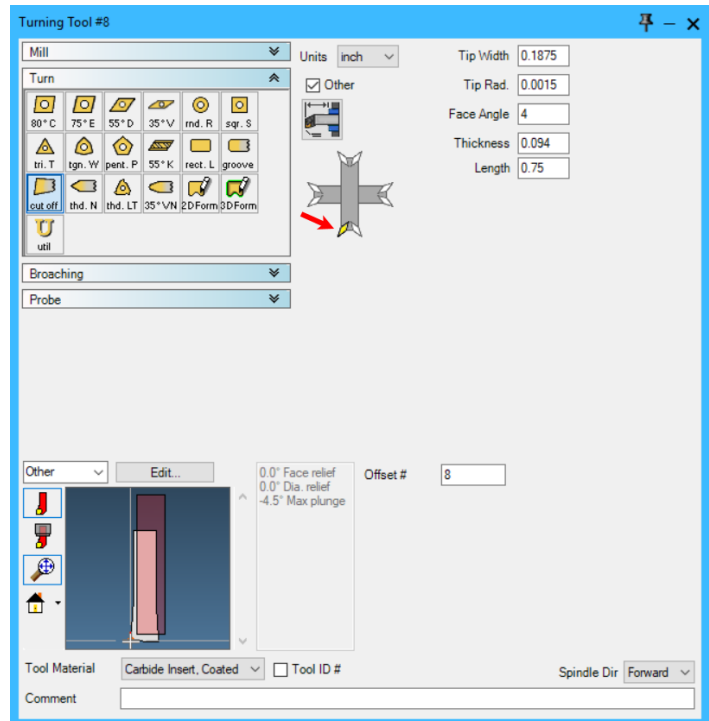
The scroll arrows at the top and bottom of tile lists allow you to easily move through the tiles in the list. To go to the top or bottom of the list, click the smaller underlined arrow button. To scroll quickly through the list, hold down the larger button and drag the mouse downward. This will change the arrow to red and scroll a "page" at a time. For more information on tile lists, refer to the [Getting Started](#) guide.

13. Create this insert as tool #7.



Once the **Style** and **TPI** (threads per inch) values are selected, the **Insert Width** and **Insert Type** will default to the desired values.

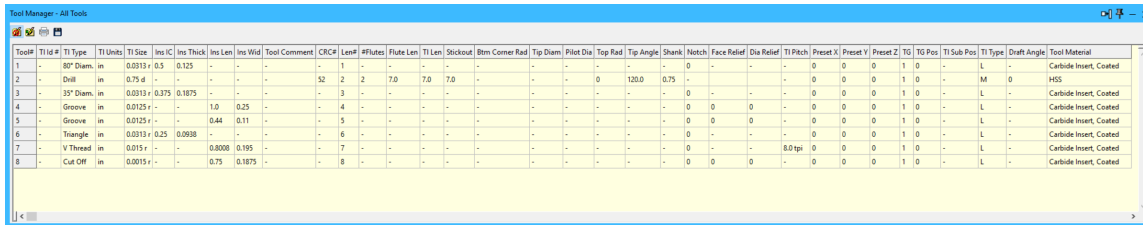
14. Create this custom insert as tool #8.



15. Close the **Turning Tool #8** dialog.

The tools necessary to machine this part have now been created. A good way to review the tools contained in the list is by using the **Window > Tool List Summary**.

16. Choose **Window > Tool Manager**.



The **Tool Manager** contains all of the current tool information in a customizable list.

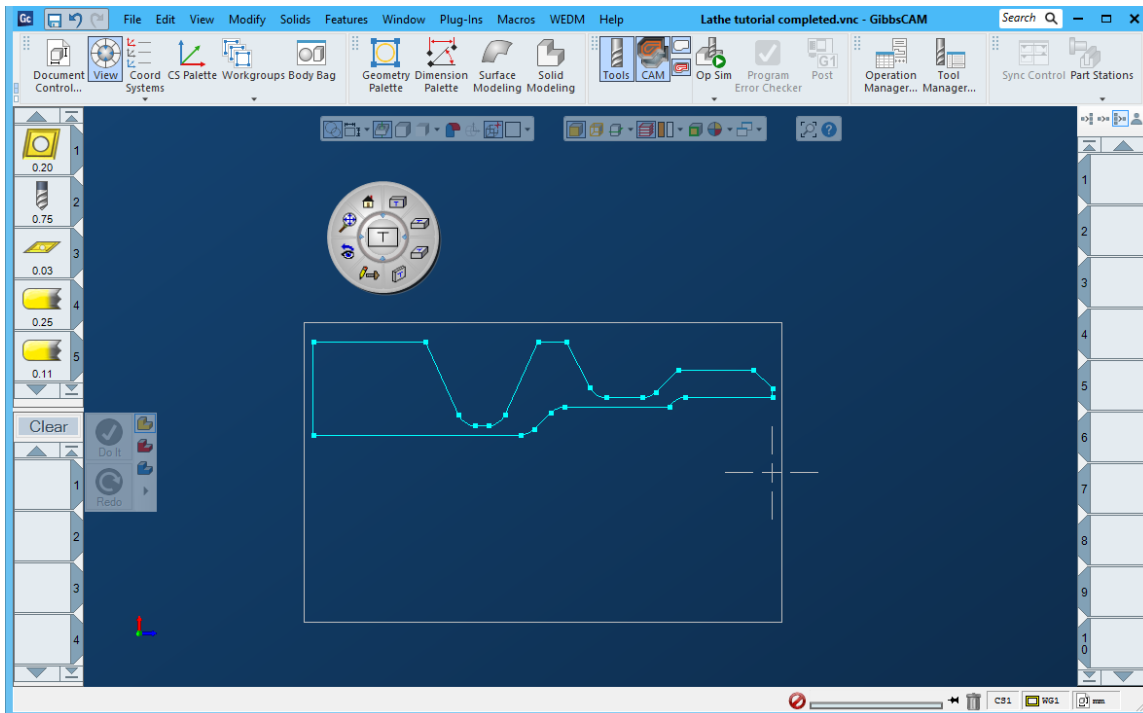
Print or **Save to file** options are available in the **Tool Manager** heading area.

17. Close the **Tool Manager**.

Creating the Operations - OD

1. **Click** the **Tool** and **CAM** buttons in the **Command palette**, (Ctrl+4) or (F4).

The **Tool**, **Process** and **Operation Tile Lists** appear.



In order to machine this part an **Operation #1** must be created. Operations are defined within **Process** dialogs.

To create a Process, one Tool and one Process type (roughing, contouring, drilling, or threading) is needed. Drop a Tool from your tool list onto a Process Tile and choose the process you require. The Process dialog will appear. This is where you define the machining parameters. If you prefer, you can also simply double-click a Process tile, select a Process type, then choose a tool from the suggested tool list (or create a new one).

Once this is done you must select the geometry to define the cut path and a click of the  Do It button creates an Operation in the operation list.

In this tutorial, we will first describe the necessary machining operation, then we will detail the steps necessary to create that Operation.

First, we will create an Operation that will face the front of the part.

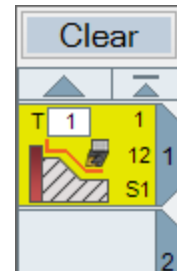
2. Double-click Process Tile #1 and select the Contouring Function Tile.



The Tool selection dialog appears.

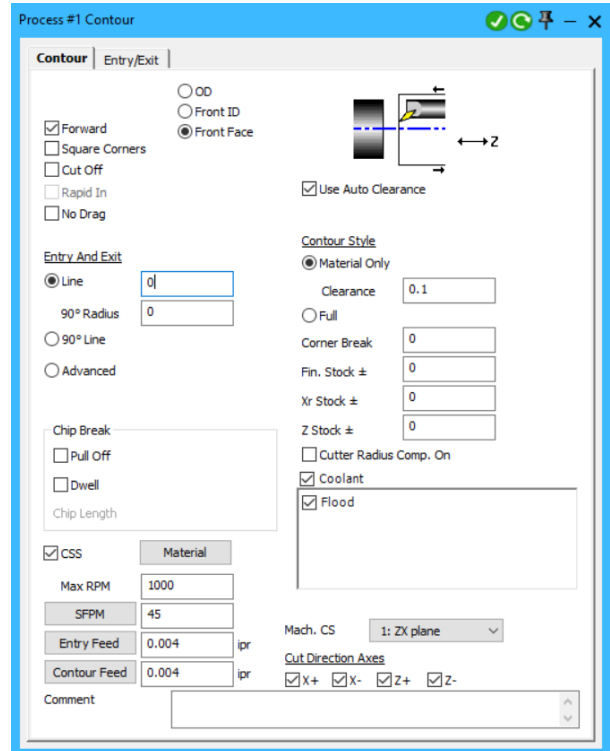
3. Choose the 80° C diamond insert (Tool #1). You will notice that the tool number is now displayed on the Contouring Process Tile.

The Contour Process dialog now appears.



4. Enter the information shown in the Contour dialog.

To position the tool to cut the face of this part, we need the tool to approach and retract along the Z axis. Therefore, the **Front Face** option is selected.



For this process, the **Material Only** option is selected so that no toolpath will be generated unnecessarily to cut the face.

5. Close the Contour Process dialog.
1. Start point
 2. Offset cut
 3. Start feature
 4. End Point
 5. End Feature
 6. Centerline
 7. Cut Direction

When the Process dialog is complete, the cut shape must be defined.

Machining Markers are used to designate a cut shape for contouring processes.

8. Select the vertical line on the X axis at Z=0.

The Machining Markers will appear on the selected feature. The black and white arrow markers allow you to specify the start and end feature and the start and end point of the cut shape. They can be “picked up” by **clicking** on them and dragging them to any location on the part

geometry. The arrows allow you to choose the cut direction. The two circles allow you to choose the side from which the tool will be offset.

In this case, the start feature and end feature markers will stay in their initial positions because only one feature, the face, will be cut. The start point and end point markers will be moved to cut the entire length of the face. The start and end point markers can rest anywhere on the selected line, even on the trimmed sections. For more information on Machining Markers, refer to the Turning user guide, "Machining" chapter.

If you find that the Machining markers are too small, the size can be changed in the Preferences dialog. Go to **File > Preferences > Display > Marker Size** and choose the size that is right for you.

9. Select the outside circle.

This is the OD side of the geometry.

10. **Click** the arrow pointing down.


We will approach the line from the OD and bring the tool just past the center to ensure a complete facing is done.

11. Drag the round white start point marker above the workspace as shown.

Because **Material Only** is used in this process, the start and end point markers do not need to be placed in exact positions as long as they clear the stock outline we defined.

12. Drag the end point marker down to the location shown.

If markers are not properly placed a process may not generate the desired toolpath. The common mistake is to chose the wrong cut side.

13. **Click** the  **Do It** button at the top of the Contour dialog.

An Operation tile will be created and added to the Operation list on the right of the screen. The approach and retract moves, and the toolpath will be drawn. The orange lines represent the toolpath, while the dashed gray lines represent positioning moves. The solid lines for both the toolpath and positioning indicate feed moves. And the dashed lines of both colors indicate rapid moves.

The picture above on the right shows the toolpath without the Machining Markers so you can see it better. To view the toolpath without the Machining Markers, simply **click** anywhere off

the part. The dashed gray line shows the tool rapiding from the tool change position (entered in the Document dialog, to the start point of the toolpath).

The angled lines at the beginning and end of the toolpath are entry and exit positioning moves that have been added because Auto Clearance is turned on. They are angled along the centerline of the insert angle. The position the tool can safely rapid to before beginning the toolpath is determined by the Auto Clearance value entered in the Document dialog.

14. Deselect the operation created by **clicking** on an empty Operation location, or by **clicking** an insertion point between Operation locations.

Tiles are considered selected when they are highlighted in yellow. Deselected tiles are the standard gray color.

Creating Operations

i

When operations have been satisfactorily completed, it is VERY IMPORTANT before proceeding that all operations are deselected. By deselecting the completed operations, they become fixed in the Operations List and will not be affected by changes made to the Process List. Operations can be edited, but to do so they must be reloaded into the Process List by **Double-clicking** on them.

1. Mouse-drag the Process in position #1 to the Trashcan to delete it. When the hand icon is over the trashcan, the trashcan lid will open. Letting go the mouse button will then deposit the Process in the Trashcan.



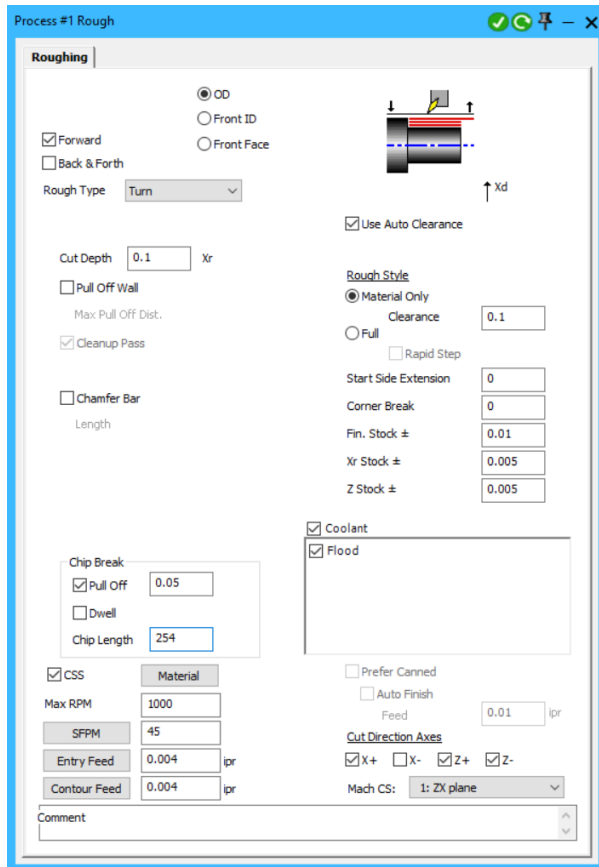
When creating a new Process, you can either throw away the old tiles in the list or modify them in order to create the new operations. In this case we will throw away the old tile and start from scratch.

The next group of operations will rough and finish the outside contour of the part. This will be accomplished using multiple process programming which allows multiple processes to be applied to one cut shape.

Multiple Process Group Op 2-4

2. Create the following Rough process by double-clicking Process Tile #1, choosing Rough from the available processes and then selecting the 80° Diamond insert tool (Tool #1).





To properly position the tool to rough the OD of this part, we need the tool to approach and retract along the X axis. Therefore, the OD option is selected for the approach type.

Lathes use different roughing (multiple cut) cycles. There are three general types: Turn, Plunge, and Pattern Shift. These three types of cycles will allow us to output the different canned cycles defined in programming manuals for lathes. The software will output these roughing cycles as canned cycles if the Prefer Canned option is selected and the lathe is capable of using canned cycles. If not, the post processor will output every move necessary to rough the part.

Turn roughing cycles are the standard contour rough cycle. For turning cycles, the selection made for the approach type (OD, ID, Front Face) determines what type of contour rough style will be generated by the system.

Plunge roughing cycles are used to generate grooving operations. These cycles are used when it is desirable to rough the part with a groove tool. Pattern Shift roughing cycles are commonly used when roughing a cast part. This type of cycle generates the part contour and shifts the toolpath on each successive cut until the specified area is roughed.

In this process, we will use a turn roughing cycle. When a turning cycle is selected, a Cut Depth value is entered which designates how much material will be removed on each pass.

For this process, the Cut Depth value entered is 0.1. The Xr label indicates that it is a radius value. This means that 0.1" of material will be removed from each side on each pass.

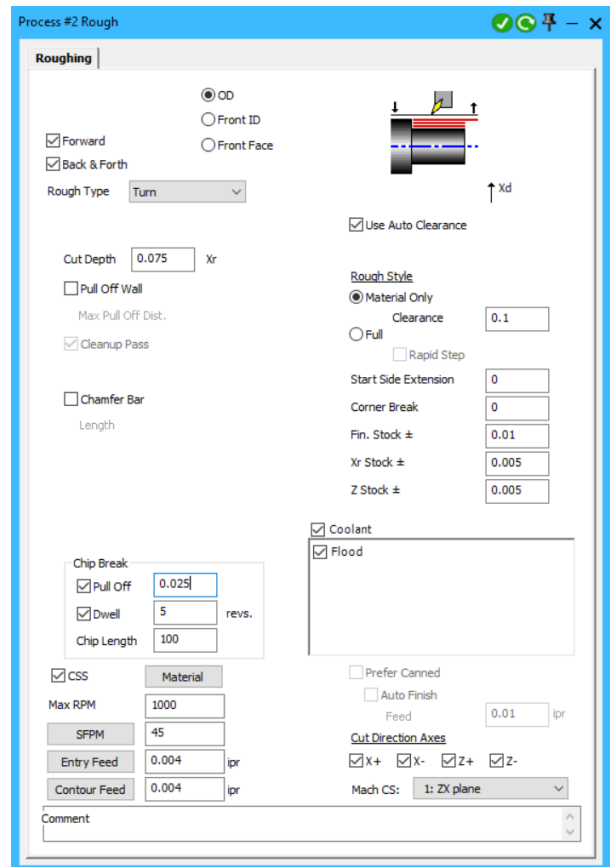
The Cut direction axes check boxes allow you to regulate the axes and directions of the cut shape. Omitting an axis (such as X-) will prevent cut shape moves in the disallowed direction. For more information on cut shapes and the Cut direction Axes buttons, refer to the Turning user guide "Machining" chapter.

For this process, the X axis in the negative direction is turned off. This will prevent the toolpath from making any moves in the X- direction so it does not go into the grooves.

The value entered in the Fin. Stock \pm text box designates the minimum amount of material that will be left outside of the part geometry after the roughing cycle is completed. The toolpath created by the system will be offset from the part geometry in Z and in Xd. We will remove this excess with the finish pass.

- Close  the Rough Process dialog.

- Create another Roughing Process with the 35° Diamond Insert (Tool #3) in position #2.



This process is included in this group to rough the neck groove on the part. Again, an OD approach type is selected so the tool will approach and retract along the X axis. And a Turn rough cycle is selected using a Cut Depth.

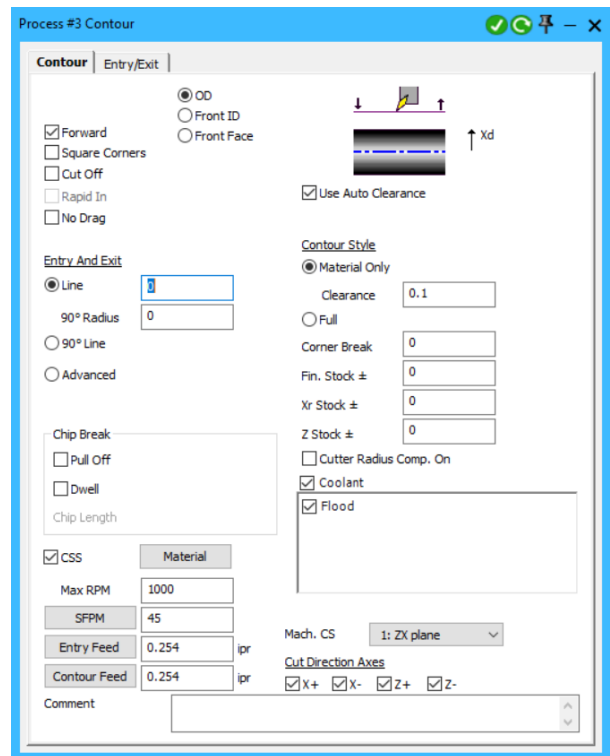
Material Only is again selected for the Rough Style so that the toolpath generated for this operation will only cut the remaining material left on the part after the previous operations have been completed. The Material Only selection is particularly useful when utilizing the multiple process programming capabilities of the system. Feed moves will only be made where it is necessary to remove material. The tool will make rapid moves whenever possible while maintaining the clearance offset.

We have also enabled chip breaking, which will occur if the chip length becomes greater than 100.

In addition, entering a finished stock amount will leave the designated amount of material around the entire cut shape.

- Close  the Rough Process dialog.

- In position #3 create this Contour process with tool #3.



- Close the Contour Process dialog.

Now that the Process List is complete, we will define the cut shape.

- Click the chamfer as shown.

Machining Markers will appear on the selected feature.

- Click the outside circle. (1)
- Click the arrow pointing up. (2)
- Drag the Start Point Marker to the position shown. (3)

The Start and End Point Markers do not need to be placed in exact locations as long as they clear the material.

- Drag the End Feature Marker (black square) and drop it on the part geometry as shown.

The End Point marker will automatically move with the End Feature marker.

Alternately the End Feature Marker can be placed by pressing (**Ctrl+Shift**) and **clicking** on the end feature geometry.

When positioning any one of the four machining markers on a geometry feature, place the tip of its arrowhead on the geometry feature.

13. Drag the end point marker to the position shown.

The Dark blue line represents the cutting path. Notice that the middle V groove and the O-ring groove are not included in this cut shape.



14. **Click** the Do It button or press (**Ctrl+**).

This will create the toolpath. If any mistakes were made, edit the process and use the Redo button to make changes to a selected operation.

Three operations are created.

The Operation list should have four operations now.

The positioning moves and toolpaths for each of the operations are also drawn on the screen. In order to get a better picture of the toolpaths generated by the system, we will view the tool moves for the first roughing operation separately.

15. Select Operation #2.

Clicking once on a tile will select and highlight the tile without opening any dialogs or selecting other tiles that were created with it.

16. Open the View Control  palette.

17. **Click** the Redraw button (**Ctrl+R**).


Only the toolpath and positioning moves for the selected operation are drawn. This is helpful if you use multiple process programming and wish to view individual toolpaths.

Notice that the custom stock outline was taken into account because **Auto Clearance** is on and the **Material Only** option was selected. Also, the toolpath created does not slope at all into the neck groove because the **X-Shape Axis** was deselected in the **Rough Process** dialog that created this operation.

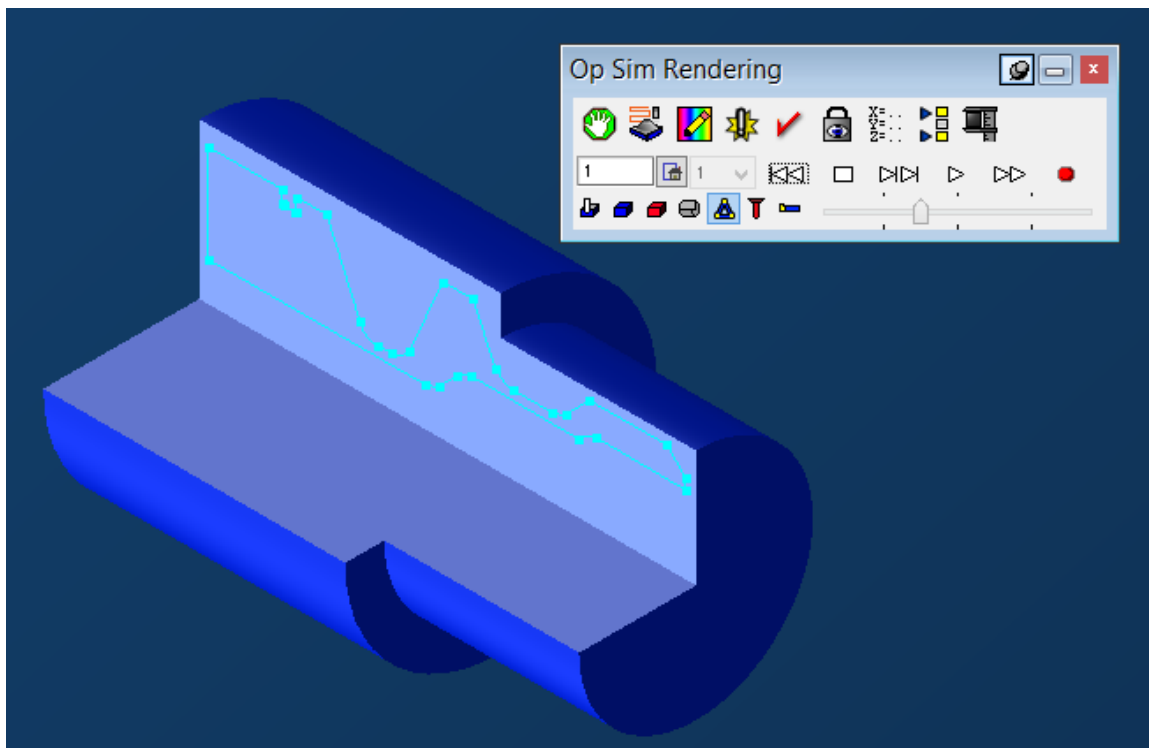
The dashed gray lines show the tool rapiding from the end point of the previous operation to the start point of this operation, and to the tool change position at the end of the operation.



Now that operations have been created, we will render the part. There are several ways to verify that the toolpaths generated by the system will safely and correctly cut the part. Looking at the toolpaths, as we have done thus far, is one way. Another way is to render the part which allows you to actually view the tool removing material and cutting the part.


Now select **Operation Tile #3**. You will see the chip breaking toolpath. You will probably need to slow the display speed dramatically to see this properly.

18. Choose **View >**  **Isometric** from the Main menu (**Ctrl+I**).

19. Click the **Op Sim** icon  on the Command palette.




The initial stock we defined at the beginning will display as a solid object. Ensure that the **Lathe stock cutaway** button is set to  **quarter cutout**, the **tool visibility** is set to  **visible tool**. Use the slider to slow down the rendering speed if necessary.

The Geometry is hidden but can be overlaid by using the  show geometry button (**Ctrl+[**). Options can be changed without resetting the rendering, or you can rewind to watch from the beginning.

20. **Click** the Play  button.

The tool is shown whilst the part is rendered. A part can be rendered at any time once operations have been created. The stock will always be shown in shades of blue. Selected operations will render in shades of yellow, while all other operations will render in shades of gray. Red will be displayed when non-cutting tool surfaces or rapid moves hit the material. For more information on Cut Part Rendering, refer to the Common Reference guide, "Cut Part Rendering" chapter.

21. Close  the Rendering palette (**F6** or (**Ctrl+6**).

22. Choose View Top from the menu or (**Ctrl+E**).

This is also the same as the Home view in CS1.

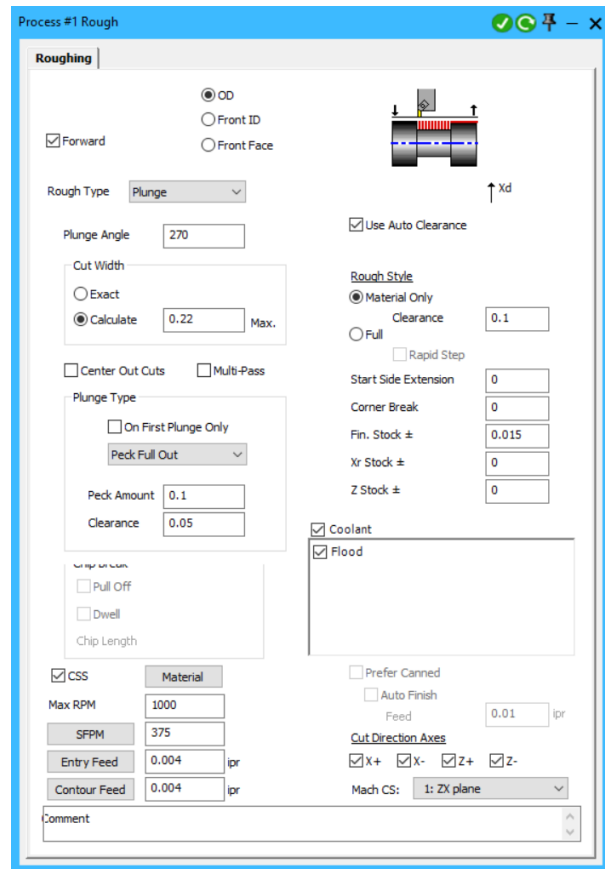
23. Deselect the operations by **clicking** on an empty operation location or by **clicking** on an insertion point between operation locations.

24. (**Shift+select**) the three tiles in the Process List and either drag them to the trash can or click the Delete key.

Multiple Process Group Op 5-8

1. For Process 1, create this Rough process with the first Groove tool (Tool #4).

An OD approach type is selected so the tool will approach and retract along the X axis. Material Only is again selected for the Rough Style. Also, a Finished Stock amount is left on this groove that will be removed in the following finish pass.



In this process, we will use a Plunge roughing cycle.

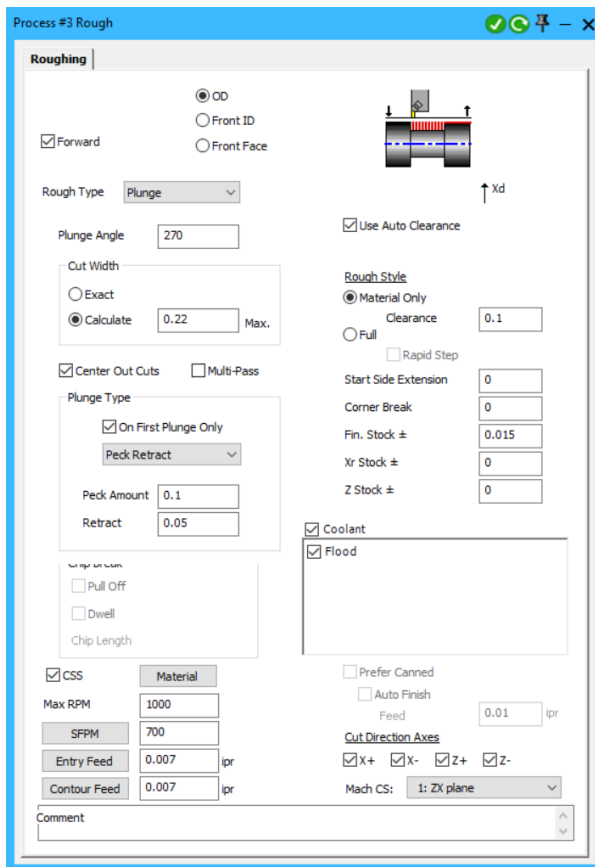
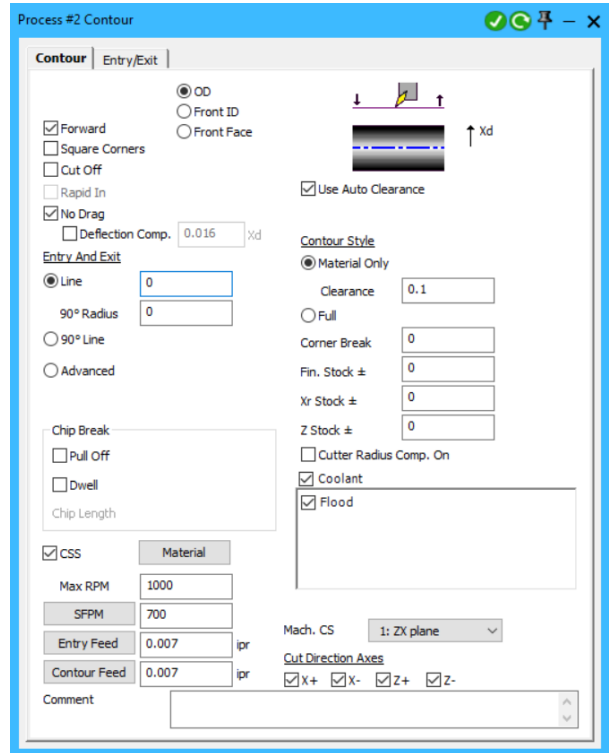
When Plunge is selected for Rough Type, the Plunge dialog options will appear. The Plunge Angle specifies the angle at which the groove tool will make its plunging moves. The default is 270° which has the tool plunging straight down.

For this process, we have selected the Details option for the Cut Width. This option will vary the Cut Width distance as necessary in order to hit the endpoints of each feature of the cut shape. The Z value entered specifies the maximum distance the tool will step over on each pass. The Center Out Cuts option is selected so that the tool will make its first plunging move in the center of the groove, and then proceed to rough each side.

For this groove, the Peck Full Out option is selected because the groove is relatively deep, and in order to remove all of the chips (material), the tool will need to come all the way out of the part after the peck. The value entered in the Peck Amt text box specifies the distance the tool will plunge into the part on each peck. The Clearance value specifies the distance away from the remaining material the next peck will begin.

- For Process 2, create this Contour process with the same Groove tool (Tool #4).

The No Drag option is turned on in this process so that the toolpath is created in such a way that only the positive insert angle of the tool cuts material. The designated cut shape is segmented into pieces to accomplish this.



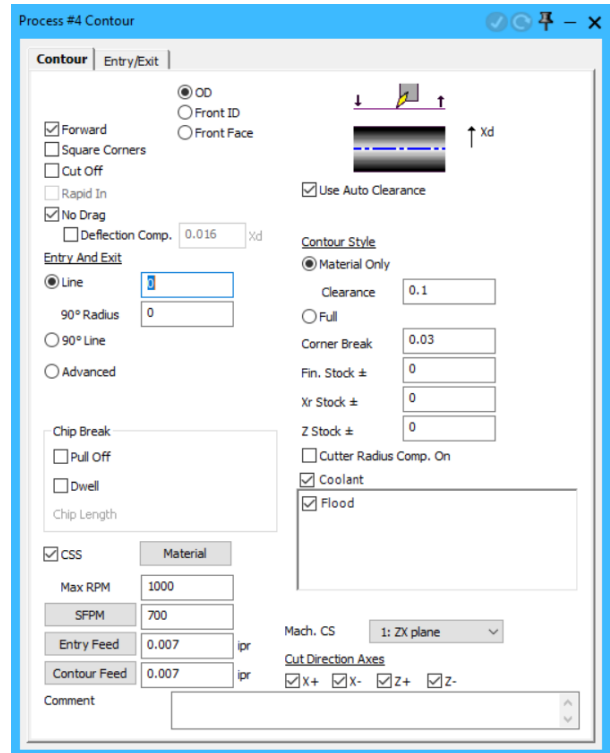
- For Process 3, create this Rough process with the second Groove tool (Tool #5).

An OD approach type is selected so the tool will approach and retract along the X axis. Material Only is again selected for the Rough Style. Also, a Finished Stock amount is left on this groove that will be removed in the following finish pass. Plunge is selected for the Rough Type.

The Exact option is selected for the Cut Width. The tool will step over on each pass regardless of the endpoints of the cut shape. The Peck Retract option is selected for the First Plunge. The Peck Amt

specifies the depth of each plunge and the **Retract** amount specifies the distance the tool will come out of the groove after each peck.

- For Process #4, create this Contour Process with the same Groove tool, (Tool #5).



Now we will define the cut shape for this process list.

- Click the part geometry as shown.

Machining Markers will appear on the selected feature.

- Right-click the last feature shown and choose Move End Feature here so the Machining Markers appear as shown.

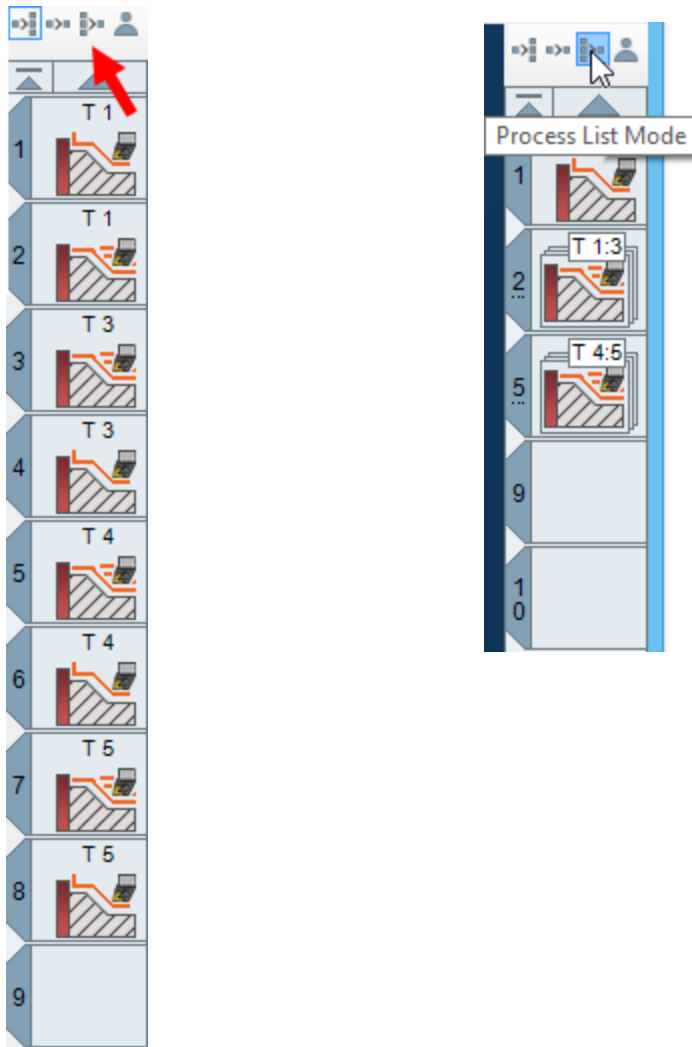
Make sure you select the outside circle and the arrow pointing left. The blue cut shape drawn should include the V groove and O-ring.

- Create the toolpath.

The toolpaths that are created will properly rough and finish both the V groove and the O-ring. Notice that the toolpath contains no unnecessary moves. **Material Only** takes full advantage of multiple process programming without making unnecessary tool moves.


Now look at the Operations list. Just above the list are a series of options.

8. Click the button as shown. This will enable Tile stacking, where consecutive operations machining the same geometry and belonging to the same flow can be stacked together, thus reducing the space they occupy on the screen.



Note that the operation tile numbers remain unchanged.

9. Now switch to the Isometric view (**Ctrl+I**).

10. Render  the Operations. Use the slider to slow the rendering if necessary.

During the rendering process, you will be able to better see the **No Drag** contour option at work. Notice the tool will finish one side of the groove, rapid up and over, and then go down and finish the other side of the groove, always cutting along the positive direction of the insert angle.

11. Exit Rendering mode.
 12. Switch to the Top view (**Ctrl+E**).
 13. Deselect any operations.

14. With the Process list selected choose **Edit** >  **Select All (Ctrl+A)** and delete the tiles or drag them to the Trashcan.

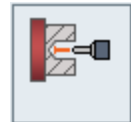
Except for the thread, the OD of our part is complete.

Creating the Operations - ID

Now, we will create the operations to drill a hole through the center of the part and rough and finish the ID.

Multiple Process Group Op 9-11

1. Create this Holes process with the 0.75mm drill tool.

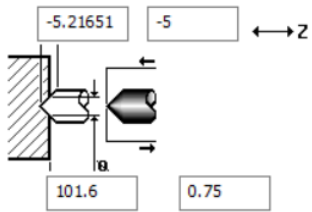


Process #1 Holes

Holes

Entry/Exit Cycle:

- Feed In - Rapid Out
- Feed In - Feed Out
- Tap
- Rigid Tap
- Peck, Full Out
- Peck, Chip Breaker



Use Auto Clearance

Material

Speed: RPM 1000

Feed: Plunge 0.01 ipr

Dwell 0 sec 0 revs

Clearance 0.05

Peck 0.5

Retract

Prefer Canned

Coolant

Flood

Mach. CS 1: ZX plane

Comment

The depth the drill will feed to is specified by the value entered in the Sharp Tip Z text box. Values may be entered in the text boxes specifying the Full Diameter Z depth and the Spot Diameter. These values will cause the Sharp Tip Z to be recalculated. The Sharp Tip Z is the value used in the posted output.

The drill cannot feed through the part in one pass. Therefore, the Peck, Full Out option is selected. Because it is a Peck, Full Out, the drill will rapid all the way out of the part after each peck.

The value entered in the Peck text box specifies the amount the tool will peck into the material on each pass. The next peck will start a Clearance amount away from the material remaining after the previous peck.

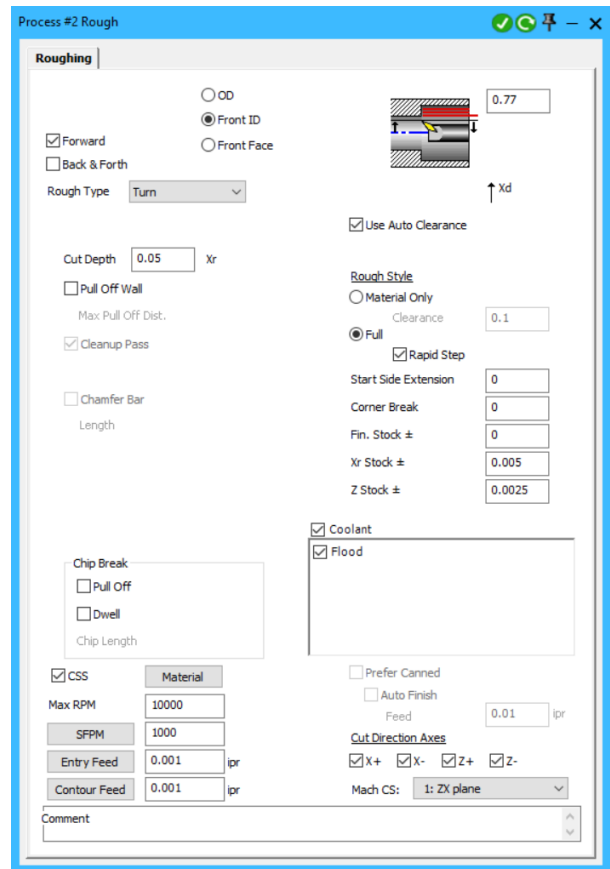
i

Remember to use the Balloons in the Help menu if you do not understand the fields.

Drilling operations do not require the selection of geometry. In a standard lathe a drill can only feeds in at X=0. Therefore, the placement of the Machining Markers will only affect the toolpath created for the Roughing and Contouring operations.

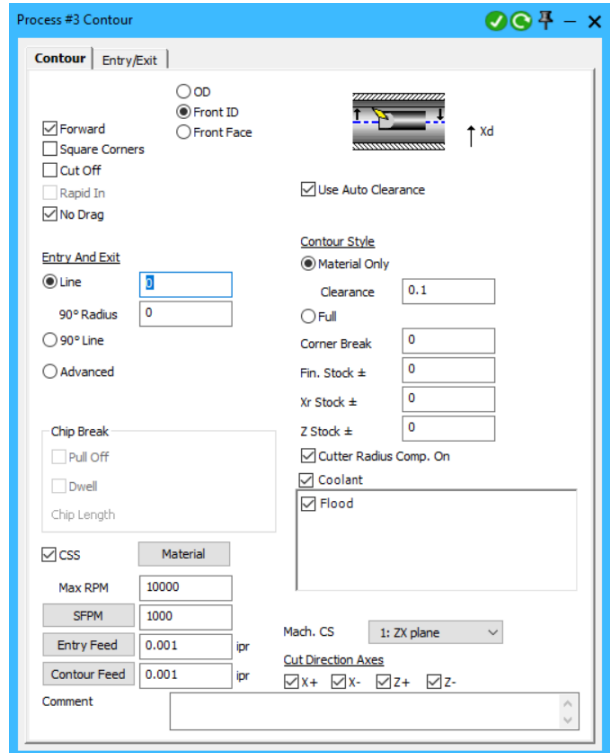
- For Process #2, create this Rough Process with the Triangle insert (Tool #6).

A Front ID approach type is selected so the tool will approach and retract from the part along the X axis. A Turn Rough Type is selected and separate stock amounts are entered for each axis.



- For Process #3, create this Contour process again using the Triangle insert (Tool #6).

The Front ID approach type is selected and Material Only is turned on in the process.



- Position the Machining Markers as shown.

- Create the toolpath.

The dashed orange line at $X = 0$ shows the tool rapiding out of the part after it has pecked in to the Sharp Tip Z depth in the Drilling operation.

- Switch to the Isometric view.
- Render the Operations.
- Deselect the operations.
- Clear the Process list.

We will skip stating these last two steps from now on.

- Switch back to the Top view.

Threading

Op 12

Next, we will create the operation that will cut the required thread.

1. Create this Thread process with the Threading tool (Tool #7).

Process #1 Thread

Thread

From Parameters
 From Shape

Xr Shift

Z- OD
 Z+ Front ID

Style: UN

Nominal Xd: 2.25
 TPI (in): 8
 Pitch (mm): 3.175
 Taper (Slope): 0
 # of Starts: 1
 Major Xd: 2.25
 Minor Xd: 2.09665
 Thrd Ht Xr: 0.07666

Material

Speed: RPM: 1000

Use Auto Clearance
 In Feed:

- Balanced
- Angle: 29.5
- Alternate

Depth of Cut:

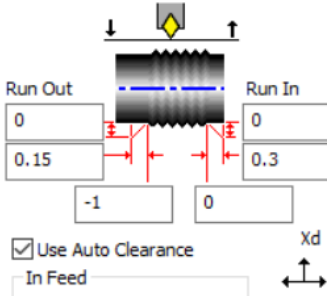
- 1st: 0.002 Xr
- One Finish Pass
- Const Cut
- Const Load
- Last Cut: 0.01 Xr
- Spring Pass: 0

Mach. CS: 1: ZX plane

Prefer Canned

Coolant
 Flood

Comment



These values can be found in the Machinery's Handbook, and are calculated from the values entered for an "ideal thread."

The Minor Xd will default to a calculated value according to the Nominal Xd and the desired pitch. The Thrd Ht Xr is calculated $[(Major - Minor)/2]$. Changing the Nominal Xd will change the values in the Major Xd, Minor Xd, and Thrd Ht Xr.

Const Load is selected to maintain the same amount of tool pressure on each pass. The Last Cut option is selected to prevent any cut from removing less than the value entered from each side. This is also the cut depth that the threading tool will remove on the last pass.

Threading operations do not require the selection of geometry. The information entered in the Thread dialog determines where the thread will be located on the part.

2. Create the toolpath.

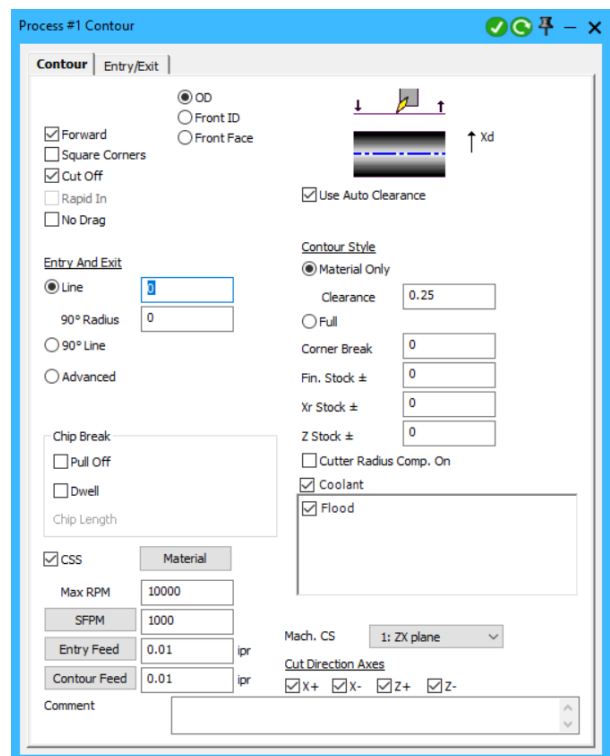
Cut Off

Op 13

Finally, we will create an operation to cut off the machined part from the bar stock.

1. Create this Contour Process with the cut-off tool (Tool #8).

An OD approach type is selected so the tool will approach and retract along the X axis. Material Only is selected for the Rough Type with a Clearance of 0.25. The Clearance value is relatively large to ensure that no tool interference occurs with our finished part.



The Cut Off option is turned on so the post processed output will contain the necessary codes for removing the part from bar stock. Not all post processors support this option.

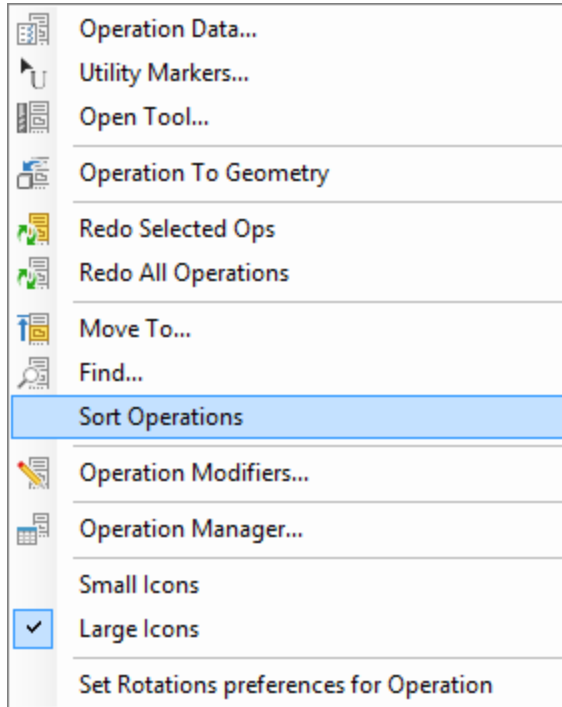
2. Close the Contour Process dialog.
3. Select the cut off line of the part and position the Machining Markers as shown.

Ensure the end point extends past X0 and the start point begins above the stock.

4. Create the toolpath.

As the last step we will sort the operations to have fewer tool changes.

5. Click **Sort Operations** from the Operations right-click menu.



The only change that occurs is the Drilling operation that uses Tool #2 will be moved up in the Operation List to location #3.



Whenever the order of operations is changed, it is good practice to verify that the positioning moves and toolpaths are still valid. When using the Auto Clearance option and/or Material Only option, the material condition is taken into account when it creates the tool moves for each operation. Changing the order of operations has the potential to change the initial material conditions for existing operations. Rendering the part is a good way to check the validity of the toolpaths.

6. Select the Isometric view. (**Ctrl+I**)
7. Render the Operations (**F6**).

For this case, changing the order of the operations does not cause any tool interference and does not appear to affect the efficiency of the toolpath and positioning moves generated by the system. Had there been any problems, we would have needed to reprocess the existing operations so that the system would have recalculated the tool moves for each operation based

Redo All Ops Redo All Ops item in the Edit menu or the Operation list context menu.

EXERCISE #2: 2D FORM TOOL

This exercise introduces you to creating and using Form Tools in GibbsCAM. We will start with an existing, very simple part file, create a Form Tool and apply a machining operation using the Form Tool.

1. Open the part file `Form Tool.vnc` located in the Tutorial Parts folder that was installed with the software.

The file contains geometry for the part, a tool and two roughing operations. The existing operations face and rough the part. First we need to create geometry for the Form Tool.

2. Create a new workgroup for the Form Tool.
3. The geometry for the form tool is provided at the end of the Tutorial, see ["2D Form Tool Blueprint" on page 45](#).

The system always uses the origin as the tool's touch-off point. The part should look like this image. In this example we have designated the non-cutting surfaces as "air" geometry. (Simply

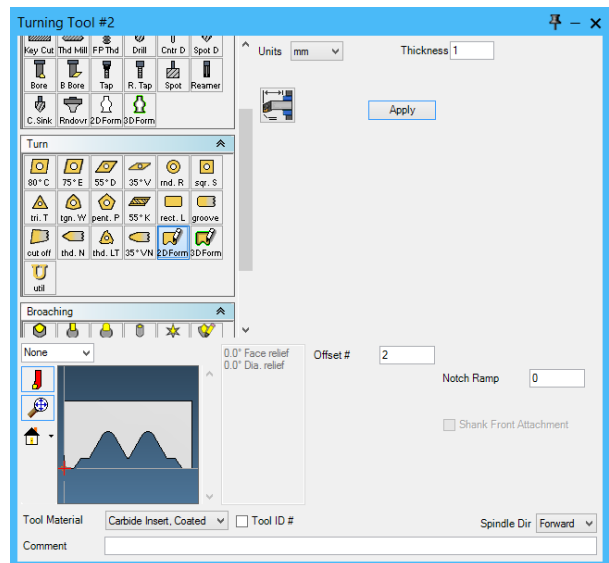
right-click the geometry and select  Change feature from "WALL" to "AIR".)

4. Open the tool dialog.
5. Select 2D Form tool as the tool type.



You are prompted to select a shape to use for the tool's profile.

6. Double-click the Form Tool geometry and click Apply.

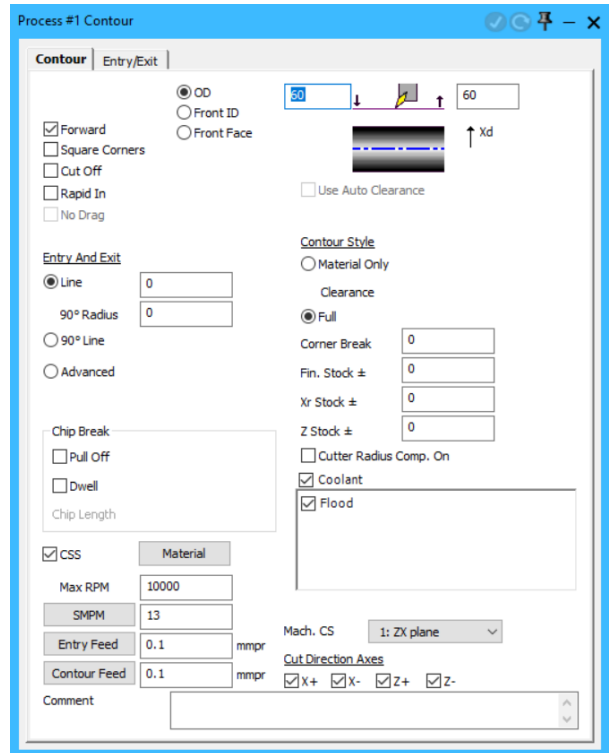


The shape of the tool is displayed in the Tool dialog window. Note the touch-off point is shown as a red cross-hair.

Form Tool Contour

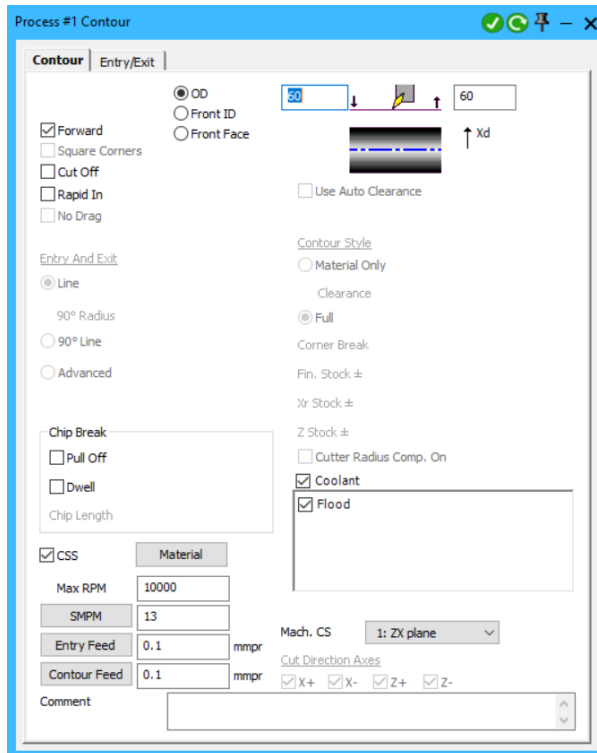
We will now make a contour process using the form tool.

1. Create this Contour Process with the form tool.



2. Position the machining markers as shown.

We have placed the start point marker above the roughed surface. The end point marker should be on the connecting point to drive the tip of the tool to that point.



Since we are contouring on a single line some options become disabled when using a Form tool.


3. Create the toolpath.

The result is a single feed line into the part.




4. Switch to the Isometric view and render the results.
5. Save this part, it is complete.

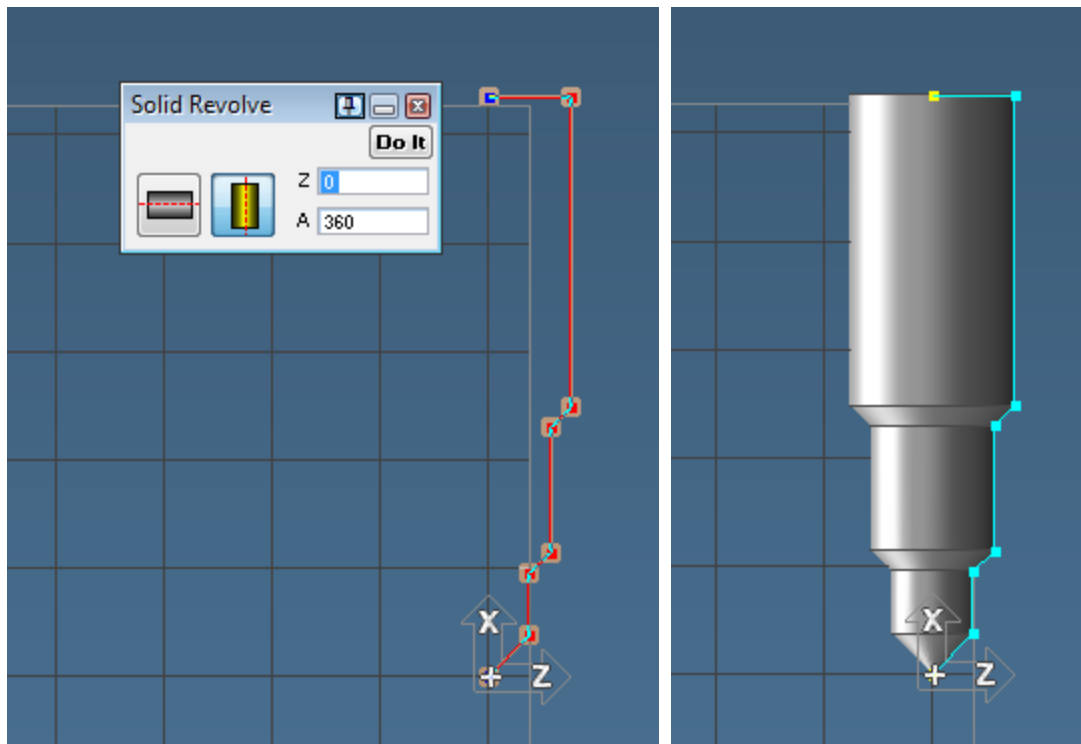
EXERCISE #3: 3D FORM TOOLS

3D Form tools are created in a similar way to 2D Form tools.

1. In part file `Form Tool.vnc`, create a new workgroup for the 3D Form Tool.
2. The geometry for the 3D Form tool is provided at the end of the Tutorial, see “[2D Form Tool Blueprint](#)” on page 45. For the purpose of this demonstration, the dimensions do not need to be exact. You could use mouse point  located in the Point section of the Geometry creation palette to create the rough shape. It is however very important that the shape originates from the X0 Y0 position.

The system uses the origin as the tool's touch-off point. The tool should look like the following image.

3. **Double-click** the geometry to select all. Open the  Solid Modeling palette and choose the first option,  Create Solid.
4. Use the  Solid Revolve tool to create a solid.

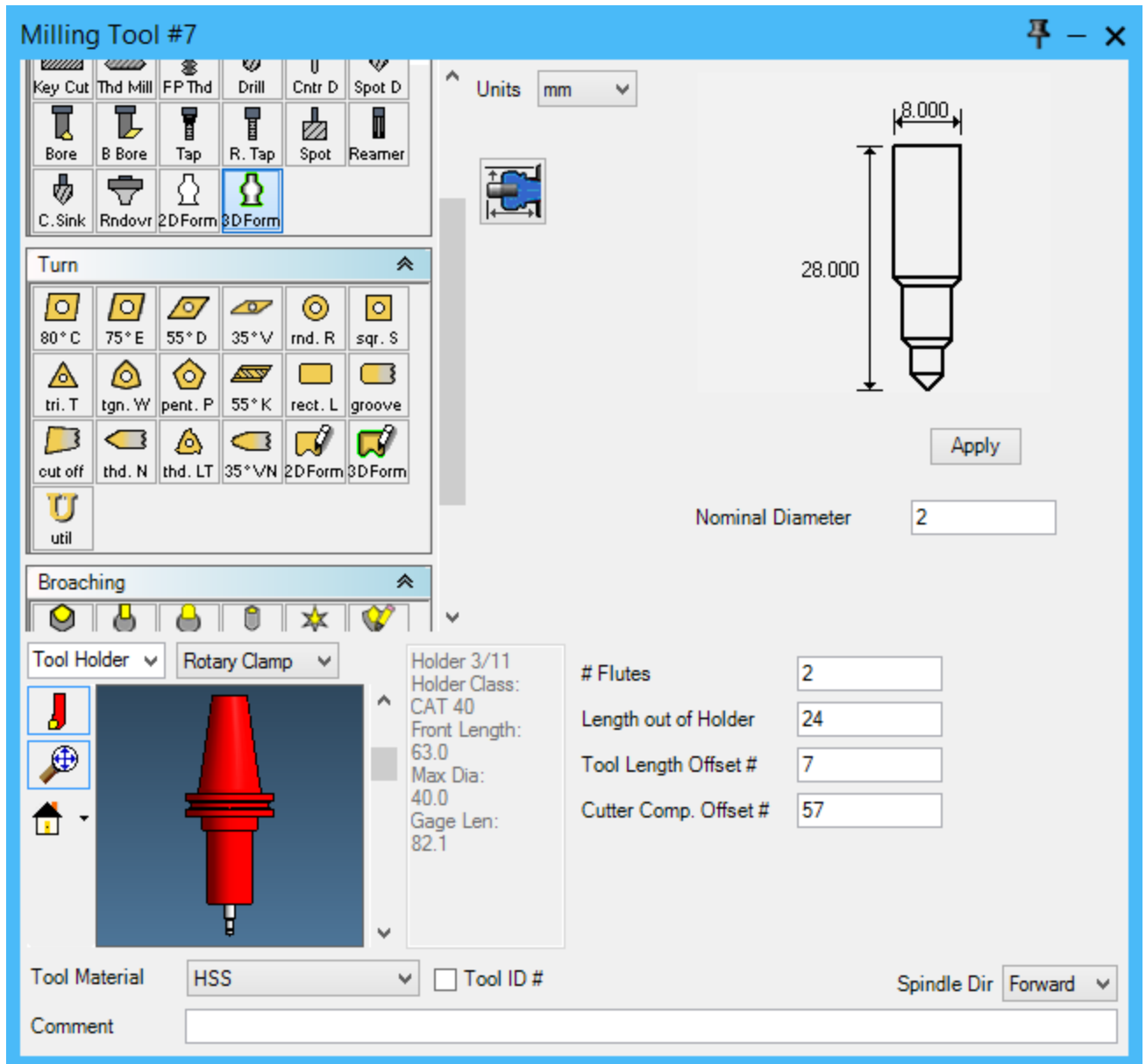


5. Open the tool dialog. Choose 3D Form tool from the Mill toolset.

You are prompted to **Select a shape**.



6. Click the Solid you just created and then Click Apply.

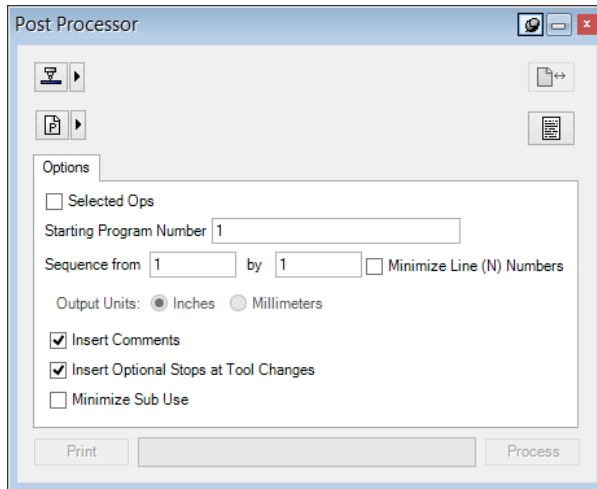


The shape of the tool is displayed in the Tool dialog.

POST PROCESSING

Now that all of the operations to machine the part have been satisfactorily created and verified, we need to post process the file. Post processing will create a text file that the CNC machine will use to cut the part.

1. Open Post Processor dialog from the Command palette.

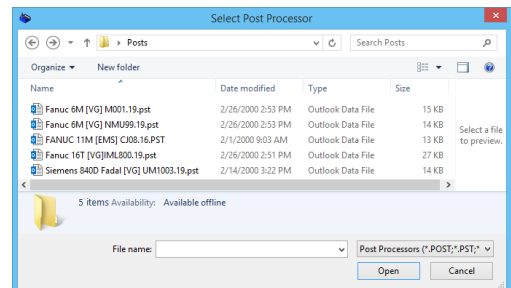


From the **Post Processor** dialog a post processor may be selected for generating a post file and communicating with a machine control. For more information on Post Processing and Communications, refer to the Turning user guide, "Post Processing" chapter.

2. **Click** the Post Processor Selection button.



3. Select a Lathe post processor.



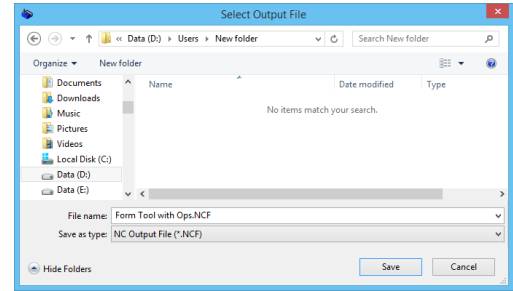
4. **Click** the **Open** button.

The name of the selected post processor will appear in the dialog next to the button.

5. **Click** the Program Name button.



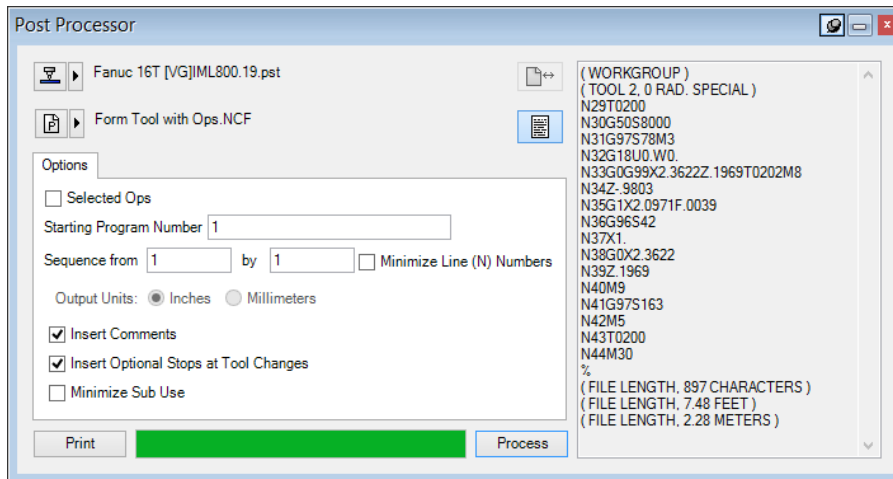
The Program Name defaults to the name of the part file. You can change the name or leave it as is.



6. Click the Save button.

Once you have selected a post processor and named and saved the program, the Process button at the bottom of the dialog becomes active.


7. Click the Text Window button.




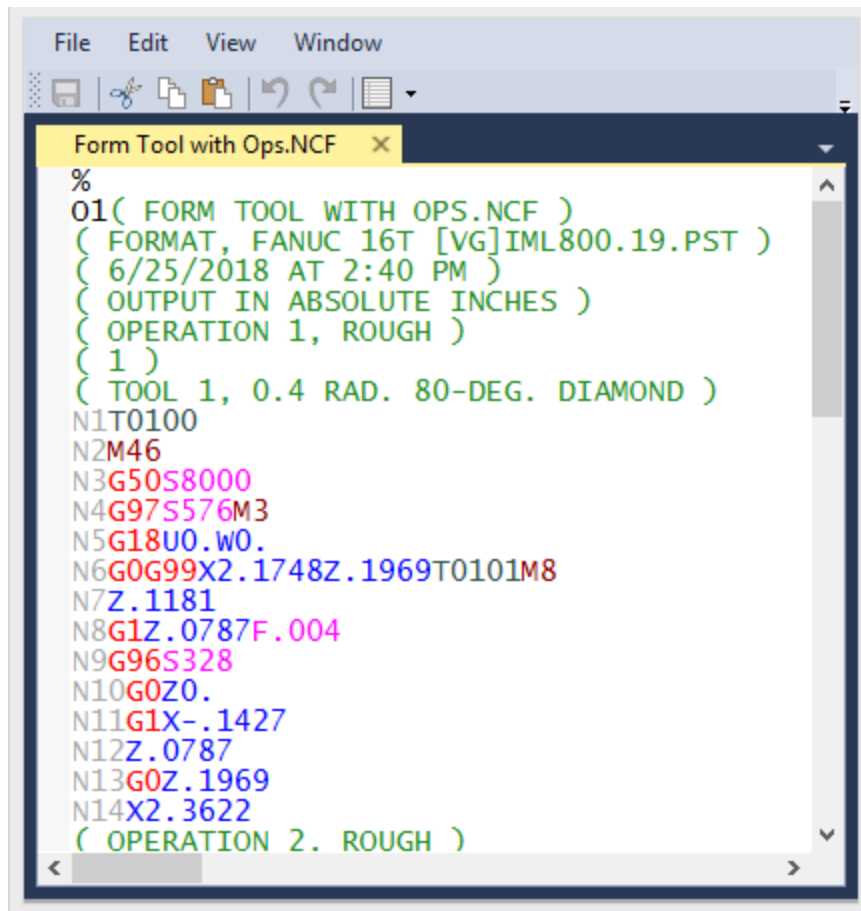
8. Click the Process button.

This will create a post file that can then be sent to the machine.

The program will scroll in the right-hand window as it is being processed. You can click the keyboard Pause button to momentarily stop the display so that you can read the program. Once the program has finished processing, the Print button becomes active and the posted code can be printed.

In addition, if you have the  Preferences > G-Code Editor Settings set to Internal Editor, the GibbsCAM Internal Post editor will appear in a separate window. This customizable, color-enhanced editor allows you to directly edit the program using cut/copy/paste etc., with multiple subs and flows clearly distinguished. You can also find a link to this internal editor under the

 Plugins > Main Tools menu, which will enable you to edit existing .ncf files without having to open the .vnc file.



The screenshot shows a software window titled "Form Tool with Ops.NCF" with a menu bar (File, Edit, View, Window) and a toolbar. The main area contains a list of NC program lines, some of which are color-coded: N3 (green), N4 (red), N5 (blue), N6 (green), N8 (red), N9 (blue), N11 (green), N13 (red), and N14 (blue). The program includes comments about format, date, operation, and tool specifications.

```
%  
O1( FORM TOOL WITH OPS.NCF )  
( FORMAT, FANUC 16T [VG]IML800.19.PST )  
( 6/25/2018 AT 2:40 PM )  
( OUTPUT IN ABSOLUTE INCHES )  
( OPERATION 1, ROUGH )  
( 1 )  
( TOOL 1, 0.4 RAD. 80-DEG. DIAMOND )  
N1T0100  
N2M46  
N3G50S8000  
N4G97S576M3  
N5G18U0.W0.  
N6G0G99X2.1748Z.1969T0101M8  
N7Z.1181  
N8G1Z.0787F.004  
N9G96S328  
N10G0Z0.  
N11G1X-.1427  
N12Z.0787  
N13G0Z.1969  
N14X2.3622  
( OPERATION 2. ROUGH )
```

PART PRINTS

Lathe Part Blueprint

2D Form Tool Blueprint

3D Form Tool Blueprint