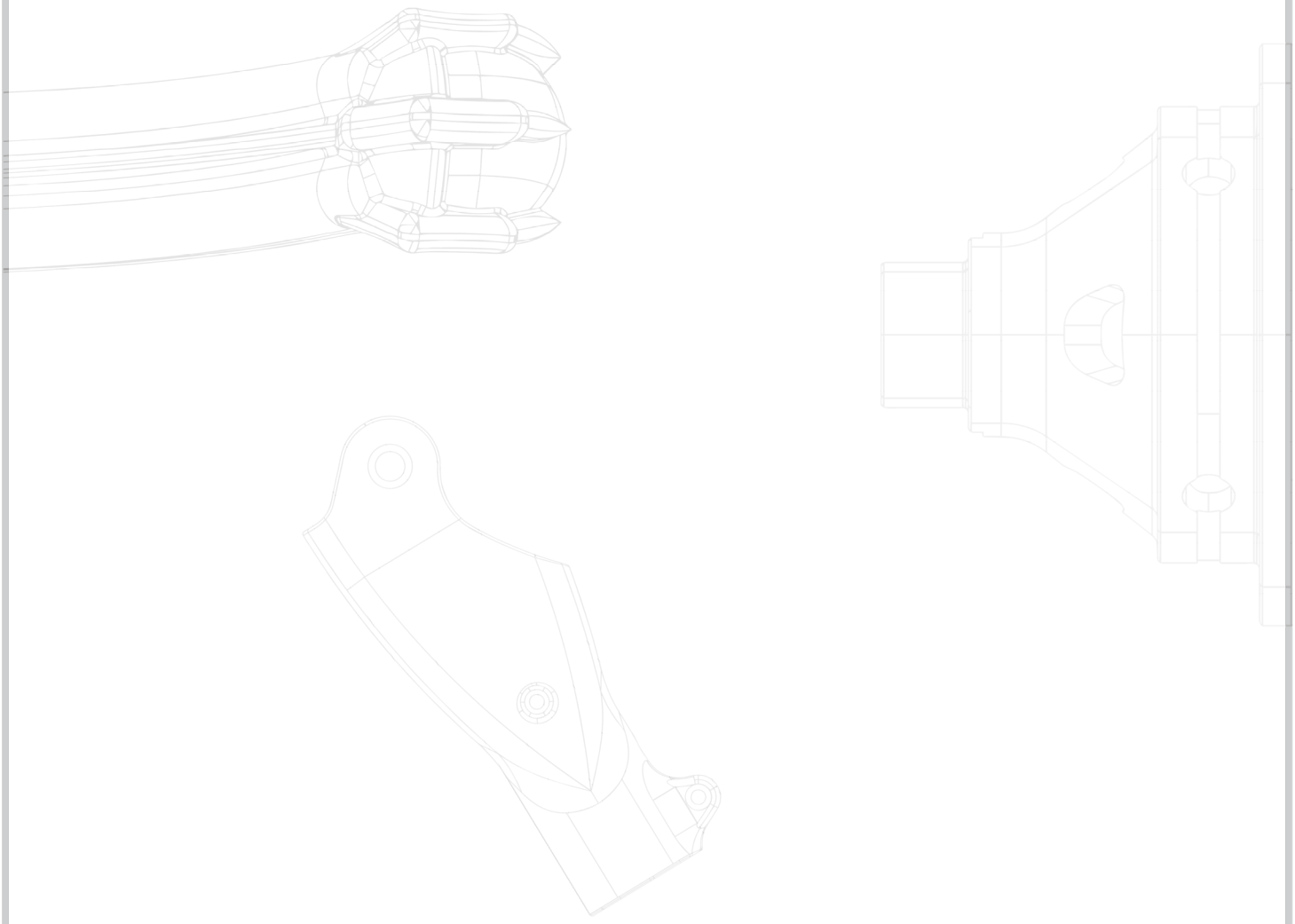
MASTERCAM LATHE FOR SWISS

March 2019



POWERED BY MASTERCAM'S DYNAMIC MOTION™ TECHNOLOGY

Mastercam

MASTERCAM LATHE FOR SWISS

March 2019

Software: Mastercam 2019

Terms of Use

Use of this document is subject to the Mastercam End User License Agreement. The Mastercam End User License Agreement can be found at:

http://www.mastercam.com/companyinfo/legal/LicenseAgreement.aspx

Be sure you have the latest information!

Information might have changed or been added since this document was published. The latest version of the document is installed with Mastercam or can be obtained from your local Reseller. A ReadMe file (ReadMe.PDF) – installed with each release – includes the latest information about Mastercam features and enhancements.

TABLE OF CONTENTS

INTRODUCTION

Welcome to *Mastercam Lathe for Swiss*. In this tutorial, you use Mastercam Lathe to program the machining for the part shown below. Along the way, you learn about the following topics:

- How to set up Mastercam to program a Swiss part.
- How to apply Mastercam toolpaths to a Swiss part.
- How to use Mastercam Lathe to machine a Swiss part on the main and sub spindles.
- How to perform cutoffs and transfers between the spindles.

Goals

- Understand the differences between Mastercam Swiss and Mastercam Lathe.
- Explore the parameters required to set up a Mastercam Swiss job.
- Successfully apply lathe toolpaths to a Swiss part.

WARNING: Screen colors in the tutorial pictures were modified to enhance image quality; they may not match your Mastercam settings or the tutorial results. These color differences do not affect the lesson or your results.

Estimated time to complete this tutorial: 4 hours

General Tutorial Requirements

All Mastercam 2019 tutorials have the following general requirements:

- You must be comfortable using the Windows® operating system.
- The tutorials cannot be used with Mastercam Demo/Home Learning Edition. The Demo/HLE file format (`emcam`) is different from Mastercam (`mcam`), and basic Mastercam functions, such as file conversions and posting, are unavailable.
- Each lesson in the tutorial builds on the mastery of the preceding lesson's skills. We recommend that you complete them in order.
- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be access from the Mastercam 2019 workstation, either with the tutorial or in any location that you prefer.
- You will need an internet connection to view videos that are referenced in the tutorials. All videos can be found on our YouTube channel: www.youtube.com/user/MastercamTechDocs
- All Mastercam tutorials require you to configure Mastercam to work in a default Metric or Inch configuration. The tutorial provides instructions for loading the appropriate configuration file.

CHAPTER 1

SETTING UP THE JOB

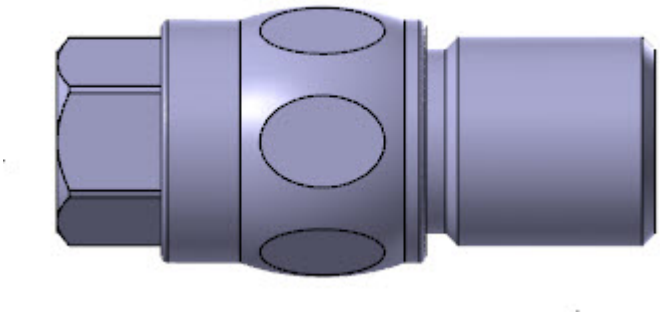
In this chapter, you set up your machining job to use Mastercam Swiss. Much of the difference between Mastercam Swiss and Mastercam Lathe is evident in the setup.

Goals

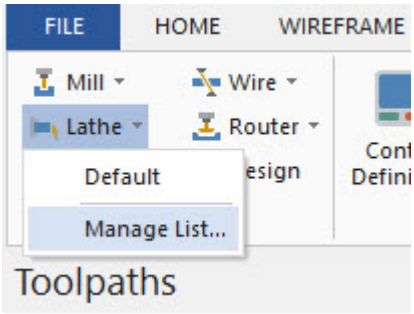
- Load the machine and tool library.
- Complete stock setup and create a turn profile.
- Create the stock advance operation.

Exercise 1: Loading the machine

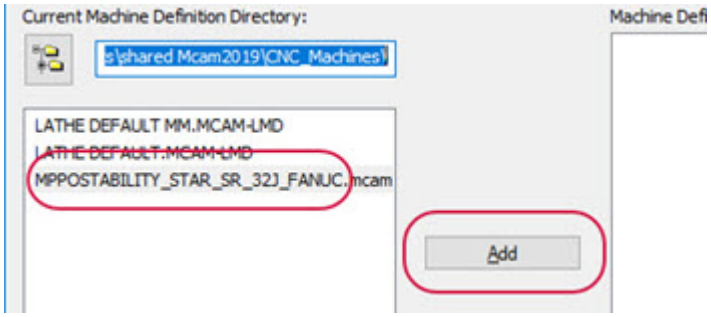
1. Load the part `Swiss Tutorial`, which is included with this tutorial.



2. Save the file as `Swiss Tutorial_XX_0`, where *XX* is your initials.
3. Select **Machine, Lathe, Manage List**.

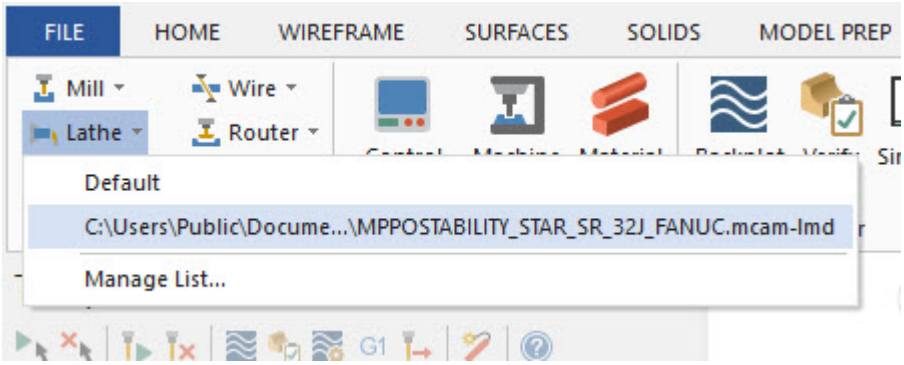


4. Add the `MPPostAbility_STAR_SR_32J_FANUC.MCAM` machine to the **Machine Definition Menu Items** list.



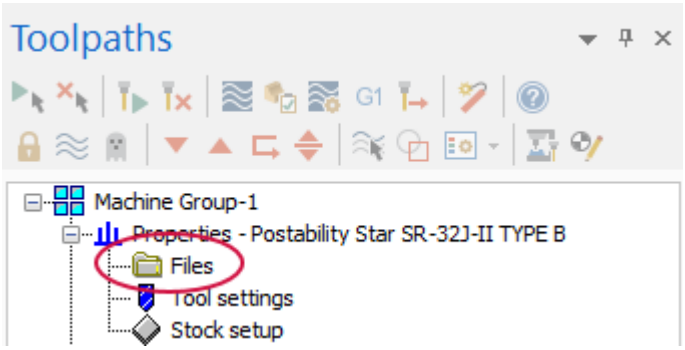
Note: The `MPPostAbility_STAR_SR_32J_FANUC.MCAM` machine is not included with this tutorial. You must have a license for this machine to access it.

5. Choose the machine from the **Lathe** menu. Mastercam creates a new lathe machine group.

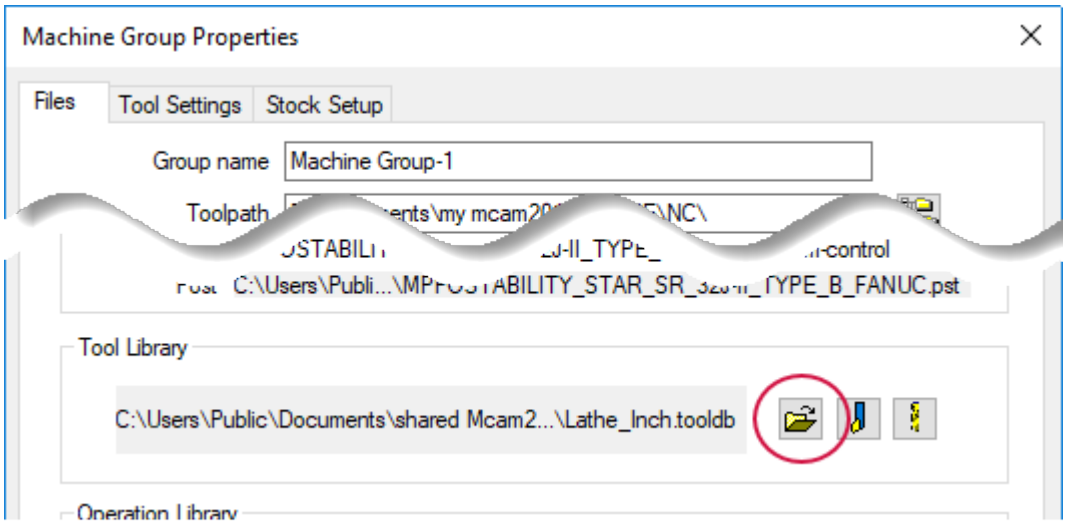


Exercise 2: Loading the tool library

1. In your new machine group, click the **Files** icon. The **Machine Group Properties** dialog box displays.



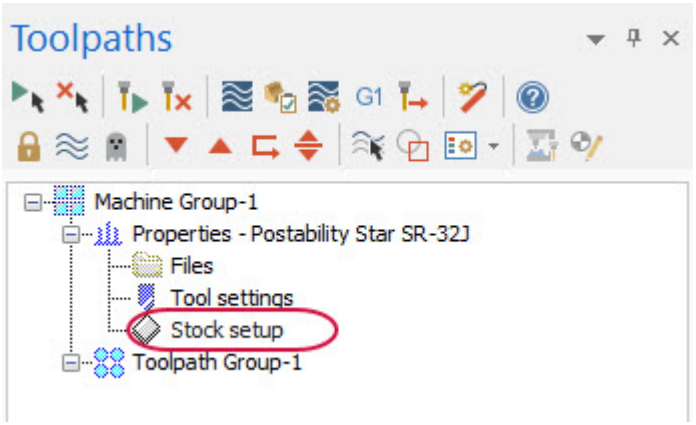
2. In the **Tool Library** section, click the folder icon. The **Select Tool Library File** dialog box displays.



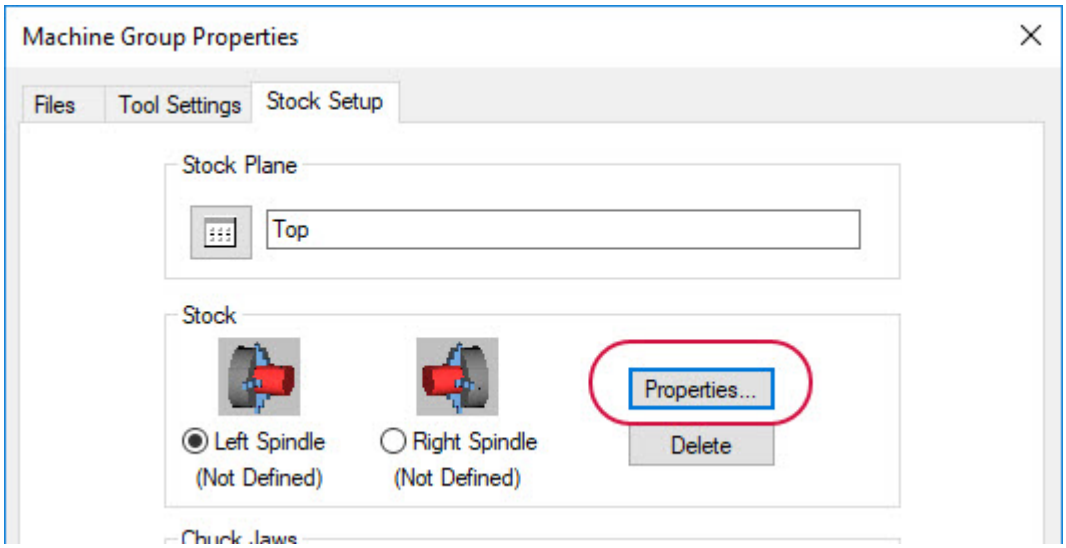
3. Navigate to and open the `Swiss.tooldb` file that's included with this tutorial.
4. Click **OK** in the **Machine Group Properties** dialog box.

Exercise 3: Completing stock setup

1. Click **Stock setup** in the machine group.

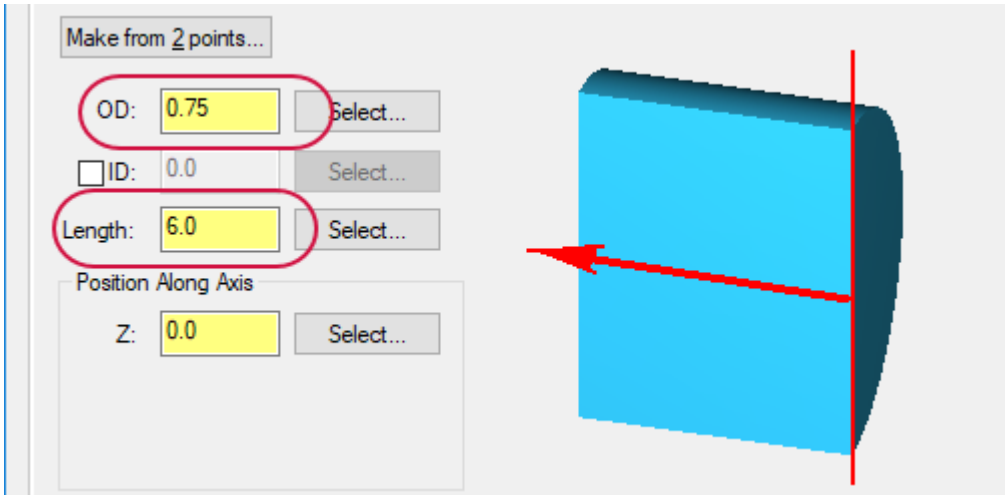


2. Select **Stock Setup, Left Spindle, Properties**.

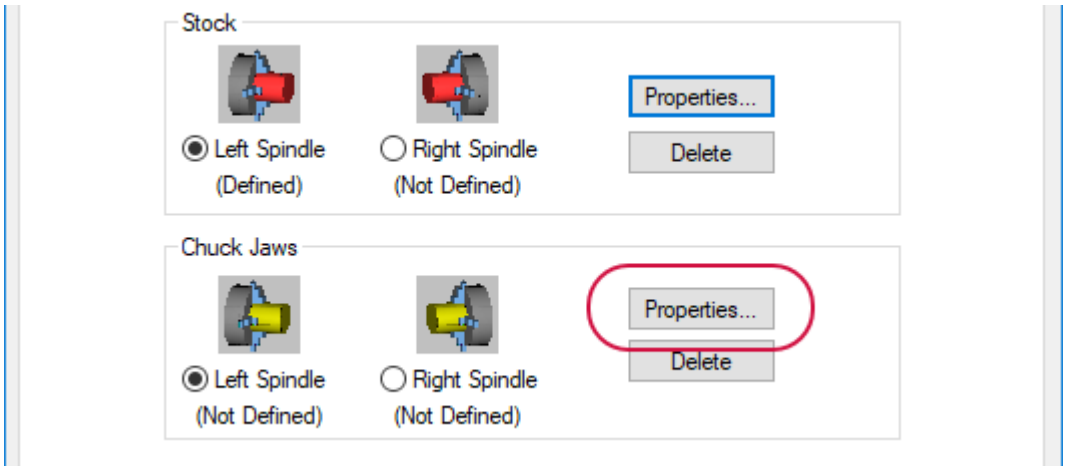


3. Enter an **OD** of **0.75** and a **Length** of **6.0**. Click **OK**.

   The stock length of 6.0 is for screen display purposes only and has no affect on the NC code.



4. Select **Chuck Jaws, Left Spindle, Properties**.



5. Enter the following values:

   Jaw width: 3.5

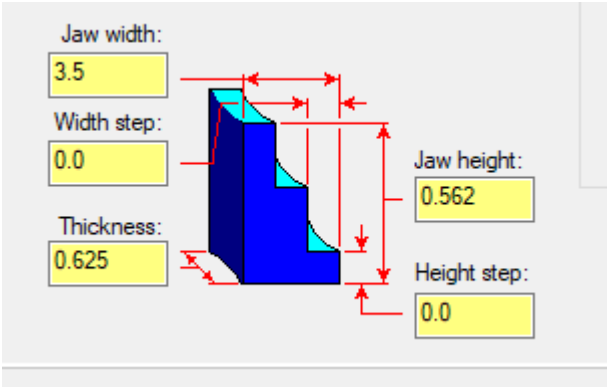
   Width step: 0.0

   Thickness: 0.625

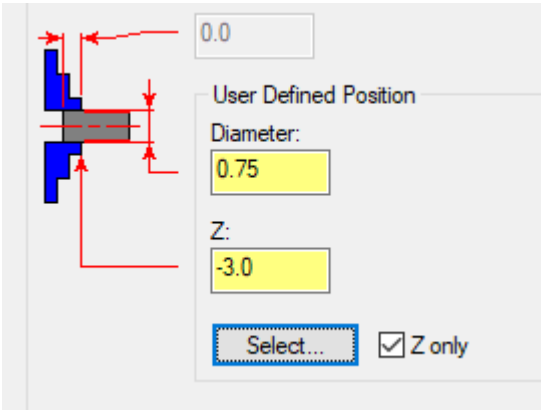
   Jaw height: 0.562

   Height step: 0.0

   These values change the default three-jaw chuck, which is not used in Swiss Lathe, to a Swiss-style chuck.

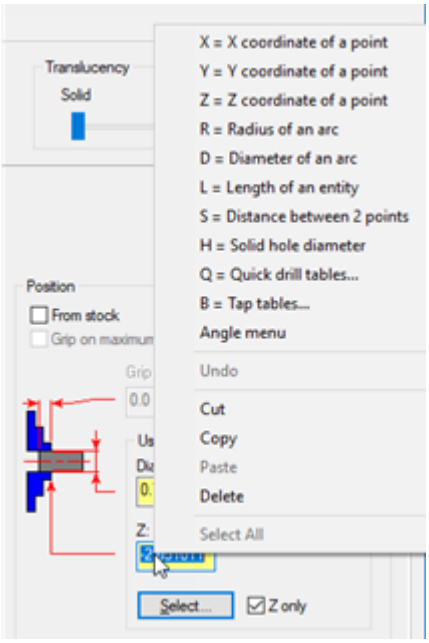


6. Enter **0.75** for **Diameter** and **-3.0** for **Z**, as shown in the following picture, and then click **OK**.

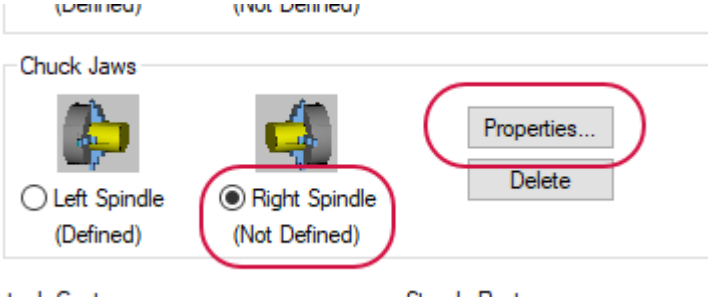


Z is the distance between the tool and the face of the chuck jaw or guide bushing and must be entered as a negative number. Typically, you subtract about **0.5** inches from the full **Z** value. This extra space leaves room for the cut-off tool to perform its operation. Failure to set this value properly can lead to collisions during machining.

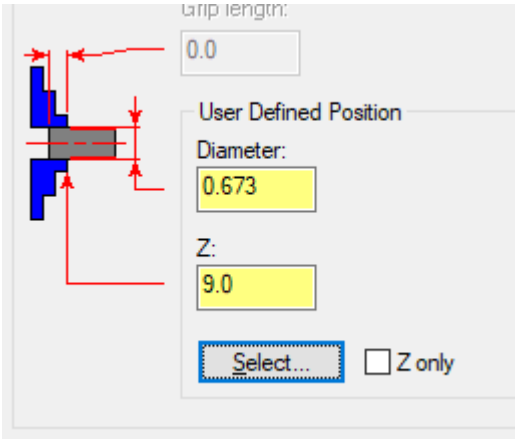
Note: To set **Z** with geometry, right-click the **Z** field, choose the geometry type (in this case, **X coordinate of a point**; see the second picture following), and then make a selection in the graphics window. You can then use the built-in calculator in the **Z** field to subtract about **0.5** inches, as described in the first part of this step.



7. Select **Chuck Jaws, Right Spindle, Properties**.

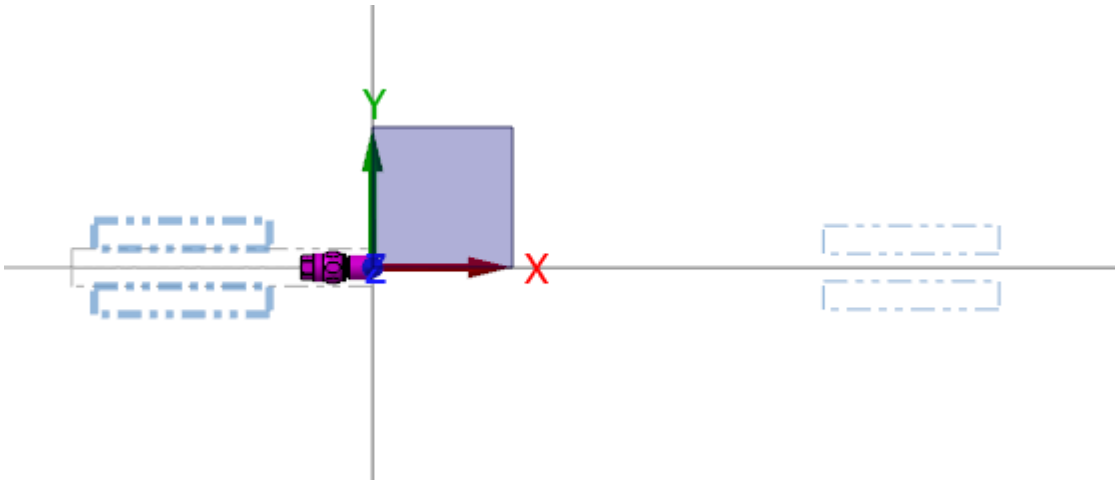


8. Set **Diameter** to **0.673** and **Z** to **9.0**, as shown in the following picture, and then click **OK** in both this dialog and in the **Machine Group Properties** dialog box.



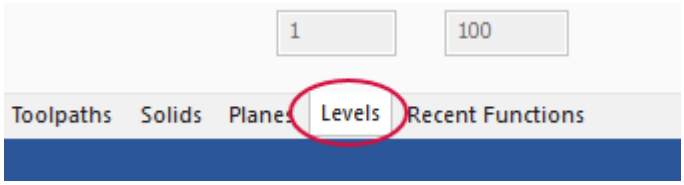
The **Z** here is the distance between the main spindle and the subspindle.

In Mastercam's graphics window, zoom out until you see the fully defined spindles, as shown in the picture following.



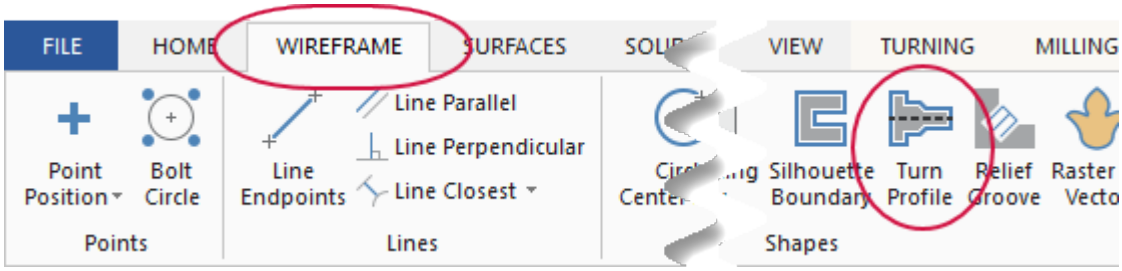
Exercise 4: Creating the turn profile

1. Click the **Levels** tab in the bottom left-hand corner of Mastercam's window.

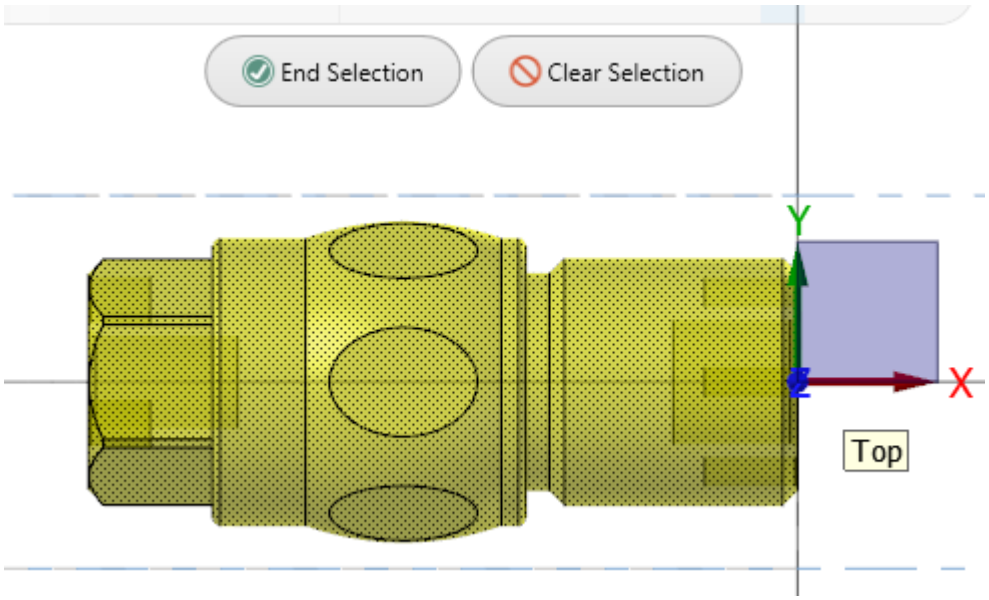


2. Type **10** in **Number** and **Wireframe** in **Name**.

3. Go to Mastercam's **Wireframe** tab, and choose **Turn Profile**.

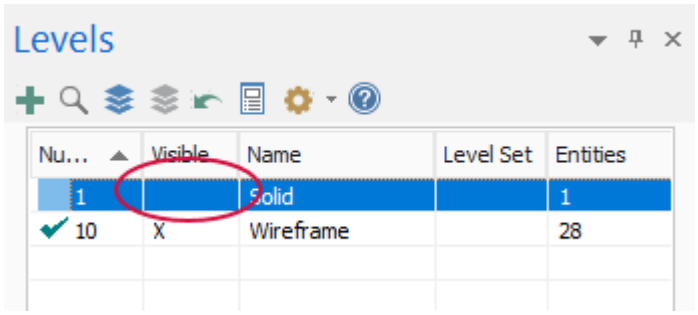


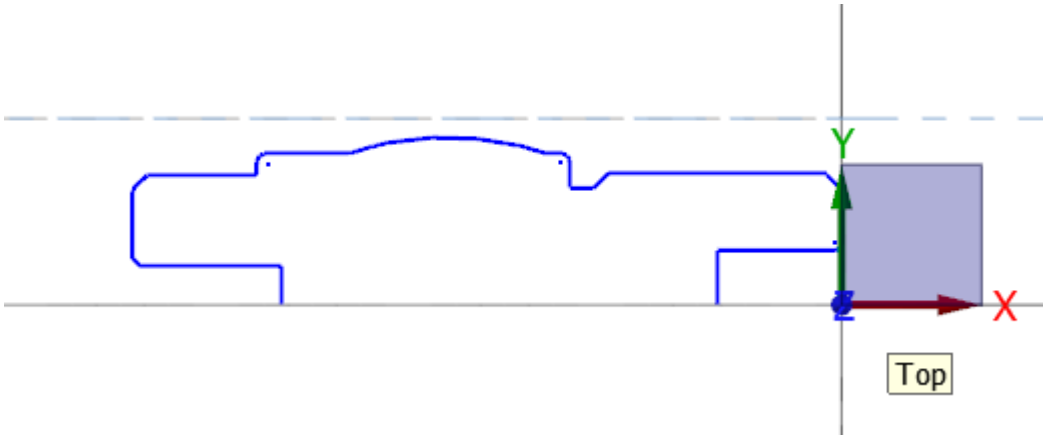
4. In the graphics window, select the solid, and click **End Selection** or press [**Enter**].



5. To finalize your selection and accept the default options, click **OK** in the **Turn Profile** function panel.

6. In the Levels Manager, hide the solid by clicking in its **Visible** column. The graphics window now shows the part's turning profile, without the solid in the way, as shown in the second picture following.

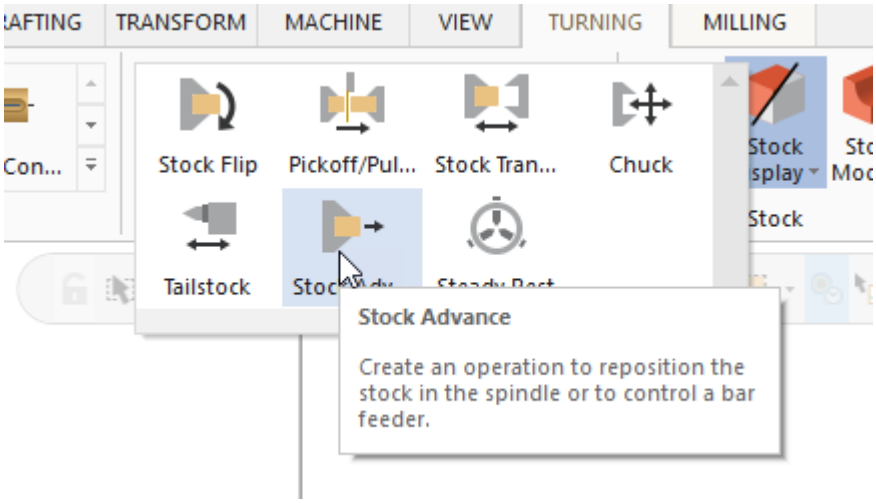




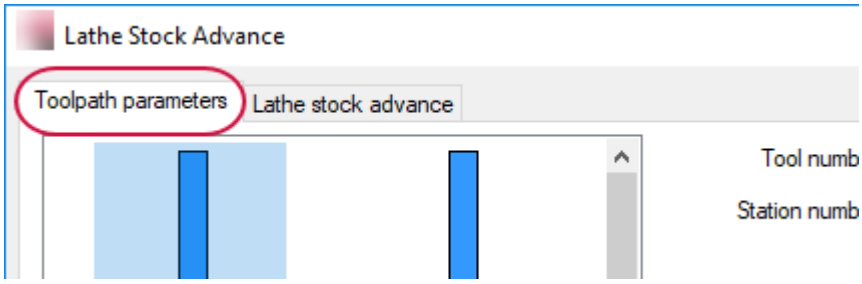
Exercise 5: The stock advance operation

With a Swiss part, the first operation must always be a stock advance. This is what tells the post that this is a Swiss part and formats how the output code will look. Also, it loads the specific machine setup variables.

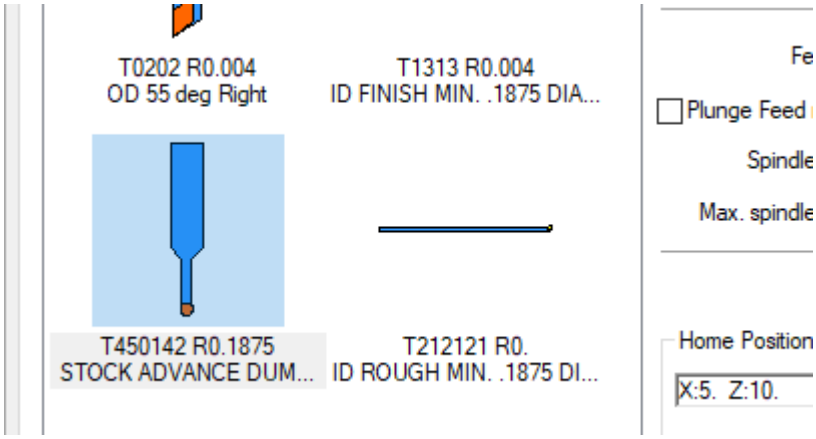
1. On the **Lathe, Turning** contextual tab, select **Part Handling, Stock Advance**.



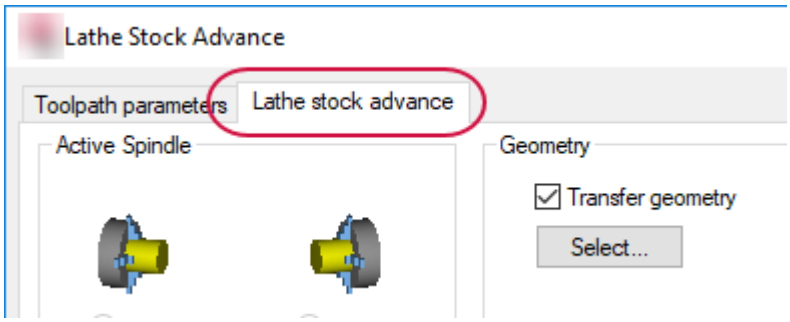
2. In the **Lathe Stock Advance** dialog box, select the **Toolpath parameters** tab.



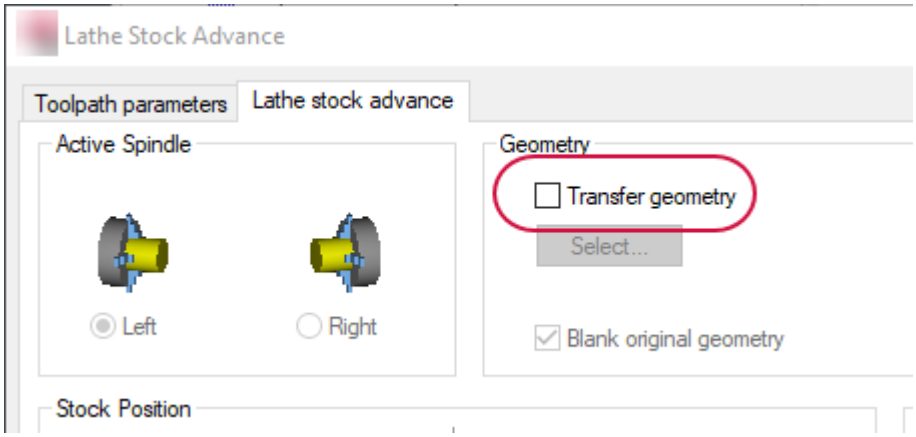
3. Select tool number **T450142**, the stock advance dummy tool. The operation requires that a tool be selected, even though it will not use it. Selecting a tool that will not be used with Swiss Lathe eliminates later conflicts in Mastercam.



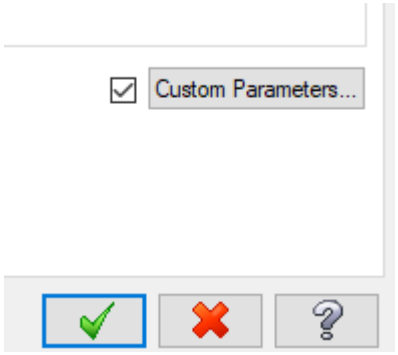
4. Select the **Lathe stock advance** tab.



5. Deselect **Transfer geometry**. This prevents the stock from repositioning as would happen with a stock pull on a lathe.



6. Select the **Custom Parameters** checkbox, and then click the now enabled **Custom Parameters** button.



7. Set the **Custom Parameters** as follows, and then click **OK** in the **Custom Parameters** and the **Lathe Stock Advance** dialog boxes.

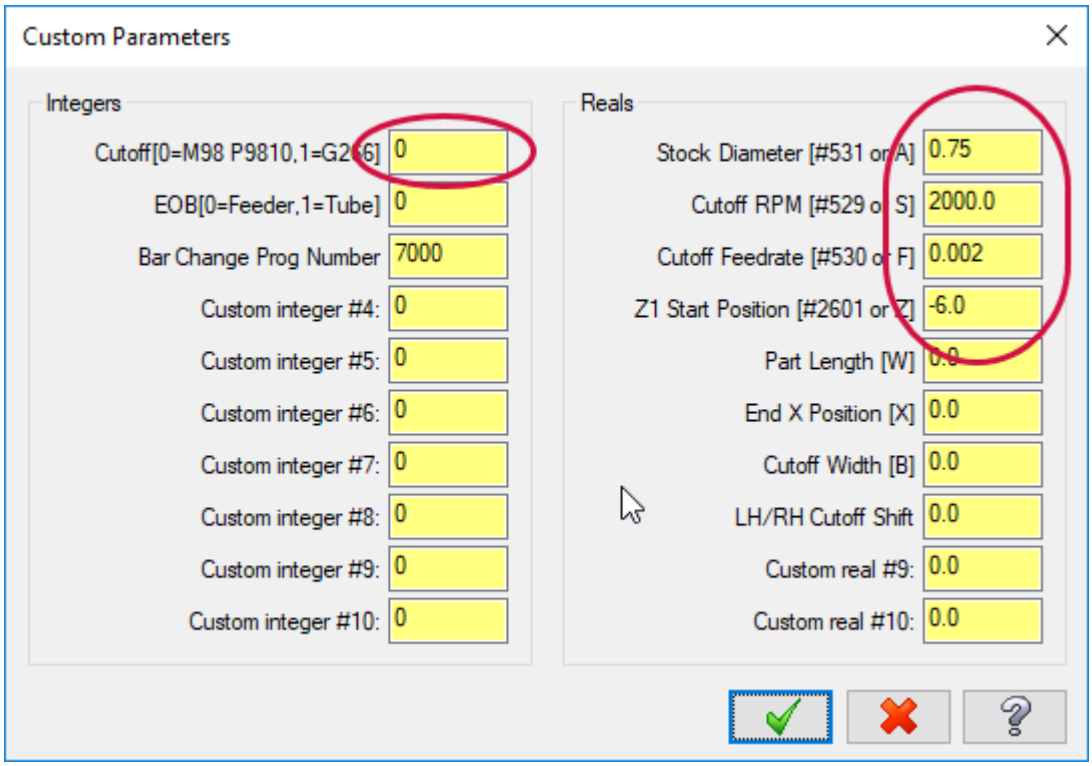
   Cutoff: 0

   Stock Diameter: 0.75

   Cutoff RPM: 2000

   Cutoff Feedrate: 0.002

   Z1 Start Position: -6.0



These values are used in the NC code's header section and are the values needed when you select **1** for **Cutoff**, the first integer in the list. By choosing **1**, you specify that the machine is equipped to handle G266.

Note: If, instead, the machine is setup for M98 and P9810, you select **0** for the first integer, and you provide values only for the parameters shown in the following picture. Notice that you can also enter values to specify a bar feeder or tube, and a Bar Change program number. These values and parameters are machine specific.

8. Save the file with the name `Swiss Tutorial_XX_1.mcam`, where *XX* is your initials.

CHAPTER 2

LEFT SPINDLE OPERATIONS

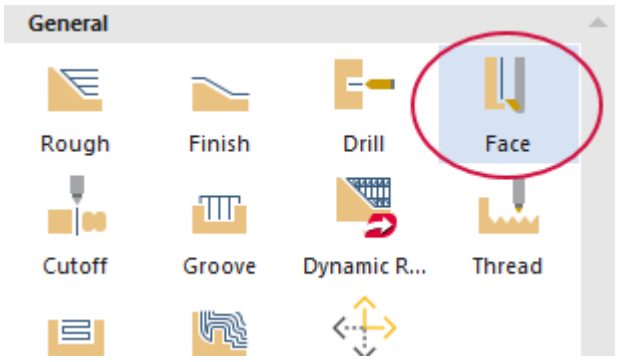
Now that you have the Swiss environment set up, you can begin to program the operations needed to machine the part. The operations you complete in this chapter are all on the left spindle.

Goals

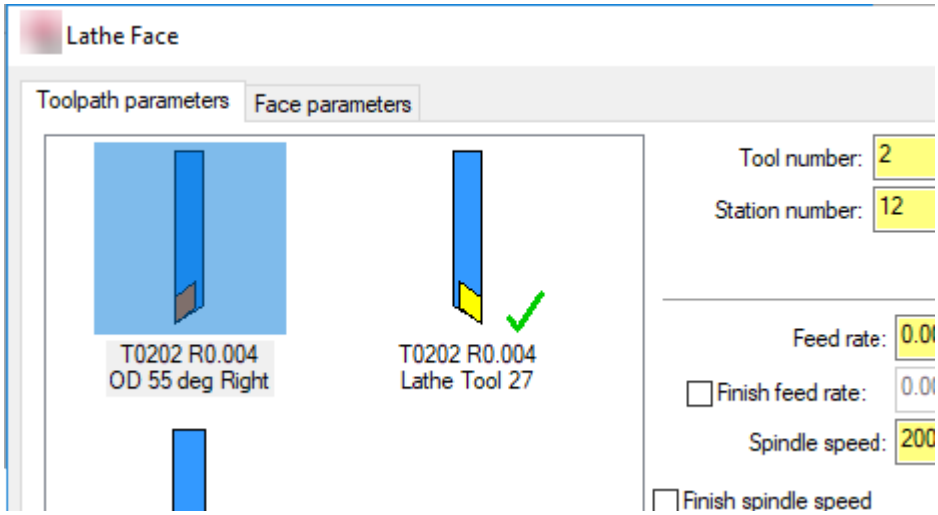
- Create a facing operation.
- Create drilling operations.
- Create finishing operations.
- Create threading and groove operations.
- Create milling operations.

Exercise 1: Programming the facing operation

1. From the **General** gallery on the **Lathe, Turning** contextual tab, choose the **Face** toolpath.



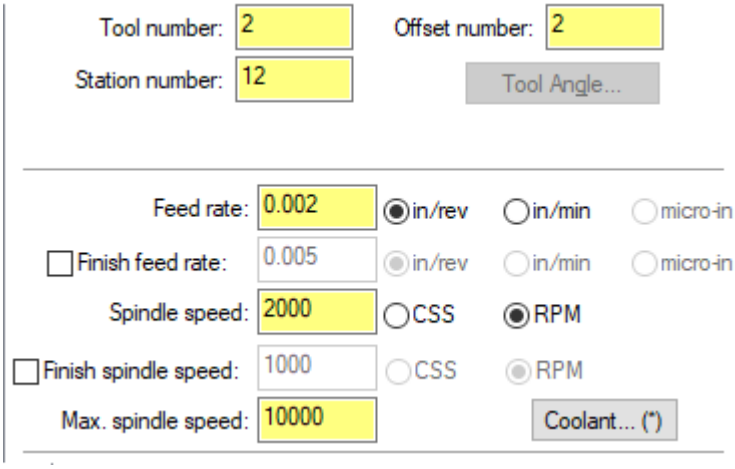
2. Select tool **T0202 R0.004 OD 55 deg Right**.



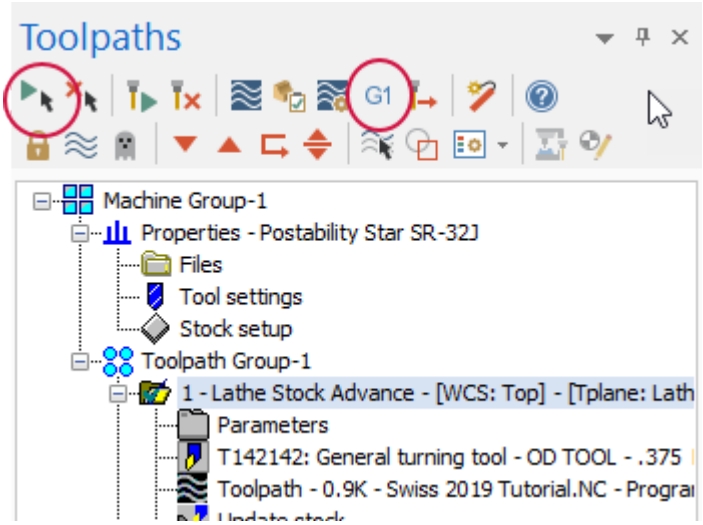
3. Set the parameters as shown in the picture below, and click **OK**.

   Tool number: 2
   Offset number: 2
   Station number: 12

Feed rate: 0.002, in.rev
Spindle speed: 2000, RPM



4. Return to the Toolpaths Manager, click **Select all operations**, and then click **Post selected operations**.



5. When they appear, click **OK** in the **Post Processing** and **Save As** dialog boxes.
6. When the **Combined Post Processor** dialog box appears, press [**Enter**] to accept the program. (You will do this twice.) You can enter any number up to four digits to name the program.
7. In the generated NC file, select the second tab. You can see the values you specified in the **Custom Parameters** dialog box, as shown in the following picture. If you use custom parameters option 1, the NC code looks as shown in the second picture following.

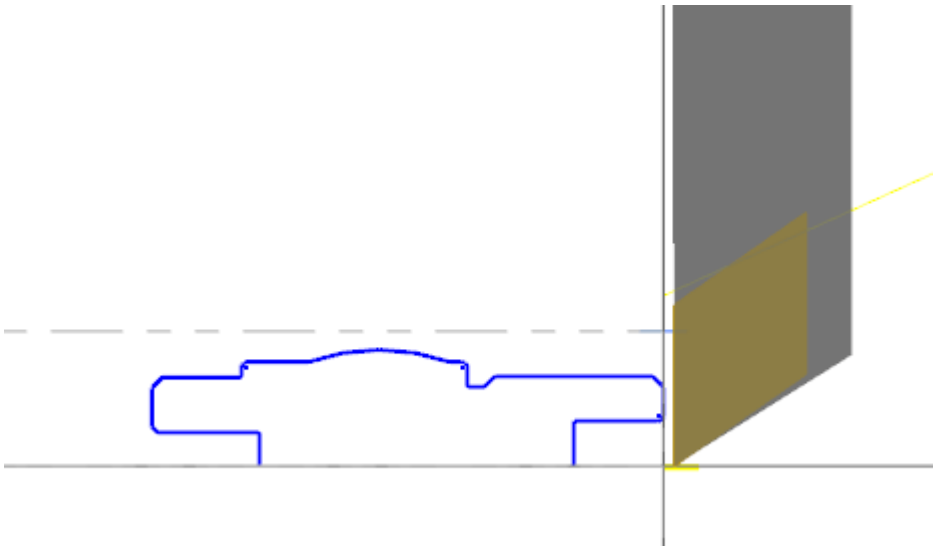
Swiss 2019 Tutorial2_CAW 2.NC | Swiss 2019 Tutorial2_CAW.NC | Start Page

```
1   %
2   O0001 (SWISS 2019 TUTORIAL2_CAW)
3   (MASTERCAM - 2019)
4   (MCX FILE  - C:\USERS\CAW\DESKTOP\MASTERCAM SWISS LATHE
5   G99 G97 G80 G40 G18 M9
6   M152 (MAGIC BUSHING OFF)
7   G266 A.75 W1.4275 S2000 X-.02 Z-6. F.002 B.118
8   (A=#531 STOCK DIAMETER)
9   (W=#530 PART LENGTH)
10  (S=#529 AUTO CUTOFF SPEED)
11  (X=#524 END X POS AT AUTO CUTOFF)
12  (Z=#525 Z1 START RECHUCK POSITION)
13  (F=#522 AUTO CUTOFF FEEDRATE)
14  (B=#528 CUTOFF WIDTH)
15
16  #515=0.  (LH/RH CUTOFF SHIFT)
```

Swiss 2019 Tutorial 2.NC | Swiss 2019 Tutorial.NC | Start Pag

```
1   %
2   O0001 (SWISS 2019 TUTORIAL)
3   (MASTERCAM - 2019)
4   (MCX FILE  - C:\USERS\CAW\DESKTOP\SWISS
5   #531=.75 (Stock Diameter)
6   #529=2500 (Cutoff RPM)
7   #530=.002 (Cutoff Feed)
8   #2601=-6. (Z1 Start Position)
9   G00 G98 G97 G40 M09
10  M150
11  IF[#5041LT0]GOTO10
```

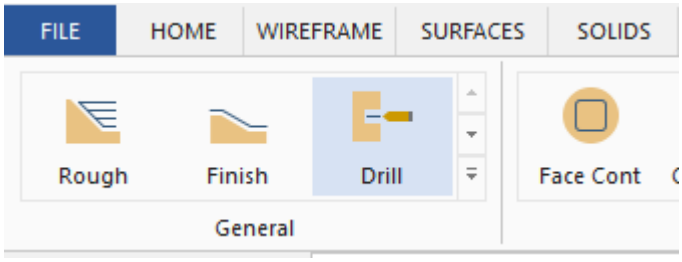
8. In Mastercam, backplot the operation to see the toolpath in action.



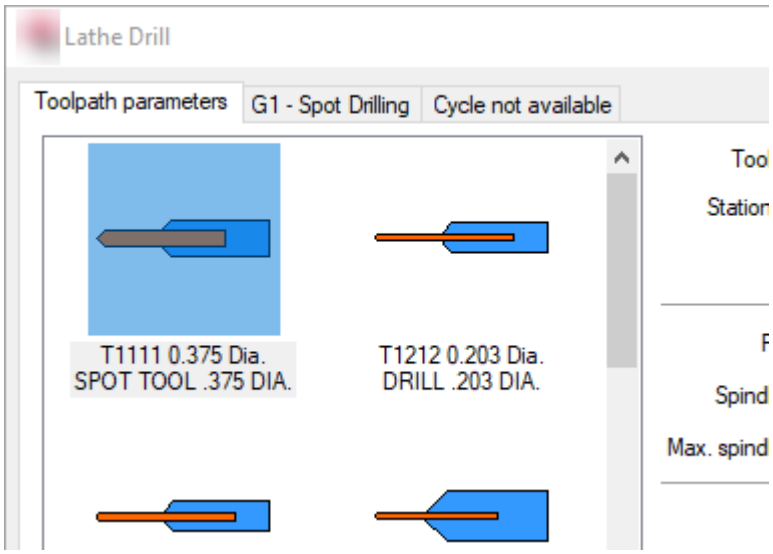
9. Save the file with the name `Swiss Tutorial_XX_2.mcam`, where *XX* is your initials.

Exercise 2: Spot drilling the front hole

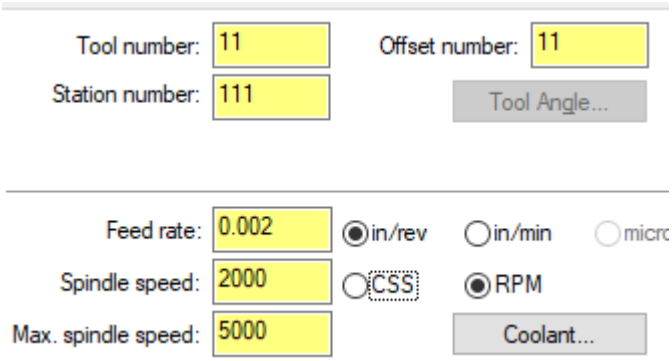
1. Select **Drill** from the **General** gallery on the **Lathe, Turning** contextual tab.



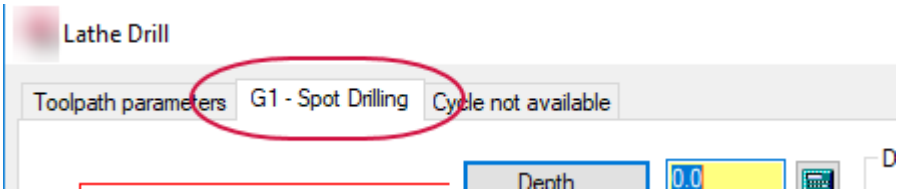
2. In the **Lathe Drill** dialog box, select the **T1111** tool.



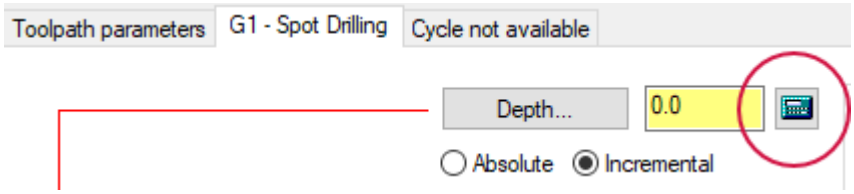
3. Make sure that your parameters match the image below.



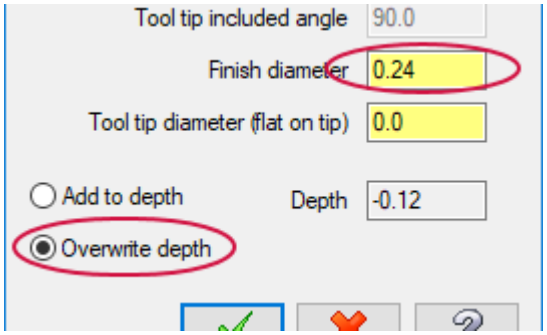
4. Click the **G1 - Spot Drilling** tab.



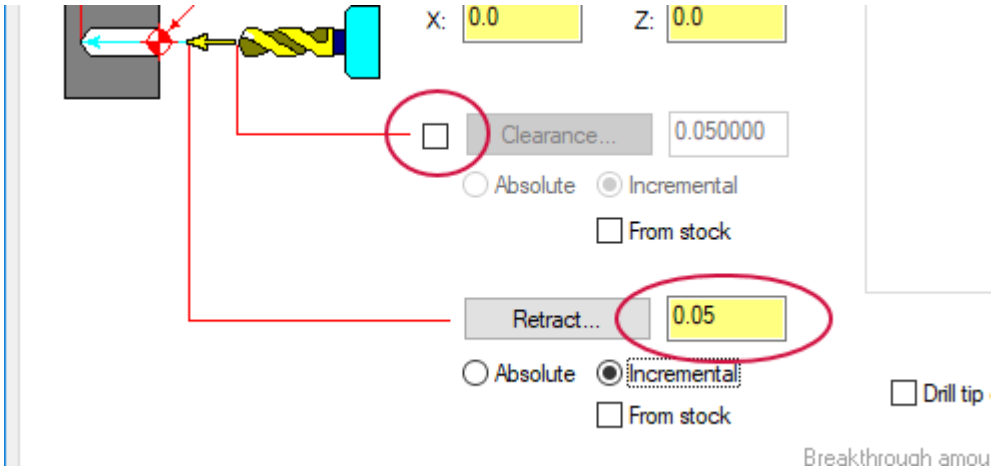
5. Click the **Depth Calculator** button.



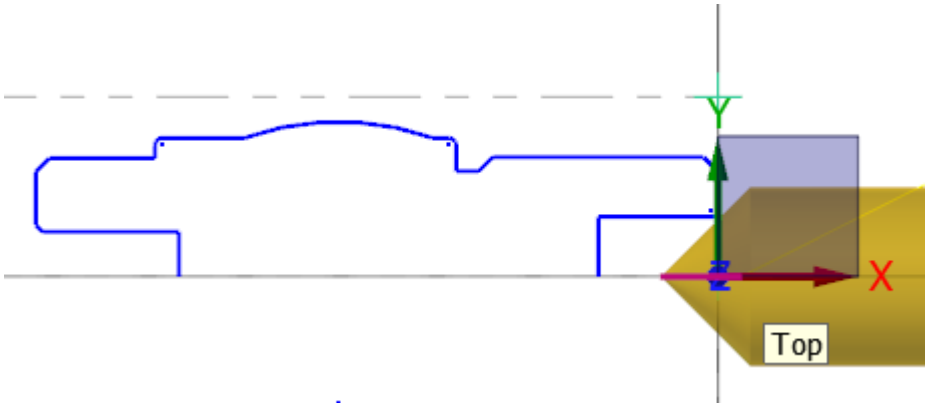
6. In the **Depth Calculator** dialog box, change **Finish diameter** to **0.24**, select **Overwrite depth**, and then click **OK**.



7. Deselect **Clearance** , set **Retract** to **0.05**, and click **OK**. Mastercam adds the toolpath to the part.



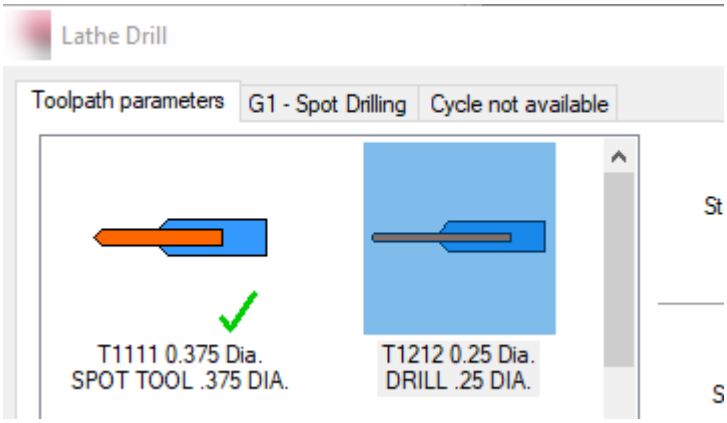
8. Backplot the toolpath to see the toolpath in action.



9. Save the file with the name `Swiss Tutorial_XX_3.mcam`, where *XX* is your initials.

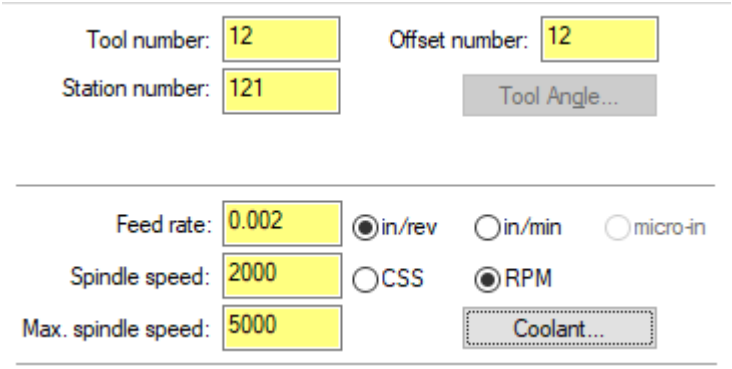
Exercise 3: Drilling the front hole

1. Select **Drill** from the **General** gallery on the **Lathe, Turning** contextual tab.
2. In the **Toolpath Parameters** tab, choose the **T1212 0.25 Dia. DRILL .25 DIA.** drill.

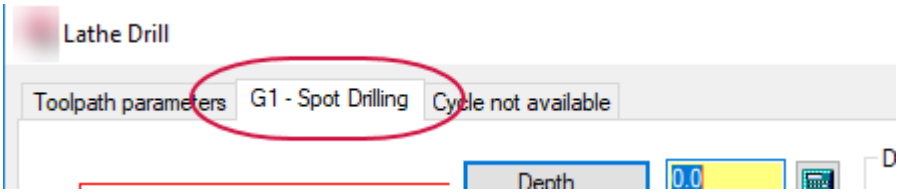


3. Set the parameters as shown in the following image.

   Tool number: 12
   Offset number: 12
   Station number: 121
   Feed rate: 0.002, in/rev
   Spindle speed: 2000, RPM

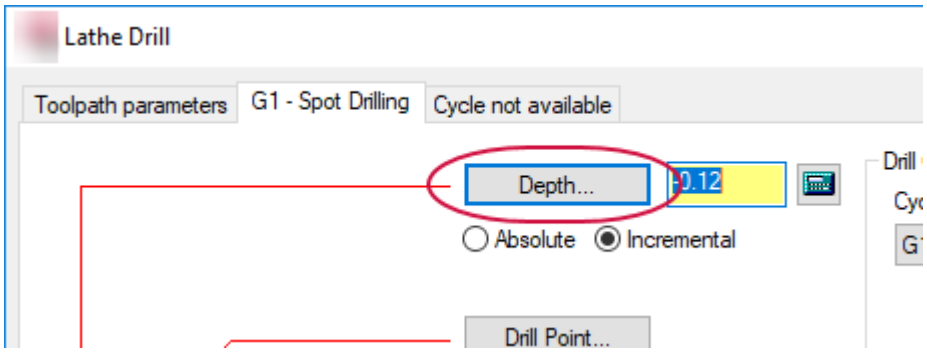


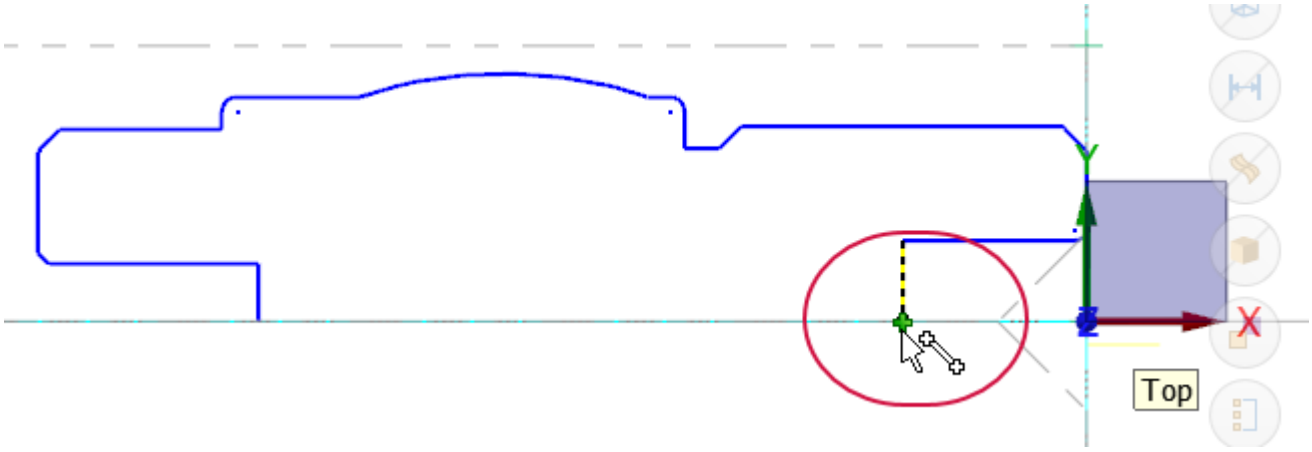
4. Select the **G1 - Spot Drilling** tab.



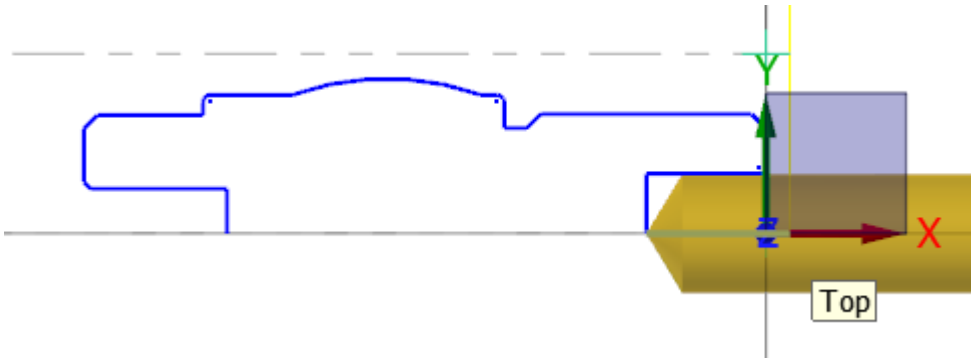
5. Ensure that the **Cycle** drop-down is set to **G1 - Spot Drilling**.

6. Click the **Depth** button, and when Mastercam returns you to the graphics window, select the point shown in the second picture following.





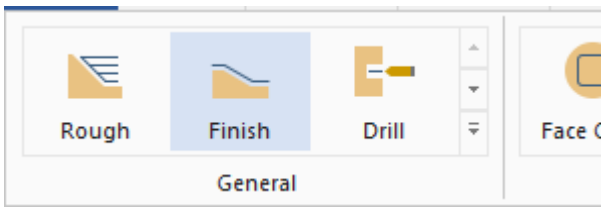
7. Click **OK** to accept your changes and to exit the dialog box.
8. Backplot the operation to see the toolpath in action..



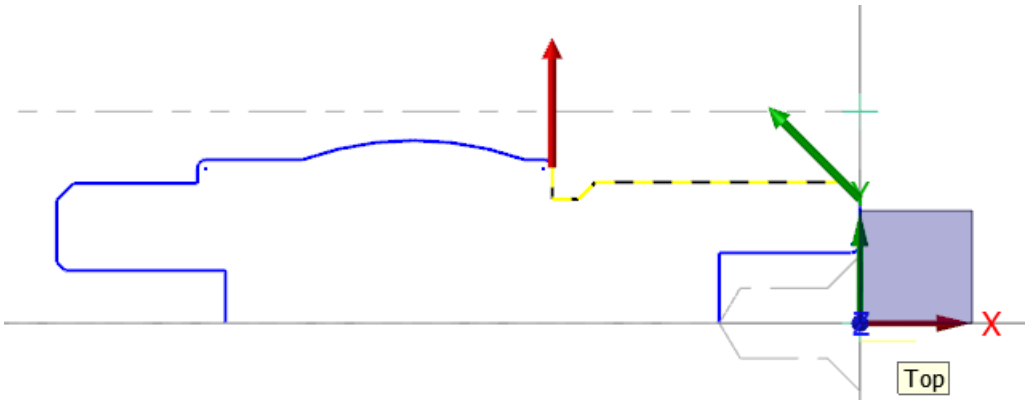
9. Save the file with the name `Swiss Tutorial_XX_4.mcam`, where *XX* is your initials.

Exercise 4: Finishing the forward OD

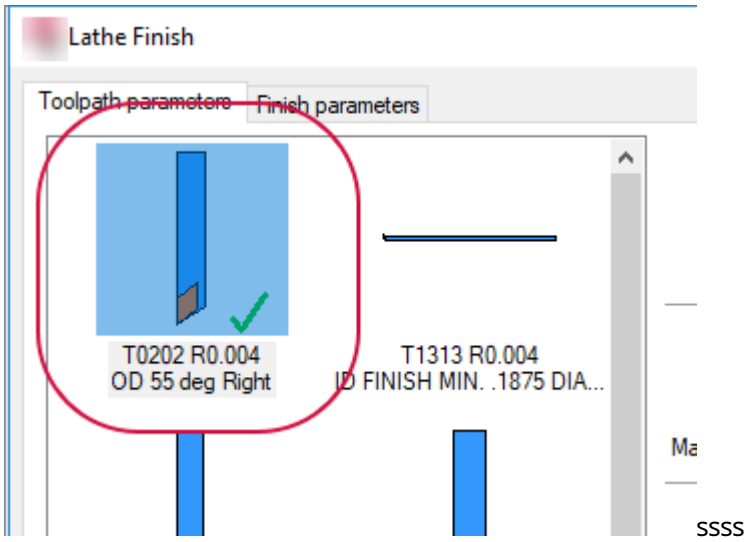
1. Select **Finish** from the **General** gallery on the **Lathe, Turning** contextual tab.



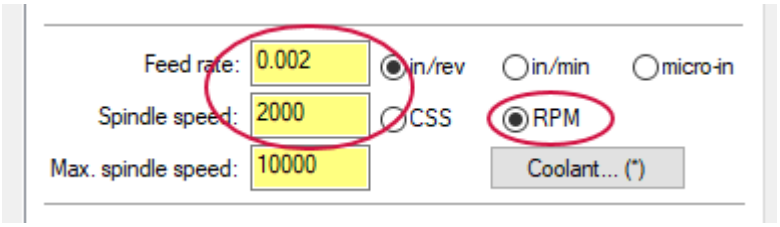
2. Chain the part as shown in the picture below, and click **OK** in the **Chaining** dialog box. If the chains point in the wrong direction, use the **Reverse** button to correct them.



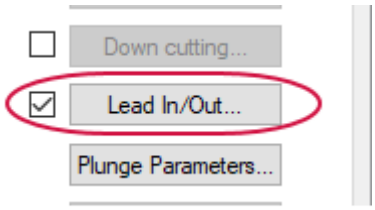
3. Select the **T0202 R0.004 OD 55 deg Right** tool.



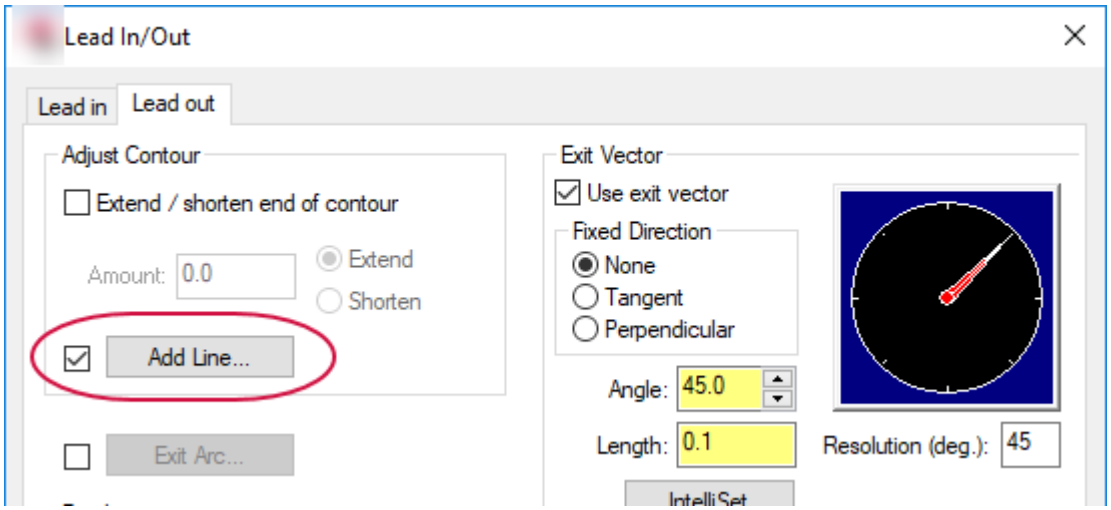
4. Set **Feed rate** to **0.002, in/rev** and **Spindle speed** to **2000, RPM**, as shown in the following picture.



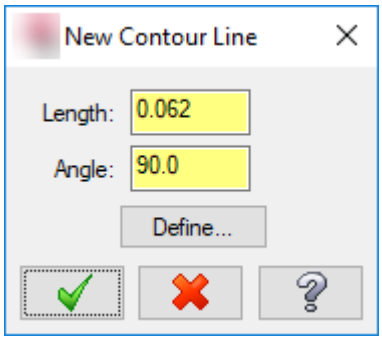
5. On the **Finish parameters** tab, select the **Lead In/Out** button.



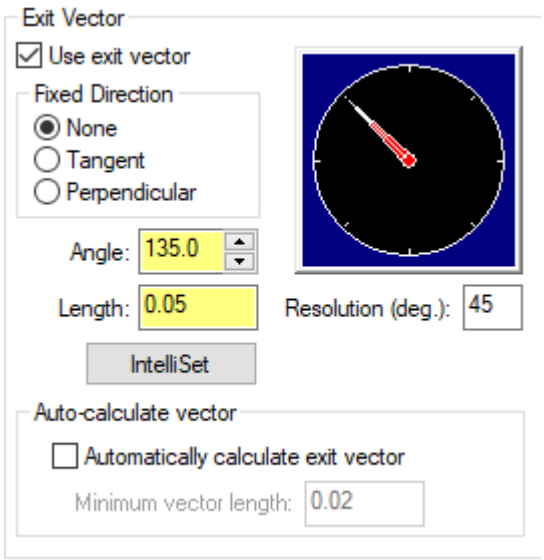
6. On the **Lead out** tab, select the checkbox next to **Add line**, and then click the **Add Line** button.



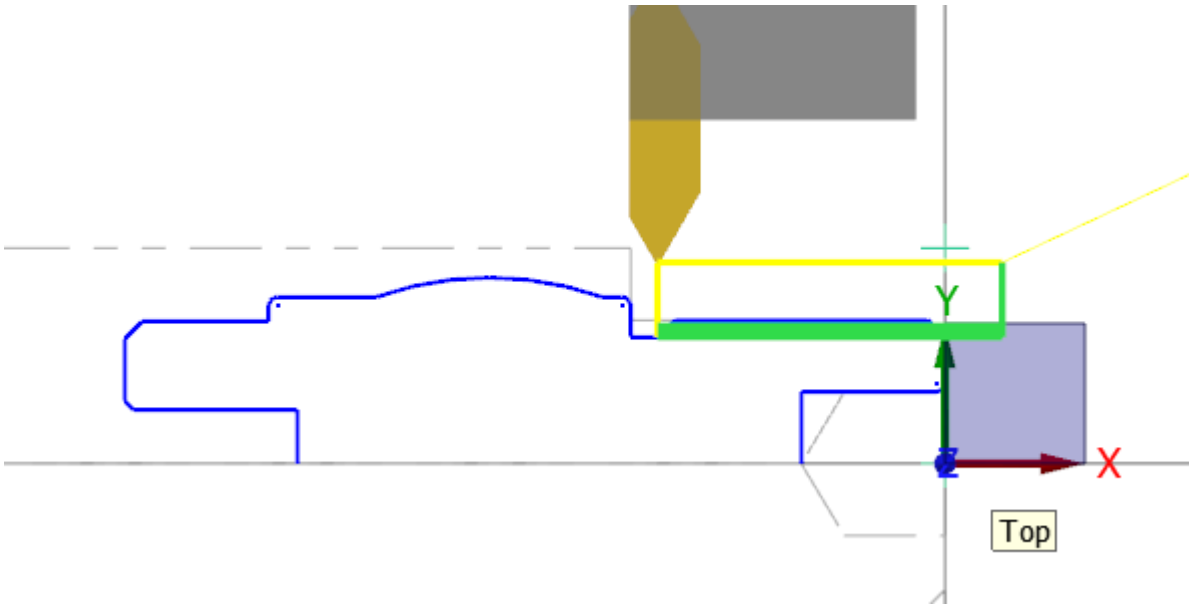
7. In the **New Contour Line** dialog box, set **Length** to **0.062** and **Angle** to **90**. Click **OK**. This allows you to create an edge break in the event of stock pullback.



8. Back in the **Lead out** tab, set the **Angle** to **135** and **Length** to **0.05**.



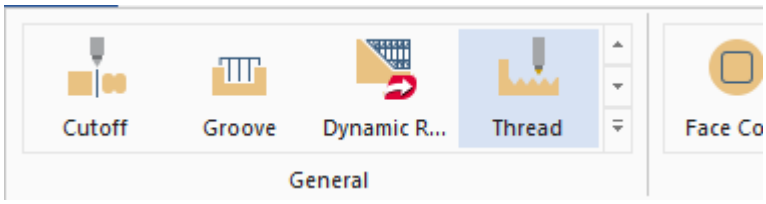
9. Click **OK** in both the **Lead In/Out** and **Lathe Finish** dialog boxes.
10. Backplot the toolpath to see it in action.



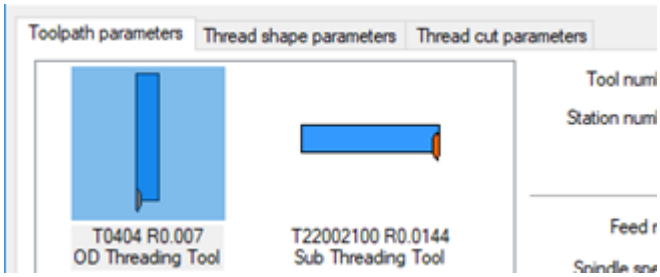
11. Save the file with the name `Swiss Tutorial_XX_5.mcam`, where *XX* is your initials.

Exercise 5: Cutting the threads

1. Select **Thread** from the **General** gallery on the **Lathe, Turning** contextual tab.

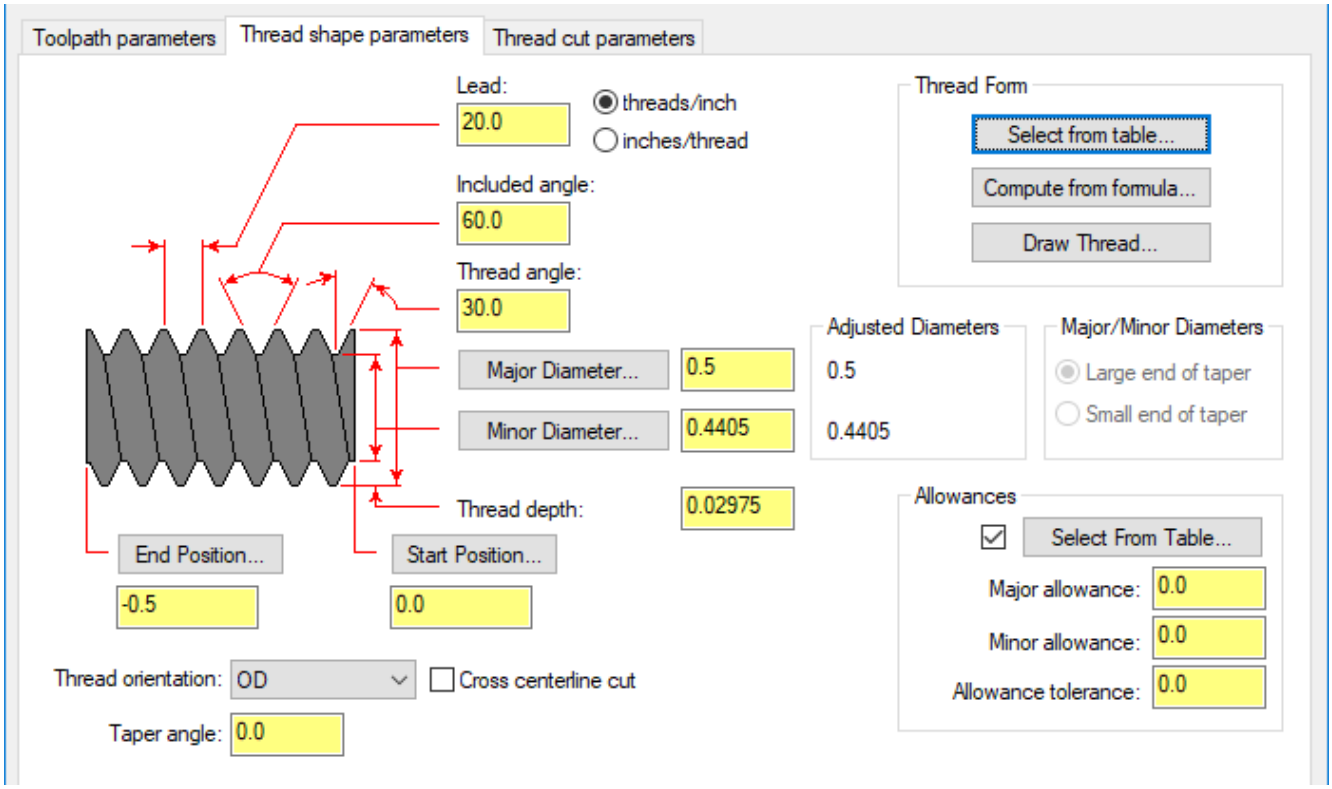


2. In the **Lathe Thread** dialog box, select tool **T0404**, and ensure that the parameters are as follows.

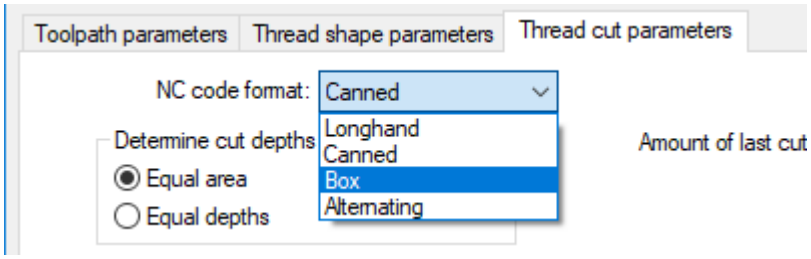


3. On the **Thread shape parameters** tab, set the parameters as shown in the following image. You can also use the **Select from table** button to choose the parameters that are appropriate for the threading operation.

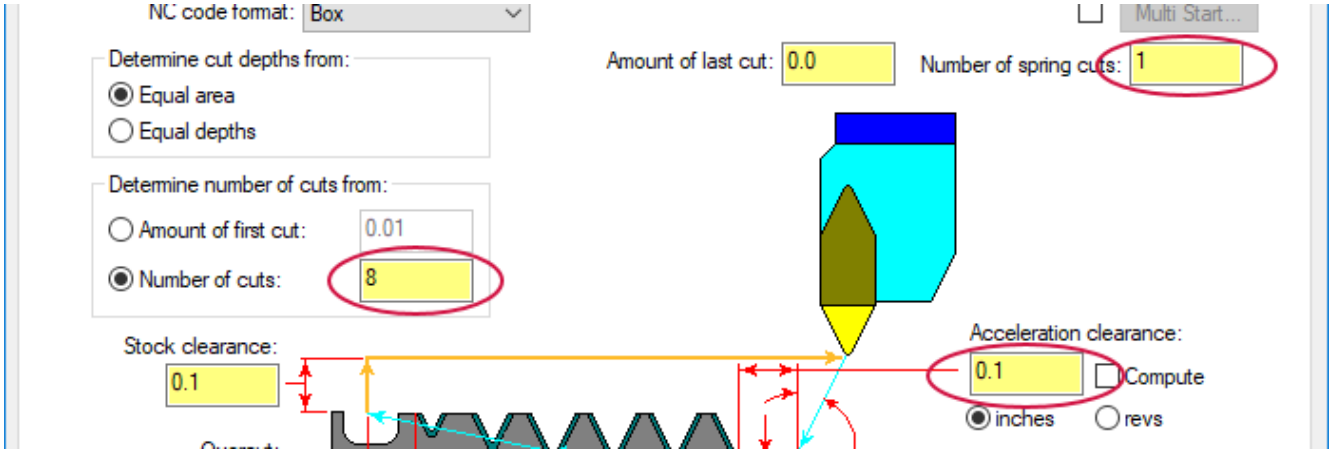
   Lead: 20
   Included angle: 60.0
   Thread angle: 30.0
   Major Diameter: 0.5
   Minor Diameter: 0.4405
   Thread depth: 0.02975
   End Position: -0.5
   Start Position: 0.0

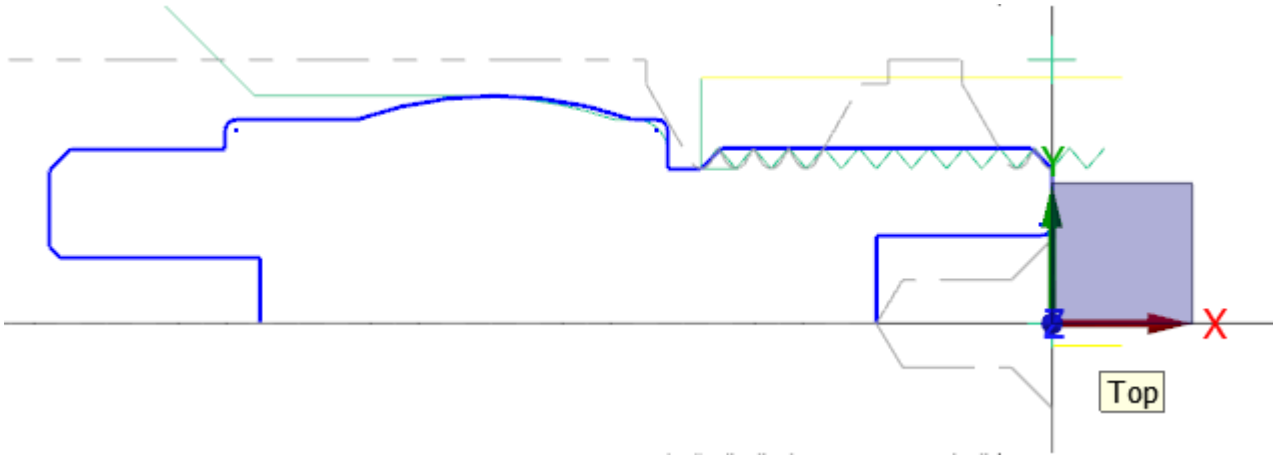


4. On the **Thread cut parameters** tab, select **Box** in the **NC code format** drop-down. **Box** is a G92, **Canned** is G76, and **Longhand** is G32.

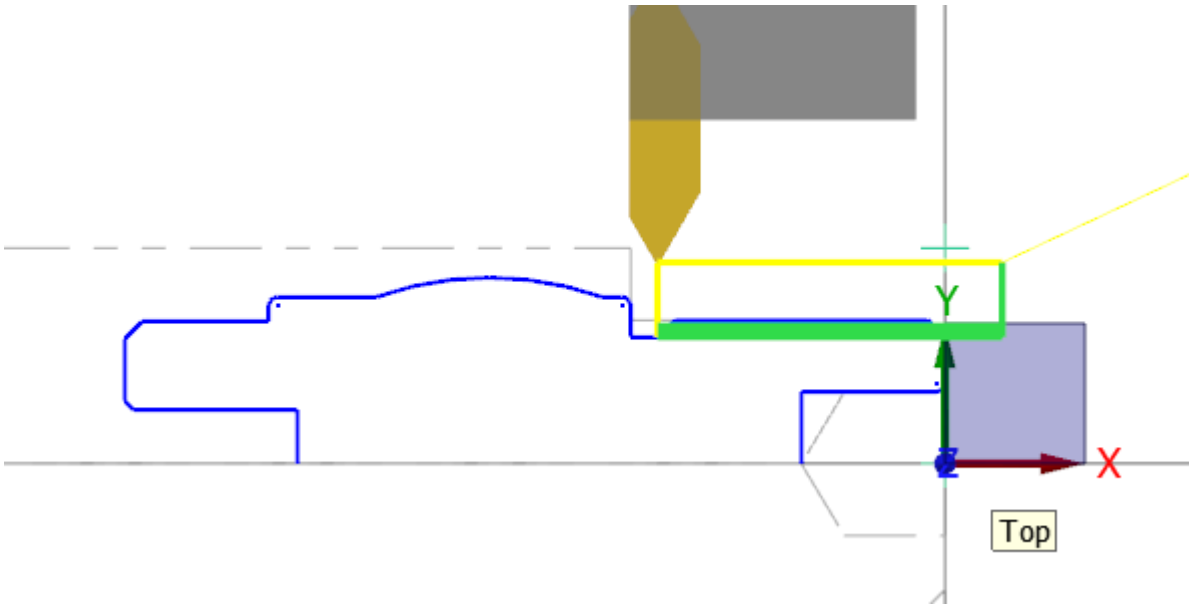


5. Set **Number of spring cuts** to **1**, **Number of cuts** to **8**, **Acceleration clearance** to **0.1**, and then click **OK**. You should see the threading toolpath shown in the second picture following.





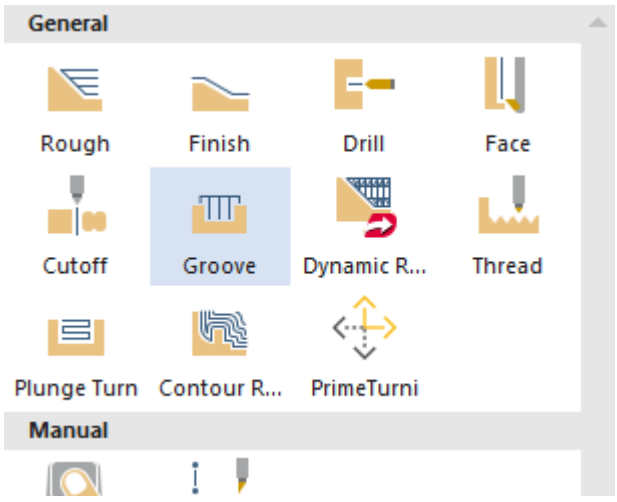
6. Backplot the toolpath to see it in action.



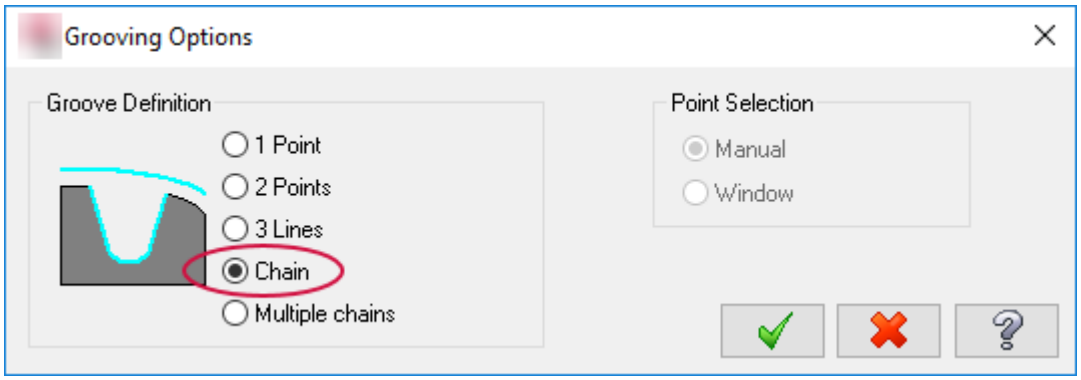
7. Save the file with the name `Swiss Tutorial_XX_6.mcam`, where *XX* is your initials.

Exercise 6: Cutting the groove

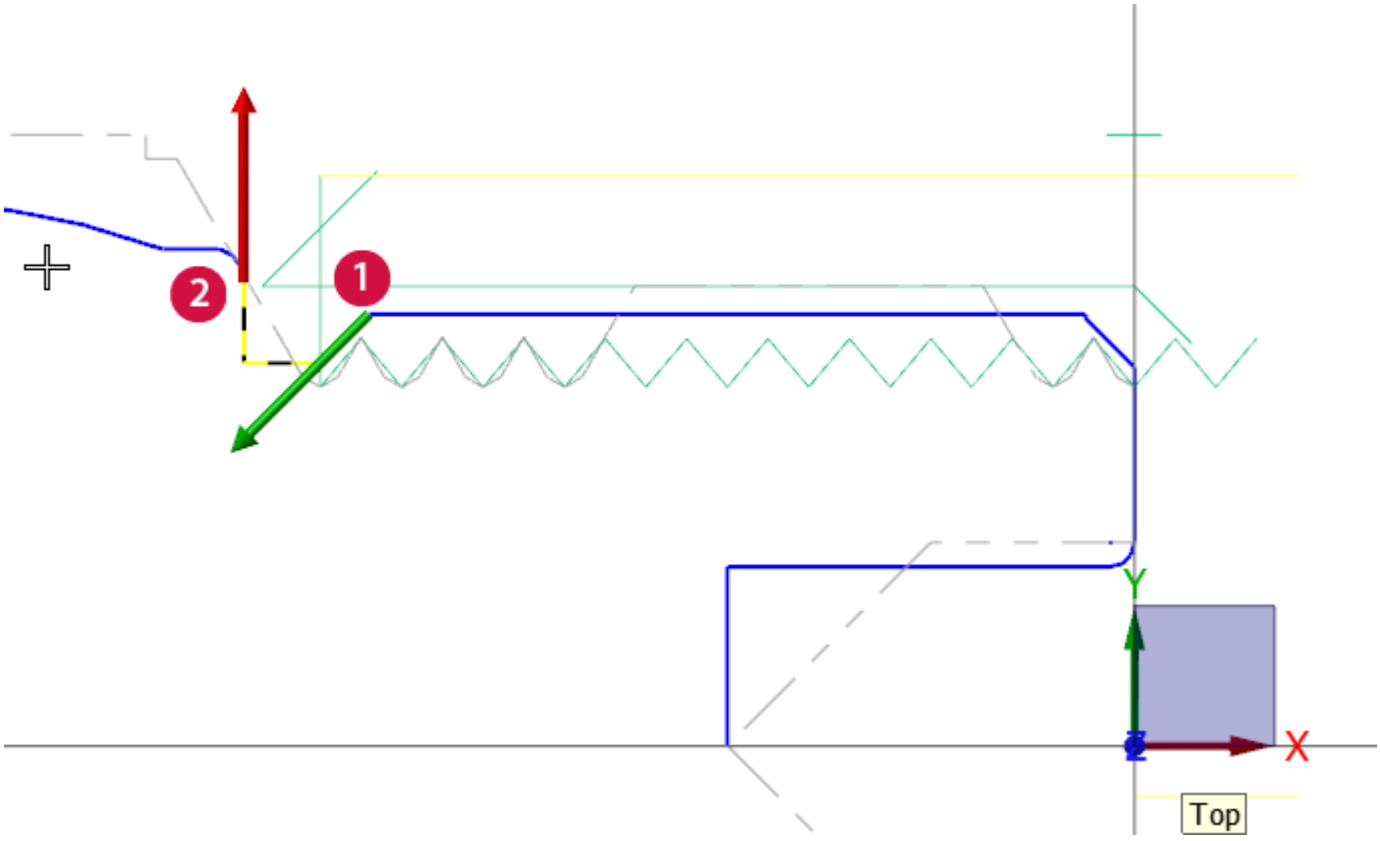
1. Select **Groove** from the **General** gallery on the **Lathe, Turning** contextual tab.



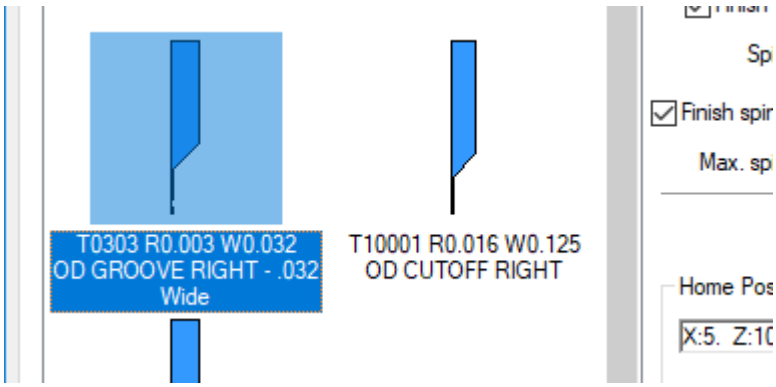
2. In the **Grooving Options** dialog box, ensure that **Chain** is selected, and then click **OK**.



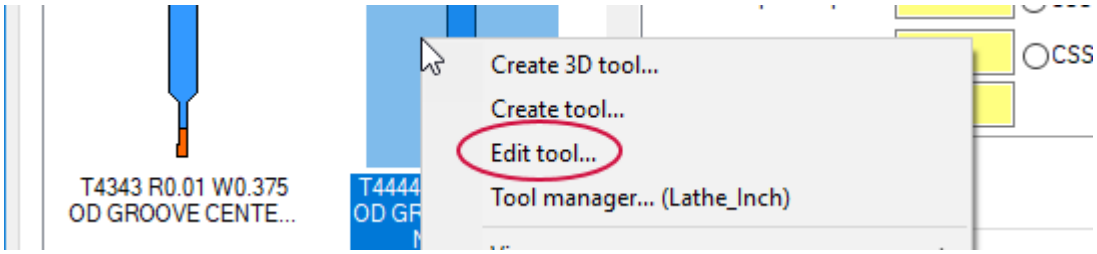
3. Make the chaining selections shown in the following picture, and click **OK** in the **Chaining** dialog box.



4. Select the **T0303 W0.032 OD GROVE RIGHT - .032 Wide** grooving tool.

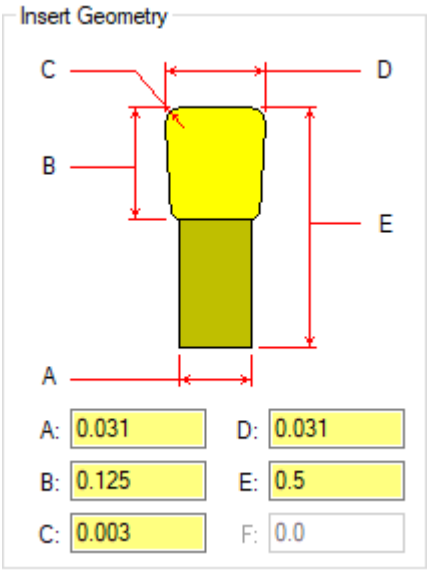


5. Right-click the tool, and select **Edit Tool** from the menu that appears.

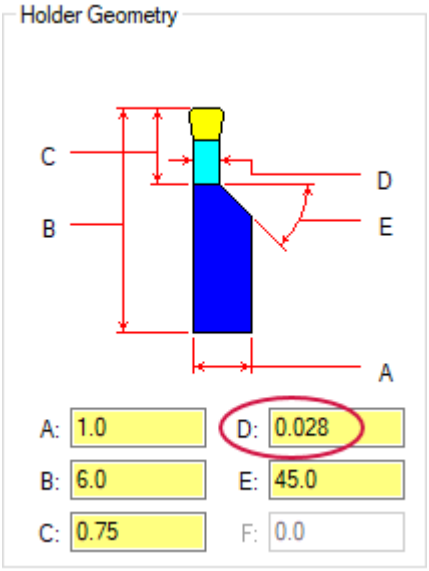


6. On the **Inserts** tab, set the **Insert Geometry** parameters as shown in the following picture.

A: 0.031
B:0.125
C: 0.003
D: 0.031
E: 0.5

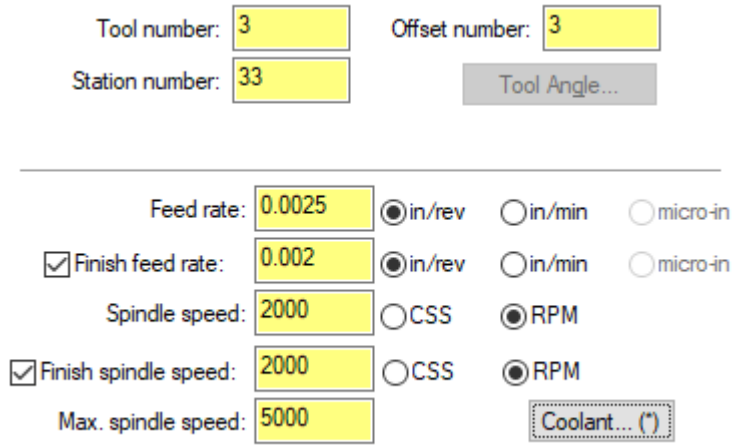


7. On the **Holders** tab, change **D** in **Holder Geometry** to **0.028**, and click **OK**.

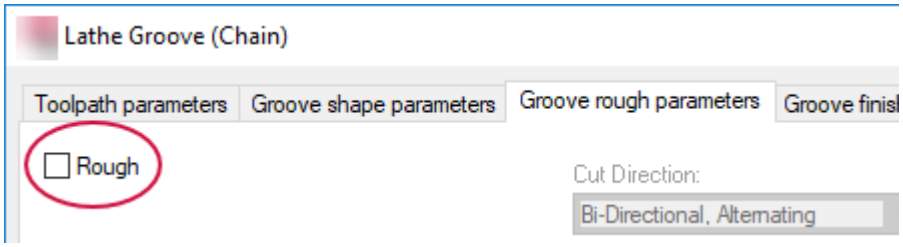


8. Back in the **Toolpath parameters** tab, set the parameters as follows:

   Tool number: 3
   Offset number: 3
   Station number: 33
   Feed rate: 0.0025
   Finish feed rate: 0.002
   Spindle speed: 2000
   Finish spindle speed: 2000
   Max spindle feed: 5000



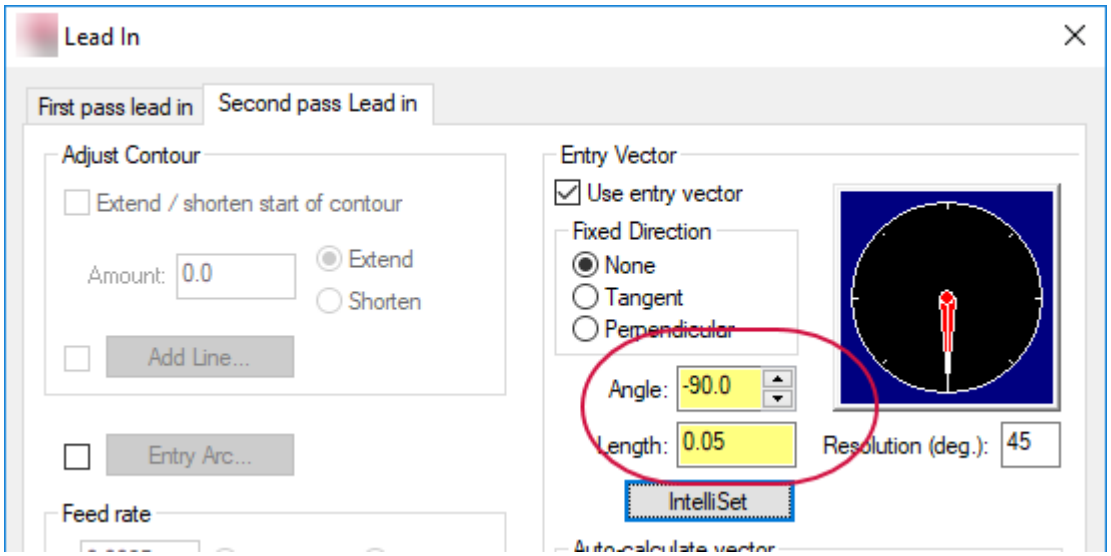
9. On the **Groove rough parameters** tab, deselect the **Rough** checkbox.



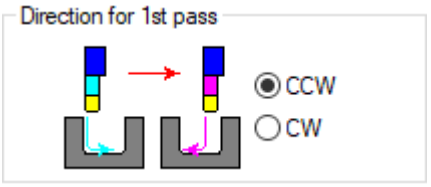
10. On the **Groove finish parameters** tab, click the **Lead In** button.

11. On the **First pass lead in** tab, set **Length** to **0.05**.

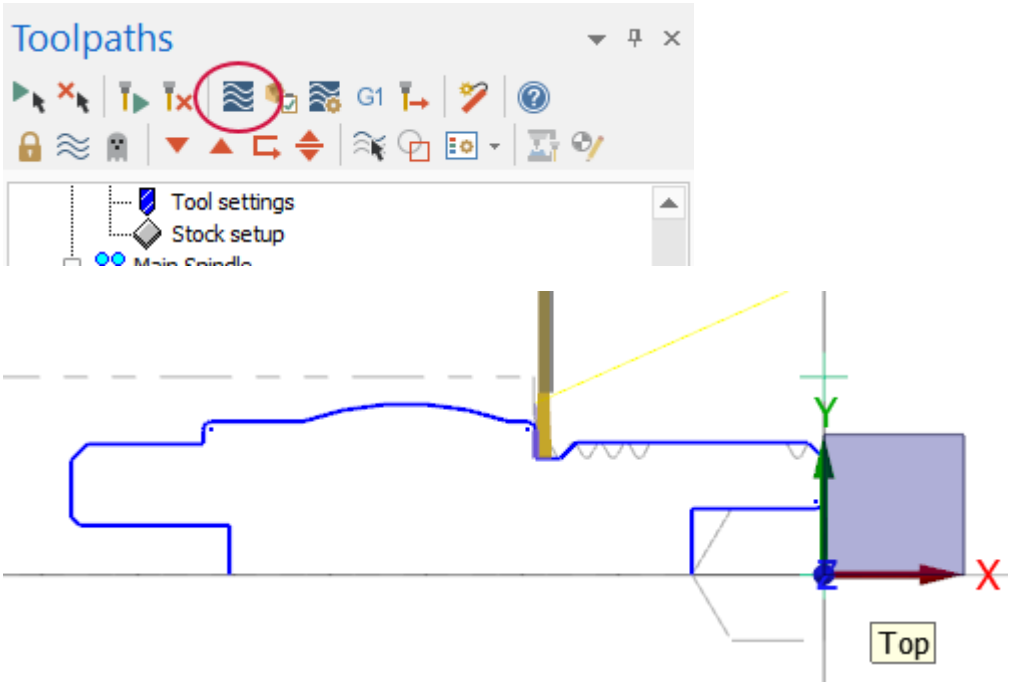
12. On the **Second pass Lead in** tab, set the direction indicator so that it is pointing straight down (-90 degrees), change **Length** to **0.05**, and click **OK**.



13. In the **Groove finish parameters** tab, change **Direction for 1st pass** to **CCW**, and click **OK**.



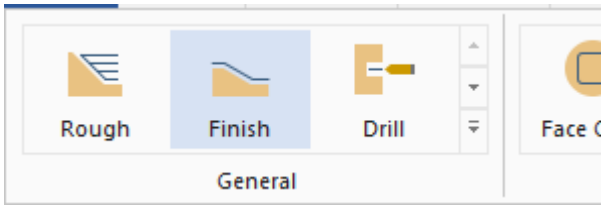
14. Backplot the new toolpath to see it in action.



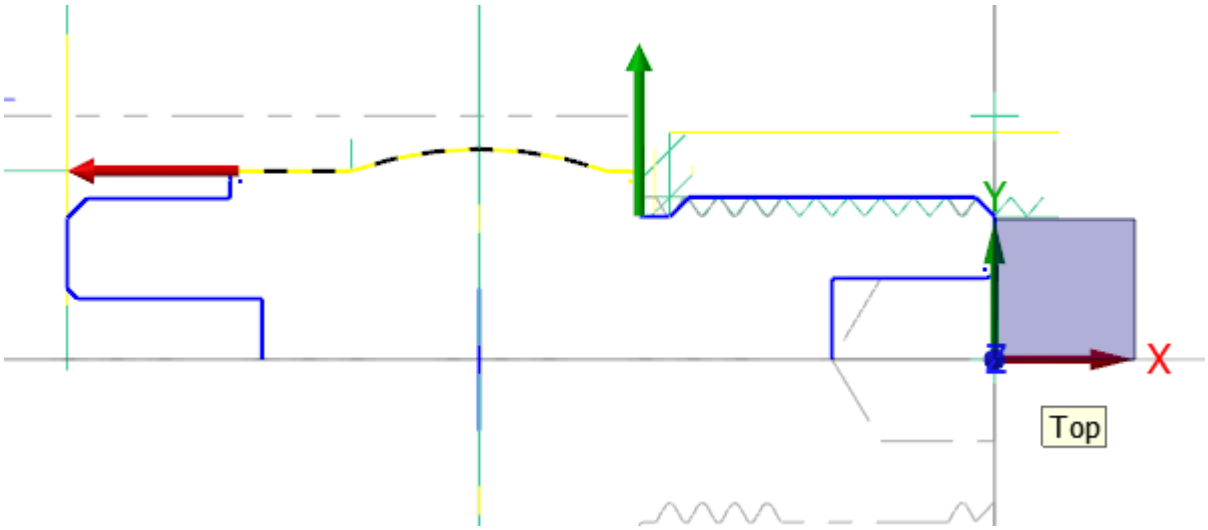
15. Save the file with the name `Swiss Tutorial_XX_7.mcam`, where *XX* is your initials.

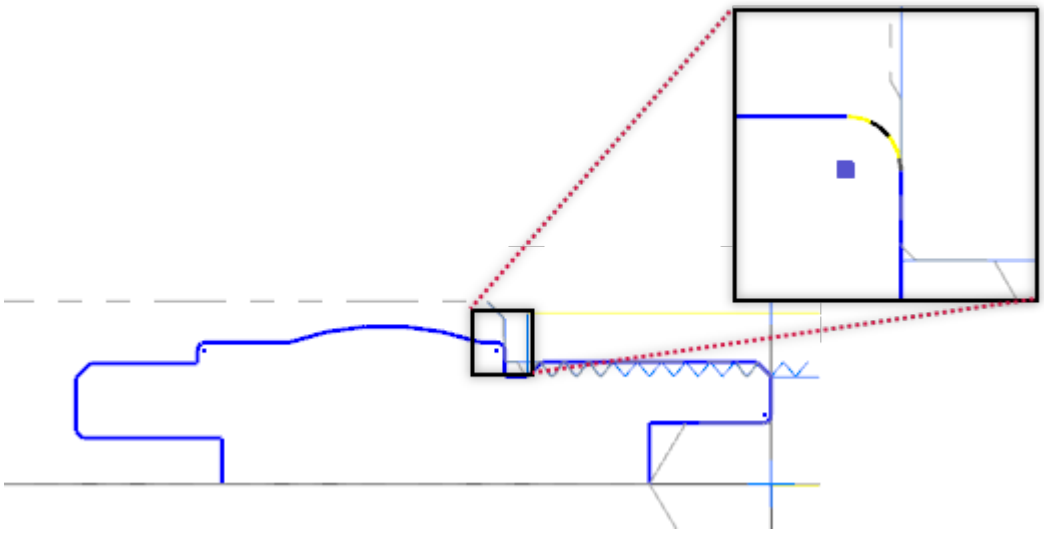
Exercise 7: Finishing the center OD

1. Select **Finish** from the **General** gallery on the **Lathe, Turning** contextual tab..

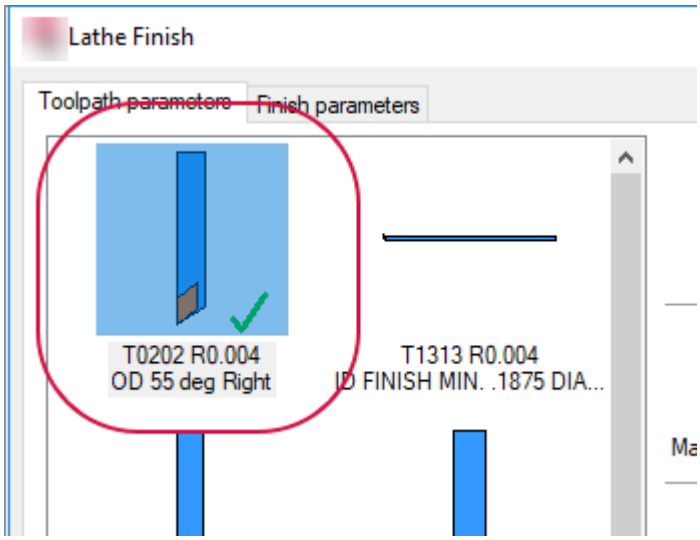


2. Chain the part as shown in the picture below, and click **OK** in the **Chaining** dialog box. Note that the first chaining selection should be the radius shown in the second picture following, and not the vertical wall.

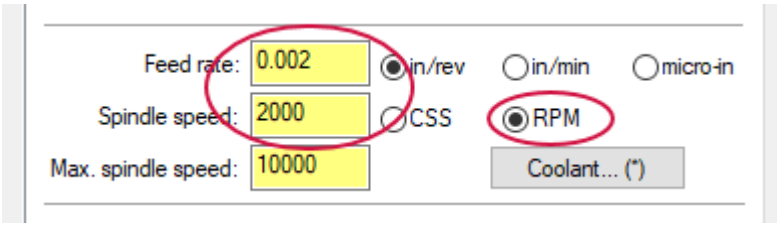




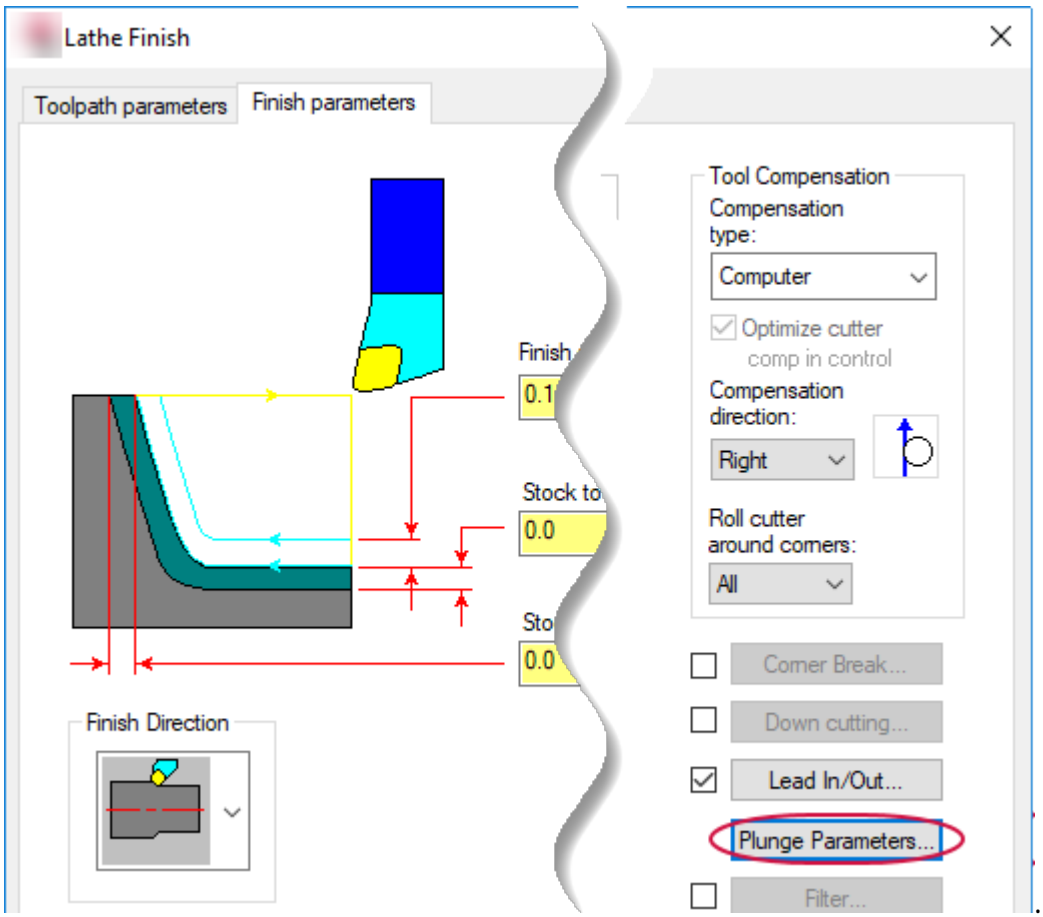
3. Select the **T0202 R0.004 OD 55 deg Right** tool.



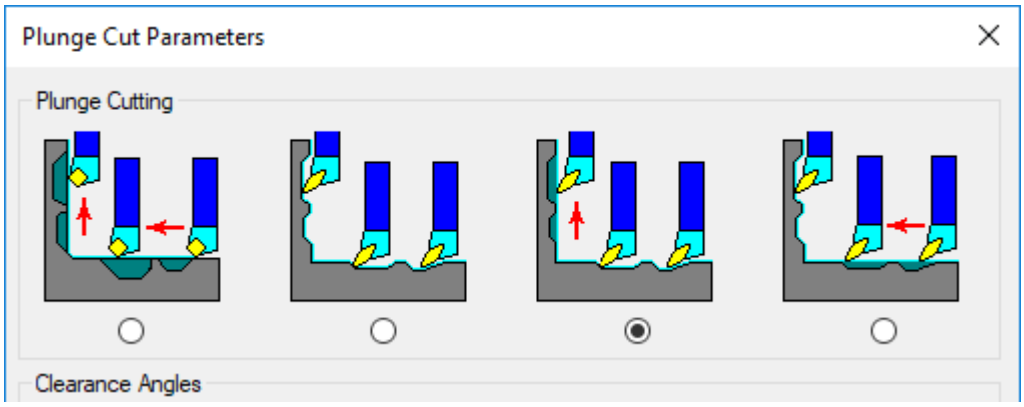
4. Set **Feed rate** to **0.002** and **Spindle speed** to **2000**.



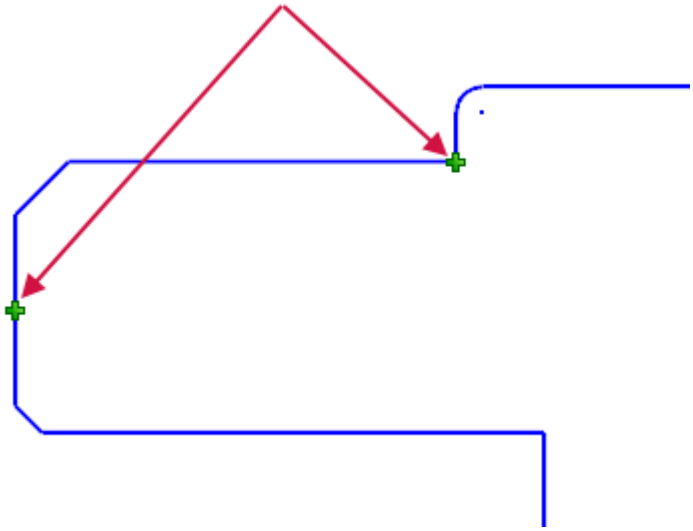
5. In the **Finish parameters** page, click the **Plunge Parameters** button.



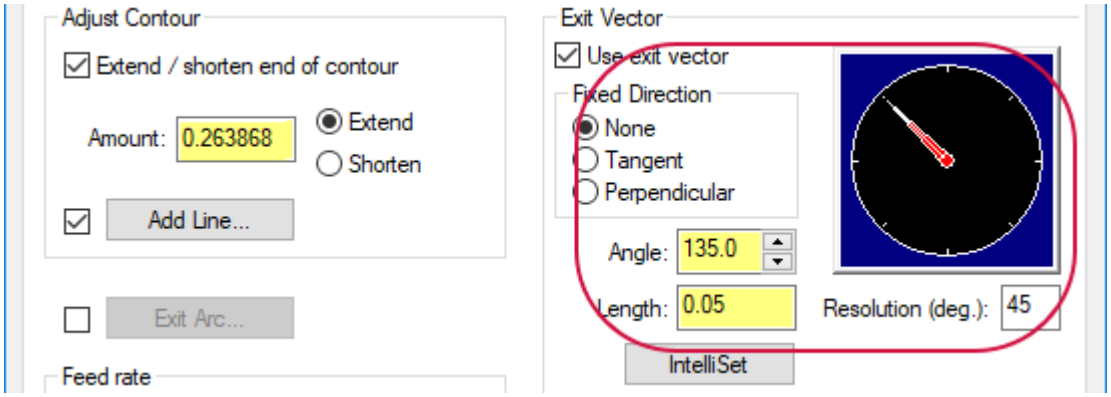
6. In the **Plunge Cut Parameters** dialog box, select the third option, and click **OK**.



7. Make the following parameter changes:
   a. In the **Finish Parameters** tab, click **Lead In/out**.
   b. On the **Lead out** tab, select the **Extend / shorten end of contour** option.
   c. Right-click in **Amount**, and select **S = Distance between 2 points** from the pop-up menu.
   d. Pick the points indicated below. You should then have **0.263868** for **Amount**.

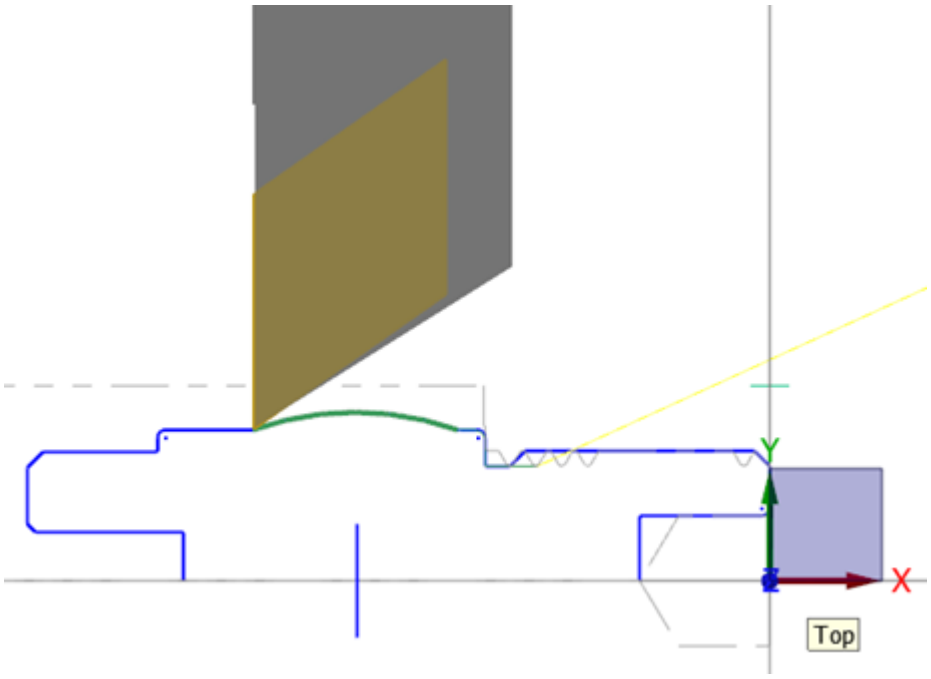


   e. Enable the **Add Line** button by clicking its checkbox, and then click the button.
   f. In the **New Contour Line** dialog box, enter **0.075** for **Length**, enter **90** for **Angle**, and click **OK**.
8. Back in the **Lead out** tab, set **Angle** to **135** and **Length** to **0.05**.



9. Click **OK** in both the **Lead In/Out** and **Lathe Finish** dialog boxes.

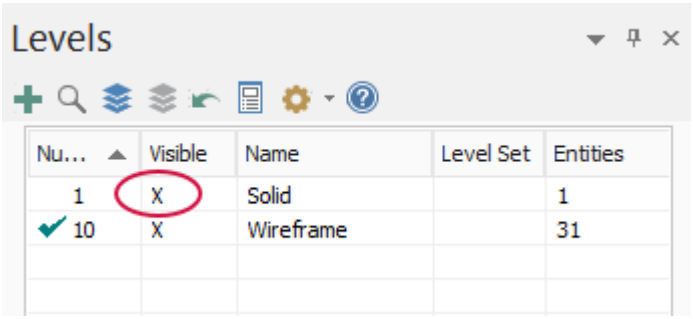
10. Backplot the toolpath.



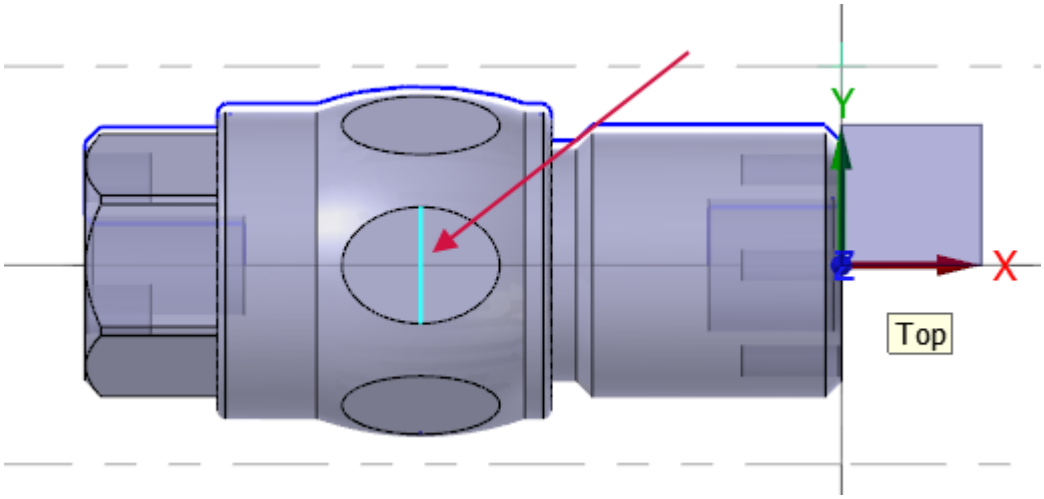
11. Save the file with the name **Swiss Tutorial_XX_8.mcam**, where *XX* is your initials.

Exercise 8: Cutting the six flat areas

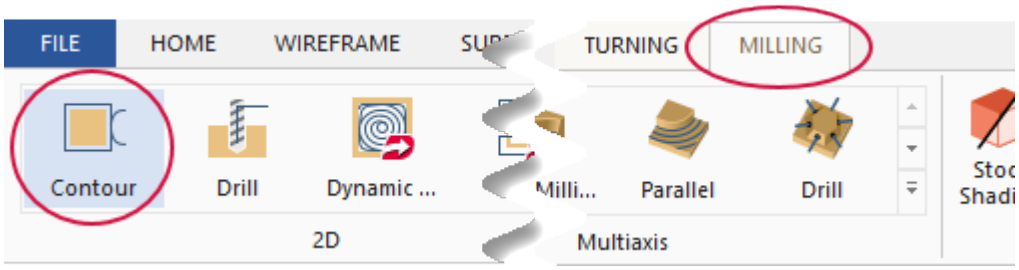
1. To display the part's solid model, in the Levels Manager, select the **Visible** column for level number 1.



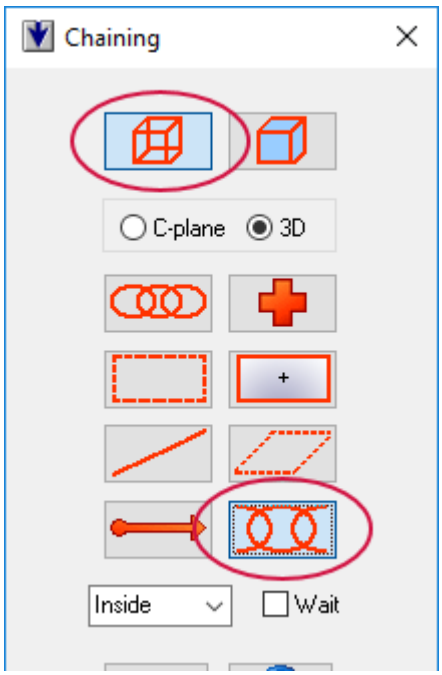
2. Select **Line Endpoints** from the **Wireframe** tab, and draw the line shown in the following image. Use the arc midpoints as anchors for the ends of the line.



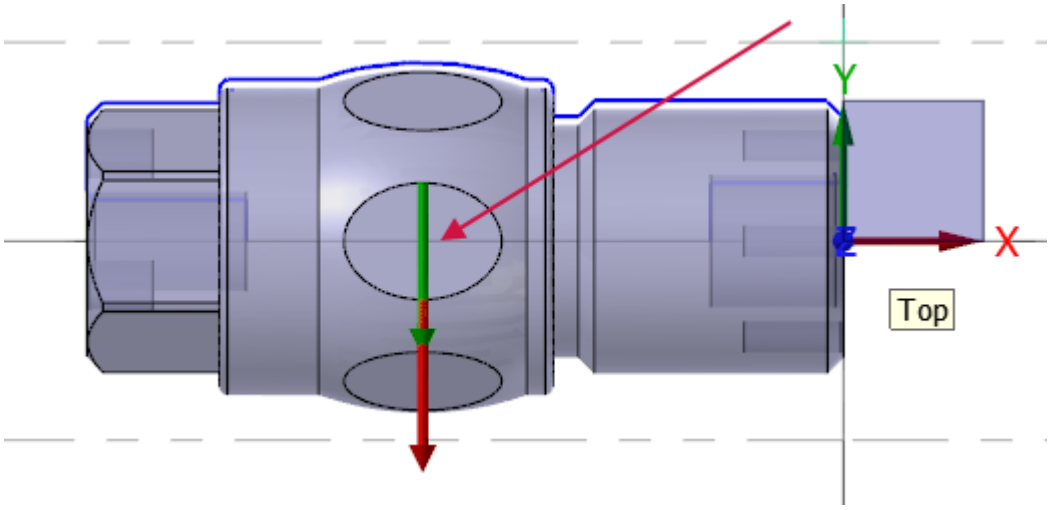
3. Select **Contour** from the 2D gallery on the **Lathe Milling** contextual tab.



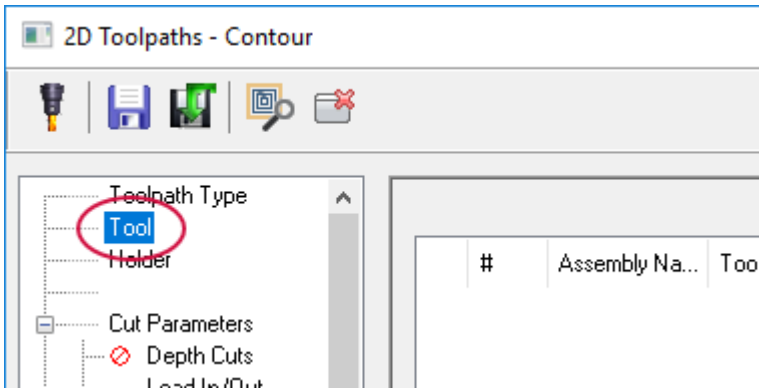
4. In the **Chaining** dialog box, select **Wireframe** and then **Partial**.



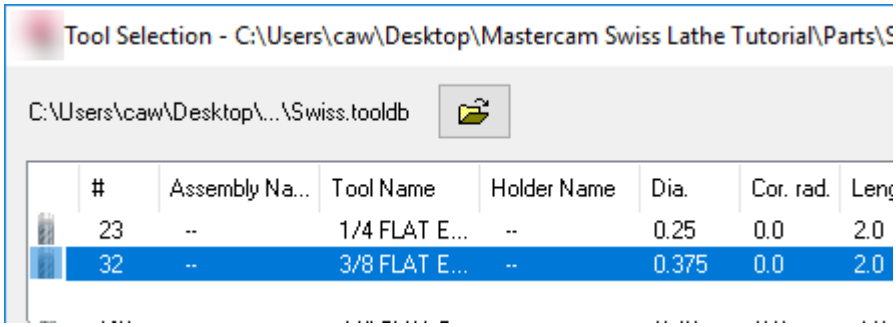
5. Select the line you created in Step 2, and click **OK** in the **Chaining** dialog box.



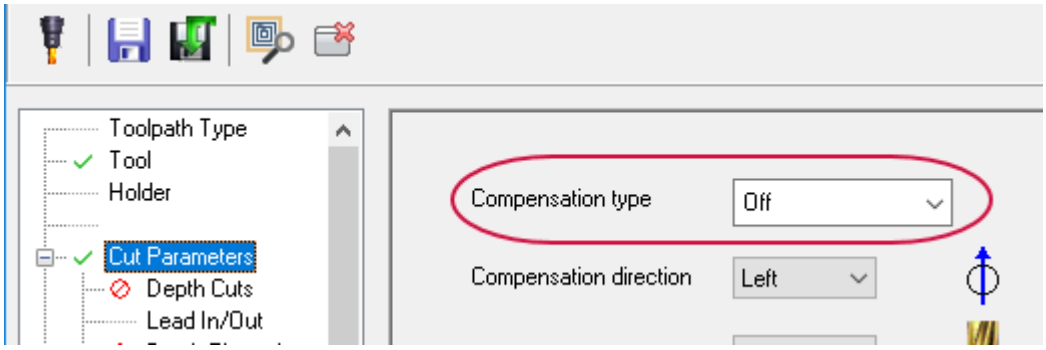
6. In the **2D Toolpaths - Contour** dialog box, click **Tool**.



7. Click **Select library tool**, choose a 3/8 flat endmill, and click **OK**.



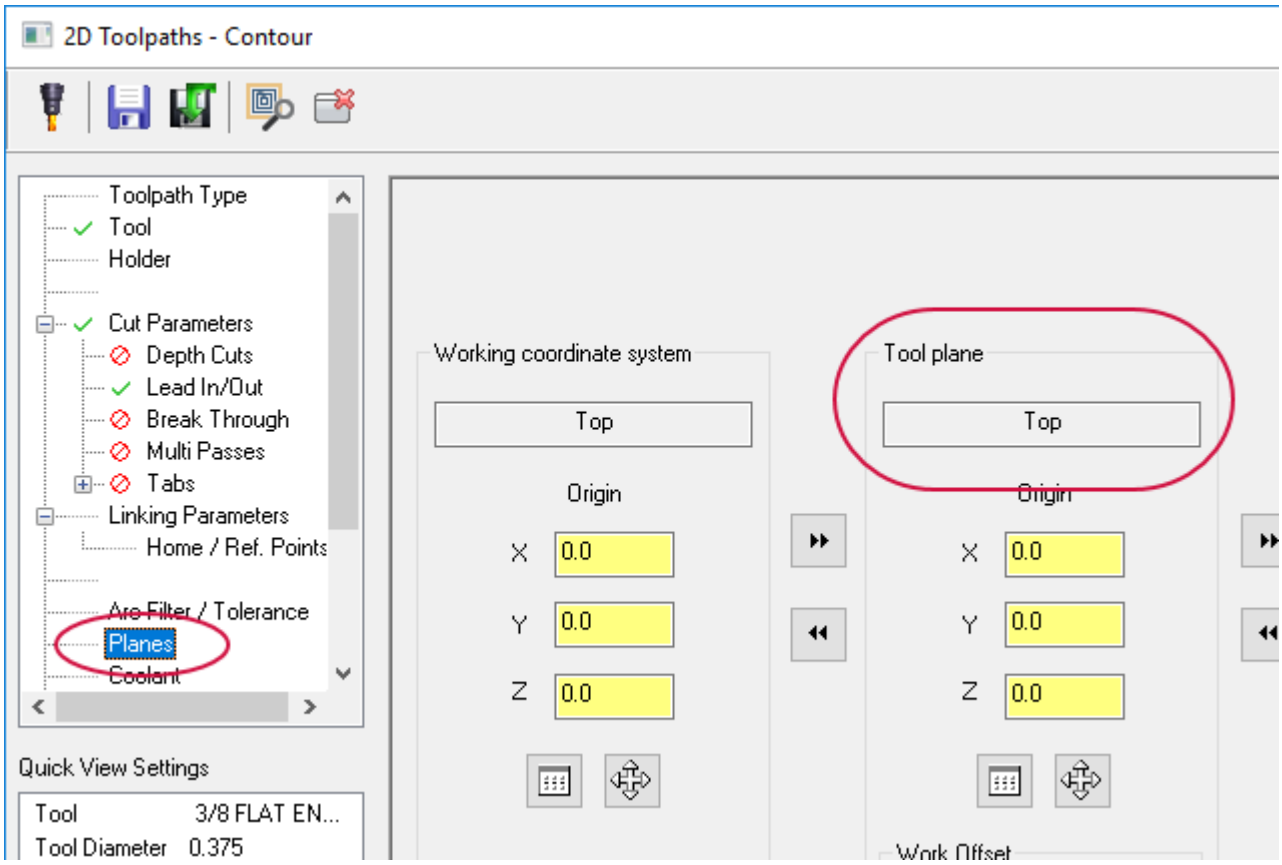
8. Select the **Cut Parameters** page, and set **Compensation type** to **Off**.



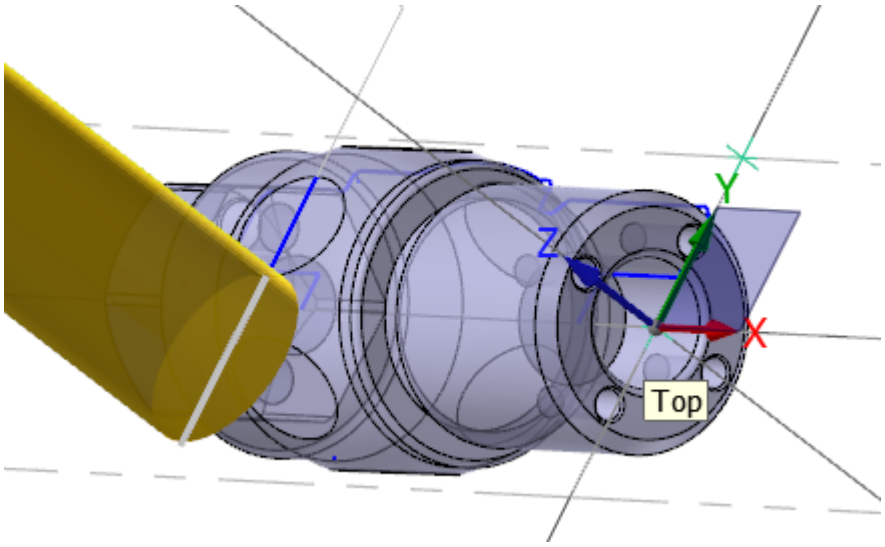
9. Select the **Lead In/Out** page, and enter the following values for both arcs indicated below.

Radius: 0.0
Sweep: 90.0
Helix height: 0.0

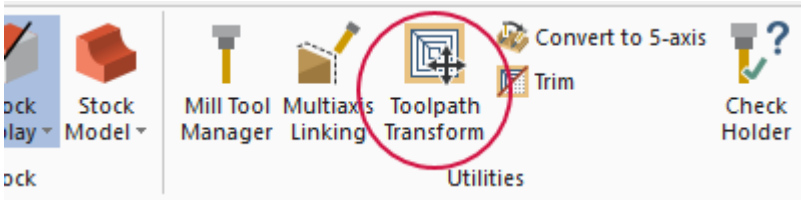
10. Select the **Planes** page, ensure that **Tool plane** is set to **Top**, and click **OK**.



11. Run Backplot to see how the operation cuts the first flat area.

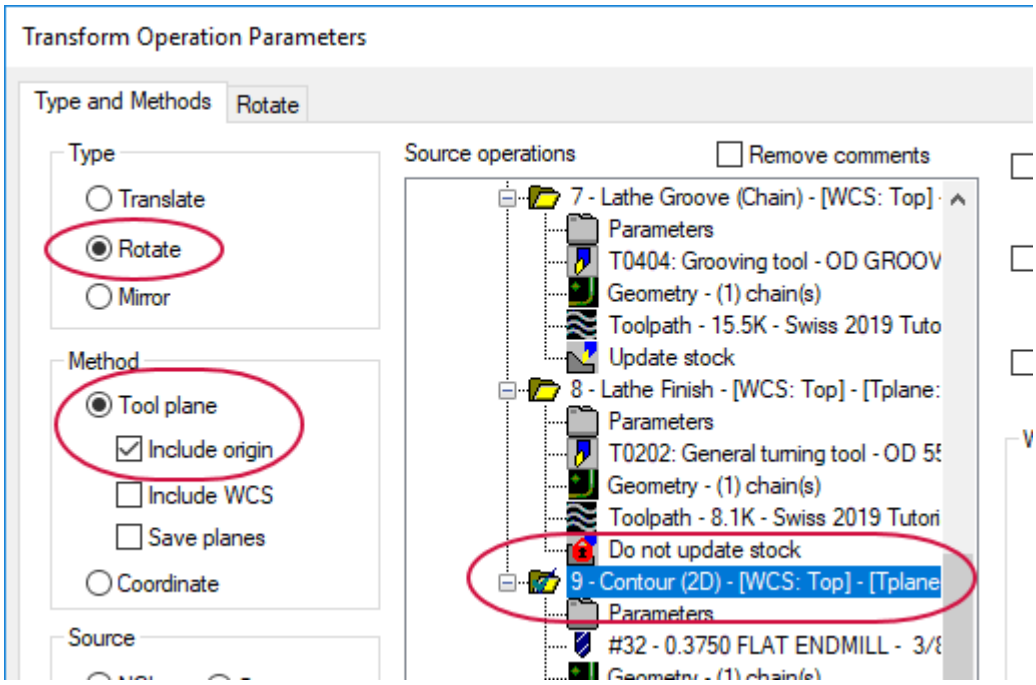


12. On the **Lathe Milling** contextual tab, select **Toolpath Transform**.

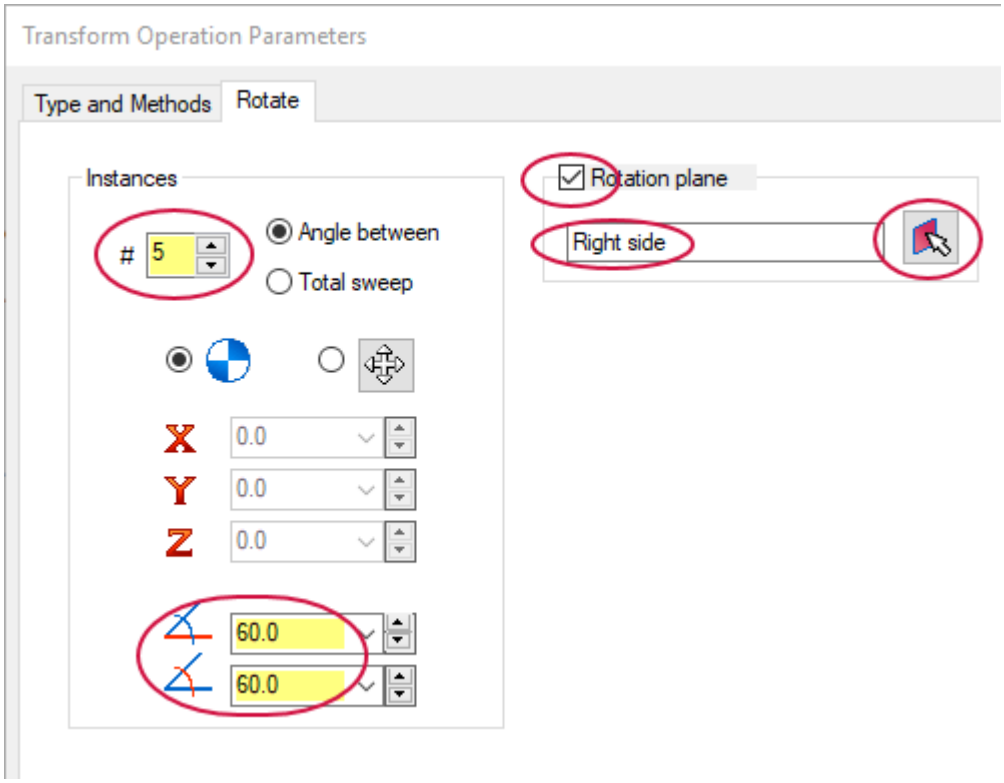


13. In the **Transfer Operation Parameters** dialog box, select **Rotate** and **Include origin**. Also, ensure that the

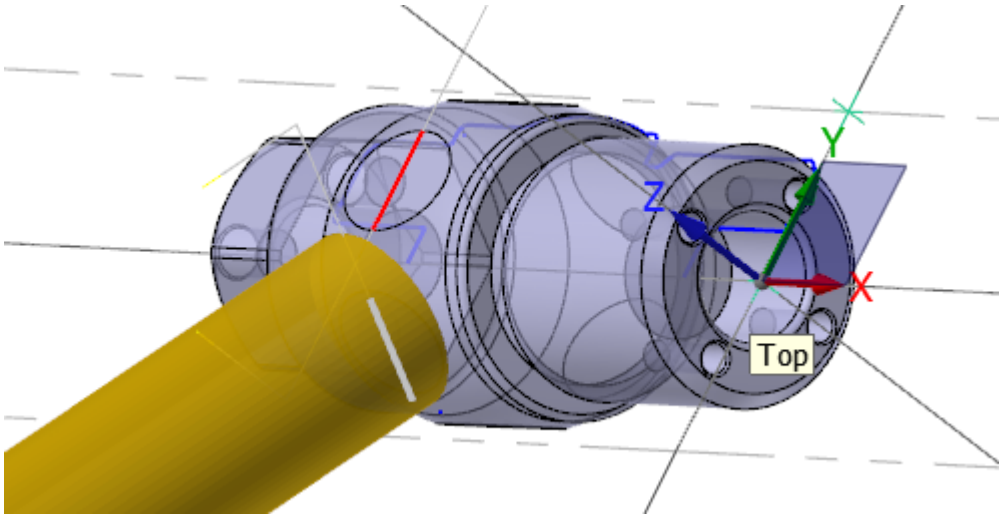
Contour operation is selected.



14. Select the **Rotate** page, set the number of instances to **5**, select **Angle between**, set both angles to **60**, and set the **Rotation plane** to **Right side** (using the **Plane selection** button). Click **OK**.



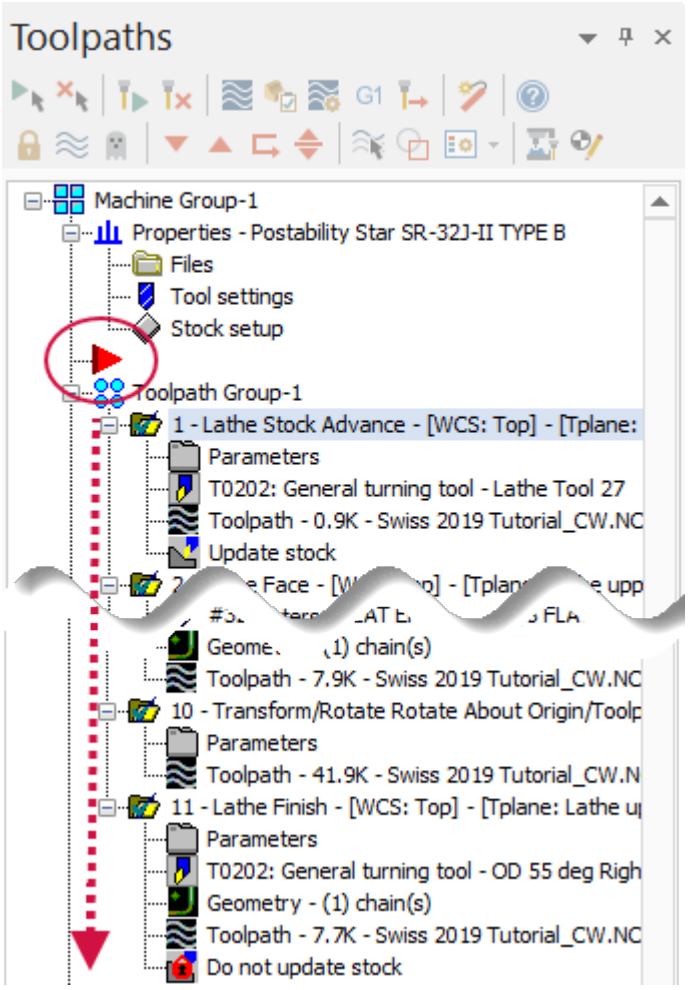
15. In Toolpaths Manager, select the two operations you created in this exercise, and run Backplot to see how the operations cut all of the flat areas.



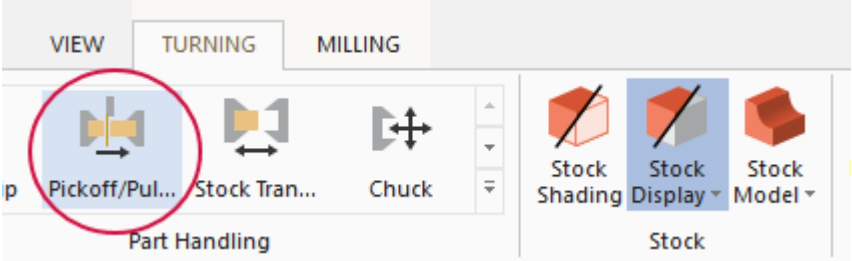
16. Save the file with the name `Swiss Tutorial_XX_9.mcam`, where *XX* is your initials.

Exercise 9: Performing the cutoff

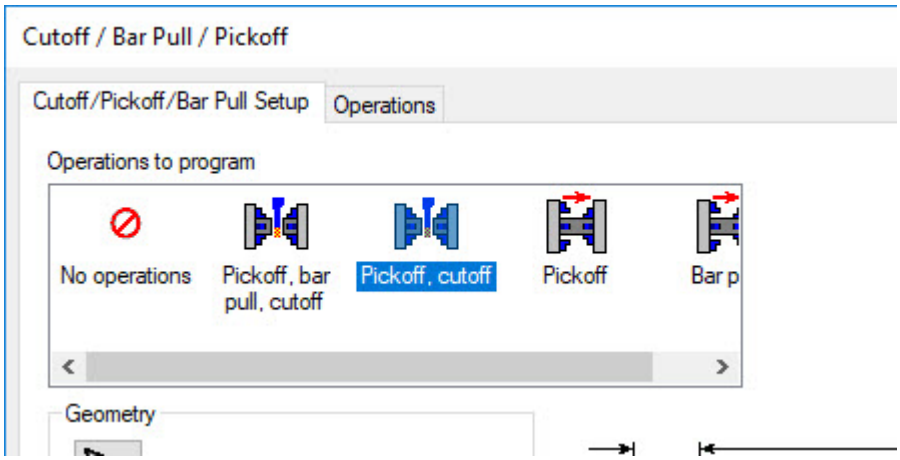
1. If necessary, drag the red arrow to below all of the operations.



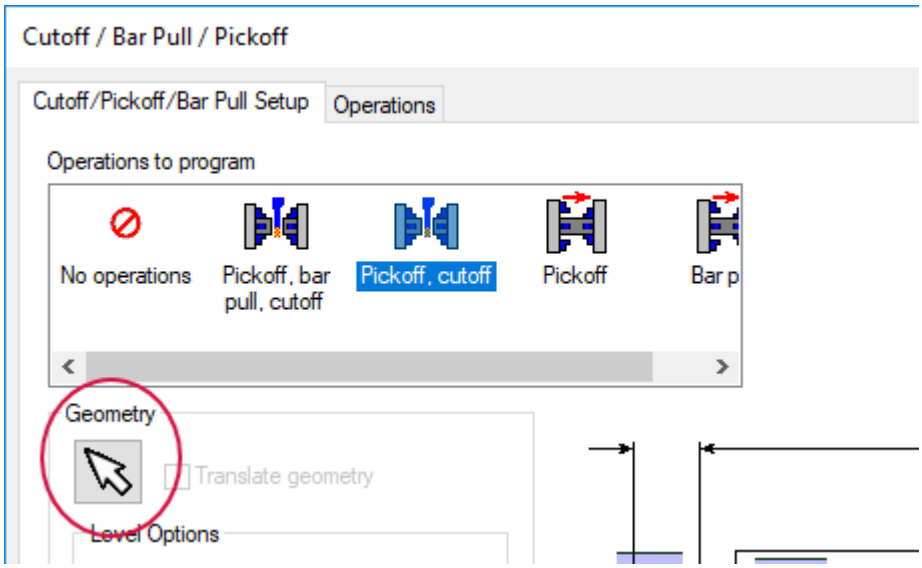
2. On the **Lathe Turning** contextual tab, select **Pickoff/Pull/Cutoff** from the **Part Handling** gallery.



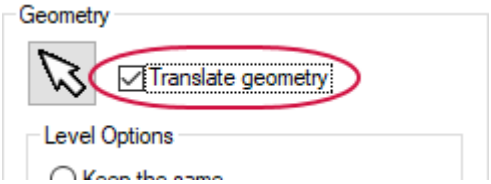
3. In the dialog box, choose **Pickoff, cutoff**. Note that **Pickoff, cutoff** is the only part-handling operation shown in the dialog box that is used with Swiss.



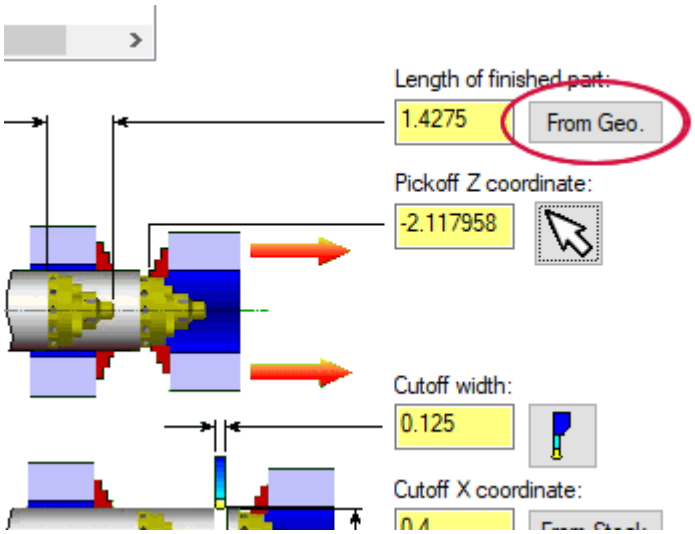
4. In the **Cutoff / Bar Pull / Pickoff** dialog box, click the **Geometry** button.



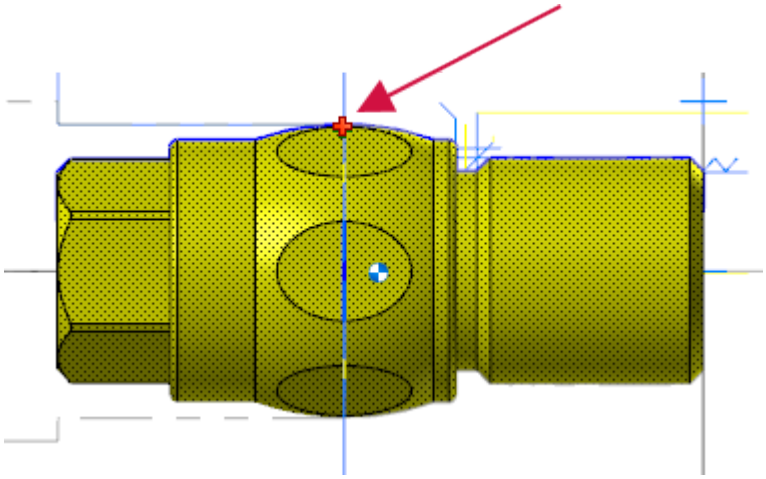
5. Select the solid model, and then press [**Enter**] or click **End Selection**.
6. Select **Translate Geometry**, which is located next to the **Geometry** button you just used.



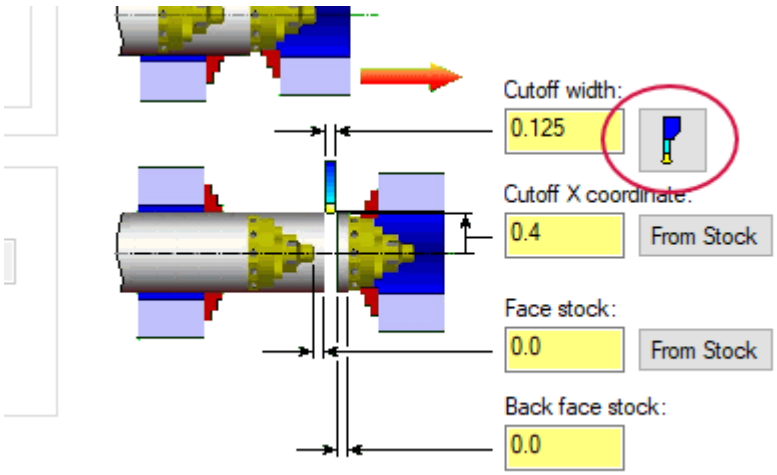
7. On the right side of the dialog box, click **From Geo.** The operation gets its length from the part geometry.



8. In the **Cutoff / Bar Pull / Pickoff** dialog box, click **Select pickoff Z coordinate**, and when returned to the graphics window, select the point shown in the second following picture.

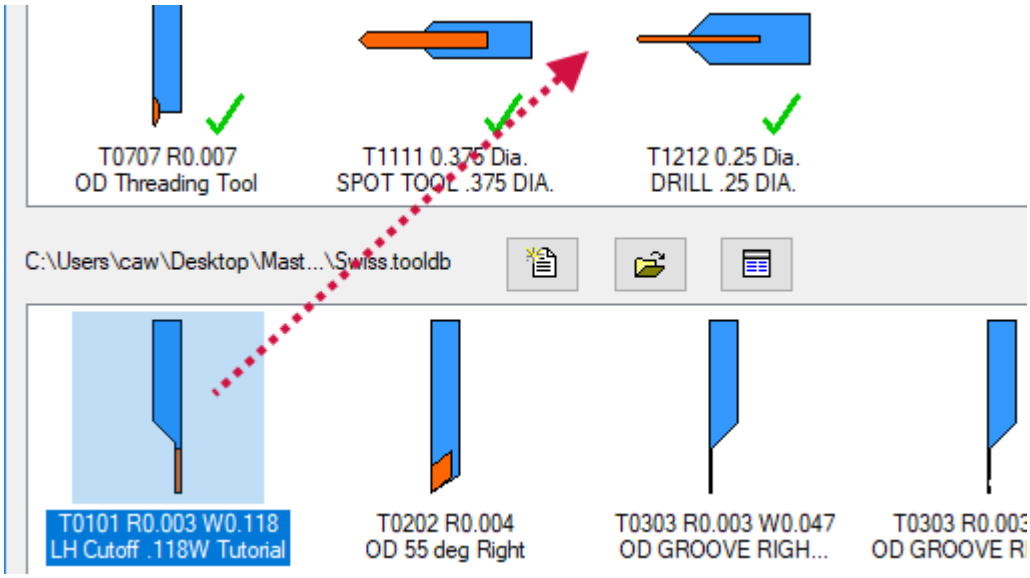


Click the **Select a tool to define cutoff width** button.

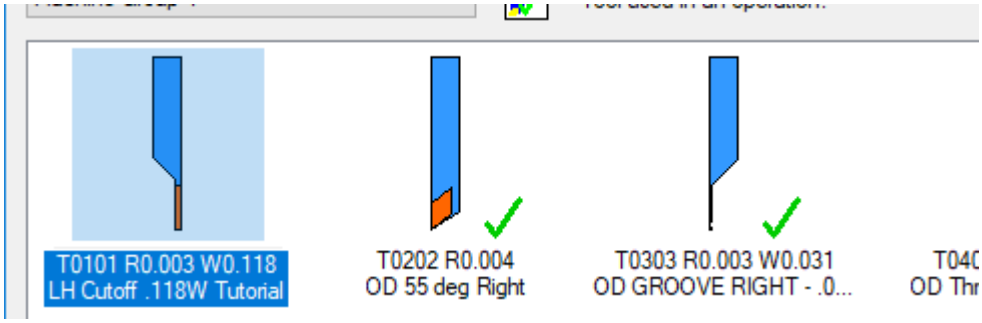


9. The cutoff X coordinate of 0.4 is derived as follows: The stock's diameter is 0.75. The 0.4 is a radius, so it represents a diameter of 0.8, which gives you 0.5 per side above the stock.

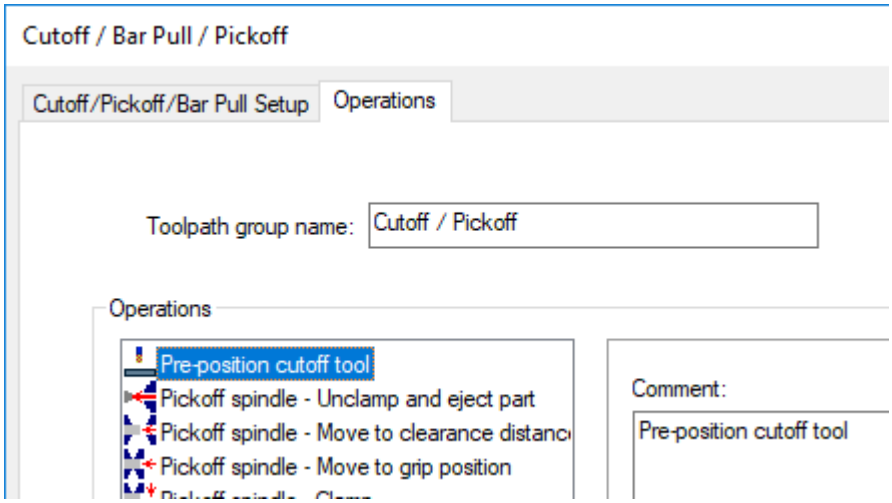
10. When the Tool Manager displays, find the **T0101 R0.003 W0.118 LH Cutoff .118W Tutorial** cutoff tool in the lower pane and drag it into the upper pane.



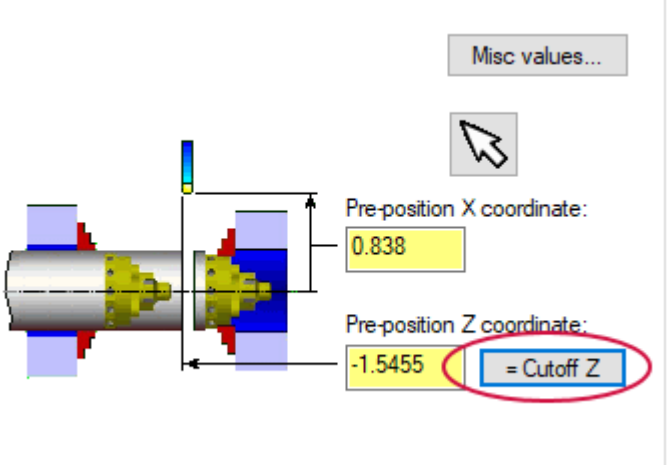
11. Select the tool in the upper pane, and click **OK**.



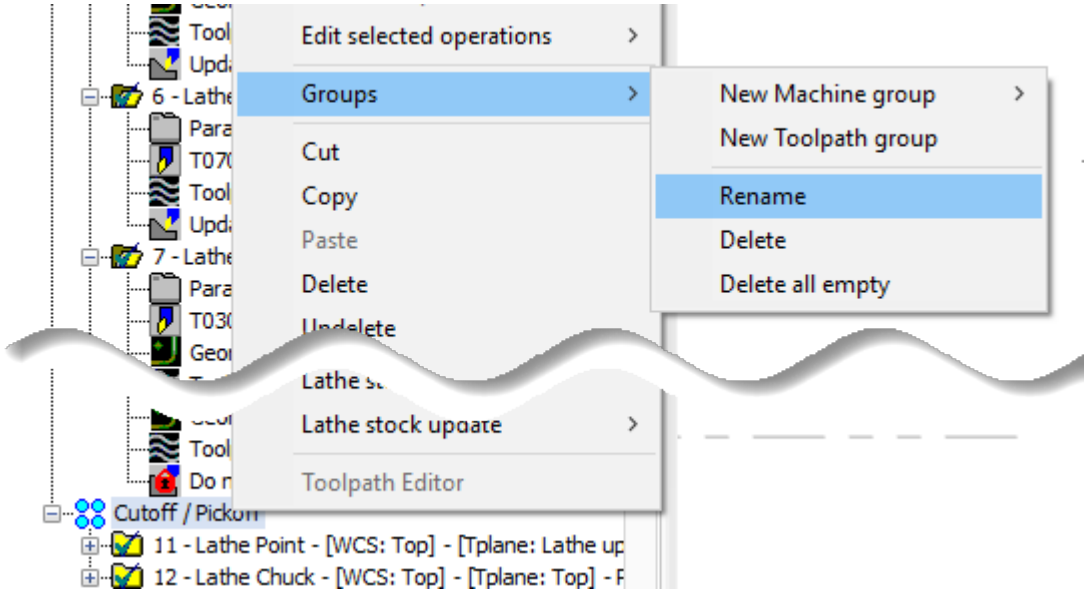
12. In the **Cutoff / Bar Pull / Pickoff** dialog box, select the **Operations** tab, and select **Pre-position cutoff tool**.



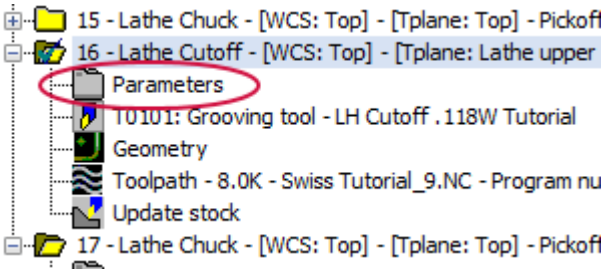
13. Click **= Cutoff Z** to set the **Pre-position Z coordinate** field and click **OK**.



14. Rename the new toolpath group from **Cutoff / Pickoff** to **Cutoff**.

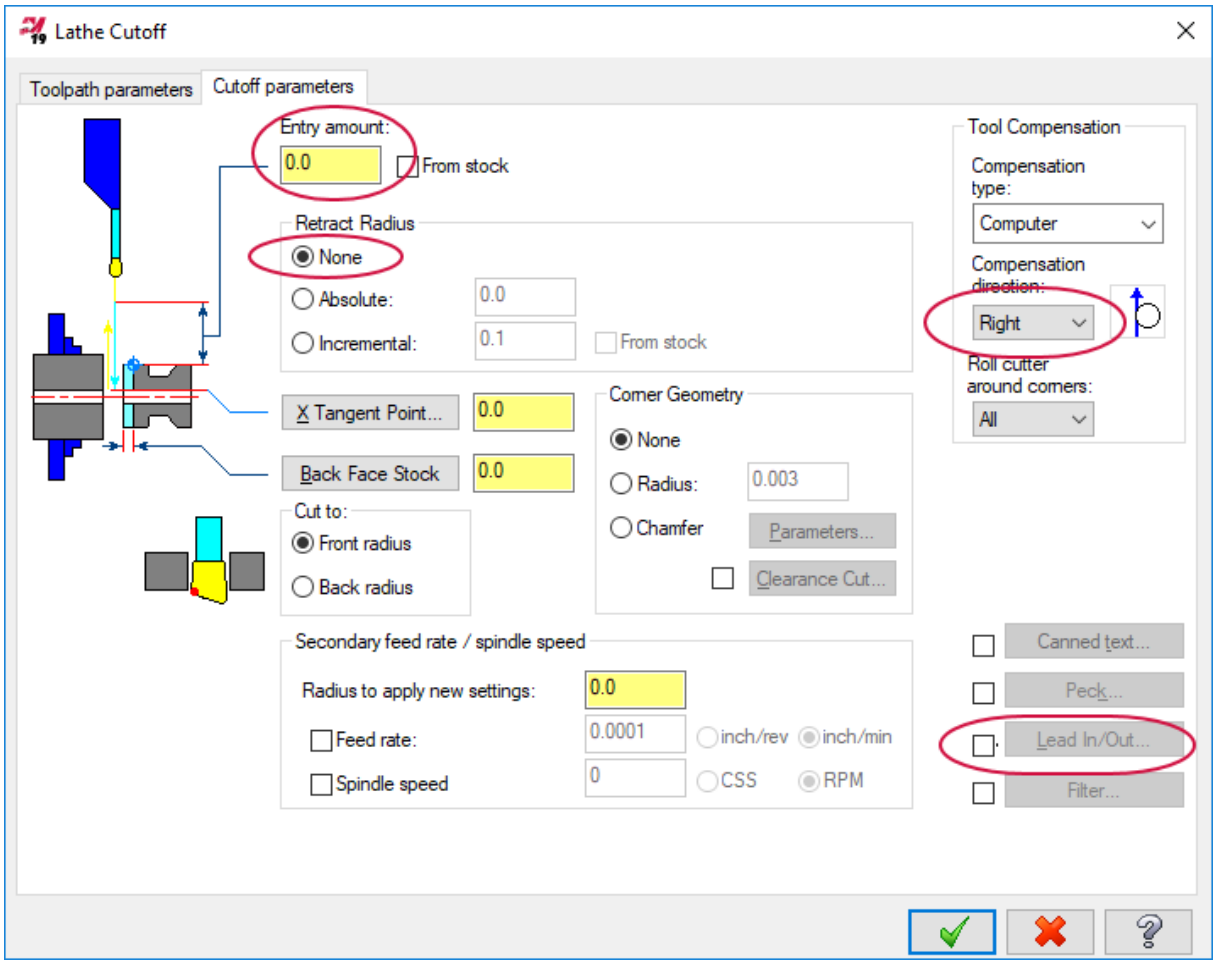


15. In operation 16, click the **Parameters** icon. The **Lathe Cutoff** dialog displays.

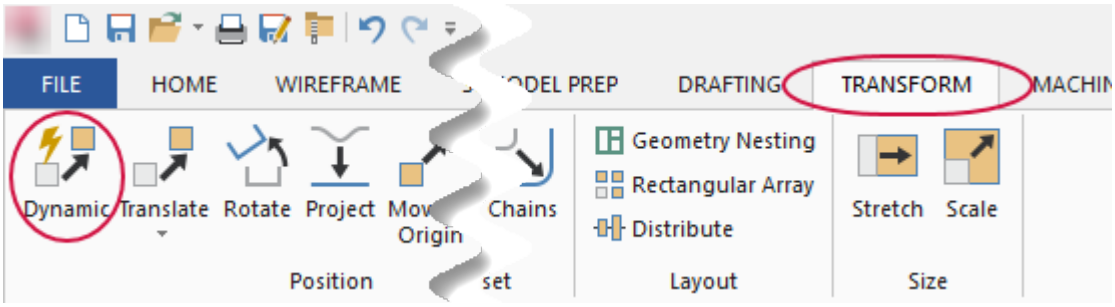


16. On the **Cutoff parameters** tab, make the following changes, and click **OK**.

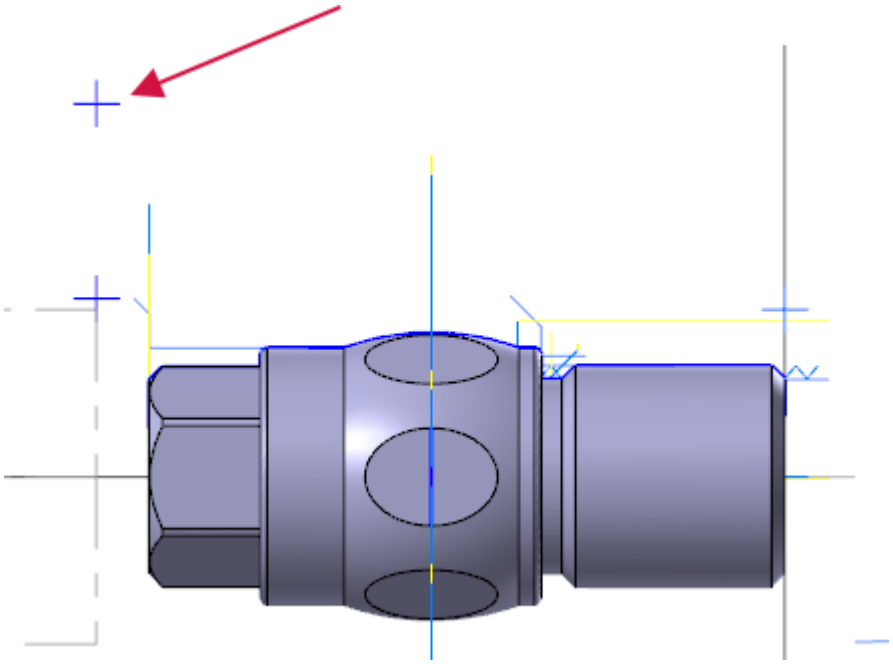
Entry amount: 0
Retract Radius: None
Compensation direction: Right
Lead In/Out: Off



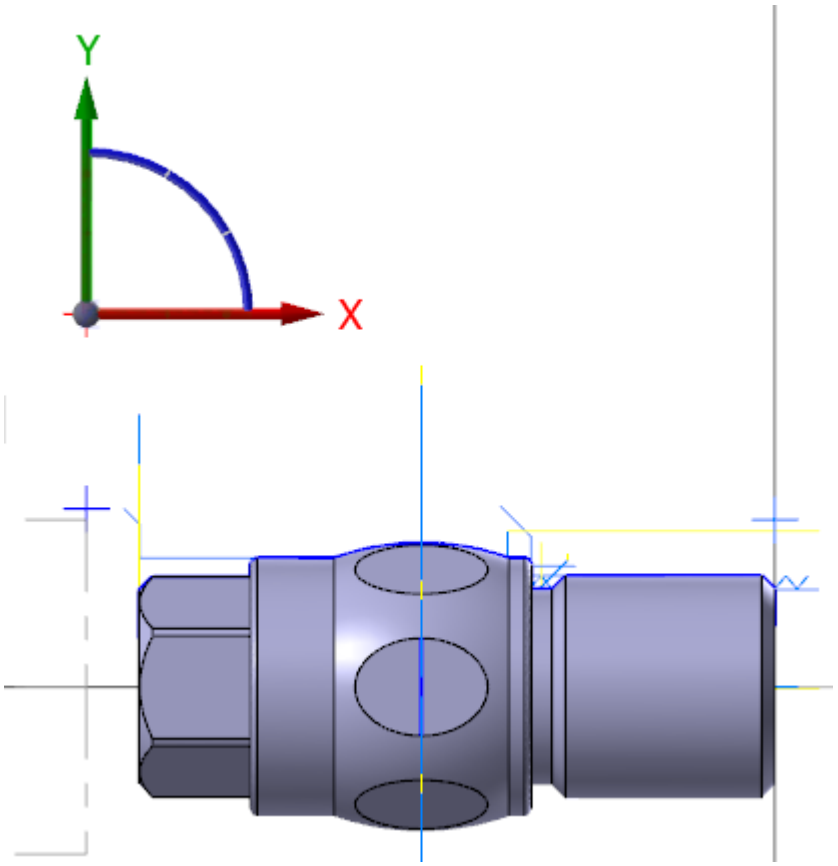
17. On the **Transform** tab, click **Dynamic**.



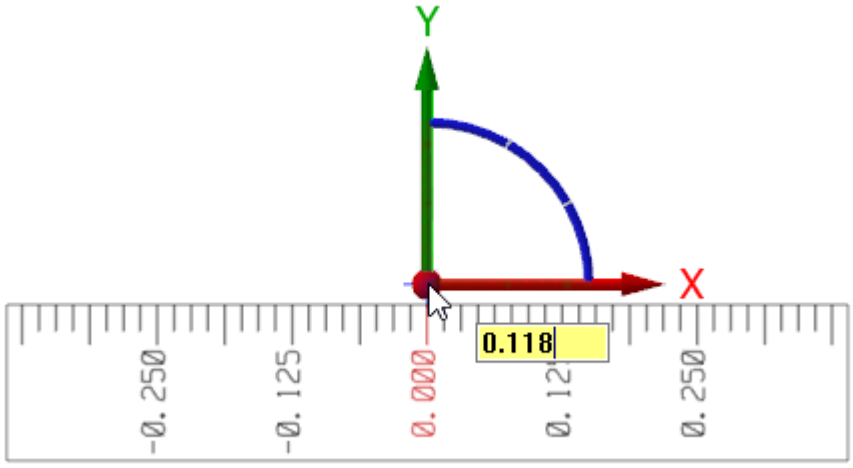
18. Select the top point, as shown in the following image, and click **End Selection**.



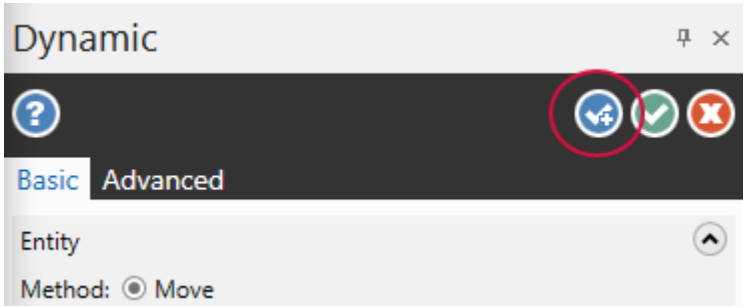
19. Place the dynamic gnomon's origin on top of the point,and click to set its location.



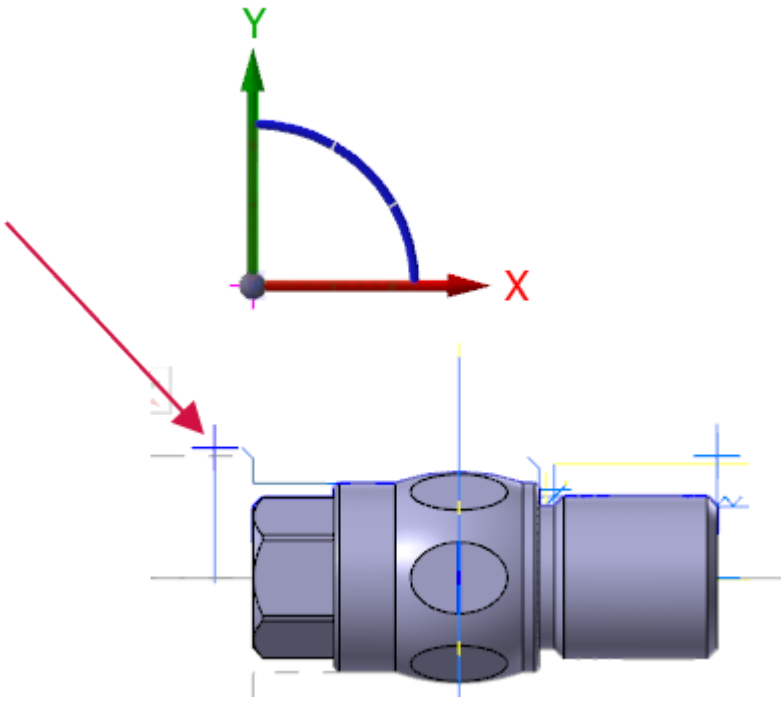
20. Click the gnomon's X axis, type **0.118**, and press [**Enter**]. The gnomon moves 0.118 to the right.



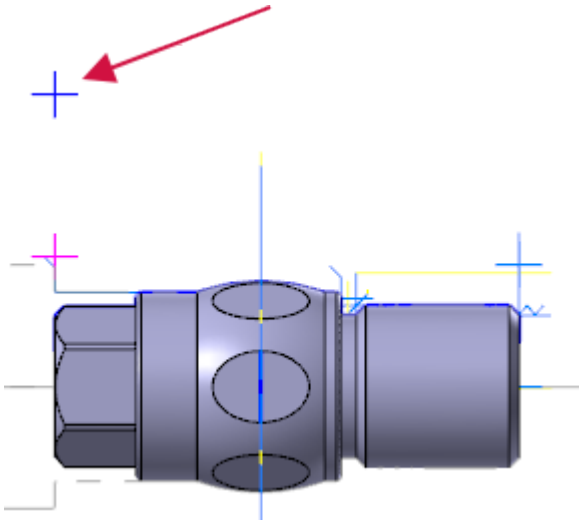
21. Press [**Enter**] a second time to verify the move, and then, in the **Dynamic** function panel, click **OK and Create New Operation**.



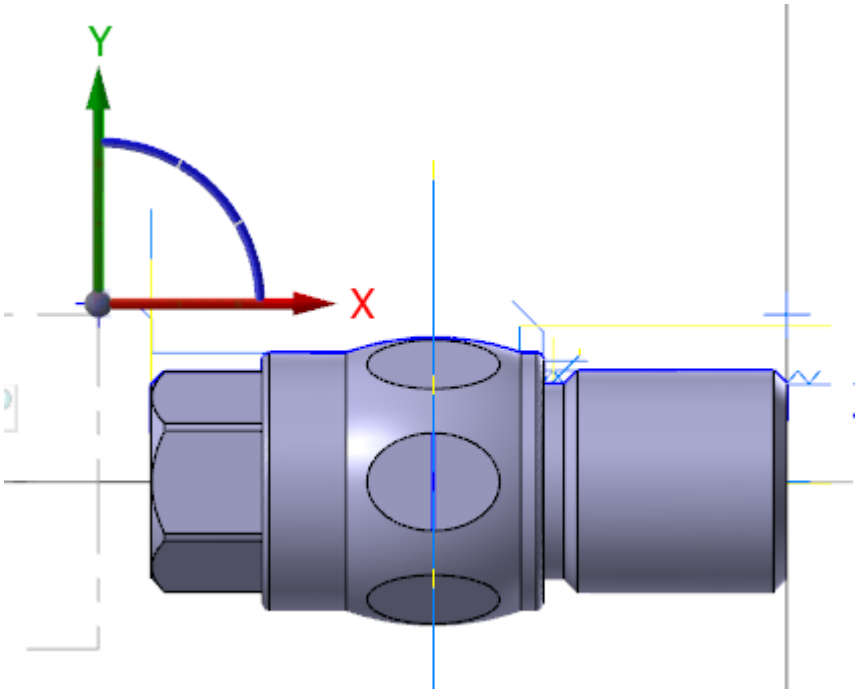
22. Using the same method (Steps 18 to 20), move the lower point, indicated in the following image, to the right **0.118** inches. Make sure to end the function by clicking **OK and Create New Operation**.



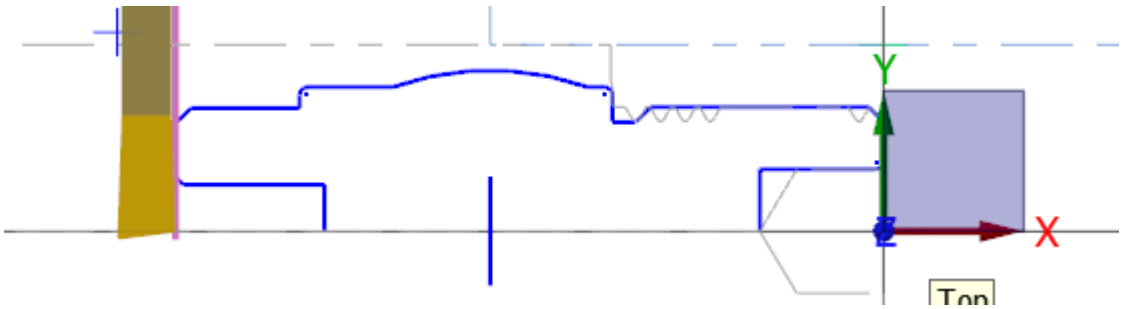
23. Select the top point, as shown in the following image, and click **End Selection**.



24. Place the dynamic gnomon on top of the point, and click to set its location.
25. Drag the ball at the gnomon's origin to the lower point and click. Then click **OK** in the **Dynamic** function panel.



26. Regenerate operations 11 through 17, and backplot the **Cutoff** group operations to see the cutoff in action.



27. Save the file with the name `Swiss Tutorial_XX_10.mcam`, where *XX* is your initials.

CHAPTER 3

SUB SPINDLE OPERATIONS

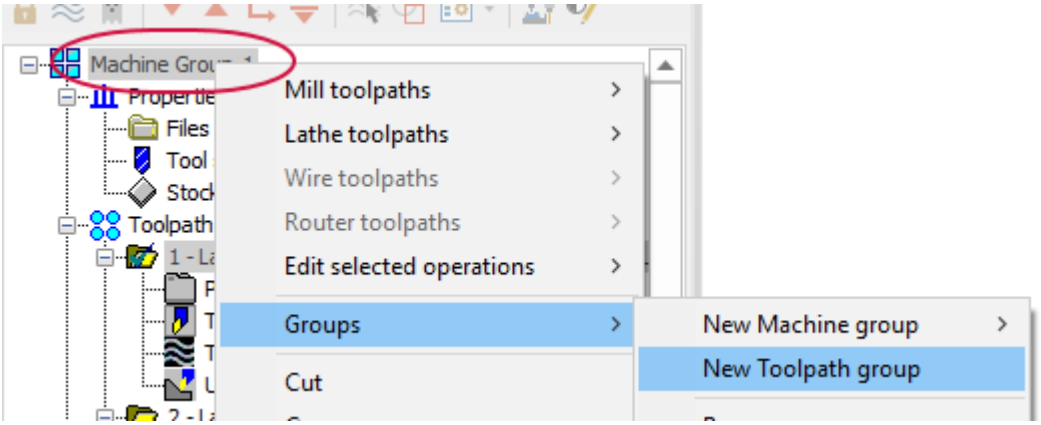
In this chapter, you create the sub spindle operations, which completes the part. These final tasks include not only programming the required toolpaths, but also setting up a new toolpath group and reordering the operations as required by Mastercam Swiss.

Goals

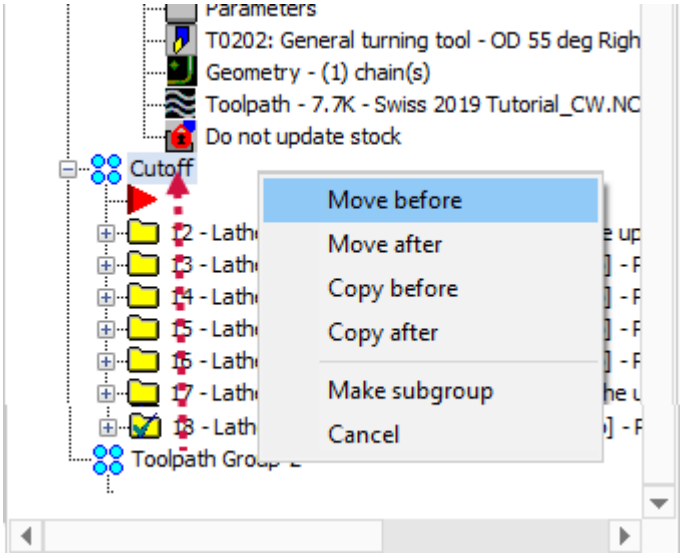
- Create a new toolpath group.
- Create drilling operations.
- Create a boring operation.
- Create a polar milling operation.

Exercise 1: Create a sub spindle group

1. In the Toolpaths Manager, right-click **Machine Group-1**, and then choose **Groups, New Toolpath Group** from the pop-up menu.



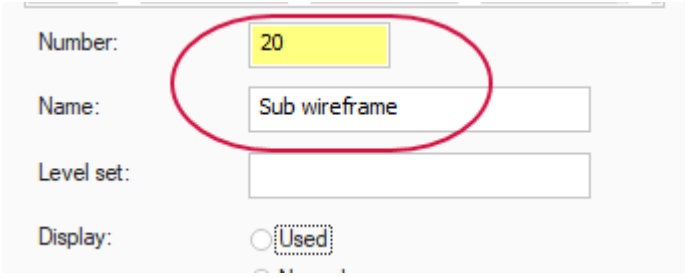
2. Drag the new **Toolpath Group-2** on top of the **Cutoff** group, and select **Move before** from the pop-up menu.



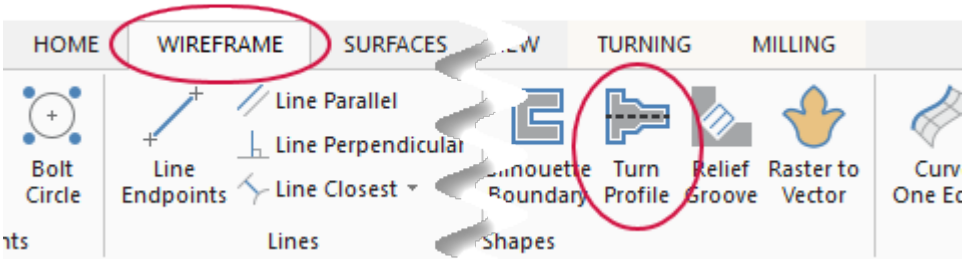
On a Swiss machine, the cutoff must be the last operation, and the order of the toolpaths in Toolpaths Manager must reflect this requirement. However, you could not create this operation before you had created the cutoff, which is why the new toolpath group must be moved now.

This operation syncs head 1 and head 2 so that both programs start simultaneously.

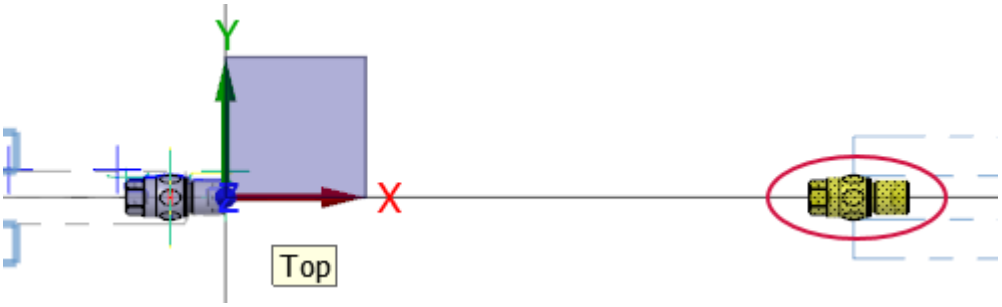
3. In the Levels Manager, create a new level **20** named **Sub wireframe**.



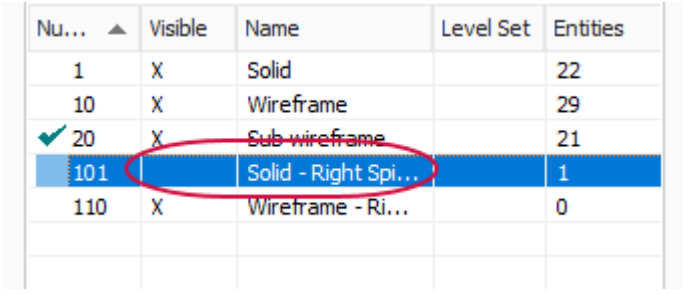
4. On the **Wireframe** tab, choose **Turn Profile**.



5. Select the part in the subspindle (be sure not to select the version of the part on the main spindle), press [**Enter**], and click **OK** in the **Turn Profile** function panel.



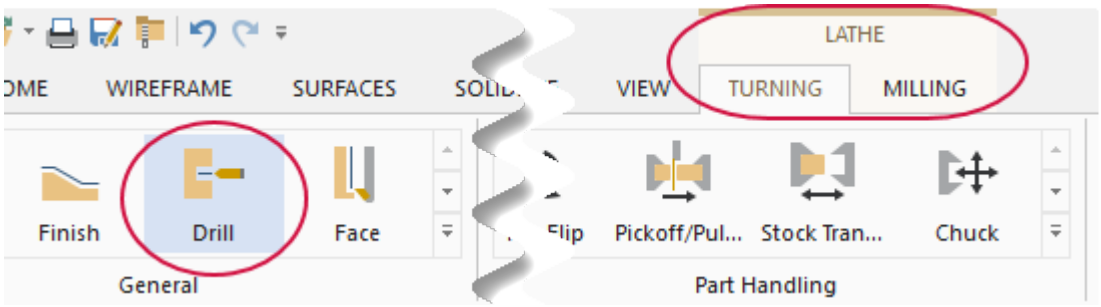
6. In the Levels Manager, turn off visibility for **Solid - Right Spindle**.



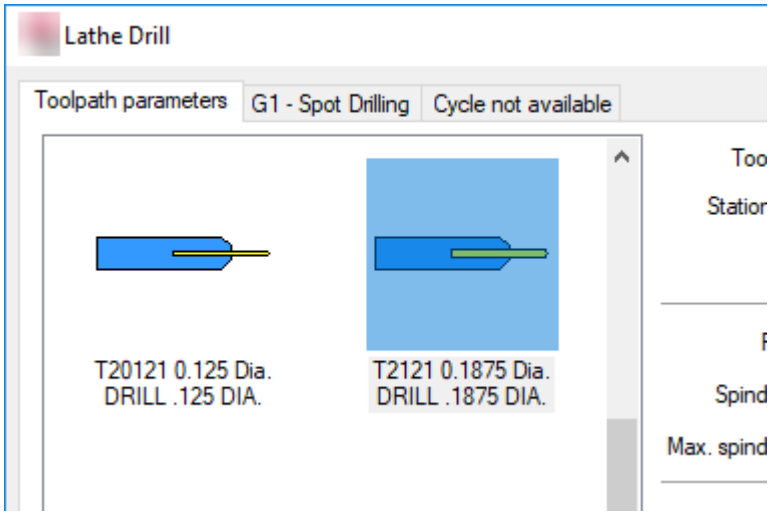
7. Save the file with the name `Swiss Tutorial_XX_11.mcam`, where *XX* is your initials.

Exercise 2: First sub spindle drill operation

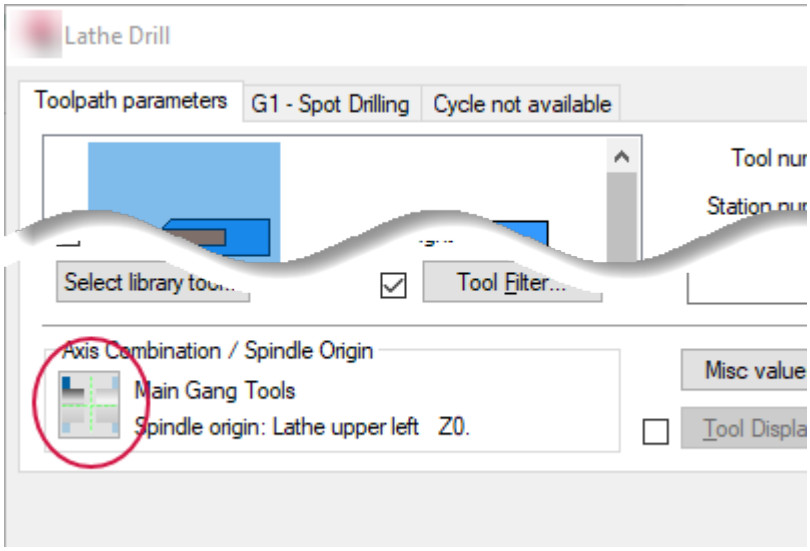
1. Select **Drill** from the **Lathe Turning** contextual tab.



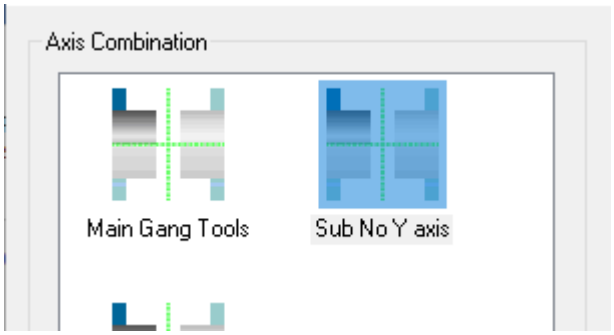
2. In the **Lathe Drill** dialog box, choose tool **T2121 0.1875**. Several tools start with T2121, so be sure you pick the correct one.



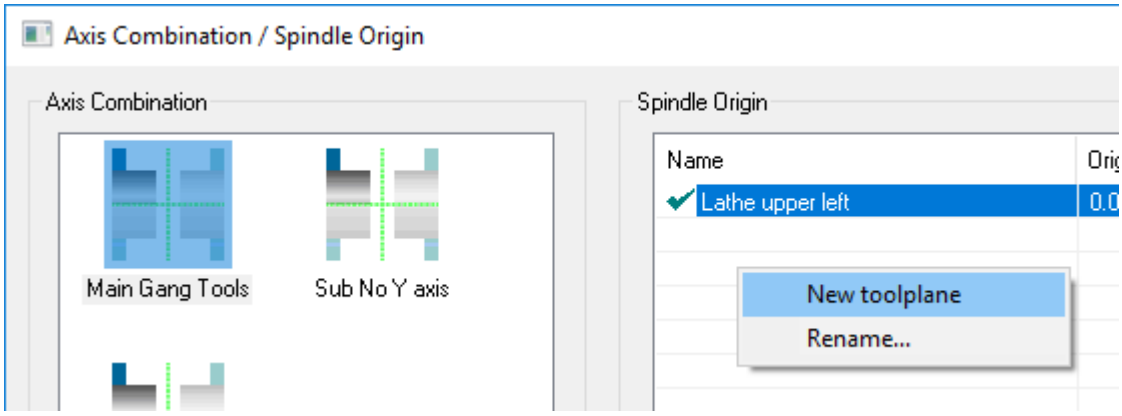
3. Click the **Axis Combination** button.



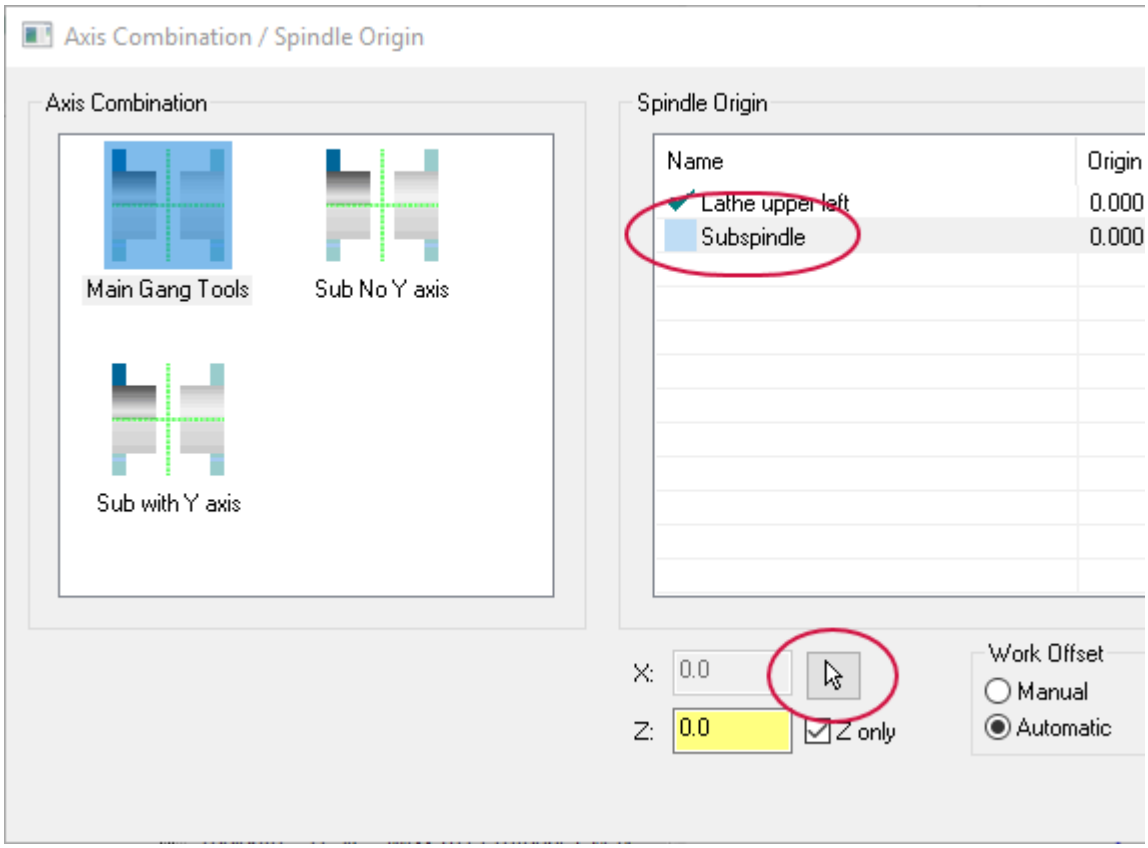
4. The machine you are using for this tutorial has no Y axis, so ensure that the **Sub No Y Axis** axis combination is selected.



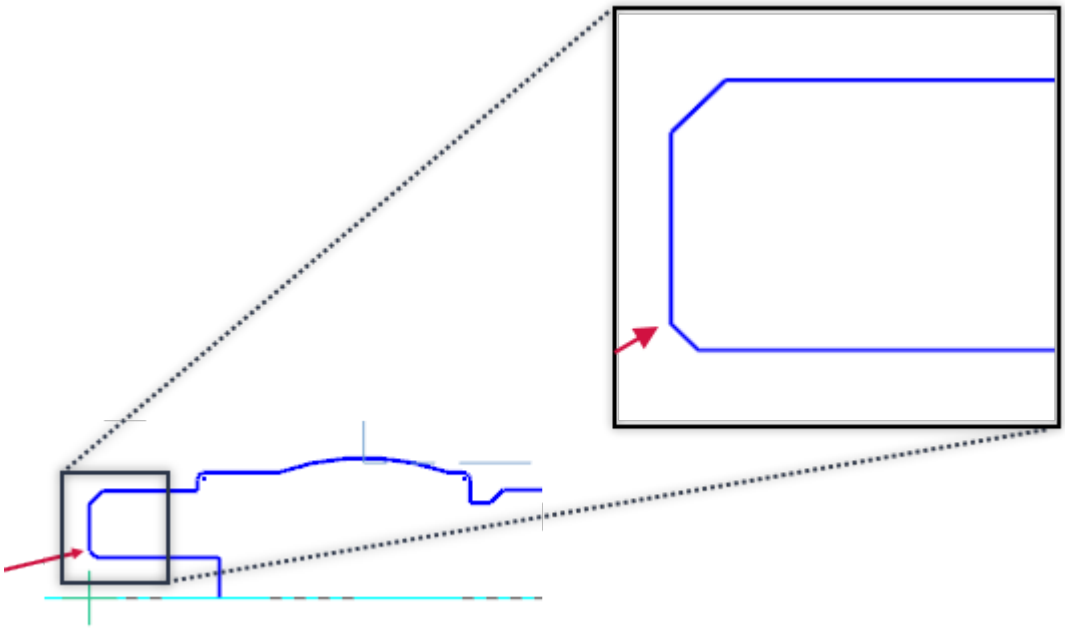
5. Right-click in the **Spindle Origin** table, and choose **New toolplane** from the pop-up menu.



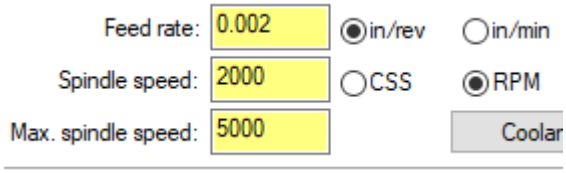
6. Name the new toolplane **Subspindle**, and then click the **Select** button, as shown in the following image.



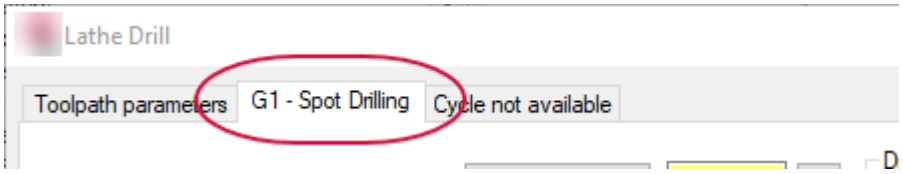
7. Select the 0 point of the part on the subspindle, as shown in the following picture, and click **OK** in the **Axis Combination / Spindle Origin** dialog box.



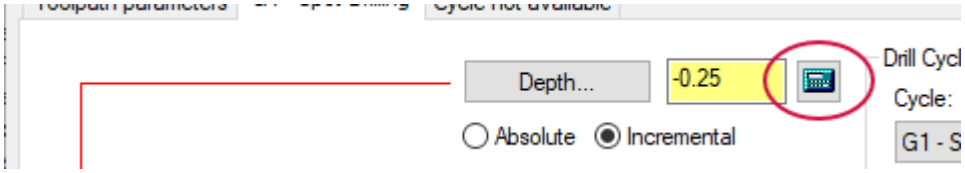
8. If necessary, change **Feed rate** to **0.002**, **Spindle speed** to **2000**, and **Max spindle speed** to **5000**.



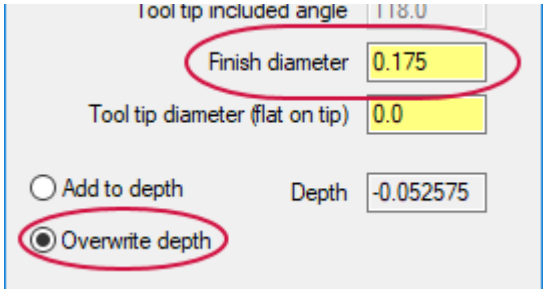
9. Select the **G1 - Spot Drilling** tab.



10. Click the **Depth Calculator** button.

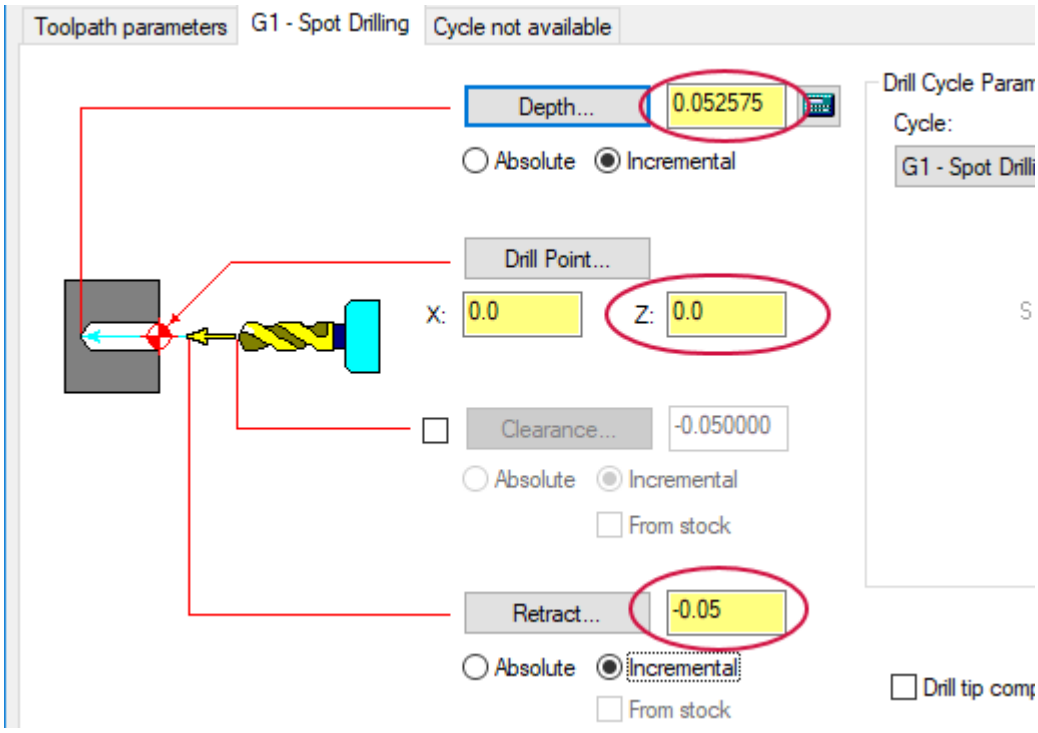


11. In the **Depth Calculator** dialog box, enter **0.175** for **Finish diameter**. Select **Overwrite depth**, and then click **OK**

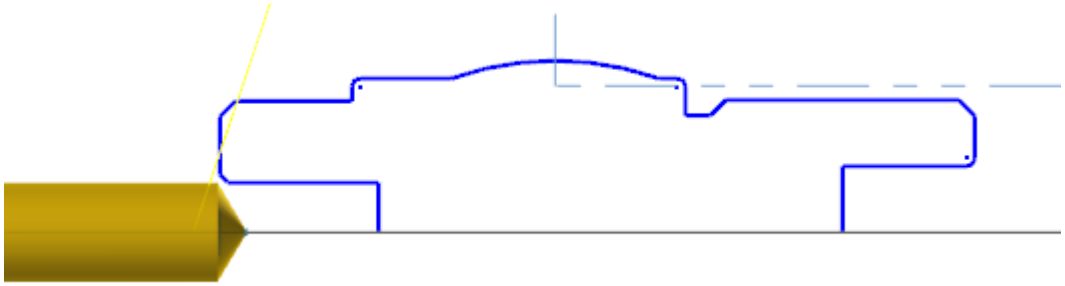


12. In the **G1 - Spot Drilling** tab, remove the negative sign from the **Depth** value, change **Z** to **0**, set **Retract** to **-**

0.05, and click **OK**.



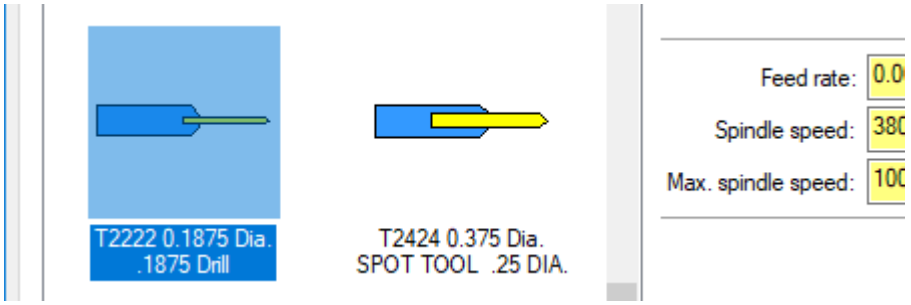
13. Backplot the operation to see the toolpath in motion.



14. Save the file with the name `Swiss Tutorial_XX_12.mcam`, where *XX* is your initials.

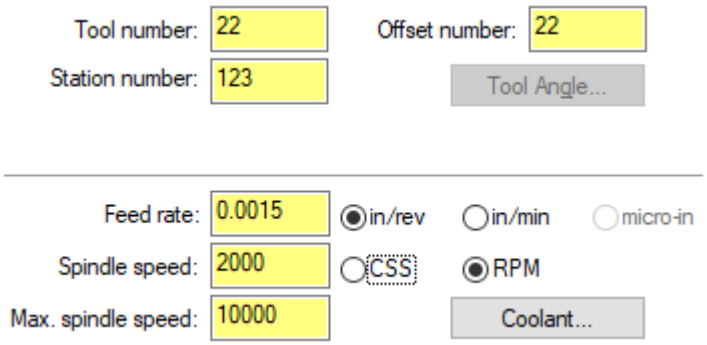
Exercise 3: Creating the second drill operation

1. Select **Drill** from the **Lathe Turning** contextual tab.
2. Choose tool **T2222**.

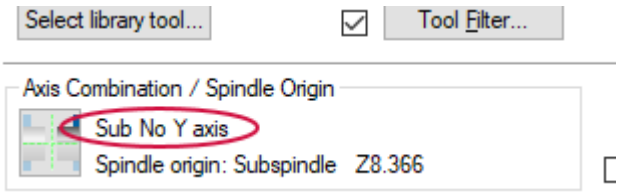


3. Ensure that the following parameters are set correctly.

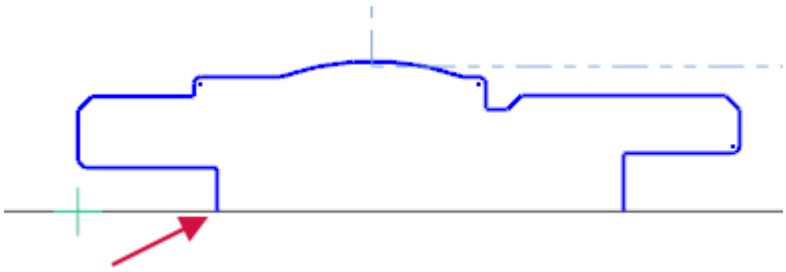
Tool number: 22
Offset number: 22
Station number: 123
Feed rate: 0.0015
Spindle speed: 2000
Max. spindle speed: 10000



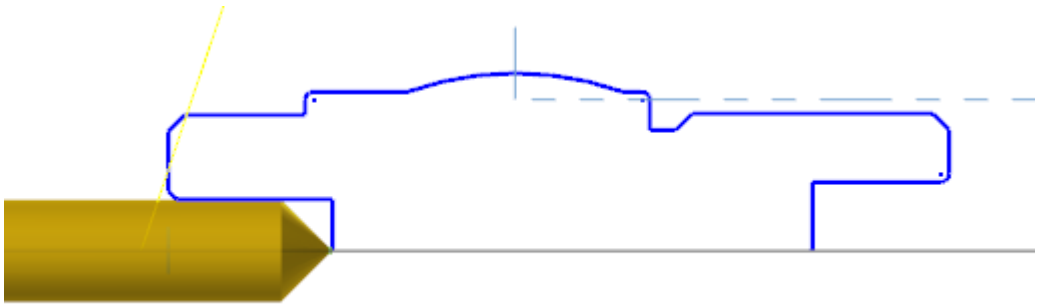
4. On the **Toolpath Parameters** page, verify that **Sub No Y axis** is selected.



5. On the **G1 - Spot Drilling** tab, click the **Depth** button, choose the point shown in the following picture, and then click **OK**.



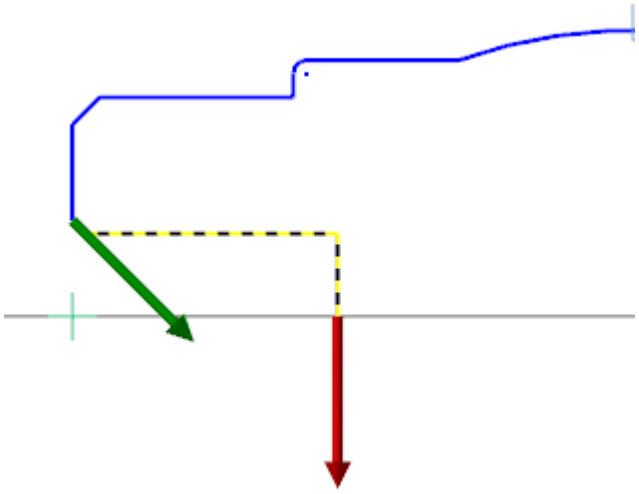
6. Use Backplot to test the toolpath.



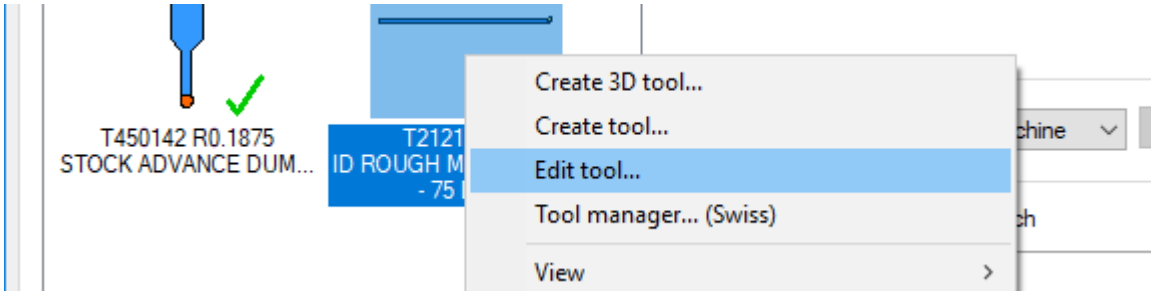
7. Save the file with the name `Swiss Tutorial_XX_13.mcam`, where *XX* is your initials.

Exercise 4: Finish operation for boring

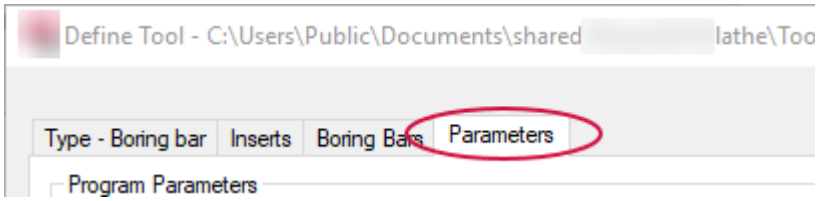
1. Select **Finish** from the **Lathe Turning** contextual tab.
2. Chain the part as shown below, and click **OK**.



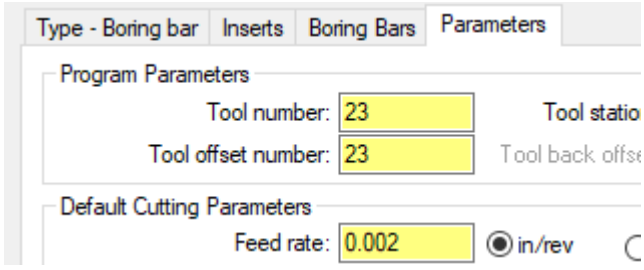
3. Right-click tool **T212121**, and choose **Edit tool** from the pop-up menu.



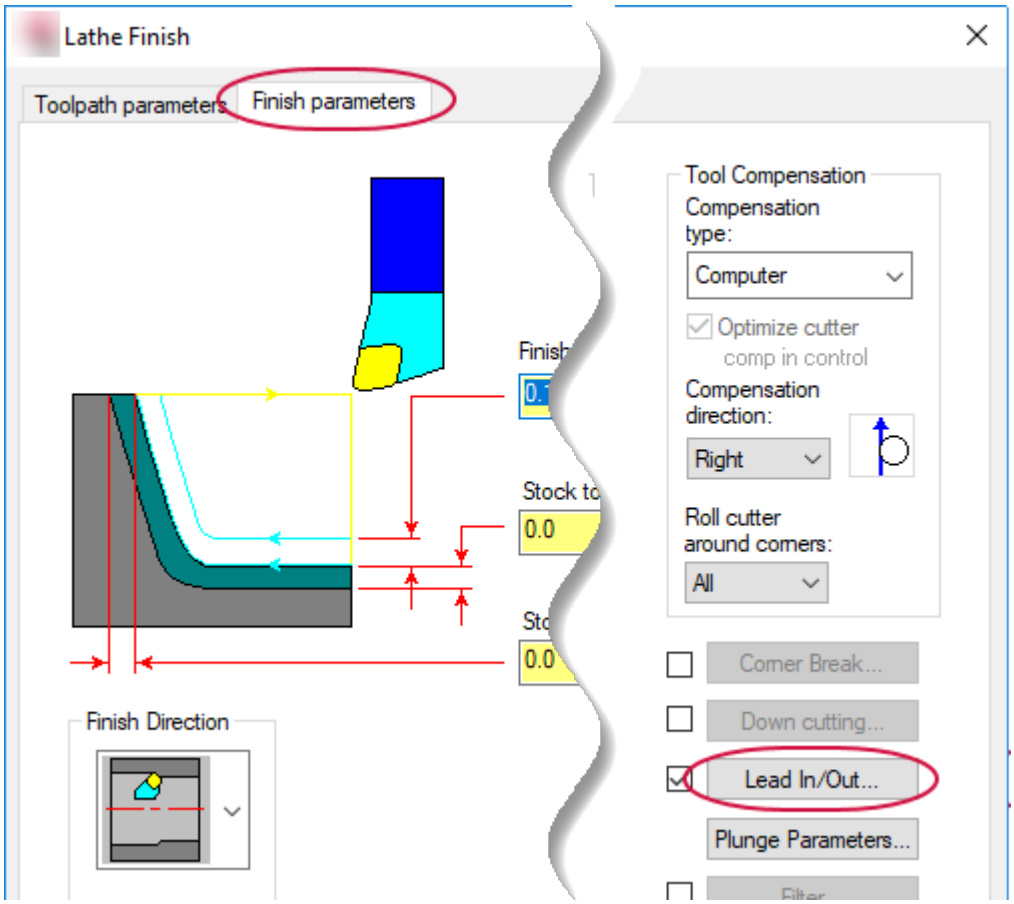
4. In the **Define Tool** dialog box, select the **Parameters** tab.



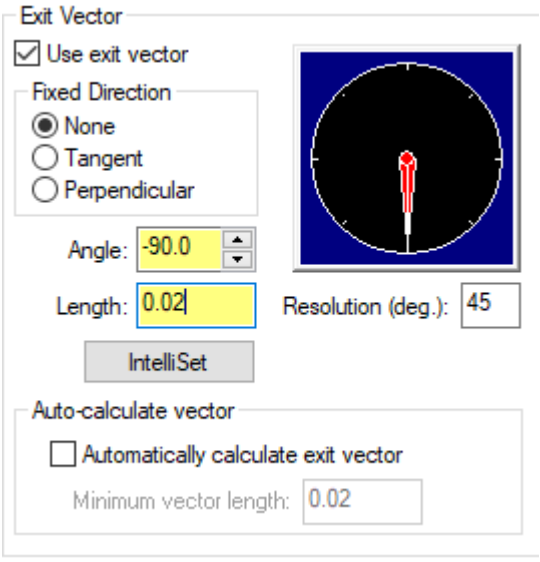
5. Set **Tool number** and **Tool offset** number to **23**, and click **OK**.



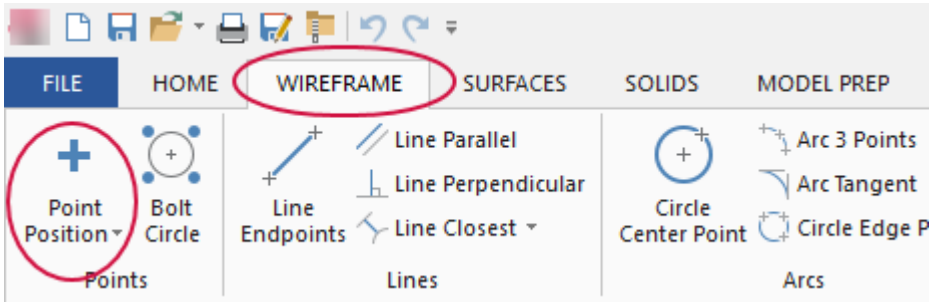
6. Select the **Finish parameters** tab, and click **Lead in/out**.



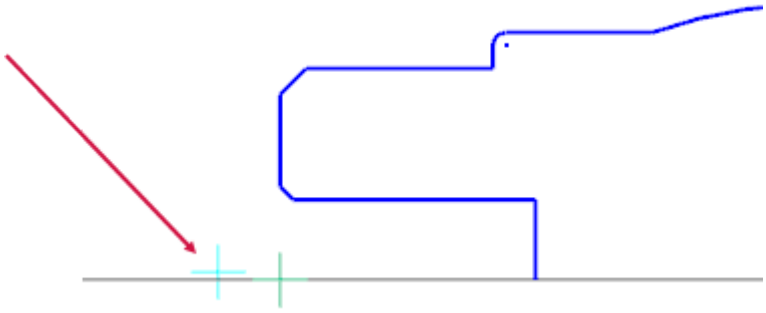
7. Select the **Lead out** tab, set **Angle** to **-90.0** and **Length** to **0.02** as shown in the following picture.



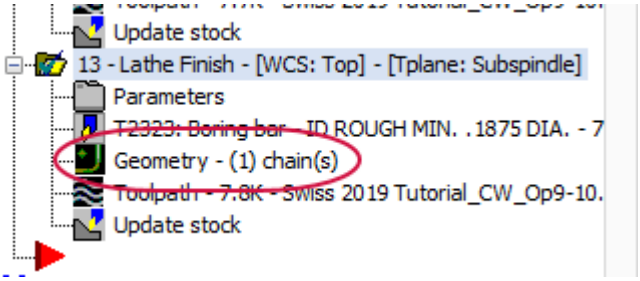
8. Click **OK** in the **Lead In/Out** page and in the **Lathe Finish** dialog box.
9. Select the **Wireframe** tab, and choose **Point Position**.



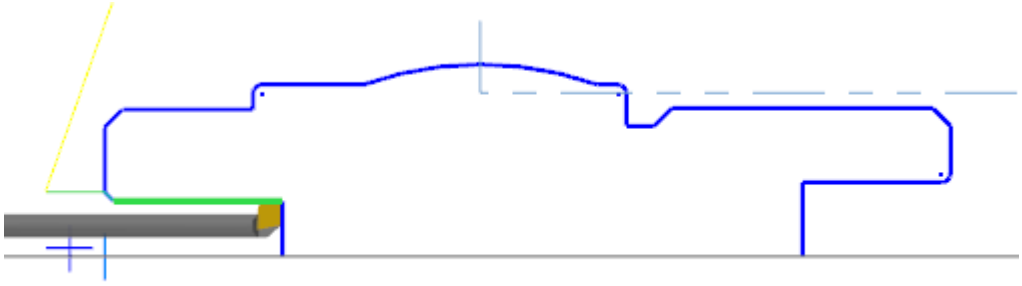
10. Create a point similar to the one shown in the following picture (in other words, your choice need not exactly match the one shown), and click **OK** in the **Point Position** function panel. Note that the point you select must be above the centerline of the part and beyond the face of the part.



11. In the Toolpaths Manager, select **Geometry** under operation 13 to display the **Chain Manager** dialog box.



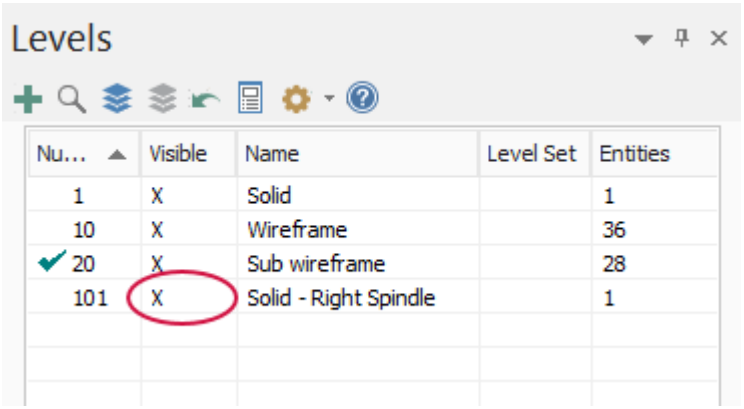
12. Right-click in the **Chain Manager** dialog box, and choose **Add chain** from the pop-up menu.
13. Select the new point from Step 10, and click **OK** in both the **Chaining** and **Chain Manager** dialog boxes.
14. Regenerate all operations and backplot operation 13 to see it in action.



15. Save the file with the name `Swiss Tutorial_XX_14.mcam`, where *XX* is your initials.

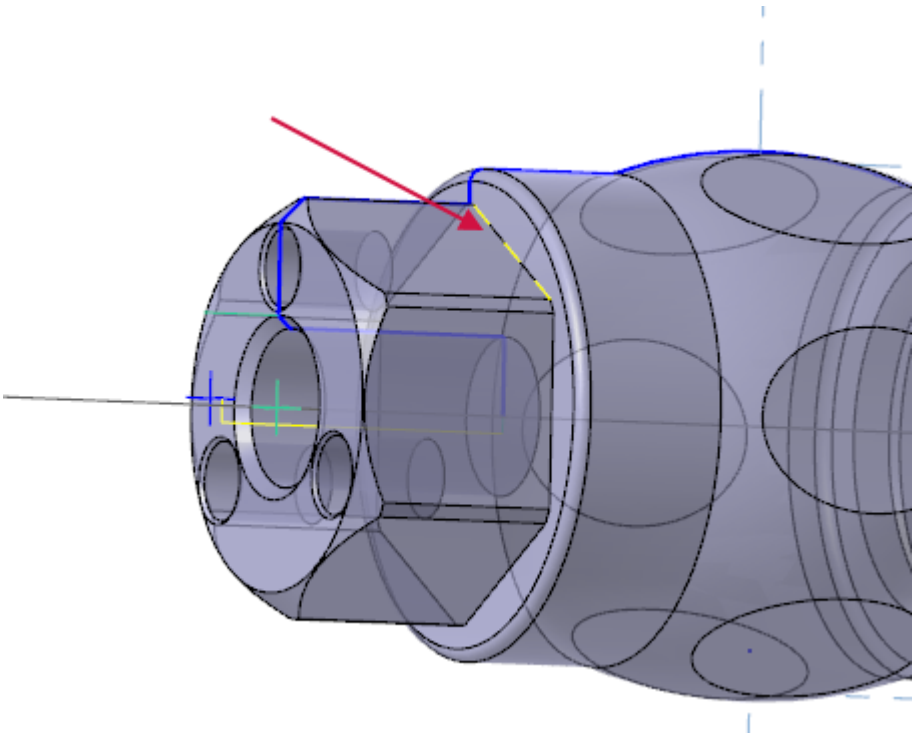
Exercise 5: Polar milling

1. Turn on the solid's visibility (level 101) in Levels Manager.



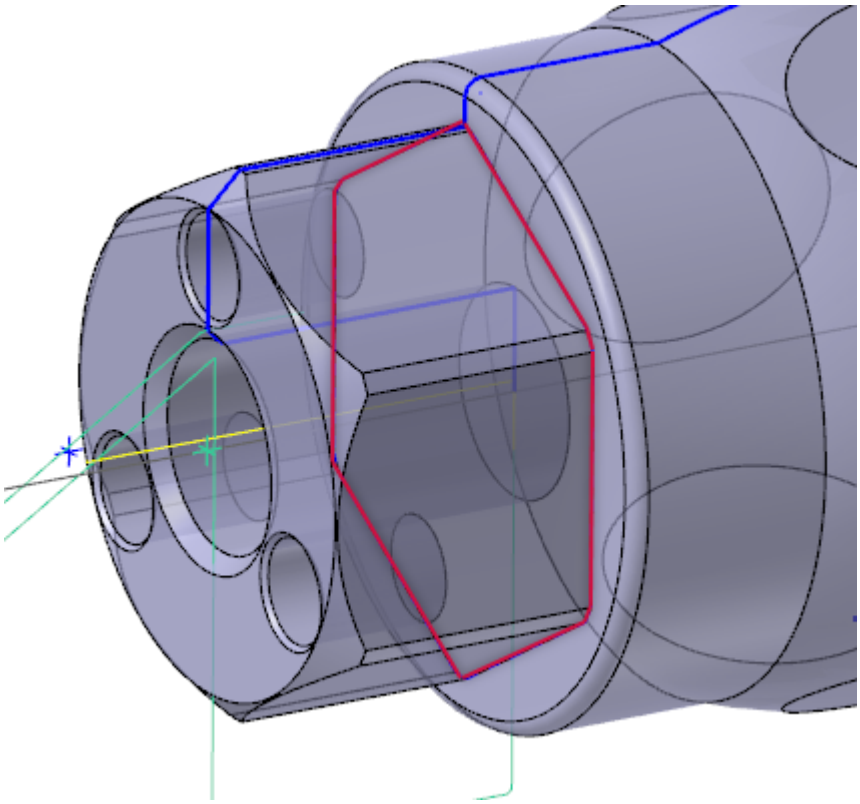
2. Select **Contour** from the **Lathe Milling** contextual tab.

3. In the **Chaining** dialog box, select the **Solids** mode. Ensure that the **Face** option is off, and that the **Loop** option is on.
4. Chain the part as shown in the following image.

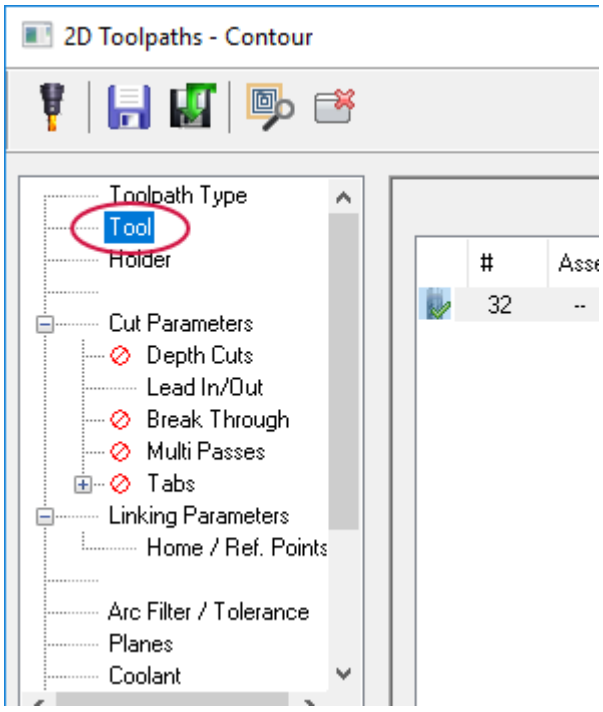


5. If necessary, in the **Pick Reference Face** dialog box, click **Other Face**. Your selection should match the

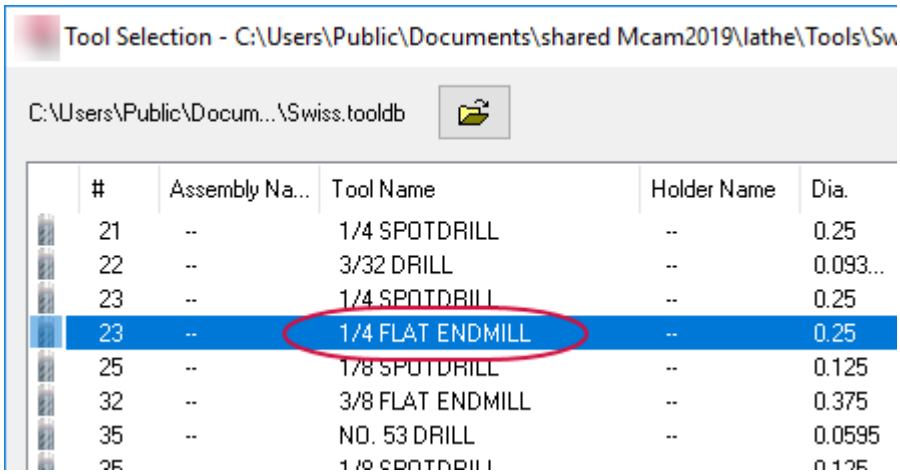
selection shown below in red.



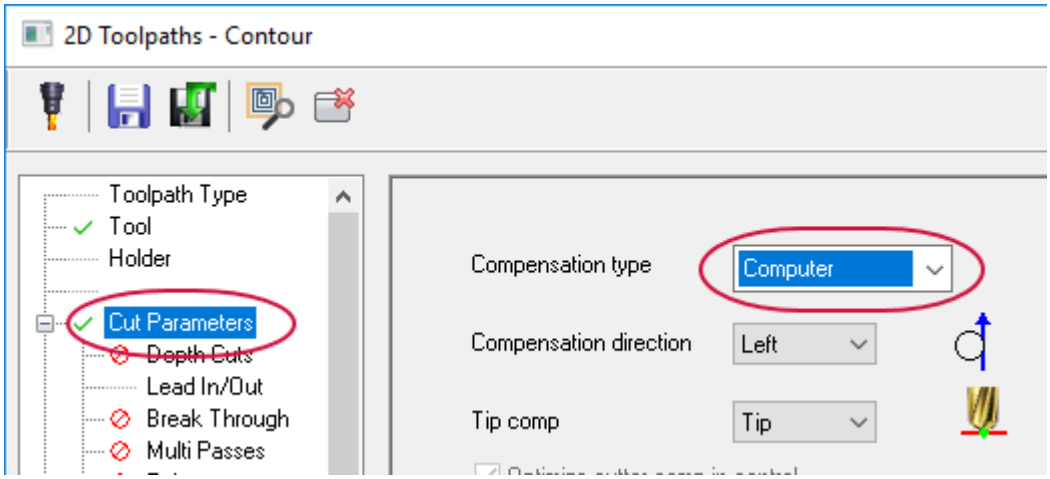
6. Click **OK** in the **Chaining** dialog box.
7. In the **2D Toolpaths - Contour** dialog box, select the **Tool** page.



8. Click **Select library tool**, choose the **1/4 flat endmill** from the **Tool Selection** dialog box, and click **OK**.

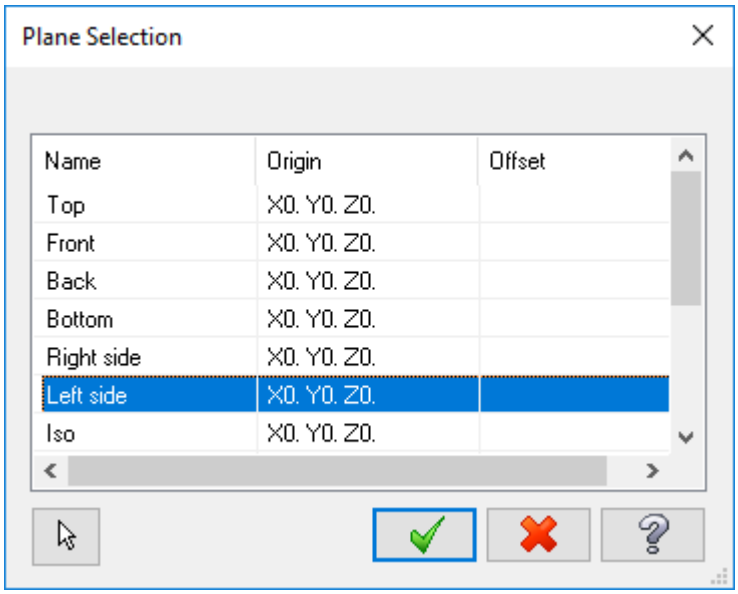


9. On the **Cut Parameters** page, change **Compensation type** to **Computer**.

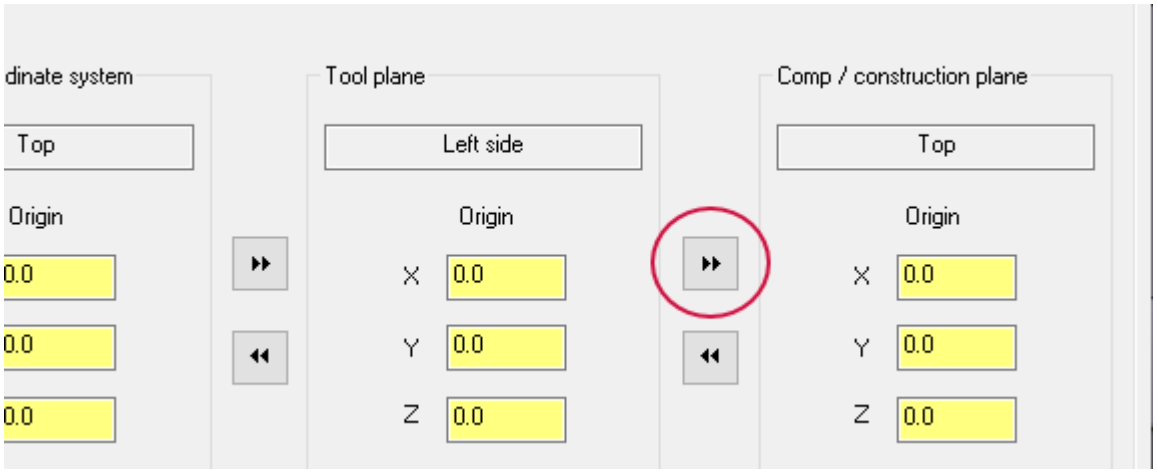


10. On the **Planes** page, click the **Select tool plane** button.

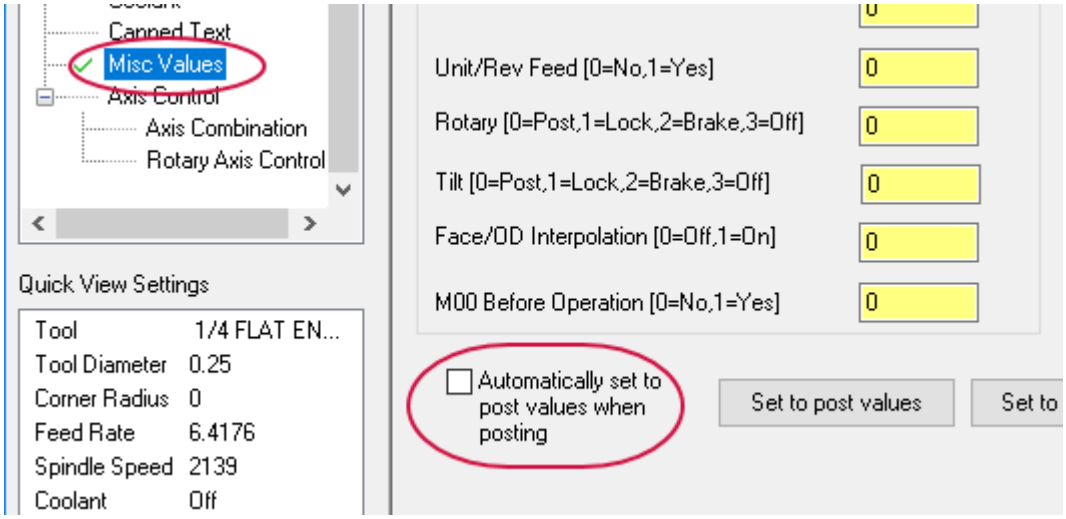
11. In the **Plane Selection** dialog box, choose **Left side**, and click **OK**.



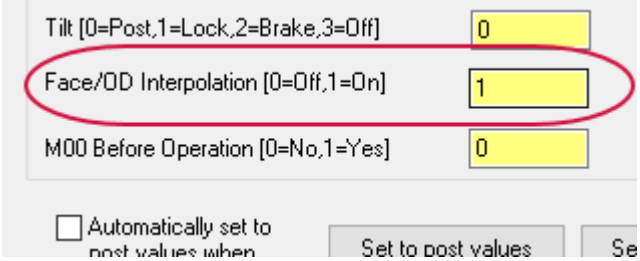
12. Click the **Copy to construction plane** button to copy the tool plane parameters to **Comp / construction plane**.



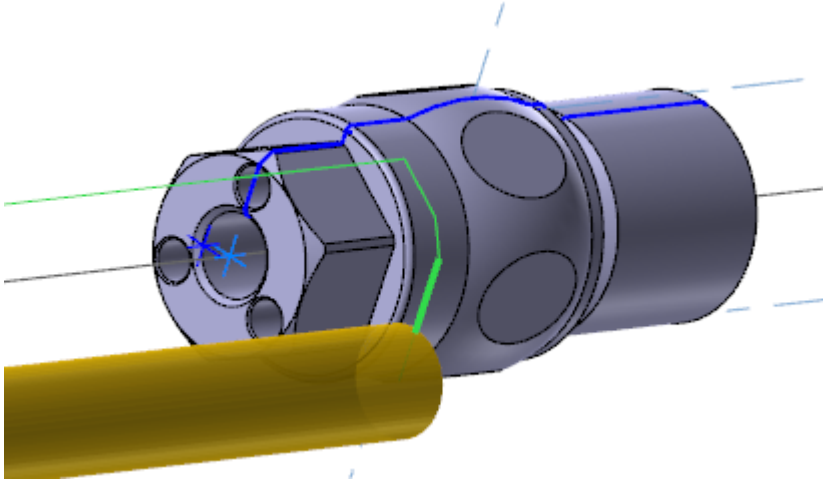
13. Select the **Misc values** page, and then deselect **Automatically set to post values when posting**.



14. Change **Face/OD Interpolation** to **1**, and click **OK**.



15. Backplot the operation to see the toolpath motion.



16. Save the file with the name `Swiss Tutorial_XX_15.mcam`, where *XX* is your initials.

CONCLUSION

Congratulations! You have completed the *Mastercam Lathe for Swiss*! Now that you have mastered the skills in this tutorial, explore Mastercam's other features and functions.

You may be interested in other tutorials that we offer. Mastercam tutorials are being constantly developed, and we will add more as we complete them. Visit our website, or select **Help**, **Tutorials** from the **File** tab.

Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Documentation*—Mastercam installs a number of helpful documents for your version of software in the Documentation folder of your Mastercam 2019 installation.
- *Mastercam Help*—Access Mastercam Help by selecting **Help**, **Contents** from Mastercam's **File** tab or by pressing **[Alt+H]** on your keyboard.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—Our Technical Support department (+1 860-875-5006 or support@mastercam.com) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- *Mastercam Tutorials*—We offer a series of tutorials to help registered users become familiar with basic Mastercam features and functions. Visit our website, or select **Help**, **Tutorials** from Mastercam's **File** tab to see the latest publications.
- *Mastercam University*—Mastercam University, an affordable online learning platform, gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master skills at your own pace and help prepare for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit university.mastercam.com/, or email training@mastercam.com.
- *Online Communities*—You can find a wealth of information at www.mastercam.com.
  - Follow us on Facebook (www.facebook.com/Mastercam), Twitter (twitter.com/Mastercam), and Instagram (www.instagram.com/mastercamcadcam/) for the latest tech tips and Mastercam news.
  - See Mastercam in action on YouTube (www.youtube.com/user/MastercamCadCam).
  - For more information on CNC Software, Inc., to find and apply to jobs, and connect with people using Mastercam, visit us on LinkedIn (www.linkedin.com/company/cnc-software/).
  - Registered users can search for information or ask questions on the Mastercam Web forum, forum.mastercam.com, or use the Mastercam Knowledgebase at kb.mastercam.com.

Contact Us

For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at techdocs@mastercam.com.

ATTENTION! UPDATES MAY BE AVAILABLE. PLEASE REFER TO MASTERCAM.COM/SUPPORT FOR THE LATEST DOWNLOADS.

CNC Software, Inc.
671 Old Post Road
Tolland, CT 06084

www.mastercam.com