



MASTERCAM 2021 LATHE FOR SWISS

January 2021

MASTERCAM 2021 LATHE FOR SWISS

January 2021

© 2021 CNC Software, Inc. – All rights reserved.

Software: Mastercam 2021

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	5
Prerequisites	5
General Tutorial Requirements	5
Setting Up the Job	7
Exercise 1: Loading the machine	7
Exercise 2: Loading the tool library	8
Exercise 3: Completing stock setup	10
Exercise 4: Creating the turn profile	17
Exercise 5: The stock advance operation	19
Left Spindle Operations	23
Exercise 1: Programming the facing operation	23
Exercise 2: Spot drilling the front hole	25
Exercise 3: Drilling the front hole	28
Exercise 4: Creating the Finish Operation	30
Exercise 5: Finishing the front side	34
Exercise 6: Cutting the groove	37
Exercise 7: Cutting the 1/2-20 thread	40
Exercise 8: Finishing the center OD	44
Exercise 9: Cutting the six flat areas	48
Exercise 10: Editing the Cutoff operation	53
Exercise 11: Adding the Cutoff operation	57
Sub Spindle Operations	65
Exercise 1: Creating a sub spindle group	65
Exercise 2: Creating the sub spindle drill operation	67
Exercise 3: Creating the second drill operation	71
Exercise 4: Creating the Finish operation for boring	73
Exercise 5: Polar milling	77
Conclusion	83

Mastercam Resources83

Contact Us83

INTRODUCTION

Welcome to Mastercam Lathe for Swiss. In this tutorial, you use Mastercam Lathe to program the machining for a Swiss part. 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

- Explore the parameters required to set up a Mastercam Swiss job.
- Successfully apply lathe toolpaths to a Swiss part.

Prerequisites

To be successful with this tutorial, you should have experience with Mastercam Lathe and a working Swiss post. You should also have a basic knowledge of Swiss machining.

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 2021 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 accessed from the Mastercam 2021 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.

This tutorial uses inch units of measure.

CHAPTER 1

SETTING UP THE JOB

In this chapter, you set up your machining job to use Mastercam Swiss. The Swiss Solutions combine Mastercam Mill and Mastercam Lathe with an optimized post processor to give you a reliable set of tools to program parts for your Swiss machines.

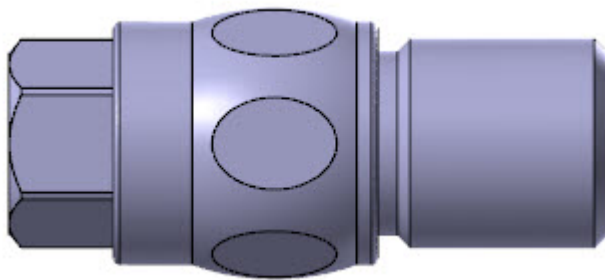
Goals

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

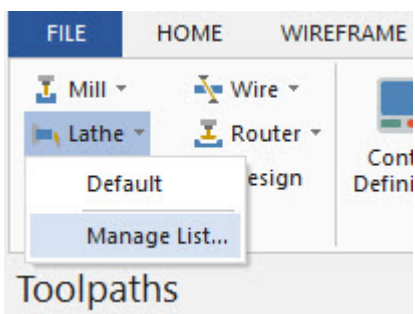
Exercise 1: Loading the machine

In this part of the tutorial, you load the solid model of the part, and load the Swiss machine on which the part will be cut.

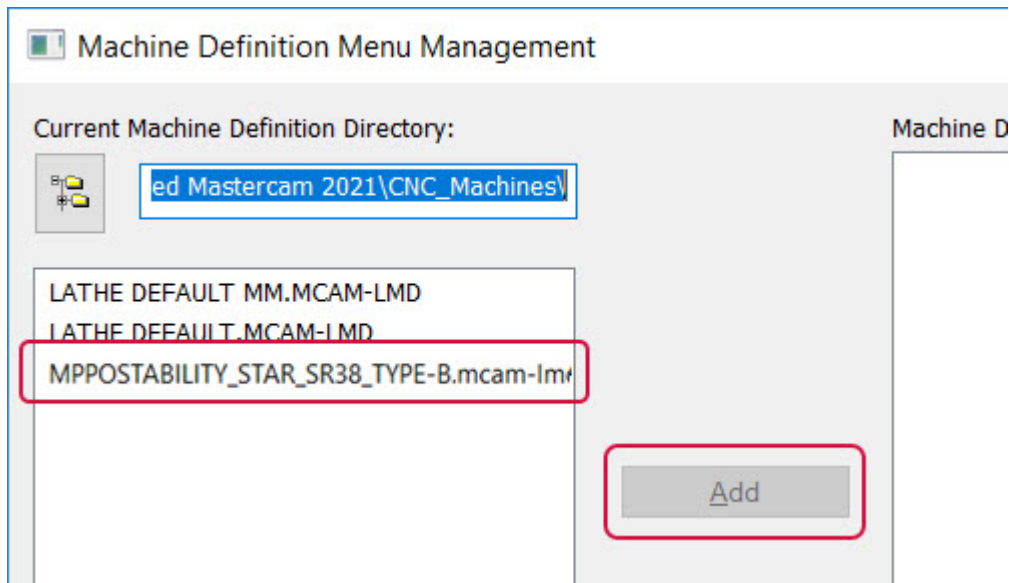
1. Load the part Swiss Tutorial, which is included with this tutorial.



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



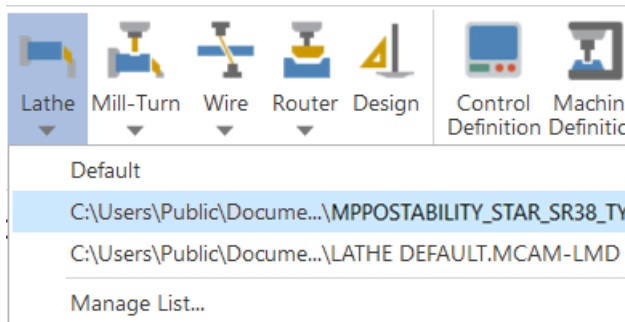
4. Add the MPPOSTABILITY_STAR_SR38_TYPE-B.mcam-lmd machine to the **Machine Definition Menu Items** list.



The MPPOSTABILITY_STAR_SR38_TYPE-B.mcam-lmd machine is not included with this tutorial. You must have a license for this machine to access it.

Note that, as of Mastercam 2021, all Swiss posts have a setting that customizes the improved pickoff-cutoff strategy. For your jobs, be sure to use a post that has been updated with this change.

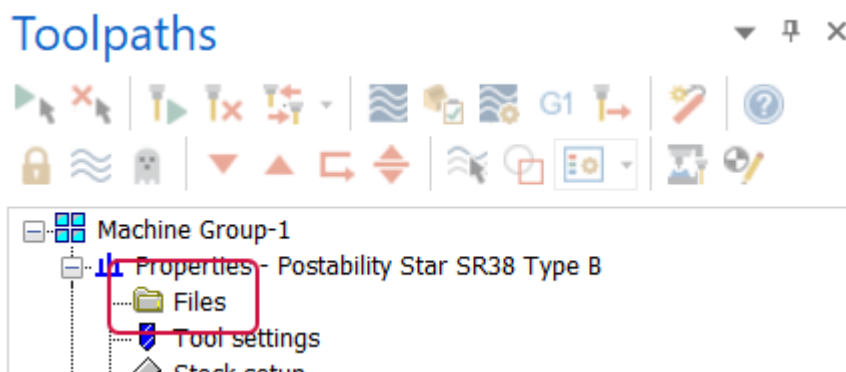
5. Choose the machine from the **Lathe** menu. Mastercam creates a new lathe machine group.



Exercise 2: Loading the tool library

In this exercise, you load the tool library that contains tools that you need to cut this part.

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.

Machine Group Properties [X]



Files | **Tool Settings** | Stock Setup

Group name:

Toolpath directory: 

Group comment:




Machine - Toolpath Copy

Edit  Replace 



Description: Postability Star SR38 Type B
From file: MPPOSTABILITY_STAR_SR38_TYPE-B.mcam-lmd

Control: MPPOSTABILITY_STAR_SR38_TYPE-B.mcam-control
Post: C:\Users\Public\Documents\Share...\MPPOSTABILITY_STAR_SR38_TYPE-B.pst



Tool Library

C:\Users\Public\Documents\Shared Mastercam 20...\Lathe_Inch.tooldb   

Operation Library

C:\Users\Public\Documents\Shared ...\LATHE_INCH.mcam-operations  

Operation Defaults

C:\Users\Public\Documents\Shared Ma...\LATHE_INCH.mcam-defaults  

Output comments to NC file

☒ Output operation comments to NC ☒ Output group name to NC
☒ Output machine name to NC ☒ Output group comments to NC

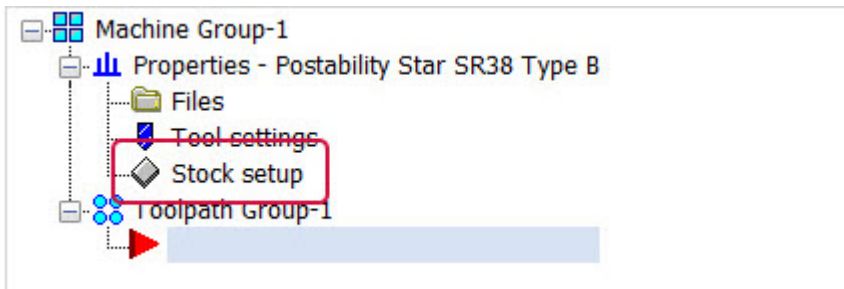
[OK] [Cancel] [Help]

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

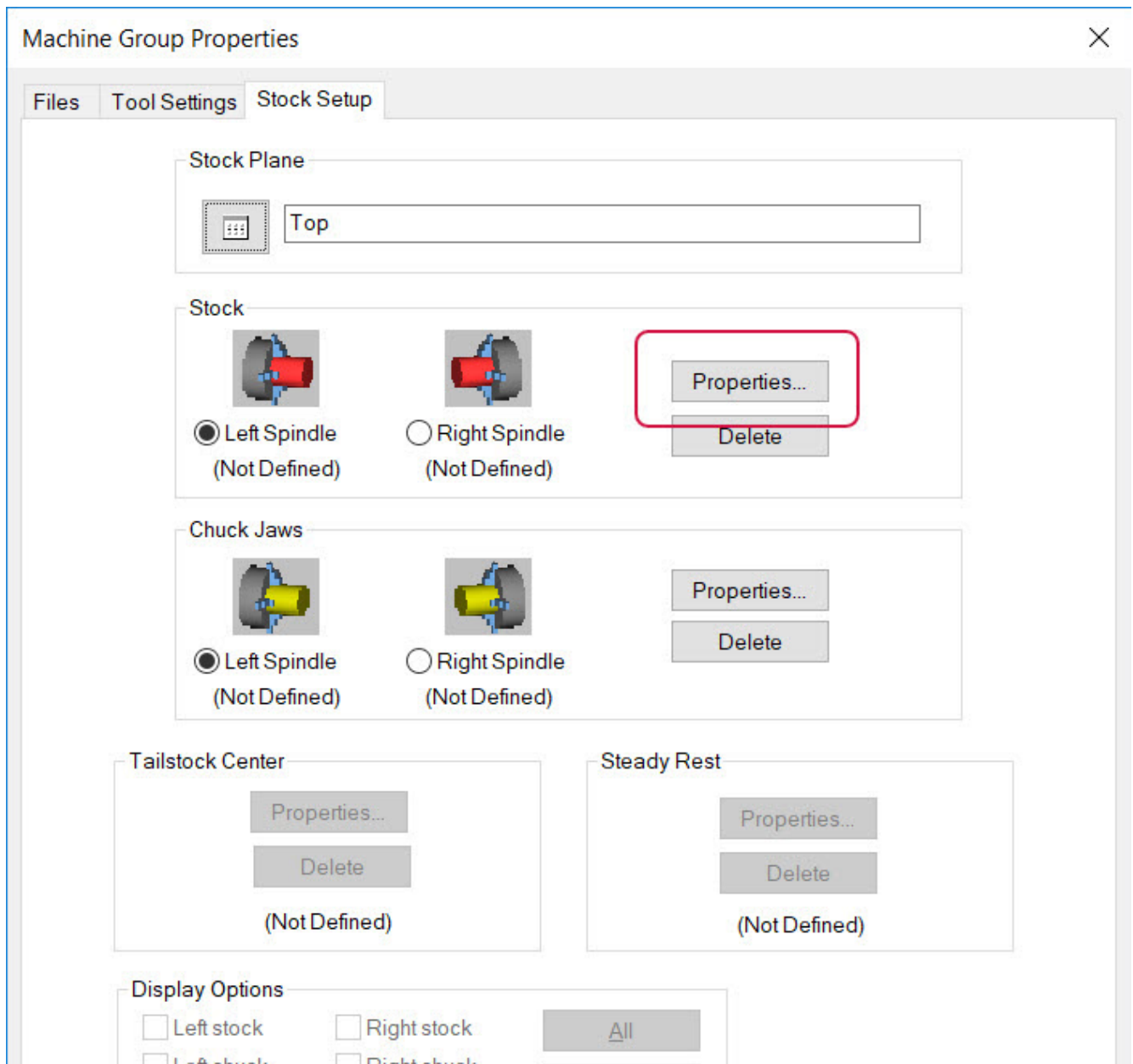
Exercise 3: Completing stock setup

In this exercise, you define the stock from which the part will be cut, as well as define the spindles that hold the part.

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

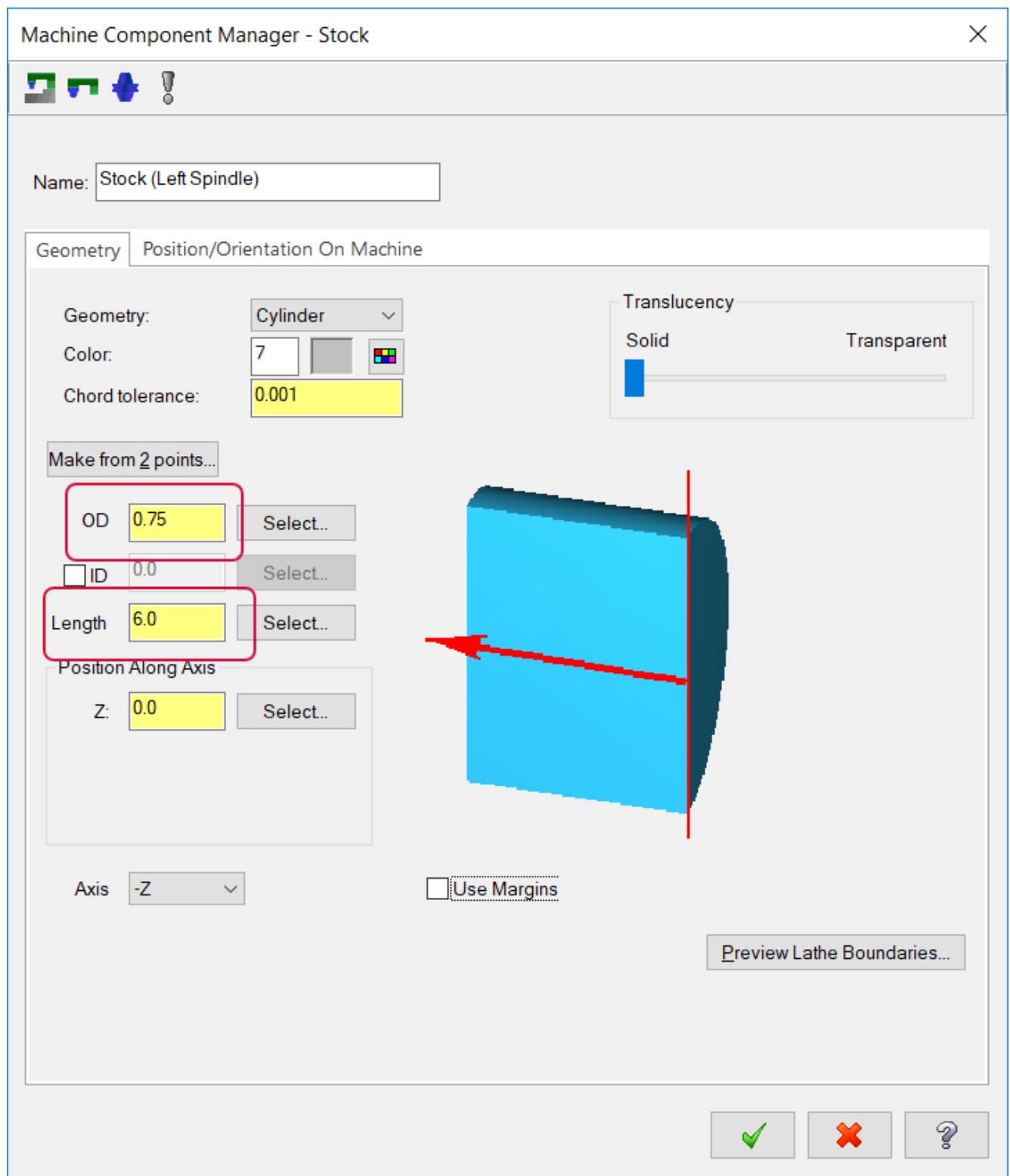


2. Select **Stock, Left Spindle, Properties**.

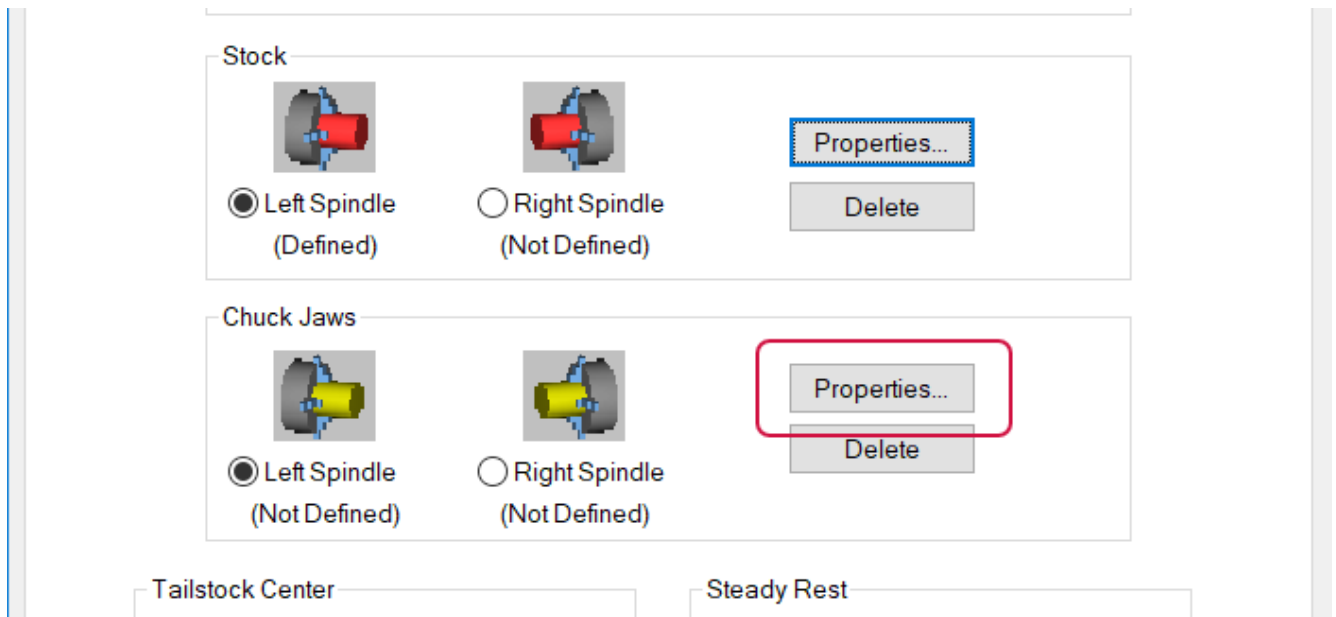


3. Enter an **OD** of **0.75** and a **Length** of **6.0**. Click **OK**.

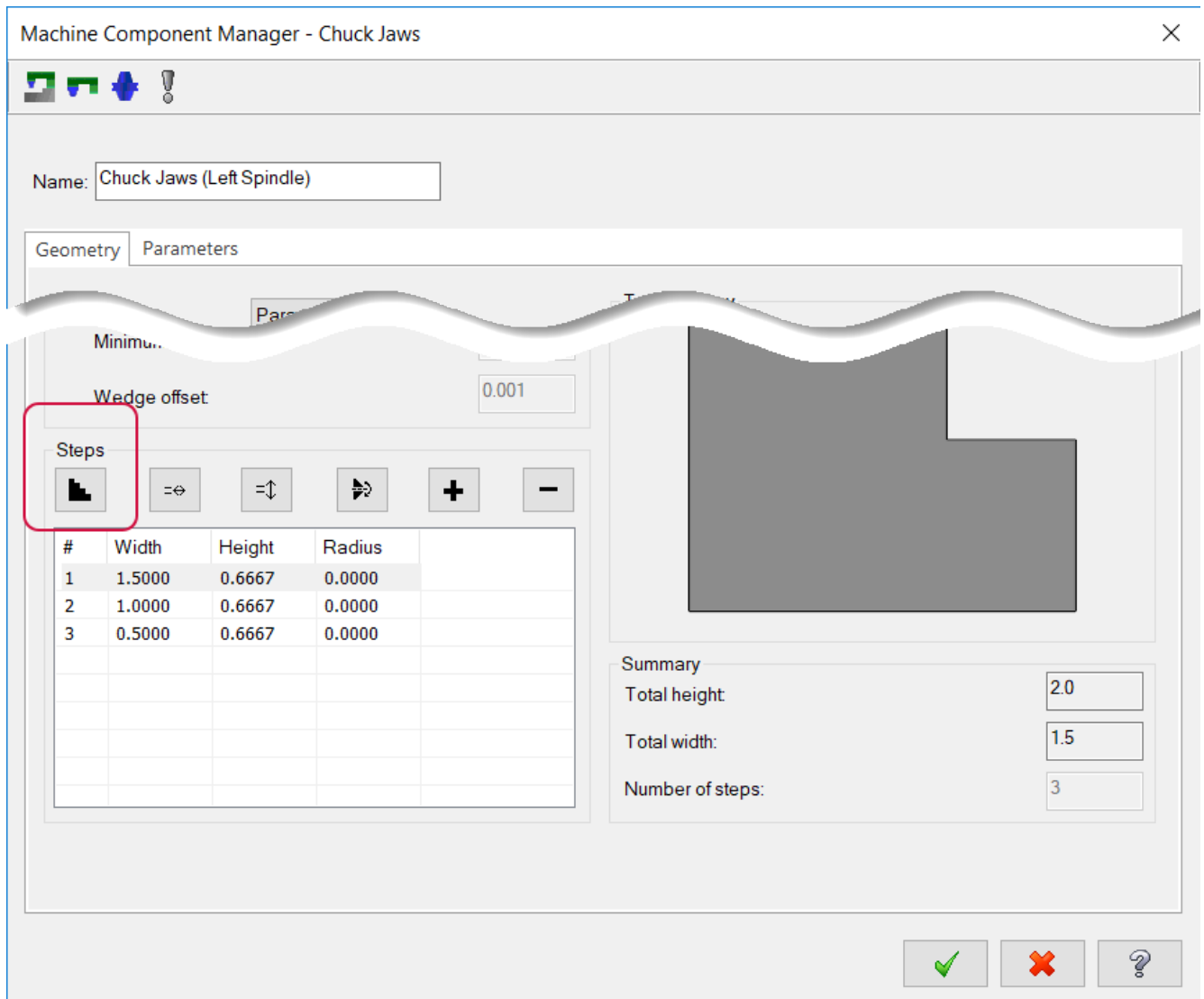
The stock length of 6.0 is for screen display purposes only and has no affect on the NC code.



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

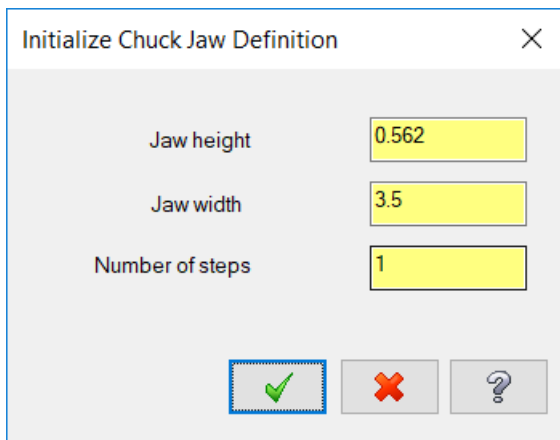


5. In the **Machine Component Manager - Chuck Jaws** dialog box, click **Steps**.



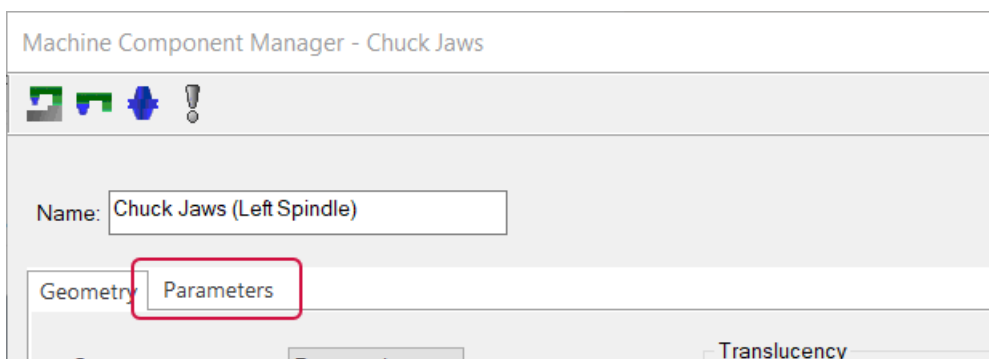
6. In the **Initialize Chuck Jaw Definition** dialog box, enter the following values and click **OK**:

- Set **Jaw height** to **0.562**.
- Set **Jaw width** to **3.5**.
- Set **Number of steps** to **1**.

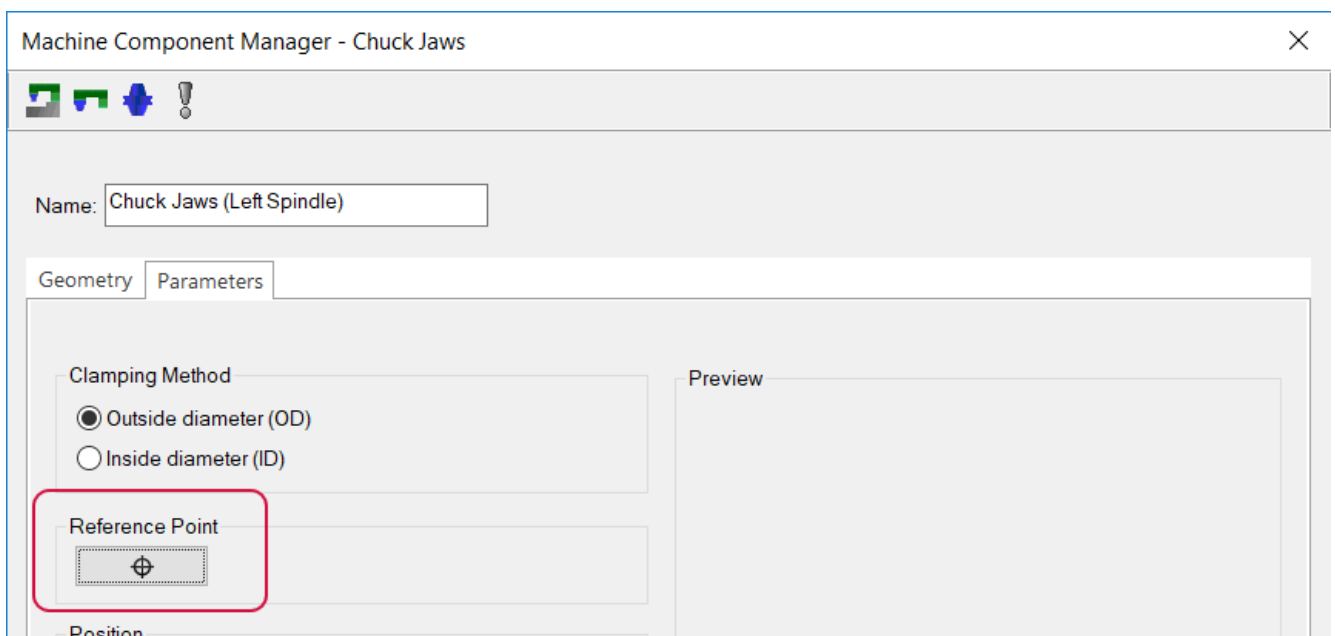


These values change the default three-jaw chuck, which is not used in Swiss Lathe, to a Swiss-style chuck.

7. In the **Machine Component Manager - Chuck Jaws** dialog box, click the **Parameters** tab.



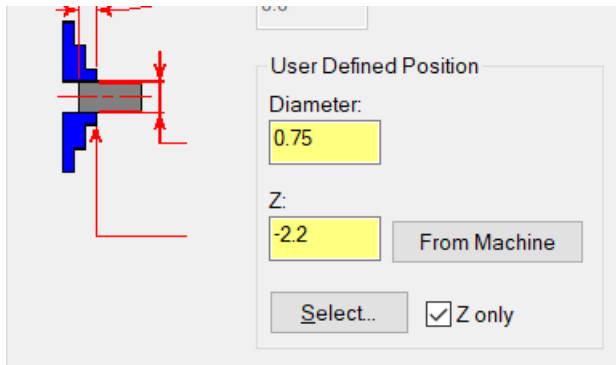
8. Click **Reference Point**. Mastercam switches to its graphics window.



9. Click the corner shown in the following image, and press **[Enter]**.

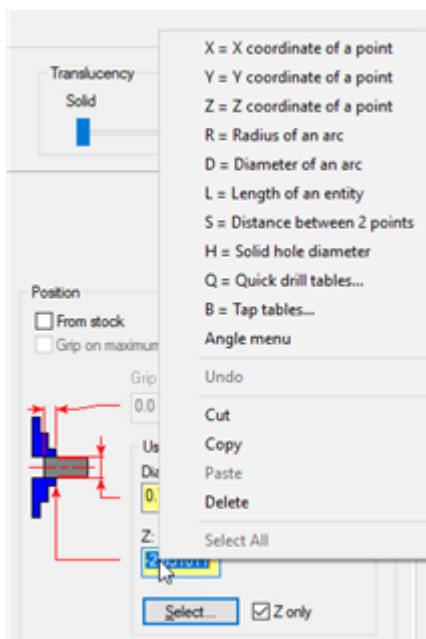


10. For **User Defined Position**, enter a **Diameter** of **0.75** and a **Z** of **-2.2**, and click **OK**.

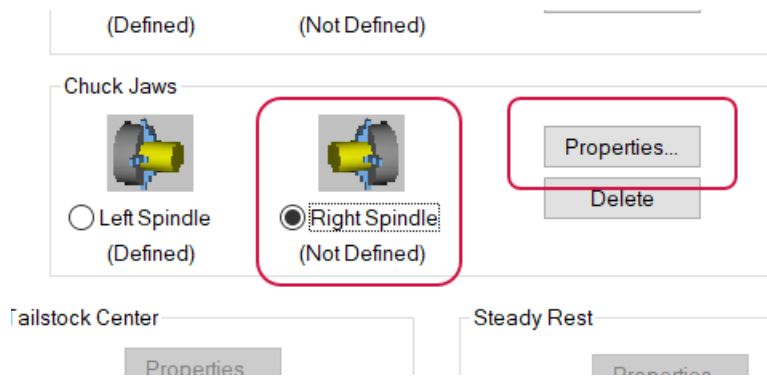


Z is the distance between the tool and the face of the chuck jaw or guide bushing and must be entered as a negative number. Typically, you subtract about **0.5** inches from the full **Z** value. This value needs to consider the overall length, width of the cutoff, facing amount and a little extra for clearance at a minimum. 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.

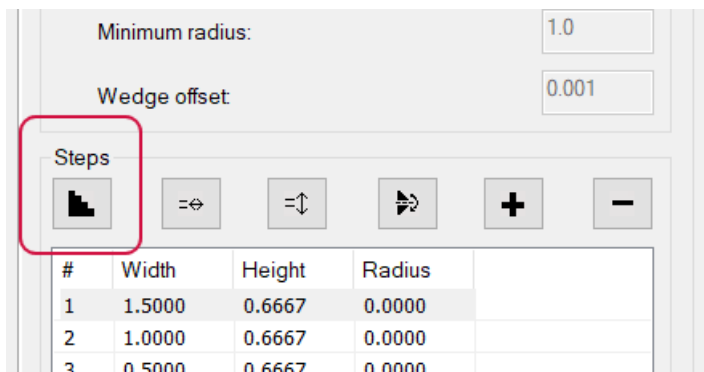
To set **Z** with geometry, right-click the **Z** field, choose the geometry type (in this case, **X coordinate of a point**; see the following picture), and then make a selection in the graphics window. You can then use the built-in calculator in the **Z** field to subtract about **0.5** inches, as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



11. Back in the **Machine Group Properties** dialog box, choose **Chuck Jaws**, **Right Spindle**, and click **Properties**.



12. In the **Machine Component Manager - Chuck Jaws** dialog box, click **Steps**.

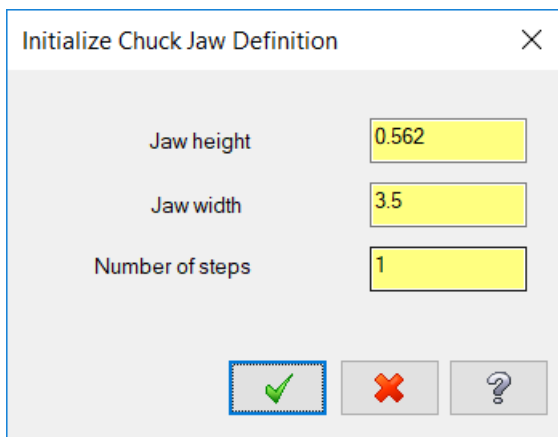


13. In the **Initialize Chuck Jaw Definition** dialog box, enter the following values and click **OK**:

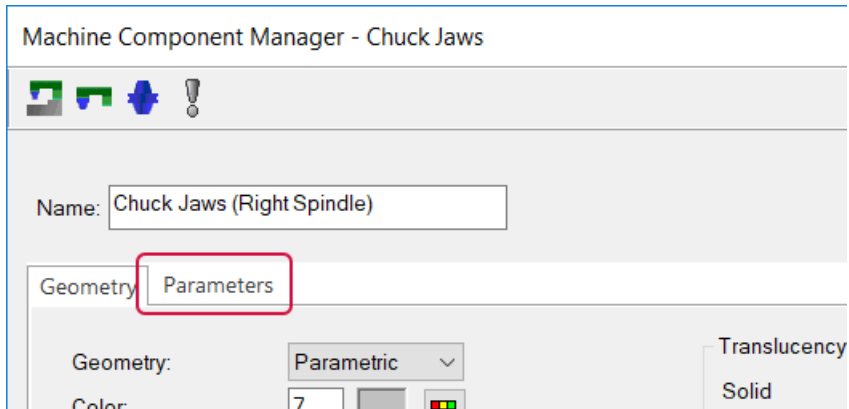
Jaw height: 0.562

Jaw width: 3.5

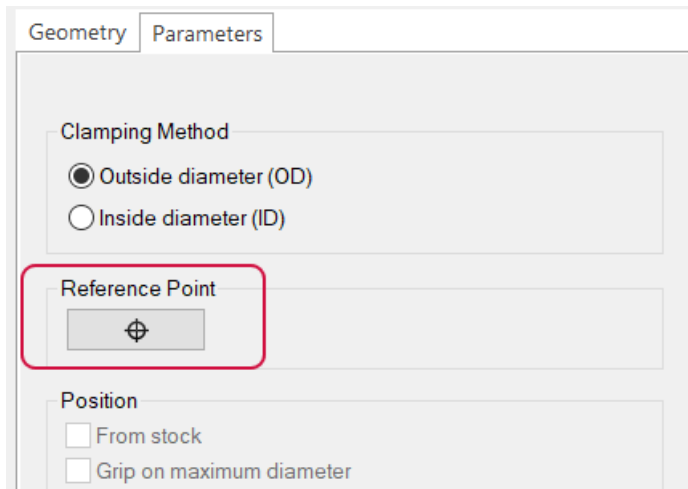
Number of steps: 1



14. Click the **Parameters** tab.



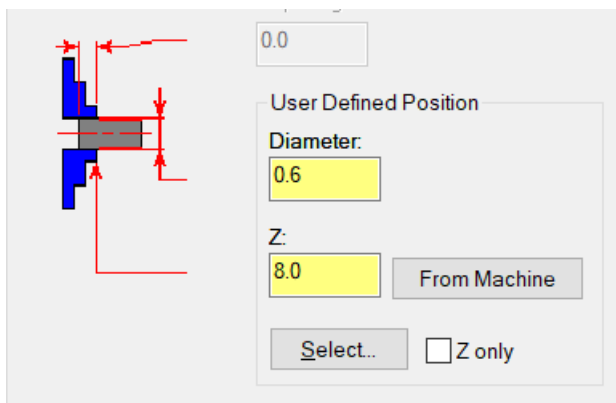
15. Click **Reference Point**. Mastercam switches to its graphics window.



16. Click the corner shown in the following image, and press **[Enter]**.



17. For **User Defined Position**, enter a **Diameter** of **0.6** and a **Z** of **8**, and click **OK**



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

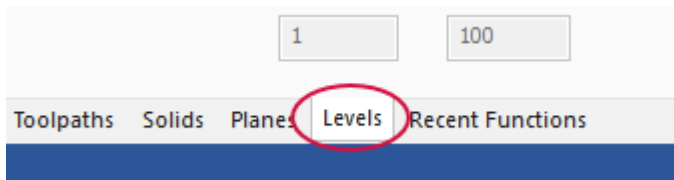
18. Click **OK** in both the **Machine Component Manager** and **Machine Group Properties** dialog boxes.
19. In Mastercam's graphics window, zoom out until you see the fully defined spindles, as shown in the picture following.



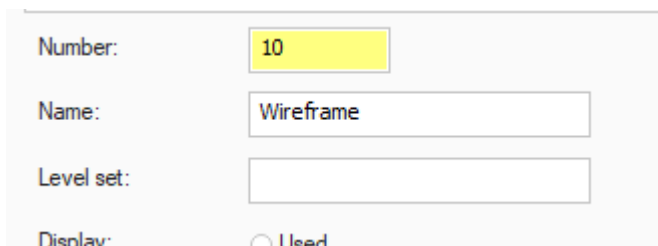
Exercise 4: Creating the turn profile

Here, you create a turning profile for the part, placing the new wireframe geometry in its own level. The turn profile provides construction geometry that you can select to help define toolpaths.

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

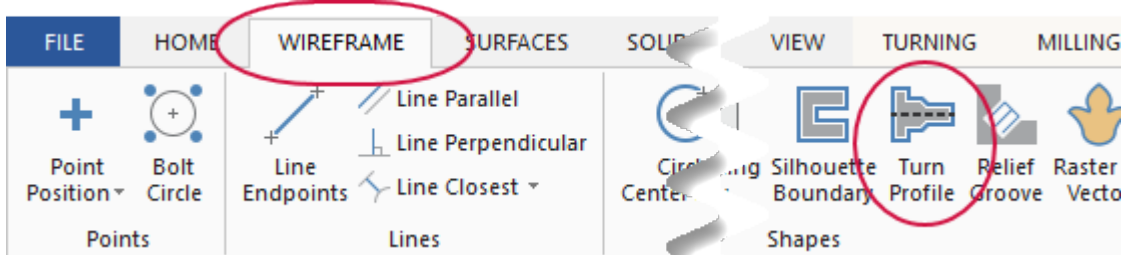


2. Type **10** in **Number** and **Wireframe** in **Name**.

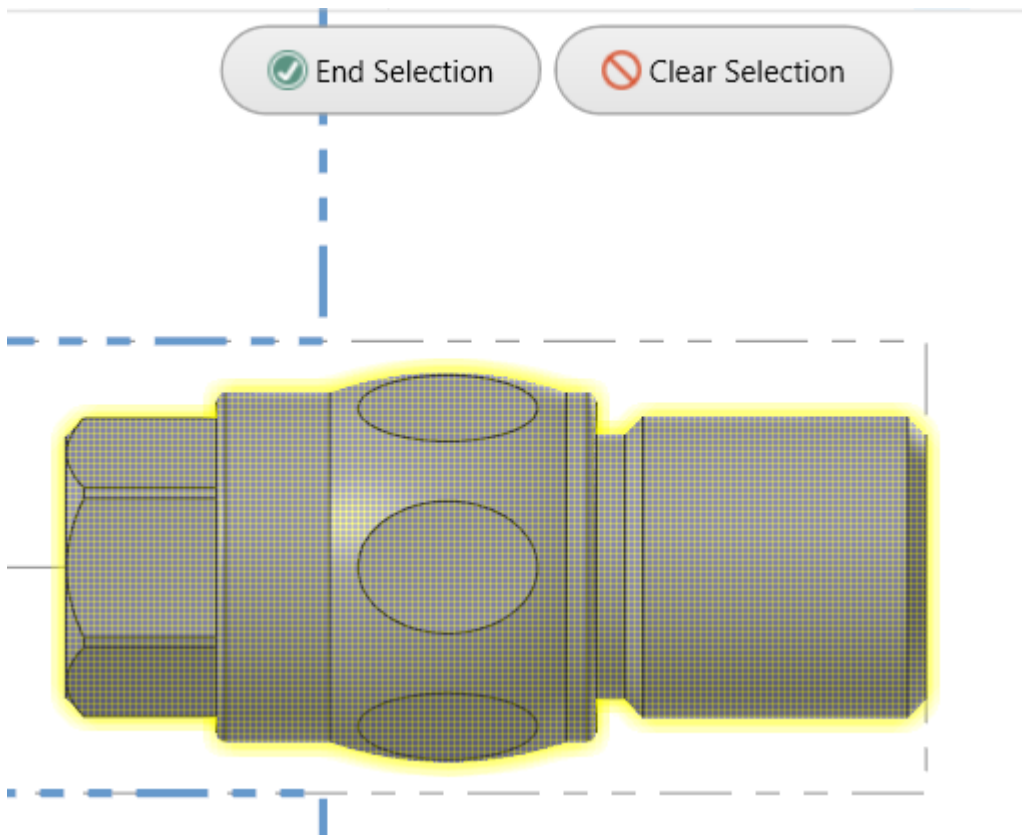


In a production piece, this is where you would make desired changes to the line thickness and color before creating the turn profile.

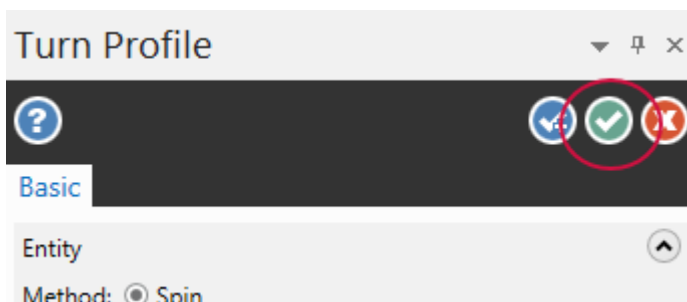
3. Go to Mastercam's **Wireframe** tab, and choose **Turn Profile**.



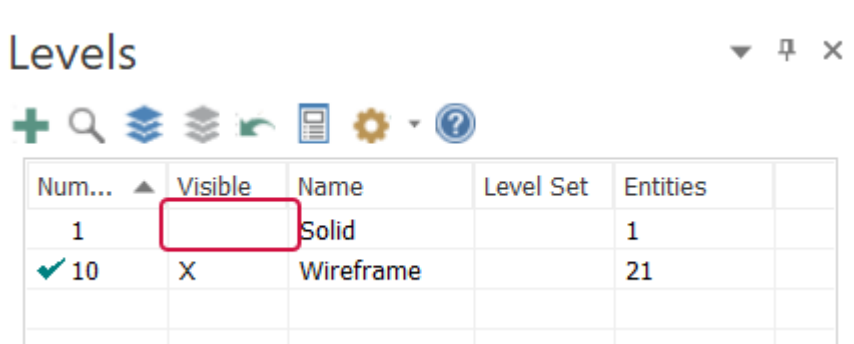
4. In the graphics window, select the solid, and click **End Selection** or press **[Enter]**.



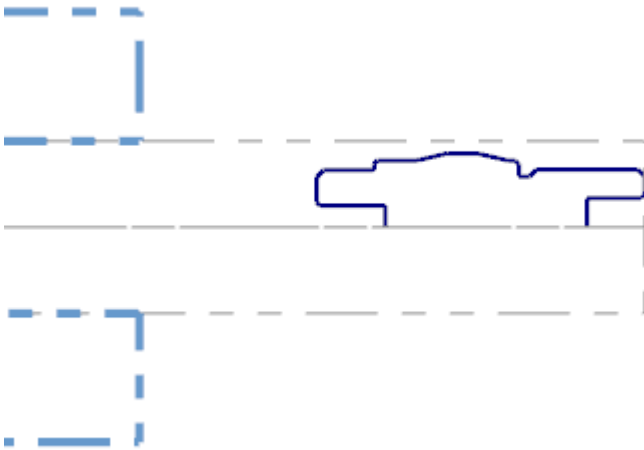
5. To finalize your selection and accept the default options, click **OK** in the **Turn Profile** function panel.



6. In the Levels Manager, hide the solid by clicking in its **Visible** column.



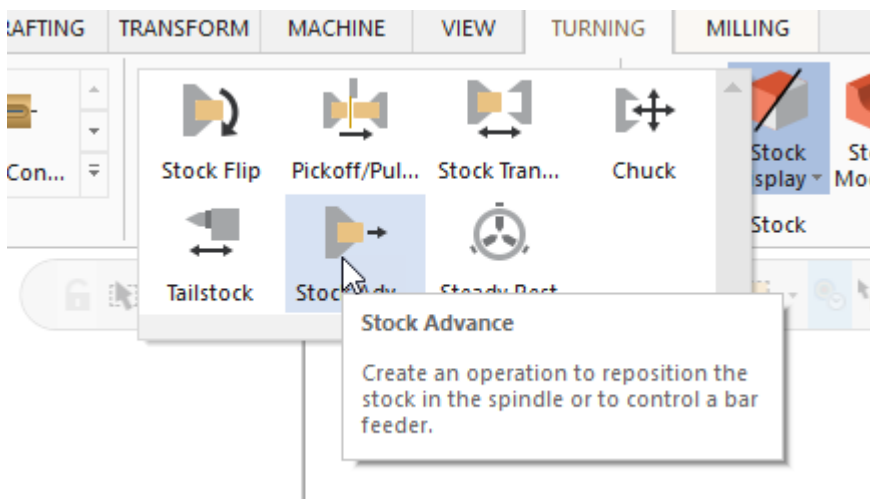
The graphics window now shows the part's turning profile, without the solid in the way, as shown in the following image.



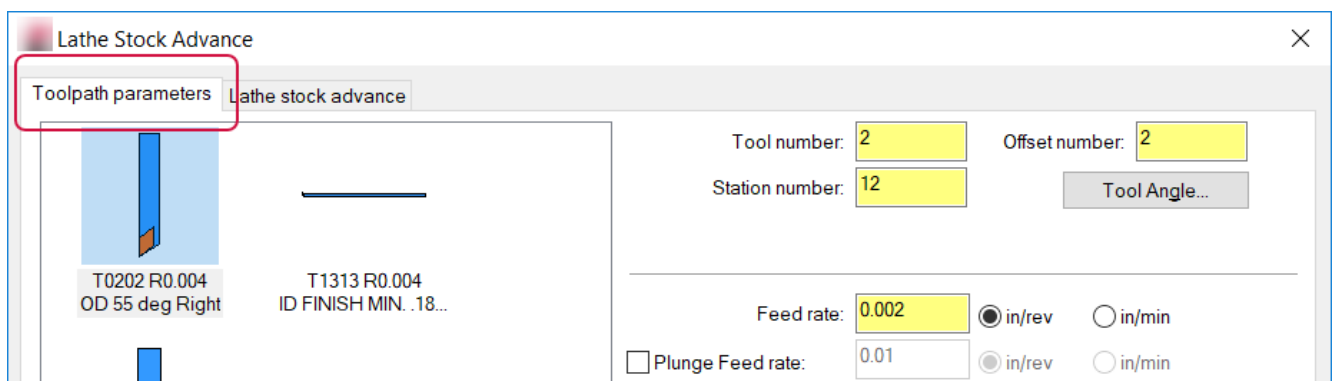
Exercise 5: The stock advance operation

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.

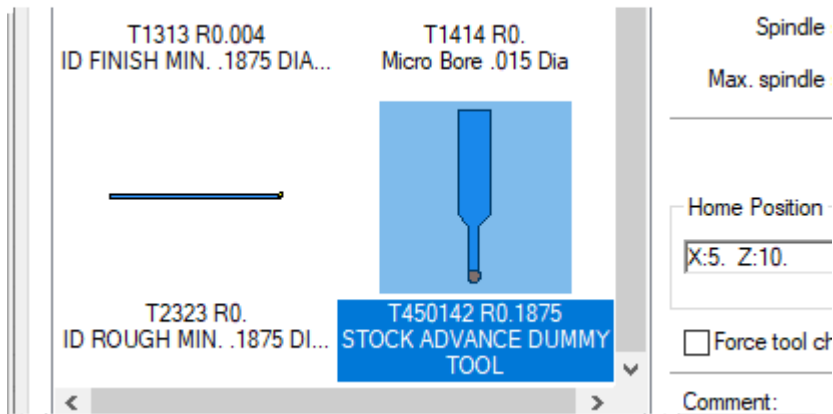
1. On the **Lathe, Turning** contextual tab, select **Part Handling, Stock Advance**.



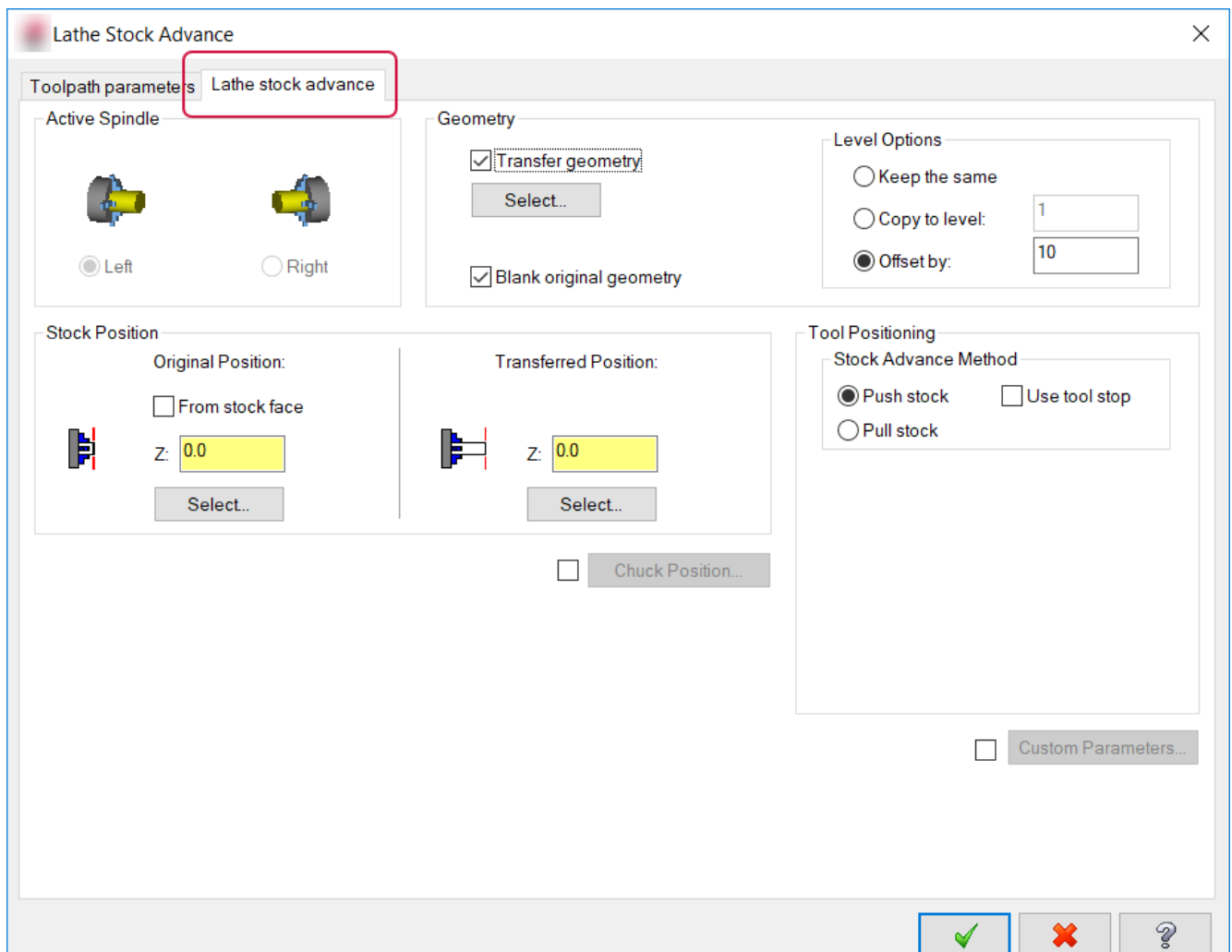
2. In the **Lathe Stock Advance** dialog box, select the **Toolpath parameters** tab.



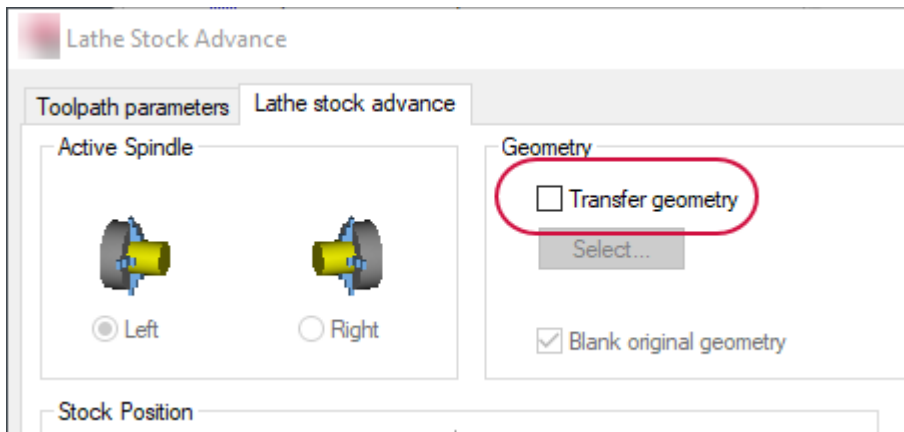
3. Select tool number **T450142**, the stock advance dummy tool. The operation requires that a tool be selected, even though it will not use it. Selecting a tool that will not be used with Swiss Lathe eliminates later conflicts in Mastercam.



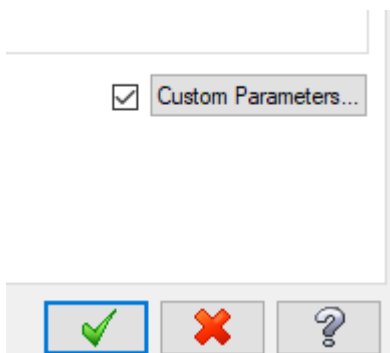
4. Select the **Lathe stock advance** tab.



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. (If you get a warning that the tool does not exist: complete the operation, and then click the operation's **Parameters** icon and reselect the tool.)

Set **Cutoff** to **1**.

Set **Stock Diameter** to **0.75**.

Set **Cutoff RPM** to **2000.0**.

Set **Cutoff Feedrate** to **0.002**.

Set **Z1 Start Position** to **6.0**.

Set **Part Length** to **1.4275**.

Set **End X Position** to **-0.04**.

Set **Cutoff Width** to **0.118**.

Custom Parameters	
Integers	Reals
Cutoff[0=M98 P9810,1=G256]	Stock Diameter [#531 or A]
EOB[0=Feeder,1=Tube]	Cutoff RPM [#529 or S]
Bar Change Prog Number	Cutoff Feedrate [#522 or F]
Custom integer #4:	Z1 Start Position [#2601 or Z]
Run Max Stroke Z [0=No,1=Yes]	Part Length [W]
Custom integer #6:	End X Position [X]
Custom integer #7:	Cutoff Width [B]
Custom integer #8:	LH/RH Cutoff Shift

Note that the values shown here are for the Star machine used in this tutorial. Other machines will show different values in this dialog box. Consult with your Reseller or machine manufacturer for more information on your specific machine.

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

CHAPTER 2

LEFT SPINDLE OPERATIONS

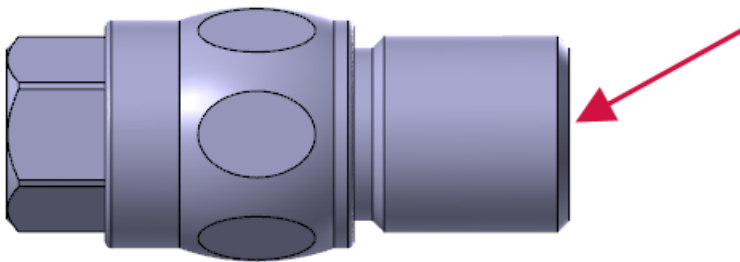
Now that you have the Swiss environment set up, you can begin to program the operations needed to machine the part. The operations you complete in this chapter are all on the left spindle.

Goals

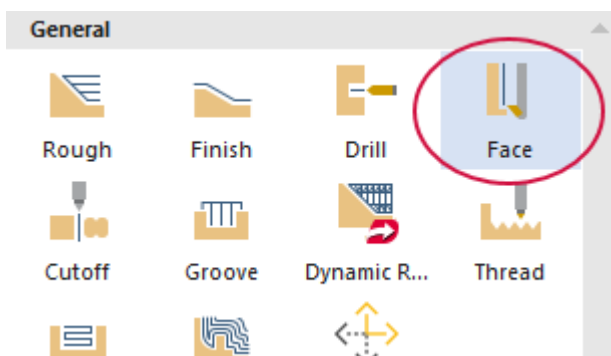
- Create a facing operation.
- Create drilling operations.
- Create finishing operations.
- Create threading and groove operations.
- Create milling operations.

Exercise 1: Programming the facing operation

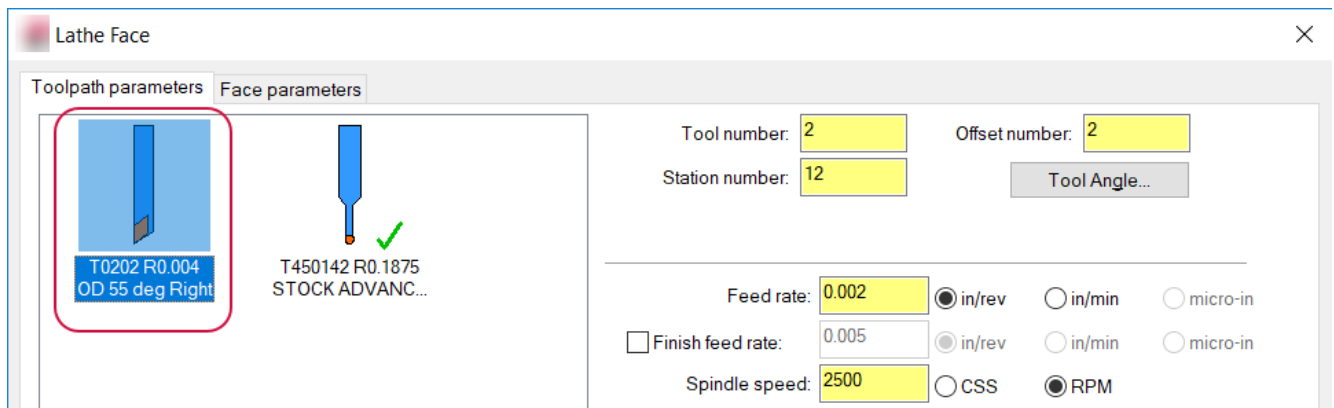
In this exercise, you program the operation that faces the front of the part.



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

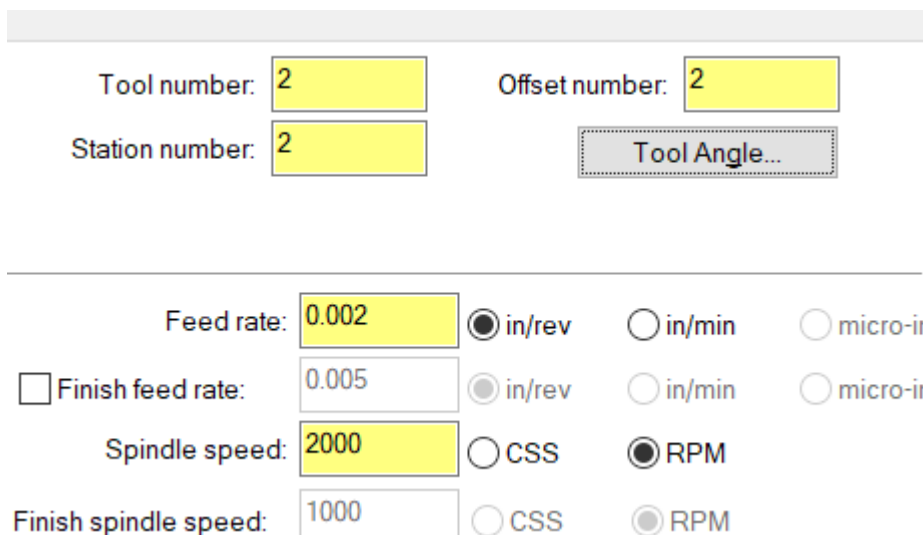


2. Ensure that the **Show library tools** box is checked, and select tool **T0202 R0.004 OD 55 deg Right**.

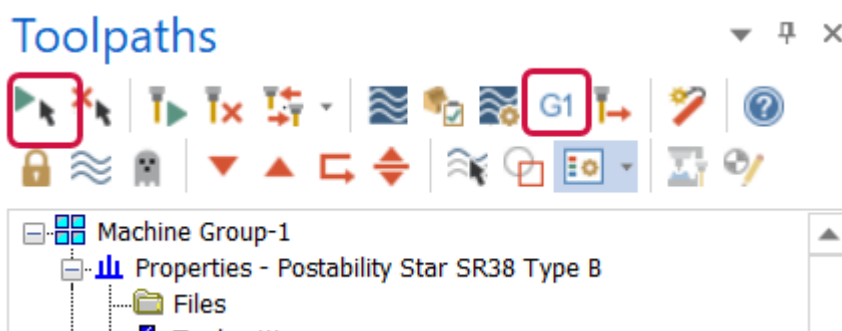


3. Set the parameters as shown in the picture below, and click **OK**.

- Set **Tool number** to **2**.
- Set **Offset number** to **2**.
- Set **Station number** to **2**.
- Set **Feed rate** to **0.002, in/rev**.
- Set **Spindle speed** to **2000, RPM**.

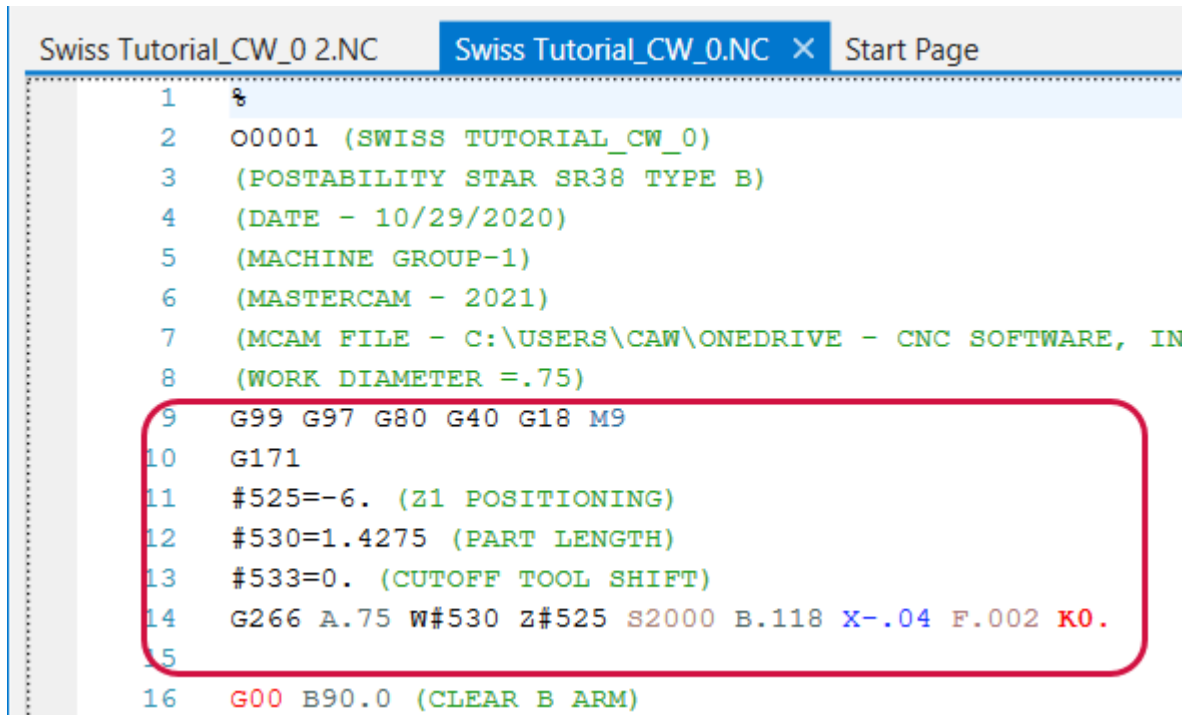


4. Return to the Toolpaths Manager, click **Select all operations**, and then click **Post selected operations**.



5. When the **Post Processing** and **Save As** dialog boxes appear, click **OK** in both.

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 Code Expert, select the second tab. You can see the values you specified in the **Custom Parameters** dialog box, as shown in the following picture.

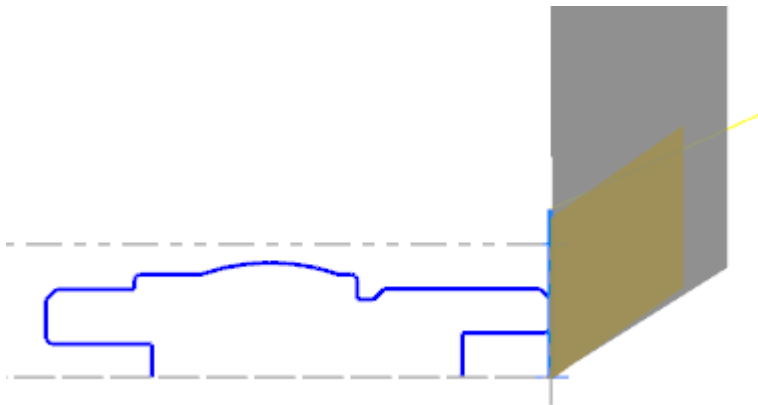


```

Swiss Tutorial_CW_0 2.NC  Swiss Tutorial_CW_0.NC  X  Start Page
1  %
2  O0001 (SWISS TUTORIAL_CW_0)
3  (POSTABILITY STAR SR38 TYPE B)
4  (DATE - 10/29/2020)
5  (MACHINE GROUP-1)
6  (MASTERCAM - 2021)
7  (MCAM FILE - C:\USERS\CAW\ONEDRIVE - CNC SOFTWARE, IN
8  (WORK DIAMETER =.75)
9  G99 G97 G80 G40 G18 M9
10 G171
11 #525=-6. (Z1 POSITIONING)
12 #530=1.4275 (PART LENGTH)
13 #533=0. (CUTOFF TOOL SHIFT)
14 G266 A.75 W#530 Z#525 S2000 B.118 X-.04 F.002 K0.
15
16 G00 B90.0 (CLEAR B ARM)

```

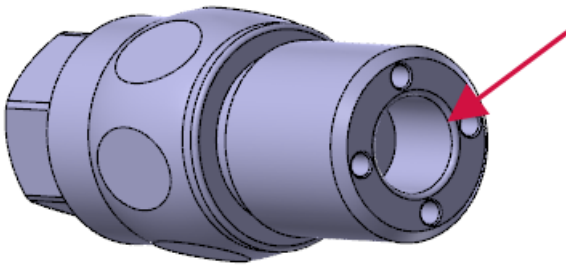
8. Close Code Expert.
9. In Mastercam, backplot the operation to see the toolpath in action.



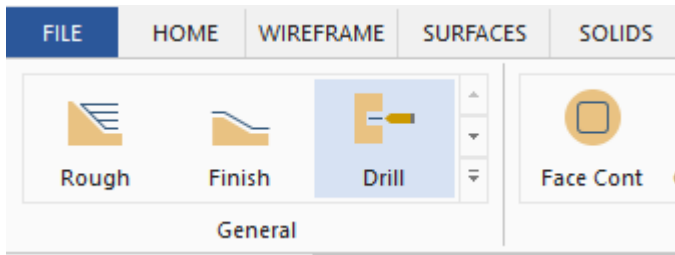
10. Save the file with the name Swiss Tutorial_XX_2.mcam, where XX is your initials.

Exercise 2: Spot drilling the front hole

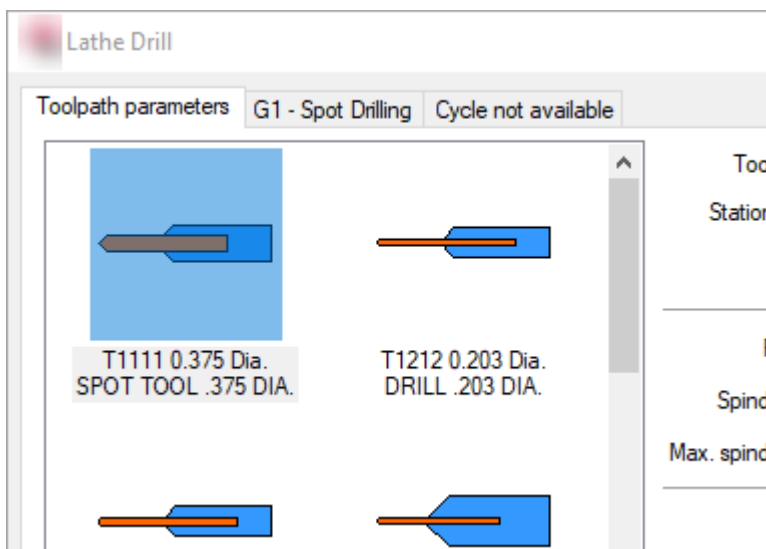
In this exercise, you spot drill on the front of the part, providing a starting hole for the next drill operation.



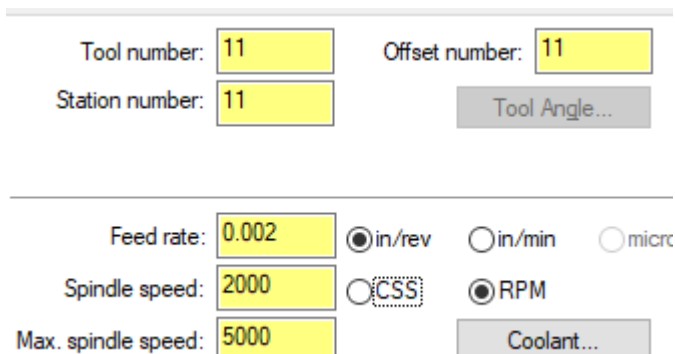
1. Select **Drill** from the **General** gallery on the **Lathe, Turning** contextual tab.



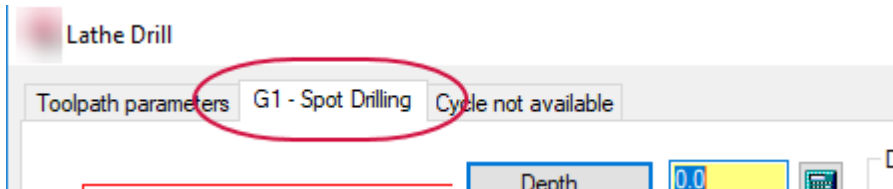
2. In the **Lathe Drill** dialog box, select the **T1111 0.375** tool. (There is more than one T1111 tool. Be sure to pick the correct one.)



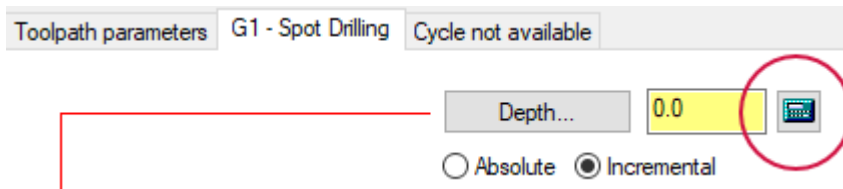
3. Make sure that your parameters match the image below.



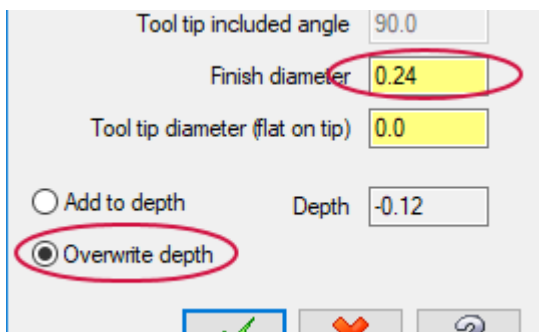
4. Click the **G1 - Spot Drilling** tab.



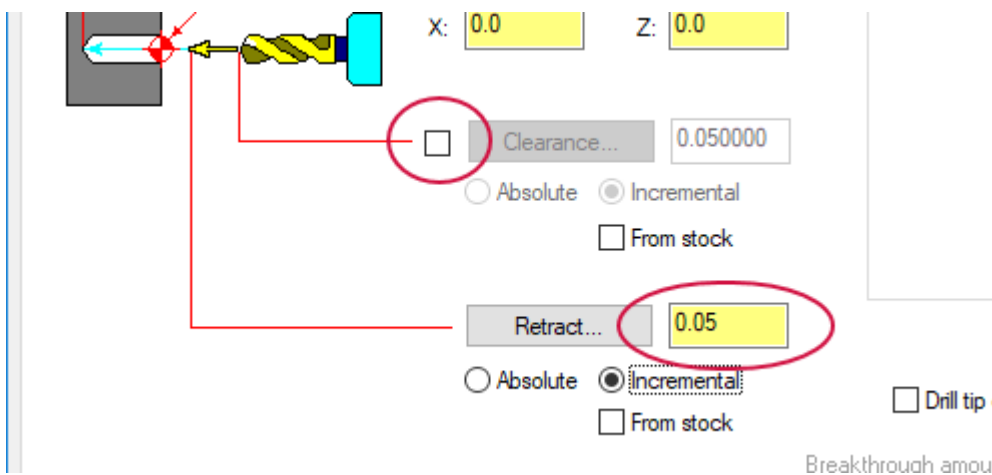
5. Click the **Depth Calculator** button.



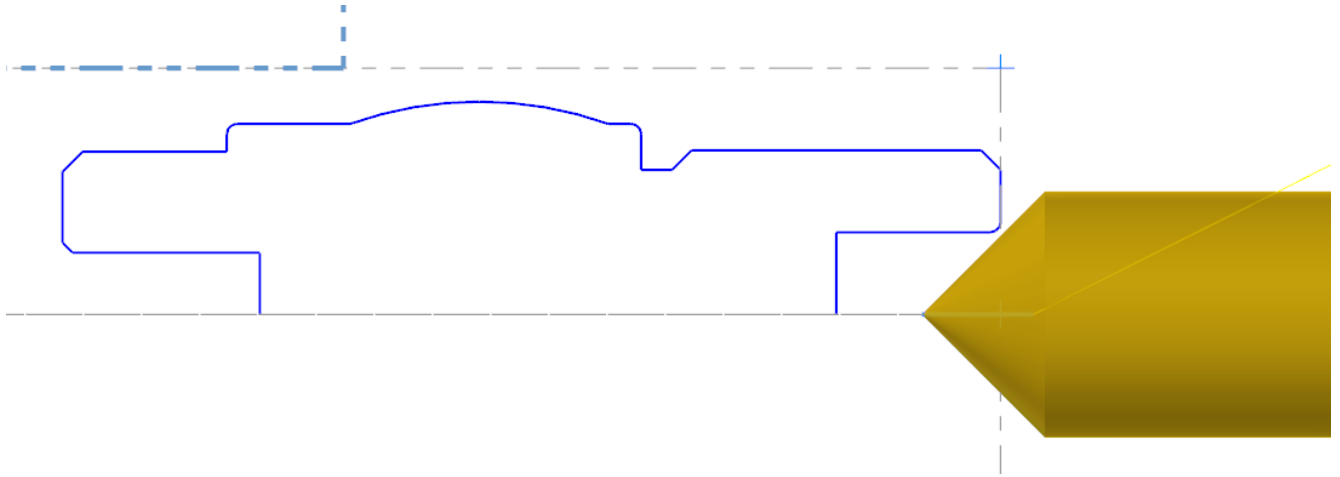
6. In the **Depth Calculator** dialog box, change **Finish diameter** to **0.24**, select **Overwrite depth**, and then click **OK**.



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



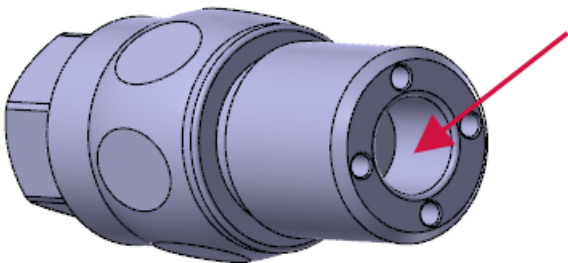
8. Backplot the toolpath to see the toolpath in action.



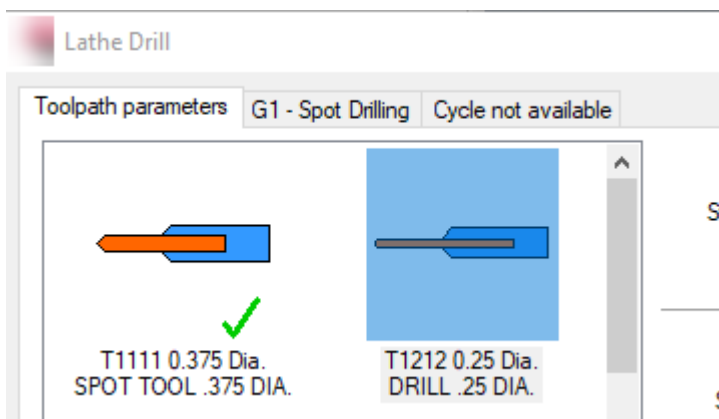
9. Save the file with the name Swiss Tutorial_XX_3.mcam, where XX is your initials.

Exercise 3: Drilling the front hole

In this exercise, you complete the hole that you spot drilled in the previous exercise.



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. (There are several T1212 tools. Be sure to pick the right one.)



3. Set the parameters as shown in the following image.
 - Set **Tool number** to **12**.
 - Set **Offset number** to **12**.
 - Set **Station number** to **12**.

- Set **Feed rate** to **0.002, in/rev**.
- Set **Spindle speed** to **2000, RPM**.

Tool number:	12	Offset number:	12
Station number:	12	Tool Angle...	

Feed rate:	0.002	<input checked="" type="radio"/> in/rev	<input type="radio"/> in/min	<input type="radio"/> micro-in
Spindle speed:	2000	<input type="radio"/> CSS	<input checked="" type="radio"/> RPM	
Max. spindle speed:	5000	Coolant...		

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

Lathe Drill

Toolpath parameters **G1 - Spot Drilling** Cycle not available

Depth 0.0

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

Drill Cycle Parameters

Cycle:

G1 - Spot Drilling

First peck 0.0

6. Click the **Depth** button, and when Mastercam returns you to the graphics window, select the point shown in the second picture following. The **Depth** field should now read -0.25.

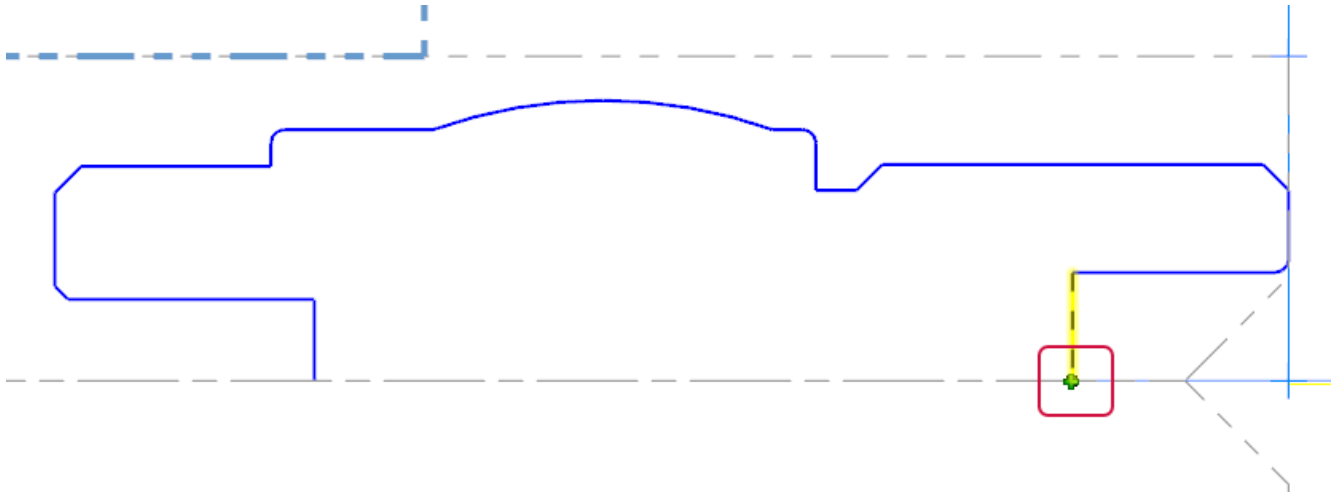
Lathe Drill

Toolpath parameters **G1 - Spot Drilling** Cycle not available

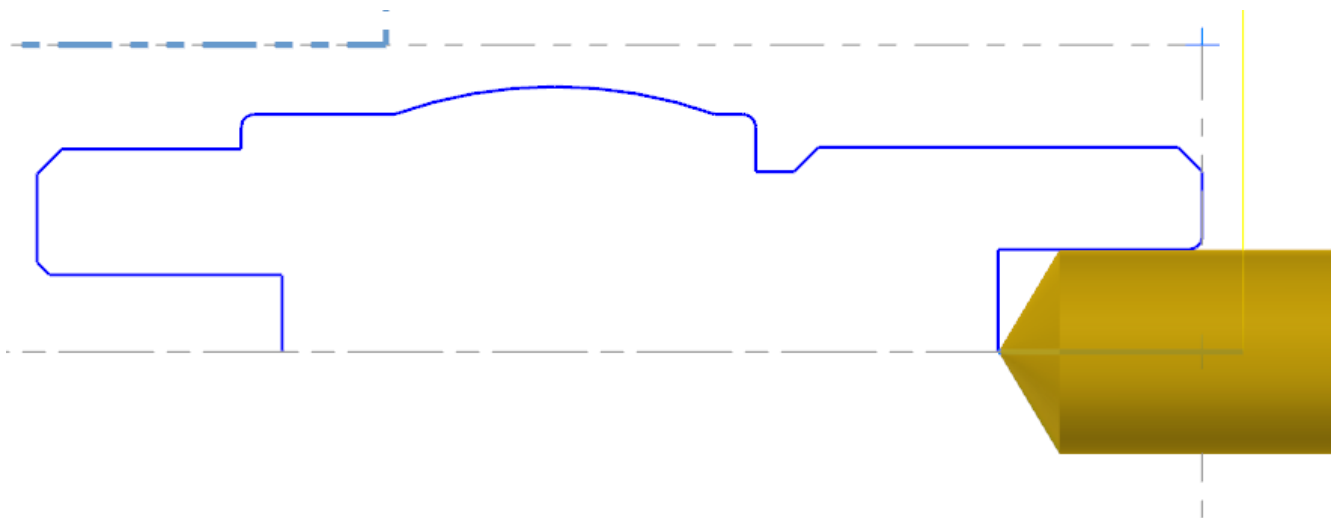
Depth... -0.12

☐ Absolute ☒ Incremental

Drill Point...



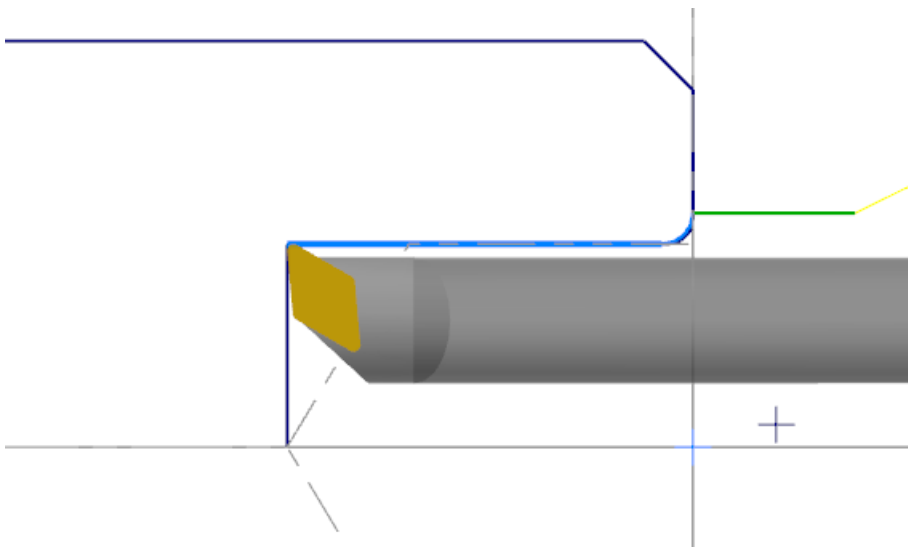
7. Click **OK** to accept your changes and to exit the dialog box.
8. Backplot the operation to see the toolpath in action..



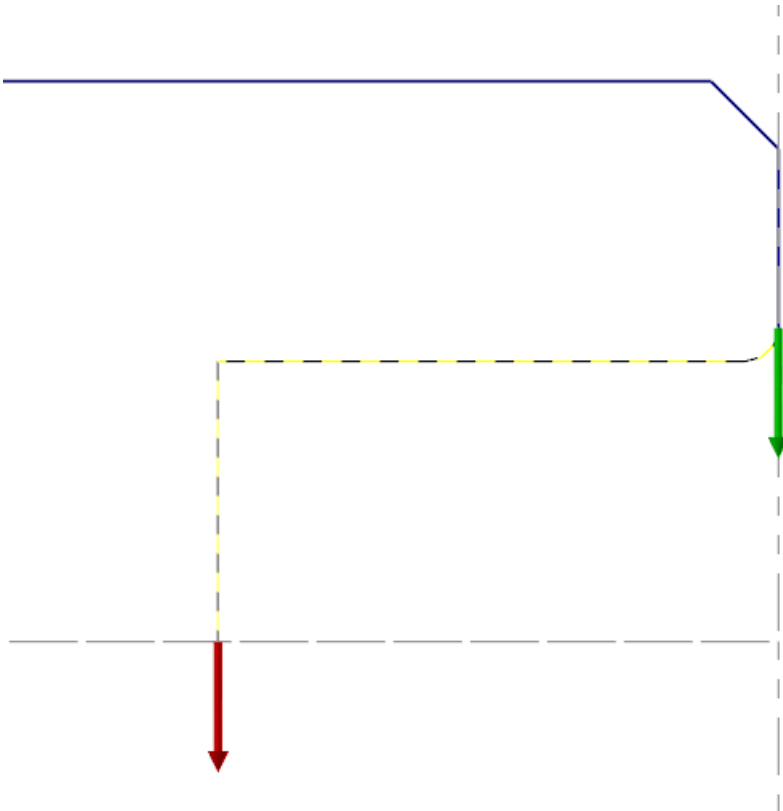
9. Save the file with the name Swiss Tutorial_XX_4.mcam, where XX is your initials.

Exercise 4: Creating the Finish Operation

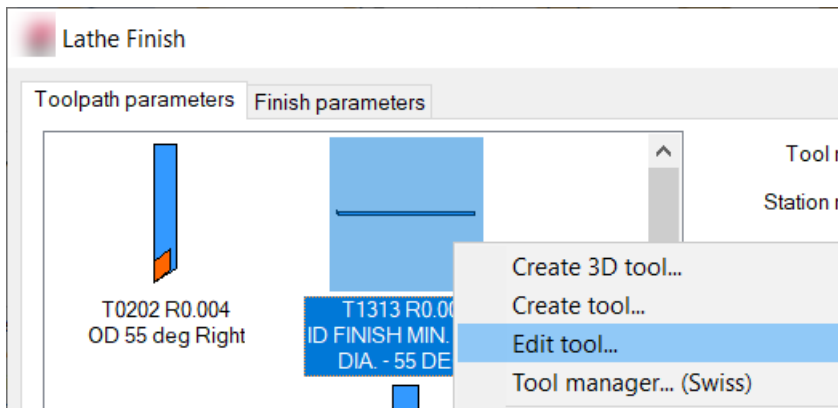
In this exercise, using a boring tool, you create a finishing operation for the hole you drilled previously.



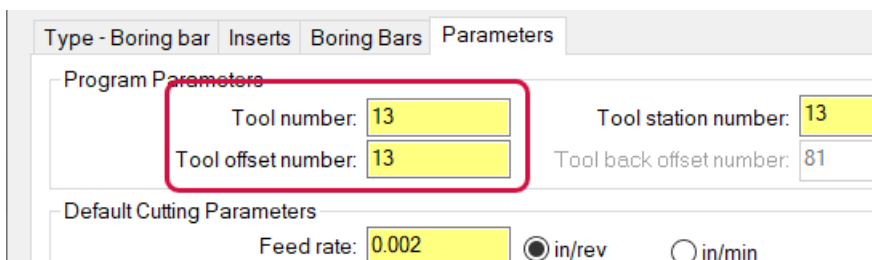
1. Select **Finish** from the **Lathe, Turning** General contextual tab.
2. Chain the part as shown below, and click **OK**.



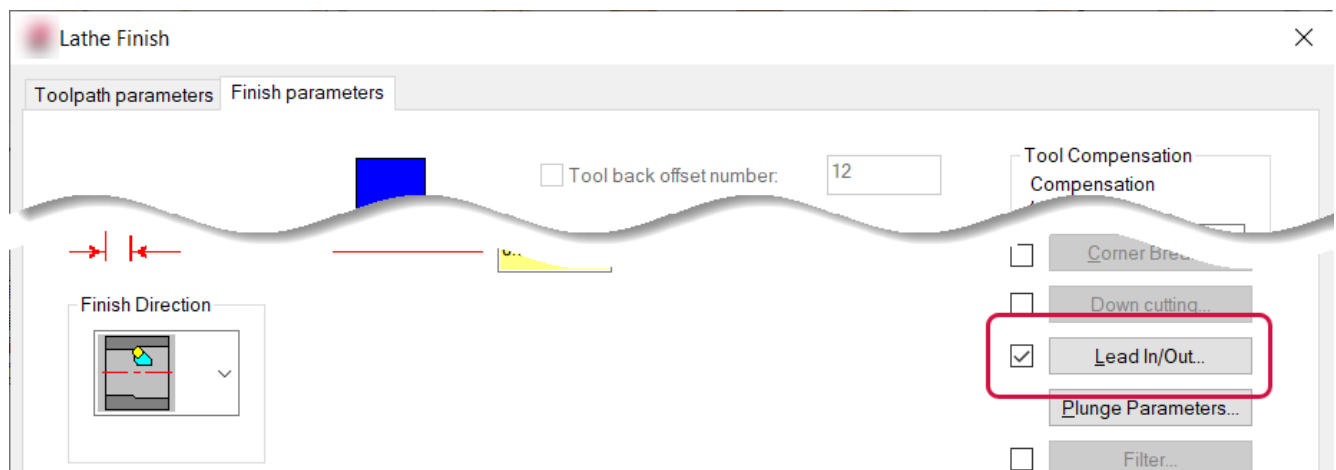
3. Right-click tool **T1313 R0.004 ID FINISH MIN..1875 DIA. - 55 DEG.**, and choose **Edit tool** from the pop-up menu. Because there is more than one T1313, make sure you choose the boring bar shown in the following picture.



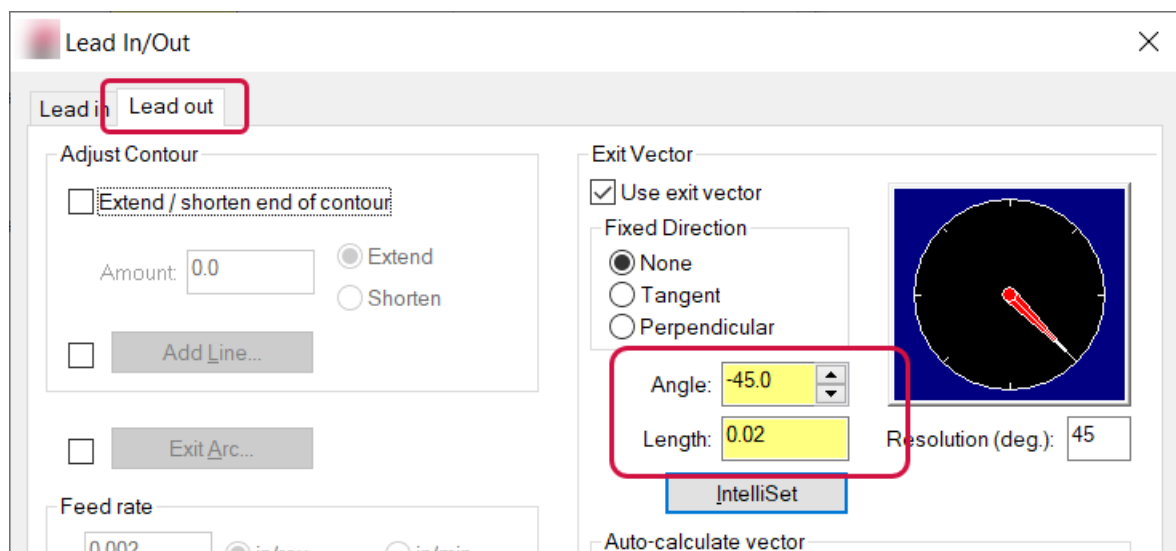
4. In the **Define Tool** dialog box, go to the **Parameters** tab, and ensure that **Tool number** and **Tool offset number** are both set to **13**. Click **OK**.



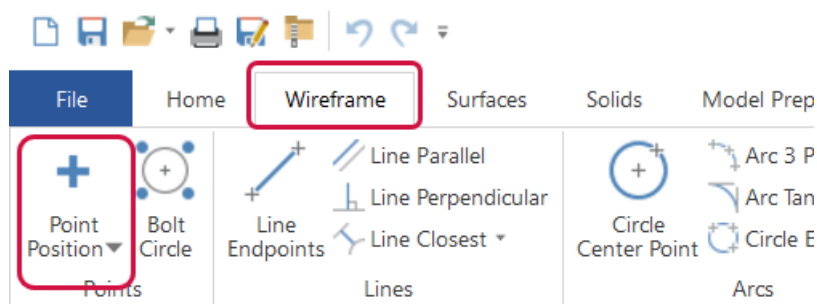
5. In the **Finish Parameters** tab, click **Lead In/Out**.



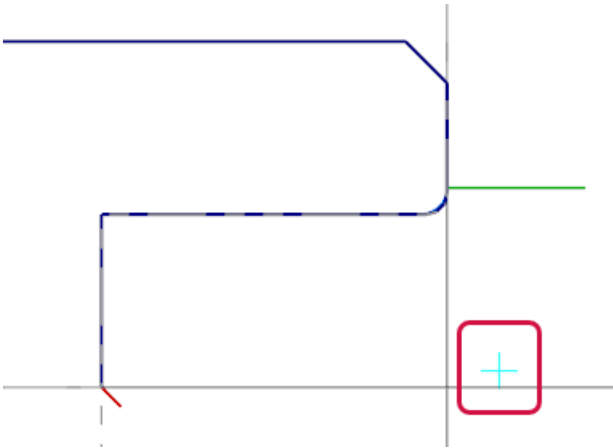
6. In the **Lead out** tab (make sure you choose **Lead out**, not **Lead in**), set **Angle** to **-45.0** and **Length** to **0.02**. Click **OK** in both the **Lead In/Out** and **Lathe Finish** dialog boxes.



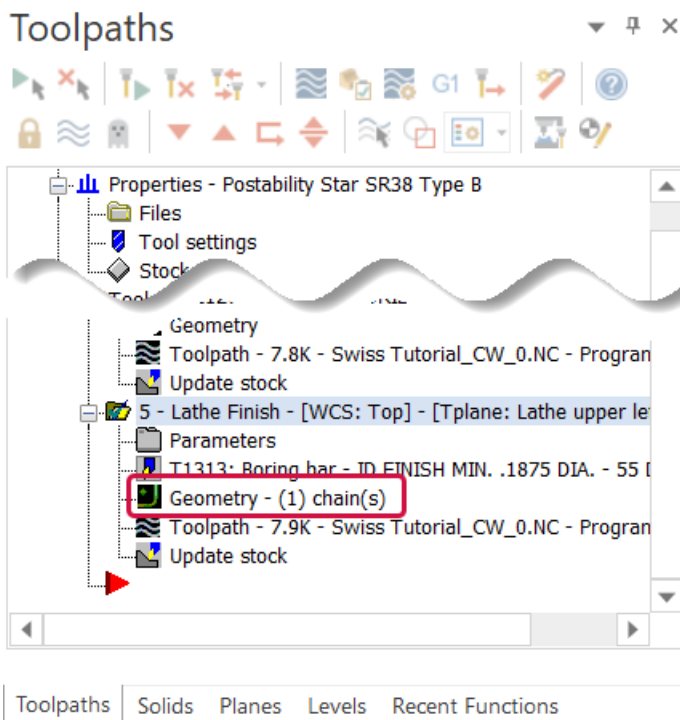
7. On Mastercam's **Wireframe** tab, select **Point Position**.



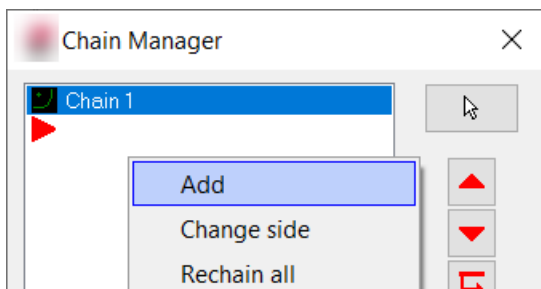
8. Create a point similar to the one shown in the following picture (in other words, your choice need not exactly match the one shown), and click **OK** in the **Point Position** function panel. Note that the point you select must be above the center line of the part and beyond the face of the part.



9. In Toolpaths Manager, select **Geometry** under operation 5 to display the **Chain Manager** dialog box.

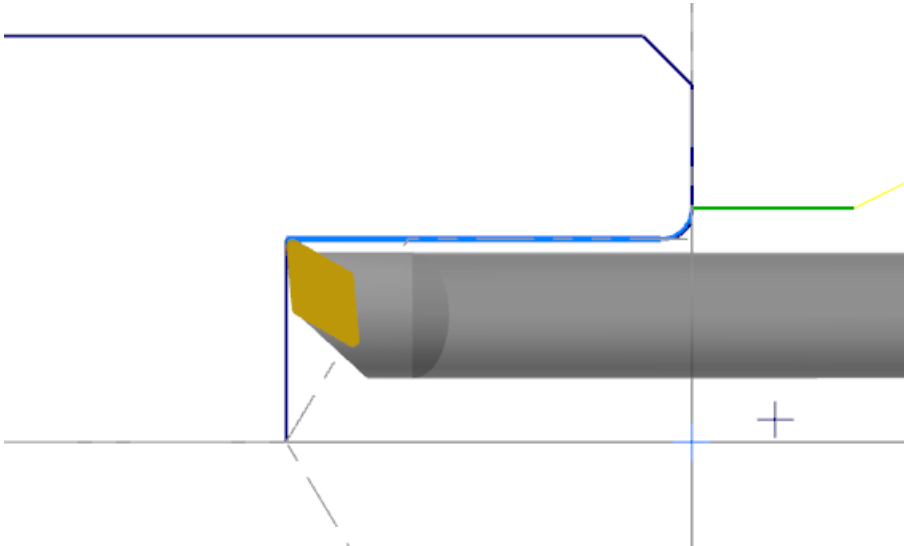


10. Right-click in the **Chain Manager** dialog box, and choose **Add** from the pop-up menu.



11. Select the new point from Step 8, and click **OK** in both the **Chaining** and **Chain Manager** dialog boxes.

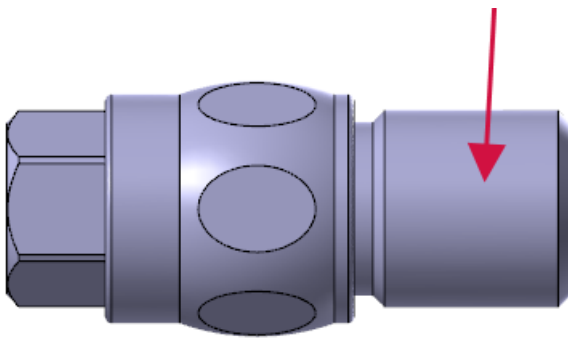
12. Regenerate all operations and backplot operation 5 to see it in action.



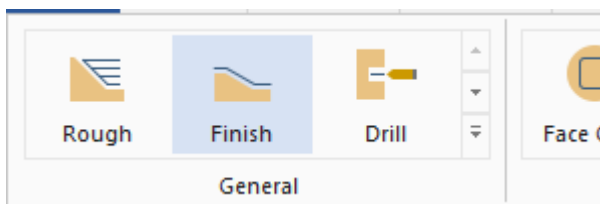
13. Save the file with the name Swiss Tutorial_XX_5.mcam, where XX is your initials.

Exercise 5: Finishing the front side

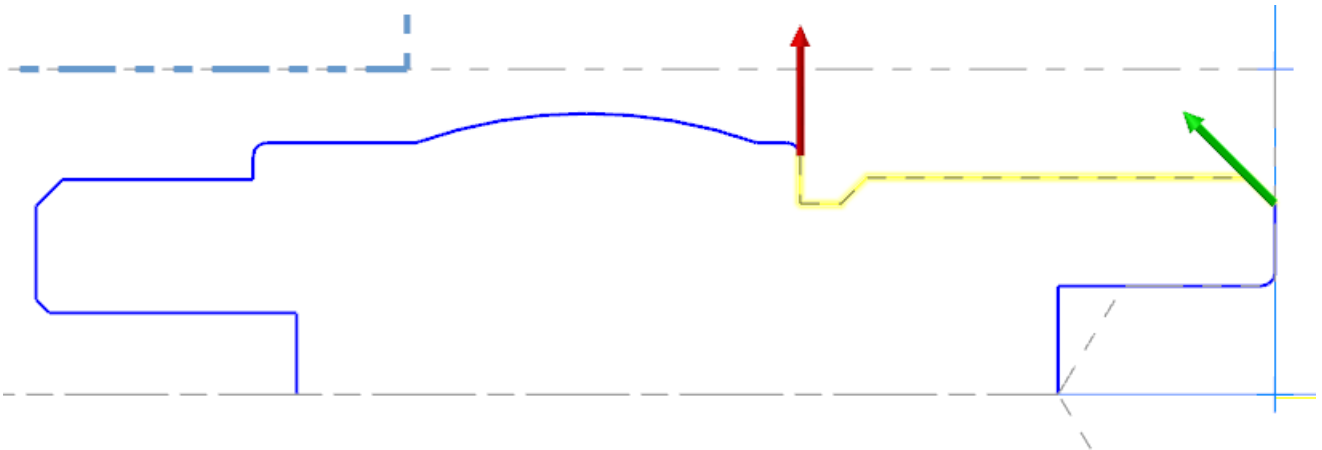
In this exercise, you program a finishing toolpath on the front side of the part in preparation for the threading operation in the next exercise.



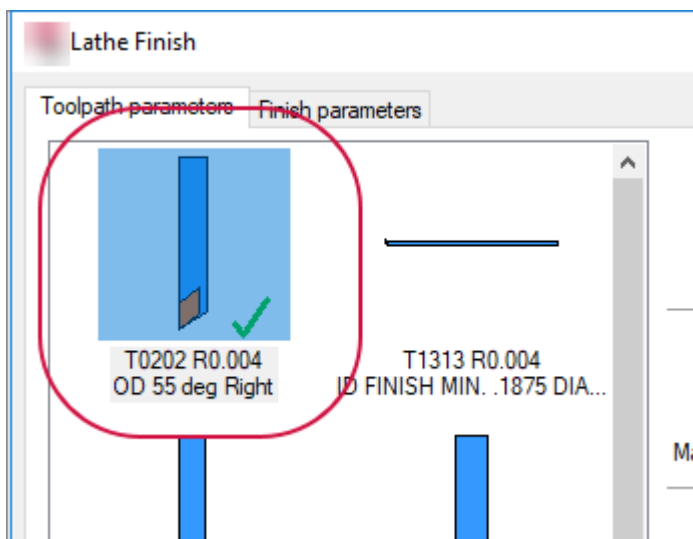
1. Select **Finish** from the **General** gallery on the **Lathe, Turning** contextual tab.



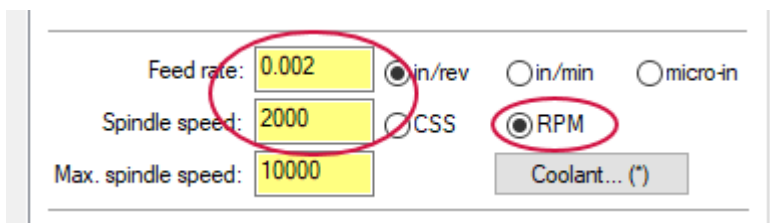
2. Chain the part as shown in the picture below, and click **OK** in the **Chaining** dialog box. If the chain points in the wrong direction, use the **Reverse** button to correct it.



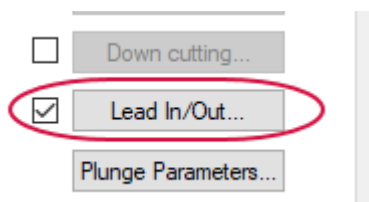
3. Select the **T0202 R0.004 OD 55 deg Right** tool.



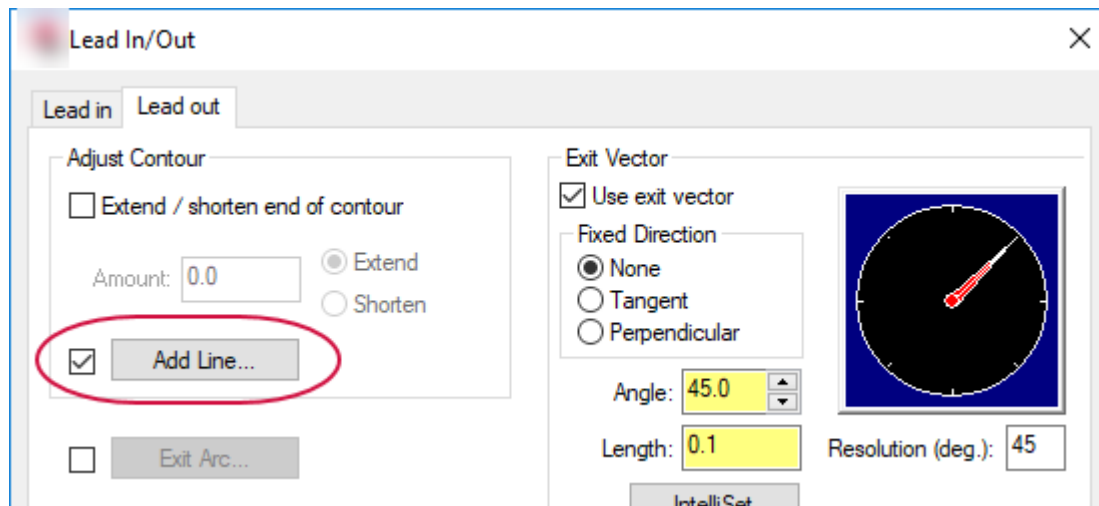
4. Set **Feed rate** to **0.002, in/rev** and **Spindle speed** to **2000, RPM**, as shown in the following picture.



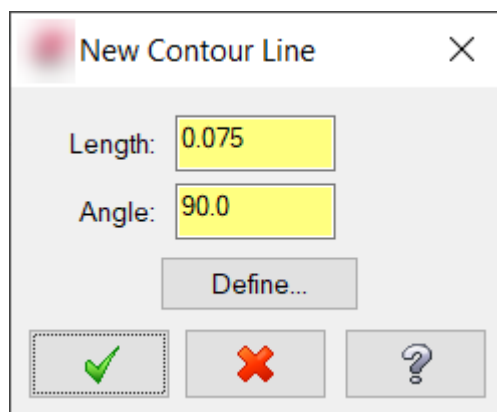
5. On the **Finish parameters** tab, select the **Lead In/Out** button.



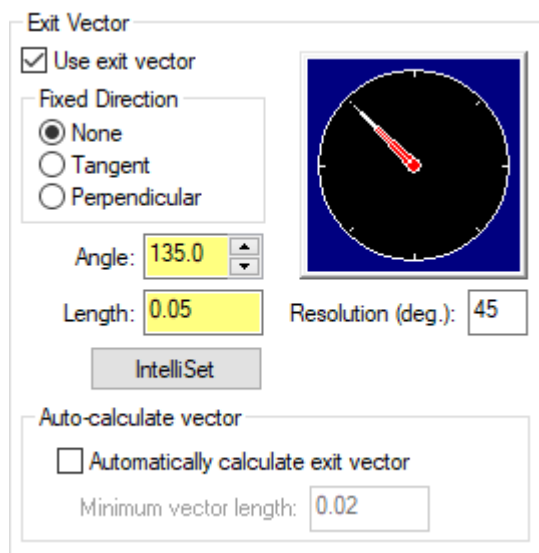
6. On the **Lead out** tab, select the checkbox next to **Add line**, and then click the **Add Line** button.



7. In the **New Contour Line** dialog box, set **Length** to **0.075** and **Angle** to **90**. Click **OK**. This allows you to create an edge break in the event of stock pullback.

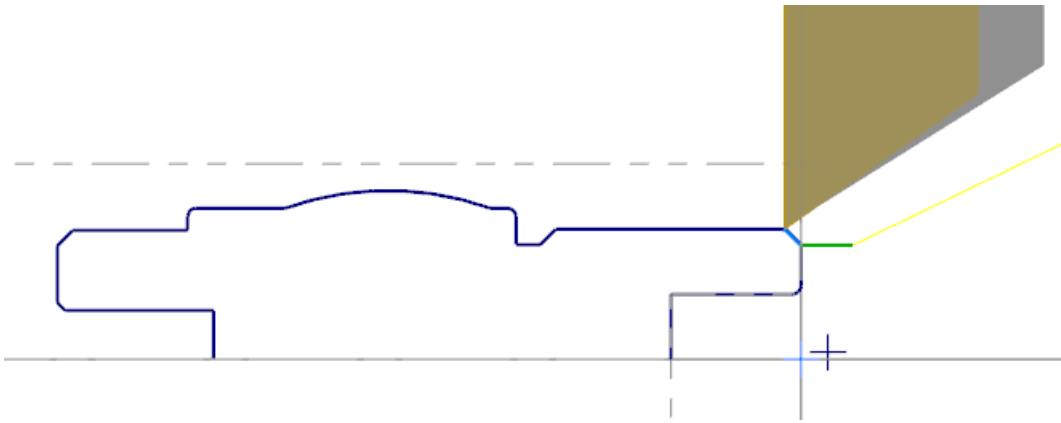


8. Back in the **Lead out** tab, set the **Angle** to **135** and **Length** to **0.05**.



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

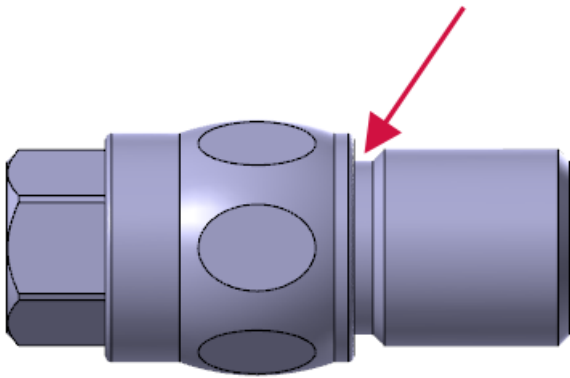
- Backplot the toolpath to see it in action.



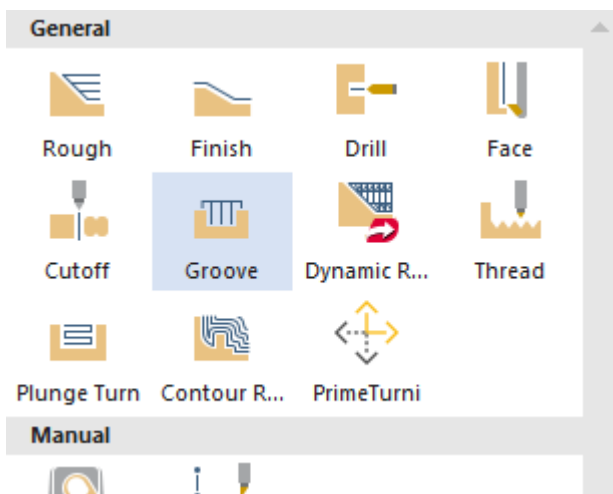
- Save the file with the name Swiss Tutorial_XX_6.mcam, where XX is your initials.

Exercise 6: Cutting the groove

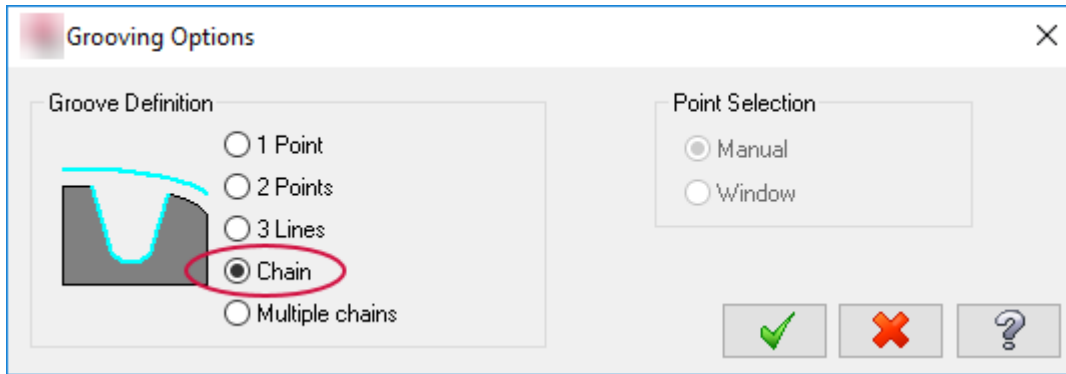
In this exercise, you program the groove toolpath, which cuts the small area between the front side and center side of the part.



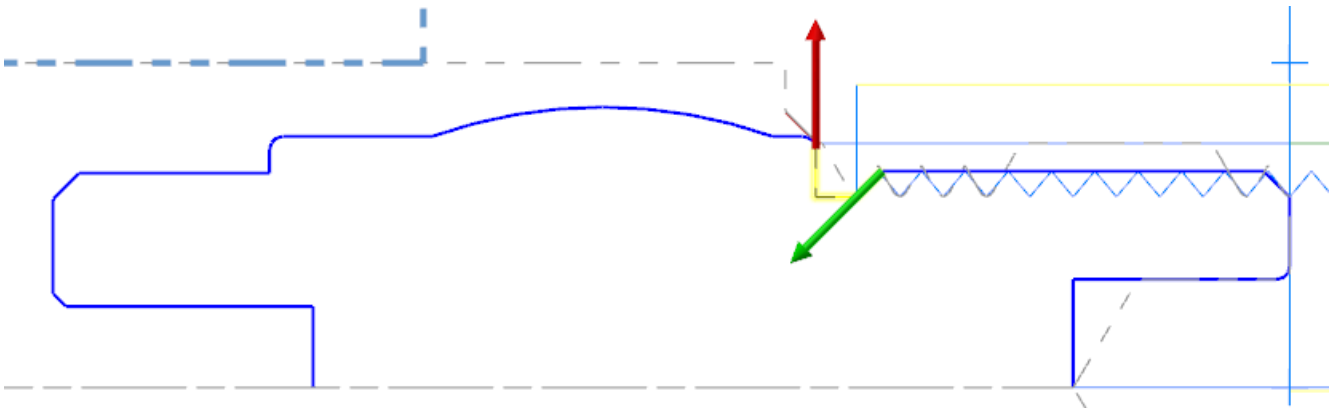
- Select **Groove** from the **General** gallery on the **Lathe, Turning** contextual tab.



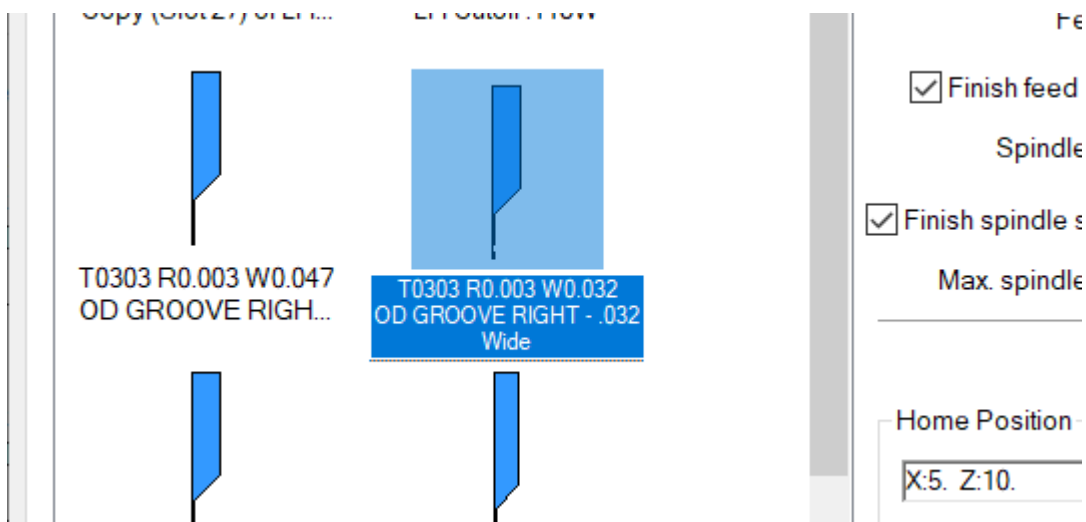
- In the **Grooving Options** dialog box, ensure that **Chain** is selected, and then click **OK**.



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



- Select the **T0303 R0.003 W0.032 OD GROOVE RIGHT - .032 Wide** grooving tool. (There are multiple T0303 tools. Be sure to pick the correct one.)



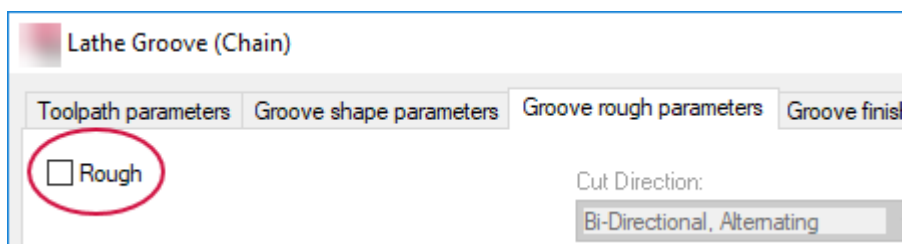
- Set the parameters as follows:
 - Set **Tool number** to **3**.
 - Set **Offset number** to **3**.
 - Set **Station number** to **3**.
 - Set **Feed rate** to **0.0025**.

- Set **Finish feed rate** to **0.002**.
- Set **Spindle speed** to **2000**.
- Set **Finish spindle speed** to **2000**.
- Set **Max spindle feed** to **5000**.

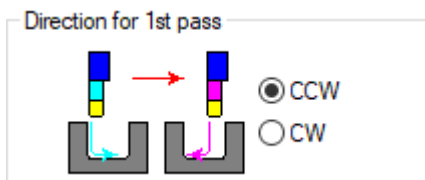
Tool number: Offset number:
 Station number: Tool Angle...

Feed rate: ☒ in/rev ☐ in/min ☐ micro-in
☒ Finish feed rate: ☒ in/rev ☐ in/min ☐ micro-in
 Spindle speed: ☐ CSS ☒ RPM
☒ Finish spindle speed: ☐ CSS ☒ RPM
 Max. spindle speed: Coolant... (*)

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



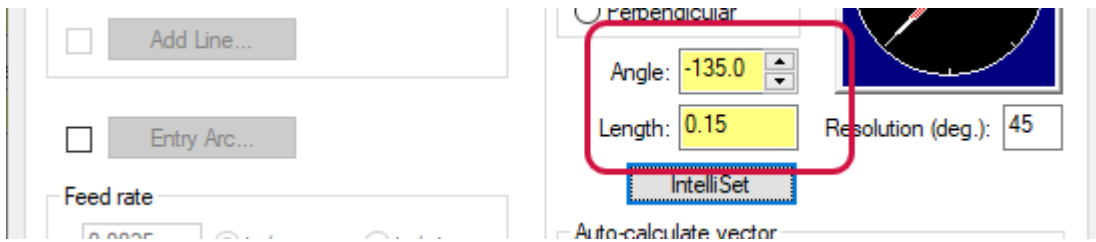
7. On the **Groove finish parameters** tab, change **Direction for 1st pass** to **CCW**.



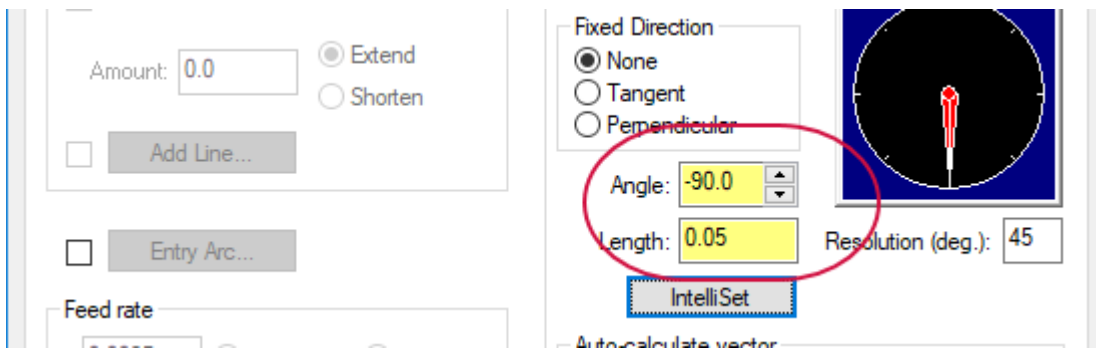
8. Click the **Lead In** button.



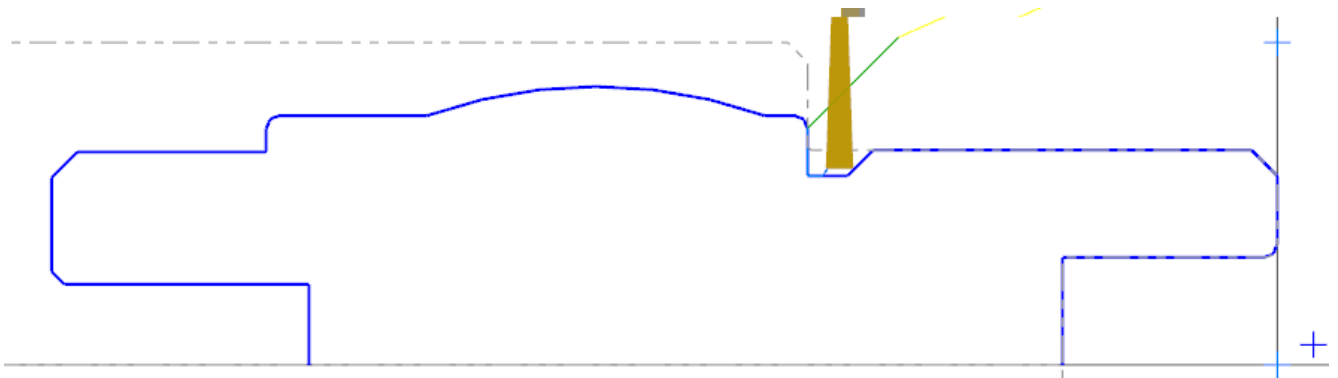
9. On the **First pass lead in** tab, set **Angle** to **-135.0** and **Length** to **0.15**.



10. 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** in both the **Lead In** and **Lathe Groove** dialog boxes.



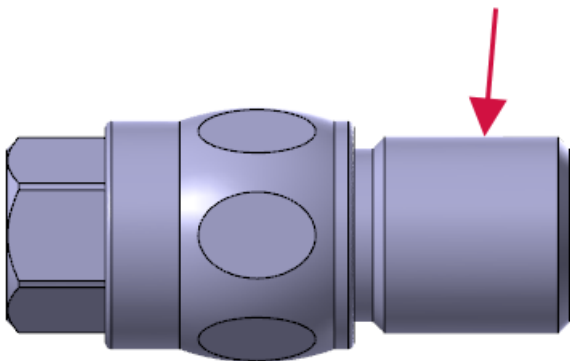
11. Backplot the new toolpath to see it in action.



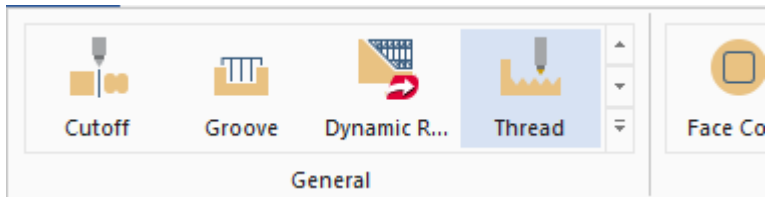
12. Save the file with the name Swiss Tutorial_XX_7.mcam, where XX is your initials.

Exercise 7: Cutting the 1/2-20 thread

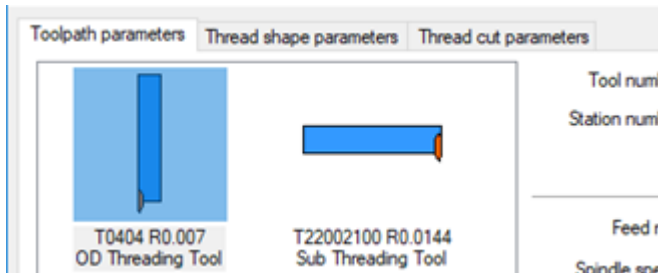
In this exercise, you cut threads onto the front side of the part, in the same area you finished in the previous exercise.



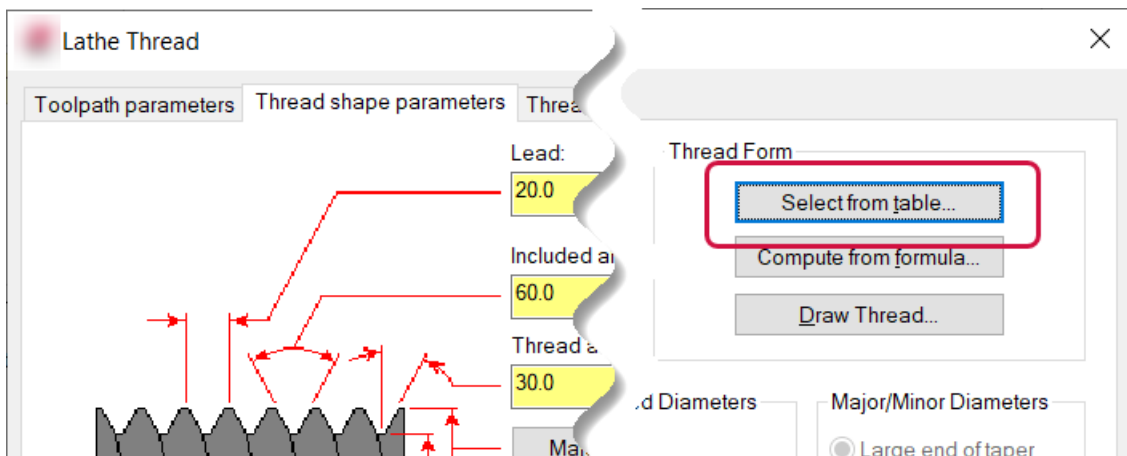
1. Select **Thread** from the **General** gallery on the **Lathe, Turning** contextual tab.



2. In the **Lathe Thread** dialog box, select tool **T0404**.



3. On the **Thread shape parameters** tab, click **Select from table**.



4. In the **Thread Table** dialog box, select the **1/2-20 thread**, and click **OK**.

Thread Table [X]

Thread form: Unified - UNRC, UNRF [v]


Common diameter/lead combinations up to 4 inches

Basic Major	Lead	Major Diameter	Minor Diameter	Comment
0.5000	13.0000	0.5000	0.4084	1/2, 13, UNRC
0.5000	20.0000	0.5000	0.4405	1/2, 20, UNRF
0.5625	12.0000	0.5625	0.4633	9/16, 12, UNRC
0.5625	18.0000	0.5625	0.4963	9/16, 18, UNRF
0.6250	11.0000	0.6250	0.5167	5/8, 11, UNRC
0.6250	18.0000	0.6250	0.5588	5/8, 18, UNRF
0.7500	10.0000	0.7500	0.6309	3/4, 10, UNRC
0.7500	16.0000	0.7500	0.6756	3/4, 16, UNRF
0.8750	9.0000	0.8750	0.7427	7/8, 9, UNRC
0.8750	14.0000	0.8750	0.7899	7/8, 14, UNRF
1.0000	8.0000	1.0000	0.8512	1, 8, UNRC
1.0000	12.0000	1.0000	0.9008	1, 12, UNRF
1.1250	7.0000	1.1250	0.9549	1-1/8, 7, UNRC
1.1250	12.0000	1.1250	1.0258	1-1/8, 12, UNRF
1.2500	7.0000	1.2500	1.0799	1-1/4, 7, UNRC
1.2500	12.0000	1.2500	1.1508	1-1/4, 12, UNRF

[✓] [✗] [?]

Mastercam fills the **Thread shape parameters** tab with the correct values from the table.

5. Enter **-0.5** into the **End Position** field.



Minor Diameter... 0.4405

Thread depth: 0.02975

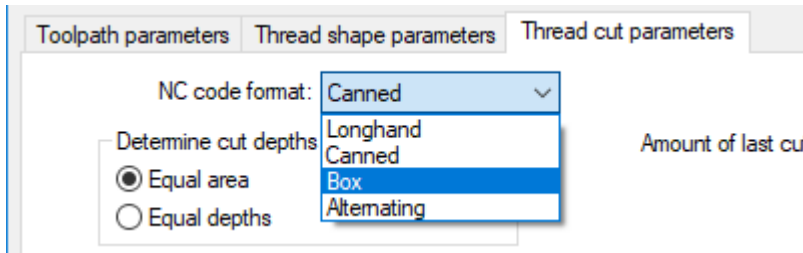
End Position... -0.5

Start Position... 0.0

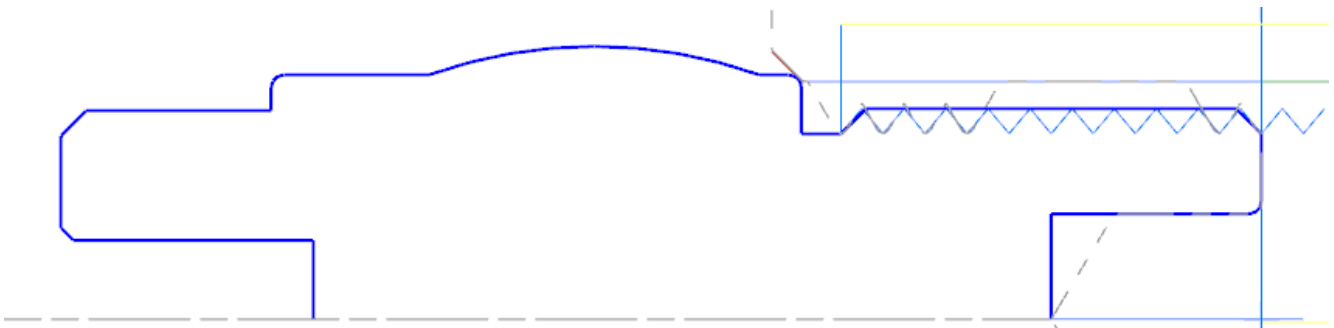
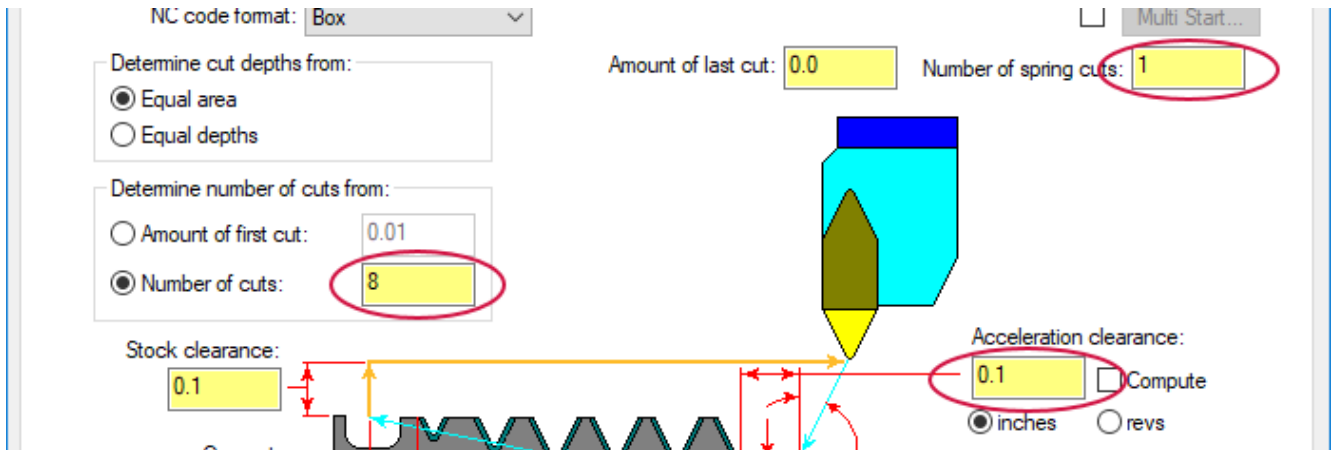
Thread orientation: OD [v] ☐ Cross centerline cut

Taper angle: 0.0

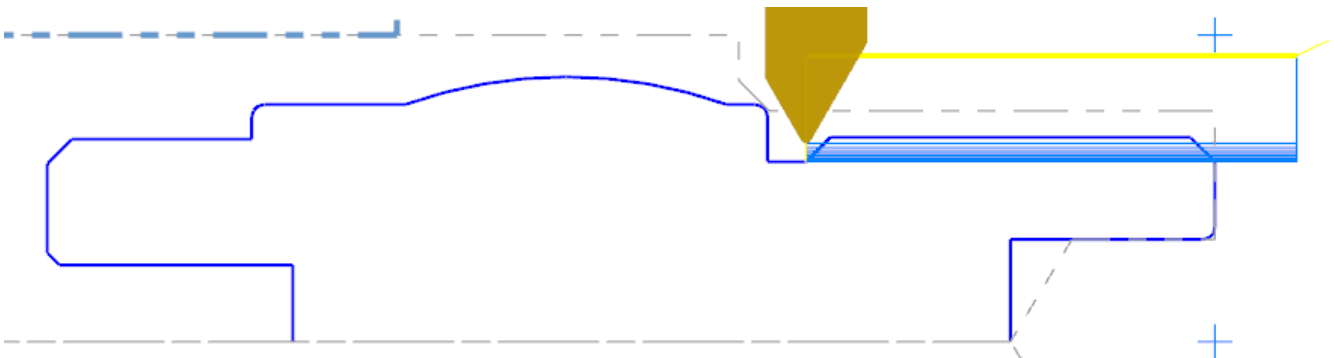
6. On the **Thread cut parameters** tab, select **Box** in the **NC code format** drop-down. **Box** is a G92, **Canned** is G76, and **Longhand** is G32.



7. 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.



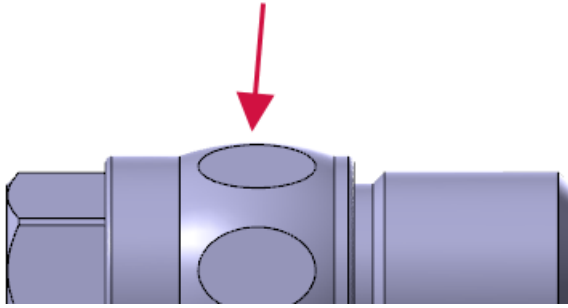
8. Backplot the toolpath to see it in action.



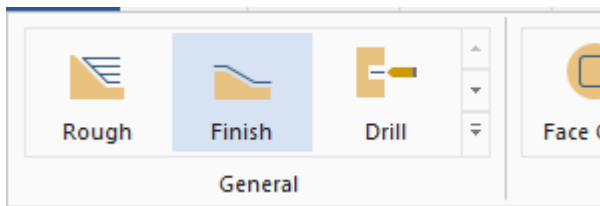
9. Save the file with the name Swiss Tutorial_XX_8.mcam, where XX is your initials.

Exercise 8: Finishing the center OD

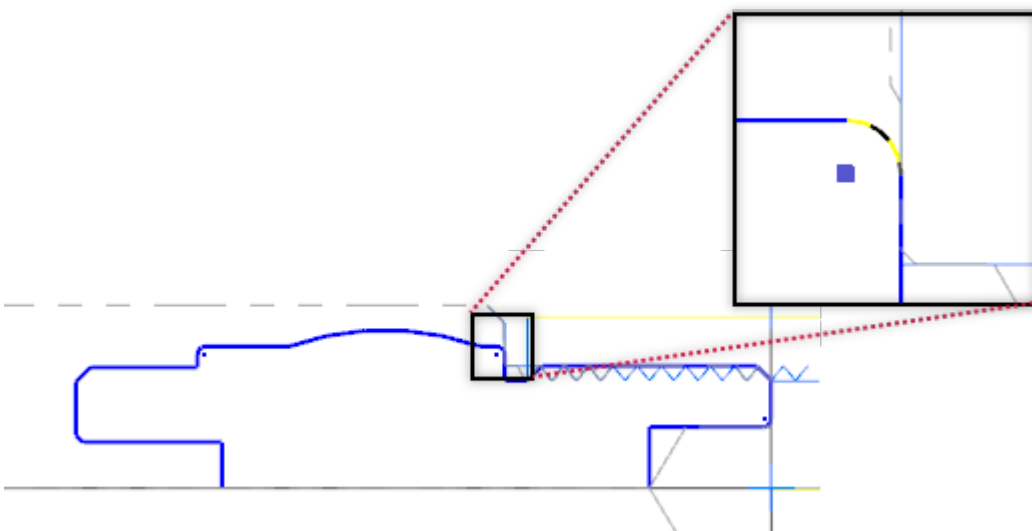
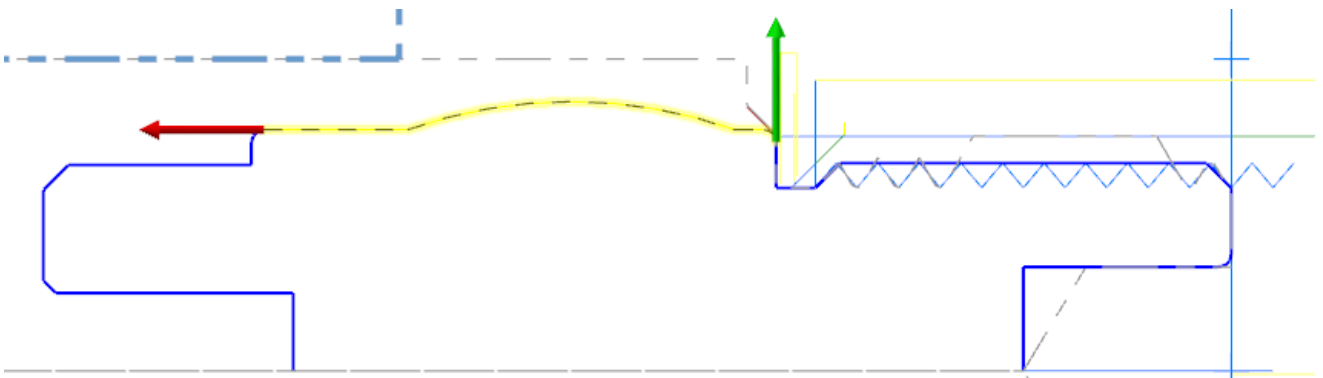
In this exercise, you program a finish cut on the center OD.



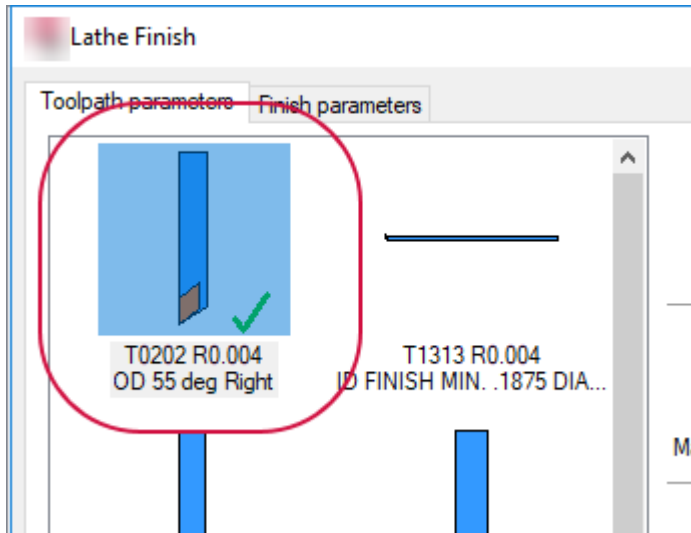
1. Select **Finish** from the **General** gallery on the **Lathe, Turning** contextual tab..



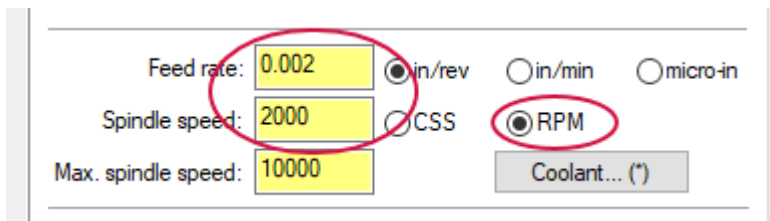
2. Chain the part as shown in the picture below, and click **OK** in the **Chaining** dialog box. Note that the first chaining selection should be the radius shown in the second picture following, and not the vertical wall.



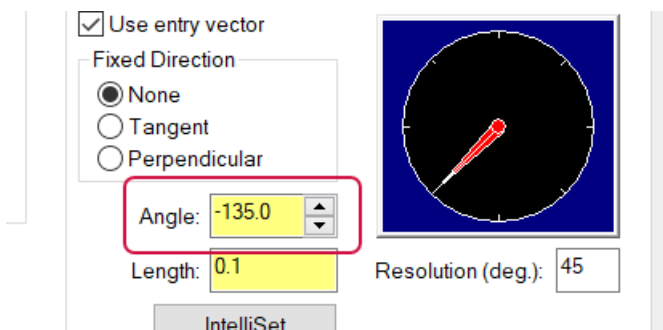
3. Select the **T0202 R0.004 OD 55 deg Right** tool.



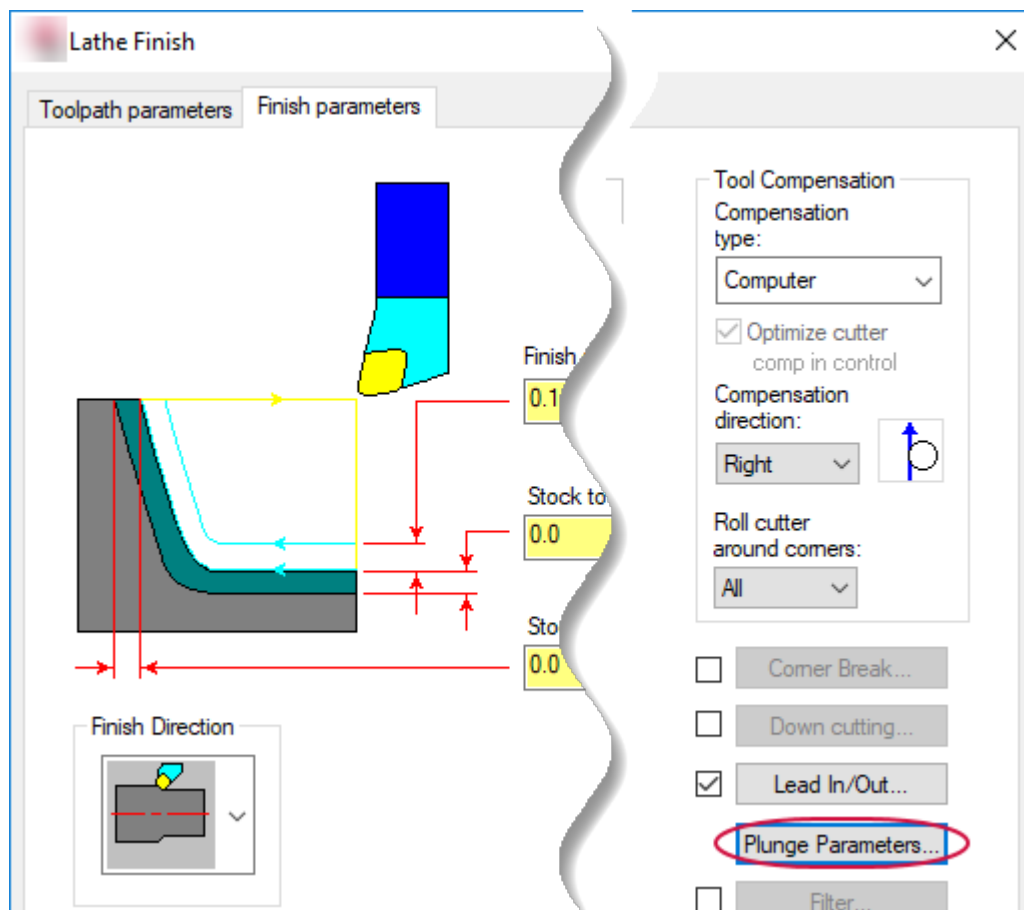
4. Set **Feed rate** to **0.002** and **Spindle speed** to **2000**.



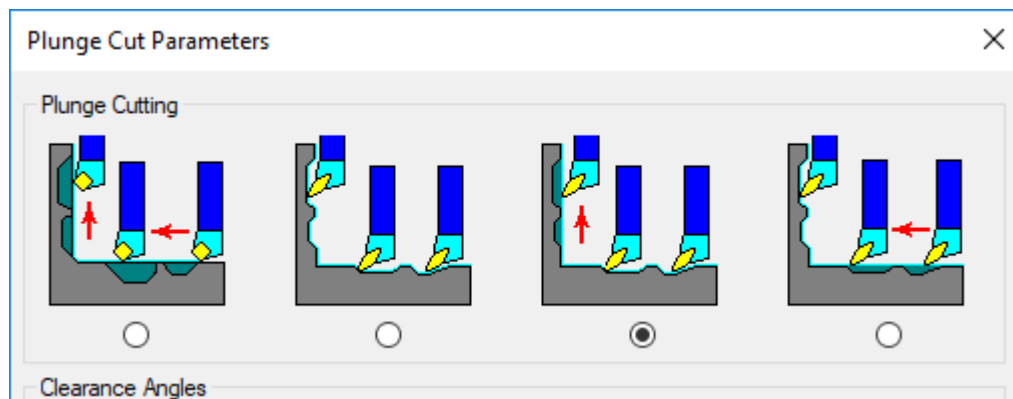
5. In the **Finish parameters** page, click **Lead In/Out**.
6. On the **Lead In** tab, change **Angle** to **-135.0**, and click **OK**.



7. Click **Plunge Parameters**.



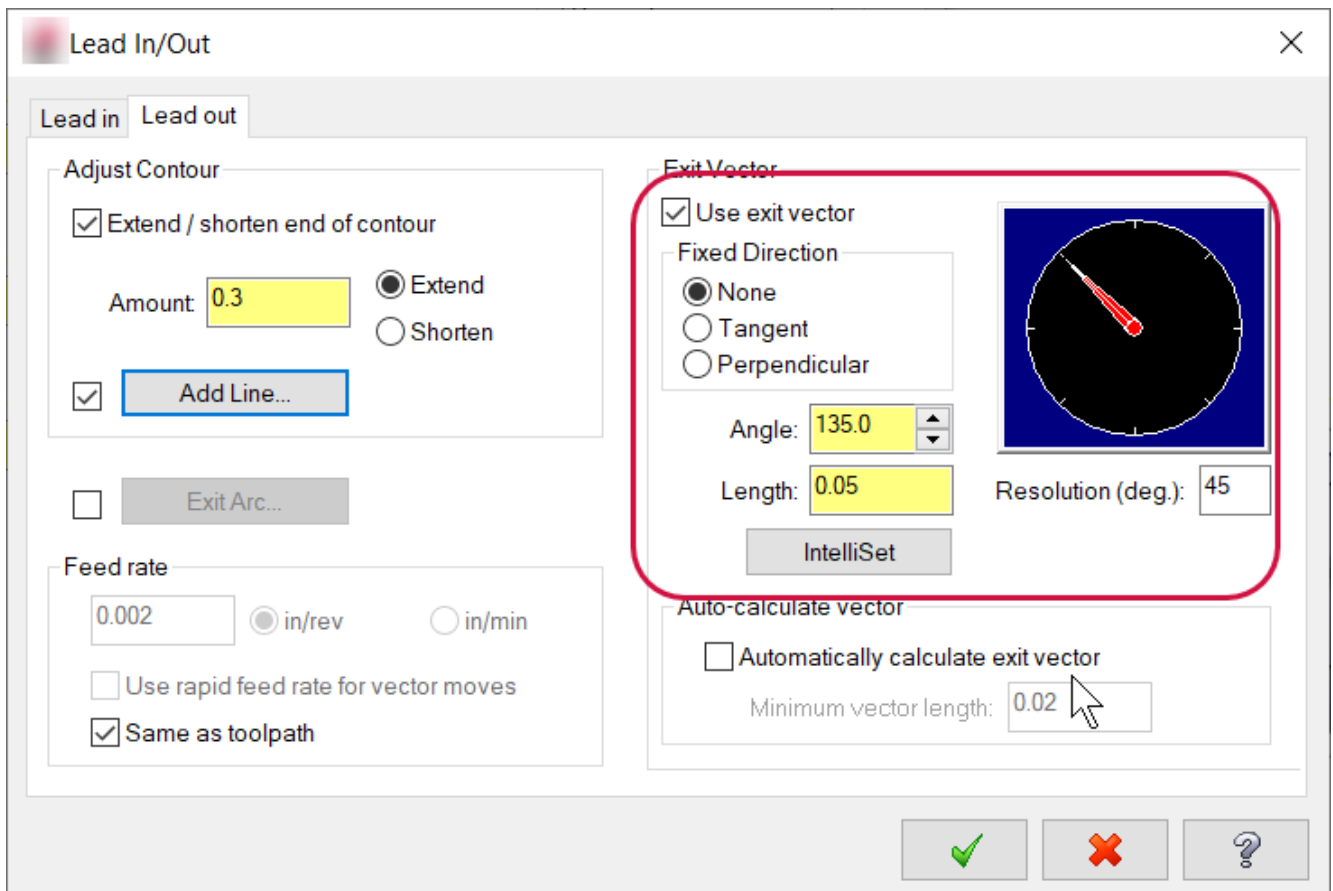
8. In the **Plunge Cut Parameters** dialog box, select the third option, and click **OK**.



9. Make the following parameter changes:

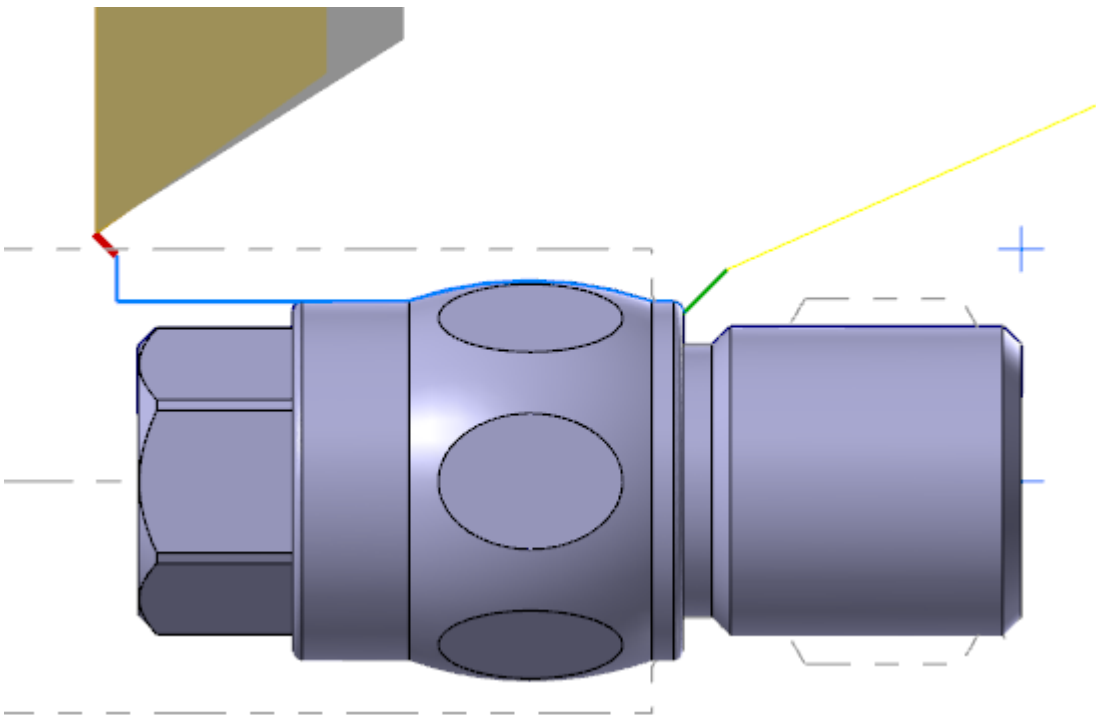
- In the **Finish Parameters** tab, click **Lead In/out**.
- On the **Lead out** tab, select the **Extend / shorten end of contour** option.
- Enter **0.3** in **Amount**.
- Enable the **Add Line** button by clicking its checkbox, and then click the button.
- In the **New Contour Line** dialog box, enter **0.075** for **Length**, enter **90** for **Angle**, and click **OK**.

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



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

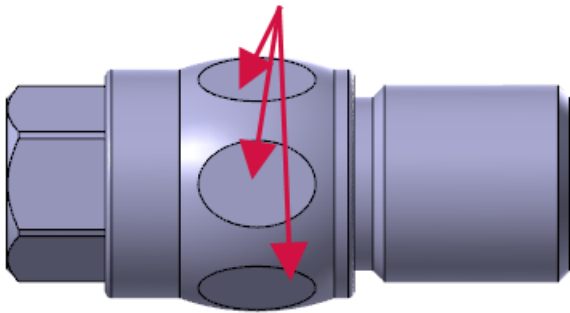
12. Backplot the toolpath.



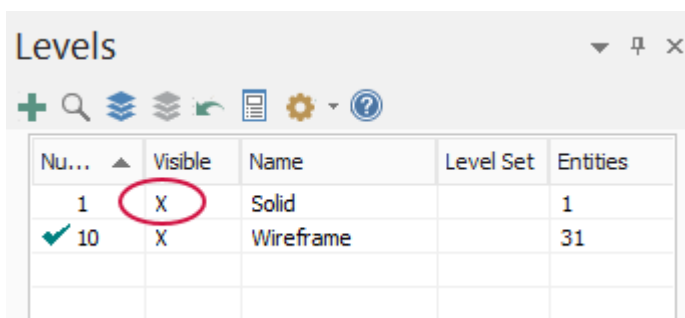
13. Save the file with the name **Swiss Tutorial_XX_9.mcam**, where **XX** is your initials.

Exercise 9: Cutting the six flat areas

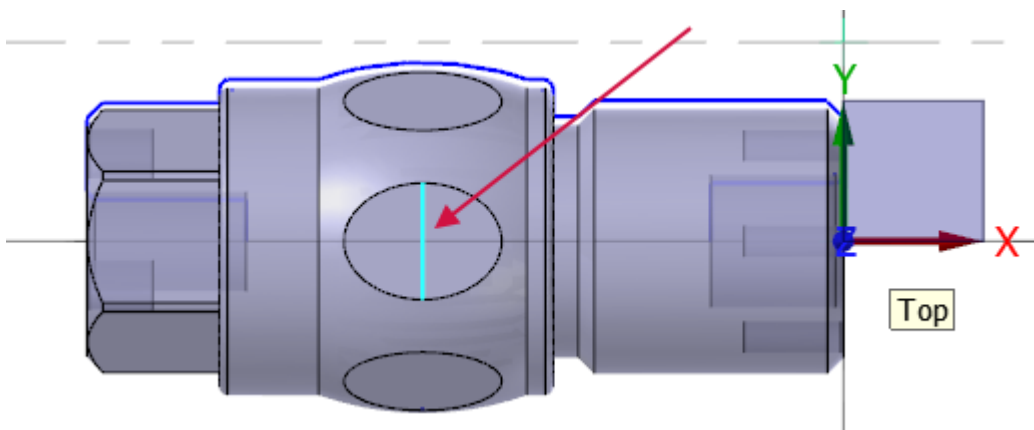
In this exercise, you program a milling toolpath for the six flat areas on the center OD. You will use a Contour toolpath, along with the Toolpath Transform function, to perform six cutting passes 60 degrees apart.



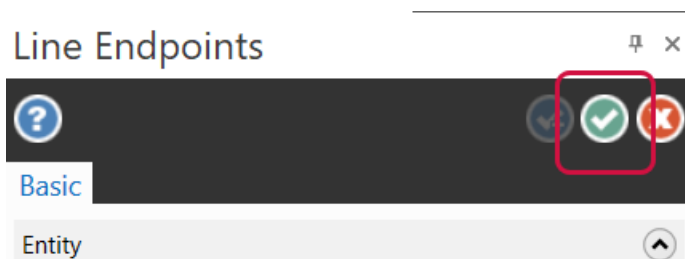
1. To display the part's solid model, in the Levels Manager, select the **Visible** column for level number 1.



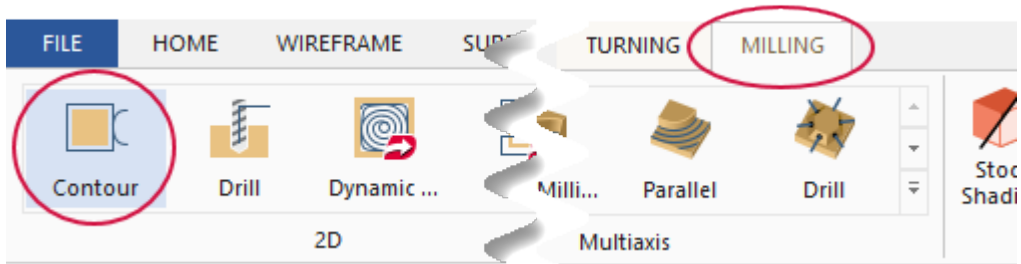
2. Select **Line Endpoints** from the **Wireframe** tab, and draw the line shown in the following image. Use the arc midpoints as anchors for the ends of the line.



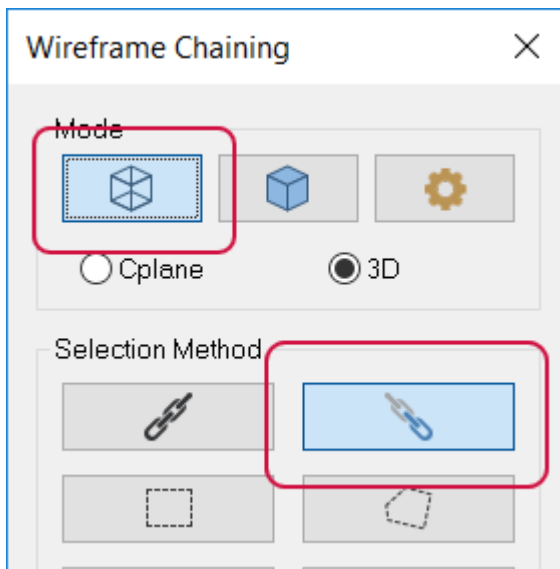
3. In the **Line Endpoints** function panel, click **OK**.



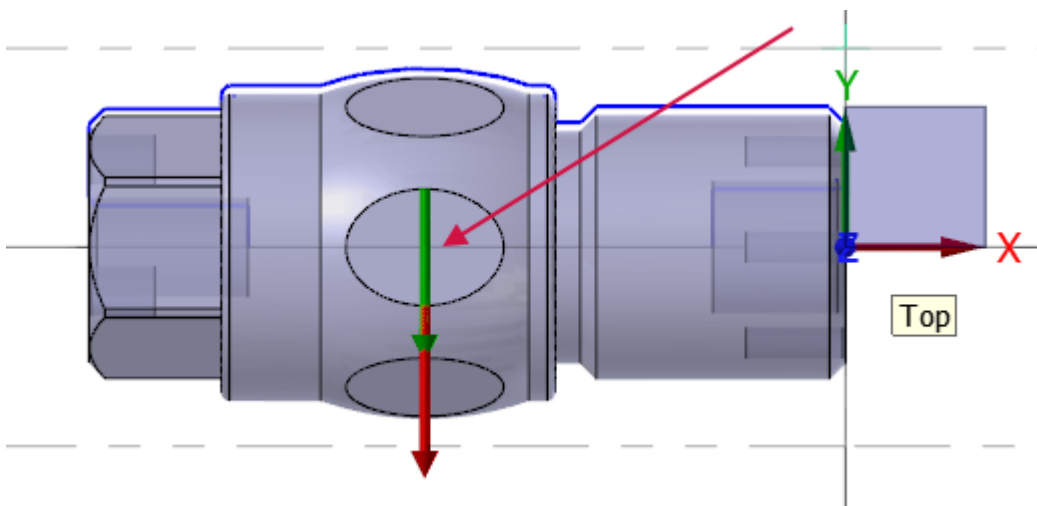
4. Select **Contour** from the 2D gallery on the **Lathe, Milling** contextual tab.



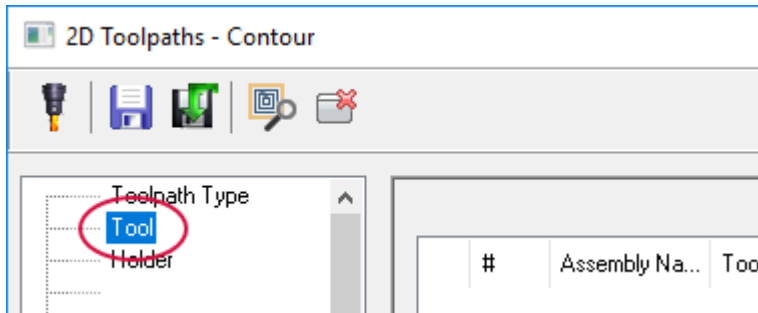
5. In the **Chaining** dialog box, select **Wireframe** and then **Partial**.



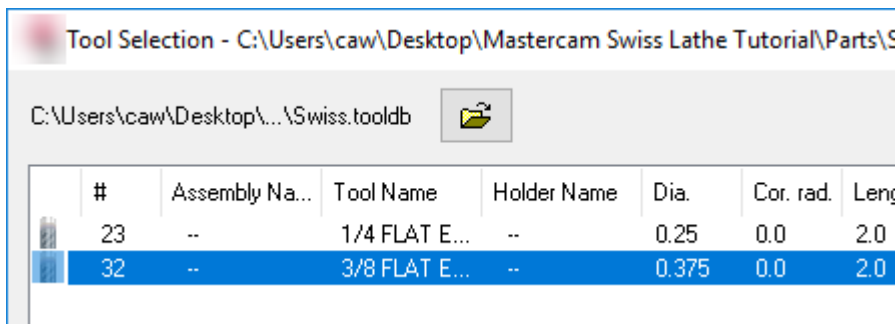
6. Select the line you created in Step 2, and click **OK** in the **Chaining** dialog box.



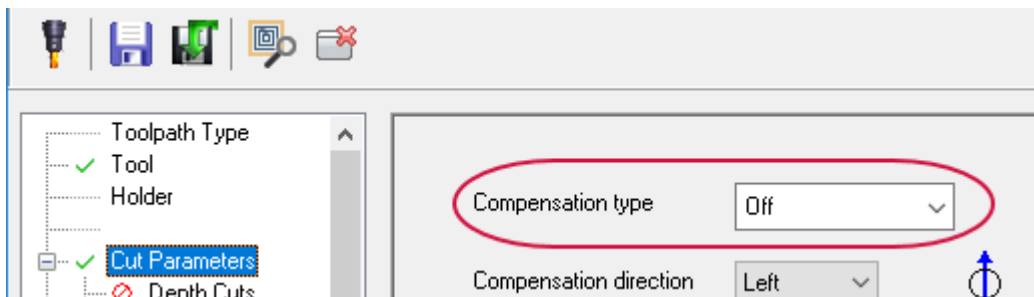
7. In the **2D Toolpaths - Contour** dialog box, click **Tool**.



8. Click **Select library tool**, choose a 3/8 flat endmill, and click **OK**. Note that, to see more tools, you might have to uncheck the **Filter Active** option.



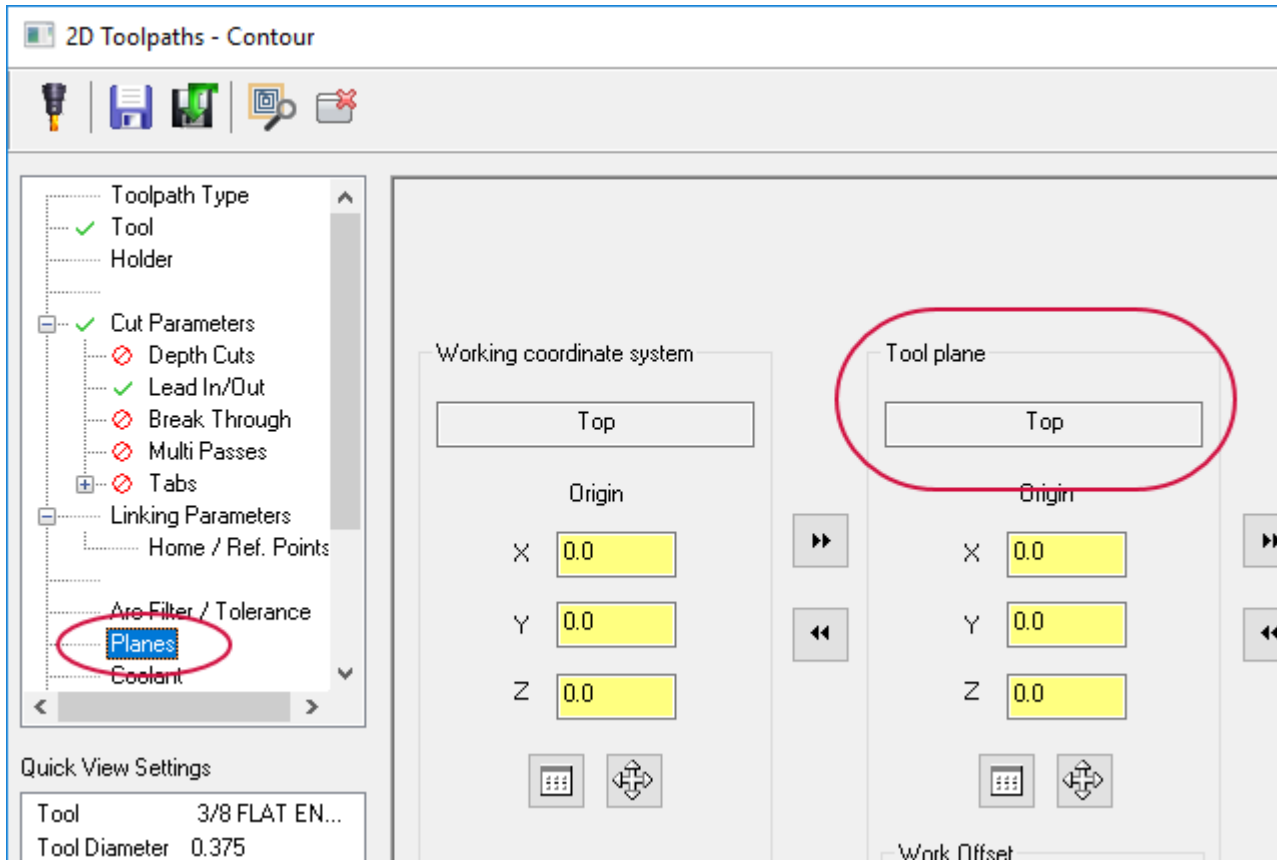
9. Select the **Cut Parameters** page, and set **Compensation type** to **Off**.



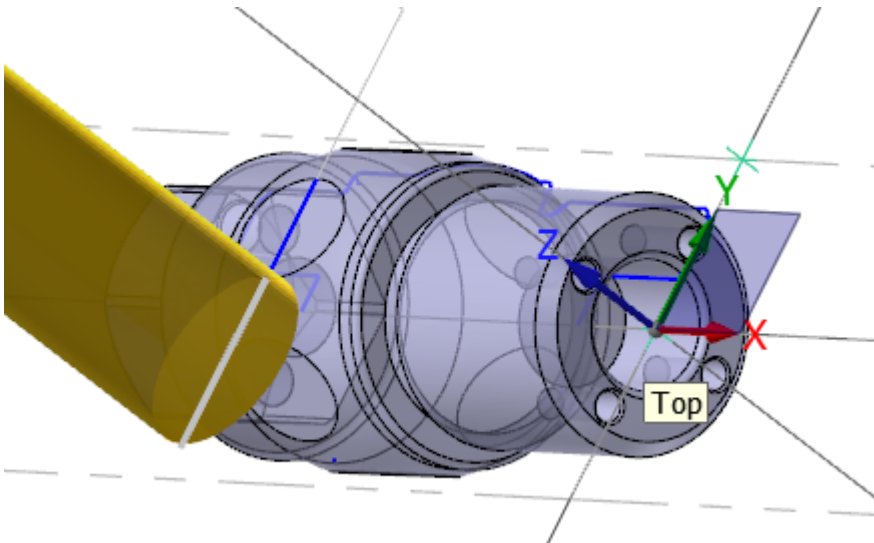
10. Select the **Lead In/Out** page, and enter the following values for both arcs indicated below.
- Set **Radius** to **0.0**.
 - Set **Helix height** to **0.0**.



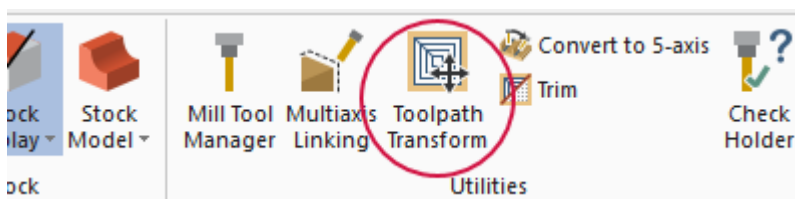
11. Select the **Planes** page, ensure that **Tool plane** is set to **Top**, and click **OK**.



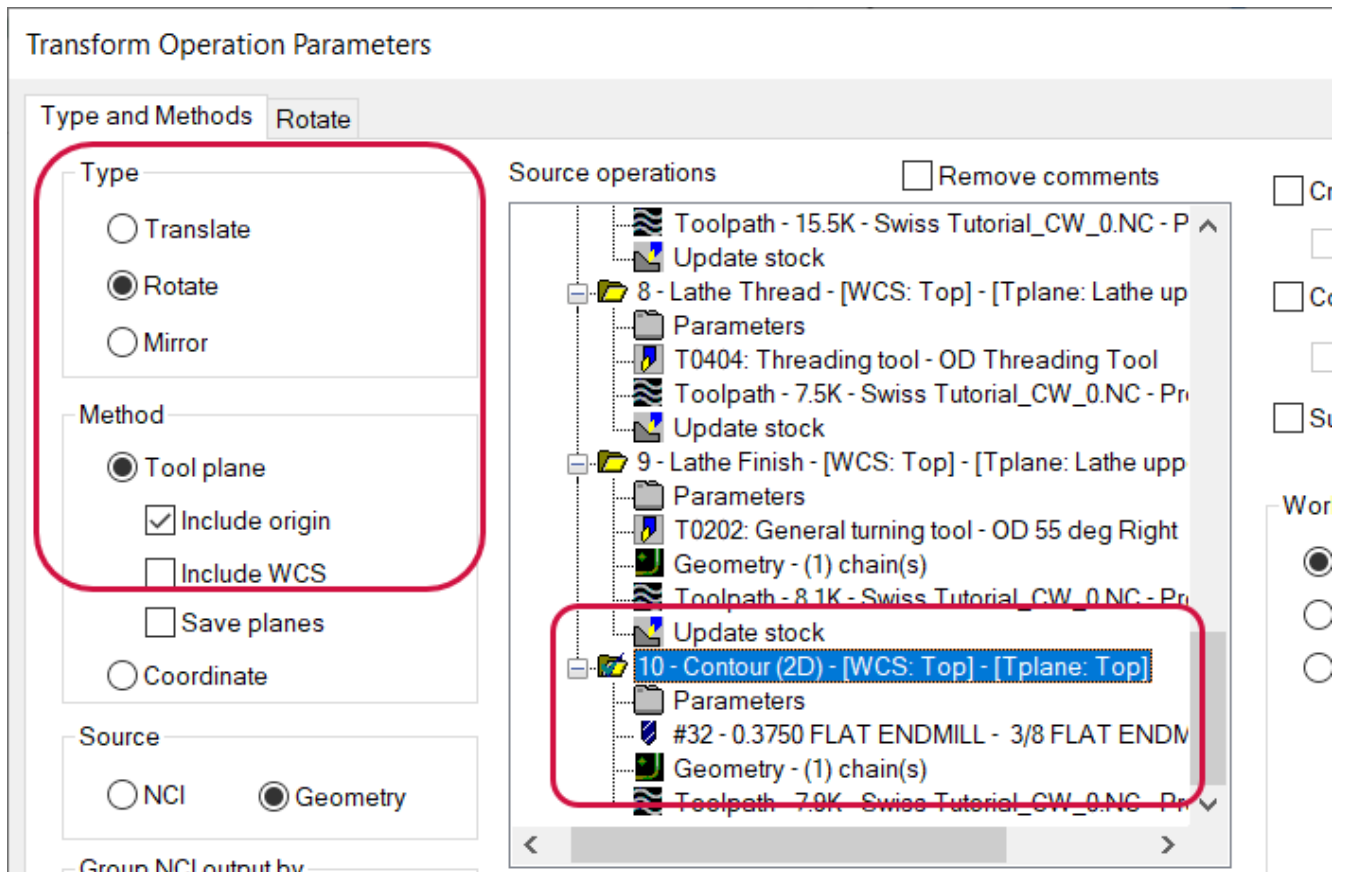
12. Run Backplot to see how the operation cuts the first flat area.



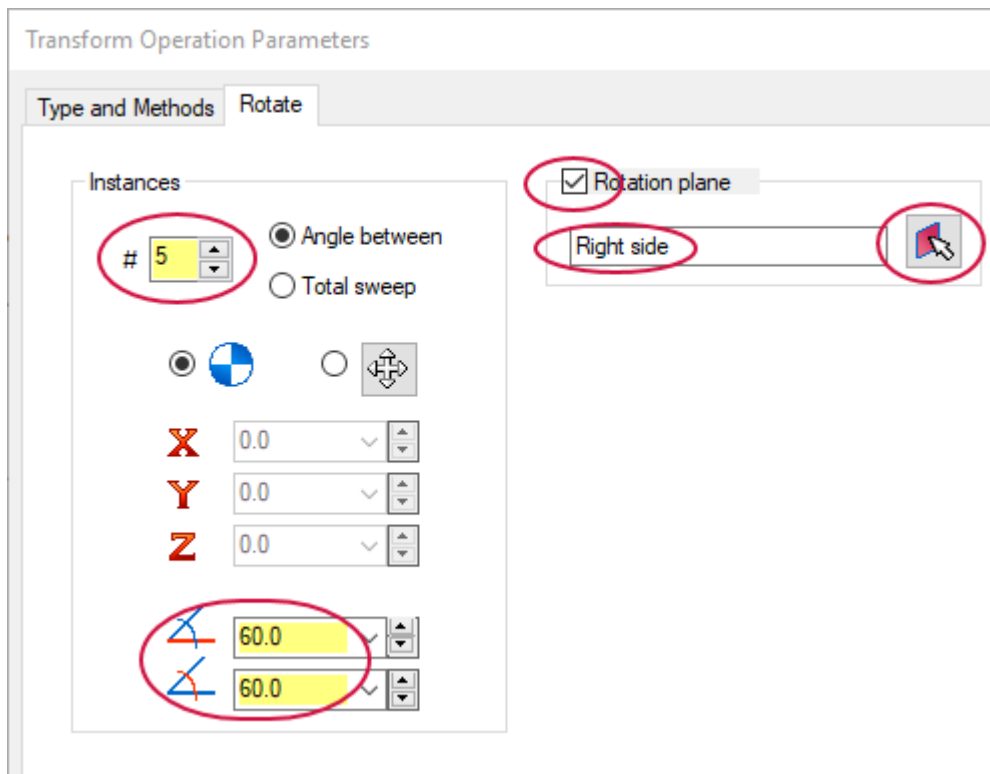
13. On the **Lathe Milling** contextual tab, select **Toolpath Transform**.



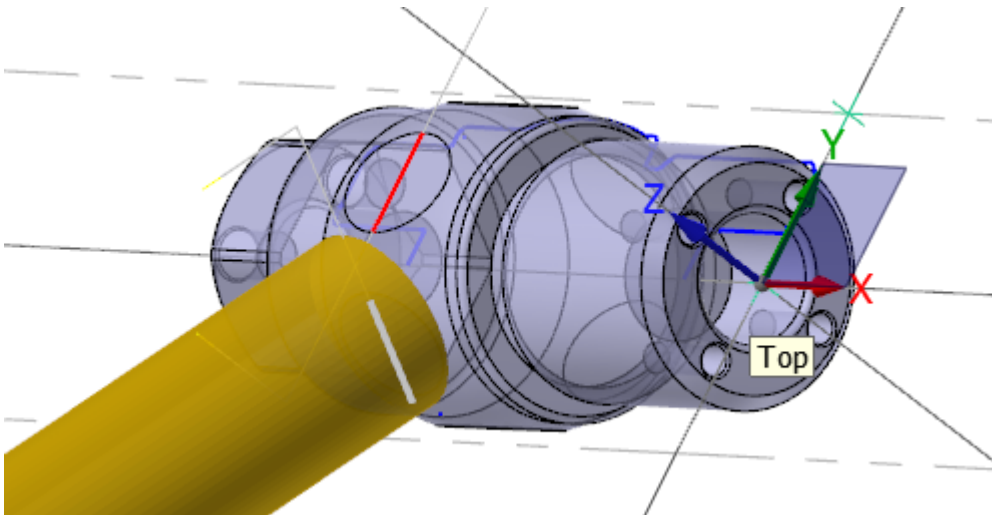
14. In the **Transform Operation Parameters** dialog box, select **Rotate** and **Tool plane/Include origin**. Also, ensure that the Contour operation is selected.



15. 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**.



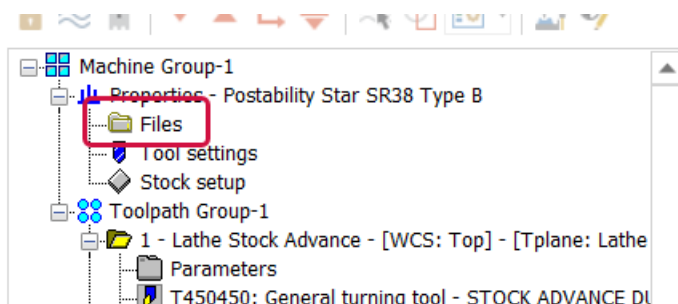
- In Toolpaths Manager, select the two operations you created in this exercise, and run Backplot to see how the operations cut all of the flat areas.



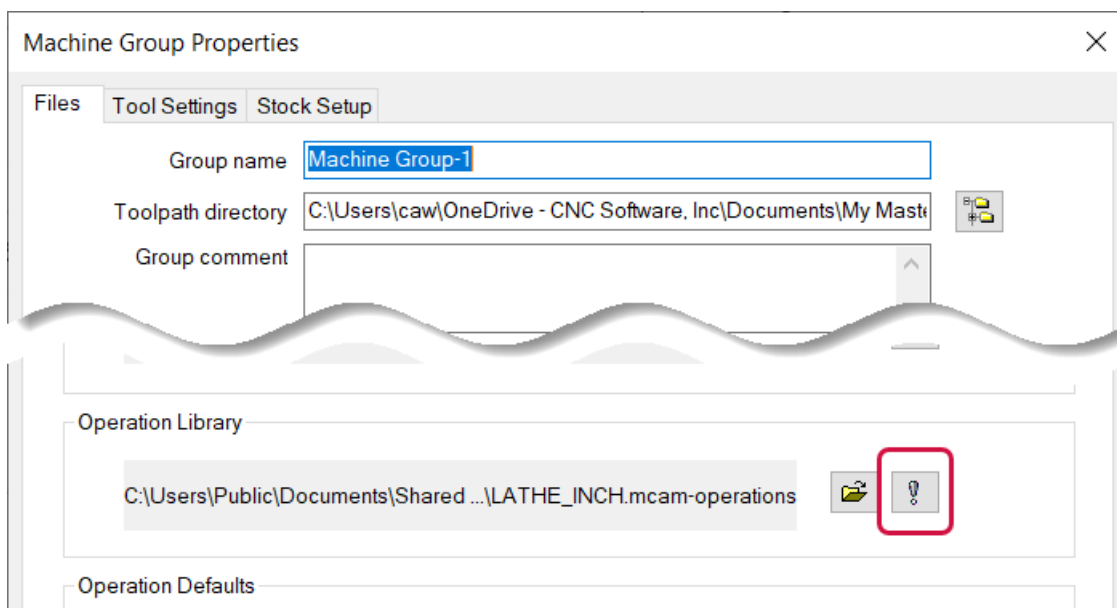
- Save the file with the name Swiss Tutorial_XX_10.mcam, where XX is your initials.

Exercise 10: Editing the Cutoff operation

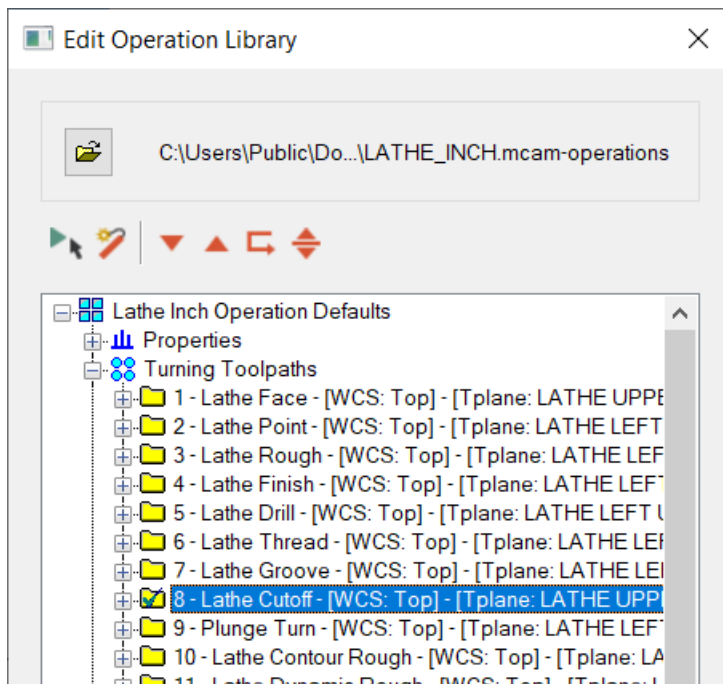
- In the Toolpaths manager, click the **Files** icon.



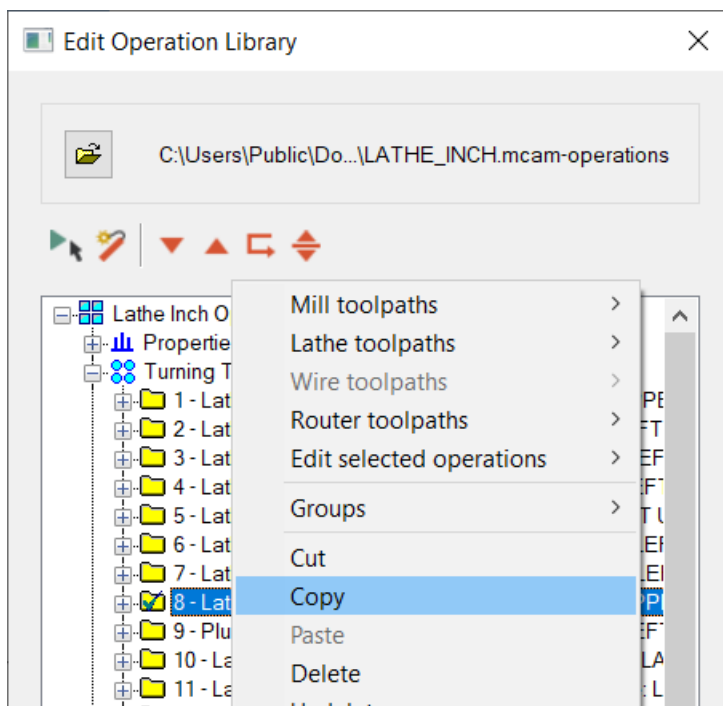
- Click the exclamation point in the **Operation Library** group.



3. In the **Edit Operation Library** dialog box, expand **Turning Toolpaths** and select **Lathe Cutoff**.

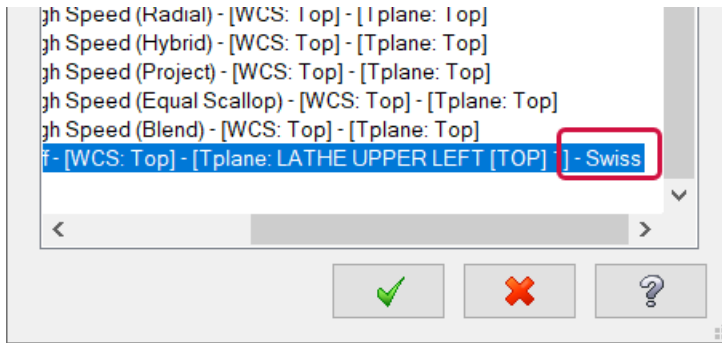


4. Right-click **Lathe Cutoff**, and select **Copy** from the pop-up menu.

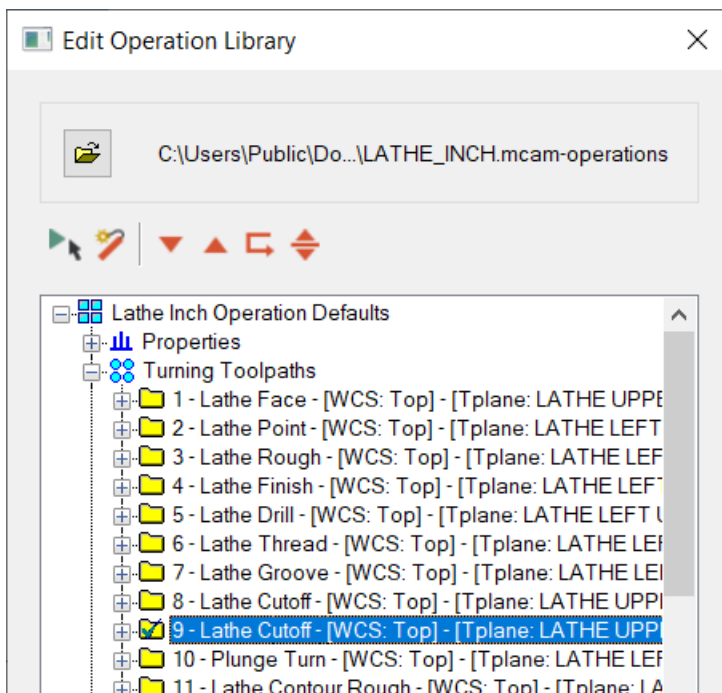


5. Right-click again, and select **Paste**.

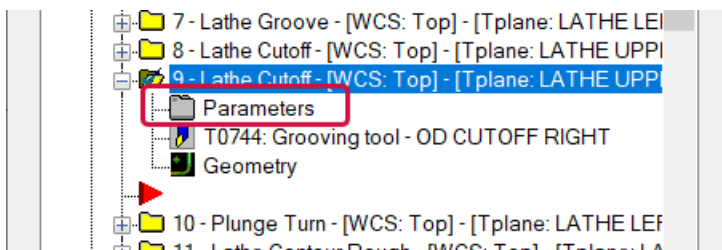
6. Click the new copy, type **Swiss**, and press **[Enter]**. Mastercam adds the word Swiss to the end of the new operation's name.



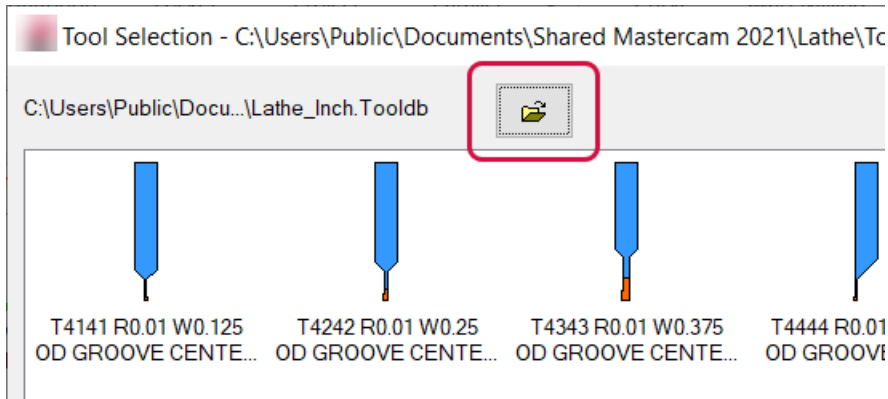
7. If necessary, drag the new operation into the **Turning Toolpaths** folder.



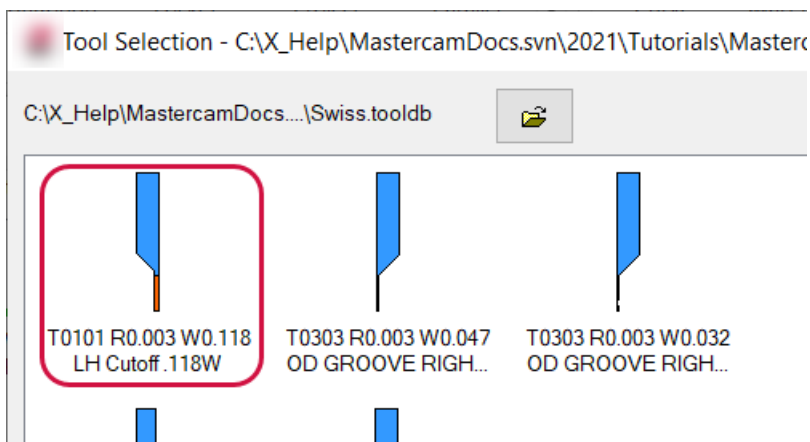
8. Click the plus sign to the left of the new operation. The operation opens to show its parameters, tool, and geometry icons.
9. Click the **Parameters** icon.



10. In the **Toolpath parameters** page of the **Lathe Cutoff** dialog box, click **Select library tool**, and in the **Tool Selection** dialog box, click the folder button.



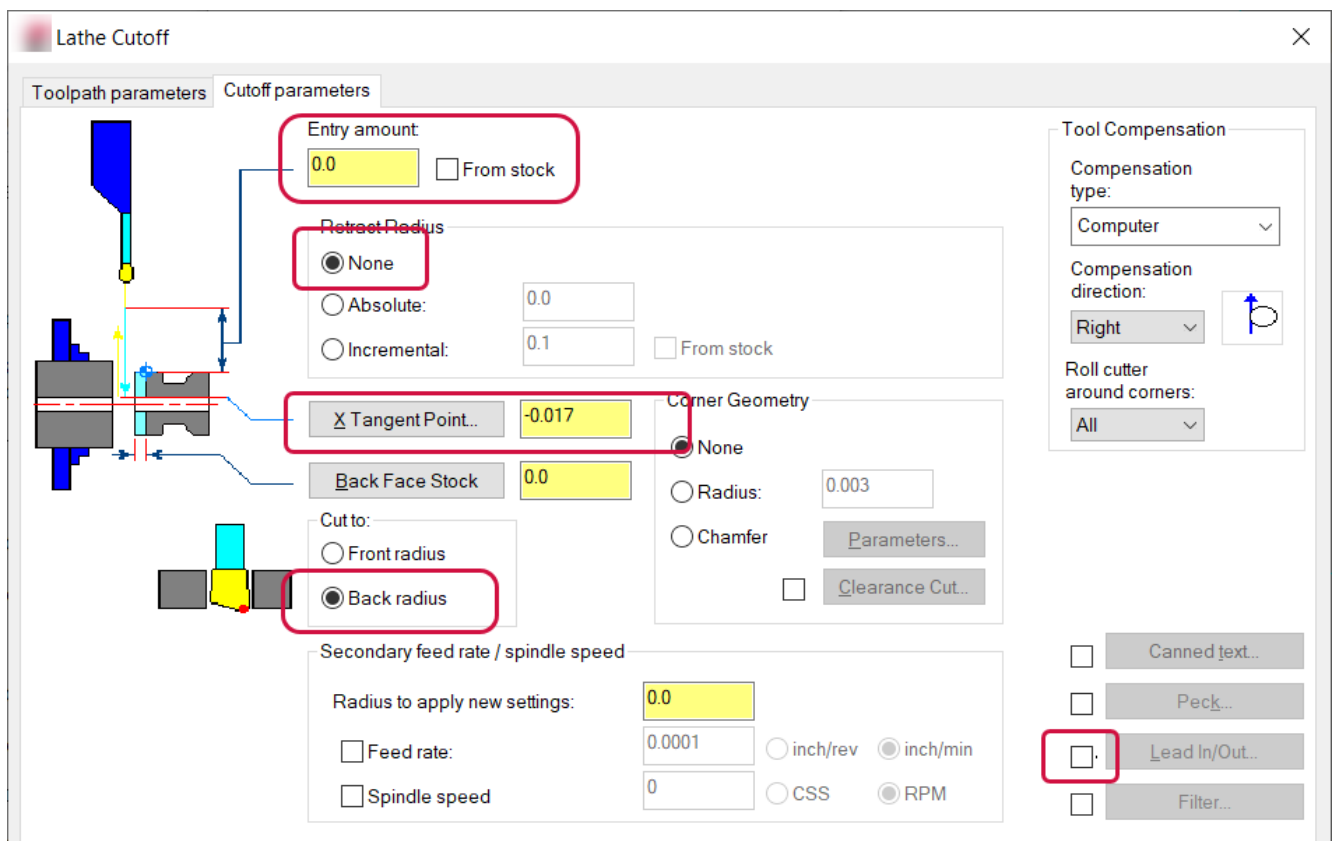
11. Load the Swiss.tooldb file that was supplied with this tutorial, select the **T0101 R0.003 W0.118** tool shown in the following image, and click **OK**. You might see multiple T0101 tools, so be sure to select the correct one.



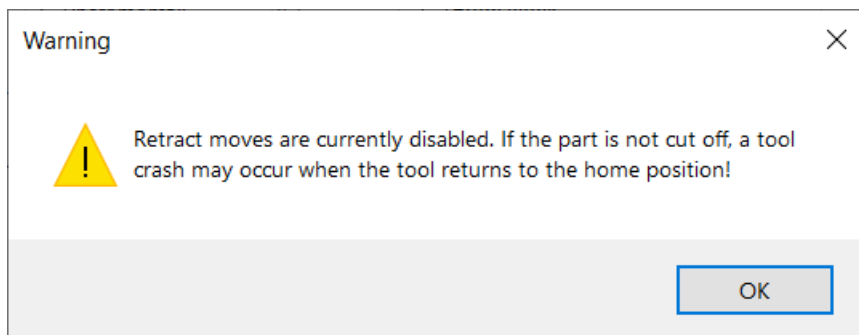
12. Set the toolpath parameters shown below.

13. Select the **Cutoff parameters** tab, and make the following changes.
- Set **Entry amount** to **0.0**.
 - Set **Retract Radius** to **None**.
 - Set **X Tangent Point** to **-0.017**.

- Set **Cut to** to **Back radius**.
- Turn off the **Lead In/Out** option.



- Click **OK** in the **Lathe Cutoff** dialog box, and click **OK** in the warning message that appears.

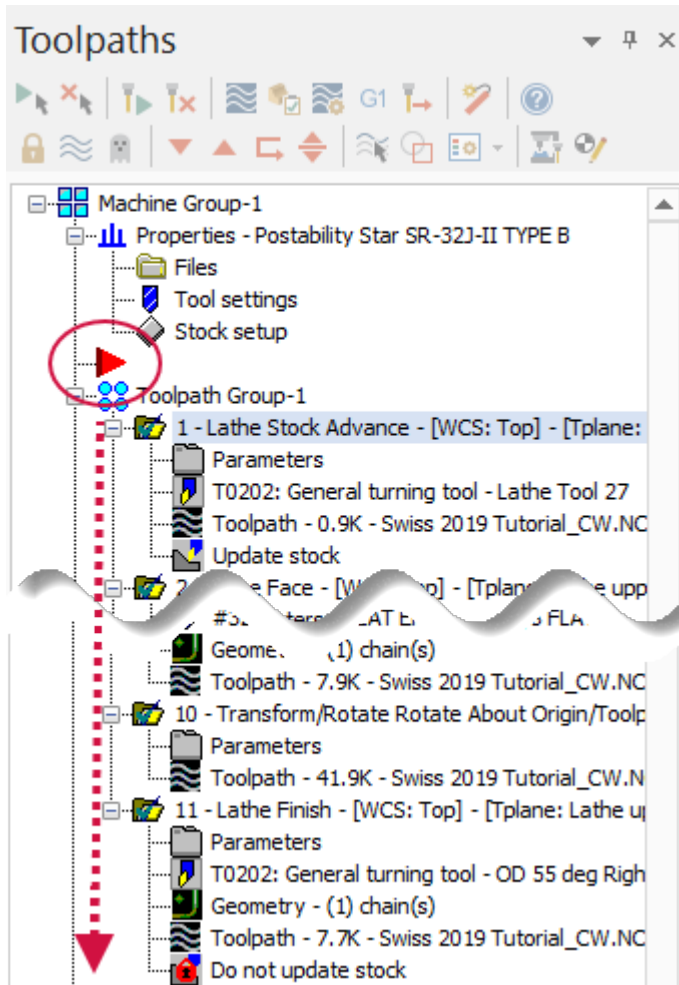


- Click **OK** in the **Edit Operation Library** dialog box and in the **Machine Group Properties** dialog box.
- Save the file with the name Swiss Tutorial_XX_11.mcam, where XX is your initials.

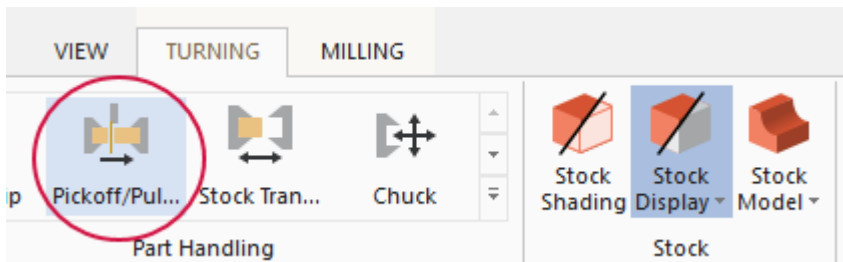
Exercise 11: Adding the Cutoff operation

In this exercise, you create the operation that cuts the part from the bar stock in preparation for transferring the part to the subspindle.

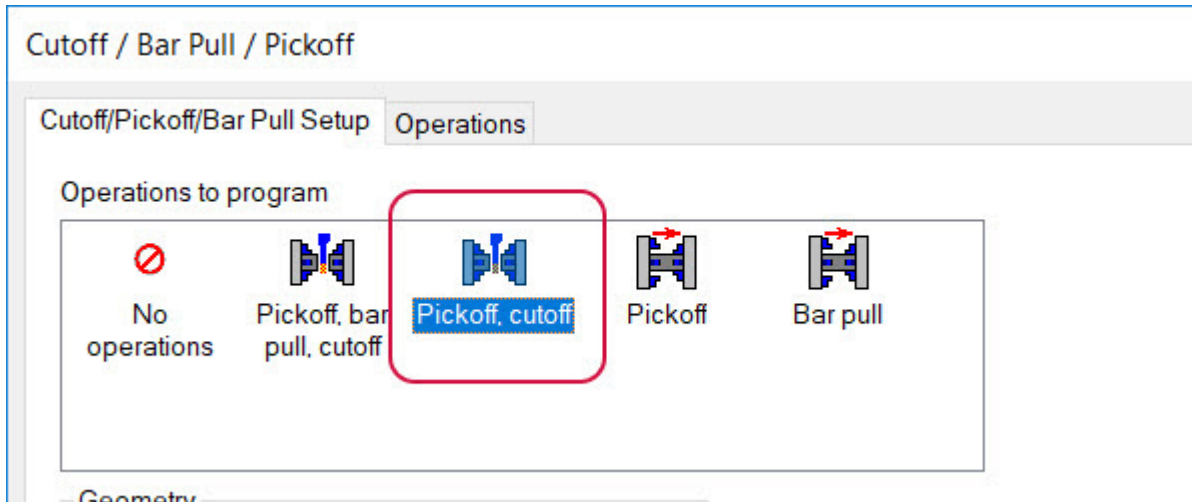
1. If necessary, drag the red arrow to below all of the operations.



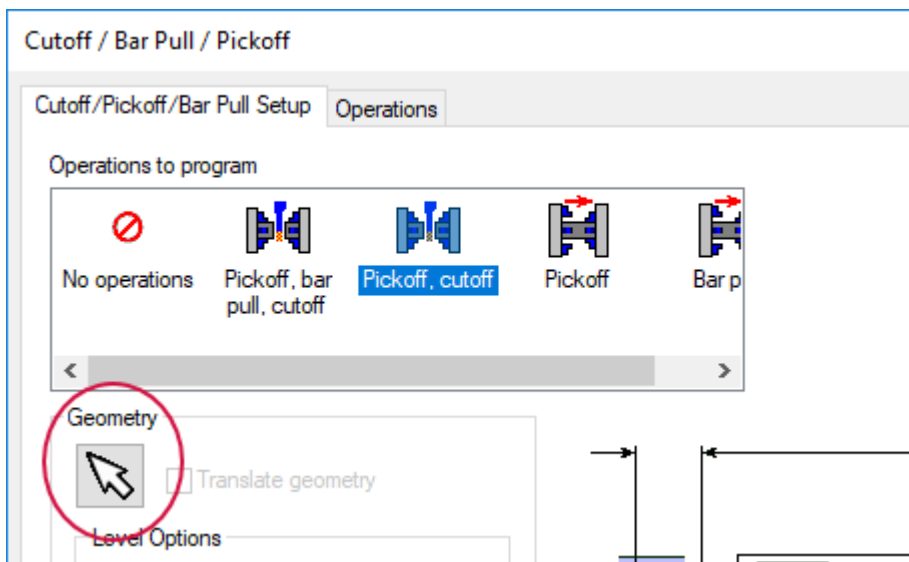
2. On the **Lathe, Turning** contextual tab, select **Pickoff/Pull/Cutoff** from the **Part Handling** gallery.



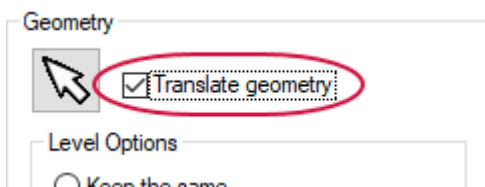
3. In the dialog box, choose **Pickoff, cutoff**. Note that **Pickoff, Cutoff** is the only part-handling operation that works correctly with a Swiss post.



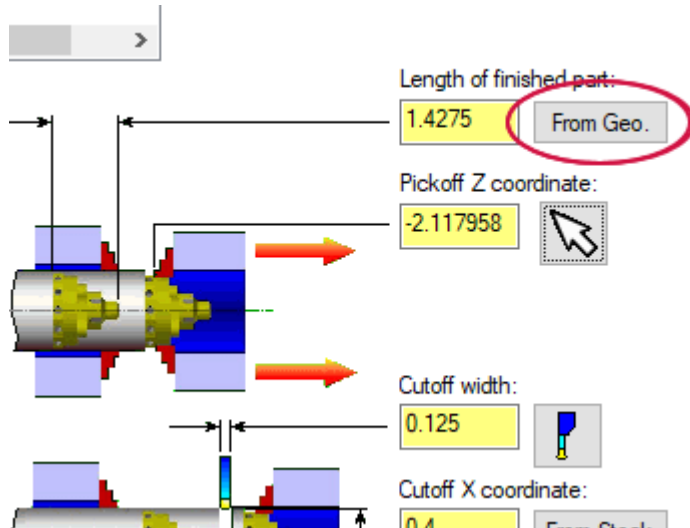
4. In the **Cutoff / Bar Pull / Pickoff** dialog box, click the **Geometry** button.



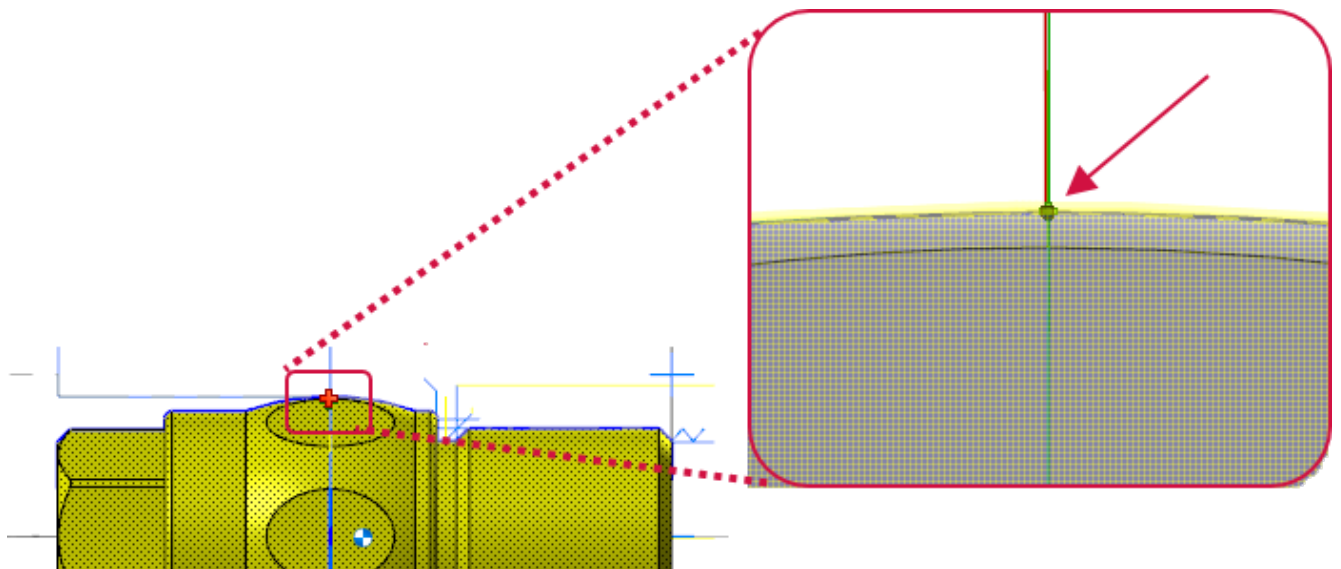
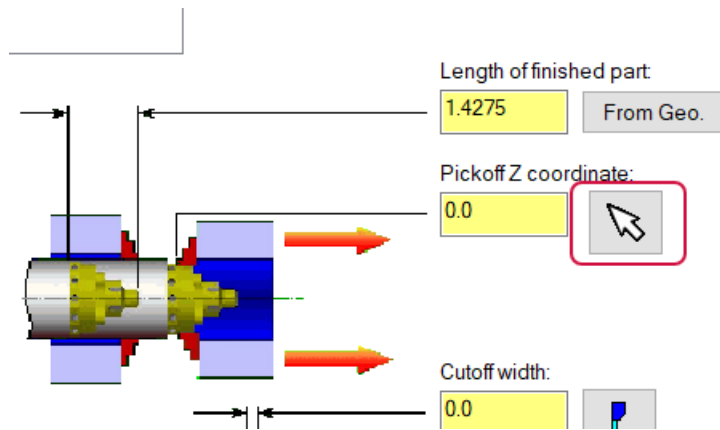
5. Select the solid model, and then press **[Enter]** or click **End Selection**.
6. Select **Translate Geometry**, which is located next to the **Geometry** button you just used.



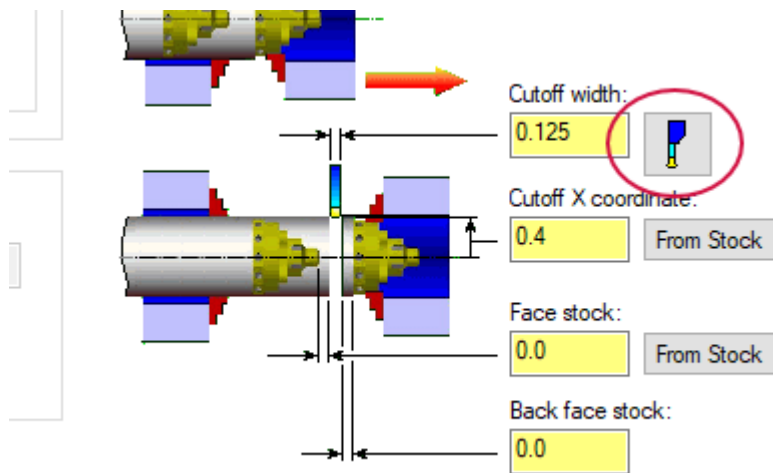
7. On the right side of the dialog box, click **From Geo.** The operation gets its length from the part geometry.



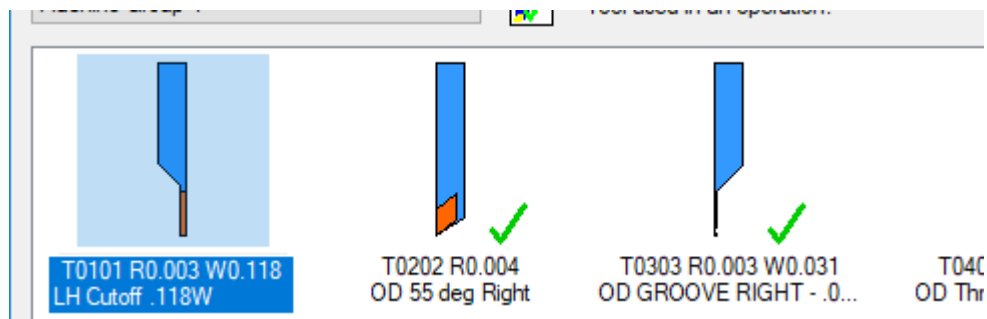
8. Click **Select pickoff Z coordinate**, and when returned to the graphics window, select the point shown in the second following picture. You should end up with a **Pickoff Z coordinate** of **-0.7935**.



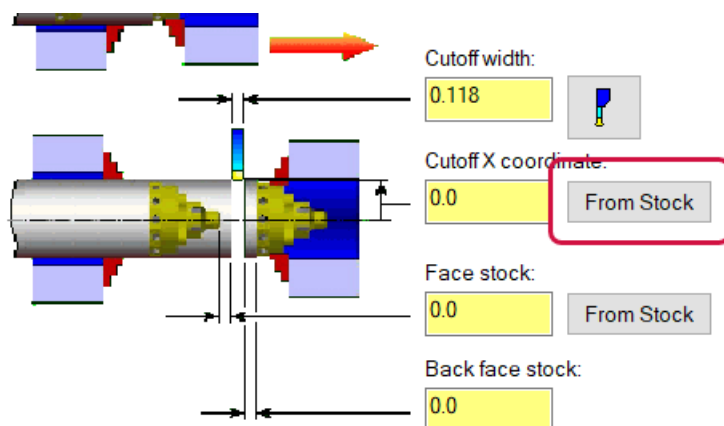
9. Click the **Select a tool to define cutoff width** button.



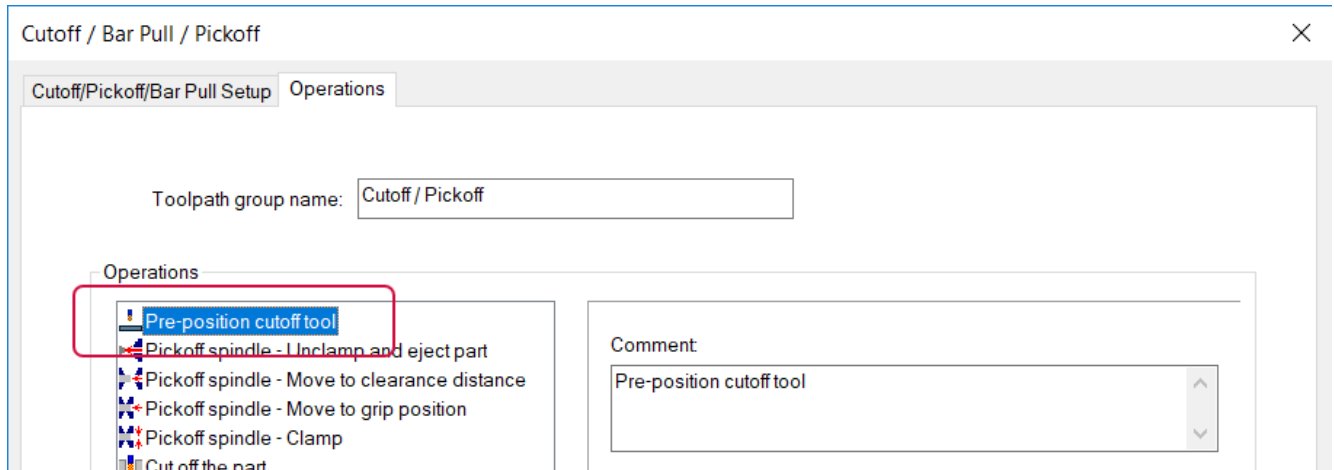
10. Select the **T0101 R0.003 W0.118 LH Cutoff .118W** tool, and click **OK**. (You may have to drag the tool up from the lower part of the dialog box.) You can also double-click the tool to select it.



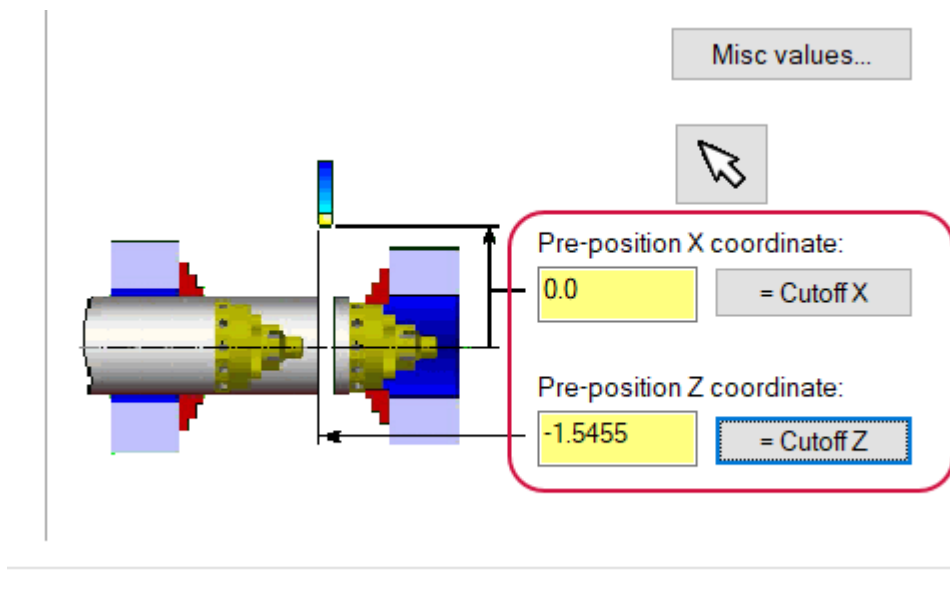
11. For **Cutoff X coordinate**, click **From Stock**.



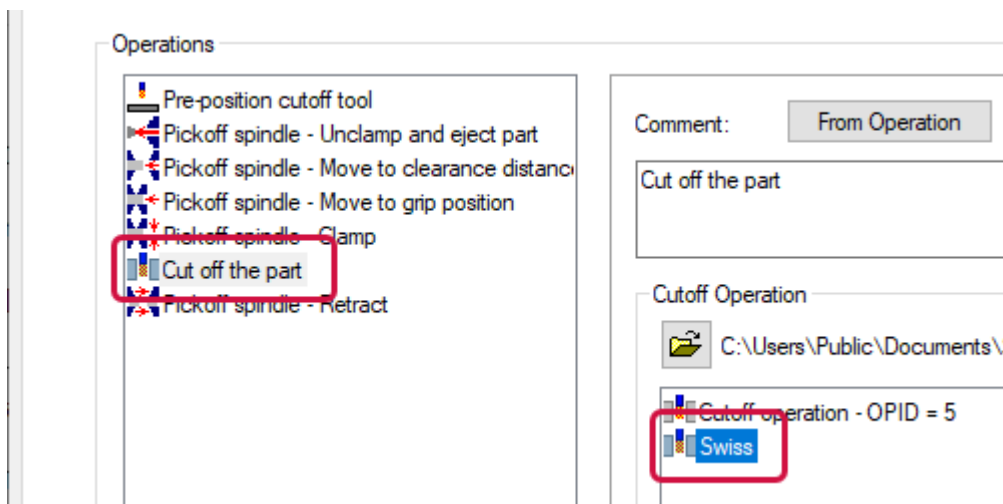
12. Select the **Operations** tab, and select **Pre-position cutoff tool**. This is new functionality that was added for Mastercam 2021.



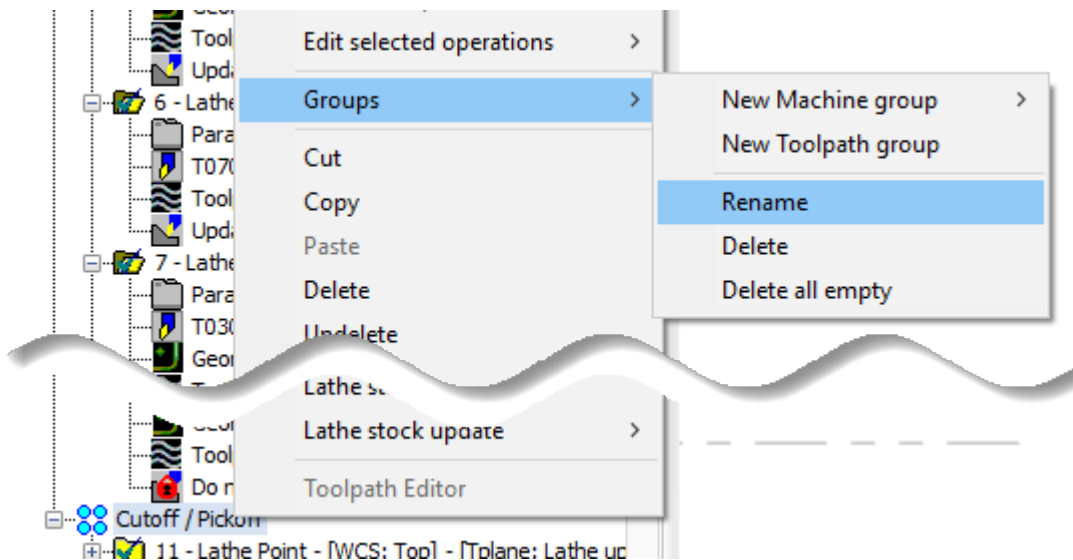
13. Click **= Cutoff X** and **= Cutoff Z** to set the pre-position coordinates.



14. Under **Operations**, click **Cut off the part**, and then select **Swiss** as the cutoff operation, and click **OK**. This activates the new Swiss cutoff operation that you created at the start of this exercise.



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



16. Save the file with the name Swiss Tutorial_XX_12.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.

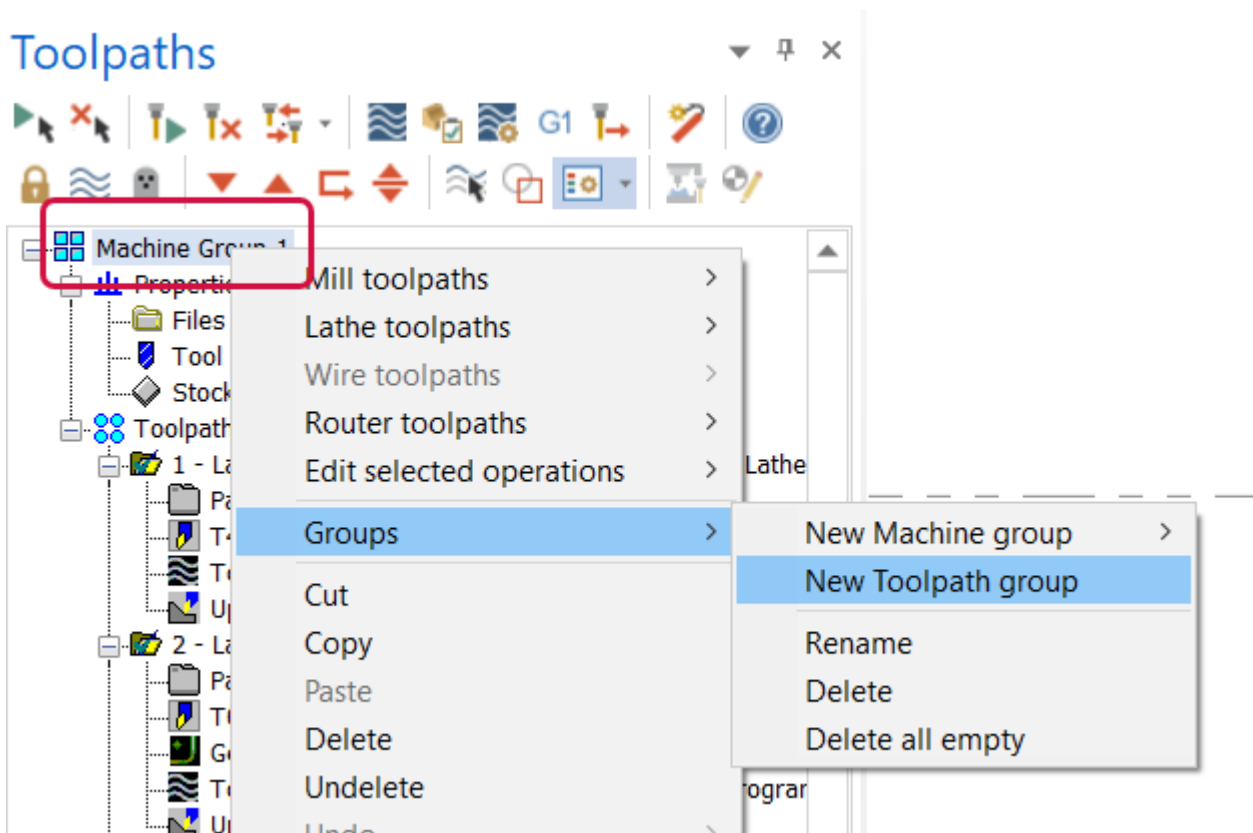
Goals

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

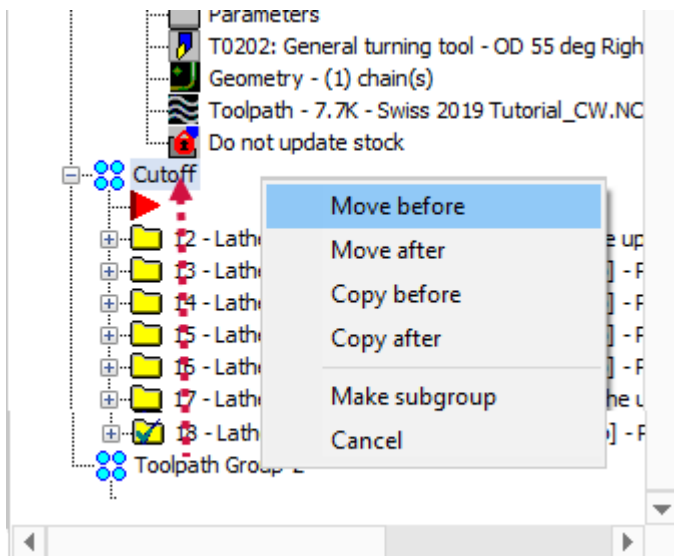
Exercise 1: Creating a sub spindle group

In this exercise, you create a subspindle toolpath group that will be placed before the cutoff. 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 subspindle operation before you had created the cutoff.

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

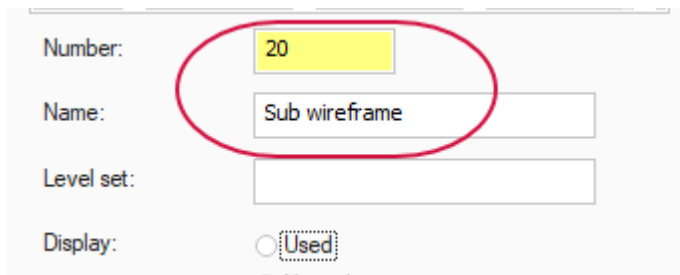


2. Drag the new **Toolpath Group-2** on top of the **Cutoff** group, and select **Move before** from the pop-up menu.

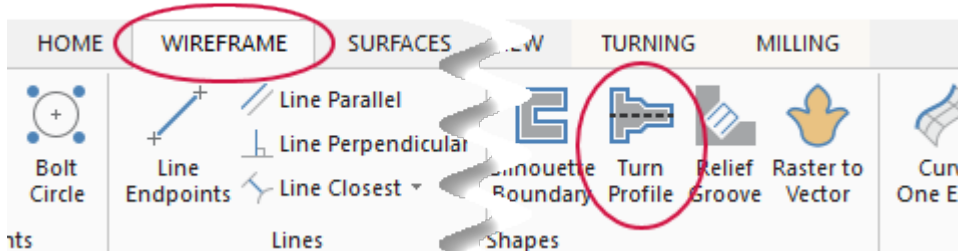


This operation syncs head 1 and head 2 so that both programs start simultaneously.

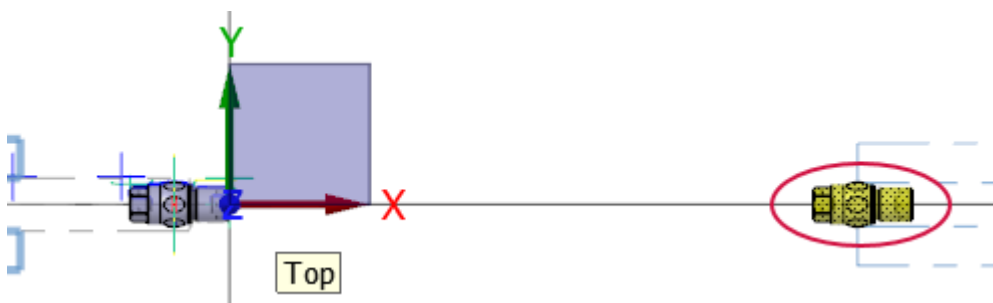
3. In the Levels Manager, create a new level **20** named **Sub wireframe**.



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.



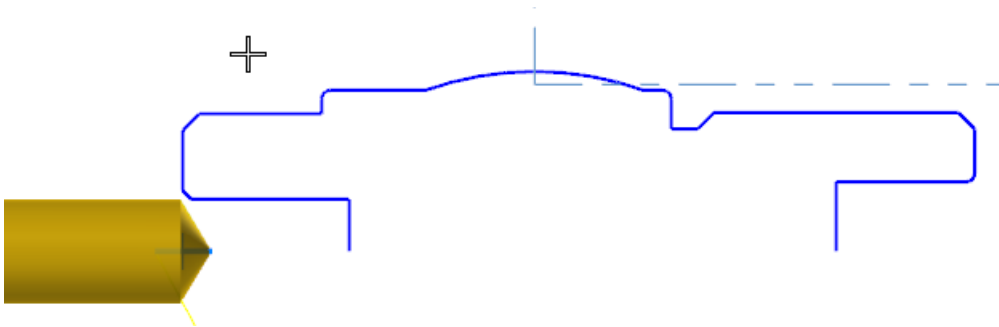
6. In the Levels Manager, turn off visibility for **Solid - Right Spindle**.

Nu...	Visible	Name	Level Set	Entities
1	X	Solid		22
10	X	Wireframe		29
✓ 20	X	Sub-wireframe		21
101		Solid - Right Spi...		1
110	X	Wireframe - Ri...		0

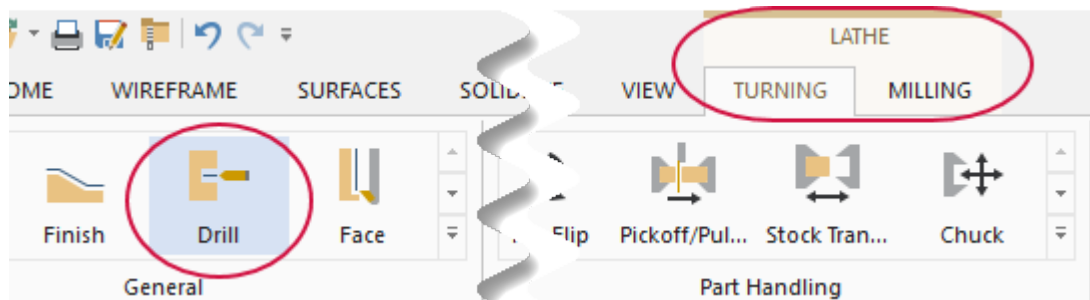
7. Save the file with the name Swiss Tutorial_XX_13.mcam, where XX is your initials.

Exercise 2: Creating the sub spindle drill operation

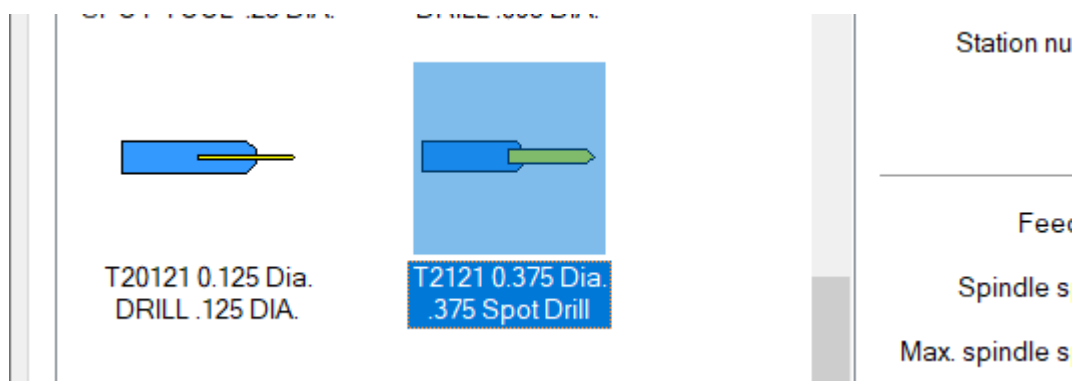
In this exercise, you create a spot drill operation to start the hole on the left end of the part.



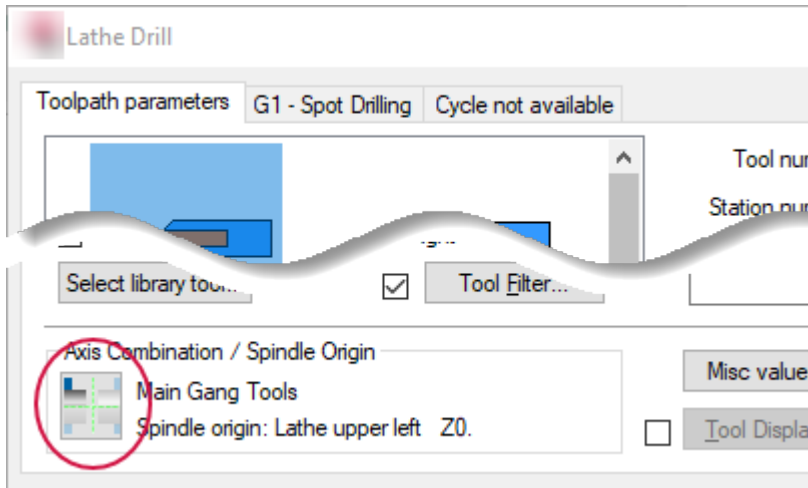
1. Select **Drill** from the **Lathe Turning** contextual tab.



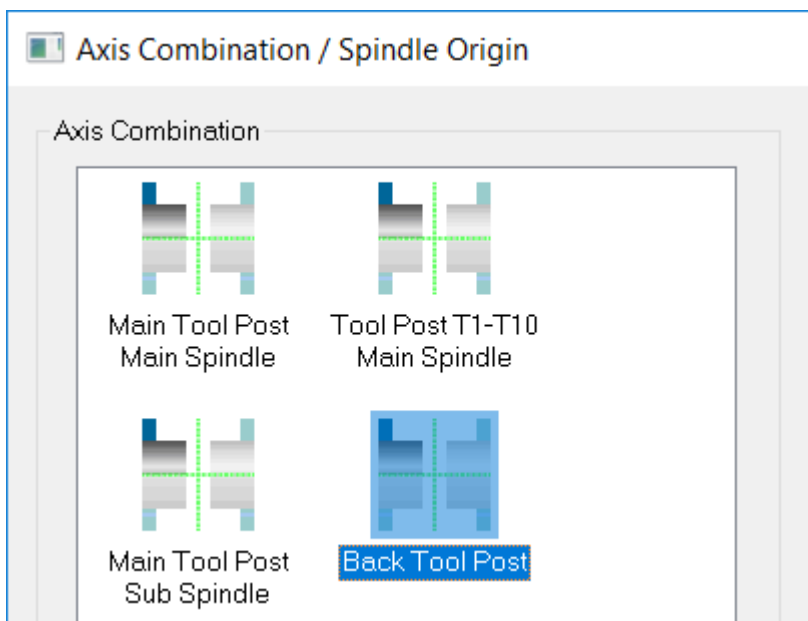
2. In the **Lathe Drill** dialog box, choose tool **T2121 0.375**. Several tools start with T2121, so be sure you pick the correct one.



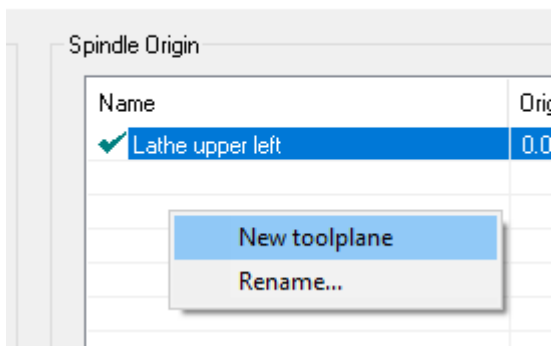
- Click the **Axis Combination** button.



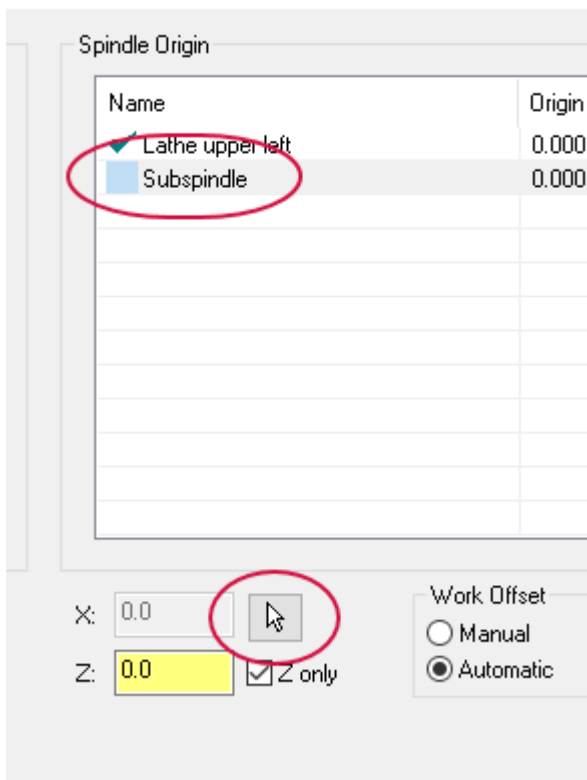
- Select the **Back Tool Post** axis combination.



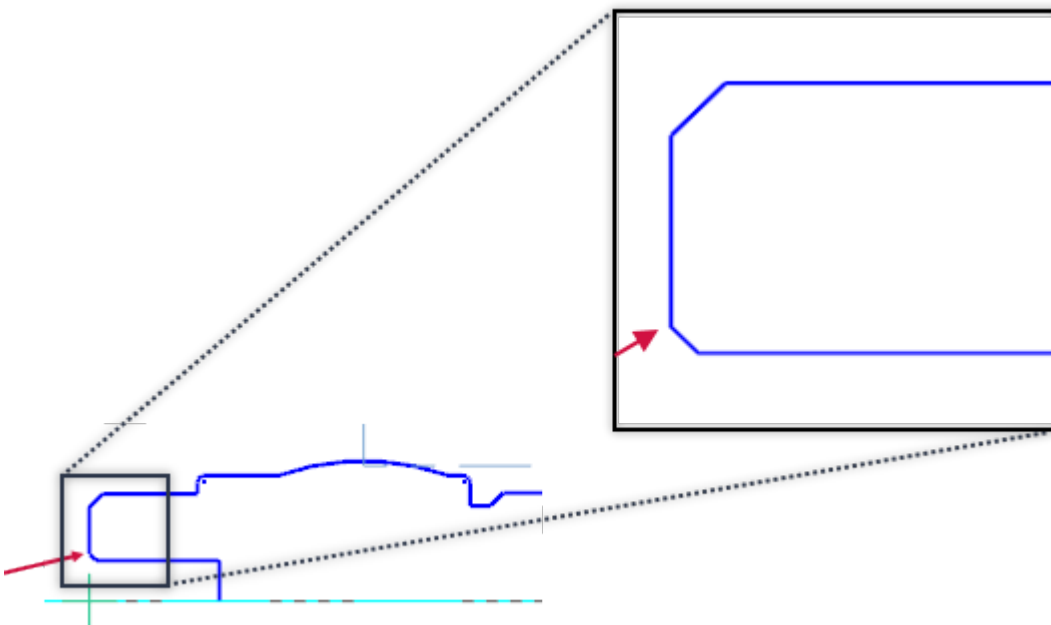
- Right-click in the **Spindle Origin** table, and choose **New toolplane** from the pop-up menu.



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. Ensure that the parameters look as follows:

Tool number:	<input type="text" value="21"/>	Offset number:	<input type="text" value="21"/>
Station number:	<input type="text" value="21"/>	<input type="button" value="Tool Angle..."/>	

Feed rate:	<input type="text" value="0.002"/>	<input checked="" type="radio"/> in/rev	<input type="radio"/> in/min	<input type="radio"/> micro-in
Spindle speed:	<input type="text" value="2000"/>	<input type="radio"/> CSS	<input checked="" type="radio"/> RPM	
Max. spindle speed:	<input type="text" value="5000"/>	<input type="button" value="Coolant..."/>		

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

Lathe Drill

Toolpath parameters **G1 - Spot Drilling** Cycle not available

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

Toolpath parameters **G1 - Spot Drilling** Cycle not available

☐ Absolute ☒ Incremental

Drill Cycle:

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

Tool tip included angle: 118.0

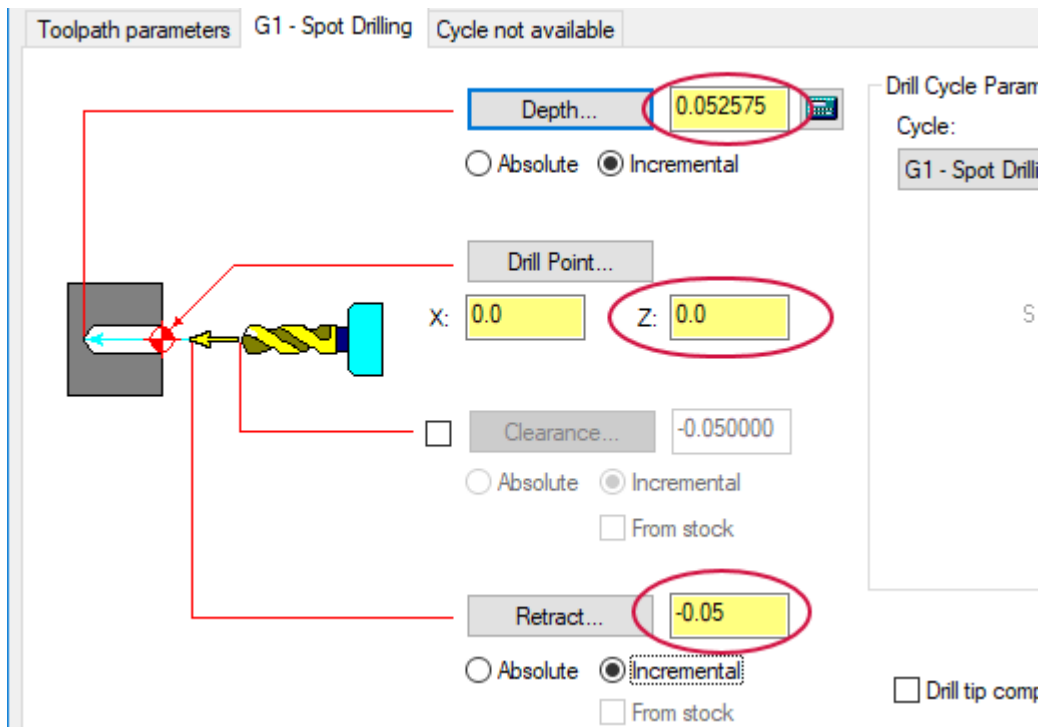
Finish diameter

Tool tip diameter (flat on tip)

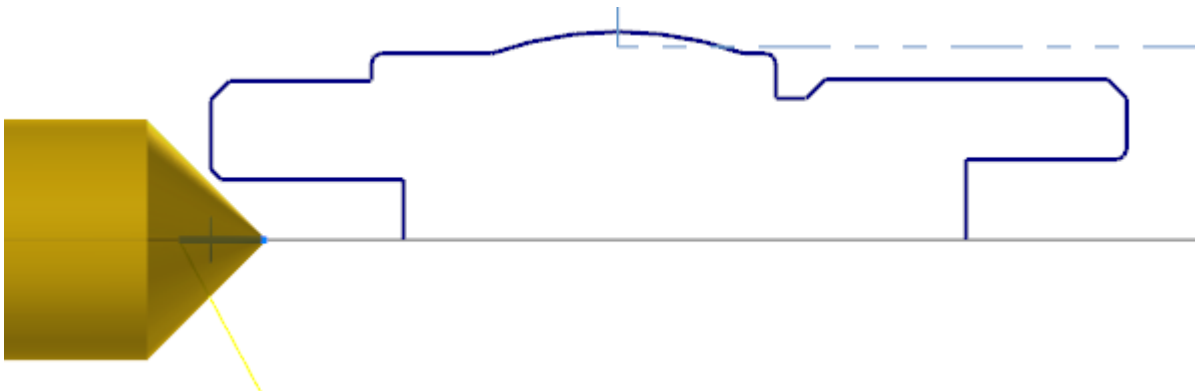
☐ Add to depth ☒ **Overwrite depth**

Depth:

12. Remove the negative sign from the **Depth** value, change **Z** to **0**, set **Retract** to **-0.05**, and click **OK**.



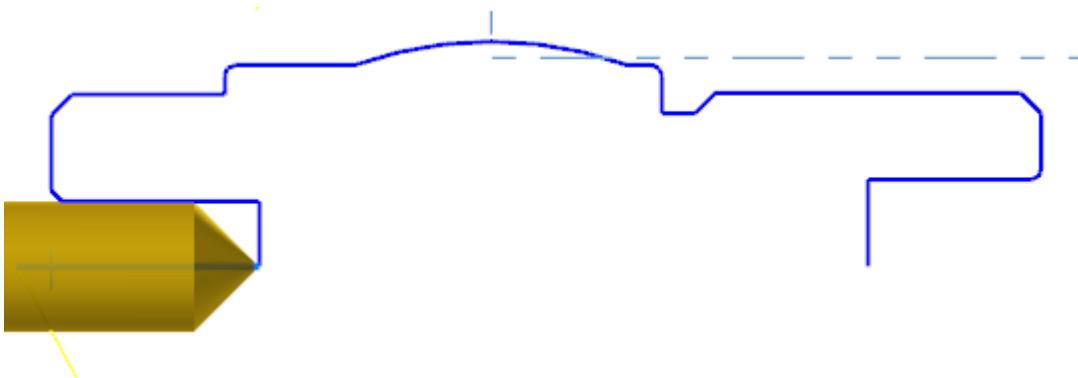
13. Backplot the operation to see the toolpath in motion.



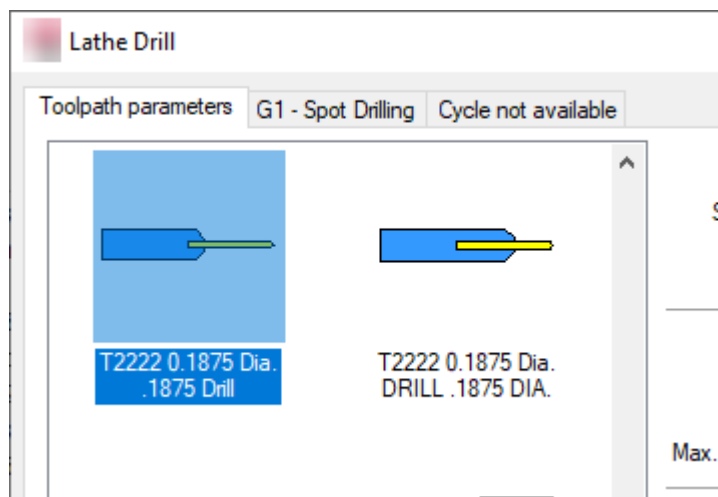
14. Save the file with the name Swiss Tutorial_XX_14.mcam, where XX is your initials.

Exercise 3: Creating the second drill operation

In this exercise, you create the second drill operation on the left side of the part.



1. Select **Drill** from the **Lathe Turning** contextual tab.
2. Choose the tool **T2222 0.1875** shown in the following image.



3. Ensure that the following parameters are set correctly.

- Set **Tool number** to **22**.
- Set **Offset number** to **22**.
- Set **Station number** to **22**.
- Set **Feed rate** to **0.0015**.
- Set **Spindle speed** to **2000**.
- Set **Max. spindle speed** to **7000**.

Tool number: Offset number:

Station number: Tool Angle...

Feed rate: ☒ in/rev ☐ in/min ☐ micro-in


Spindle speed: ☐ CSS ☒ RPM

Max. spindle speed: Coolant...

4. Set the **Axis Combination** to **Back Tool Post**.

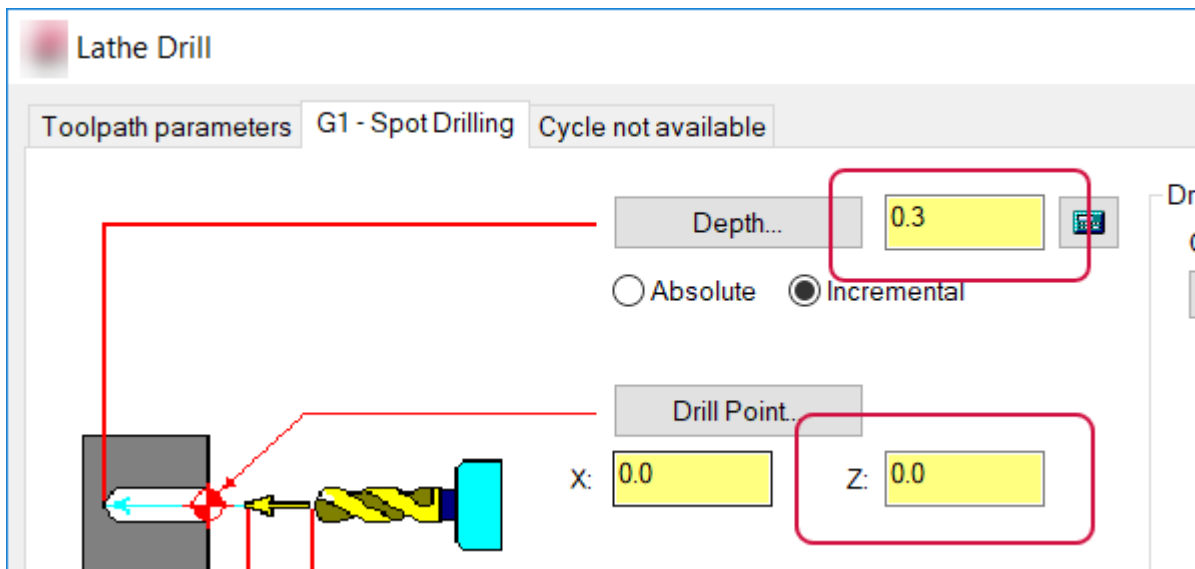
Select library tool... ☒ Tool Filter...

Axis Combination / Spindle Origin

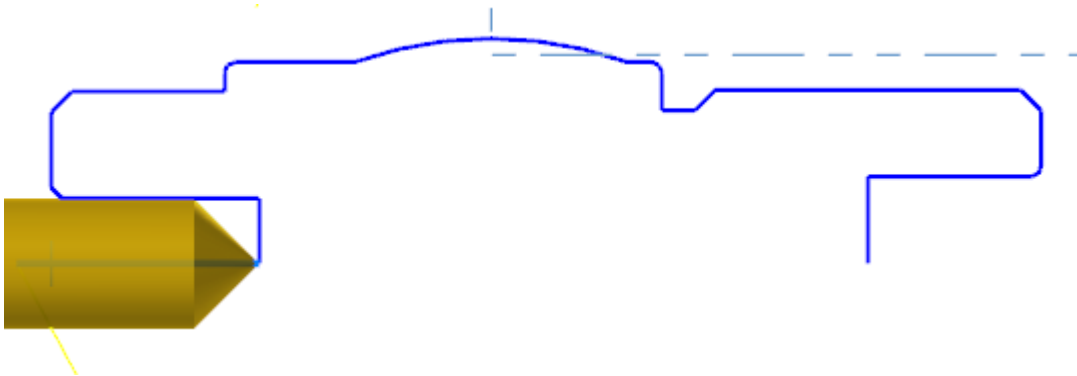
 **Back Tool Post**

Spindle origin: Subspindle Z7.3661

5. On the **G1 - Spot Drilling** tab, set **Depth** to **0.3** and **Z** to **0.0**, and then click **OK**.



6. Use Backplot to test the toolpath.

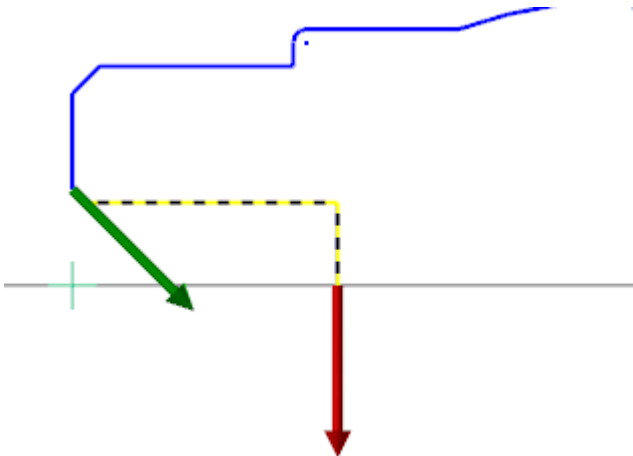


7. Save the file with the name Swiss Tutorial_XX_15.mcam, where XX is your initials.

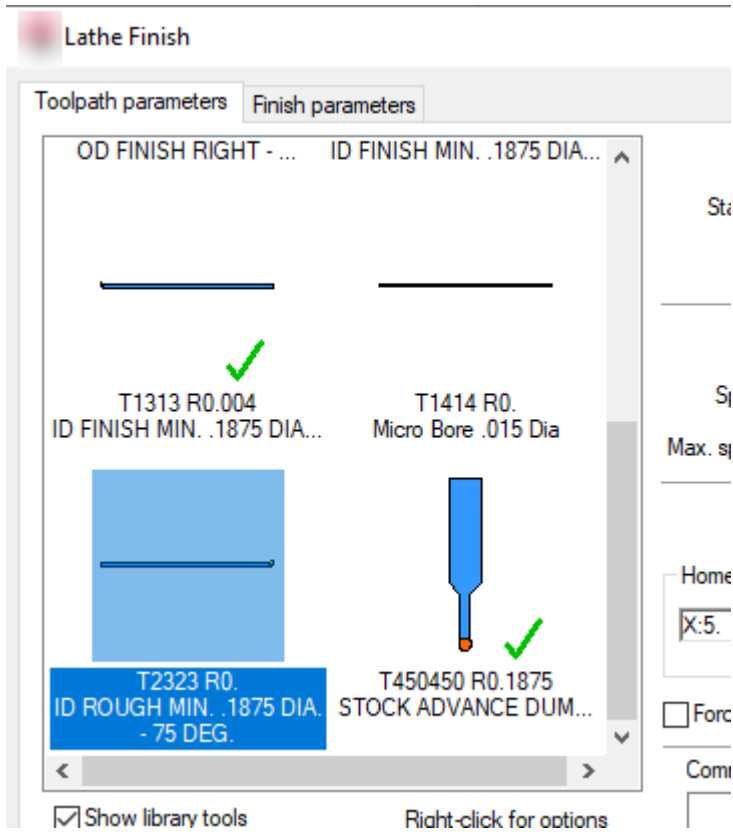
Exercise 4: Creating the Finish operation for boring

In this exercise, you create a finish operation for the hole you drilled previously.

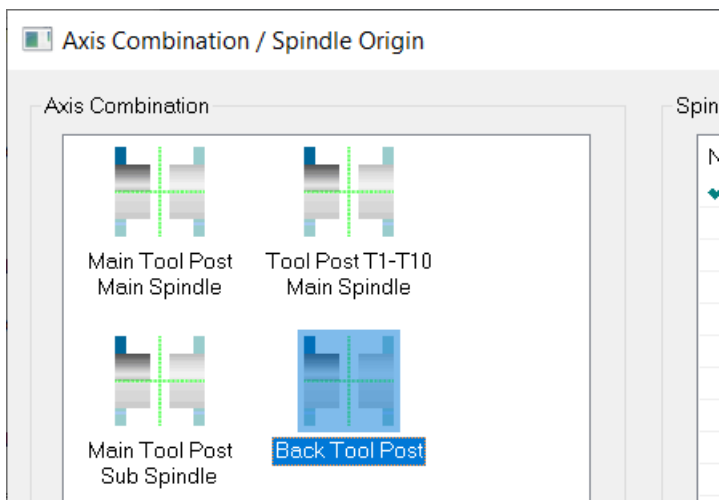
1. Select **Finish** from the **Lathe Turning** contextual tab.
2. Chain the part as shown below, and click **OK**.



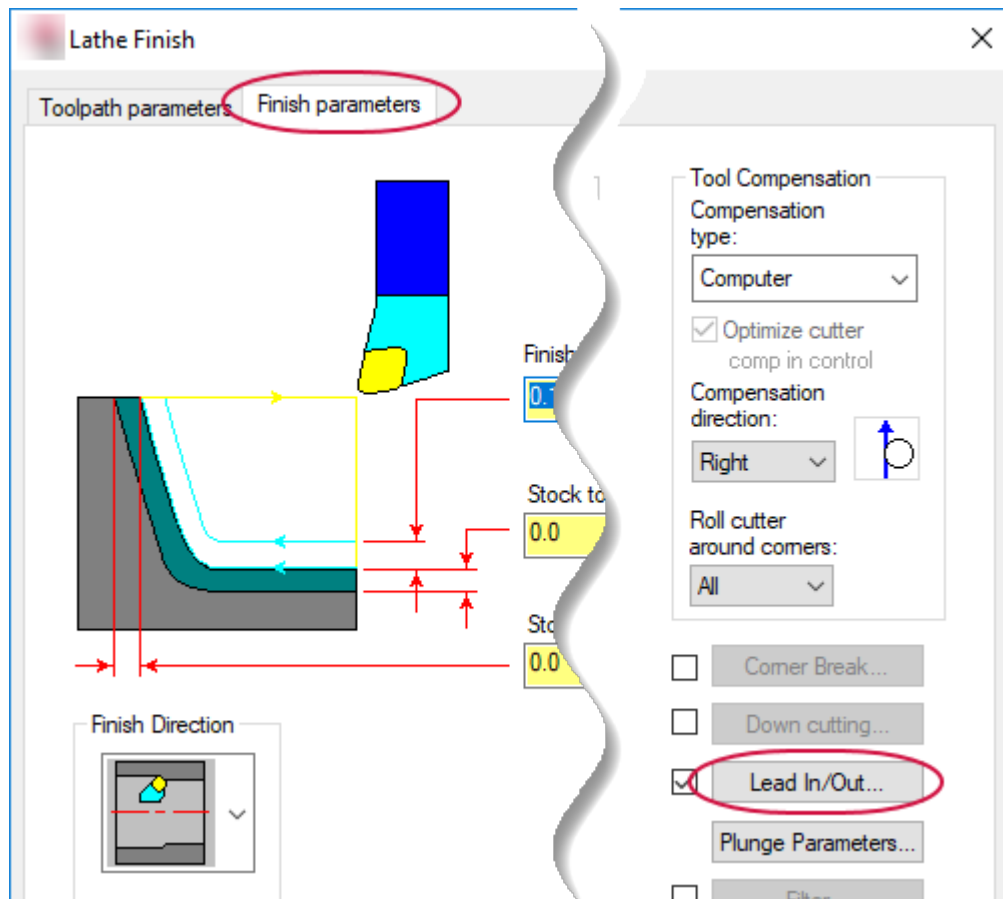
3. Select the tool **T2323**.



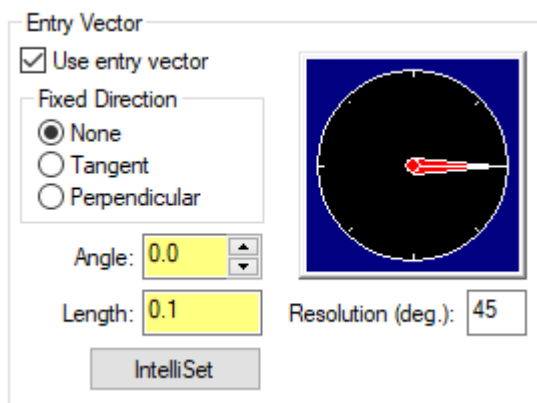
4. Click the **Axis Combination/Spindle Origin** button, set **Axis Combination** to **Back Tool Post**, and click **OK**.



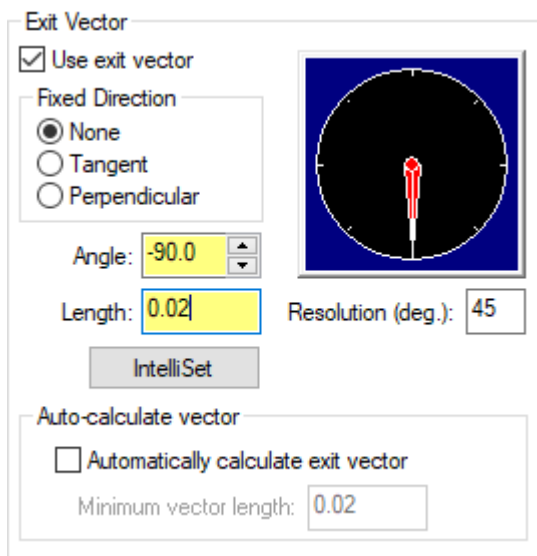
5. Select the **Finish parameters** tab, and click **Lead in/out**.



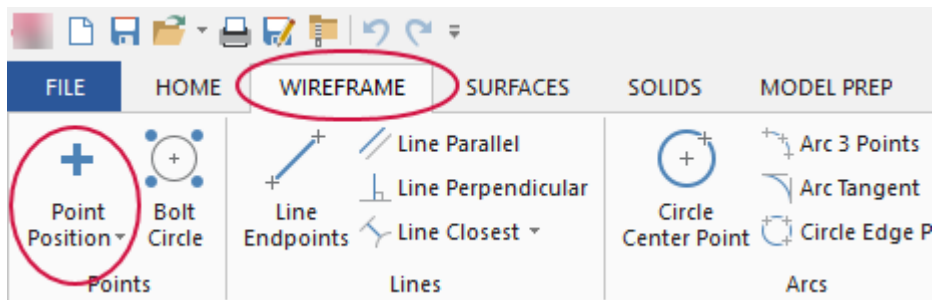
6. Select the **Lead in** tab, and set **Angle** to **0.0** and **Length** to **0.1**.



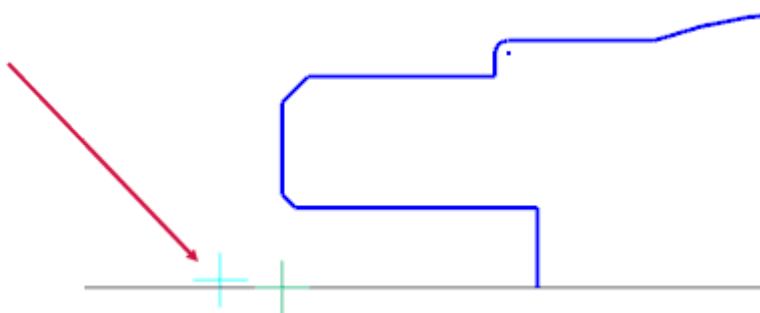
7. Select the **Lead out** tab, set **Angle** to **-90.0** and **Length** to **0.02** as shown in the following picture.



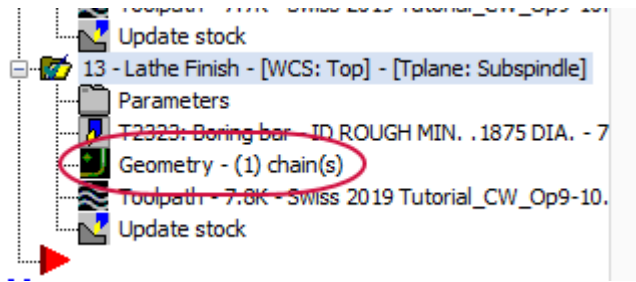
8. Click **OK** in the **Lead In/Out** and the **Lathe Finish** dialog boxes.
9. Select the **Wireframe** tab, and choose **Point Position**.



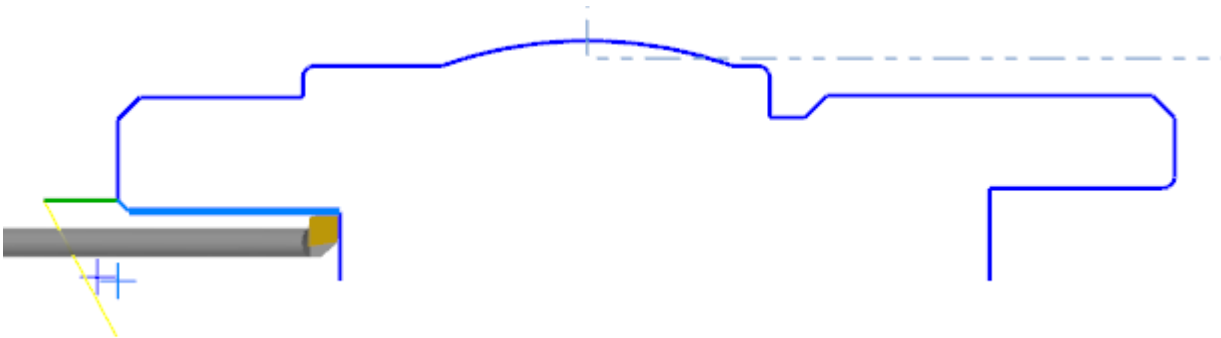
10. Create a point similar to the one shown in the following picture (in other words, your choice need not exactly match the one shown), and click **OK** in the **Point Position** function panel. Note that the point you select must be above the center line of the part and beyond the face of the part.



11. In the Toolpaths Manager, select **Geometry** in operation 14 to display the **Chain Manager** dialog box.



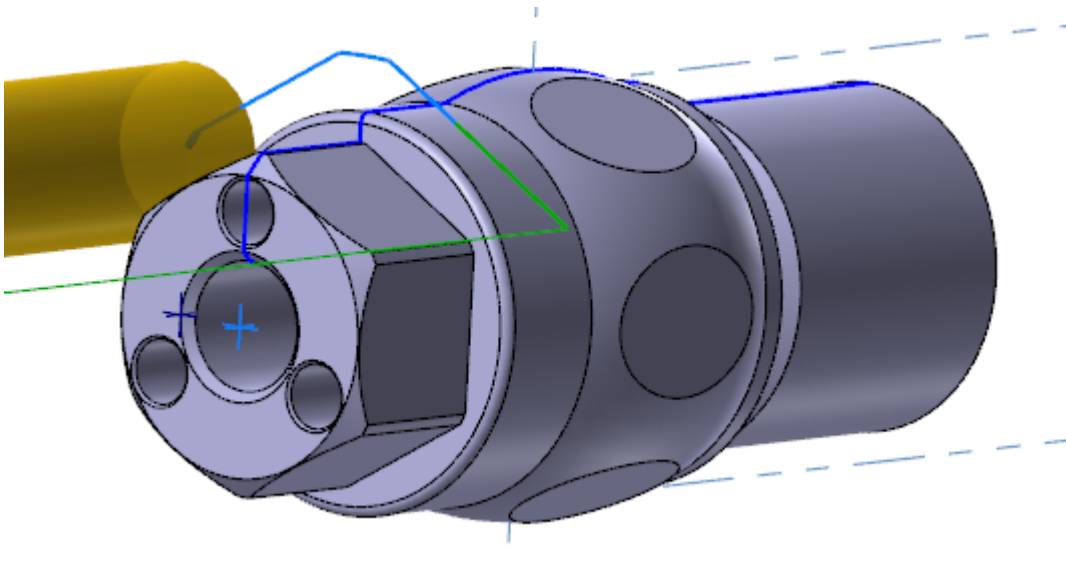
12. Right-click in the **Chain Manager** dialog box, and choose **Add** from the pop-up menu.
13. Select the new point from Step 10, and click **OK** in both the **Chaining** and **Chain Manager** dialog boxes.
14. Regenerate all operations, select operation 14, and backplot to see it in action.



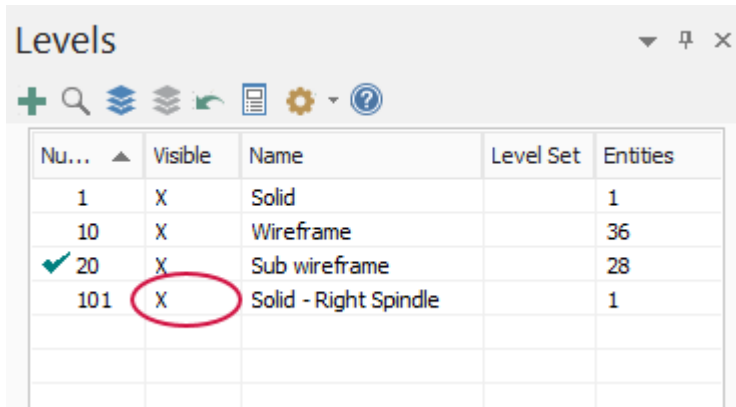
15. Save the file with the name Swiss Tutorial_XX_16.mcam, where XX is your initials.

Exercise 5: Polar milling

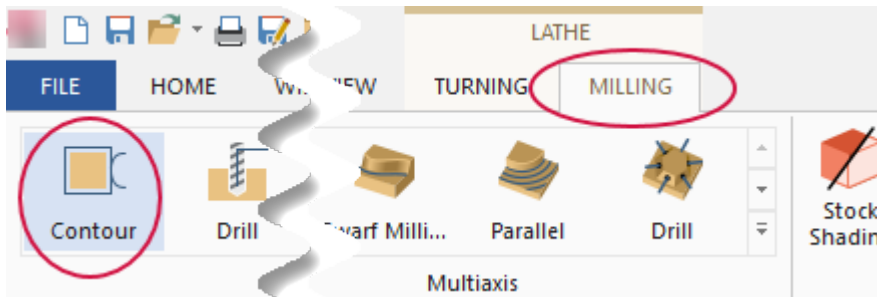
In this exercise, you create a toolpath to mill the flat areas on the left end of the part. For machines without a Y axis on sub spindle, you need to use Polar Milling to get this done.



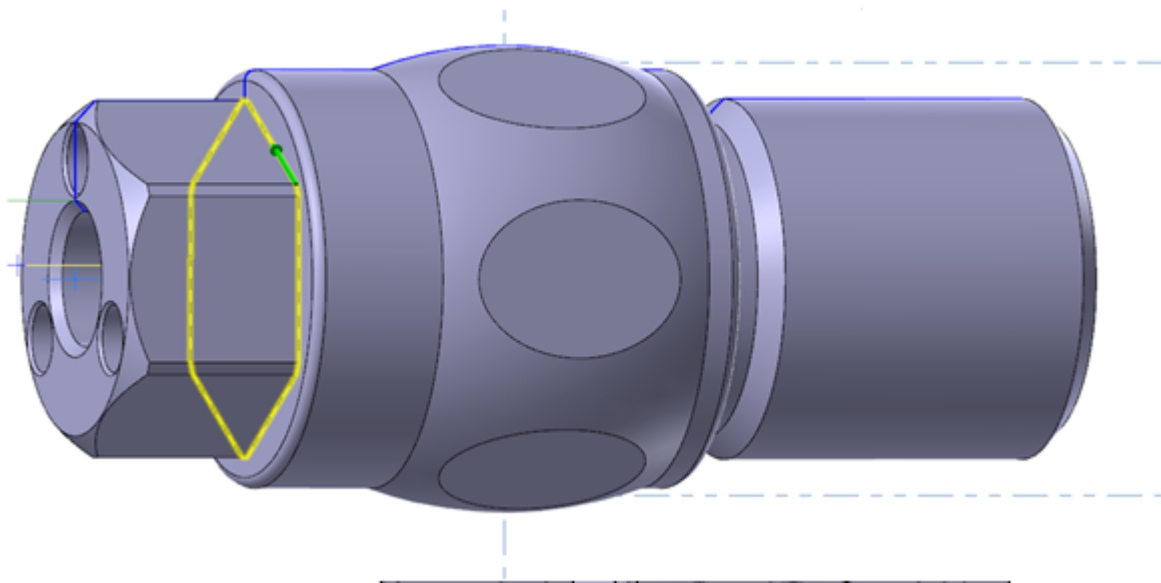
1. Turn on the solid's visibility (level 101) in Levels Manager.



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

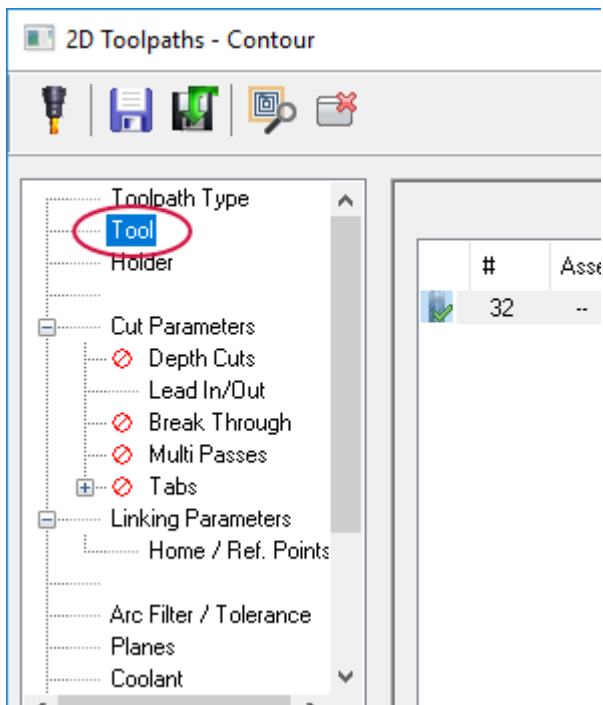


3. In the **Chaining** dialog box, select the **Solids** mode. Ensure that the **Face** option is off, and that the **Loop** option is on.
4. Ensure that **3D** is selected on Mastercam's status bar, and then chain the part as shown in the following image.

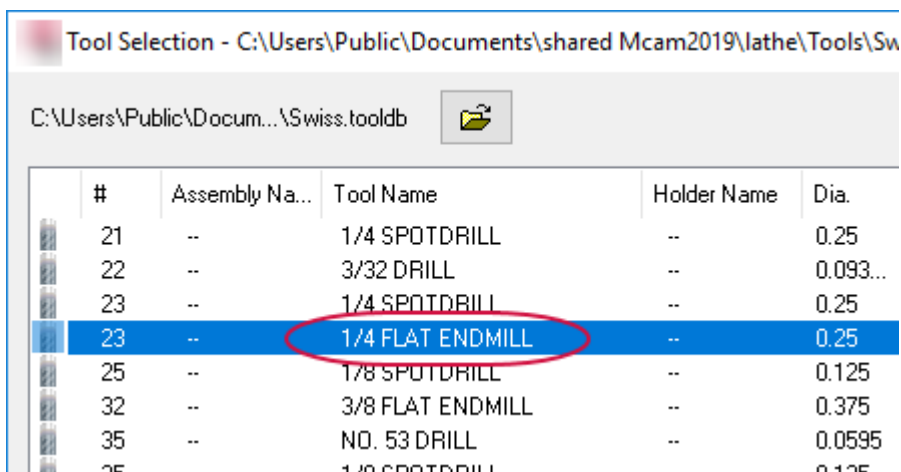


5. Click **OK** in the **Chaining** dialog box.

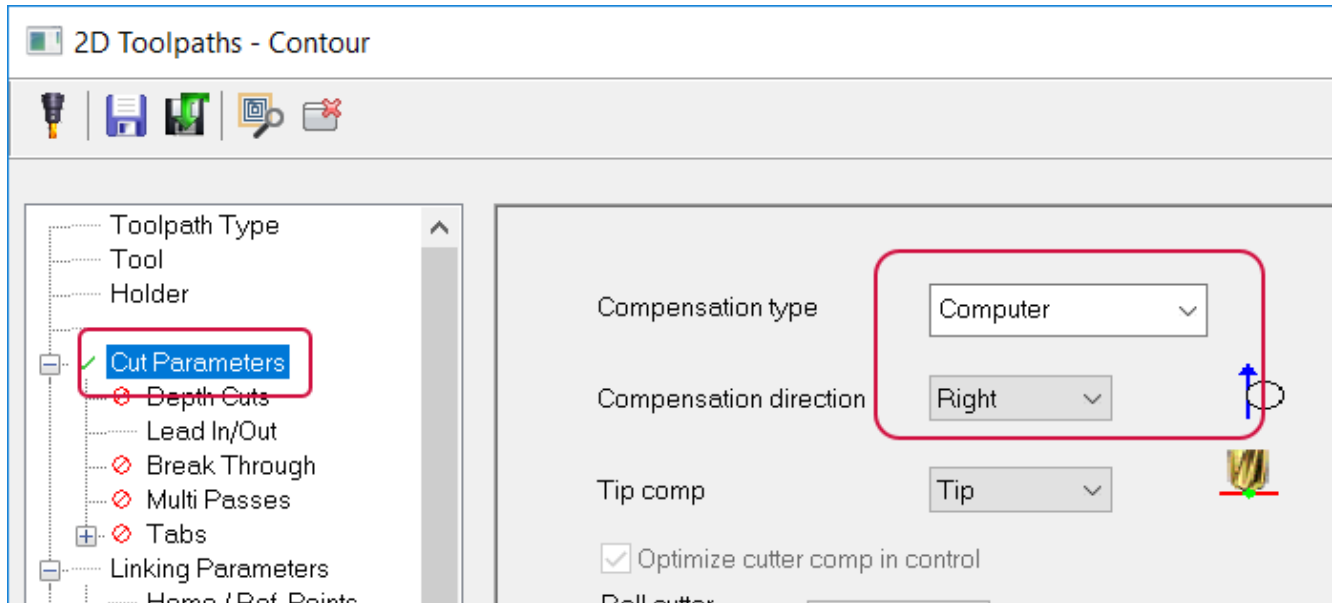
6. In the **2D Toolpaths - Contour** dialog box, select the **Tool** page.



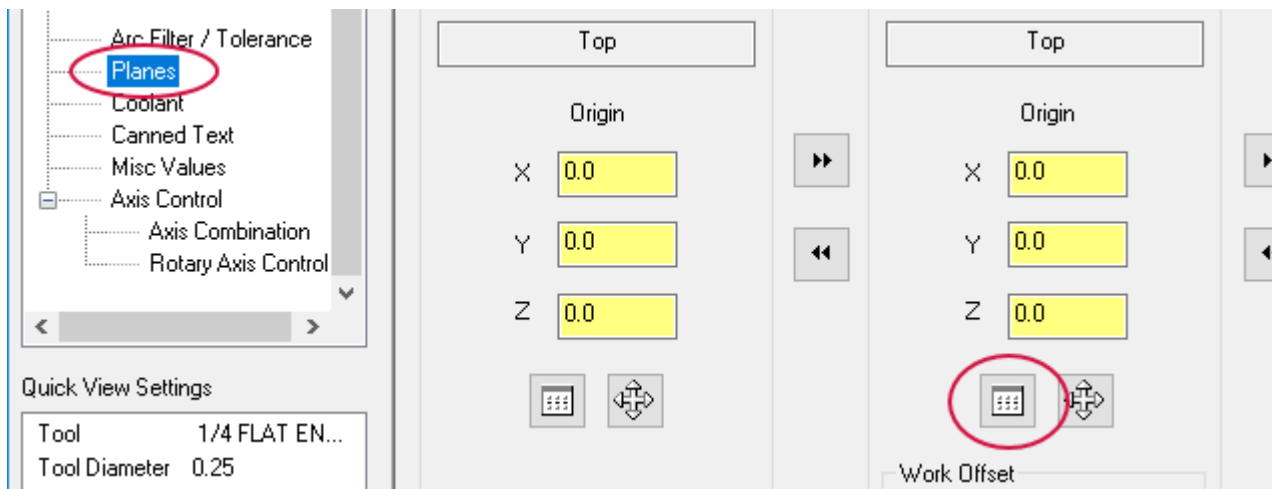
7. Click **Select library tool**, choose the **1/4 flat endmill** from the **Tool Selection** dialog box, and click **OK**.



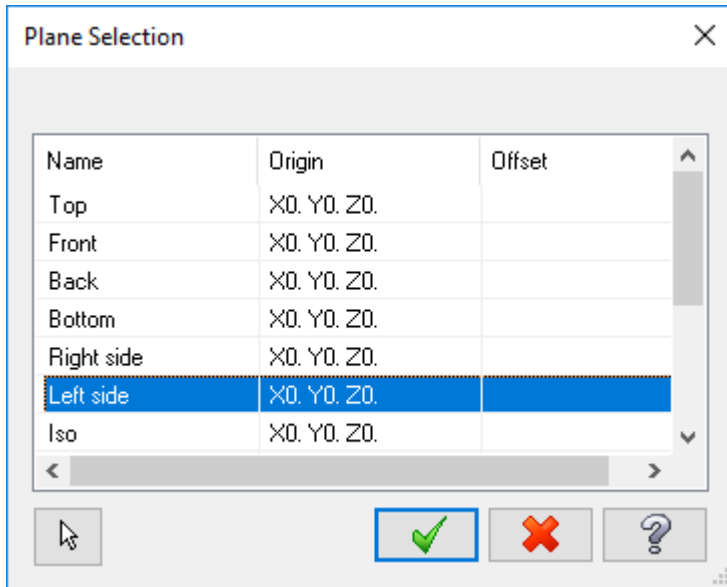
8. On the **Cut Parameters** page, change **Compensation type** to **Computer** and **Compensation direction** to **Right**.



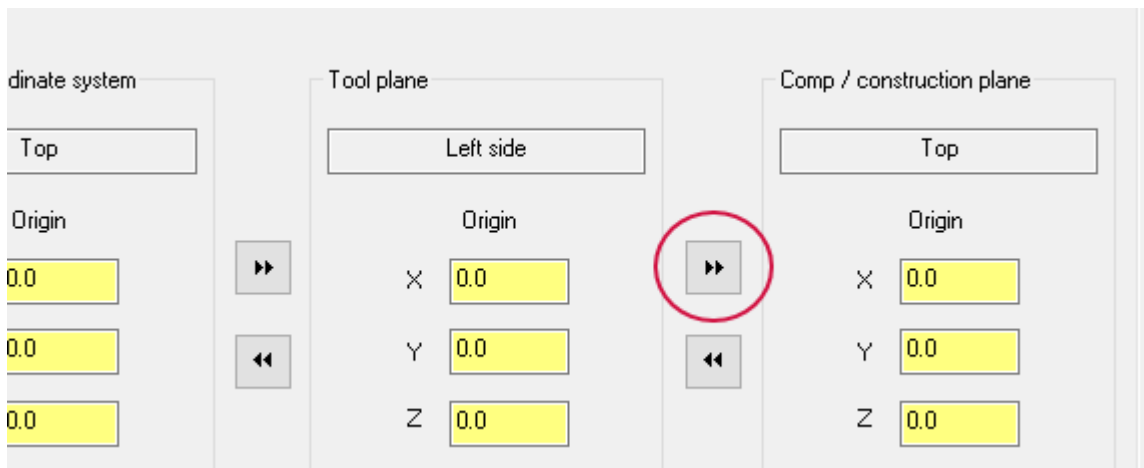
9. On the **Planes** page, click the **Select tool plane** button.



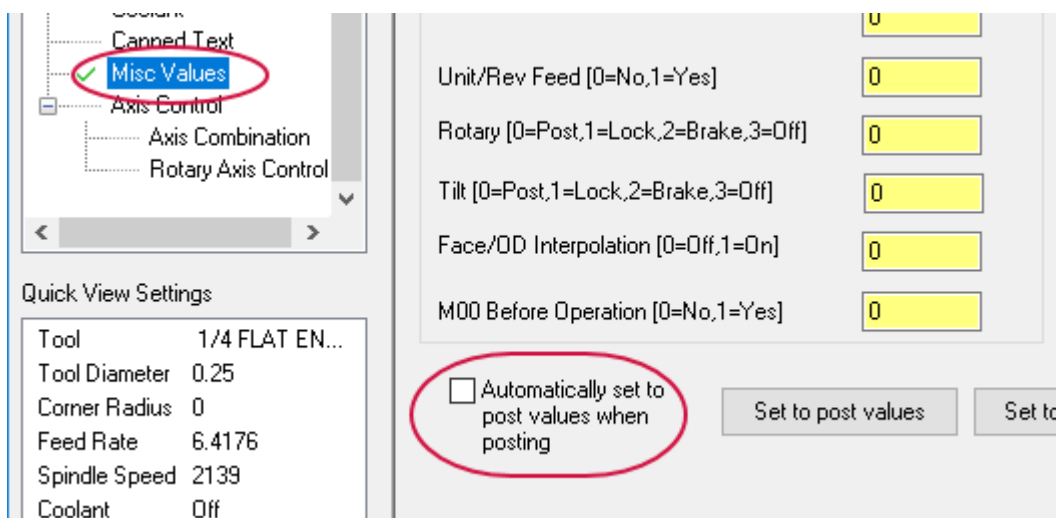
10. In the **Plane Selection** dialog box, choose **Left side**, and click **OK**.



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



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



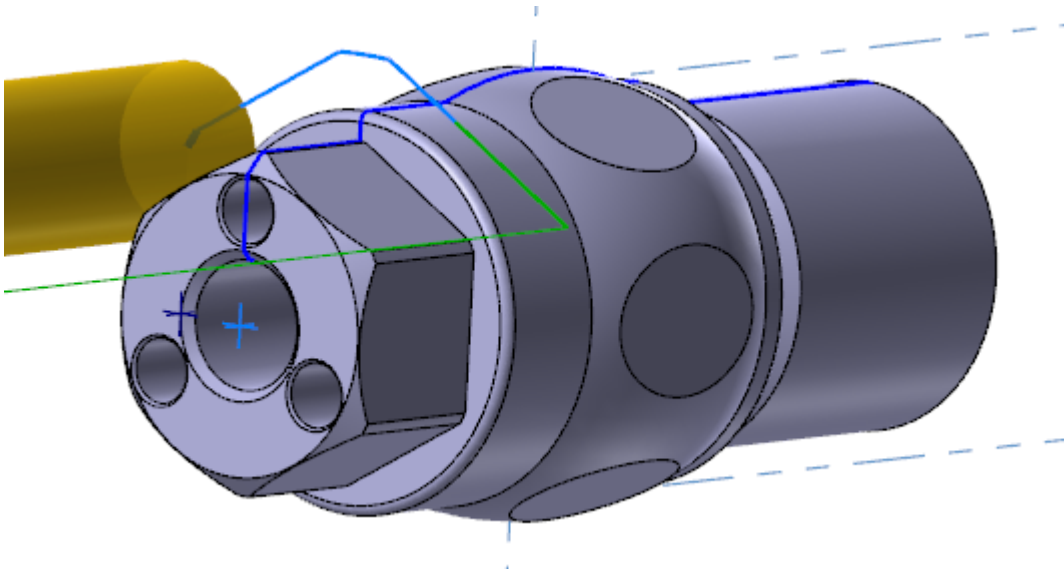
13. Change **Face/OD Interpolation** to **1**, and click **OK**.

Tilt [0=Post,1=Lock,2=Brake,3=Off]	0
Face/OD Interpolation [0=Off,1=On]	1
M00 Before Operation [0=No,1=Yes]	0

☐ Automatically set to post values when

Set to post values

14. Backplot the operation to see the toolpath motion.



15. Save the file with the name Swiss Tutorial_XX_17.mcam, where XX is your initials.

CONCLUSION

Congratulations! You have completed the *Mastercam 2021 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 2021 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](https://www.mastercam.com/support)
FOR THE LATEST DOWNLOADS.**

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



www.mastercam.com

Mastercam[®]

Mastercam[®] is a registered trademark of CNC Software, Inc. ©1983-2019. All rights reserved.
All other trademarks are property of their respective owners.