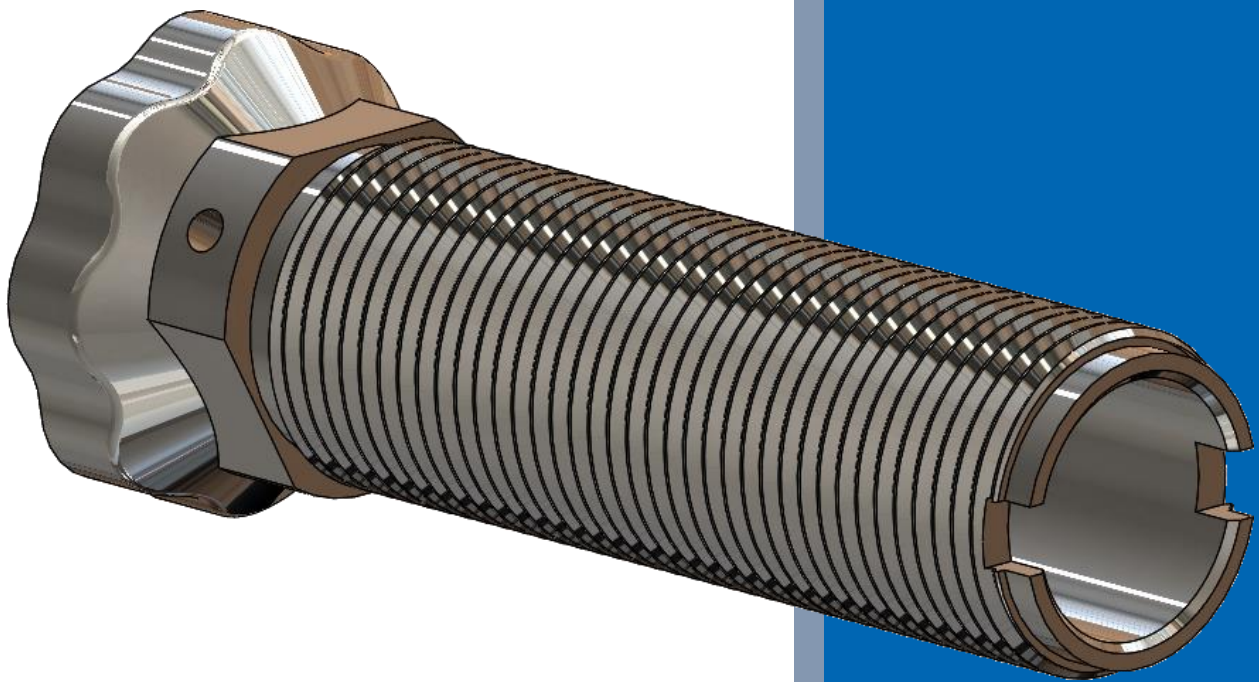


HCL CAMWorks

Swiss Turn for Launchpad

2026



CERTIFIED
Gold
Product

SOLIDWORKS

LAUNCH  PAD

CAMWorks ANNUAL SALES PARTNER SUMMIT 2025

Disclaimer

Geometric Americas, Inc. makes no warranties, either express or implied, with respect to this manual. Geometric Americas, Inc. reserves the right to revise and improve products as it sees fit, and to revise the specifications and information contained herein without prior notice. Due to continuing product development, specifications and capabilities described in this manual are subject to change without notice.

Trademarks

The following and other product names and corporate references herein are registered or proprietary trademarks of their respective owners.

CAMWorks® is a registered trademark of Geometric Americas, Inc.

All other brands and names are the property of their respective owners.



Copyright © 2019 Geometric Americas, Inc. All Rights Reserved.

Product Version: CAMWorks 2026

Table of Contents

Disclaimer	1
Trademarks	1
Table of Contents	2
Introduction to training material	3
About this course and manual	3
Prerequisites	3
Lab exercises	3
Course Length	3
Training files	3
Technology Database for this course	4
Introduction – CAMWorks Swiss Turn Basics	5
What is CAMWorks?	6
CAMWorks Add-in	6
User interface	7
What is Swiss Turn Machining?	9
Lesson 1 – CAMWorks Swiss Turn Workflow and User Interface	12
Process overview	13
Case Study – Generate Toolpaths and NC Code	14
Lesson 2 – Programing a Part for Swiss-Turn Advanced Options Part 1	39
Case Study – Advanced Swiss Turn Part Programing Part 1	40
Lesson 3 – Programming a Part for Swiss-Turn Advanced Options Part 2	80
Case Study – Advanced Swiss Turn Part Programing Part 2	81
Lesson 4 – G-Code Simulation with CAMWorks Virtual Machine	125
CAMWorks Virtual Machine	126
Case Study – G-Code Simulation with CAMWorks Virtual Machine	128

Introduction to training material

About this course and manual

This course is intended to teach the user how to use CAMWorks to create toolpaths for the machining of part and assembly files created in SOLIDWORKS and CAMWorks Solids for CNC mills.

This manual is intended to be used in a classroom environment with an experienced CAMWorks instructor. The instructor is expected to cover the lesson material as part of a live demonstration and provide additional instruction and guidance during the lab exercises. It is not intended to be a self-paced tutorial.

Self-paced tutorials are available with every installation of CAMWorks. They are in **C:\Program Files\CAMWorks20XXx64\CAMWorks_VC140\lang\English\Manuals**. It is recommended to review this material before attending the course.

Since it is hard to cover every minute detail of CAMWorks 2.5 Axis Milling in a short class time, this course will examine the fundamental skills and concepts needed for the successful use of CAMWorks.

The case studies and lab exercises in this course are intended to show you how to use CAMWorks to set up your files for Milling and Drilling operations and may not correspond to actual machining practices.

It is recommended to use this course material as a supplement to and not a replacement for the documentation and help files.

Prerequisites

Students attending this course are expected to have the following:

- Experience with SOLIDWORKS Design Software
- Experience with Windows operating system
- Experience with CNC Machining
- Completed the CAMWorks 2.5 Axis Essentials training class
- Completed the Turn and Mill/Turn training class

Lab exercises

Lab exercises provide the opportunity to apply and practice the concepts covered in each lesson. They are modest in scope to allow the user enough time to complete in the allotted time

Course Length

Recommend to be a minimum course length is 3 days.

Training files

A complete set of training files will be provided by the instructor.

The files are organized by lesson number. The Case Study folder contains all the files the instructor will use during the lessons. The *Exercises* folder will contain the files that are required for the lab

exercises. In each of these folders there is a *Finished* folder containing the files in their completed state.

Technology Database for this course

This course was designed to be used with the default technology database that is installed with a new installation of CAMWorks. If a different technology database is used, some of the resulting toolpaths might be different from the manual.

Your instructor can provide you with a default technology database if necessary.

Introduction – CAMWorks Swiss Turn Basics

Upon successful completion of this lesson, you will be able to:

- Load the CAMWorks Add-in.
- Identify the elements of the CAMWorks User Interface.
- Understand the basics of Swiss Turn Machining

What is CAMWorks?

CAMWorks is a fully integrated, knowledge-based technology that allows you to create toolpaths and generate NC Code.

Fully integrated with SOLIDWORKS/CAMWorks Solids

CAMWorks is fully integrated into SOLIDWORKS and works within the SOLIDWORKS environment.

As a result of this integration, you can:

- Use the same user interface (Windows) for design and CAM.

CAMWorks converts the "design view" of parts in SOLIDWORKS to a "manufacturing view". The SOLIDWORKS FeatureManager Design tree shows the design tree. The CAMWorks Feature and Operation trees show the manufacturing view for the same part.
- Use the same part file for storing the design and the machining information (machinable features, operations, and toolpaths). For a list of compatible and supported part/assembly file formats, read the topic: Supported File Formats.
- Eliminate file transfers using time-consuming standard file formats such as IGES, SAT and Parasolid. The IGES and SAT file formats are a close approximation of the part within a tolerance and as such are prone to errors.
- Generate toolpaths on the actual SOLIDWORKS part, not on an imported approximation.
- Generate toolpaths that are associative with SOLIDWORKS. This means that if the design model is changed, the toolpaths are changed automatically with minimal user intervention.

Feature-based

Machining is done on features such as holes, pockets, and bosses.

Knowledge-based

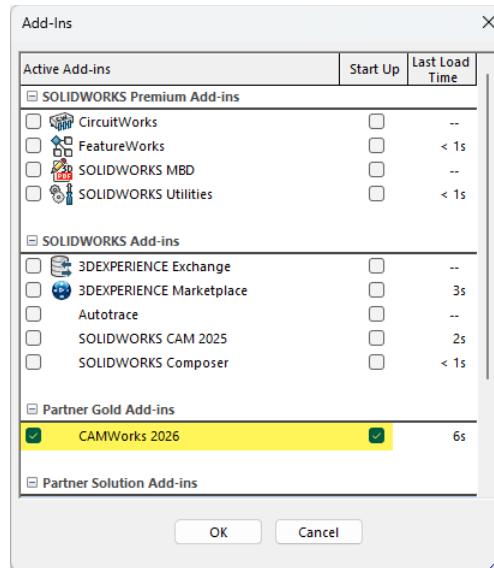
A Technology Database (TechDB) allows you to capture your facility's manufacturing knowledge and apply this information to further automate the NC programming process. The Technology database also encourages facility-wide consistency and rules-based manufacturing.

Associative

Within limits, if the part changes, the features and toolpaths reflect the change.

CAMWorks Add-in

To use CAMWorks, it must be activated using **Tools, Add-Ins**. Click both **Active Add-ins** and **Start Up** for **CAMWorks 2026** and click **OK**.



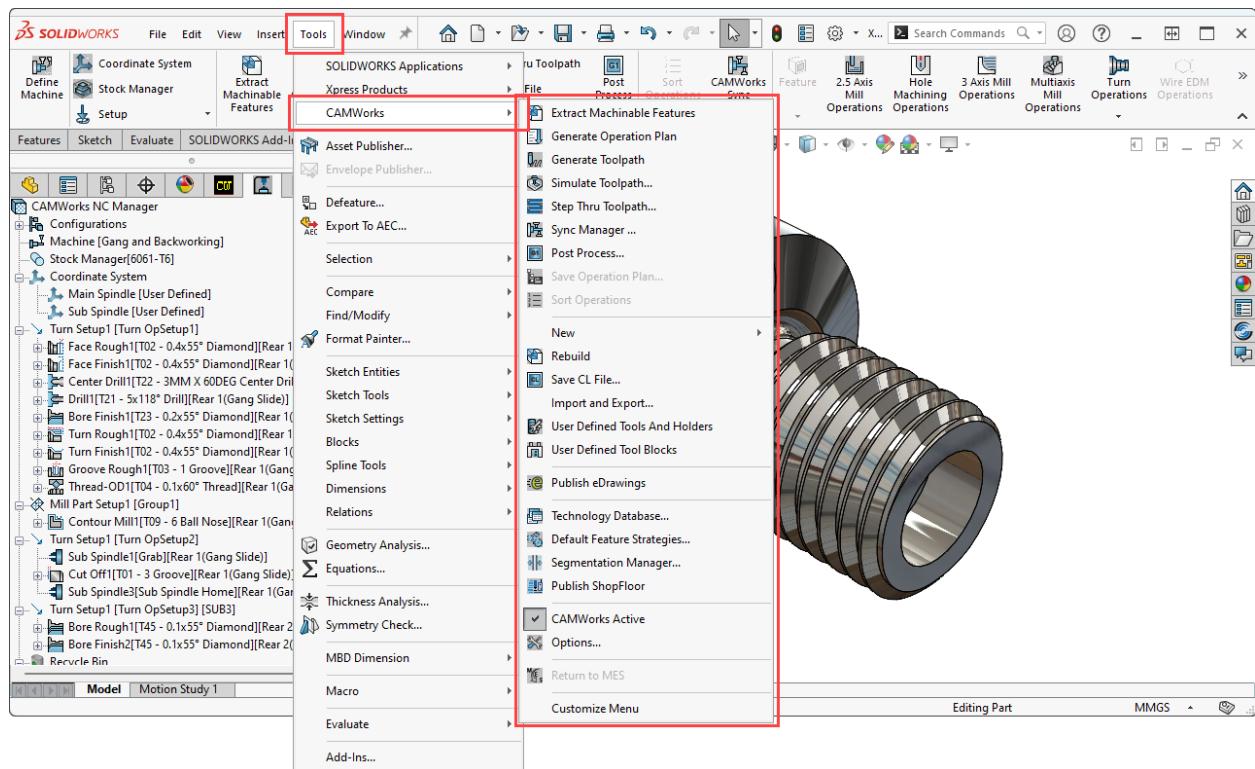
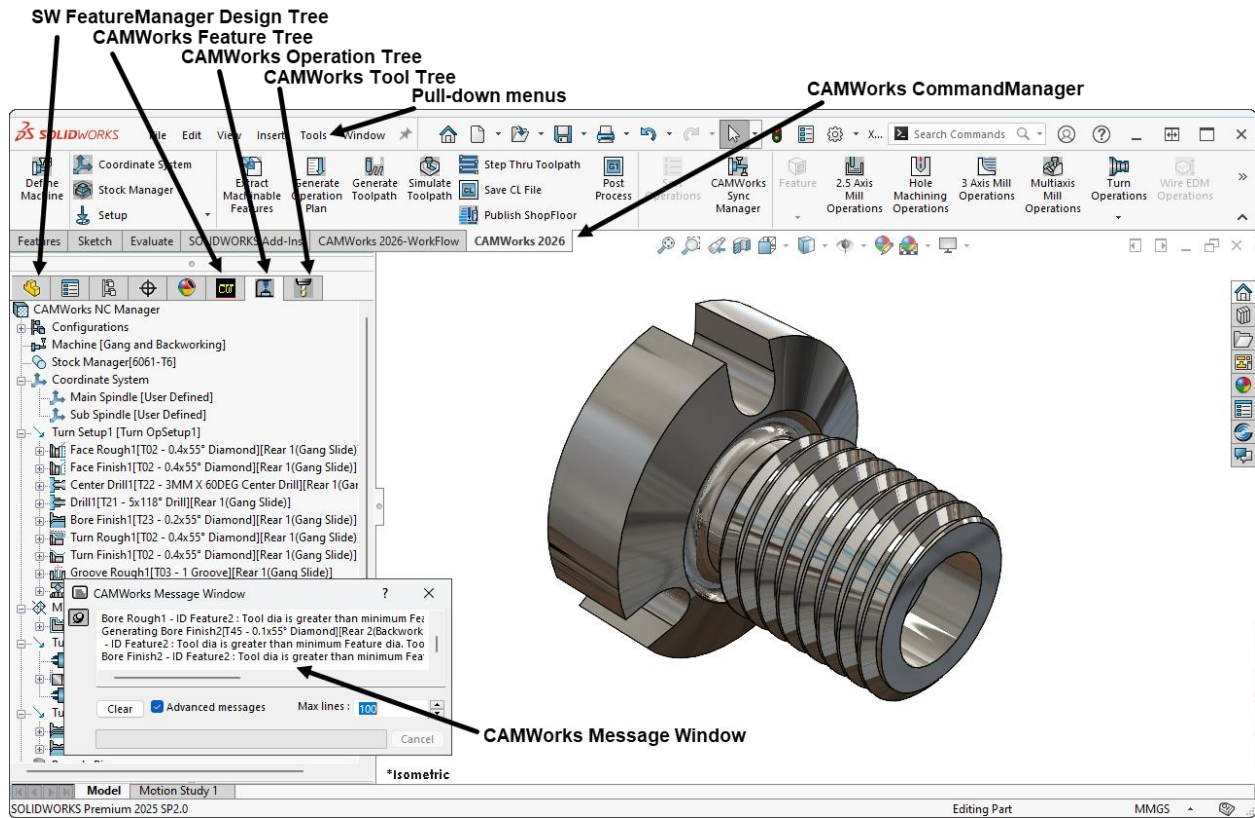
To turn on the CAMWorks Add-in:

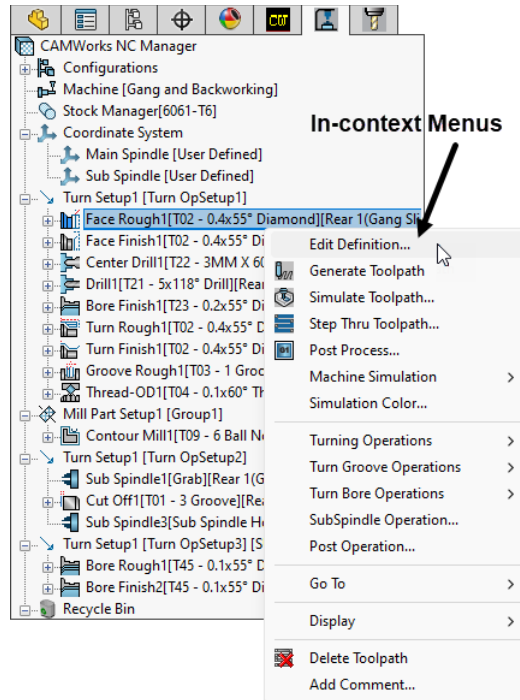
- Menu bar: **Options** , **Add-ins, CAMWorks 2026**
- CommandManager: **SOLIDWORKS Add-ins > CAMWorks 2026**
- Menu: **Tools, Add-Ins, CAMWorks 2026**

User interface

CAMWorks User Interface is an extension to the SOLIDWORKS native Windows interface and behaves in much the same way. Here is a list of the important parts of the interface:

- **SOLIDWORKS Feature Manager Design Tree** – Displays all the features of the part or assembly file in SOLIDWORKS.
- **CAMWorks Feature Tree** – Displays the machinable features and setups of the part or assembly.
- **CAMWorks Operations Tree** – Displays the machining operations of the part or assembly.
- **CAMWorks Tool Tree** – Show the tool crib, or list of available tools in the part or assembly file.
- **CAMWorks CommandManager Tool bar** – Contains easy access to the CAMWorks commands.
- **CAMWorks Pull Down Menu** – Provides access to the CAMWorks commands.
- **CAMWorks Message Window** – Reports on what processes have occurred.





What is Swiss Turn Machining?

Swiss turn machining, also known as Swiss screw machining or sliding headstock turning, is a high-precision CNC machining process specifically designed for producing small, complex, and intricate parts with extremely tight tolerances. It was originally developed in Switzerland in the 19th century for the watchmaking industry, which required miniature components with exceptional accuracy.

How it Works

The fundamental difference between Swiss turning and conventional turning lies in how the raw material (typically bar stock) is supported and moved during the machining process.

Sliding Headstock

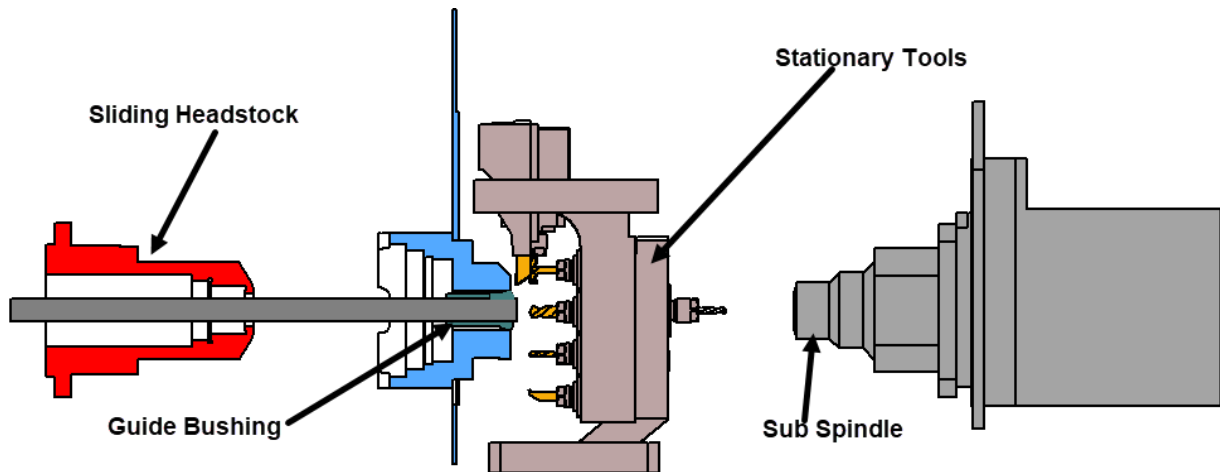
Unlike conventional lathes, the workpiece in a Swiss turn machine is held in a collet and moves linearly along the Z-Axis (Sliding Headstock) while being supported by a guide bushing. This minimizes deflection, enabling high precision for long, slender parts.

Guide Bushing

The guide bushing provides support to the bar stock directly at the point of cutting, minimizing deflection and vibration. This allows for extremely accurate cuts, even on long and slender parts with high length-to-diameter ratios (up to 20:1 or more).

Stationary Tools

The cutting tools themselves remain largely stationary, positioned very close to the guide bushing. As the bar stock is fed through the bushing, the tools engage the material to perform various operations.



Simultaneous Operations

Modern CNC Swiss machines often feature multiple axes (up to 13) and can have several tooling stations, including a main spindle and a sub-spindle. This enables them to perform multiple operations simultaneously, such as turning, milling, drilling, threading, and even laser cutting, all in a single setup. This significantly reduces cycle times and eliminates the need for secondary operations.

Advantages of Swiss Turn Machining

Swiss turn machining offers several key advantages, particularly for specific types of parts and production needs:

- **Exceptional Precision and Tight Tolerances:** The primary benefit is the ability to achieve extremely high precision, often with tolerances as tight as ± 0.0001 inches. The guide bushing support virtually eliminates part deflection, leading to superior accuracy and repeatability.
- **Ideal for Small, Complex, and Slender Parts:** It excels at manufacturing miniature components, parts with intricate geometries, and long, thin shafts or pins that would be prone to whipping or bending on a conventional lathe.
- **Reduced Cycle Times:** By performing multiple operations concurrently in a single setup, Swiss machines drastically reduce overall production time, especially for complex parts.
- **High-Volume Production:** With automatic bar feeders and the ability to run unattended for extended periods, Swiss turning is highly efficient for large production runs.
- **Excellent Surface Finishes:** The rigid support and precise cutting lead to very smooth surface finishes, often eliminating the need for post-machining finishing processes.
- **Versatility in Materials:** Swiss machines can effectively process a wide range of materials, including various metals (aluminum, titanium, stainless steel, brass, copper, nickel alloys, carbon steel) and even plastics.

Applications

Due to its precision and efficiency, Swiss turn machining is indispensable in industries where the highest standards are required for small, intricate components. Common applications include:

- **Medical Devices:** Surgical screws, implants, dental instruments, and other miniature medical components requiring biocompatible materials and strict tolerances.
- **Aerospace:** Lightweight, high-precision components for aircraft engines, hydraulic systems, and communication equipment.
- **Electronics:** Connectors, pins, shafts, and other tiny parts for consumer electronics and industrial applications.
- **Automotive:** Fuel injector components, valve parts, and sensors.
- **Watchmaking:** Gears, pins, and other intricate parts for timepieces, a nod to the origin of the technology.
- **Firearms:** Firing pins, bolts, and triggers.
- **Musical Instruments:** Slender and detailed parts for woodwind and brass instruments.

Swiss turn machining has revolutionized the manufacturing of micro-components, allowing for the creation of smaller, lighter, and more efficient parts across a multitude of critical industries.

Lesson 1 – CAMWorks Swiss Turn Workflow and User Interface

Upon successful completion of this lesson, you will be able to:

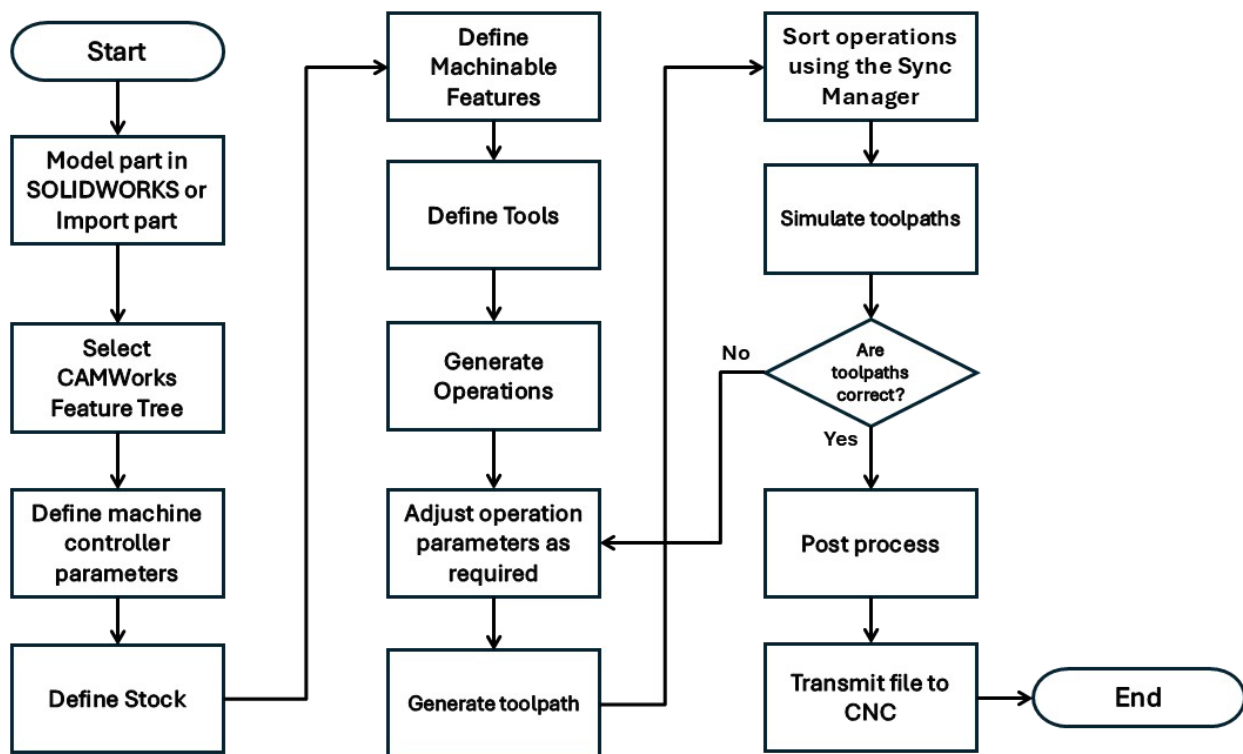
- Describe the process for creating and post processing toolpath for 3D models in SOLIDWORKS.
- Define the machine.
- Define the stock.
- Define a coordinate system.
- Describe methods for creating machinable features.
- Create milling and turning operations
- Generate toolpath
- Simulate toolpaths.
- Post process toolpaths

Process overview

With CAMWorks, we can create toolpaths through automated and interactive methods. Using the knowledgebase that can be captured in the Technology database, toolpaths can be automatically inserted based on the features.

Here is the process we use to generate toolpaths in CAMWorks for Swiss Turn Machining:

1. Model the part in SOLIDWORKS or CAMWorks Solids.
2. Select the CAMWorks Feature Tree.
3. Define Machine Controller Parameters.
4. Define Stock.
5. Define Machinable Features.
6. Define Tools available.
7. Generate Operations.
8. Adjust Operation Parameters as required.
9. Generate toolpath.
10. Sort operations using the Sync Manager.
11. Simulate toolpath.
12. Post process.
13. Transmit the file to CNC.



Case Study – Generate Toolpaths and NC Code

In this Case study, we will use the process described above to create milling and turning toolpaths and generate NC code on a SOLIDWORKS part file. This case study uses an existing SOLIDWORKS part file.

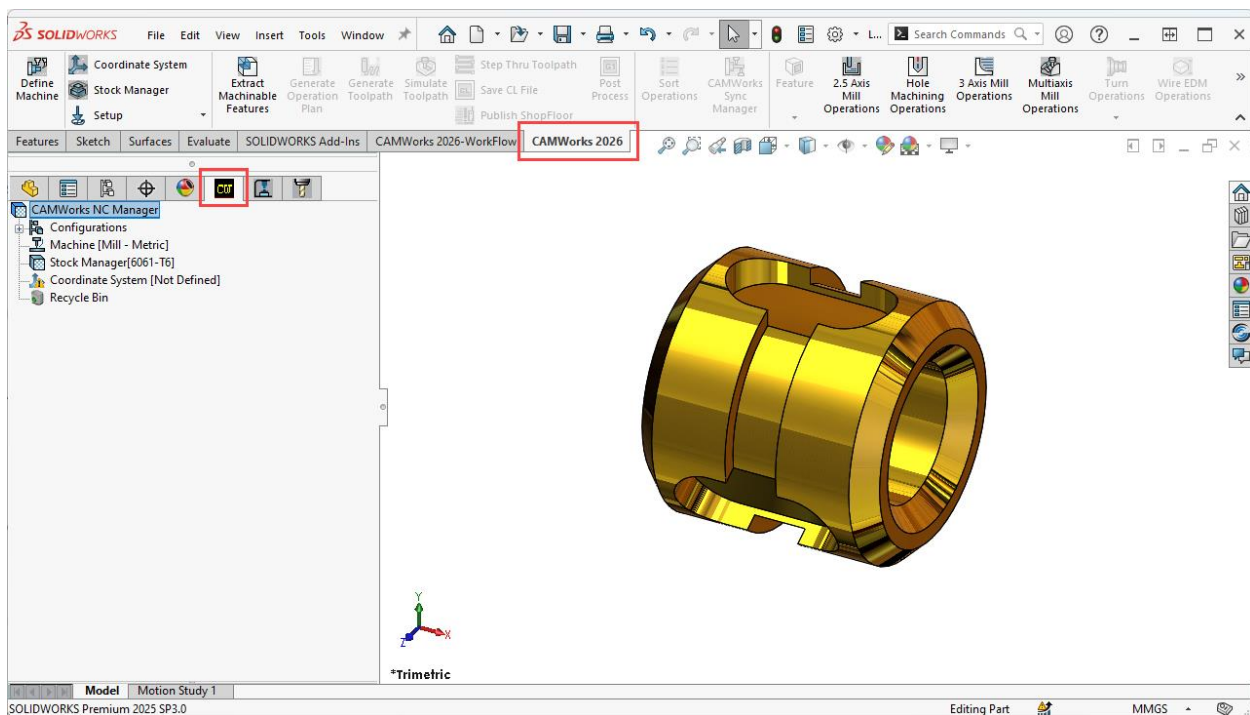
The CAMWorks menus discussed earlier will only become available when a part file is open in SOLIDWORKS.

1. Open Part.

- a. Open the part **LESSON_01.SLDPRT** in the ...\\Lesson_01\\ folder.

2. Select the CAMWorks Tabs.

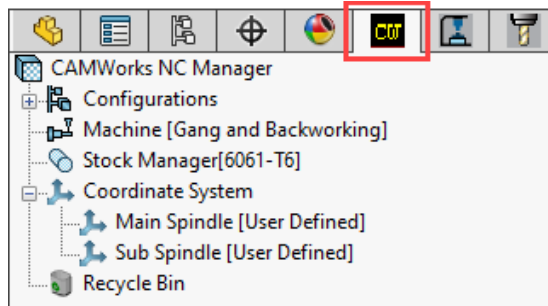
- a. Click on **CAMWorks 2026 CommandManager** tab and the **CAMWorks Feature Tree** tab.



CAMWorks Feature Tree

On the CAMWorks Feature tree we set the machining information for the model. Here, we setup the environment for Swiss Turn machining. This includes setting the Machine, Stock, coordinate system and tools used on the part. We start this by editing the machine definition and selecting a Swiss turn machine.

Initially, the CAMWorks Feature Tree lists the CAMWorks NC Manager, Machine, Stock Manager, Coordinate Systems and Recycle bin items. As you follow the steps to generate an NC program, this tree expands to include Turn Setups, Mill Part Setups and their respective machinable features.

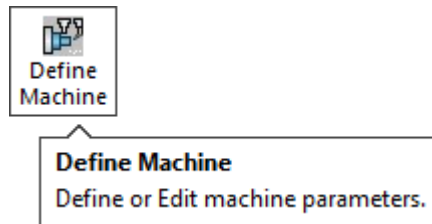



Define Machine

The machine dialog has multiple tabs that define what machine, tools, and post processor will be used to machine the part.

To access the **Machine** dialog:

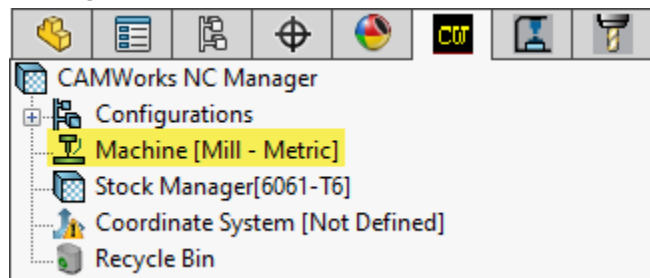
- CommandManager: **CAMWorks 2026 > Define Machine**



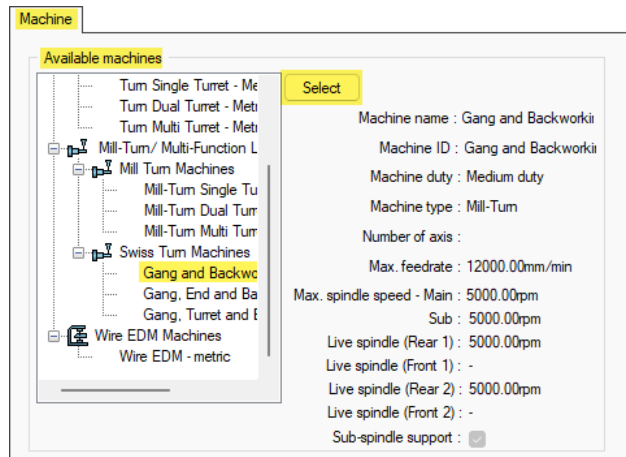
- CAMWorks Feature Tree: Right-click **CAMWorks NC Manager** and select **Define Machine**
- Toolbar: **Define Machine** 
- CAMWorks Feature tree: Double click the **Machine** item.

3. Define Machine

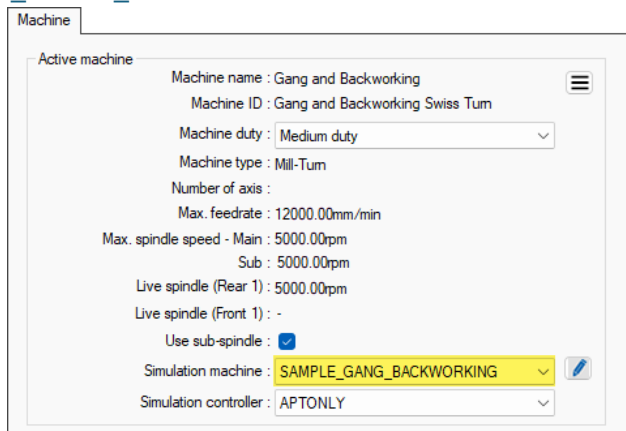
- Double click on the **Machine** item in the CAMWorks Feature Tree to open the Machine dialog.



- Click on the **Machine** tab.
- In the **Available machines** box, under **Mill-Turn/Multi-Function Lathe**, highlight **Gang and Backworking** under the **Swiss Turn Machines** area and then click the **Select** button.



- d. Under **Active machine**, set the **Simulation machine** to **SAMPLE_GANG_BACKWORKING**



This sets the machine that is used for Machine simulation and Machine Aware visualization.

- e. Click on the **Tool Crib** tab
- f. Ensure that the **Active tool crib** for **Turret: Rear 1** is **Gangslide GB (Metric)** and for **Turret: Rear** is **Backworking GB (Metric)** respectively.

Tool Crib

Active tool crib : Gangslide 1 Turret : **Rear 1**

☐ Moves In Z Axis

Usage	Stn. No.	Tool Type	ID	Comment	Dia. (mm)
1	1	Groove	240	CTPS15FR HOLDER	0
4	2	Diamond	237	DCGT 11T304 TURN HOLDER	9.53
	3	Diamond	238	DCMT 11T302 TURN HOLDER	9.53
2	4	Groove	231	GTMX32150 T01 OD HOLDER	0
	5	Groove	229	GTMH32 105RGX OD HOLDER	0
	6	Thread	232	CSV111FRP60-035A TURN HOLDER	0
	7	Center Drill	1	3MM X 60DEG HSS CENTERDRILL	3
	8	Bore	-1		
3	9	Flat End	11	6MM CRB 4FL 19 LOC	6
3	10	Flat End	8	4MM CRB 4FL 8 LOC	4
	21	Center Drill	4	6MM X 60DEG HSS CENTERDRILL	6

Add Tool... Edit Tool... Remove Tool Update Tool Save Tool Crib...

☐ Tool crib has sub stations

☒ Tool crib priority

☐ Use tool crib tools only

☒ Do not create new tool stations

☒ Allow Duplicate Station Numbers

Available tool cribs

Gangslide GB (M
Backworking GB
End Working GB
Gangslide CTR #

Select

Name : Gangslide GB (Metric)

No. of stations : 14

Tool Crib

Active tool crib : Backworking 1 Turret : **Rear 2**

☐ Moves In Z Axis

Usage	Stn. No.	Tool Type	ID	Comment	Dia. (mm)	Rad
2	41	Diamond	207	VCMT 070204 BORE BAR	3.97	0.4
	42	Flat End	7	4MM CRB 2FL 14 LOC	4	0
4	43	Diamond	247	VCMT 070204 BORE BAR	3.97	0.4
	44	Center Drill	9	6MM X 90DEG CRB SPOT DRILL	6	0
	45	Bore	-1			

Add Tool... Edit Tool... Remove Tool Update Tool Save Tool Crib...

☐ Tool crib has sub stations

☒ Tool crib priority

☐ Use tool crib tools only

☒ Do not create new tool stations

☒ Allow Duplicate Station Numbers

Available tool cribs

Gangslide GB (M
Backworking GB
End Working GB
Gangslide CTR #

Select

Name : Backworking GB (Metric)

No. of stations : 5

- g. Click on the **Post Processor** tab.
- h. For the **Active Post processor**, select **SW_GANG_BACK_TUTORIAL_FANUC** from the list of available post processors in the **C:\CAMWorksData\CAMWorks2026x64\posts** directory.

Post Processor

Active post processor :
C:\CAMWorksData\CAMWorks2026x64\posts\SW_GANG_BACK_TUTORIAL_FANUC

Available

C:\CAMWorksData\CAMWorks2026x64\posts\SW_GANG_BACK_TUTORIAL_FANUC

MT2AXIS-SIEMENS-TUTORIAL
MT2AXIS-TUTORIAL
MT4AXIS-TUTORIAL
MT5AXIS-TUTORIAL
SW_GANG_BACK_TUTORIAL_FANUC
SW_GANG_END_BACK_TUTORIAL_FANUC

Browse...
Select
☐ APT CL

☐ Do not show license expired post processors

- i. Click on the **Fixture** tab.
- j. Set the Main and Sub Spindle collets to the **TF 30 Collet**.

Fixture

Main Spindle Information

Shape : Collet

Name : TF 30

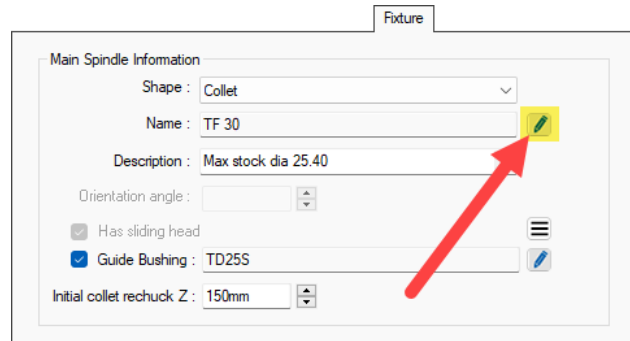
Description : Max stock dia 25.40

Orientation angle :

☒ Has sliding head

☒ Guide Bushing : TD25S

Initial collet rechuck Z : 150mm



- k. In the **Collet Parameters**, uncheck the **Collet Housing Parameters** check box.

Collet Parameter : [Main Spindle]

✓ ✗

Collet Management

Type/Style : [Active] SC

Available Collets : [Active] TF 30 ✗

Name : TF 30

Collet Parameters

Stock Diameter (ID) : 25.4mm

Max Diameter (OD) : 35mm

Nose Diameter : 30.1mm

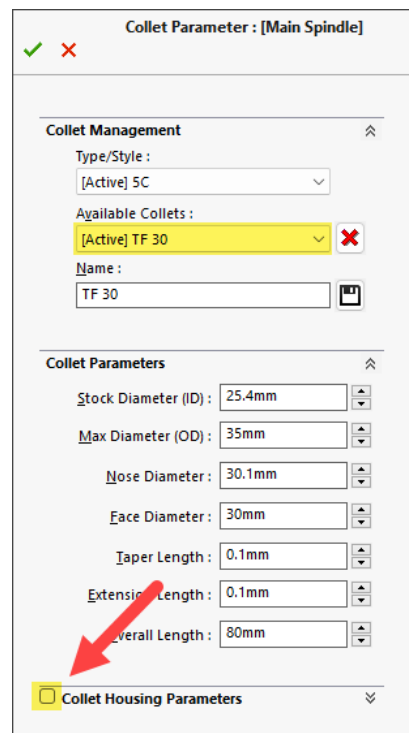
Face Diameter : 30mm

Taper Length : 0.1mm

Extension Length : 0.1mm

Overall Length : 80mm

☐ Collet Housing Parameters



- l. Do the same to the **Sub Spindle Information**.
- m. Edit the **Guide Bushing**.

The screenshot shows the 'Fixture' dialog box with two sections: 'Main Spindle Information' and 'Sub Spindle Information'. Both sections have fields for Shape (Collet), Name (TF 15 4630), Description (None), and Orientation angle. The 'Main Spindle Information' section has checkboxes for 'Has sliding head' and 'Guide Bushing' (set to TD25S), and a field for 'Initial collet rechuck Z' (100mm). The 'Sub Spindle Information' section has a diagram showing the distance D between the spindles, with a field for 'Distance (D)' (300mm) and a 'Reference for D' (Front of the fixture face). A red arrow points to the 'Guide Bushing' dropdown menu in the 'Main Spindle Information' section.

- n. Under **Guide Bushing Management**, select **P2553D** from the **Available Guide Bushing list**.

The screenshot shows the 'Guide Bushing' dialog box. It has a 'Guide Bushing Management' section with a dropdown menu for 'Available Guide Bushing' (set to P2553D) and a 'Name' field (set to P2553D). There are green and red checkmarks at the top left of the dialog box.

- o. Set the **Initial collet rechuck Z**: to **150mm**.

The screenshot shows the 'Fixture' dialog box with the 'Main Spindle Information' section. The 'Initial collet rechuck Z' field is now set to 150mm. The 'Guide Bushing' dropdown menu is also set to P2553D. The 'Name' field is now TF 30 and the 'Description' is Max stock dia 25.40.

- p. Click **OK**.

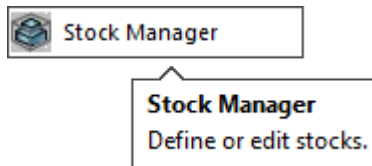
Define Stock


The stock in the material from which the part will be machined.

The default stock shape for Mill/Turn and Swiss Turn parts in the smallest cylinder (Round Bar Stock) that the part will fit into along the Z axis. This is most likely not the size of the actual stock you will be using. You can change the stock definition by offsetting the Bar stock parameters from the part. Other options to define the stock shape include defining the stock from a revolved sketch, a 2d WIP file, an STL file, or a part file.

To Access the Stock Manager:

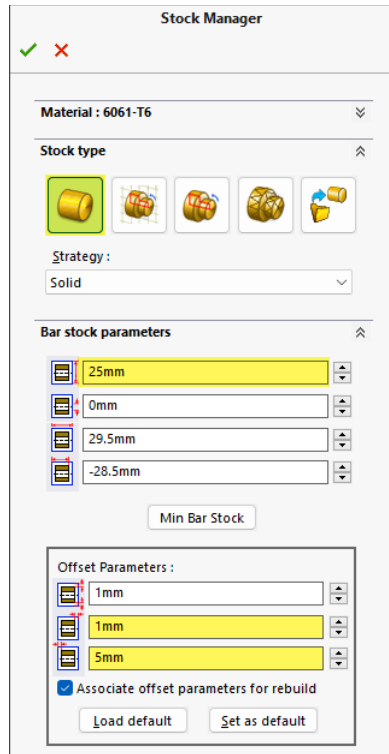
- CommandManager: **CAMWorks 2026> Stock Manager**



- Toolbar: **Stock Manager** 
- CAMWorks Feature Tree: Right Click the **Stock Manager** item and select **Edit Definition**
- CAMWorks Feature Tree: Double-click the **Stock Manager** item

4. Define Stock

- a. Double-click on the **Stock Manager** in the CAMWorks Feature Tree.
- b. Set the **Stock type** to **Round Bar Stock**.
- c. Under **Bar stock parameters**, set the **Outside Diameter** to **25mm**.
- d. Under **Offset Parameters**, set the **Front Face of Stock Offset** to **1mm**.
- e. Set the **Back Face of Stock Offset** to **5mm**.



f. Click **OK**.

Define Tools

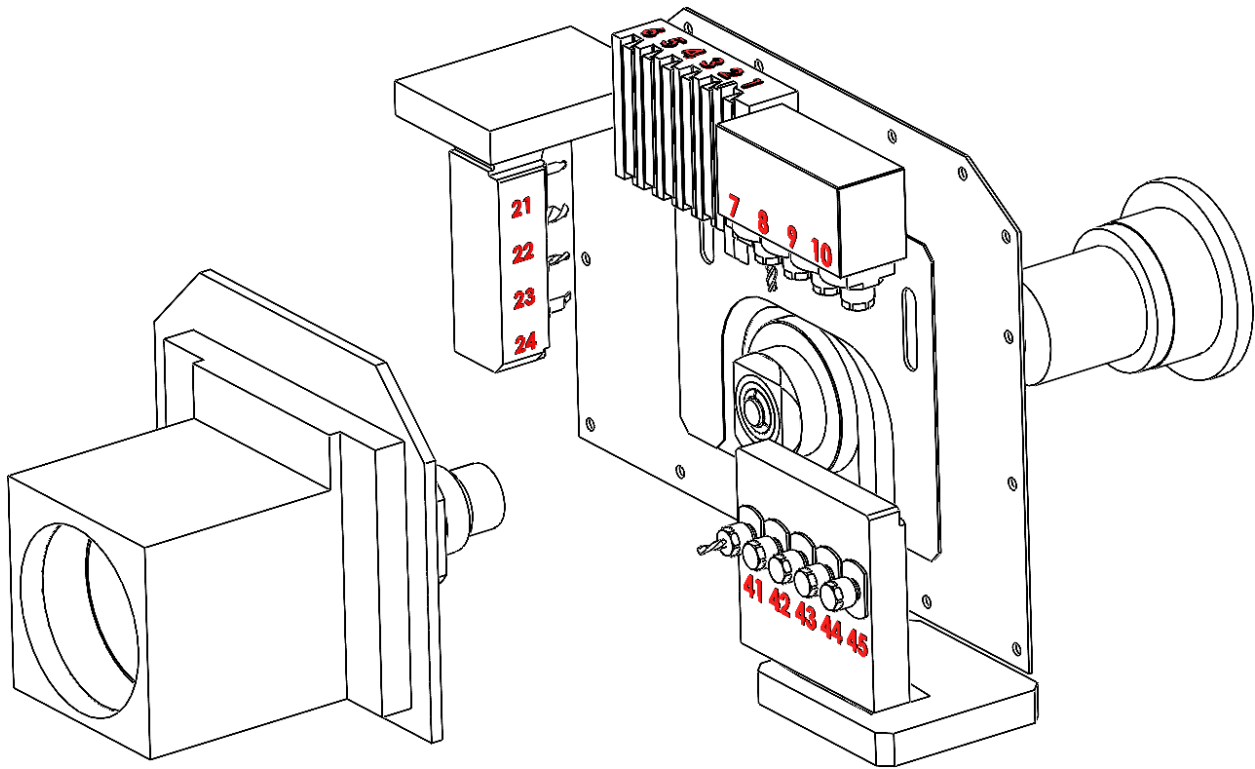
Selecting tools for a Swiss Turn machining project, especially considering the limited tool positions and orientations, is a highly strategic and often challenging aspect of programming and setup. Unlike conventional lathes where a turret can have many tool stations, Swiss machines often have a more compact tooling zone, requiring careful planning to maximize efficiency and part quality.

Understanding the Constraints of Swiss Tooling:

- **Gang-Type Tooling:** Many Swiss machines utilize a "gang-type" tool post, where tools are mounted in a linear arrangement on a slide. This means tools are generally fixed in their X and Z positions relative to each other, and their cutting edges are oriented for specific operations (e.g., OD turning, ID drilling). The number of positions is finite.
- **Limited Work Envelope:** The cutting zone in a Swiss machine is very close to the guide bushing, which provides supreme rigidity. While this is a huge advantage for slender parts, it also means there's a confined space for tools to operate without collision.
- **Live Tooling Positions (if applicable):** While Swiss machines are known for their gang slides, many modern machines also incorporate live tooling (rotary tools for milling, drilling off-center, etc.). These live tools are typically mounted in specific positions (axial or radial) and may have limitations on the size of the tool and the length of its reach. Some advanced machines feature a B-axis or even a Y-axis on the live tool block, offering more flexibility, but still within defined limits.

- **Sub-Spindle Operations:** The sub-spindle allows for back-side machining, but the tools used for these operations are distinct and located in different physical positions than the main spindle tools.
- **Simultaneous Machining:** The ability to perform simultaneous operations (e.g., turning on the main spindle while drilling on the sub-spindle) is a core advantage of Swiss machines, but it dramatically increases the complexity of tool selection and collision avoidance.

For this example, the following stations are available on the machine we are programming:



Tools 1 – 6: These are static turn tools for Face, OD Turning, OD Grooving, and Threading. Typically, Tool station 1 is dedicated to the cut-off tool so Tools 2 through 6 will be used for turning external geometry.

Tools 7 – 10: These are live tools for machining milling features on the OD of the part.

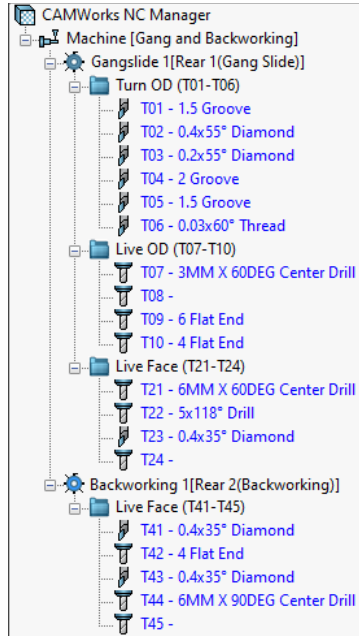
Tools 21 – 24: These are live and statics tool for machining features on the face and ID of the part

Tools 41 – 45: These are live and static tools for machining features on the backside of the part after it has been transferred to the sub spindle.

Other machines will have different sets of tools stations and orientations available.

5. Define Tools

- Click on the **CAMWorks Tool Tree** tab.
- Examine the tools available



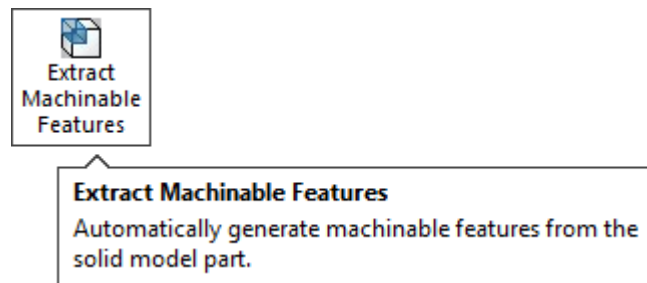
For now, the tools defined in the current sets of tool cribs are what we are going to use for this part.


Define Machinable features

In CAMWorks, machining is done on machinable features. These features are defined either interactively, through Interactive Feature Recognition (IFR), or automatically using Automatic Feature recognition, or AFR.

To access Extract Machinable Feature command:

- CommandManager: **CAMWorks 2026 > Extract Machinable Features**



- Menu: **Tools, CAMWorks, Extract Machinable Feature**
- Toolbar: **Extract Machinable Features** 
- CAMWorks Feature tree: Right-click on **CAMWorks NC Manager** and select **Extract Machinable Feature**
- Interactive Feature Recognition (IFR)**

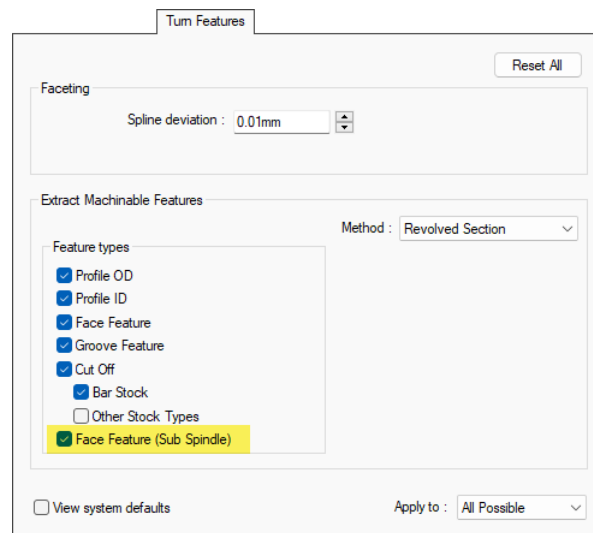
AFR cannot recognize every feature on complex parts and does not recognize some types of features. To machine these areas, you need to define features interactively using the New 2.5 Axis Feature command.

We will start by defining the features to be defined from running AFR in the CAMWorks Options to define features more optimally for Swiss-Turn machining.

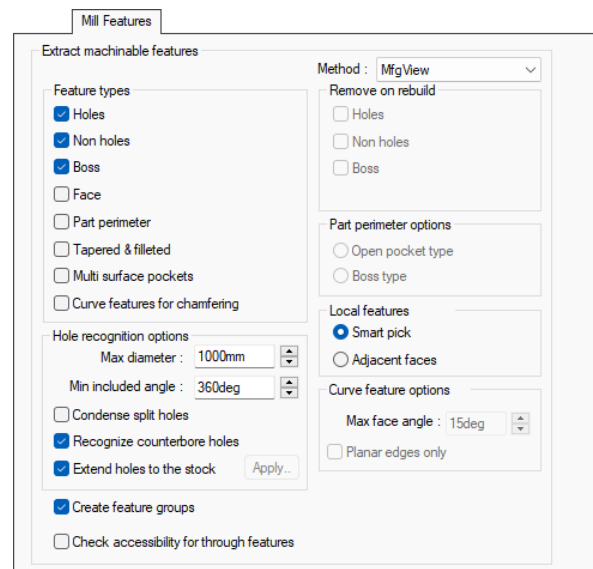
One of the operations we want to do is after the part is cut off and transferred to the sub-spindle, there should be an extra facing operation to finish the face.

6. Set AFR options.

- Right-click on **CAMWorks NC Manager** and select **Options**.
- Click on the **Turn Features** tab.
- Under **Feature types** in the **Extract Machinable Features** area check on the following options:



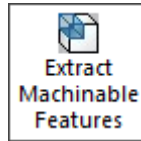
- Go to the **Mill Features** tab and ensure that the only feature types selected are **Holes**, **Non holes**, and **Boss**.



- Click **OK**.

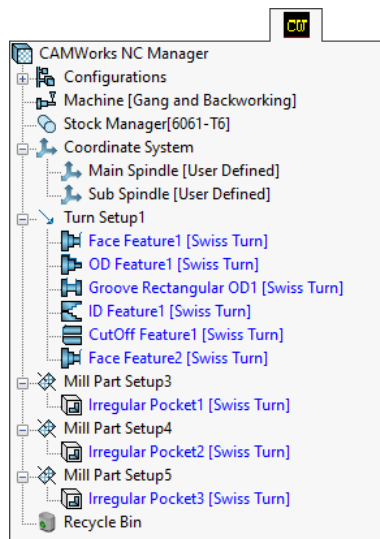
7. Extract Machinable Features.

- Click **Extract Machinable Feature** from the CAMWorks CommandManager toolbar.



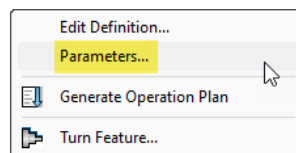
8. Examine the Results.

- Click on the CAMWorks Feature Tree to see the features that were defined.

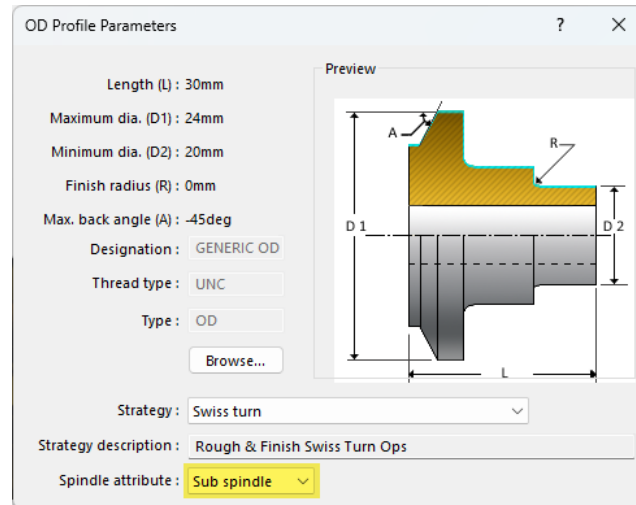


9. Interactively define additional features.

- CRTL+Drag and drop on **OD Feature1** to copy it.
- CRTL+Drag and drop on **ID Feature1** to copy it.
- Rename **OD Feature1 – Copy** to **OD Feature2**.
- Rename **ID Feature1 – Copy** to **ID Feature2**.
- Right-click on **OD Feature2** and select **Parameters** from the list.



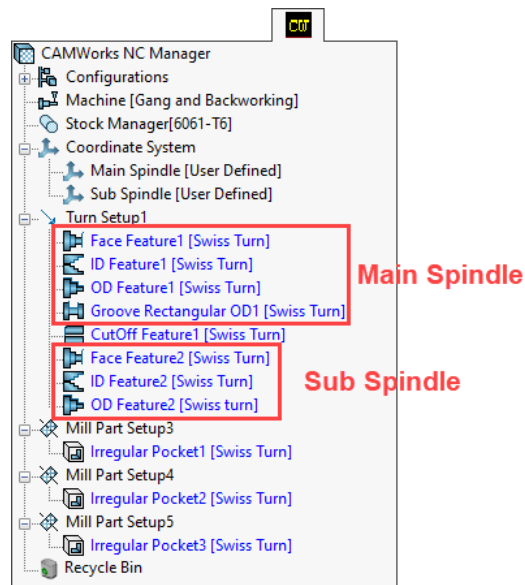
- Set the **Spindle attribute** to **Sub Spindle**.



- g. Do the same for **ID Feature2**.

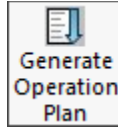
Generally, in Swiss-turn machining, after the initial facing operations, we process the ID operations before the OD operations. To better facilitate this, it can be easier to reorder the features in the CAMWorks Features tree so that after the Face feature, the ID feature is next, and then the OD features.

- h. Re-order the turn features so that the ID Features are directly after the Face Features on both main and sub spindles.



Generate Operations

The next step is to generate the operations for this part. This is done using the Generate Operation plan command on the CAMWorks Command manager

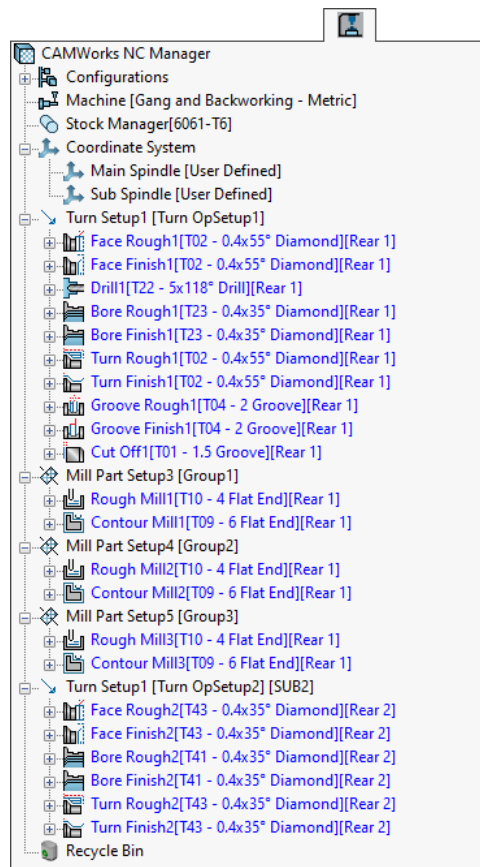


In the CAMWorks Feature tree, we can see that the strategy is set to [Swiss Turn]. This strategy for all the features has been selected automatically because of the Swiss turn machine type being selected earlier. With this strategy type selected, the operations that generates from the Generate Operation Plan command will be better optimized from Swiss Turn machining. This includes parameters such as tool selection, start and end point for the toolpath, etc.

Additional operations can be defined later.

10. Generate Operations

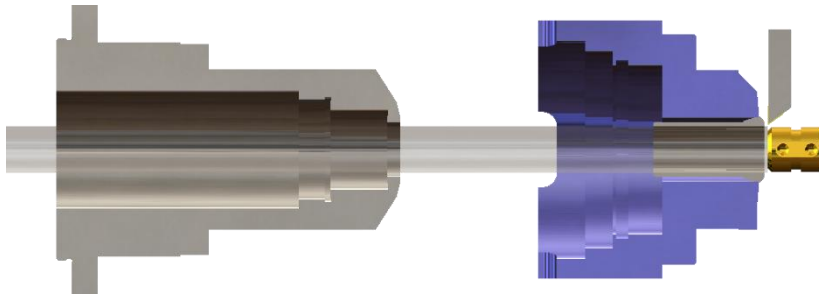
- a. Run the Generate operation plan command on the CAMWorks Command Manager toolbar.



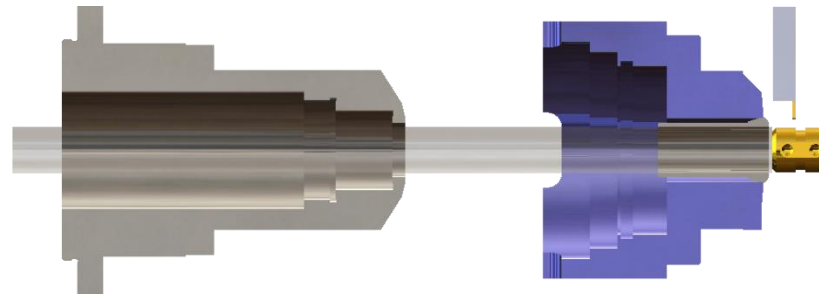
Define the sub-spindle transfer/cutoff operation

After the Milling and Turning operations are completed on the main spindle, the part will need to be separated from the barstock using a cutoff operation and transferred to the sub spindle to cut the back side of the part. Typically, the following steps occur as the part is cut off and transferred to the sub spindle:

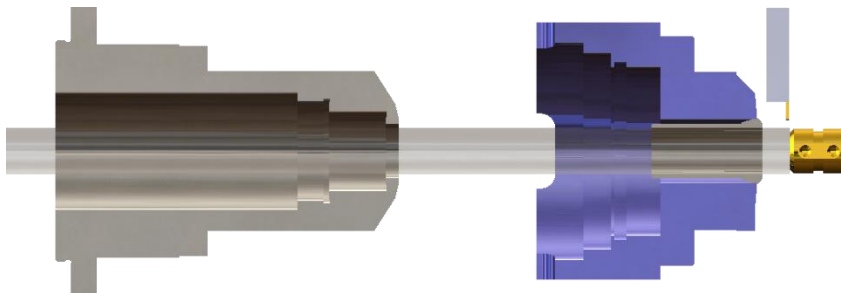
Turn and Milling operations finish.



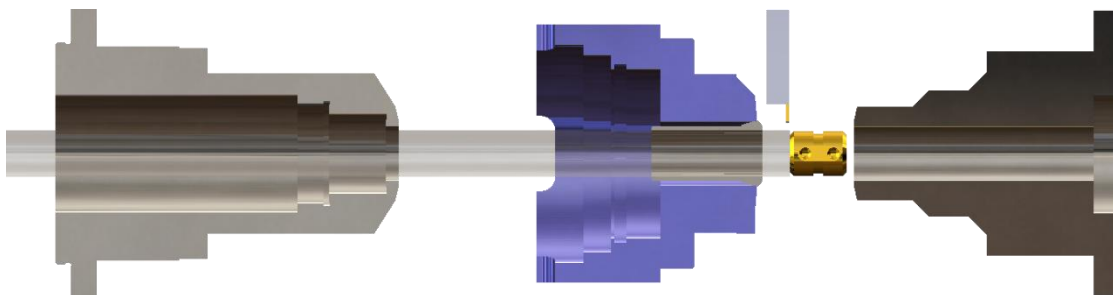
Position the cutoff tool – The cutoff tool is moved into position for the cutoff operation.



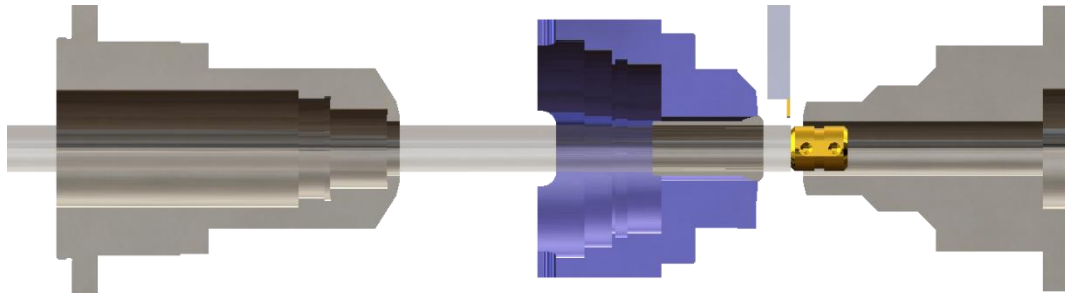
Position the part/stock – the stock and part move in Z to the position of when it will be cut.



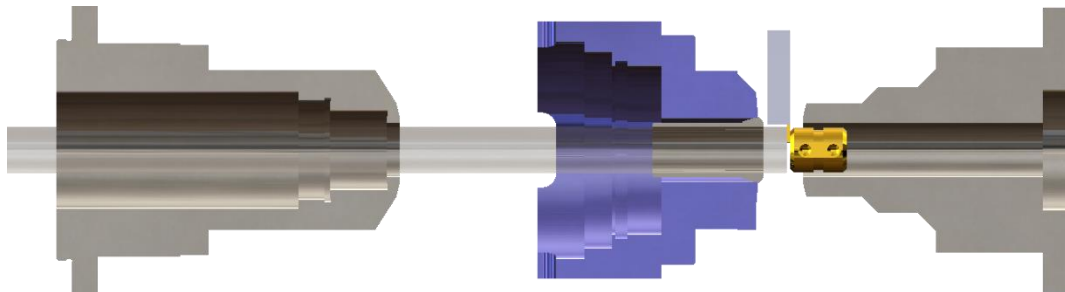
Sub Spindle rapids to clearance



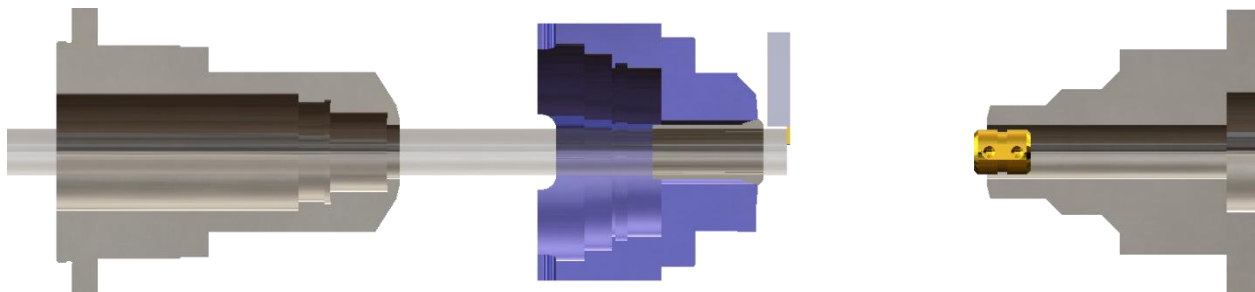
Sub Spindle Feeds to grip position and clamps onto workpiece.



Cut off operation –



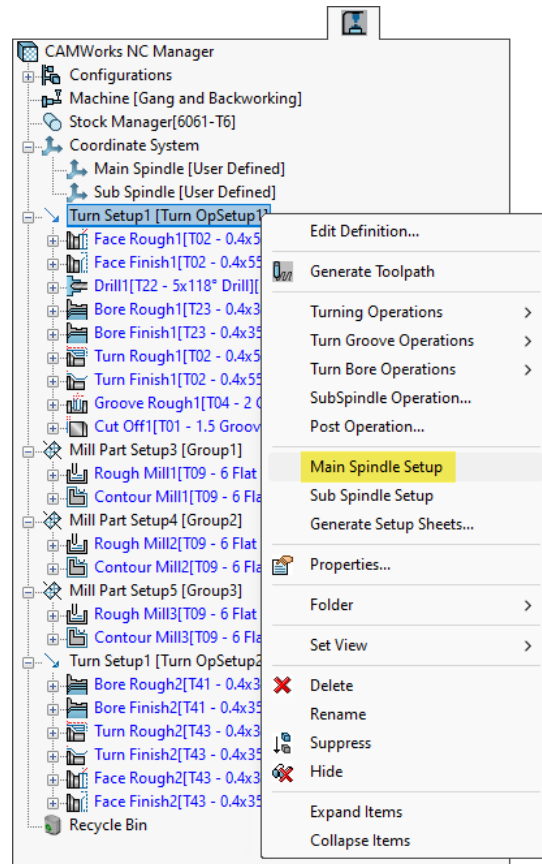
Sub Spindle rapids home and machining operations continue on the sub spindle.



For this to happen, we need a separate turn setup for the Cut off operation so that it can be reordered after the milling operations on the OD. We will then insert a sub spindle operation.

11. Insert a new Turn setup.

- a. **Right-click** on **Turn Setup1** and select **Main Spindle Setup**.

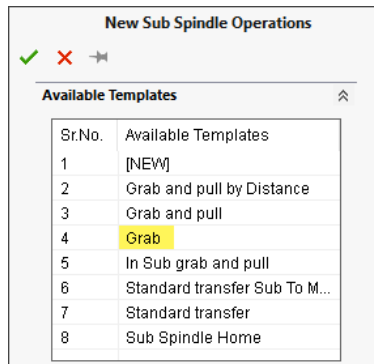


- Drag and drop **Cut Off1** to the new **Turn Setup**.
- Drag the new **Turn Setup** below the last Mill Part Setup.



12. Insert Sub-Spindle Operation.

- Right-Click** on the new **Turn Setup1** and select **Sub Spindle Operations**.
- Under the **Available Templates**, select **Grab**.



- Click **OK**.

As mentioned above, the cut off operation happens after the part is grabbed by the sub spindle. To initialize this, the positioning of the cutoff tool and the sub spindle need to occur simultaneously between the G-Code controlling the gang slide where the cutoff tool is and the Sub spindle.

This will launch the Sub Spindle operation Parameters for the Grab operation. The first step we will do is associate this operation to the Cutoff feature.

Associate to an operation (Checkbox Option)

This option is used to associate the Sub Spindle operation to another operation machined on the Main Spindle.

Associating a Sub Spindle operation with another operation ensures that both these operations are automatically synchronized in the CAMWorks Sync Manager. This functionality is especially useful for synchronizing Cut Off operation with Sub Spindle operations when machining using Turn, Swiss Turn, Mill-Turn or Multi-Function Lathe machines.

- d. Check on the **Associate to an operation** checkbox.
- e. Set the **Associated to:** to **Cut Off1**.
- f. Under **Steps**, set the **Spindle Feed Move** to **15mm** from the Front face of the part.

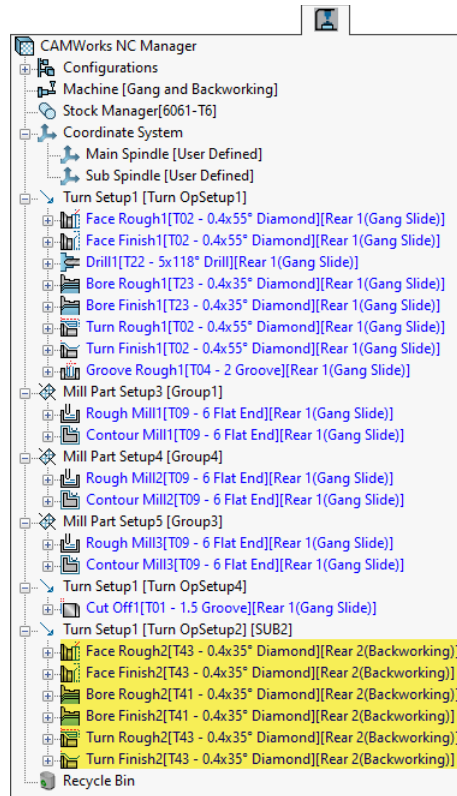
The screenshot shows the 'Sub Spindle Parameters' dialog box with the following settings:

- Name and Description:**
 - Name: Grab
 - Description: This will position to grab for a cutoff
 - ☒ Associate to an operation
 - Associated to: Cut Off1
- Steps:**
 - Spindle Speed [Main, Off]
 - Spindle Speed [Sub, Off]
 - Spindle Orient [Main, C:0°]
 - Spindle Orient [Sub, C:0°]
 - Spindle Rapid [Z:-20]
 - Spindle Feed [Z:-15, F:125.00]**
 - Spindle Fixture Clamp [Sub]
 - Spindle Sync Turn mode [Speed]
 - Spindle Speed [Main, CW, S:1000.00]
- Turret:** Rear 2(Backworking)
- Spindle Feed Move:**
 - ☒ Absolute to: Front Face of Part
 - Clearance: 15mm
 - ☐ Incremental by: Distance
 - Clearance: 0mm
 - Feedrate: 125.00mm

- g. Click **OK**.

13. Re-Order operations

- a. Reorder the operations **under Turn Setup1[Turn OpSetup2][SUB2]** so that the Face operations are first, flowed by the Bore Operations, and then the OD Operations.



14. Generate toolpath

- Generate the toolpaths.

Modify Operation Parameters

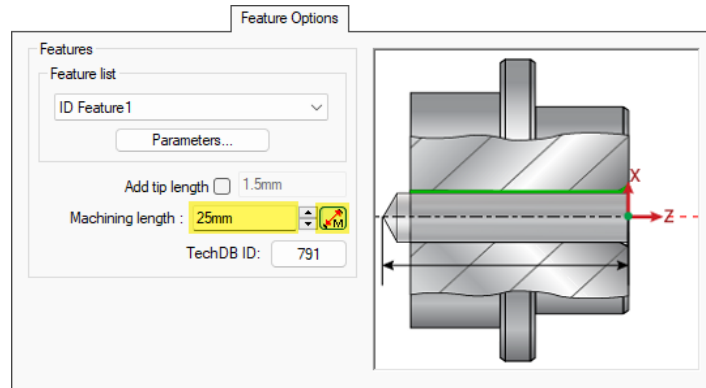
Under the Turn Setup for the operations on the sub-spindle, we can see that some of the operations didn't generate. This is because some of the previous operations have already removed some of the stock material intended for these operations. For example, the Cut Off1 operation cuts off the material at the stock.

We might want that operation to leave extra material so it can be faced off more accurately from the back working tools while on the Sub Spindle.

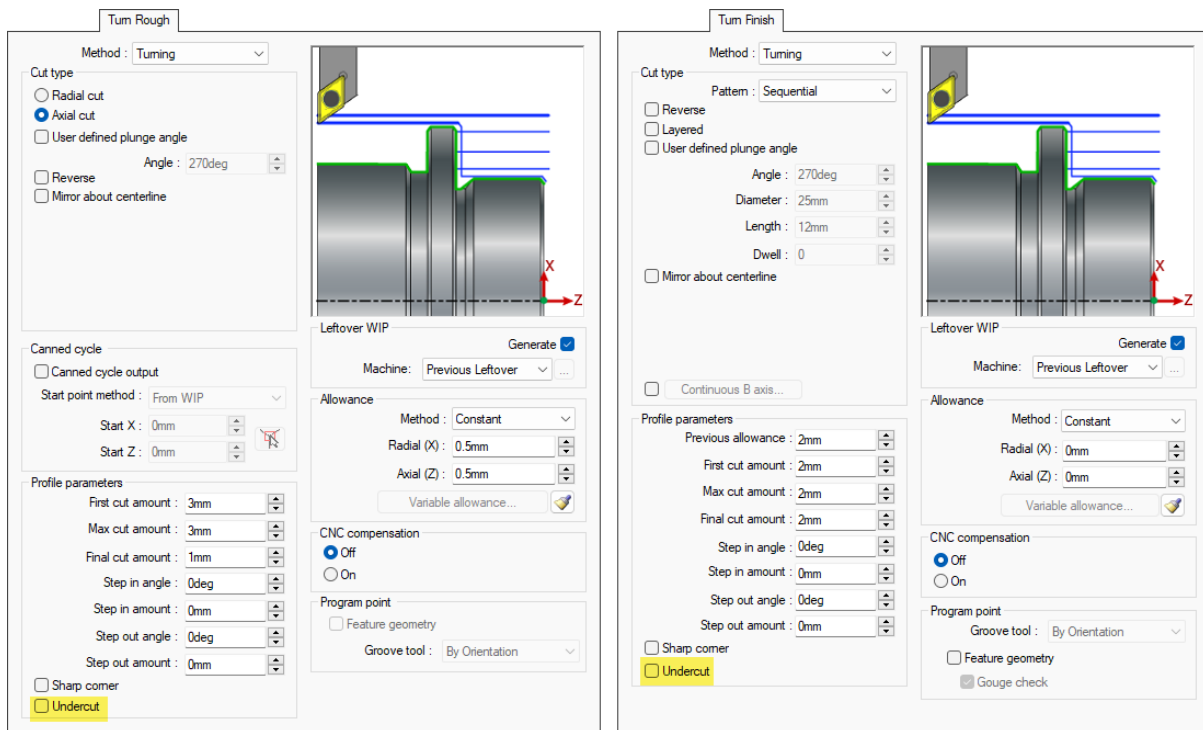
We can go back and edit some of these operations earlier in the tree to ensure enough material is left to machine. We will start with the Hole to machine sure that it machines deep enough.

15. Modify Operation Parameters.

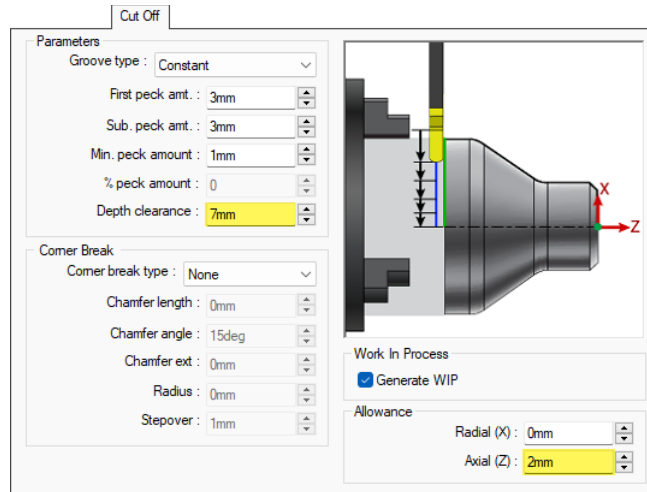
- Edit the operation **Drill1**.
- Go to the **Feature Options** tab
- Override the machining depth and enter **25mm** for the **Machining length**.



- d. Click **OK**.
- e. Edit **Turn Rough1** and go to the **Turn Rough** tab.
- f. Uncheck the **Undercut** checkbox.
- g. Click **OK**.
- h. Edit **Turn Finish1** and go to the **Turn Finish** tab.
- i. Uncheck the **Undercut** checkbox.
- j. Click **OK**.



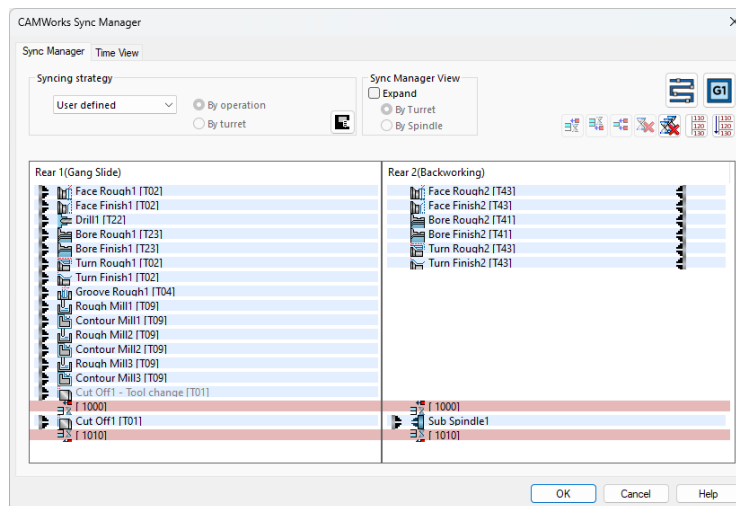
- k. Edit **Cut Off1** and go to the **Cut Off** tab.
- l. Under **Parameters**, set the **Depth clearance** to 7mm.
- m. Under **Allowance**, set the **Axial (Z)** to 2mm.



n. Click **OK**.

Sync Manager

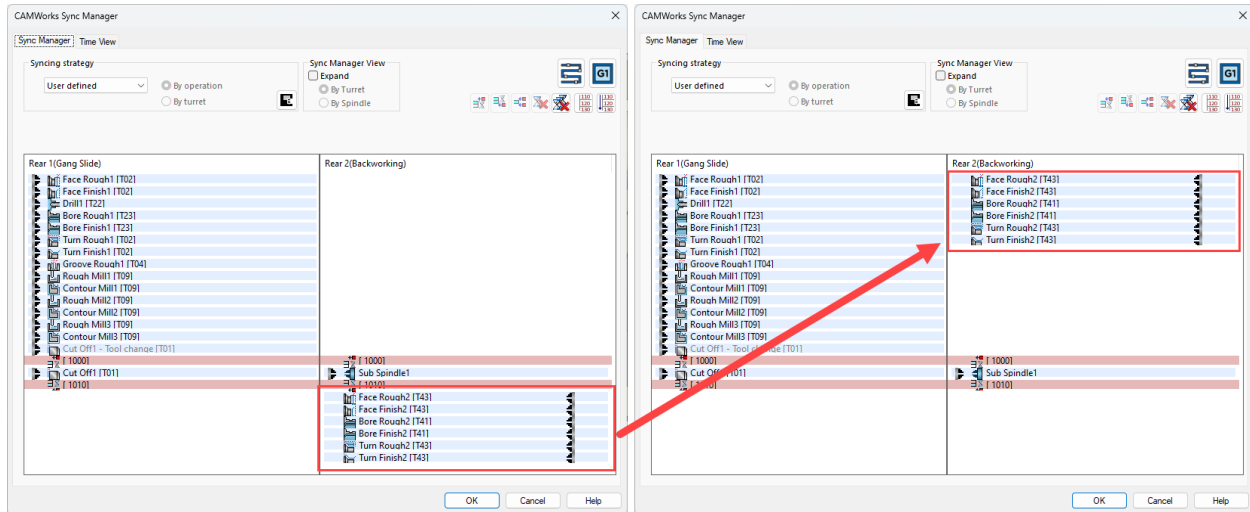
The order of operations and how the channels (Main and Sub Spindle, Gang Slide tools and Back Working tools) communicate with each other is controlled in the Sync Manager. Generally, we want operations performed on the main spindle and sub spindle to be placed at the top of the sync manager and the transfer operation at the bottom.



By doing this, it might raise concern about the movement of the sub spindle when the program is first run. The Machine will know if there is any stock in the sub spindle for this first time through the program and it won't run those operations. Subsequent run-throughs of the program will machine the operations on the sub spindle.

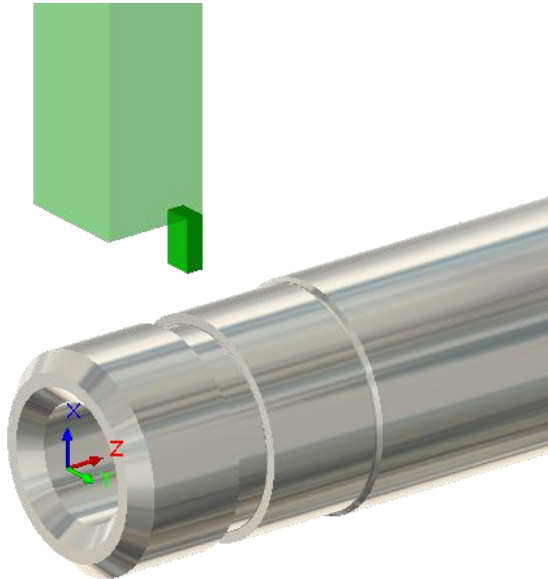
16. Launch the Sync Manager.

- Move all the turning operations on the sub spindle to the top.



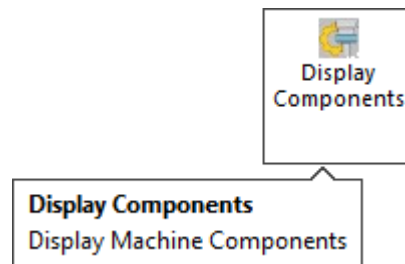
17. Simulate toolpath.

- Simulate the toolpath.

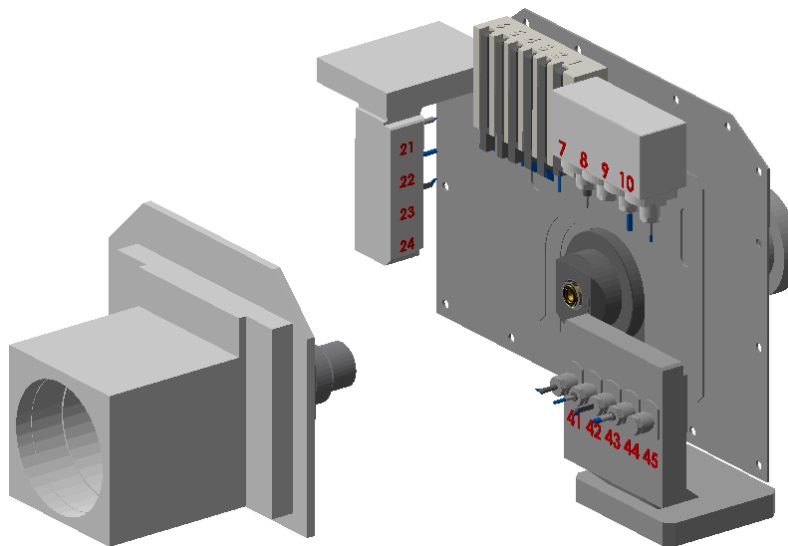


Machine Aware Programming

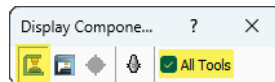
Before we run the simulation to verify the toolpath, we can display the machine to better visualize how the part and the tools will move during the machining. This can be shown using the Display Components command from the CAMWorks CommandManager toolbar.



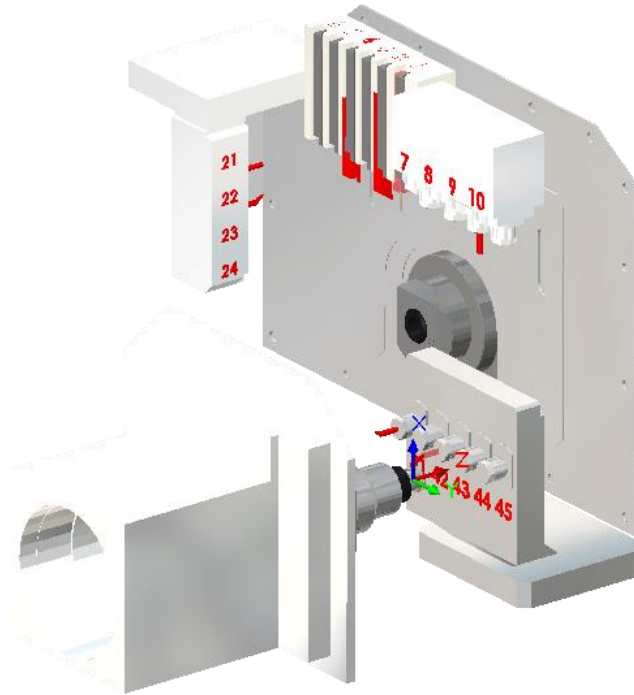
This Command will enable the Display Components toolbar that allows us to show the machine components. Here is how the current machine displays.



- b. Click on the **Display Components** toolbar.
- c. Click on the **Display Machine** Icon.
- d. Check on the **All Tools** checkbox. This will display the tools loaded in their turret stations on the machine.



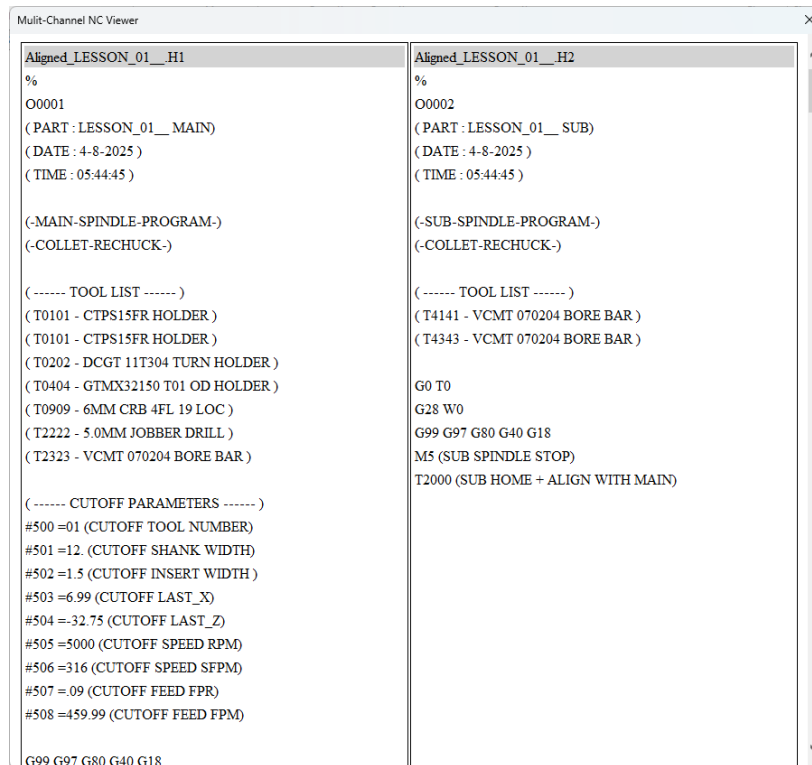
- e. Run **Simulation**.



f. Click **OK**.

18. Post Process.

- Post process the G-Code.
- Examine the code in the **Mult-Channel NC Viewer**.



19. Save and Close the Part.

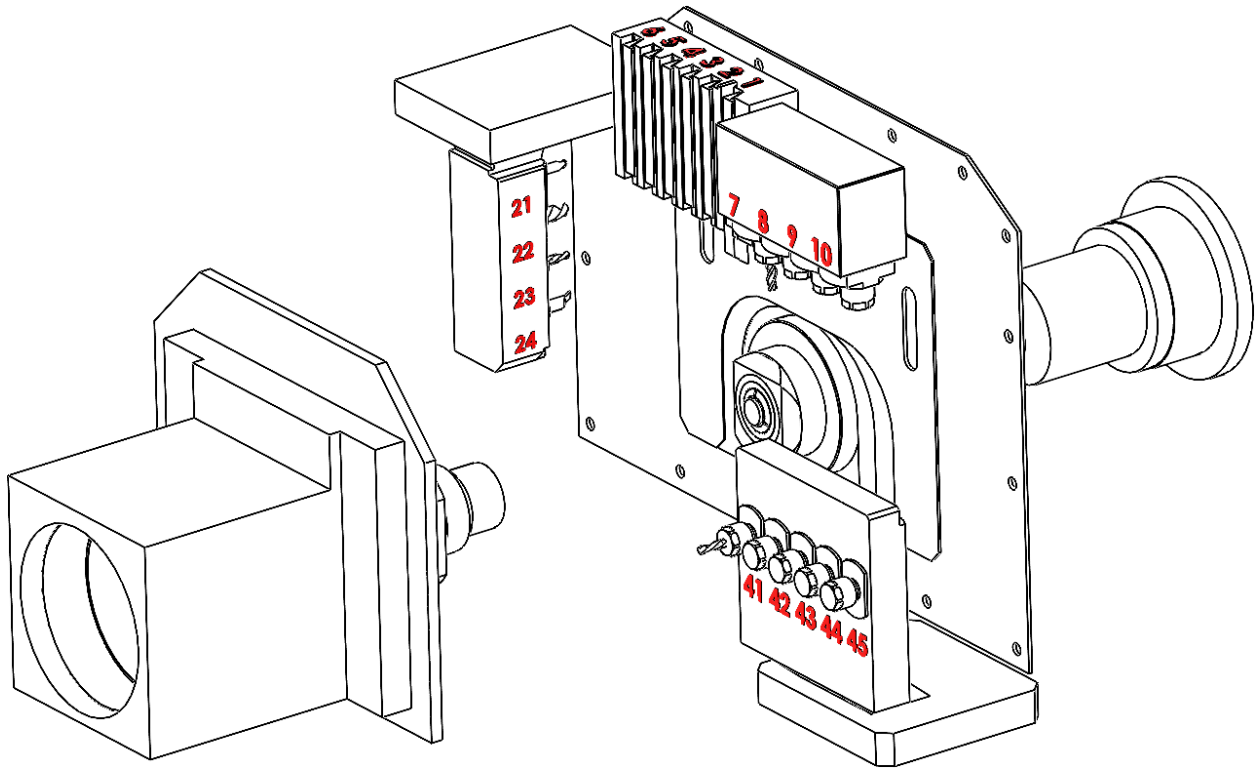
Lesson 2 – Programing a Part for Swiss-Turn Advanced Options Part 1

Upon successful completion of this lesson, you will be able to:

- Understand how CAMWorks Virtual Machine is used in CAMWorks Swiss Turn Machining
- Understand the Following Machine Setup Parameters
- Program with or without a Guide Bushing
- Understand the Coordinate System settings
- Understand how to define the Stock
- Set the turn feature option
- Define Features
- Default Feature Strategies
- Define Operations
- Understand the Bar Break command.
- Run Simulation
- Post processing

Case Study – Advanced Swiss Turn Part Programing Part 1

In this exercise we will apply the principles discussed in previous lessons to program a part file for a Swiss turn machine. For this example, the machine used will be the 2-channel Gang and Back Working sample machine that is already set up in the default Technology Database. As we showed earlier, this machine has the following layout for the tools used:



We will use the CAMWorks Workflow discussed earlier to program the part.

1. Open Part.

- Open the part file **LESSON_02.SLDPRT** from the **Lesson_02** folder.

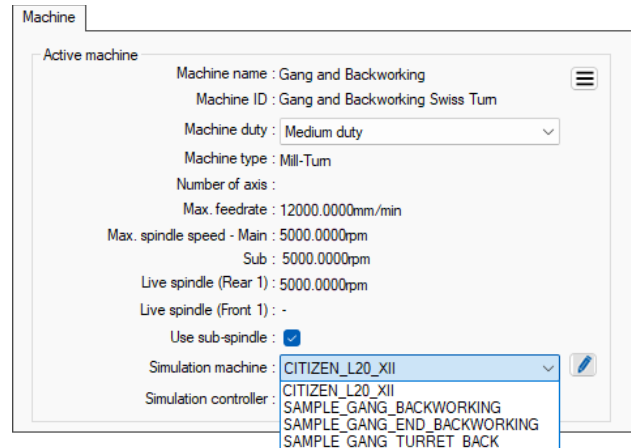
2. Define the Machine.

- Edit the machine definition and go to the **Machine** tab.
- Select the **Gang and Backworking** machine under the **Swiss Turn Machines** area in the **Available machines** area.
- Click **Select**.

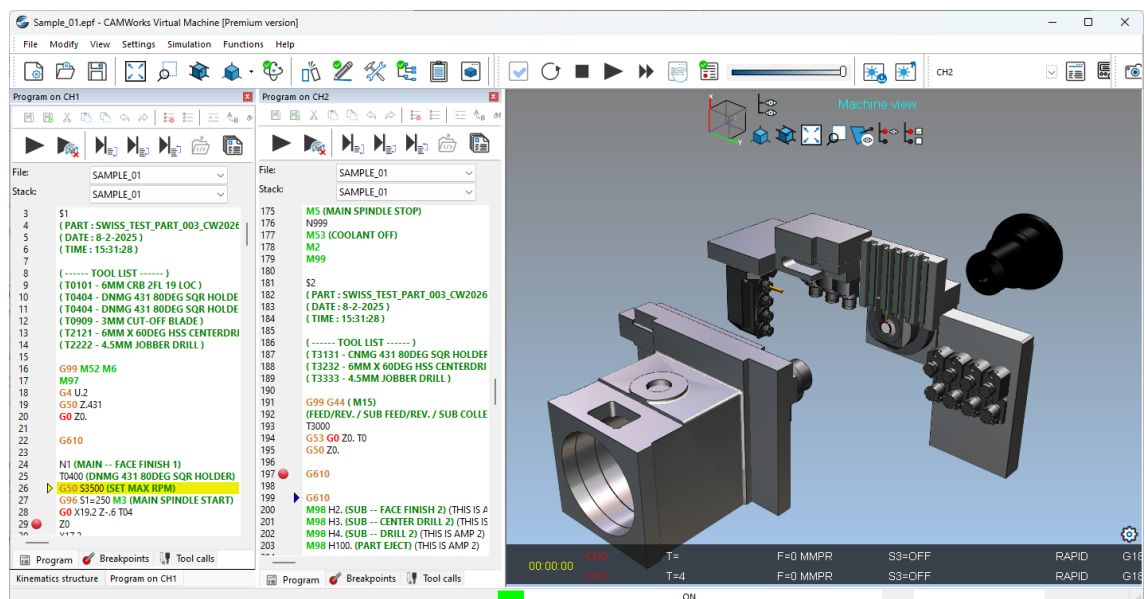
Define the Machine

CAMWorks Virtual Machine

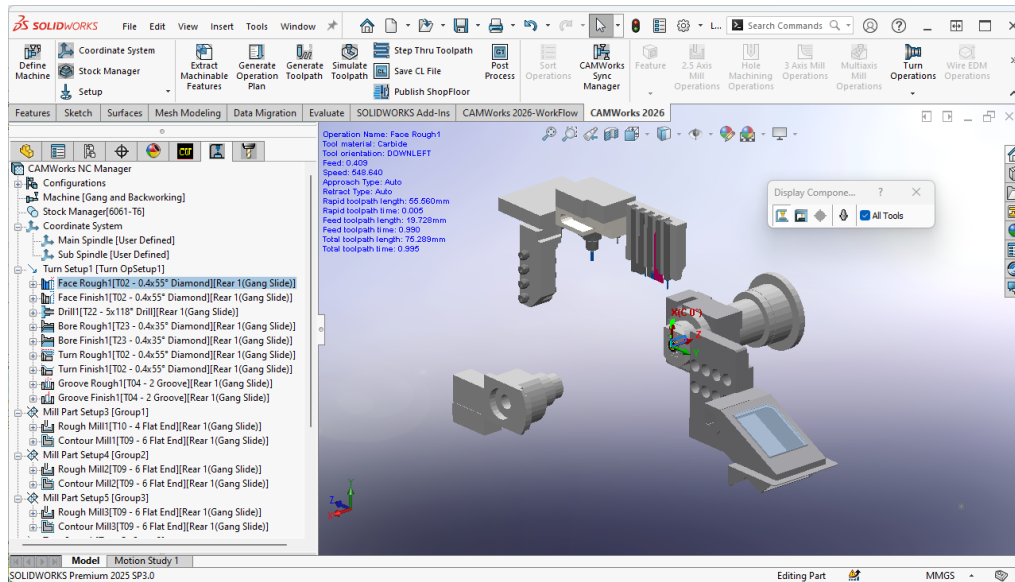
At the bottom of the Machine tab there are settings for the Simulation machine. If a customized simulation machine has been purchased and installed that machine will be available on the Simulation machine dropdown list.



If the customized machine is set up to do so, running G-Code machine simulation will launch CAMWorks Virtual Machine to simulate the G-Code.



CAMWorks will also use this machine when the Display Machine Components are enabled.



For this exercise, we will use the **SAMPLE_GANG_BACKWORKING** machine for the simulation machine to visualize the machine using Machine Aware.

- d. Under **Simulation Machine**, select **SAMPLE_GANG_BACKWORKING**.
- e. Click on the **Tool Crib** tab
- f. Ensure that the **Gangslide GB** tool crib is used for the **Rear 1** turret.
- g. Also ensure that the **Backworking GB** tool crib is used for the **Rear 2** turret.

Tool Crib

Active tool crib : Gangslide 1 Turret : **Rear 1**

☐ Moves In Z Axis

Usage	Stn. No.	Tool Type	ID	Comment	Dia. (mm)
1	1	Groove	240	CTPS15FR HOLDER	0
4	2	Diamond	237	DCGT 11T304 TURN HOLDER	9.53
	3	Diamond	238	DCMT 11T302 TURN HOLDER	9.53
2	4	Groove	231	GTMX32150 T01 OD HOLDER	0
5	5	Groove	229	GTMH32 105RGX OD HOLDER	0
6	6	Thread	232	CSVT11FRP60-035A TURN HOLDER	0
	7	Center Drill	1	3MM X 60DEG HSS CENTERDRILL	3
	8	Bore	-1		
3	9	Flat End	11	6MM CRB 4FL 19 LOC	6
3	10	Flat End	8	4MM CRB 4FL 8 LOC	4
	21	Center Drill	4	6MM X 60DEG HSS CENTERDRILL	6

Add Tool... Edit Tool... Remove Tool Update Tool Save Tool Crib...

☐ Tool crib has sub stations

☒ Tool crib priority

☐ Use tool crib tools only

☒ Do not create new tool stations

☒ Allow Duplicate Station Numbers

Available tool cribs

Gangslide GB (V
 Backworking GB
 End Working GB

Select Name : Gangslide GB (Metric)

No. of stations : 14

Tool Crib

Active tool crib : Backworking 1 Turret : **Rear 2**

☐ Moves In Z Axis

Usage	Stn. No.	Tool Type	ID	Comment	Dia. (mm)	Rad
2	41	Diamond	207	VCMT 070204 BORE BAR	3.97	0.4
	42	Flat End	7	4MM CRB 2FL 14 LOC	4	0
4	43	Diamond	247	VCMT 070204 BORE BAR	3.97	0.4
	44	Center Drill	9	6MM X 90DEG CRB SPOT DRILL	6	0
	45	Bore	-1			

Add Tool... Edit Tool... Remove Tool Update Tool Save Tool Crib...

☐ Tool crib has sub stations

☒ Tool crib priority

☐ Use tool crib tools only

☒ Do not create new tool stations

☒ Allow Duplicate Station Numbers

Available tool cribs

Gangslide GB (V
Backworking GB
 End Working GB


Select Name : Backworking GB (Metric)

No. of stations : 5

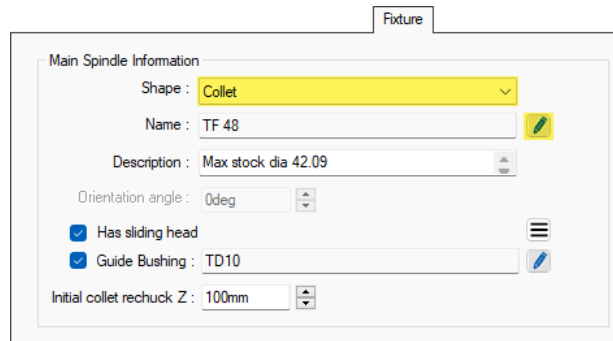
- h. Click on the **Post Processor** tab.
- i. Select the **SW_GANG_BACK_TUTORIAL_FANUC** post processor.
- j. Click on the **Fixture** tab.

Defining the Main Spindle Collet

Generally, collets are used to hold the workpiece in Swiss Turn so under the Main Spindle Information, the shape would be set to Collet. A library of standards collet is available by default inside of CAMWorks. Additional collets can be defined in the Technology Database.

Clicking on the **Edit**  button will let us define a custom collet size.

- k. Under the **Main Spindle Information** section, ensure the **Shape** is set to **Collet**.
- l. Click on the **Edit** button.



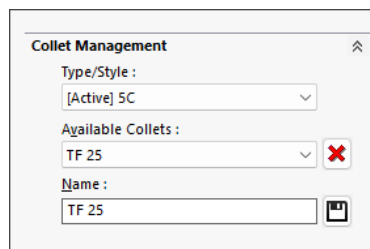
Collet Parameters

The **Collet Parameters** dialog is displayed when you select 'Collet' from the Shape dropdown list under the Fixture tab and click the adjacent Edit button. Separate dialog boxes exist for the Main Spindle and Sub Spindle.

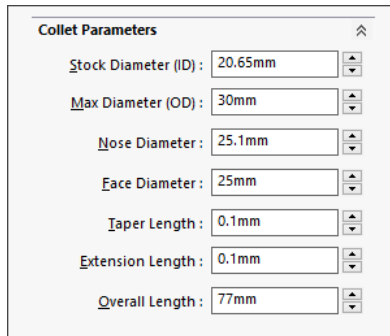
The purpose of this dialog box is to allow users to view, create, edit, and delete collets and their associated parameters.

Collet Management – A collet's Type/Style can be chosen from a list of Available Collets stored in the Technology Database (TechDB).

A new collet can be assigned a name and saved or deleted.



Collet Parameters – Users can view and edit various dimensional parameters of the collet, such as Stock Diameter, Max. Diameter, Nose Diameter, Face Diameter, and different lengths (Taper Length, Extension Length, and Overall Length).

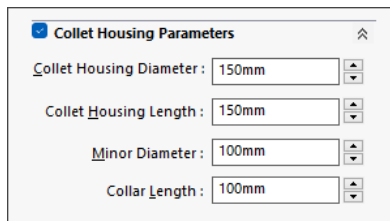


The **Collet Parameters** dialog box contains the following fields:

Parameter	Value
Stock Diameter (ID)	20.65mm
Max Diameter (OD)	30mm
Nose Diameter	25.1mm
Face Diameter	25mm
Taper Length	0.1mm
Extension Length	0.1mm
Overall Length	77mm

Collet Housing Parameters – An optional group box allows you to define the geometry of the collet housing, which is the fixture that holds the collet. This is for visualization purposes during simulation and includes parameters like Collet Housing Diameter, Length, Minor Diameter, and Collar Length.

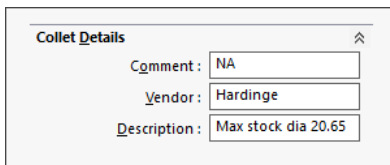
Most of the time in Swiss turn, this option should be unchecked.



The **Collet Housing Parameters** dialog box is checked and contains the following fields:

Parameter	Value
Collet Housing Diameter	150mm
Collet Housing Length	150mm
Minor Diameter	100mm
Collar Length	100mm

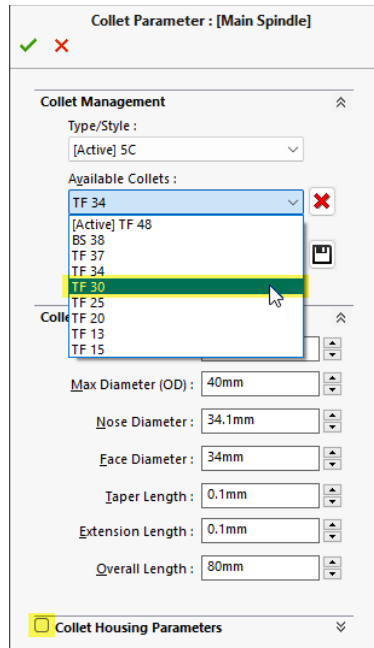
Collet Details – Users can also add descriptive information about the collet, including a Comment, Vendor, and Description.



The **Collet Details** dialog box contains the following fields:

Parameter	Value
Comment	NA
Vendor	Hardinge
Description	Max stock dia 20.65

- m. In the **Collet Management** section under **Available Collets**, select **TF 30**.
- n. Uncheck the checkbox next to **Collet Housing Parameters**.



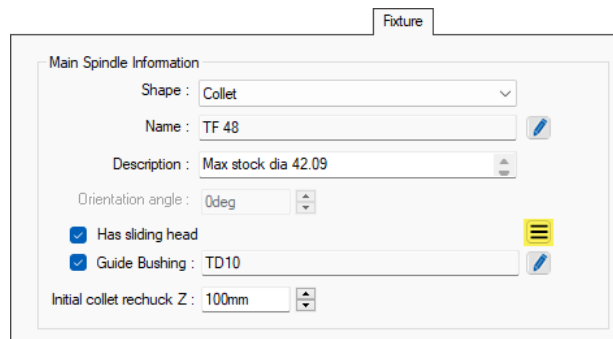
o. Click **OK**.

Has Sliding Head

The **Has sliding head** checkbox determines if a machine is a traditional fixed-head lathe (Mill/Turn) or a sliding-head (Swiss Turn) lathe. If unchecked, the machine is defined as a Mill/Turn machine and has a fixed-head. If checked, then it is assumed that the machine is a SwissTurn machine.

Enabling the option affects other settings in the Fixture tab. The **Guide Bushing** checkbox is only enabled when a sliding-head machine is selected, and the shape drops down on the main spindle is set to 'Collet'. The **Initial collet rechuck Z** field also becomes active.

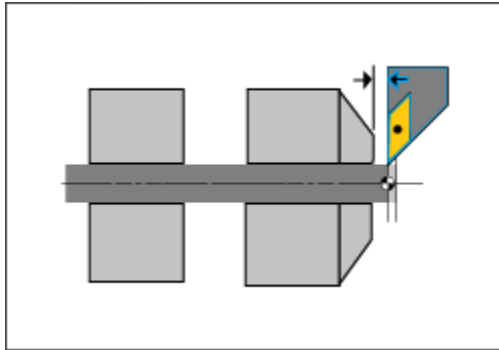
The **Additional Parameters For Sliding Head** dialog box is accessed by clicking a button next to the **Has sliding head** checkbox on the Fixture tab.



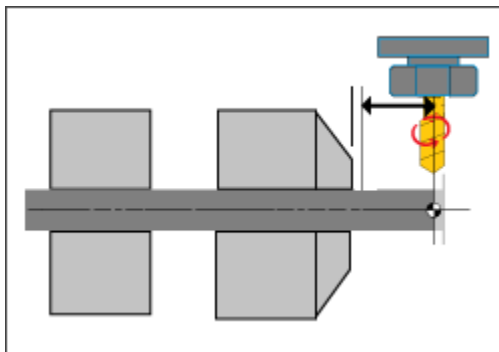
Its primary purpose is to set the default cutting locations for tools during simulation on a sliding-head lathe. This is necessary because tools need to be visualized at a specific distance from the guide bushing.

The dialog box contains a Default Cutting Locations group with the following parameters:

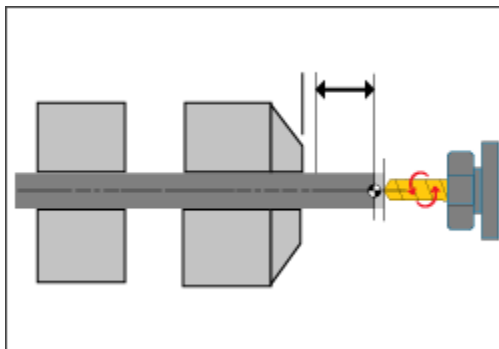
- **Zero Plane** – This parameter sets the distance from the part's origin to the face of the guide bushing.



- **OD Mill Tools** – This parameter sets the distance from the guide bushing's face to the milling tool used for machining the outside diameter (OD).

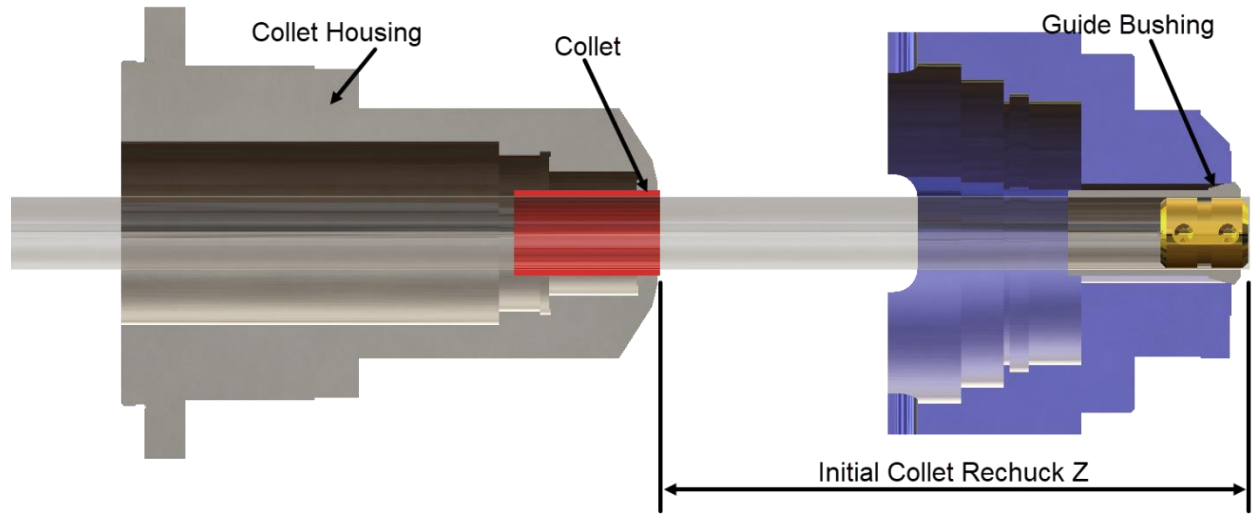


- **Z Tools** - This parameter sets the distance from the guide bushing's face to the tool used for machining in the Z direction.

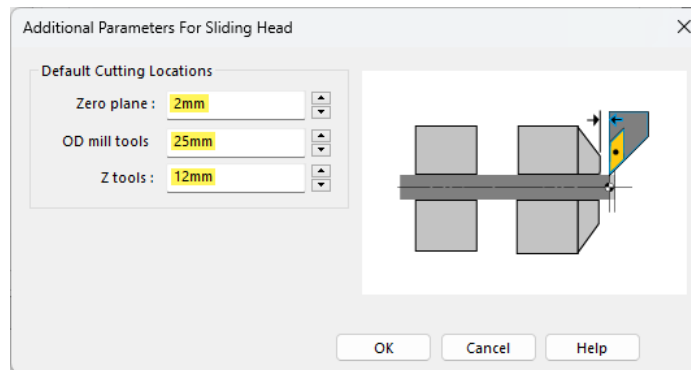


Initial Collet Rechuck Z

The **Initial Collet Rechuck Z** parameter in CAMWorks is used to set the distance from the face of the collet to the part's origin. This value, which must be zero or a positive number, is crucial for preventing the collet from colliding with the guide bushing during machining. The default value is pulled from the Technology Database (TechDB) for the active machine.



- p. Ensure the checkbox next to **Has sliding head** is checked on.
- q. Click on the **Additional Parameters for Sliding Head** button.
- r. Set the **Zero plane** to **2mm**, set **OD Mill tools** to **25mm** and **Z tools** to **12mm**.



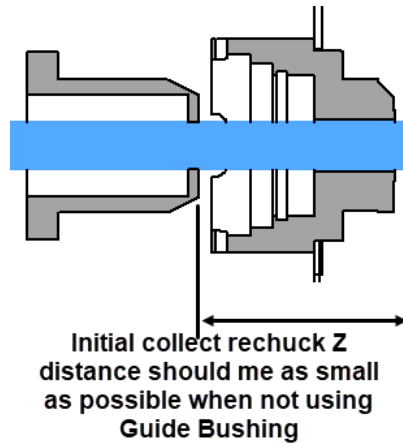
- s. Click **OK**.

Guide Bushings

Guide bushings are a key component that differentiates a CNC Swiss Multi-Function Lathe from a conventional Mill-Turn machine. They are supporting devices for the bar stock, allowing it to move axially through the bushing while being rotated and machined. This continuous support ensures the turning tool can stay close to the workpiece, leading to high-quality finishes and extreme accuracy.

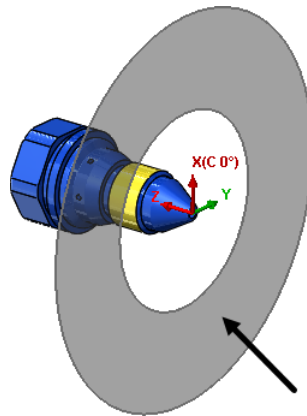
In certain cases, the guide bushing can be omitted when the part is short, for example, less than three to one length diameter ratio and the stock isn't as accurate.

In these cases, we want the collet and collet housing that holds the part to be as close as possible to the guide bushing housing.



The minimum **Initial Collet Rechuck Z** distance can be calculated automatically by entering **0mm** in the **Initial Collet Rechuck Z** field, clicking OK and opening the **Machine Definition** again.

For this example, we will remove the Guide bushing. When we turn off the guide bushing, a reference plane is displayed as a ring where the face of the guide bushing housing is.



- t. Uncheck the checkbox next to the **Guide Bushing**.
- u. For **Initial collet rechuck Z**, type in **0mm**.

Fixture

Main Spindle Information

Shape : Collet

Name : TF 30

Description : Max stock dia 25.40

Orientation angle : 0

☒ Has sliding head

☐ Guide Bushing : TD25S

Initial collet rechuck Z : 0mm

- v. Click **OK** and then edit the **Machine Definition** again. Note the calculated **Initial collet rechuck Z** is now set to **70mm**.

Fixture

Main Spindle Information

Shape : Collet

Name : TF 30

Description : Max stock dia 25.40

Orientation angle : 0

☒ Has sliding head

☐ Guide Bushing : TD25S

Initial collet rechuck Z : 70mm

- w. Under **Sub Spindle Information**, set the Collet to be the **TF30**.
- x. Uncheck the **Collet Housing Parameters** checkbox.

Collet Parameter : [Sub Spindle]

✓ ✗

Collet Management

Type/Style : [Active] SC

Available Collets :

- TF 34
- [Active] TF 48
- BS 38
- TF 37
- TF 34
- TF 30
- TF 25
- TF 20
- TF 13
- TF 15

Collet Housing Parameters

Max Diameter (OD) : 40mm

Nose Diameter : 34.1mm

Face Diameter : 34mm

Taper Length : 0.1mm

Extension Length : 0.1mm

Overall Length : 80mm

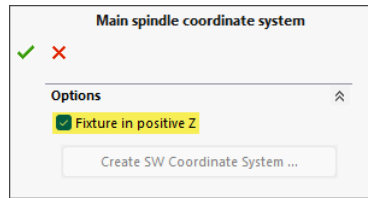
- y. Click **OK**.
- z. Click **OK** to the **Machine Definition** dialog.

Define Coordinate System

By default, CAMWorks sets the Z-axis for Swiss turn machines so the main spindle is in the negative direction. The **Fixture in positive Z** checkbox, located in the **Main Spindle Coordinate System Dialog Box**, allows you to reverse this setting. If you check this box, the main spindle will be oriented in the positive Z direction, and all toolpath coordinates will be calculated based on this new orientation. This affects how coordinates are generated for simulation and NC code.

3. Define Coordinate System.

- a. Edit the **Main Spindle coordinate system**.
- b. Under **Options**, check on the **Fixture in positive Z** checkbox.



- c. Click **OK**.

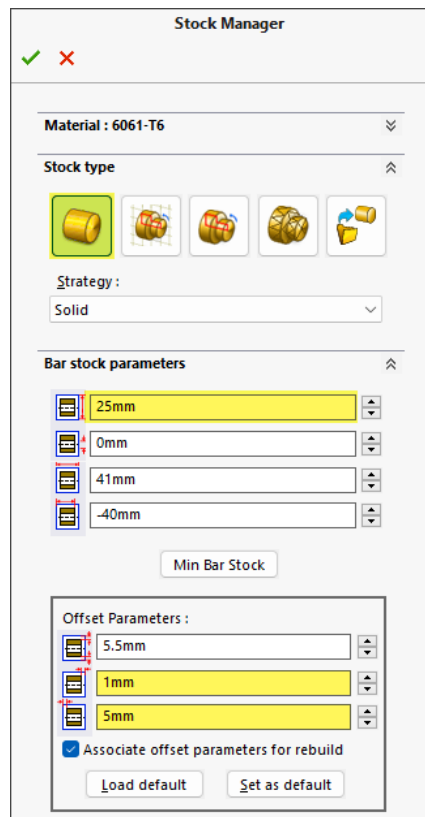
Define Stock

For this part we will define the stock so that the diameter matches the diameter of the bar stock being used. This value should match the inside diameter of the collet and guide bushing selected earlier.

For the face and back of the part, we can leave enough to face off the with a facing operation and for the back there should be enough material for a cut off operation and a subsequent facing operation that is performed on the sub spindle.

4. Define Stock.

- Edit the **Stock Manager**.
- Set the **Outside diameter** to **25mm**.
- Under **Offset Parameters**, set the **Front Face of Stock Offset** to **1mm**.
- Set the **Back Face of Stock Offset** to **5mm**.



- e. Click **OK**.

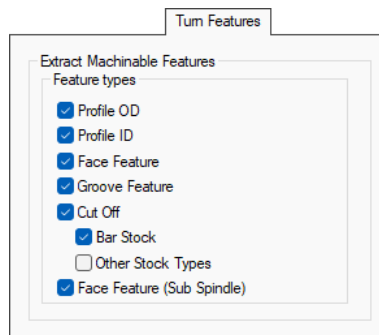
Turn Features Options

The CAMWorks **Options** dialog box, specifically the **Turn Features** tab, allows you to customize the settings for automatic feature recognition (AFR) on Swiss Turn machines.

Feature Types

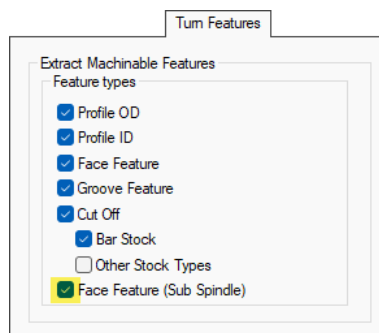
This section gives you granular control over which types of features AFR will automatically search for and recognize. For better performance, it's recommended to only select the features you expect to find on your part.

- **Profile OD/ID** – When these checkboxes are selected, AFR will look for outer diameter (OD) and inner diameter (ID) features.
- **Face Feature** – The behavior of this option depends on the Stock Type. For a Round Bar Stock, a single face feature is created at the start of the part. For other stock types, two face features are created (at the start and end of the part).
- **Groove Feature** – When checked, AFR will search for and recognize groove features.
- **Cut Off** – This option has sub-options for Bar Stock and Other Stock Types. When enabled, AFR will create a cut-off feature at the end of the part.
- **Face Feature (Sub Spindle)** – This option creates a face feature at the end of the part, typically where the cut-off feature is generated. It will be assigned to either the main or sub-spindle depending on the machine configuration.



5. Edit the CAMWorks Options.

- a. Go to the **Turn Features** tab.
- b. Check on the checkbox next to **Face Features (Sub Spindle)**.

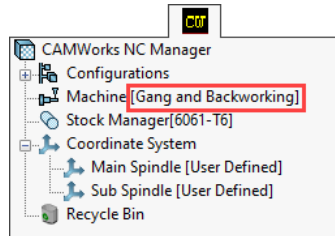


- c. Click **OK**.

Define the Machinable Features

For Swiss Turn programming we can use a combination of Automatic Feature Recognition (AFR) and Interactive Feature Recognition (IFR) to define the machinable features.

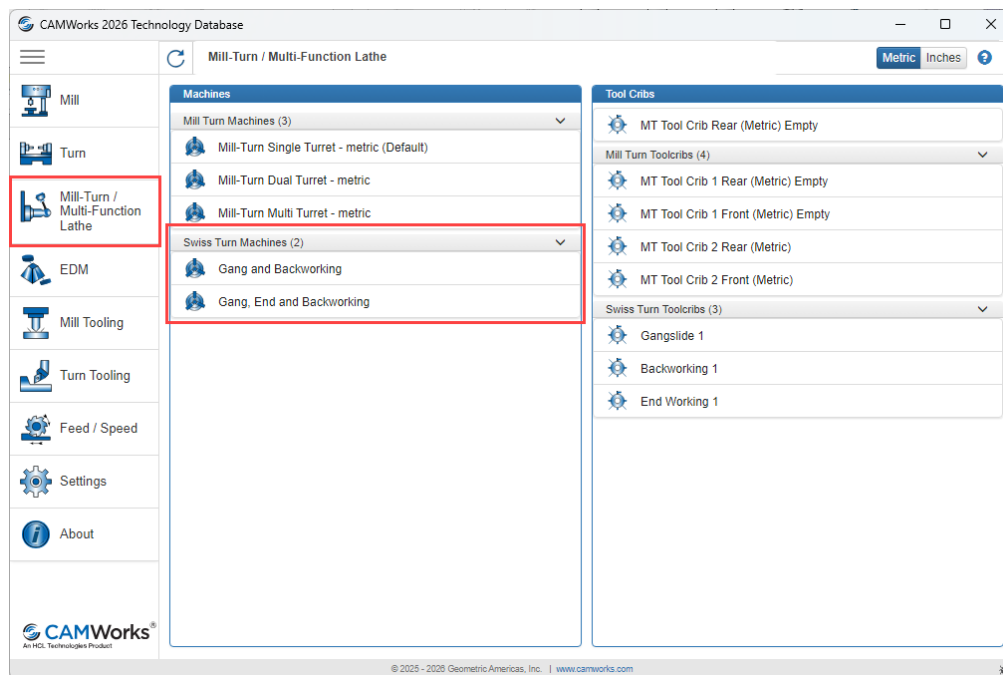
As part of the Swiss turn machine (Gang and Backworking) defined in the TechDB and selected in the exercise, running AFR will define features with strategies that will produce operations optimized for Swiss Turn machining.



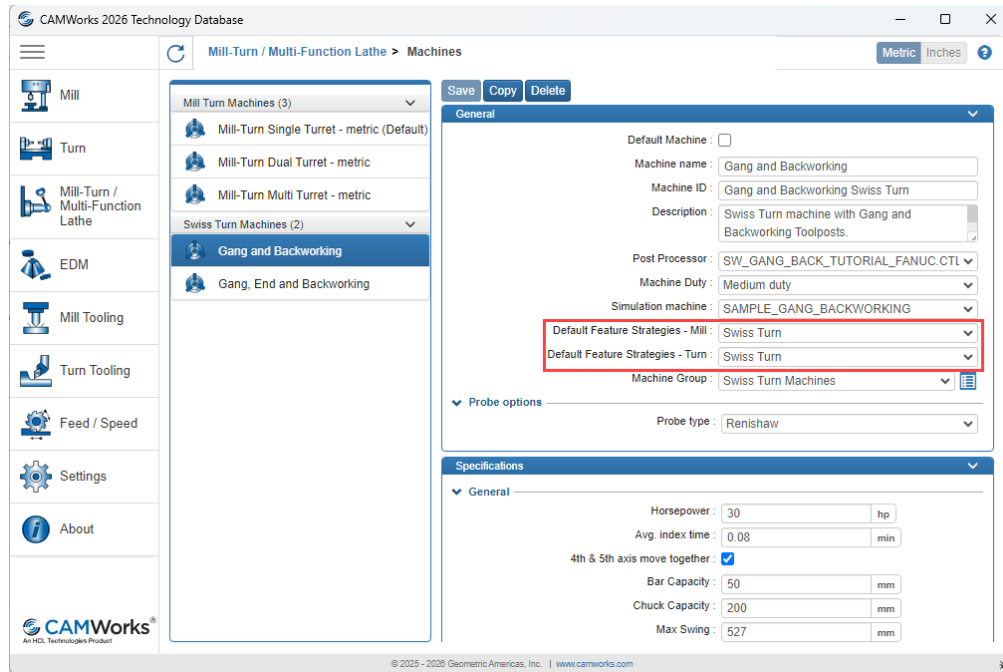
This is done by using **Default Feature Strategies**.

The Default Feature Strategies menu in the CAMWorks Technology Database (TechDB) lets you manage "feature strategies," which are sets of machining instructions for milling and mill-turn machines.

In the TechDB, we have a section for **Mill-Turn/Multi-Function Lathes**. In which there is a section for the **Swiss Turn Machines**.

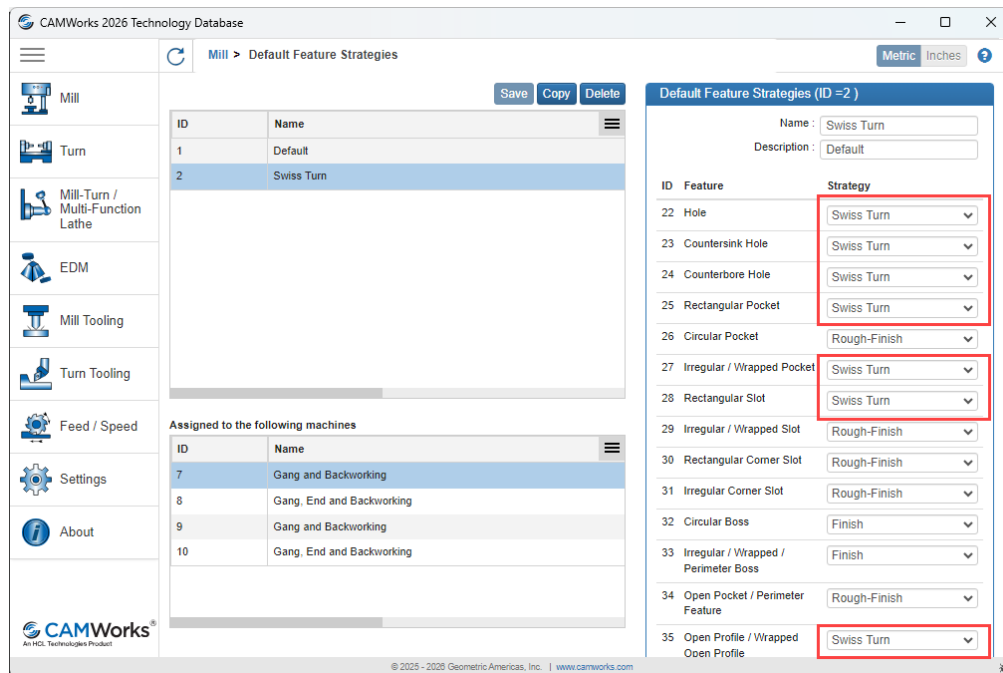


As part of the machine definition for these machines a set of **Default feature strategies** are set for mill and turning features. When you select this machine in the CAMWorks interface, this chosen scheme will be automatically loaded.

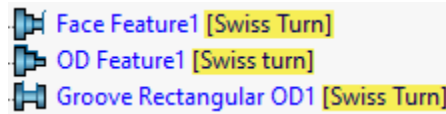


The **Default Feature Strategies** menu in the CAMWorks Technology Database (TechDB) is where we manage sets of machining instructions, known as "feature strategies," for milling, mill-turn and Swiss turn machines.

For every feature, a "Swiss Turn" strategy has been defined with the generally optimal tool selection and parameter definition for Swiss turn machining. These strategies have been set for the "Swiss Turn" Default strategy.



The result of all this is that when we select a Swiss Turn machine in the TechDB, the features that get automatically generated are optimized for Swiss turn machining.

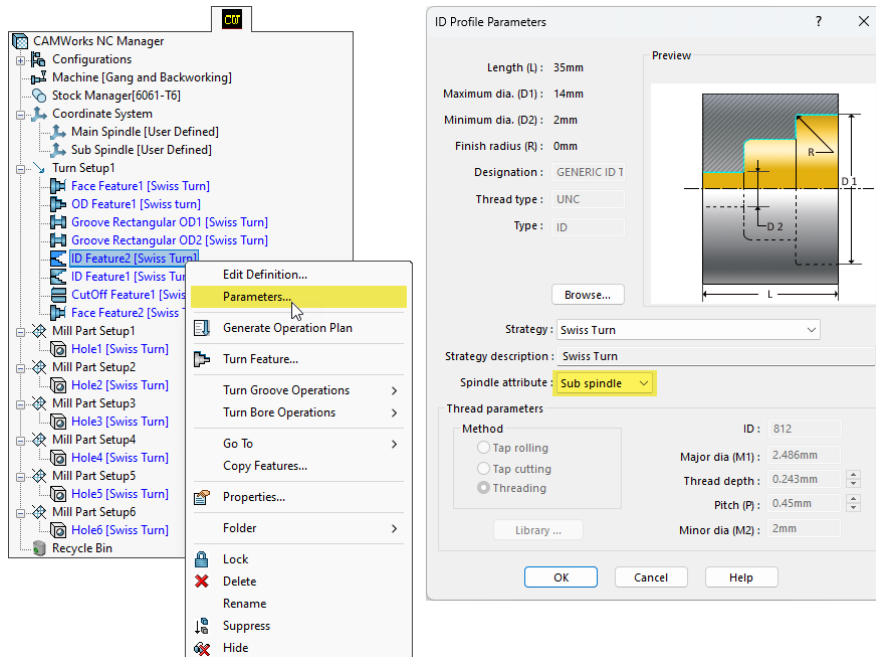


We will now proceed to define the machinable features of this part starting with automatic feature recognition.

We can then interactively recognize the OD Feature that defines the threaded portion of the part. Another mill set up and boss feature can be defined for the hexagonal geometry that will be machined on the back side of the part.

6. Define Machinable Features (AFR).

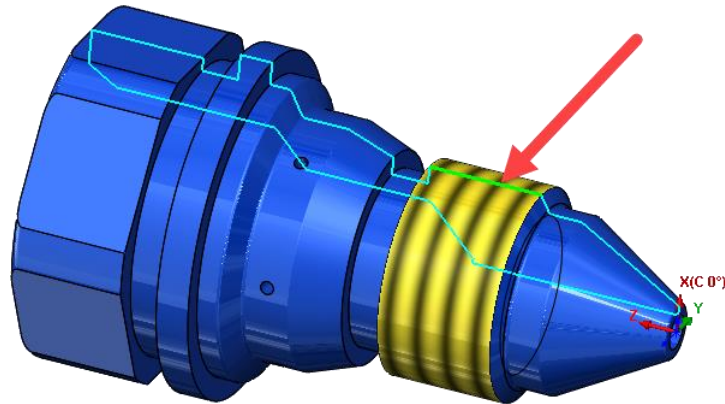
- Run **Extract Machinable Features**.
- CRTL+Drag and Drop** on **ID Feature1** to copy it.
- Rename **ID Feature1 – Copy** to **ID Feature2**.
- Right-click on **ID Feature2** and select **Parameters** from the list.
- Set the **Spindle attribute** to **Sub Spindle**.



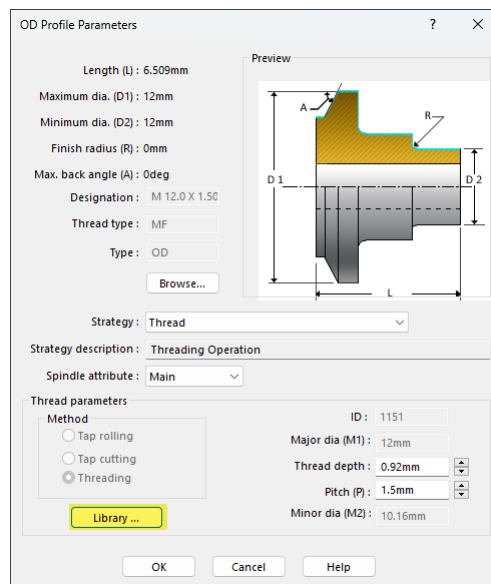
- Click **OK**.

7. Define Machinable Features (IFR).

- Create a new OD feature for the threaded portion of the part.
- Set the **Strategy** to **Thread**.
- For the **Selected entities** field select the cosmetic threaded cylinder on part file in the graphics area of SOLIDWORKS.



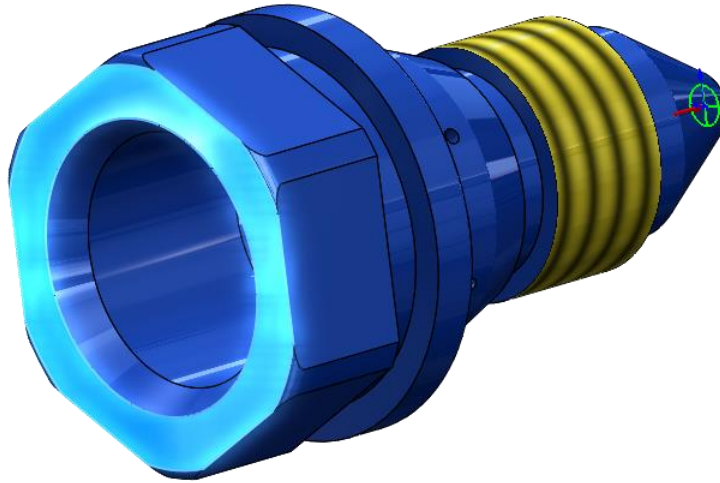
- d. Click **OK**.
- e. **Right-Click** on **OD Feature2** and select **Parameters...**
- f. Click on the **Library** button.



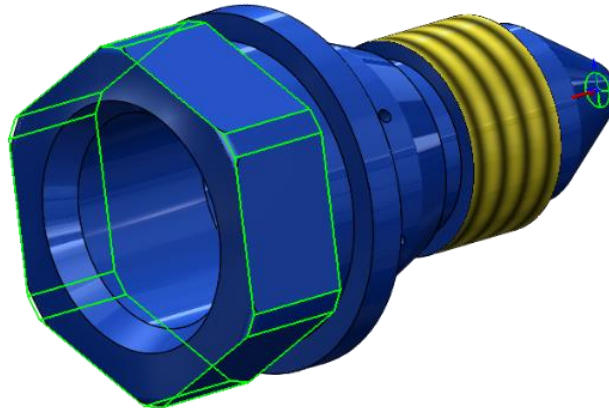
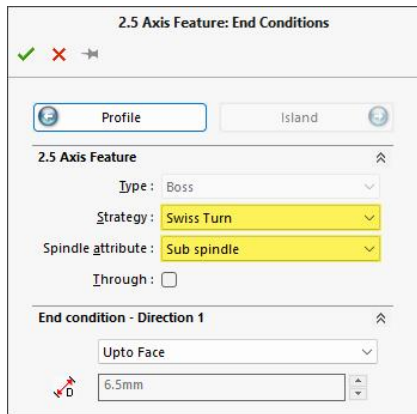
- g. Select the **M 12 X 1.50** thread condition from the **Tools Database** list.

ID	Type	Designation	Pitch	EndPitch	DepthOfThread	ProcessMethod	Units	Spins
117	1025	MC	M 1.8 X 0.35	0.350000	0.000000	0.214200	1	1
118	1137	MF	M 10.0 X 0.75	0.750000	0.000000	0.459800	1	1
119	1139	MF	M 10.0 X 1.00	1.000000	0.000000	0.613400	1	1
120	1141	MF	M 10.0 X 1.25	1.250000	0.000000	0.767100	1	1
121	1051	MC	M 10.0 X 1.50	1.500000	0.000000	0.919500	1	1
122	1143	MF	M 11.0 X 0.75	0.750000	0.000000	0.459400	1	1
123	1145	MF	M 11.0 X 1.00	1.000000	0.000000	0.613000	1	1
124	1053	MC	M 11.0 X 1.50	1.500000	0.000000	0.920400	1	1
125	1147	MF	M 12.0 X 1.00	1.000000	0.000000	0.613900	1	1
126	1149	MF	M 12.0 X 1.25	1.250000	0.000000	0.767600	1	1
127	1151	MF	M 12.0 X 1.50	1.500000	0.000000	0.920000	1	1
128	1055	MC	M 12.0 X 1.75	1.750000	0.000000	1.073700	1	1
129	1153	MF	M 14.0 X 1.00	1.000000	0.000000	0.613200	1	1
130	1155	MF	M 14.0 X 1.25	1.250000	0.000000	0.766800	1	1
131	1157	MF	M 14.0 X 1.50	1.500000	0.000000	0.920500	1	1
132	1057	MC	M 14.0 X 2.00	2.000000	0.000000	1.226600	1	1
133	1159	MF	M 15.0 X 1.00	1.000000	0.000000	0.614100	1	1
134	1161	MF	M 15.0 X 1.50	1.500000	0.000000	0.920100	1	1

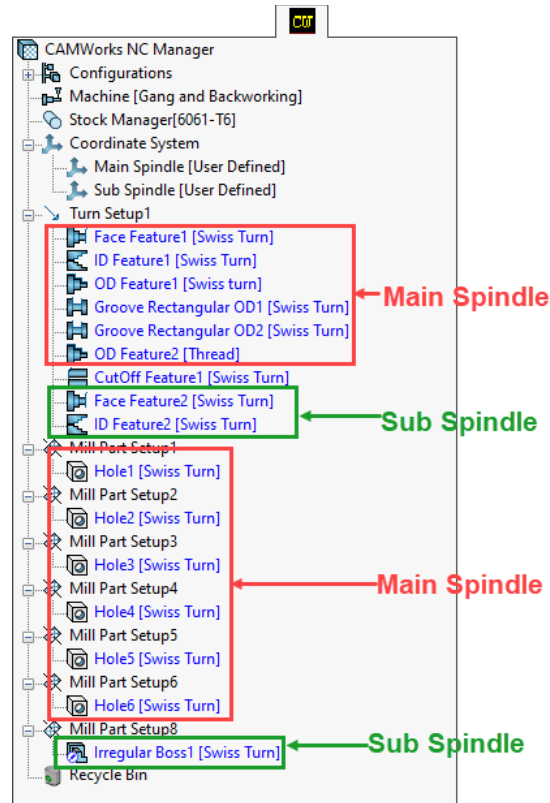
- h. Click **OK**.
- i. Define Mill part Setup on the backside of the part.



- j. Using that new Mill Part Setup, define a **Boss** feature for the hexagonal geometry.
- k. Set the **Strategy** to **Swiss Turn** and the **Spindle attribute** to **Sub spindle**.



- l. Click **OK**.
8. **Reorder Feature in the tree.**
- a. Reorder the features in the tree to the following order:



Generate Operations

We can generate the operations for these features that we have just defined.

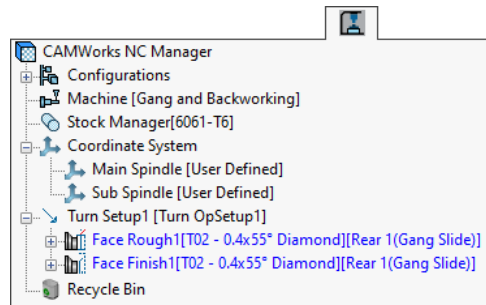
Note that the strategy name that is list next to the feature name is **Face Feature1 [Swiss Turn]**. This means that when we run the **Generate Operation Plan** command, the operations and associated parameters in those operations that are inserted will be optimized for the Swiss Turn machining strategy.

These operations and parameters have been saved previously inside the TechDB and can be further modified by the user.

We will start at the top of the list with the First face feature and examine the parameters associated with the operation that are relevant to Swiss Turn machining. Rather than clicking on the **Generate Operation Plan** command from the CAMWorks CommandManager toolbars, which will generate the operation plan for all the features, we will generate the operations individually for each feature.

9. Generate operations – Face Feature.

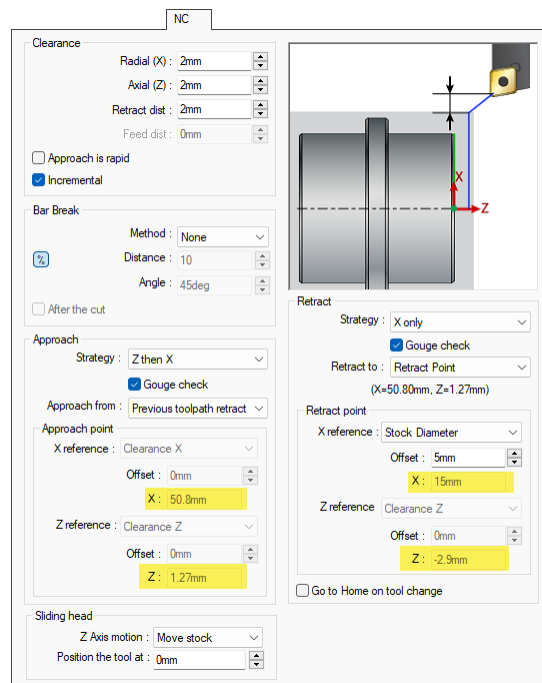
- Right-click on **Face Feature1** and select **Generate Operation Plan**.
- On the CAMWorks Operation tree, examine the operations that were generated.



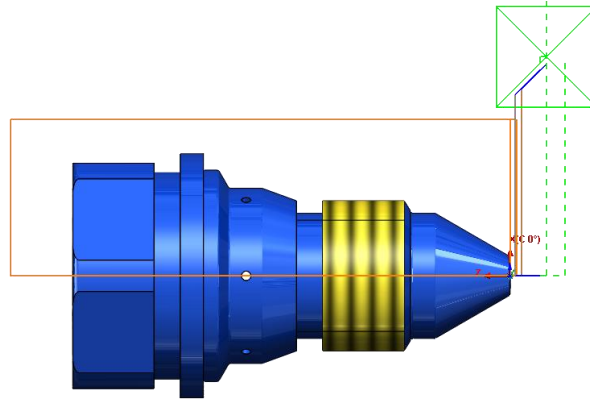
c. Edit **Face Rough1**.

Modify operation parameters.

By editing the Face Rough and Face Finish operations, we can see the parameters that are preset from the TechDB that are optimized for Swiss turn machining. Most notably, the Approach and retract parameters will be set to a value that is closer to the spindle compared to the parameters that would result from the typical Mill Turn strategy. This is because the gang slide with all of the tooling is located closer to the spindle and the limitations of the X, Y and Z travel of the machine.



- d. Click **OK** to the operation parameters.
- e. Generate the toolpaths for these operations by right-clicking on the turn setup and selecting **Generate Toolpath**.



10. Generate operations for ID Feature.

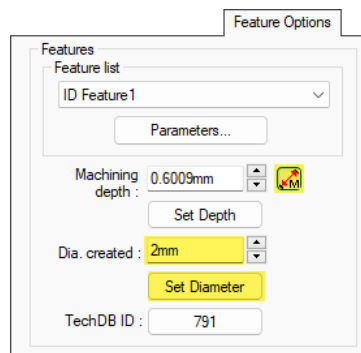
- a. Generate the operation plan for **ID Feature1**.

For this feature we won't need the Bore Rough and Bore Finish operations that are generated by TechDB. This is because the hole at the end of the part is too small for the bore tools we have available to fit and the drill operation will be enough to machine this feature.

However, we will need to add a drill tool to the gang slide and apply it to this ID turn feature to drill the hole.

We can go ahead and delete these features.

- b. Delete **Bore Rough1** and **Bore Finish1**.
- c. Edit **Center Drill1** and go to the **Feature Options** tab.
- d. Click on the **Override Machining Depth** button.
- e. For **Dia created**, type in **2mm**.
- f. Click on the **Set Diameter** button.

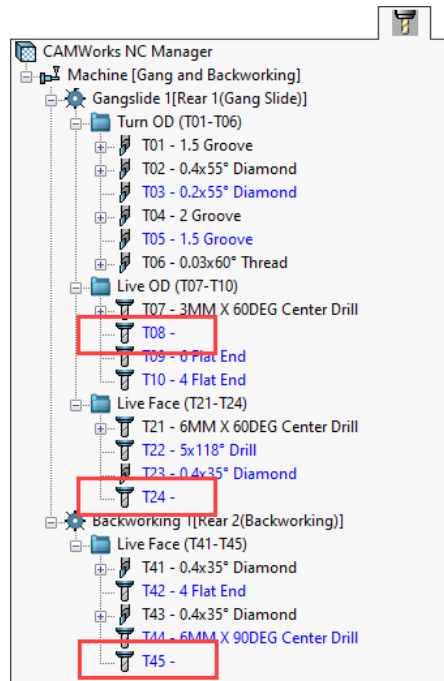


- g. Click **OK**.
- h. Click on the **CAMWorks Tool Tree** tab.

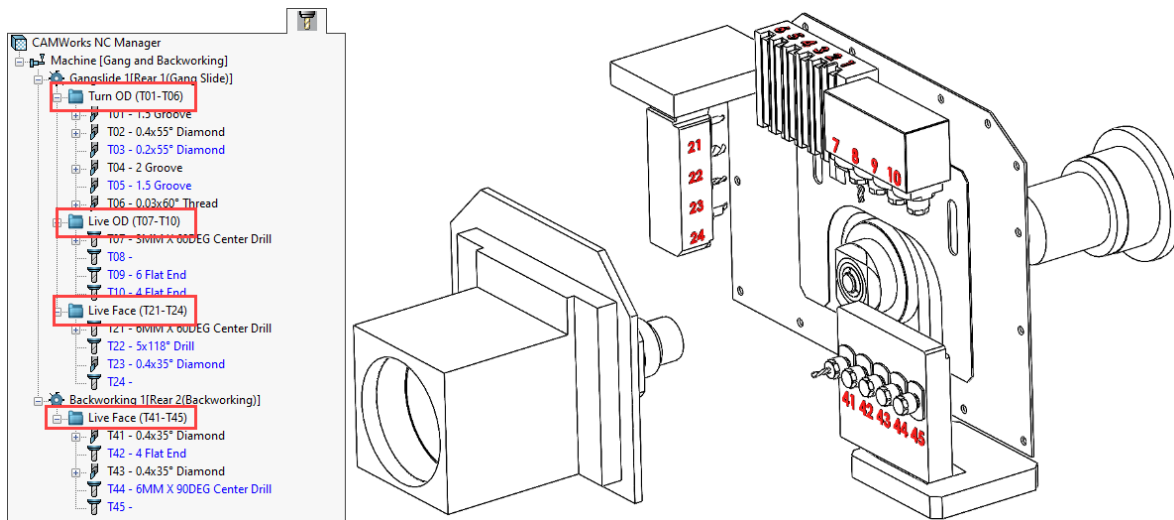
Tool Definition

Swiss turn machines have a limited number of tool stations in their tool stations that can be used for machining. Also, within this limited number of tools, there is a limitation of what kind of tool and how that tool is oriented.

If we look at the CAMWorks tool tree tab, we can not only see the tools defined from the tool crib, but also the entire tool stations available even if there aren't tools defined in these stations.



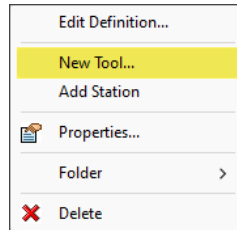
In addition to this, the tools have been separated into subfolders. The tools within these folders correspond to the tool numbers on the machine and have been set up to match the orientation.



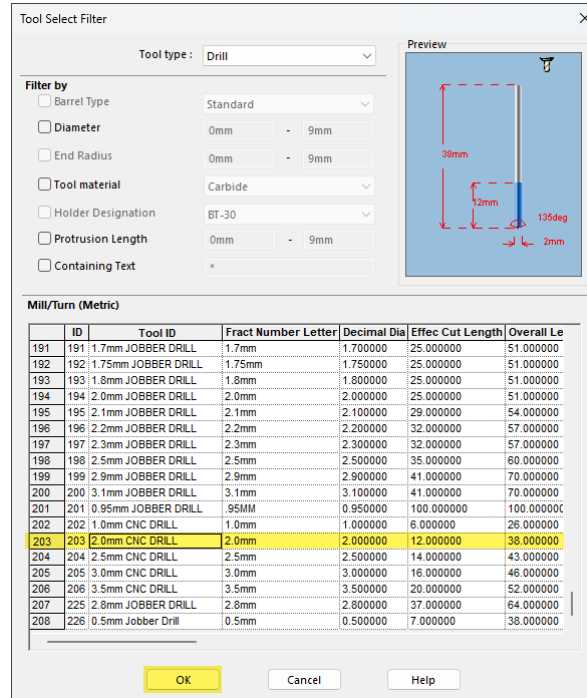
For example, tool in the back working tool post (tool numbers 41 through 45) can only point towards the sub spindle. Any tool that doesn't match that orientation won't be allowed to be assigns the tool station number 41 through 45.

For this part file, we will add a 2mm drill to the empty tool station in the "Live Face" tool post (Tool numbers 21-24).

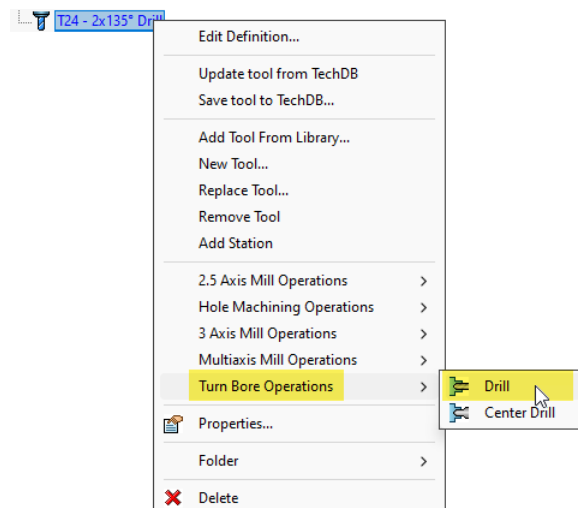
- i. Right -click on **T24** under **Live Face (T21-T24)** and select **New Tool**.



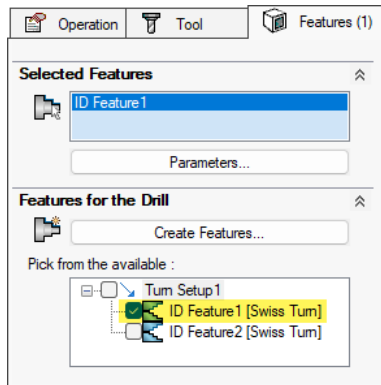
- j. For **Tool type**, select **Drill**.
- k. Select the **2.0mm CNC DRILL** from the list of drills.



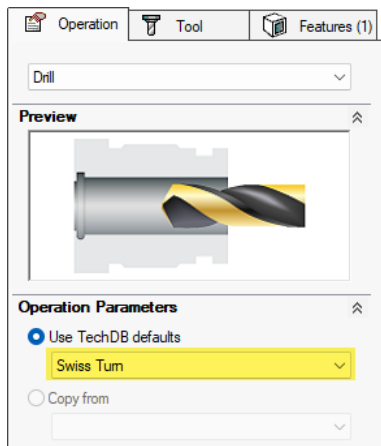
- l. Click **OK**.
- m. Right-click on **T24** and select **Turn Bore Operations** and then **Drill**.



- n. On the **Features** tab select **ID Feature1** from the list of features under **Turn Setup1**.



- o. Click on the **Operation** tab, under **Operation Parameters**, select **Swiss Turn** from the **Use TechDB defaults** list.

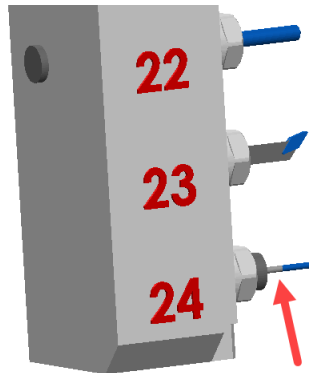


- p. Click **OK**.

11. Edit Operation Parameters

- a. Edit the Operation parameters of the Drilling operation.

For this tool we will set the orientation and length of the tool as it sits in the tool post.



The holder that was selected for this tool comes from the database and will need to be modified so it will fit into station 24 and display correctly.

- b. On the **Tool** tab, **Holder** tab, set the **Top diameter** to **12mm**, the **Bottom diameter** to **12mm**, the **Overall length** to **20mm**. and the **Bottom length** to **10mm**.

Station Drill Tool Holder Tool Crib

Holder number : Default ☒ Preview

Holder ID : -1

Basic

Holder type : None

Holder spec : None

Top diameter (D1) : 12mm

Bottom diameter (D2) : 12mm

Overall length (L1) : 20mm

Bottom length (L2) : 10mm

Protusion (L3) : 20mm

Name : None

Comment : Add Comment

- c. On the **Station** tab set the **Gage offset (XYZ), Z** value to **-25mm**.

Station Drill Tool Holder Tool Crib

Station

Tool number : 24 . 0

Station ID :

Gage offset (XYZ) : 0mm 0mm -25mm Reset

Station type : Any

- d. Go to the **Feature Options** tab, **Override the Machining Depth** and set the **Machining length** to **15mm**.

Feature Options

Features

Feature list

ID Feature1

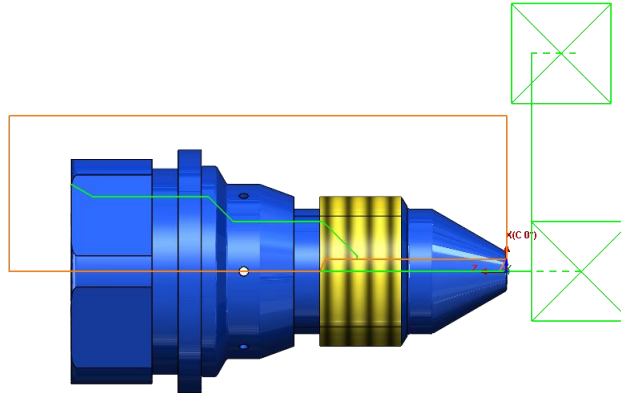
Parameters...

Add tip length ☐ 0.4142mm

Machining length : 15mm

TechDB ID: -1

- e. Click **OK**.
- f. Generate the toolpath for this operation.

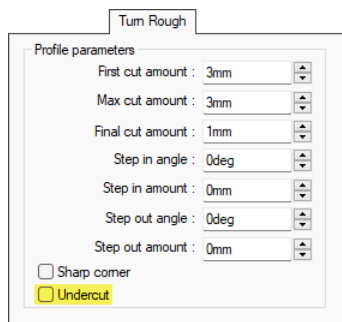


12. Generate Operation Plan.

- Generate Operation Plan for **OD Feature1**, **Groove Rectangular OD1** and **Groove Rectangular OD2**.

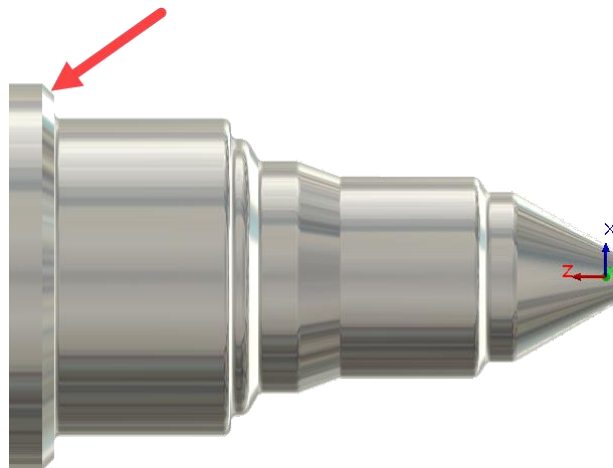
13. Edit OD Turn Operations.

- Edit **Turn Rough1** and go to the **Turn Rough** tab.
- Under **Profile parameters** uncheck the checkbox next to **Undercut**.



Bar Break

In a Swiss turning operation, a turning tool can be programmed to make a **bar break** move, which is a special cut used to clean up the uneven edges of a cylindrical workpiece.

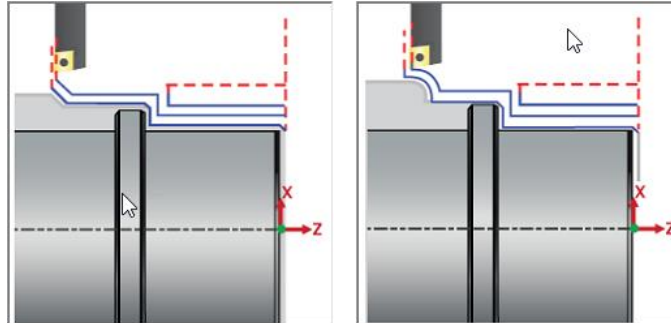


Bar break moves are intelligently added to the passes that intersect with the **maximum diameter** of the stock. Typically, this means they're applied to the first and last passes of the toolpath. If a

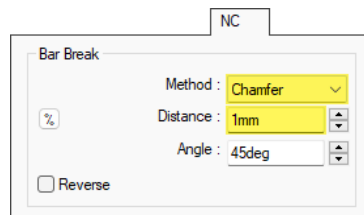
final cut amount is specified, the bar break move will also be added to the last pass. This ensures that the deformed edges of the stock are removed, which protects the guide bushings from damage.

To prevent tool damage, there is a built-in safety feature. If the user-defined distance for a bar break move exceeds a safe limit, the system will automatically limit the vertical depth of the cut. This depth will be restricted to the depth of the first cut, ensuring the tool remains protected.

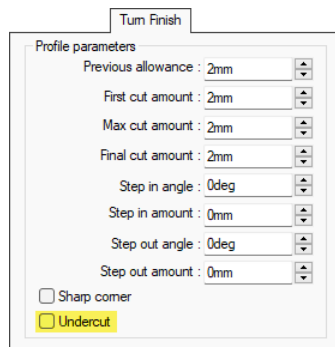
The Bar Break move can either be a Chamfer or a Radius.



- c. Go to the **NC** tab
- d. Under the **Bar Break** section, set the **Method** to **Chamfer**.
- e. Set the **Distance** to **1mm**.

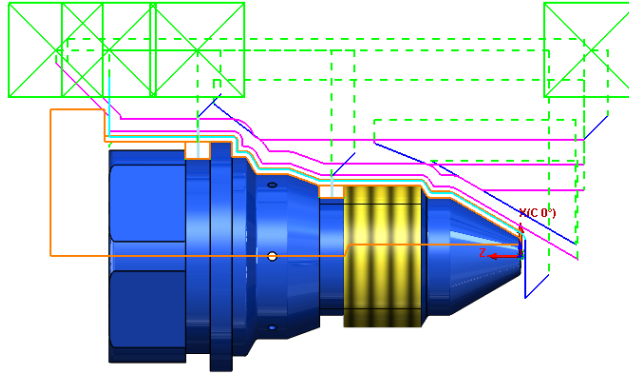


- f. Click **OK**.
- g. Edit **Turn Finish1** and go to the **Turn Finish** tab.
- h. Click on the **Turn Finish** tab and under **Profile parameters**, uncheck the checkbox next to **Undercut**.



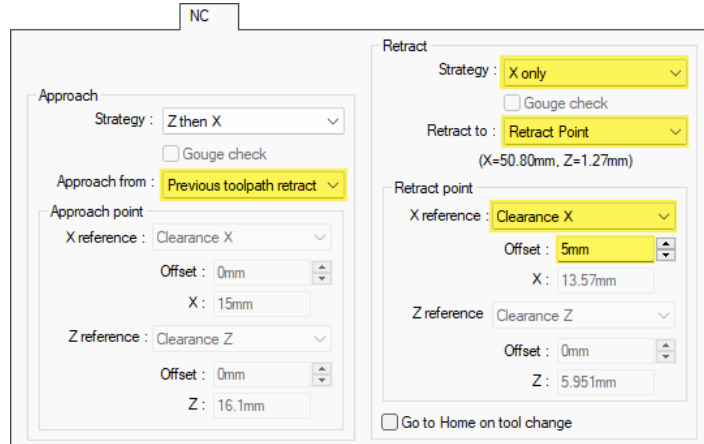
- i. Click **OK**.
- j. Delete the operations **Groove Rough1** and **Grove Rough2**.

- k. Edit **Groove Finish2**, on the **Lead In/Out** tab, change the **Leadin amount** to **0.75mm**.
- l. Click **OK**.
- m. Generate the toolpath for these operations.

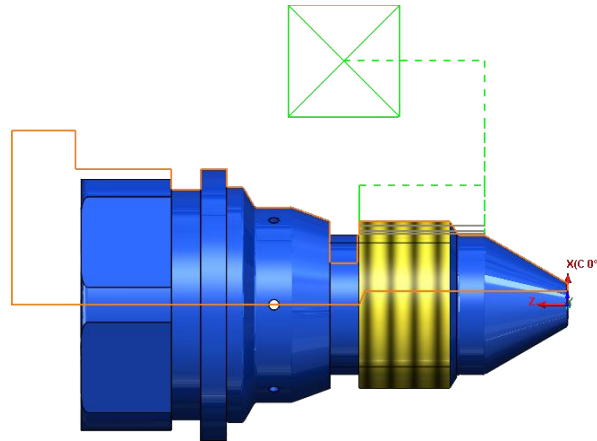


14. Generate Operation Plan and Toolpath for OD Thread Feature

- a. Generate Operation plan for **OD Feature2 [Thread]**.
- b. Edit the **Thread-OD1** operation.
- c. On the **NC** tab, under **Approach**, set the **Approach from** to **Previous toolpath retract**.
- d. Under **Retract**, set the **Strategy** to **X only** and **Retract to** to **Retract Point**.
- e. Under **Retract point**, set the **X reference** to **Clearance X** and the **Offset** to **5mm**.



- f. Click **OK**.
- g. Generate the Toolpath for this operation.



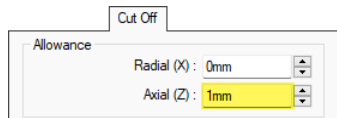
15. Generate Operation Plan for the Rest of the Turn Features.

- a. Generate Operation plan for **Cutoff Feature1**, **Face Feature2** and **ID Feature2**.

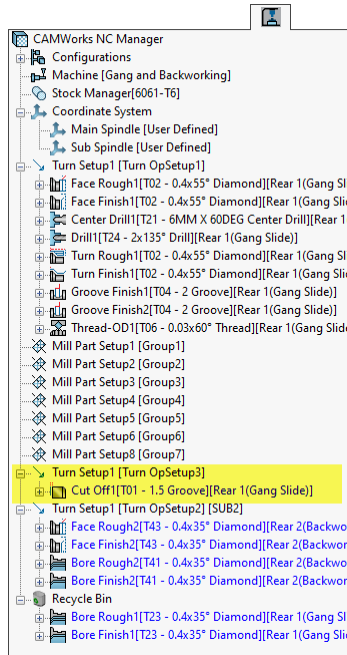
The cutoff operation will happen after the milling operation and simultaneously with a sub spindle transfer operation like we did in the first exercise. We will need to edit this operation to give it some allowance so that the facing operations on the sub spindle will have some stock to remove.

As we did before on the previous example, we will place this operation into its own turn setup and drag it below the mill part setups in the tree.

- b. Edit **Cut Off1** and on the **Cut Off** tab, under **Allowance**, set the **Axial (Z)** to **1mm**.



- c. Click **OK**.
- d. Right-click on **Turn Setup1** and select **Main Spindle Setup**.
- e. Drag **Cut Off1** into the new Turn Setup and drag it below the Mill Part setups.



- f. Generate toolpath for the **Cut Off1**, **Face Rough2** and **Face Finish2**.

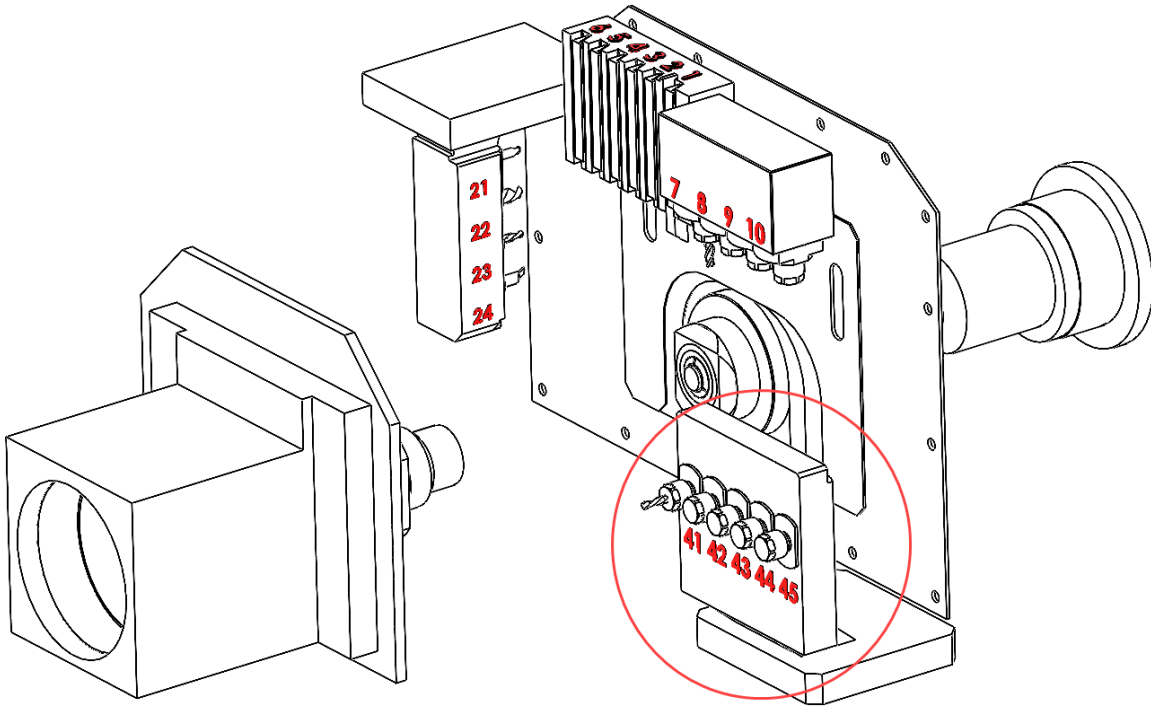
In most cases, the Face Rough operation won't be necessary because of the previous cut off operation. We can go ahead and delete this operation.

- g. Delete **Face Rough2**.

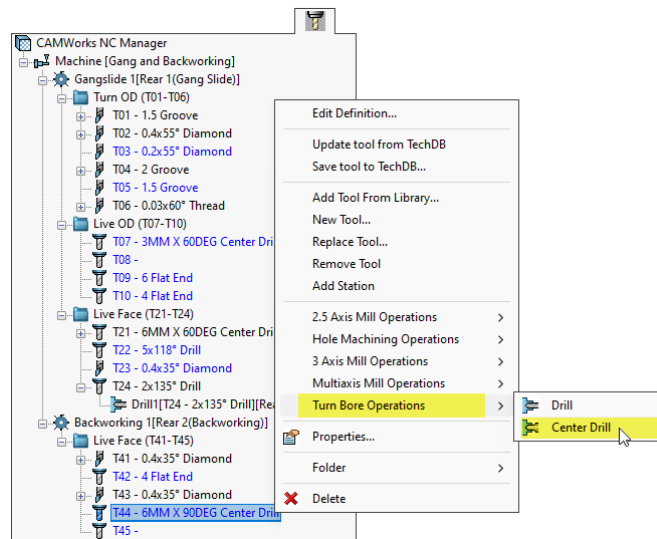
Add tools to the tool post.

The ID Bore operations that were generated on the sub spindle didn't have a drill and center drill operation that generated with them. We will need to interactively insert these operations and define the tools for them.

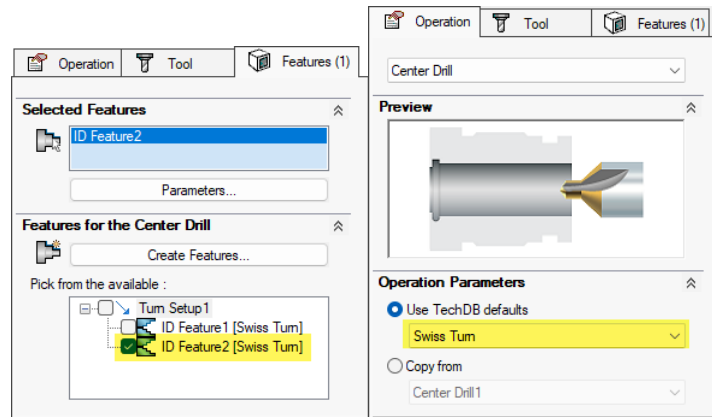
These operations will be done with tools that are mounted on the Back Working tool post of this machine. We can define these tools from the Tool tree.




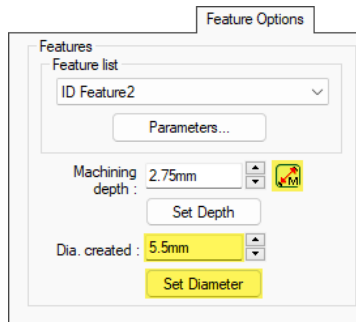
- h. Go to the **CAMWorks Tool Tree** tab.
- i. Under **Backworking 1**, right click on **T44 – 6MM X 90DEG Center Drill** and select **Turn Bore Operations, Center Drill**.



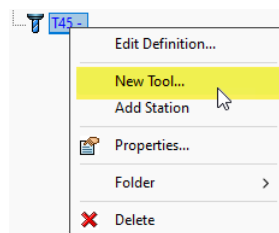
- j. On the **Features** tab, select **ID Feature2**.
- k. Click on the **Operation** tab, under **Operation Parameters**, select **Swiss Turn** under the **Use TechDB defaults** list.



- l. Click **OK**.
- m. Go to the **Feature Options** tab and click on the **Override Machining Depth**  button.
- n. Change the **Dia. Created** to **5.5mm** and click on the **Set diameter button**.



- o. Click **OK**.
- p. Go to the CAMWorks Tool tree, Right-Click on **T45** - under **Backworking 1** and select **New Tool...** from the menu.



- q. Select **7.8mm Jobber Drill** from the **Tool Select Filter** and click **OK**.

Tool Select Filter						
Mill/Turn (Metric)						
	ID	Tool ID	Fract Number Letter	Decimal Dia	Effec Cut Length	Overall Le
43	43	7.4mm JOBBER DRILL	7.4mm	7.400000	78.000000	111.000000
44	44	7.5mm JOBBER DRILL	7.5mm	7.500000	78.000000	111.000000
45	45	7.6mm JOBBER DRILL	7.6mm	7.600000	78.000000	111.000000
46	46	7.7mm JOBBER DRILL	7.7mm	7.700000	81.000000	114.000000
47	47	7.8mm JOBBER DRILL	7.8mm	7.800000	81.000000	114.000000
48	48	7.9mm JOBBER DRILL	7.9mm	7.900000	81.000000	114.000000
49	49	8.0mm JOBBER DRILL	8.0mm	8.000000	81.000000	114.000000
50	50	8.1mm JOBBER DRILL	8.1mm	8.100000	84.000000	117.000000
51	51	8.2mm JOBBER DRILL	8.2mm	8.200000	84.000000	117.000000

- r. Insert Drill operation using this tool. In the **New Operation: Drill** dialog, go to the **Operation** tab and set the **Use TechDB defaults** to **Swiss Turn** under **Operation Parameters**.
- s. In the **Operation Parameters**, go to the **Drill** tab, **Drill Tool** tab, change the following tool dimensions.

Station	Drill Tool	Holder	Tool Crib
<div> <div> Tool Dimensions Diameter (D1): 7.8mm Tip angle (A): 118deg Tip length: 2.3434mm Flute length (L2): 25mm Overall length (L1): 50mm No. of flutes: 2 </div> <div> <input checked="" type="checkbox"/> Preview </div> </div>			

- t. On the **Holder** tab, change the following dimensions.

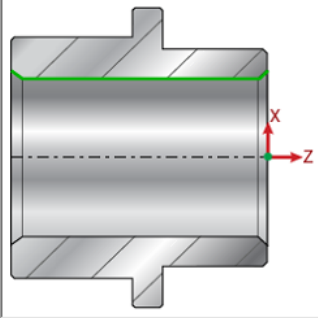
Station	Drill Tool	Holder	Tool Crib
<div> <div> Holder number: Default Holder ID: -1 Basic Holder type: None Holder spec: None Top diameter (D1): 12mm Bottom diameter (D2): 12mm Overall length (L1): 25mm Bottom length (L2): 10mm Protrusion (L3): 25mm Name: None Comment: Add Comment </div> <div> <input checked="" type="checkbox"/> Preview </div> </div>			

- u. On the **Station** tab, set the **Gage offset (XYZ)**, **Z offset** to **25mm**.

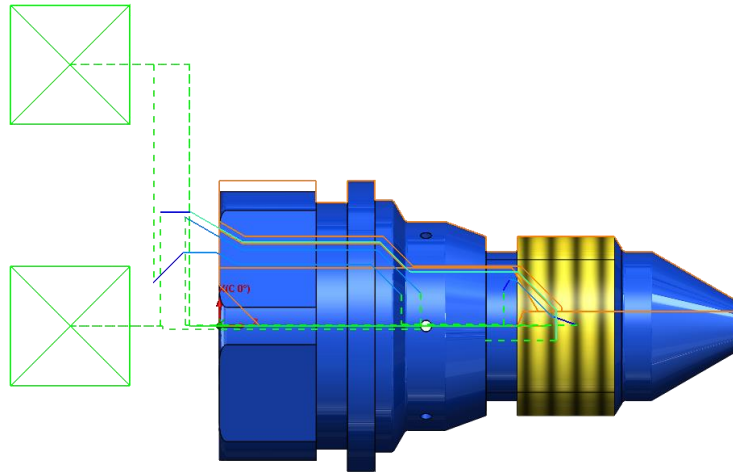
Station	Drill Tool	Holder	Tool Crib
Station			
Tool number : 45 . 0			
Station ID :			
Gage offset (XYZ) : 0mm 0mm 25mm Reset			
Station type : Any			

- v. Go to the **Feature Options** tab and set the **Machining length** to **22mm**.

Feature Options	
Features	
Feature list	
ID Feature2	
Parameters...	
Add tip length <input type="checkbox"/> 2.3434mm	
Machining length : 22mm	
TechDB ID: -1	



- w. Click **OK**.
x. Generate toolpath.

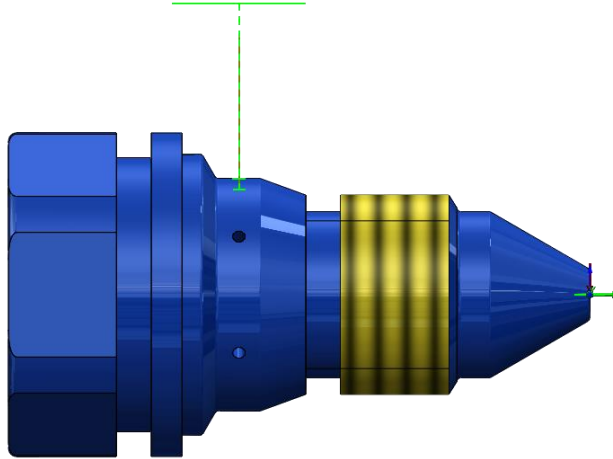


The holes on the OD are the same diameter and spaced equally apart. We can simplify the CAMWorks Feature and Operations trees by deleting all but one instance of the holes and patterning it around the part.

16. Define Hole Operations on the OD.

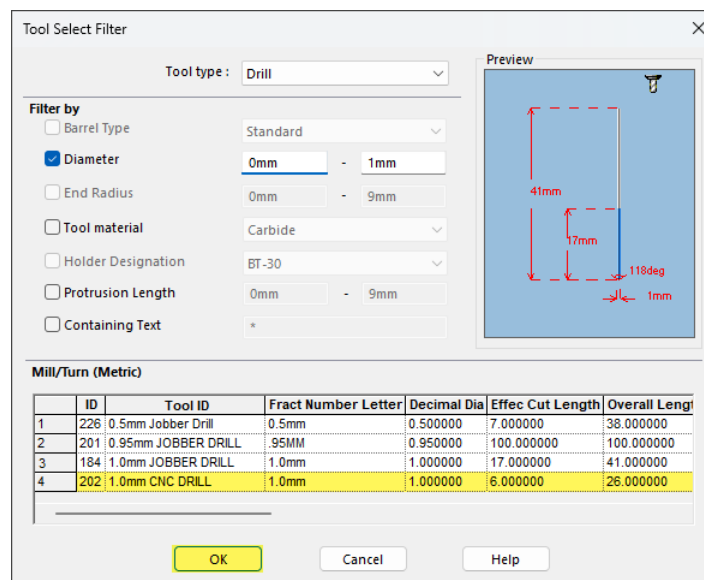
- Go to the CAMWorks Feature Tree.
- Delete **Mill Part Setup2**, **Mill Part Setup3**, **Mill Part Setup4**, **Mill Part Setup5**, and **Mill Part Setup6**.
- Generate Operation Plan for Hole1.
- Edit **Center Drill3** on the CAMWorks Operation Tree.

- e. On the **Feature Options** tab, **Override the Machining Depth** and set the **Dia. Created** to **1mm**. Click on the **Set Diameter** button.
- f. Click **OK**.
- g. Generate toolpath.

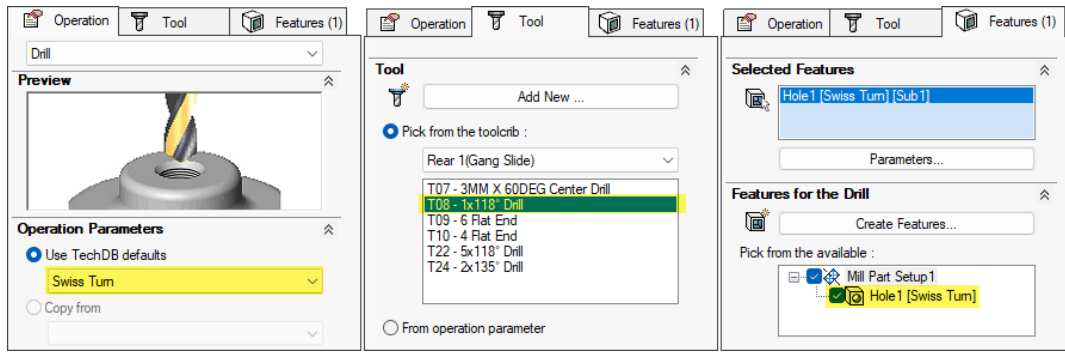


We will need to add another drill too to the tool post to drill these holes on the OD. We can do this the same way we did it with the other drill tools in previous steps.

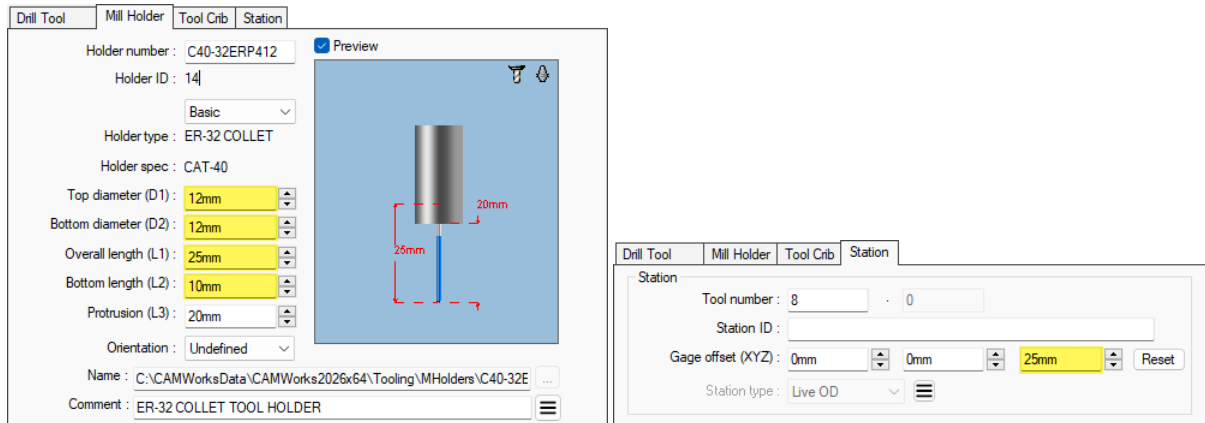
- h. Go to the CAMWorks Tool Tree tab.
- i. Right-Click on **T08** – and select **New Tool....**
- j. In the **Tool Select Filter**, select **1.0mm CNC DRILL** and click **OK**.



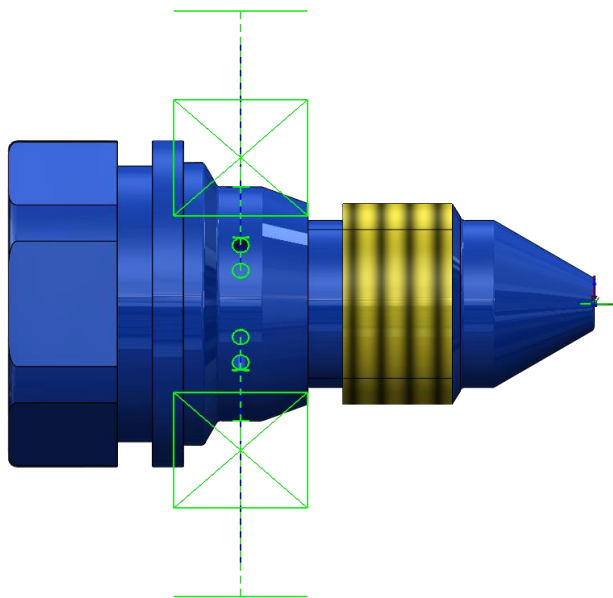
- k. Define Drilling operation using this tool.



- l. Modify the tool and tool holder dimensions so that the tool fits on the toolpost.



- m. Click **OK**.
- n. Generate toolpath.
- o. Pattern the hole feature in the CAMWorks Feature Tree.



The final machinable feature we need to create operations for is the Irregular boss to be machined on the Sub Spindle. If we **Right-Click** on the feature and select **Generate Operation Plan** nothing will happen.

This is because of the settings set in the **Machine Definition** for the tool cribs. The option to **Do not create new tool stations** makes it so that no new tools will be added to the tool crib from the tool library.

Machine | Tool Crib | Post Processor | Posting | Setup | Fixture

Tool crib
Active tool crib : Gangslide 1
Turret : Rear 1(Gang Slic)

☐ Moves In Z Axis

Usage	Stn. No.	Tool Type	ID	Comment	Dia. (mm)
1	1	Groove	240	CTPS15FR HOLDER	0
4	2	Diamond	237	DCGT 11T304 TURN HOLDER	9.525
	3	Diamond	238	DCMT 11T302 TURN HOLDER	9.525
2	4	Groove	231	GTMX32150 T01 OD HOLDER	0
	5	Groove	229	GTMH32 105RGX OD HOLDER	0
1	6	Thread	232	CSV111FRP60-035A TURN HOLDER	0
1	7	Center Drill	1	3MM X 60DEG HSS CENTERDRILL	3
1	8	Drill	184	1.0mm JOBBBER DRILL	1
	9	Flat End	11	6MM CRB 4FL 19 LOC	6
	10	Flat End	8	4MM CRB 4FL 8 LOC	4
1	21	Center Drill	4	6MM X 60DEG HSS CENTERDRILL	6

Buttons: Add Tool... Edit Tool... Remove Tool Update Tool Save Tool Crib...

☐ Tool crib has sub stations
☒ Tool crib priority
☐ Use tool crib tools only
☒ Do not create new tool stations
☒ Allow Duplicate Station Numbers

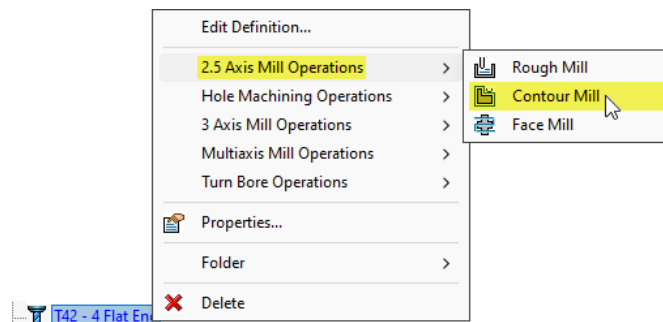
Available tool cribs
 Gangslide 1 [Swi]
 Backworking 1 [S]
 End Working 1 [S]
 Select
 Name : MT Tool Crib Rear (Metric)
 No. of stations : 12

Tool library
 New Tool... Save Tool... Delete Tool

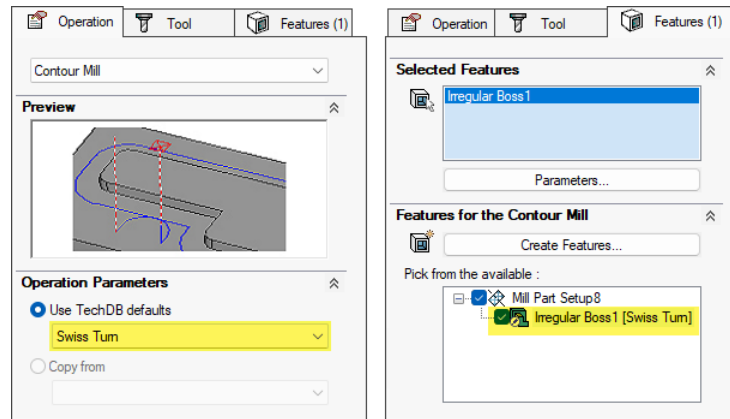
We can define this operation interactively.

17. Generate Contour Mill Operation for Irregular boss.

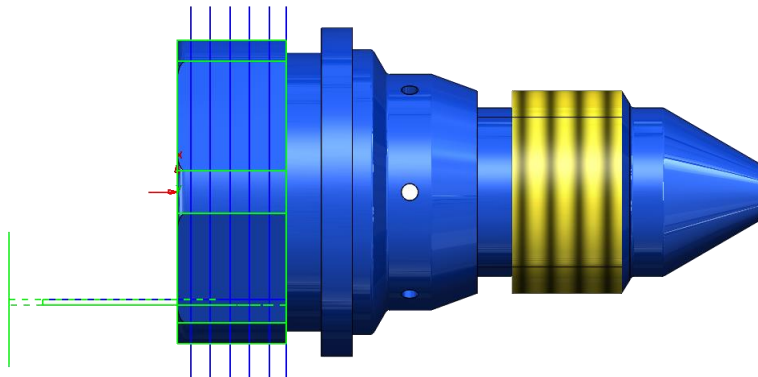
- On the **Tool Tree** tab, under **Backworking 1**, right-click on **T42** and select **2.5 Axis Milling Operations, Contour Mill**.



- On the **Features** tab, select **Irregular Boss1** from the list of features.
- On the **Operation** tab, select **Swiss Turn** from the **Use TechDB defaults** list.

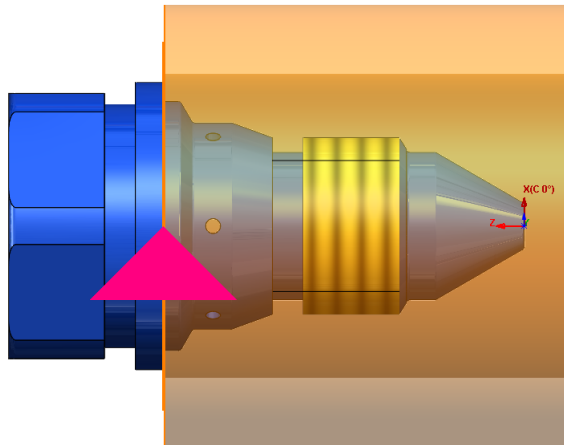
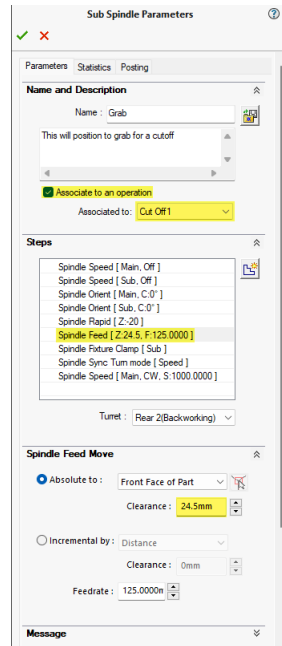


- d. Click **OK** to the operation parameters.
- e. Generate toolpath.

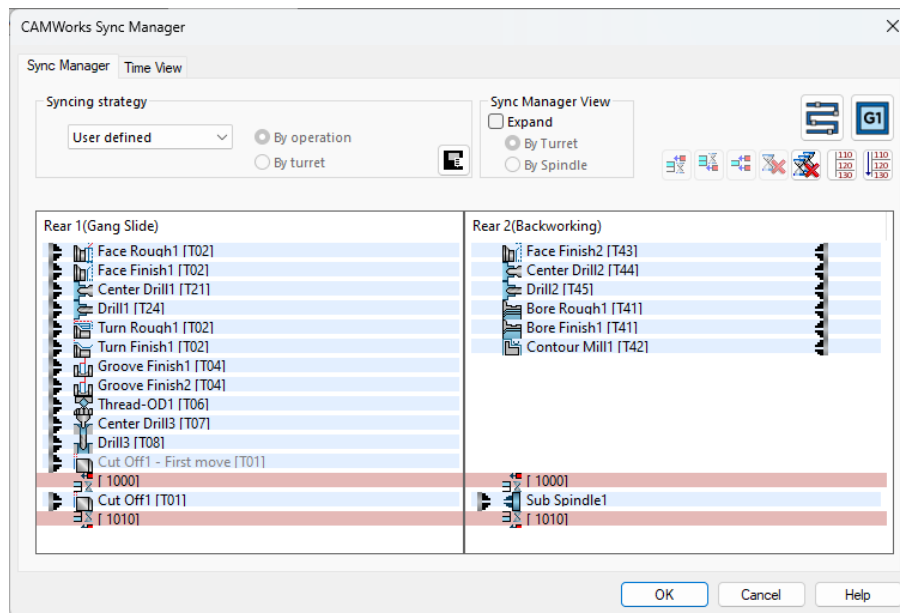


18. Add Sub Spindle Operation.

- a. Right-click on the second Turn Setup1 in the tree and select on **Sub Spindle Operation**.
- b. Select **Grab** from the **Available Templates** and click **OK**.
- c. In the **Sub Spindle Parameters** dialogue, under **Name and Description**, click on the **Associate to an operation** checkbox.
- d. Make sure **Cut Off1** is selected under the **Associated to** list.
- e. In the **Steps** list, click on **Spindle Feed** step and set the **Clearance** to **24.5mm**.



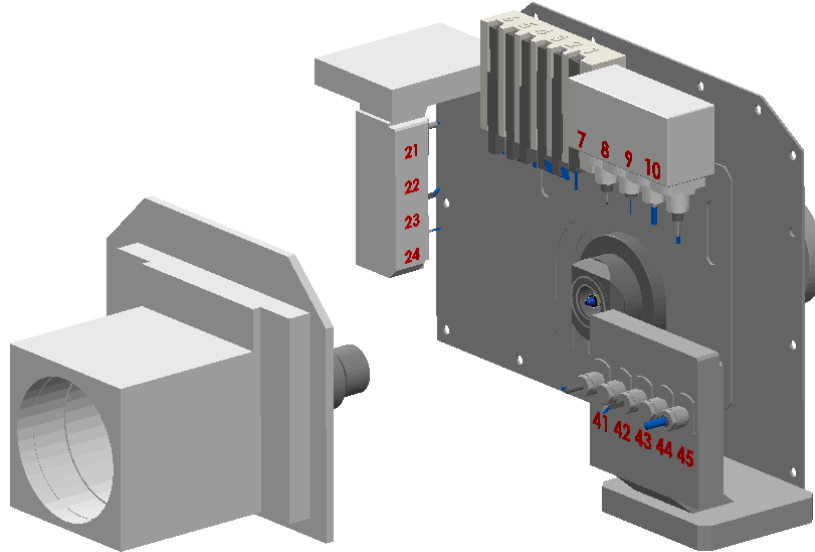
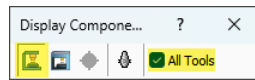
- f. Click **OK**.
19. **Modify the Order of Operations with the Sync Manager.**
 - a. Launch the **Sync Manager**.
 - b. Drag the operations on the Sub Spindle to the top of the Rear2(Backworking) list.



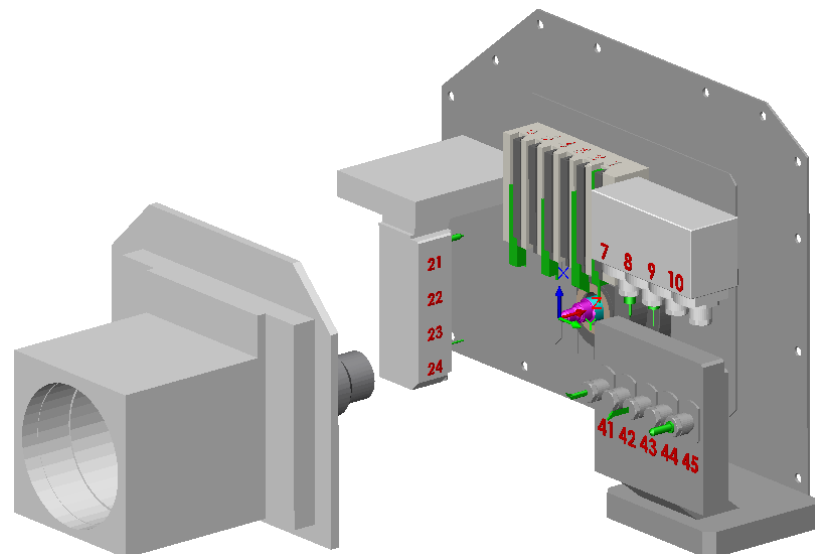
- c. Click **OK**.
20. **Simulate Toolpath.**
 - a. Turn on the **Display Components** toolbar.



- b. Activate the **Display Machine** command.
- c. Check on the **All tools** checkbox.



- d. Run toolpath simulation.



21. Post Process the G-Code.

Aligned_LESSON_02__H1	Aligned_LESSON_02__H2
%	%
O0001	O0002
(PART : LESSON_02__MAIN)	(PART : LESSON_02__SUB)
(DATE : 13-8-2025)	(DATE : 13-8-2025)
(TIME : 16:05:37)	(TIME : 16:05:37)
(-MAIN-SPINDLE-PROGRAM-)	(-SUB-SPINDLE-PROGRAM-)
(-COLLET-RECHUCK-)	(-COLLET-RECHUCK-)
(----- TOOL LIST -----)	(----- TOOL LIST -----)
(T0101 - CTPS15FR HOLDER)	(T4141 - VCMT 070204 BORE BAR)
(T0101 - CTPS15FR HOLDER)	(T4242 - 4MM CRB 2FL 14 LOC)
(T0202 - DCGT 11T304 TURN HOLDER)	(T4343 - VCMT 070204 BORE BAR)
(T0404 - GTMX32150 T01 OD HOLDER)	(T4444 - 6MM X 90DEG CRB SPOT DRILL)
(T0606 - CSVT11FRP60-035A TURN HOLDER)	(T4545 - 7.8MM JOBBER DRILL)
(T0707 - 3MM X 60DEG HSS CENTERDRILL)	
(T0808 - 1.0MM JOBBER DRILL)	G0 T0
(T2121 - 6MM X 60DEG HSS CENTERDRILL)	G28 W0
(T2424 - 2.0MM SCREW MACH DRILL)	G99 G97 G80 G40 G18
	M5 (SUB SPINDLE STOP)
(----- CUTOFF PARAMETERS -----)	T2000 (SUB HOME + ALIGN WITH MAIN)
#500 =01 (CUTOFF TOOL NUMBER)	
#501 =12. (CUTOFF SHANK WIDTH)	
#502 =1.5 (CUTOFF INSERT WIDTH)	
#503 =-.05 (CUTOFF LAST_X)	
#504 =-36.75 (CUTOFF LAST_Z)	
#505 =5000 (CUTOFF SPEED RPM)	
#506 =316 (CUTOFF SPEED SFPM)	
#507 =.09 (CUTOFF FEED FPR)	
#508 =497.29 (CUTOFF FEED FPM)	

22. Save and Close the Part.

Lesson 3 – Programming a Part for Swiss-Turn Advanced Options Part 2

Upon successful completion of this lesson, you will be able to:

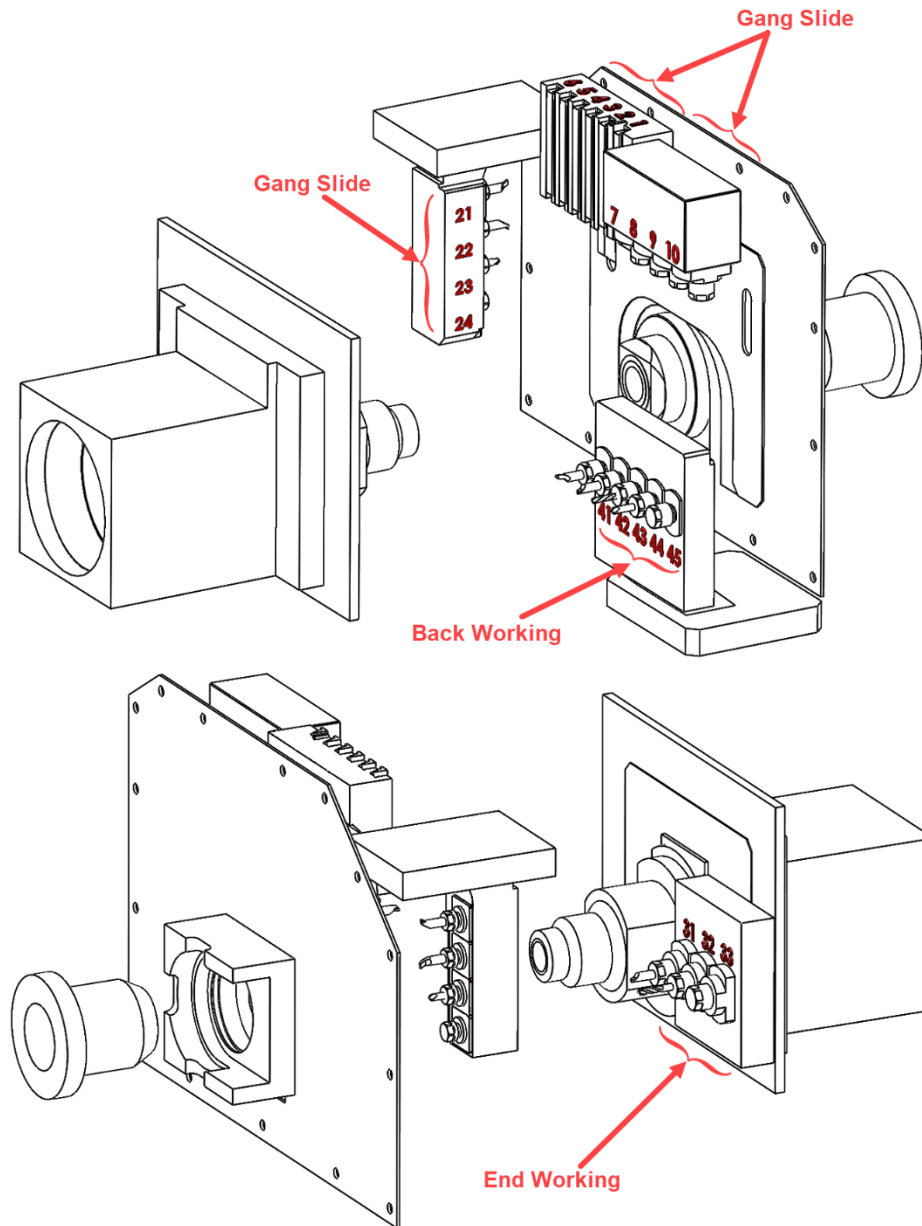
- Program a part using a three path machine (Gang, End and Back working)
- Set up tooling and advanced tooling parameters.
- Understand Sliding Head Parameters
- How to implement segmentation Manager to segment toolpaths.
- How to segment an OD thread operation.
- Advanced Sync manager options for three post machining.
- Tool Shift parameters.

Case Study – Advanced Swiss Turn Part Programming Part 2

In this exercise we will further apply the principles discussed in previous lessons to program a part file for a Swiss turn machine. In the previous exercise we used a simple machine with a gang slide tool post and a back working tool post for features machined on the sub spindle.

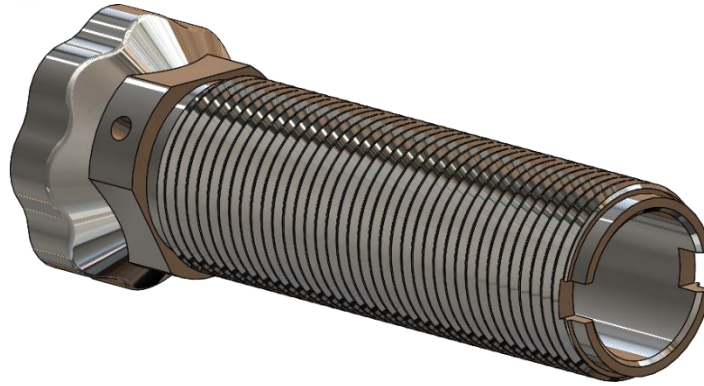
Now we will use a machine that has an additional end working tool post mounted to the sub spindle. In this lesson we will show how to set up the machine to use tools on all three posts.

Here is a basic diagram of the machine we will be using.



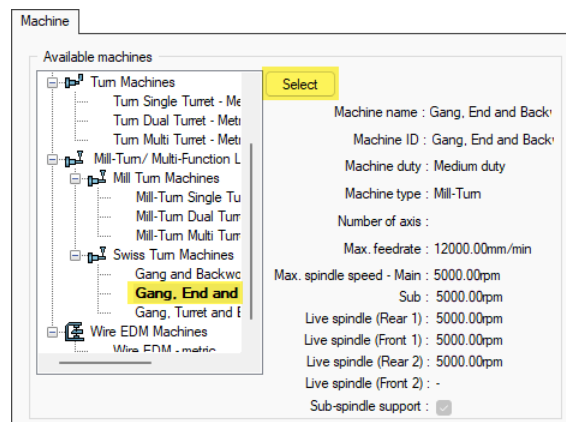
1. Open Part.

- Open the part file **LESSON_03.SLDPRT** from the **Lesson_03** folder.

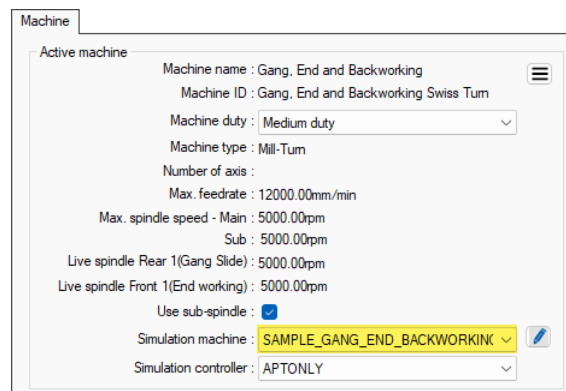


2. Define the Machine.

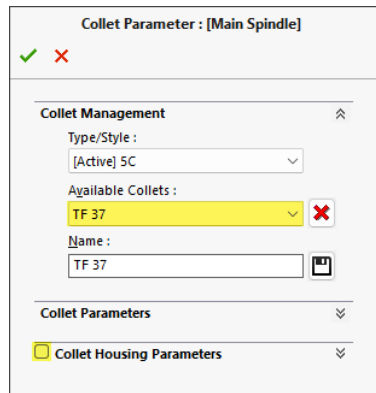
- On the **SOLIDWORKS Feature** tree, suppress the feature **Thread1**.
- Go to the **CAMWorks Feature** tree.
- Edit the machine definition and go to the **Machine** tab.
- Select the **Gang, End and Backworking** machine under the **Swiss Turn Machines** area in the **Available machines** area.
- Click **Select**.



- Under the **Active machine** section, ensure that **SAMPLE_GANG_END_BACKWORKING** is selected as the **Simulation machine**.



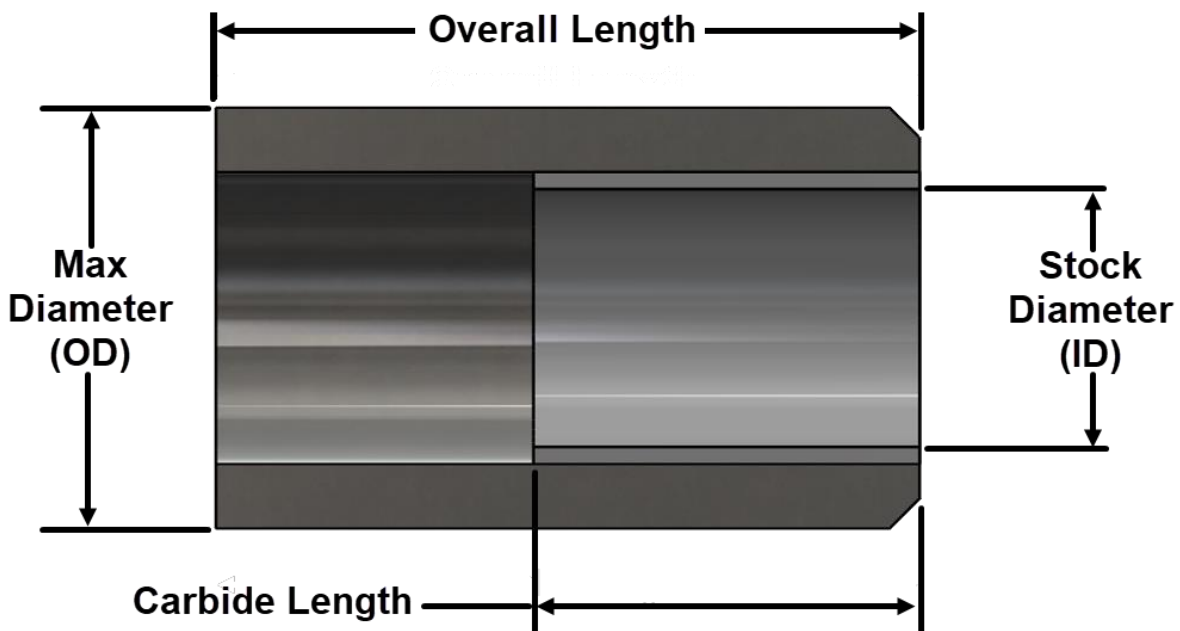
- g. On the **Fixture** tab, under the **Main Spindle Information**, edit the Collet and select the **TF 37** collet.
- h. Make sure the **Collet Housing Parameters** selection is unchecked.



- i. Click **OK**.
- j. Edit the Guide Bushing.

Carbide Length

The following parameters define the geometry of the Guide Bushing:



Carbide Length is the length of the tungsten carbide insert that lines the inside of a guide bushing. The length of the carbide insert within the guide bushing provides a greater contact surface with the bar stock. A longer carbide length is particularly beneficial for:

- **Machining Long Parts** – For parts with a high length-to-diameter ratio, a longer carbide insert provides additional support along the length of the bar stock. This increased contact

area helps to minimize vibration and deflection, allowing for more stable cutting and preventing issues like chatter or bent parts.

- Enhanced Rigidity – The longer contact area ensures the bar is held more securely, always keeping the cutting tool close to the support point. This rigidity is the fundamental principle behind Swiss-type machining and is what allows for the creation of long, slender, and highly accurate components.

Later in this lesson, this length will affect how we use the Segmentation manager.

- Under the **Guide Bushing Management** section, select **TD32S** from the list of the **Available Guide Bushings**.
- Under the **Guide Bushing Parameters**, set the **Stock Diameter** to **30mm**.
- Set the **Carbide Length** to **35mm**.

Guide Bushing

✓ ✗

Guide Bushing Management

Available Guide Bushing : TD32S ✗

Name : TD32S

Guide Bushing Parameters

Stock Diameter (ID) : 30mm

Max Diameter (OD) : 49mm

Carbide Length : 35mm

Overall Length : 82mm

- Click **OK**.
- Set the **Initial collet rechuck Z** to **150mm**.

Initial collet rechuck Z : 150mm

- Edit the **Sub Spindle Collet** parameters.
- Set the **Available Collets** to **TF 25**.
- Uncheck the **Collet Housing Parameters** checkbox.

Collet Parameter : [Sub Spindle]

✓ ✗

Collet Management

Type/Style : [Active] SC

Available Collets : TF 25 ✗

Name : TF 25

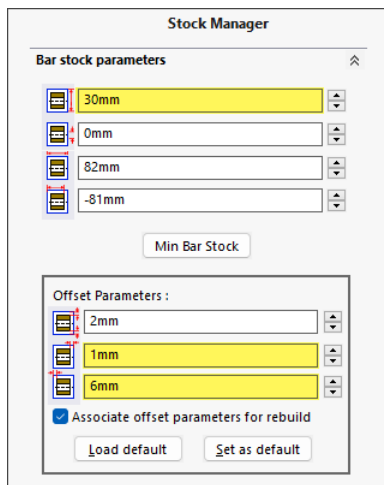
Collet Parameters

☐ Collet Housing Parameters

- s. Click **OK**.
- t. Click **OK** to the Machine Definition.

3. Edit Stock Definition.

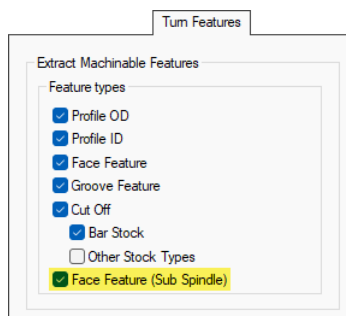
- a. Edit the **Stock Manager**.
- b. Set the **Outside Diameter** to **30mm**.
- c. Set the **Front Face Of Stock Offset** to **1mm**.
- d. Set the **Back Face Of Stock Offset** to **6mm**.



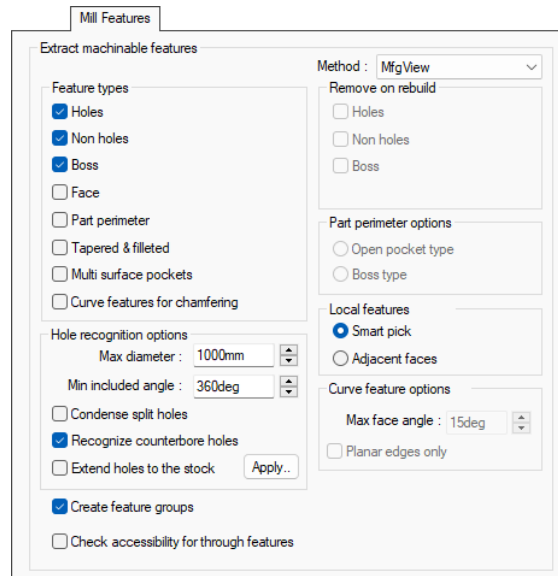
- e. Click **OK**.


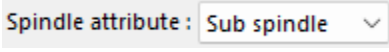
4. Define Machinable Features.

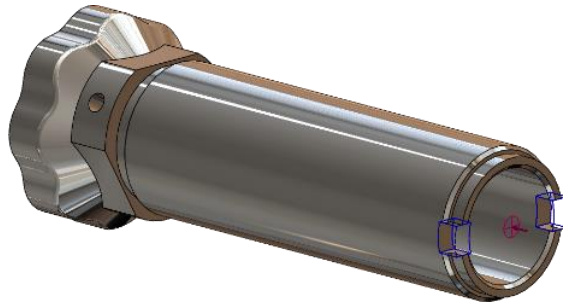
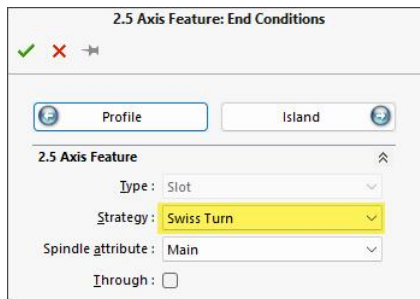
- a. In the CAMWorks Options, on the **Turn Features** tab, click on the checkbox next to **Face Feature (Sub Spindle)**.



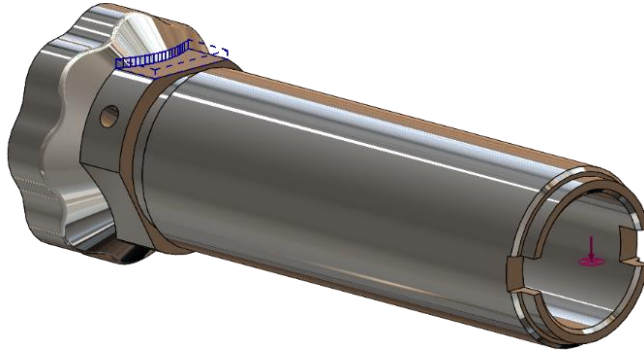
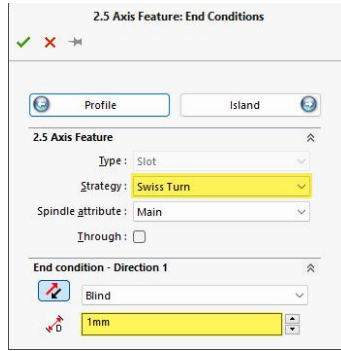
- b. Set the **Mill Features** options as shown below:



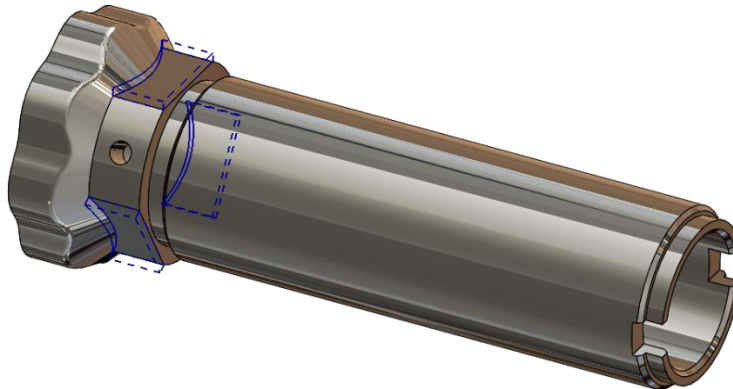
- c. Click **OK**.
- d. Run the **Extract Machinable Features** command from the CAMWorks CommandManager toolbar.
- e. Copy **ID Feature1** (CTRL+Drag and Drop). Rename the copied ID Feature to **ID Feature2**.  **ID Feature2 [Swiss Turn]**
- f. Edit the **ID Feature2**'s feature parameters and Change the Spindle attribute to Sub spindle. 
- g. Click **OK**.
- h. Create face milling **Mill Part Setup** and generate slot features as shown below. Use the **Swiss Turn** strategy for both these features.



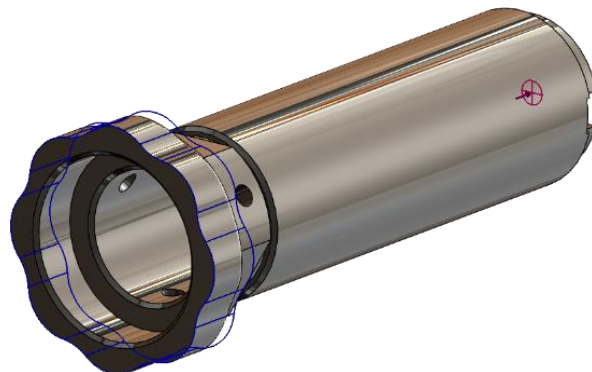
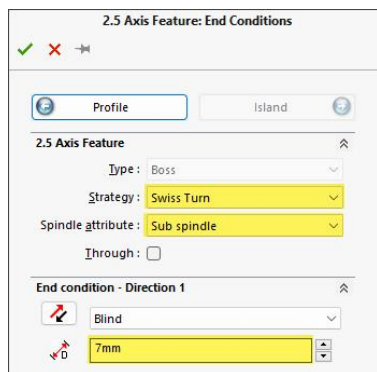
- i. Create **Mill Part Setup** for the flat on the top of the part as shown below. Create slot feature for the flat. Use the **Swiss Turn** strategy. Set the **Blind distance** to **1mm**.



- j. Pattern this slot feature around the axis of the part.



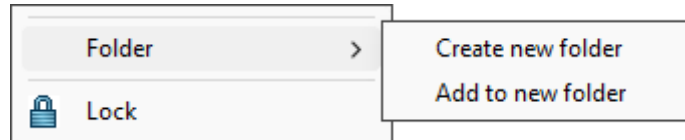
- k. Create a **Mill Part Setup** on the back of the part. Create a Boss feature as shown below. Set the **Strategy** to **Swiss Turn**. Set the **Spindle attribute** to **Sub Spindle**. Set the **Blind Feature Depth** to **7mm**.



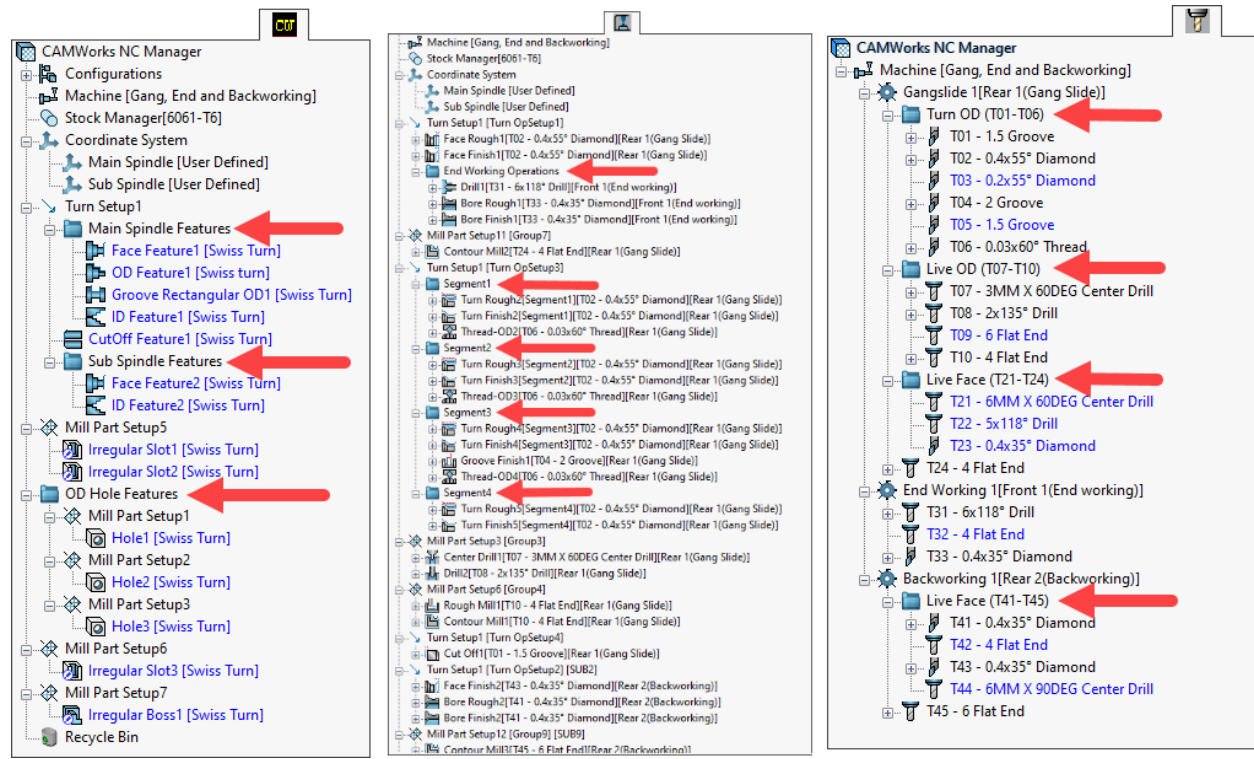
Folders

Folders can be created within the Feature, Operation, and Tools trees. This functionality helps with organization and makes it easier to navigate complex projects.

Two options, **Create New Folder** and **Add to New Folder**, included on the right-click menu. Both options create a new folder above the selected item. The difference is that the **Add to New Folder** option automatically places the selected item into the new folder, while the **Create New Folder** option does not.

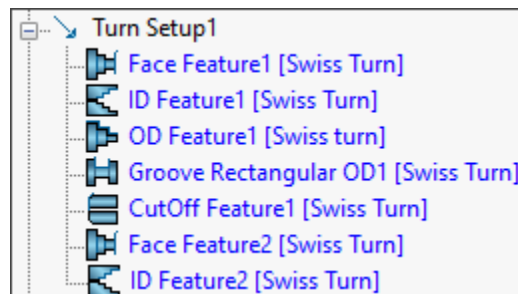


Once created, you can use **drag-and-drop** to add and arrange features, operations, part setups, and tools into these folders. You can also drag and drop the folders themselves to different locations within the trees. This makes it much easier to group similar items together, like all features with the same machining direction, and ultimately makes managing projects in CAMWorks more efficient.

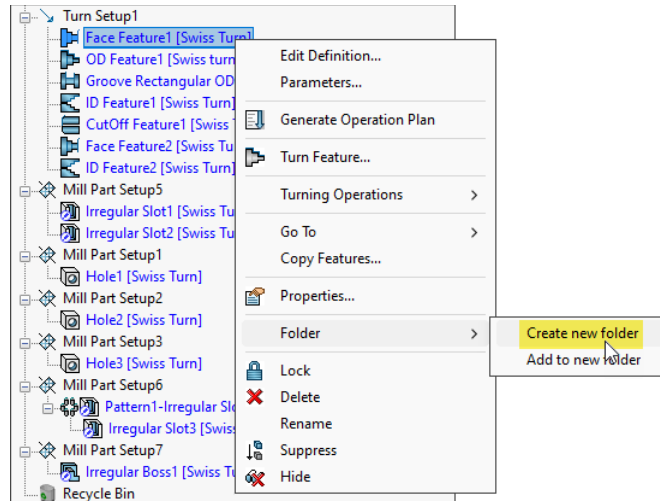


5. Add Folders to the CAMWorks Feature Tree.

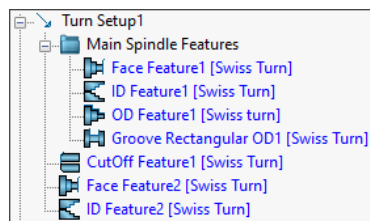
- Under Turn Setup1, order the features in the following order:



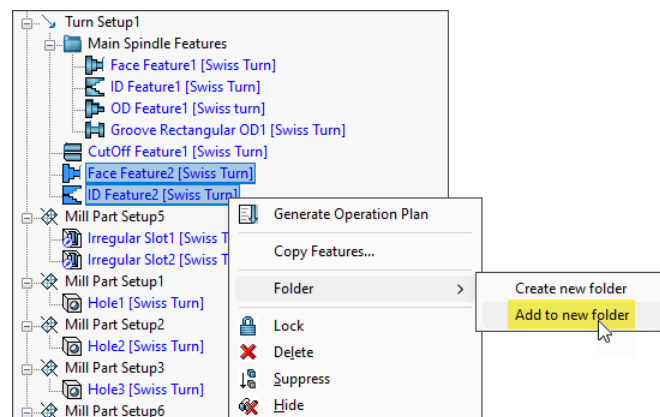
- Right-Click on **Face Feature1** and select Folder, **Create new folder**.



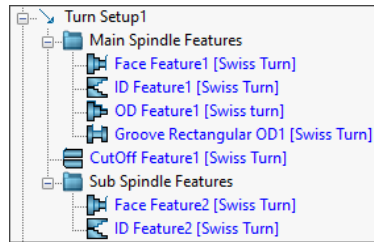
- c. Rename the new Folder1 to **Main Spindle Features**.
- d. Drag the features **Face Feature1**, **ID Feature1**, **OD Feature1**, and **Groove Rectangular OD1** into this **Main Spindle Features** folder.



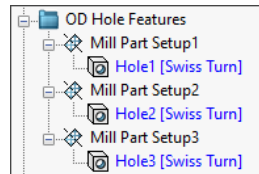
- e. CTRL+select **Face Feature2** and **ID Feature2**.
- f. Right-click on these features and select **Folders, Add to new folder**.



- g. Rename the new folder to **Sub Spindle Features**.



- h. Add **Mill Part Setup 1**, **Mill Part Setup2**, and **Mill Part Setup3** to a folder and rename the folder to **OD Hole Features**.

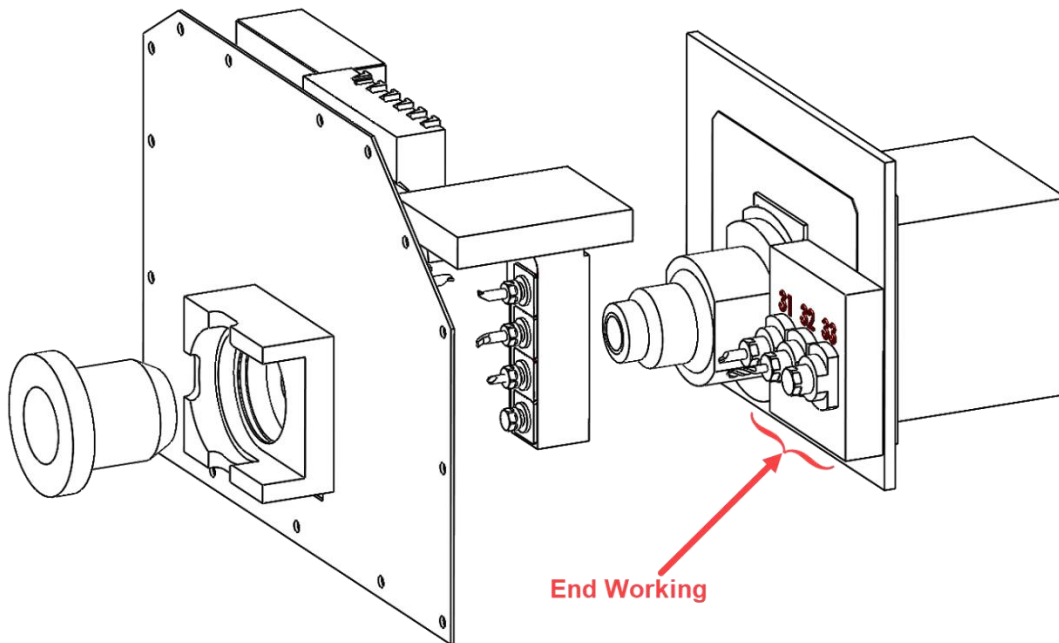


End Working Tool Definition

After generating the operations, we can organize the operations into folders.

The next step is to generate the operations for these features.

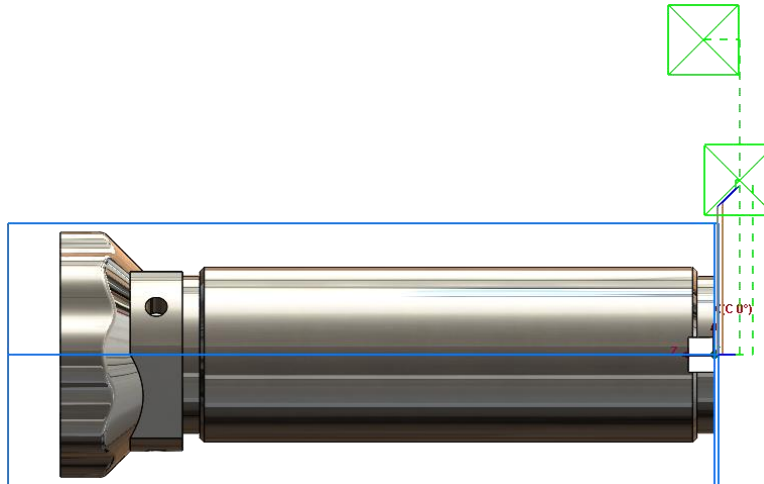
We will start by facing the part followed by drilling, and boring the ID. These ID operations will be done using tools mounted in the stations on the end working tool post.



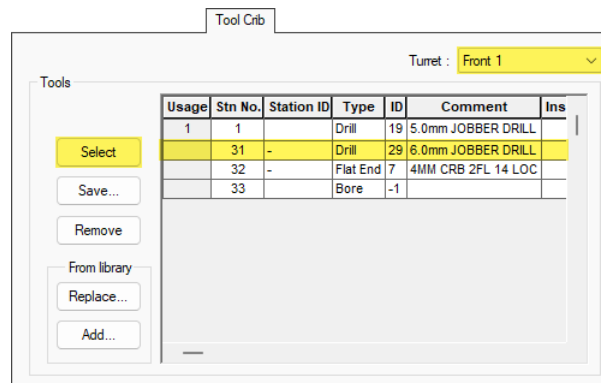
This will allow us to use a longer tool machine to the depth of the feature.

6. Generate Turn Operations.

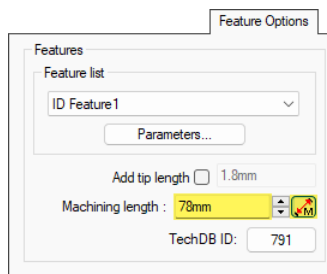
- Right-click on **Face Feature1** and select **Generate Operation Plan**.
- Generate the toolpath for **Face Rough1** and **Face Finish2**.



- Generate Operation Plan for **ID Feature1**.
- Edit **Drill1** and go to the **Tool** tab, **Tool Crib** tab.
- Set the **Turret** to **Front1**.
- Select tool **Stn No.31** and click **Select**.



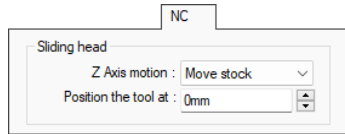
- Go to the **Feature Options** tab. Press the **Override Machining Depth** button and set the **Machining length** to **78mm**.



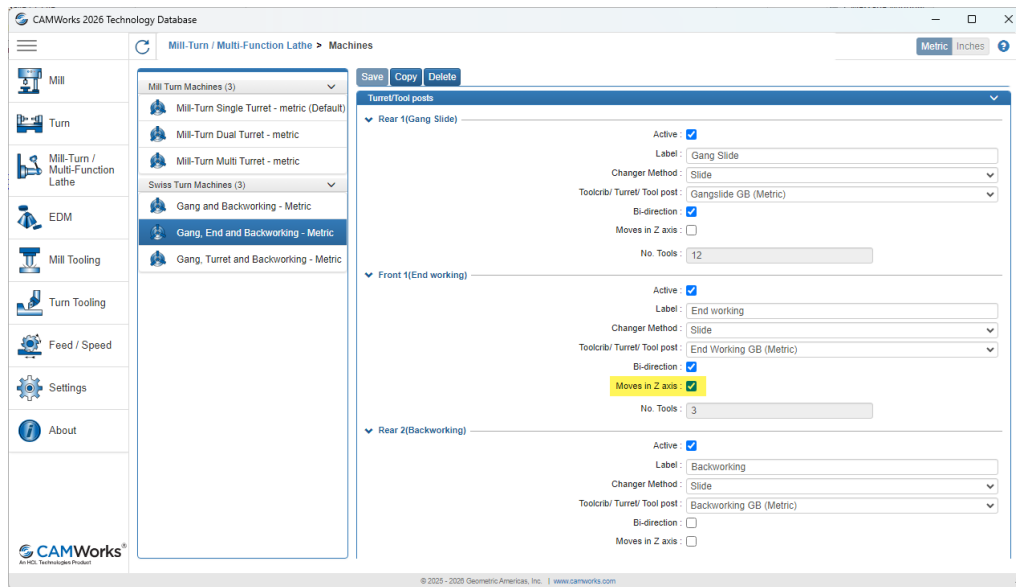
Sliding Head Parameters

Since this current tool is mounted on the End Working tool post that has a Z axis, we have the option to control the motion of the Z axis with the **Sliding head** parameters.

In the **Sliding Head** group box, you'll find parameters for certain machining operations.

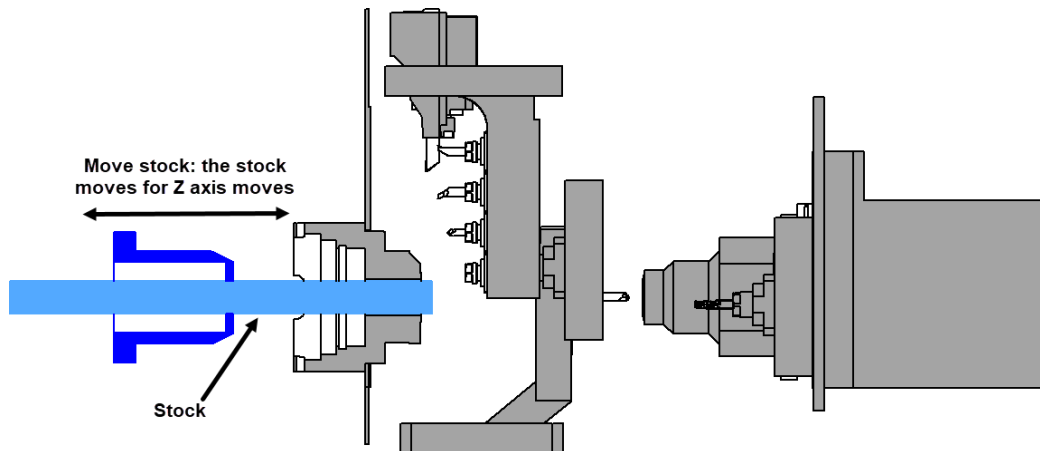


This section is only visible on the **NC tab** if these three conditions are met: the machine is a **Swiss Turn/Multi-Function Lathe** with a sliding head, and the operation is on the **Main Spindle**, and the tool post is defined in the TechDB as able to move in the Z axis.

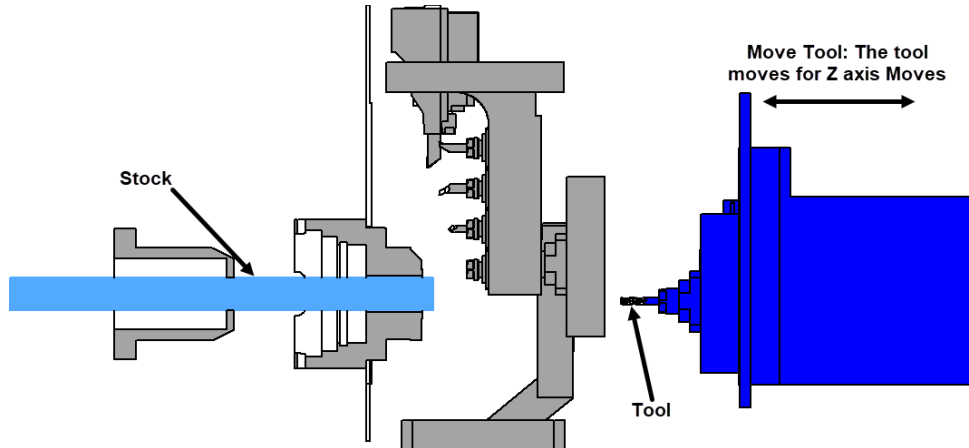


The **Z Axis Motion** setting, a dropdown menu with two choices, determines how Z-axis movement is handled. The default option is set in the TechDB.

- **Move Stock:** The workpiece itself moves to provide the Z-axis motion during machining.



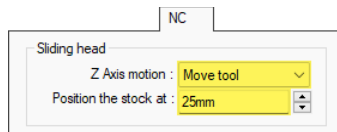
- **Move Tool:** The tool moves along the Z-axis to machine the part.



Position the tool at

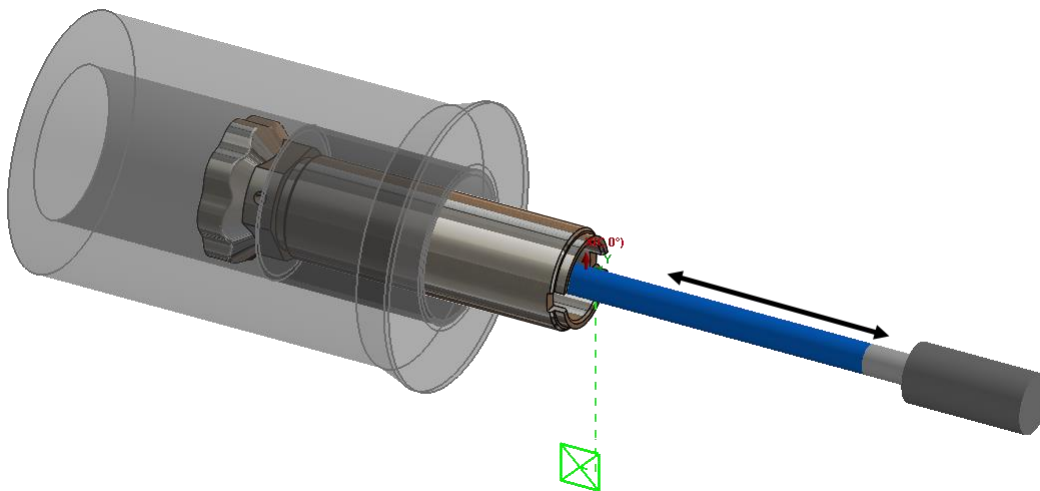
This parameter sets the absolute Z-axis position of the tool relative to the face of the **Guide Bushing**. You must enter a value greater than zero. Like the previous setting, the default value is pulled from the TechDB.

- h. Go to the **NC** tab, under **Sliding head**, Set the **Z Axis motion** to **Move tool**.
- i. Set the **Position the tool at** to **25mm**.

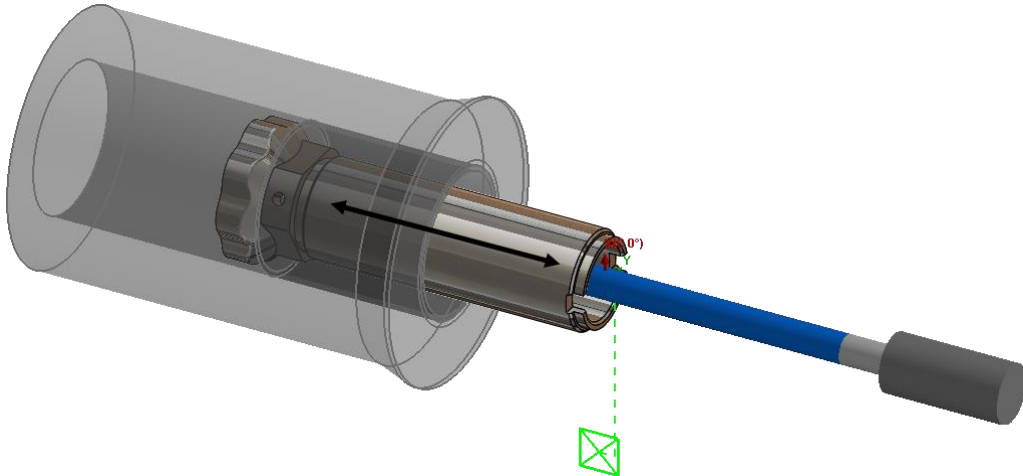


- j. Click **Preview**.
- k. Run the **Step Trough Toolpath** command and examine the toolpath.

Notice that the tool moves during the step through toolpath.



If we set the **Z Axis motion** to **Move stock**, the step through would show the tool post moving rather than the stock.



- l. Edit **Bore Rough1**. Go to the **Tool** tab, **Tool Crib** tab.
- m. Set the **Turret** to **Front 1**. Turret : Front 1
- n. Click on the **Station** tab.
- o. Change the **Tool Number** to **33**.

Station

Station

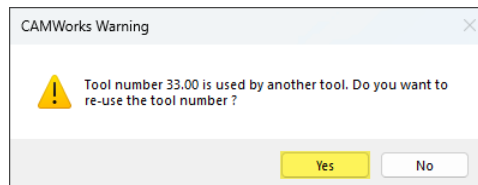
Tool number : 33 - 0

Station ID :

Gage offset (XYZ) : -5.1mm 0mm -54.45mm Reset

Station type : Any

- p. Click **Yes** to the **CAMWorks Warning** message to reuse the tool number.



- q. Go to the **Bore Rough** tab and under the **Cut type** section, check on the **Mirror about centerline** checkbox. ☒ Mirror about centerline
- r. Under the **Profile parameters** section, uncheck the **Undercut** checkbox.

Bore Rough

Profile parameters

First cut amount : 3mm

Max cut amount : 3mm

Final cut amount : 1mm

Step in angle : 0deg

Step in amount : 0mm

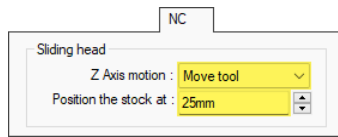
Step out angle : 0deg

Step out amount : 0mm

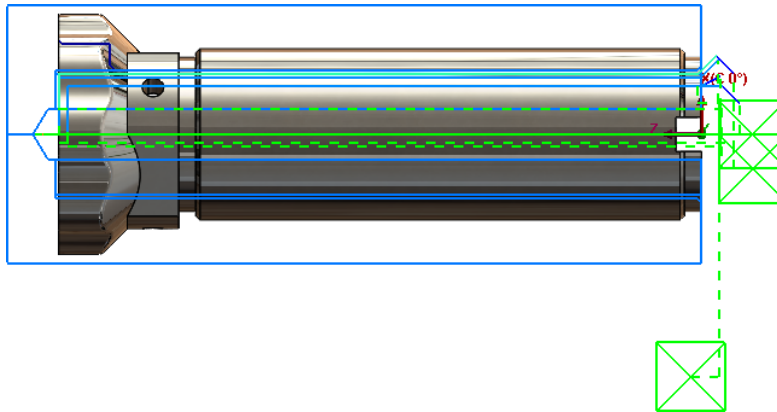
☐ Sharp corner

☐ Undercut

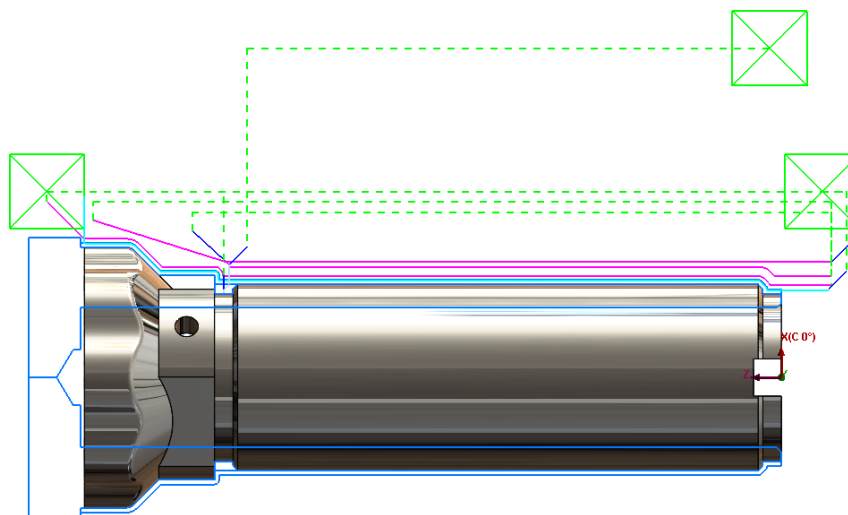
- s. On the **NC** tab, under **Sliding head**, set the **Z Axis motion** to **Move tool** and the **Position the tool at** to **25mm**.



- t. Click **OK**.
- u. Do the same thing for **Bore Finish**: change the tool to **T33**, on the **Bore finish** tab, check on the **Mirror about centerline** checkbox ☒ **Mirror about centerline**, and uncheck the **Undercut** checkbox.
- v. On the **NC** tab, under **Sliding head**, set the **Z Axis motion** to **Move tool** and the **Position the tool at** to **25mm**.
- w. Click **OK**.
- x. Generate the toolpath for **Drill1**, **Bore Rough1** and **Bore Finish1**.



- y. Generate Operations for **OD Feature1** and **Groove Rectangular OD1**.
- z. Delete **Grove Rough1** and generate toolpath for **Turn Rough1**, **Turn Finish1** and **Grove Finish1**.



7. Add Folders to the operation tree.

- Create a folder for the facing operations and change the name to **Face Operations**.
- Create a folder for the end working tools and change the name to **End Working Operations**.

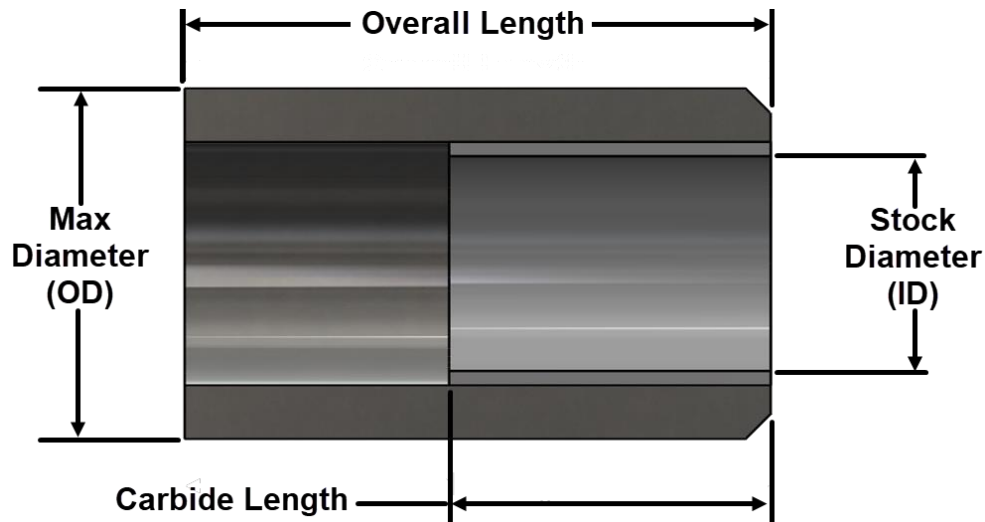


Segmentation Manager

For the OD Turning operations, we will be using the Segmentation Manager to automatically break the OD operations into predefined segments according to the length of the part in Z.

Segmentation Manager

In **Swiss turning**, a part's machining length must be shorter than or equal to the guide bushing's carbide length to ensure the part is held securely. If the part is too long, it needs to be machined in smaller sections.

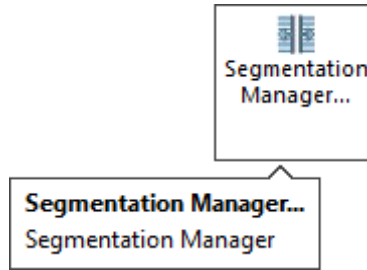




The **Segmentation Manager Wizard** is a tool that helps users divide these long turning operations into multiple segments. This interactive wizard lets you split and position each segment to ensure the part is properly clamped throughout the machining process.

Only Turn Roughing and Turn finishing operations on the outside diameter can be segmented in the segmentation manager

To Access the Stock Manager:

- CommandManager: **CAMWorks 2026> Segmentation Manager...**



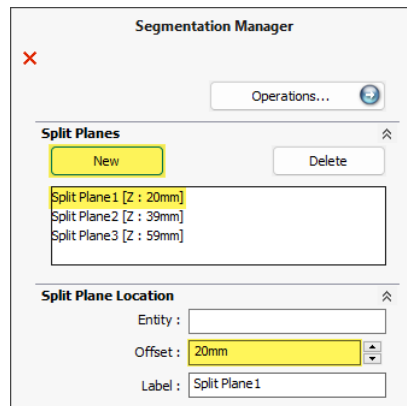
- Toolbar: **Segmentation Manager** 
- CAMWorks Feature Tree: Right Click the **CAMWorks NC Manager** item and select **Segmentation Manager...**  **Segmentation Manager...**
- Tools pull down menu, **CAMWorks>Segmentation Manager...**

Earlier, when we defined the machine for this part, we defined the Carbide Length as 35mm. Each segment on this part should be less than 35mm to ensure the stock is held securely and accurately within the guide bushing as it moves along the Z motion of the of the cut.

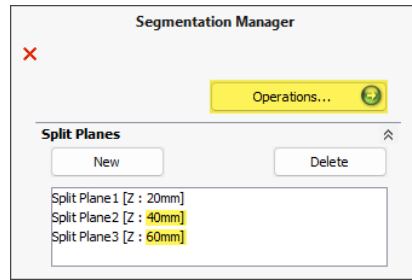
For this example, we will set each segment to 20mm.

8. Launch the Segmentation Manager.

- a. Generate the toolpath for the **Turn Rough1**, **Turn Finish1** and **Groove Finish1** operations.
- b. Click on the **Segmentation Manager...** command on the CAMWorks 2026 CommandManager toolbar.
- c. Under **Split Planes** click on the **New** button three times.
- d. Select the **Split Plane1** segment in the list. In the **Split Plane Location** area, set the **Offset** to **20mm**.

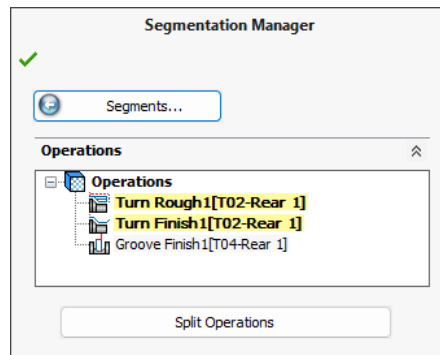


- e. Set the **Split Plane2** to an **Offset** of **40mm**.
- f. Set the **Split Plane3** to an **Offset** of **60mm**.
- g. Click on the **Operations...** button.

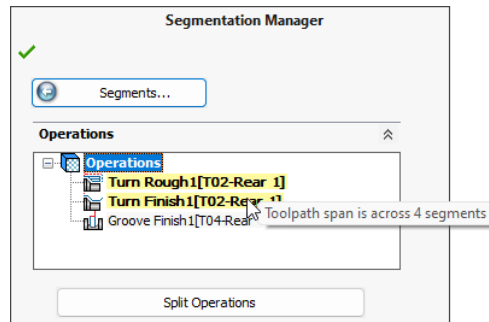


The **Operations list box** in the Segmentation Manager Wizard displays all operations with a generated toolpath, including those from the Sub Spindle. The list does not include specific operation types such as **Sub Spindle, Probe, Post, Face, Cut Off, Suppressed, or Thread operations**, nor does it list operations for ID features or those using **VoluTurn** or **Prime Turning** methods.

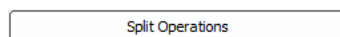
Operations that are already split will also appear in this list, including any that have been deleted but are still in the Recycle Bin.



Operations that span across one or more segment boundaries will be **highlighted** in the list. This highlighting indicates that they are candidates for splitting. When you hover over a highlighted operation, a tooltip will show how many segments its toolpath crosses.

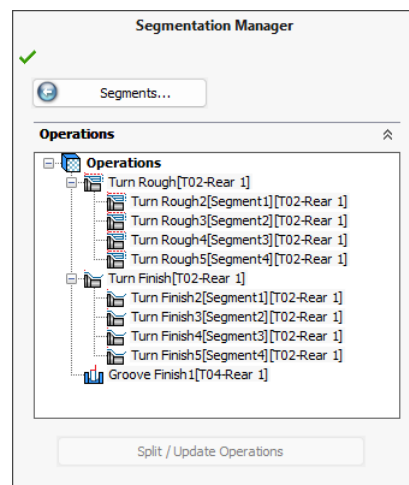


To split these operations, simply select one or more highlighted operations and click the **Split Operations** command.

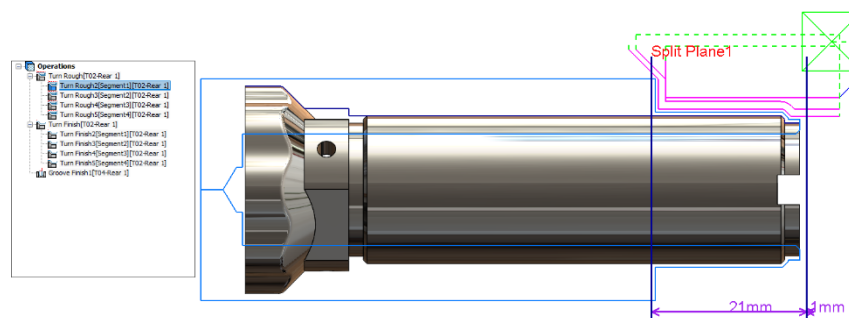


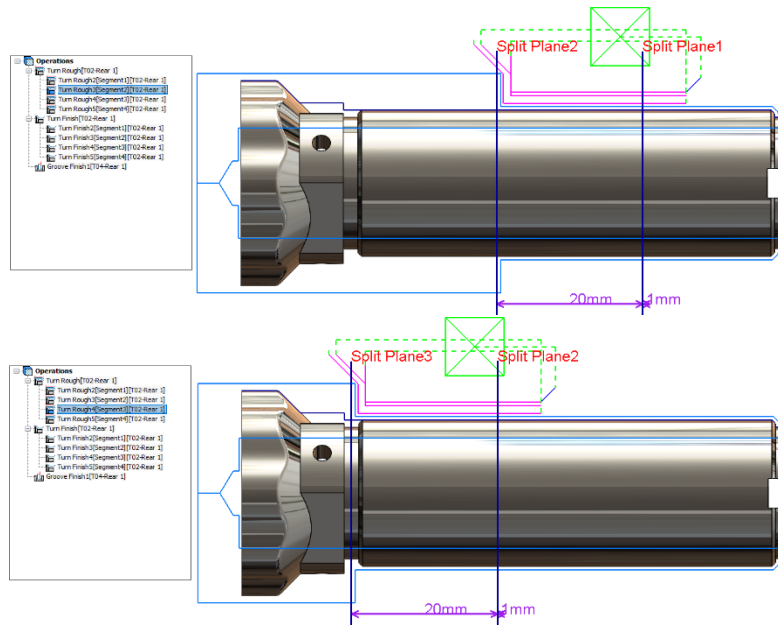
The command will be enabled as soon as you select a valid operation. Each selected operation will be split into the number of new operations equal to the number of segments it spans. For instance, if an operation's toolpath crosses three segments, it will be split into three new operations.

- h. From the **Operations** list, CTRL+select **Turn Rough1** and **Turn Finish1**.
- i. Click on the **Split Operations** button.
- j. Examine the results.



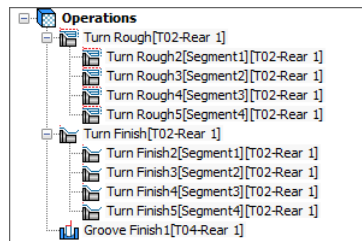
After being split, the new operations appear as nested nodes under their original parent operation in the list. Selecting a nested operation will display its corresponding toolpath in the graphics area.





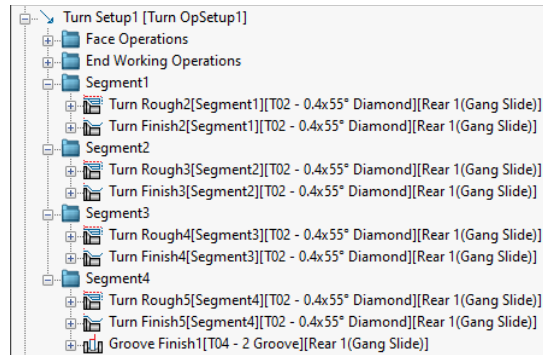
The naming convention for these split operations is: **<Operation Name><Segment #>[Tool Number - Turret Name]**.

- **<Operation Name>** refers to the name of the original operation.
- **<Segment #>** indicates the segment number the new operation corresponds to.
- **[Tool Number - Turret Name]** specifies the tool number and which turret it's from.



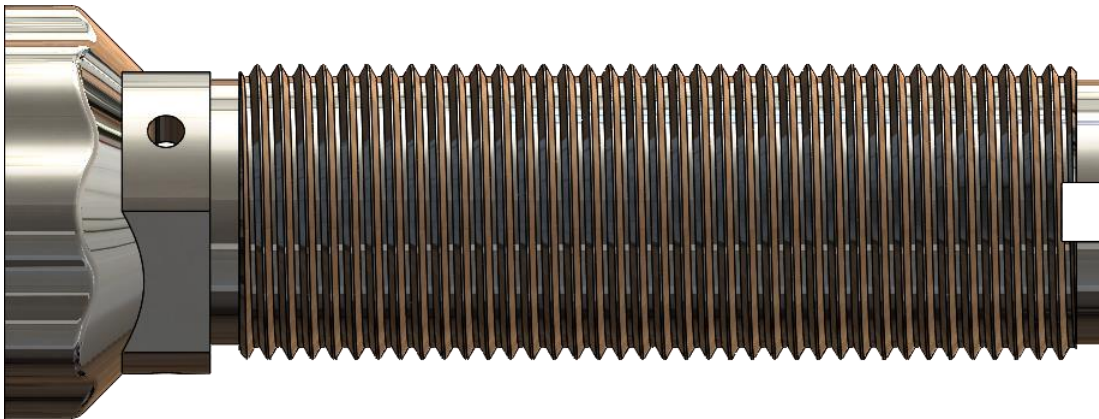
If we need to adjust the split planes, click on the **Segments...** button to return to the previous screen.

- Click **OK**.
- Examine the operations under **Turn Setup1**.
- Add folders for each segment named **Segment1**, **Segment2**, **Segment3** and **Segment4**.
- Drag the segments of the operations into their respective folders.



Manually Segmenting a Thread

On this example part there is an external thread on the OD of the part that needs to be machined in segments as we did the OD Turning operations.

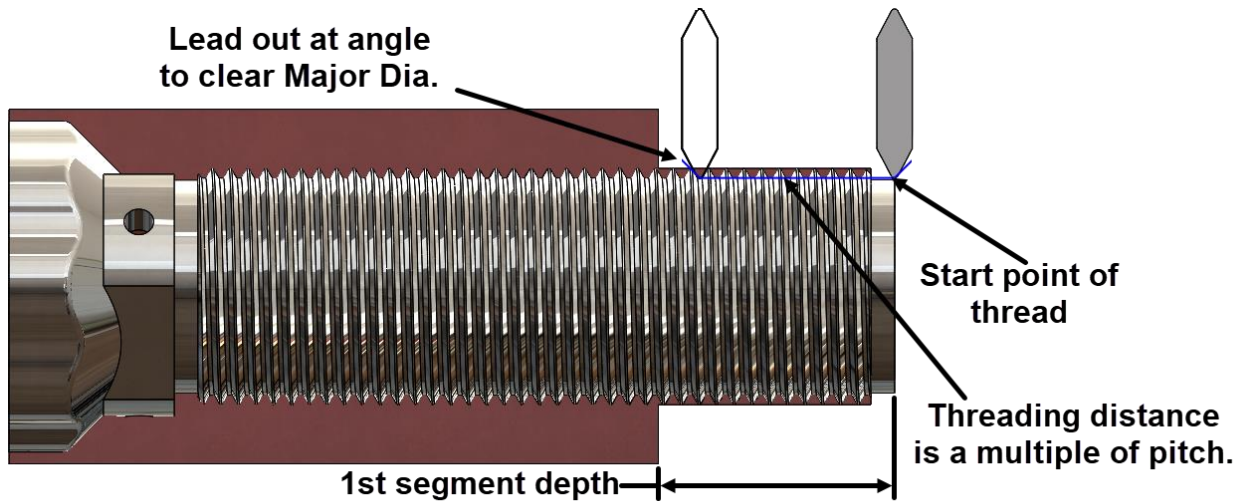


OD threads are not supported operations for the Segmentation Manager at this time for automatically segmentation, but we can manually add each segment of the threads.

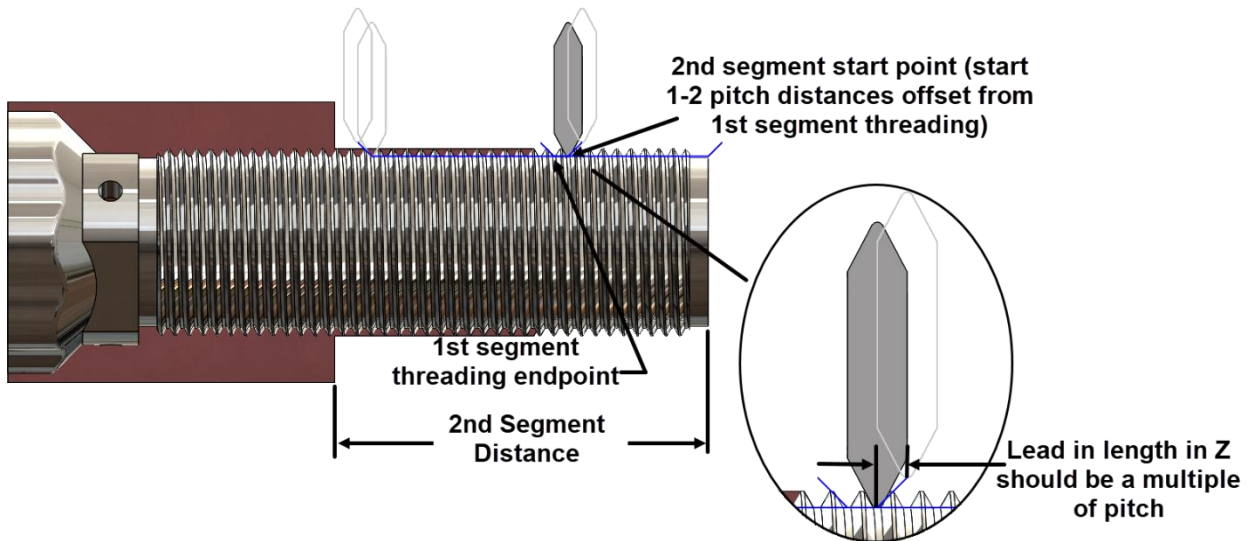
To do this, we will need to define multiple OD turn features with at the same distances as the segments defined in the segmentation manager.

On the model the thread on the OD is an M20x1.5. The pitch, 1.5mm on this part, is a critical dimension to consider when defining the OD Features. This is because the subsequent threading operations after the first threading operation must start at a specific place along the Z direction based on the Pitch dimension for the threads to line up.

The first segment will start at the front face edge of the part and go to distance that is a multiple of the pitch of the part. The tool will have to lead-out at an angle until it clears the major diameter of the thread.



Subsequent thread segments will start one to two pitch distances before the previous segment's end point to ensure the threads line up with the previous segment. There should be an angular lead-in where the Z length of the lead-in is a multiple of the pitch distance.

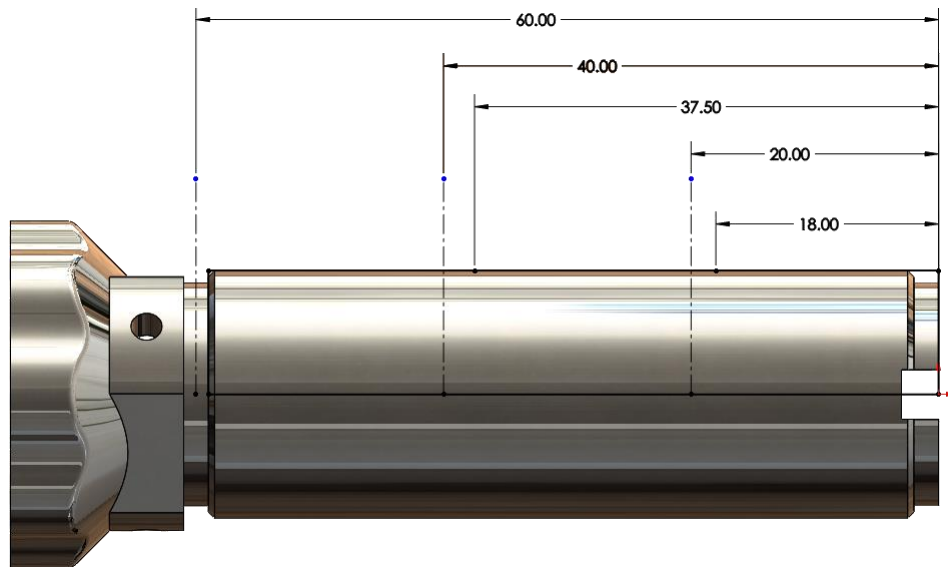


Because there needs to be an angular lead-in and lead-out to these threading operations, only G32 thread can be used. We cannot use any threaded canned cycle like G76. This will be set in the operation parameters of the threading operation.

On this part, let's look at the sketch *Screw_Sketch*.

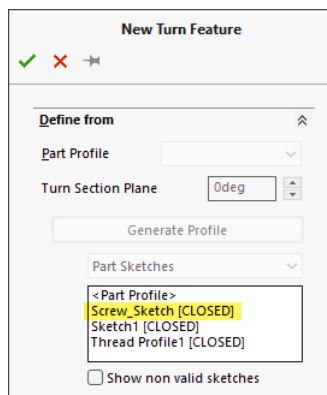
9. Create Segmented Thread Features.

- a. On the SOLIDWORKS feature tree, edit ***Screw_Sketch***.

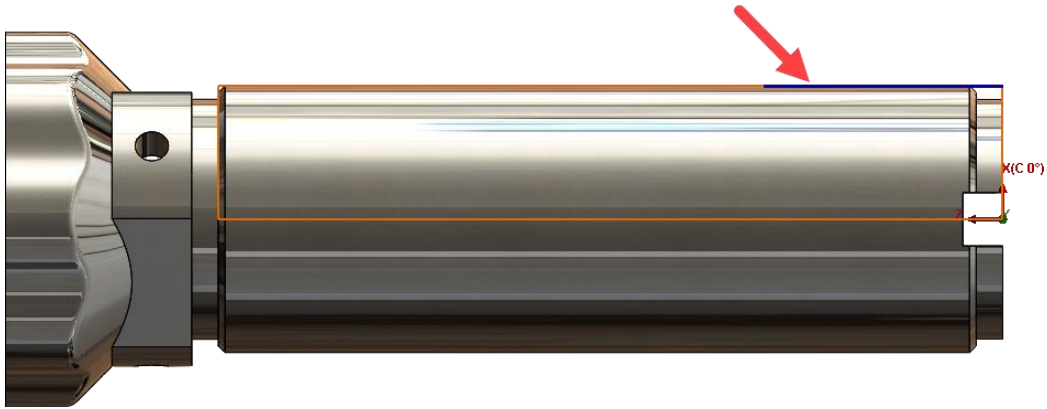


The vertical construction lines represent the split plane distances defined in the segmentation manager. The **37.5mm** and **18mm** dimensions are the distances in Z of the individual threaded segments. The values are multiples of the pitch of the thread, **1.5mm**.

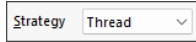
- b. Exit the sketch definition.
- c. On the CAMWorks Operation Tree, insert an **OD feature** under **Turn Setup1**.
- d. Under **Define from** select the **Screw_Sketch** part sketch.

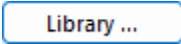


- e. In the graphics area, select the first segment of the sketch.



- f. Set the **Strategy** to **Thread**. Click **OK**.
- g. Create two more OD Features with the thread strategy for the other segments of **Screw_Sketch**.



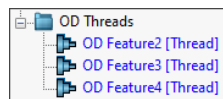
- h. Edit the parameter of each of these new OD Features, click on the Library  button at the bottom of the page, and select **M20.0 X 1.50** from the **Thread Condition** list.

Tools Database - Thread Condition (metric)

	ID	Type	Designation	Pitch	EndPitch	DepthOfThread	ProcessMethod	Units	Spindle
185	1175	MF	M 18.0 X 2.00	2.000000	0.000000	1.227600	1	1	1
186	1177	MF	M 20.0 X 1.00	1.000000	0.000000	0.613400	1	1	1
187	1179	MF	M 20.0 X 1.50	1.500000	0.000000	0.919500	1	1	1
188	1181	MF	M 20.0 X 2.00	2.000000	0.000000	1.226800	1	1	1
189	1183	MF	M 22.0 X 1.00	1.000000	0.000000	0.613900	1	1	1
190	1185	MF	M 22.0 X 1.50	1.500000	0.000000	0.920000	1	1	1
191	1187	MF	M 22.0 X 2.00	2.000000	0.000000	1.227400	1	1	1
192	1189	MF	M 24.0 X 1.00	1.000000	0.000000	0.613200	1	1	1
193	1191	MF	M 24.0 X 1.50	1.500000	0.000000	0.920500	1	1	1

OK Cancel

- Click **OK**. Do this for the other two threads.
- Add these OD Features to a folder and name the folder **OD Threads**.



10. Generate operations for the threaded features.

- Generate operation plan for the threaded features.
- Edit the first **Thread OD1** operation parameters.
- On the **Thread** tab, under **Parameters**, set the **Start length** to **0mm**.

Thread

Parameters

Depth per cut : 0.38mm

Final cut amount : 0.18mm

Spring passes : 0

Step in angle : 90deg

Step out angle : 90deg

Start length : 0mm

End length : 0mm

Pitch : 1.5mm

Thread depth : 0.92mm

Major diameter : 20mm

Minor diameter : 18.16mm

Library ... ID : 1179

Designation : M 20.0 X 1.50

Thread type : MF

- Under **Program point**, uncheck the **Canned cycle output** checkbox.

Thread

Program point

☐ Canned cycle output

Groove tool : By Orientation

Other Tools : Tool Nose Center

- On the **NC** tab, under **Approach** set the **Approach from** to **Previous toolpath retract**. Under **Retract**, set the **Retract to** to **Retract Point**.

NC

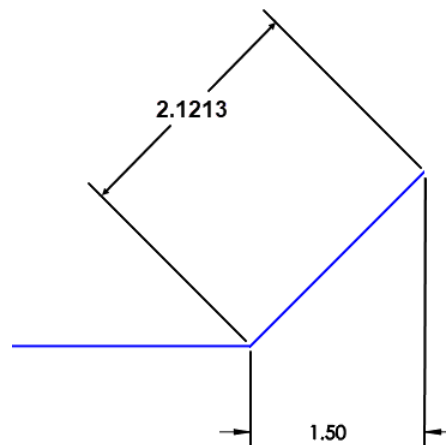
Approach	Retract
Strategy : Z then X	Strategy : X then Z
<input checked="" type="checkbox"/> Gouge check	<input checked="" type="checkbox"/> Gouge check
Approach from : Previous toolpath retract	Retract to : Retract Point
	(X=50.80mm, Z=1.27mm)
Approach point	Retract point
X reference : Clearance X	X reference : Clearance X
Offset : 0mm	Offset : 0mm
X : 19.99mm	X : 50.8mm
Z reference : Clearance Z	Z reference : Clearance Z
Offset : 0mm	Offset : 0mm
Z : 23.64mm	Z : 1.27mm
<input type="checkbox"/> Go to Home on tool change	

- f. On the **Lead In/Out** tab, set the **Leadin type** to **Parallel**. Set the **Leadin amount** to **2.1213mm**, set the **Lead angle** to **45deg**.

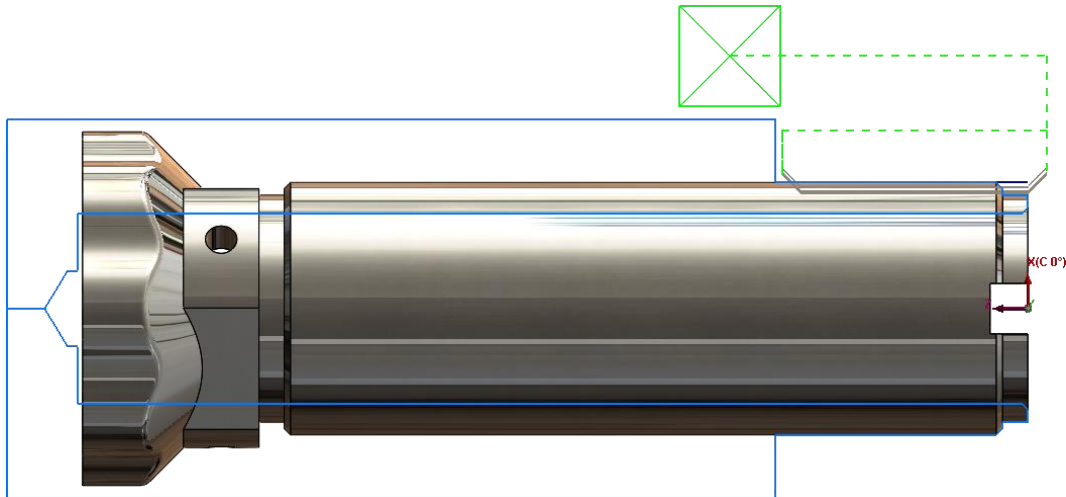
Lead In/Out

Leadin type : Parallel	Leadout type : Same as leadin
Leadin amount : 2.12mm	Leadout amount : 2.12mm
Lead angle : 45deg	Lead angle : 45deg
Start Thread cycle at : Before leadin	End Thread cycle at : After leadout

The 2.12mm value is the distance that would make the Z length of the leading move match the pitch.



- g. Click **OK**. Drag **Thread-OD1** into the **Segment1** folder. Generate the toolpath for **Thread-OD1**.



- h. Edit **Thread-OD2**, on the **Thread** tab, under **Parameters**, set the **Start length** to **1.5mm**. Uncheck the **Canned cycle output** checkbox.

Thread	
Parameters	
Depth per out :	0.38mm
Final cut amount :	0.18mm
Spring passes :	0
Step in angle :	90deg
Step out angle :	90deg
Start length :	1.5mm
End length :	0mm
Pitch :	1.5mm
Thread depth :	0.92mm
Program point	
<input type="checkbox"/> Canned cycle output	
Groove tool :	By Orientation
Other Tools :	Tool Nose Center


- i. On the NC tab, under **Approach** set the **Approach from** to **Previous toolpath retract**. Under **Retract**, set the **Retract to** to **Retract Point**.

NC	
Approach	
Strategy : Z then X	
<input checked="" type="checkbox"/> Gouge check	
Approach from : Previous toolpath retract	
Approach point	
X reference :	Clearance X
Offset :	0mm
X :	19.99mm
Z reference :	Clearance Z
Offset :	0mm
Z :	23.64mm
Retract	
Strategy : X then Z	
<input checked="" type="checkbox"/> Gouge check	
Retract to : Retract Point	
(X=50.80mm, Z=1.27mm)	
Retract point	
X reference :	Clearance X
Offset :	0mm
X :	50.8mm
Z reference :	Clearance Z
Offset :	0mm
Z :	1.27mm
<input type="checkbox"/> Go to Home on tool change	

- j. On the **Lead In/Out** tab, set the **Lead in type** to **Parallel**. Set the **Lead in amount** to **2.1213mm**, set the **Lead angle** to **45deg**.

Lead In/Out

Leadin type : **Parallel** Leadout type : Same as leadin

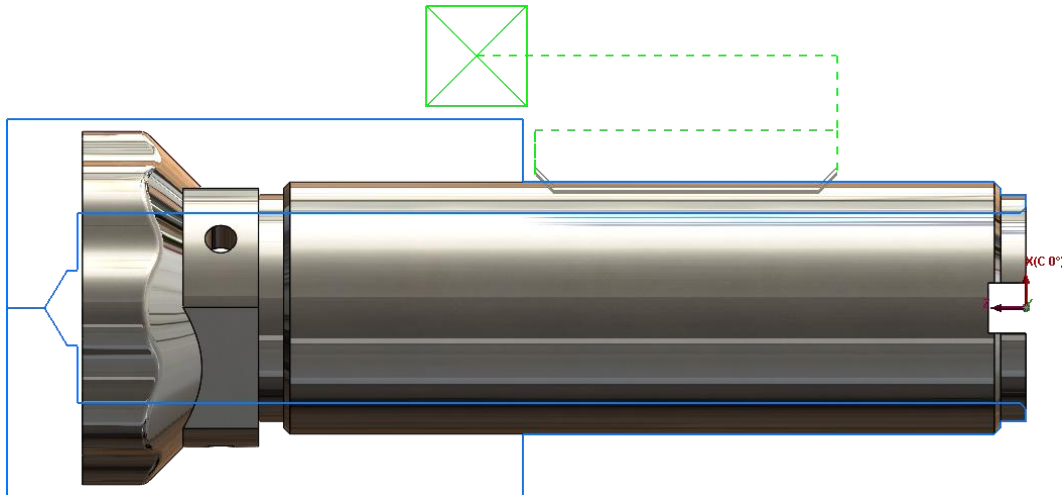


Leadin amount : **2.12mm** Leadout amount : 2.12mm

Lead angle : **45deg** Lead angle : 45deg

Start Thread cycle at : Before leadin End Thread cycle at : After leadout

- k. Click **OK**. Drag **Thread-OD2** into the **Segment2** folder. Generate the toolpath for **Thread-OD2**.



- l. Do the same thing with **Thread-OD3**. Edit **Thread-OD2**, on the **Thread** tab, under **Parameters**, set the **Start length** to **1.5mm**. Uncheck the **Canned cycle output** checkbox.

Thread

Parameters

Depth per cut : 0.38mm

Final cut amount : 0.18mm

Spring passes : 0

Step in angle : 90deg

Step out angle : 90deg

Start length : **1.5mm**

End length : 0mm

Pitch : 1.5mm

Thread depth : 0.92mm

Program point

☐ Canned cycle output

Groove tool : By Orientation

Other Tools : Tool Nose Center

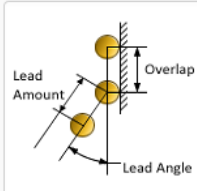
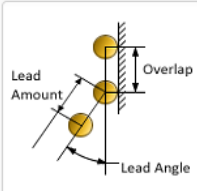
- m. On the NC tab, under **Approach** set the **Approach from** to **Previous toolpath retract**. Under **Retract**, set the **Retract to** to **Retract Point**.

NC

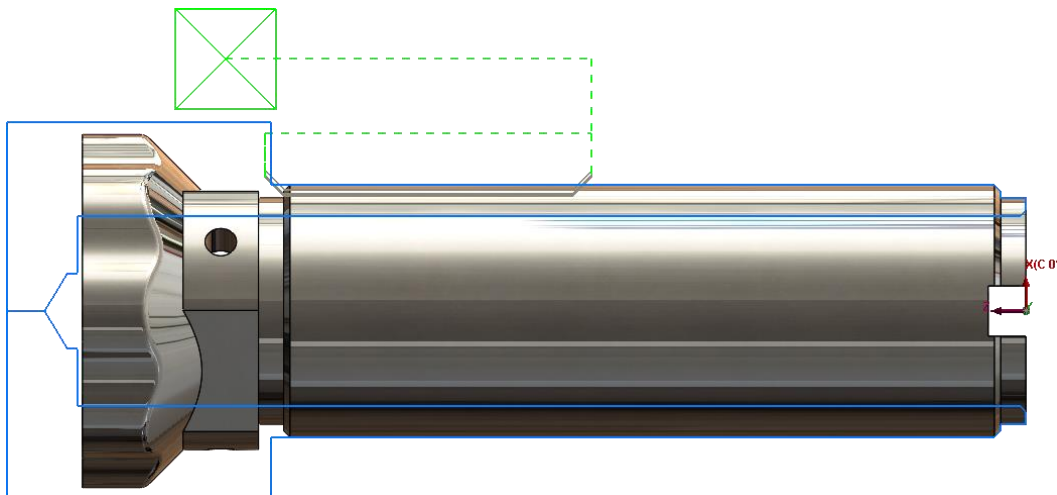
Approach	Retract
Strategy: Z then X	Strategy: X then Z
<input checked="" type="checkbox"/> Gouge check	<input checked="" type="checkbox"/> Gouge check
Approach from: Previous toolpath retract	Retract to: Retract Point
(X=50.80mm, Z=1.27mm)	
Approach point	Retract point
X reference: Clearance X	X reference: Clearance X
Offset: 0mm	Offset: 0mm
X: 19.99mm	X: 50.8mm
Z reference: Clearance Z	Z reference: Clearance Z
Offset: 0mm	Offset: 0mm
Z: 23.64mm	Z: 1.27mm
<input type="checkbox"/> Go to Home on tool change	

- n. On the **Lead In/Out** tab, set the **Leadin type** to **Parallel**. Set the **Leadin amount** to **2.1213mm**, set the **Lead angle** to **45deg**.

Lead In/Out

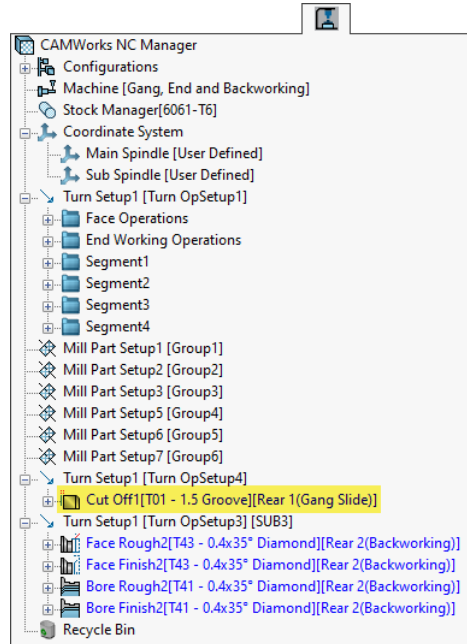
Leadin type: Parallel	Leadout type: Same as leadin
	
Leadin amount: 2.12mm	Leadout amount: 2.12mm
Lead angle: 45deg	Lead angle: 45deg
Start Thread cycle at: Before leadin	End Thread cycle at: After leadout

- o. Click **OK**. Drag **Thread-OD3** into the **Segment3** folder. Generate the toolpath for **Thread-OD3**.

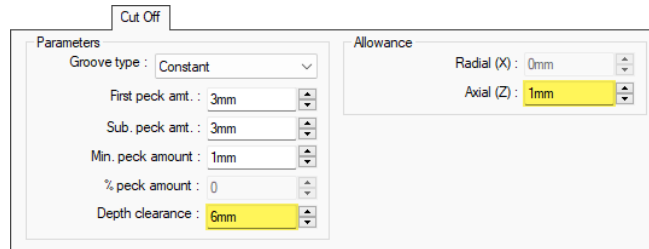


11. Generate Operation Plan for Cutoff Feature1 and the Sub Spindle Features.

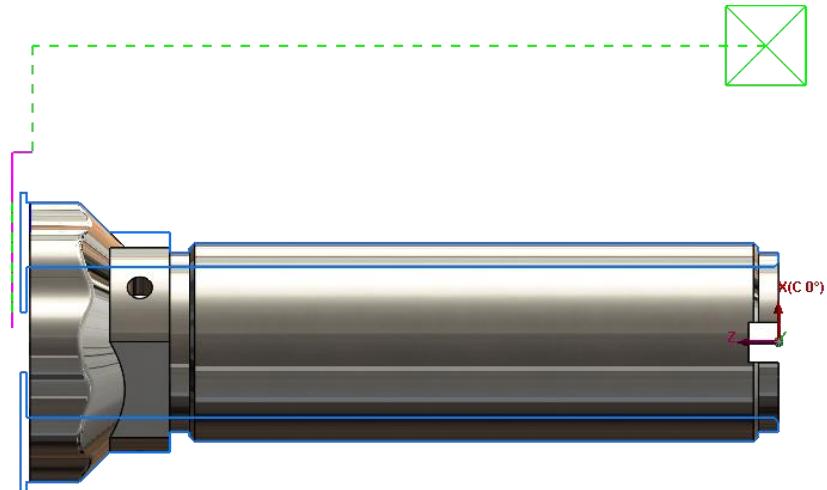
- Generate the operations to the **CutOff Feature1** and the features defined in the **Sub Spindle** folder.
- Right-click on **Turn Setup1** and select **Main Spindle Setup**.
- Drag **CutOff Feature1** into the new **Turn Setup1** and drag **Turn Setup1** below the Mill Part Setups.



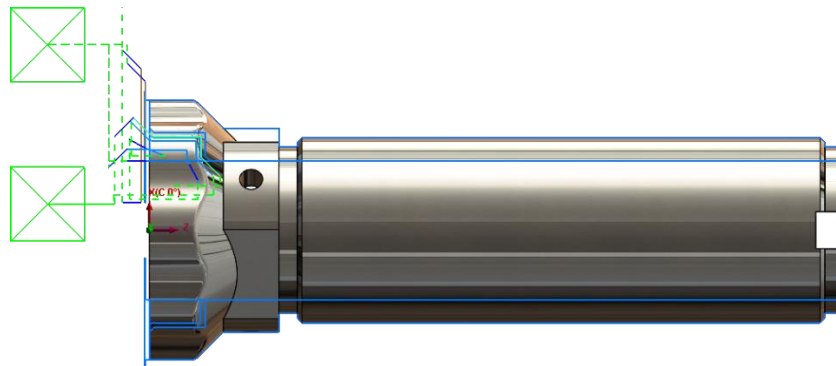
- Edit the **CutOff1** operation. On the **Cut Off** tab, under **Parameters** set the **Depth clearance** to **6mm**. Under **Allowance**, set the **Axial (Z)** to **1mm**.



- Click **OK** and generate the toolpath of the cutoff operation.

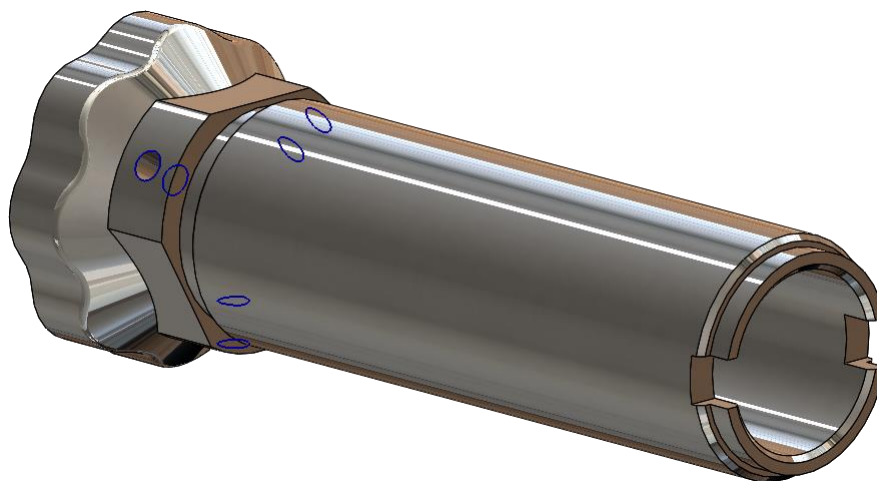


- f. Generate the toolpath on the operations on the Sub Spindle.



12. Generate the Milling Operations.

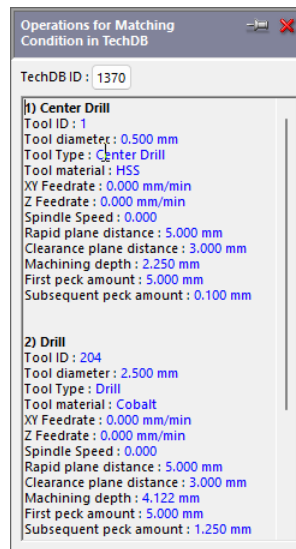
- Delete **Mill Part Setup2** and **Mill Part Setup3**.
- Apply circular pattern to **Hole1**. Pattern the hole three times.



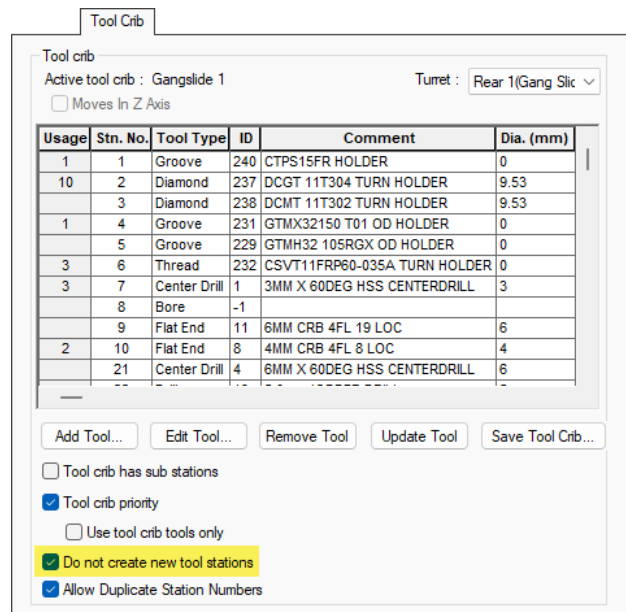
- c. Generate the Operation plan for the OD Hole Feature.

Add Tools to the Tool Crib.

Only the center drill operation got generated, despite the TechDB ID of that Hole feature allowing for a drill operation to be generated from GOP.

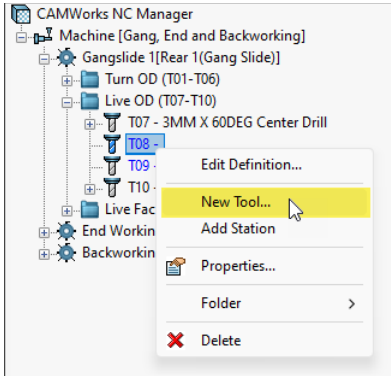


This is because of the tool crib settings to **Do not create new tool stations**. The tool defined for the drilling operation would have had to add a drill tool to the tool crib for **Generate Operation Plan** to create a Drill operation automatically.

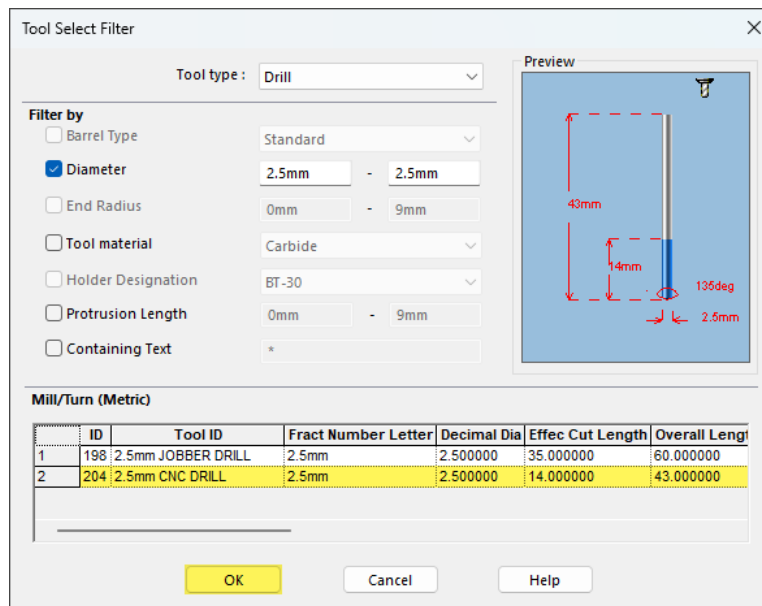


We can add this tool to the tool crib.

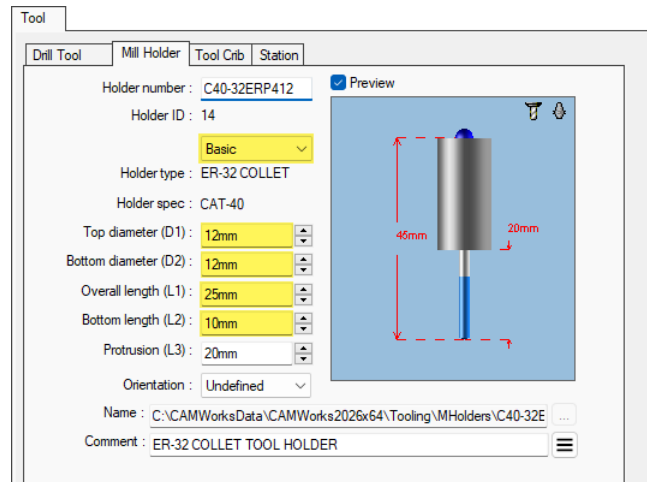
- d. Go to the CAMWorks Tool Tree.
- e. Right-click on **T08** – and select **New Tool**.



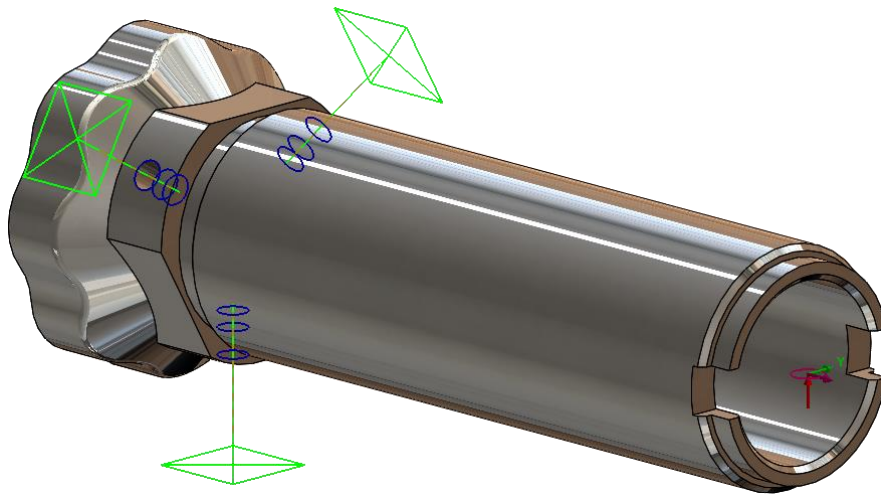
- f. Select the **2.5mm CNC Drill** from the **Tool Select Filter**. Click **OK**.



- g. Generate Operation plan for **Hole1**.
 h. Generate toolpath for both hole operations
 i. Edit **Drill2** and go to the **Tool** tab, **Mill Holder** tab.
 j. Set the **Holder** type to **Basic**, **Top diameter(D1)** to **12mm**, **Bottom diameter(D2)** to **12mm**. **Overall length(L1)** to **25mm**, and **Bottom length(L2)** to **10mm**.

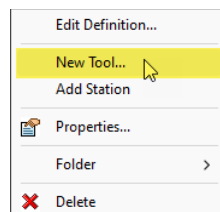


Both **Center Drill** and **Drill** operations are generated for this operation now.

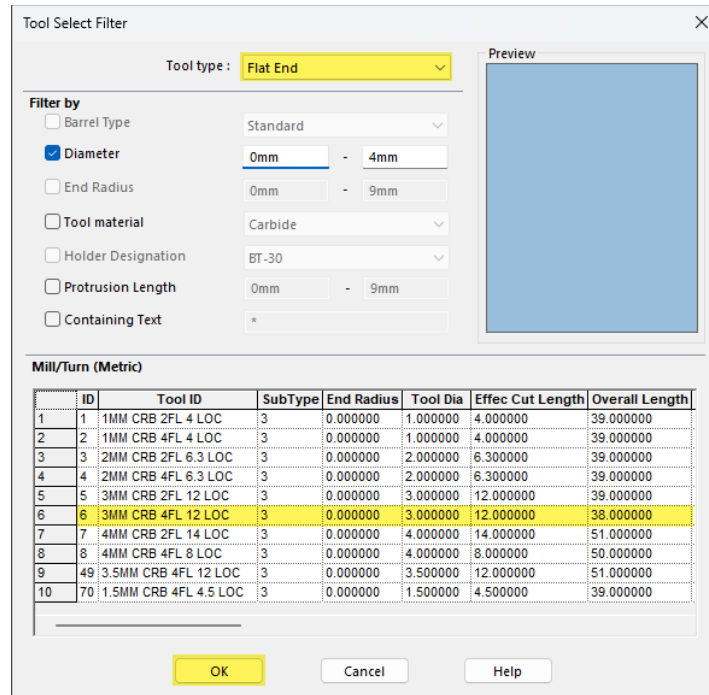


Next, we will need to add a tool to the Gang slide to machine the slots on the face of the part.

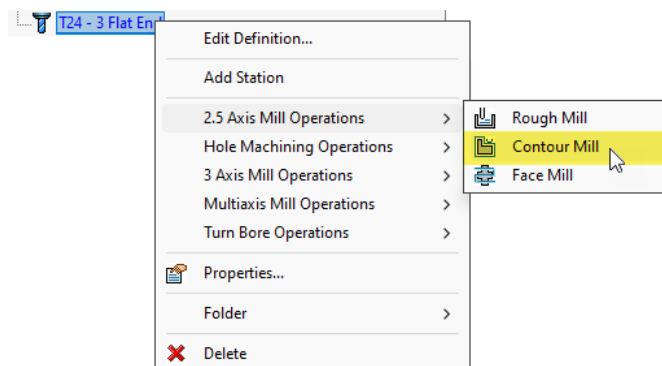
- k. Go to the **CAMWorks Tool Tree** tab.
- l. Right-click on **T24** – and select **New Tool...**



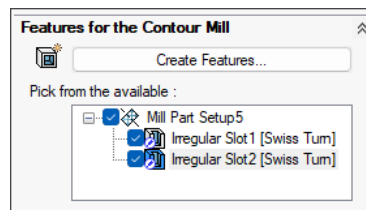
- m. In the **Tool Select Filter**, set the **Tool type** to **Flat End**.
- n. Select **3MM CRB 4FL 12 LOC** from the list.



- o. Click **OK**.
- p. Right-click on **T24 – 3 Flat End** and select **2.5 Axis Mill Operations, Contour Mill**.



- q. On the **Features** tab select the **Irregular Slot1** and **Irregular Slot2** features from the **Pick from the available** list.



- r. On the **Operation** tab, under **Operation Parameters**, select **Use TechDB defaults**. Select **Swiss Turn** from the drop-down list.

Operation Parameters

☒ Use TechDB defaults

Swiss Turn

☐ Copy from

Contour Mill1

- s. Click **OK**.
- t. In the operation parameters of this contour mill operation, go to the **Tool** tab, **Mill Holder** tab.
- u. Change the **Holder** type to Basic, set the **Top diameter (D1)** to **12mm**, set the **Bottom diameter (D2)** to **12mm**, set the **Overall length (L1)** to **25mm**, and set the **Bottom length (L3)** to **10mm**.

Mill Holder

Top diameter (D1) : 12mm

Bottom diameter (D2) : 12mm

Overall length (L1) : 25mm

Bottom length (L2) : 10mm

Protrusion (L3) : 15mm

Orientation : Undefined

- v. Go to the **Station** tab and set the **Gage offset (XYZ)** Z value to **25mm**.

Station

Tool number : 24 · 0

Station ID :

Gage offset (XYZ) : 0mm · 0mm · 25mm · Reset

Station type : Any

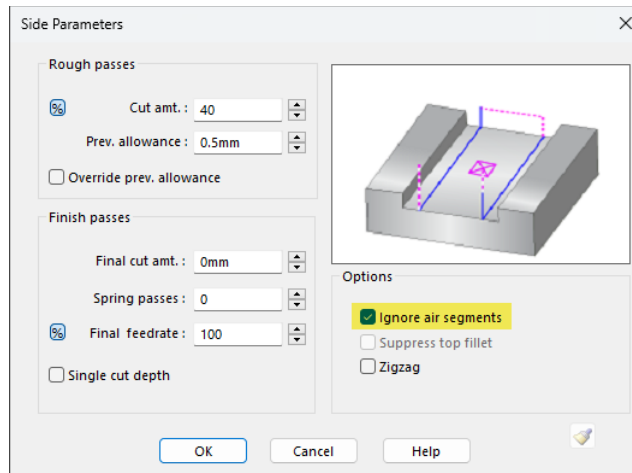
- w. Click **OK**.
- x. Generate the toolpath for the contour mill operation.



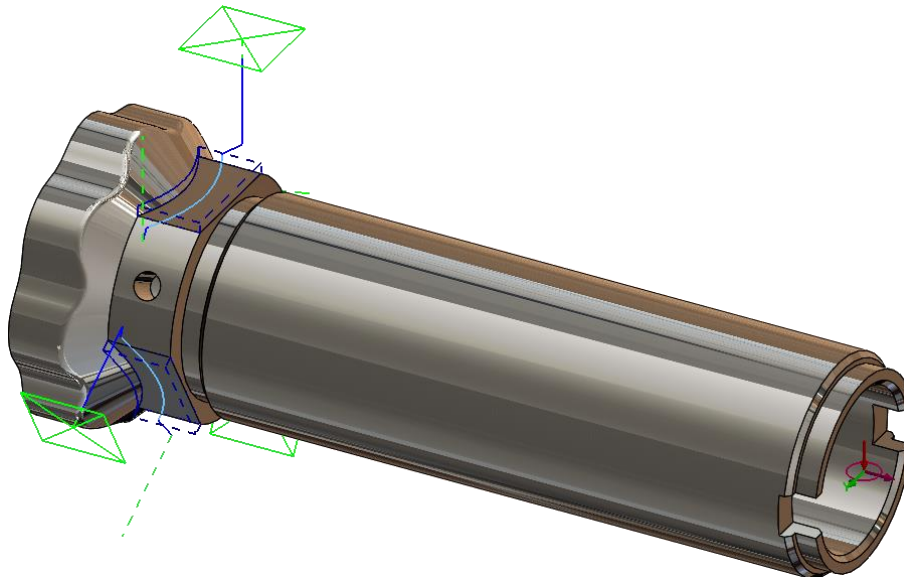
13. Generate the operations for the Pattern1-Irregular Slot3.

- a. Generate the Operation Plan to **Pattern-Irregular Slot3**.
- b. Delete **Rough Mill1**.

- c. Edit **Contour Mill2**.
- d. On the **Tool** tab, **Tool Crib** tab, select **Stn. No. 10 4MM CRB 4FL 8 LOC**.
- e. On the **Contour** tab, click the **Settings** button. Under **Options**, click on the **Ignore air segments** checkbox.

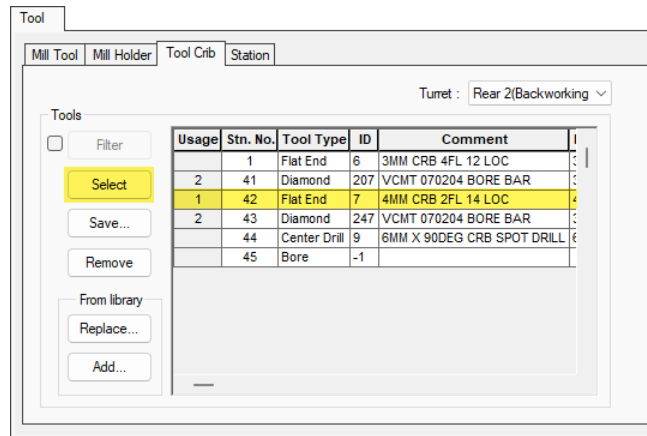


- f. Click **OK**.
- g. Generate the toolpath.

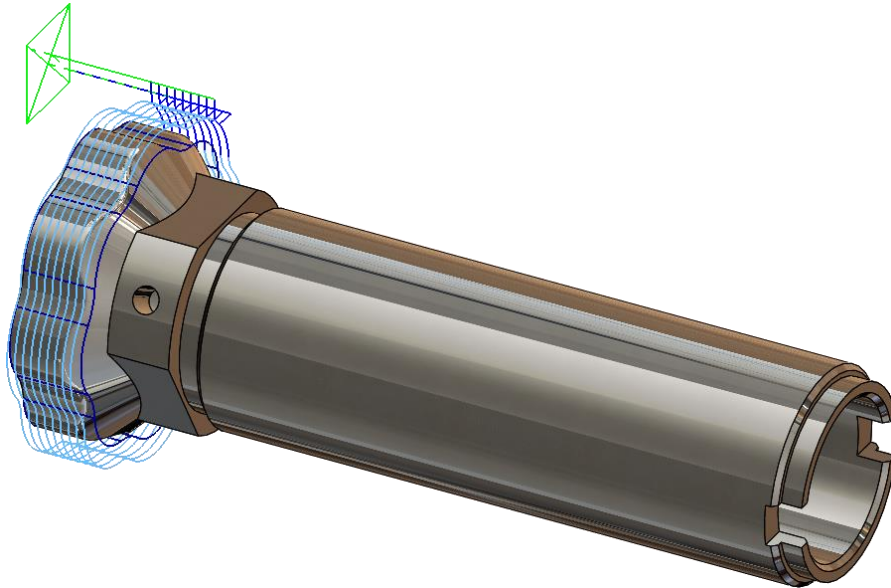


14. Generate operation for Irregular Boss1 feature.

- a. Generate Operation Plan for **Irregular Boss1**.
- b. Edit the Contour Mill operation. Go to the **Tool** tab, **Tool Crib** tab.
- c. Select **Stn. No. 42, 4MM CRB 2FL 14 LOC** and click **Select**.



- d. Click **OK**.
- e. Generate Toolpath.

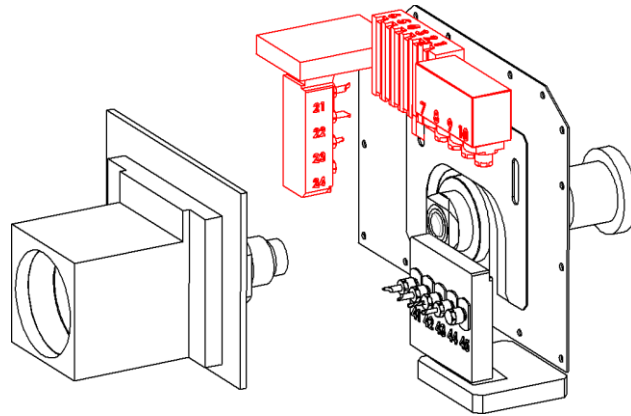


Order of Operations

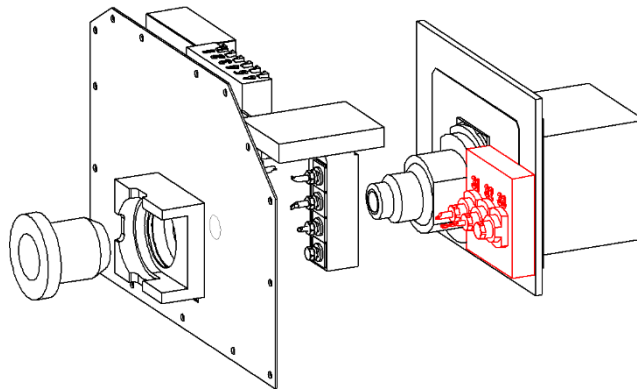
Next, we will order the operations in the correct order for Swiss turn machining.

For this machine, the best order for the operations will be the following:

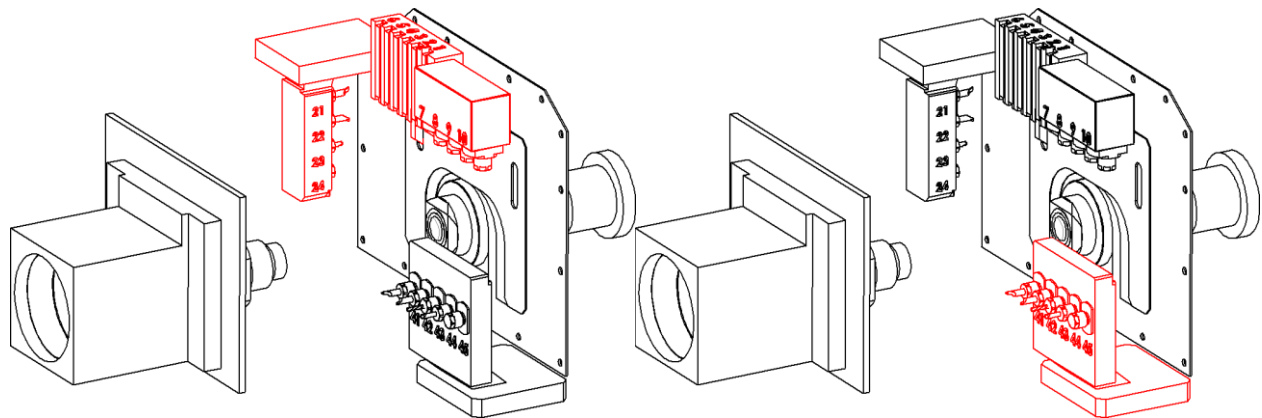
- Facing operations using the tools on the Gang Slide.



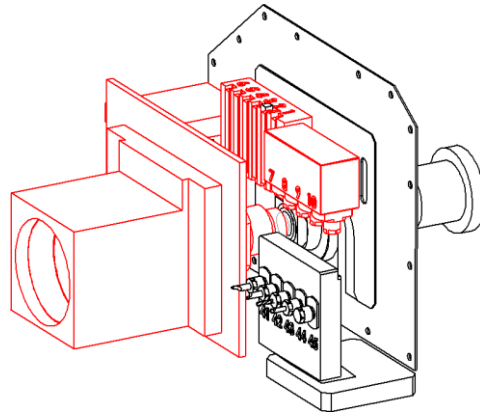
- Inside diameter operations with tools from the End working tool post.



- Outside diameter operations synced with the operations on the sub spindle done with the back working tool post.



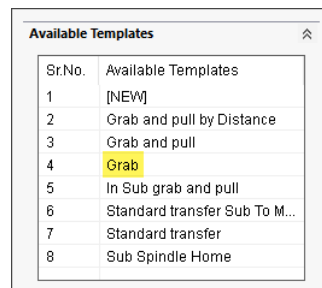
- Sub spindle transfer/cutoff.



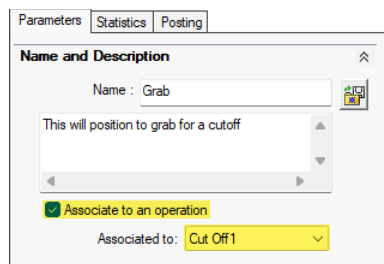
We will start this process by adding a **Sub Spindle Operation** to the Cut Off operation. Then we will use the CAMWorks Sync Manager to put the operations in the order described above.

15. Add Sub Spindle Operation.

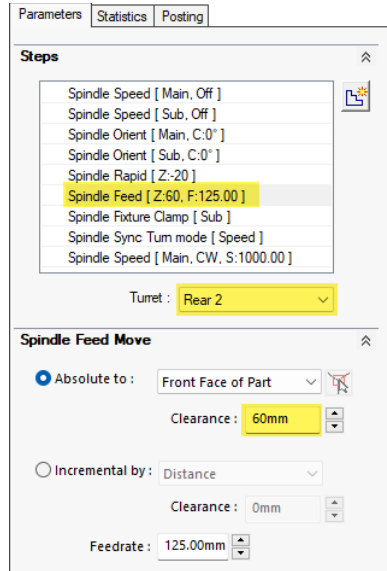
- a. Right-click on the **Cut Off1** operation and select **SubSpindle Operation**.
- b. In the **New Sub Spindle Operations** dialogue, under the **Available Templates**, select **Grab**.



- c. Click **OK**.
- d. Under the **Name and Description** section, click on the **Associate to an operation** checkbox and select **Cut Off1** from the **Associated to** dropdown menu.



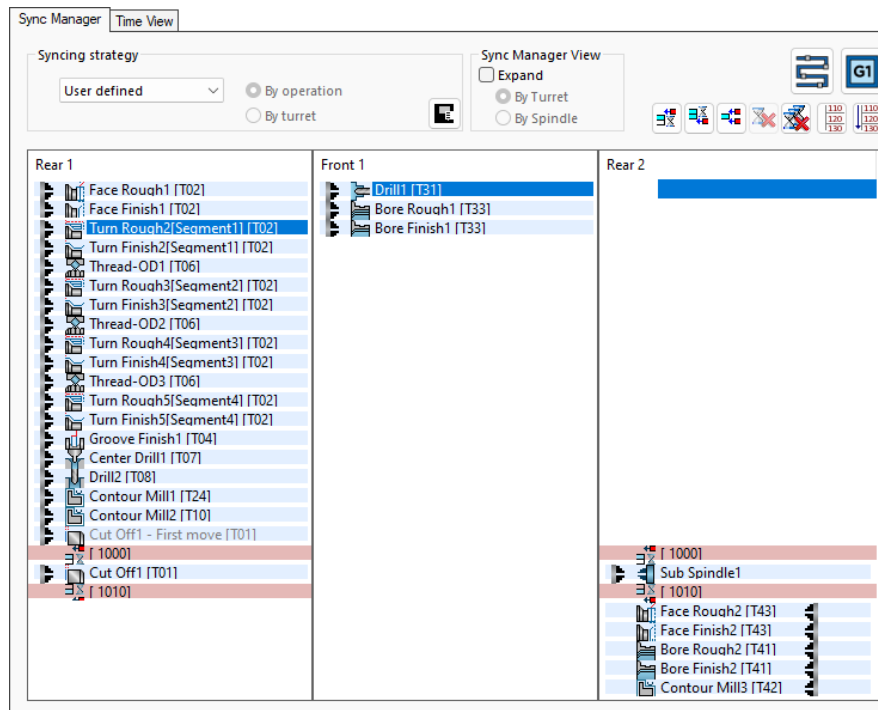
- e. Under the **Steps** section, select on the **Spindle Feed** step.
- f. Under the **Spindle Feed Move** section, set the clearance to **60mm**.
- g. Set the **Turret** to **Rear2**.



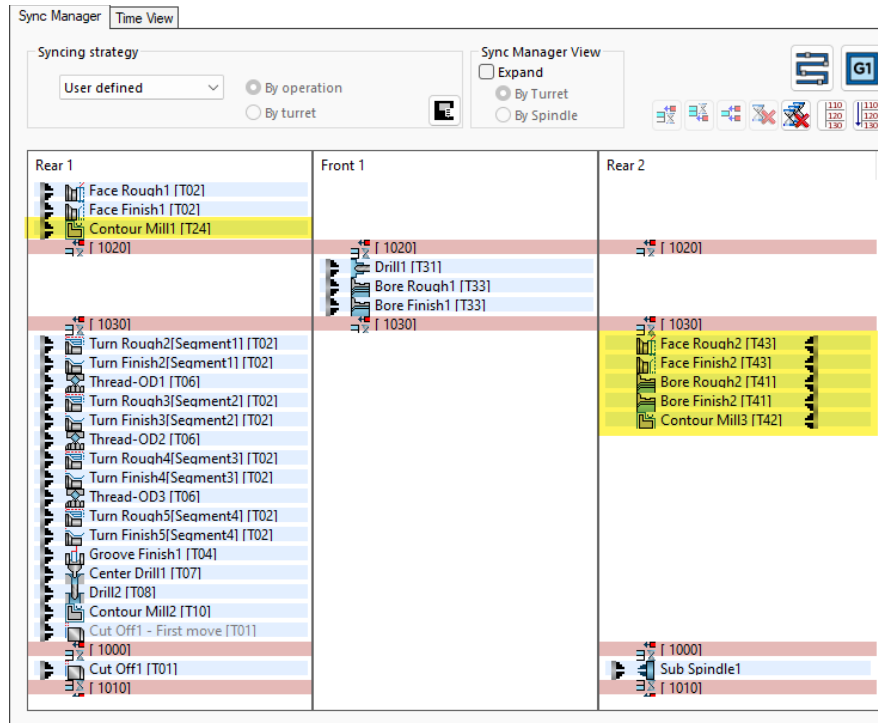
h. Click **OK**.

16. Launch the Sync Manager.

- Launch the CAMWorks Sync Manager.
- Under the **Rear1** column, select **Turn Rough2[Segment1][T02]**, Under the **Front 1** column, select **Drill1[T31]** and select the top cell in the **Rear 2** column.



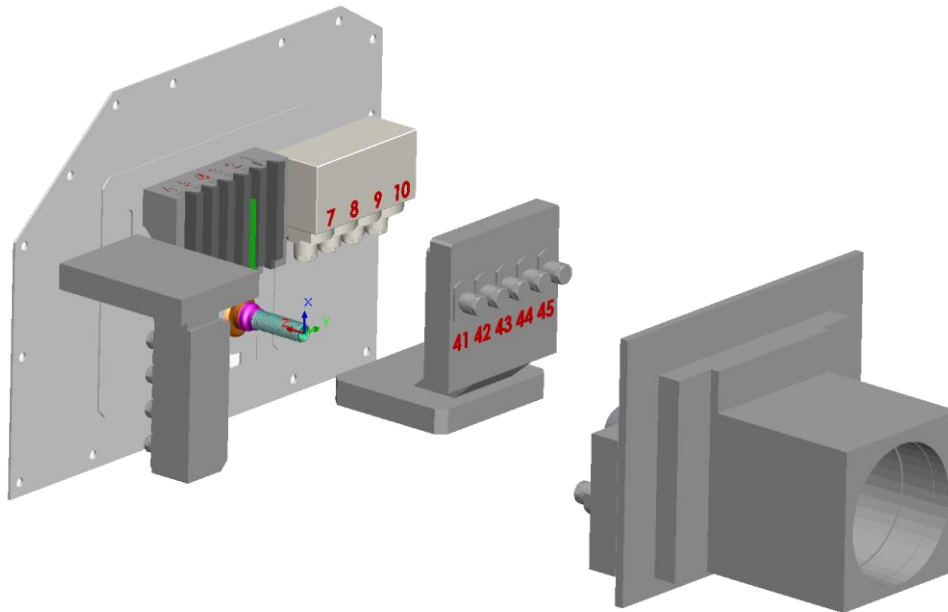
- Click on the **Insert wait code before the selected operation** button.



h. Click **OK**.

17. Simulate toolpath.

- Activate the **Display Components** toolbar from the CAMWorks CommandManager toolbar.
- Run the toolpath simulation.



18. Post process the G-Code.

Aligned_LESSON_03__H1	Aligned_LESSON_03__H2
%	%
O0001	O0002
(PART : LESSON_03__ MAIN)	(PART : LESSON_03__ SUB)
(DATE : 25-8-2025)	(DATE : 25-8-2025)
(TIME : 05:39:43)	(TIME : 05:39:43)
(-MAIN-SPINDLE-PROGRAM-)	(-SUB-SPINDLE-PROGRAM-)
(-COLLET-RECHUCK-)	(-COLLET-RECHUCK-)
(..... TOOL LIST)	(..... TOOL LIST)
(T0101 - CTPS15FR HOLDER)	(T4141 - VCMT 070204 BORE BAR)
(T0101 - CTPS15FR HOLDER)	(T4242 - 4MM CRB 2FL 14 LOC)
(T0202 - DCGT 11T304 TURN HOLDER)	(T4343 - VCMT 070204 BORE BAR)
(T0404 - GTMX32150 T01 OD HOLDER)	
(T0606 - CSV11FRP60-035A TURN HOLDER)	G0 T0
(T0707 - 3MM X 60DEG HSS CENTERDRILL)	G28 W0
(T0808 - 2.5MM SCREW MACH DRILL)	G99 G97 G80 G40 G18
(T1010 - 4MM CRB 4FL 8 LOC)	M5 (SUB SPINDLE STOP)
(T2424 - 3MM CRB 4FL 12 LOC)	T2000 (SUB HOME + ALIGN WITH MAIN)
(T3131 - 6.0MM JOBBER DRILL)	
(T3333 - VCMT 070204 BORE BAR)	
(..... CUTOFF PARAMETERS)	
#500 =01 (CUTOFF TOOL NUMBER)	
#501 =12. (CUTOFF SHANK WIDTH)	
#502 =1.5 (CUTOFF INSERT WIDTH)	
#503 =1.45 (CUTOFF LAST_X)	
#504 =-76.75 (CUTOFF LAST_Z)	
#505 =3593 (CUTOFF SPEED RPM)	
#506 =316 (CUTOFF SPEED SFPM)	

Lesson 4 – G-Code Simulation with CAMWorks Virtual Machine

Upon successful completion of this lesson, you will be able to:

- Run CAMWorks Virtual Machine G-Code simulation to verify the toolpath.

CAMWorks Virtual Machine

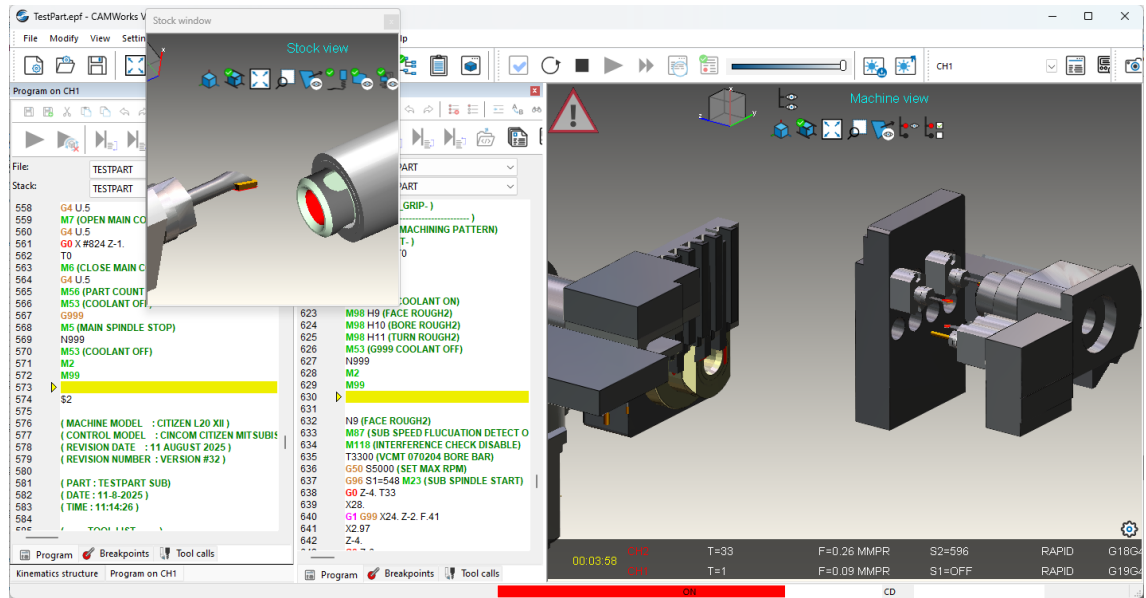
CAMWorks Virtual Machine is a machine simulation application that's part of the CAMWorks software suite.

The application supports various machine types for simulation, including Mill, Turn, Mill-Turn, and Swiss Turn machines. To simulate a machine, you need to select the corresponding Simulation Machine template file and Controller emulation template file. These are specified in the CAMWorks Options and Machine dialog boxes. To launch the simulation, you must generate G-code and ensure the **Run Machine Simulation** checkbox is selected in the Post Process dialog box.

CAMWorks Virtual Machine is highly beneficial for Swiss Turn programming because it provides a realistic, digital simulation of the machining process before you ever cut a part. This is especially crucial for Swiss machines due to their complexity, which involves multiple tools, turrets, and spindles working simultaneously.

The Key Benefits for Swiss Turn Programming are:

- **True G-code Simulation:** The software simulates the actual G-code that will run on the CNC machine, not just a generic toolpath. This is a vital difference because it ensures the program is precisely validated, eliminating the need for time-consuming and costly dry runs on the physical machine.
- **Collision Detection and Avoidance:** Swiss machines often perform simultaneous milling and turning operations with multiple tools and spindles. This increases the risk of collisions between the tool, tool holder, chucks, fixtures, and the part itself. CAMWorks Virtual Machine provides full collision detection, allowing you to identify and fix these potential errors in a virtual environment, thereby preventing expensive machine damage, broken tools, and scrapped parts.
- **Reduced Setup and Cycle Time:** By proving out the program in a virtual environment, you can significantly reduce the amount of time spent on machine setup and verification. Additionally, the software's synchronization manager helps you optimize the timing of simultaneous operations between multiple spindles and turrets, leading to a shorter overall cycle time.
- **Support for Complex Machines:** The software fully supports complex multi-tasking machines like Swiss turns, including 4-axis dual-turret, dual-spindle lathes, and multi-axis mill-turn machines with upper milling heads and sub-spindles. This comprehensive support ensures that even the most intricate Swiss machine configurations can be accurately simulated and programmed.



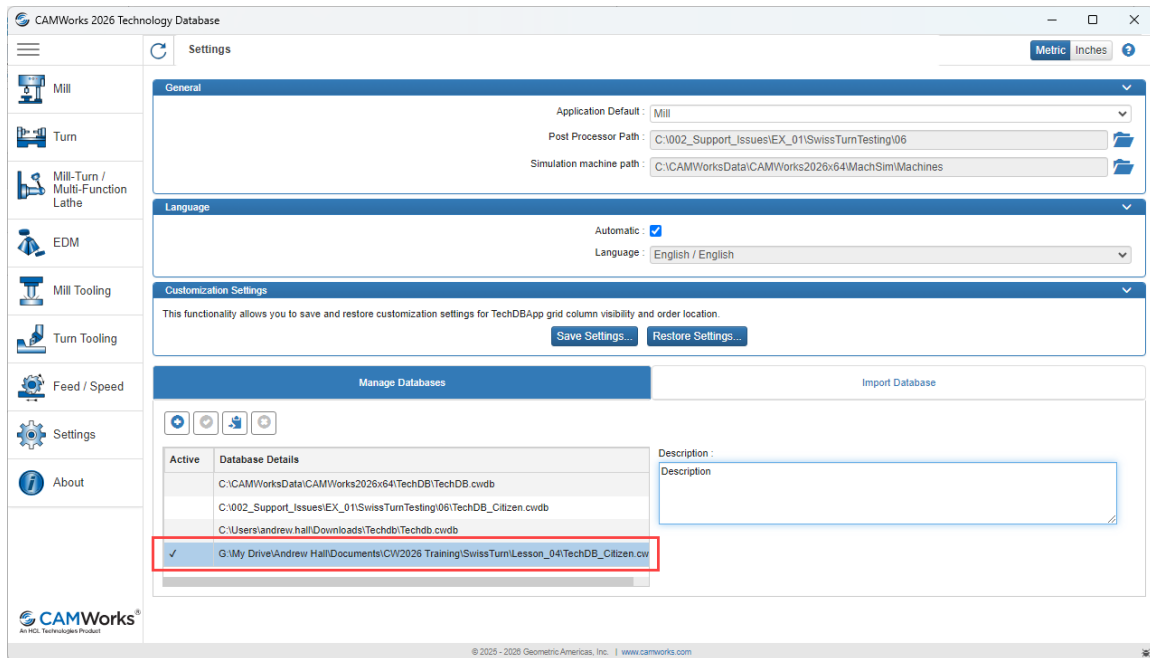
Case Study – G-Code Simulation with CAMWorks Virtual Machine

In this lesson we will use CAMWorks Virtual Machine to validate G-Code on a previously programmed part.

We will start by setting up CAMWorks to be able to use a custom machine.

1. Add custom Technology Database

- Launch the TechDB Application from the Windows Start Menu with SOLIDWORKS closed.
- Go to **Settings** and under **Manage Database**, add the **TechDB_Citizen.cwdb** from the **...Lesson_04** folder from the training files.
- Set this database as the active database.



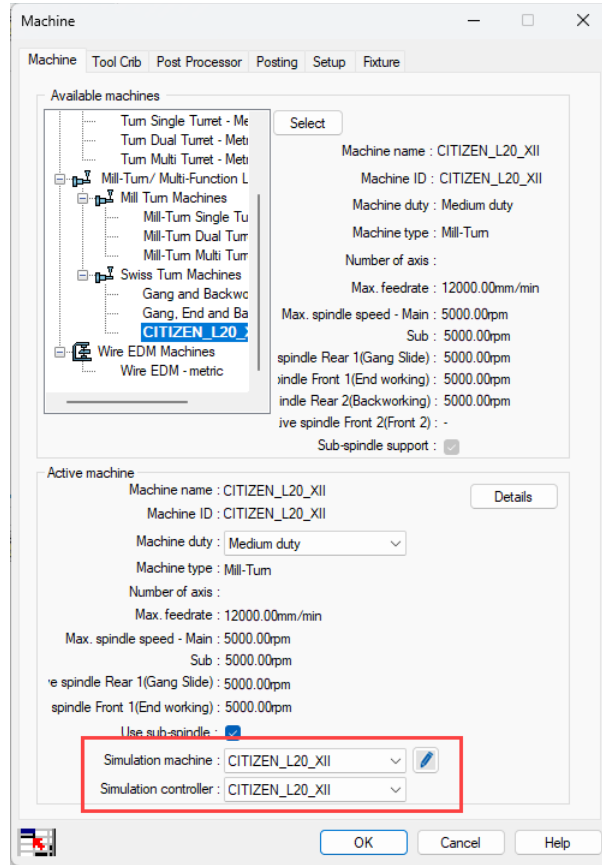
2. Copy files for CAMWorks Virtual Machine

- Copy the folder **...Lesson_04\CITIZEN_L20_XII** to **C:\CAMWorksData\CAMWorks2026x64\MachSim\Machines\MillTurn**.

3. Open part

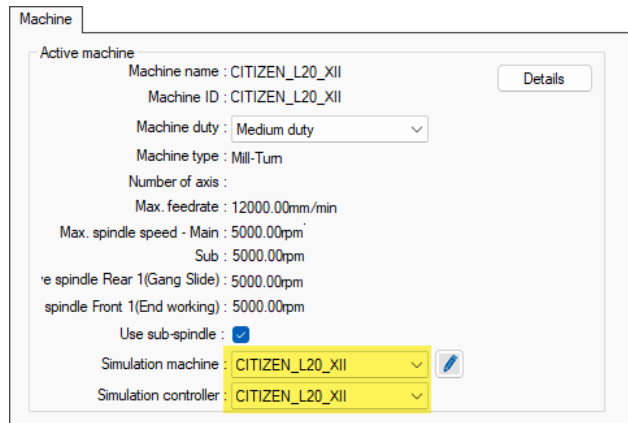
- Launch the SOLIDWORKS application.
- Open the part **LESSON_04.SLDPR**T from the **...Lesson_04** directory.

To ensure the proper machine is selected for this part, we can edit the machine definition and set the Simulation machine. This also sets the machine display for machine aware programming.



4. Assign Virtual Machine

- Go to the CAMWorks Operation Tree.
- Edit the **Machine Definition**.
- On the **Machine** tab, ensure that the **CITIZEN_L20_XII** machine is selected as the **Simulation machine**.



5. Assign Custom Post processor

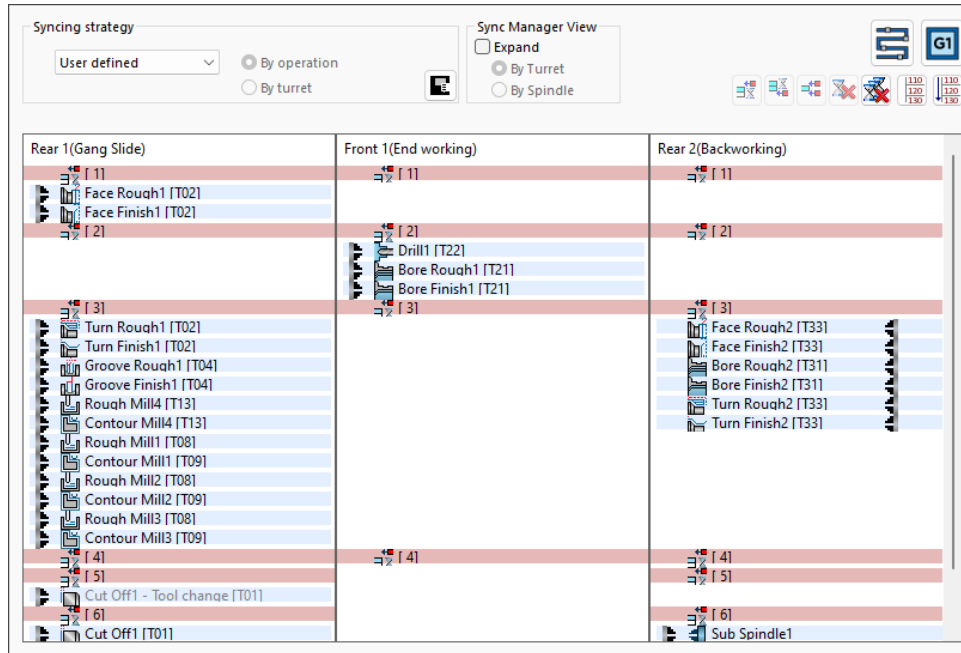
- Go to the **Post Processor** tab

- b. Click **Browse** and browse to the *CITIZEN_L20_XII_R2_R2_V33.cdl* post processor file in the **...Lesson_04** folder.
- c. Click **OK**.

6. Examine Part programming

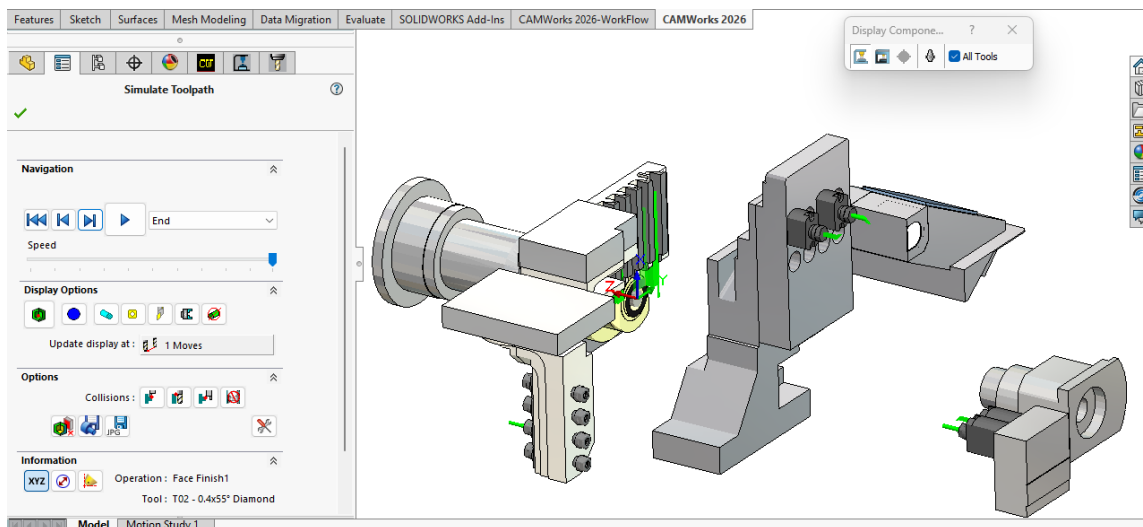
- a. Examine the part programming.
- b. Launch the CAMWorks Sync Manager to see how the part is programmed.

The part starts with turn facing operations on the Rear Gang Slide. Then the ID work is done with the end working tools before the rest of the turning operations on the main spindle.



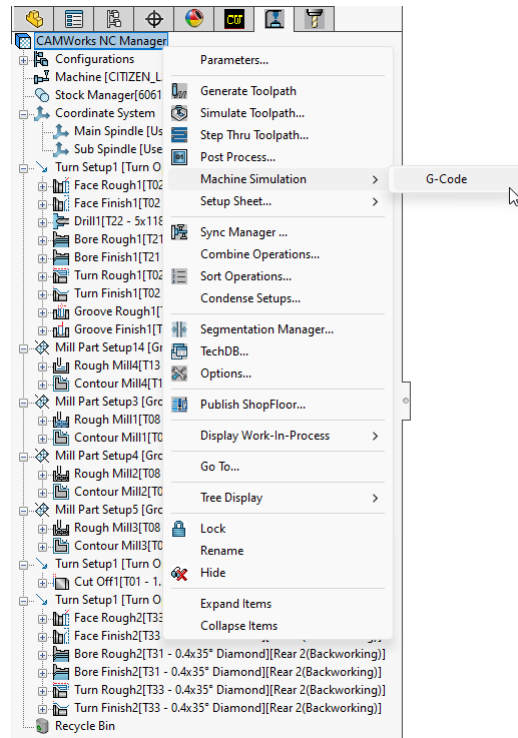
7. Toolpath Simulation

- a. Run Toolpath Simulation

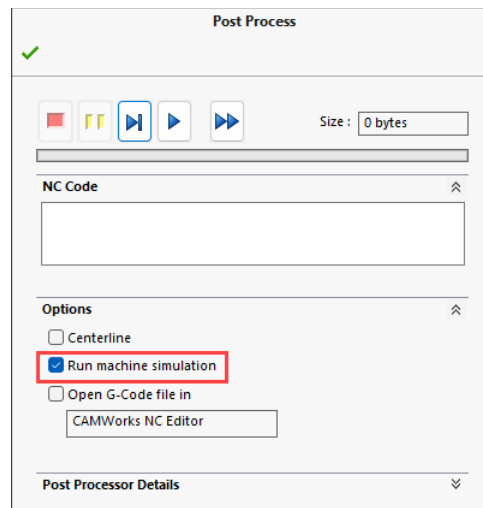


- b. Click **OK**.

CAMWorks Virtual Machine can be run from the Right-Click menu in the CAMWorks Operation tree. We can simulate individual setups, operations, or the entire part depending on which node we right-click on. Typically for Swiss Turning machine simulation, we want to simulate the entire G-code so this will be done from the Right-Click menu from the CAMWorks NC Manager node.

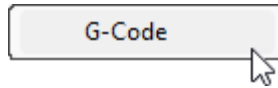


Alternatively, CAMWorks Virtual Machine can also be run from the Post Processing dialog box. Under the Options section there is a checkbox for **Run machine Simulation**. After the G-Code is posted, CAMWorks Virtual Machine will launch.



8. Run CAMWorks Virtual Machine

- a. **Right-Click** on the **CAMWorks NC Manager** and select **Machine Simulation, G-Code**.



- b. Save the **Post Output File** and click **Save**.
 c. Post the G-Code.
 d. Run the Simulation in CAMWorks Virtual Machine.

