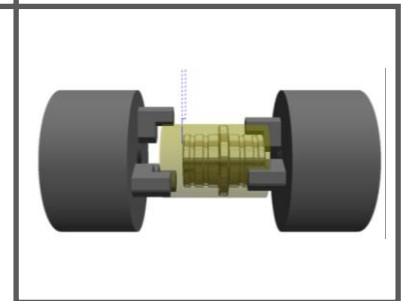
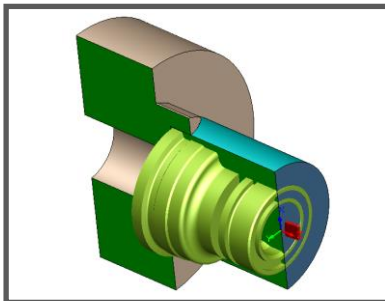
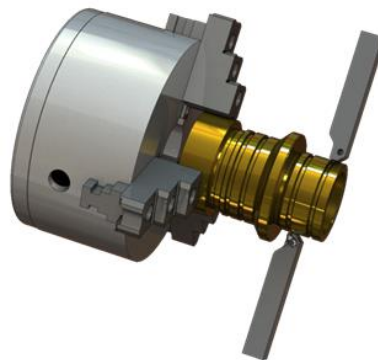
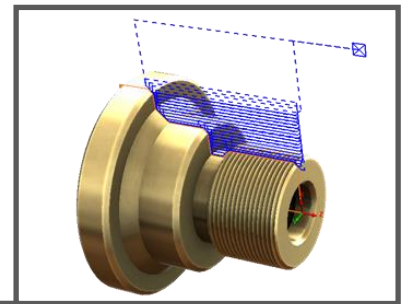
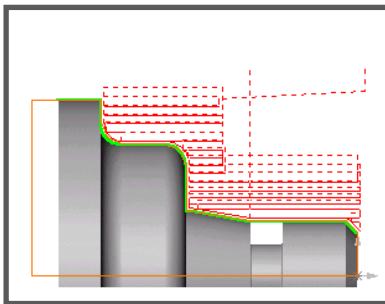




CAMWorks



Turn Tutorial



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.

SOLIDWORKS® is a registered trademark of Dassault Systèmes SOLIDWORKS Corp.

FeatureManager™ is a trademark owned jointly by DS SOLIDWORKS and Bentley Systems, Inc.

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



Copyright ©1998-2021 Geometric Americas, Inc. All Rights Reserved.





Table of Contents

CHAPTER 1: LEARNING 2 AXIS TURN	7
Turn 1	8
Steps to Generate Turn Toolpaths and NC Code	8
Step 1: Model Part in SOLIDWORKS/CAMWorks Solids or Import Part	9
Viewing the FeatureManager Design Trees	10
Step 2: Change to CAMWorks Feature Tree	10
CAMWorks Machining Trees	11
CAMWorks Menu and CAMWorks NC Manager	12
CAMWorks Command Manager	12
CAMWorks Workflow Toolbar	13
CAMWorks Options	13
Step 3: Define the Machine	14
Step 4: Edit the Stock Definition	20
Step 5: Define Machinable Features	21
Using Automatic Feature Recognition (AFR)	22
Defining Machinable Features Automatically	22
Defining Features Interactively (IFR)	25
Step 6: Generate Operation Plan and Adjust Operation Parameters	28
Step 7: Generate Toolpaths	31
Step 8: Simulate Toolpaths	32
Change the Machining Order	34
Step 9: Post Process Toolpaths	35
Setup Sheets	37
Turn 2	38
Turn Feature Types Recognized by AFR and IFR	38
AFR & IFR	38
Supported Turn Feature Types	38
Step 1: Open the Part	39
Step 2: Defining the Machine and Editing the Stock Definition	39
Define the Machine	39
Editing the Tool Crib	40
Selecting the Post Processor	44
Editing the Stock Definition	44
Step 3: Extracting Machinable Features and Editing Feature Parameters	45
Extracting Machinable Features	45
Feature Strategy	46
Step 4: Generating Operations	48
Step 5: Generating NC Code	50



Turn 3	51
Step 1: Open the Part, Define the Machine, Edit the Stock Definition and Extract Machinable Features.....	51
Define the Machine Parameters	51
Editing the Stock Definition	52
Extract machinable Features	52
Step 2: Interactively Inserting an OD Profile for Threading.....	52
Step 3: Reorganizing Machinable Features.....	54
Step 4: Suppressing Machinable Features.....	54
Step 5: Saving the File.....	55
Step 6: Generating an Operation Plan and Toolpaths	55
CAMWorks Message Window	56
Step 7: Simulate Toolpaths.....	56
Viewing XZ cutter coordinates of the Tool in CAMWorks Message Window	57
Step 8: Post Processing Toolpaths.....	58
Turn 4	60
Step 1: Open the Part and Define the Machine	60
Step 2: Defining the Stock as a Forging or Casting	60
Step 3: Recognizing Machinable Features using AFR.....	61
Deleting Unwanted Features Recognized by AFR	62
Step 4: Defining a Rectangular OD Groove Feature Interactively	62
Step 5: Deleting an Operation	64
Step 6: Adjusting Operation Parameters	64
Step 7: Displaying the Chuck in Graphics Area.....	65
Step 8: Defining Program Zero	65
Step 9: Simulate Toolpath	68
Step 9: Post Processing Toolpaths.....	68
Turn 5	69
Step 1: Opening the Part and Defining the Machine and Stock.....	69
Open the Part.....	69
Define the Machine Parameters	69
Defining the Stock	70
Step 2: Extract Machinable Features.....	70
Deleting Features	71
Step 3: Adding an ID Groove Feature Interactively	71
Step 4: Adding OD Thread and ID Thread Features Interactively	72
Interactively Inserting a Threaded OD Feature	73
Interactively Inserting a Threaded ID Feature	75
Step 5: Changing Feature Parameters, Generating Operations and Modifying Operation Parameters	77
Changing Feature Parameters and Renaming Features	77
Generating Operations	77
Modifying Operation Parameters.....	78
Step 6: Defining the Machining Sequence and Generating Toolpaths	80
Step 7: Displaying the Chuck.....	80



Step 8: Defining the Chuck	81
Step 9: Defining the Chuck Location	83
Step 10: Simulate Toolpath	85
Step 11: Post Processing Toolpaths.....	86
Turn 6	87
Step 1: Opening the Part.....	87
Step 2: Defining Machine Parameters.....	87
Step 3: Establishing Part Zero	88
Step 4: Defining the Stock from a Sketch for Double Chucking	91
Step 5: Defining Machinable Features	91
Changing Feature Definitions for OD and ID Profiles	92
Step 6: Changing the Origin Machining Direction	94
Step 7: Generating Operations and Editing Operation Parameters	96
Step 8: Defining the Chuck Configuration.....	96
Step 9: Setting the Chuck Display State.....	97
Step 10: Setting the Chuck Location	99
Step 11: Simulating the Toolpaths for Double Chucking	101
Step 12: Post Processing Toolpaths.....	102
Turn 7	103
Step 1: Opening the Part and Defining the Machining Parameters.....	104
Defining Machining Parameters	104
Step 2: Extracting Machinable Features using Plane Section Method.....	104
Setting the Method for AFR	104
Extracting Machinable Features	105
Step 3: Using the Plane Section to Extract Machinable Features Correctly	105
Viewing the Feature Relative to the Standard Orientation	107
Turn 8	109
Step 1: Open the Part and Defining the Machine Parameters	109
Step 2: Defining a Thread Feature	109
Step 3: Cutting Multiple Start Threads.....	112
Step 4: Enabling the 'Process by Level' Option for a Threading Operation	113
Step 5: Step Through Toolpath	113
Step 6: Simulating the Threading Toolpath	114
CHAPTER 2: LEARNING 4 AXIS TURN	116
4 Axis Turn 1.....	117
Step 1: Opening the Part.....	117
Step 2: Define the Machine Parameters for 4 Axis Machining.....	118
Step 3: Defining the Stock.....	119
Step 4: Defining Features Automatically and Interactively	120
Step 5: Editing Machinable Features.....	121
Extends of the OD feature.....	121
Extends of the ID feature	122
Interactively inserting Face Feature and OD Feature in Turn Setup2	123



Step 6: Generating Operations and Toolpaths.....	124
Step 7: Defining the Chuck Location for Turn Setup1	127
Disabling Chuck Display in Graphics Area.....	127
Defining the Geometry of the chuck.....	128
Step 8: Defining the Chuck Location for Turn Setup2	129
Step 9: Simulating the Toolpaths for Multiple Turn Setups	132
4 Axis Turn 2.....	134
Step 1: Opening the Part and Defining Machining Parameters	134
Activating the Use sub-spindle option.....	135
Step 2: Defining Features Automatically	136
Step 3: Changing Feature Definitions for OD and ID Profiles	136
Step 4: Generating the Operation Plan	140
Step 5: Enabling the Chuck Display option	141
Step 6: Defining the Chuck Configuration for a Twin Spindle Setup	141
Setting the Chuck Location for Turn Setup machined from Main spindle.....	141
Setting the Chuck Location for Turn Setup machined from the Sub Spindle	142
Step 7: Adjusting the Parameter and Generating the Toolpaths	143
Step 8: Simulating the Toolpaths for Turn Setups	144



Chapter 1: Learning 2 Axis Turn

This chapter provides an opportunity to learn CAMWorks 2 Axis Turn through a step by step hands-on tour of the features and functions.

The tutorials in this chapter are intended to show you how to use CAMWorks and may not correspond to actual machining practices.

The exercise parts are installed when you install CAMWorks and are in the *\Examples\Tutorial_Parts\Turn* folder. This folder is inside the CAMWorks data folder.

Typical location: Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Turn).

IMPORTANT! CAMWorks uses a set of knowledge-based rules to assign machining operations to features. The Technology Database contains the data for the machining process plans and can be customized for your facility's machining methodology. When you do these exercises, your results may not be exactly the same as described in the steps and illustrated in the figures. This is because the machining sequences and operations data in your Technology Database may be different from the database used to produce the documentation.

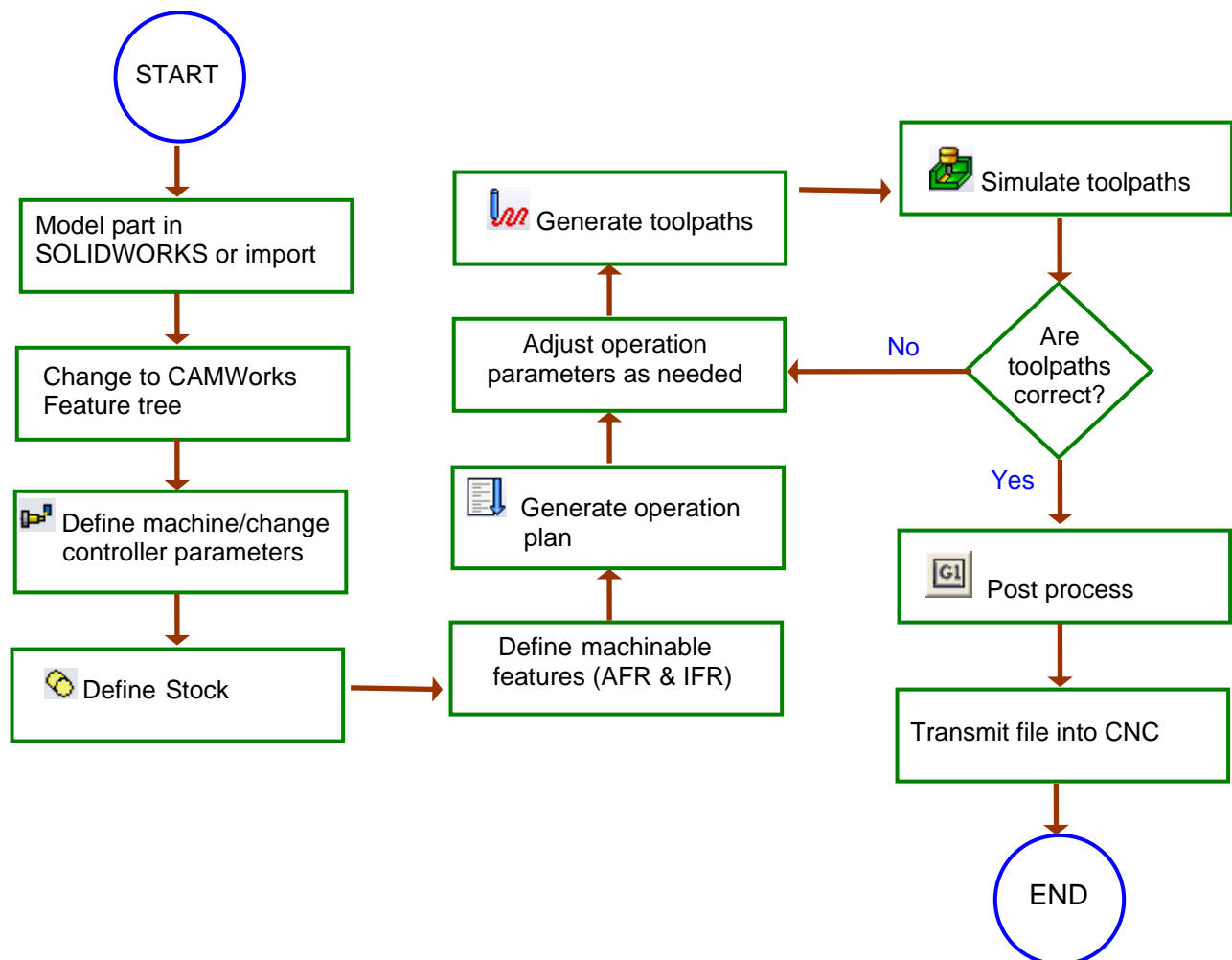


Turn 1

Steps to Generate Turn Toolpaths and NC Code

The following steps are used to generate Turn toolpaths and NC code:

1. Model the part or open the part file in SOLIDWORKS/CAMWorks Solids.
2. Click on the CAMWorks Feature tree.
3. Define the Machine and modify the post processor parameters.
4. Edit the Stock definition.
5. Define machinable features and adjust feature parameters.
6. Generate the operation plan and adjust operation parameters.
7. Generate toolpaths.
8. Simulate Material Removal.
9. Post process the toolpaths.





The following series of tutorials show you how to generate finish toolpaths on a SOLIDWORKS/CAMWorks Solids part model. In order to give you a general understanding of how to use CAMWorks, you work with a part that was previously modeled in SOLIDWORKS. When you define the operations and toolpaths, you will follow steps that are not explained in depth. This is done to show you the basics of generating toolpaths from start to finish without getting into the details at this time.

Sample parts are provided for the tutorials in this manual. When you install CAMWorks, these files are installed automatically.

Topics covered in this tutorial:

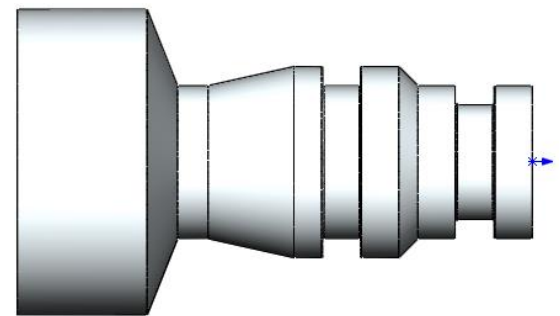
- Using the [CAMWorks Command Manager](#) or [Workflow toolbar](#) to execute commands
- [Editing the Machine Definition](#)
- [Editing the Stock Definition](#)
- Changing Default Settings using [Options dialog box](#)
- Using the CAMWorks [Feature Tree](#) and [Operation Tree](#)
- [Defining Machinable Features Automatically](#)
- [Interactively inserting an OD Feature](#)
- [Generating Toolpaths](#)
- [Simulate Toolpaths](#)
- [Post Process Toolpaths](#)

Step 1: Model Part in SOLIDWORKS/CAMWorks Solids or Import Part

A part is a solid that is created with SOLIDWORKS/CAMWorks Solids or imported into SOLIDWORKS/CAMWorks Solids from another CAD system via an IGES, Parasolid, SAT file, etc. This tutorial uses an existing SOLIDWORKS part.

Open the part file **TURN2AX_1.SLDPRT** in the following folder.


*Drive:\CAMWorksData\CAMWorks202x\Examples\
Tutorial_Parts\Turn*







TURN2AX_1.SLDPRT



Viewing the FeatureManager Design Trees

The FeatureManager design tree  displays the list of the features, sketches, planes and axes related to the part.


To use CAMWorks, you need to move between SOLIDWORKS/CAMWorks Solids trees and the CAMWorks Feature trees. Different tabs are provided to access the SOLIDWORKS/CAMWorks Solids trees and the CAMWorks Feature trees. Click the  *Pin* button to continuously view this Tree area.

If the CAMWorks tabs , ,  are not visible, you can expand the size of the tree. Position the cursor on the line that divides the tree area from the graphics area. When the cursor changes to a bar, drag the bar to the right until the tabs display.

Step 2: Change to CAMWorks Feature Tree

 Click the *CAMWorks Feature Tree* tab.

When the CAMWorks Feature tree is displayed, it initially lists *Configurations*, *Stock Manager*, *Machine* and *Recycle Bin* items.

 The icons that display for the 'Stock Manager' and 'Machine' are indicative of the machine which is currently selected. [Step 3 of this tutorial](#) explains how to select a turn machine.





CAMWorks
Feature Tree

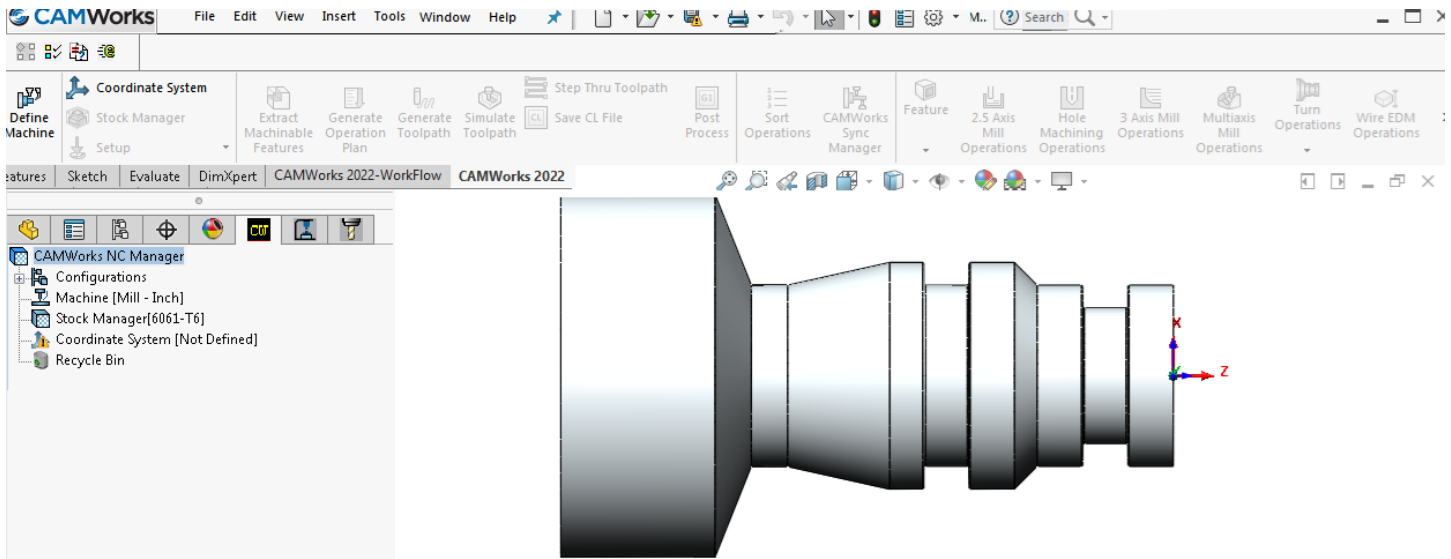
If you work mainly with turn parts, you can set the default to *Turn* in the TechDB so that CAMWorks will select a Turn machine and stock whenever you open part files.

Changing default machine in TechDB

To change the default machine in the TechDB:



Click the *TechDB* button  on the CAMWorks Command Manager. Alternatively, click on the CAMWorks menu and select 'Technology Database' from the dropdown list. Once

the Technology Database Application is launched, click on the *Settings menu*  on the left hand side of the Technology Database application, the user interface for *settings* will be displayed. Observe that the *Application Default* dropdown list is displayed in the *General tab*. Change the Application Default to *Turn* from the dropdown list. and then click *Cancel* . When you open a part file, CAMWorks will select a Turn machine by default.



CAMWorks Machining Trees

The CAMWorks machining trees provide an outline view of the machining information for the model. Initially, the CAMWorks Feature tree shows only the NC Manager, Configurations, Stock Manager, Machine and Recycle Bin items. As you follow the steps to generate an NC program, this tree expands to include Turn Setups and machinable features. The tabs are for moving between the SOLIDWORKS/CAMWorks Solids trees and the CAMWorks trees.

-  **Configurations**
Multiple CAMWorks datasets are supported. Each dataset is called a configuration. You can use configurations to support multiple machines and SOLIDWORKS configurations.
-  **Stock Manager**
The turn stock is the material from which the part will be machined. You can define the stock as a cylinder (for bar stock) or as a closed sketch (for a forging or casting) and specify the type of material.



- **Machine**

Turn
 Mill
 Mill-Turn
 Wire EDM

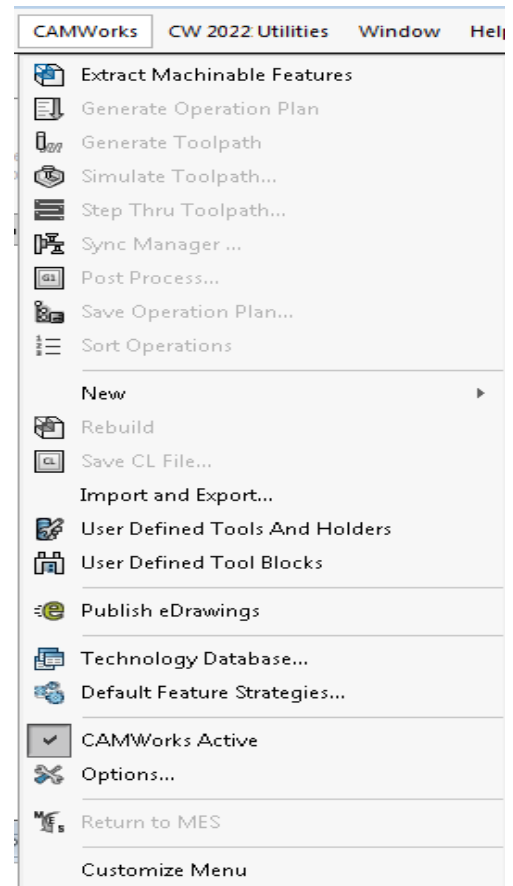
The Machine item defines the machine tool that will be used to machine the part. The machine definition includes tool definitions and the post processor. These machines are set up in the Technology Database.

- **Recycle Bin**

The Recycle Bin is used to store machinable features that you do not intend to machine.

CAMWorks Menu and CAMWorks NC Manager

1. Click *Tools* on the SOLIDWORKS menu bar and select *CAMWorks* from the dropdown menu./Click *CAMWorks* on the *CAMWorks Solids* menu bar.
2. The CAMWorks NC Manager item is present in both the Feature tree and the Operation tree. Right click on the *CAMWorks NC Manager* in the tree. A list of executable commands is displayed on the context menu. These right-click context menus provide access to a variety of commands. The commands displayed on the context menu for *CAMWorks NC Manager* item in the Feature tree is different from that in the Operation tree.
3. Refer the CAMWorks context-based Help for detailed explanation of the CAMWorks commands and functionalities.



CAMWorks Menu

CAMWorks Command Manager

Click *Tools* on the SOLIDWORKS menu bar and select *CAMWorks* from the dropdown menu./Click *CAMWorks* on the *CAMWorks Solids* menu bar. This action displays the CAMWorks Command Manager. The CAMWorks Command Manager provides access to the main CAMWorks commands. The commands are explained in the CAMWorks context-based Help.



CAMWorks Command Manager

Customization of CAMWorks Command Manager

Command Manager is a context-sensitive toolbar that can be dynamically updated based on the toolbar you want to access. It provides access to the main CAMWorks commands found on the CAMWorks menu. By default, it has toolbars embedded in it.

The CAMWorks Command Manager can be customized. Right click anywhere on the CAMWorks Command Manager and select *Customize* from the RMB context menu. The *Customize* dialog box will be displayed. Use the *Toolbars*, *Commands*, *Menus*, *Keyboard shortcut*, *Mouse gesture* and *Options* tab of this dialog box to customize the Command Manager as per your requirements.


Alternative Access to CAMWorks Commands

All the commands executed from the CAMWorks Command Manager can also be alternatively accessed from the RMB context menu of the CAMWorks NC Manager. This is a context menu. To execute the command, right click on the CAMWorks NC Manager item in the tree and select the desired command from the RMB context menu. In addition to the Command Manager commands, this context menu also provides access to a variety of commands.

CAMWorks Workflow Toolbar

The CAMWorks Workflow toolbar contains all the major commands of CAMWorks in a separate toolbar. When executed in sequential order, they define the workflow in CAMWorks. The workflow includes, setting up of the machine, stock and subsequent features, operations and toolpaths.


CAMWorks Options

1.  Click the *Options* button on the CAMWorks Command Manager. OR
Click *Tools* on the SOLIDWORKS menu bar and select *CAMWorks* from the dropdown menu./Click *CAMWorks* on the *CAMWorks Solids* menu bar.



Options button on the CAMWorks Command Manager

The Options dialog box is displayed. This dialog box contains various tabs to customize settings and options related to Saving data, Turn Features, Display, Simulation, Update and File locations.

2. Click on each different tab in this dialog box and click the *Help* button. Each tab is explained in the context-based Help.
3.  To close the context-based Help, click the *Close* button in the upper right corner of the Help window to close the window.



- Click *OK/Cancel* to close the *Options* dialog box.

Step 3: Define the Machine

The machine definition specifies the type of machining that will be done for the current model and the associated machine tool control (post processor) for proper generation of the NC program. Based on the selected machine, the *Extract Machinable Features* command will recognize mill, turn or EDM features. The machine can be changed to produce NC code for an alternate machine tool of the same type. The icons that display in the tree identify the machine:



Turn



Mill



Mill-Turn



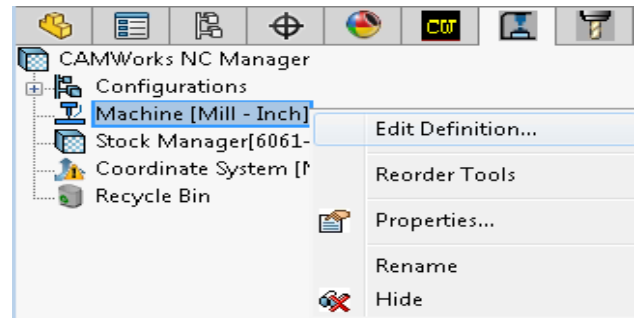
Wire EDM

- Right click the *Machine* item in the CAMWorks Feature tree and select the *Edit Definition* from the context menu.

OR

Double click the *Machine* item in the Feature tree to edit the machine definition.

The *Machine* dialog box is displayed.



Select 'Edit Definition' on the context menu



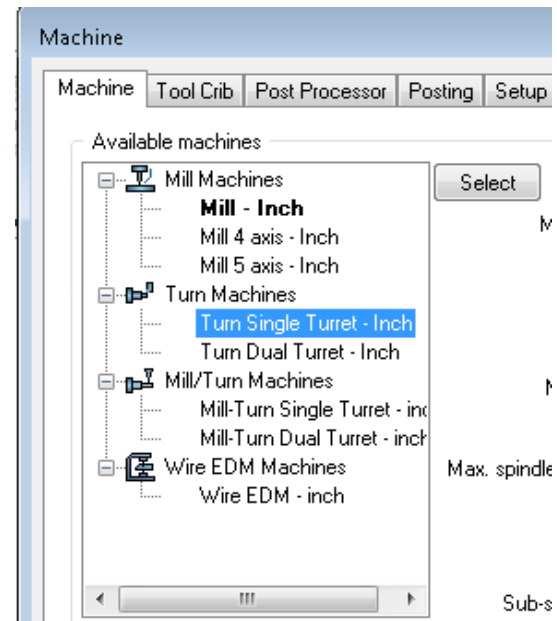
Did You Know ...

In the Feature and Operation trees, instead of right clicking items and selecting *Edit Definition* on the context menu, you can double-click the item to open the corresponding dialog box for editing the Stock Manager, Machine, Setups, Features and Operations.

- On the Machine tab, select *Turn Single Turret- inch* in the list of Available machines and click the *Select* button.

In this tutorial, the turn part will be machined using only the Main Spindle. The Sub spindle will not be used. Hence, ensure that the *Use sub-spindle* option at the bottom of this tab is unchecked.

Milling is the default machining type that is set when CAMWorks is first installed. The default machining type is specified in the Technology Database. This is the machine used for all the tutorials for Turn machining in this manual. When you use CAMWorks to machine your own parts, select the machine tool you want to use.



Selecting the Machine



Machines and the Machine tools are set up in the TechDB. Before using CAMWorks to machine your parts, make sure you define the machine and the machine tools available in your facility within the TechDB.



For details on customizing the TechDB to suit your facility's requirements, refer the manual 'Technology Database Tutorial'.

3. Click on the *Tool Crib* tab.
 - a. Make sure the *Tool crib priority* option is unchecked.
 - b. In the *Available tool cribs* list at the bottom of this tab, highlight *Tool Crib 2 Rear (Inch)* and click the *Select* button. This action switches the active tool crib for the part to the tool crib named *Tool Crib 2 Rear (Inch)*.

The name of the selected tool crib will be displayed below the *Select* button.



Machine

Machine Tool Crib Post Processor Posting Setup Chuck/Fixture

Tool crib

Active tool crib : Tool Crib 2 Rear (Inch) Turret : Rear Turret

Usage	Stn No.	Station ID	Type	ID	Comment	Insert
	1		Turn Tool	3	CNMG 431 80DEG SQR HOLDER	Diam
	2		Turn Tool	5	CNMG 431 80DEG BORE BAR	Diam
	3		Turn Tool	4	DNMG 431 55DEG SQR HOLDER	Diam
	4		Turn Tool	6	DNMG 431 55DEG BORE BAR	Diam
	5		Center Drill	5	#3 60DEG HSS CENTERDRILL	
	10		Turn Tool	9	0.118W CUT-OFF BLADE	Gro

Add... Remove Edit... Save...

☐ Tool crib has sub stations

☐ Tool crib priority

☐ Use tool crib tools only

Save Tool Crib...

Available tool cribs

Tool Crib 1 Rear (Inch) Empty
Tool Crib 1 Front (Inch) Empty
Tool Crib 2 Rear (Inch)
Tool Crib 2 Front (Inch)

Select

Name : Tool Crib 2 Rear (Inch)

No. of stations : 12

OK Cancel Help

Selecting the Tool Crib

The *Tool Crib* page allows you to choose the tool crib (set of tools) that is used with the machine you have chosen. These are not all the tools that are available but a subset that you can modify to represent the actual set of tools that the machine has loaded.

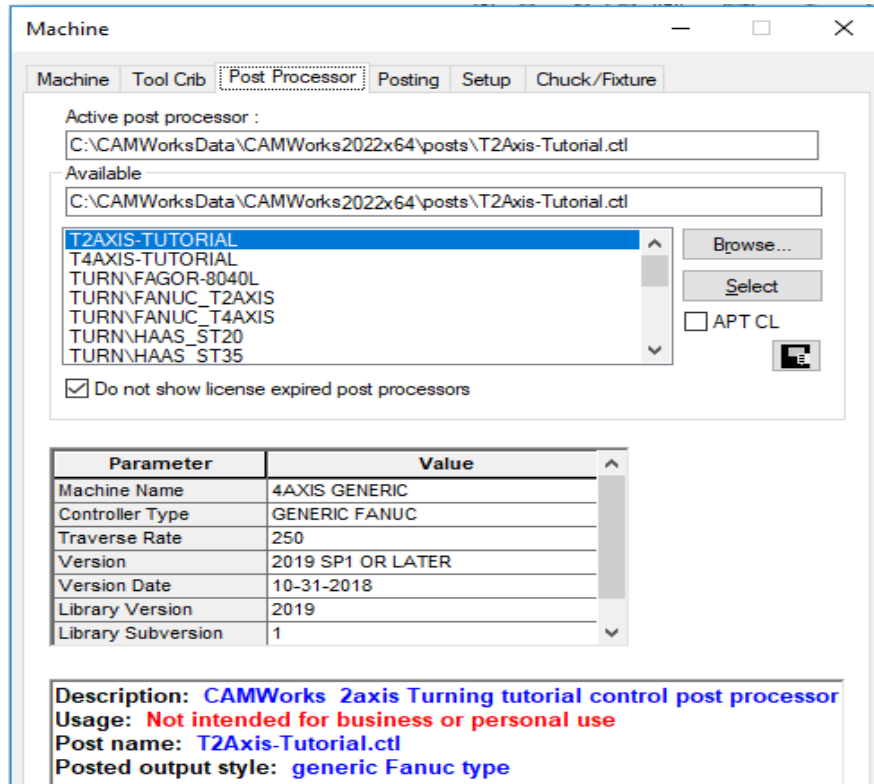
Tool Crib 2 Rear (Inch) is not the default tool crib that has been set up for the sample turn machine. As this tutorial makes use of this tool crib, you have to switch the tool crib to make it the active tool crib. When you define your machine tools in the Technology Database, you can set up your own tool cribs.

4. Click the *Post Processor* tab.



Highlight *T2AXIS-TUTORIAL* (the tutorial post processor) in the list and click the *Select* button. This action assigns *T2AXIS-TUTORIAL* as the Active post processor.

If the post processors do not display, use the *Browse* button to locate the folder containing the files (*.ctl). By default, tutorial post processors are located in the following folder. *Drive:\CAMWorksData\ CAMWorks201x\Posts*



Selecting the Post Processor

Information displays about *T2AXIS-TUTORIAL*. A short description displays in the window. This window contains information only if an optional file has been created for the post processor.

T2AXIS-TUTORIAL is the post processor used for the Turn machining related tutorials in this manual. When you use CAMWorks to machine your own parts, select your post processor.



Contact your CAMWorks reseller for more information on obtaining and/or customizing post processors for your machine tool.

5. Click the *More* button.

A longer description displays. The *More* button is activated only if a second optional file has been created. This information is intended for use in training or as a detailed description of post processor attributes that can be created.



Information files are provided for the sample post processor. Your CAMWorks reseller or your company manager may be able to supply these files if they are available for your post processor. If files are not available, you can create post information files as explained in the context-based Help.



6. Click the *Posting* tab.

The parameters in this tab provide information required to generate the NC program. The parameters are machine-dependent and different parameters may display for the selected post processor. The value for a parameter is output in the NC code if the machine requires it.

Parameter	Value
Program number	1001
Z Preset Rear Main	5.00000"
X Preset Rear Main	10.00000"
Z Preset Rear Sub	-5.00000"
X Preset Rear Sub	10.00000"

Posting tab: Displays information and parameter values related to generating NC code

If you have installed the Feed/Speed Library, you can specify a material on this tab or when you define the Stock. If the Feed/Speed Library is not installed, you can specify the material only on this tab.

The *Z Preset* and *X Preset* values are used by the system for a return position. These values are absolute numbers from X0, Z0. If your machine requires an absolute preset to be output in the program, these values are used. If you always change tools at the same position, setting these values saves you time later.

The *Maximum RPM* defines the maximum RPM that your machine allows when running Constant Surface Feed per Minute (CSFM). This prevents an over-speed spindle alarm as the tool moves to centerline. If necessary, you can set a lower maximum RPM for individual operations.

7. Type **1001** for the *Program Number*.



- Click on the *Chuck/Fixture* tab.

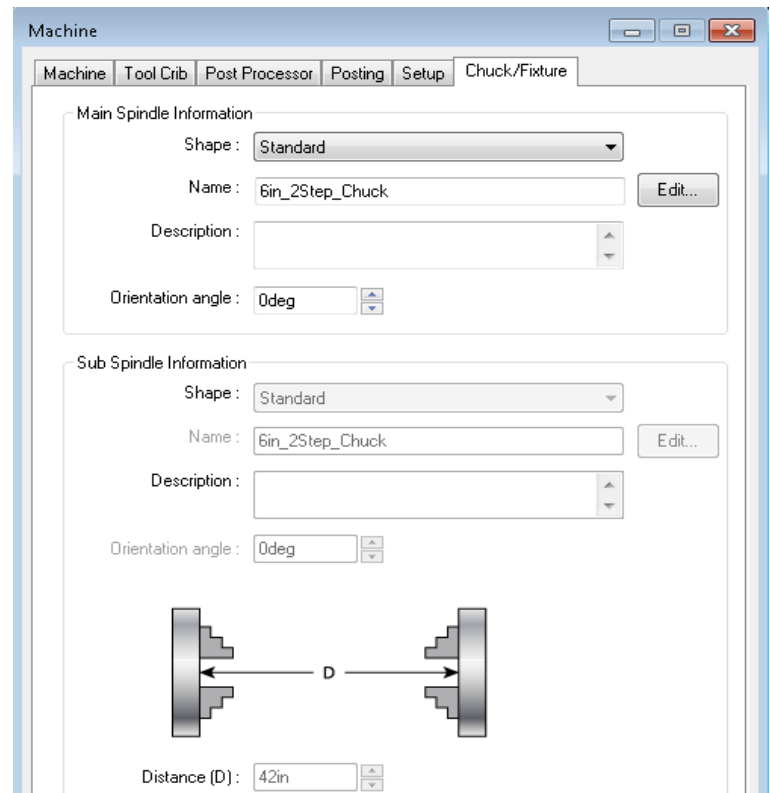
The Chuck tab allows you to define the Chuck geometry (size of the chuck base and the jaws) during toolpath simulation.

The chuck base and jaws can be viewed in the Graphics area of SOLIDWORKS/CAMWorks Solids when the *Chuck* Tab is selected within the *Machine* dialog box.

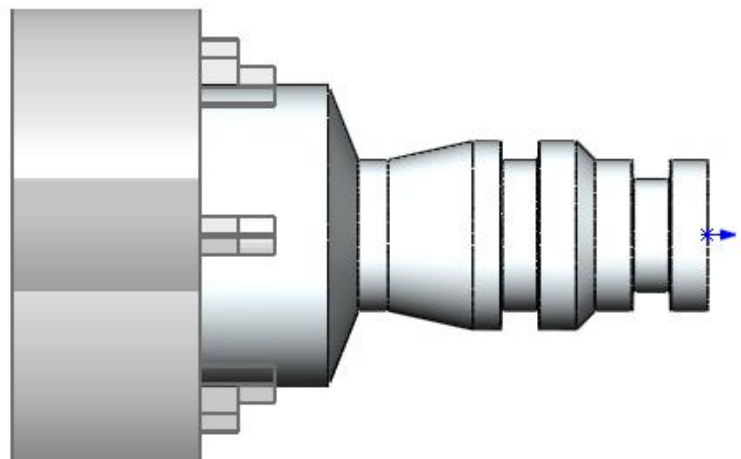
The *Main Spindle Information* and *Sub Spindle Information* group boxes display information related to the Main and Sub Spindle respectively. The *Sub Spindle Information* group box is enabled only when the *Use Sub Spindle* option in the *Machine* tab of the *Machine* dialog box is checked.

In this tutorial, the turn part will be machined using only the Main Spindle. Sub spindle will not be used.

- Click *OK* to close the *Machine* dialog box.
- Switch to CAMWorks Operation tree. Right-click on the *Machine [Turn Single Turret -inch]* node in this tree and select ***Chuck/Fixture Display>> Transparent*** from the context menu. Observe that the chuck/Fixture is displayed on the part in the SOLIDWORKS/CAMWorks Solids Graphics area.
- Switch to CAMWorks Feature tree.



Chuck/Fixture related information of the Main and Sub Spindle



Chuck/Fixture displayed on the part in the graphics area



To control the chuck display settings, right click on the **Machine** item in the **CAMWorks Operation** tree and choose the appropriate Display state within the Chuck Display cascading menu. The display states for the chuck include **Wireframe**, **Transparent**, **Shaded** and **Shaded with Edges**.

If you do not wish to view the chuck in the graphics area, right click on the **Machine** item in the tree and select the **Chuck Display** state as 'None'.

Step 4: Edit the Stock Definition

Stock is the material from which the part will be machined. The default Stock is the smallest cylinder (bar stock) that the part will fit into. Typically, this is not the size of the stock you will be using. You can change the Stock definition either by offsetting the length and/or diameter of the bar stock from the part or by defining the Stock from a closed sketch (for a casting or forging). Currently, the sketch has to be in the same plane as the Stock in terms of the X and Z plane and must be a closed profile. No revolved line is needed for the geometry.

In this tutorial, you will define the Stock as a cylinder (bar stock) offset from the part in length.

Following are the steps:

1. Right click *Stock Manager* item in the Feature tree and select *Edit Definition* on the context menu.

OR

 Double click *Stock Manager* item in the Feature tree.

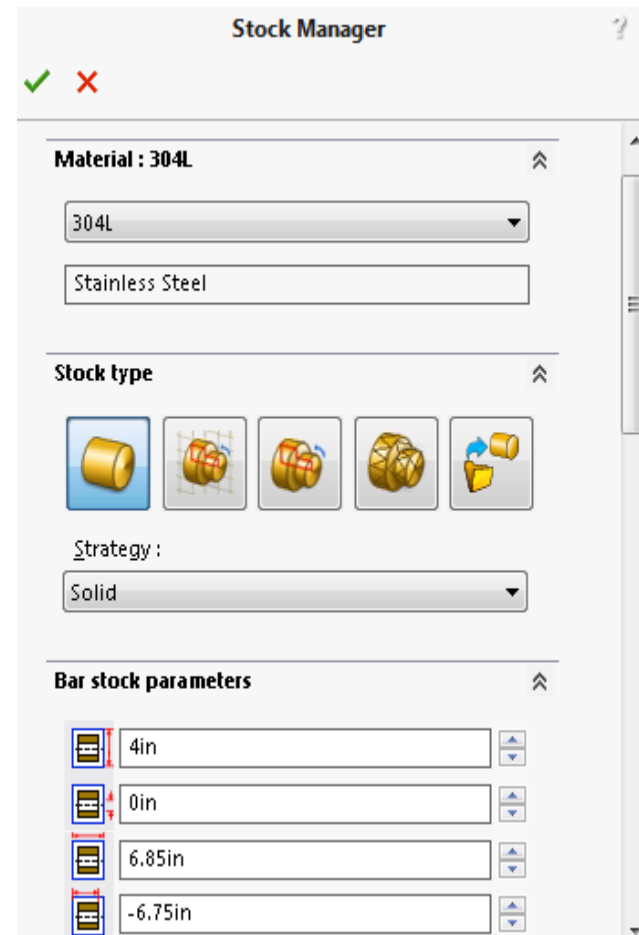
OR



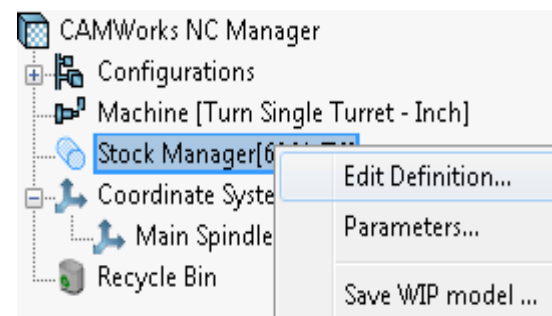
Click the *Stock Manager* button on the CAMWorks Workflow toolbar.

The *Stock Manager* dialog box is displayed.

2. Select **304L** as the material from the *Material* dropdown list.
3. Leave the Stock type set to *Round Bar Stock*.
The stock type can be defined from a revolved closed sketch, WIP sketch file or an STL file.






Stock Manager Dialog Box

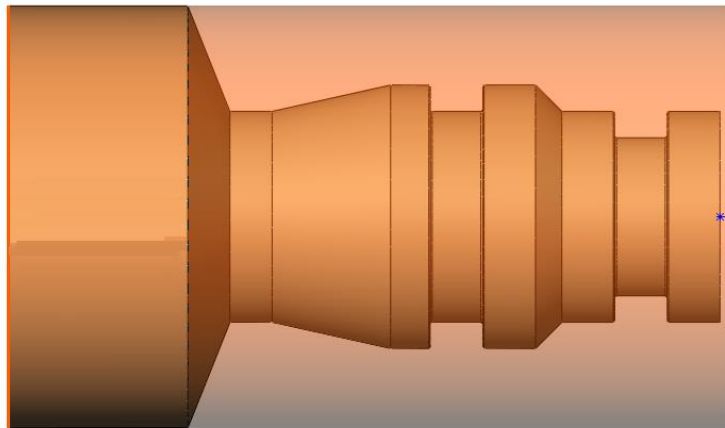


Command to open the Stock Manager dialog box



Defining the stock using these types will be explored in later tutorials of this manual

4. In the *Bar Stock Parameters* group box, change the  *Stock Length* dimension to **6.85in.**
5. Leave the  *Outside Diameter* set to **4in.**
6.  Click *OK* to close the *Manage Stock* dialog box.
The Stock will be updated in the graphics area.
7. Right click on the *Stock Manager* in the Feature tree and select *Display>>Translucent* in the cascading context menu. The stock will now be displayed in the graphics area whenever you select the *Stock Manager* item in both the *Feature* tree and *Operation* tree.
Notice the Stock line moved out from the face by **0.1in** (in the +Z-axis direction).



Stock displayed over the part in translucent shade

Step 5: Define Machinable Features

In CAMWorks, machining can be done only on machinable features. You must use either of the following two methods to define machinable features:

- **Automatic Feature Recognition (AFR)**

Automatic Feature Recognition analyzes the part shape and attempts to define most common machinable features such as grooves, outside and inside profile features, and face features. Depending on the complexity of the part, AFR can save considerable time in defining machinable features for turn solid parts.



Use the *Extract Machinable Features* command available on the CAMWorks Command Manager/Workflow Toolbar/ CAMWorks menu or the RMB context menu of the *CAMWorks NC Manager* to recognize features by AFR.

- **Interactive Feature Recognition (IFR)**

If AFR does not recognize a feature you want to machine, you can define the feature interactively using the *New Turn Feature* command.



Using Automatic Feature Recognition (AFR)

Automatic Feature Recognition is one of CAMWorks's most powerful features. The idea of AFR is to scan the part for features that can be machined. This process is much the same as what you would do if you were to pick up a part that you had to machine. You would look it over, take measurements and begin deciding what machining processes you would need.

CAMWorks does not machine the SOLIDWORKS/CAMWorks Solids features directly. Instead, it creates a separate list of Machinable Features. This is because a single SOLIDWORKS or CAMWorks Solids feature may have several areas that need to be machined in different ways with different tools.

The part used in this tutorial contains the following machinable features, which are defined automatically by CAMWorks:

- Face Feature
- OD Profile Feature
- Rectangular Groove Features
- Cutoff Feature

Defining Machinable Features Automatically

Automatic Feature Recognition



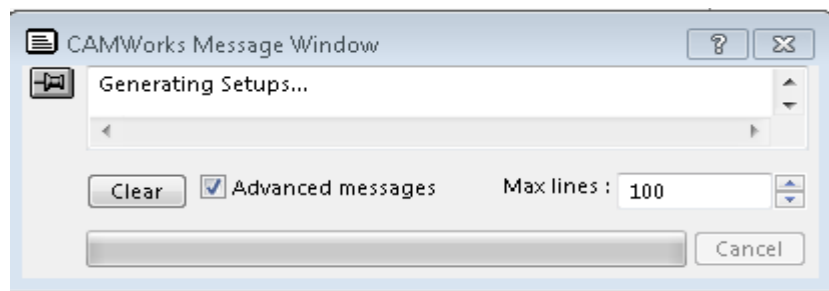
Click the *Extract Machinable Features* button on the CAMWorks Command Manager/Workflow toolbar.

OR


Right click *CAMWorks NC*

Manager in the Feature tree and select *Extract Machinable Features* on the context menu.

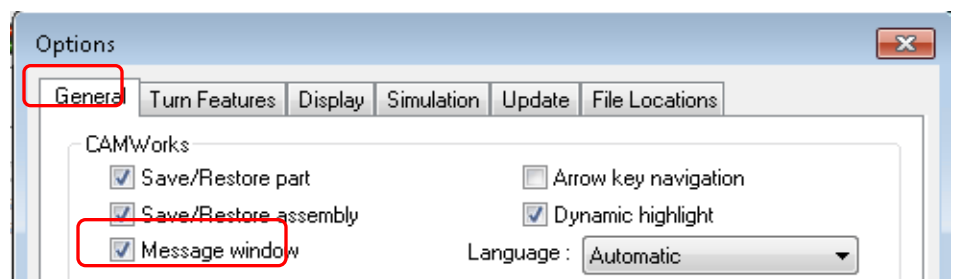
The Message Window is displayed. This window displays automatically to report the progress of the current process. Generating Setups is always the last item during Automatic Feature Recognition. When you see the line 'Generating Setups' in the message window, you can be sure that the process is almost complete.



CAMWorks Message Window

You can control whether the Message window displays temporarily or permanently. 

Click on the *Options* button on the CAMWorks Command Manager. This action opens the Option dialog box. On the General tab of this dialog box, check the *Message Window* option.



CAMWorks Options Dialog box - General Tab



To view the Options dialog box, you can also click **CAMWorks** on the menu bar OR right click **CAMWorks NC Manager** in the Feature tree and select the **Options** command.

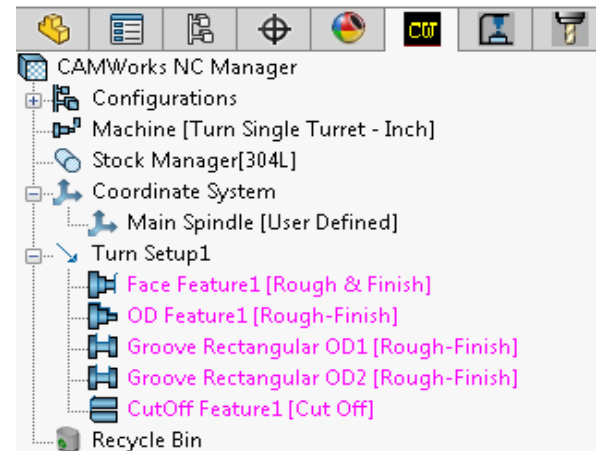
On execution of the *Extract Machinable Features* command, CAMWorks generates the Turn Setup and the machinable features. The items are displayed in the Feature tree.

Turn Setup: The Turn Setup is the 2 axis (X and Z) plane that the tool movement will be based on. It has an origin location, and X, Y, and Z direction vectors. The Turn Setup is created automatically; however, you can move the origin and change the direction and angles of the axis. Most turning applications work in the X and Z plane, which is relative to the SOLIDWORKS/CAMWorks Solids Y and X axis plane respectively.

When you define the Stock as bar stock, only one Turn Setup (X and Z) is created and all features will be machined using tool orientations that are relevant for a rear turret machine configuration.

Feature Tree: The Feature tree allows you to:

- Copy, rename, suppress, delete and combine machinable features
- Change machinable feature parameters
- Change the order in which the features are machined
- Insert Turn features
- Search for a feature based on item name
- Hide or show feature display in graphics area
- Generate an Operation Plan and find the first operation for a feature



Features recognized by AFR listed in Feature tree



Did You Know ...

When you recognize features by Automatic Feature Recognition (AFR) or Interactive Feature Recognition (IFR), the features listed in the Feature tree will display in different color **Magenta color** (by default) till you generate operations for these features. Once a valid operation is generated, the color of the corresponding feature item will change **Black color** (by default) indicating successful generation of the operation(s). If operations could not be generated for a feature (because the feature conditions have not been defined in the Technology Database for that particular feature type), then the feature will continue to display in the initial color (**Magenta color**), thus indicating that they have no operations defined.

You can set these colors on the Display tab in the Options dialog box.

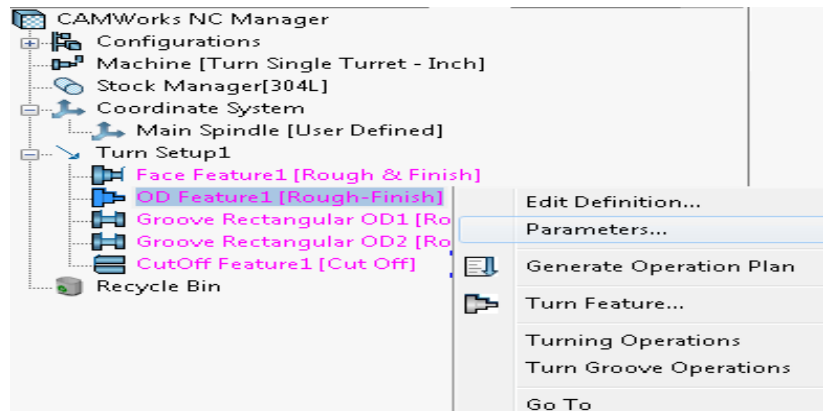


Changing Feature Parameters

Most feature parameters are fixed; however, some parameters can be changed using the *Parameters* command on the context menu for the feature item in the Feature tree.

1. Right click *OD Feature1* in the Feature tree and select *Parameters* from the context menu.

The *OD Profile parameters* dialog box is displayed. Since there is no physical information about the type of profile, CAMWorks allows you to define a *Strategy* for the profile.



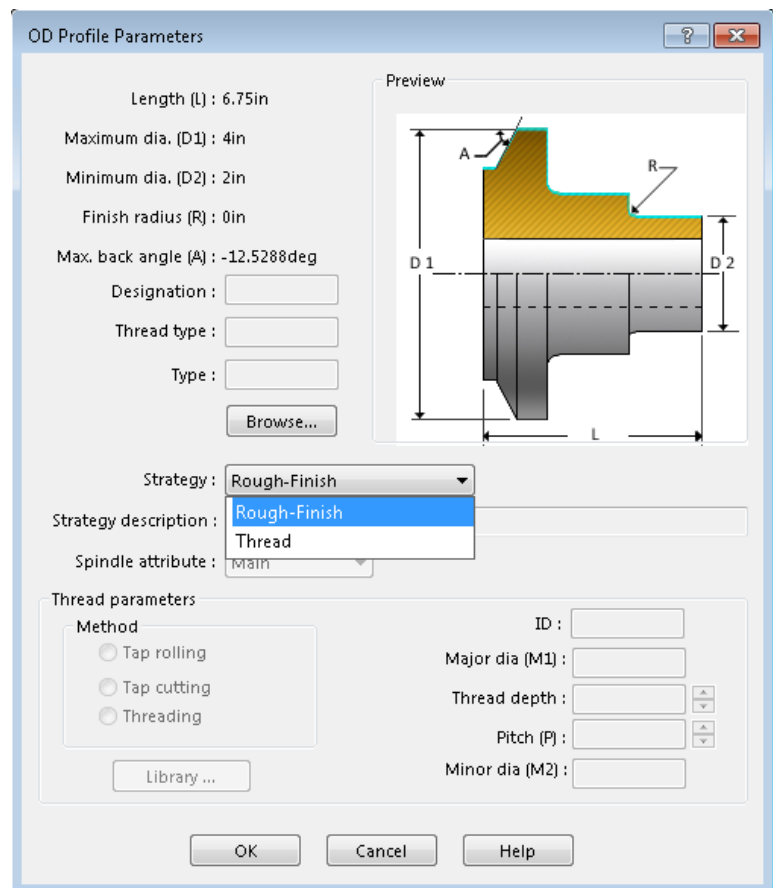
Select Parameters from the RMB context menu

2. Click the down arrow next to *Rough-Finish* in order to view the *Strategy* dropdown list containing the choices.

Rough-Finish and *Thread* are system Strategies.

User-defined Strategies are also listed, if defined. For example, RF80-80 is a shortcut name for roughing and finishing the profile using 2 tools (80 degree diamond tools).


3. Leave the Strategy set to *Rough-Finish* and click *OK* or *Cancel* to close the dialog box.

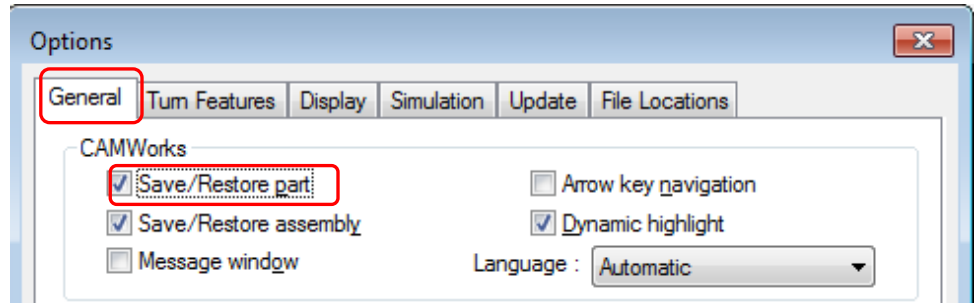


Feature Parameters Dialog Box



Save the part with the machinable feature data

1.  Click the CAMWorks *Options* button the CAMWorks Command Manager to open the options dialog box.
2. Make sure the *Save/Restore part* option on the *General* tab is checked, then click *OK* to close the dialog box.
 - If this option is checked, when you save and close a part document, the machining data is saved. When the part document is opened again, the machining data is restored along with the part design information.
 - If this option is not checked, when you save and close a part document that contains at least one CAMWorks Setup, a message indicates that *Save/Restore* is disabled. If you click *Yes*, CAMWorks saves all CAMWorks data before closing the file. If you click *No*, CAMWorks closes the part and discards any new CAMWorks data since the last save.
3. Click on the *File* menu and select *Save As* on the dropdown menu.
4. In the *Save As* dialog box, type in the desired file name and click the *Save* button.



CAMWorks Options Dialog box - General Tab



Make sure you save frequently. When you open a file, you are actually working on a copy of the file. The original is still stored on disk. Periodically saving your file ensures that your latest work is retained.

Defining Features Interactively (IFR)

Automatic Feature Recognition can save a significant amount of time; however, AFR does have its limitations. AFR cannot recognize every feature on complex parts and does not recognize some features such as certain types of grooves. To machine these areas, you need to define machinable features interactively using the *New Turn Feature* command.

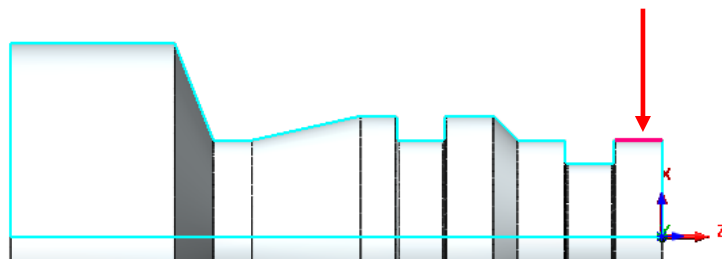
CAMWorks also provides the flexibility to edit machinable features to accommodate machining requirements. For example, if you want the OD machined in segments to take into account possible machining rigidity, you can define features for the segments in the OD Feature as shown in this tutorial as well as subsequent tutorials in this chapter.

In this tutorial, you will insert an OD Feature so that you can rough turn the first OD step of the part. This OD segment will be finished in the Turn Finish operation for *OD Feature1*.



Steps to interactively insert OD feature:

1. Right click *OD Feature1* in the Feature tree and select *Turn Feature* on the context menu.
The *New Turn Feature* dialog box is displayed. The part profile is shown in the graphics area. Rotate the part to view the part profile in case it is not visible.
2. In the *Features* group box this dialog box, select *OD Feature* for Type.
3. Leave the *Strategy* set to *Rough-Finish*.
4. In this tutorial, we will interactively define a feature picking the segment(s) on the highlighted geometry of the part. Hence, in the *Define from* group box, leave the method set to *<Part Profile>*.
5. Since the part does not contain any non-revolved features, leave the *Part Profile Method* set to *Plane Section*.
6. In the graphics area, the profile of *OD Feature1* is highlighted on the part. Pick the first OD profile segment (face) to the right of the groove closest to the face to select it. The single segment is highlighted.



Pick the first OD profile Segment

The selected segment (face) will be listed in the *Selected entities* list box of the dialog box as *CW Face-1*. *Extend 1* and *Extend 2*, which are parameters used to determine how the toolpath is computed at the ends of the feature, are listed in this box.

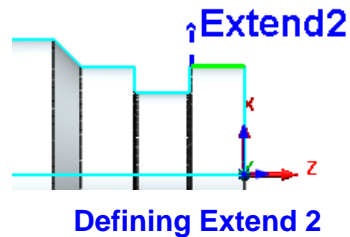
7. In the *Extend 2* group box, set the Extend to *Along X*.


This terminates the feature at the end of the

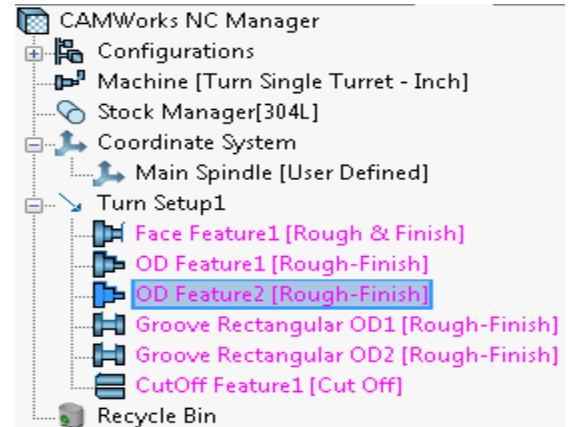
New Turn Feature Dialog Box



segment and up to the Stock major OD.



8.  Click *OK* to apply the changes. *OD Feature2* displays in the CAMWorks Feature tree. You have now defined all the machinable features in this part.

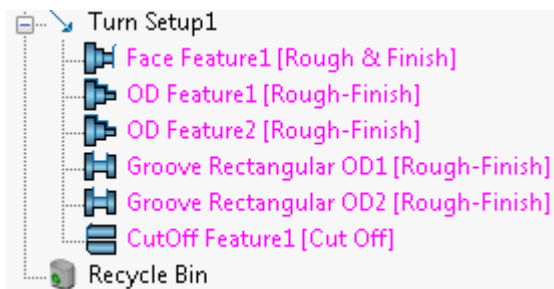


OD Feature 2 inserted interactively

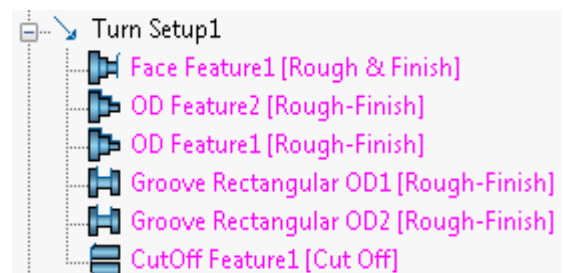
Reordering the features in the Feature tree

In the CAMWorks Feature tree, drag *OD Feature2* above *OD Feature1* in the tree using the mouse.

You need to reorder these features because you want to rough *OD Feature2* before cutting the entire OD. If you generated operations and toolpaths without reordering, CAMWorks would generate toolpaths for *OD Feature1* first, which includes the segment in *OD Feature2*. Toolpaths would not be generated for *OD Feature2* since there would be no material left to remove.



Before Reordering



After Reordering




Step 6: Generate Operation Plan and Adjust Operation Parameters

An Operation Plan contains information on how each machinable feature is to be machined and how the NC code will be output. When *Generate Operation Plan* command is run, operations for each machinable feature are created automatically based on information in the TechDB.



Did You Know ...

In some situations, the operations defined for a feature in the TechDB may not be sufficient and additional operations may be required. You can insert operations interactively using the New Turn Operation command. This command is explained in detail in the CAMWorks context-based Help.

1.  Click the *Generate Operation Plan* button on the CAMWorks Command Manager/Workflow toolbar.

OR

Right click *Turn Setup1* in the Feature tree and click *Generate Operation Plan* on the context menu.


CAMWorks generates the operation plan for all the machinable features in *Turn Setup1*.

The Operation tree is displayed and it lists the operations that were generated. The listed operations are displayed in **magenta** color (default color setting). This color indicates that toolpaths have not yet been generated for the operations.

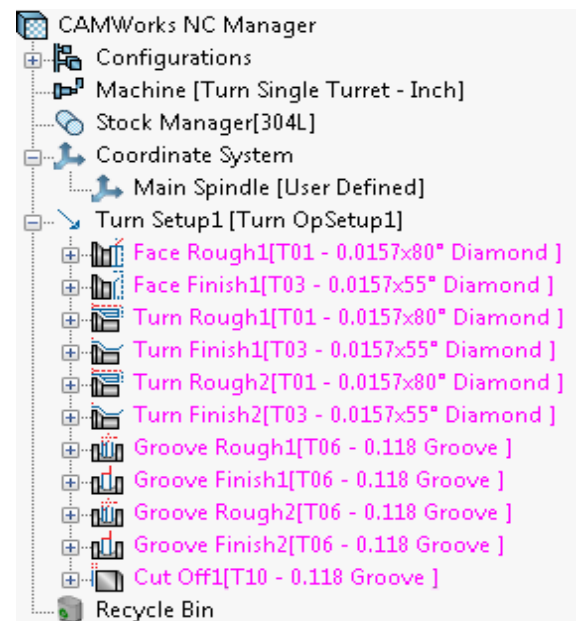
2. Click on the *Feature tree* tab. Observe that the font color of all the listed features has changed from **magenta** to **black** (default color settings). This change in color for all the features indicates that operations were generated for all the features.



You can set the color for the features listed in the Feature tree on the *Display* tab of the *Options* dialog box.

3.  Click the *Operation Tree* tab to display the Operation tree again.

Operation tree: The Operation tree provides an outline view of the operations for the machinable features. Operations are listed under the Turn Setup in the same order as the machinable features. At the top of the tree is the NC Manager. The Stock Manager and Machine items are the same as in the Feature tree. You can



Generated Operations listed in Operation Tree



Operation Tree tab



change the definition of your stock and the post processor used by CAMWorks to produce G-Code.

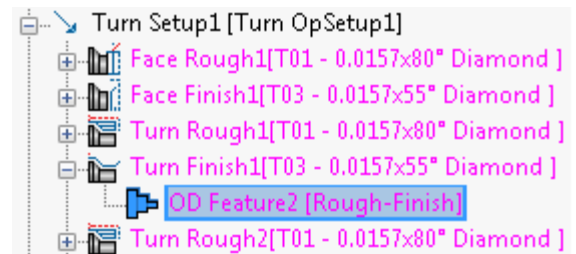
The Operation tree allows you to:

- Insert, rename, suppress and delete operations
- Change operation parameters
- Edit the feature list
- Change the machining order
- Generate toolpaths
- Simulate toolpaths
- Hide or show toolpath display
- Post process the toolpaths
- Search based on an item name

4. Click the plus sign (+) next to *Turn Finish1*.

This action displays the name of the machinable feature that this operation is going to machine.

5. Right click *Turn Finish1*, select *Delete* on the context menu and click *Yes* to confirm the deletion.



Expand the Operation item to view the feature it machines

- Based on the information in the TechDB, rough and finish operations were generated for *OD Feature2*. In this tutorial, you want to finish turn this OD segment in the finish operation for the entire OD (*Turn Finish2*) so you can delete the finish operation.
- If you typically machine the OD in segments, you can customize the TechDB and set up a Strategy to generate only a rough turn operation for this type of OD feature.

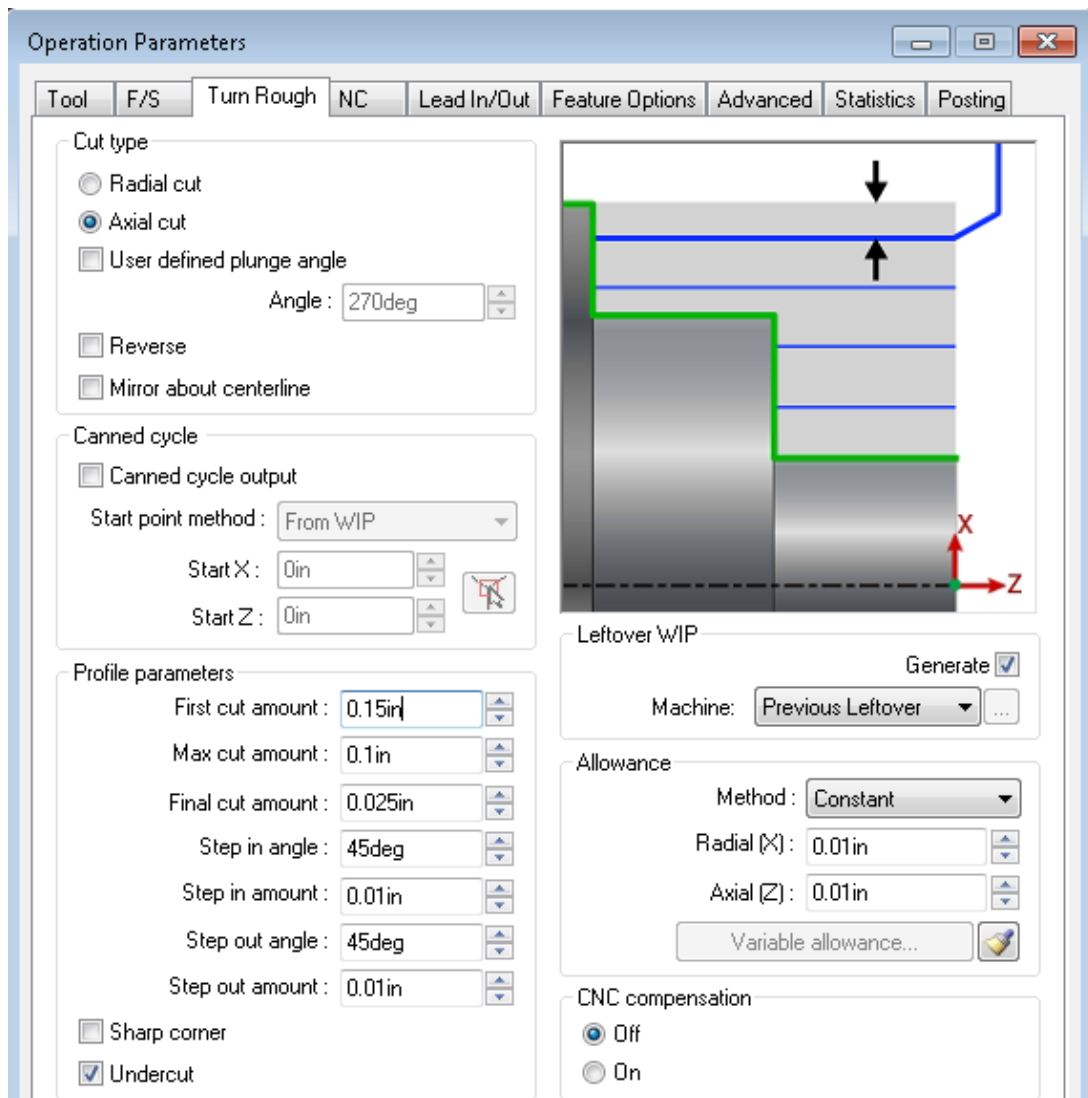
The operations that are generated by CAMWorks are based on information stored in the Technology Database. Each operation contains parameters that affect how the toolpath is created and specific parameters that will be output to the NC program. These parameters can be edited before generating the toolpaths and post processing the part.

6. Double click *Turn Rough1* (Roughing operation for *OD Feature2*) in the tree.

The *Operation Parameters* dialog box for the operation is displayed.

The *Operation Parameters* dialog box gives you access to all the parameters used to define the toolpath. This dialog box also provides access to the parameters for the tool you are using and allows you to select a different tool.

7. Click on the *Turn Rough* tab. In the *Profile Parameters* group box, change the *First cut amount* to **0.15in**.
8. Click on the other tabs and view the parameters. When you cut your own parts, you can edit these values as per your machining requirements.
9. Click *OK* to apply the changes made and close the dialog box.




The Operation Parameters dialog box



Step 7: Generate Toolpaths

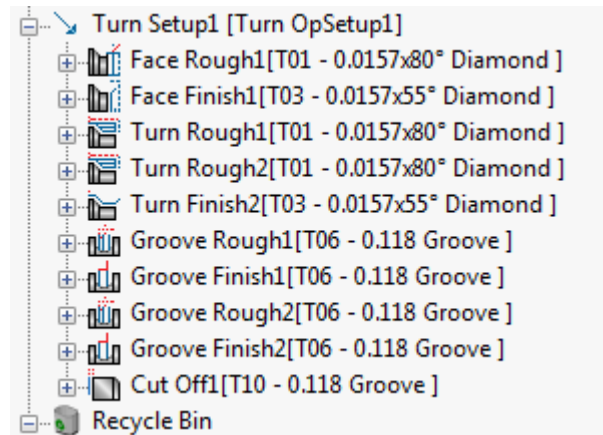
CAMWorks calculates toolpaths using the operation parameters to define how to machine each machinable feature.

1.  Click the *Generate Toolpath* button on the CAMWorks Command Manager/Workflow toolbar.

OR

Right click *Turn Setup1* in the Operation tree and select *Generate Toolpath* on the context menu.

On executing the *Generate Toolpath* command, CAMWorks calculates the toolpaths for each operation in the Setup. The font color of all the listed operations in the Operation tree changes from **magenta** to **black**. This change in color indicates that toolpaths were successfully generated.



Black font color of the operations indicates that toolpaths were successfully generated



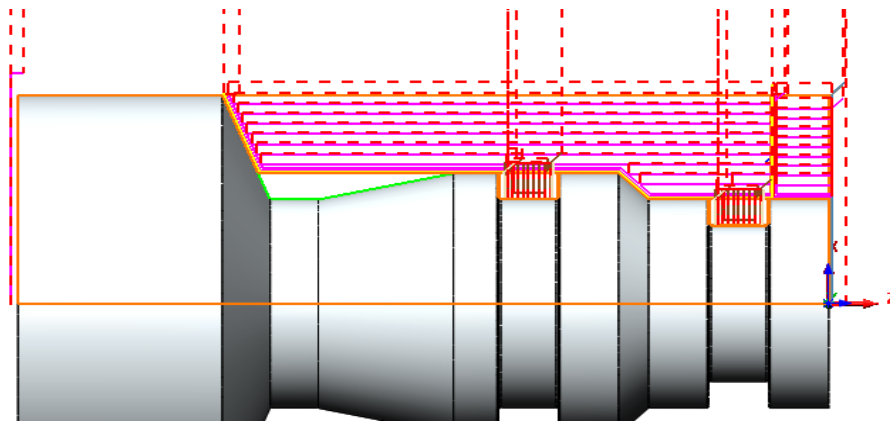
Did You Know ...

If an operation displays in a magenta color instead of black, then toolpaths have not been generated. This might occur in one of the following situations:

- i. When you insert a new operation interactively;
 - ii. When you insert a new feature interactively and then generate operations for the new feature;
 - iii. When CAMWorks cannot generate the toolpath for an operation because of an error in the toolpath algorithm or a parameter is not correct.
2. Left click any operation in the Operation tree. That operation will be highlighted in the Operation tree.
 - The toolpath for that highlighted operation will be displayed in the graphics area. As you highlight each operation in the tree, the toolpaths for that corresponding operation will be displayed.
 - Turning operation parameters can be edited and the operation can be renamed, moved, suppressed, deleted, etc. after toolpaths have been generated. These commands are available in the RMB context menu.
 - If you make any changes, the toolpaths must be updated by selecting Generate Toolpath again at the Setup level.
 3. Hold down the *Shift* key and select the first and last operation in the tree. This action selects all the operations.




The toolpaths for all the operations will be displayed on the part showing the centerline of the toolpath.



Toolpaths for all the operations displayed on the part when all the operations are selected in the operation tree

Step 8: Simulate Toolpaths

CAMWorks provides the ability to simulate the toolpaths showing the tool movement and the resulting shape of the part.

1.  Click the *Simulate Toolpath* button on the CAMWorks Command Manager/Workflow toolbar.

OR

Right click on *Turn Setup1* in the operation tree and select *Simulate Toolpath* on the context menu.

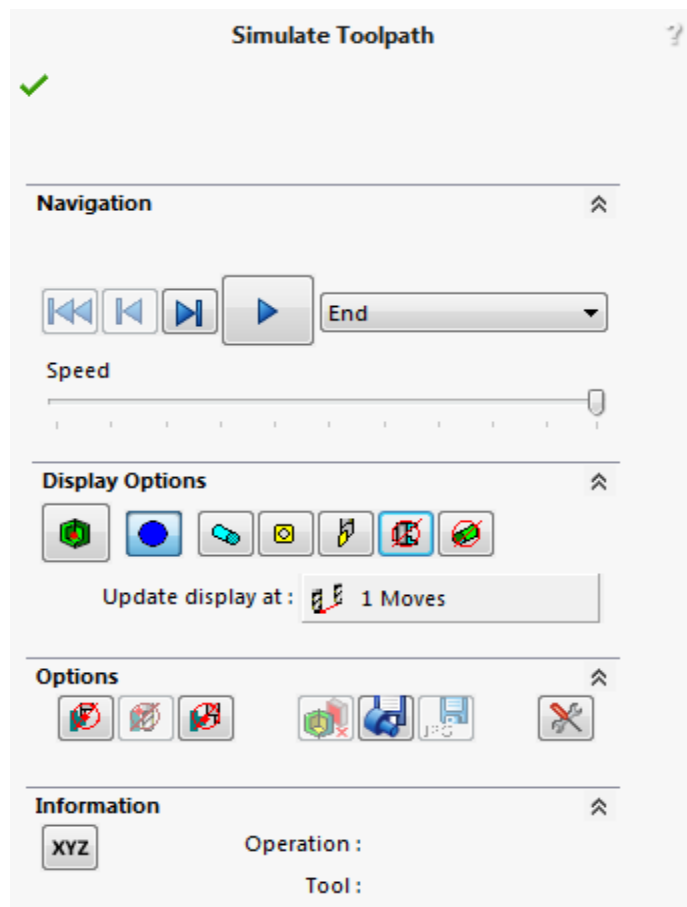
On executing this command, the *Toolpath Simulation* toolbar is displayed.

Some of the options you can select to customize the simulation include:

- Update the Stock after each cut or show the completed part at the end of the simulation.
- Change the display of the stock, tool, tool holder, and target part (wireframe, translucent, shaded, or no display).
- Run the simulation to the end or advance by single step or by feature.
- Compare the design part and the simulated part during simulation.
- Show a cross section of the material removal.
- Show holder and fixture collisions.
- Control the simulation speed by dragging the Simulation Speed Control slider.










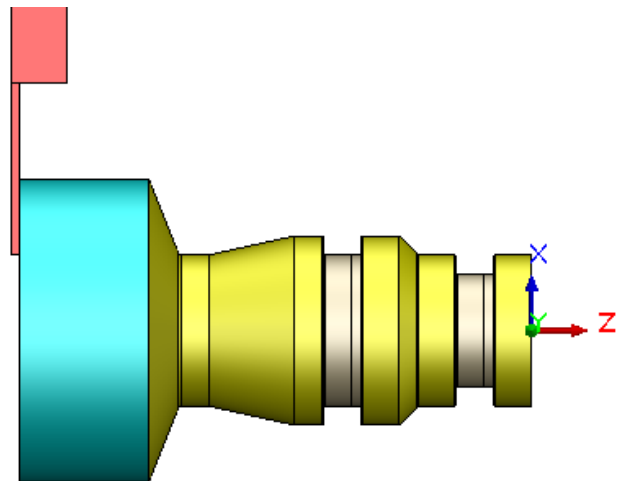
If you want to simulate only the toolpath for a given operation, you can right click that operation and then select *Simulate Toolpath* from the context menu.




Simulate Toolpath Dialog Box

2. When you click on the display control buttons of the Toolpath Simulation toolbar, the available settings associated with that button are displayed in a dropdown list.
3. Set these toolbar buttons as below:

-  Section View: *Full*
-  Stock: *Shaded with Edges*
-  Tool: *Shaded with Edges*
-  Tool Holder: *Shaded with Edges*
-  Fixture: *No display*
-  Tool Collision: *Tool Ignore Collision*
-  Tool Holder Collision: *Tool Holder Ignore Collision*



Toolpath Simulation



4. In the End Condition dropdown list, select *Next Operation*.
5.  Click the *Run* button.



The simulation is run with the tool displayed during simulation. As the End Condition is set to Next Operation, only the first operation is simulated. Click the *Run* button each time you wish to simulate the subsequent toolpaths. The toolpaths will be simulated in the serial order of the operations listed in the Operation tree.



6. Use the *Simulation Speed Control* slider to control the speed of the Simulation.

7.  To pause the simulation while it is running, click on the *Pause* button. When you click *Run* button again, the simulation will continue from the point where it was paused.
8. Click the *OK*  button to exit the simulation mode and return to the SOLIDWORKS/CAMWorks Solids display.




Did You Know ...

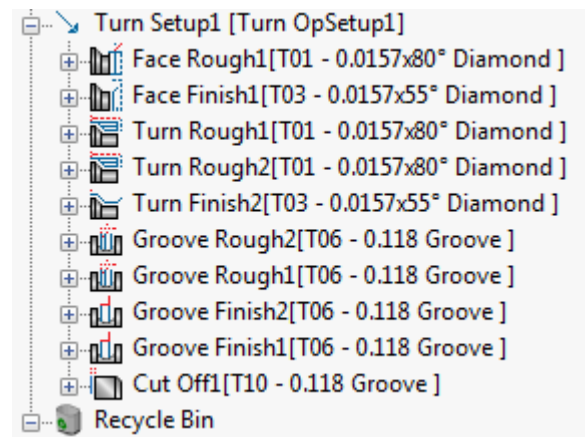
CAMWorks provides an option to display the XZ position of the tool in the Message Window during simulation. To activate this option, select the Options command on the CAMWorks Command Manager to open the Options dialog box. On the General tab, ensure that the Message Window option is checked. On the Simulation tab, select the Display Coordinates option.

Change the Machining Order

Operations are generated in the same order that the machinable features are listed in the tree. This may not cut the part in the order you think is most efficient or practical for your machining requirements. You can drag and drop operations in the tree to reorder them.



For example, it is more practical to first machine the groove closest to the face rather than any other groove. Similarly, it is prudent to first rough machine the grooves followed by the Finish operations.

1. Drag and drop the *Groove* operations in the order shown in the tree on the right.
2.  Click the *Generate Toolpath* button on the CAMWorks Command Manager/Workflow toolbar.



Operation Tree after reordering Groove operations

If you change the order of the operations after you generate toolpaths, you must regenerate the toolpaths so that the work in process can be updated.

3.  Simulate the toolpaths.  Run the simulation and observe that the groove closest to the face is cut first and both grooves are rough cut and then finished.



Step 9: Post Process Toolpaths

Post processing is the final step in generating the NC program file. This step translates generalized toolpath and operation information into NC code for a specific machine tool controller. CAMWorks creates NC code for each toolpath in the order the toolpath operation appears in the Operation tree. When you post process a part, CAMWorks creates two files: the NC program and the Setup Sheet. These are text files that you can read, edit and print using a word processor or text editor.

In this exercise, you post process all the operations and generate the NC program:



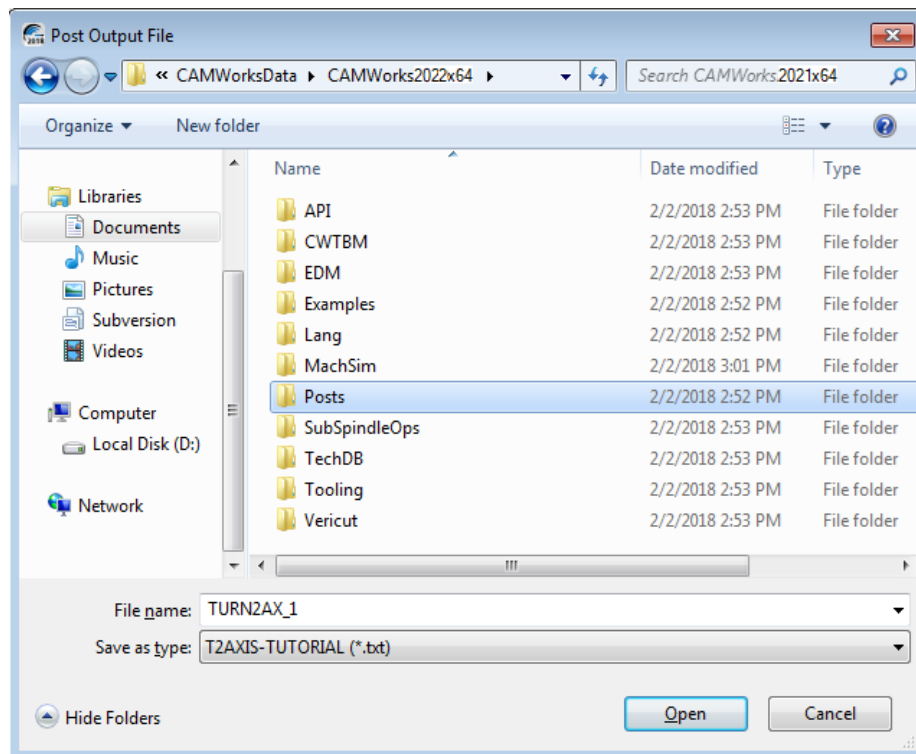
Click the *Post Process* button on the CAMWorks Command Manager/Workflow Toolbar. OR

Right click the *CAMWorks NC Manager* in the Operation tree and select *Post Process* on the context menu.

The *Post Output File* dialog box will be displayed. Use this dialog box to name and save the NC program file.



If you are running CAMWorks in Demo mode, the Post Output File dialog box does not display because you cannot save NC code in Demo mode.



Post Output File dialog box



Turn Tutorial

Typically, the NC program and Setup Sheet files are stored in the folder that contained the last part that was opened. If you want these files in another location, you can change the folder location.

If Post Process button is grayed out on the CAMWorks menu, Command Manager or the CAMWorks Workflow toolbar, [make sure that you have selected a post processor](#) and generated the toolpaths.

1. Click the down arrow to the right of the *Save as type* box.

CAMWorks provides a list of commonly used extensions that you can select. For this tutorial, use the default *.txt* extension.




Did You Know ...

If you want change the default extension from *.txt* to one of the ones in the list or if you want a different file name extension for NC program files, you can edit or create a *.pinf* file and specify the new extension. For more information on making these changes, see the context-based Help.

2. If *TURN2AX_1* is not in the *File name* text box, type **TURN2AX_1**, then click *Save*.
You do not have to type the extension if you are using the default *.txt*. Naming the post output file the same as the part file is the most common way of saving parts and NC programs. Both files can have the same name because they have different extensions.


3. After executing *Save* command, the *Post Process*

Output dialog box is displayed. Click the *Step* button  at the top of the Post Process Output dialog box.

CAMWorks starts to generate the NC program and the first line of NC code displays in the NC code output view box. When you click the *Step* button, CAMWorks generates one line of code.

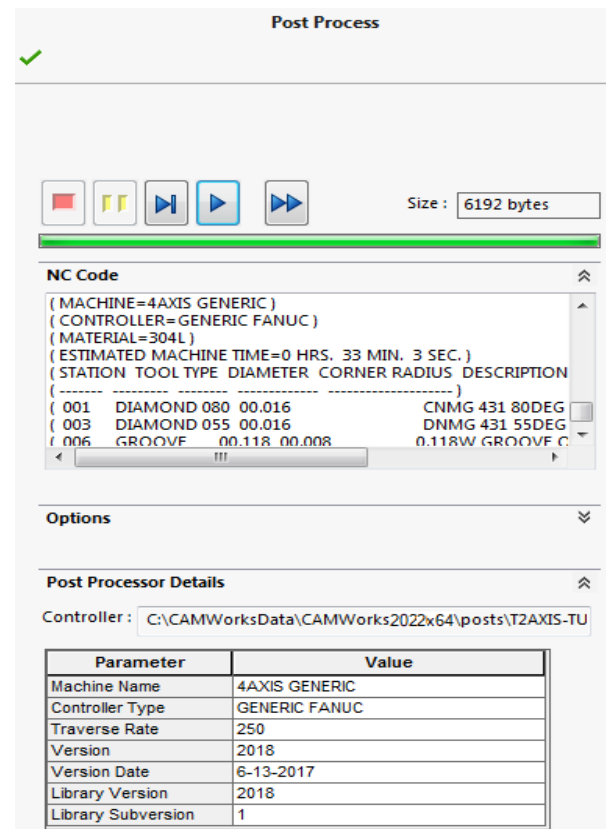
4.  Click the *Step* button again.

The next line of NC code is displayed.

5.  Click the *Run* button.

Post processing continues until it is completed. When the post processing is finished, you can view the code using the vertical scroll bar.

6. Click *OK* to close the dialog box.



Post Process Dialog Box



Setup Sheets

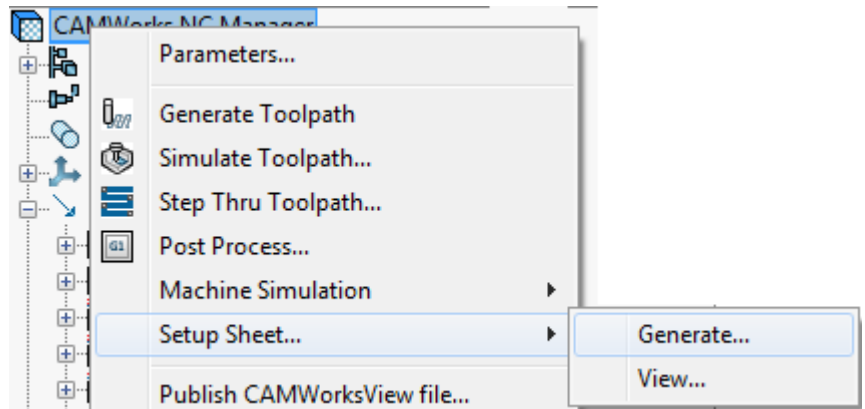
The Setup Sheet is a printable file that contains information the machine tool operator can use to set up the part and the tools required to produce a part. The information includes the machine, the controller, estimated machine time, the part material, and the tooling used to machine the part.

CAMWorks provides two methods for creating Setup Sheets:

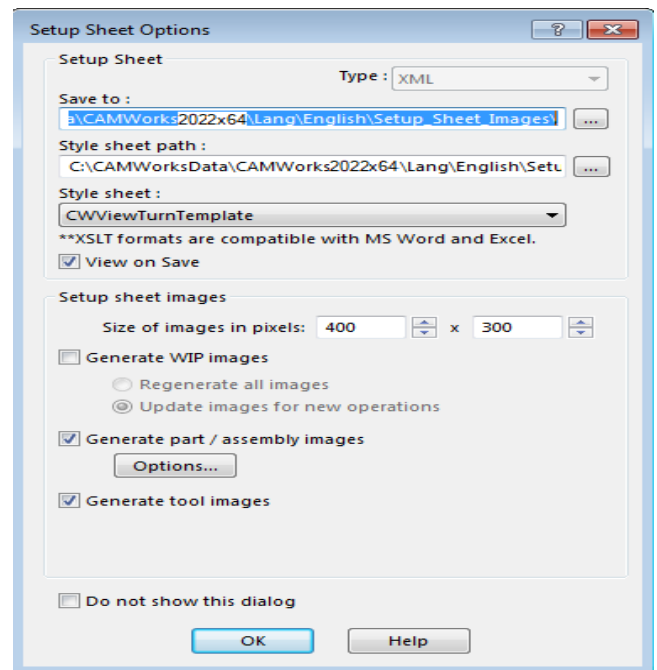
- During post processing, CAMWorks automatically creates a simple text file with a .set extension.
- Click *Setup Sheet* command on the CAMWorks NC Manager context menu and the *Generate Setup Sheets* command on the Setup context menu provide a choice of two formats:
 - i. **Access:** When you select Access, the Setup Sheet is based on an Access database report template and the information is stored in the Report Database to view at any time. CAMWorks supplies several report templates that can be used as is. You can also open the Report Database in Access and create customized reports based on these templates or design your own original reports.
 - ii. **XML:** When you select XML, the information is saved in an XML file, which allows the Setup Sheet to be formatted with an HTML style sheet and displayed in a web browser.



Refer the manual 'CAMWorks Setup Sheet Tutorial' for detailed Help on how to generate Setup Sheets.



Setup Sheet Command on CAMWorks NC Manager context menu



Setup Sheet Options dialog box

Click Help to view context-based help on the topic



Turn 2

Topics covered in this tutorial:

- [Turn feature types recognized by AFR and IFR](#)
- [Editing a tool in the active tool crib](#)
- [Removing a tool from the active tool crib](#)
- [Adding a tool to the active tool crib](#)
- [Saving the changes made to the tools in the tool crib](#)
- [Changing the Feature Strategy](#)

Turn Feature Types Recognized by AFR and IFR

AFR & IFR

CAMWorks is a feature-based machining system. Feature-based machining provides numerous benefits because the definition of the feature enables a higher level of automation when creating machining operations and associated toolpaths.

CAMWorks provides two methods for extracting features:

- i. An automatic method called Automatic Feature Recognition (AFR),
- ii. An interactive method using the *New Turn Feature* function.

A CAMWorks file can contain features recognized by any one or both of the above methods.

Supported Turn Feature Types

The following turn feature types are currently recognized:

- **Face Feature:** A Face feature exists on the outermost portion of the part.
- **OD Profile Feature:** An OD profile feature covers the entire outer surface of the part.
- **ID Profile Feature:** An ID profile feature covers the entire inner surface of the part.
- **Groove Feature with vertical walls:** A Groove is a feature that is totally closed and below the surrounding features.
 - Grooves are further broken down into one of two categories: Rectangular and Outbound.
 - Three types of grooves are supported:
 - ✓ OD Groove
 - ✓ ID Groove
 - ✓ Face Groove
- **Cutoff Feature:** A Cutoff feature is generated on the opposite side of the Face feature when the Stock is defined as bar stock. A Cutoff feature is similar to a face and can be converted to a Face feature for two-step turning operations.

This tutorial demonstrates the types of features identified automatically by the *Extract Machinable Features* command.




Step 1: Open the Part

In this tutorial, you will extract features and generate operations and toolpaths for a turn part that was imported into SOLIDWORKS/CAMWorks Solids.

1. Open the part file **TURN2AX_2.SLDPRT** located in the following folder.

Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Turn.

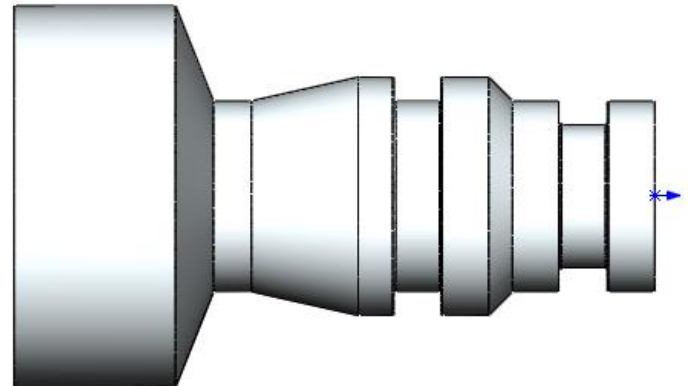
2.  Click on *FeatureManager Design Tree*. Notice that the design tree contains no design features. This part was imported into SOLIDWORKS.

3. Examine the part.

You can recognize various sizes and shapes of faces, profiles and grooves.

It would appear that CAMWorks extracts features based on SOLIDWORKS/CAMWorks Solids features used to create the part.

As you will see in this tutorial, this is not necessarily so. The part used in this tutorial is an imported part. This tutorial will demonstrate how CAMWorks determines machinable features.



TURN2AX_2.SLDPRT


Step 2: Defining the Machine and Editing the Stock Definition

Define the Machine

1. Click the *CAMWorks Feature Tree* tab.
2. Ensure that *Turn Single Turret - inch* is selected as the Machine. If not, then following are the steps:

- i.  or  Double click the *Machine* item in the Feature tree.

OR

-  Click on the *Define Machine* button on the CAMWorks Workflow toolbar.

The Machine tab of the Machine dialog box is displayed.



CAMWorks Feature Tree

- ii. Highlight *Turn Single Turret - inch* in the *Available Machines* list and click the *Select* button.
- iii. Click *OK* button to apply the changes and close the dialog box.



This machine definition has been created for the CAMWorks exercises. When you use CAMWorks to machine your own parts, select the machine tool you want to use to machine the part.



Editing the Tool Crib

1. Click the *Tool Crib* tab of the Machine dialog box.
From this tab, you can add, remove and edit tools in the Tool Crib.
2. Uncheck the *Tool crib priority* option in case this option is checked.
3. In the *Available tool cribs* list, make sure *Tool Crib 2 Rear (Inch)* is the Active tool crib.
To do so, highlight it in the *Available tool cribs* list and then click the *Select* button.

Machine

Machine Tool Crib Post Processor Posting Setup Chuck/Fixture

Tool crib

Active tool crib : Tool Crib 2 Rear (Inch) Turret : Rear Turret

Usage	Stn No.	Station ID	Type	ID	Comment	Insert
	1		Turn Tool	3	CNMG 431 80DEG SQR HOLDER	Diam
	2		Turn Tool	5	CNMG 431 80DEG BORE BAR	Diam
	3		Turn Tool	4	DNMG 431 55DEG SQR HOLDER	Diam
	4		Turn Tool	6	DNMG 431 55DEG BORE BAR	Diam
	5		Center Drill	5	#3 60DEG HSS CENTERDRILL	
	10		Turn Tool	9	0.118W CUT-OFF BLADE	Gro

Add... Remove Edit... Save...

☐ Tool crib has sub stations
☐ Tool crib priority
☐ Use tool crib tools only

Save Tool Crib...

Available tool cribs

Tool Crib 1 Rear (Inch) Empty
Tool Crib 1 Front (Inch) Empty
Tool Crib 2 Rear (Inch)
Tool Crib 2 Front (Inch)

Select

Name : Tool Crib 2 Rear (Inch)
No. of stations : 12

OK Cancel Help

Tool Crib Tab of Machine dialog box



Editing a Tool

1. Select any tool from the Active tool crib list and click the *Edit* button.

To select a tool in the *Active tool crib* grid, click on any field in the row containing the tool.

2. When you click the *Edit* button for a selected tool, the *Edit Tool Parameters* dialog box is displayed. This dialog box contains three tabs that allow you to change the parameters for the selected tool.

3. Click the tabs to view the tool and holder parameters. If you make any changes to the parameters, click *OK* to apply those changes and close the *Edit Tool Parameters* dialog box.

The changes you make in this dialog box affect only the tool crib for the current part.

To change the tool definition for all future jobs, you need to click on the *Save* button in the Tool crib tab else the changes will be applicable only for the current model part.

Alternatively, you can edit the Tool Crib definition in the Technology Database.

The screenshot shows the 'Edit Tool Parameters' dialog box with the 'Station' tab selected. The 'Diamond Insert' and 'Holder' tabs are also visible. The 'Station' section includes a 'Tool number' field with the value '1', a 'Station ID' field with the value '0', and a 'Gage offset (XYZ)' field with the value '4.2666in'. Below these are 'TechDB ID : 3', a 'Comment' field containing 'CNMG 431 80DEG SQR HOLDER', and 'Tool usage : 0'. At the bottom, there is a section titled 'Operations using this tool' with an empty text area.

Edit Tool Parameters dialog box

Removing a Tool

To remove a tool from the Active tool crib, select the tool in the Active tool crib grid and then click the *Remove* button.



Note that the tool removal from the tool crib is effective only for the current part. To make this change available for all future jobs, click on the *Save Tool Crib* button to permanently save the changes.

Adding a Tool

Use the *Add* button to add a tool to the Active Tool crib.

1. Click the *Add* button in the Tool Crib tab.
2. The *Tool Select Filter* dialog box is displayed. This dialog box allows you to set filters for displaying and selecting tools. At the bottom of this dialog box is a *Tools database* grid listing all the tools entries from the TechDB which fulfill the tool selection criteria that you select.

In this tutorial, you will insert a Turn tool to the active tool crib.



Turn Tutorial

3. In the *Tool Select Filter* dialog box, select *Turn Tool* from the dropdown list for the *Tool type*.
4. The *Filter by* group box within the *Tool Select Filter* dialog box will now display tool selection parameters for the selected Tool type. The *Tools database* grid too will now display only Turn tools.
5. You can choose to further filter down your tool selection criteria by defining additional filters. For example, in this tutorial, we will select a Turn tool made of material *Carbide*. To do so:
 - a. Check the option *Tool material*. This enables the *Tool material* dropdown list.
 - b. Select *Carbide* from the dropdown list.
 - c. The *Tools database* grid will now display only Turn tools made of material *Carbide*. From this grid, highlight the tool you wish to add to the active Tool Crib.
 - d. You can view the Preview of the highlighted tool in the Preview window.
 - e. Click the *OK* button.



Tool Select Filter

Tool type : **Turn Tool**

Filter by

☐ Turn Insert Type **Diamond**

☐ Insert Radius **0in** - **9in**

☒ Tool material **Carbide**

☐ Turn Holder **Boring Bar**

☐ Insert/Holder Summary

Preview

Turn (Inches)

	ID	Insert Type	InsertID	InsertSummary	InsertTabRec	Holder Type	Holder ID	Ho
1	1	3	1	TIP-UN 60Deg w/	TurnThreadIns	1	10	RH C
2	2	3	1	TIP-UN 60Deg w/	TurnThreadIns	2	8	LH C
3	3	1	1	CNMG-431	TurnDiamondIn	1	1	RH E
4	4	1	2	DNMG-431	TurnDiamondIn	1	3	RH E
5	5	1	1	CNMG-431	TurnDiamondIn	2	2	RH E
6	6	1	2	DNMG-431	TurnDiamondIn	2	4	RH E
7	7	2	1	0.118" Width Gro	TurnGrooveIns	1	8	RH C
8	8	2	1	0.118" Width Gro	TurnGrooveIns	2	10	RH C
9	9	2	1	0.118" Width Gro	TurnGrooveIns	1	12	RH C
10	19	1	6	VNMG-432	TurnDiamondIn	1	5	RH C
11	20	1	6	VNMG-432	TurnDiamondIn	2	5	LH 3
12	21	2	5	0.079 WIDE GRO	TurnGrooveIns	1	8	RH C
13	22	2	5	0.079 WIDE GRO	TurnGrooveIns	2	10	RH C
14	25	4	1	TNMG 221	TurnTriangleIns	2	7	LH C
15	26	5	1	SNMG 322	TurnSquareIns	1	10	RH C
16	27	6	1	WNMG 331	TurnTrigonInse	1	10	RH C
17	28	7	1	HEX .5" Dia w/.01	TurnHexagonIn	1	10	RH C
18	29	8	1	RNMG 32	TurnCircularIns	1	10	RH C

OK **Cancel** **Help**

Tool Select Filter dialog box

Note: To exit without adding any tool using the Tool Select Filter dialog box, click on the **Cancel** button.

6. A new tool selected to be added to the active tool crib using the Tool select Filter dialog box will be added to the bottom of the Active Tool crib. It can be viewed in the *Tool Crib* tab of the *Machine* node.



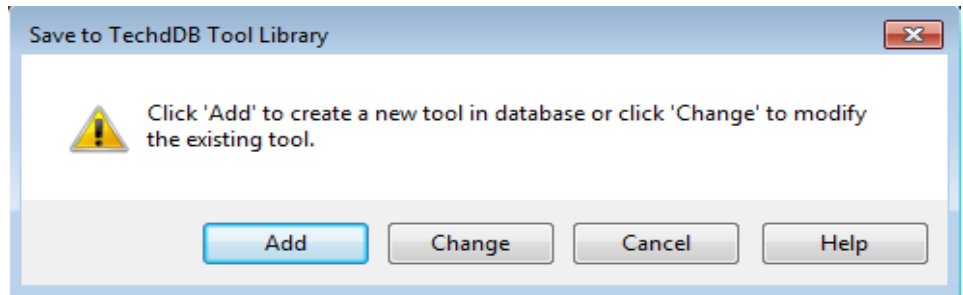
Tool addition to the active tool crib is effective only for the current part. To make this tool addition available for all future jobs, click on the **Save Tool Crib** button to permanently save the changes.



Saving the changes made to a Tool

If you make changes to any tool in the active tool crib, the changes are effective only for the current part and not for any other part. The edits made to a tool to make available for all future jobs, highlight the tool in the *Active tool crib* grid and click on the *Save* button.

When you click *Save*, CAMWorks displays a message prompting you to select whether you wish modify the existing tool or add a new tool to the database.



Message displayed when you try to Save a Tool

- If you select '**Change**', then the modifications made to the tool will be saved to the Technology Database, thus making them permanent. The modified tool will now be available for all future jobs.
- If you select '**Add**', then the original tool parameters and Tool ID of the edited tool will be retained. Instead, a new tool with the edited parameters will be created and saved to the Technology Database. The edited tool will continue to show the newly edited values. These values will be displayed only for the current part and not any future jobs. The newly created tool will not be visible in the *Active tool crib*. If you wish to add this new tool to the active tool crib, use the functionality of [Adding a tool](#) to add it.

Selecting the Post Processor

1. Click the *Post Processor* tab in the Machine dialog box.
2. Make sure *T2AXIS-TUTORIAL* (the tutorial post processor) is selected as the active post processor.



T2AXIS-TUTORIAL is used for the exercises in this manual. When you use CAMWorks to machine your own parts, select your machine tool controller or post processor.

3. Click *OK* to apply the changes and close the *Machine* dialog box.

Editing the Stock Definition

1. Double click *Stock Manager* in the CAMWorks Feature tree.

OR




Click on the *Stock Manager* button of the CAMWorks Workflow toolbar.

The *Stock Manager* dialog box is displayed.

2. By default, the Stock shape is a cylinder whose dimensions enclose the part. For this tutorial, the part is machined from a bar stock that has **0.10in** material to be removed




from the face. Hence, in the *Bar Stock Parameters* group box, change the *Stock Length* of the bar stock to **6.85in**.

3.  Click *OK* to apply the changes and close the dialog box.

Step 3: Extracting Machinable Features and Editing Feature Parameters

Extracting Machinable Features

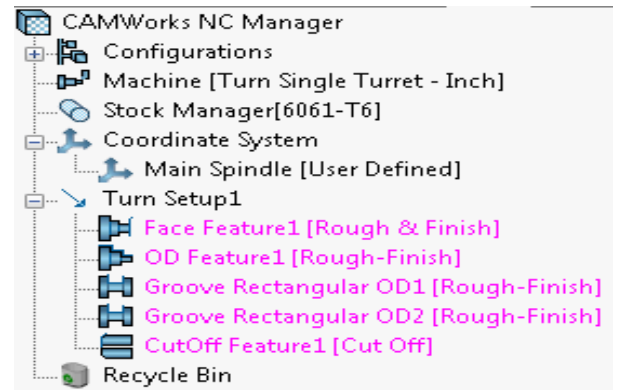
1.  Click the *Extract Machinable Features* button on the CAMWorks Command Manager/Workflow toolbar.

OR

Select the *Extract Machinable Features* command from the CAMWorks menu.

The CAMWorks Message Window is displayed. This window shows the progress of the process.

When Automatic Feature Recognition ([AFR](#)) is finished, the [Feature tree](#) displays the five machinable features recognized by AFR.



Turn features recognized by AFR as listed in the Feature Tree



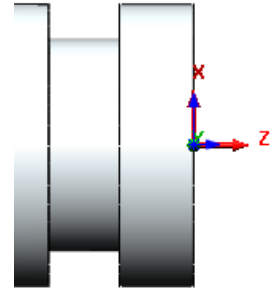
Turn Setup

Observe that AFR created *Turn Setup1*. The Setup defines the machining direction necessary to machine these features. The origin of the Setup is program zero for the Setup. By default, the origin is set to the front of the part.

For this tutorial, you will use the current machining direction.



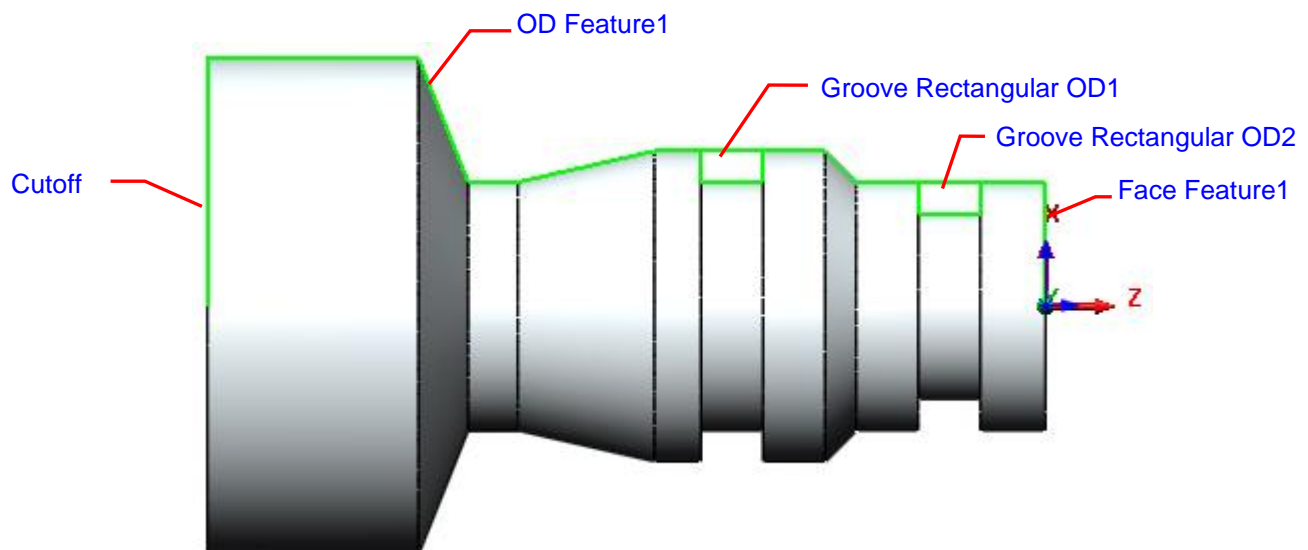
By default, the origin of the Turn setup is set to the front of the part. You can change the machining direction using the *Edit Definition* command on the Turn Setup context menu. Tutorial [Turn 4](#) explains how to change this origin to another location.



Origin of setup is set at the front of the part

- Click each machinable feature in the Feature tree.

As you pick each machinable feature, the geometry for that feature is highlighted on the part.



Turn features are highlighted on the part as you pick each feature in the Feature tree

Feature Strategy

Machinable features are recognized based on the part's geometry (the shape of a feature) and topology (how the features are related to one another). However, not all attributes of a feature can be determined from the geometry and topography. For example, AFR cannot determine that an ID profile has been threaded, reamed or bored. CAMWorks allows you to specify additional Strategies to a feature so that more accurate operations can be created.

- Right click on *OD Feature1* in the Feature tree and choose *Parameters* in the context menu.



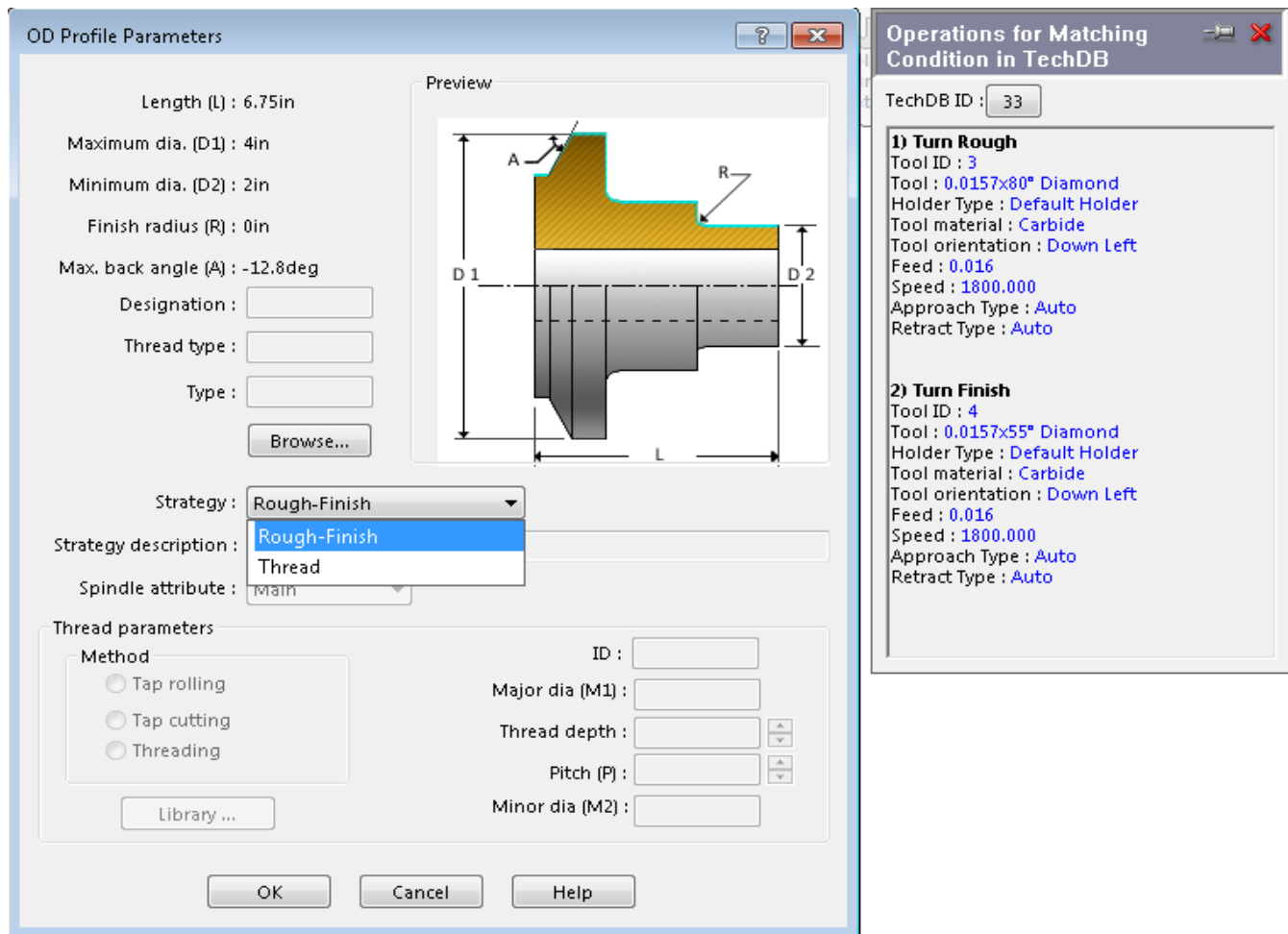
2. The *Parameters* dialog box for the selected feature is displayed.

This dialog box provides a geometric and dimensional report on the feature and allows you to assign additional information to the feature. Depending on the Strategy, a different machining process can be assigned. You can change how each feature is processed by defining the sequence in the Technology Database.

3. View the *Strategy* dropdown list.

The Strategies in the list are user-defined Strategies that have been set up in the TechDB for other operation sequences. After you have learned how to use CAMWorks, you can modify or delete these user-defined Strategies and you can define additional Strategies to customize the TechDB.

Note: Whenever you select a Strategy from the Strategy dropdown list, the operations sequence and associated details will be displayed in a pop-up window next to the Feature Parameters dialog box.



Parameters Dialog Box with Pop-up window displaying operations for selected Strategy



4. Leave the Strategy set to *Rough-Finish*.
This Strategy has been defined to generate Turn Rough and Turn Finish operations using an 80-degree and 55-degree diamond insert tool respectively.
5. Click *OK* to close the dialog box.
6. Review the parameters of other machinable features to understand more about the features.

Step 4: Generating Operations

In this tutorial, you will first generate operations and toolpaths for *OD Feature1*, change its Feature strategy and then regenerate operations and toolpaths. After successfully generating toolpaths for *OD Feature1*, operations will be generated for the remaining turn features.

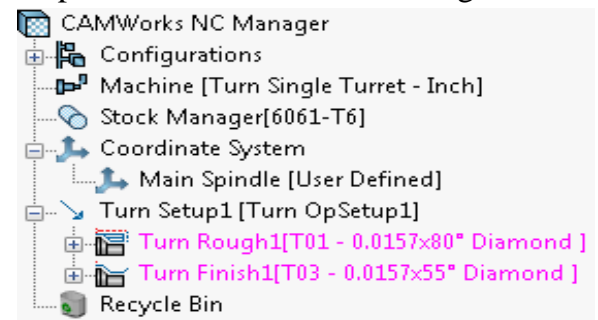
1. Right click *OD Feature1* in the Feature tree and select *Generate Operation Plan* from the context menu.

The Operation tree lists Turn Rough and Turn Finish operations for the OD feature.

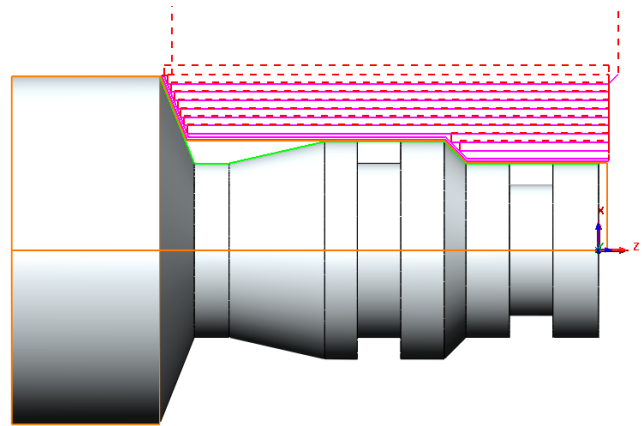
2. Left click the *Turn Rough1* operation in the Operation tree.

Notice the amount of material remaining after the Turn Rough operation. The back angle of the insert does not allow the tool to machine the entire feature.

In order to machine these areas, an additional Turn Rough operation need to be interactively inserted to machine the leftover material.



Operations generated for OD Feature1



Turn Rough Toolpath for OD Feature1

3. Steps to insert Turn Rough operation interactively

- In the CAMWorks Operation tree, right-click on the *Turn Rough1* operation and select *Turning Operations>>Turn Rough* from the context menu.

The *New Operation: Turn Rough* dialog box will be displayed.

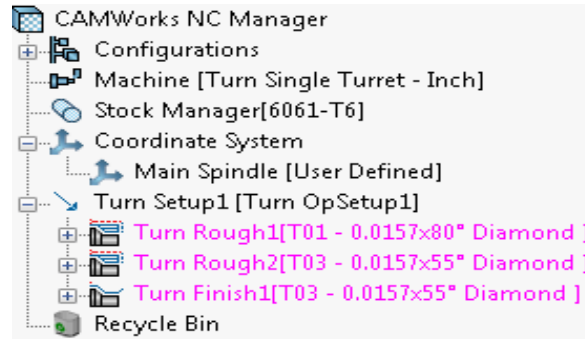
- In the Tool group box, click on the dropdown list and select *Rear Turret*.
- The Tool dropdown list becomes active. Select the *T03 – 0.0157x55.0000deg Diamond* tool.
- Click *OK* to insert the operation.



- The *Operation Parameters* dialog box will be displayed. Click *OK* to close this dialog box.

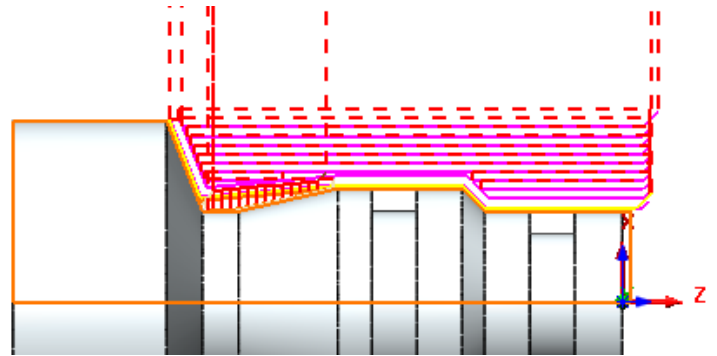
Observe that the *Turn Rough2* operation has been added to the Operation tree.

The Operation tree now lists four operations for *OD Feature1*.




Operations for OD Feature1


4. Right click *Turn Setup1* in the Operation tree and select *Generate Toolpath* from the context menu. Alternatively, you can click on the *Generate Toolpath* command on the CAMWorks Command Manager.
5. Press the *Shift* key and select all the operations listed in the Operation tree.



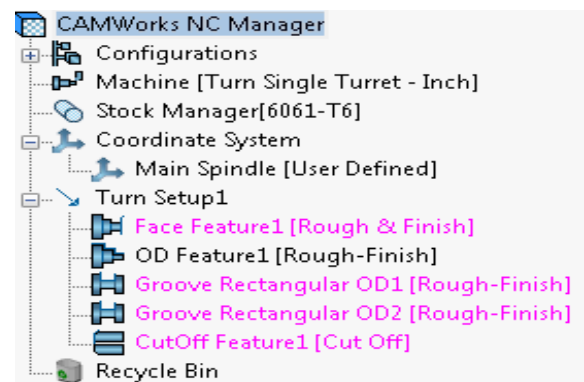
Toolpaths to machine OD Feature1

The toolpaths generated are displayed in the graphics area. Observe the toolpaths for the Turn rough operation. These toolpaths ensure that there is no material leftover.

6.  Click the *CAMWorks Feature Tree* tab to return to the Feature tree.
7. You now need to generate operations for the remaining four features in the Feature tree.

 Click the *Generate Operation Plan* button on the CAMWorks Command Manager/Workflow toolbar.

8. The *Generate Operation Plan* message window is displayed again. Click either *Update* or *Regenerate*. The operations generated when you click either button will be the same.



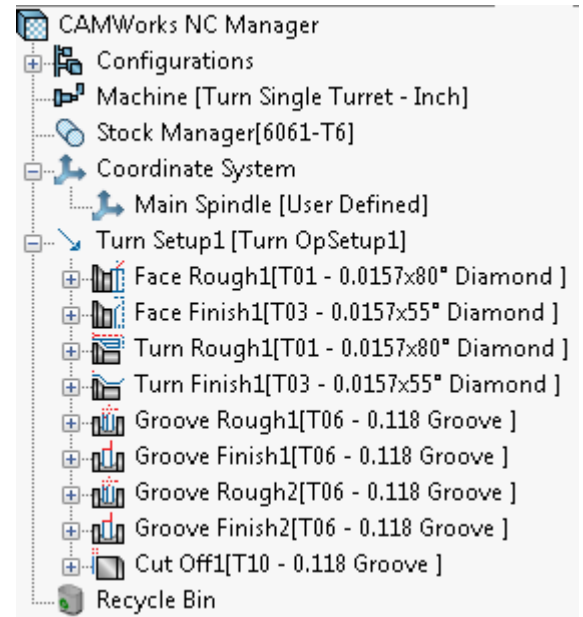
Operations regenerated for OD Feature1



The Operation tree now contains operations for all machinable features in the Turn Setup.

9. Reorder the operations so that the *Face Rough1* and *Face Finish1* operations are at the top of the list.

- **If you have not changed any machining parameters for the operations that have already been generated, you can select either *Regenerate* or *Update*.**
- **If you have modified the operations, select *Regenerate* to delete all previous operations and generate new operations for all the features.**
- **If you have modified the operations, select *Update* to generate operations only for those features that do not have operations.**



Operations generated for all machinable features

10.  Click on the *Generate Toolpath* button on the CAMWorks Command Manager/Workflow toolbar.

Step 5: Generating NC Code

To generate the NC code for this part, you must:

- Adjust the operation parameters as required
- Reorder the operations in the tree if necessary (Refer [Step 8-b](#) of Tutorial 1)
- Generate toolpaths (Refer [Step 7](#) of Tutorial 1)
- Run the Toolpath Simulation (Refer [Step 8](#) of Tutorial 1)
- Readjust operation parameters if necessary
- Post process the part (Refer [Step 9](#) of Tutorial 1)



Did You Know ...

Depending on your machining preferences, you could delete the OD feature and the two rectangular groove features found by AFR, then interactively insert an OD feature that includes the grooves. This feature could be machined completely with the operations generated by the RF80-GR Strategy.



The [next tutorial](#) explains how to insert Turn features interactively.



Turn 3



Topics covered in this tutorial:

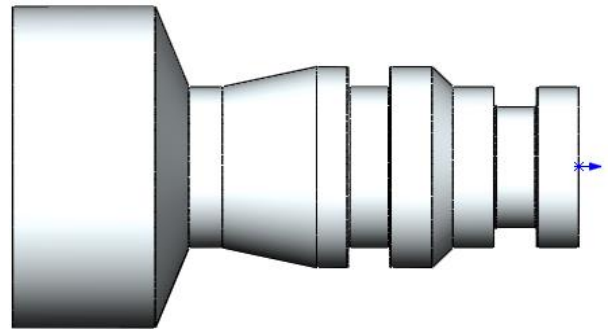
- [Inserting an OD Profile for Threading](#)
- [Reorganizing Machinable Features](#)
- [Suppressing Machinable Features](#)
- [Significance of the CAMWorks Message Window](#)
- [Viewing XZ coordinates of the tool in the CAMWorks Message Window](#)
- [Post Processing Toolpaths](#)

In this tutorial, assume that the part starts out as a casting and machining is required only for the features up to the larger diameter.

In *Turn Setup1*, you face the part, rough and finish the OD profile, rough and finish the rectangular grooves and cut the part off.



Step 1: Open the Part, Define the Machine, Edit the Stock Definition and Extract Machinable Features

1. Open part file **TURN2AX_3.SLDPRT** in the following folder.
Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Turn
2.  Click the *CAMWorks Feature Tree* tab.
3. Under this tree, observe that *Machine [Turn Single Turret- inch]*  is the active machine. In this tutorial, the part will be machined on a Single Turret Turn machine.



TURN2AX_3.SLDPRT

Define the Machine Parameters

1.  Double click the *Machine* item in the Feature tree. OR
 Click on the *Define Machine* button on the CAMWorks Workflow toolbar. The *Machine* dialog box is displayed.
 - On the Machine tab, ensure that the Use sub-spindle option is unchecked.
 - Click the *Tool Crib* tab, ensure *Tool Crib 2 Rear (Inch)* is the active tool crib.
 - Click the *Post Processor* tab and make sure *T2AXIS-TUTORIAL* is selected.
 - Click *OK* to apply the changes and exit the *Machine* dialog box.



Editing the Stock Definition

1. Double click *Stock Manager* item in the Feature tree. OR



Click the *Stock Manager* button in the CAMWorks Workflow toolbar.

The *Stock Manager* dialog box will be displayed.

2. In the *Bar stock parameters* group box, change the Stock Length of the bar stock to **6.85in.**

By default, the Stock shape is a cylinder whose dimensions enclose the part. For this tutorial, the part is machined from bar stock that has **0.10in** material to be removed from the face. Assume that the largest OD of the part is to size and is not machined in this Setup. Since CAMWorks currently defines the Stock as round bar stock, you will use the default round shape and offset the minimum Stock size by **0.10in** in length.

4. Click *OK* to apply the changes and close the dialog box.

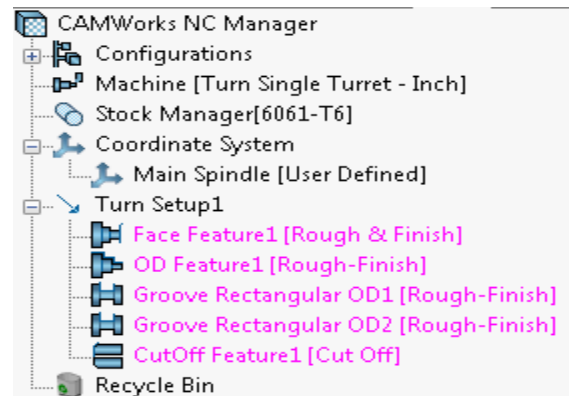
Extract machinable Features



Click the *Extract Machinable Features* button on the CAMWorks Command Manager/Workflow toolbar. OR

Right click *CAMWorks NC Manager* in the Feature tree and select *Extract Machinable Features* from the context menu.

On executing this action, AFR creates *Turn Setup1*. It recognizes five features and lists them in the Feature tree.



Turn features recognized by AFR as listed in the Feature Tree

Step 2: Interactively Inserting an OD Profile for Threading

Now that a Turn Setup has been defined, you can create an OD profile feature to thread the part. This OD feature with thread will be inserted interactively using the *New Turn Feature* command.

1. Right click *OD Feature1* in the Feature tree and select *Turn Feature* from the context menu.

The *New Turn Feature* dialog box is displayed. The part profile is shown in the graphics area. Notice that as you move your cursor over the profile, the entities change color and are highlighted.

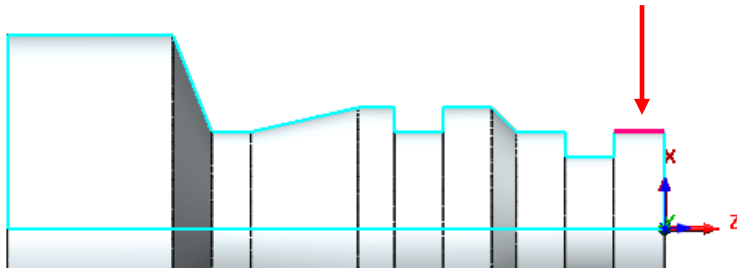
2. In the *Features* group box of this dialog box, assign the following parameters:

- Select *OD Feature* for Type.
- Click on the dropdown list for *Strategy* and select *Thread*.

(The Thread Strategy has been defined in the TechDB to generate a thread operation.)



3. Since the part does not contain any non-revolved features, leave the *Part Profile* set to *Plane Section*.
4. In this tutorial, we will interactively define a feature picking the segment(s) on the highlighted geometry of the part. Hence, in the *Define from* group box, leave the method set to *<Part Profile>*.
5. Turn features can be defined by an outside shape and optionally inside shapes to represent profiles. These shapes can be sketches or part



Pick the first OD profile Segment

profiles that are normal to the current Setup.


- In the graphics area, pick the *OD profile* segment on the right at the part face to select it. The single segment is highlighted.
 - The selected segment (face) will be listed in the *Selected entities* list box of the dialog box as *CW Face-1*. *Extend 1* and *Extend 2*, which are parameters used to determine how the toolpath is computed at the ends of the feature, are also listed in this box.
6. Make sure that *Extend1* and *Extend 2* are set to *None* in their respective group boxes.



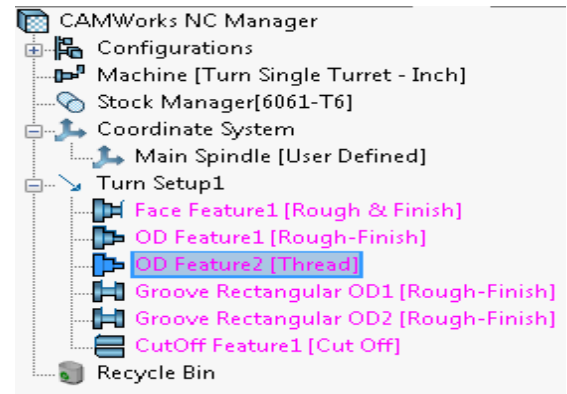
If CAMWorks is calculating the Work In Process (WIP), then the ID and OD Thread features should not be extended in either Z or X direction. If you extent the Thread features, CAMWorks determines the feature length relative to the WIP and may extent the feature to start threading closer to the face than you want. Typically, this would occur when the thread feature is not located at the face of the part.

New Turn Feature Dialog box



7.  Click *OK* to insert the feature and close the *New Turn Feature* dialog box.

A feature labeled *OD Feature2* is displayed in the Feature tree under *OD Feature1*. You have now defined all the machinable features in this part.



OD Feature2 in Feature Tree

Step 3: Reorganizing Machinable Features

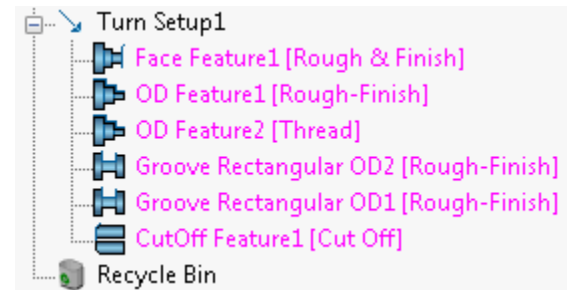
To change the machining order, you can reorder either the machinable features or the operations that are generated for the features. You can drag a feature in the Feature tree from the current location to a position above or below another feature in the same Setup.

1. Left click *Groove Rectangular OD1*, then click *Groove Rectangular OD2*. Observe the order the grooves are highlighted on the part model in the graphics area.

This may not be the order you want to machine these features.

2. Highlight *Groove Rectangular OD2*; hold down the left mouse button and drag and release it over *OD Feature2*.

Notice that as you drag the item, an arrow displays to indicate where the item will be positioned. *Groove Rectangular OD2* will now be listed before *Groove Rectangular OD1* and will be machined first.

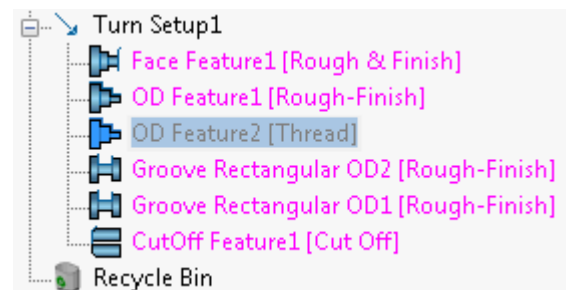


Reorganized list of Machinable Features

Step 4: Suppressing Machinable Features

You can suppress a machinable feature and machine the feature later. Suppressing a feature prevents operations (and associated toolpaths) from being created for that feature. For example, if there is not enough tool capacity to do threading on the machine which you are using, you can suppress *OD Feature2* and output the thread operation separately.

1. Right click *OD Feature2* in the Feature tree and select *Suppress* on the context menu. This action grays out *OD Feature2* icon in the tree.



Suppressed OD Feature2



Step 5: Saving the File

1. Click on the *File* menu and select *Save As* on the dropdown menu.
2. The *Save As* dialog box is displayed. Browse to the location where you wish to save the file.
3. If you wish to save the file with another name rather than the default name, assign another file name
4. Click the *Save* button.

Step 6: Generating an Operation Plan and Toolpaths



1.  Click the *Generate Operation Plan* button on the CAMWorks Command Manager/Workflow toolbar.

OR

Right click *CAMWorks NC Manager* in the Feature tree and select *Generate Operation Plan* on the context menu.

Operations are created for the machinable features in the Turn Setup.

The generated operations are listed in the Operation tree.

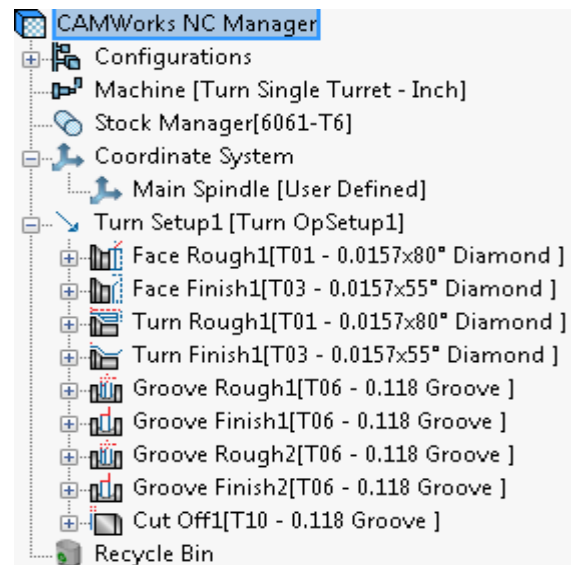
2. Click on the  sign against each operation listed in the Operation tree to view the feature machined by that selected operation. Observe that no operations were generated for *OD Feature2* as this feature was suppressed.
3.  Click the *Generate Toolpath* button on the CAMWorks Command Manager/Workflow toolbar.

OR

Right click *Turn Setup1* in the Operation tree and select *Generate Toolpath* from the context menu.

The change in the font color of the listed operations from **Magenta** to **Black** (default color settings) indicates that toolpaths have been generated for all operations.

4. Experiment by changing parameters for the operations and regenerating the toolpaths for all operations in the Setup.



Operations with toolpaths generated for all Turn features listed in the Operation tree



Did You Know ...


When you change operation parameters after generating toolpaths, we recommend that you regenerate toolpaths at the Setup level so the work in process can be updated correctly.

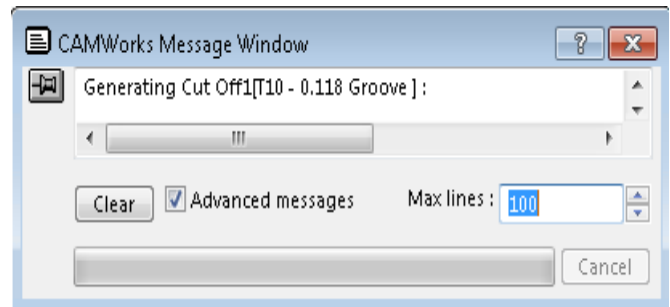


CAMWorks Message Window




Check the CAMWorks Message Window.

- This CAMWorks window displays automatically whenever CAMWorks commands such as *Extract Machinable Features*, *Generate Operation Plan*, *Generate Toolpath* etc. are executed.
- The Message Window provides information you may find helpful. For example, based on the information in this window, you may want to change a tool or insert a finish operation in order to cut a machinable feature completely.
- If the Message Window is not displayed, you can do any one the following to display the Message Window:
 - Select *Options* on the CAMWorks menu or the *CAMWorks NC Manager* context menu. The Options dialog box will be displayed.
 - On the *General* tab, check the *Message Window* check box.
 -  Click on the *Message Window* button on the CAMWorks Command Manager.
- You can move and resize the window. To change the size, point to a border or corner of the window (the pointer changes to a two-headed arrow). Drag the corner or border until the window is the size you want and then release the mouse button.
- For turn and mill-turn parts, CAMWorks provides an option to display the XZ position of the tool in the Message Window during simulation. How to use this functionality is explained in the Step 7 of this tutorial in the section [Viewing XZ Cutter coordinates of the Tool in the Message Window](#).











CAMWorks Message Window

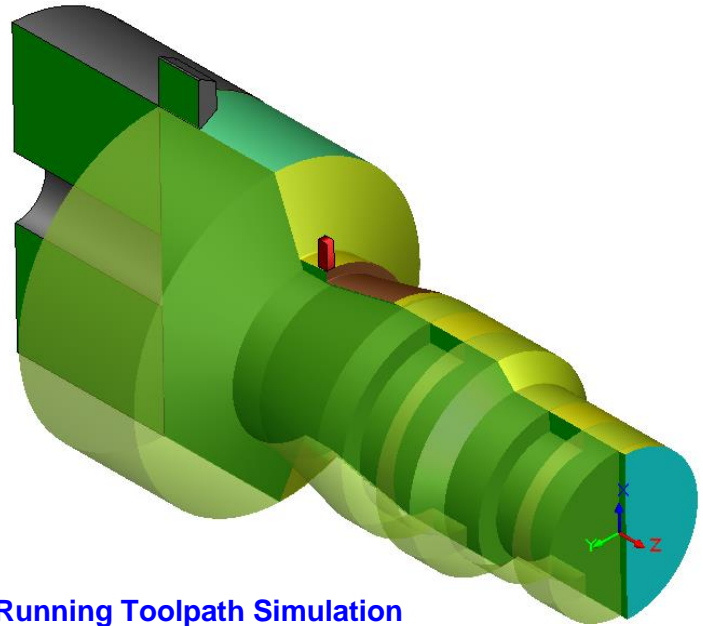
Step 7: Simulate Toolpaths

1.  Click the *Simulate Toolpath* button on the CAMWorks Command Manager/Workflow toolbar. OR
Select *Simulate Toolpath* from the CAMWorks menu.
OR
Right click *CAMWorks NC Manager* in the Operation tree and select *Simulate Toolpath* on the context menu.
2. The *Toolpath Simulation* toolbar is displayed. The graphics area enters Simulation mode.
3. Select the following display options:



-  Stock: *Shaded Display*
-  Tool: *Shaded with Edges Display*
-  Tool Holder: *No Display*
-  Target Part: *Translucent Display*
-  Fixture: *Shaded with Edges*
-  Section View: *Half*


4. Rotate the part, if necessary, so that you can see the simulation.
5.  Click the *Run* button.
6. Click the *OK*  button in the upper right corner of the *Toolpath Simulation* toolbar to exit Simulation mode.

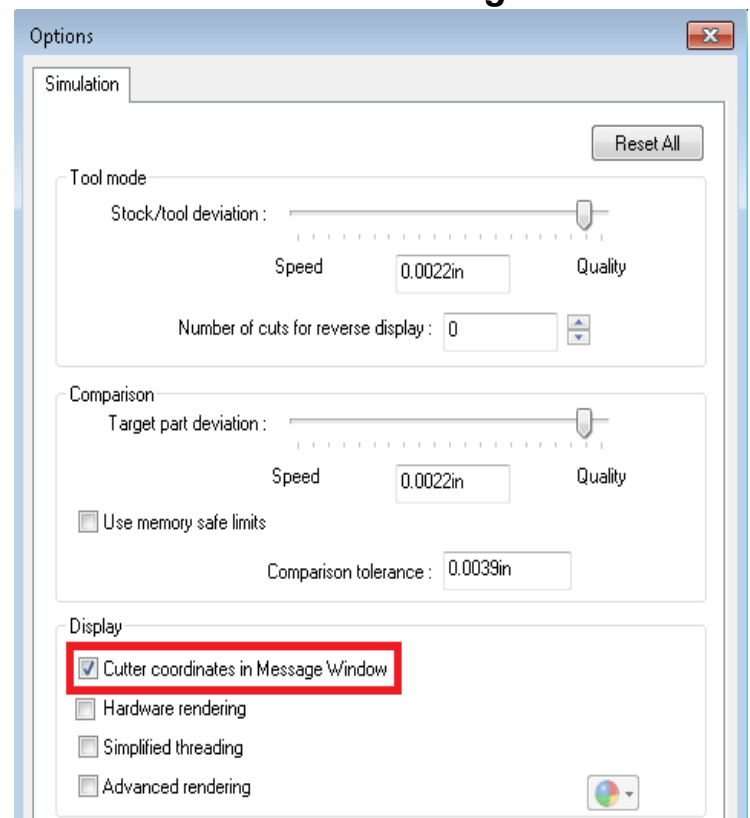


Running Toolpath Simulation

Viewing XZ cutter coordinates of the Tool in CAMWorks Message Window




For turn and mill-turn parts, CAMWorks provides an option to display the XZ position of the tool in the Message Window during toolpath simulation. To view the coordinates,

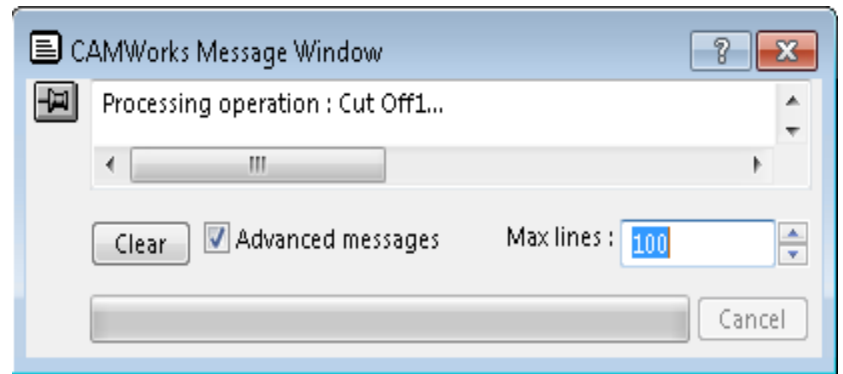
1.  Click on the *Options* button in the *Toolpath Simulation* toolbar.
2. The Simulation Tab of the Options dialog box is displayed. In the Display group box, ensure that the option *Cutter Coordinates in Message Window* is checked.
3. Click *OK* to close the *Options* dialog box.
4. Click *Yes* to restart the Simulation mode when CAMWorks prompts a warning message in order to apply the new simulation settings.



Simulation Tab of Options dialog box.






5.  In the Simulation toolbar, click the *Cutter coordinates* button.
6.  The X coordinate is displayed as a Radial value by default. Click on the *Radial or diameter coordinate display* button to view the X coordinate as a diameter value.
7.  The XZ coordinates are output to the tool center by default. To output these coordinates to the theoretical Tool Tip, click on the *Tip or center coordinate display* button.
8. Run the Simulation. Observe that the XZ coordinates of the tool are output in the CAMWorks Message Window.




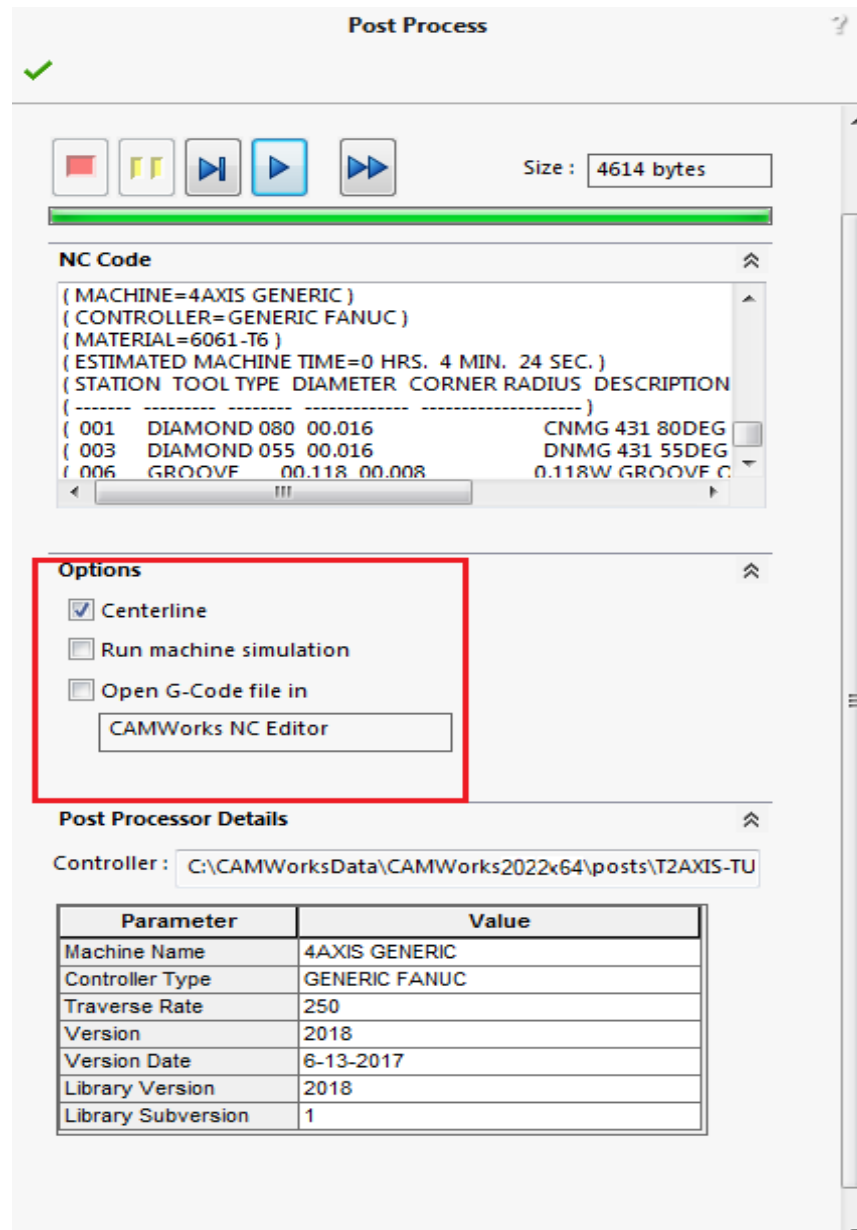
CAMWorks Message Window displaying XZ coordinates

Step 8: Post Processing Toolpaths

1.  Click the *Post Process* button on the CAMWorks Command Manager/Workflow Toolbar.
OR
Right click *Turn Setup1* in the Operation tree and select *Post Process* from the context menu.
OR
Click the *Post Process* on the CAMWorks menu.
2. The *Post Output File* dialog box is displayed. Browse to the folder where you wish to save the file.
 **If you are running CAMWorks in Demo mode, the Post Process functionality will be disabled.**
3. If you do not wish to save the file with the default file name, assign another file name in the *File name* field.
4. Click *Save* to save the file.
5. The *Post Process Output* dialog box is displayed. In the Options group box, check the *Centerline* option so that the toolpath will be highlighted as each line of code is generated.
6.  Click the *Play* button.
This command generates the NC code. The generated NC code can be viewed in the *NC code output* area of the dialog box.



7. After viewing the code, click **OK**  to close the dialog box.



The screenshot shows the 'Post Process' dialog box. It has a title bar with a question mark icon. Below the title bar is a green checkmark icon. There are five buttons: a red square, a yellow square, a blue square with a left arrow, a blue square with a right arrow, and a blue square with a double right arrow. To the right of these buttons is a 'Size' field showing '4614 bytes'. Below the buttons is a green progress bar. The main area is divided into three sections: 'NC Code', 'Options', and 'Post Processor Details'. The 'NC Code' section shows a list of G-code commands. The 'Options' section has three checkboxes: 'Centerline' (checked), 'Run machine simulation' (unchecked), and 'Open G-Code file in' (unchecked). The 'Post Processor Details' section has a 'Controller' field and a table of parameters.

NC Code

```
( MACHINE=4AXIS GENERIC )  
( CONTROLLER=GENERIC FANUC )  
( MATERIAL=6061-T6 )  
( ESTIMATED MACHINE TIME=0 HRS. 4 MIN. 24 SEC. )  
( STATION TOOL TYPE DIAMETER CORNER RADIUS DESCRIPTION )  
( ----- )  
( 001 DIAMOND 080 00.016 CNMG 431 80DEG )  
( 003 DIAMOND 055 00.016 DNMG 431 55DEG )  
( 006 GROOVE 00.118 00.008 0.118W GROOVE C )
```

Options

- ☒ Centerline
- ☐ Run machine simulation
- ☐ Open G-Code file in
CAMWorks NC Editor

Post Processor Details

Controller : C:\CAMWorksData\CAMWorks2022x64\posts\T2AXIS-TU

Parameter	Value
Machine Name	4AXIS GENERIC
Controller Type	GENERIC FANUC
Traverse Rate	250
Version	2018
Version Date	6-13-2017
Library Version	2018
Library Subversion	1

Post Process dialog box



Did You Know ...

You should save the part frequently. If you want the CAM information saved with the part, make sure that the Save/Restore part option is checked on the General tab in the Options dialog box before you save. When you open the part, make sure that Save/Restore is checked or the CAM information will not be restored.

The General tab in the Options dialog box also has an Auto save option for automatically saving your CAMWorks data.



Turn 4

Topics covered in this tutorial:




- [Defining the Stock as a Forging or Casting](#)
- [Defining a Rectangular OD Groove Feature interactively](#)
- [Deleting an Operation](#)
- [Adjusting Operation Parameters](#)
- [Defining Program Zero](#)

This tutorial guides you through the steps for the sequence of machining the turn part when the stock is defined from a forging or casting. The machining sequence is:


- Rough and finish the face of the part
- Rough and finish the OD profile
- Rough and finish the groove

Step 1: Open the Part and Define the Machine

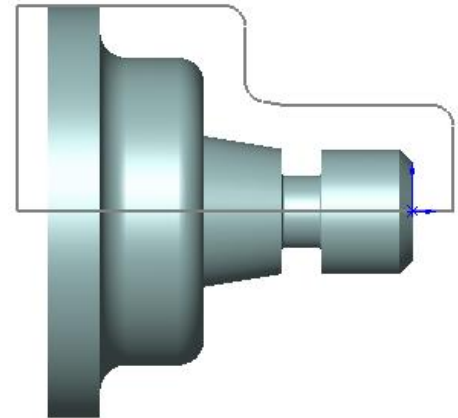
1. Open part file **TURN2AX_4.SLDPRT** in the following folder.
Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Turn

2.  Click the *CAMWorks Feature Tree* tab.
3. Under this tree, observe that *Machine [Turn Single Turret- inch]*  is the active machine. In this tutorial, the part will be machined on a Single Turret Turn machine.
4.  Double click the *Machine* item in the Feature tree.

OR

-  Click on the *Define Machine* button on the CAMWorks Workflow toolbar. The *Machine* dialog box is displayed.

- On the Machine tab, ensure that the *Use sub-spindle* option is unchecked.
- Click the *Tool Crib* tab, ensure *Tool Crib 2 Rear (Inch)* is the active tool crib.
- Click the *Post Processor* tab and make sure *T2AXIS-TUTORIAL* is selected.
- Click *OK* to apply the changes and exit the *Machine* dialog box.



TURN2AX_4.SLDPRT

Step 2: Defining the Stock as a Forging or Casting

In this tutorial, assume that the stock of the part is a forging. A sketch named *Stock Profile* has already been created in this part to represent the forging.



1. In the Feature tree,  double click the *Stock Manager* item to open the *Stock Manager* dialog box.

OR



Click the *Stock Manager* button in the CAMWorks Workflow toolbar.

By default, CAMWorks automatically calculates the Stock size based on the smallest cylinder possible. (Default Stock Type: *Round Bar Stock*). You can change this definition by specifying a sketch to use (Stock Type: *From Revolved Sketch*).

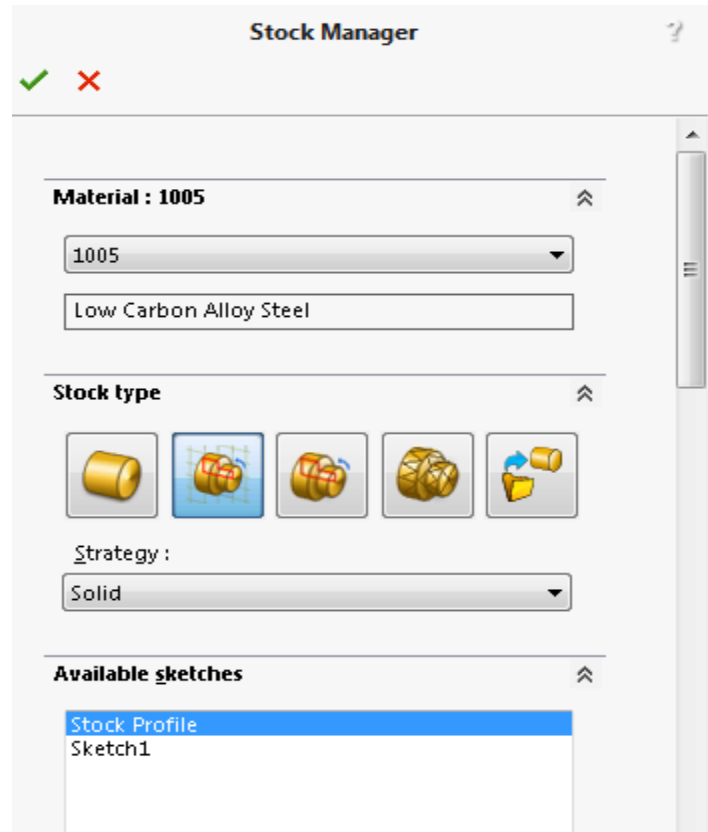


2. Select *From Revolved Sketch* for the Stock type.

3. In the *Available sketches* list, select *Stock Profile*.

The profile sketch representing the shape of the stock is highlighted on the part model in the graphics area.

4.  Click *OK* to apply the changes and close the *Manage Stock* dialog box.



Stock Manager dialog box

Step 3: Recognizing Machinable Features using AFR



Click the *Extract Machinable Features* button on the CAMWorks Command Manager/Workflow Toolbar.

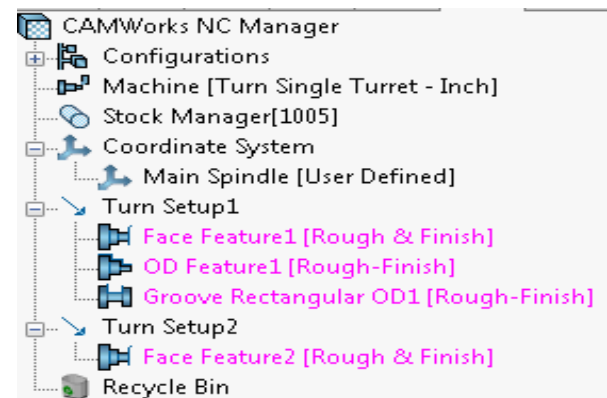
OR

Select *Extract Machinable Features* from the CAMWorks menu.

OR

Right click *CAMWorks NC Manager* in the Feature tree and select *Extract Machinable Features* from the context menu.

[AFR](#) creates *Turn Setup1* and *Turn Setup2* and lists all the recognized features in Feature tree. The recognized features under *Turn Setup1* include a Face Feature, an OD Feature and a Rectangular OD groove.



Rectangular OD groove feature recognized by AFR



Turn Tutorial

Turn Setup2 is created as a separate setup for the second side of the part, which includes a Face feature.



When the Stock is defined from a sketch, CAMWorks creates a second setup with a Face feature.


Deleting Unwanted Features Recognized by AFR

When AFR is run on some parts, all the features you want to machine might not be recognized. In certain cases, AFR may recognize a feature that might not be suitable for the intended machining process. When this occurs, you can delete unwanted features and define features interactively.

For learning purposes, in this tutorial, you will delete the *Rectangular OD groove* feature and insert the same feature interactively.

1. Right click *Groove Rectangular OD1* in the tree and select *Delete* on the context menu.
2. Click *Yes* to confirm the deletion.

The deleted feature is moved to the Recycle Bin.

3. Click the  sign to the left of the Recycle Bin in the Feature tree to collapse it.



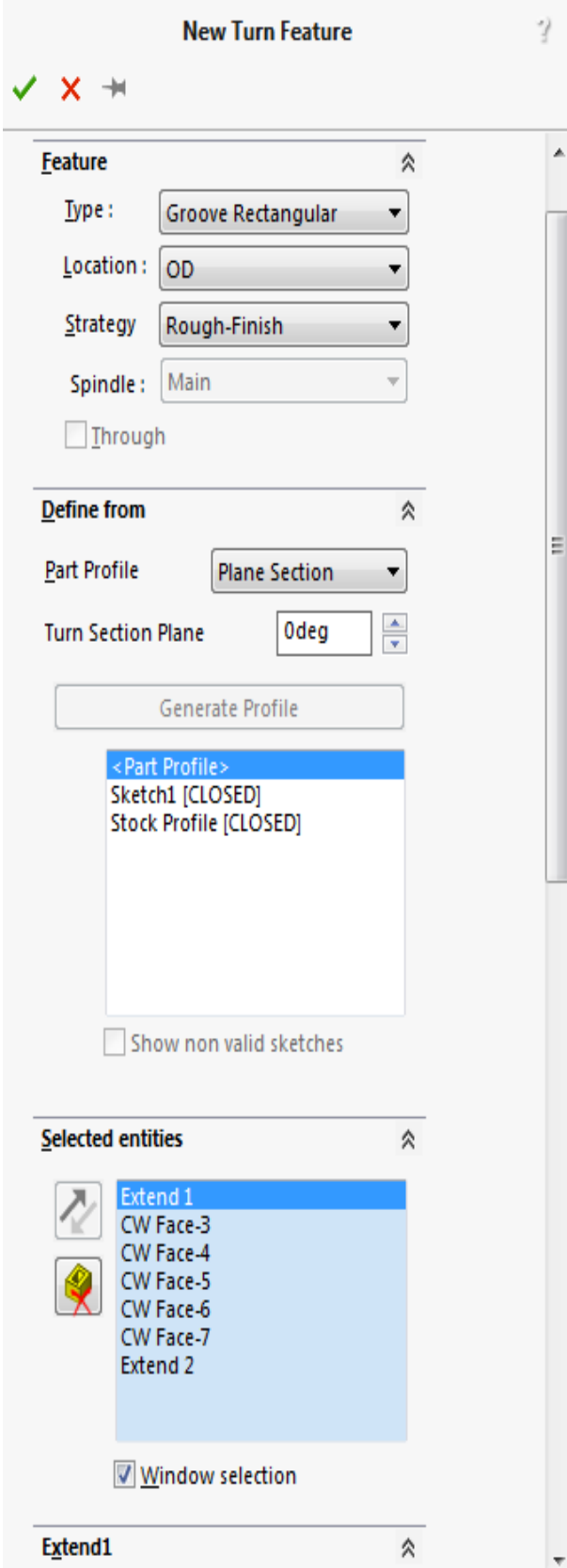
Deleted features are automatically placed in the Recycle Bin, which is used to store machinable features that you do not intend to machine.

Step 4: Defining a Rectangular OD Groove Feature Interactively

Following are the steps to interactively insert a *Rectangular OD groove* feature. This feature is defined in order to machine the undercut.

1. Right click *OD Feature1* under *Turn Setup1* in the Feature tree and select *Turn Feature* from the context menu.

The *New Turn Feature* dialog box is



The **New Turn Feature** dialog box is shown with the following settings:

- Feature**
 - Type: Groove Rectangular
 - Location: OD
 - Strategy: Rough-Finish
 - Spindle: Main
 - ☐ Through
- Define from**
 - Part Profile: Plane Section
 - Turn Section Plane: 0deg
 -
 - Part Profile list:
 - <Part Profile>
 - Sketch1 [CLOSED]
 - Stock Profile [CLOSED]
 - ☐ Show non valid sketches
- Selected entities**
 - ☒ Extend 1
 - CW Face-3
 - CW Face-4
 - CW Face-5
 - CW Face-6
 - CW Face-7
 - Extend 2
 - ☒ Window selection

At the bottom, the feature name **Extend1** is displayed.

New Turn Feature Dialog box



displayed.

2. In the *Feature* group box:
 - Set the Feature Type to *Groove Rectangular*.
 - Leave the Location set to *OD*.
 - Make sure the Strategy is set to *Rough-Finish*.
3. In the *Define from* group box,
 - Ensure that the Part Profile is set to *Plane Section*.



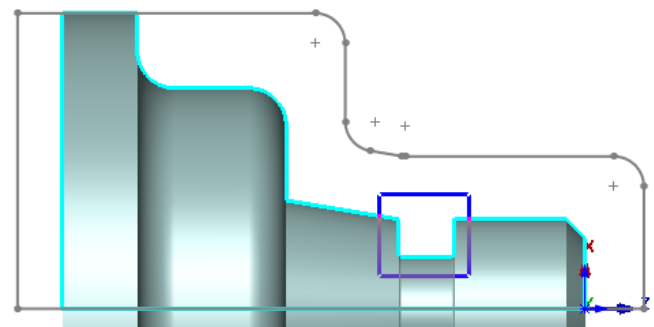
If the part does not contain any non-revolved feature(s), then you can set the Part Profile to either *Plane Section* or *Revolved Section*. The Part Profile created in both cases will be the same. Ideally:

- Use *Plane section* method when the part does not contain any non-revolved features
 - Use *Revolved section* method when the part contains non-revolved features or mill features.
- In the list box, ensure *<Part Profile>* is selected.



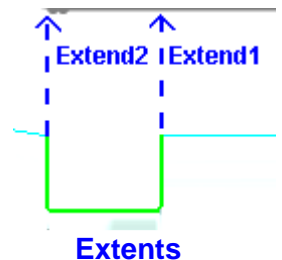
When *<Part Profile>* is selected in the Define From list box, all profiles in the current section plane are highlighted on the model in the graphics area. The profiles recognized are based on the *Part Profile Method* selected.


4. In the *Selected entities* list box,
 - Check the *Windows selection* check box.
 - In the graphics area, use the mouse to select the geometry for the groove by drawing a window around the groove entities as shown in the figure on the right.
 - The *Selected entities* list box will be updated to contain 5 selected faces entities and 2 extent entities.
 - Highlight *Extentd1* in the *Selected entities* list box.



Window Select the Rectangular OD Groove Feature

- The *Extend1* and *Extend2* group boxes will be expanded.
 - In the *Extend1* group box, set the direction to *Along X*.
 - In the *Extend2* group box, set the direction to *Along X*.
- The extents will be displayed on the model part in the graphics area.



5.  Click *OK* to insert the feature.



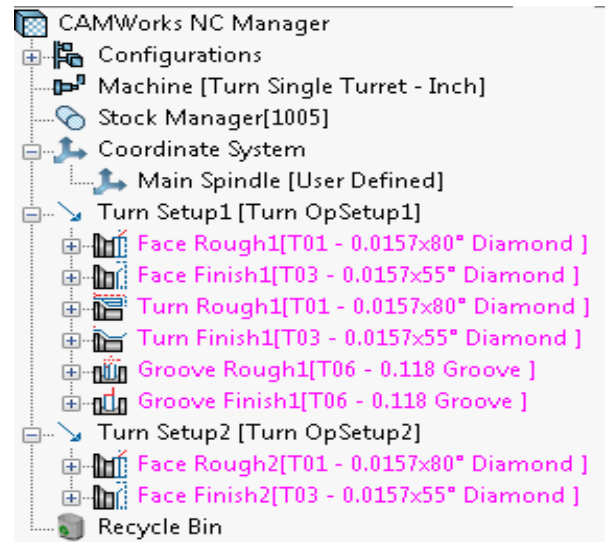
Groove Rectangular OD2 is added to the Feature tree.

- Click the *Generate Operation Plan* button on the CAMWorks Command Manager/Workflow Toolbar.

OR

Right click *CAMWorks NC Manager* and select *Generate Operation Plan* from the context menu.

The operations for *Turn Setup1* and *Turn Setup2* are listed in the CAMWorks Operation tree.



CAMWorks Operation Tree

Step 5: Deleting an Operation

In some cases, it may be desirable to delete operations that are created automatically. In this tutorial, all of the operations that were generated for *Turn Setup1* are required to machine the part.

If the stock is a casting or forging and you use a sketch to define the Stock, CAMWorks generates a second setup for the other side of the part. Depending on whether or not a second setup is needed for this machining session, you can delete or suppress the operations that were generated for *Turn Setup2*. In this tutorial, you will delete the second setup and all the operations under it.


- In the Operation tree, right click *Turn Setup2* and select *Delete* on the context menu.
- Click *Yes* to confirm you want to delete the Setup and all dependent operations under it.

The Setup is deleted and the operations are moved to the *Recycle Bin*.


- Click the  sign to the left of the Recycle Bin in the Operation tree to collapse it.

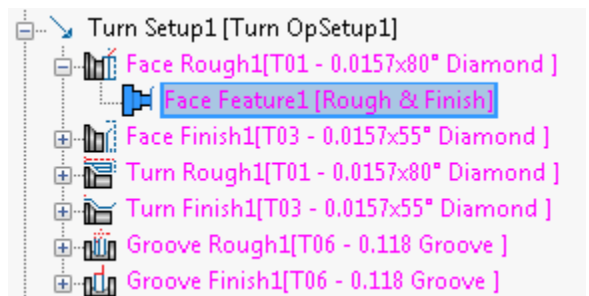
Step 6: Adjusting Operation Parameters

Each operation contains machining parameters that affect how the toolpath is created and specific parameters that will be output to the NC program. These parameters can be edited before generating the toolpaths and post processing the part.

- Click the  sign next to *Face Rough1* in the Operation tree.

The tree expands to identify the machinable feature the operation was generated to machine (*Face Feature1*).

- Click the  sign next to *Face Rough1* in order



Expanding the tree



to collapse it.

3. Double click *Face Rough1*.

The Operation Parameters dialog box is displayed. In the Operation Parameters dialog box:

- Click on the *NC* tab.
- Within the *Clearance* group box, change the *Retract dist.* to **0.15in**.

This parameter specifies the distance away from the part to start the cycle in the X and Z axis. This is where the tool rapids to at the beginning of the cycle and the distance the tool retracts from the part after each cutting pass.

- Click *OK* to apply the changes and close the dialog box.

Step 7: Displaying the Chuck in Graphics Area

You can choose to display the chuck in the graphics area in any one of the following display states:

- None (*no display*)
- Wireframe
- Translucent
- Shaded
- Shaded with Edges

In this tutorial, you will display the chuck in a *Shaded with edges* display state.

Following are the steps to display the chuck:

1. Switch to the Operation tree.
2. Right click the machine item *Machine [Turn Single Turret-inch]* in the Operation tree and select *Chuck/Fixture Display>>Shaded with Edges* from the cascading context menu.
3. Once again, right click on this item and ensure that *Chuck/Fixture Display>>Machine Node* and *Chuck/Fixture Display>>Setup Node* options in the cascading context menu are checked.

To check/uncheck this option, left click on the desired option in the cascading menu.

The above action ensures that the chuck is displayed whenever the Machine item or Turn Setup is selected in the Operations tree.

Step 8: Defining Program Zero

Each Setup has a different origin. The Turn Setup defines the tool direction and Program Zero. Program Zero can be changed in the Setup by picking one of the following that is closest to the desired work *Z0* face:

- An OD that is concentric to the part. OR
- Angle that is concentric to the part. OR

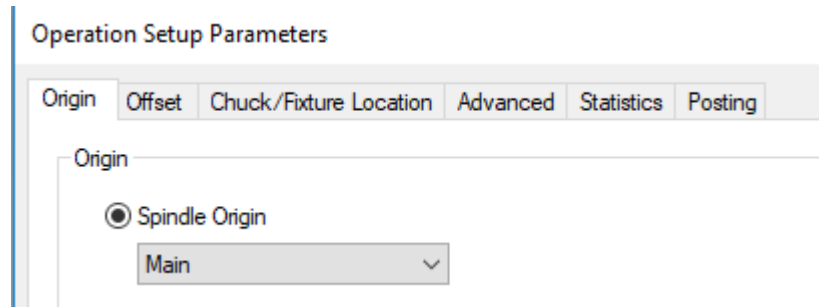


- ID face that is concentric to the part.

Following are the steps to change the Program Zero:

1. Double click *Turn Setup1* in the Operation tree.

The *Operation Setup Parameters* dialog box is displayed.



Operation Setup Parameters dialog box

2. On the *Origin* tab:
 - Select *Spindle Origin* in the *Origin* list box.
and select *Main* from the *Spindle Origin* dropdown list, change *offset along spindle axis* to 0in.
 - The *Chuck Location* and *Chuck Definition* tabs are explained in the next lesson.
 - Click *OK* to apply the changes and close the dialog box.

3. Click on the *Yes* button if you get a pop-up message prompting you to regenerate the toolpaths.

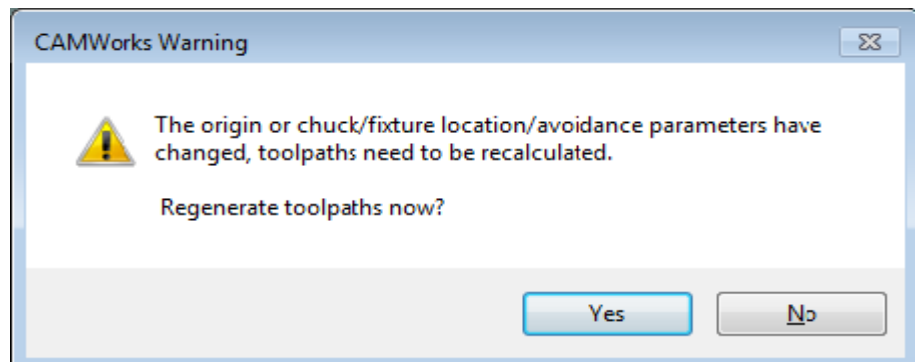
OR



Click the *Generate Toolpath* button on the CAMWorks Command Manager/Workflow toolbar.

Toolpaths are generated for all operations.

4. Click each operation in the tree to view the toolpaths.



Warning message to regenerate toolpaths



Operation Setup Parameters

Origin Offset Chuck/Fixture Location Advanced Statistics Posting

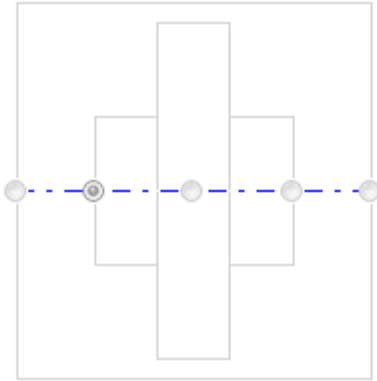
Oigin

☒ Spindle Origin
Main

☐ Entity

☐ Part Vertex

☐ Stock vertex



Offset along spindle axis:
0in

Origin Coordinates

X: -0in
Y: -0in
Z: -0in










Z Axis machining direction

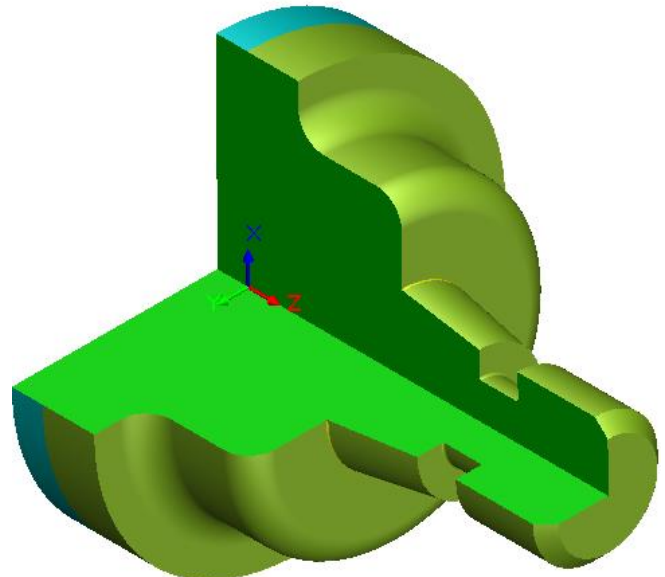
☐ Reverse direction

OK Cancel Help



Step 9: Simulate Toolpath

1.  Click on the View Orientation and select  the *Isometric* view orientation.
2. Right click *Turn Setup1* and select *Simulate Toolpath* from the context menu.
The Toolpath Simulation toolbar is displayed.
3. Select the following display options:
 -  Stock: *Shaded Display*
 -  Tool: *Shaded with Edges Display*
 -  Target: *Translucent Display*
 -  Fixture: *No Display*
 -  Section View: *Three Quarter*
4.  Click the *Run* button.
Since you deleted *Turn Setup2* and the Face Feature, material is left on the locating face. This could be machined using another setup.
5. Click the *OK*  button to exit the simulation mode.
6. Using 'drag and drop' feature, reorder the operations to cut the part in the order you think is most efficient.
7. Click on the *File* menu and select *Save As* to save the part using an appropriate file name.



Toolpath Simulation

Step 9: Post Processing Toolpaths

Post process the toolpaths in the same manner as explained in [Step 8 of the previous tutorial](#) (Tutorial 3) of this chapter.



Turn 5

Topics covered in this tutorial:





- [Adding an ID Groove Feature](#)
- [Adding OD Thread and ID Thread Features](#)
- [Displaying the Chuck](#)
- [Defining the Chuck](#)
- [Defining the Chuck Clamping Location](#)
- [Viewing the Chuck in Toolpath Simulation](#)

Step 1: Opening the Part and Defining the Machine and Stock

Open the Part

1. Open part file **TURN2AX_5.SLDPRT** in the following folder.
Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Turn

Define the Machine Parameters

1.  Click the *CAMWorks Feature Tree* tab.
2.  In the *Machine* item of the Feature tree, observe that the *Turn Single Turret- inch*  is selected.
3. Under this tree, observe that *Machine [Turn Single Turret- inch]*  is the active machine. In this tutorial, the part will be machined on a Single Turret Turn machine.
4. Double-click on the Machine item.

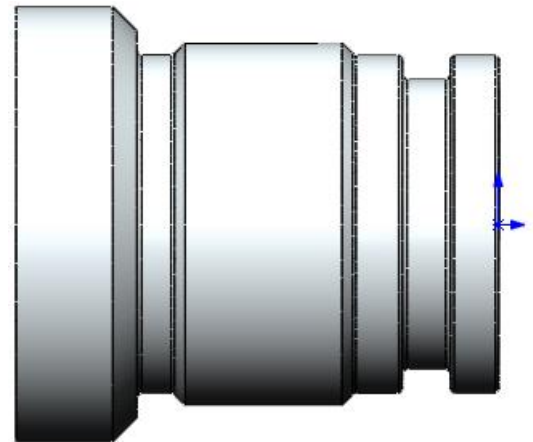
OR



Click on the *Define Machine* button on the CAMWorks Workflow toolbar.

The *Machine* dialog box will be displayed.

- On the Machine tab, ensure that the *Use sub-spindle* option is unchecked.
- Click the *Tool Crib* tab, ensure *Tool Crib 2 Rear (Inch)* is the active tool crib.
- Click the *Post Processor* tab and make sure *T2AXIS-TUTORIAL* is selected.
- Click *OK* to apply the changes and exit the *Machine* dialog box.

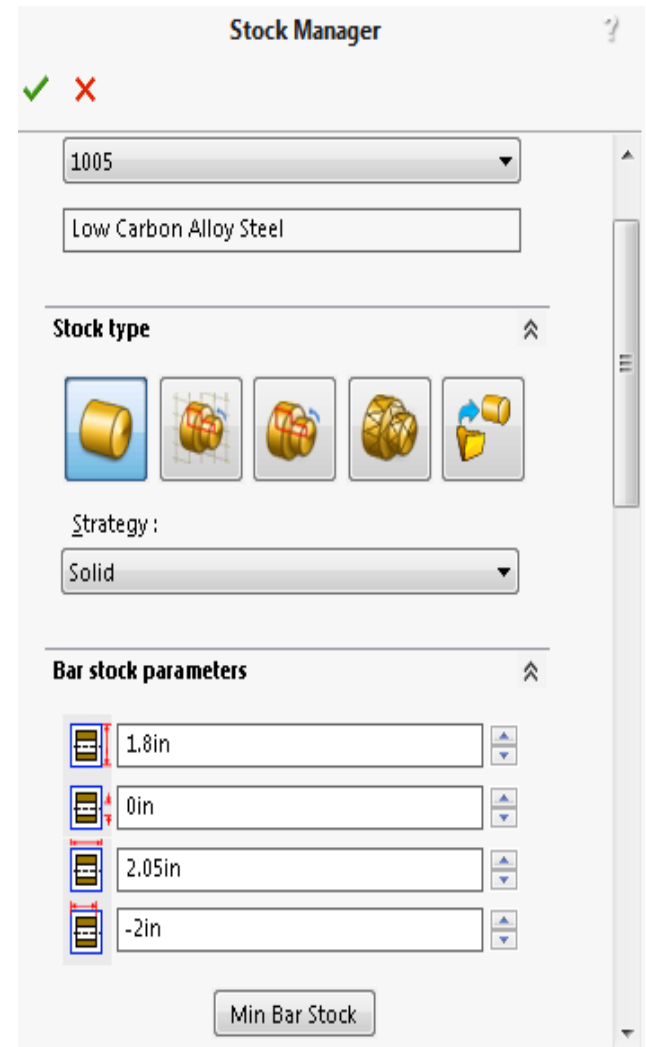


TURN2AX_5.SLDPRT



Defining the Stock

1. Double click on *Stock Manager* item in the Feature tree.
OR
 Click the *Stock Manager* button in the CAMWorks Workflow toolbar.
2. The Stock Manager dialog box is displayed. In this dialog box,
 - Leave the Stock Type set to *Bar stock*.
 - Change the Stock Length to **2.05in**
 - Click the down arrow to the right of the *Material* list and select the desired material from the dropdown list or use the default material.
 - Click *OK* to apply the changes and close the dialog box.



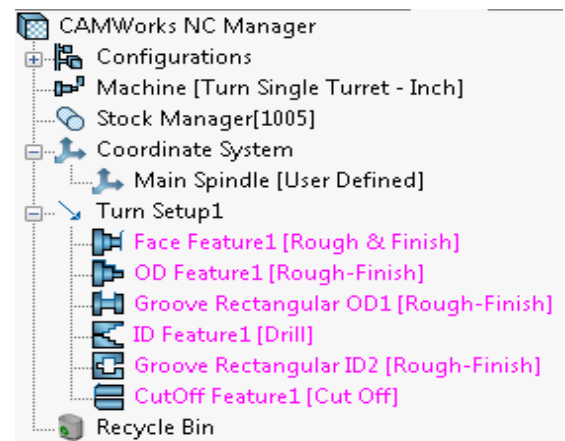
Stock Manager Dialog box

Step 2: Extract Machinable Features

- Click the *Extract Machinable Features* button on the CAMWorks Command Manager/Workflow toolbar.

AFR analyzes the model for features to machine. When AFR is completed, the tree displays *Turn Setup1*, which establishes the workplane for turning, and the machinable features that were recognized.


When AFR is run on some parts, CAMWorks may not recognize all the features you want to machine or AFR may recognize a feature that might not be suitable for the intended machining process. When this occurs, you can delete unwanted features and define required features interactively.



Features Recognized using EMF



Deleting Features

1. Right click the *Groove Rectangular ID2* feature and select *Delete* on the context menu. For learning purposes, in this tutorial, you will delete the *Groove Rectangular ID2* feature and insert it interactively.
2. Since you will be defining a chuck for this part, also delete *Cutoff Feature1*. The deleted features will be moved to the *Recycle Bin*.
3. Click the  sign to the left of the Recycle Bin in the Feature tree to collapse it.

Step 3: Adding an ID Groove Feature Interactively

In this tutorial, you are going to add three features interactively. The first feature will be the groove that you deleted in the previous step.

CAMWorks understands the shape of a groove and knows how it is to be machined. When you insert a Groove feature for this undercut area, CAMWorks knows how to machine the area using the standard Rectangular Groove machining techniques.

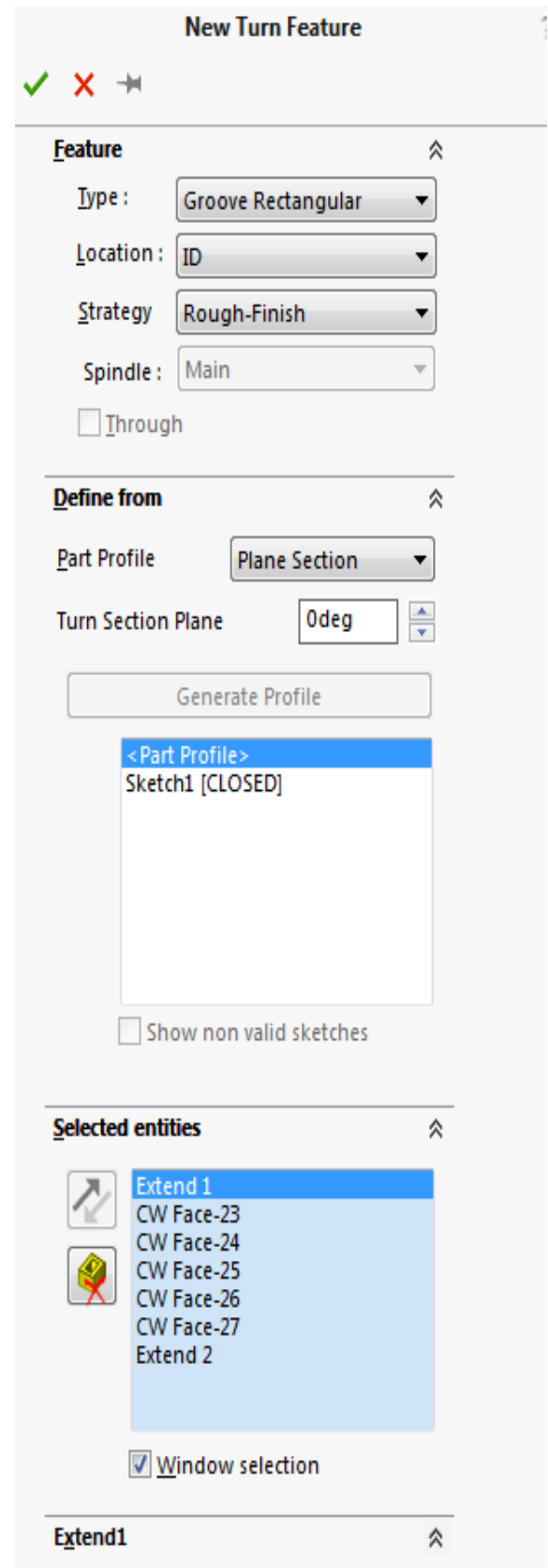


Within CAMWorks, the typical groove configuration is defined as a rectangular shape consisting of two equal length walls, which may or may not have fillet radii and may or may not have corner breaks.

Steps to interactively insert an ID Groove Feature:

1. Right click *Turn Setup1* in the Feature tree and select *Set View>>Turn ZX* from the context menu.
2. The new feature will be inserted after the feature *ID Feature1*. Right click on *ID Feature1* and select *Turn Feature* from the context menu.

The *New Turn feature* dialog box will be displayed.



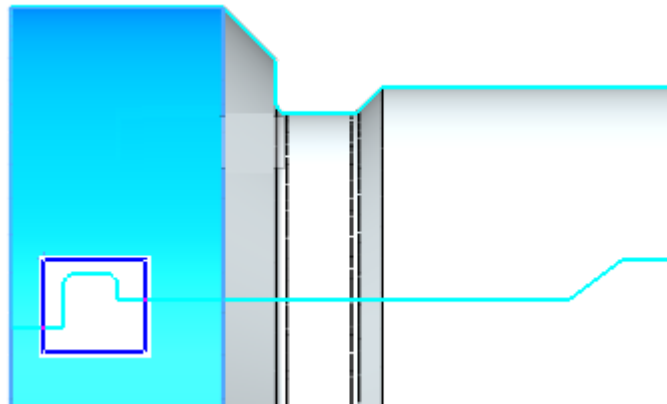
The image shows the 'New Turn Feature' dialog box in CAMWorks. It has a title bar with a green checkmark, a red X, and a grey arrow icon. The dialog is divided into several sections: 'Feature' with dropdowns for Type (Groove Rectangular), Location (ID), Strategy (Rough-Finish), and Spindle (Main), and a checkbox for 'Through'; 'Define from' with a dropdown for Part Profile (Plane Section) and a spinner for Turn Section Plane (0deg), followed by a 'Generate Profile' button; a list box showing '<Part Profile>' and 'Sketch1 [CLOSED]' with a 'Show non valid sketches' checkbox below it; 'Selected entities' with a list of faces (CW Face-23 to CW Face-27) and 'Extend 1', 'Extend 2', and a 'Window selection' checkbox; and a final 'Extend1' label at the bottom.

New Turn Feature Dialog box



Turn Tutorial

3. In the *Feature* group box:
 - Select *Groove Rectangular* for Feature Type
 - Set the Location to *ID*.
 - Leave the Strategy set to *Rough-Finish*.
4. In the *Selected Entities* list box, make sure that the *Window selection* check box is checked.
5. In the graphics area, use the mouse to select the geometry for the groove by drawing a window around the groove entities as shown in the below figure.

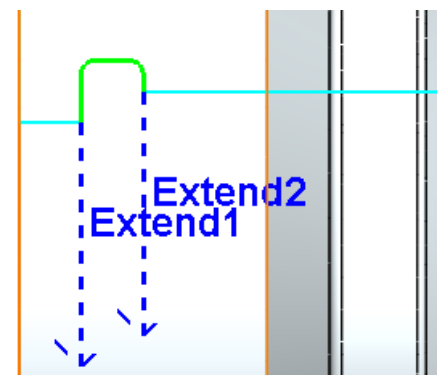


Window Select the Rectangular ID Groove Feature


The *Selected entities* list box will be updated to contain 5 selected face entities and 2 extent entities.

6. Highlight *Extend2* in the *Selected entities* list box.
 - The *Extend1* and *Extend2* group boxes will be expanded.
 - In the *Extend1* group box, set the direction to *Tangential*.
 - In the *Extend2* group box, set the direction to *Tangential*.

The extents will be displayed on the model part in the graphics area.



Setting the direction of the Extent entities

7.  Click *OK* to insert the feature.

Groove Rectangular ID3 is added to the Feature tree.

Step 4: Adding OD Thread and ID Thread Features Interactively

Thread information comes in many different formats and has many parameters. For a CAD/CAM system, it is not practical to model threads on a solid. Even though this can be done, it is very difficult for software to interpret a helical cut in the OD of a part and decipher what type of thread it is.

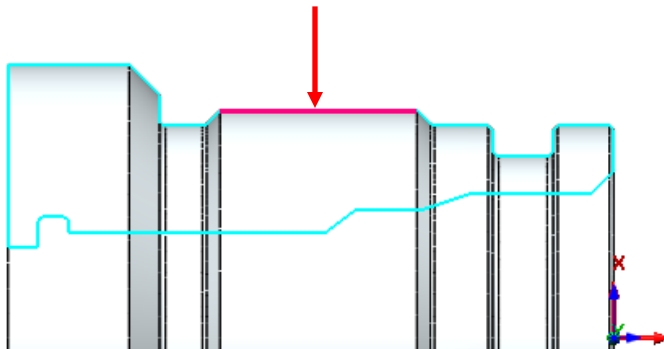


In CAMWorks, you can assign a Strategy to define a thread on a feature.

In this tutorial, you will define OD and ID features to be threaded.

Interactively Inserting a Threaded OD Feature

1. In the Feature tree, right click *OD Feature1* and select *Turn feature* on the context menu. The *New Turn Feature* dialog box will be displayed.
2. In the *Feature* group box:
 - Leave the Feature Type set to *OD Feature*.
 - Set the Strategy to *Thread*.
3. In the graphics area, use the mouse button to pick the OD segment where the thread will be machined (indicated by arrow in the below figure).



Pick the OD segment to be threaded

The *Selected entities* list box will be updated to contain 1 selected face entity and two Extend entities.

4. In the *Selected entities* list box, highlight *Extend 1* and *Extend 2*. Make sure *None* is selected in the corresponding *Extend* group boxes.



If CAMWorks is calculating the WIP (work in process), ID and OD Thread features should not be extended in either Z or X. If you extent Thread features, CAMWorks determines the feature length relative to the WIP and may extent the feature to start threading closer to

New Turn Feature

✓ ✕ ↶

Feature

Type : OD Feature

Location : OD

Strategy : Thread

Spindle : Main

☐ Through

Define from

Part Profile : Plane Section

Turn Section Plane : 0deg

Generate Profile

<Part Profile>
Sketch1 [CLOSED]

☐ Show non valid sketches

Selected entities

Extend 1
CW Face-13
Extend 2

☐ Window selection

Extend1


New Turn Feature Dialog box



the face than you want. Typically, this would occur on thread features such as this one where the feature is not located at the face of the part.

To adjust the machining length for a Thread feature, you can:

- Specify a Start length in the Thread operation machining parameters.
- Set up the TechDB to define an amount that you want the tool to go beyond the thread length into a thread relief area.

5.  Click *OK* to insert the feature.

OD Feature2 is listed in the Feature tree after *OD Feature1*.

6. Right click on *OD Feature2* and select *Parameters* on the context menu.

The *OD Profile Parameters* dialog box is displayed.

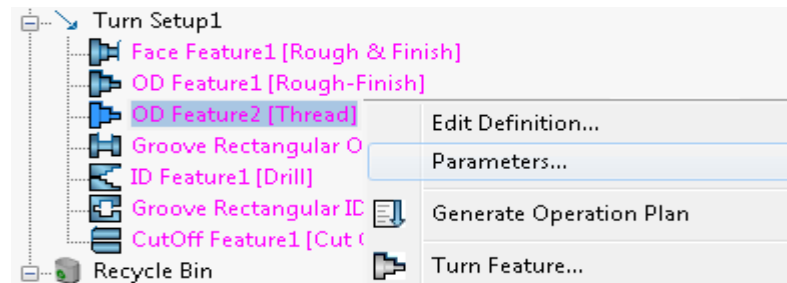
7. Observe that the *Maximum dia. (D1)* for this feature is **1.5in.**



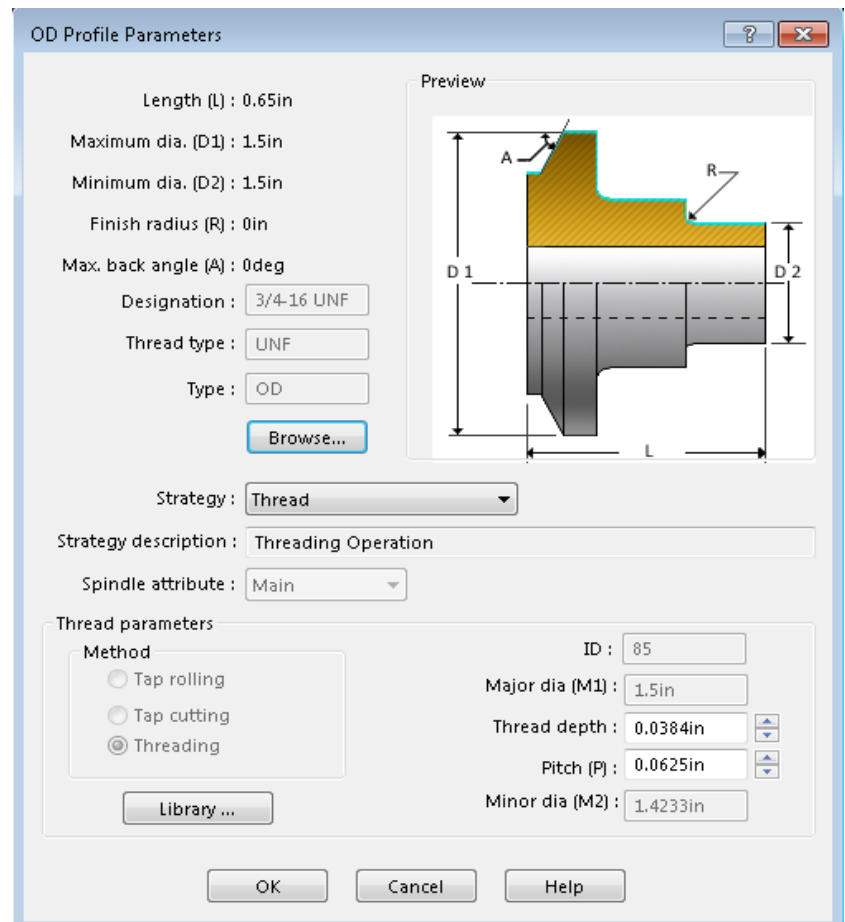
In order to generate a Thread operation, the corresponding thread condition must be selected from the TechDB. The major diameter in the TechDB must match the feature maximum diameter.

Initially, the Thread Parameters display the values in the first record in the TechDB; however, this record has not been selected.

8. Click the *Library* button. The *Tools Database - Thread Condition* form is displayed.



Context menu to open Parameters dialog box



OD Profile Parameters dialog box



Several thread conditions have been set up in the TechDB. When you machine your own parts, you can add data for additional threads.



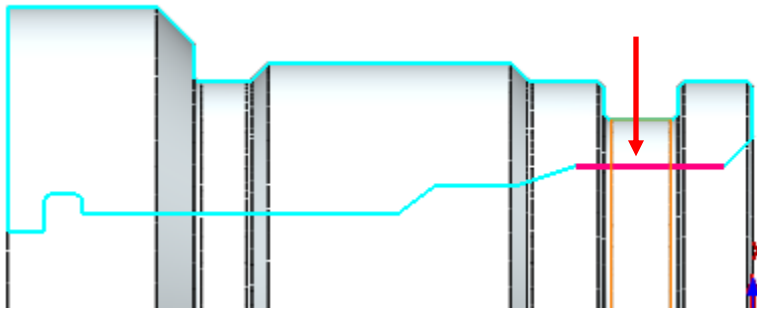
9. In this form:
 - Select too with **ID** of '**43**'.
 - Click *OK* to assign the selected Thread Condition.
10. Click *OK* to close the *OD Profile Parameters* dialog box.

	ID	Type	Designation	Pitch	EndPitch	DepthOfThread	ProcessMethod	Units	Spin
117	31	UNC	3/4-10 UNC	0.100000	0.000000	0.061400	1	2	1
118	33	UNC	7/8-9 UNC	0.111100	0.000000	0.068200	1	2	1
119	35	UNC	1-8 UNC	0.125000	0.000000	0.076700	1	2	1
120	37	UNC	1 1/8-7 UNC	0.142800	0.000000	0.087700	1	2	1
121	39	UNC	1 1/4-7 UNC	0.142800	0.000000	0.087700	1	2	1
122	41	UNC	1 3/8-6 UNC	0.166700	0.000000	0.102300	1	2	1
123	43	UNC	1 1/2-6 UNC	0.166700	0.000000	0.102300	1	2	1
124	45	UNC	1 3/4-5 UNC	0.200000	0.000000	0.122700	1	2	1
125	47	UNC	2-4.5 UNC	0.222200	0.000000	0.136300	1	2	1
126	49	UNC	2 1/4-4.5 UNC	0.222200	0.000000	0.136300	1	2	1
127	51	UNC	2 1/2-4 UNC	0.250000	0.000000	0.153400	1	2	1
128	53	UNF	1-72 UNF	0.013900	0.000000	0.008500	1	2	1
129	55	UNF	2-64 UNF	0.015600	0.000000	0.009600	1	2	1
130	57	UNF	3-56 UNF	0.017800	0.000000	0.011000	1	2	1
131	59	UNF	4-48 UNF	0.020800	0.000000	0.012800	1	2	1
132	61	UNF	5-44 UNF	0.022700	0.000000	0.014000	1	2	1
133	63	UNF	6-40 UNF	0.025000	0.000000	0.015400	1	2	1
134	65	UNF	8-36 UNF	0.027800	0.000000	0.017100	1	2	1

Tools Database - Thread Condition form

Interactively Inserting a Threaded ID Feature

1. Right click *ID Feature1* in the Feature tree and select *Turn feature* from the context menu.
The *New Turn Feature* dialog box will be displayed.
2. In the *Feature* group box:
 - Set the Feature Type set to *ID Feature*.
 - Set the Strategy to *Thread*.
3. In the graphics area, use the mouse button to pick the ID segment at the front of the part where the thread will be machined (indicated by the arrow in the figure).



Pick the ID segment to be threaded

The *Selected entities* list box will be updated to contain 1 selected face entity and 2 extend entities.

- In the *Selected entities* list box, highlight *Extend 1* and *Extend 2*. Make sure *None* is selected in the corresponding Extent group box.

- ✓ Click *OK* to insert the feature.

ID Feature2 is added to the Feature tree after *ID Feature1*.

- Right click on *ID Feature2* and select *Parameters* from the context menu.

The *ID Profile Parameters* dialog box is displayed. You need to select the thread condition for *ID Feature2*.

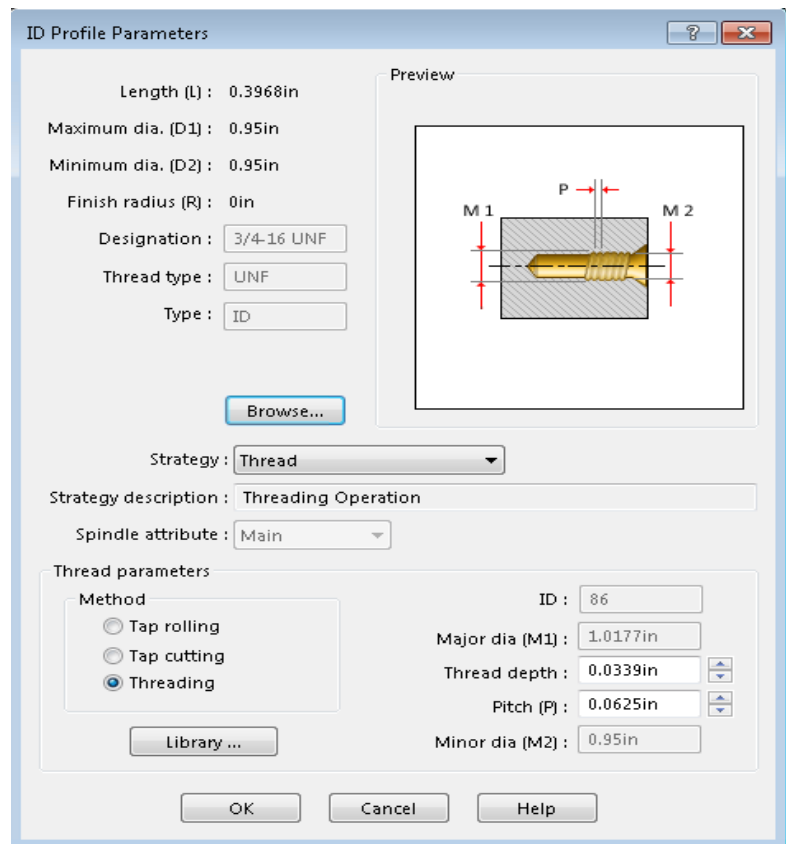
- Observe that the *Maximum dia. (M1)* for this feature is **1.01in**.

- Click the *Library* button in this dialog box.

The *Tools Database - Thread Condition* form is displayed.

- In this form:

- A tool with **1.01 Major Dia.** is not available.
- Hence, select tool with **ID of '34'**.



ID Profile Parameters dialog box

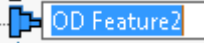



- Click *OK* to assign the selected Thread Condition.
10. Click *OK* to close the *ID Profile Parameters* dialog box.


Step 5: Changing Feature Parameters, Generating Operations and Modifying Operation Parameters

Changing Feature Parameters and Renaming Features

Before generating operations, you can change feature parameters and rename features so that they are more descriptive.

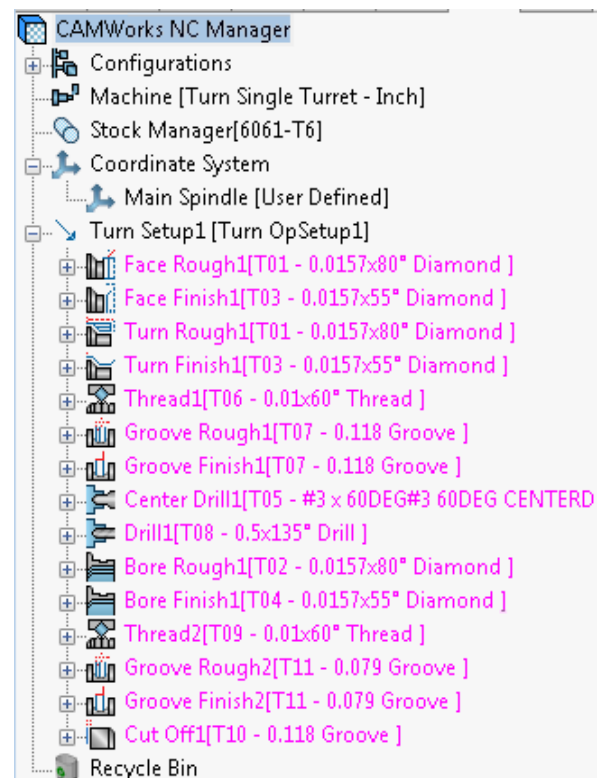
1. Right click *ID Feature1* in the Feature tree and select *Parameters* from the context menu.
The *ID Profile Parameters* dialog box is displayed.
 - Change the Strategy to *Rough-Finish*.
 - Click *OK* to close the ID Profile Parameter dialog box.
2. Right click *OD Feature2* in the Feature tree and select *Rename* from the context menu.
 - A text box displays around the item. .
 - Type **OD Thread** and press *Enter* key.
3. Right click *ID Feature2* and select *Rename* from the context menu.
 - A text box displays around the item. .
 - Type **ID Thread** and press *Enter* key.

Generating Operations

 Click the *Generate Operation Plan* button on the CAMWorks Command Manager/Workflow Toolbar.

OR

Right click *Turn Setup1* in the tree and select *Generate Operation Plan* from the context menu.
The Operations for all the features are generated and listed in the Operation tree.



Operations listed in Operation tree



Modifying Operation Parameters



You can modify operation parameters before or after generating toolpaths. Choosing parameters for machining depends on your machining practices.

The following examples show you how to make changes.

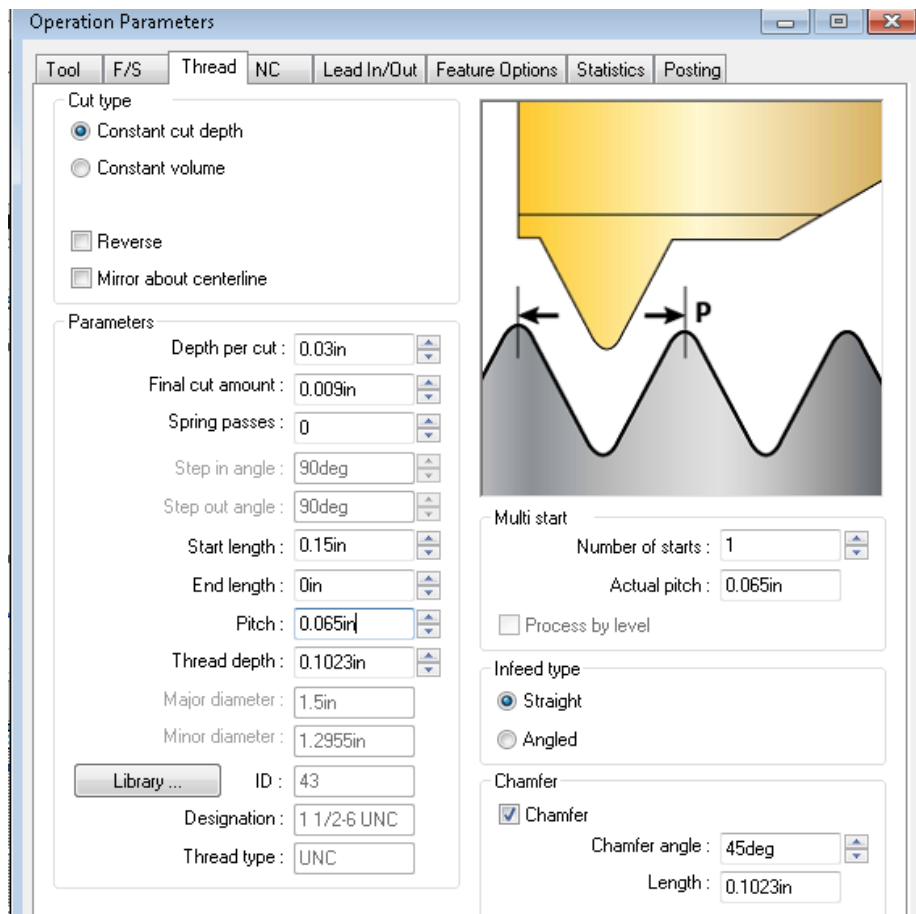
1. Double click *Thread1* in the Operation tree. The *Operation Parameters* dialog box is displayed.
2. Click on the *Thread* tab. Change the following values in the *Parameters* group box:

- Depth per cut = **0.03in**
- Final cut amount = **0.009in**
- Start length = **0.15in**

The start length is measured from the start of the OD Thread feature that you created and is a distance in front or to the right of the feature.

Since this is an OD feature and the tool is on a rear turret, the tool orientation is down and left. (Click on the *Holder* page of the *Tool* tab to view the *Tool Orientation*.)

- Pitch = **0.065in**

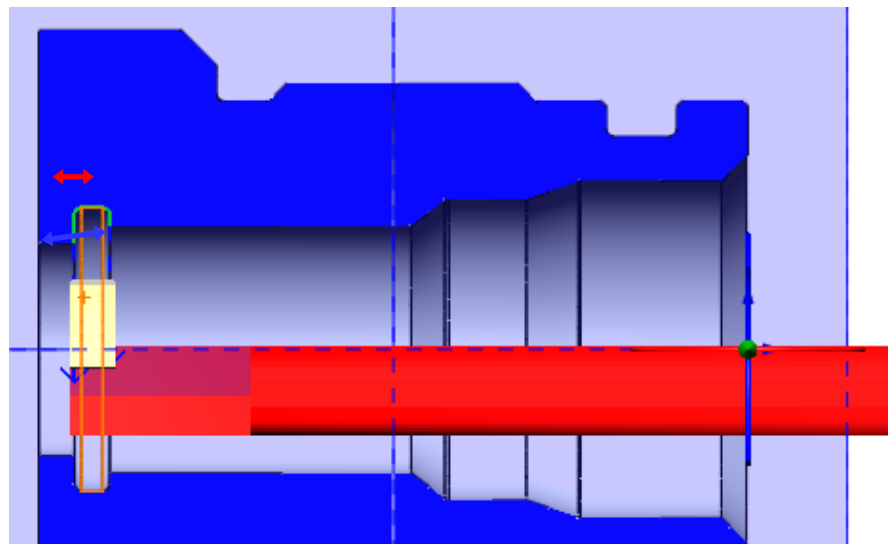


Modifying Thread Parameters



- Click *OK* to close the dialog box.
 - 3. Double click *Groove Rough1* in the tree. The *Operation Parameters* dialog box is displayed.
 - 4. Click on the *Groove Rough* tab.
 - In the *Allowance* group box, change the Radial (X) and Axial (Z) allowances to **0.0in**.
 - Click *OK* to close the dialog box.
 - 5. Right click *Groove Finish1* in the Operation tree and select *Delete* from the context menu. Click *Yes* to confirm the deletion.
- By setting the Groove Rough allowances to **0.0in**, you can rough and finish the groove in one operation. Hence, this Groove Finish operation was deleted.
- 6. Double click *Groove Rough2* in the Operation tree. The *Operation Parameters* dialog box is displayed.
 - 7. Click on the *Groove Rough* tab.

- In the *Allowance* group box, change the Radial (X) and Axial (Z) allowances to **0.0in**.
- Click on the *Tool* tab. Select the *Boring Bar* page. In the graphics area, observe that the width of the Groove Insert used to machine this feature is lesser than the width of the feature. Hence, toolpaths will be generated correctly for this Groove feature.



Width of the Groove Insert is greater than the width of the feature




In cases where the width of the Groove Insert used to machine the feature is greater than the width of the feature, toolpaths will not be generated. You will have to either assign suitable Groove insert from the active Tool Crib or add a Groove Insert Tool from the Tool Library to the active Tool Crib and assign it to the operation.

- 8. Right click *Groove Finish2* and select *Delete* from the context menu. Click *Yes* to confirm the deletion.



Step 6: Defining the Machining Sequence and Generating Toolpaths

The order that the operations are listed is relative to the feature order. Both the Feature tree and the Operation tree can be reordered to achieve the specific machining sequences that are needed in terms of tool order and material removal.

1. In the Operation tree, drag *Groove Rough2* above *Bore Finish1*.
2.  Click the *Generate Toolpath* button on the CAMWorks Command Manager/Workflow Toolbar.

OR

Right click on *Turn Setup1* in the Operation tree and select *Generate Toolpath* from the context menu.

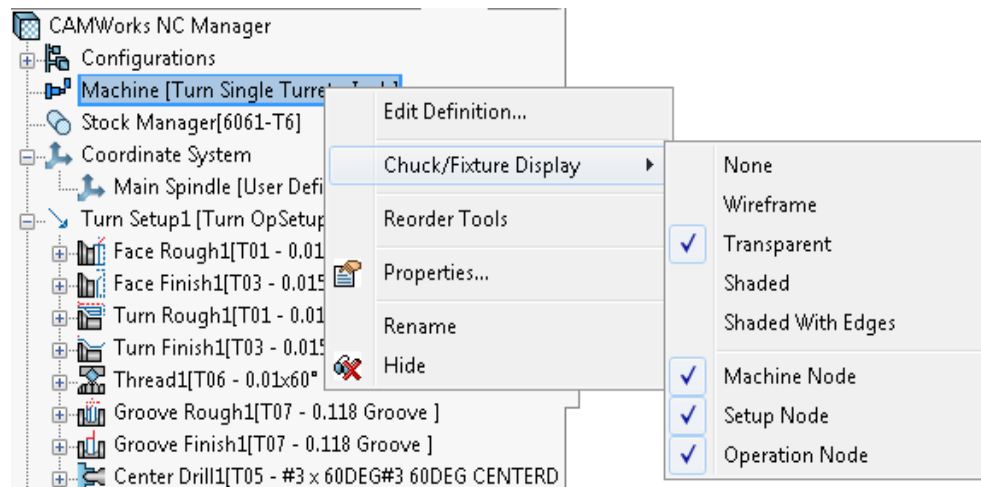
3. Click on each operation to view the toolpath.

Step 7: Displaying the Chuck/Fixture

You can choose to display/not display the chuck in which the part will be clamped.

1. In the Operation tree, right click on the *Machine [Turn Single Turret -inch]* and select *Chuck Display*.

An expandable cascading menu displays the various display options on how the chuck will be displayed in the Graphics area.



Chuck Display Options


- **Non-display:** When set to *None*, the chuck will not be displayed in the Graphics area. Instead, two locating triangles will be displayed. These triangles provide a graphical representation of where the part will be clamped in the chuck.
- **Chuck Display States:** When the other display option *Wireframe*, *Transparent*, *Shaded* or *Shaded with Edges* is chosen, the chuck will be displayed accordingly in the graphics area.
- **Node Display:** You can select the Operation tree item, which on being selected/highlighted in the Operation tree, will display the Chuck in the selected display option state. You can select one or more among the options of *Machine node*, *Setup node* and *Operation node*.

2. In the *Chuck/Fixture Display* cascading menu:



- Select *Transparent* for Chuck/Fixture Display state.
 - Select *Machine Node* and *Setup node* by clicking on these options. A checkmark next to these options indicates that they have been selected.
 - Ensure *Operation node* is not selected.
3. In the Operation tree, left click on *Turn Setup1*. Observe that the Chuck is displayed in the graphics area.
 4. Next, left click on the *Machine [Turn Single Turret - inch]*. The chuck is once again displayed in the graphics area.
 5. Left click on the different operations. Observe that the chuck is not displayed. This is because the *Operation Node* display option was not selected in the *Chuck Display* menu.

Step 8: Defining the Chuck/Fixture

1.  Double click the *Machine [Turn Single Turret - inch]* in the Operation tree.

OR



Click on the *Define Machine* button in the CAMWorks Workflow toolbar.

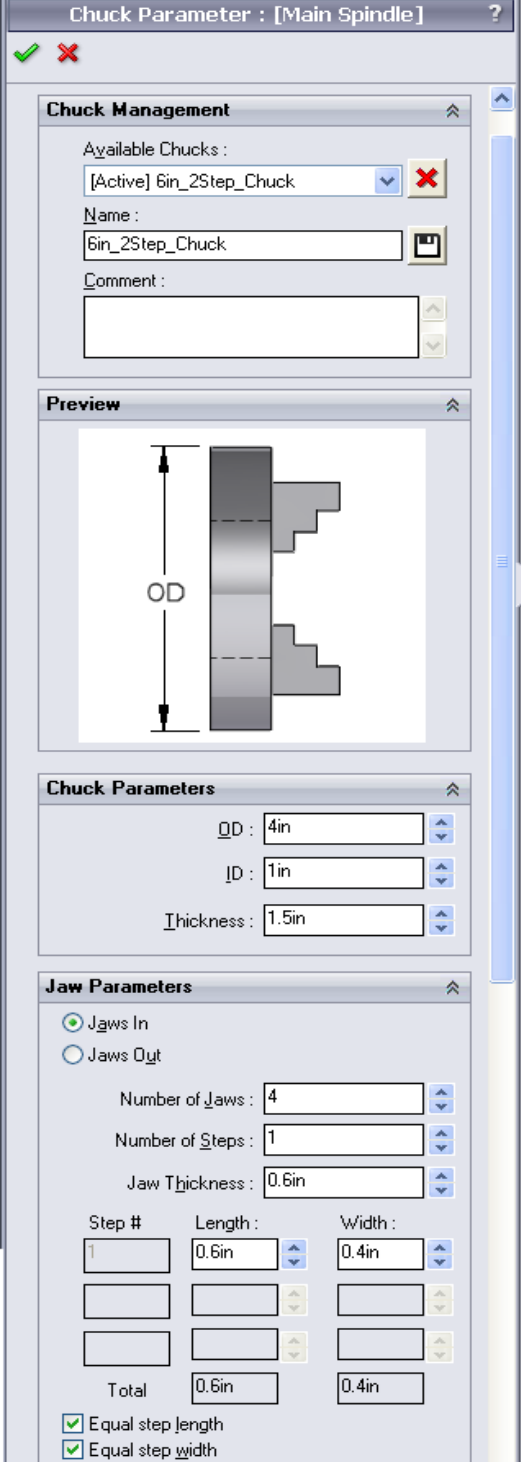
The *Machine* Dialog box is displayed.

2. Click on the *Machine* tab. Ensure that the *Use Sub-spindle* option is not checked.
3. Click on the *Chuck/Fixture* tab.



The Chuck/Fixture tab allows you to define the Chuck geometry (size of the chuck base and the jaws) to be displayed in the graphics area and during toolpath simulation.

4. The *Chuck/Fixture* tab displays the *Main Spindle Information* and *Sub Spindle Information*. Since only a single Spindle (main spindle) is used for machining this part, the *Sub Spindle Information* group box is grayed out.
5. In both the *Main Spindle Information* and *Sub Spindle Information* group boxes, the *Name* field indicates the name of the current Chuck Configuration. By default, the Chuck Configuration *6in_2Step_Chuck* is displayed.



Step #	Length :	Width :
1	0.6in	0.4in
Total	0.6in	0.4in

☒ Equal step length
☒ Equal step width

Chuck Parameter dialog box



A Chuck Configuration is a set of chuck parameter values defined and saved in the TechDB. CAMWorks allows multiple configurations of chuck parameters to be saved in the TechDB and reused. You must select/create/edit a chuck configuration as per your requirements for the turn part.

A ready set of commonly used chuck configurations are already available within CAMWorks.

6. In the *Main Spindle Information* group box, click on the *Edit* button. The *Chuck Parameter: [Main Spindle]* dialog box is displayed.



Use the Chuck Parameter dialog box to edit existing chuck configurations and to define additional chuck configurations.

This dialog box allows you to define the size of the chuck base and the jaws.

7. In the Chuck Parameters group box, assign the following values:

- OD = **4in**
- ID = **1in**
- Thickness = **1.5in**

The *Preview* area of the dialog box provides a graphical description of the active Chuck Parameter.

Observe that as soon as you shift focus from the active Chuck parameter, the graphical representation of the chuck is immediately updated in the graphics area to match the new values you have entered.

8. In the Jaw Parameters group box, assign the following values:


- Number of Jaws= **4**
- Number of Steps = **1**
- Jaw Thickness = **0.6in**
- Step Length = **0.6in**
- Step Width = **0.4in**



The Jaw Thickness value should be less than or equal to the Step Length else CAMWorks will not display the chuck and will display an error message indicating that the Jaw Chamfer is larger than the Step Length.


The *Equal step length* and *Equal Step width* options apply only when there are multiple steps.

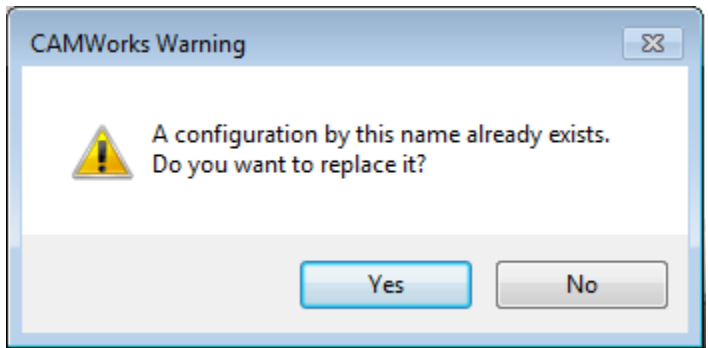
9. The changes you have made to the active chuck configuration can either be saved to the TechDB or applied only for the current part.

-  If you directly click *OK* after making the changes, then the changes made to the chuck configuration will be applicable only to the current part. For all other machining jobs, the original set of values will be applied.
- **Overwriting existing Chuck Configuration:**



In the *Chuck Management* group box, the name of the active chuck configuration is displayed in the name field. If

you click on the *Save* button  next to this field after making the changes, CAMWorks will display a warning message indicating that a chuck configuration by that name already exists.





CAMWorks Warning message displayed when you try to save an edited Chuck Configuration

- If you click *Yes* within the message, then the active chuck configuration will be replaced with the new set of values and saved to the TechDB. The new set of values will be available for all future jobs.
- If you click *No* within the message, CAMWorks will return to the Chuck Parameter dialog box.

- **Defining a new Chuck Configuration:**

If you want the changes made saved as a new chuck configuration which will be available for future jobs, enter a new name in the *Name* field in the *Chuck*

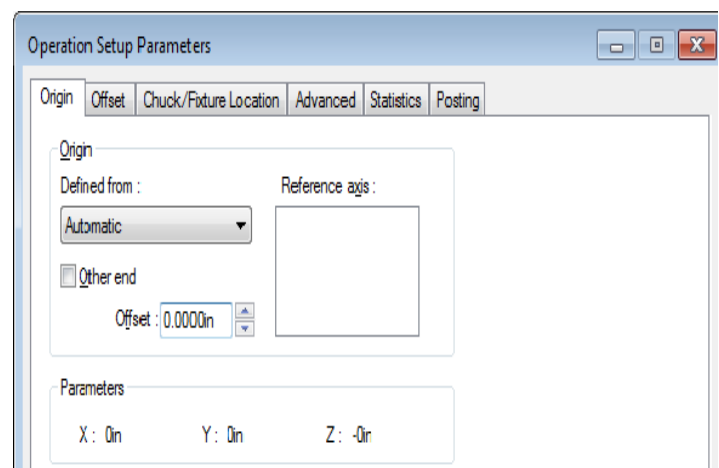
Management group box. After making the changes, click on the *Save* button . This new chuck configuration will be available in the *Available Chucks* drop down list.

10. In this tutorial, you will not save the edited configuration. Click the *Cancel* button  to exit the *Chuck Parameter: [Main Spindle]* dialog box.
11. Click *OK* to close the *Machine* dialog box.

Step 9: Defining the Chuck Location

1. Double click *Turn Setup1* in the Operation tree.
The *Operation Setup Parameters* dialog box is displayed.
2. In the *Origin* tab, select *Automatic* in the *Defined from* dropdown list.
This is where the Origin or Part Zero is located.
3. Click the *Chuck Location* tab.

The WIP Sketch/Stock Boundary is displayed on the part in the graphics area.



Operation Setup Parameters Dialog Box




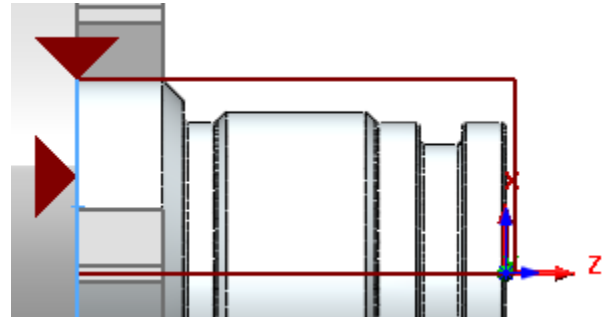
Two locating triangles display at Part Zero by default. Use the parameters in the *Clamping Location* group box of the *Chuck Location* tab to position these two triangles in order to reflect where the chuck will be located.



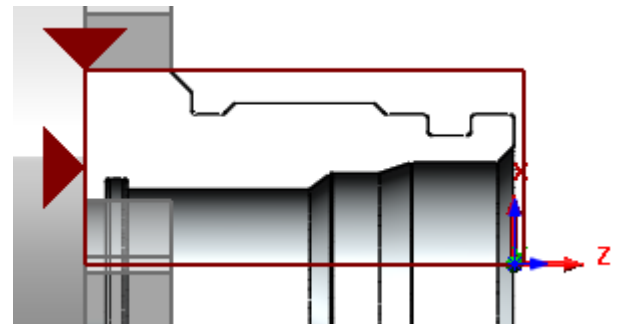
For a Main Spindle Turn setup, the Clamping Location (represented by the locating triangles) is located at the most negative Z end point of the WIP sketch/Stock Boundary.

4. Use the *Select Entity for Z Clamping Location* field in the Clamping Location group box to select the entity to clamp along Z.

-  Click on *Select Entity for Z Clamping Location* field to set the focus.
- In the graphics area,
- pick any circular edge of the part whose normal is parallel to the turn axis and center is concentric with turn axis. The Clamping Location changes.
- Now click on the left-most vertical edge of the stock WIP sketch. The Clamping Location moves back to its original location.



Selecting a Circular edge



Selecting a Left-most vertical WIP entity



An entity fulfilling any one of the following criteria can be selected as a Z Clamping location:

- Circular edge whose normal is parallel to the turn axis and center is concentric with turn axis
- Vertical WIP entity
- Vertex (for mill-turn features)

5. You can also change the size of the triangles. Use the *Triangle display* group box in the *Chuck/Fixture Location* tab to change the size. Change the size to **0.2in**.
6. Observe the *Chuck Properties* group box in the *Chuck/Fixture Location* tab. You can define the chuck from the *Machine* level or *Setup* level.
 - For parts created in CAMWorks 2013 and later versions, the default setting is Chuck defined from *Machine*. This setting uses the globally defined chuck parameter values assigned on the *Chuck/Fixture* tab of the Machine dialog box.
 - Defining the chuck from Setup may be preferred when machining a part with multiple turn setups where the chuck may change from setup to setup.



For legacy parts (parts created in CAMWorks 2012 or earlier versions), the default setting is Chuck defined from Setup. This allows the Chuck Geometry to be defined in the Setup node, as was the case in CAMWorks 2012 and earlier versions.

- Click *OK* to close the dialog box.



A graphic of the part can be added to the setup sheet to give the shop floor a better understanding of the setup.



Did You Know ...

If you change the Chuck Configuration, you will need to position the locating triangles again using the Chuck/Fixture Location tab of the Operation Parameters dialog box.

Step 10: Simulate Toolpath

- Change the *View Orientation*  to *Isometric*



You do not need to simulate this operation.

- Right click *Turn Setup1* in the Operation tree and select *Simulate Toolpath* from the context menu.






OR




Click on the *Simulate Toolpath* button on the CAMWorks Command Manager/Workflow toolbar.

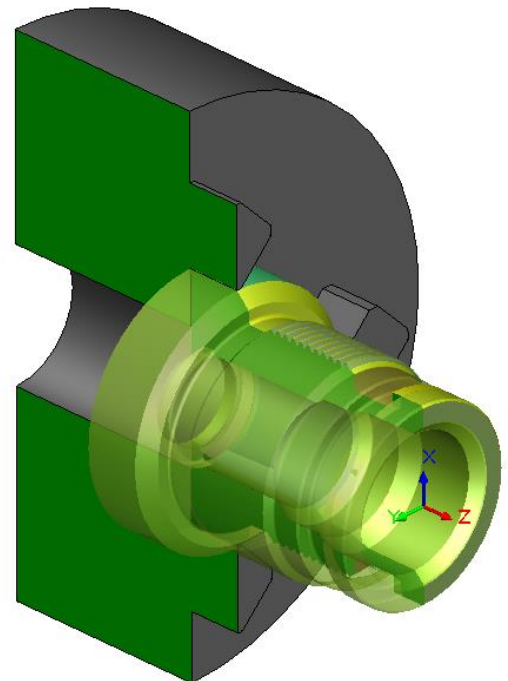
The *Toolpath Simulation* toolbar is displayed.

- Select the following display options:

-  Fixture: *Shaded With Edges*
-  Tool holder: *Shaded With Edges*
-  Tool: *Shaded With Edges*
-  Target part: *Translucent Display*
-  Section view: *Half*

-  Click on the *Run* button.



Observe that the OD feature is threaded.



Toolpath Simulation



Turn Tutorial

5. Click *OK*  *button* to exit the simulation.
6. Double click on the *Machine [Turn Single Turret - inch]* in the Operation tree.
7. Click on the *Chuck* tab. Edit the Chuck Configuration and try different settings: change the number of steps and the size of the steps and the base, then click *OK*.
8.  Run the Toolpath Simulation again and view the changes.

Step 11: Post Processing Toolpaths

Post process the toolpaths in the same manner as explained in [Step 8 of the Tutorial 3](#) of this chapter.



Turn 6

Double chucking is a method that allows the NC programmer to machine a Turn Stock on both sides of the part using the Main Spindle. Such double chucking operations are supported in CAMWorks turning.

The methodology for doing a double chucking operation using the Main spindle is explained in this tutorial.

Topics covered in this tutorial:

- [Establishing Part Zero](#)
- [Defining a stock from a sketch for double chucking](#)
- [Changing Feature Definitions for OD & ID Profiles](#)
- [Changing the origin machining direction](#)
- [Defining the Chuck Configuration](#)
- [Setting the Chuck Display State](#)
- [Setting the chuck location](#)
- [Simulating toolpaths for double chucking](#)

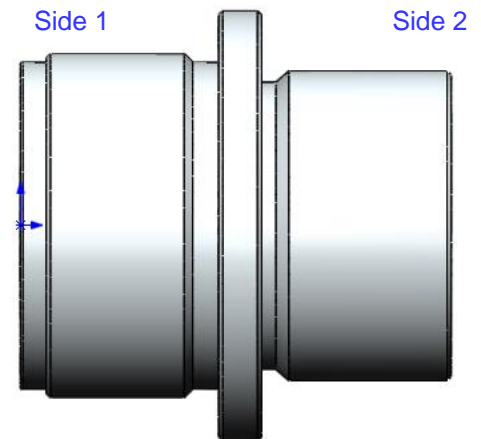
Step 1: Opening the Part

1. Open part file **TURN2AX_6.SLDPRT** in the following folder.

Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Turn



In this tutorial, assume that *Side 1* and *Side 2* are established as shown on the right.

- *Side 1* will be machined first and you will be chucking on *Side 2*.
- Since the material is a solid billet, you will be locating on the billet OD on *Side 2*.



TURN2AX_6.SLDPRT

Step 2: Defining Machine Parameters

1.  Click the *CAMWorks Feature Tree* tab.
2. Under this tree, observe that *Machine [Turn Single Turret- inch]*  is the active machine. In this tutorial, the part will be machined on a Single Turret Turn machine.

Double click the *Machine* item in the Feature tree. OR



Click on the *Define Machine* button on the CAMWorks Workflow toolbar.

The *Machine* dialog box is displayed.



Turn Tutorial

- On the Machine tab, ensure that the *Use sub-spindle* option is unchecked.
 - Click the *Tool Crib* tab, ensure *Tool Crib 2 Rear (Inch)* is the active tool crib.
 - Click the *Post Processor* tab and make sure *T2AXIS-TUTORIAL* is selected.
3. Do not close the *Machine* dialog box.


Step 3: Establishing Part Zero

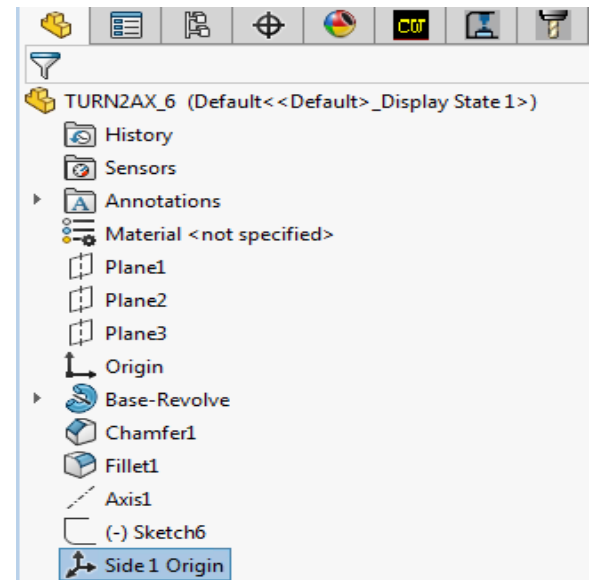
Since this part is a cylinder, CAMWorks already knows where the centerline is. CAMWorks automatically sets the origin to a default setting and establishes a default centerline.

For a single chucking operation, the default origin may be acceptable. Since you are doing a double-chucking operation, however, you need change the default to set the system origin for *Side 1*.

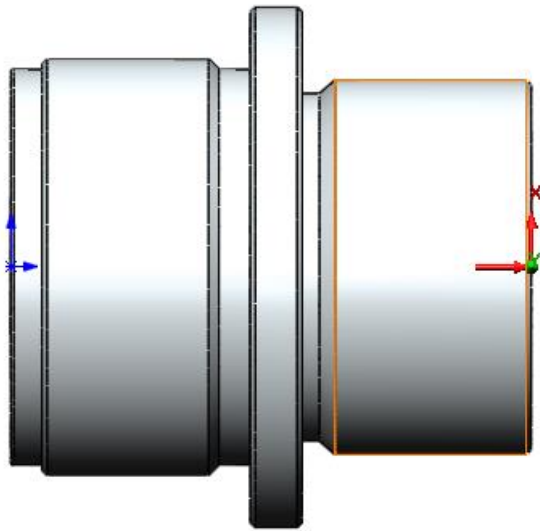
1. Click the *Setup* tab in the *Machine* dialog box.

Observe the part in the graphics area. The origin X and Z symbol displays on the part in the default location (at the face of the part) for *Side 2*.

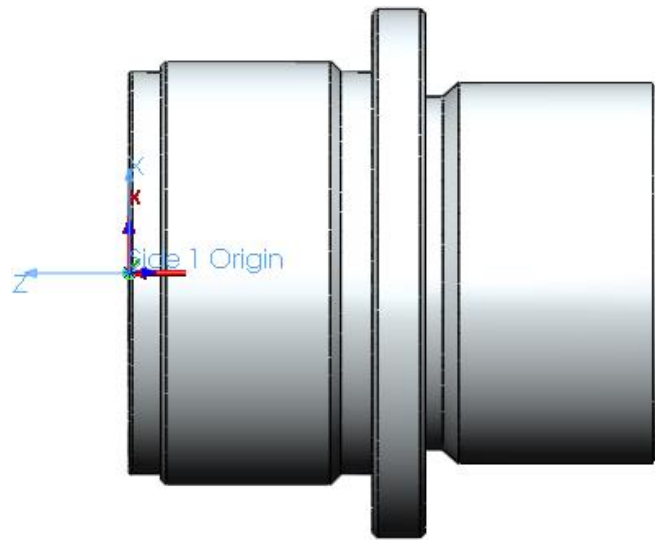
2. In the *Setup definition* group box, select *Coordinate System* for the Method. The *Machine* dialog box minimizes.
3. In the Main spindle coordinate system Group box, Click on the Edit button. The Main Spindle Coordinate system dialog box is displayed. In the Method group box select SOLIDWORKS coordinate system from the drop down list and click on side1 origin in the Available Coordinate Systems group box. Side1 origin is automatically selected under the coordinate system group box.
4.  Click the *FeatureManager Design Tree* tab and pick *Side 1 Origin* in the tree. This action selects the entity *Side 1 Origin*.
The origin marker moves to *Side 1*.
5. Click *OK* button in the *Machine* dialog box to apply the changes and close the dialog box.



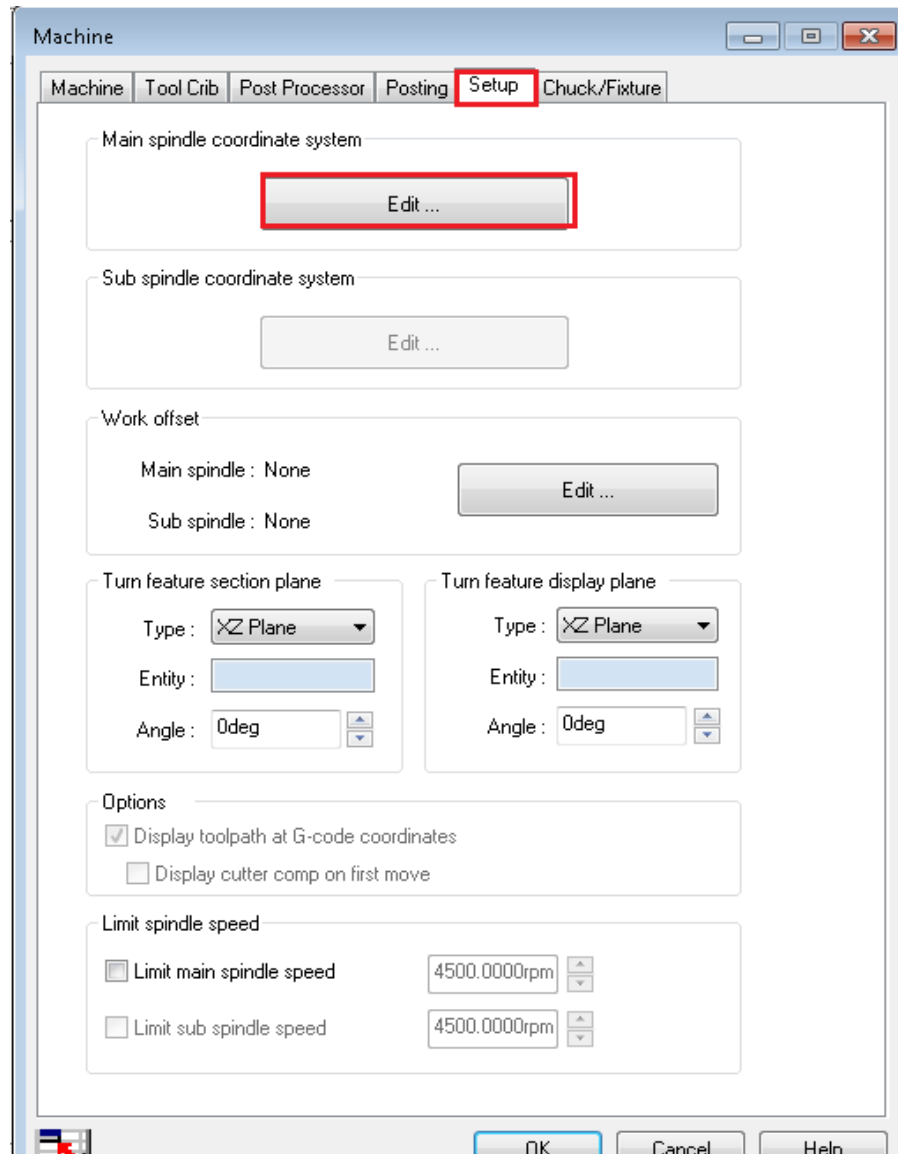
Feature Manager Design Tree



Default Origin at Side 2



Origin changed to Side 1



Setup Tab of Machine Dialog Box



Step 4: Defining the Stock from a Sketch for Double Chucking

The turn part model in this tutorial will be machined by double chucking. For double chucking, two Turn Part Setups will be required. To define two setups, a sketch (which is longer than the part on both the sides) must be used to define the stock.



Always define the stock from a sketch when machining a turn part using double chucking. Defining a stock from a sketch establishes 2 distinct faces (or ends of the part) which is necessary for double chucking. Since bar stock has an infinite side length and one face, you cannot select Bar stock for double chucking.

In this tutorial, you will define the stock from a sketch for the purpose of generating two setups for double chucking.



1.  Click the *CAMWorks Feature Tree* tab.
2.  Double click *Stock Manager* in the Feature tree.

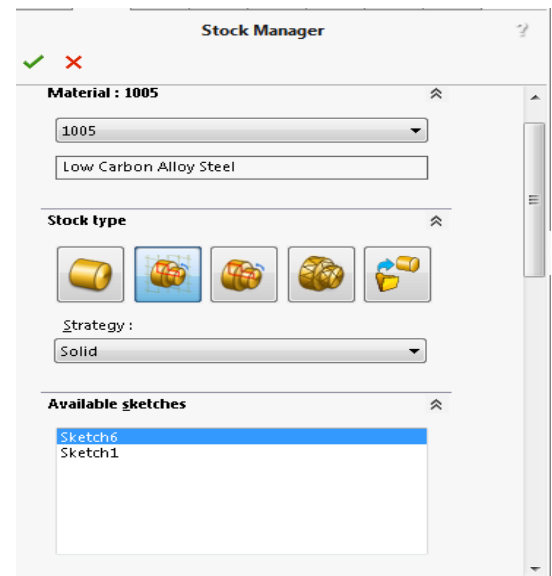
OR



Click the *Stock Manager* button in the CAMWorks Workflow toolbar.


In the displayed Stock Manager dialog box:

-  Set the Stock Type to *Revolved Sketch*.
- In the *Available sketches* list, select *Sketch6*.
-  Click *OK* to apply the changes and close the dialog box.



Stock Manager Dialog Box

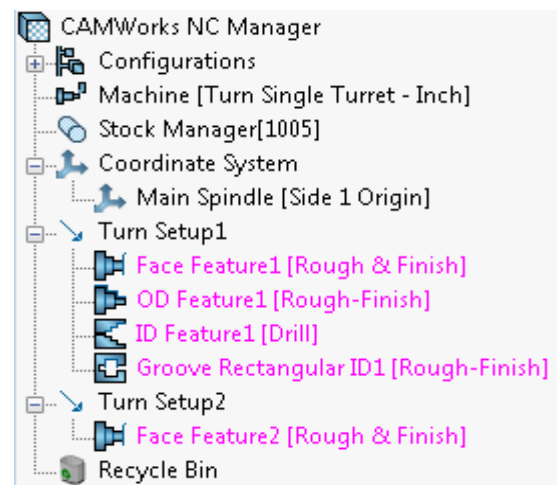
Step 5: Defining Machinable Features

 Click the *Extract Machinable Features* button on the CAMWorks Workflow toolbar or Command Manager. OR

Right click *CAMWorks NC Manager* in the Feature tree and select *Extract Machinable Features* from the context menu.

On executing this command, two Turn Setups are created and AFR recognizes the features that can be machined in each Setup.

AFR has found an OD Feature and an ID Feature in *Setup 1* that each extent from the front face to the back face.

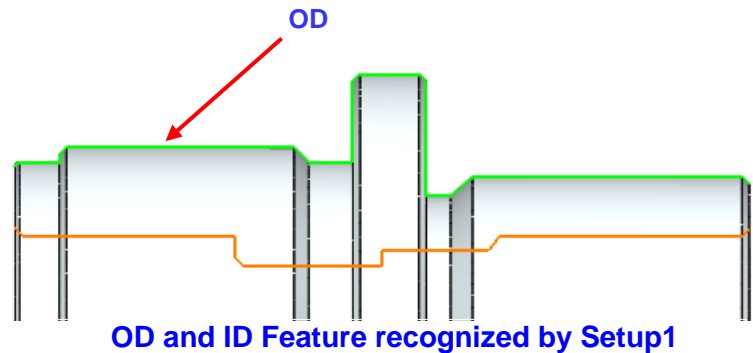


Turn features recognized by EMF



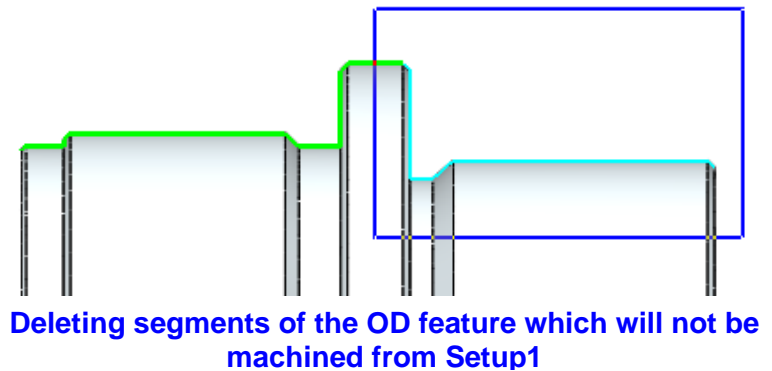
Changing Feature Definitions for OD and ID Profiles

In this tutorial, you will edit the feature definitions in Setup 1 and create new features in Setup 2 such that half of the OD profile and ID profile is machined from *Setup1* while the other half of these profiles is machined from *Setup2*. The double-chucking environment makes this possible.



Editing OD Feature in Turn Setup1:

1. Double click *OD Feature1* in the tree. The *Edit Turn Feature* dialog box will be displayed.
2. In this dialog box, in the *Selected Entities* list box, make sure the *Window selection* option is checked.
3. Window-pick the segments of the part as shown in the figure.



This action deletes all the segments (in blue highlights) captures by the window.

The *Selected entities* list will now display only those segments (in green highlights) which will be machined.

4.  Click *OK* to close the dialog box.

Interactively Inserting OD Feature in Turn Setup2:


The next step would be to interactively insert the part of the OD feature the segments of which were deleted in the previous step) which will be machined from *Turn Setup2*.

1. Under *Turn Setup2*, right-click *Face Feature2* and select *Turn Feature* from the context menu.


The *New Turn Feature* dialog box will be displayed.

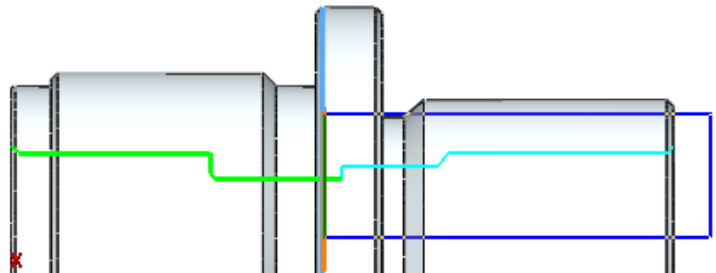
2. In this dialog box:
 - Select *OD Feature* as the *Feature Type*.
 - Select *Rough-Finish* as the *Strategy*.
 - In the *Selected Entities* list box, make sure the *Window selection* option is checked.
 - Window-pick the same segments (faces) that you removed from *OD Feature1* as shown in the figure on the previous page.



3.  Click *OK* to insert the feature.
OD Feature2 is added to *Turn Setup2* after *Face Feature2*.

Editing ID Feature in Turn Setup1:

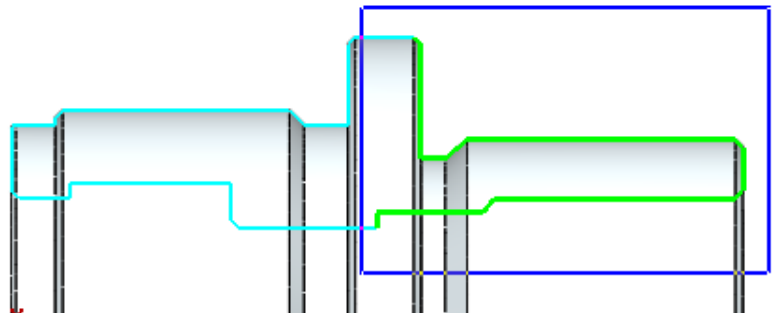
1. Double click *ID Feature1* in the tree. The *Edit Turn Feature* dialog box is displayed.
2. Change the Strategy to *Rough-Finish*. This is necessary as the default strategy *Drill* only generates *Center Drill* and *Drill* operations while the *Rough-Finish* strategy, in addition to the *Drill* operation, also generates *Rough* and *Finish Boring* operations required to completely machine the feature
3. In this dialog box, in the *Selected Entities* list box, make sure the *Window selection* option is checked.
4. Window-pick the segments of the part as shown in the figure.
The *Selected entities* list will now display only the segments (in green highlights) which will be machined from *Turn Setup1*.
5.  Click *OK* to close the dialog box.



Deleting segments of the ID feature which will not be machined from Setup1


Inserting ID Feature in Turn Setup2:

1. Right click *OD Feature2* in *Turn Setup2* and select *Turn Feature* from the context menu.
The *New Turn Feature* dialog box will be displayed.
2. In this dialog box:
 - Select *ID Feature* for the Feature Type.
 - Select *Rough-Finish* as the Strategy.
 - In the *Selected Entities* list box, make sure the *Window selection* option is checked.
 - Window-pick the same segments (faces) that you removed from *ID Feature1* as shown in the right side figure.

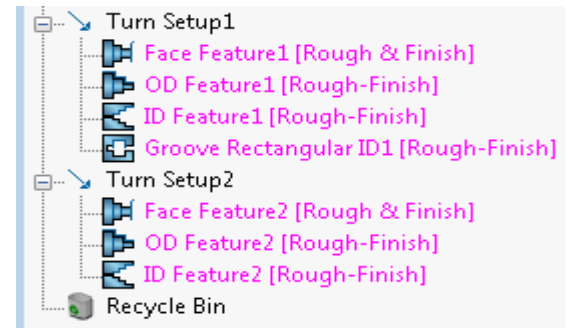


Inserting segments of the ID feature which will be machined from Setup2

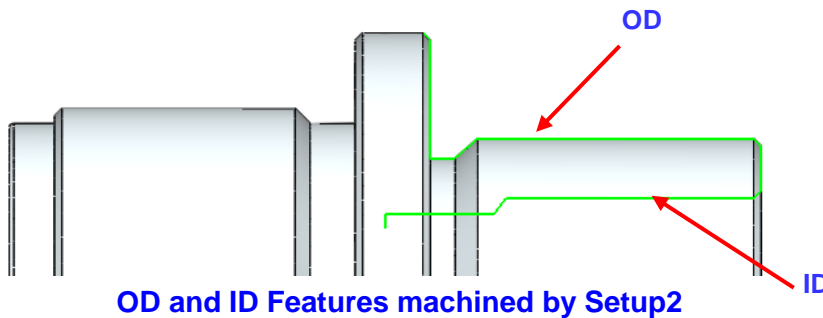


- The selected segments should be CW Face-25 through 29. If any extra face(s) are listed, highlight that face name in the *Selected Entities* list and press the *Delete* button.
 -  Click *OK* to insert the feature.
3. *ID Feature2* is added to *Turn Setup2*.

The OD and ID features in *Turn Setup2* should look like the features in the figure on the right.



ID Feature2 inserted in Setup2



OD and ID Features machined by Setup2

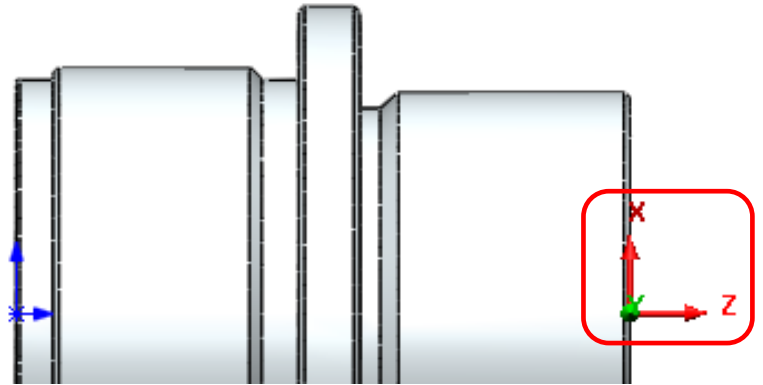
Step 6: Changing the Origin Machining Direction

1. Left click each Turn Setup in the Feature tree and notice the Origin of the machining direction indicated by the **3D Origin marker** that displays on the part model.
2. The Origin is at the back of the part for both Turn Setups. For *Turn Setup1*, this is correct; however, the origin needs to be changed for *Turn Setup2*.
3. In the CAMWorks Operation tree, double click *Turn Setup2*.

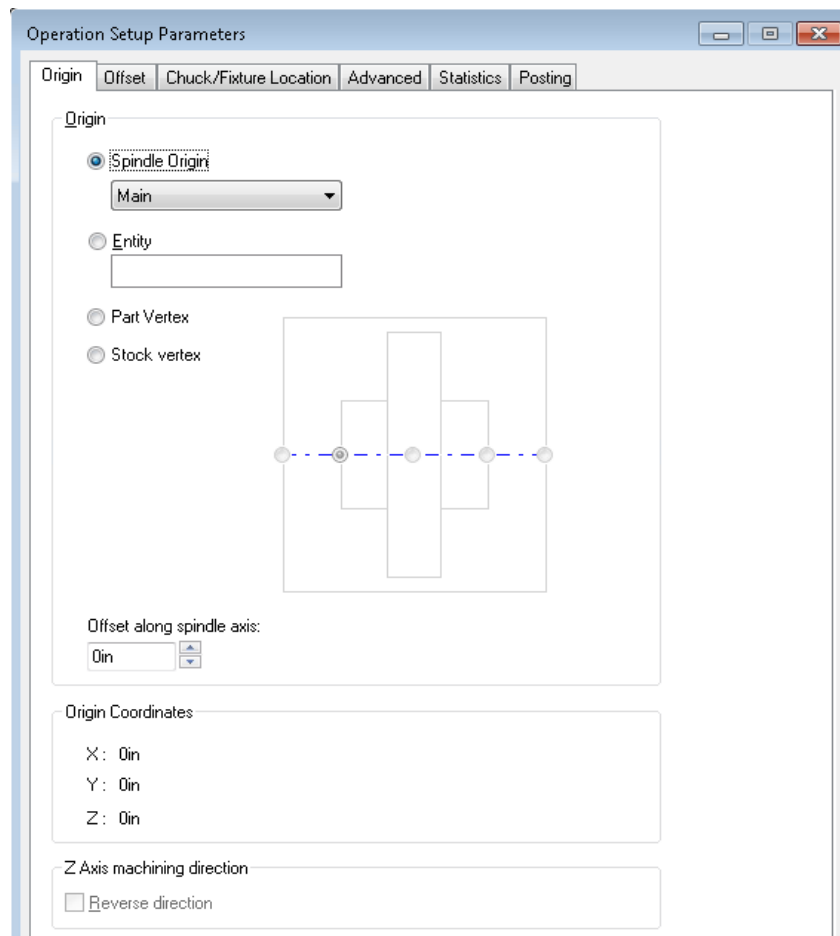
The *Operation Setup Parameters* dialog is displayed.

4. On the *Origin* tab, set the *Defined from* option to *Automatic*.
5. Click *OK* to close the dialog box.

In the graphics area, observe that the **3D Origin marker** for *Turn Setup2* has moved to the front face.



3D Origin marker displayed on the part model



Operation Setup Parameters Dialog Box

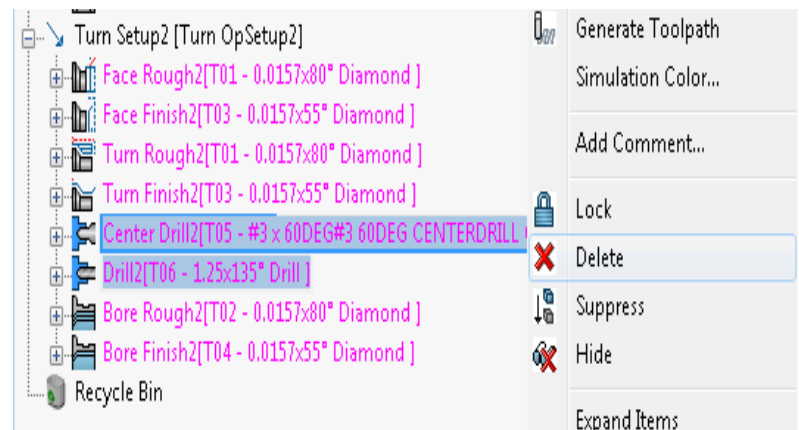


Step 7: Generating Operations and Editing Operation Parameters

1. Click the *Generate Operation Plan* button on the CAMWorks Workflow toolbar or Command Manager.
OR
Right click *CAMWorks NC Manager* in the Feature tree and select *Generate Operation Plan* from the context menu.
Operations are created for the machinable features in the Turn Setups and are listed in the Operation tree.
2. Double click *Drill1* (for *ID Feature1*) under *Turn Setup1* in the Operation tree. The *Operation Parameters* dialog will be displayed.
3. On the *Feature Options* tab:
 - Click on the *Override Machining Depth* button. Change the Machining length to **5.2in**. Increasing this length will drill the part all the way through.
 - Place a check in the *Add tip length* check box. With this option selected, CAMWorks takes into account the drill point angle when generating the toolpath.

4. Click *OK* to close the dialog box.
5. Under *Turn Setup2*, hold down the Shift key and select *Center Drill2* and *Drill2* operations in the tree.
6. Click on the *Delete* key.
7. Click *Yes* to confirm the deletion.

As a result of the changes to the drill operations in *Turn Setup1*, you can delete these operations because there will no longer be stock on the face.



8. Click the *Generate Toolpath* button on the CAMWorks Workflow toolbar or the Command Manager.


Deleting Drill Operations in Turn Setup2

9. Click each operation in the tree to view the toolpaths.




Step 8: Defining the Chuck Configuration



You are now ready to define the chuck location for each Setup and run the Toolpath Simulation.

1. Double click the *Machine [Turn Single Turret - inch]* in the Operation tree. The *Machine* dialog box is displayed.
2. Click on the *Chuck/Fixture* tab.
 - In the *Main Spindle Information* group box, click on the *Edit* button.
The *Chuck Parameter [Main Spindle]* dialog is displayed.
 - In the *Chuck Management* group box, select *[Active] 6in_2Step_Chuck* from the *Available Chucks* list.
 - In the *Jaw Parameters* group box, ensure that the *Length* and *Width* is set to **0.5in**.
 -  Click *OK* to apply the changes and close the dialog box.
3. Click *OK* to close the *Machine* dialog box.

Step 9: Setting the Chuck Display State

1.  Right click the *Stock Manager* item in the Operations tree and select *Display>>Translucent* in the cascading menu.
2.  Right click on the *Machine [Turn Single Turret - inch]* item in the Operation tree and select *Chuck/Fixture Display>> Shaded with Edges* in the cascading menu.
3. Once again,  right click on the *Machine [Turn Single Turret - inch]* item in the Operation tree and this time, select *Chuck/Fixture Display>>Machine Node* from the cascading menu. The active selection is indicated by a check next the *Machine Node* label.

When the *Machine Node* option is selected, the chuck is displayed in the graphics area whenever the *Machine* item is selected in the Operation tree.


Turn Tutorial

Chuck Parameter : [Main Spindle]

✓ ✗

Chuck Management ⌵

Available Chucks :
 ✗

Name :
 

Comment :

Preview ⌵

Chuck Parameters ⌵

Jaw Parameters ⌵

☒ Jaws In
☐ Jaws Out

Number of Jaws : ⌵

Number of Steps : ⌵

Jaw Thickness : ⌵

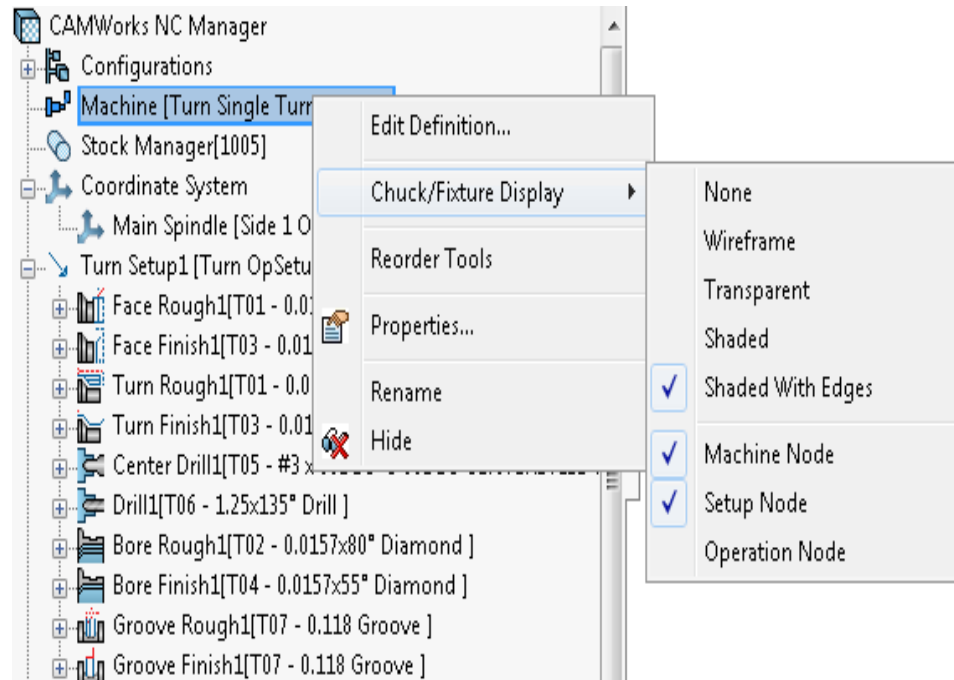
Step #	Length :	Width :
<input type="text" value="1"/>	<input type="text" value="0.5in"/> ⌵	<input type="text" value="0.5in"/> ⌵
<input type="text" value="2"/>	<input type="text" value="0.5in"/> ⌵	<input type="text" value="0.5in"/> ⌵
<input type="text"/>	<input type="text"/> ⌵	<input type="text"/> ⌵
Total	<input type="text" value="1in"/>	<input type="text" value="1in"/>

Chuck Parameters Dialog box



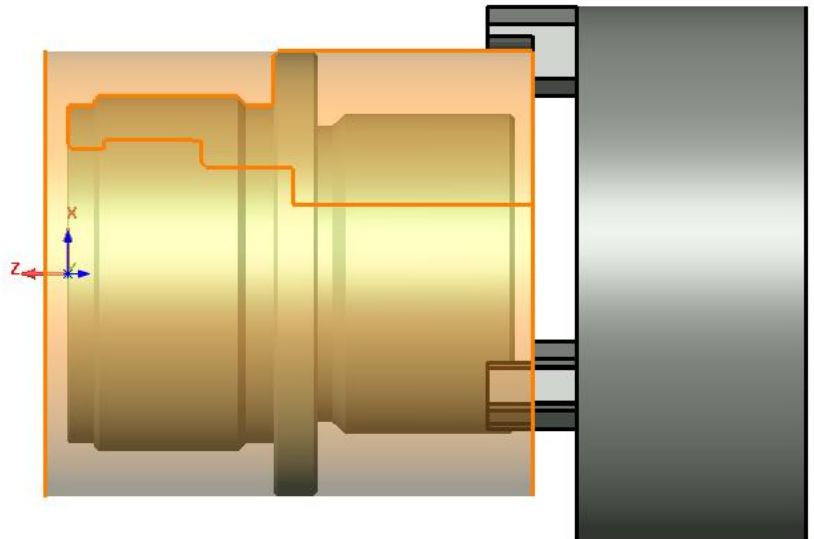
4. Once again, right click on the Machine item in the Operation tree and this time, select *Chuck/Fixture Display>>Setup Node* from the cascading menu. The active selection is indicated by a check next to the *Setup Node* label.

When the *Setup Node* option is selected, the chuck is displayed in the graphics area whenever the any of the Setups is selected in the Operation tree.



Setting Chuck Display Options

5. Left click on the *Turn Setup1* in the Operation tree. Observe that the Chuck is displayed as *Shaded with Edges* in the graphics area. As *Turn Setup1* machines *Side 1* of the part, the location of the chuck at the back of the part is correct.
6. Now, mouse over the *Stock Manager* item in the Operation tree. A translucent display of the stock is displayed in the graphics area.
7. Left click on the *Turn Setup2* in the Operation tree. Observe that the Chuck is displayed as *Shaded with Edges* in the graphics area. As *Turn Setup2* machines *Side 2* of the part, the location of the chuck displayed is incorrect. The chuck should be located at the opposite end, which is the front of the part. The chuck location needs to be changed.




Chuck Displayed as 'Shaded with Edges' for Turn Setup1



Step 10: Setting the Chuck Location

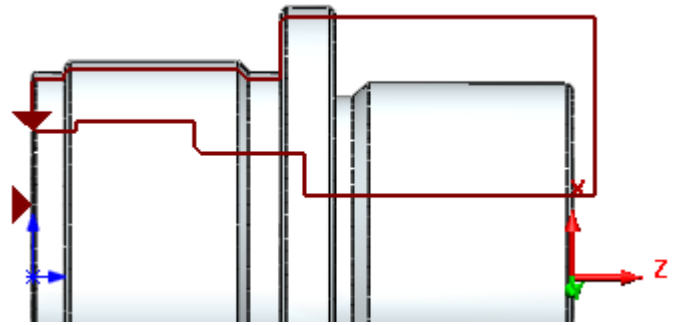
As seen in the previous step, the location of the Chuck for *Turn Setup2* needs to set at the front of the part (Side 1). The Chuck location for a given step can be changed using the *Chuck Location* tab of the *Operation Setup Parameters* dialog box.

Steps to set the Chuck Location:

1.  Right click on the *Machine* item in the Operations tree and select *Chuck /FixtureDisplay>>None* from the cascading context menu.
2. Double click *Turn Setup2* in the Operation tree. The *Operation Setup Parameters* dialog box is displayed.
3. Click the *Chuck/Fixture Location* tab.

Observe that, in the graphics area the outline of the WIP stock is displayed over the part model.

Datum triangles also display over the part model. These datum triangles indicate the location and diameter of the chuck on the WIP stock sketch.



WIP stock sketch displayed over the part model




Operation Setup Parameters

Origin Offset Chuck/Fixture Location Advanced Statistics Posting

☒ Define chuck/fixture location:

Clamping diameter (X)


Clamp on : OD with jaw : Back of Jaw

 WIP Diameter : 2.5in

Select Entity For Clamping Diameter.

Clamping location (Z)

Use : Back of jaw : Back of Jaw

 CW Edge-30 Z offset : 0in

Chuck/Fixture face : -4in Select Entity For Z Clamping Location.

Chuck/Fixture properties

Chuck/Fixture defined from : Machine

Edit....

Triangle display

Triangle size : 0.197in

Sub spindle


Spindle Z pickup : -4in Pickup offset : 0in

Transfer dist : 0in


OK Cancel Help

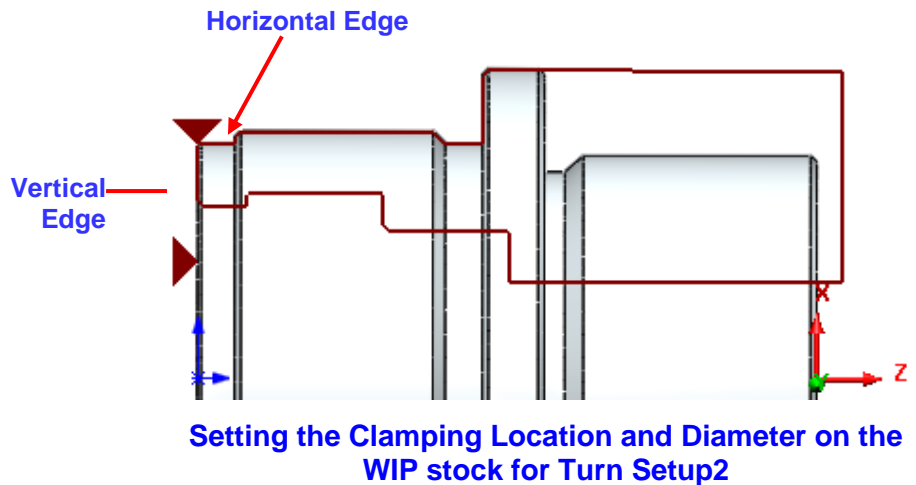
Operation Setup Parameters Dialog Box



In the *Clamping Diameter* group box of the *Operation Setup Parameters* dialog box, click on the field next to the  symbol. The mouse tip indicates that this is the *Select Entity for Clamping Diameter* field. The stock *WIP* is the default selected entity. Click in this field to set the focus.

4. In the graphics area, pick the horizontal edge representing the OD of the stock WIP. This action sets the clamping diameter of the chuck.

In the *Clamping Location* group box, click on the field next to the  symbol. The mouse tip indicates that this is the *Select Entity for Z Clamping Location* field. The stock *WIP* is the default selected entity. Click in this field to set the focus.




5. In the graphics area, pick the vertical edge representing the machined face of the part as shown in the image on the right. This action sets the chuck face. Observe that the *Entity* field displays *CW Edge-30*.

Note that you may need to zoom in to this area in order to pick these segments easily.

The datum triangle representing the chuck clamping location and the chuck (if display is enabled) move as shown in the figure.




6. Click *OK* to apply the changes and close the *Operation Setup Parameters* dialog box.

Step 11: Simulating the Toolpaths for Double Chucking

1.  Click the *Simulate Toolpath* button on the CAMWorks Workflow toolbar/Command Manager.





The *Toolpath Simulation* toolbar is displayed.

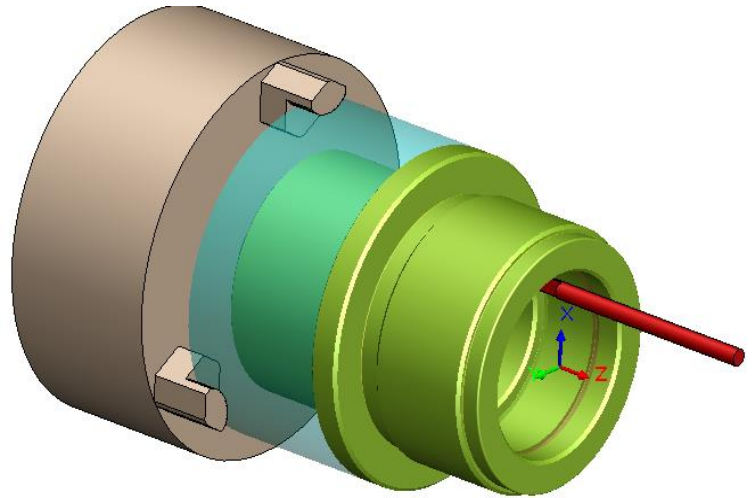
2. Click the down arrow next to *End* and select *Next Setup*.
3. Select the following display options:

-  Fixture: *Shaded With Edges*
-  Stock: *Translucent*
-  Target Part: *Shaded Display*



Turn Tutorial

-  Tool Holder: *Shaded With Edges*
 -  Tool: *Shaded with Edges*
4. Rotate the part so you can see the part being machined in *Turn Setup1*.
 5.  Click *Run* button. The simulation runs through the operations in *Turn Setup1*, then the chuck display changes to *Turn Setup2*.
 6. Rotate the part so you can see the part being machined in *Turn Setup2*.
 7.  Click *Run*. The toolpaths for the operations in *Turn Setup2* are simulated.



Toolpath Simulation for Turn Setup1

Step 12: Post Processing Toolpaths

Post process the toolpaths in the same manner as explained in [Step 8 of the Tutorial 3](#) of this chapter.



Turn 7

Turn features that are recognized automatically and created interactively are based on the spindle centerline and turn section plane. CAMWorks provides two methods for Automatic Feature Recognition (AFR):

- **Revolved Section:** This method uses a more advanced method of recognizing OD features and in most cases is capable of calculating the minimum rotary silhouette of the OD of the part. When this method is selected, the part profile is generated using an automatic method of rotating all model geometry around the turning axis and then creating a section through the revolved model. For parts with both mill and turn features, this method will generate a more complete part profile by calculating the minimum silhouette of the model as it rotates. However, if the Turn part you intend to machine does not contain any mill features, then you can use the Plane Section method.



Revolved section is the default AFR method.

- **Plane Section:** When this method is selected, the part profile is based on the section plane through the centerline of the part and along the X axis. The section plane is defined in the *Setup* tab under the *Machine* node. Any non-revolved features present on the part are ignored if they are not in line with the section plane selected in the *Setup* tab. If this section plane does not cross through all areas of the part, the resulting features will not be correct. The **Define Turn Section Plane** command allows you to define the section plane so the features will be correct.



The spindle centerline and origin are automatically determined properly for most turning parts. By default, CAMWorks will position the machining origin at one end of the part and set the positive Z machining direction to be in the same direction as the positive SOLIDWORKS/CAMWorks Solids X, Y, or Z axis.

When necessary, use the *Define Turn Section Plane* command to modify the automatically determined origin, spindle centerline, machining direction and AFR section plane (for Plane Section).

The Turn part model in this tutorial contains protrusions or ears (mill feature) on the OD of the part. If the **Revolved section** method were to be used, then the OD would have included the shape of the protrusion. However, in order to learn how to define a Turn section plane, you will use **Plane section** method.

Topics covered in this tutorial:

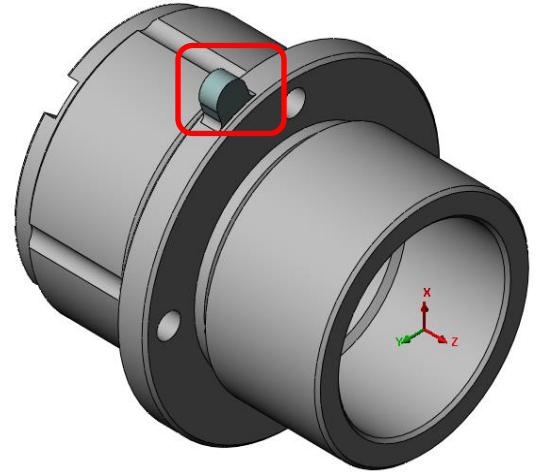
- [Setting the AFR method to Plane section](#)
- [Using Plane section to extract features correctly](#)
- [Viewing the feature relative to standard orientation](#)



Step 1: Opening the Part and Defining the Machining Parameters

1. Open part file **TURN2AX_7.SLDPRT** in the following folder.
Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Turn

Notice the protrusion on the OD of the large diameter of the part.



TURN2AX_7.SLDPRT

Defining Machining Parameters

1. Click the *CAMWorks Feature Tree* tab.
2. Under this tree, observe that *Machine [Turn Single Turret- inch]* is the active machine. In this tutorial, the part will be machined on a Single Turret Turn machine.
3. Double click the *Machine* item in the Feature tree. OR



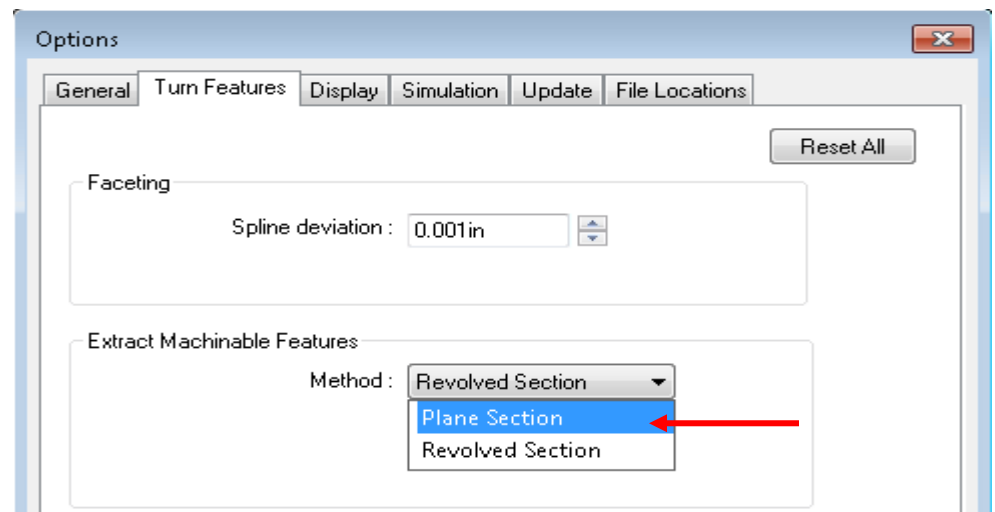
Click on the *Define Machine* button on the CAMWorks Workflow toolbar. The *Machine* dialog box is displayed.

- On the Machine tab, ensure that the *Use sub-spindle* option is unchecked.
- Click the *Tool Crib* tab, ensure *Tool Crib 2 Rear (Inch)* is the active tool crib.
- Click the *Post Processor* tab and make sure *T2AXIS-TUTORIAL* is selected.
- Click *OK* to apply the changes and close the dialog box.

Step 2: Extracting Machinable Features using Plane Section Method

Setting the Method for AFR

1. Click the *CAMWorks Options* button on the CAMWorks Command Manager.
OR
Select *Options* from the CAMWorks menu.
The *Options* dialog box will be displayed.




Setting the AFR method in Turn Features Tab of Options dialog box



2. Click on the *Turn Features* tab.
3. On this tab, in the *Extract Machinable Features* group box, set the Method to *Plane Section*.
4. Click *OK* to apply the changes and close the dialog box.

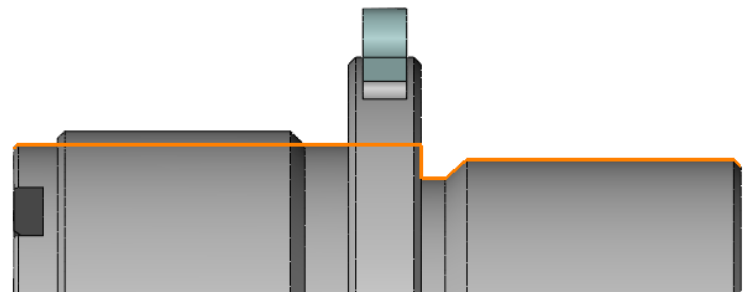
Extracting Machinable Features

1.  Click on the *Extract Machinable Features* button on the CAMWorks Workflow toolbar or Command Manager.
2. The features recognized are listed in the CAMWorks Feature tree.
3. Change the *View Orientation*






4. Left click *OD Feature1* in the tree and notice that the OD feature profile does not include the shape of the protrusion.

You need to define a section plane that cuts through the protrusion.



OD profile (highlighted in orange color) that does not include the shape of the protrusion

Step 3: Using the Plane Section to Extract Machinable Features Correctly

1. Change the *View Orientation*  to *Right* .
2. Right click *CAMWorks NC Manager* item in the  CAMWorks Feature tree and select *Define Turn Section Plane* from the context menu.
3. The *Setup* tab of the *Machine* dialog box is displayed. For this part, the origin is on the front right side of the part.



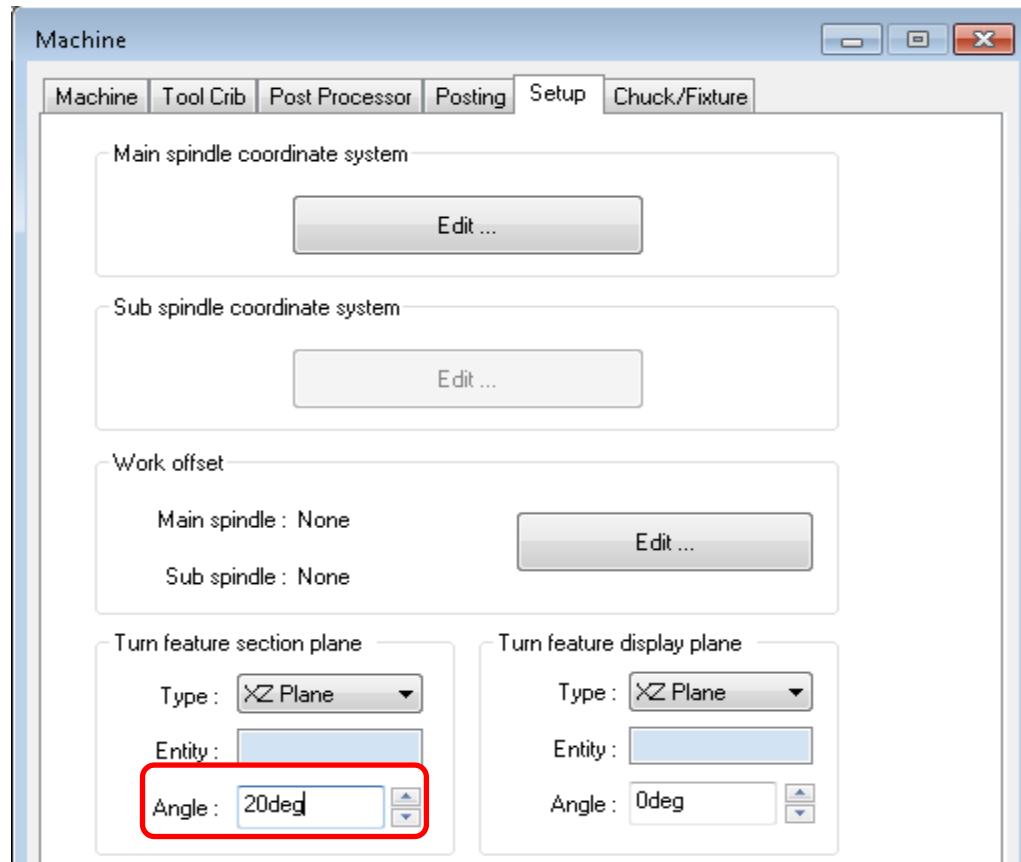
On the *Setup* tab in the *Machine* dialog box, the *Setup* definition options define the machining origin and the location of the spindle centerline for the part. The origin specifies program zero and the Z axis identifies the location and direction of the spindle axis, which determines the features that will be recognized automatically and created interactively. If the part contains more than one centerline direction, you can define the desired centerline by selecting a *SOLIDWORKS/CAMWorks Solids coordinate system*.

In this tutorial, there is only one logical centerline direction and the Automatic setting finds it properly.

4. The center of the protrusion is 20 degrees counterclockwise off the vertical, so you need to rotate the section plane to cut through the protrusion. In the *Turn feature section plane*



group box, set the Angle to **20deg**. In the graphics area, notice that the X axis now points through the protrusion.



Setting the Angle to 20 degrees for the Turn Feature section plane



When recognizing features using *Plane Section*, CAMWorks sections the part through the centerline and along the CAMWorks X axis. If this section plane does not cross through all areas of the part, the resulting recognized features will not be correct.

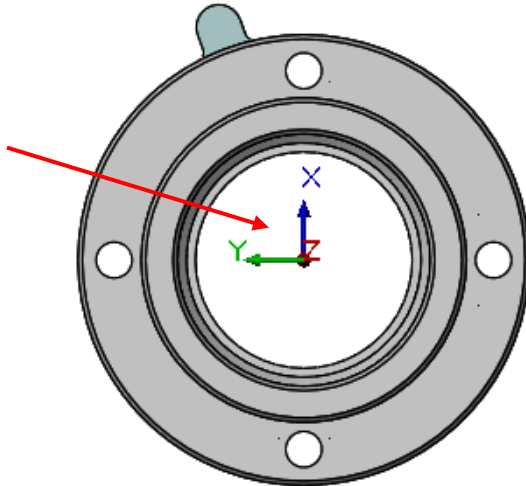
By default, the X axis, which defines the section plane, is vertical and will not cut through the protrusion, so the protrusion would not be included in the OD feature.

5. Press the tab button to the keyboard to switch focus from the *Angle* field of *Turn feature section plane* to that of *Turn feature display plane*. (Alternatively, you may click on any field within the *Turn feature display plane* group box to shift the focus.)

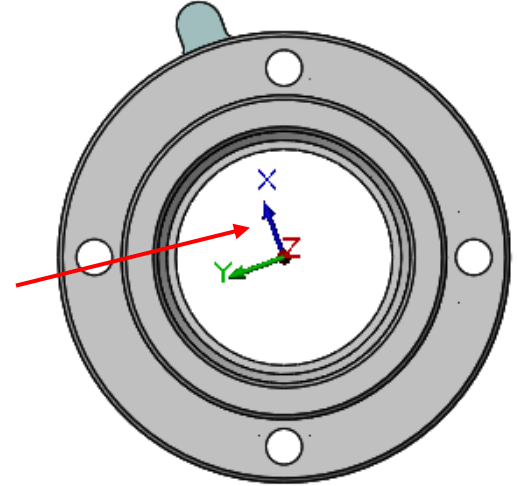
Notice that the X axis is vertical again. This is because the *Turn feature display plane* section of the dialog box is currently set to 0 degrees.



The *Turn feature display plane* is the plane that the features are displayed parallel to. This plane can be set separately from the *Turn section plane*. The display plane is provided for viewing purposes only and has no effect on AFR.

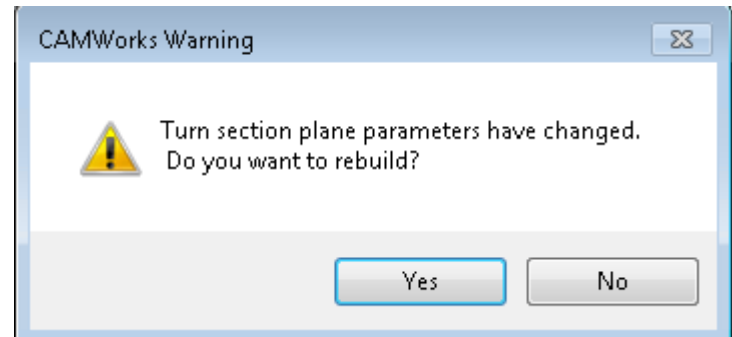


Protrusion at 20degrees
counterclockwise to the X axis

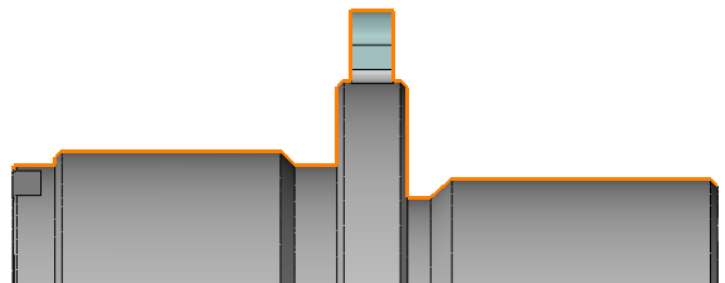


X axis points through the
protrusion after aligning

6. To graphically verify that the rebuilt OD feature includes the protrusion, set the Angle for the *Turn feature display plane* also to **20deg**.
7. Click *OK* to apply the change and close the dialog box.
8. Click *Yes* when a warning message is displayed asking if you want to rebuild.
9. In the CAMWorks Feature tree, right click *Turn Setup1*, select *Set View* on the context menu, then select *Turn ZX* from the cascading menu.
CAMWorks displays turn features from a single view that is normal to the *Turn feature display plane*.
10. Click on the *OD feature1* in the tree to verify that the protrusion is included in the feature.



Warning message



OD profile recognized by Plane section now
includes the protrusion



Viewing the Feature Relative to the Standard Orientation

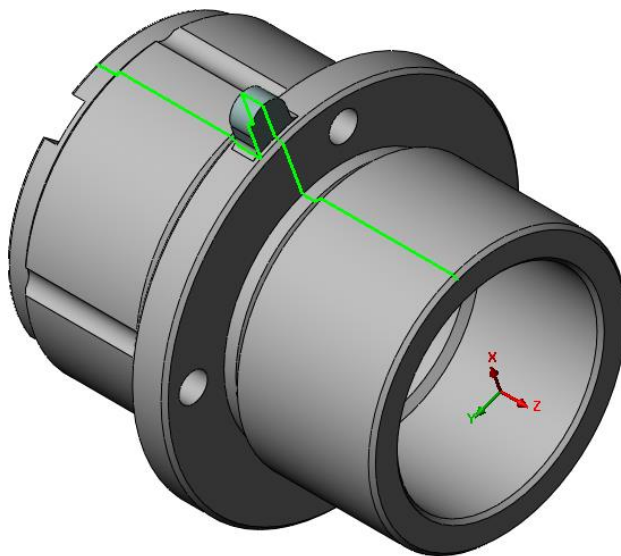
In many cases, you may not have a SOLIDWORKS/CAMWorks Solids orientation that is normal to the Turn section plane and you may want to view the features relative to a standard orientation. To do this, you can set the *Turn feature display plane* to be parallel to a standard orientation.



Turn Tutorial

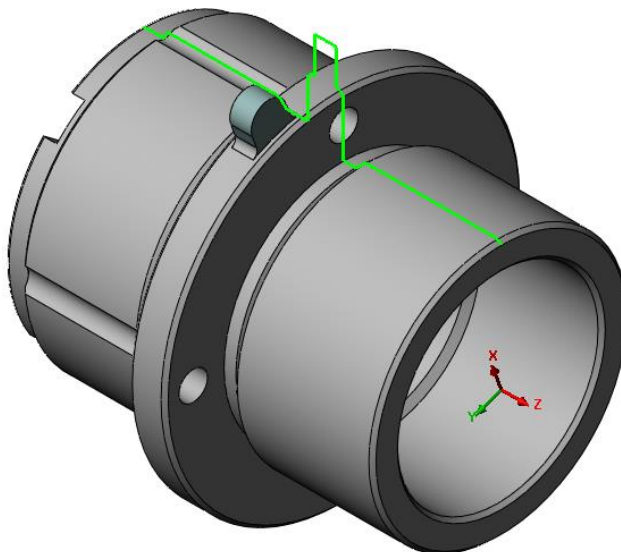
Following are the steps:

1. Double click the *Machine [Turn Single Turret – inch]* in the Feature tree.
The *Machine* dialog box will be displayed.
2. Click on the *Setup* tab. (This is another way to access the Setup tab.)
3. On the *Setup* tab in the *Turn feature display plane* group box, set the *Angle* back to **0deg**.
4. Click *OK*.
5. Change the *View Orientation*  to *Isometric* .
6. Click the *OD feature1* in the Feature tree and notice the difference in the display.



Turn feature section plane	Turn feature display plane
Type: <input type="text" value="YZ Plane"/>	Type: <input type="text" value="YZ Plane"/>
Entity: <input type="text"/>	Entity: <input type="text"/>
Angle: <input type="text" value="20.00deg"/>	Angle: <input type="text" value="20.00deg"/>

OD profile view with orientation aligned with
Turn section plane



Turn feature section plane	Turn feature display plane
Type: <input type="text" value="YZ Plane"/>	Type: <input type="text" value="YZ Plane"/>
Entity: <input type="text"/>	Entity: <input type="text"/>
Angle: <input type="text" value="20.00deg"/>	Angle: <input type="text" value="0.00deg"/>

OD profile viewed relative to standard
orientation



Turn 8




In this tutorial, you will learn to define a thread feature and associated operations to cut double and triple lead threads.

Topics covered in this tutorial:

- [Defining a Thread Feature](#)
- [Cutting Multiple Start Threads](#)
- [Using the 'Process by Level' option for a Threading Operation](#)
- [Simulating a Threading toolpath](#)

Step 1: Open the Part and Defining the Machine Parameters

1. Open part file **TURN2AX_8.SLDPRT** in the following folder.
Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Turn

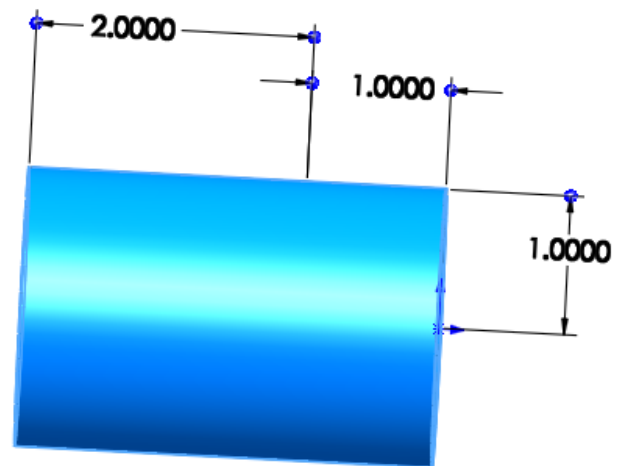
2.  Click the *CAMWorks Feature Tree* tab.
3. Under this tree, observe that *Machine [Turn Single Turret- inch]*  is the active machine. In this tutorial, the part will be machined on a Single Turret Turn machine.
4.  Double click the *Machine* item in the Feature tree.

OR






Click on the *Define Machine* button on the CAMWorks Workflow toolbar. The *Machine* dialog box is displayed.

- On the Machine tab, ensure that the *Use sub-spindle* option is unchecked.
- Click the *Tool Crib* tab, ensure *Tool Crib 2 Rear (Inch)* is the active tool crib.
- Click the *Post Processor* tab and make sure *T2AXIS-TUTORIAL* is selected.
- Click *OK* to apply the changes and close the dialog box.



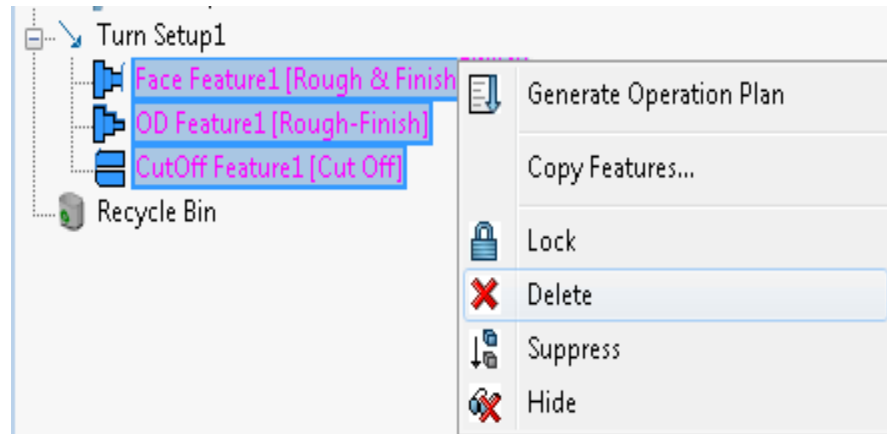
TURN2AX_8.SLDPRT


Step 2: Defining a Thread Feature

1.  Click on the *FeatureManager Design* tree.
2. Right click  *Revolve1* in the FeatureManager design tree and select  *Edit Sketch* from the context menu.
3. In the graphics area, observe that two separate line segments are displayed.




Notice that the length of the cylindrical turn part is 3 inches and radius is 1 inch. In this tutorial, you will be threading only the first inch at the front of the part. A separate line segment of 1 inch length has been created so that you can define the feature being machined for the length of the thread. You could also use one long line and edit the operation's feature length to cut only the first inch.



4.  Click the *Exit Sketch* command button to exit the sketch.

Deleting the three features recognized by EMF command

5.  Click the *Extract Machinable Features* button on the CAMWorks Workflow toolbar or Command Manager.

AFR finds a Face Feature, an OD Feature and a Cutoff Feature.

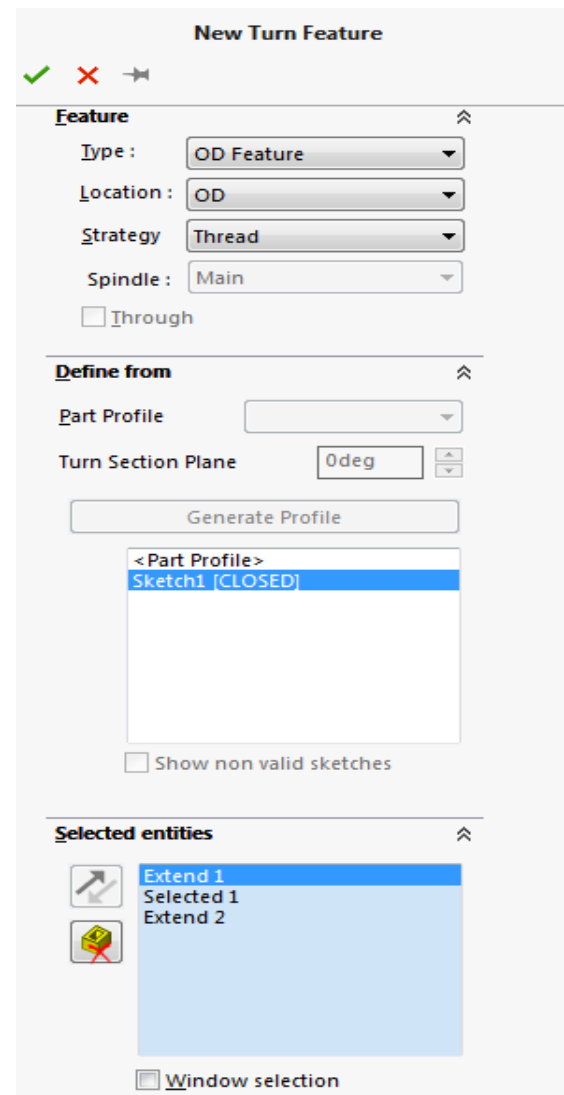
6. In the CAMWorks Feature tree, select the three features then right click and select *Delete* from the context menu.

As these features are not going to be machined, they are to be deleted.

7. Click *Yes* to confirm the deletion.
8. Right click the *Recycle Bin* and select *Empty* from the context menu. Click *Yes* to confirm the deletion.
9. Right click *Turn Setup1* in the tree and select *Turn Feature* from the context menu.



The *New Turn Feature* dialog box is displayed.

10. In the *Feature* group box,
- Ensure that the Feature Type is set to *OD Feature*.
 - Set the Strategy to *Thread*.
11. In the *Define from* group box, highlight *Sketch1* in the list.




New Turn Feature Dialog box

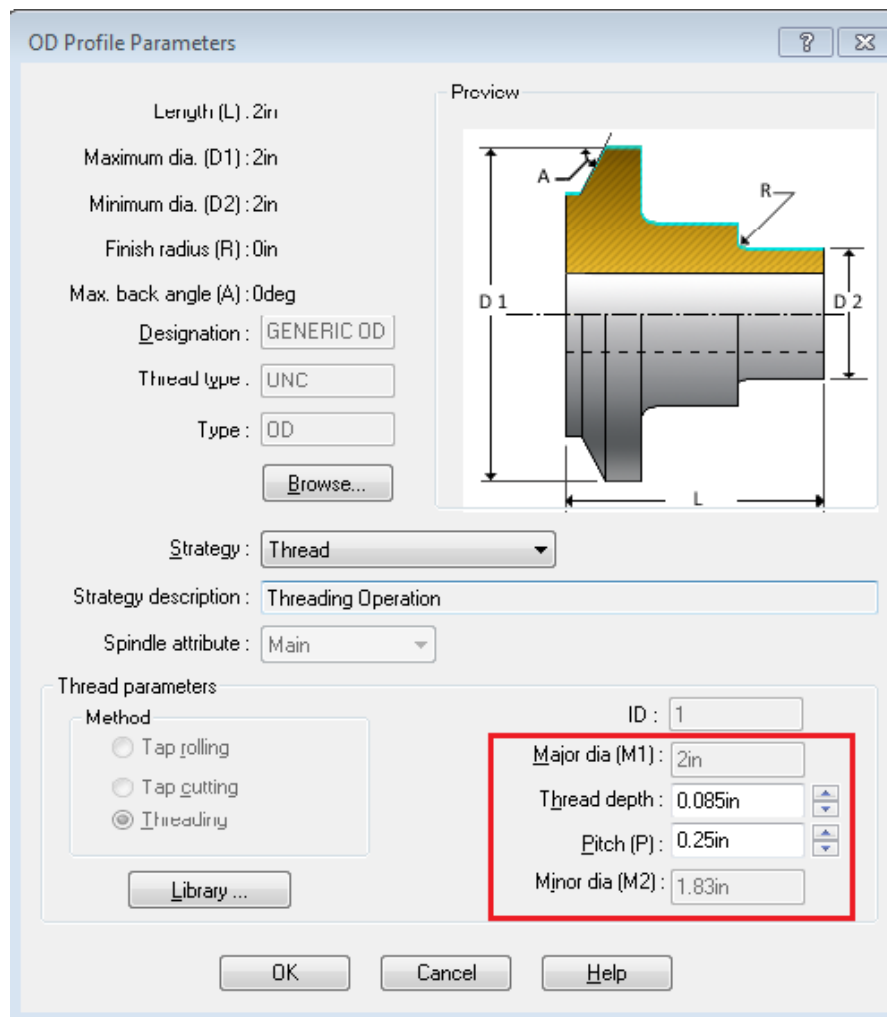
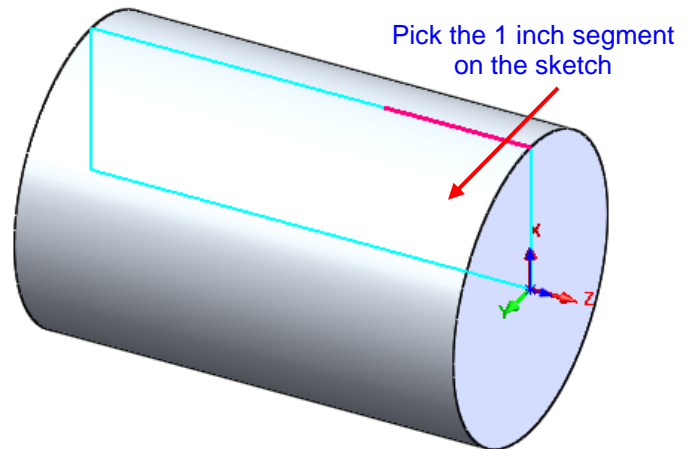


12. In the graphics area, either change the *View Orientation*  to *Isometric* .
- OR

Rotate the part so that you can see the sketch.

13. Click in the *Selected Entities* list box of the dialog box to set the focus.
14. Move the pointer over the sketch and pick the **1 inch line segment**. The segment *Selected 1s* is added to the *Selected Entities* list in the dialog box.
15.  Click *OK* to insert the feature.

Observe that *OD Feature2* is added to the Feature tree.



OD Profile Parameters dialog box



16. Right click *OD Feature2* in the tree and select *Parameters* in the context menu.


The *OD Profile Parameters* dialog box will be displayed.

17. In the *OD Profile Parameters* dialog box, set the following in the Thread Parameters group box:

- Thread Depth = **0.085in**
- Pitch = **0.25in**

18. Click *OK* to apply the changes and close the dialog box.

Step 3: Cutting Multiple Start Threads

1.  Click the *Generate Operation Plan* button on the CAMWorks Workflow toolbar or Command Manager.

2. Right click *Turn Setup1* and select *Set View>>Turn ZX* from the context menu.

3. Double click *Thread1* in the Operation tree.

The Operation Parameters dialog box is displayed.

4. Click on the *Thread* tab.

5. In the Parameters group box:

- Change the Depth per cut to **0.050in**.
- Set the Final cut amount to **0.0in**.

6. In the Multi start group box, leave the *Number of starts* set to **1**.

7. Click the *Preview* button to generate and view the toolpath.

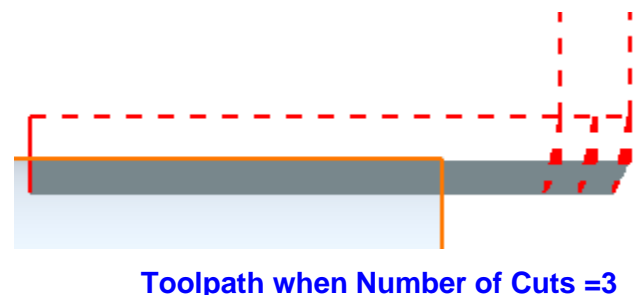
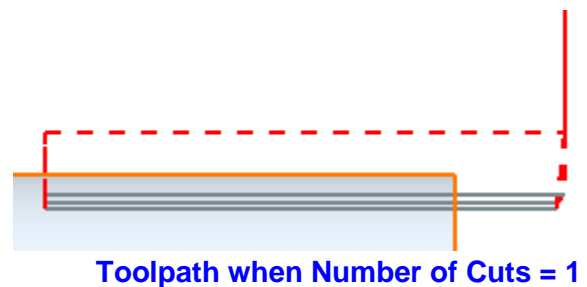
8. Double click the blue title bar to expand the dialog box.

9. On the Thread tab in the Multi start group box, set the *Number of starts* to **3**.

10. Ensure that you leave the *Process by level* option unchecked. When this option is not checked, CAMWorks generates each start to depth.

11. Click the *Preview* button to generate and view the toolpath. Notice the three leads.

12. Double click the blue title bar of the Operation Parameters dialog box to expand the dialog box.






13. Click *OK* to apply the changes and close the dialog box.

Step 4: Enabling the 'Process by Level' Option for a Threading Operation




1. Double click the *Thread1* operation in the Operation tree. The *Operation Parameters* dialog box is displayed.
2. Click on the *Thread* tab.
3. In the *Multi start* group box, check the *Process by level* option.



When the 'Process by Level' option is selected for a threading operation, CAMWorks generates all starts at the first depth level, then processes all starts at the next level and so on for each depth.

4. Click *OK* to apply the changes and close the dialog box.
5.  Regenerate the toolpath by clicking on the *Generate Toolpath* button on the CAMWorks Command Manager or Workflow Toolbar.

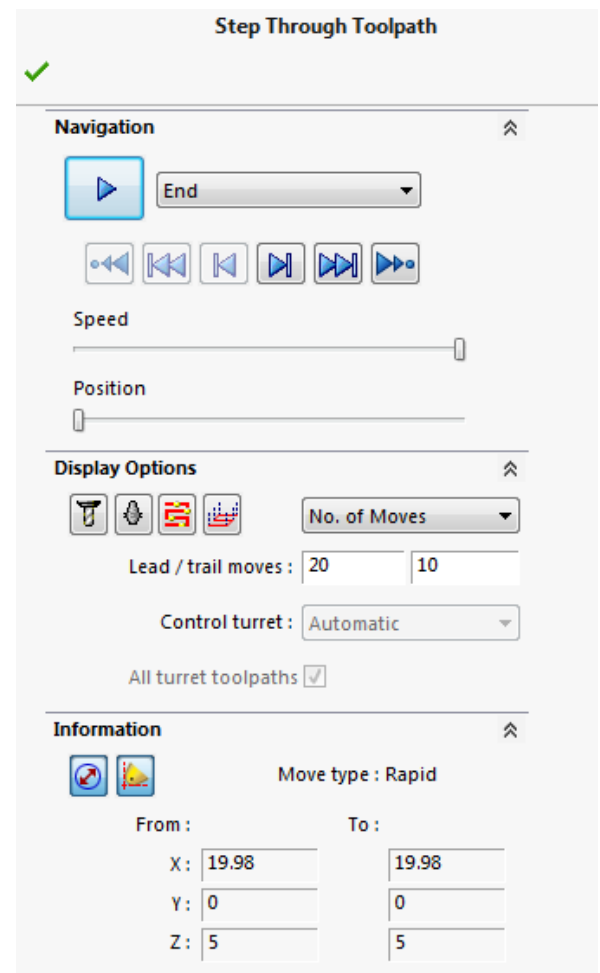
Step 5: Step Through Toolpath

1.  Click the *Step Thru Toolpath* button on the CAMWorks Workflow toolbar or Command Manager. The *Step Through Toolpath* dialog box is displayed.
2. In the *Display Options* group box, set the following:
 -  Tool: *Shaded with Edges*
 -  Tool Holder: *Shaded with Edges*
 - From the *Display Number of Toolpath records* dropdown list, select *No. of Moves*.
 - Assign a value of **20** in the *Number of Leading moves* field.
 - Assign a value of **10** in the *Number of Trailing moves* field.



Use the *Number of Leading moves* field to input the number of toolpath records ahead of the current tool position which are to be displayed.





Use the *Number of Trailing moves* field to input the number of toolpath records following the current tool position




Step Through Toolpath



which are to be displayed.

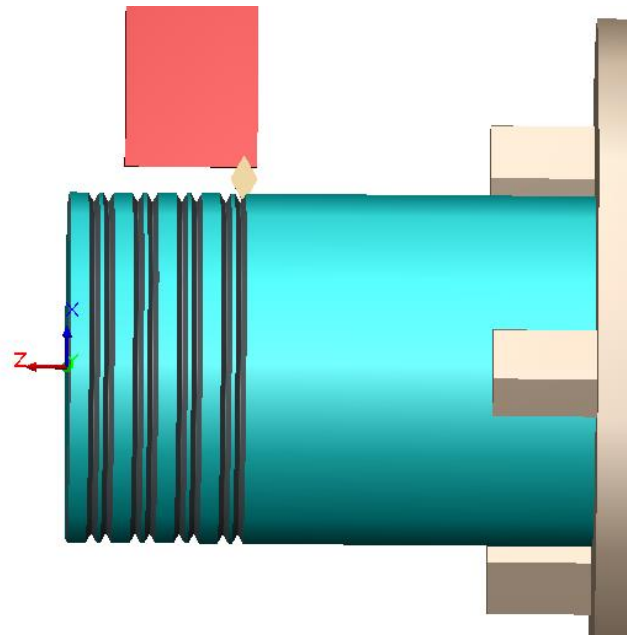
3. In the Information group box:
 -  Ensure that the *Radial or Diameter Coordinate Display* button is indented. This will output the XZ coordinates to the tool tip.
 -  Ensure that the *Tip or Center Coordinate Display* button is indented. This will output the X coordinates in Diameter mode.
4.  Click on the *Forward Single Step* button in order to step through the toolpath to view the processing order.
5.  Click the *OK* button to close the *Step Through Toolpath* dialog box.

Step 6: Simulating the Threading Toolpath



1.  Click the *Options* button on the CAMWorks Command Manager. OR Select *Options* from the CAMWorks menu. The *Options* dialog box is displayed.
2. Click on the *Simulation* tab.
3. On this tab, check the *Simplified threading* option in the *Display* group box.









The 'simplified Threading' option can be used to simulate a thread in a simplified form to improve performance. When this option is not checked, CAMWorks simulates the full thread, which can result in very slow simulation speeds. When this option is checked, CAMWorks simulates the thread as if the cut is a normal turning cut and thread simulation should be much faster.

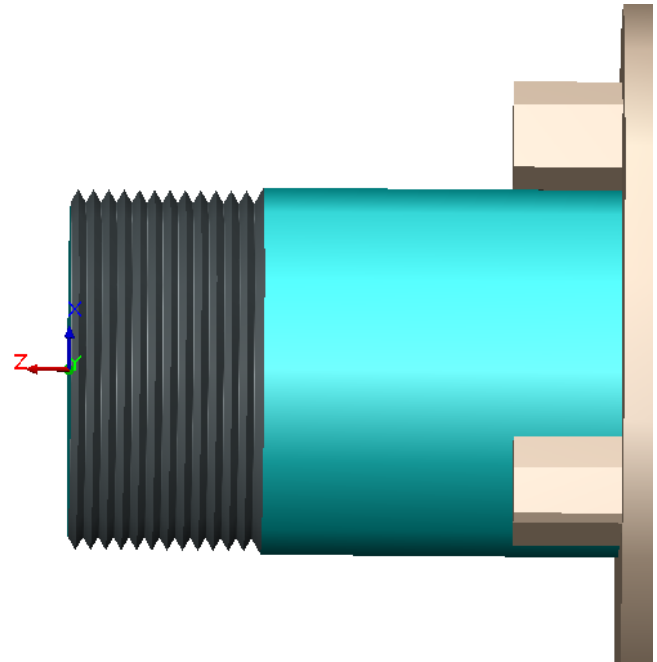


Thread Toolpath Simulation in progress

4. Click *OK* to apply the changes and close the dialog box.
5. Right click *Turn Setup1* in the tree, select *Set View>>Turn -ZX* from the cascading context menu.
6.  Click the *Simulate Toolpath* button on the CAMWorks Workflow toolbar or Command Manager.
7. Set the following Display Options:
 -  Fixture: *No display*.



-  Target Part: *No display*
 -  Tool: *Shaded with Edges*
 -  Tool Holder: *Shaded with Edges.*
 -  Stock: *Shaded with Edges*
8.  Use the *Zoom to Area* button to zoom to the front of the part.
 9.  Click the *Single Step* button repeatedly. View how each toolpath cuts into the part to generate thread feature.



Completed Thread Feature after Toolpath Simulation



Chapter 2: Learning 4 Axis Turn

CAMWorks uses the Rear turret for **2 Axis Turn machining** and as the default for 4 Axis Turn machining. All tool motion and output is displayed and output from a Rear turret perspective.

When the machine has **4 Axis capability**, you can define the machine in the Technology Database to use both Rear and Front turrets. Then, you can choose to have the display and output for any of the operations in the Operation tree generated from the Rear or Front turret (providing there is a tool available on the Front turret).

This chapter provides an opportunity to learn CAMWorks 4 Axis Turn. The exercises in this chapter are intended to show you how to use CAMWorks and may not correspond to actual machining practices.

The exercise parts are installed when you install CAMWorks and are in the `\Examples\Tutorial_Parts\Turn` folder. This folder is inside the CAMWorks data folder (Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Turn).

IMPORTANT! CAMWorks uses a set of knowledge-based rules to assign machining operations to features. The Technology Database contains the data for the machining process plans and can be customized for your facility's machining methodology. When you do these exercises, your results may not be exactly the same as described in the steps and illustrated in the figures. This is because the machining sequences and operations data in your Technology Database may be different from the database used to produce the documentation.



4 Axis Turn 1

CAMWorks provides support for machines with front and rear turret configurations allowing **2+2 axis** machining. For machines that are defined as **4 axis** in the Technology Database, an option is provided to select either Front or Rear turret for each operation.

Topics covered in this tutorial:

In this tutorial, you will explore how to machine a turn part using 4axis machining. The following aspects will be covered in detail:

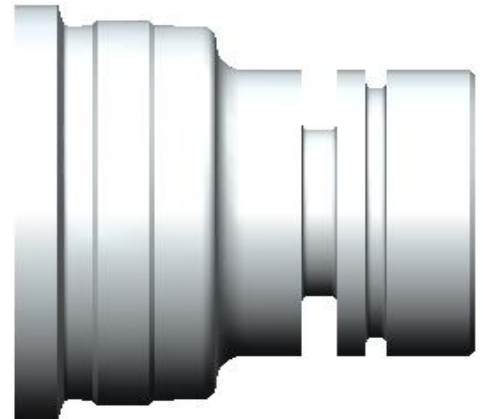
- [Defining a 4 Axis machine](#)
- [Defining machinable turn features interactively](#)
- [Defining the Chuck location for multiple Turn Setups](#)
- [Simulating toolpaths for operations machined from Multiple Turn Setups](#)

Step 1: Opening the Part

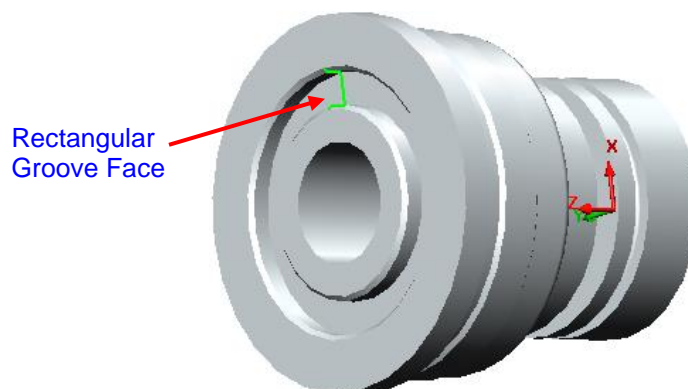
1. Open part file **TURN4AX_1.SLDPRT** in the following folder.

Drive:\CAMWorksData\CAMWorks202x\Examples\
Tutorial_Parts\Turn

2. Observe that there are 5 grooves on the part in this tutorial: 2 face grooves, 2 OD grooves and 1 ID groove. There is also an OD thread that will be machined. In this tutorial, all the features will be machined from the Main Spindle. However, the rear end of the part contains a rectangular groove on the face feature which cannot be machined from the same turn setup as other features (unless Sub Spindle functionality is used). This feature will be machined from a different Turn Setup along with an OD feature and Face feature.



TURN4AX_1.SLDPRT



Rectangular
Groove Face



Step 2: Define the Machine Parameters for 4 Axis Machining

1. Click the *CAMWorks Feature Tree* tab.
2. Under this tree, observe that *Machine [Turn Dual Turret- inch]* is the active machine. In this tutorial, the part will be machined on a Dual Turret Turn machine.
3. Double click the *Machine* item in the Feature tree.

OR



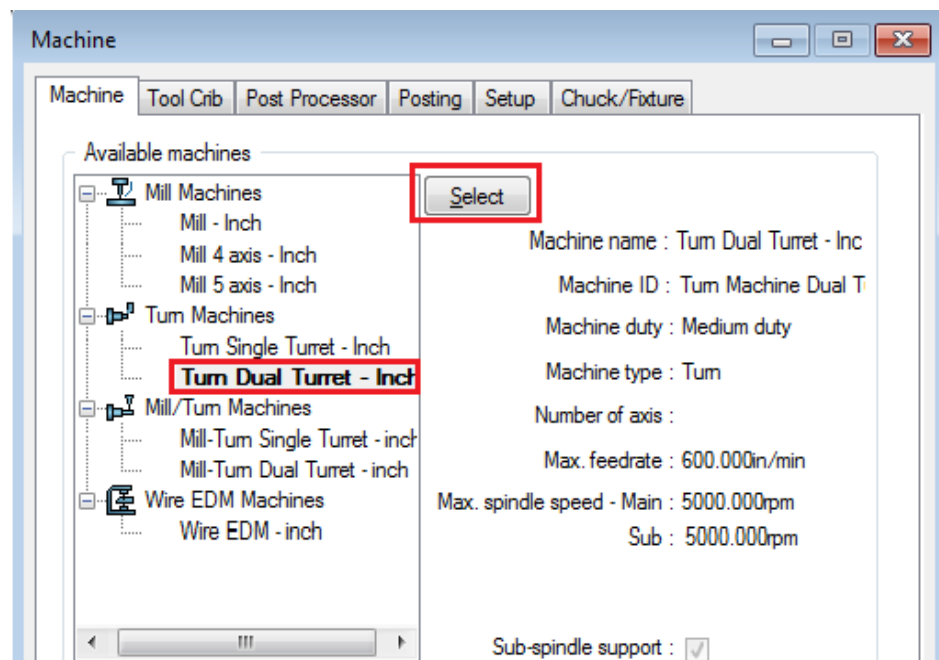
Click on the *Define Machine* button on the CAMWorks Workflow toolbar. The *Machine* dialog box is displayed.

4. On the Machine tab:

- Ensure that the *Turn Dual Turret - inch* machine is selected.

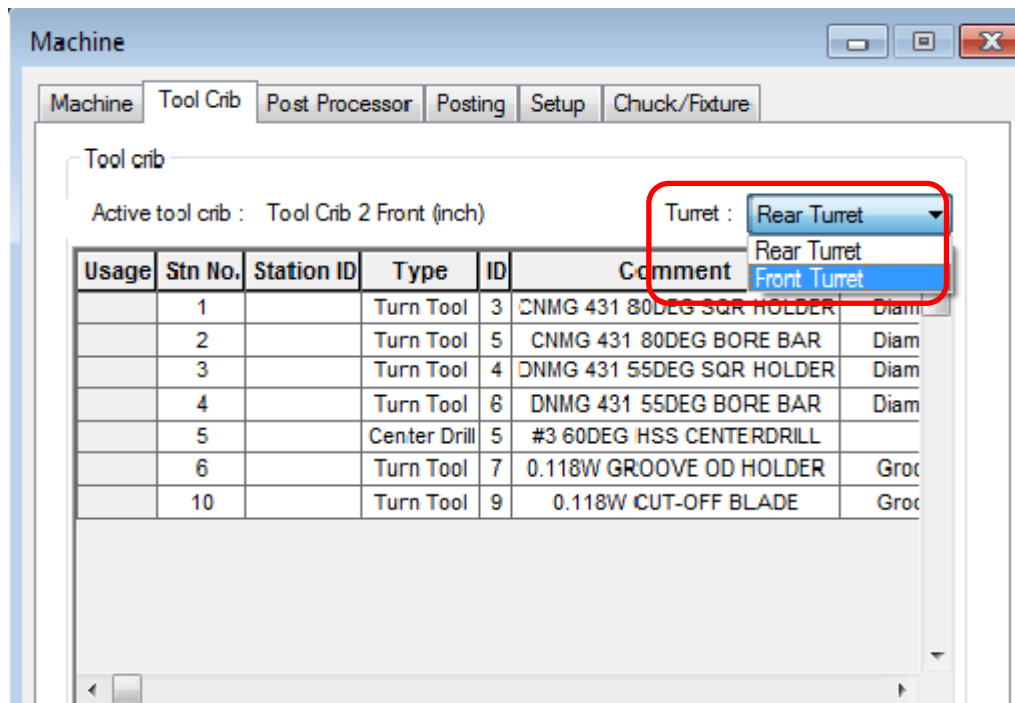
This machine has been defined as a 4 axis machine in the TechDB. Before using CAMWorks to machine your parts, make sure you define the machine tools available in your facility.

- On the Machine tab, ensure that the *Use sub-spindle* option is unchecked.



Selecting Turn Twin Turret - inch Machine



5. Click on the *Tool Crib* tab.
 - At the top-right corner, in the *Turret* dropdown list, ensure *Rear Turret* is selected. For this Rear Turret, highlight *Tool Crib 2 Rear (Inch)* in the *Available tool cribs* list and click *Select*.
 - In the *Turret* dropdown list, change the Turret to *Front Turret*.
 - Highlight *Tool Crib 2 Front (Inch)* in the *Available tool cribs* list and click *Select*.



Assigning Tool crib to the Front turret

6. Click the *Post Processor* tab and make sure *T4AXIS-TUTORIAL* is selected.
This tutorial post processor supports 2+2 axis output. When you use CAMWorks to machine your own parts, select your machine tool controller or post processor.
7. Click *OK* to apply the changes and exit the *Machine* dialog box.

Step 3: Defining the Stock


1. Double click *Stock Manager* in the Feature tree.
2.  Leave the Type set to *Bar stock*.
3. Set the following parameters:
 - *Outside Diameter* to **4.25in**
 - *Stock Length* to **4.1in**
 - *Back Of Stock Absolute* to **-4.05in**
4.  Click *OK* to apply the changes and close the dialog box.



Step 4: Defining Features Automatically and Interactively

In order to define the machinable features on this part, you will run the *Extract Machinable Features* command.

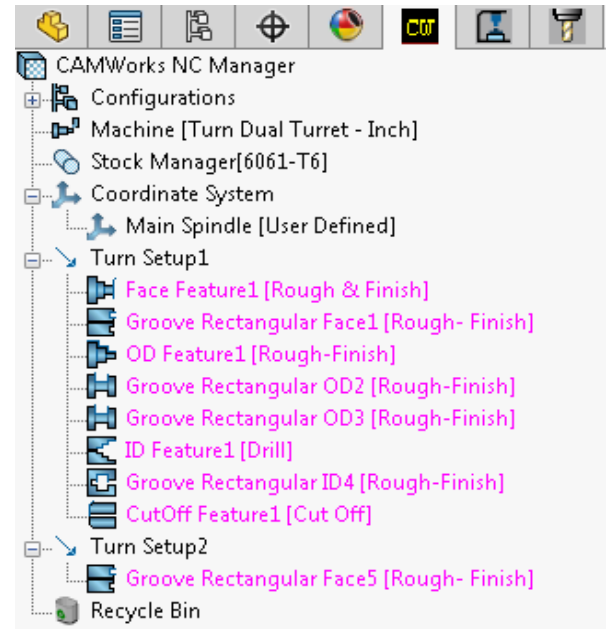
Since this turn part will be chucked, two different setups will be used to machine the features. The rear portion of the part will be machined from a different setup from the front portion of the part. Also, a portion of the OD feature will be threaded.

1.  Click the *Extract Machinable Features* button on the CAMWorks Workflow toolbar or Command Manager.

OR

Right click *CAMWorks NC Manager* and select *Extract Machinable Features* from the context menu.

Two Turn Setups are created and AFR recognizes the features that can be machined in each Setup.

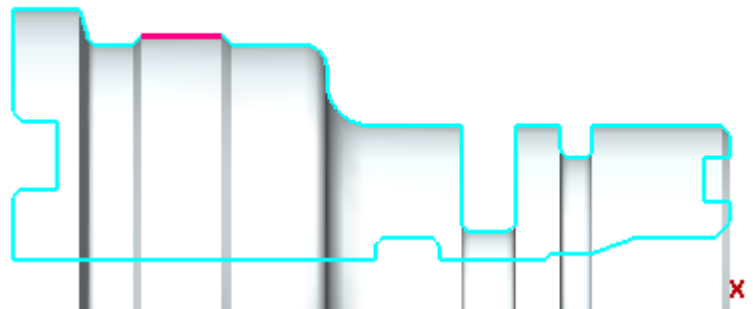


Features recognized by EMF


2. Since you will be chucking, right click on *Cutoff Feature1* in the *Turn Setup1*, select *Delete* on the context menu and click *Yes* to confirm the deletion.
3. Right click on *Groove Rectangular OD3* feature in the tree and select *New Turn Feature* from the context menu.

The *New Turn Feature* dialog box will be displayed.

- In the Feature group box:
 - Select *OD Feature* for Type.
 - For the Strategy, select *Thread* from the list.
- Click in the *Selected entities* list box to set the focus.
 - Pick the horizontal segment as shown in the figure.
 - The selected face entity (*CW Face-24*) and two Extend entities display in the *Selected entities* list. You do not need to extend either end of this segment.



Highlight the segment to be threaded

4.  Click *OK* to insert the feature.

OD Feature2 [Thread] is listed in the Feature tree after *Groove Rectangular OD3*.



5. Right click *OD Feature2* and select *Parameters* from the context menu.
The *OD Profile Parameters* dialog box is displayed.
 - In this dialog box, set the following thread parameters:
 - Major dia = **3.85in**
 - Thread depth = **0.05in**
 - Pitch = **0.05in**
 - Minor dia = **3.75in**
6. Click *OK* to apply the changes and close the *OD Profile Parameters* dialog box.

Step 5: Editing Machinable Features

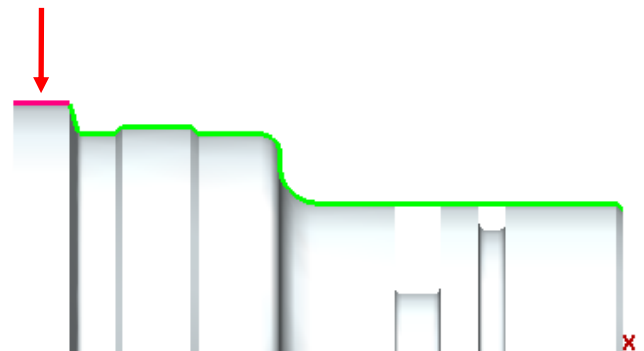
One of the advantages of CAMWorks is that WIP (work-in-process) is updated after generation of every toolpath, which reduces the amount of air cutting. The feature geometry defines which portions of the part to cut and is based on the part itself. In almost all cases the initial stock will be larger than the finished part size and CAMWorks must determine how to intersect the feature geometry to the stock WIP. By editing the feature geometry you can modify which portion of the part to machine and how the feature will extent to the WIP. In the following steps, you will learn how to remove segments from a feature and learn how to set the extent direction for each end of the feature.

Extends of the OD feature

1. Double click *OD Feature1* in the Feature tree.
The *Edit Turn Feature* dialog box is displayed.
2. Observe the *Selected entities* list box and note that the feature is defined by 13 geometry segments and two *Extend* segments.
3. Since the rear of the part will be machined from a separate setup, remove the segment that will machine the largest diameter at the rear of the part. This segment will be later [machined as an OD feature from Turn Setup2](#).

Removing this segment from the list can be done in either of the following two ways:

- By highlighting the segment in the *Selected entities* list and pressing the *Delete* key.
- By simply picking the entity on the OD profile of the part in the graphics area.



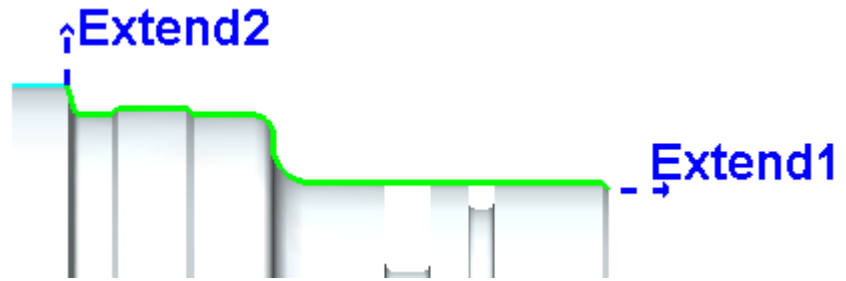
Pick the segment indicating the largest diameter

After you remove the segment using one of the above two methods, note that the feature now contains only 12 geometry segments.



4. The *Extend* segments define how the start and end of the feature will project to meet the WIP. Now that you have removed the last geometry segment from the feature, you want to define the extend segment at the rear of the part to project in direction X towards the OD of the stock. To determine which extend segment is associated to the last geometry segment, pick either the first or last *Selected entity* in the list to see which graphically highlights. In most cases the last segment in the list will be towards the rear of the part.

- You would want the feature extend direction at the front of the part to be along the Z direction. Highlight *Extend1* in the Selected Entities group box. In the *Extend1* group box (representing the extent segment at the front of the feature), select *Along Z*.




Extending the segments for OD Feature1

- Highlight *Extend1* in the Selected Entities group box. In the *Extend2* group box, (representing the extent segment at the back of the feature), select *Along X*.



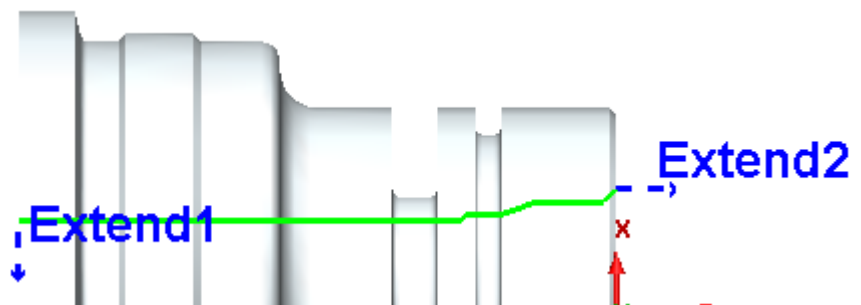
To determine which Extent Entity group box (between Extent1 and Extent2) represent the front and back of the part, pick different options in these group boxes. The change in extent direction as represented in the graphics area helps you to determine which Extent represents the Front and Rear.

5.  Click *OK* to apply the changes and close the dialog box.

Extends of the ID feature


1. Double click *ID Feature1* to change its Strategy and define the extend directions for the ID feature. The *Edit Turn Feature* dialog box is displayed.

2. Change the Strategy to *Rough-Finish*.
3. At the start of the feature, you would want to extend *Along Z*. In the *Extend2* group box, select *Along Z*.



Extending the segments for ID Feature1

4. The extend segment for the end of the feature should extent toward the centerline of the part. In the *Extend1* group box, select *Along X*. If necessary, select the *Reverse* option to point the extent direction toward the centerline.

5.  Click *OK* to apply the changes and close the dialog box.



Interactively inserting Face Feature and OD Feature in Turn Setup2

When you will machine the rear of the part while holding with the Main Spindle, you will need to turn down the diameter at the rear of the part, face the back of the part and machine the rear groove. AFR already found the groove at the rear of the part (*in Turn Setup2*), but not the face feature or OD feature.

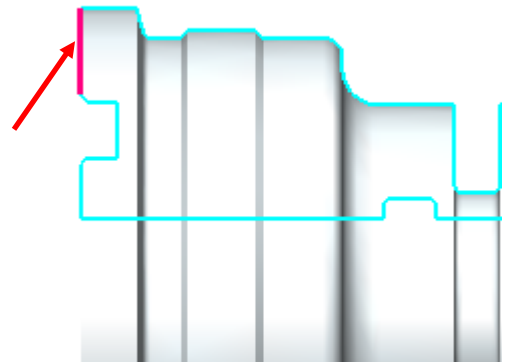
(The OD feature is the OD segment of *OD Feature1* which was deleted with the intent of machining it *Turn Setup2*.)

In this step, you will interactively insert these features.


1. Right click *Turn Setup2* and select *Turn Feature* from the context menu.

The *New Turn Feature* dialog box is displayed.

2. In the Feature group box, select *Face Feature* for Type.
3. Click in the *Selected entities* list box to set the focus.
 - Pick the vertical segment as shown in the figure.
 - The selected entity (*CW Face-30*) and two extend entities displays in the *Selected entities* list. You do not need to extend either end of the segment.

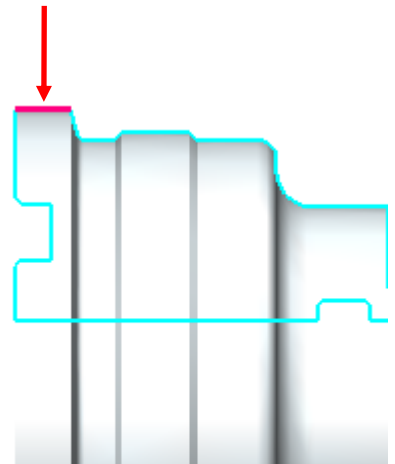


Picking the vertical segment indicating the face feature


4.  Click *OK* to insert the feature.
Face Feature2 will be listed under *Turn Setup2* in the Feature tree.
5. Once again, right click *Turn Setup2* and select *Turn Feature* from the context menu.

The *New Turn Feature* dialog box will be displayed.

6. In the Feature group box:
 - Select *OD Feature* for Type.
 - Leave the Strategy set to *Rough-Finish*.
7. Click in the *Selected entities* list box to set the focus.
 - Pick the horizontal edge as shown in the figure.
 - The selected entity (*CW Face-29*) and two extend entities displays in the *Selected entities* list. You do not need to extend either end of the segment.



Picking the segment at the rear end to define OD Feature

8.  Click *OK* to insert the feature.

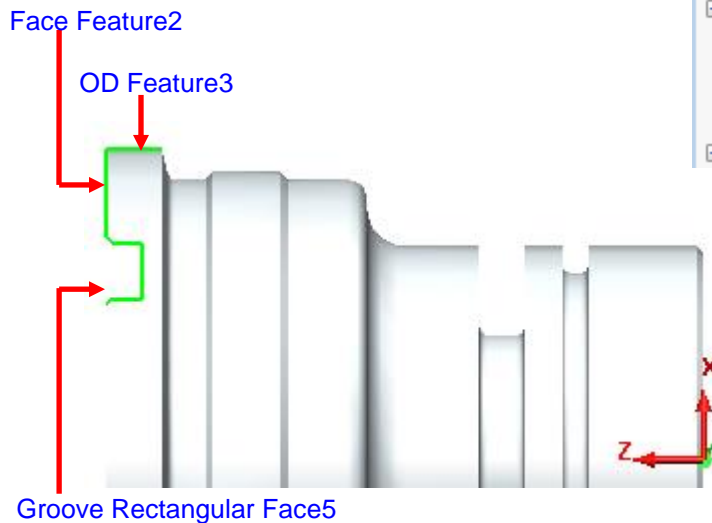
OD Feature3 will be listed under *Turn Setup2* in the Feature tree.



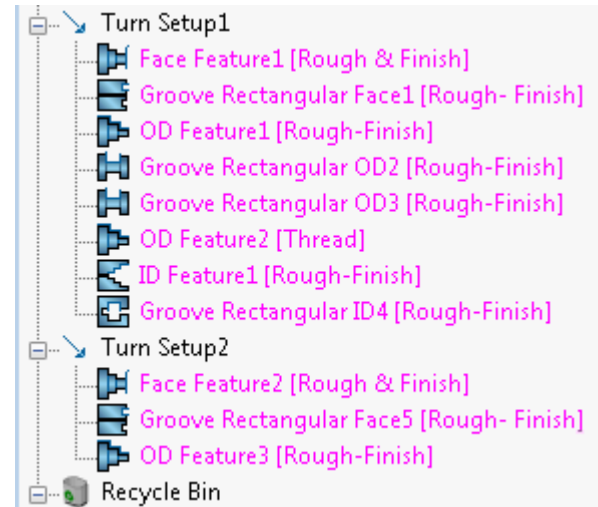
Turn Tutorial

9. Drag and drop *Face Feature2* so that it is the first feature listed under *Turn Part Setup2*.
10. Ensure that features are listed in *Feature tree* as shown in the image to the right.

The features at the rear end of the part which will be machined from *Turn Setup2* are as shown in the image on the right.



Features machined from Turn Setup2



Order of Features

Step 6: Generating Operations and Toolpaths

1.  Click the *Generate Operation Plan* button on the CAMWorks Command Manager or Workflow toolbar.

OR

Right click *CAMWorks NC Manager* in the Feature tree and select *Generate Operation Plan* from the context menu.

Operations are created for the machinable features in both the Turn Setups.

2. Double click *Turn Rough1* (for *OD Feature1*) in the Operation tree to edit the operation.
 - On the *Turn Rough* tab, ensure that the *Undercut* option in the *Profile parameters* group box is checked.




The Undercut option defines whether the toolpath can create an undercut up to the back angle of the tool. When checked, the toolpath follows the entire toolpath observing the back angle.

- Click *OK* to apply the changes and close the dialog box.
3. Double click *Turn Finish1* operation (for *OD Feature1*).




- On the *Turn Finish* tab, ensure that the *Undercut* option is checked.
 - Click *OK* to apply the changes and close the dialog box.
4. Double click *Groove Rough2* operation (for *Groove Rectangular OD2* feature).
The *Operation Parameters* dialog box is displayed.
5. On the *Groove Rough* tab of the *Operation Parameters* dialog box, make the following changes:
- Ensure that the *Stepover* is set to **0.05in.**
 - In the *Grooving styles* group box:
 - Change the *Groove peck type* to *Constant*.
With this option selected, the tool feeds in the *First peck amount*, then continues feeding in the *Sub peck amount* until the depth is reached.
 - Make sure the *Groove style* is set to *Normal*.
This option generates a toolpath where the tool plunges perpendicular to the bottom of the groove. Multiple parallel plunge cuts will be produced when the width of the groove is wider than the tool. The sequence of the cuts is based on the specified order. Using the optional *Deep groove cycle*, the tool can be retracted between each depth of cut to allow chip removal.
 - Check the *Deep groove cycle* option.
With this option selected, the tool pecks at a specific depth (much like in the *Cutoff cycle*). This cycle is used for machining grooves that are unusually deep and require using a thin insert that would break if you could not peck the groove.
 - Activating the *Groove Peck type* enables the *First peck amt.*, *Sub Peck amt.* and *Min. Peck amount* fields in the *Parameters* group box.
 - Set the *First peck amt.* to **0.05in.**
 - Set the *Sub peck amt* to **0.05in.**
6. Click the *Tool* tab.
- Select the *Tool Crib* page,
 - Select *Front Turret* for the *Turret* option.
 - Highlight the tool in **Stn.No.04** (*Groove Insert tool of 0.008in. radius and 0.062in. Width*) and click *Select*.
 - Click *Yes* to replace the holder.
 - On the *Groove Insert* page,
 - Set the *Length* to **1in.**
 - Click the *Holder* page:
 - In the *Holder Edge* group box, make sure the *Holder edge* is set to *Side*.
 - In the *Orientation* group box, make sure the orientation is set to *Up right* or *Up left*.
 - Change the *Protrusion* to **0.65in.**
 - Click *OK* to apply the changes and close the dialog box.



- If CAMWorks displays a warning stating that Tool Parameters have changed and prompt you to either change the Tool parameters or the add it as a new tool. Select *Add*.
7. Double click *Groove Finish2* operation (for *Groove Rectangular OD2* feature).
The *Operation Parameters* dialog box is displayed.
- Select the *Tool Crib* page of the *Tool* tab.
 - Select *Front Turret* for the Turret option.
 - Highlight the tool in **Stn.No.04** (Groove Insert tool of **0.008in.** *radius* and **0.062in.** *Width*) and click *Select*.
 - Click *Yes* to replace the holder.
 - Click the *Holder* tab and verify that the holder is the same as for *Groove Rough2* and has the same orientation.
 - Click *OK* to apply the changes and close the dialog box.
 - If CAMWorks displays a warning stating that Tool Parameters have changed and prompt you to either change the Tool parameters or the add it as a new tool. Select *Add*.
8. Double click *Groove Rough4* operation (for *Groove Rectangular ID4* feature). The *Operation Parameters* dialog box is displayed.
- On the *Tool* tab:
 - Select the *Boring Bar* page.
 - Ensure that *Lead angle* is set to **0deg.**
 - On the *Groove Rough* tab:
 - Change the *Radial (X)* and *Axial (Z)* allowances to **0.0in.**
 - In the *Cleanup* group box, set the *Cleanup pass* option to *Full Groove*.
 - Click *OK* to apply the changes and close the dialog box.
 - CAMWorks might display a warning stating that Tool Parameters have changed and prompt you to either change the Tool parameters or the add it as a new tool. Select *Add*.
9. Right click *Groove Finish4* (operation for *Groove Rectangular ID4* feature) in the *Operation tree* and select *Delete*. Click *Yes* to confirm the deletion.
10. Double click the *Drill1* operation (for *ID Feature1*).
- On the *Feature Options* tab:
 - Check the *Add tip length* option.
 -  Click on the *Override Machining Depth* button. Change the Machining length to **4.05in.**
Increasing this length ensures that the full diameter of the drill will pass through the back of the part.

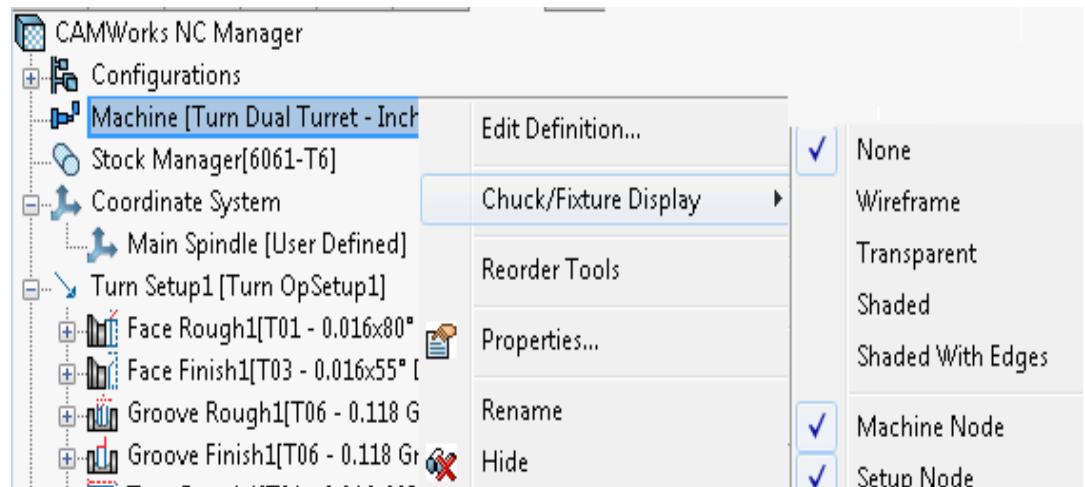


- Click *OK* to apply the changes and close the dialog box.
11. Double click the *Thread1* operation.
- On the Thread tab,
 - Set the *Start length* to **0.075in.**
 - Set the *End length* to **0.05in.**
 - Click *OK* to apply the changes and close the dialog box.
12. Under the *Turn Setup2*, double click the *Face Rough2* operation (for *Face Feature 2*).
- On the Holder Page of the Tool tab:
 - Change the Lead Angle to **0deg.**
 - Ensure that the Orientation is *Down left*.
 - Click *OK* to apply the changes and close the dialog box.
 - CAMWorks will display a warning stating that Tool Parameters have changed and prompt you to either change the Tool parameters or the add it as a new tool. Select *Add*.
13. Repeat the process mentioned in above Step no. 12. for *Face Finish2* operation.
14.  Click the *Generate Toolpath* button on the CAMWorks Command Manager or Workflow toolbar.

Step 7: Defining the Chuck Location for Turn Setup1

Disabling Chuck/Fixture Display in Graphics Area

Right-click on the *Machine [Turn Twin Turret – inch]* in the Operation tree and set the Chuck/Fixture Display to *None*.



Setting the Chuck Display option to

This action disables the chuck/Fixture display. The chuck/Fixture display has been disabled at this stage because the chuck configurations and locations have not yet been properly defined. Once defined, the chuck/Fixture display will be enabled again.




Defining the Geometry of the chuck

1. Double click the *Machine [Turn Dual Turret – inch]* in the Operation tree.
The *Machine* dialog box is displayed.
2. Click on the *Chuck/Fixture* tab.
3. *Turn Setup1* is machined from the Main Spindle. Hence, in the *Main Spindle Information* group box, click on the *Edit* button.
4. The *Chuck Parameter : [Main Spindle]* dialog box is displayed. This dialog box is used to defined the geometry of the chuck.

- In the Chuck Management group box, select *8in_2Step_Chuck* from the list of *Available Chucks*.

This is one of several chuck configurations that has been saved in the TechDB. You can define additional configurations by setting the parameters on this tab, then typing a name in the input box and clicking the Save button.

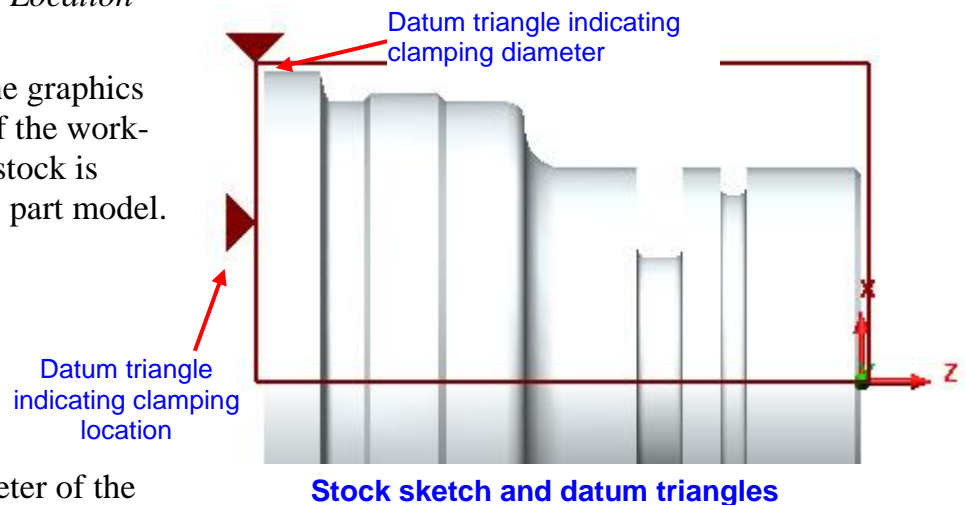
- In the Jaw Parameters group box:
 - Uncheck the *Equal step width* option.
 - Set the Step 2 Width to **0.5in**.
 - Set the Step 1 Length to **0.6in**.

5. Click  *OK* to apply the changes and close the dialog box.
6. Click *OK* to close the *Machine* dialog box.
7. Double click on *Turn Setup1* in the Operation tree.

The *Operation Setup Parameters* dialog box is displayed.

8. On the *Chuck/Fixture Location* tab,

- Observe that, in the graphics area, the outline of the work-in-process (WIP) stock is displayed over the part model. Datum triangles also display over the part model. These datum triangles indicate the location and diameter of the chuck on the WIP stock sketch.



- In the graphics area, observe that the datum triangle representing the Clamping diameter and Clamping location are located correctly on the stock sketch. The triangle



indicating the clamping location is located on the vertical edge representing the back (locating) face.

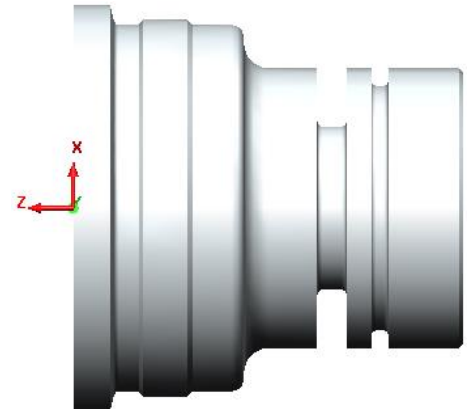
9. Click *OK* to close the dialog box.

Step 8: Defining the Chuck Location for Turn Setup2

1. Double click on *Turn Setup2* in the Operation tree.


The *Operation Setup Parameters* dialog box is displayed.

2. On the Origin tab, change the *Origin Defined from* option to *Automatic*.
3. Ensure that the *Other end* option is checked.
4. The origin marker displays as shown in the image on the right.



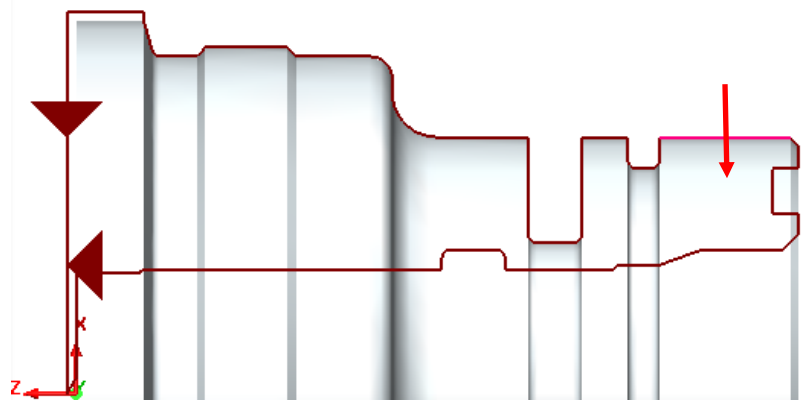
Change in location of Origin marker

5. Click on the *Chuck Location* tab,
 - In the graphics area, observe that, the outline of the work-in-process (WIP) stock is displayed over the part model. The location of the datum triangles indicating the clamping diameter and location are incorrect. This is because the *Clamping diameter* and *Clamping location* for the turn setup have not been defined in the Chuck Location tab. These parameters need to be defined.
 - To set the *Clamping diameter*, following are the steps:

- In the *Clamping Diameter* group box of the *Chuck location* tab, click in the *Select Entity for Clamping Diameter* field next to the  symbol in order to set the focus.


In the graphics area, pick the right-most horizontal edge of the WIP stock sketch (indicated by the arrow in the image on the right). Observe that the location of the triangle representing the *Clamping diameter* changes.

- Observe that *WIP* is listed as the entity *Select Entity for Clamping Diameter* field.



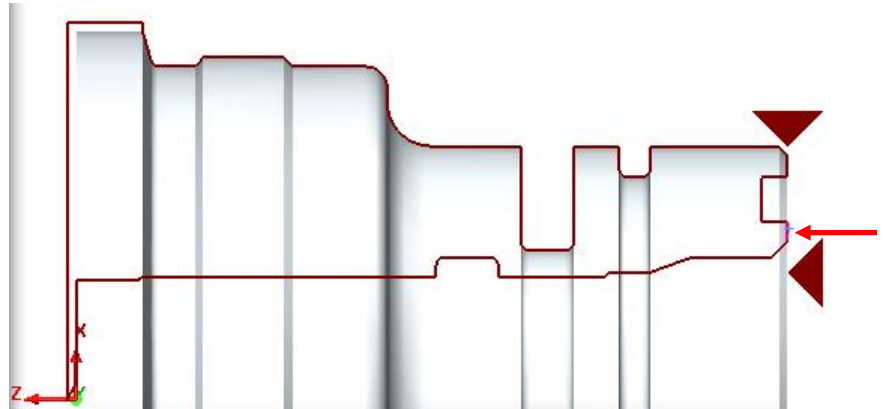
Selecting Entity for Clamping diameter



- To set the *Clamping location*, following are the steps:
 - In the *Clamping Location* group box of the *Chuck location* tab, click in the *Select Entity for Z Clamping Location* field next to the  symbol in order to set the focus.

- In the graphics area, pick the WIP stock sketch at the back vertical edge of the small diameter for the chuck face as shown in the image on the right.

Note that you may need to zoom in to this area in order to pick easily.



Corrected location of the datum triangles

- Observe that the location of the triangle representing the *Clamping Location* changes as well as the triangle representing *Clamping Diameter* changes.
 - Observe that *WIP* is listed as the entity *Select Entity for Z Clamping Location* field.
6. In the Chuck properties group box, set *Chuck defined from to Setup*.



The screenshot shows the 'Operation Setup Parameters' dialog box with the 'Chuck/Fixture Location' tab selected. The 'Define chuck/fixture location' checkbox is checked. The 'Clamping diameter (X)' section has 'Clamp on' set to 'OD', 'with jaw' set to 'Back of Jaw', and 'Diameter' set to '2.5in'. The 'Clamping location (Z)' section has 'Use' set to 'Back', 'of jaw' set to 'Back of Jaw', 'Z offset' set to '0in', 'Chuck/Fixture face' set to '-4in', and 'Jaw face' set to '-3in'. The 'Chuck/Fixture properties' section has 'Chuck/Fixture defined from' set to 'Machine' and an 'Edit....' button. The 'Triangle display' section has 'Triangle size' set to '0.197in'. The 'Sub spindle' section has 'Spindle Z pickup' set to '-4in', 'Pickup offset' set to '0in', and 'Transfer dist' set to '0in'. At the bottom are 'OK', 'Cancel', and 'Help' buttons.


Operation Setup Parameters dialog box

7. As observed, the geometry of the chuck is defined from the Chuck Parameters dialog box in the Machine node. By default, this globally defined chuck location is used for all the turn setups. This is indicated in the Chuck Properties group box on the Chuck Location tab.
 - If the *Chuck properties* group box indicates Chuck defined from Machine, then the chuck geometry defined from the Machine node is currently being used for the Turn Setup.
 - If the group box indicates Chuck defined from Setup, then the chuck geometry defined at the Setup level. This option is preferable when machining the part using multiple turn setups where the chuck may change from setup to setup.
8. Click on the *Edit* button in the *Chuck properties* group box.


The *Chuck Parameters : [Main Spindle]* dialog box is displayed.



Turn Tutorial

- In the Jaw Parameters group box, select the *Jaws out* option.
 - Click  to apply changes and close the dialog box.
9. Click *OK* to close the *Operation Setup Parameters* dialog box.
 10. Click *Yes* when CAMWorks displays a message stating that toolpaths need to be recalculated.

OR






 Click the *Generate Toolpath* button on the CAMWorks Command Manager/Workflow toolbar to regenerate toolpaths for both Setups.

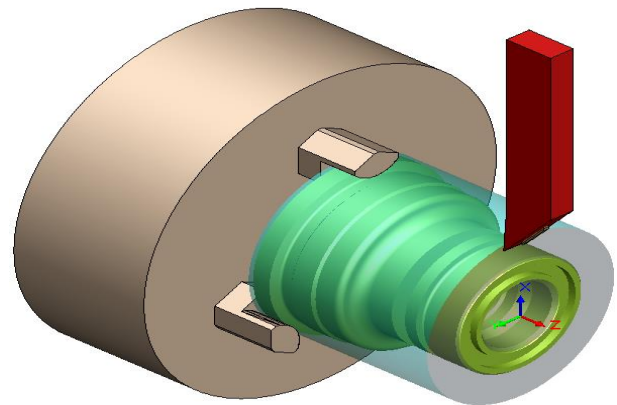


If you change the Chuck Configuration in the Chuck tab of the *Machine* dialog box, you will need to pick the Chuck Location again on the Chuck Location tab of the *Operation Setup Parameters* dialog box.

11. Right click on the *Machine [Turn Twin Turret – inch]* in the Operation tree and set the *Chuck Display* option to *Shaded with Edges*.
12. Once again right click on the *Machine [Turn Twin Turret – inch]* in the Operation tree and set the *Chuck Display* option to *Machine node and Setup node*.
This action ensures that the Chuck is displayed in the graphics area whenever a machine node or Setup node is highlighted in the Operation tree.
13. Left click *Turn Setup1* in the Operation tree and observe the *Chuck* in the graphics area.
14. Left click *Turn Setup2* in the Operation tree and observe the *Chuck* in the graphics area.








Step 9: Simulating the Toolpaths for Multiple Turn Setups

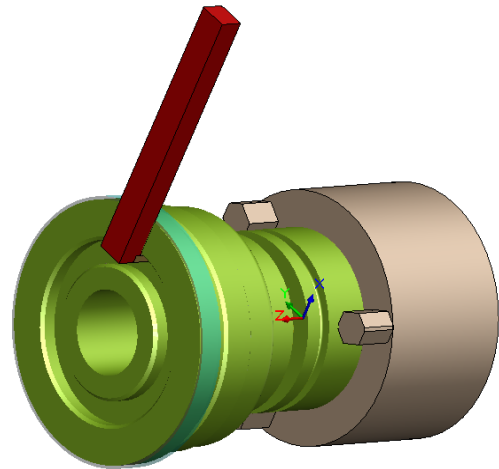
1.  Change the View Orientation to *Isometric* view .
2.  Click the *Simulate Toolpath* button on the CAMWorks Command Manager/Workflow toolbar.
The *Toolpath Simulation* toolbar is displayed.
3. Click the down arrow next to *End* and select *Next Setup* from the dropdown list.
4. Select the following display options:
 -  Fixture: *Shaded with Edges*
(The chuck defined for *Turn Setup1 [TurnOpSetup1]* will be displayed)
 -  Stock: *Translucent Display*



Operations machined from Turn Setup1



-  Tool: *Shaded Display*
 -  Holder: *Shaded Display*
 -  Target: *Shaded Display*
5.  Click the *Options* button on the Toolpath Simulation toolbar.
 6. In the displayed *Options* dialog box, check the *Simplified Threading* option.
 7. When this option is checked, CAMWorks simulates the thread as if the cut is a normal turning cut and thread simulation should be much faster. When this option is not checked, CAMWorks simulates the full thread, which can result in very slow simulation speeds. Click *OK* to apply the changes and close the dialog box.
 8. Click *Yes* when CAMWorks displays a warning stating that Simulation must be restarted for the new settings to take effect.
 9.  Click *Run* button on the *Toolpath Simulation* toolbar.
The simulation runs through the operations in *Turn Setup1* which is machined from the Main Spindle.
 10. After simulating all the operation in this Setup, it will then display the chuck for *Turn Setup2* which is also machined from the Main spindle.
The simulation stops because the end condition was specified as *Next Setup*.
 11. Rotate the part so you can see the face with the groove.
 12.  Click *Run* button on the *Toolpath Simulation* toolbar. The operations in *Turn Setup2* are simulated.
 13. After simulation, click the *OK*  button on the *Toolpath Simulation* toolbar to exit Simulation mode.



Operations machined from Turn Setup2



4 Axis Turn 2

Twin Spindle Setup


Certain Turn machines have a twin Spindle Setup comprising the **Main Spindle** and **Sub Spindle**. A twin Spindle Setup allows Turn features to be machined when the part is chucked in the Main Spindle or Sub Spindle.

Topics covered in this tutorial

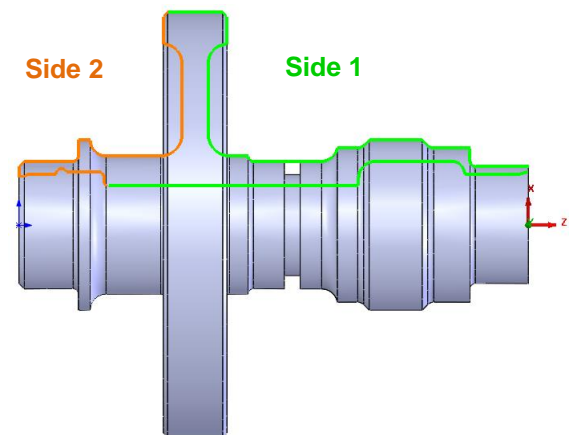
This tutorial explores the machining of a Turn part on a Turn machine with Twin Spindle Setup. The following aspects will be covered in detail:

- [Activating the Sub Spindle Machining functionality by enabling the ‘Use Sub Spindle’ option](#)
- [Editing the OD and ID Feature definitions, including their Spindle Attribute](#)
- [Automatic generation of operations for Turn features when Turn stock is chucked on Sub Spindle](#)
- [Setting the chuck location for Turn Setup when stock is chucked in the Sub Spindle](#)

Step 1: Opening the Part and Defining Machining Parameters

1. Open part file **TURN4AX_2.SLDPRT** in the following folder:
Drive:\CAMWorksData\CAMWorks202X\Examples\Tutorial_Parts\Turn
2. The stock for this part has already been defined as a Bar Stock with diameter of **5.25in** and stock length of **6.1in**. The Back Face of the stock has been offset by **0.05in**.
3. In this tutorial, the OD and ID features on *Side 1* and *Side 2* will be machined as follows:
 - The features on the *Side 1* will be machined in the Main Spindle Setup (*Side 2* will be chucked in the Main Spindle).
 - The features on *Side 2* will be machined in the Sub Spindle Setup (*Side 1* will be chucked in Sub Spindle).
4.  On the CAMWorks *Feature Tree* tab, double click the *Machine [Turn Dual Turret – Inch]* item in the Feature tree to open the Machine dialog box.
 - On the Machine tab, the example machine *Turn Dual Turret – Inch* has been selected for machining this turn part. This example machine has been defined as a 4 axis machine in the TechDB.
 - On the *Tool Crib* tab, ensure that *Tool Crib 2 Rear (Inch)* is selected as the Active Tool Crib for the Rear Turret.

Before using CAMWorks to machine your parts, make sure you define the machine tools available in your facility.



TURN4AX_2.SLDPRT



- Also ensure that the *Tool crib priority* option is checked.
- On the *Post Processor* tab, the example post processor *T4AXIS-TUTORIAL* has been selected.
- On the *Chuck/Fixture* tab, the distance between the two chucks has been set to **10in**.

Activating the Use sub-spindle option

5. On the *Machine* tab of the *Machine* dialog box, in the *Active machine* group box, ensure that the *Use sub-spindle* option is checked.

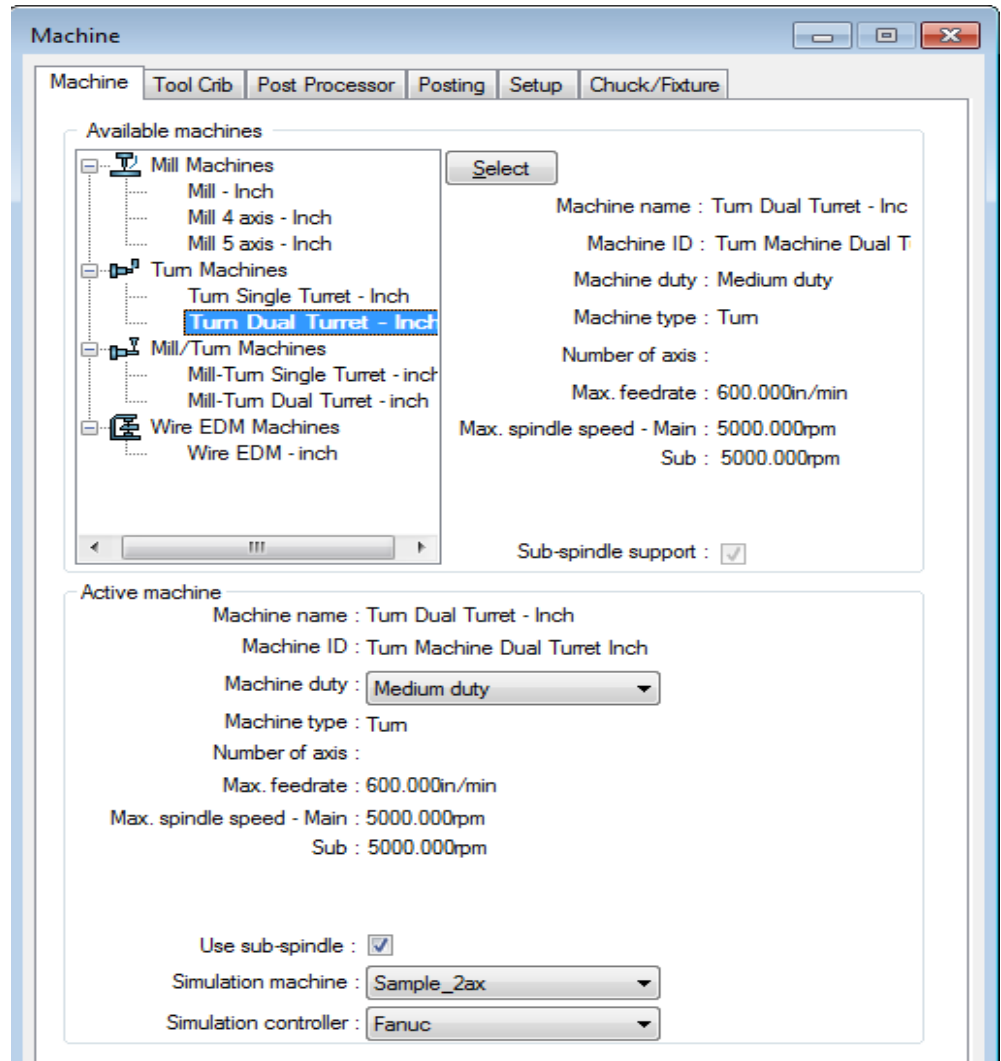
This action allows the current part model to use Sub Spindle functionalities for machining operations. The *Use sub-spindle* option enables the following Sub Spindle functionalities:

- Automatic generation of operations for Turn features when Turn stock is chucked on Sub Spindle. (Achieved by defining the spindle attribute for the feature as *Sub Spindle*.)
- Geometric definition of the Sub spindle chuck and jaws
- Insertion of Sub Spindle Setups in CAMWorks Operation tree
- Interactive insertion of operations under Sub Spindle Setups
- Manual definition of the Z location of the Sub Spindle



If the **Use sub-spindle** option is unchecked and you proceed to generate features, operations and toolpaths for a Turn or Mill-Turn part model, then the above-mentioned Sub spindle functionalities will not be available in CAMWorks.


6. Click *OK* to apply the changes and exit the *Machine* dialog box.



Checking the Use Sub-spindle option in Machine Tab



Step 2: Defining Features Automatically

Define the machinable features for this part automatically by executing the  *Extract Machinable Features* command.

On executing this command, Turn Setup is created and AFR recognizes the features that can be machined in the Setup.

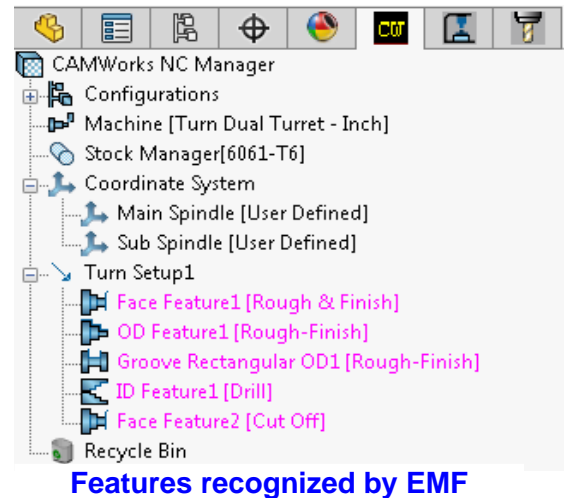
AFR has found an OD Feature and an ID Feature in *Turn Setup 1*.

Converting the CutOff feature to Face Feature

Since you will be chucking the part, there will be no need for the Cutoff feature. Instead a Face feature will be required to machine the stock offset of 0.05in.

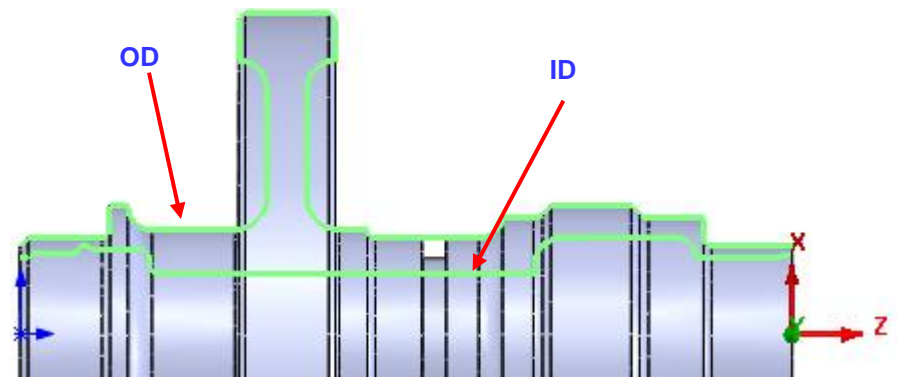
1. Right click *CutOff Feature1* in the *Turn Setup1* and select the *Convert to Face Feature* on the context menu.

This action converts the Cut off feature into a Face feature (*Face Feature2*).

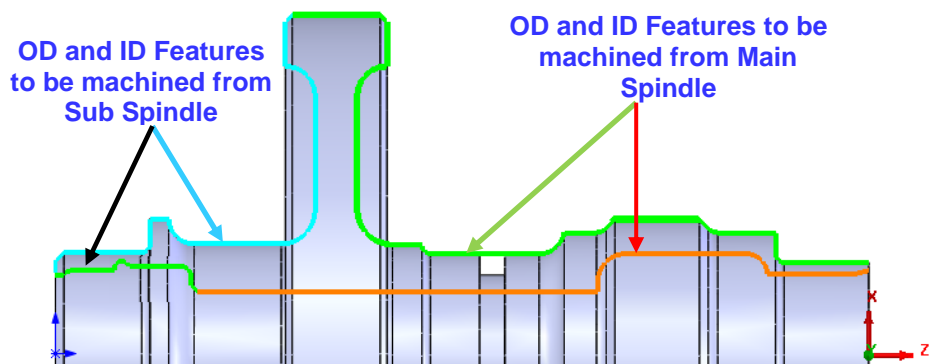


Step 3: Changing Feature Definitions for OD and ID Profiles

Observe that both the OD and ID Feature profiles extend from the front face to the back face. Since you will be double-chucking, machining the entire length of these profiles is not possible. Therefore, you will edit the OD and ID profile feature definitions in *Turn Setup1* and create new OD and ID features such that both the OD and ID profiles are machined from *Main Spindle* as well as the *Sub spindle*. The double-chucking environment makes this possible.



OD and ID Feature recognized by Turn Setup1



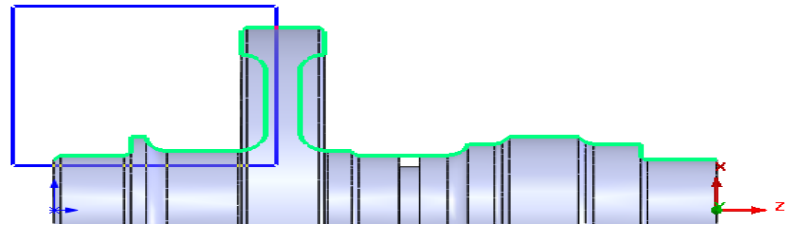
OD and ID Features to be machined from Main & Sub Spindle




Editing OD Feature in Turn Setup1 in order to machine it from Main Spindle

The OD Feature extends from the front face to the back face of the part. In this step, you will edit the OD feature to delete those segments which will not be machined when the stock is chucked in the Main Spindle. (The deleted segments will be machined from the Sub Spindle).

1. Double click *OD Feature1* in the tree. The *Edit Turn Feature* dialog box is displayed.
2. In the *Feature* group box, observe that the Spindle attribute is set to *Main spindle*.
3. The *Selected Entities* list box lists all the entities that comprise the OD feature. Ensure that the *Window selection* option in this group box is checked.
4. Using the mouse, window pick the segments of the part as shown in the image.



Deleting the segments of the OD feature which will not be machined from Main spindle

- This action removes the entities from the OD feature which were captured by the window. The *Selected entities* list will now display an updated list of entities comprising of 20 segments and two extends.
5.  Click *OK* to close the dialog box.


Interactively Inserting and OD Feature in Turn Setup1:

In this step, the OD feature entities deleted in the previous step will be interactively inserted back. The OD feature thus inserted will be machined from the Sub Spindle.

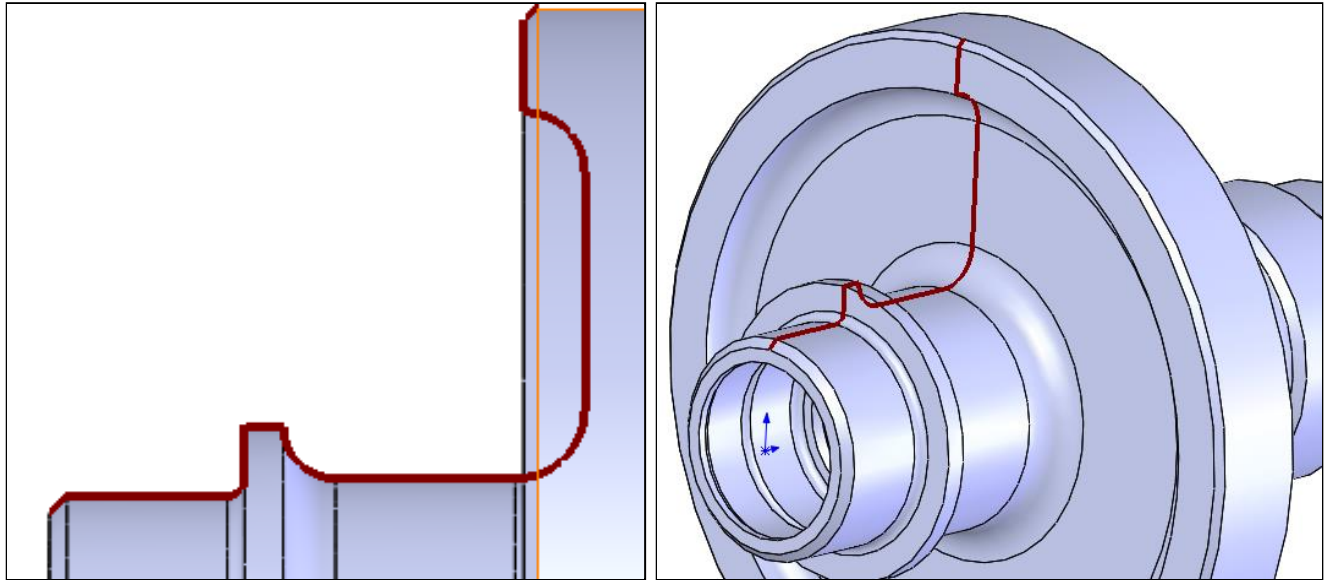
1. Right click *Turn Setup1* in the CAMWorks Feature tree and select *Turn Feature* on the context menu.
The *New Turn Feature* dialog box is displayed.
2. In this dialog box, within the *Feature* group box:
 - Ensure that *OD Feature* is set for the Feature Type.
 - Ensure that the strategy is set to *Rough-Finish*.
 - Set the Spindle attribute to *Sub spindle* from the dropdown list. (This action ensures that the operations generated for this feature are on the Sub Spindle.)
3. In the *Selected Entities* group box:
 - Make sure the *Window selection* option is checked.
 - Window pick the same segments (faces) that you removed from *OD Feature1* as shown in the above image.
 - The entities that you window picked will be listed in the *Selected Entities* group box.
 - There is a high chance that some ID profile entities may get window picked accidentally. Ensure that you delete such segments from the *Selected Entities* list box.



When you highlight a listed entity in the *Selected Entities* list box, that entity is highlighted in the graphics area. Press the Delete button in order to delete a highlighted entity.

4.  Click *OK* to insert the feature.

OD Feature2 will now be listed under *Turn Setup1* of the CAMWorks Feature tree.



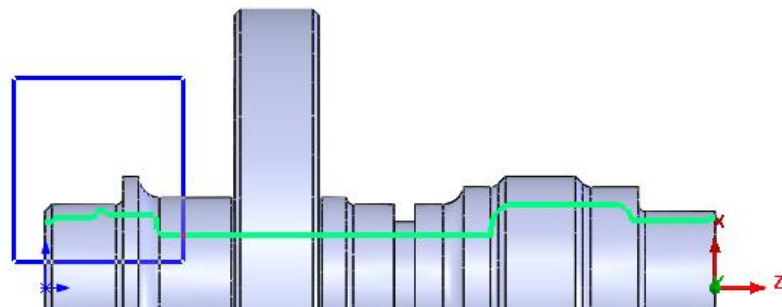
Front view and Side View of the OD Profile which will be machined from Sub Spindle

Editing ID Feature in Turn Setup1 in order to machine it from Main Spindle

The ID feature too extends from the front face to the back face of the part. In this step, the automatically recognized ID feature will be edited such that only those entities which can be machined when the stock is chucked in the Main Spindle are retained. (The deleted segments will be machined from the Sub Spindle).

1. Double click *ID Feature1* in the tree. The *Edit Turn Feature* dialog box is displayed.
2. In the *Feature* group box, observe that the Spindle attribute is set to Main Spindle.
3. Change the Strategy to *Rough-Finish*.
4. In this dialog box, within the *Selected Entities* group box, make sure the *Window selection* option is checked.
5. Window pick the segments of the part as shown in the image or right.

These selected entities will be deleted from the ID feature. The *Selected entities* list will be updated and will now contain only seven segments and two extend entities.



Deleting the segments of the ID feature which will not be machined from Main Spindle



6. Click *OK* to close the dialog box.

Interactively Inserting an ID Feature that will be machined from Sub Spindle

In this step, the ID feature entities deleted in the previous step will be interactively inserted back. The ID feature thus inserted will be machined from the Sub Spindle.

1. Right click *Turn Setup1* and select *Turn Feature* on the context menu.

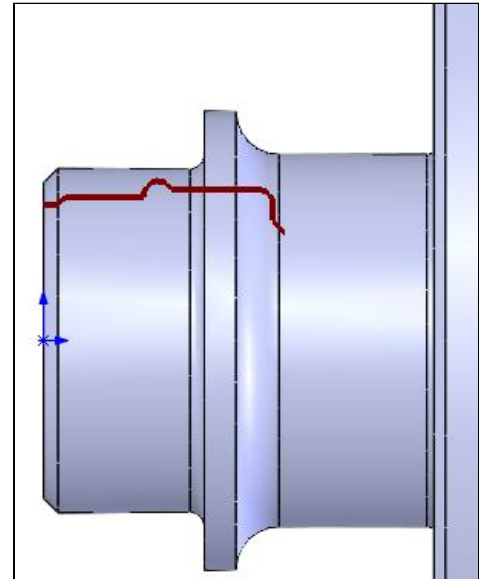
The *New Turn Feature* dialog box is displayed.

2. In this dialog box, select *ID Feature* for the Feature Type.
3. Change the Strategy to *Rough-Finish*.
4. Set the Spindle attribute as *Sub spindle* from the dropdown list. (This action ensures that the operations generated for this feature are on the Sub Spindle.)
5. In the *Selected Entities* list box, make sure the *Window selection* option is checked.
6. Window pick the same segments (faces) that you removed from *ID Feature1* as shown in the above image. The *Selected Entities* list box will display the selected entities.
7. Some OD profile entities may get window picked accidentally. Ensure that you delete such segments from the *Selected Entities* list box.

The *Selected Entities* list box must contain two extend entities and 9 segments (*CW Face-33* to *CW Face-41*) in reverse order.

8. Click *OK* to insert the feature.

ID Feature2 will be added to *Turn Setup1*.



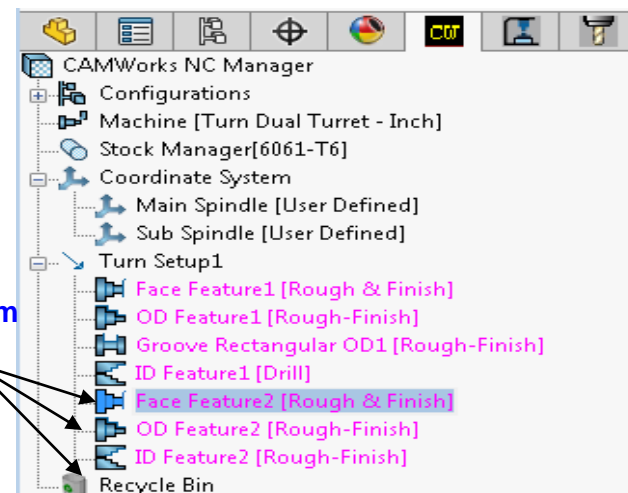
Front view of the ID Profile which will be machined from Sub Spindle

Editing the Spindle Attribute of the Face Feature

1. Double-click *Face Feature2* in the CAMWorks Feature tree. The *Edit Turn Feature* dialog box is displayed.
2. In the Feature group box, change the Spindle attribute to Sub Spindle.
3. Click *OK* to apply the changes and close the dialog box.

Face Feature2 will now be machined from the Sub Spindle.


Machined from Sub Spindle



Final list of Features



Step 4: Generating the Operation Plan

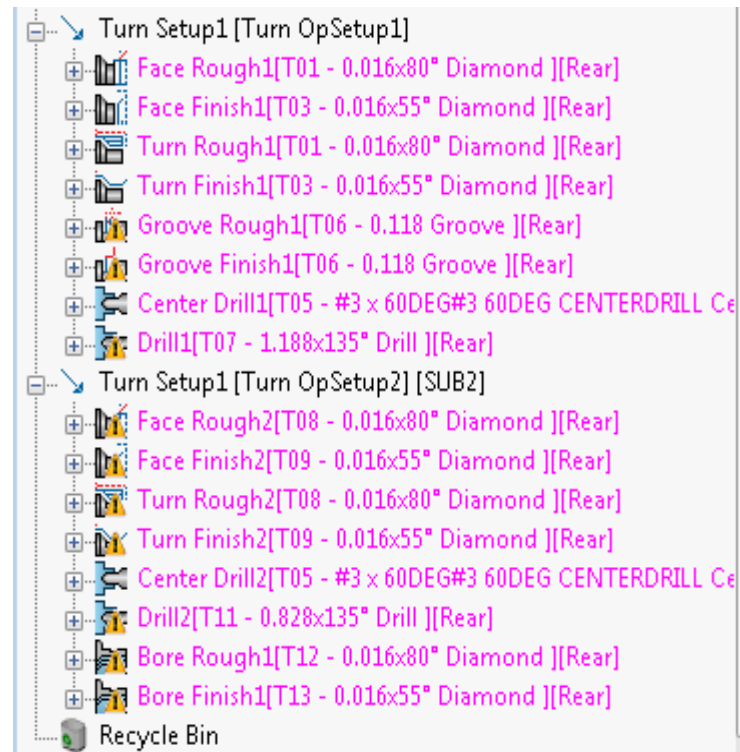
 Click the *Generate Operation Plan* button on the CAMWorks Workflow toolbar or Command Manager.

Operations are created for the machinable features in the Turn Setups and are listed in the Operation tree.

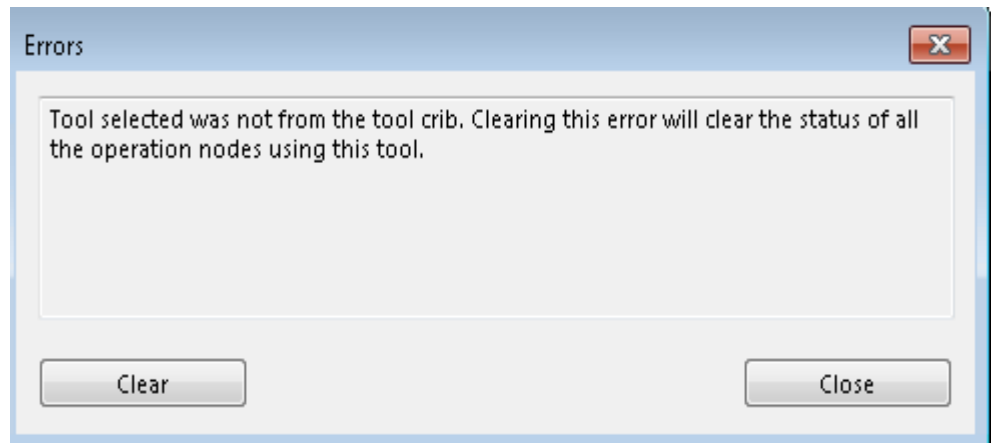
- Operations listed in *Turn Setup1 [Turn OpSetup1]* will be machined when the stock is chucked in the Main Spindle.
- Operations listed in *Turn Setup1 [Turn OpSetup2] [SUB2]* will be machined when the stock is chucked in the Sub Spindle.

Exclamation mark symbol over certain operations

- Some of the operations may have a red exclamation mark symbol over them. The exclamation mark indicates that no suitable tool was available in the active tool crib to machine that particular feature and hence a suitable tool was added from the tool library and assigned to the operation.
- To get rid of the exclamation mark:
 - a. Right-click on the operation with an exclamation mark and select *What's wrong?* from the context menu.
 - b. A warning message indicating that tool selected was not from the active tool crib will be displayed.
 - c. Click on the *Clear* button in this dialog box to get rid of the exclamation mark.
 - d. Repeat above steps for all operations with the yellow exclamation mark.



List of Generated Operations

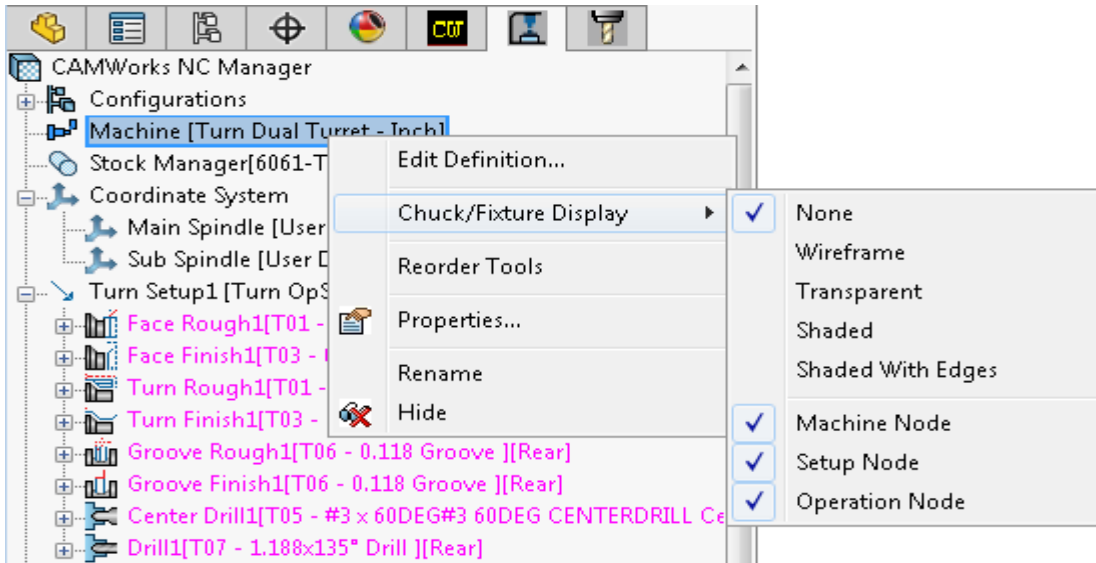


Message indicating What's wrong with the selected Operation



Step 5: Enabling the Chuck Display option

1. In the CAMWorks Operation tree, right-click on the *Machine [Turn Twin Turret - inch]* and observe that the *Chuck Display* option is set to *None*.
2. Enable the Chuck display by selecting on the *Chuck Display* options such as *Transparent*, *Shaded* or *Shaded With Edges*.



Chuck Display Options on Machine node of CAMWorks Operation tree



Step 6: Defining the Chuck Configuration for a Twin Spindle Setup

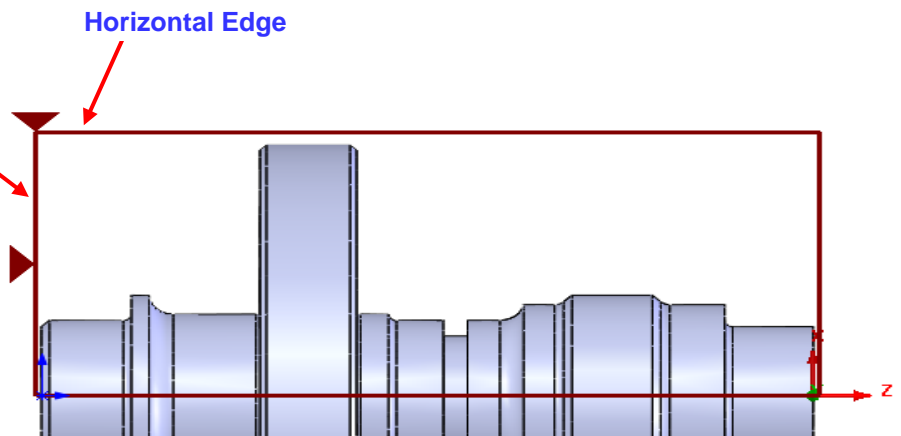
In this step, you will define the chuck location for each Turn Setup.

Setting the Chuck Location for Turn Setup machined from Main spindle

1. Double click the *Turn Setup1 [Turn OpSetup1]* in the Operation tree. The *Operation Setup Parameters: [Main Spindle]* dialog box is displayed.
2. Click on the *Chuck/Fixture Location* tab.

In the graphics area, observe that the outline of the WIP stock is displayed over the part model. Datum triangles also display over the part model. These datum triangles indicate the location and diameter of the chuck on the WIP stock sketch.

3. In the *Clamping Diameter* group box, within the  **Vertical Edge**  **Horizontal Edge**, the stock **WIP** is the entity selected by default. (The horizontal edge representing the OD of the stock WIP sets the clamping diameter of the

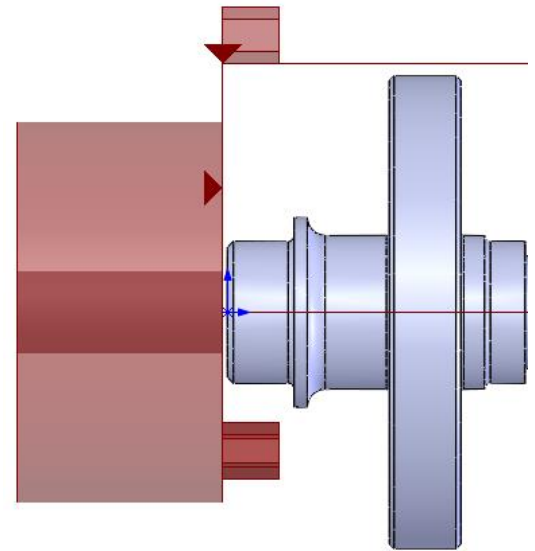


Selection the Clamp Location and Diameter for Turn Setup1



chuck.)

4. In the *Clamping Location* group box, within the *Select Entity for Z Clamping Location* field, the stock WIP is the default selected entity. (The vertical edge representing the OD of the stock WIP sets the chuck face.
5. The datum triangle representing the chuck clamping location and the chuck (after *Chuck Display* is enabled) will be as shown in the image below.
6. Click *OK* to close the *Operation Setup Parameters* dialog box.



Chuck location after selection
Clamping Location and Diameter
and setting the Z offset value for
Turn Setup1

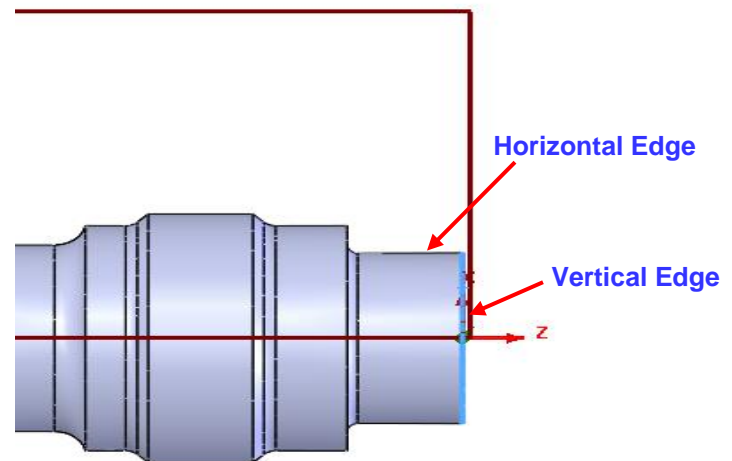
Setting the Chuck/Fixture Location for Turn Setup machined from the Sub Spindle

The stock WIP will be chucked in the Sub Spindle after machining from the Main Spindle Setup is over. [Side 1](#) was machined from the Main Spindle Setup. Hence, when the stock WIP is chucked in the Sub Spindle, it will be chucked on the OD profile of Side 1 which has already been machined.

1. Double click the *Turn Setup1 [Turn OpSetup2][Sub2]* in the Operation tree. The *Operation Setup Parameters: [Sub Spindle]* dialog box is displayed.

2. Click the *Chuck/Fixture Location* tab.
3. Within the *Clamping Diameter* group box, click on the *Select Entity for Clamping Diameter* field to set the focus.

4. In the graphics area, pick the horizontal edge or face of the OD profile of the Turn part as shown in the image on the right. This action sets the clamping diameter of the chuck. Observe that the field now displays *CW Face-76*.



Selection the Clamp Location and
Diameter for Turn Setup2

5. Within the *Clamping Location* group box, click on the *Select Entity for Z Clamping Location* field next to the symbol to set the focus.
6. In the graphics area, pick the vertical edge of the part as shown in the image above (highlighted in blue). This action sets the chuck face. Observe that the field displays *CW Edge-0*.

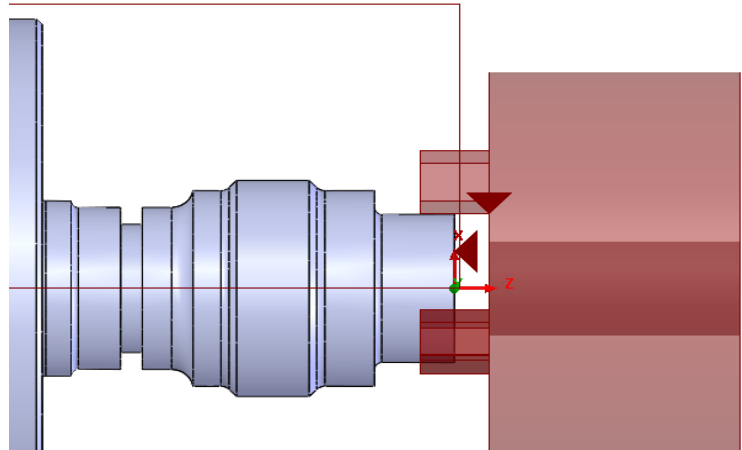
Note that you may need to zoom in to this area in order to pick these segments easily.



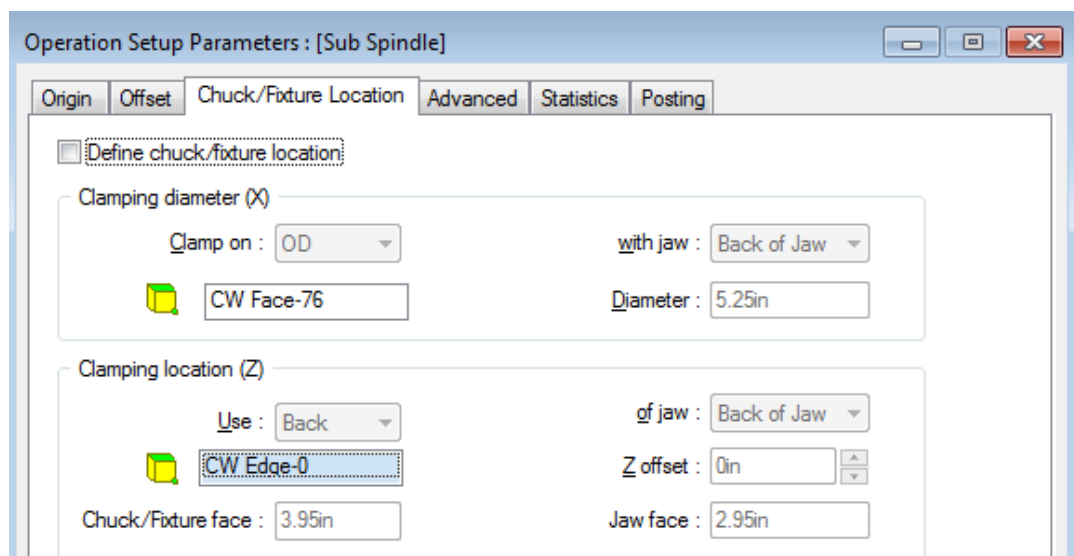
7. The datum triangle representing the chuck clamping location and the chuck/Fixture display will now relocate as shown in the image on the right.
8. Click OK to close the Operation Setup Parameters dialog box.



You can change the geometry of the Sub spindle Chuck in the same manner as explained in [Step 7 of the previous tutorial](#).



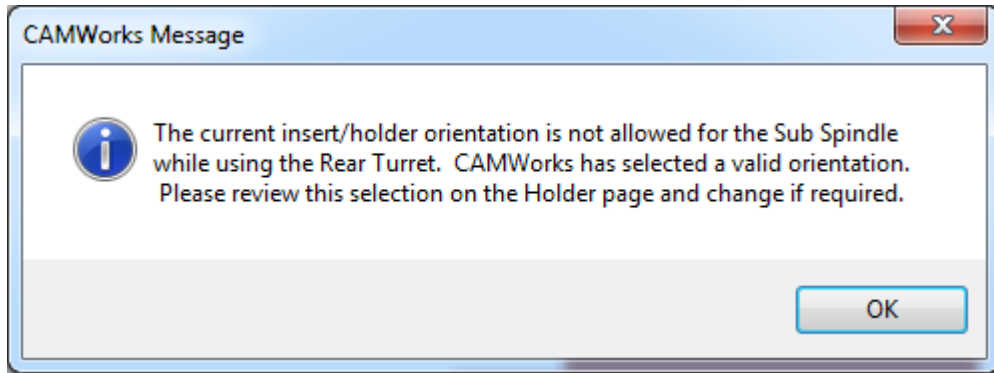
Chuck/Fixture location after selecting the Clamp Location and Diameter and setting Z offset value for Turn Setup2





Operation Setup Parameters: [Sub Spindle] dialog box

Step 7: Adjusting the Parameter and Generating the Toolpaths




1. Double click on the *Bore Finish2* of the Turn Setup1 [Turn OpSetup2] of Sub spindle.
2. The warning message displayed which prompt that the current holder orientation is not allowed and CAMWorks have selected a valid orientation. Click *OK* to review the orientation.









Warning Message

3. The Operation Parameters dialog box is displayed.
4. Click the *Preview* button and observe the orientation.
5.  Click the button to restore the dialog box and click *OK* to close.
6. Observe the CAMWorks Operation tree that the toolpath for *Bore Finish* is created automatically since you have approved for the holder orientation.
7.  Click the *Generate Toolpath* button on the CAMWorks Command Manager.
CAMWorks calculates the toolpaths for each operation in the Turn Setup.

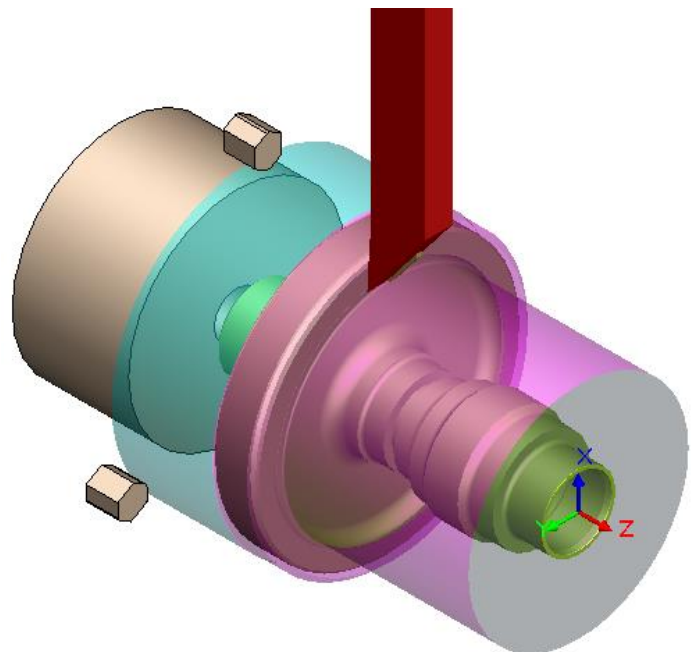
Step 8: Simulating the Toolpaths for Turn Setups

1.  Change the View Orientation to *Isometric* view .
2.  Click the *Simulate Toolpath* button on the CAMWorks Command Manager or Workflow toolbar.

The *Toolpath Simulation* toolbar is displayed.

3. Select the following display options:
 -  Fixture: *Shaded with Edges*
 -  Stock: *Translucent Display*
 -  Tool: *Shaded Display*
 -  Holder: *Shaded Display*
 -  Target: *Shaded Display*
4.  Click *Run* button on the *Toolpath Simulation* toolbar.



The simulation runs through the operations in *Turn Setup1* [*Turn OpSetup1*] which are machined with the

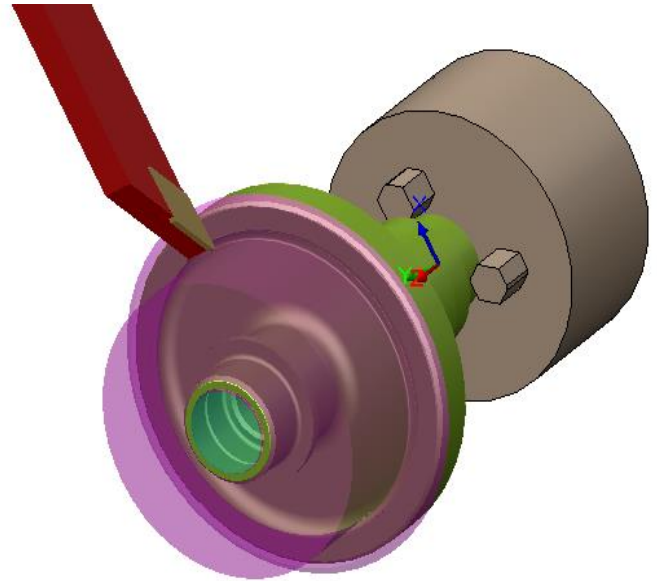


Operations machined from Main Spindle



stock chucked in Main Spindle.

5. After simulating all the operation in this Setup, it will then simulate operations which are machined with the WIP stock chucked in the Sub Spindle.
6. You can rotate the part during simulation in order to get a better view of the section being simulated.
7.  Click *Run* button on the *Toolpath Simulation* toolbar. The operations in *Turn Setup1 [Turn OpSetup8] [SUB2]* are simulated.
8. After simulation, click the *OK*  button on the *Toolpath Simulation* toolbar to exit Simulation mode.



Operations machined from Sub Spindle