Lathe Machining

Overview
- Conventions

What's New?

Getting Started
- Open the Part to Machine
- Create a Rough Turning Operation
- Replay the Toolpath
- Create a Groove Turning Operation
- Create Profile Finish Turning Operation
- Generate NC Code

User Tasks
- Turning Operations
  - Longitudinal Rough Turning
  - Face Rough Turning
  - Parallel Contour Rough Turning
  - Recess Turning
  - Groove Turning
  - Profile Finish Turning
  - Groove Finish Turning
  - Ramp Rough Turning
  - Ramp Recess Turning
  - Thread Turning
  - Sequential Turning
  - Turning on a Milling Center with Facing Head
  - Manage Local Information on Finishing Operations
  - Update Input Stock - Manual Mode
  - Update Input Stock - Automatic Mode
- Axial Machining Operations
- Auxiliary Operations
- Part Operations, Manufacturing Programs and Machining Processes
  - Machining Process for Lathe Machining
- Machining Resources and Other Entities
  - Edit the Insert Holder of a Turning Operation
  - Edit the Tool Assembly of a Turning Operation
  - Edit the Insert of a Turning Operation
  - Edit an Insert Holder in the Resource List
  - Edit a Tool Assembly in the Resource List
  - Edit an Insert in the Resource List
- Verification, Simulation and NC Output

Workbench Description
- Menu Bar
- Toolbars

Specification Tree

Customizing

General
Resources
Operation
Output
Program
Photo/Video

Reference Information

Rough Turning Operations
Recess Turning Operations
Groove Turning Operations
Profile Finish Turning Operations
Groove Finish Turning Operations
Ramp Rough Turning Operations
Ramp Recess Turning Operations
Thread Turning Operations
Sequential Turning Operations
Tool Assembly Conventions for Turning

Methodology

Cutter Compensation and Finish Operations
How to Change the Output Point
How to Update Input Stock
Turning on a Milling Center with Facing Head

Glossary

Index

Overview

Welcome to the *Lathe Machining User's Guide*. This guide is intended for users who need to become quickly familiar with the Lathe Machining Version 5 product.

This overview provides the following information:

- Lathe Machining in a Nutshell
- Before Reading this Guide
- Getting the Most Out of this Guide
- Accessing Sample Documents
- Conventions Used in this Guide.

Lathe Machining in a Nutshell

Lathe Machining easily defines NC programs dedicated to machining 3D cylindrical parts using 2-axis turning and drilling operations, for both horizontal and vertical spindle lathe machines.

Quick tool path definition is ensured thanks to an intuitive user interface based on graphic dialog boxes. Tools can be easily created and integrated to tool catalogs. Tool path can be generated, simulated and analyzed.
Whole manufacturing process is covered from tool path definition to NC data generation thanks to an integrated postprocessor execution engine. Shop floor documentation is automatically created in HTML format.
Finally, associativity with Version 5 design products allows productive design change management.

Suitable for all kinds of cylindrical machined parts, Lathe Machining fits the needs of Fabrication & Assembly industry, as well as all industries where lathe machining techniques are involved.
It can be used in shop-floors as a stand-alone product for CAM-centric customers, who will particularly appreciate the product's ease-of-use and high level of manufacturing capabilities.
Lathe Machining can be combined with DELMIA products for overall manufacturing process integration, simulation and optimization, particularly for bigger customers concerned by high quality and quick time-to-market.

Before Reading this Guide

Before reading this guide, you should be familiar with basic Version 5 concepts such as document windows, standard and view toolbars. Therefore, we recommend that you read the *Infrastructure User's Guide* that describes generic capabilities common to all Version 5 products. It also describes the general layout of V5 and the interoperability between workbenches.

You may also like to read the following complementary product guides, for which the appropriate license is required:

- *NC Manufacturing Infrastructure User's Guide*: explains how to use common Machining functionalities
- *Prismatic Machining User's Guide*: provides useful information about axial machining operations.

Getting the Most Out of this Guide

To get the most out of this guide, we suggest that you start reading and performing the step-by-step Getting Started tutorial. This tutorial will show you how to produce an NC program for turning.

Once you have finished, you should move on to the User Tasks section, which gives more complete information about the product's functionalities. The Reference section provides useful complementary information.

The Workbench Description section, which describes the commands that are specific to Lathe Machining, and the Customizing section, which explains how to customize settings, and the Methodology section, which provides useful information about recommended work methods, will also certainly prove useful.

Accessing Sample Documents

To perform the scenarios, you will be using sample documents contained in the doc/online/lmgug_C2/samples or doc/online/lmgug_D2/samples folder. For more information about this, refer to Accessing Sample Documents in the *Infrastructure User's Guide*.

Conventions

Certain conventions are used in CATIA, ENOVIA & DELMIA documentation to help you recognize and understand important concepts and specifications.

Graphic Conventions

The three categories of graphic conventions used are as follows:

- Graphic conventions structuring the tasks
- Graphic conventions indicating the configuration required
- Graphic conventions used in the table of contents

Graphic Conventions Structuring the Tasks

Graphic conventions structuring the tasks are denoted as follows:

This icon...	Identifies...
	estimated time to accomplish a task
	a target of a task
	the prerequisites
	the start of the scenario
	a tip
	a warning
	information
	basic concepts
	methodology
	reference information
	information regarding settings, customization, etc.
	the end of a task
	functionalities that are new or enhanced with this release
	allows you to switch back to the full-window viewing mode

Graphic Conventions Indicating the Configuration Required

Graphic conventions indicating the configuration required are denoted as follows:

This icon...	Indicates functions that are...
	specific to the P1 configuration
	specific to the P2 configuration
	specific to the P3 configuration

Graphic Conventions Used in the Table of Contents

Graphic conventions used in the table of contents are denoted as follows:

This icon...	Gives access to...
	Site Map
	Split View Mode
	What's New?
	Overview
	Getting Started
	Basic Tasks
	User Tasks or Advanced Tasks
	Interoperability
	Workbench Description
	Customizing
	Administration Tasks
	Reference
	Methodology
	Frequently Asked Questions
	Glossary
	Index

Text Conventions

The following text conventions are used:

- The titles of CATIA, ENOVIA and DELMIA documents *appear in this manner* throughout the text.
- **File** -> **New** identifies the commands to be used.
- Enhancements are identified by a blue-colored background on the text.

How to Use the Mouse

The use of the mouse differs according to the type of action you need to perform.

Use this mouse button...	Whenever you read...
	• Select (menus, commands, geometry in graphics area, ...) • Click (icons, dialog box buttons, tabs, selection of a location in the document window, ...) • Double-click • Shift-click • Ctrl-click • Check (check boxes) • Drag • Drag and drop (icons onto objects, objects onto objects)
	• Drag • Move
	• Right-click (to select contextual menu)

What's New?

Enhanced Functionalities

Support turning operations on milling center machines

For a lathe center machine, the turning plane is defined once at part operation level. To provide turning operations on a milling center machine with facing heads, it is now possible to select several turning machining planes at machining operation level.

Enhancements brought by the NC Manufacturing Infrastructure

This product benefits from enhancements to the infrastructure's general functions (NC program review, material removal simulation, and so on). Please refer to the *NC Manufacturing Infrastructure User's Guide* for more information.

Getting Started

Before getting into the detailed instructions for using Lathe Machining, this tutorial is intended to give you a feel of what you can accomplish with the product.

It provides the following step-by-step scenario that shows you how to use some of the key functionalities.

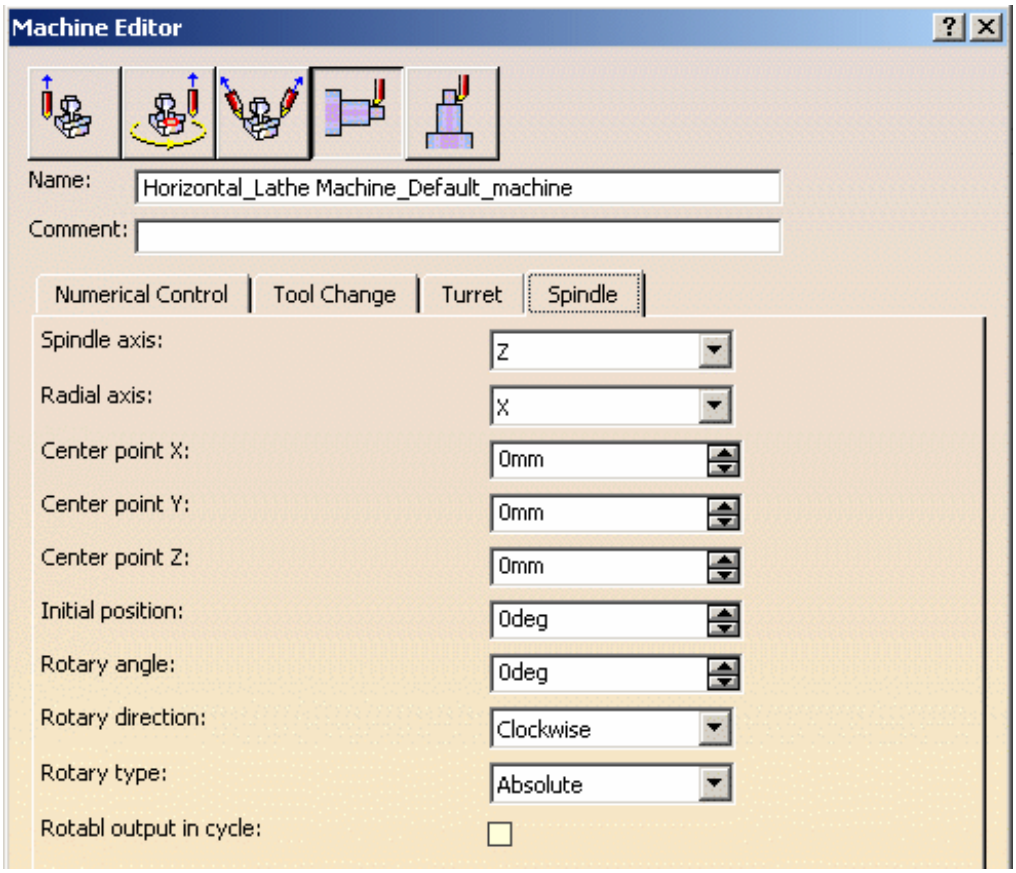
Open the Part to Machine
Create a Rough Turning Operation
Replay the Toolpath
Create a Groove Turning Operation
Create Profile Finish Turning Operation
Generate NC Code

Open the Part to Machine

This first task shows you how to open a part, enter the Lathe Machining workbench and make basic modifications to the Part Operation.

1. Select **File** > **Open** then select the Lathe01.CATPart document.
2. Select **Machining** > **Lathe Machining** from the **Start** menu.

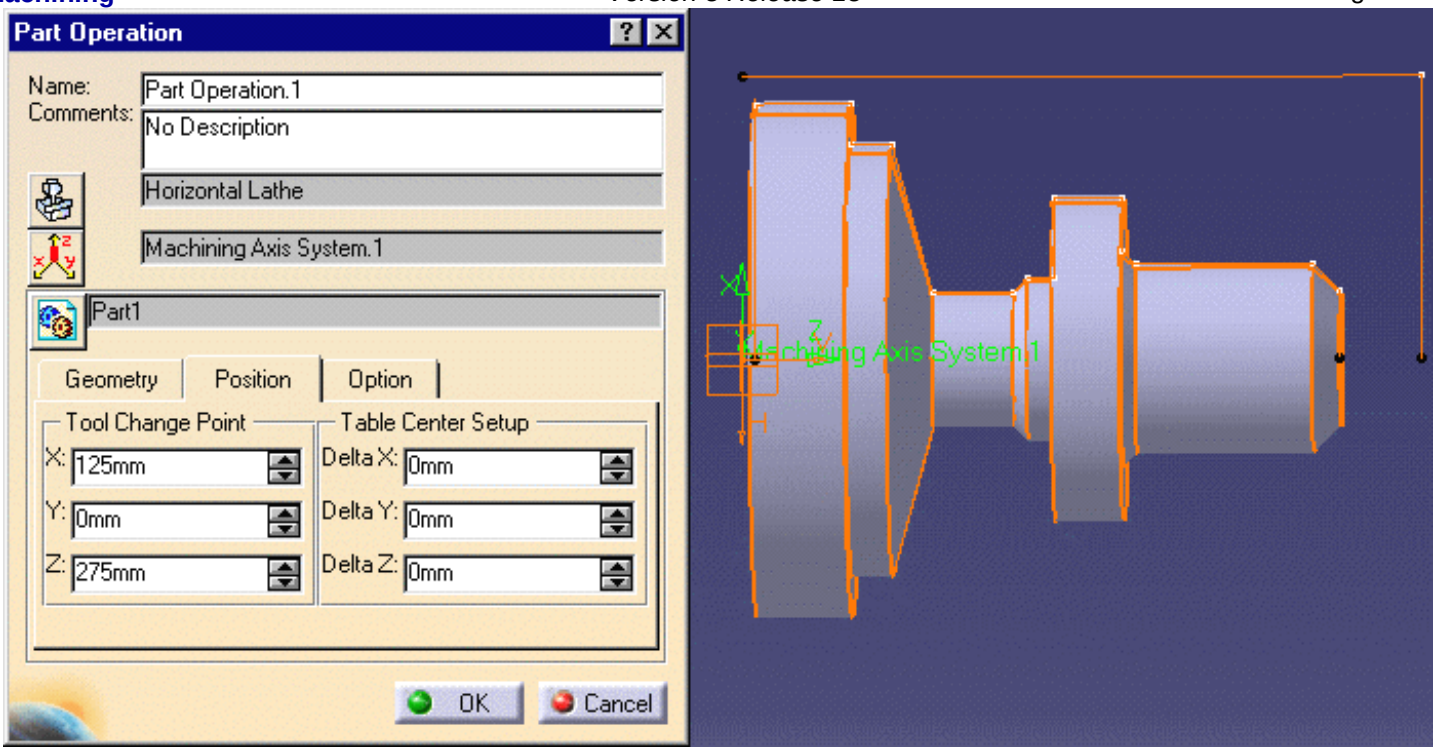
   The Lathe Machining workbench appears. The part is displayed in the Setup Editor window along with the manufacturing specification tree.
3. Double click Part Operation.1 in the tree to display the Part Operation dialog box.
4. Click the Machine icon. The Machine Editor dialog box appears.



- Select the Horizontal Lathe Machine icon.
- Set the spindle axis to Z and the radial axis to X.
- Click OK.

Setting the spindle axis to Z defines the C-axis (that is, rotary motion about the Z-axis). This allows creation of indexed machine rotations for milling and drilling operations.

5. Set the tool change point in the Position tab page as shown below.



6. Click OK to confirm your modifications to the Part Operation.
7. Select Manufacturing Program.1 in the tree to make it the current entity.

To insert program entities such as machining operations, tools and auxiliary commands you can either:

- make the program current before clicking the *insert program entity* command
- click the *insert program entity* command then make the program current.

Create a Rough Turning Operation

This task shows you how to create a Longitudinal Rough Turning operation for machining part of the workpiece.

This operation will use the tool proposed by the program, so you just need to specify the geometry to be machined and set some of the machining parameters.

1. Select the Rough Turning icon .

   A Rough Turning.1 entity along with a default tool is added to the program.

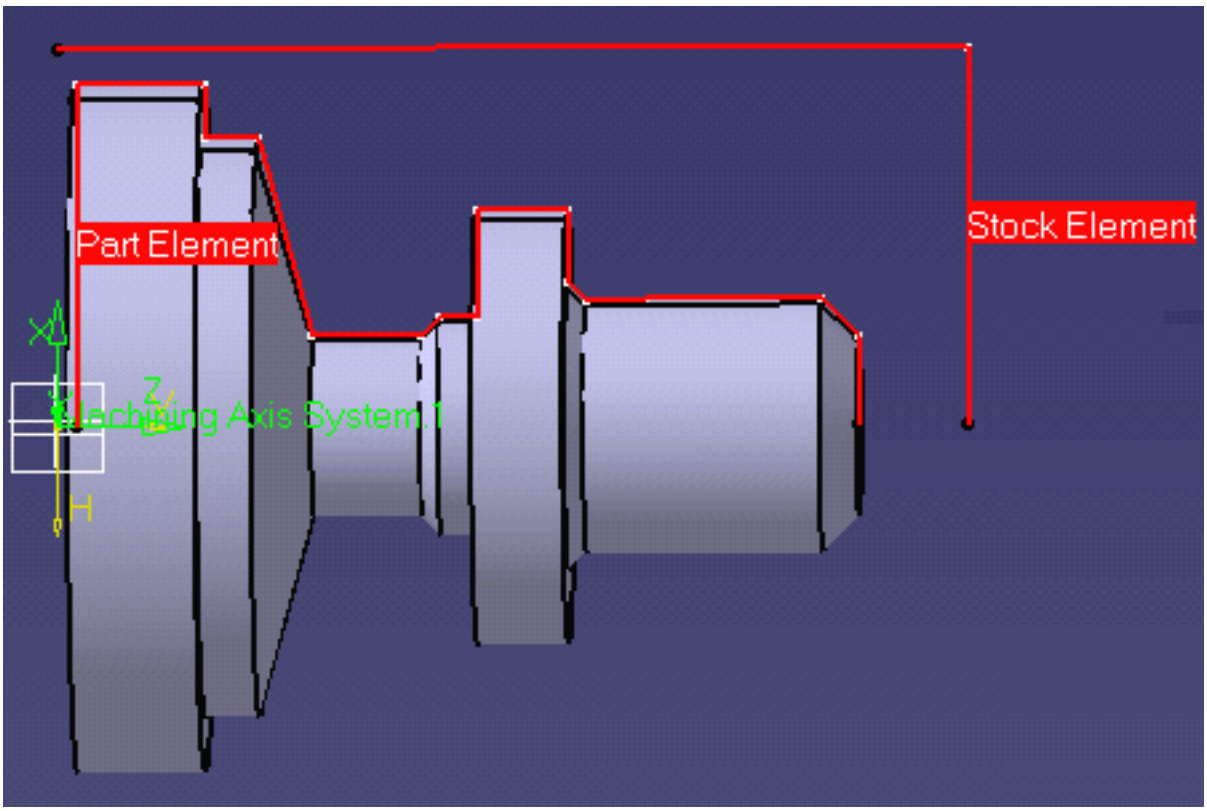
   The Rough Turning dialog box appears directly at the Geometry tab page .

2. Click the red Stock area in the icon, then select the stock profile as shown.

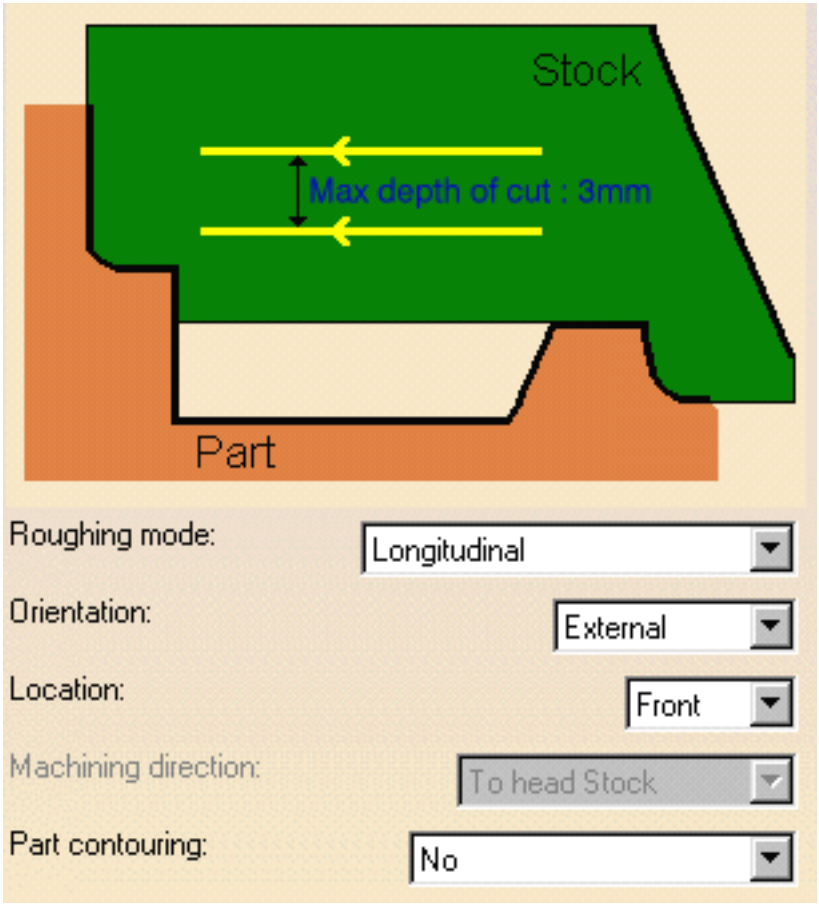
   Click OK in the Edge Selection toolbar to end your selection.

3. Click the red Part area in the icon, then select the part profile as shown.

   Click OK in the Edge Selection toolbar to end your selection.



4. Select the Strategy tab page  and set the parameters as shown.



5. Click OK to create the operation.

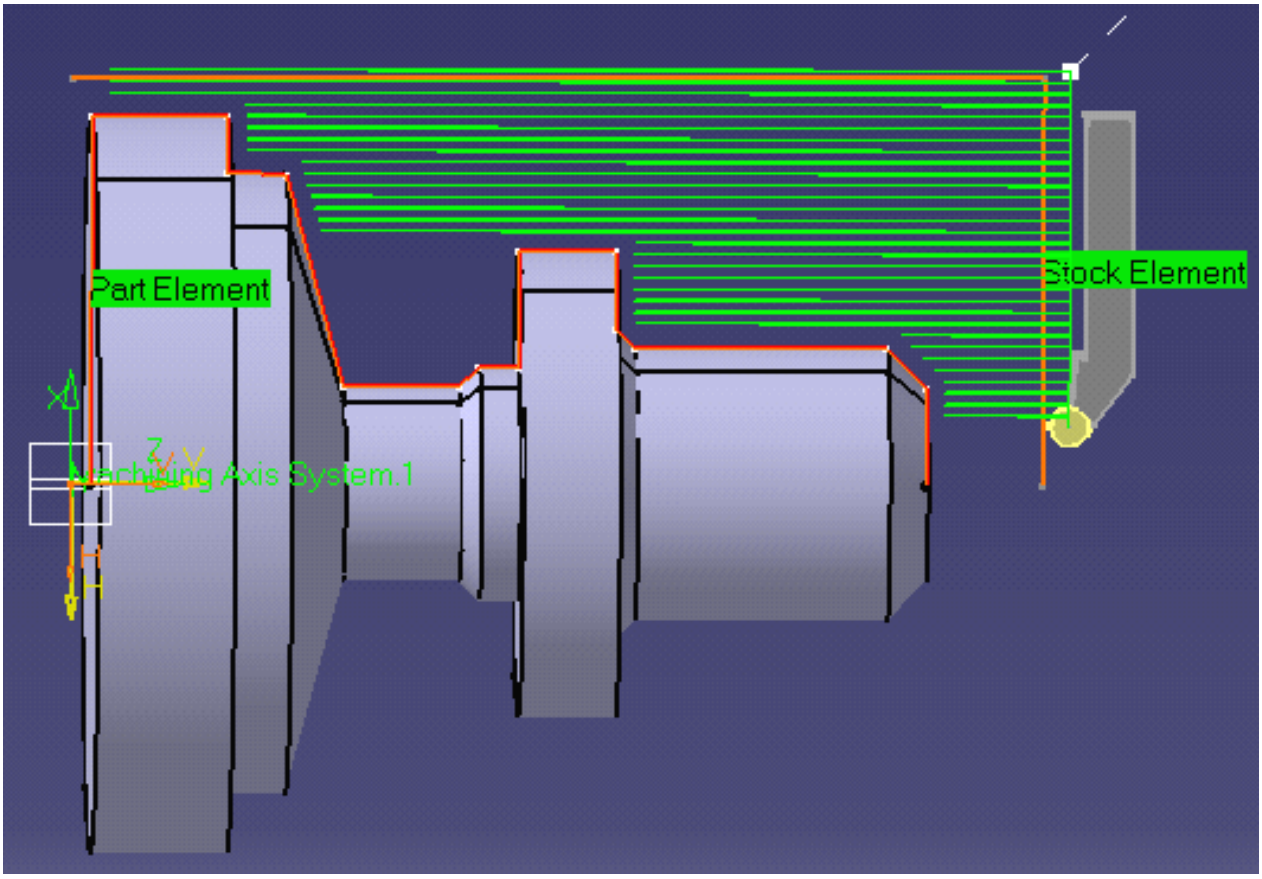
Replay the Tool Path

This task shows you how to replay the tool path of the Roughing operation.

1. Select the Roughing operation in the tree then select the Replay Tool Path icon .

   The Replay dialog box appears.

2. Choose the Continuous replay mode by means of the drop down icon .



3. Click the button to position the tool at the start point of the operation.
4. Click the button to start the replay. The tool moves along the computed trajectory.
5. Click OK to quit the replay mode.

Create a Groove Turning Operation

This task shows you how to create a Groove Turning operation to machine part of the workpiece.

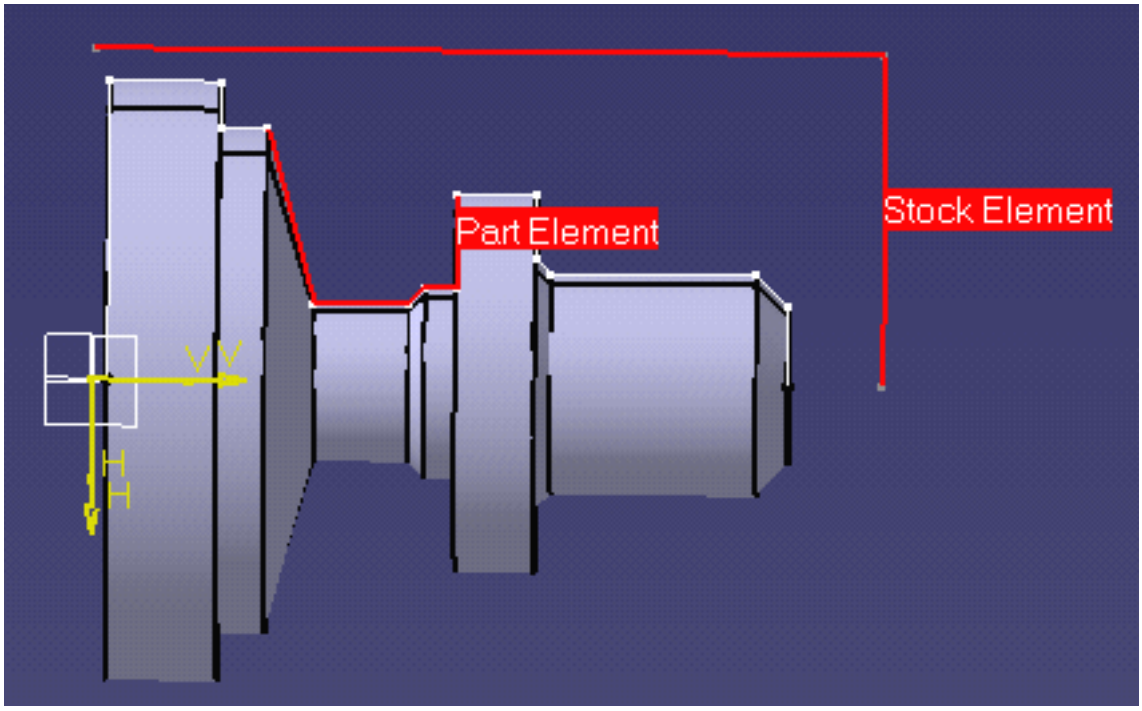
You will specify the geometry to be machined, set some of the machining parameters and select a new tool.

Make sure that the Rough Turning operation is the current entity in the program.

1. Select the Groove Turning icon .

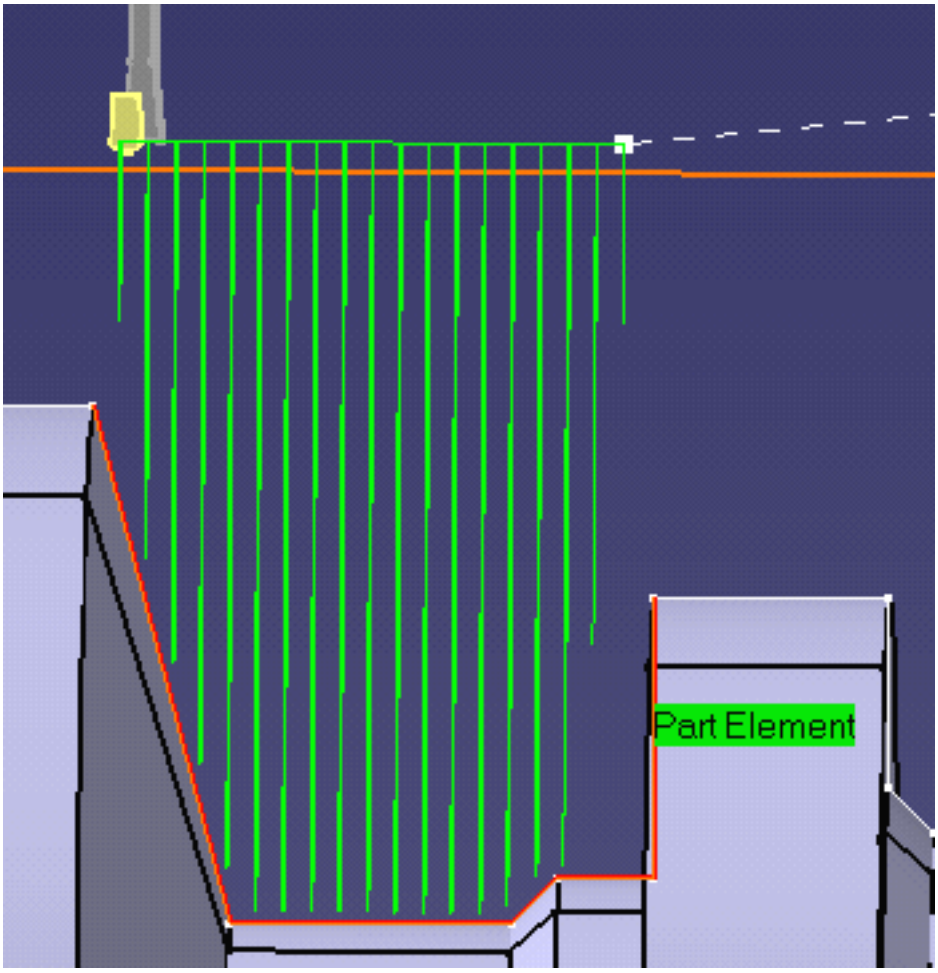
   The Groove Turning dialog box appears directly at the Geometry page .

2. Click the red Stock area in the icon, then select the stock profile as shown.
3. Click the red Part area in the icon, then select the groove profile as shown.



4. Select the Strategy tab page and check machining parameters. Set the Gouging Safety Angle to 10 degrees.
5. Select the Insert Holder tab in the Tooling page .
   - Enter a name of the new tool (for example, Grooving Tool).
   - Double click the l2 (shank length 2) parameter in the icon, then enter 60mm in the Edit Parameter dialog box.
   - Set the Max cutting depth Technology parameter to 80mm.

6. Click Replay in the dialog box to visually check the operation's tool path.



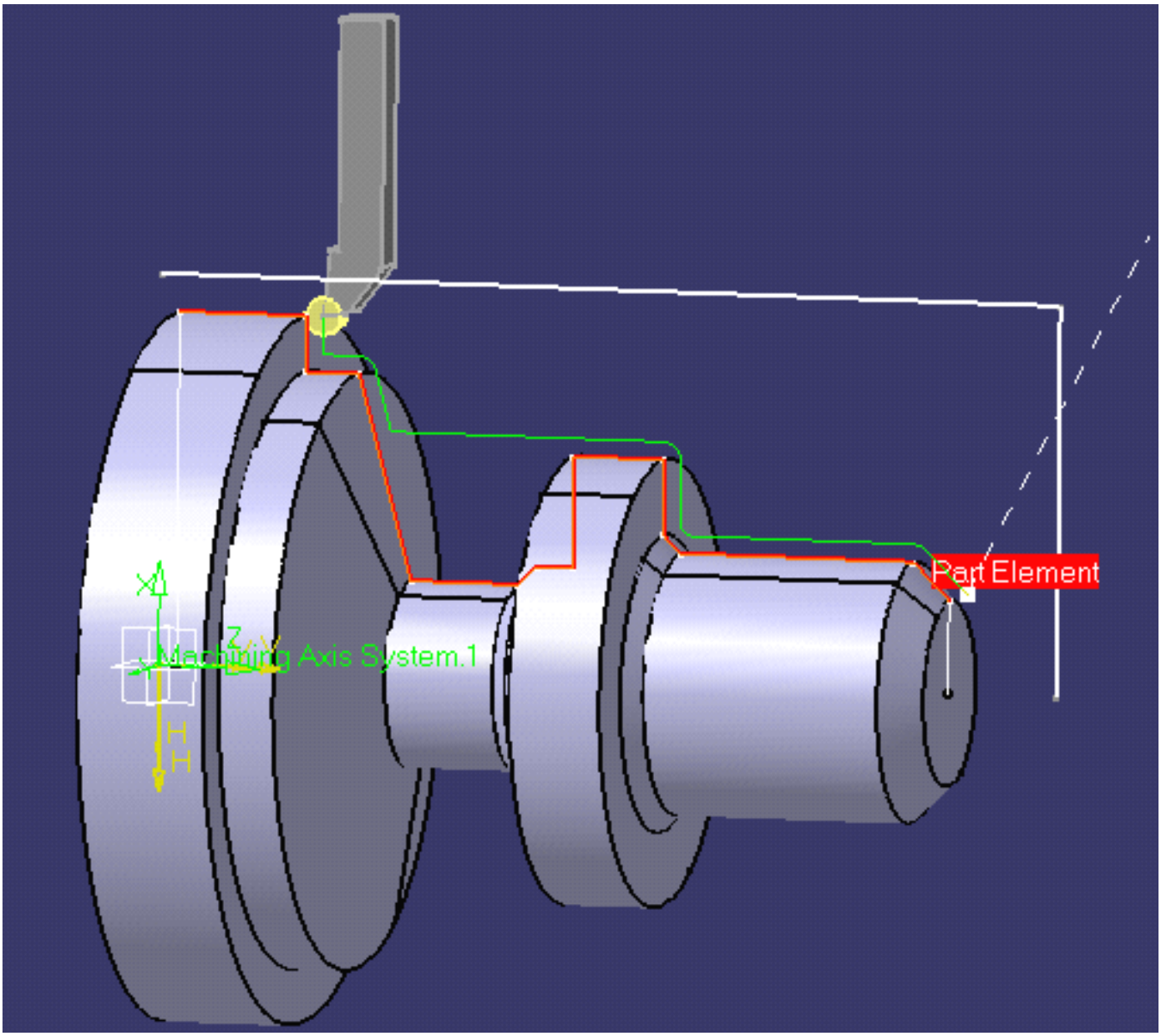
Click OK to exit the replay mode and return to the Groove Turning dialog box.

7. Click OK to create the operation.

Create a Profile Finish Turning Operation

This task shows you how to insert a Profile Finish Turning operation in the program.

1. Select the Profile Finish Turning icon . The Profile Finish Turning dialog box appears directly at the Geometry page .
2. Select the red part in the sensitive icon then select the part profile.
3. Select the Strategy tab page and set the Leading Safety Angle to 0 degrees.
4. Click Replay to replay the operation as described previously.



Click OK to exit the replay mode and return to the Profile Finish Turning dialog box.

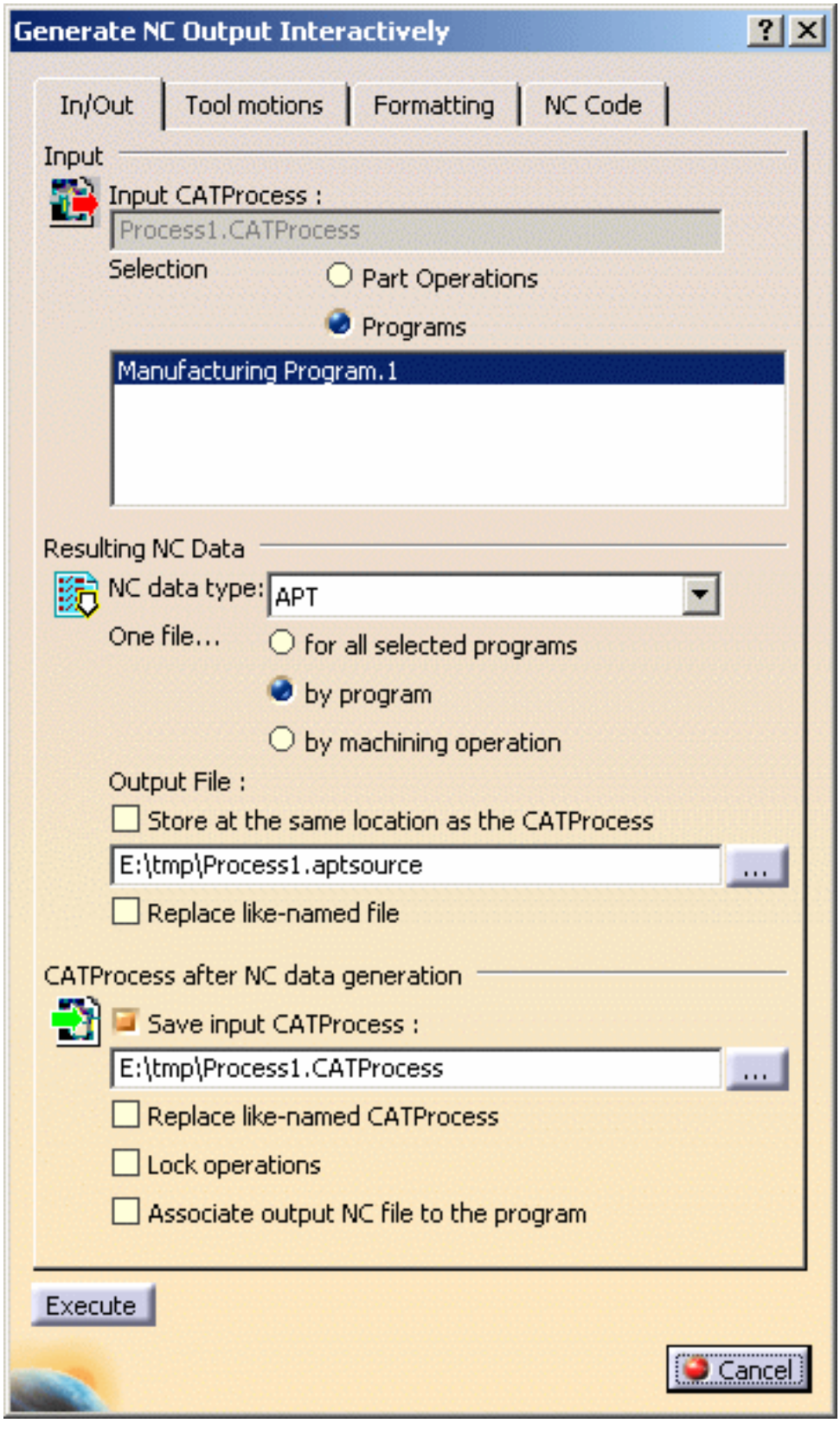
5. Click OK to create the operation in the program.

Generate NC Code

This task shows you how to generate NC code from the program. An APT source file will be generated in this example.

Before doing this task, double click the Part Operation entity in the tree and, in the dialog box that appears, click the Machine icon to access the Machine Editor dialog box. Make sure that you have selected a Horizontal lathe machine and that the desired NC data format is set to Axis (X, Y, Z).

1. Use the right mouse key on the Manufacturing Program.1 entity in the tree to select **Generate NC Code Interactively**. The Generate NC Output Interactively dialog box appears.



2. Select APT as the desired NC data type.
3. Click the Output File button to select the folder where you want the file to be saved and specify the name of the file.
4. Click **Execute** to generate the APT source file.

An extract from a typical APT source file is given below.

```
$$ ------------------------------------------------------------
$$ Generated on Wednesday, April 07, 2004 11:16:08 AM
$$ CATIA APT VERSION 1.0
$$ ------------------------------------------------------------
$$ Manufacturing Program.1
$$ Part Operation.1
$$*CATIA0
$$ Manufacturing Program.1
$$ 1.00000 0.00000 0.00000 0.00000
$$ 0.00000 1.00000 0.00000 0.00000
$$ 0.00000 0.00000 1.00000 0.00000
PARTNO Part Operation.1
$$ OPERATION NAME : Turning Tool Change.1
$$ Start generation of : Turning Tool Change.1
$$ TOOLCHANGEBEGINNING
CUTTER/ 10.000000
TOOLNO/1,TURN,1,0,9, 5.000000,$
0.000000, 0.000000, 0.400000,MMPR, 70.000000,RPM,$
CCLW,ON, 0.000000,NOTE
TPRINT/T1 External Insert-Holder,T1 External Insert-Holder,Turning Tool$
Assembly.1
LOADTL/1,1,1
$$ TOOLCHANGEEND
$$ End of generation of : Turning Tool Change.1
$$ OPERATION NAME : Rough Turning.1
$$ Start generation of : Rough Turning.1
SWITCH/9
FEDRAT/ 0.3000,MMPR
SPINDL/ 70.0000,RPM,CCLW
GOTO / 107.08333, 0.00000, 257.00000
GOTO / 107.08333, 0.00000, 255.00000
...
FEDRAT/ 0.8000,MMPR
GOTO / 40.21213, 0.00000, 150.21213
$$ End of generation of : Rough Turning.1
$$ OPERATION NAME : Turning Tool Change.2
$$ Start generation of : Turning Tool Change.2
$$ TOOLCHANGEBEGINNING
CUTTER/ 2.400000
TOOLNO/1,TURN,1,0,9, 1.200000,$
0.000000, 0.000000, 0.400000,MMPR, 70.000000,RPM,$
CCLW,ON,,NOTE
TPRINT/T3 External Groove Insert-Holder,T3 External Groove Insert-Holde$
r,Turning Tool Assembly.1_1
LOADTL/1,1,1
$$ TOOLCHANGEEND
$$ End of generation of : Turning Tool Change.2
$$ OPERATION NAME : Groove Turning.1
$$ Start generation of : Groove Turning.1
SWITCH/9
FEDRAT/ 0.3000,MMPR
SPINDL/ 70.0000,RPM,CCLW
GOTO / 108.20000, 0.00000, 96.70000
```

```
GOTO / 106.20000, 0.00000, 96.70000
...
RAPID
GOTO / 108.20000, 0.00000, 57.91213
$$ End of generation of : Groove Turning.1
$$ OPERATION NAME : Turning Tool Change.3
$$ Start generation of : Turning Tool Change.3
$$ TOOLCHANGEBEGINNING
CUTTER/ 10.000000
TOOLNO/1,TURN,1,0,9, 5.000000,$
0.000000, 0.000000, 0.400000,MMPR, 70.000000,RPM,$
CCLW,ON, 0.000000,NOTE
TPRINT/T1 External Insert-Holder,T1 External Insert-Holder,Turning Tool$
Assembly.1
LOADTL/1,1,1
$$ TOOLCHANGEEND
$$ End of generation of : Turning Tool Change.3
$$ OPERATION NAME : Profile Finish Turning.1
$$ Start generation of : Profile Finish Turning.1
SWITCH/9
FEDRAT/ 0.3000,MMPR
SPINDL/ 70.0000,RPM,CCLW
GOTO / -2.00000, 0.00000, 225.00000
GOTO / 0.00000, 0.00000, 225.00000
...
GOTO / 40.00000, 0.00000, 145.00000
FEDRAT/ 0.8000,MMPR
GOTO / 40.21213, 0.00000, 144.78787
$$ End of generation of : Profile Finish Turning.1
FINI
```

User Tasks

The user tasks you will perform with Lathe Machining involve creating, editing and managing machining operations and other Machining entities.

Turning Operations
Axial Machining Operations
Auxiliary Operations
Part Operations, Manufacturing Programs and Machining Processes
Machining Resources and Other Entities
Verification, Simulation and NC Output

Turning Operations

The tasks in this section show you how to create turning operations in your manufacturing program.

Create a Rough Turning operation:
Select the Rough Turning icon and choose the desired roughing mode. You can then select the part and stock geometry and specify the tool to be used. Specify machining parameters, feeds and speeds, and NC macros as needed.

Basic tasks illustrate the following roughing modes:

- Longitudinal
- Face
- Parallel Contours.

Create a Recess Turning operation:
Select the Recess Turning icon and choose the desired recessing mode. You can then select the part and stock geometry and specify the tool to be used. Specify machining parameters, feeds and speeds, and NC macros as needed.

Create a Groove Turning operation:
Select the Groove Turning icon then select the part and stock geometry and specify the tool to be used. Specify machining parameters, feeds and speeds, and NC macros as needed.

Create a Profile Finish Turning operation:
Select the Profile Finish Turning icon then select the part profile and specify the tool to be used. Specify machining parameters, feeds and speeds, and NC macros as needed.

Create a Groove Finish Turning operation:
Select the Groove Finish Turning icon then select the part geometry and specify the tool to be used. Specify machining parameters, feeds and speeds, and NC macros as needed.

Create a Ramp Rough Turning operation.
Select the Ramp Rough Turning icon then select the part and stock geometry and specify the tool to be used. Specify machining parameters, feeds and speeds, and NC macros as needed.

Create a Ramp Recess Turning operation.
Select the Ramp Recess Turning icon then select the part and stock geometry and specify the tool to be used. Specify machining parameters, feeds and speeds, and NC macros as needed.

Create a Thread Turning operation:
Select the Thread Turning icon and choose the desired thread type. You can then select the part geometry and specify the tool to be used. Specify machining parameters, feeds and speeds, and NC macros as needed.

Create a Sequential Turning operation:
Create and manage a sequence of basic Go Standard, Go-Go, Go InDirv, Go Delta, and Follow tool motions in a single operation.

Turning on a Milling Center:
Create turning operations to machine large diameter holes on a milling center machine equipped with a facing head and rotary table.

Manage Local Information:
Assign local feedrates and local offsets to Lathe Profile Finishing and Groove Finishing operations.

Update Input Stock: choose to either update the stock automatically by setting an option on the Part Operation or update the stock manually for each lathe operation or axial machining operation along the spindle axis.

Create a Longitudinal Rough Turning Operation

This task illustrates how to create a Longitudinal Rough Turning operation in the program.

To create the operation you must define:

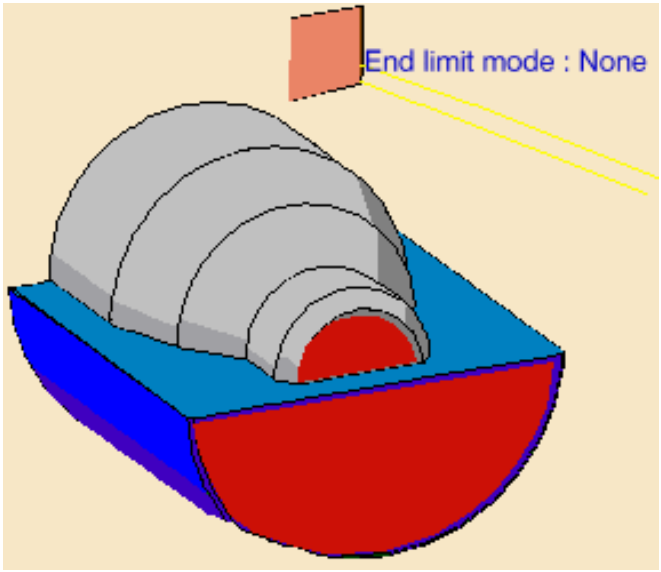
- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the Lathe01.CATPart document, then select **Machining > Lathe Machining** from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Rough Turning icon.

   A Rough Turning entity along with a default tool is added to the program.

   The Rough Turning dialog box appears directly at the Geometry tab page.
   This tab page includes a sensitive icon to help you specify the geometry to be machined.

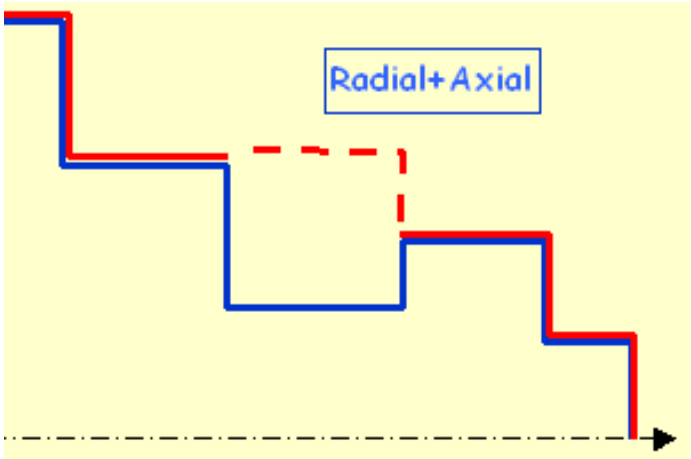


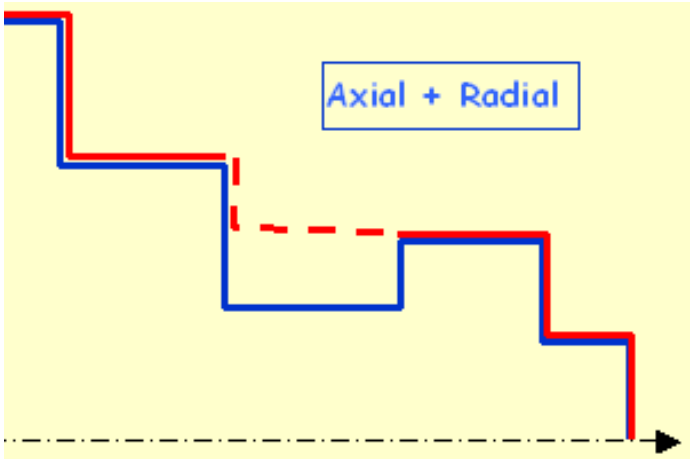
The part and stock of the icon are colored red indicating that this geometry is required. All other geometry is optional.

2. Click the red part in the icon then select the desired part profile in the 3D window.
   The Edge Selection toolbar appears to help you with contour selection.

   The Automatic Linking option allows you to select a first element then the element to navigate to in order to complete the profile selection.

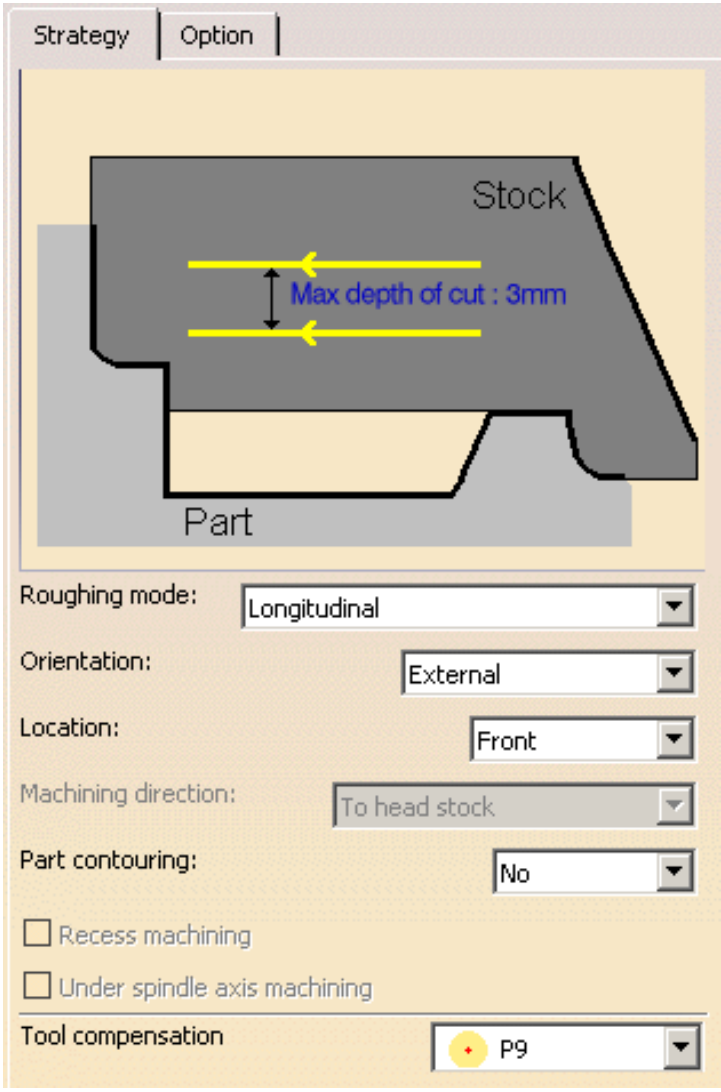
   The Axial/Radial and Radial/Axial Linking options are also useful for profile selection.





The part of the icon is now colored green indicating that this geometry is now defined.

3. Click the red stock in the icon then select the desired stock profile in the 3D window.
4. Set **Part Offset** to 5mm.
5. Select the Strategy tab page  to specify the main machining strategy parameters:
   - Roughing mode: **Longitudinal**
   - Orientation: **External**
   - Location: **Front**.



6. Double click **Max depth of cut** in the icon.

   Set this value to 15mm in the Edit Parameter dialog box and click OK.

Other optional parameters can be set in the Options tab (lead-in and so on).

A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

Please refer to Edit the Tool of a Lathe Operation.

7. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

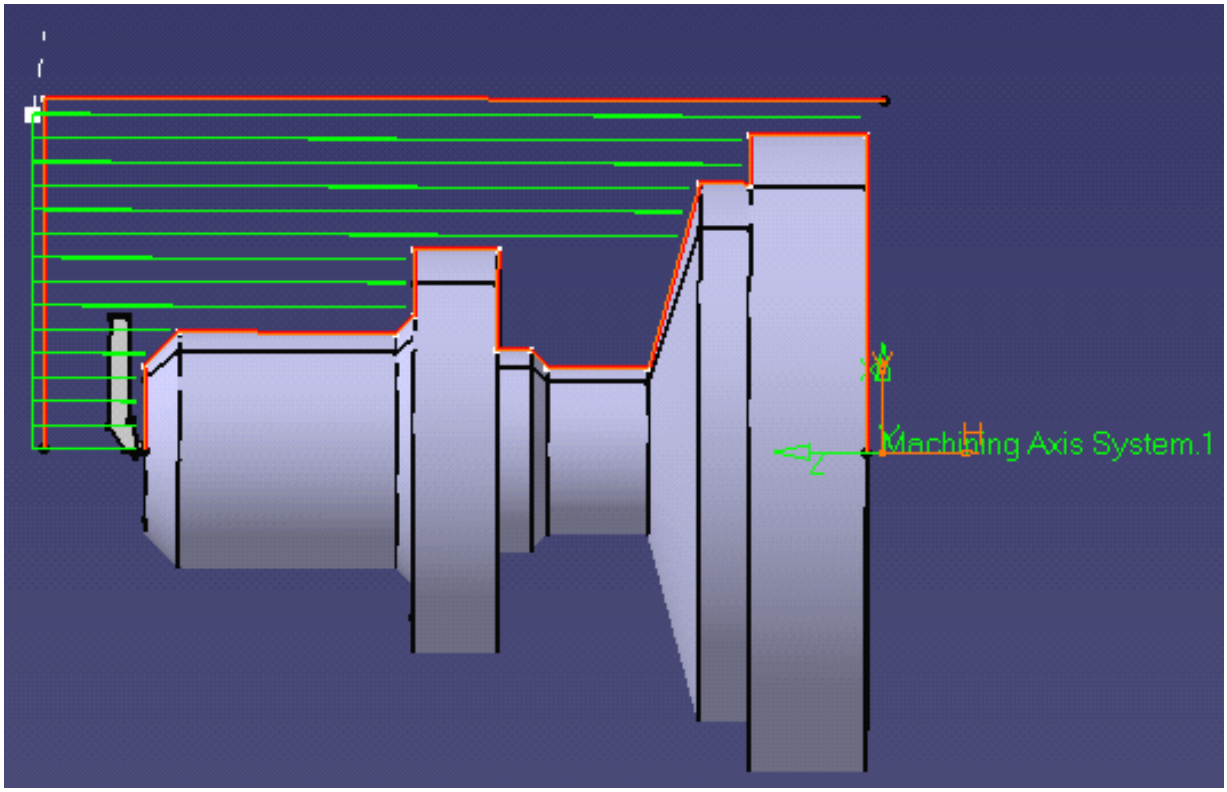
   See Feeds and Speeds for Rough Turning for more information.

8. Select the Macros tab page  to specify the operation's transition paths (approach and retract motion, for example).

   Approach linking and retract linking motions are interruptable for this type of operation.

   See Define Macros on a Lathe Operation for more information.

Before accepting the operation, you should check its validity by replaying the tool path.



9. Click OK to create the operation.

Create a Face Rough Turning Operation

This task shows how to insert a Face Rough Turning operation in the program.

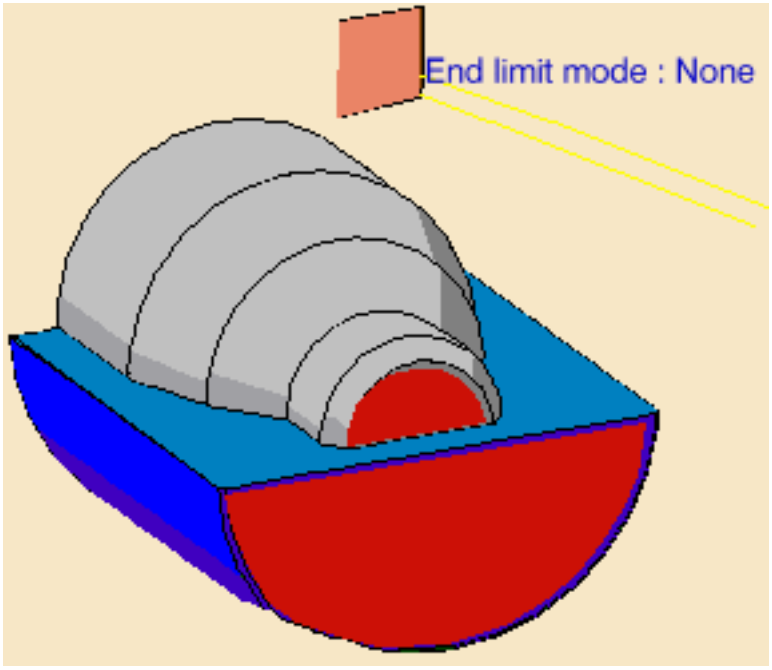
To create the operation you must define:

- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the Lathe01.CATPart document, then select **Machining > Lathe Machining** from the **Start** menu. Make the Manufacturing Program current in the specification tree.

1. Select the Rough Turning icon.

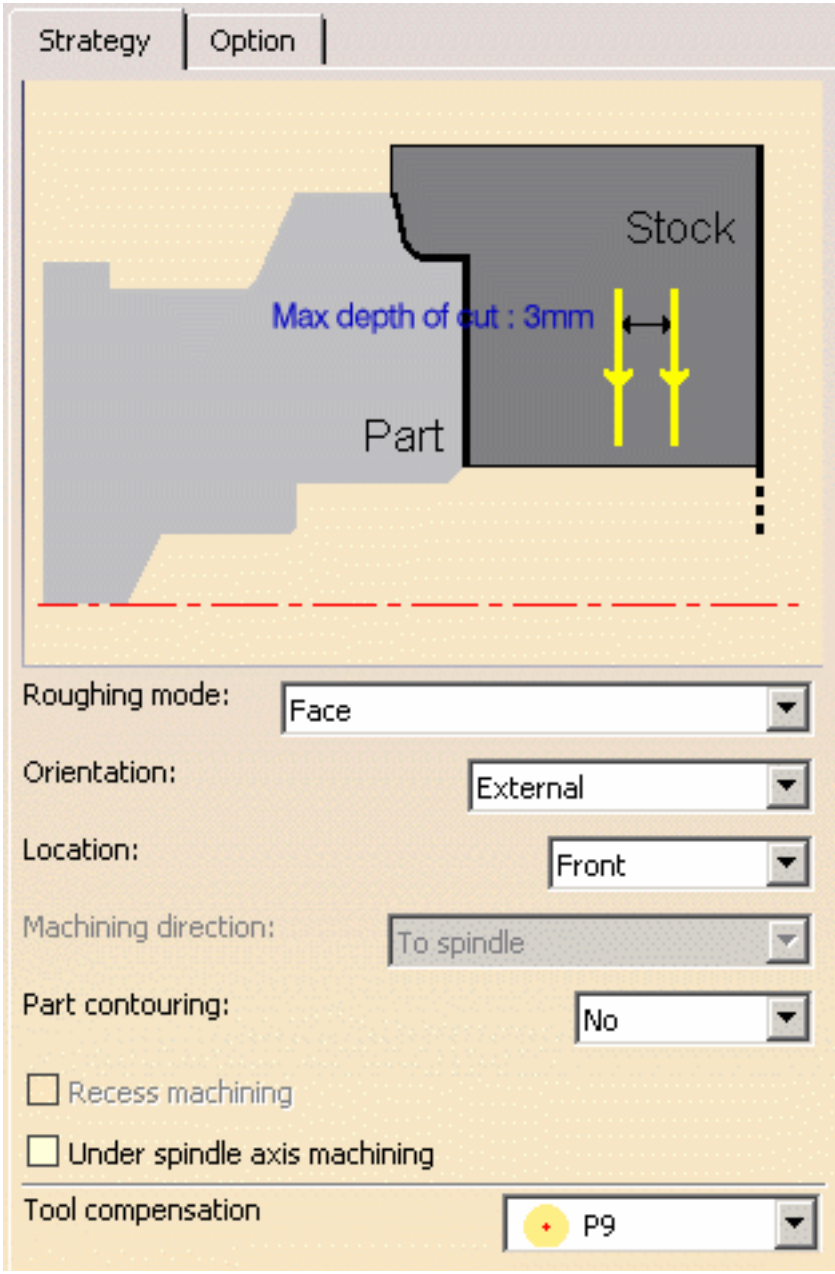
   The Rough Turning dialog box appears directly at the Geometry tab page.
   This page includes a sensitive icon to help you specify the geometry to be machined.



   The part and stock in the icon are colored red indicating that this geometry is required for defining the operation.

2. Click the red part in the icon then select the desired part profile in the 3D window.
   The part of the icon is now colored green indicating that this geometry is now defined.
3. Click the red stock in the icon then select the desired stock profile in the 3D window.
4. Set **Part Offset** to 5mm.

5. Select the Strategy tab page  to specify the main machining strategy parameters:
   - Roughing mode: **Face**
   - Orientation: **External**
   - Location: **Front**.



6. Double click **Max depth of cut** in the icon.

   Set this value to 10mm in the Edit Parameter dialog box and click OK.

7. In the Options tab, set the lift-off distance to 1.5mm.

A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use. This is described in Edit the Tool of a Lathe Operation.

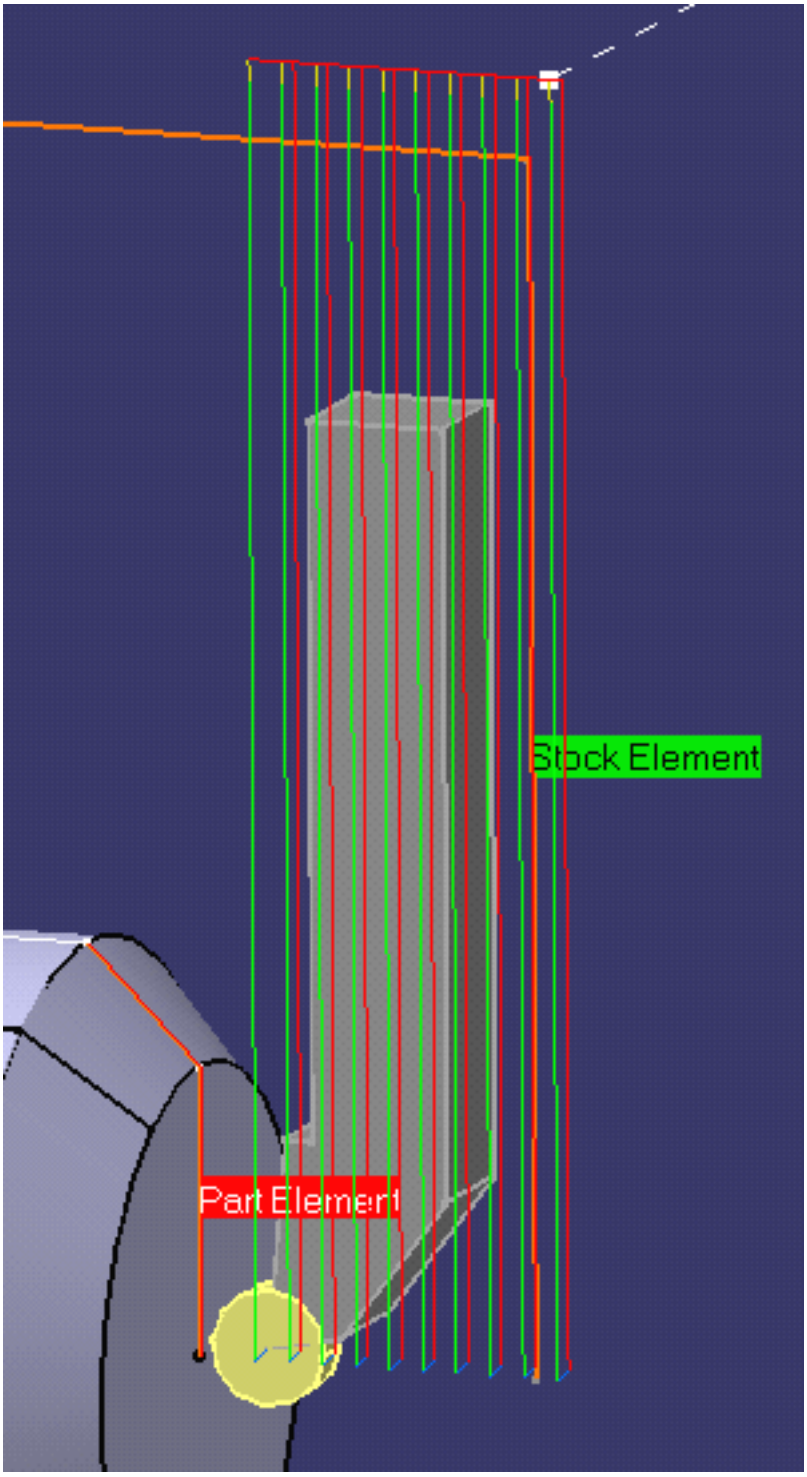
8. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation. See Feeds and Speeds for Rough Turning for more information.

If you want to specify approach and retract motion for the operation, select the Macros tab page  to specify the desired transition paths.

Approach linking and retract linking motions are interruptable for this type of operation.

See Define Macros on a Lathe Operation for more information.

9. Check the validity of the operation by replaying the tool path.



10. Click OK to create the operation.

Create a Parallel Contour Rough Turning Operation

This task shows how to insert a Parallel Contour Rough Turning operation in the program.

To create the operation you must define:

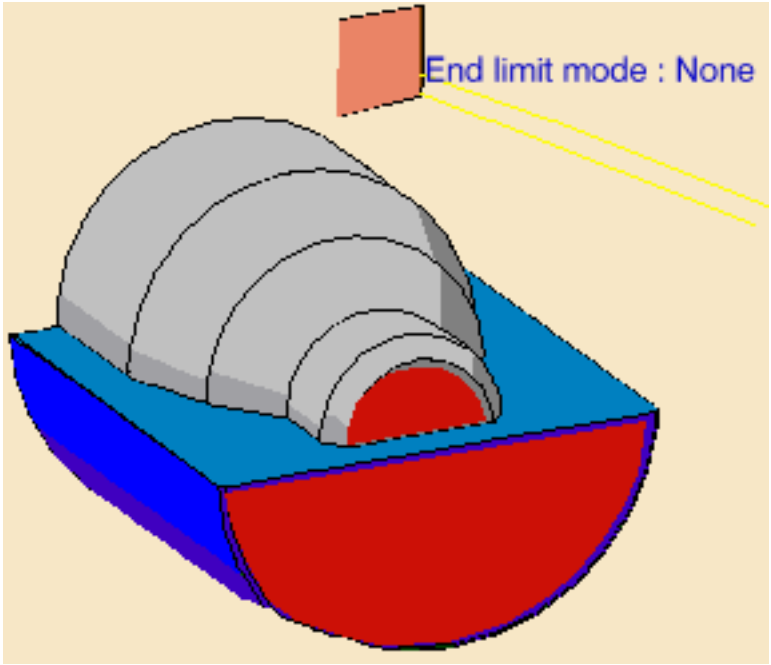
- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the Lathe01.CATPart document, then select **Machining > Lathe Machining** from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Rough Turning icon .

   A Rough Turning entity along with a default tool is added to the program.

   The Rough Turning dialog box appears directly at the Geometry tab page .
   This tab page includes a sensitive icon to help you specify the geometry to be machined.

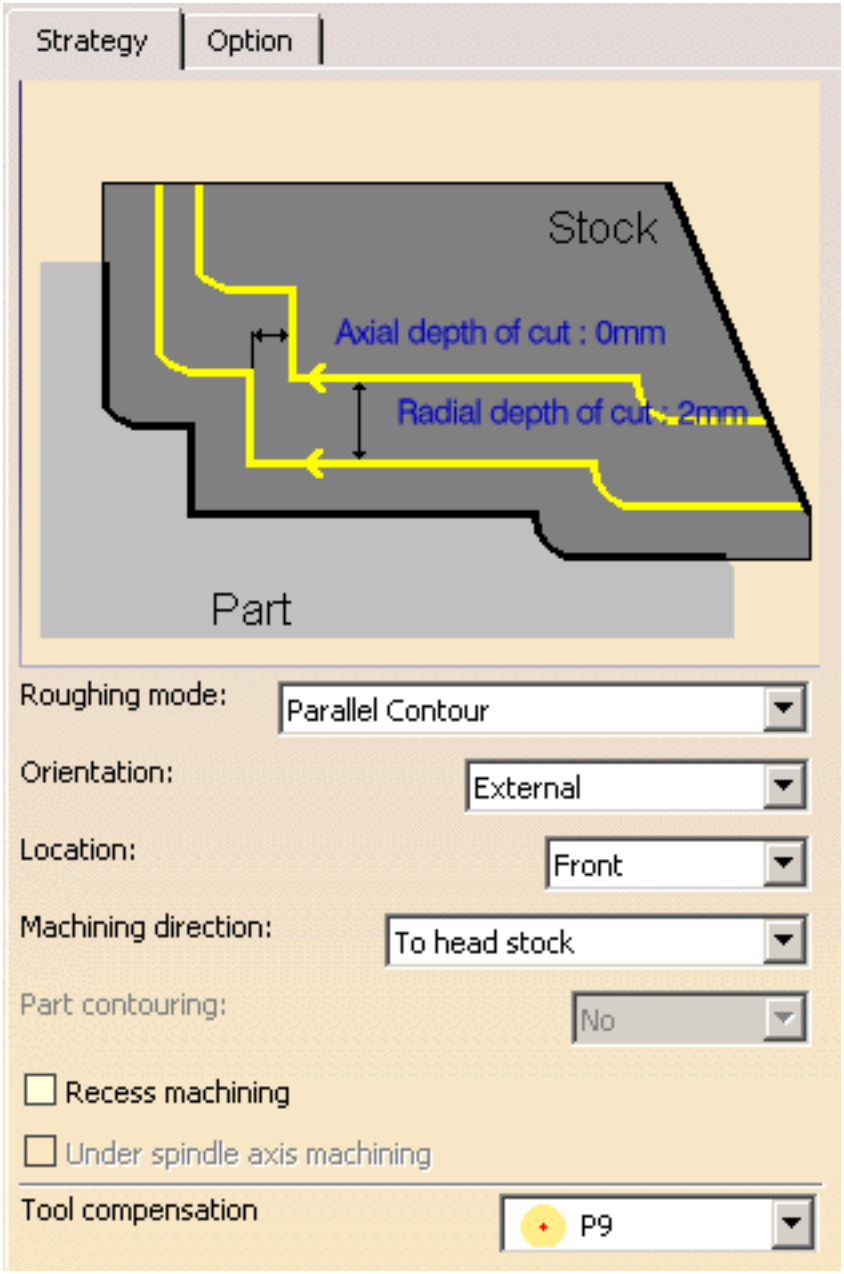


The part and stock of the icon are colored red indicating that this geometry is required.

2. Click the red part in the icon, then select the desired part profile in the 3D window.

   Select the stock in the same way.

3. Select the Strategy tab page  to specify the main machining strategy parameters:
   - Roughing mode: **Parallel Contour**
   - Orientation: **External**
   - Location: **Front**
   - Machining direction: **To head stock**.



4. Double click **Axial depth of cut** in the icon.

   Set this value to 3mm in the Edit Parameter dialog box and click OK.

   Set **Radial depth of cut** to 3mm in the same way.

   Other optional parameters can be set in the Options tab (lead-in and so on).

When recess machining is active in Parallel Contour Rough Turning, Axial and Radial Depth of Cut must have suitable values to ensure a collision free toolpath. See Recommendations for more information.

5. A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

   This is described in Edit the Tool of a Lathe Operation.

6. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

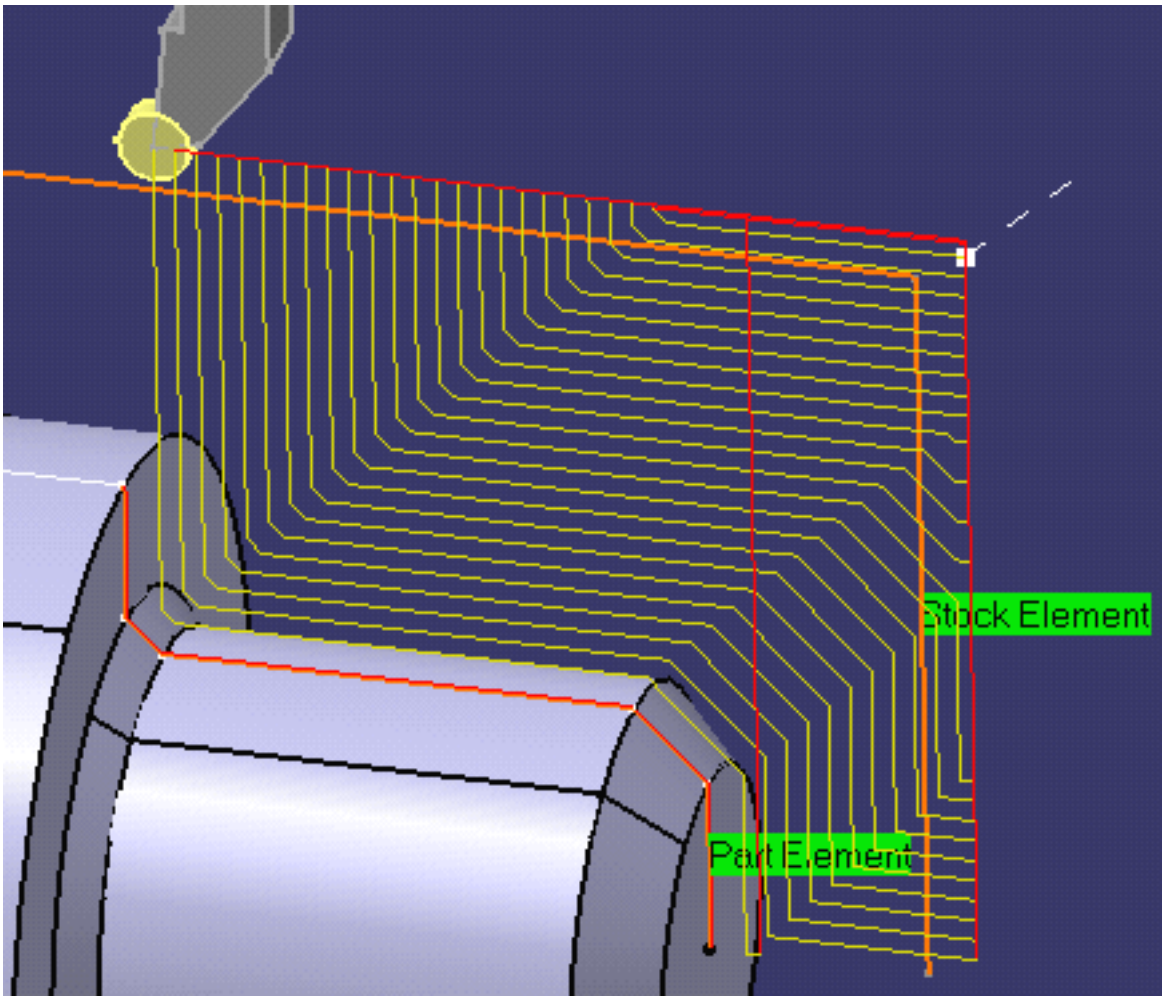
   See Feeds and Speeds for Rough Turning for more information.

7. If you want to specify approach and retract motion for the operation, select the Macros tab page  to specify the desired transition paths.

   Approach linking and retract linking motions are interruptible for this type of operation.

   See Define Macros on a Lathe Operation for more information.

8. Check the validity of the operation by replaying the tool path.



9. Click OK to create the operation.

Create a Recess Turning Operation

This task shows how to insert a Recess Turning operation in the program.

To create the operation you must define:

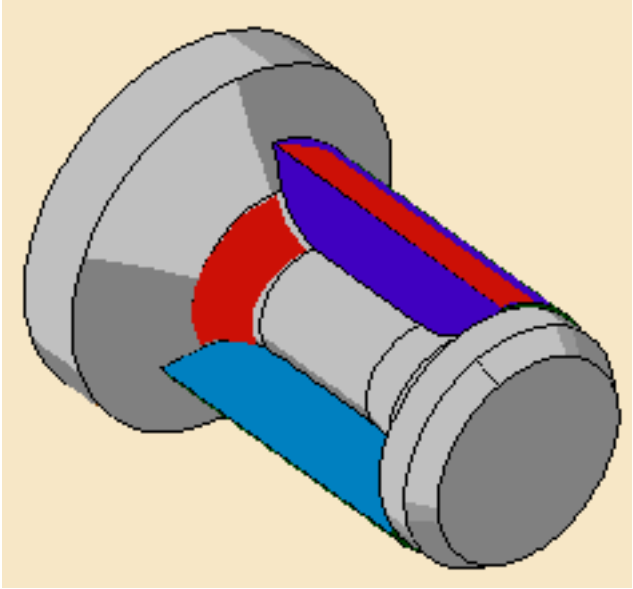
- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the Lathe01.CATPart document, then select **Machining > Lathe Machining** from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Recess Turning icon.

   A Recess Turning entity along with a default tool is added to the program.

   The Recess Turning dialog box appears directly at the Geometry tab page.
   This tab page includes a sensitive icon to help you specify the geometry to be machined.



The part and stock in the icon are colored red indicating that this geometry is required.

2. Click the red part in the icon then select the desired part profile in the 3D window.

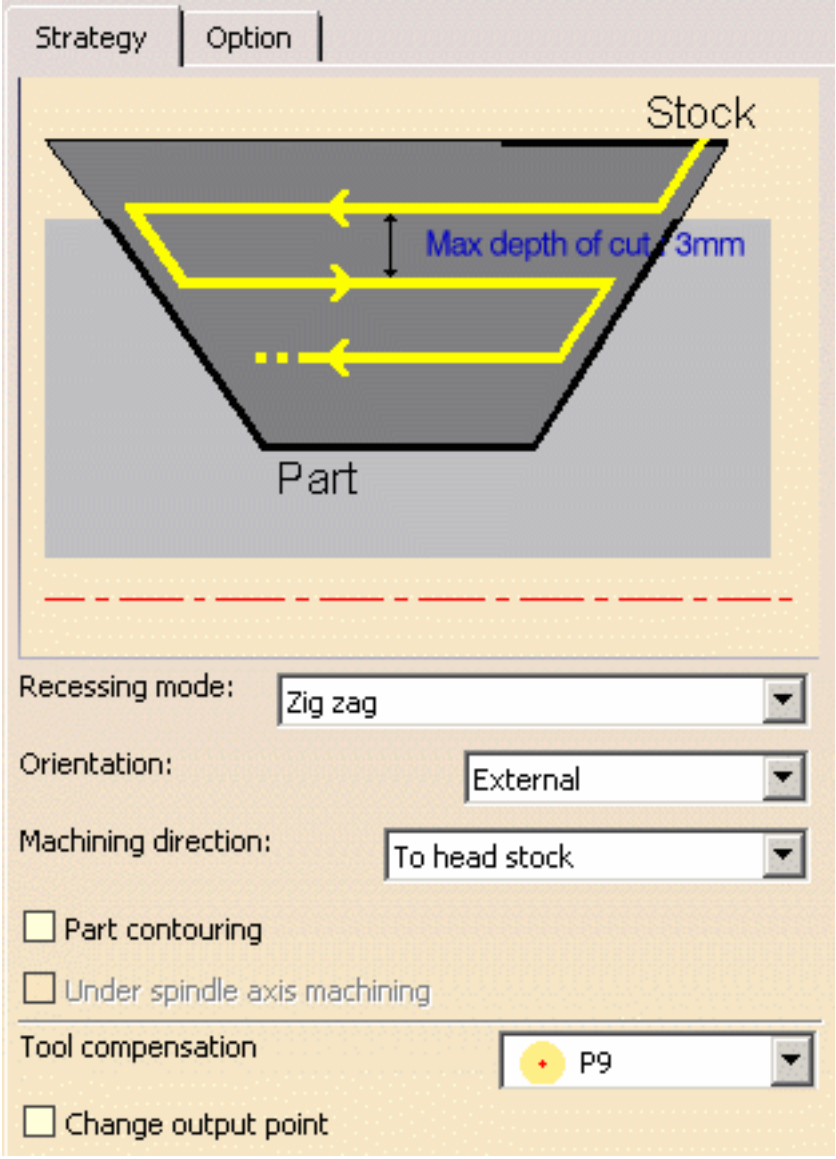
   Select the stock in the same way.

3. Select the Strategy tab page  to specify the main machining strategy parameters:
   - Recessing mode: **Zig zag**
   - Orientation: **External**
   - Machining direction: **To head stock**.

4. Double click **Max depth of cut** in the icon.

   Set this value to 10mm in the Edit Parameter dialog box and click OK.

   Other optional parameters can be set in the Options tab (lead-in and so on).



A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

This is described in Edit the Tool of a Lathe Operation.

5. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

   See Feeds and Speeds for Recess Turning for more information.

6. If you want to specify approach and retract motion for the operation, select the Macros tab page  to specify the desired transition paths.

   Approach linking and retract linking motions are interruptable for this type of operation.

   See Define Macros on a Lathe Operation for more information.

Before accepting the operation, you should check its validity by replaying the tool path.

7. Click OK to create the operation.

Create a Groove Turning Operation

This task shows how to insert a Groove Turning operation in the program.

To create the operation you must define:

- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the Lathe01.CATPart document, then select **Machining > Lathe Machining** from the **Start** menu. Make the Manufacturing Program current in the specification tree.

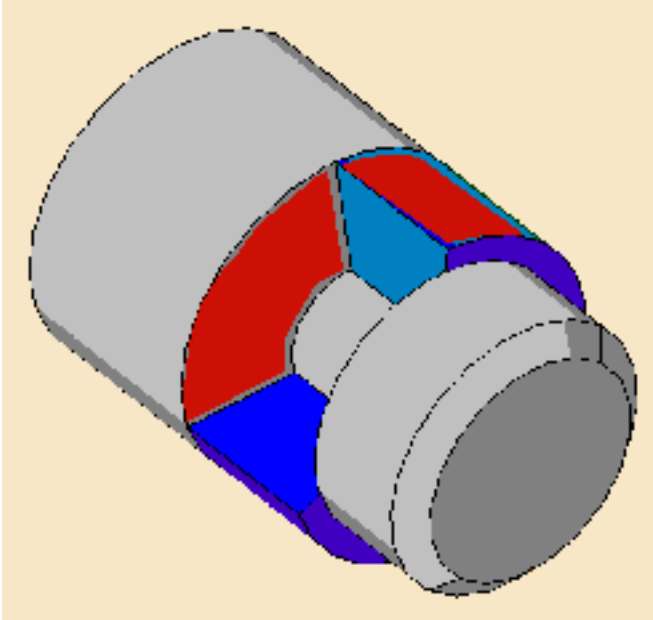
1. Select the Groove Turning icon .

   The Groove Turning dialog box appears directly at the Geometry tab page .

   This page includes a sensitive icon to help you specify the geometry to be machined.

   The part and stock in the icon are colored red indicating that this geometry is required for defining the operation.

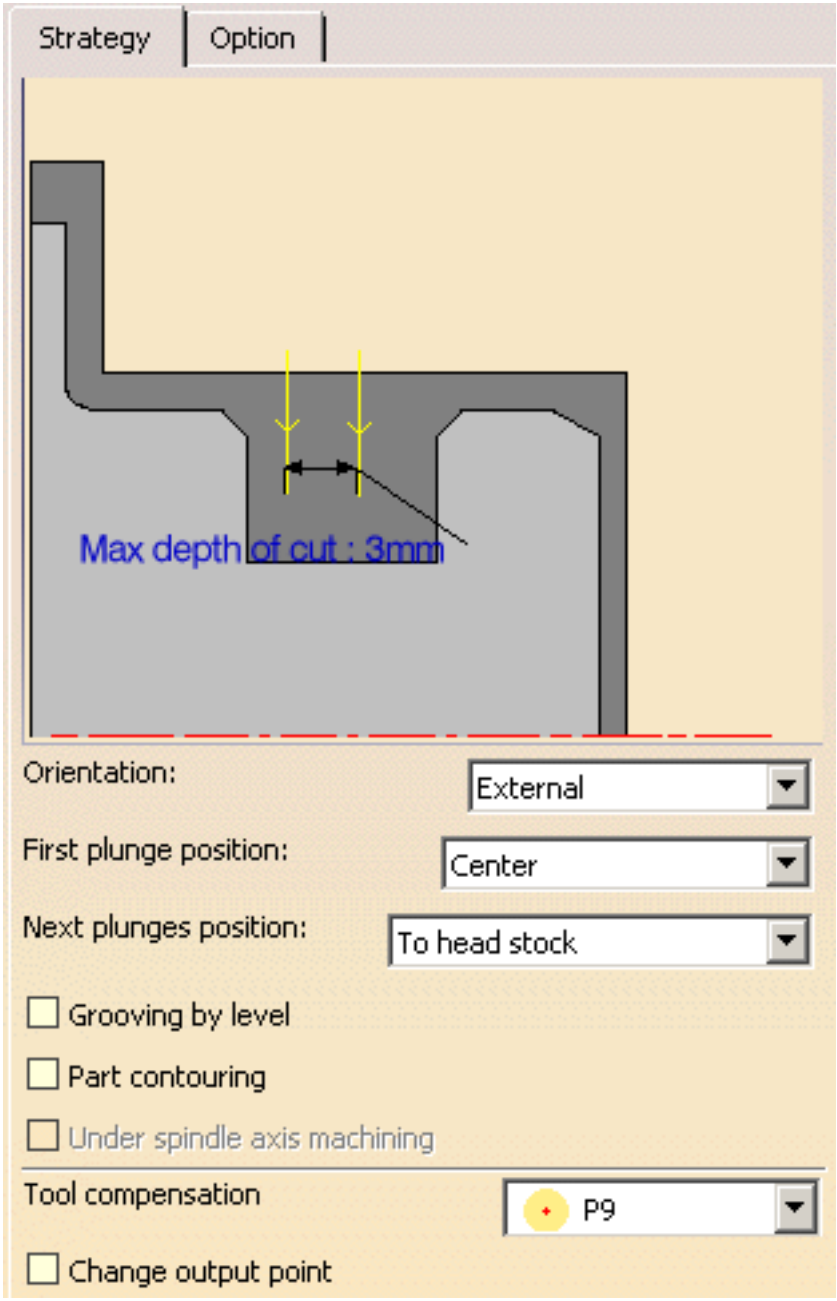
2. Click the red part in the icon, then select the desired part profile in the 3D window.



   Select the stock in the same way.

The part and stock of the icon are now colored green indicating that this geometry is now defined.

3. Select the Strategy tab page  to specify the main machining strategy parameters:
   - Orientation: **External**
   - First plunge position: **Center**
   - Next plunges position: **To head stock**.



4. Double click **Max depth of cut** in the icon.

   Set this value to 10mm in the Edit Parameter dialog box and click OK.

   Other optional parameters can be set in the Options tab (lead-in and so on).

A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

This is described in Edit the Tool of a Lathe Operation.

5. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

   See Feeds and Speeds for Groove Turning for more information.

You can add approach and retract motions to the operation in the Macros tab page  .

Approach linking and retract linking motions are interruptable for this type of operation.

See Define Macros on a Lathe Operation for more information.

6. Check the validity of the operation by replaying the tool path.
7. Click OK to create the operation.

Create a Profile Finish Turning Operation

This task shows how to insert a Profile Finish Turning operation in the program. To create the operation you must define:

- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the Lathe01.CATPart document, then select **Machining > Lathe Machining** from the **Start** menu. Make the Manufacturing Program current in the specification tree.

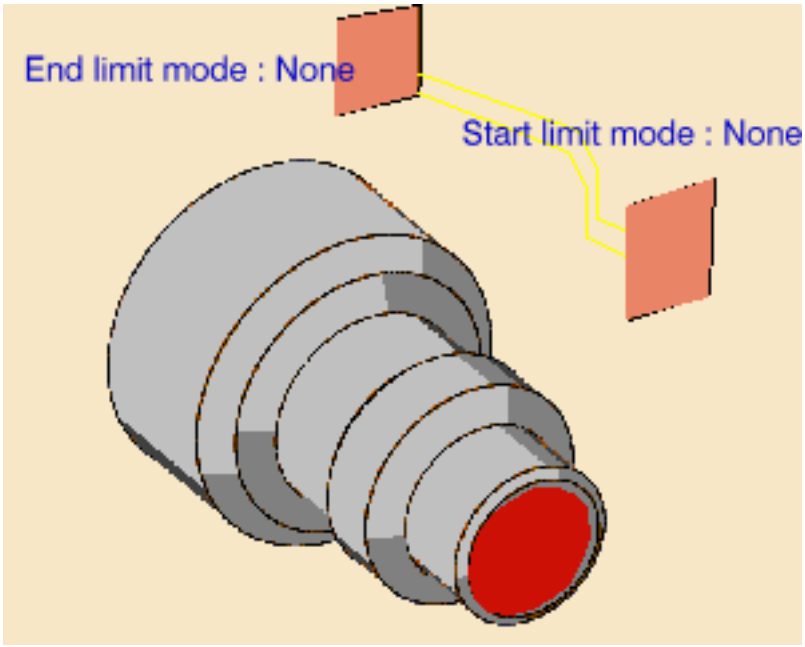
1. Select the Profile Finish Turning icon.

   The Profile Finish Turning dialog box appears directly at the Geometry tab page.

   This page includes a sensitive icon to help you specify the geometry to be machined.

   The part in the icon is colored red indicating that this geometry is required for defining the operation.

2. Click the red part in the icon, then select the desired part profile in the 3D window.



In addition to the global offsets that you can assign to the selected profile, you can also add local values.

Right click the geometry to be assigned the local value, and select the **Add Local Information** contextual command. A dialog box appears allowing you to assign the desired local values.

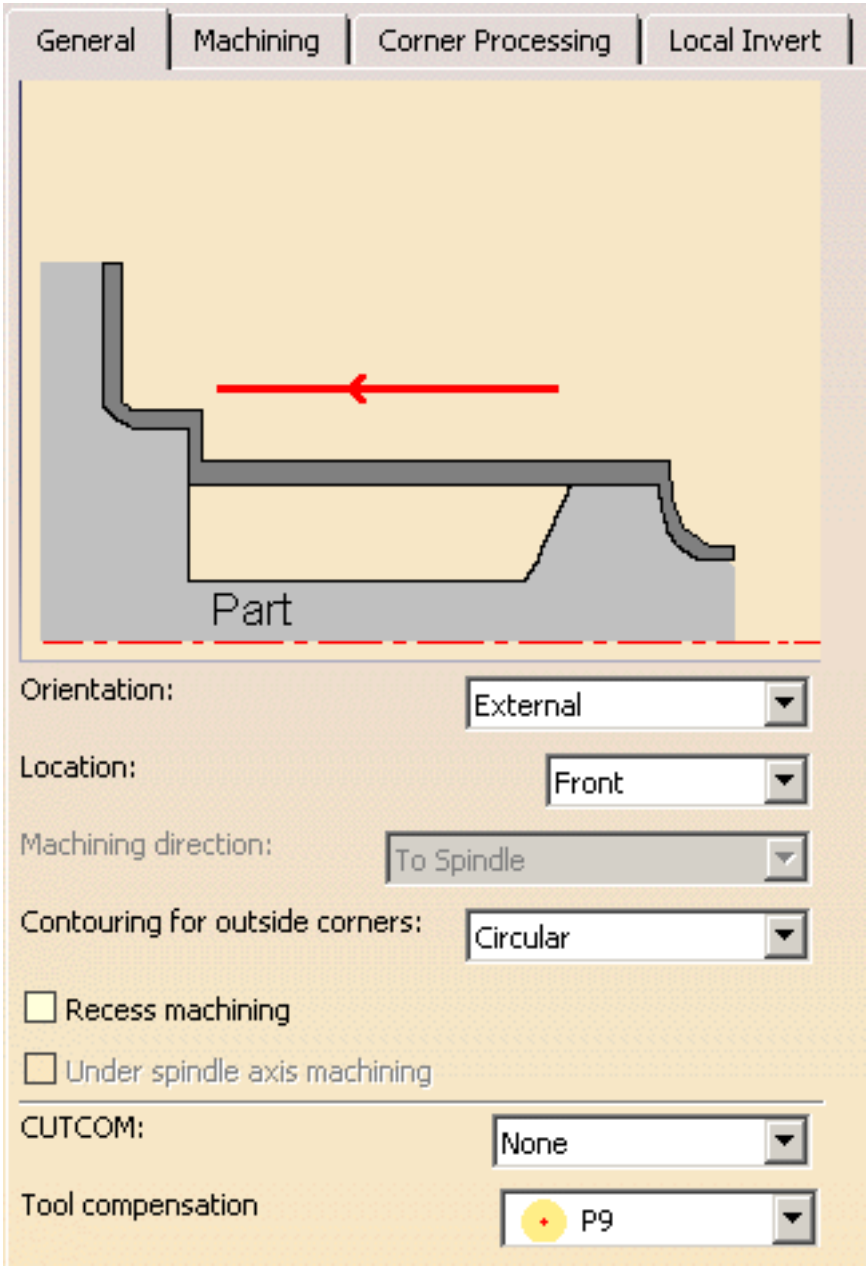
Other contextual commands are available for analyzing and resetting local information. Please refer to Local Information for more details.

The part of the icon is now colored green indicating that this geometry is now defined.

4. Select the Strategy tab page  to specify the general machining strategy parameters:
   - Orientation: **External**
   - Location: **Center**
   - Select the **Recess machining** checkbox
   - Machining direction is set automatically **To spindle.**

   Note that you can locally invert machining directions using Local Information facilities.

   Other optional parameters can be set in the Machining, Corner Processing, and Local Invert tabs.



A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use. This is described in Edit the Tool of a Lathe Operation.

5. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation. See Feeds and Speeds for Profile Finishing for more information.

   In addition to the global feedrates that you can assign for the operation, you can also add local feedrates to portions of the profile.

   Right click the geometry to be assigned the local value, and select the **Add Local Information** contextual command. A dialog box appears allowing you to assign the desired local values.

   Other contextual commands are available for analyzing and resetting local information. Please refer to Local Information for more details.

You can add approach and retract motions to the operation in the Macros tab page  . See Define Macros on a Lathe Operation for an example.

6. Check the validity of the operation by replaying the tool path.
7. Click OK to create the operation.

Create a Groove Finish Turning Operation

This task shows how to insert a Groove Finish Turning operation in the program. To create the operation you must define:

- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the Lathe01.CATPart document, then select **Machining > Lathe Machining** from the Start menu. Make the Manufacturing Program current in the specification tree.

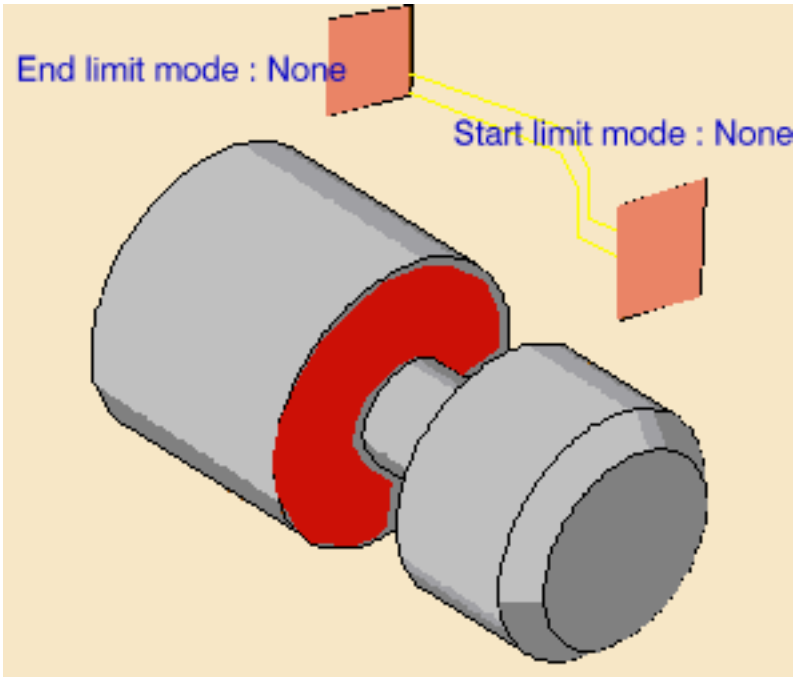
1. Select the Lathe Groove Finish Turning icon.

   A Groove Finish Turning entity along with a default tool is added to the program.

   The Groove Finish Turning dialog box appears directly at the Geometry tab page.

   This tab page includes a sensitive icon to help you specify the geometry to be machined.

   The part in the icon is colored red indicating that this geometry is required.



2. Click the red part in the icon then select the desired part profile in the 3D window.

   In addition to the global offsets that you can assign to the selected profile, you can also add local values.

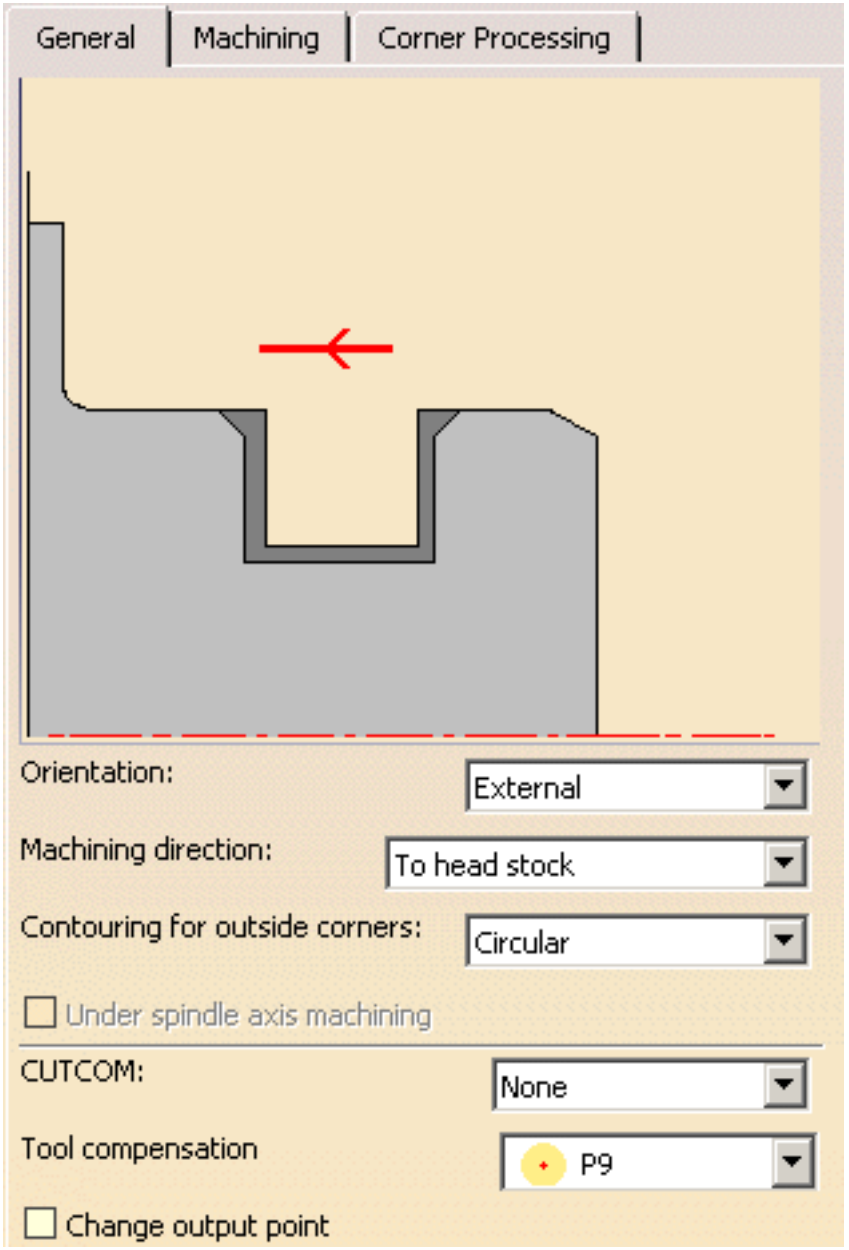
   Right click the geometry to be assigned the local value, and select the **Add Local Information** contextual command. A dialog box appears allowing you to assign the desired local values.

   Other contextual commands are available for analyzing and resetting local information. Please refer to Local Information for more details.

3. Select the Strategy tab page 

to specify the main machining strategy parameters:

- Orientation: **External**
- Machining direction: **To head stock**
- Contouring for outside corners: **Circular**.



Other optional parameters can be set in the Machining and Corner Processing tabs.

A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use. This is described in Edit the Tool of a Lathe Operation.

4. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation. See Feeds and Speeds for Finish Grooving for more information.

In addition to the global feedrates that you can assign for the operation, you can also add local feedrates to portions of the profile.

Right click the geometry to be assigned the local value, and select the **Add Local Information** contextual command. A dialog box appears allowing you to assign the desired local values.

Other contextual commands are available for analyzing and resetting local information. Please refer to Local Information for more details.

5. Select the Macros tab page  to specify the operation's transition paths (approach and retract motion, for example). See Define Macros on a Lathe Operation for an example.

Before accepting the operation, you should check its validity by replaying the tool path.

6. Click OK to create the operation.

Create a Ramp Rough Turning Operation

This task illustrates how to create a Ramp Rough Turning operation in the program. This type of operation is suitable for machining hard materials using round ceramic inserts, thereby minimizing wear and cutting stress.

To create the operation you must define:

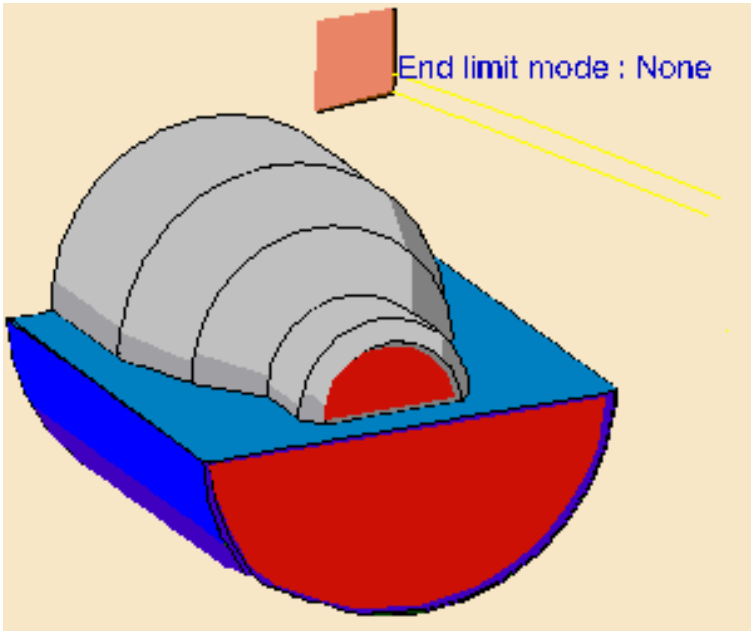
- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the Lathe01.CATPart document, then select **Machining > Lathe Machining** from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Ramp Rough Turning icon.

   A Ramp Rough Turning entity along with a default tool is added to the program.

   The Ramp Rough Turning dialog box appears directly at the Geometry tab page.
   This tab page includes a sensitive icon to help you specify the geometry to be machined.



The Part and Stock areas of the icon are colored red indicating that this geometry is required. All other geometry is optional.

2. Click the red Part area in the icon then select the desired part profile in the 3D window.

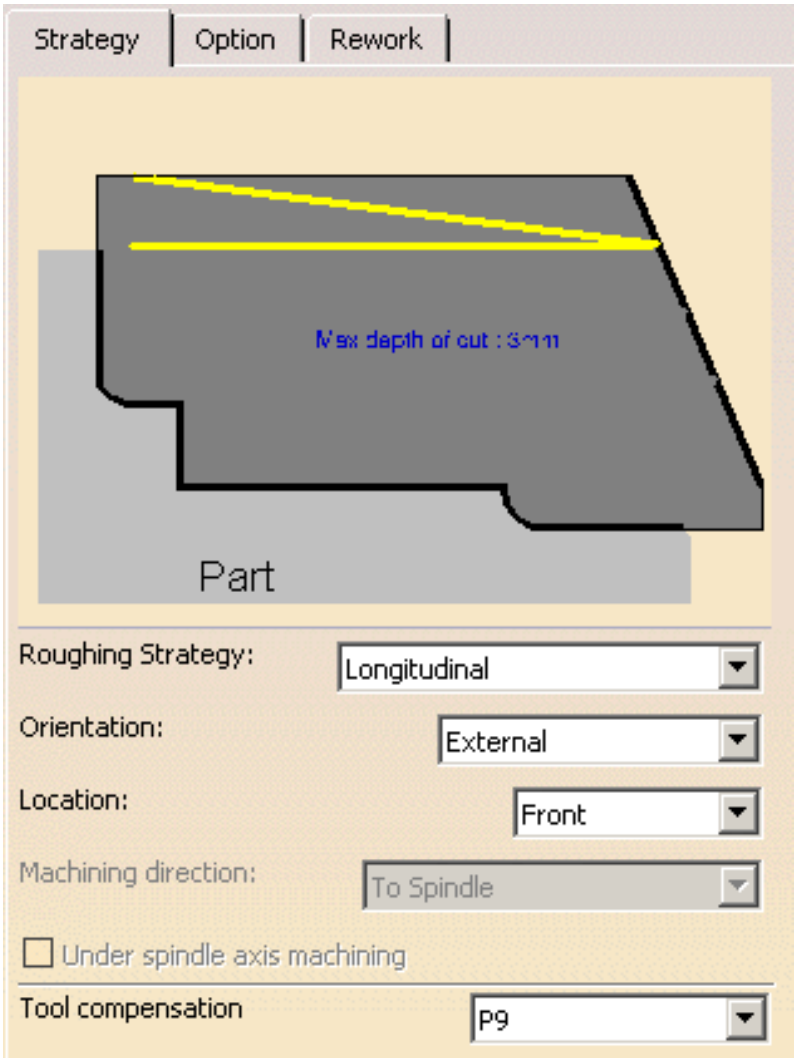
The Edge Selection toolbar appears to help you with contour selection.

3. Click the red Stock in the icon then select the desired stock profile in the 3D window.

The Part and Stock areas of the icon is now colored green indicating that this geometry is now defined.

4. Set **Part Offset** to 5mm.

5. Select the Strategy tab page  to specify the machining strategy parameters as shown below.



6. Double click **Max depth of cut** in the icon. Set this value to 15mm in the Edit Parameter dialog box and click OK.

Other parameters can be set in the Options and Rework tabs.

7. A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

Please refer to Edit the Tool of a Lathe Operation.

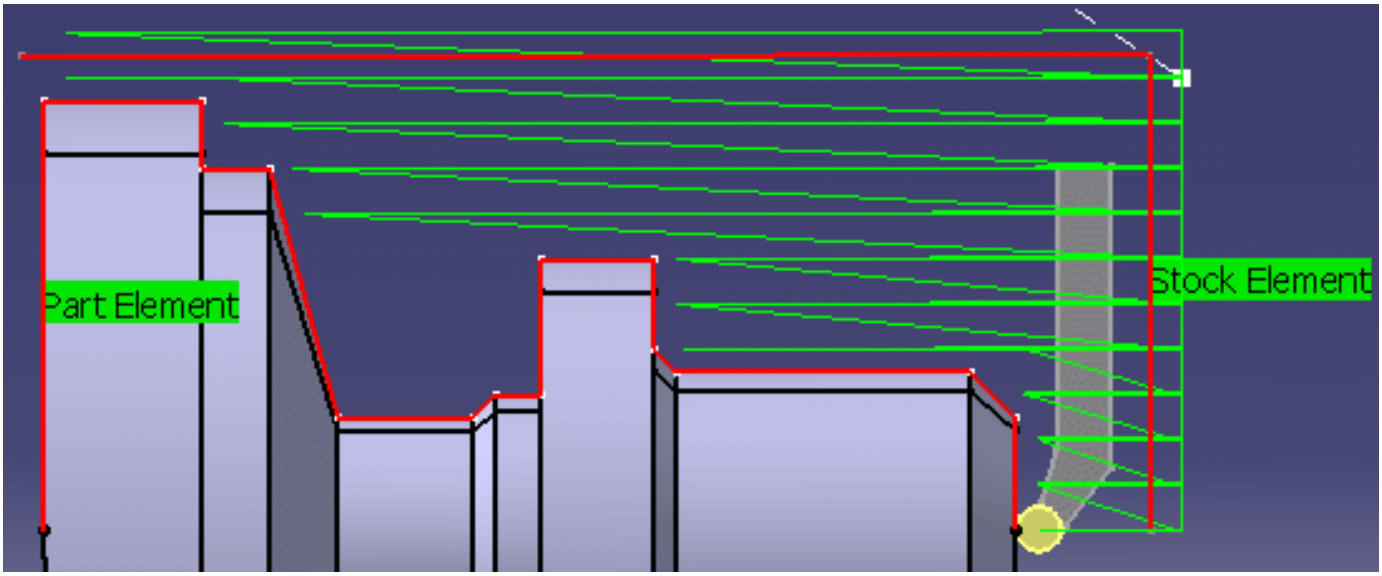
8. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

See Feeds and Speeds for Ramp Rough Turning for more information.

9. Select the Macros tab page  to specify the operation's transition paths (approach and retract motion, for example).

See Define Macros on a Lathe Operation for more information.

Before accepting the operation, you should check its validity by replaying the tool path.



10. Click OK to create the operation.

Create a Ramp Recess Turning Operation

This task shows how to insert a Ramp Recess Turning operation in the program. This type of operation is suitable for machining hard materials using round ceramic inserts, thereby minimizing wear and cutting stress.

To create the operation you must define:

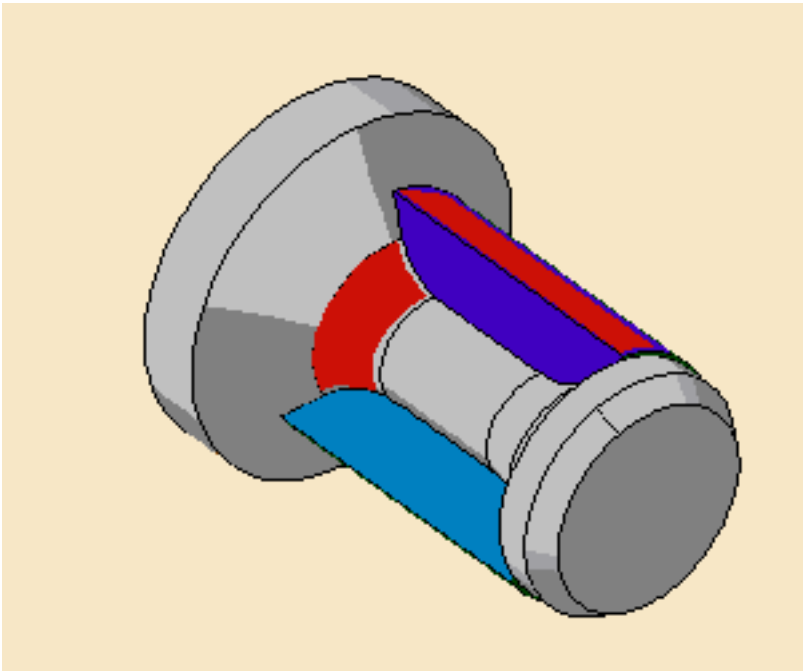
- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the Lathe01.CATPart document, then select **Machining > Lathe Machining** from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Ramp Recess Turning icon.

A Ramp Recess Turning entity along with a default tool is added to the program.

The Ramp Recess Turning dialog box appears directly at the Geometry tab page. This tab page includes a sensitive icon to help you specify the geometry to be machined.



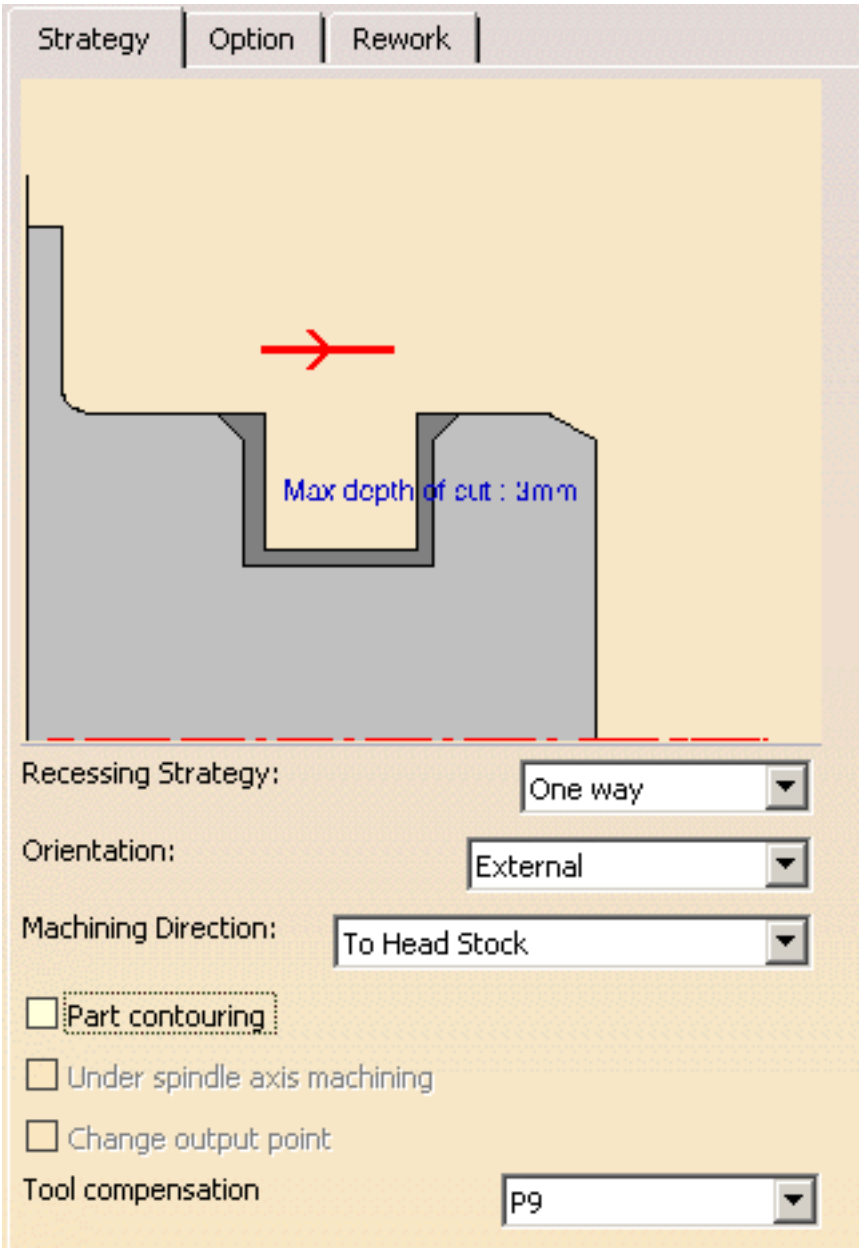
The Part and Stock areas in the icon are colored red indicating that this geometry is required.

2. Click the red Part area in the icon then select the desired part profile in the 3D window.

   Select the stock in the same way.

   The Part and Stock areas in the icon are now colored green indicating that the required geometry is selected.

3. Select the Strategy tab page and specify the main machining strategy parameters as shown below.



4. Double click **Max depth of cut** in the icon. Set this value to 10mm in the Edit Parameter dialog box and click OK.

   Other parameters can be set in the Options and Rework tabs.

A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

This is described in Edit the Tool of a Lathe Operation.

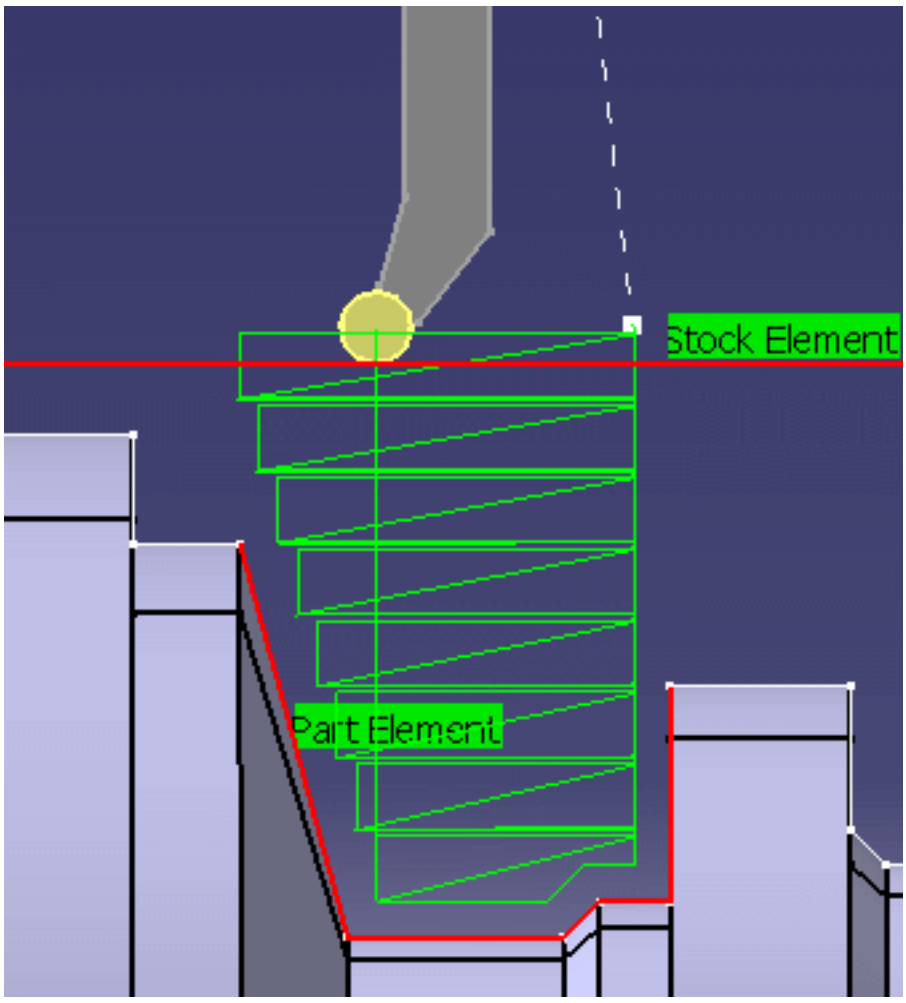
5. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

See Feeds and Speeds for Ramp Recess Turning for more information.

6. If you want to specify approach and retract motion for the operation, select the Macros tab page  to specify the desired transition paths.

See Define Macros on a Lathe Operation for more information.

Before accepting the operation, you should check its validity by replaying the tool path.



7. Click OK to create the operation.

Create a Thread Turning Operation

This task shows how to insert a Thread Turning operation in the program. To create the operation you must define:

- the geometry to be machined
- the tool that will be used
- the parameters of the machining strategy
- the feedrates and spindle speeds
- the macros (transition paths).

Open the Lathe01.CATPart document, then select **Machining > Lathe Machining** from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Thread Turning icon .

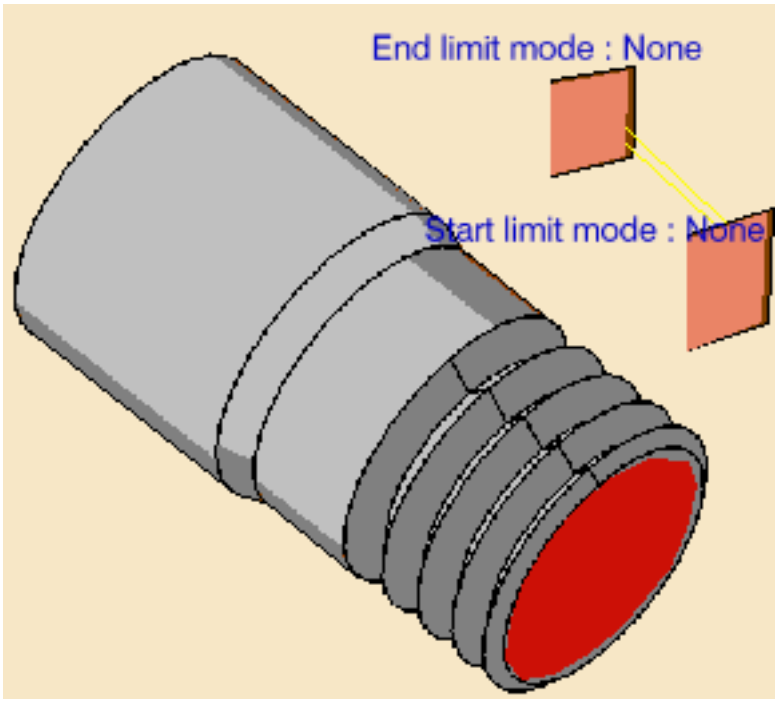
   A Thread Turning entity along with a default tool is added to the program.

   The Thread Turning dialog box appears directly at the Geometry tab page .

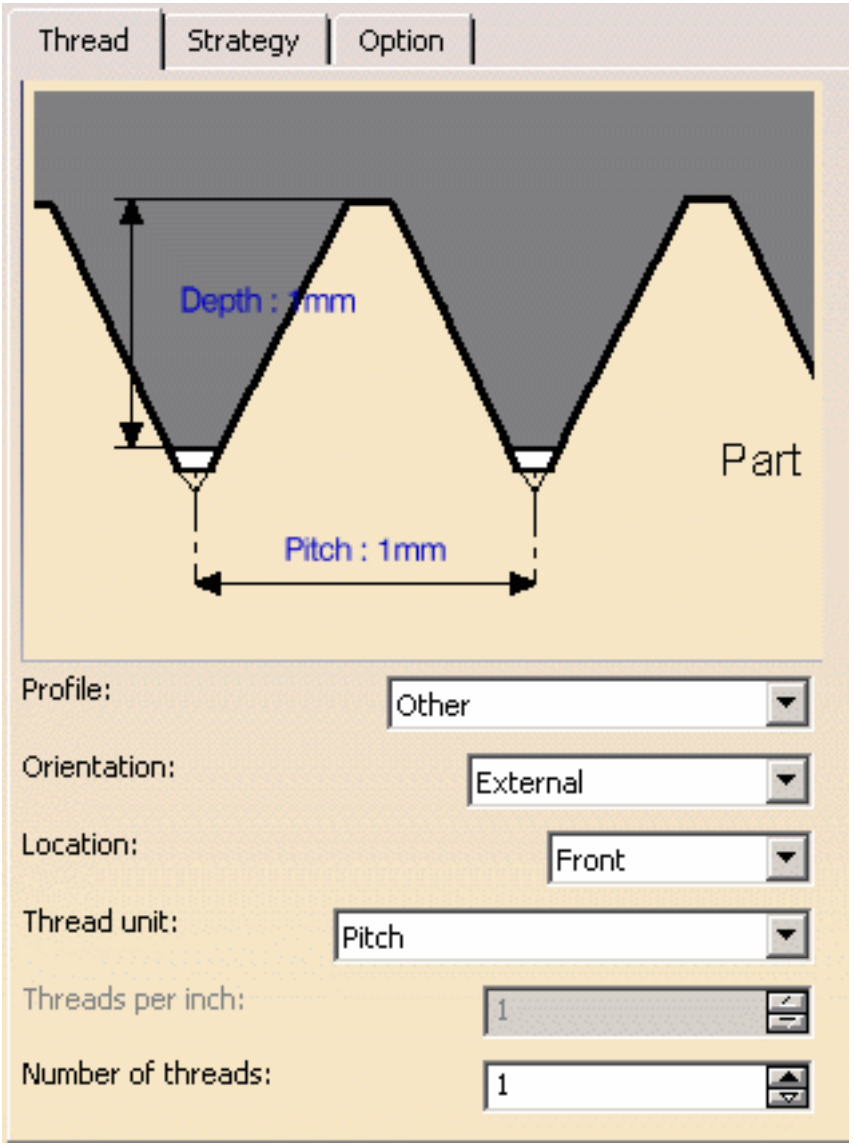
   This tab page includes a sensitive icon to help you specify the geometry to be machined.

   The part in the icon is colored red indicating that this geometry is required.

2. Click the red part in the icon then select the desired part profile in the 3D window.
3. Specify the desired length of threading.



4. Select the Strategy tab page  to specify the main machining parameters that are organized in three tabs: Thread, Strategy and Options.



Set the following values in the Thread tab:

- Profile: **Other**
- Orientation: **External**
- Location: **Front**
- Thread unit: **Pitch**
- Number of threads: **1**
- Thread depth: **10mm**
- Thread pitch: **10mm**.

Other optional parameters can be set in the Strategy and Options tabs.

5. If you want to generate CYCLE statements, you must select the **Output CYCLE syntax** checkbox in the Options tab and set the Syntax Used option to Yes in the NC Output generation dialog box. Otherwise, GOTO statements will be generated.

You can display and edit CYCLE syntaxes by clicking the **Edit Cycle** command.

6. A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use.

This is described in Edit the Tool of a Lathe Operation.

7. Select the Feeds and Speeds tab page  to specify the machining spindle speed for threading.

Feedrates in units per minute are available for air cutting such as macro motions and path transitions.
Note that RAPID feedrate can be replaced by Air Cutting feedrate in tool trajectories (except in macros) by selecting the corresponding checkbox.

8. Select the Macros tab page  to specify the operation's transition paths (approach and retract motion, for example).

See Define Macros on a Lathe Operation for an example.

9. Before accepting the operation, you should check its validity by replaying the tool path.
10. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Thread Turning operations:

```
CYCLE/THREAD,%MFG_THREAD_PITCH
```

A typical NC data output is as follows:

```
CYCLE/THREAD, 10.000000
```

The parameters available for PP word syntaxes for this type of operation are described in the NC_LATHE_THREADING section of the Manufacturing Infrastructure User's Guide.

Create a Sequential Turning Operation

This task illustrates how to create a Sequential Turning operation in the program.

To create the operation you must define:

- the list of elementary motions making up the operation 
- the tool that will be used 
- the feedrates and spindle speeds 
- the macros (transition paths)  .

Open the Lathe01.CATPart document, then select **Machining > Lathe Machining** from the Start menu. Make the Manufacturing Program current in the specification tree.

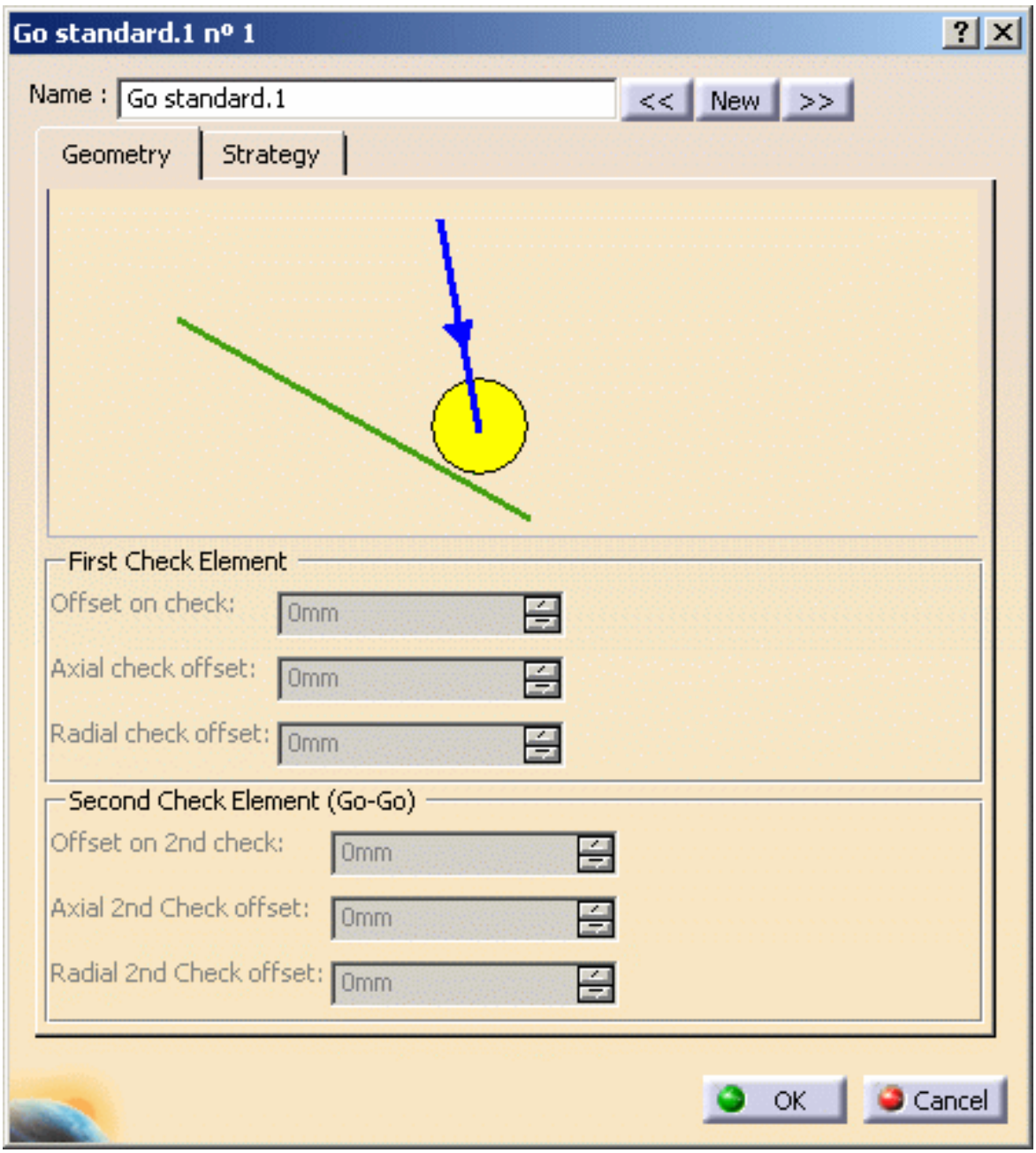
1. Select the Sequential Turning icon  . A Sequential Turning entity along with a default tool is added to the program. The dialog box appears directly at the List of Motions tab page  .

   The first motion must be a Go motion to a point.

2. Select the Go icon  . The dialog box for defining the first motion appears.

   In the Geometry tab:

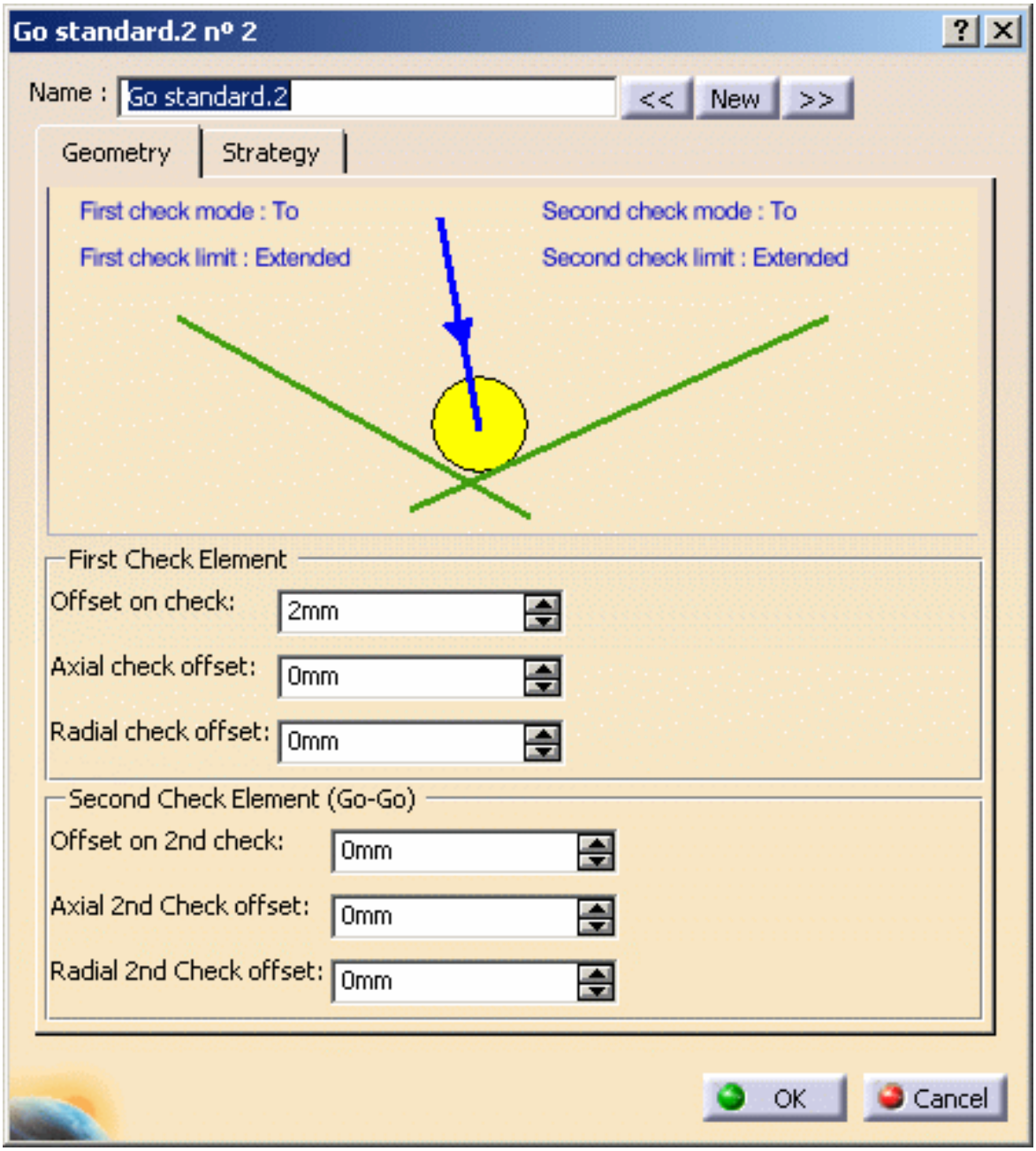
   - Select a check representation in the dialog box then the desired check element in the 3D view.
   - Click OK to define the motion.



3. Select the Go icon again to define the second motion.

In the Geometry tab:

- Successively select the two check representations in the dialog box then the desired check elements in the 3D view.
- Set the first and second check modes to **To**.
- Set a offset on the first check element to 2mm.
- Click OK to define the motion.



4. Select the Go icon again to define the next motion.

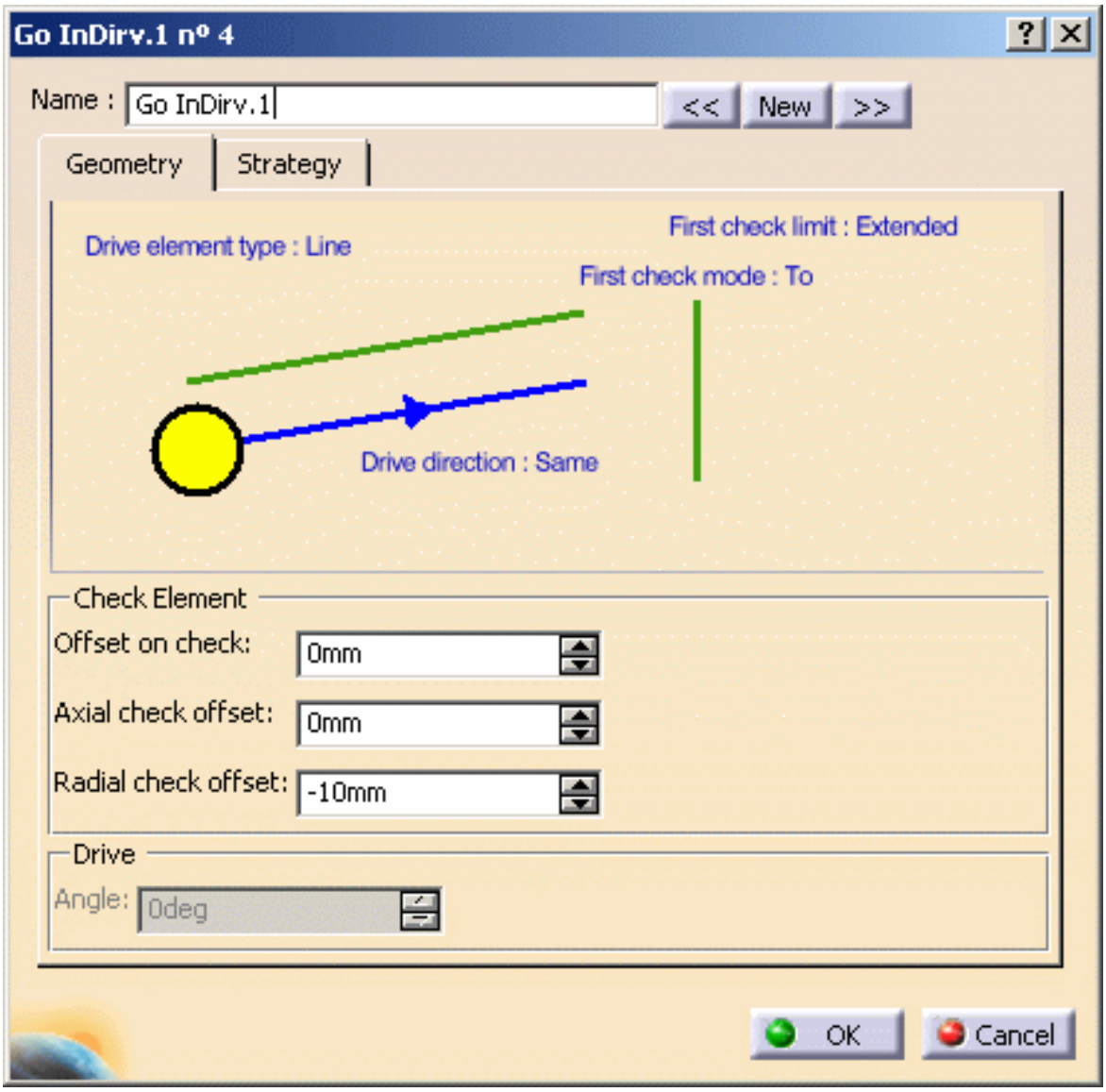
In the Geometry tab:

- Select a check representation in the dialog box then the desired check element in the 3D view.
- Set the check mode to **Past**.
- Set a offset on the check element to 5mm.
- Click OK to define the motion.

5. Select the Go InDirv icon .

In the Geometry tab:

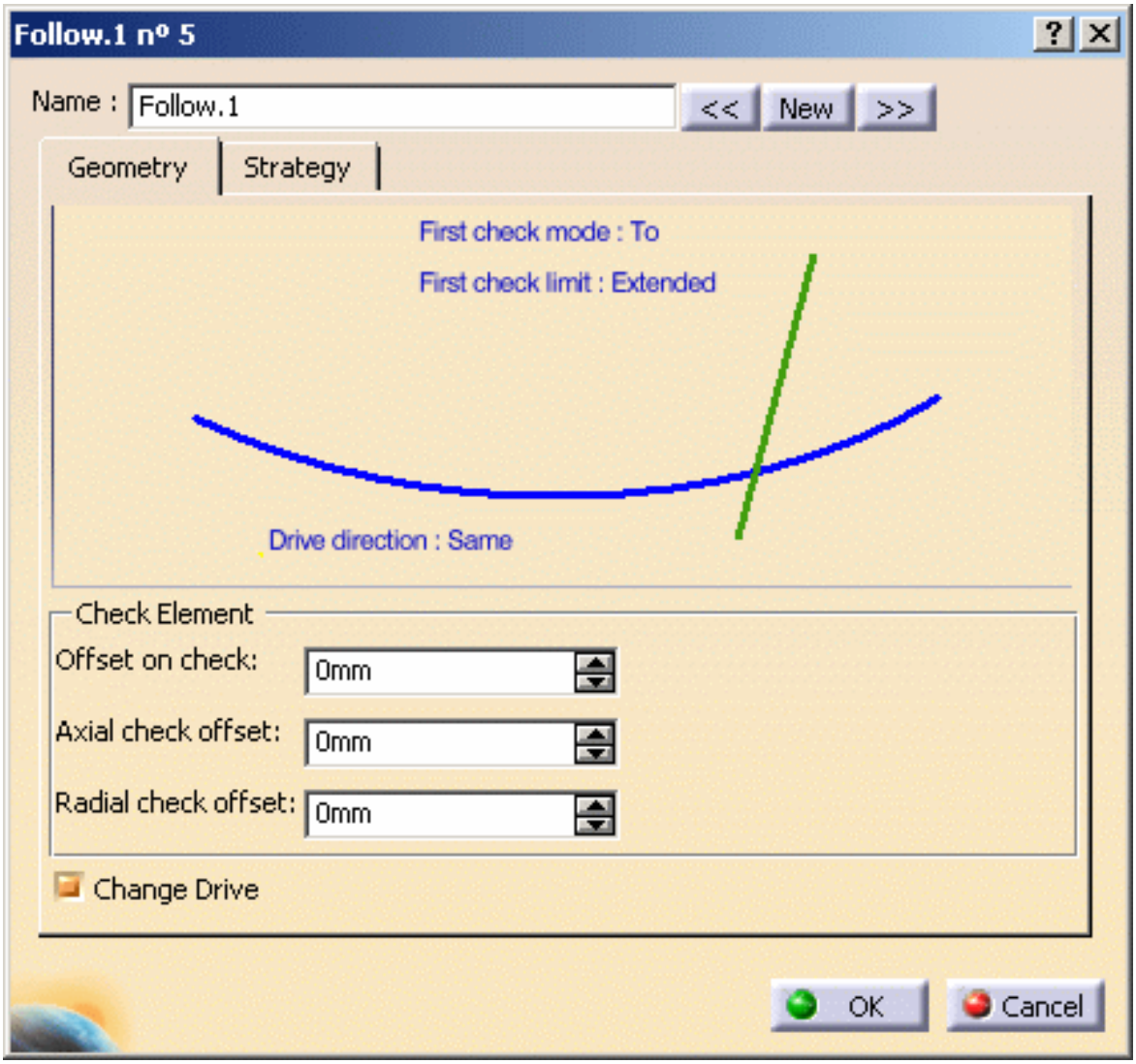
- Select the drive direction in the dialog box then the desired linear element in the 3D view.
- Select a check representation in the dialog box then the desired check element in the 3D view.
- Set the check mode to **To**.
- Click OK to define the motion.



6. Select the Follow icon . The check curve of the previous motion is used as drive curve. This drive element is highlighted in the 3D view.

In the Geometry tab:

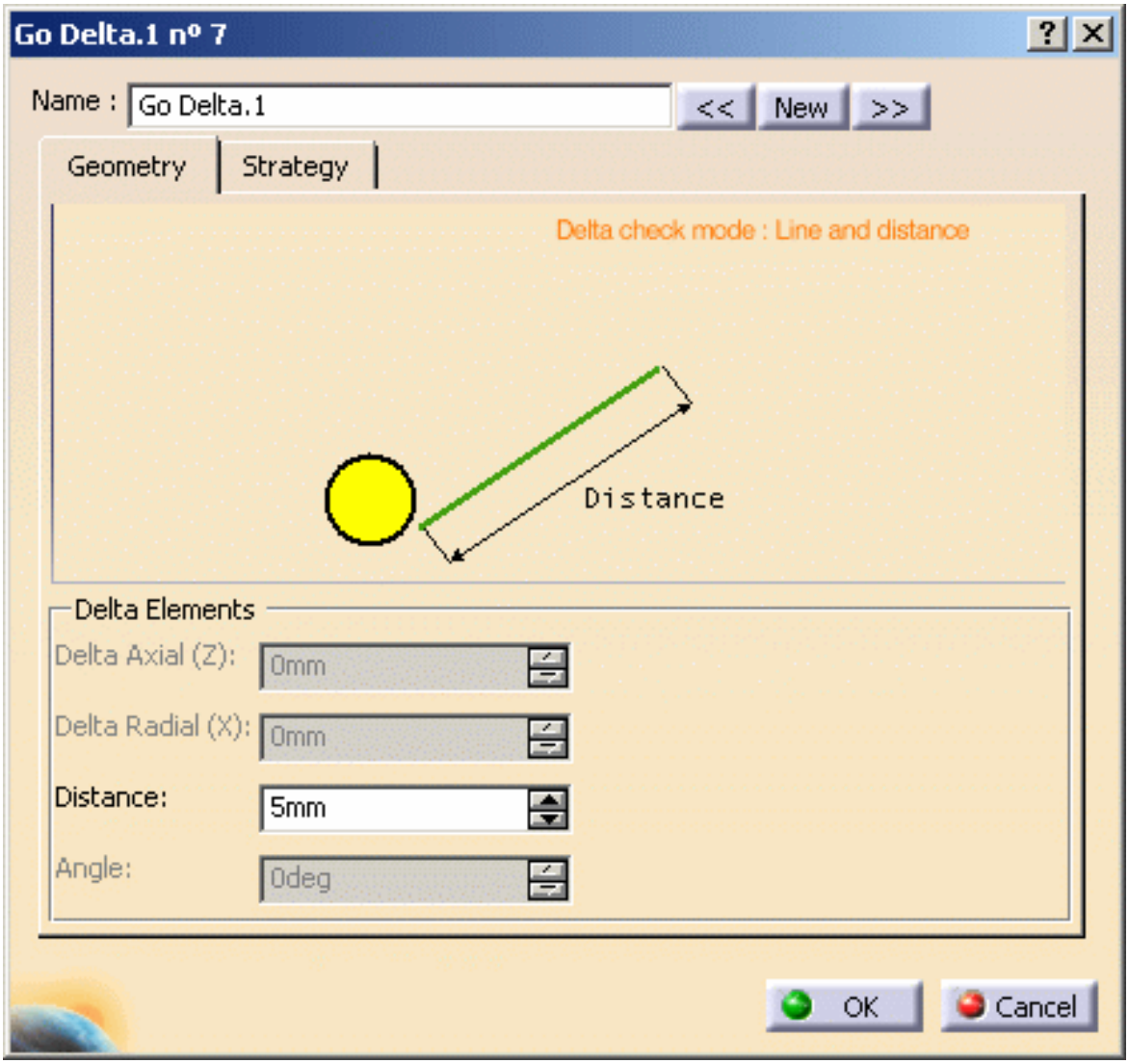
- Select a check representation in the dialog box then the desired check element in the 3D view.
- Set the check mode to **To**.
- Click OK to define the motion.



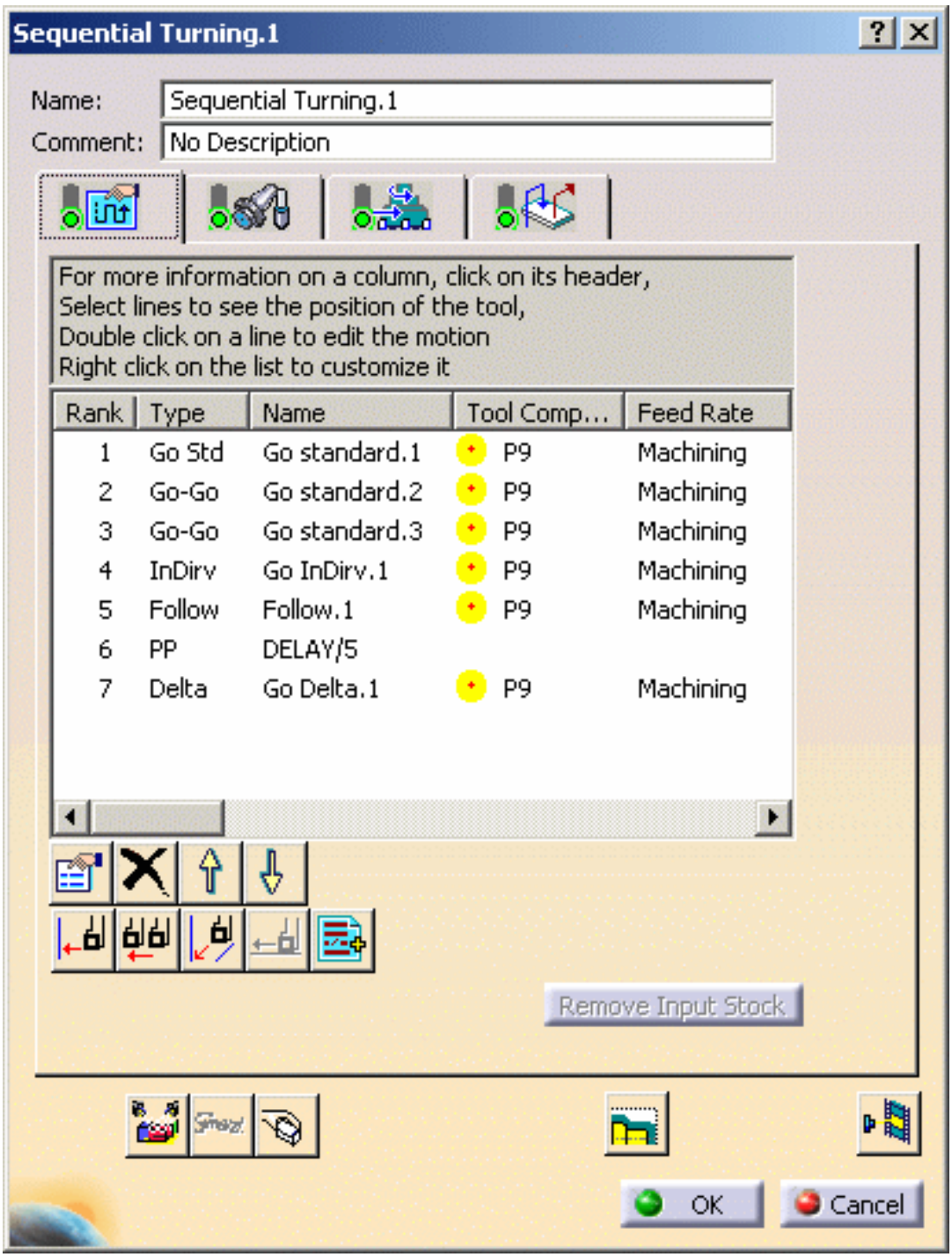
7. Select the PP word icon  , then specify a PP word in the dialog box that appears (DELAY/5, for example).
8. Select the Go Delta icon  .

In the Geometry tab:

- Set the check mode to **Line and distance**.
- Select the line representation in the dialog box then the desired linear element in the 3D view.
- Enter a distance.
- Click OK to define the motion.



Other optional parameters can be set in the Strategy tab page for each of the motions.

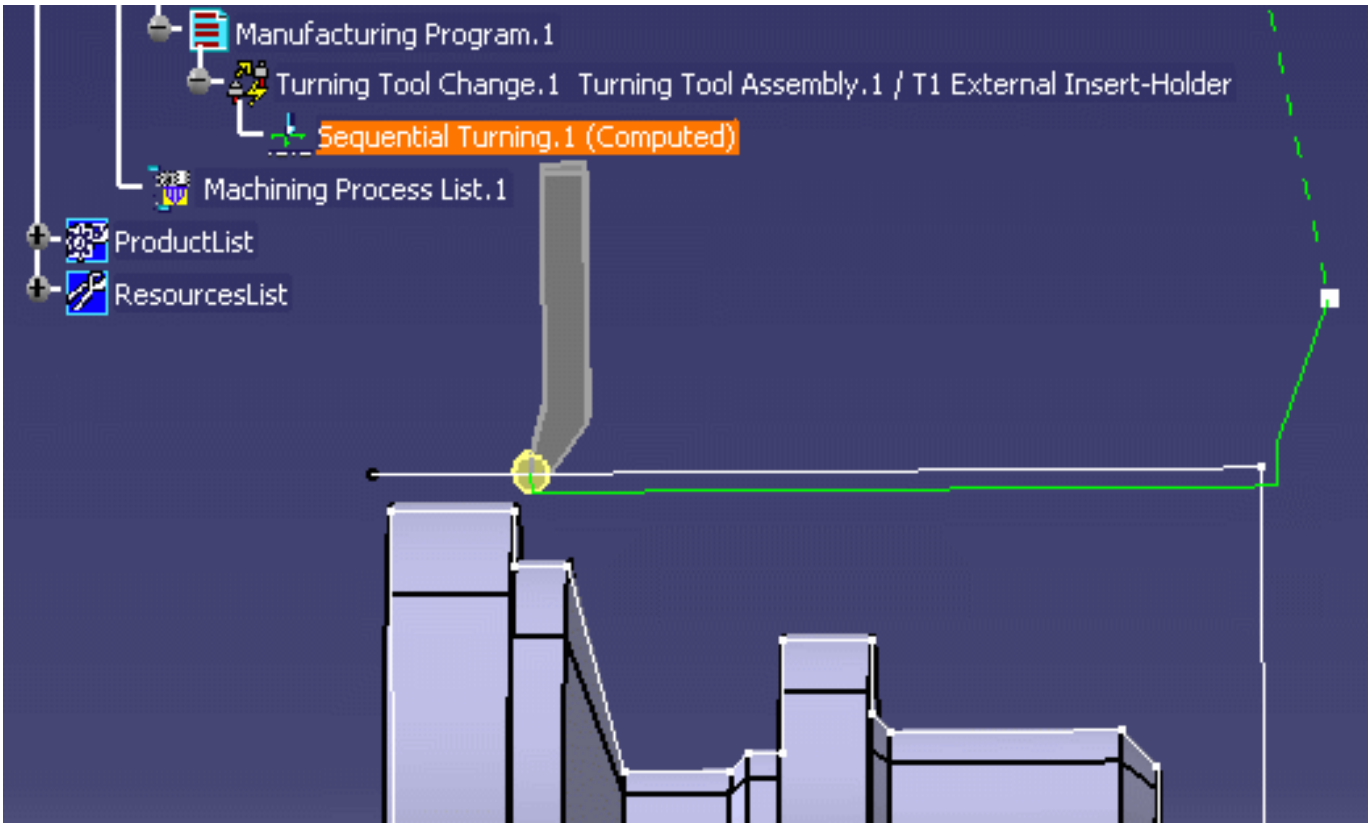


A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page to specify the tool you want to use. Please refer to Edit the Tool of a Lathe Operation.

9. Select the Feeds and Speeds tab page to specify the feedrates and spindle speed for the operation. See Feeds and Speeds for Sequential Turning for more information.

10. Select the Macros tab page to specify the operation's transition paths (approach and retract motion, for example). See Define Macros on a Lathe Operation for an example.

Before accepting the operation, you should check its validity by replaying the tool path.



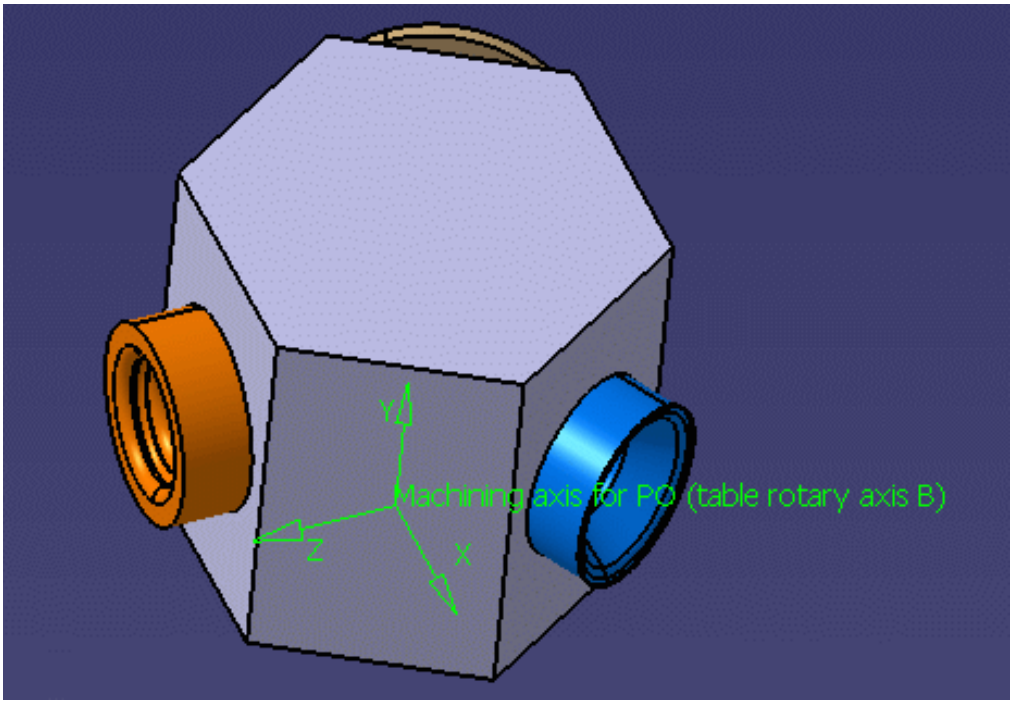
11. Click OK to create the operation.

Turning on a Milling Center with Facing Head

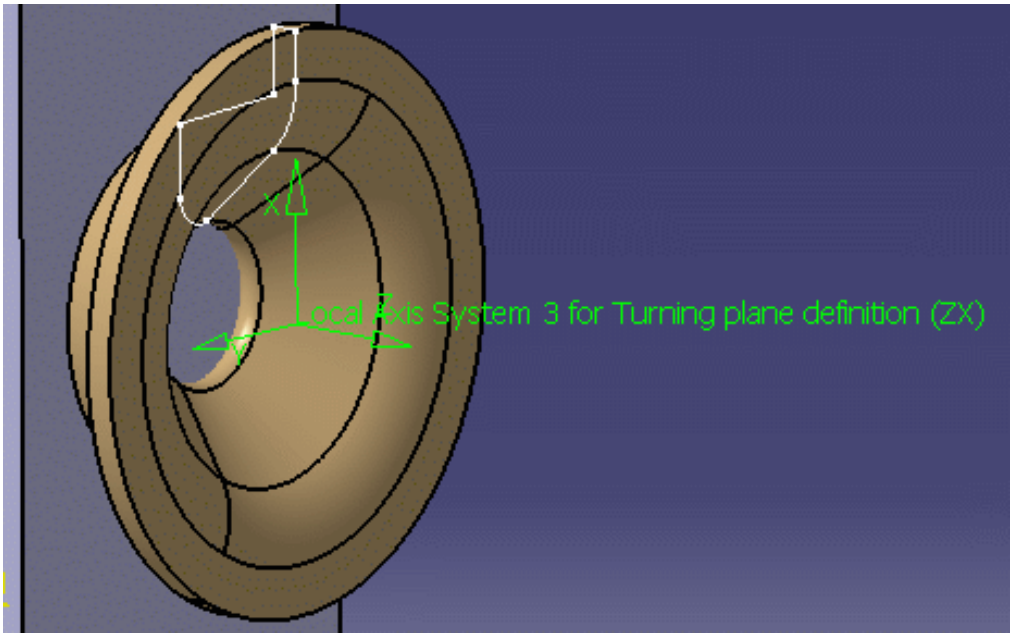
This task illustrates how to create turning operations to machine large diameter holes on a milling center machine equipped with a facing head and rotary table.

The facing head is a mechanism mounted on the head of the milling center to support lathe tools. This configuration can be used to machine large diameter holes with better quality than milling.

In this scenario, the 3 large holes to be machined are located in different turning planes around the table rotary axis (C).

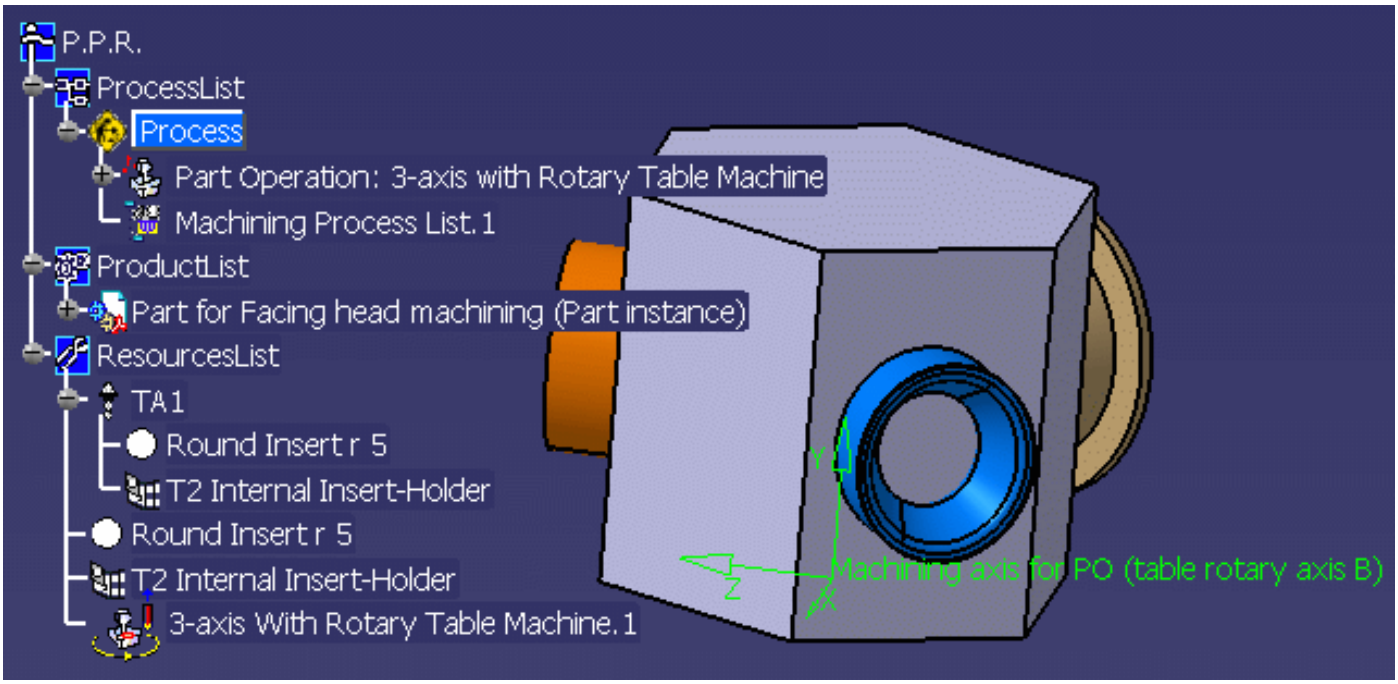


Each turning plane is defined by two axes (ZX) of a local machining axis system, so that each hole can be reached by rotating the table.

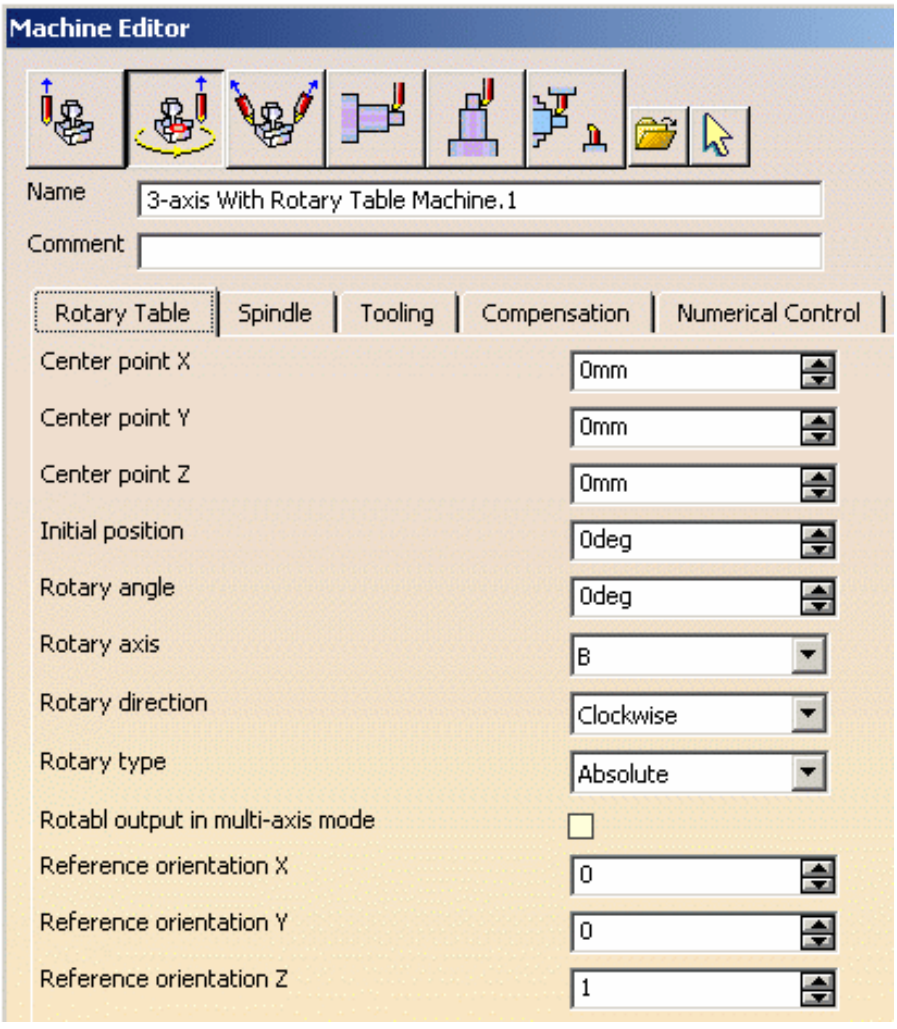


The tool mounted on the facing head is piloted along the U-axis to machine along the profile of the hole.

Open the TurningOperationOnMillMachine.CATProcess document, then select **Machining** > **Lathe Machining** from the Start menu.



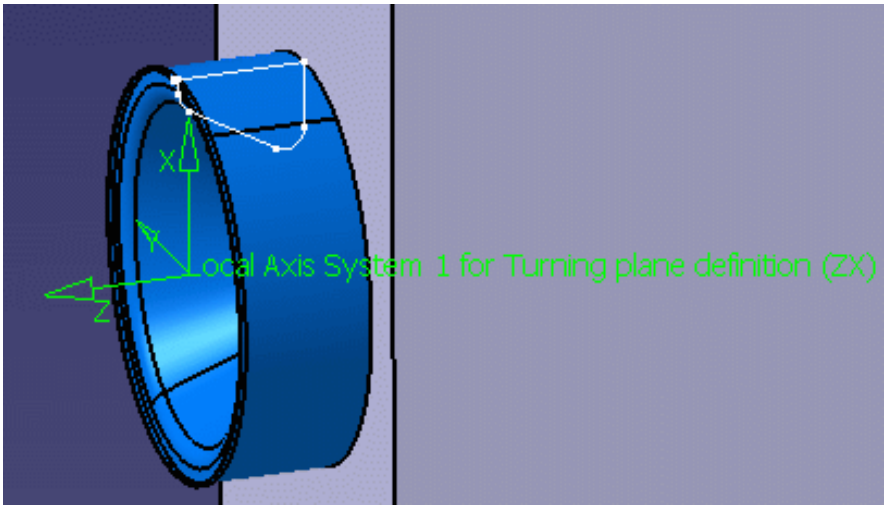
1. Double click the Part Operation and access the Machine editor. A 3-axis rotary table machine is already selected. Make the following settings in the Rotary Table tab.



2. Select the Machining Axis Change icon to create a local machining axis for the first hole.

Position the origin of the axis by selecting the outer circle of the hole. The axis origin will be positioned at the hole center.

Set the orientation of the X-axis (radial) and Z-axis (axial) by using one of the proposed methods (Selection or Manual Components, for example).
See Insert Machining Axis Change for more information about these methods.

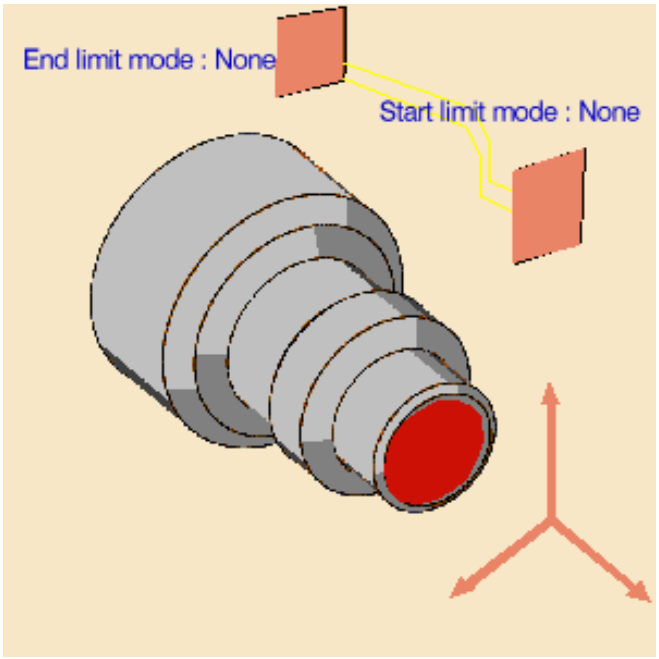


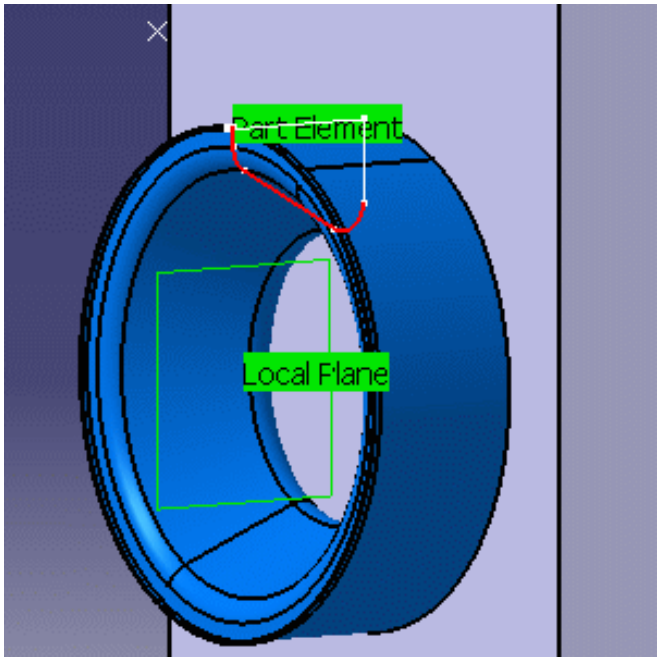
3. Select the Profile Finish Turning icon .

Use the Geometry tab to:

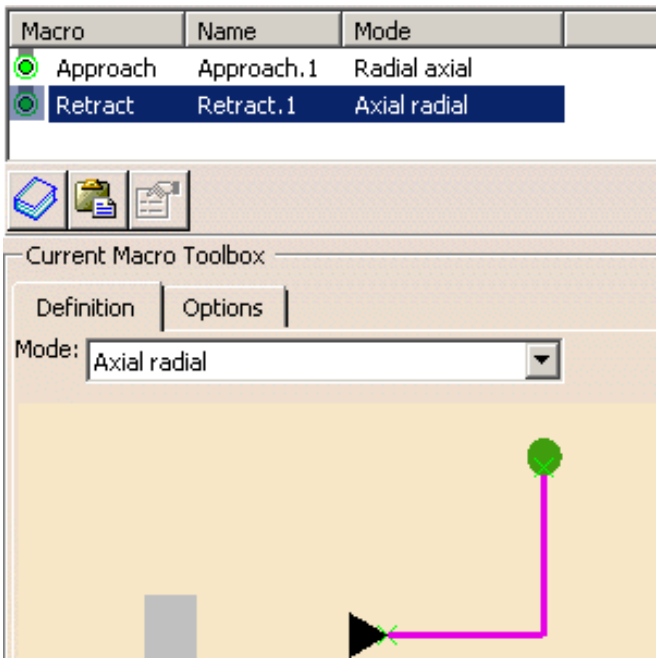
- select the profile to machine making use of the sensitive Part Element symbol.
- specify the local plane making use of the sensitive axis system symbol.
  Select the local machining axis you created previously. The axial and radial axes (ZX) define the turning plane.

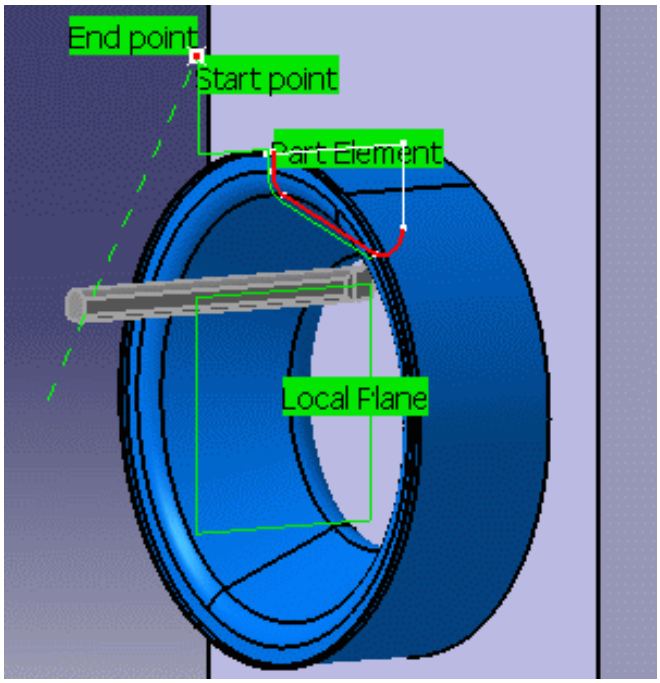
Use the Macro tab to define an approach macro by selecting a start point and a retract macro by selecting an end point.



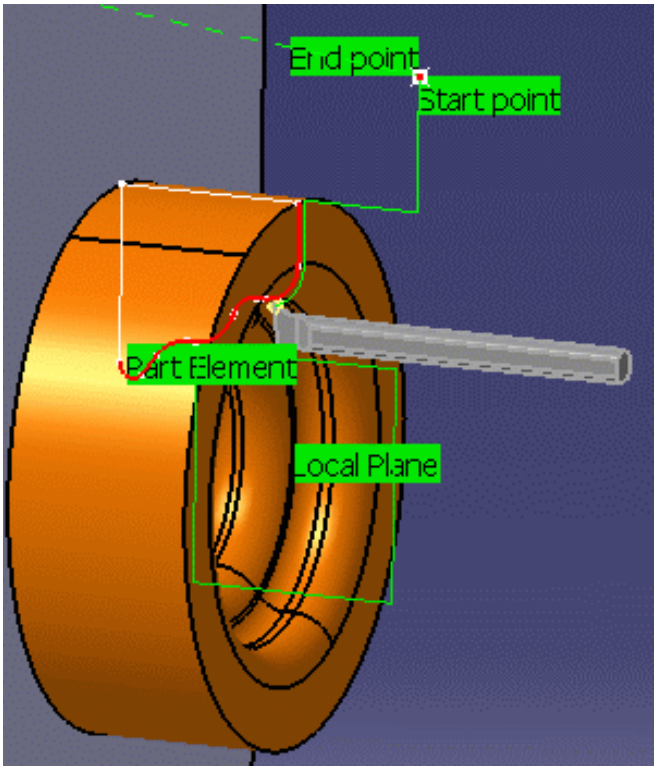


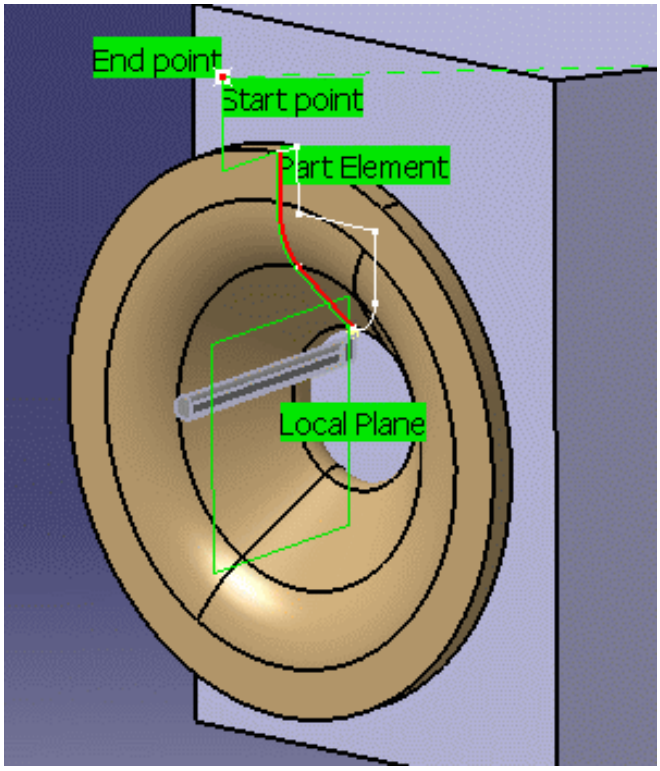
Use the Macro tab to define an approach macro by selecting a start point and a retract macro by selecting an end point.
Check the tool path by means of a replay.



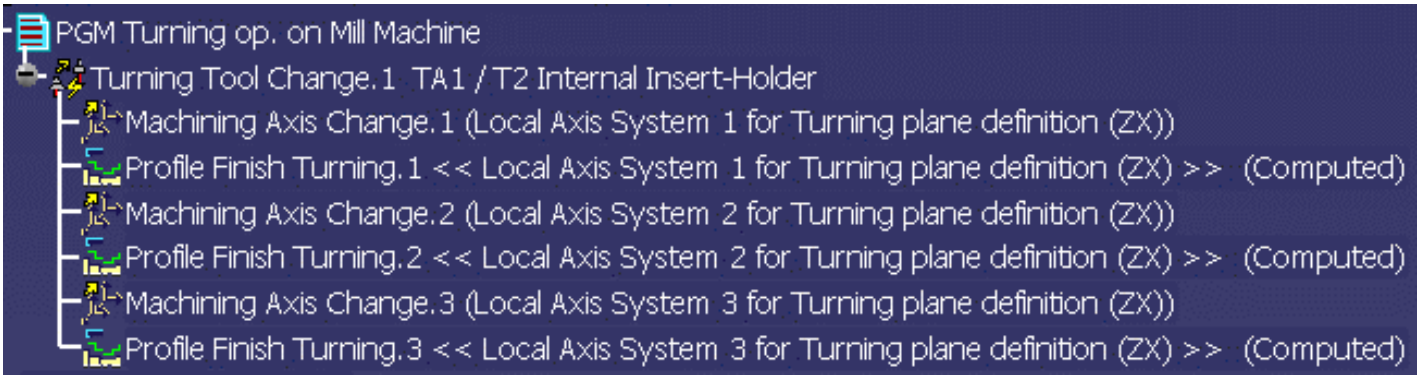


4. Repeat the steps above to create a local machining axis and a Profile Finish Turning operation for the other two holes.





5. When the 3 turning operations are computed your program should look something like this:



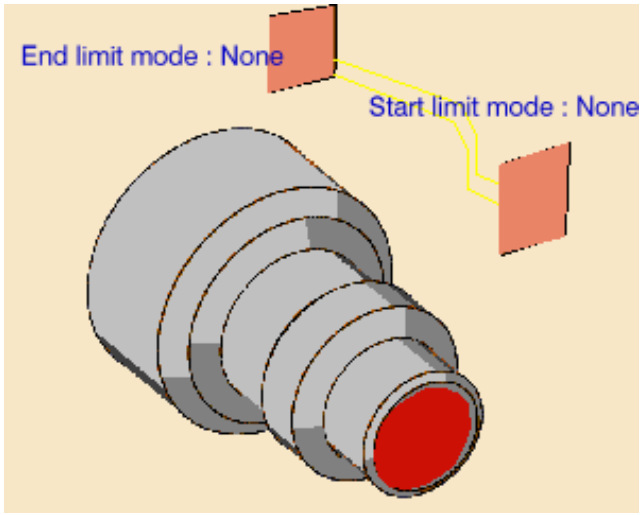
Please note the following points. They are dealt with in more detail in the Methodology section.

- At each change of turning plane, you must define:
  - Machining Axis System Change: this is needed for NC data generation
  - Local Machining Axis on turning operation: this is needed for processing geometry and visualizing the tool assembly.
- NC data output can be generated in XYZ or XYZIJK format.
  For XYZ data, you must generate table rotations in your program.
  For XYZIJK data, the value that is output for IJK is taken on the Z axis (spindle) of the local machining axis system.
- The following NC data statements will be generated at the start of each Turning operation:
  - The coordinates of the origin of the Local Machining Axis
  - An order to switch from the X-axis to the U-axis.
    This is done through parameterized syntaxes in the PP table (NC_SPINDLE_LATHE or NC_LATHE_MO_START_COMMENT).

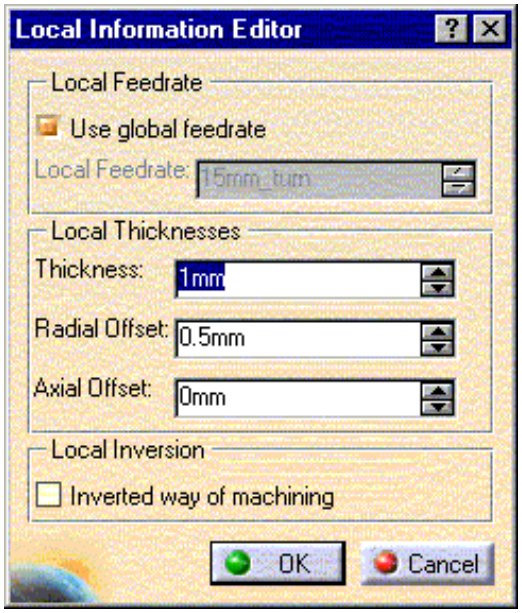
Manage Local Information on Finishing Operations

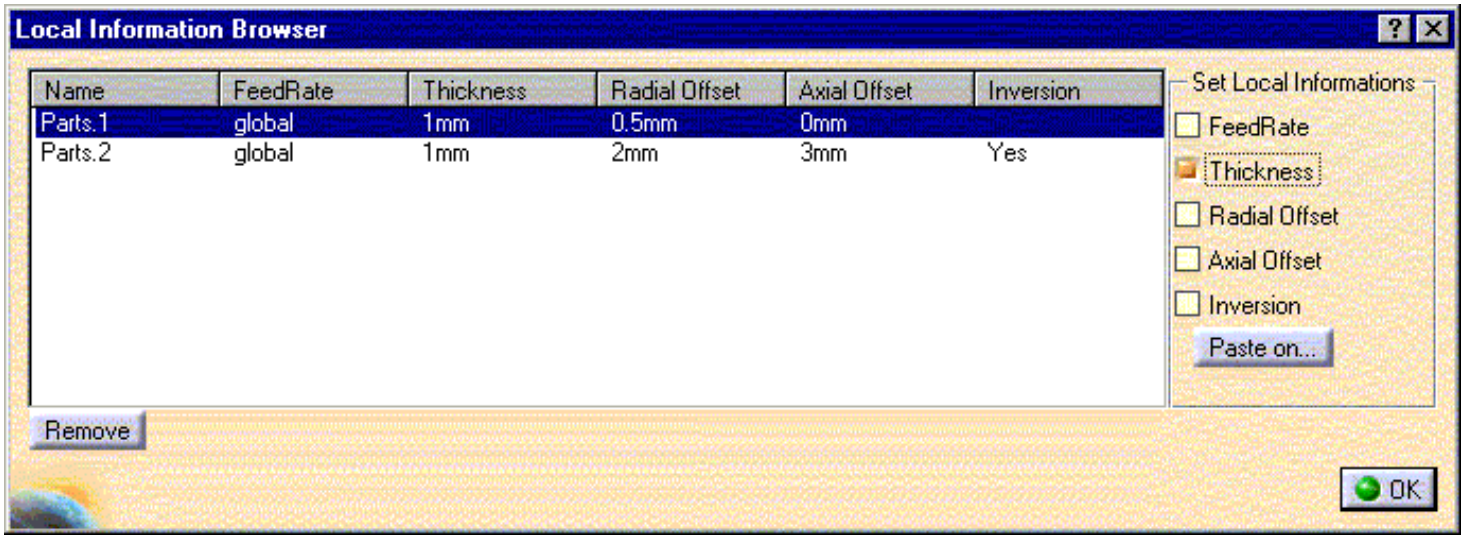
This task illustrates how to manage Local Information (local feedrates and offsets) on a Profile Finishing or Groove Finishing operation. To create Local Information, you must define the geometry to be machined .

Create a Profile Finishing operation in the program. In the Profile Finishing dialog box that appears, select the Geometry tab page .



Local Information can be added on lines, arcs and curves if they are part of a finished profile.
Click the red part in the icon then select the desired finish profile in the 3D window.





To create or edit local information

To add local information on an element, right click on it and select the **Add Local Information** contextual command.

To edit Local Information from an element, right click on it and select the **Edit Local Information** contextual command.

In both cases, set the information you want then click OK.

To copy the local information of an element and paste it on another element of the finish profile, right click on it and select the **Copy Local Information** contextual command, then right click on the target element and select the **Paste Local Information** contextual command.

In the Local Information Browser, you can edit local information by double clicking on the row of the list you are interested in.

To copy local information from an element and paste it on another element of the finished profile, select an element in the list of the Local Information Browser and select the information you want to copy in the right-frame. Then click **Paste on** and select the target element.

Local information is associated to the selected profile and is supported by that profile. If you delete the profile, the associated information is also deleted.

To view local information

To browse local information on a finished body, right click on an element of the finished body or on the representation of the Part Body in the Geometry tab, then select the **Browse Local Information** contextual command.

In the Local Information Browser, you can see on the 3D view of the finished body defined local information by clicking on the column header.

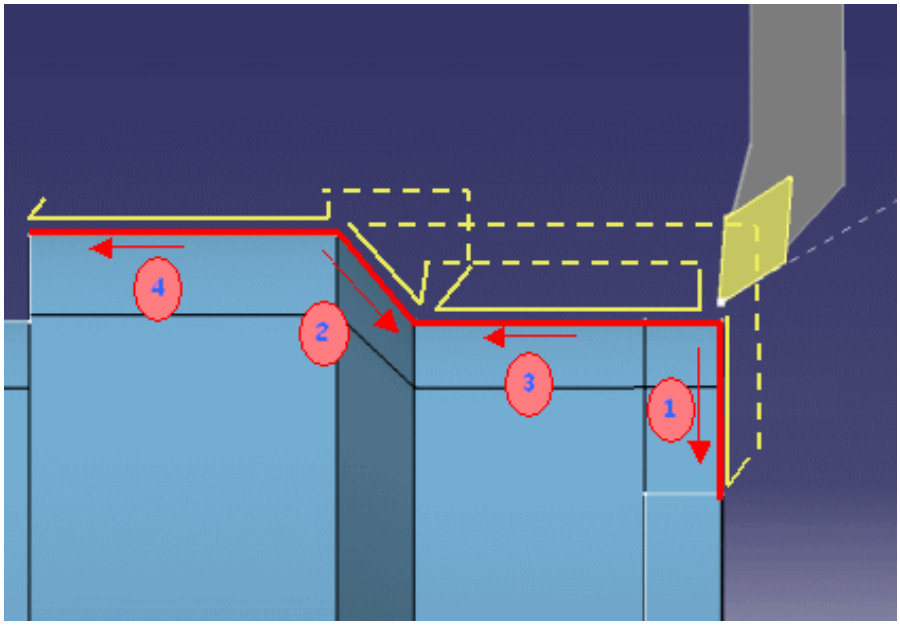
To remove local information

To remove Local Information from an element, right click on it and select the **Remove Local Information** contextual command.

In the Local Information Browser, click on the local information you want to delete and click **Remove**.

Options for Inverted Element (for Profile Finishing only)

The inversion of elements is possible for Profile Finishing operations. This is illustrated in the figure below:



In the Strategy tab page, click on the Local Invert tab.

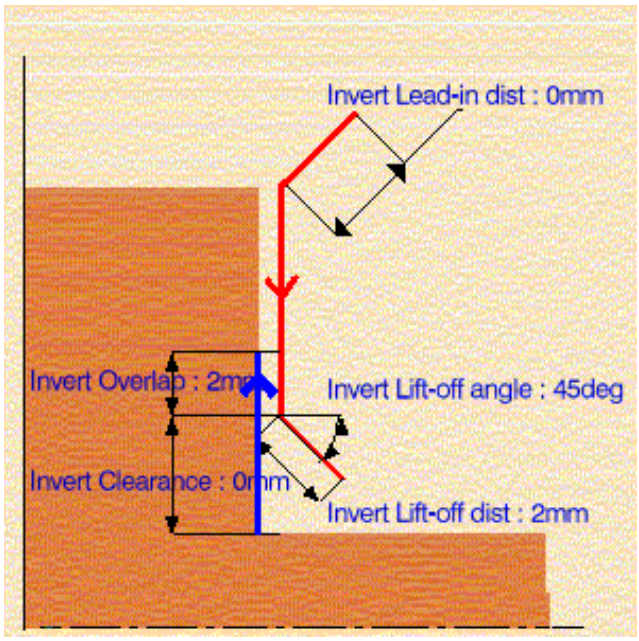
To machine part with inverted elements, several strategies are available:

- **None**: the profile is machined with inverted ways of machining.
- **Thickness**: a given thickness is let on inverted element and the remaining material is removed when the inverted element is

machined again.

- **Overlap**: a given length of an element is machined twice when the profile is machined and then when inverted elements are machined. In this case, you can choose to machine inverted elements first or later.

The lift-off can be linear or circular as specified in the Machining tab.



Update Input Stock - Manual Mode

This task shows you how to update the input stock manually for the machining operations in a lathe manufacturing program. These machining operations can be lathe operations, and axial operations along the spindle axis.

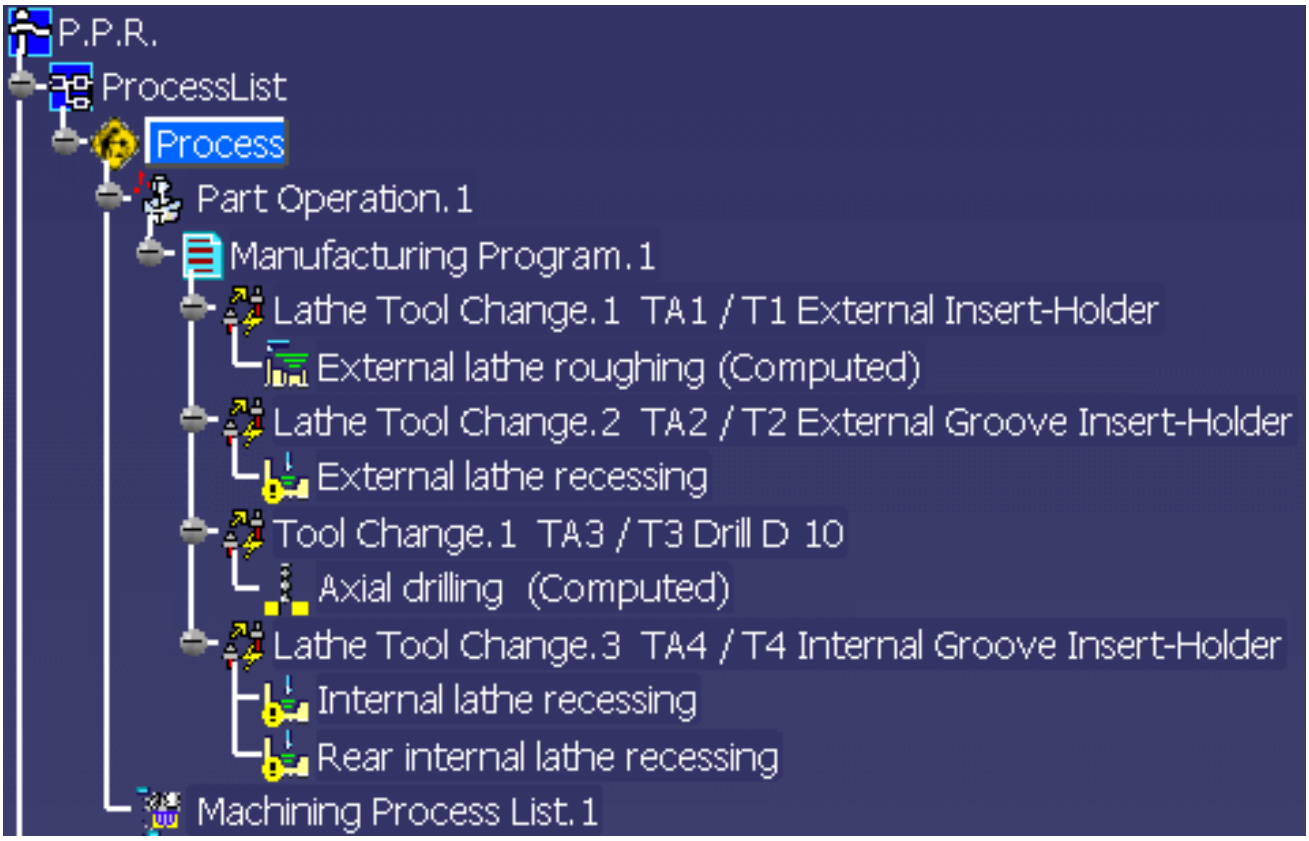
In Manual mode, the input stock takes into account all the previous operations. The lathe and axial operations must be completed.

Please refer to How to Update Input Stock for more information about this capability.

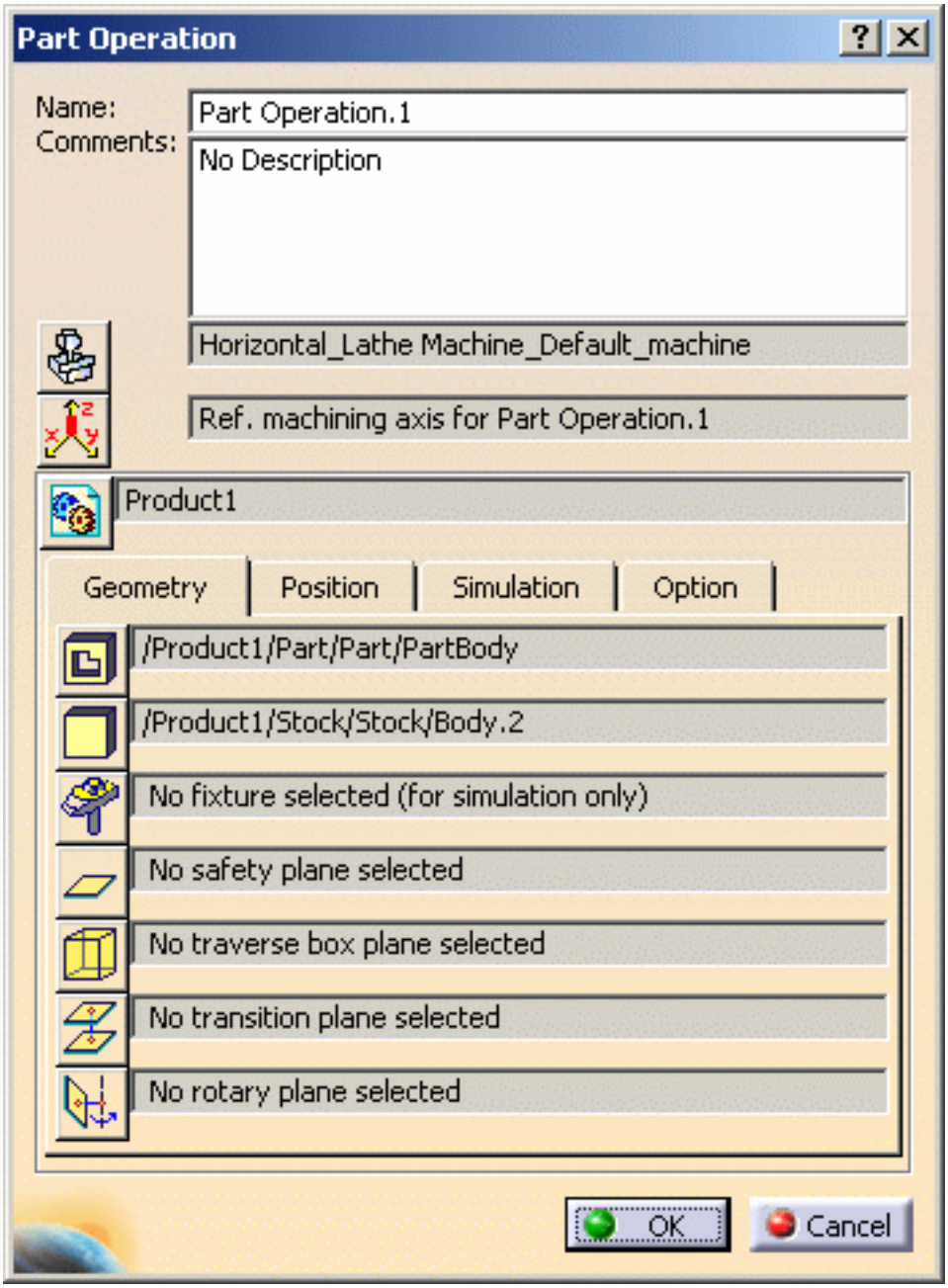
1. Select **File** > **Open** then select the StartStockUpdateManualMode.CATProcess document

2. Select **Machining** > **Lathe Machining** from the **Start** menu.

   The Lathe Machining workbench appears. The part is displayed in the Setup Editor window along with the manufacturing specification tree.



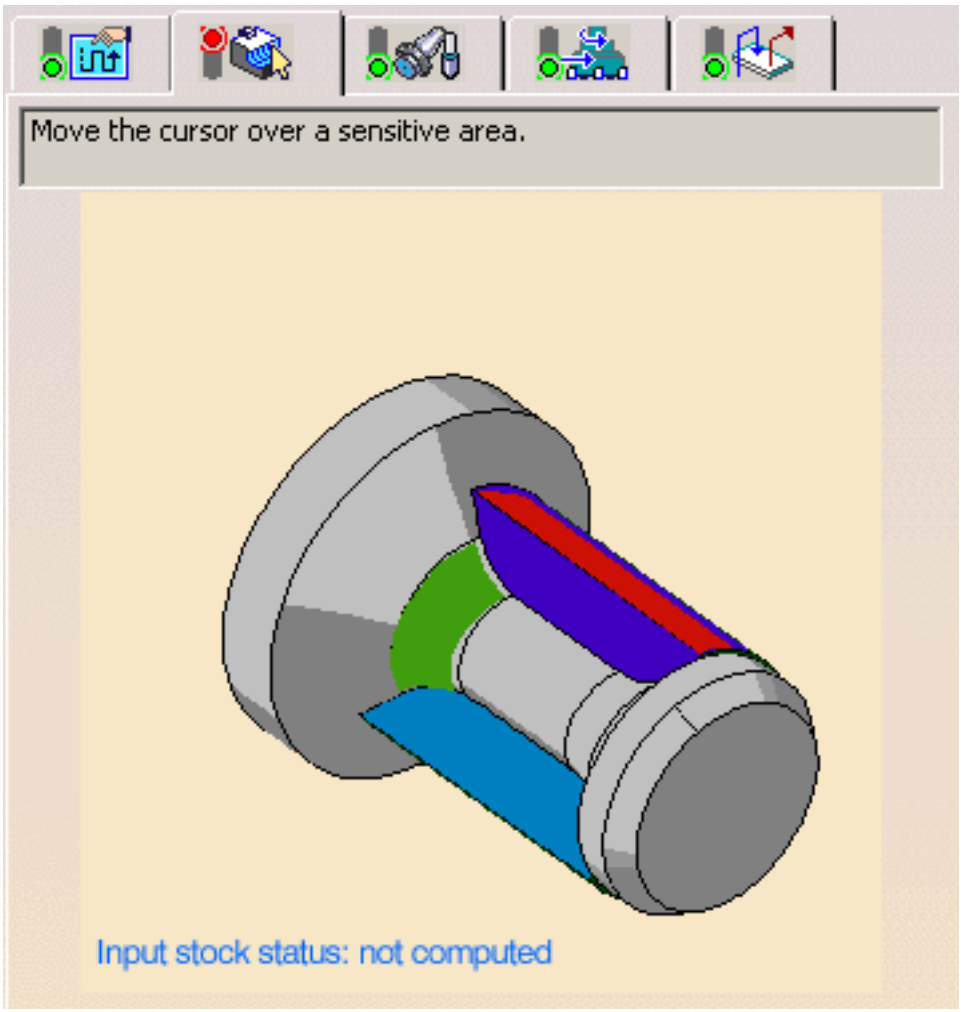
3. Double click Part Operation.1 in the tree to display the Part Operation dialog box.



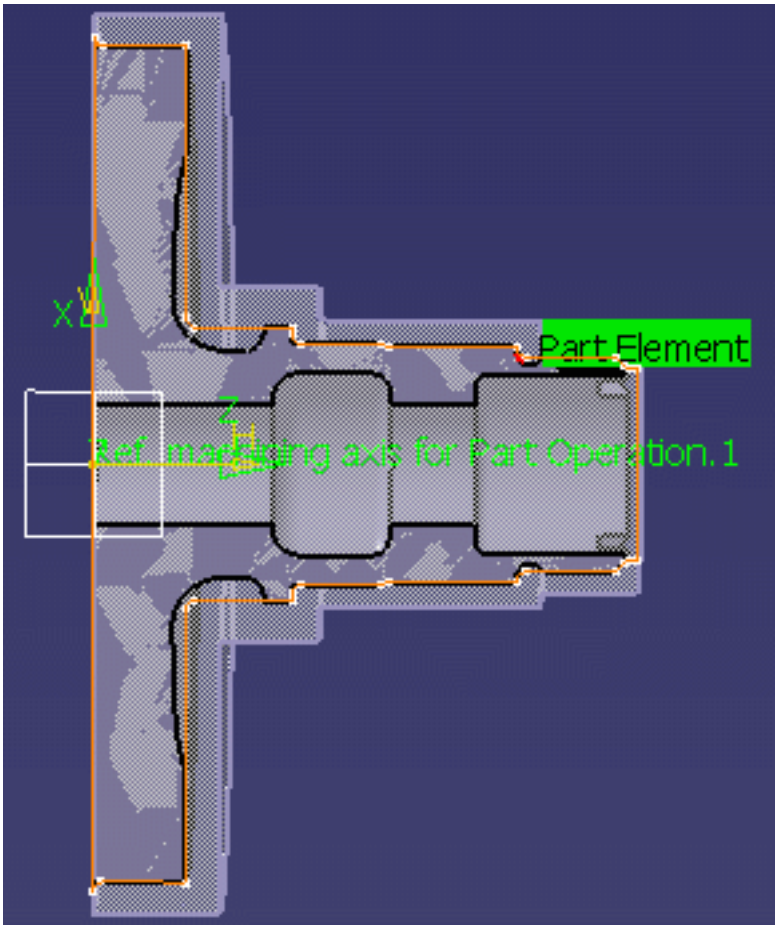
Right-click the Stock field to make sure that **Automatic Stock Selection for Turning Operations** is not activated in the contextual menu.

4. Double click the second lathe operation in the program: **External lathe recessing**.

5. In the Geometry page, the status is red because there is no input stock defined for the operation.

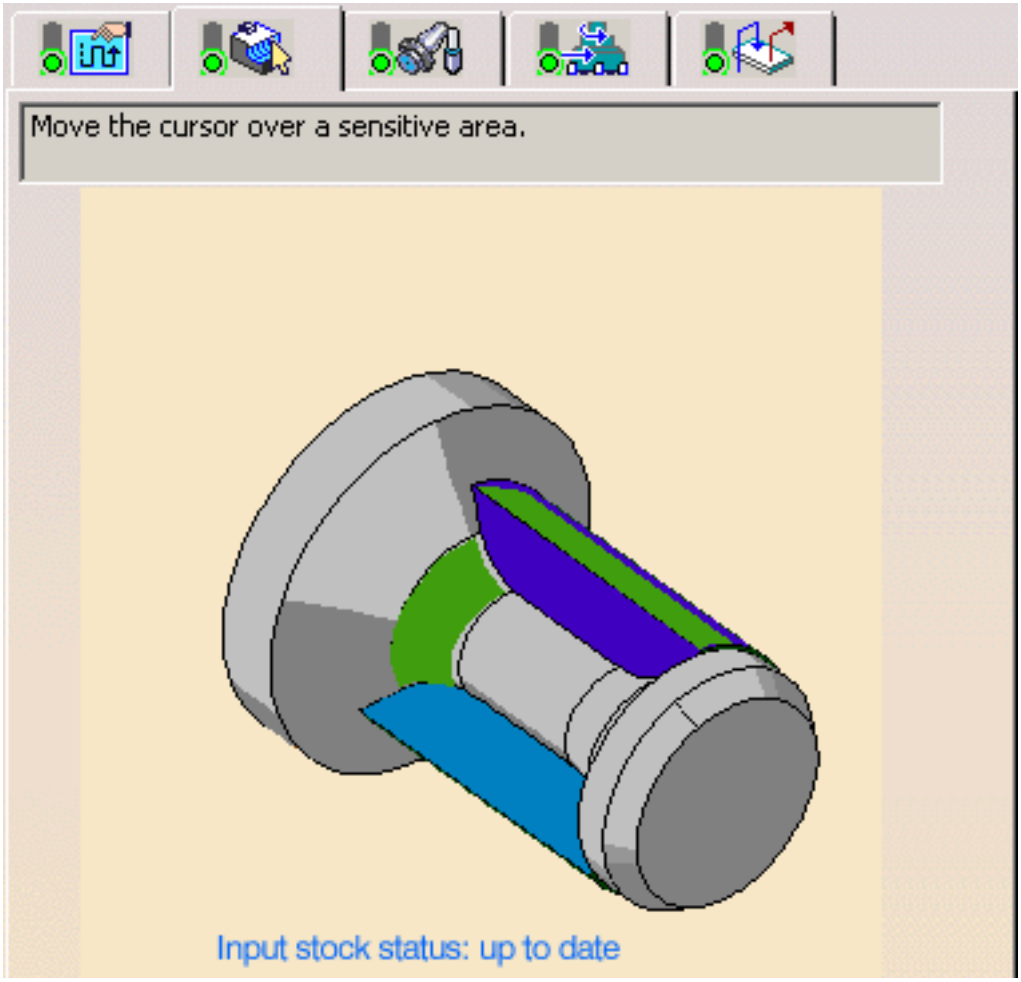


6. Select the **Update Input Stock** command by either clicking the icon  or right-clicking the **Input stock status** field in the sensitive icon. The stock is computed and the corresponding sketch profile is displayed in the viewer.



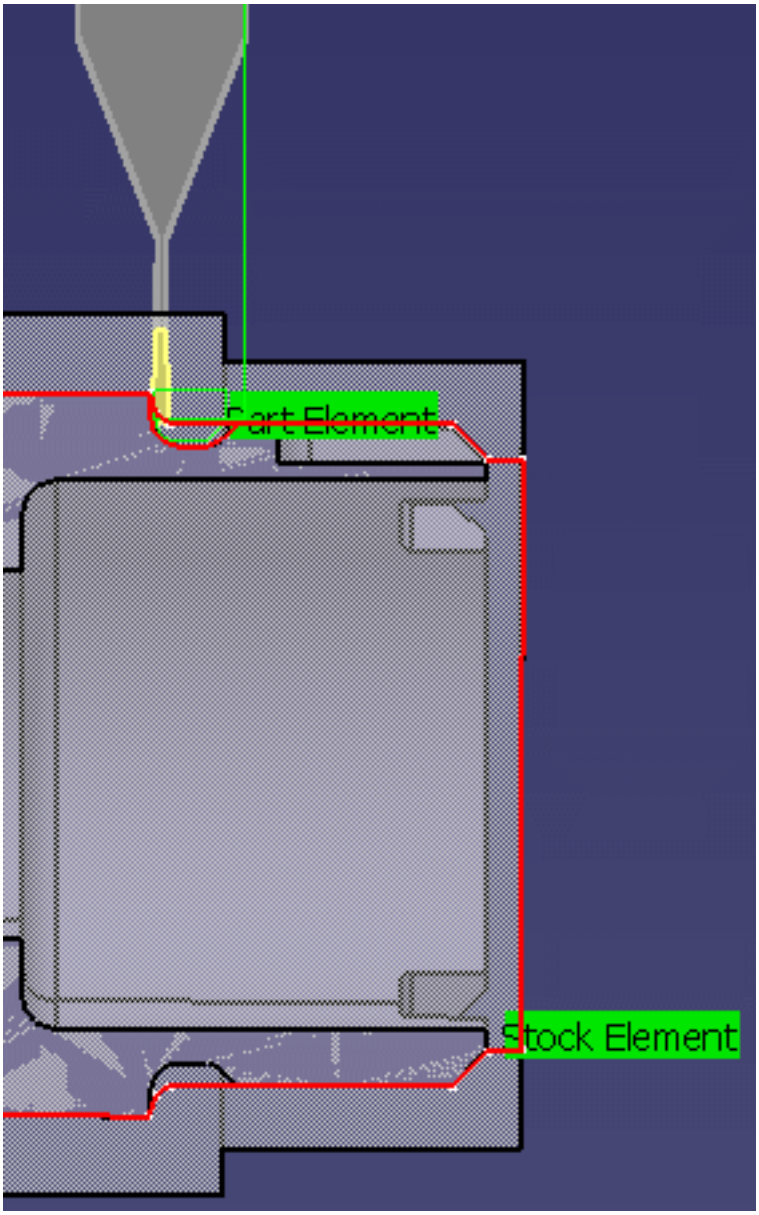
7. Select this profile as the input stock for the operation.

The Geometry page is updated with this stock



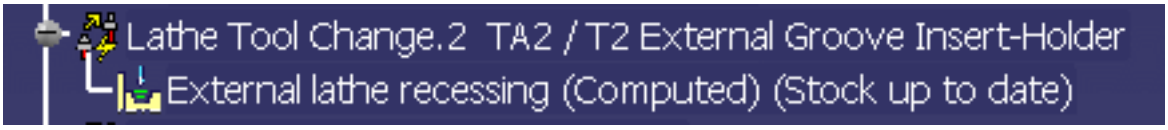
You can use the **Analyze** contextual command (right-click the stock area in the sensitive icon) to check that an input stock has been selected.

8. Click the **Replay** icon  to verify the operation.



Click OK to accept the operation.

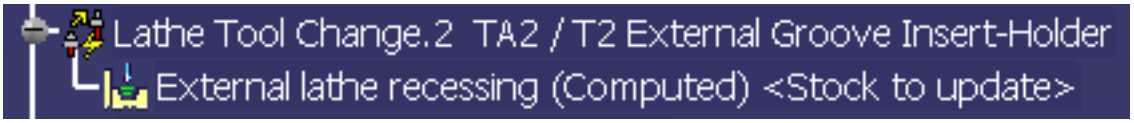
9. The status of the operation is as follows:



10. The input stock of the other Lathe operations in the program can be updated in the same way.

Updating Computed Stock Status

For modifications to the Part Operation, a previous operation in the program (edit), or a previous sequence of the program (delete or copy/paste), the stock status becomes **Stock to update**.



After modifications, there is no automatic refresh of the stock status of the operations. You can right click the Program in the tree and select the **Update Computed Stock Status** contextual command. This will refresh the stock status of all the operations.

Please note that when you copy/paste an operation, the input stock is not copied.

Removing Input Stock

If you right-click the **Input stock status** field in the Geometry tab page, you can use the **Remove Input Stock** contextual command to remove the operation's input stock. The corresponding sketch profile will be deleted. The status of the previously selected geometry from the deleted sketch will be *Not found*, so new geometry must be selected in this case. You cannot use the operation's **Cancel** button to recover the deleted sketch.

Update Input Stock - Automatic Mode

This task shows you how to update the input stock automatically for the machining operations in a lathe manufacturing program. These machining operations can be lathe operations, and axial operations along the spindle axis.

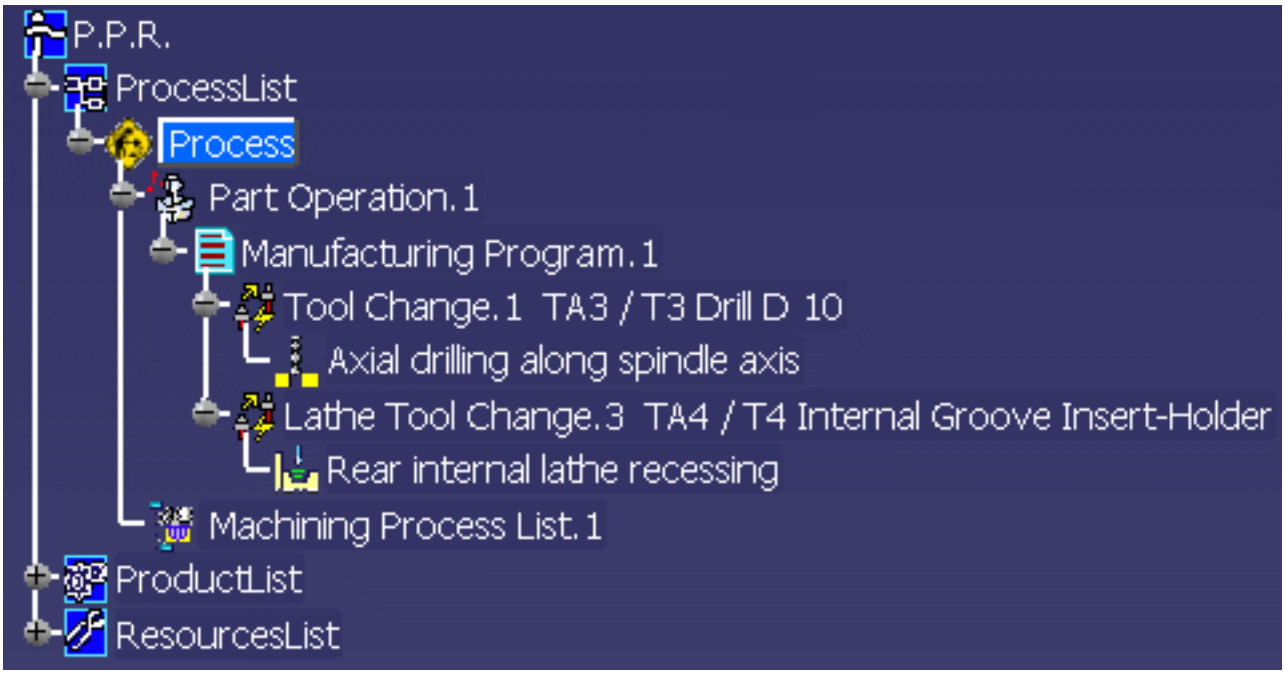
In Automatic mode, the input stock takes into account all the previous operations. The axial operations must be completed and the lathe operations that use a stock must have either a manually selected stock or a saved input stock.

Please refer to How to Update Input Stock for more information about this capability.

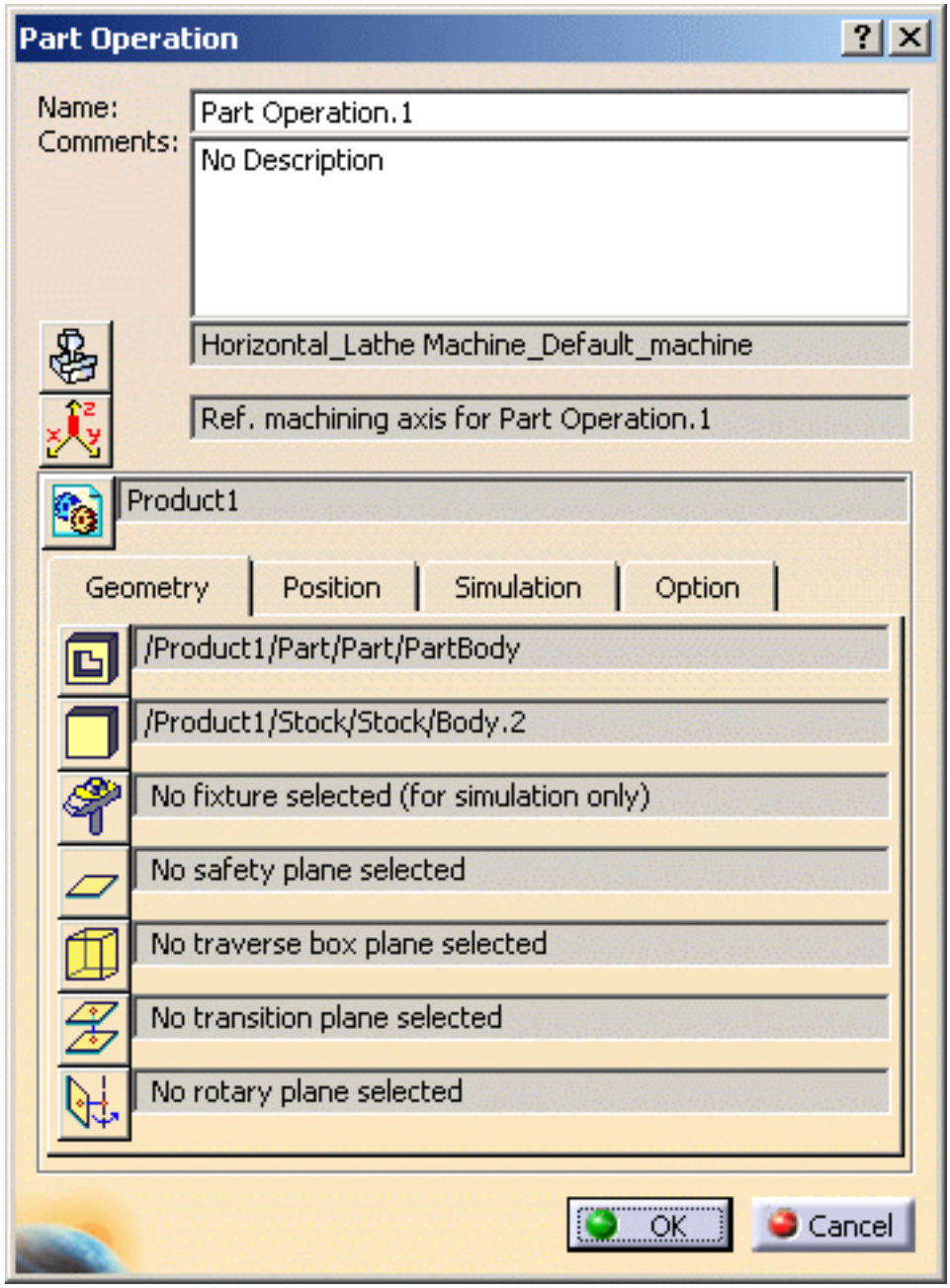
1. Select **File** > **Open** then select the StartStockUpdateAutoMode.CATProcess document

2. Select **Machining** > **Lathe Machining** from the **Start** menu.

   The Lathe Machining workbench appears. The part is displayed in the Setup Editor window along with the manufacturing specification tree.

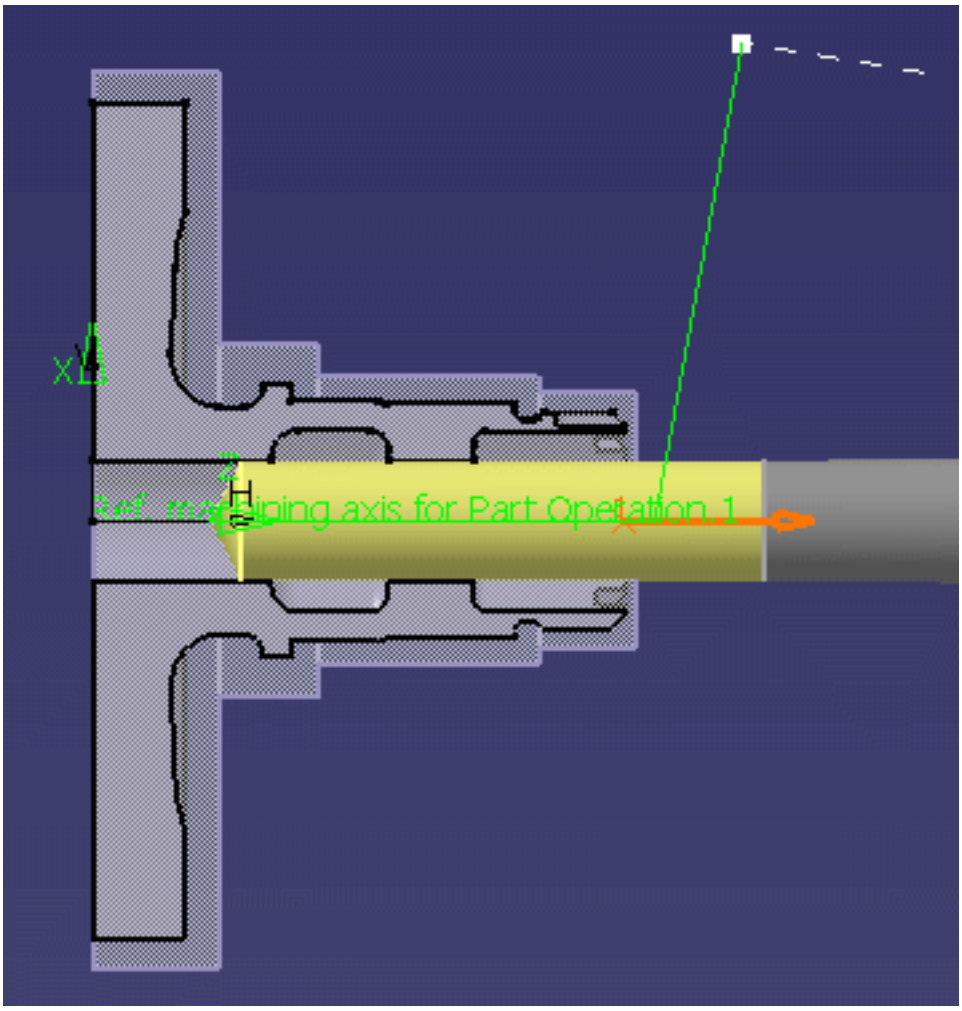


3. Double click Part Operation.1 in the tree to display the Part Operation dialog box.



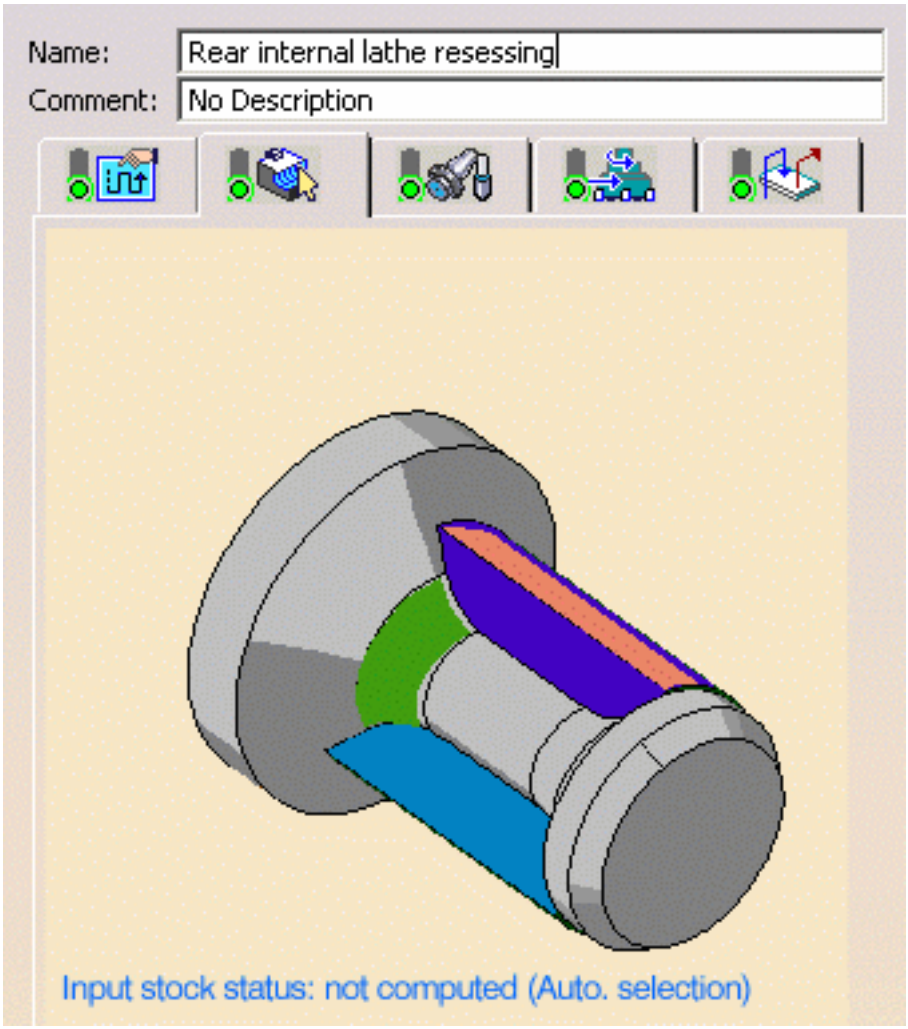
Right-click the Stock field to make sure that **Automatic Stock Selection for Turning Operations** is activated in the contextual menu.

4. Right click the drilling operation in the tree and select the Replay Tool Path icon to replay the operation



Click OK in the Replay dialog box. The operation status in the tree is now **Computed**.

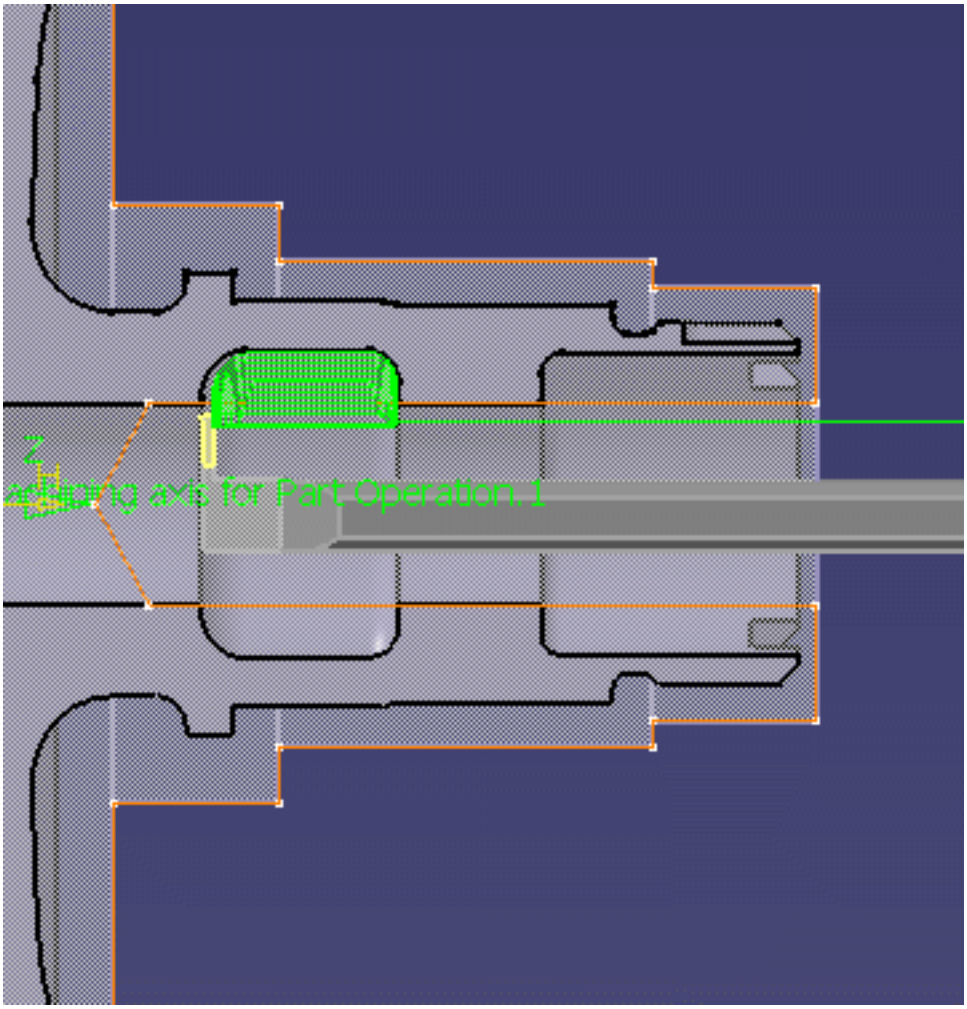
5. Double click the lathe recessing operation in the tree. Select the Strategy tab in the machining operation editor.



Note that:

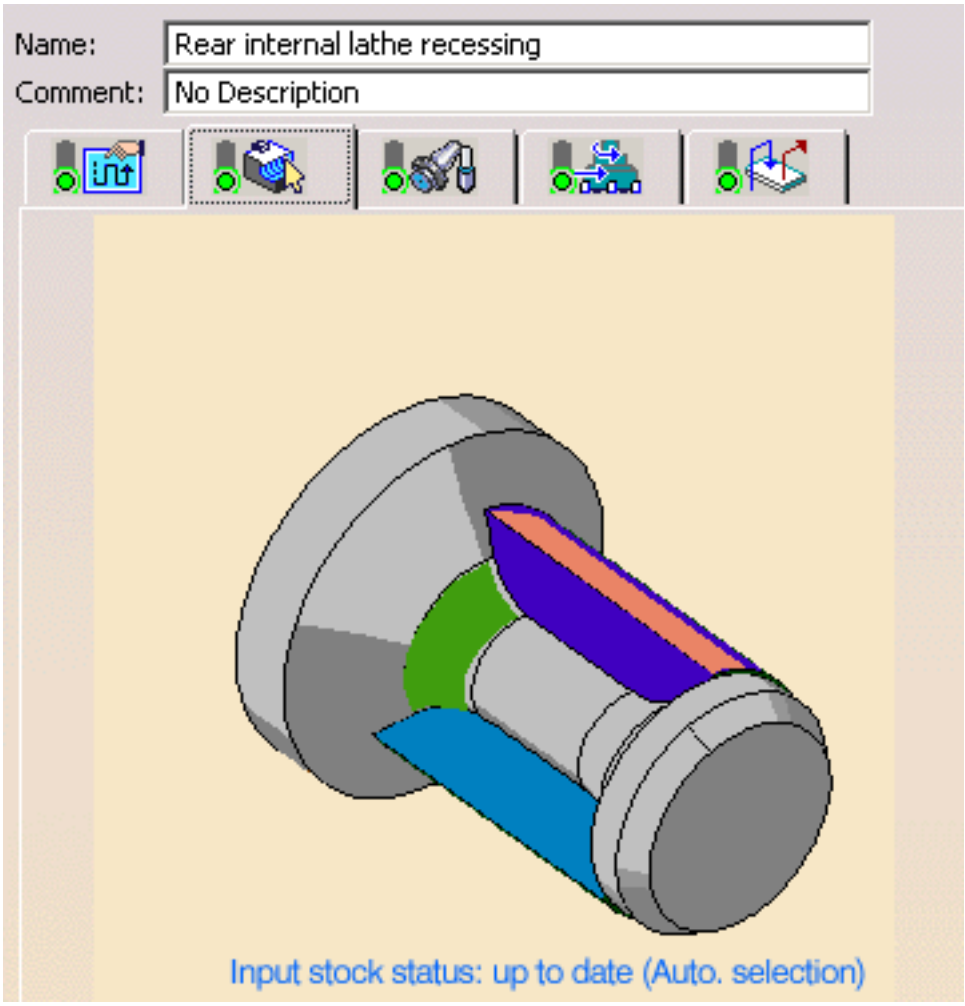
- the color of the stock area indicates that stock selection is optional
- the Input stock status is **Not computed** and that the automatic Stock selection is activated (**Auto. selection**).

6. Click the Tool Path Replay icon  to replay the operation.



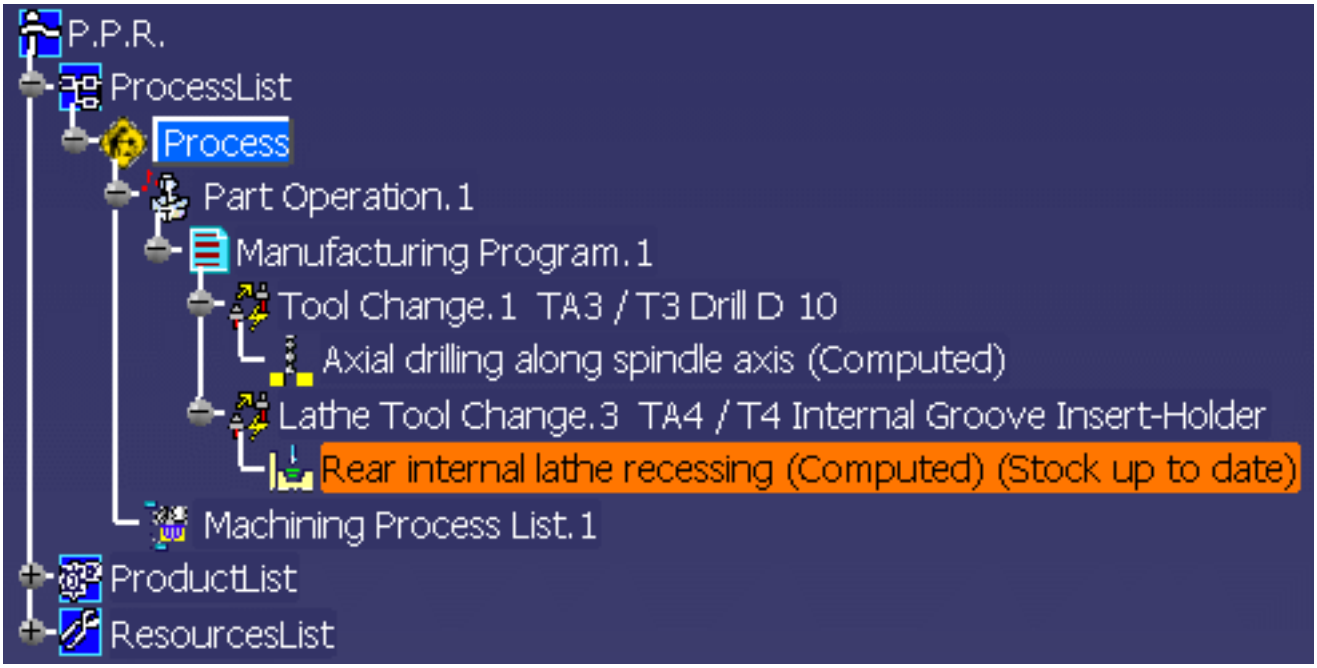
Click OK in the Replay dialog box.

7. In the Strategy tab  the Input stock status is now **Up to date**.



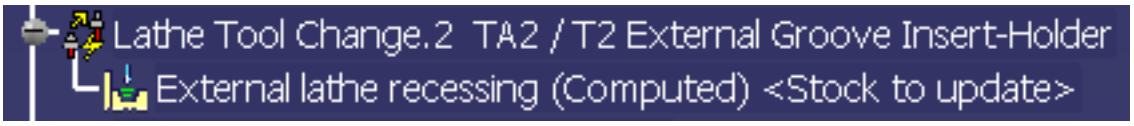
Click OK in the machining operation editor.

In the manufacturing specification tree, the operation status is **Computed** and **Stock up to date**.

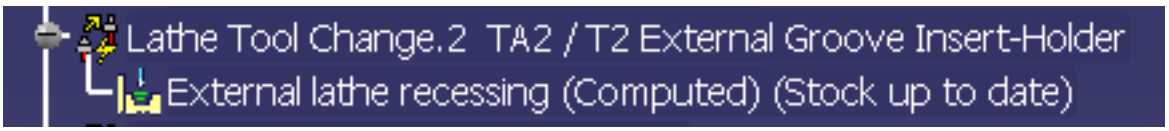


Computing Stock and Tool Path

For modifications to the Part Operation, previous operation in the program (edit) or previous sequence of the program (delete or copy/paste), the stock status becomes **Stock to update**.



In this case you can right click the Manufacturing Program in the tree and select the **Compute Stock and Tool Path** contextual command. All input stocks and tool paths of the operations in the program are recomputed.



Please note that when you copy/paste an operation, the input stock is not copied.

Removing Input Stock

If you right-click the **Input stock status** field in the Geometry tab page, you can use the **Remove Input Stock** contextual command to remove the operation's input stock. The corresponding sketch profile will be deleted. The status of the previously selected geometry from the deleted sketch will be *Not found*, so new geometry must be selected in this case. You cannot use the operation's **Cancel** button to recover the deleted sketch.

Axial Machining Operations

The tasks for creating axial machining operations are documented in the *Prismatic Machining User's Guide*.

Spot Drilling Operation

Create a Spot Drilling Operation: Select the Spot Drilling icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.

Drilling Operations

Create a Drilling Operation: Select the Drilling icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.

Create a Drilling Dwell Delay Operation: Select the Drilling Dwell Delay icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.

Create a Drilling Deep Hole Operation: Select the Drilling Deep Hole icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.

Create a Drilling Break Chips Operation: Select the Drilling Break Chips icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.

Hole Finishing Operations

Create a Reaming Operation: Select the Reaming icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.

Create a Counterboring Operation: Select the Counterboring icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.

Boring Operations

Create a Boring Operation: Select the Boring icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.

Create a Boring Spindle Stop Operation: Select the Boring Spindle Stop icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.

Create a Boring and Chamfering Operation: Select the Boring and Chamfering icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.

Create a Back Boring Operation: Select the Back Boring icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.

Threading Operations

Create a Tapping Operation: Select the Tapping icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.


Create a Reverse Threading Operation: Select the Reverse Threading icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.


Create a Thread without Tap Head Operation: Select the Thread without Tap Head icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.


Create a Thread Milling Operation: Select the Thread Milling icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.

Countersinking and Chamfering Operations


Create a Countersinking Operation: Select the Countersinking icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.


Create a Chamfering Two Sides Operation: Select the Chamfering Two Sides icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.

T-Slotting and Circular Milling


Create a T-Slotting Operation: Select the T-Slotting icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros and feeds and speeds as needed.


Create a Circular Milling Operation: Select the Circular Milling icon then select the hole or hole pattern to be machined and specify the tool to be used. Specify machining strategy parameters, macros, and feeds and speeds as needed.

Auxiliary Operations

The tasks for inserting auxiliary operations in the manufacturing program are documented in the *NC Manufacturing Infrastructure User's Guide*.


Insert Tool Change: Select the Tool Change icon then select the tool type to be referenced in the tool change.


Insert Machine Rotation: Select the Machine Rotation icon then specify the tool rotation characteristics.


Insert Machining Axis Change: Select the Machining Axis Change icon then specify the characteristics of the new machining axis system.


Insert PP Instruction: Select the PP Instruction icon then enter the syntax of the PP instruction.


Insert COPY Operator **(P2 functionality)**: Select the COPY Operator icon then select the reference operation. You can then specify the number of copies and the characteristics of the transformation.


Insert TRACUT Operator **(P2 functionality)**: Select the TRACUT Operator icon then select the reference operation. You can then specify the characteristics of the transformation.


Insert Copy Transformation Instruction **(P2 functionality)**: Select the Copy Transformation icon then select the reference operation. You can then specify the number of copies and the characteristics of the transformation.


Opposite Hand Machining: for machining symmetrical parts.

Part Operations, Manufacturing Programs and Machining Processes

The tasks for creating and managing Part Operations, Manufacturing Programs and Machining Processes are documented in the *NC Manufacturing Infrastructure User's Guide*.

Create and Edit a Part Operation: Select the Part Operation icon then specify the entities to be referenced by the part operation: machine tool, machining axis system, tool change point, part set up, and so on.

Create and Edit a Manufacturing Program: Select the Manufacturing Program icon to add a program to the current part operation then insert all necessary program entities: machining operations, tool changes, PP instructions, and so on.

Create a Machining Process **(P2 Functionality)**: Select the Machining Process icon to create a machining process, which can then be stored in a catalog.

Apply a Machining Process **(P2 Functionality)**: Select the Open Catalog icon to access the machining process to be applied to selected geometry.

Create a Machining Process for Lathe Machining

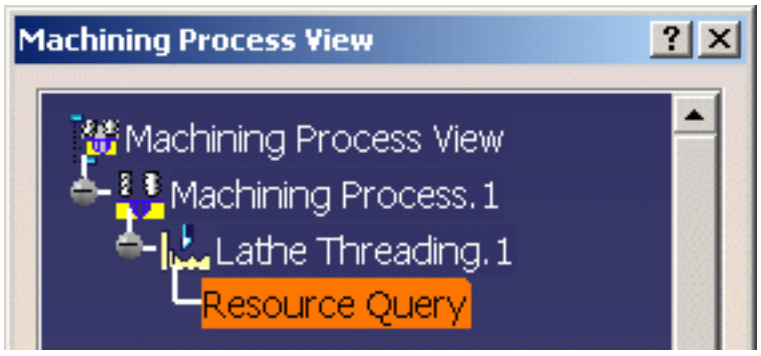
This task shows how to create a machining process for a lathe Threading operation. In this scenario you will specify a **tooling query** to find an appropriate tool and insert for the operation.

Select an Machining workbench from the Start menu.
No CATPart or CATProcess is needed at this stage.
If the Machining Process toolbar is not already displayed, select it using View > Toolbars.
Make sure that Start Edit mode is selected in the Operations tab page of Tools > Options > Machining.

Initialize the Machining Process

1. Select the Machining Process View icon  . The Machining Process View dialog box appears.
2. Select the Machining Process icon  . The dialog box is updated with a new machining process as shown.
3. Select the Lathe Threading icon.
   The Lathe Threading dialog box appears.

   At this stage you can set certain parameters such as feeds and speeds and machining strategy. However, there is only limited access to geometry parameters and it is not possible to specify a tool.
4. Just click OK to add a reference Lathe Threading operation to the machining process.

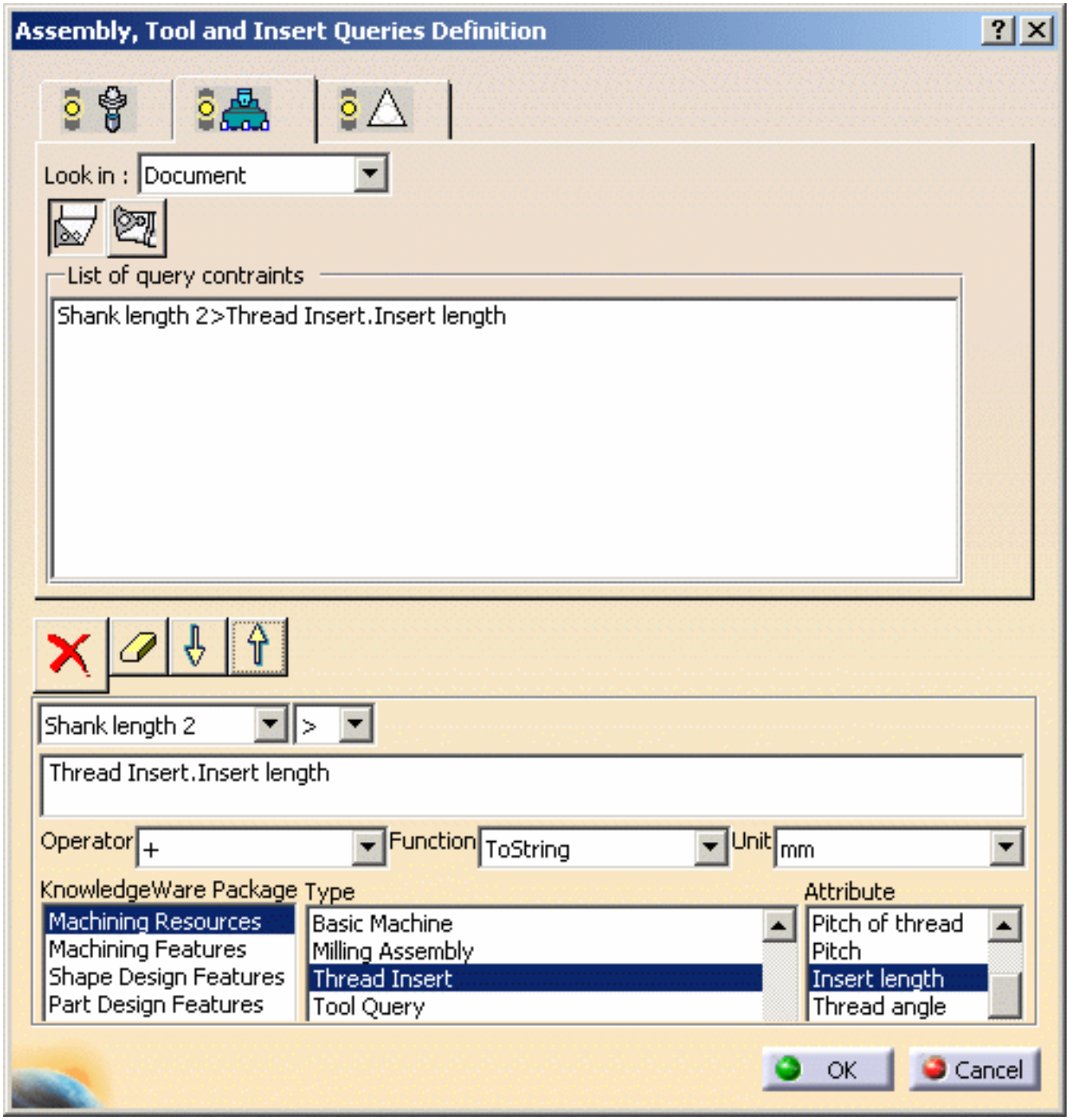


The reference operation has an associated Tooling Query.

Define the Tooling Query

5. Double click the Tooling Query associated to the Lathe Threading operation. The Query Definition dialog box appears.

   Define a simple tooling query as shown below. It corresponds to the criteria:
   find a spot drill in the ToolsSampleMP tool repository whose name is Spot Drill D10.



6. Click OK to assign the tooling query to the Lathe Threading operation.
7. Select File > Save As to save the machining process in a CATProcess document (called LatheThreadingProcess1.CATProcess, for example).

8. Right click the Machining Process in the Machining Process View and select the **Save in Catalog** contextual command.

   The Save in Catalog dialog box appears. Click the [...] button and specify a new catalog name (catalogAxialMP1.catalog, for example).

   Click OK to save the machining process as a component in the specified catalog.

   The following are initialized automatically:

   - family name: Machining Process
   - component name: name given to the machining process using File > Save As.

   However, you can change family or component in the Catalog Editor workbench. Click here to see how you can organize machining processes in a catalog using the Catalog Editor workbench.

   Then you can Apply the Machining Process to the geometry to be machined. The tooling query will be resolved at this stage.

Click here to see how you can make use of Knowledgeware functionalities in Machining Processes.

Machining Resources and Other Entities

Tasks involving machining entities that are specific to the Lathe Machining product can be found in this section.

Edit the Insert Holder of a Turning Operation
Edit the Tool Assembly of a Turning Operation
Edit the Insert of a Turning Operation
Edit an Insert Holder in the Resource List
Edit a Tool Assembly in the Resource List
Edit an Insert in the Resource List

The tasks for creating and managing specific entities of the Machining environment are documented in the *NC Manufacturing Infrastructure User's Guide*.

- Edit the Tool of a Milling or Drilling Operation: Double click the machining operation in the program and select the Tool tab page to edit the characteristics or search for a new tool.
- Edit a Mill or Drill Tool in the Resource List: Double click a tool referenced in the resource list and edit the characteristics in the Tool Definition dialog box.
- Edit a Mill or Drill Tool Assembly in the Resource List: Double click a tool assembly referenced in the resource list and edit the characteristics in the Tool Assembly Definition dialog box.
- Specify Tool Compensation Information: Double click a tool referenced in the program or resource list and specify the tool compensation information in the Compensation tab page of the Tool Definition dialog box.
- Create and Use Machining Patterns: Select Insert > Machining Feature > Machining Pattern then select a pattern of holes to be machined.
- Feature Based Programming: Select a feature using the Manufacturing view and create operations based on this feature.
- Define Macros on a Turning Operation: Select the Macros tab page when creating or editing a turning operation, then specify the transition paths of the macros to be used in the operation.
- Define Macros on an Axial Machining Operation: Select the Macros tab page when creating or editing an axial machining operation, then specify the transition paths of the macros to be used in the operation.
- Manage the Status of Manufacturing Entities: Use the status lights to know whether or not your operation is correctly defined.

Edit the Insert Holder of a Turning Operation

This task shows you how to edit the insert holder of a turning operation.

You can do this in several ways:

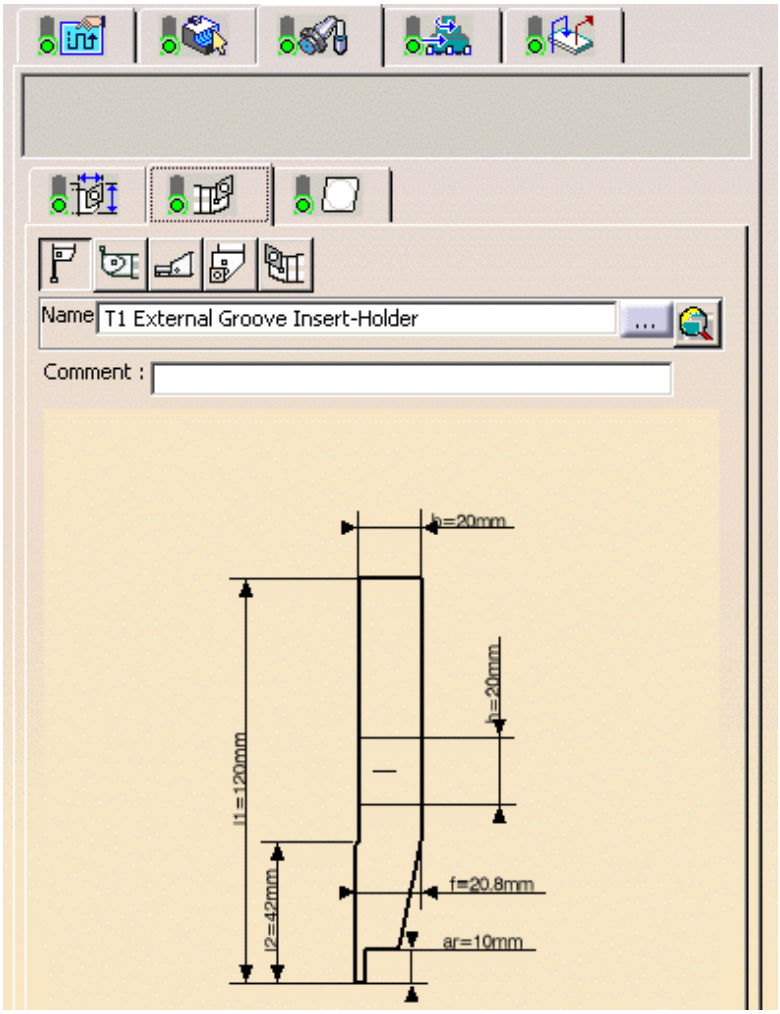
- edit its characteristics, thereby creating a new insert holder
- replace it by selecting another insert holder that is already used in the document
- replace it by selecting another insert holder by means of a query.

1. Double click the operation in the program, then select the Insert Holder tab in the Tooling page .

2. **To create a new insert holder:**

   If you want to change insert holder type, select the icon corresponding to the desired type:

| | | | | | | |
| | | | | | | |
| | | | | | | --> Internal Thread Insert Holder
| | | | | | --> External Thread Insert Holder
| | | | | --> Frontal Groove Insert Holder
| | | | --> Internal Groove Insert Holder
| | | --> External Groove Insert Holder
| | --> Internal Insert Holder
| --> External Insert Holder



In this case the corresponding insert holder representation appears in the 2D viewer.

- Double click the geometric parameter that you want to modify in the 2D viewer, then enter the desired value in the Edit Parameters dialog box that appears.

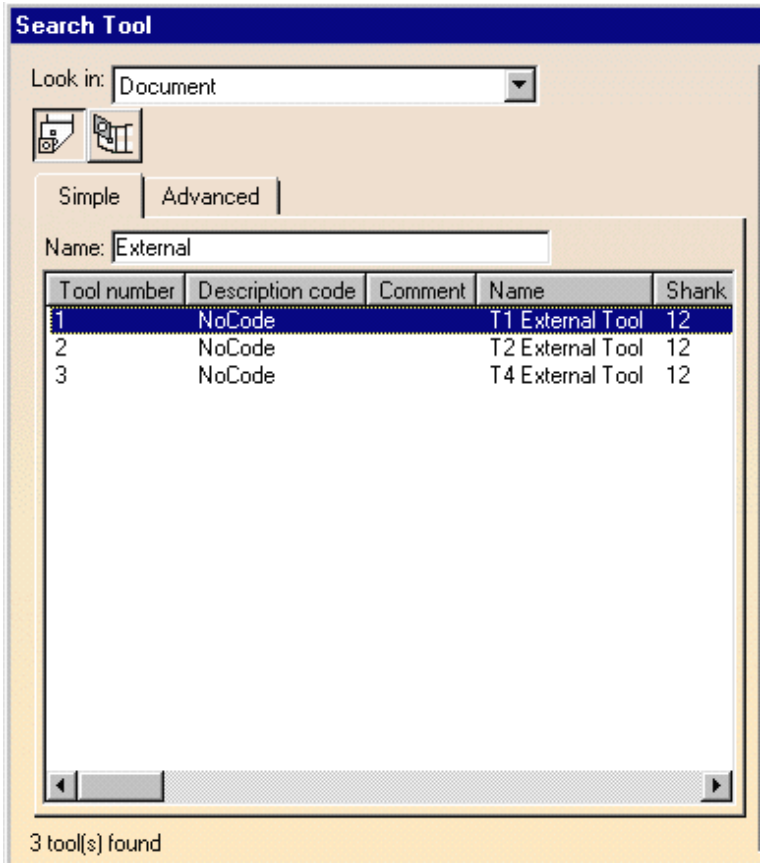
  Modify other parameters in the same way.
  The insert holder representation is updated to take the new values into account.
- Click More to expand the dialog box to access all the insert holder's parameters.
  Modify the values as desired.
- Enter a name for the new insert holder.

If the same tooling is used on more than one operation, the parameters of the resource are not accessible for modification (they are greyed out).

In this case, you can create a new resource by entering a new identifier in the Name field. You can then specify the parameters of the new resource.

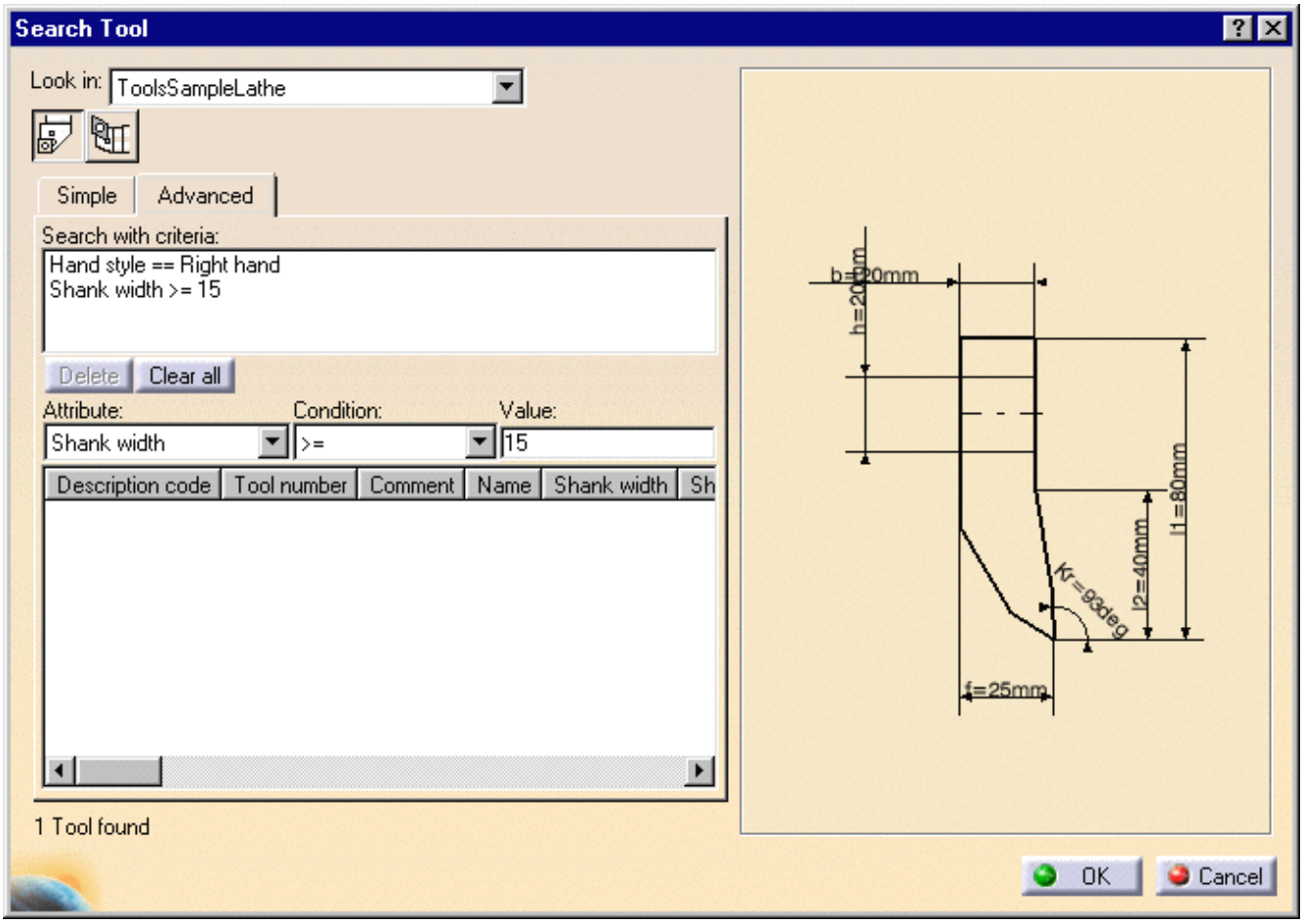
3. **To select an insert holder that is already used in the document:**
   - Select the [...] button opposite Name. Then select the desired insert holder from the list of insert holders already present in your document.
   - In the Name field, enter the name of an insert holder already present in your document.

   The insert holder representation is displayed in the 2D viewer. It can be edited as described above.
4. **To select another insert holder by means of a query:**
   - Click the **Select an insert holder with query** icon opposite Name.
     The Search Tool dialog box appears.



   - Use the **Look in** combo to specify where you want to search for the insert holder:
     - in the current document
     - in a tool catalog
     - in an external tool database such as the TDM (Tool Data Management) or CATIA Version 4 Manufacturing database.
   - If you want to change insert holder type, select the icon corresponding to the desired insert holder.
   - You can do a quick search in the **Simple** tab page by entering a character string in the Name field. The insert holders meeting the simple search criteria are listed.
   - Select the desired insert holder from the list and click OK.
     The insert holder representation is displayed in the 2D viewer. It can be edited as described above.

- You can search an insert holder using finer constraints by selecting the **Advanced** tab page. The example below shows the result of a search for a right hand insert holder with shank width of 15mm or more in the catalog ToolsSampleLathe.



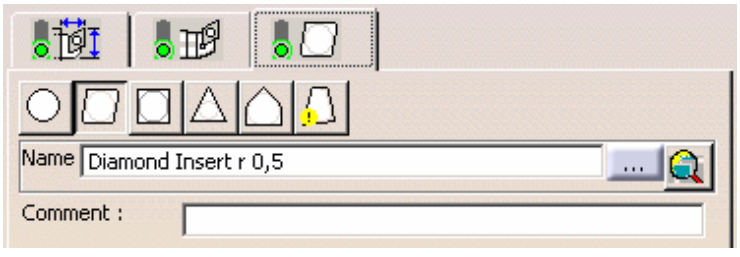
5. Click OK to confirm using this new insert holder in the operation.

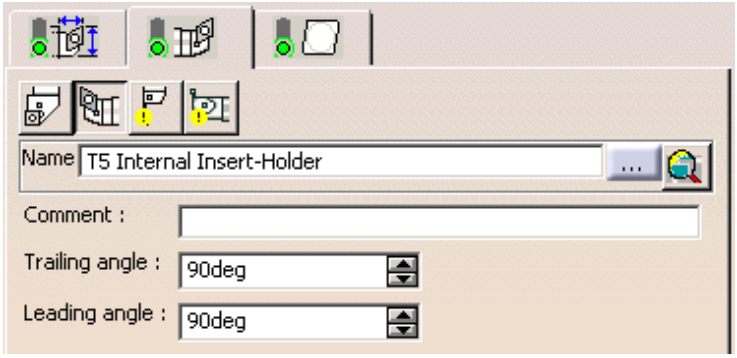
Note that compatibility is checked between the insert and the insert holder.

If there is incompatibily, this is indicated by an exclamation mark symbol on the icon.

In the example below, the selected diamond insert  **is compatible** with the selected Internal insert holder  .

However, it cannot be used on a groove internal insert holder  or a groove external insert holder  .





If, for example, you select a groove internal insert holder, the diamond insert will no longer be referenced by the tool assembly (the status light on the Insert tab turns red). Then you will have to select a compatible insert type (such as groove insert ).

Edit the Tool Assembly of a Turning Operation

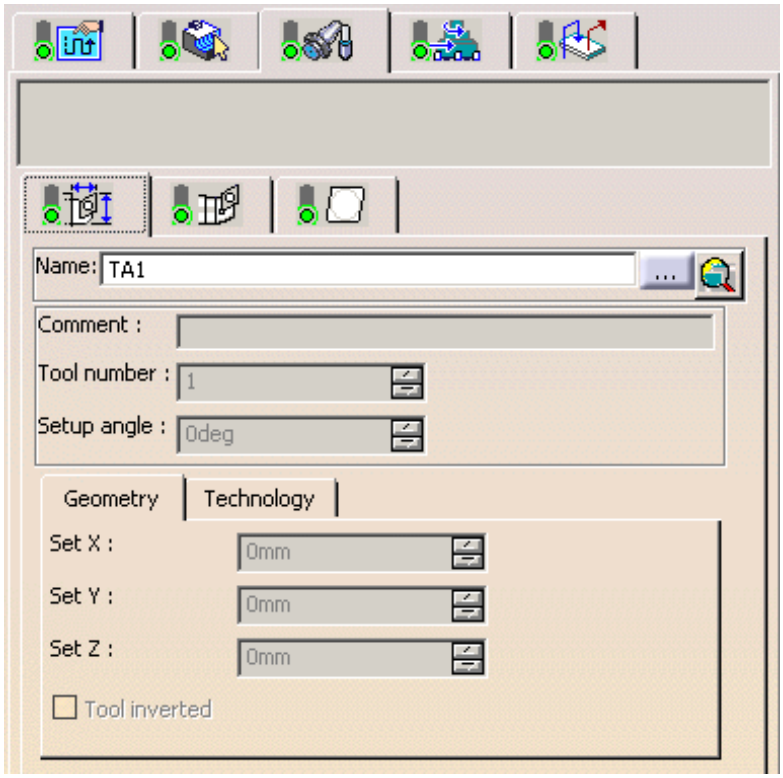
This task shows you how to edit the tool assembly of a turning operation.

You can do this in several ways:

- edit its characteristics, thereby creating a new tool assembly
- replace it by selecting another tool assembly that is already used in the document
- replace it by selecting another tool assembly by means of a query.

1. Double click the operation in the program, then select the Tool Assembly tab in the Tooling page.
2. **To create a new tool assembly:**

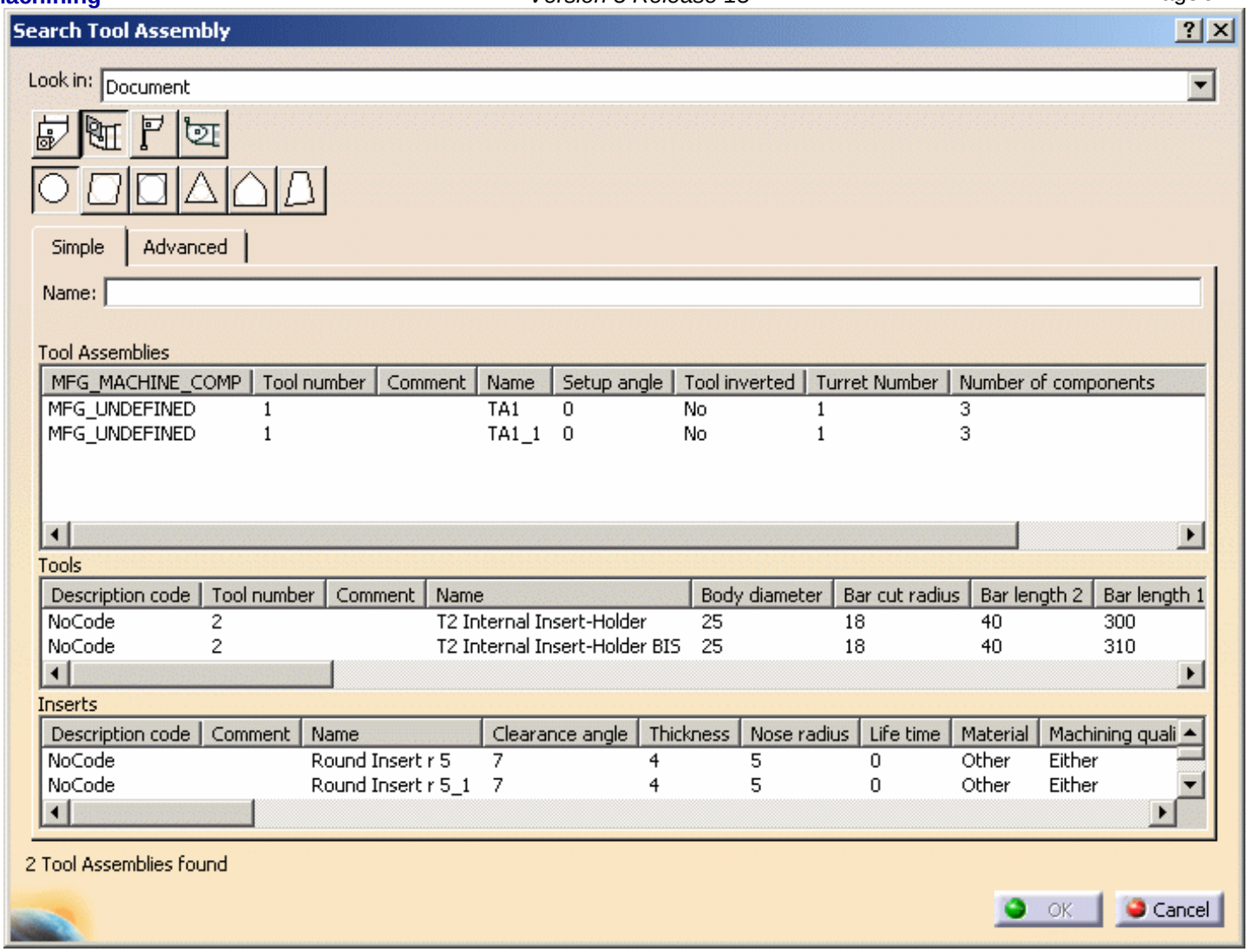
   The parameters of the current tool assembly are shown but are not accessible (greyed out).



   Enter a name for the new tool assembly. The parameter fields on the page become accessible.

   If the same tooling is used on more than one operation, the parameters of the resource are not accessible for modification (they are greyed out).

   In this case, you can create a new resource by entering a new identifier in the Name field. You can then specify the parameters of the new resource.
3. **To select a tool assembly that is already used in the document:**
   - Select the [...] button opposite Name. Then select the desired tool assembly from the list of tool assemblies already present in your document.
   - In the Name field, enter the name of a tool assembly already present in your document.
4. **To select another tool assembly by means of a query:**
   - Click the **Select a tool assembly with query** icon opposite Name.
     The Search Tool Assembly dialog box appears.



- Use the **Look in** combo to specify where you want to search for the tool assembly:
  - in the current document
  - in a tool catalog
  - in an external tool database such as the TDM (Tool Data Management) database.
- You can do a quick search in the **Simple** tab page by entering a character string in the Name field. The tool assemblies meeting the simple search criteria are listed.
- Select the desired tool assembly from the list and click OK.
- You can search a tool assembly using finer constraints by selecting the **Advanced** tab page.

5. Click OK to confirm using this new tool assembly in the operation.

Edit the Insert of a Turning Operation

This task shows you how to edit the insert of a turning operation.

You can do this in several ways:

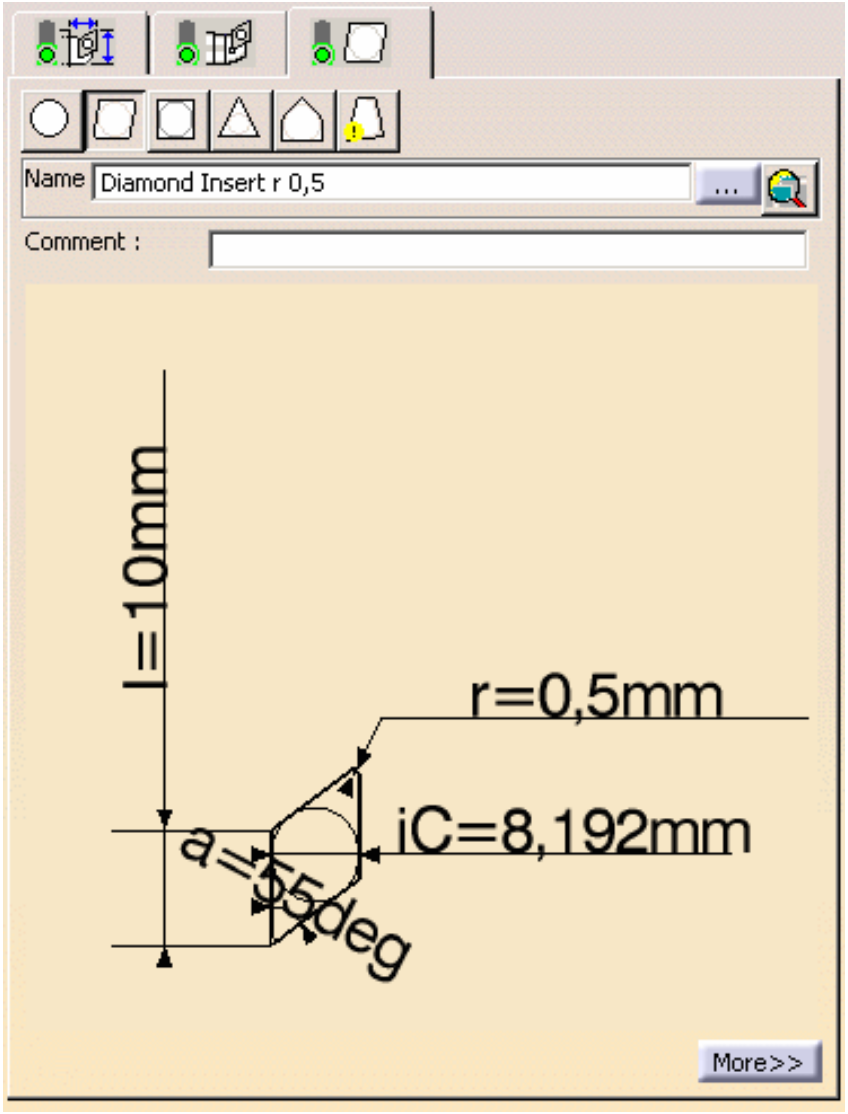
- edit its characteristics, thereby creating a new insert
- replace it by selecting another insert that is already used in the document
- replace it by selecting another insert by means of a query.

1. Double click the operation in the program, then select the Insert tab in the Tooling page .

2. **To create a new insert:**

   If you want to change insert type, select the icon corresponding to the desired type:

						--> Thread Insert
					--> Groove Insert	
				--> Trigon Insert		
			--> Triangular Insert			
		--> Square Insert				
	--> Diamond Insert					
--> Round Insert						



In this case the corresponding insert representation appears in the 2D viewer.

- Double click the geometric parameter that you want to modify in the 2D viewer, then enter the desired value in the Edit Parameters dialog box that appears.

  Modify other parameters in the same way.
  The insert representation is updated to take the new values into account.
- Click More to expand the dialog box to access all the insert's parameters.
  Modify the values as desired.
- Enter a name for the new insert.

If the same tooling is used on more than one operation, the parameters of the resource are not accessible for modification (they are greyed out).

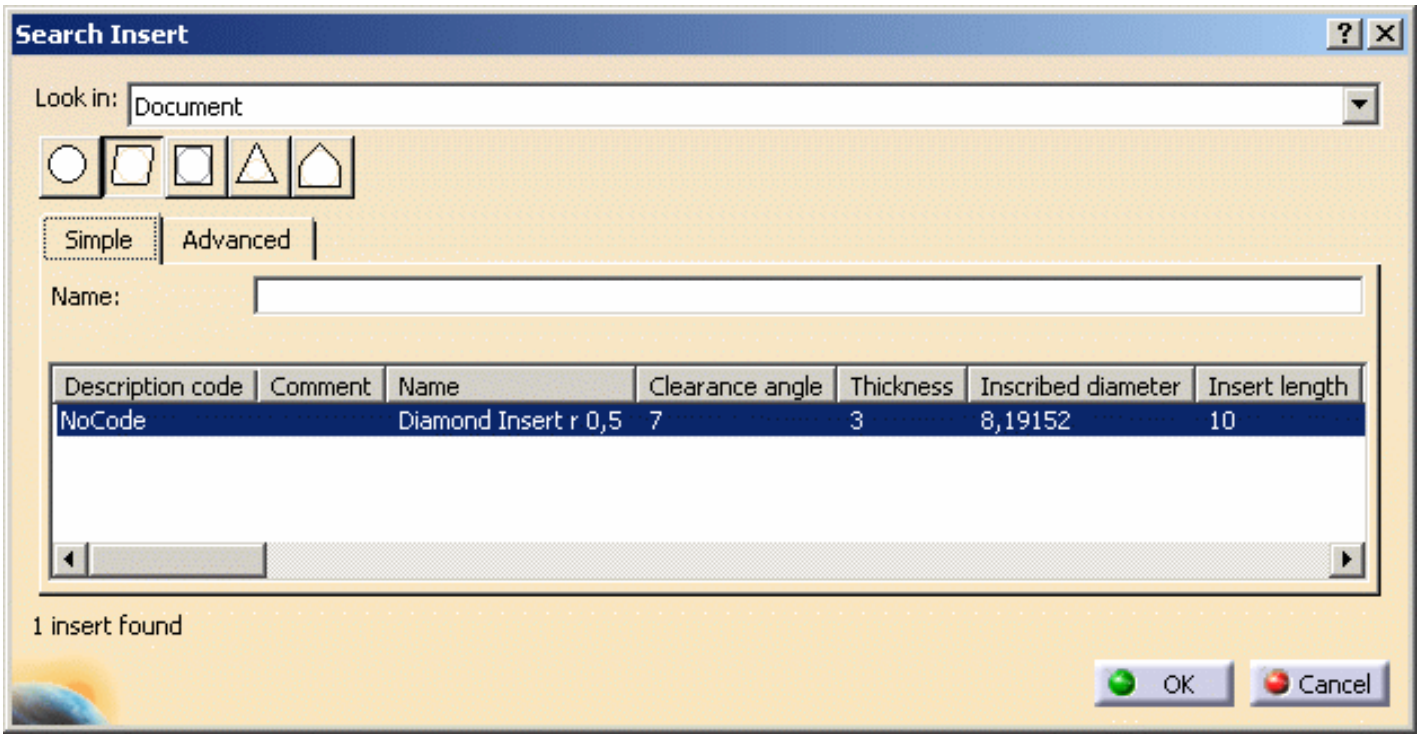
In this case, you can create a new resource by entering a new identifier in the Name field. You can then specify the parameters of the new resource.

3. **To select an insert that is already used in the document:**
   - Select the button opposite Name. Then select the desired insert from the list of inserts already present in your document.
   - In the Name field, enter the name of an insert already present in your document.

The insert representation is displayed in the 2D viewer. It can be edited as described above.

4. To select another insert by means of a query:

- Click the **Select an insert with query** icon opposite Name.
  The Search Tool dialog box appears.



- Use the **Look in** combo to specify where you want to search for the insert:
  - in the current document
  - in a tool catalog
  - in an external tool database such as the TDM (Tool Data Management) or CATIA Version 4 Manufacturing database.

  Note that Cutting conditions (feeds and speeds) can be included in a tool catalog and in the TDM. Please refer Feeds and Speeds for more information.

- If you want to change insert type, select the icon corresponding to the desired insert.
- You can do a quick search in the **Simple** tab page by entering a character string in the Name field. The inserts meeting the simple search criteria are listed.
- Select the desired insert from the list and click OK.
  The insert representation is displayed in the 2D viewer. It can be edited as described above.
- You can search an insert using finer constraints by selecting the **Advanced** tab page.

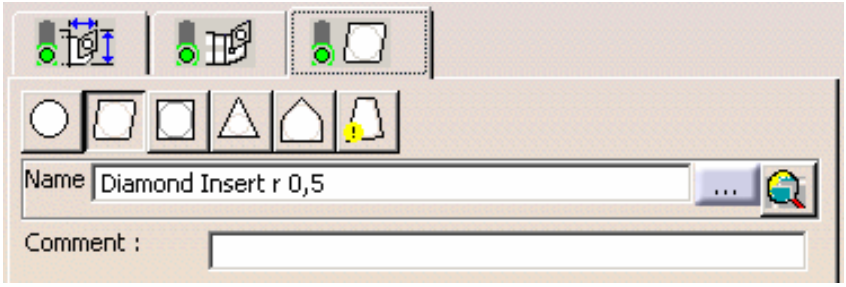
5. Click OK to confirm using this new insert in the operation.

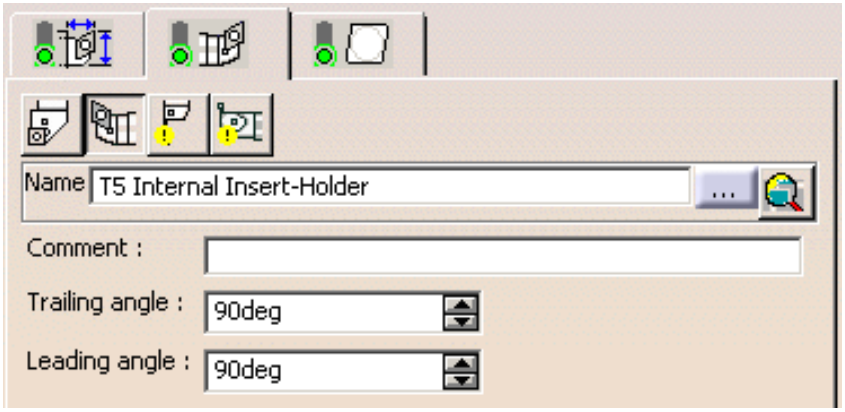
Note that compatibility is checked between the insert and the insert holder.

If there is incompatibily, this is indicated by an exclamation mark symbol on the icon.

In the example below, the selected diamond insert  **is compatible** with the selected Internal insert holder  .

However, it cannot be used on a groove internal insert holder  or a groove external insert holder  .





If you select a groove insert  , the internal insert holder will no longer be referenced by the tool assembly (the status light on the Insert Holder tab turns red). Then you will have to select a compatible insert holder type (such as groove internal insert holder).

Edit an Insert Holder in the Resource List

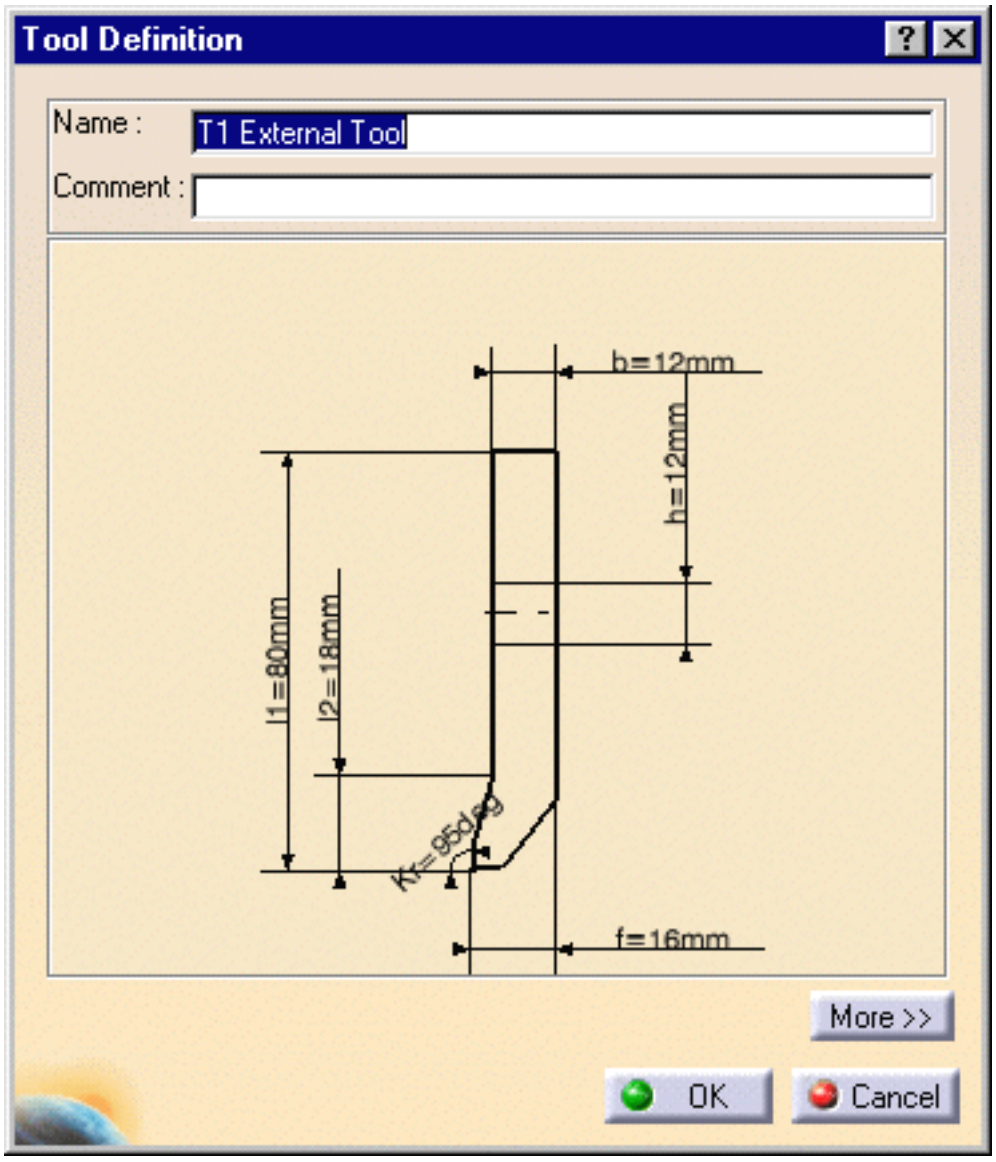
This task shows you how to edit an insert holder that is already used in your document.

Note that if you edit a resource that is used by one or more machining operations, then these operations will be impacted by the modification.

1. Double click the desired insert holder in the Resource List.

   You can also right-click it and select the **Edit NC Resource** contextual command.

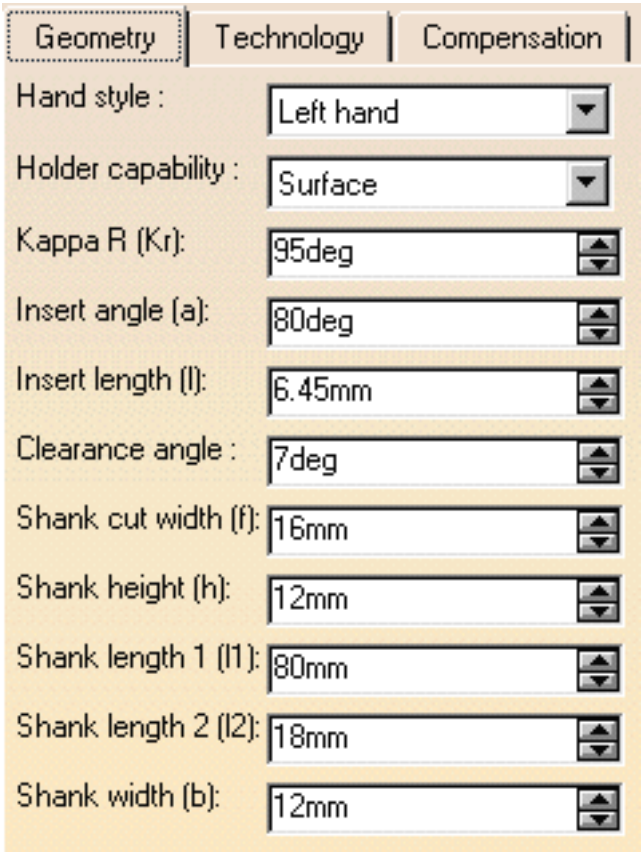
   The Tool Definition dialog box is displayed allowing you to edit the insert holder's geometric, technological, and compensation parameters.



Please refer to Tools/Insert Holders for Turning Operations for a description of the available parameters for this resource.

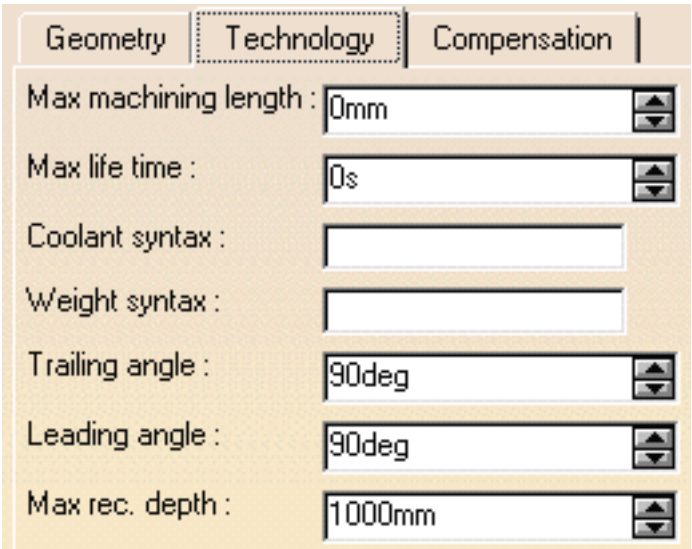
2. If needed, enter a new name and comment for the insert holder.
3. Click More to expand the dialog box to access the Geometry, Technology, and Compensation tabs.

4. You can specify the insert holder geometry in two ways:
   - double click a parameter in the large insert holder icon and enter the desired value in the Edit Parameter dialog box that appears
   - or enter the desired values in the **Geometry** tab page.



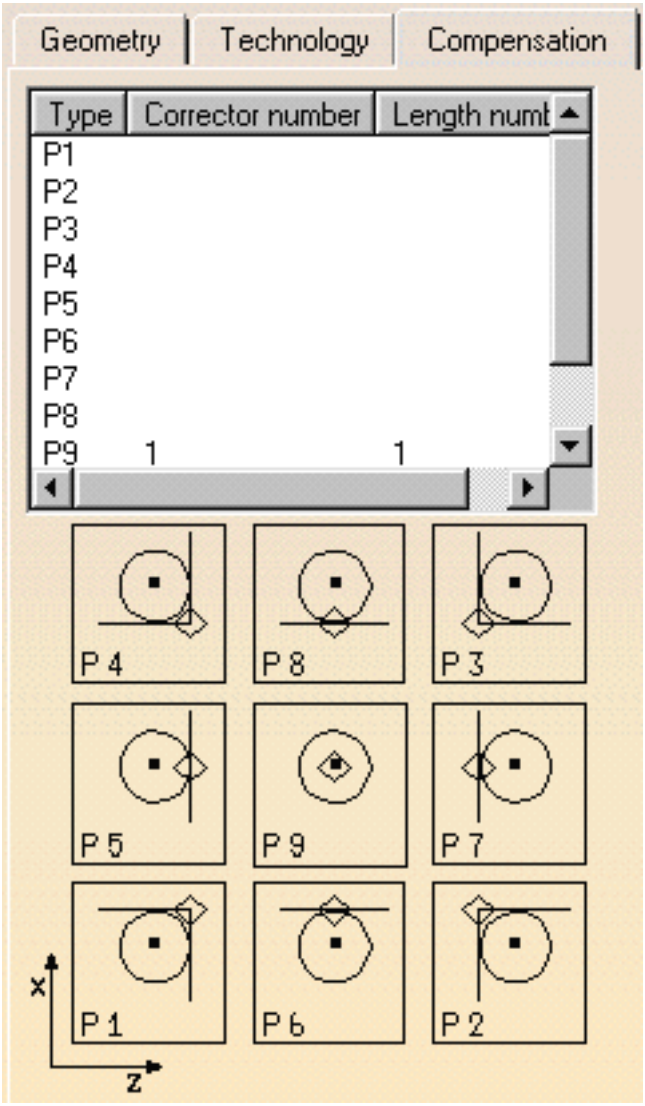
The icon representation of the insert holder is updated with these values.

5. Click the **Technology** tab and enter the desired values for the insert holder's technological parameters.



6. If cutter compensation is required, click the **Compensation** tab.

   You can either edit an existing compensation site or add another site, if other sites are proposed.



7. Right click the desired line to either edit or add cutter compensation data.

   The Compensation Definition dialog box appears.

8. Enter the desired values for the cutter compensation sites.

   See Specify Tool Compensation for more information.

9. Click OK to accept the modifications made to the insert holder.

   A CATPart or CATProduct representation can be assigned to the insert holder by means of the **Add User Representation** contextual command in the Resource List.

   A mask symbol on the bottom right of the Insert Holder icon in the PPR tree indicates that a user representation has been added to the resource. For example:

   For more information about user representations, please refer to User Representation of Lathe Tools.

Edit a Tool Assembly in the Resource List

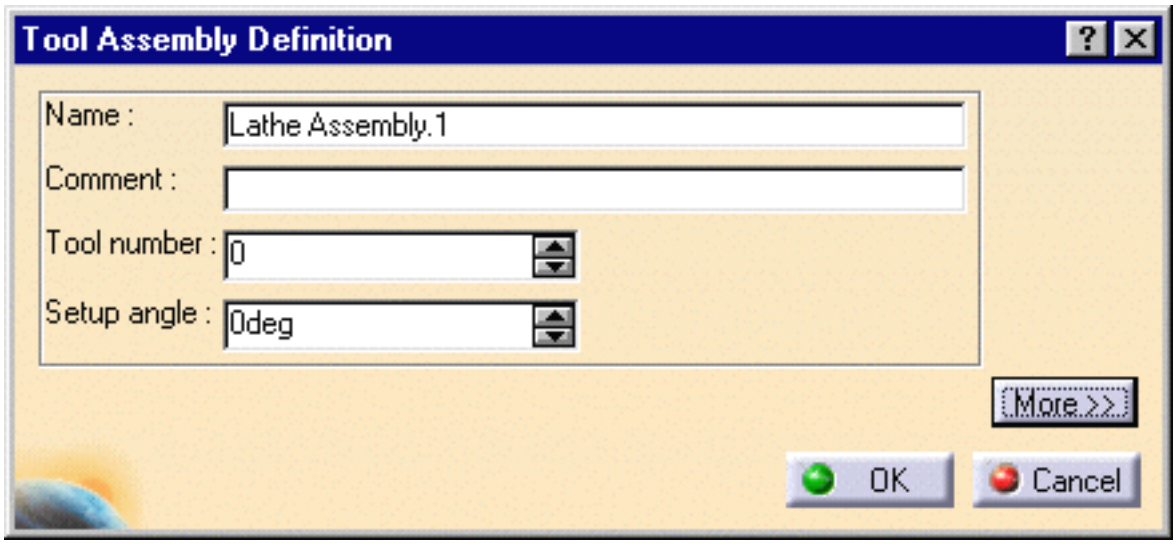
This task shows you how to edit a tool assembly that is already used in your document.

Note that if you edit a resource that is used by one or more machining operations, then these operations will be impacted by the modification.

1. Double click the desired tool assembly in the Resource List.

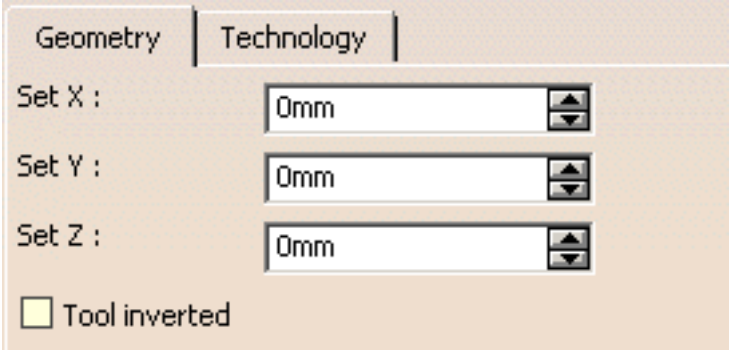
   You can also right-click it and select the **Edit NC Resource** contextual command.

   The Tool Assembly Definition dialog box is displayed allowing you to edit the tool assembly's geometric and technological parameters.

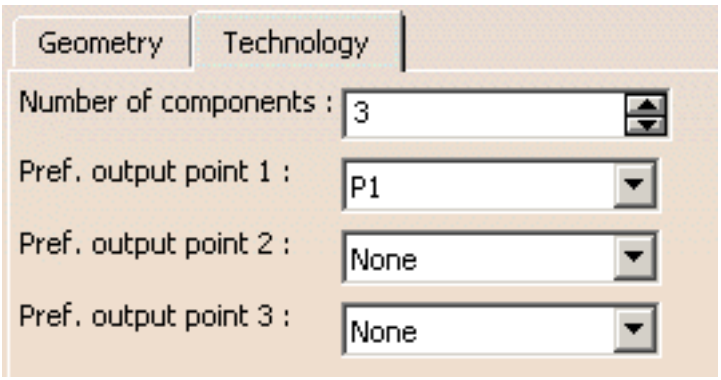


Please refer to Lathe Assembly Resources for a description of the available parameters.

2. If needed, enter a new name and comment for the tool assembly.
3. If needed, use the spinner to change the Tool number. This parameter identifies tool in the tool storage magazine.
4. Click More to expand the dialog box to access the Geometry and Technology tabs.
5. You can specify the tool assembly geometry in two ways:
   - double click a parameter in the large tool assembly icon and enter the desired value in the Edit Parameter dialog box that appears
   - or enter the desired values in the **Geometry** tab page.



6. Click the **Technology** tab and enter the desired values for the tool assembly's technological parameters.

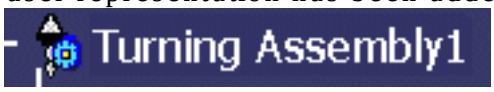


7. Click OK to accept the modifications made to the tool assembly.

A CATPart or CATProduct representation can be assigned to the tool assembly by means of the **Add User Representation** contextual command in the Resource List.

If a component (insert holder or insert) of the assembly has a user representation, then that representation will not be taken into account. The assembly representation has priority.

A mask symbol on the bottom right of the Tool Assembly icon in the PPR tree indicates that a user representation has been added to the resource. For example:



For more information about user representations, please refer to User Representation of Lathe Tools.

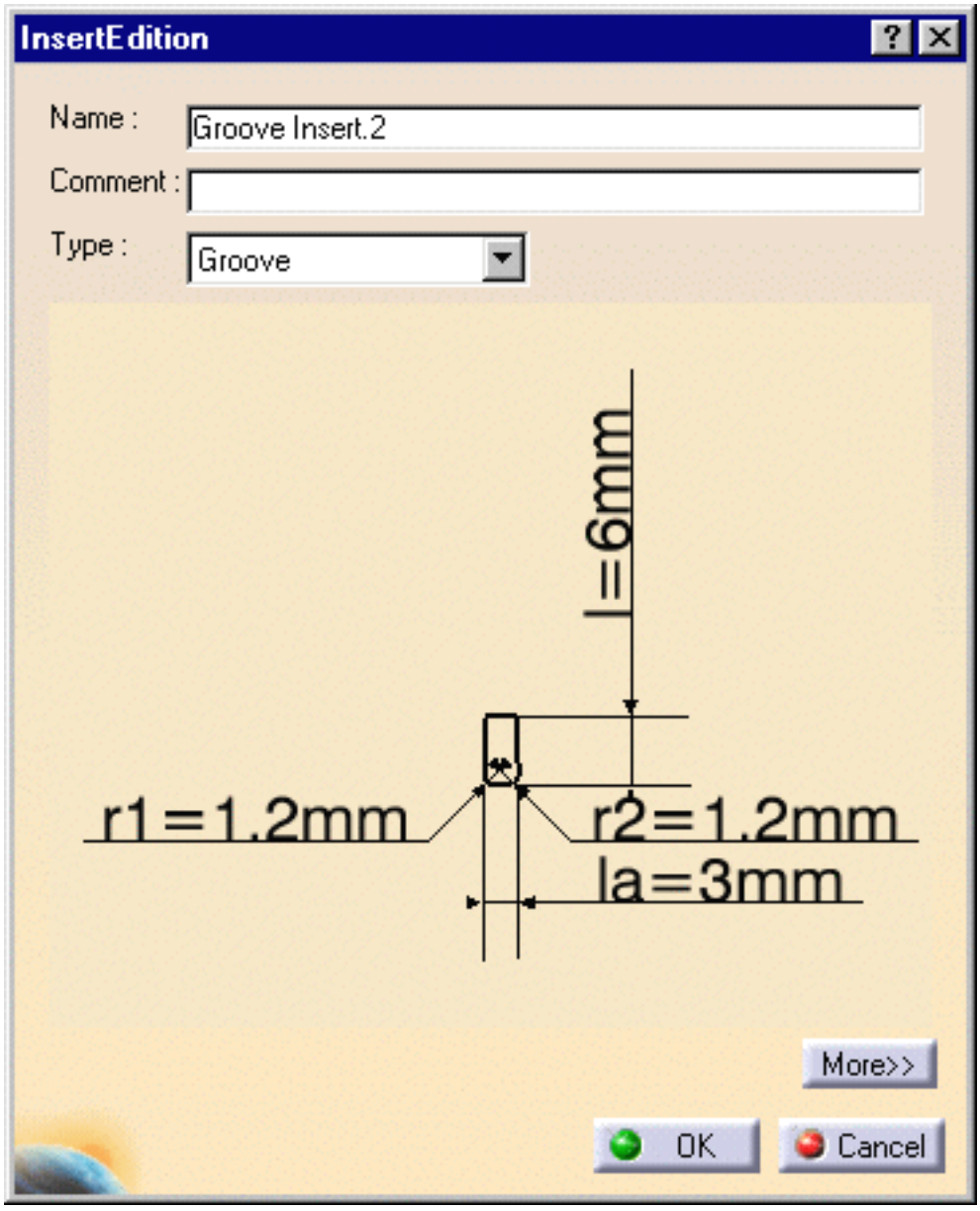
Edit an Insert in the Resource List

This task shows you how to edit a lathe insert that is already used in your document.

1. Right-click the desired insert in the resource list, and select the **Edit NC Resources** contextual command.

   Note that if you edit a resource that is used by one or more machining operations, then these operations will be impacted by the modification.

   The Insert Definition dialog box is displayed allowing you to edit the insert's characteristics.



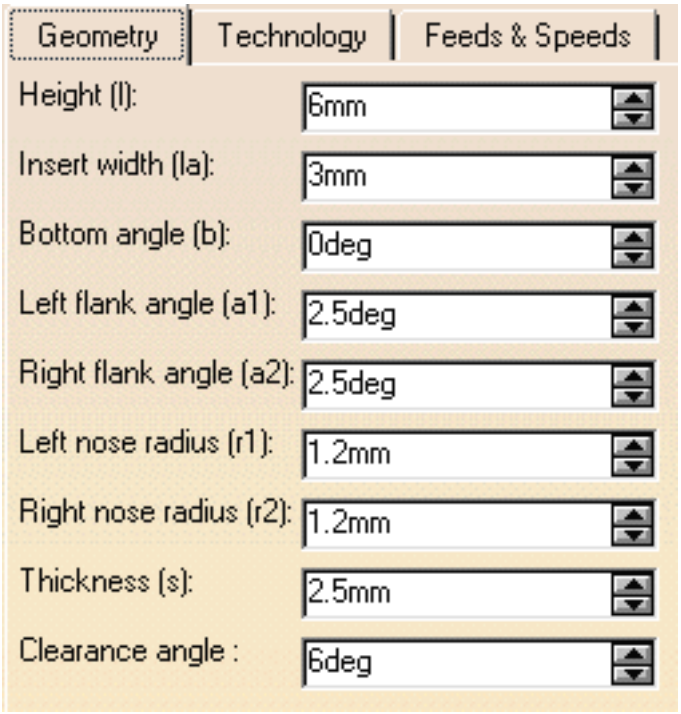
2. If needed, enter a new name for the insert.

   You can also assign a comment.

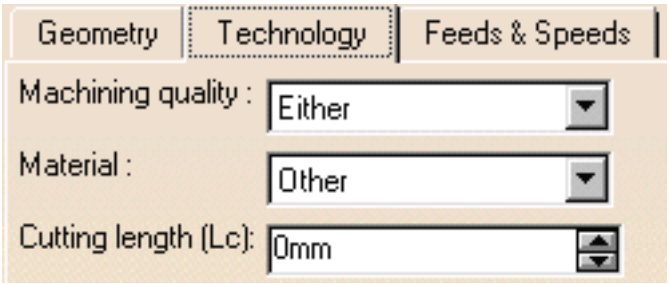
3. If needed, use the spinner to change the Tool number.
4. Click More to expand the dialog box to access the Geometry, Technology, and Feeds & Speeds tabs.

Please refer to Insert Resources for a description of the available parameters.

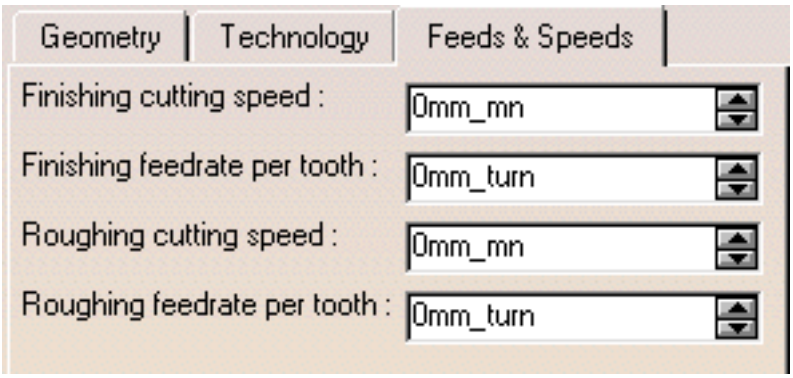
5. You can specify the insert geometry in two ways:
   - double click a parameter in the large icon representing the insert and enter the desired value in the Edit Parameter dialog box that appears
   - or enter the desired values in the **Geometry** tab page.



6. Click the **Technology** tab and enter the desired values for the insert's technological parameters.



7. Click the **Feeds and Speeds** tab and enter the desired values for the insert's feed and speed parameters.



8. Click OK to accept the modifications made to the insert.

A CATPart or CATProduct representation can be assigned to the insert by the **Add User Representation** contextual command in the Resource List.

A mask symbol on the bottom right of the Insert Holder icon in the PPR tree indicates that a user representation has been added to the resource. For example:

For more information about user representations, please refer to User Representation of Lathe Tools.

Verification, Simulation and Program Output

The tasks for using capabilities such as tool path verification, material removal simulation, and production of NC output data are documented in the *NC Manufacturing Infrastructure User's Guide*.


Replay Tool Path: Select the Tool Path Replay icon then specify the display options for an animated tool path display of the manufacturing program of machining operation.


Simulate Material Removal **(P2 functionality)**: Select the Video icon in the Tool Path Replay dialog box to run a material removal simulation in Video mode.

- Generate APT Source Code in Batch Mode: Select the Generate NC Code in Batch Mode icon then select the manufacturing program to be processed and define the APT source processing options.
- Generate NC Code in Batch Mode: Select the Generate NC Code in Batch Mode icon then select the manufacturing program to be processed and define the NC code processing options.
- Generate Clfile Code in Batch Mode: Select the Generate NC Code in Batch Mode icon then select the manufacturing program to be processed and define the Clfile processing options.
- Generate a CATProduct In-Process Model in Batch Mode **(P2 functionality)**: Select the Generate NC Code in Batch Mode icon then select the manufacturing program to be processed and specify the processing options for generating a CATProduct in-process model.
- MfgBatch Utility that allows you to generate NC data files from a manufacturing program by means of an executable program under Windows or a shell under UNIX.


Generate NC Code in Interactive Mode: Select the Generate NC Code Interactively then select the manufacturing program to be processed and define processing options.


Batch Queue Management: Manage tool path computation outside the interactive CATIA session, with the possibility of scheduling the execution of several batch jobs.

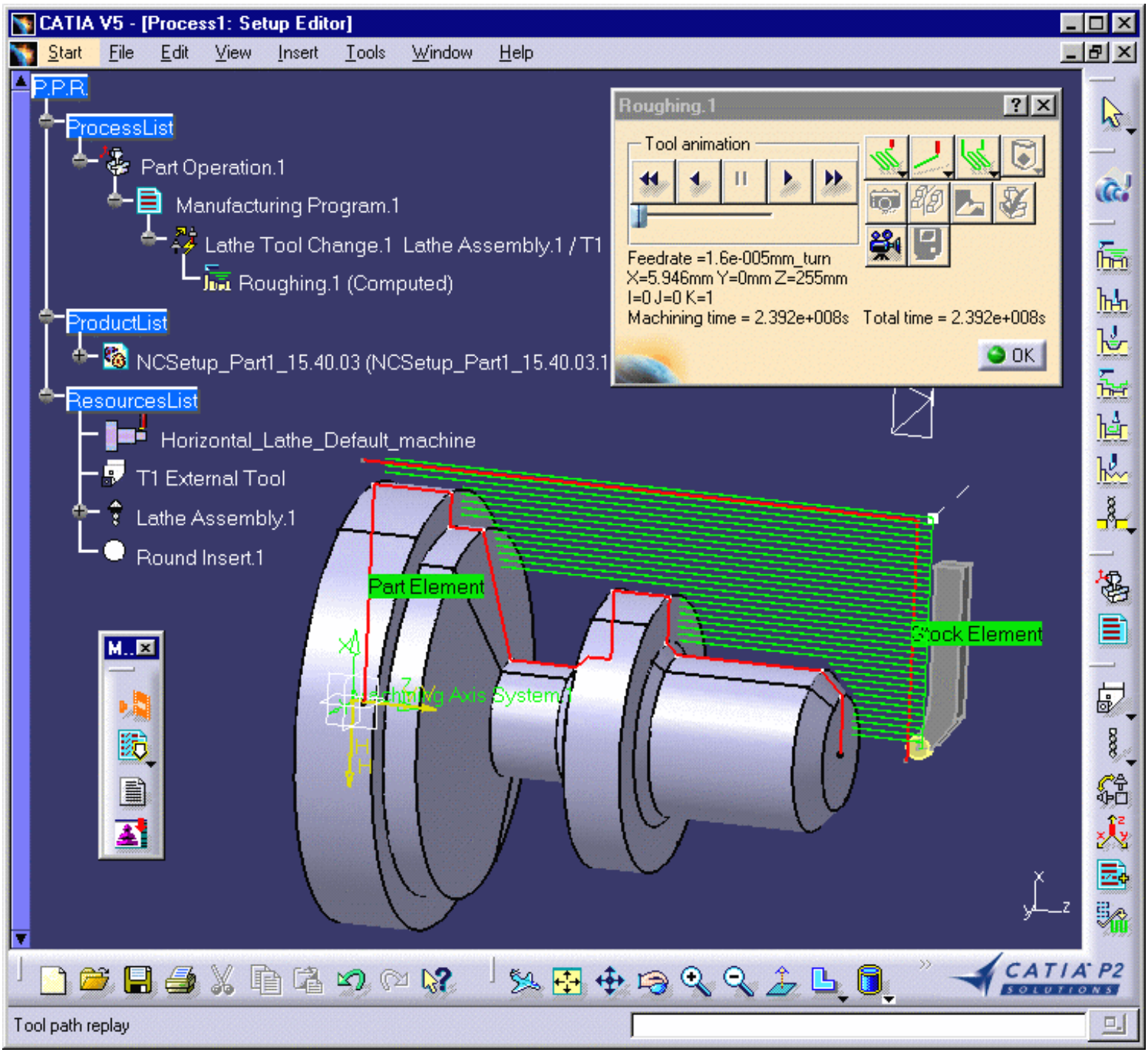

Generate Documentation: Select the Generate Documentation icon to produce shop floor documentation in HTML format.

Import an APT Source into the Program: Select the APT Import contextual command to insert an existing APT source into the current manufacturing program.

Workbench Description

This section contains the description of the menu commands and icon toolbars that are specific to the Lathe Machining workbench.

Menu Bar
Toolbars
Specification Tree



Lathe Machining Menu Bar

The menu commands that are specific to Lathe Machining are described below.

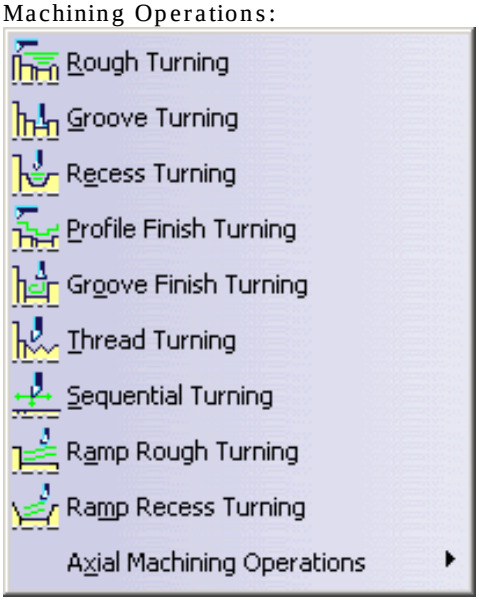
Start | File | Edit | View | Insert | Tools | Windows | Help

Tasks corresponding to general menu commands are described in the *CATIA Version 5 Infrastructure User's Guide*.

Tasks corresponding to menu commands that are common to all Machining products are described in the *NC Manufacturing Infrastructure User's Guide*.

Insert Menu

Machining Operations:

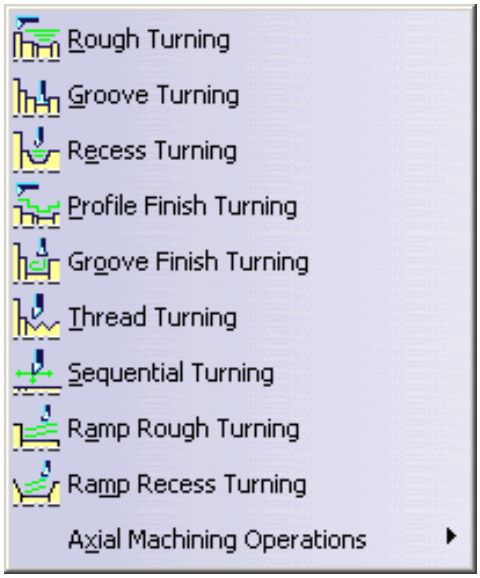


Auxiliary Operations:

Machining Features:

Insert > Machining Operations

Command...	Description...
Rough Turning	Create a Longitudinal Rough Turning Operation Create a Parallel Contour Rough Turning Operation Create a Face Rough Turning Operation
Groove Turning	Create a Groove Turning Operation
Recess Turning	Create a Recess Turning Operation



Profile Finish Turning Create a Profile Finish Turning Operation

Groove Finish Turning Create a Groove Finish Turning Operation

Thread Turning Create a Thread Turning Operation

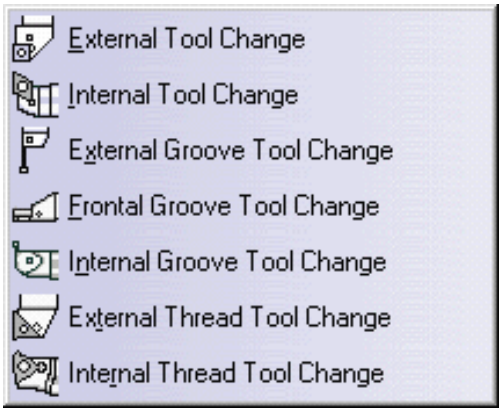
Sequential Turning Create a Sequential Turning Operation

Ramp Rough Turning Create a Ramp Rough Turning Operation

Ramp Recess Turning Create Ramp Recess Turning Operation

Axial Machining Operations Create Axial Machining Operations

Insert > Auxiliary Operations > Turning Tool Change



Description...

Allows inserting turning tool changes in the program.

Lathe Machining Toolbars

The Lathe Machining workbench includes a number of icon toolbars, some of which are common to all NC workbenches and some of which are specific to Lathe Machining. The common toolbars are described in the Toolbars section of the *NC Manufacturing Infrastructure User's Guide*.

The following toolbar is specific to the Lathe Machining workbench.

It contains commands to create and edit turning operations as follows.


Create a Rough Turning operation.
Basic tasks illustrate the following roughing modes:

- Longitudinal
- Face
- Parallel Contours.


Create a Recess Turning operation.


Create a Groove Turning operation.


Create a Profile Finish Turning operation.


Create a Groove Finish Turning operation.


Create a Thread Turning operation.


Create a Sequential Turning operation.


Create a Ramp Rough Turning operation.


Create a Ramp Recess Turning operation.


Create Axial Machining Operations.

The following specific toolbar is accessed from the drop-down icon in the Auxiliary Operations toolbar.

It contains icons for creating and editing Turning Tool Change operations as follows.


See External tool for more information about this resource


See Internal tool for more information about this resource


See External Groove tool for more information about this resource


See Frontal Groove tool for more information about this resource

 See Internal Groove tool for more information about this resource

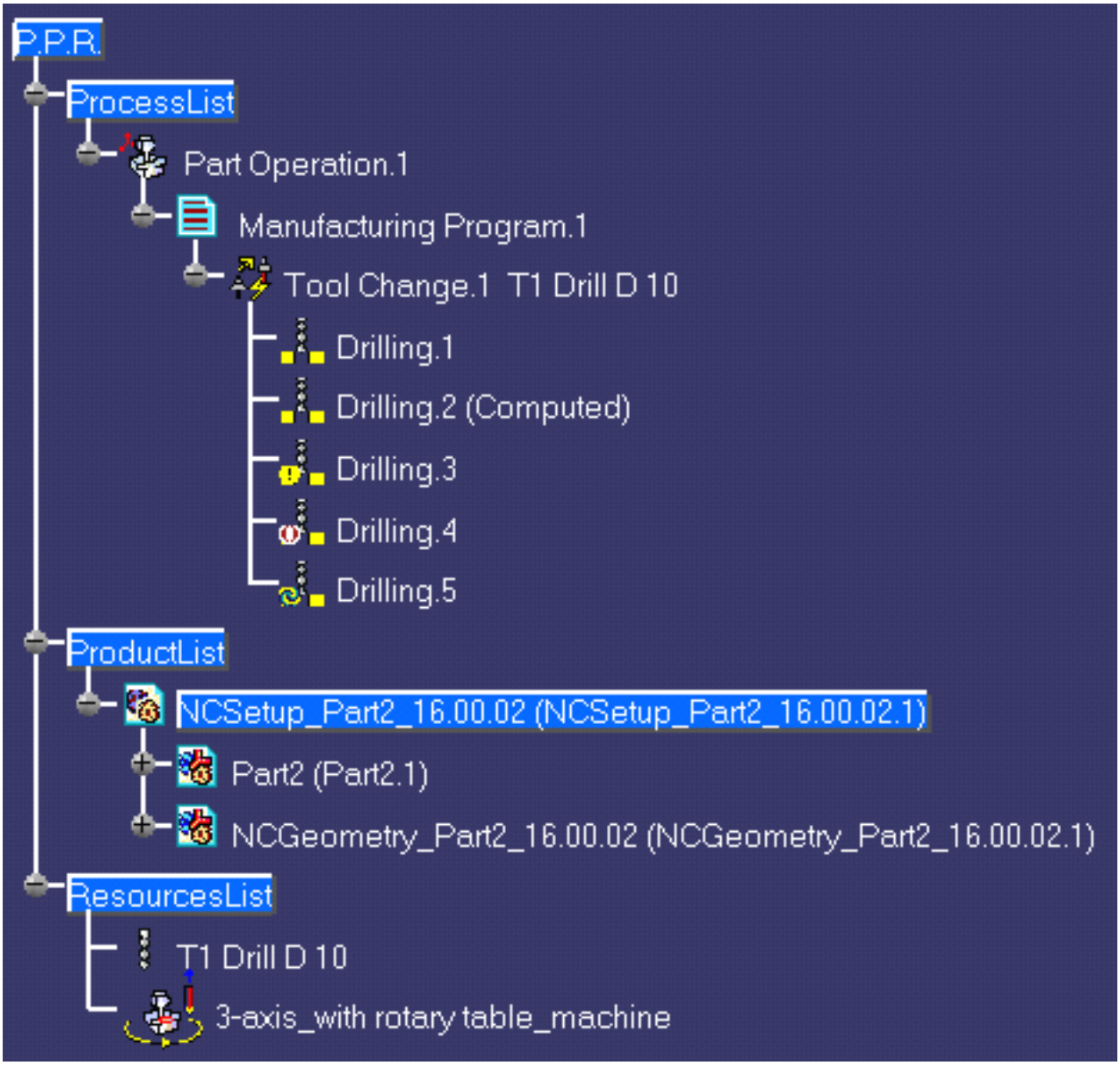
 See External Thread tool for more information about this resource

 See Internal Thread tool for more information about this resource.

Please note that the icon representing a Tool Change operation in the PPR tree looks like this:  .

Specification Tree

Here is an example of a Process Product Resources (PPR) specification tree for Machining products.



Process List is a plan that gives all the activities and machining operations required to transform a part from a rough to a finished state.

- **Part Operation** defines the manufacturing resources and the reference data.
- **Manufacturing Program** is the list of all of the operations and tool changes performed. The example above shows that:
  - Drilling.1 is complete and has not been computed
  - Drilling.2 is complete but has been computed (by means of a replay)
  - Drilling.3 does not have all of the necessary data (indicated by the exclamation mark symbol)
  - Drilling.4 has been deactivated by the user (indicated by the brackets symbol)
  - Drilling.5 has been modified and needs to be recomputed (indicated by the update symbol).

Product List gives all of the parts to machine as well as CATPart documents containing complementary geometry.

Resources List gives all of the resources such as machine or tools that can be used in the program.

Customizing

This section describes how to customize settings for Machining.

Before you start your first working session, you can customize the settings to suit your working habits. Your customized settings are stored in permanent setting files: they will not be lost at the end of your session.

Other tasks for customizing your Machining environment are documented in the *NC Manufacturing Infrastructure User's Guide*:

Build a Tools Catalog
Access External Tools Catalogs
Add User Attributes on Tool Types
PP Word Syntaxes
NC Documentation
Workbenches and Tool Bars

1. Select **Tools** > **Options** from the menu bar: the Options dialog box appears.
2. Select the **Machining** category in the tree to the left. The options for Machining settings appear, organized in tab pages.

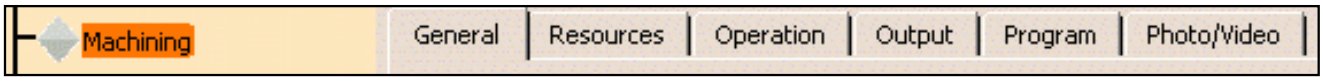
3. Select the tab corresponding to the parameters to be customized.

Parameters in this tab...	Allow you to customize...
General	general settings for all Machining products
Resources	tooling, feeds&speeds and resource files
Operation	machining operations
Output	PP files and NC data output
Program	manufacturing programs (sequencing, and so on)
Photo/Video	material removal simulation

4. Set these options according to your needs.
5. Click **OK** to save the settings and quit the Options dialog box.

General

This document explains how to customize general settings for Machining products.

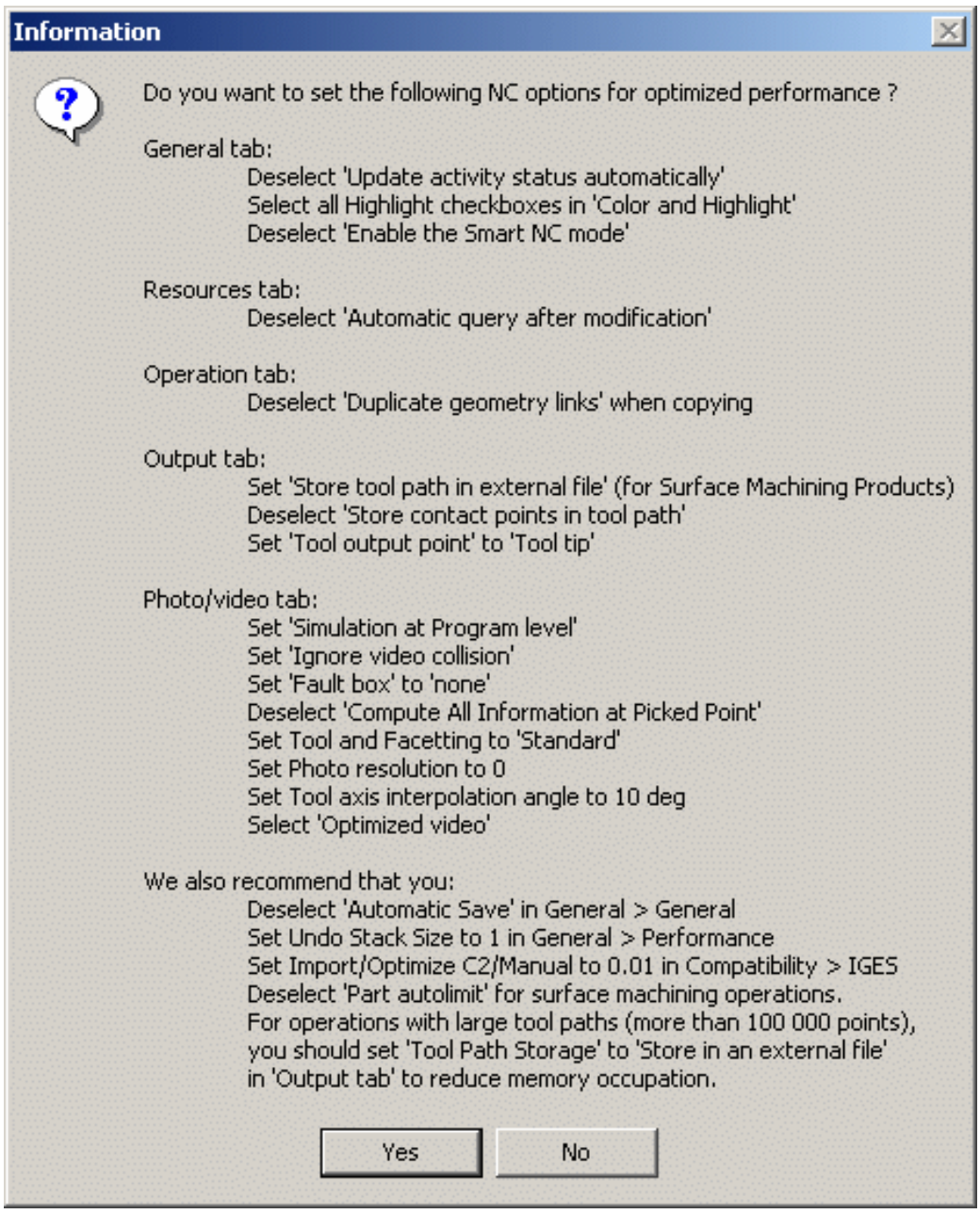


Select the **General** tab, which is divided up into areas.

Parameters in this area...	Allow you to customize...
Performance	settings for optimized performance
Tree Display	display of the specification tree
Color and Highlight	colors of displayed geometry and parameters
Tool Path Replay	tool display during tool path replay
Complementary Geometry	handling of geometry necessary for manufacturing
Design Changes	use of the Smart NC mode and enhanced detection of design changes.

Performance

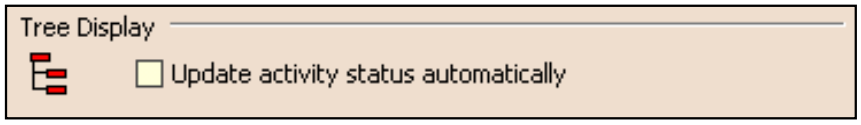
Click the **Optimize** button in order to automatically set a number of the Machining options for optimized performance. These options are listed in the Information dialog box that appears:



If you click **Yes**, these options will be set as described in the dialog box. Note that, if needed, you may locally reset any of these options.
If you click **No**, the options will remain with their current settings.

The Information box also lists some recommendations for manually setting other options that have an influence on performance.

Tree Display

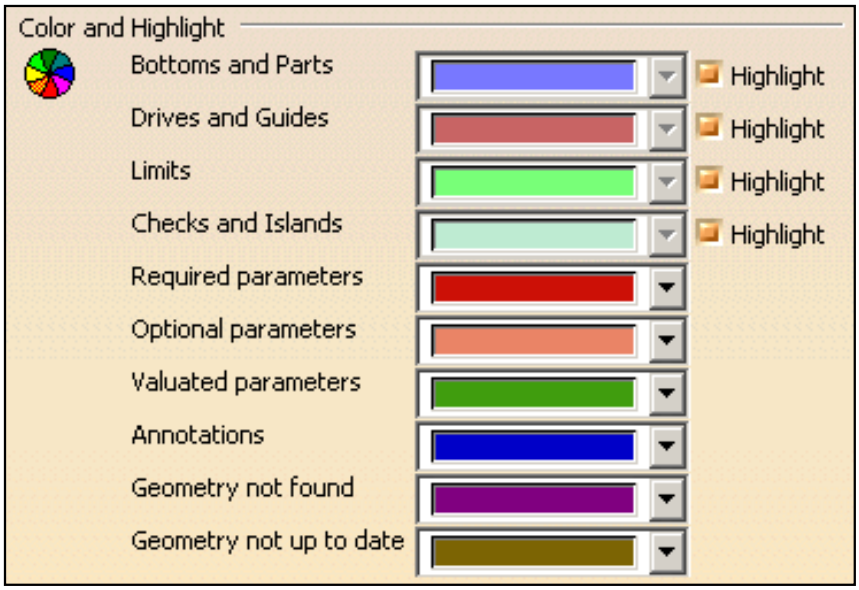


- Select the checkbox if you want the status of activities in the tree to be updated automatically.
- If this checkbox is not selected:
  - you can update activity status manually in your workbench using the Update Status icon  in the Auxiliary Commands toolbar.
  - the status of the activity after a manual update is masked at the first action on the node (for example, edit, replay, collapse/expand of a parent node). To retrieve the status of the activity you must select the Update Status icon again.

If this checkbox is not selected, performance is improved.

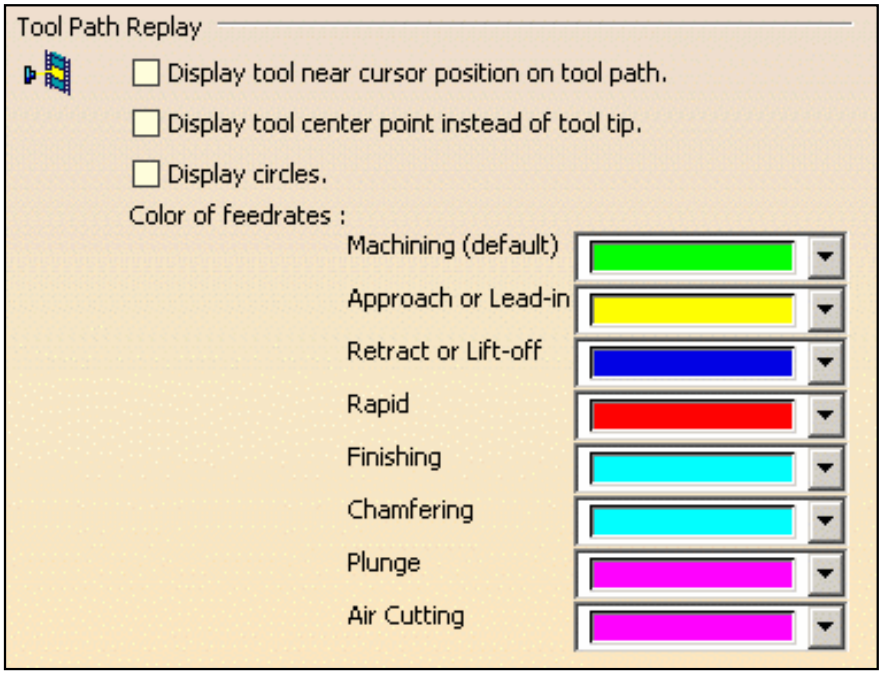
By default, the checkbox is not selected.

Color and Highlight



- Select the colors to be used for identifying the various manufacturing entities by means of the combos.
  Note that for Geometry that is not found or not up to date, you can select the colors used to display the valuated parameters in the corresponding Operation or Feature dialog boxes.
- For certain entities, you can select the corresponding checkbox to use highlighting.
  Performance is improved when all the Highlight checkboxes are selected.

Tool Path Replay



Display tool near cursor position on tool path

Select this checkbox if you want to display the tool near your cursor position on the trajectory during a tool path replay. You can display the tool at a specific point by clicking on the tool path. The tool will then be positioned on the nearest computed point on the trajectory.

Display tool center instead of tool tip

Select this checkbox if you want to display the tool center point instead of the tool tip during a tool path replay.

Display circles

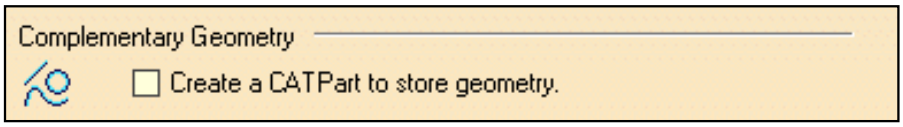
Select this checkbox if you want to display each circular trajectory as a circular arc instead of a set of discretization points. The extremities of the circular arc are indicated by means of 'O' symbols.
This allows better control of the Point by Point replay mode, where it is necessary to make several interactions to replay a circle (because of its representation by a set of points). With the graphic representation as a circle, only one interaction is necessary to perform the replay.

By default, these checkboxes are not selected.

Color of feedrates

Select the colors to be used for identifying the various feedrate types by means of the combos. The selected colors will be displayed in the **Different colors** replay mode.

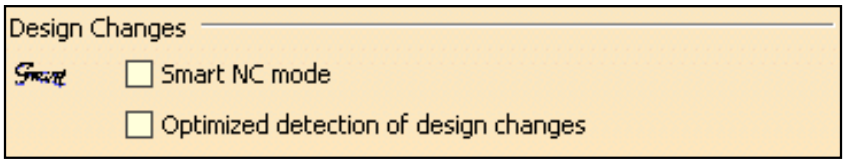
Complementary Geometry



Select the checkbox to create a CATPart dedicated to manufacturing-specific geometry in the Product List of the PPR tree.

By default, the checkbox is not selected.

Design Changes



Smart NC mode

Select this checkbox to activate the Smart NC mode. In this mode, an image of the geometry selected in machining operations is kept to allow analysis of design changes.
Performance is improved when this checkbox is not selected.

Optimized detection of design changes

Select this checkbox to enable a geometrical comparison mode in order to more precisely determine the design change status of machining operations.

By default, these checkboxes are not selected.

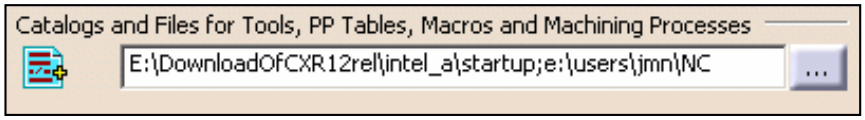
Resources

This document explains how to customize resource settings for Machining products.

Select the **Resources** tab, which is divided up into areas.

Parameters in this area...	Allow you to customize...
Catalogs and Files	the path name for resource files
Tool Selection	the selection of tools
Automatic Compute from Tool Feeds and Speeds	the update of feeds and speeds according to tooling data
Tool Query Mode in Machining Processes Instantiation	tool queries in machining processes

Catalogs and Files



Enter the path of the folder containing tool catalogs, PP tables, macros, and machining processes. You can choose a folder by clicking the **[...]** button.

You can concatenate paths using:

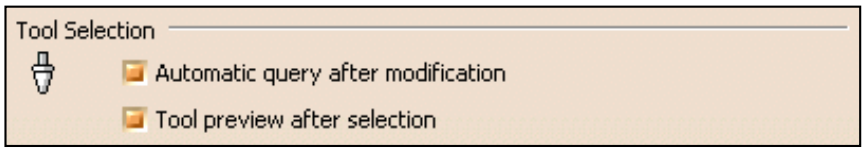
- a semi colon (;) character for Windows platforms
- a colon (:) character for UNIX platforms.

For example, if the concatenated folders E:\DownloadOfCXR12rel\intel_a\startup and e:\users\jmn\NC in the figure above contain PP tables, then those PP tables will be available for selection in the Part Operation's Machine Editor dialog box.

Please note that:

- PP tables must be contained in folders named Manufacturing\PPTables
- tools must be contained in folders named Manufacturing\Tools.

Tool Selection



Automatic query after modification

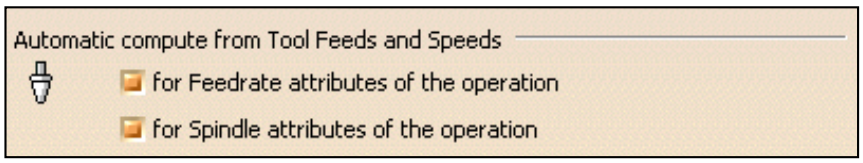
Select this checkbox if you want to to activate an automatic query after each modification of a tool parameter. Performance is improved when this checkbox is not selected.

Tool preview after selection

Select this checkbox if you want to preview the tool after selection.

By default, these checkboxes are selected.

Automatic Compute from Tool Feeds and Speeds



Feedrate attributes of the operation

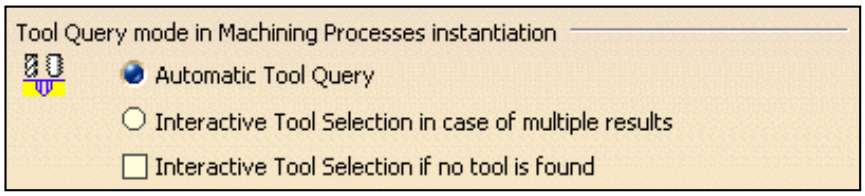
Select this checkbox if you want the Automatic Update of Feedrates option to be set by default in the Feeds and Speeds tab page of machining operations.
This option allows feedrates of operations to be automatically updated whenever feedrate information on the tool is modified.

Spindle attributes of the operation

Select this checkbox if you want the Automatic Update of Speeds option to be set by default in the Feeds and Speeds tab page of machining operations.
This option allows spindle speeds of operations to be automatically updated whenever speed information on the tool is modified.

By default, these checkboxes are selected.

Tool Query mode in Machining Processes Instantiation



Select the type of Tool Query to be executed when a Machining Process is instantiated:

- automatically computed Tool Query
- interactively defined Tool Selection in case of multiple results
- interactively defined Tool Selection if no tool is found.

Depending on the selected option, the Advanced tab page of the Search Tool dialog box shows the solved Tool Query for each operation in the Machining Process.

By default, the **Automatic Tool Query** option is selected.
In the example below, you can choose one of the tools found in the ToolsSampleMP, or use the **Look in** combo to select a tool from the current document or another tool catalog.

Search Tool

Look in: ToolsSampleMP

Simple | Advanced

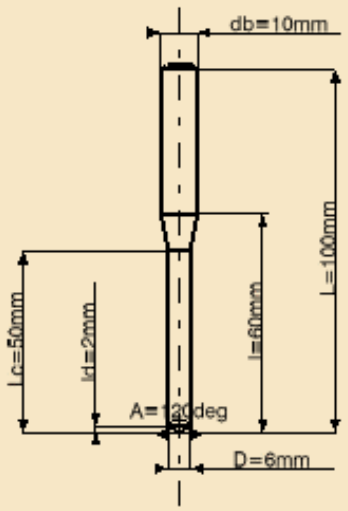
Search with criteria:

Nominal diameter<=10mm
Nominal diameter>=5mm

Delete | Clear all

Attribute: | Condition: | Value:

Tool number	Comment	Name	Cutting angle	Length	Nominal diameter
10	Drill Tool	Drill D6	120	60	6
11	Drill Tool	Drill D6,5	120	60	6
12	Drill Tool	Drill D8	120	60	8
13	Drill Tool	Drill D8,5	120	60	8
14	Drill Tool	Drill D10	120	60	10
15	Drill Tool	Drill D10,5	120	60	10



6 tool(s) found

OK | Cancel

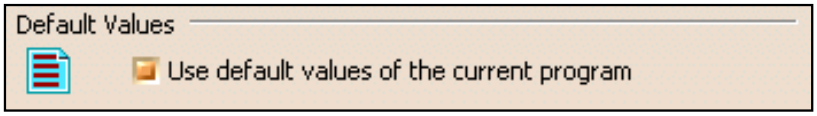
Operation

This document explains how to customize machining operation settings for Machining products.

Select the **Operation** tab, which is divided up into areas.

Parameters in this area...	Allow you to customize...
Default Values	the use of default values
After Creation or Machining Process (MP) Instantiation	what happens after creating machining operations or machining processes
When Copying	the duplication of geometry links
Display	tool path displays of operations
User Interface	dialog boxes of 3-axis surface machining operations.

Default Values

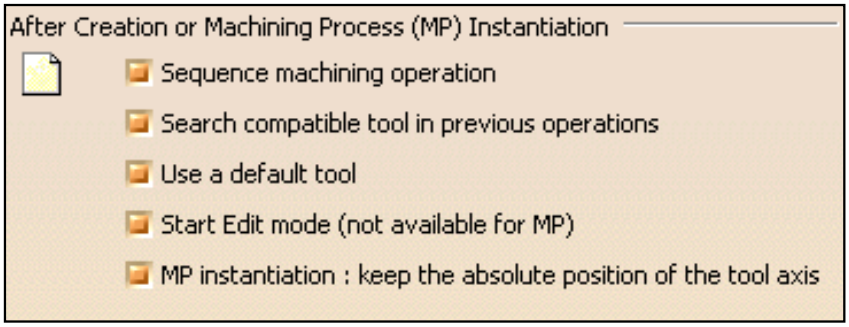


Select the checkbox if you want operations to be created with the values used in the current program.
The values and units of attributes at the creation step of an operation are set to the values and units of the last edited and validated operation whatever its type (that is, exit the operation definition dialog box using OK).

Otherwise the default settings delivered with the application are used.

By default, this checkbox is selected.

After Creation or Machining Process (MP) Instantiation



Select the desired checkboxes to specify conditions to be applied when you create machining operations or machining processes.

Sequence machining operation

Machining operations are automatically sequenced in the current program after creation.
Otherwise, sequencing can be managed in the feature view.

Search compatible tool in previous operations

When creating an operation, if a compatible tool exists in a previous operation of the current program, it will be set in the

new operation.
Otherwise, the operation will be incomplete.

Use a default tool

When creating an operation, a search is done in the document to find a compatible tool. If no compatible tool exists, a default one is created in the document and set in the created operation.
If checkbox is not selected, no tool will be defined on the operation.

Start edit mode (not available for machining processes)

When creating a machining operation, Edit mode is automatically started to allow modifying parameters of the created operation.
Otherwise, the operation is added to the program but the machining operation editor is not started.

MP instantiation: keep the absolute position of the tool axis

When a machining process is instantiated, the tool axes of the activites in the MP keep their absolute positions. Otherwise, if the checkbox is not selected, these positions are changed in order to keep the relative components of the tool axis.

By default, these checkboxes are selected.

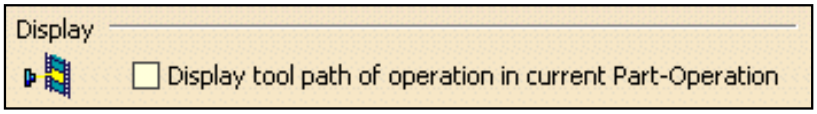
When Copying

Select the checkbox if you want geometry links to be duplicated in a copied operation.

Otherwise the geometry must be defined for the copied operation. Performance is improved when this checkbox is not selected.

By default, this checkbox is selected.

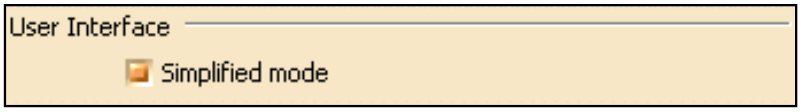
Display



Select the checkbox if you want to display tool paths of operations in the current Part Operation.

By default, this checkbox is not selected.

User Interface



Select the checkbox if you want to have the possibility of simplifying the dialog boxes of machining operations (that is, you can display the minimum number of parameters necessary for a correct tool path). This setting is available for 3-axis surface machining operations only.

By default, this checkbox is not selected.

Output

This document explains how to customize data output settings for Machining products.

Select the **Output** tab, which is divided up into areas.

Parameters in this area...	Allow you to customize...
Post Processor	the type of PP files to be used for generating NC code output and the path where these files are located
Tool Path Storage	the tool path storage capability
Tool Path Edition	the tool path edition capability
During Tool Path Computation	contact point storage
Tool Output Point	type of tool output point
Tool Output Files ... Location	default paths for NC output files storage.

Post Processor

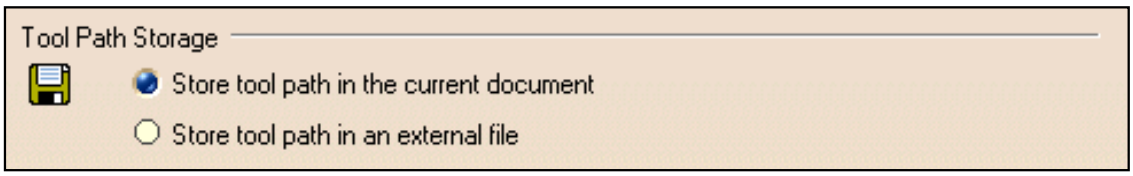
Select the desired Processor option:

- None: no Post Processor is defined. NC code output is not possible in this case
- Cenit: you can choose from among the Post Processor parameter files proposed by Cenit to generate your NC code
- IMS: you can choose from among the Post Processor parameter files proposed by Intelligent Manufacturing Software (IMS) to generate your NC code
- ICAM: you can choose from among the Post Processor parameter files proposed by ICAM Technologies Corporation (ICAM) to generate your NC code.

Enter the path of the folder containing Post processors. You can choose a folder by clicking the **[...]** button. File concatenation is possible.

By default, the **None** option is selected.

Tool Path Storage

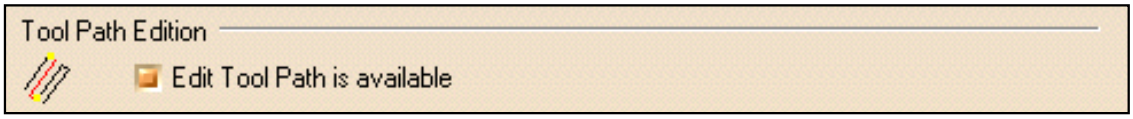


Select the desired option to store tool path data either in the current document or in an external file (as a tpl file).

For operations with large tool paths (more than 100 000 points), tool path storage in an external file is recommended.

By default, the **Store tool path in the current document** option is selected.

Tool Path Edition

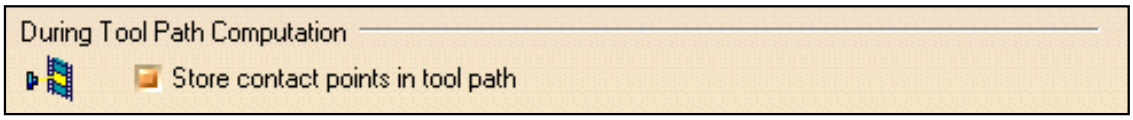


Select the checkbox if you want to be able to edit tool paths even when the operation is locked.

This capability is available only for activities with a tool path node in the specification tree.

By default, this checkbox is selected.

During Tool Path Computation

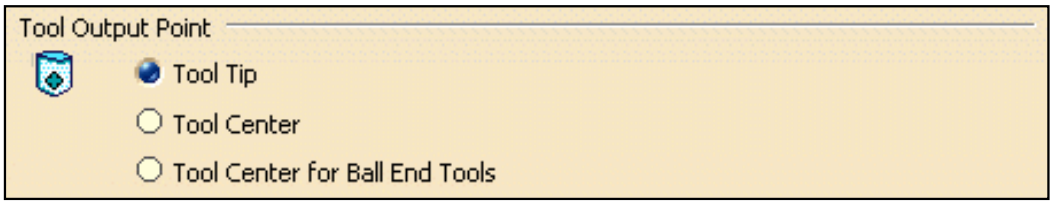


Select the checkbox if you want to store contact points in the tool path.

Performance is improved when this checkbox is not selected.

By default, this checkbox is selected.

Tool Output Point



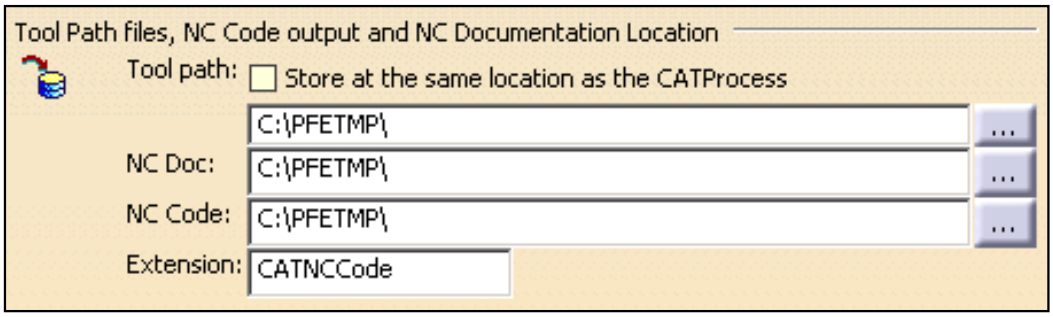
Select the desired option to select one of the following as output point:

- tool tip
- tool center point
- tool center point for ball end tools (that is, any tool with the Ball-end tool attribute selected or an end mill whose nominal diameter is equal to twice the corner radius).

Performance is better when the Tool Tip option is selected.

By default, the **Tool Tip** option is selected.

Default File Locations



Specify default locations for storing **Tool Path files**, **NC Documentation**, and **NC Code output**.

You can store tool paths files (tpl files) in the same folder as the CATProcess by selecting the checkbox. This allows you to store these files according to your CATProcess context. Otherwise, you can choose another location by clicking the **[...]** button.

For NC Documentation, and NC Code output you can choose a folder easily by clicking the **[...]** button.

You can customize the extension to be used for NC Code output (by default, the suffix used is CATNCCode).

Please note that Video results are stored in the NC Code output directory. This is done by using the **Associate Video Result to Machining Operation** icon  in the Tool Path Replay dialog box.

By default, the **Tool path: Store at same location as the CATProcess** checkbox is not selected.

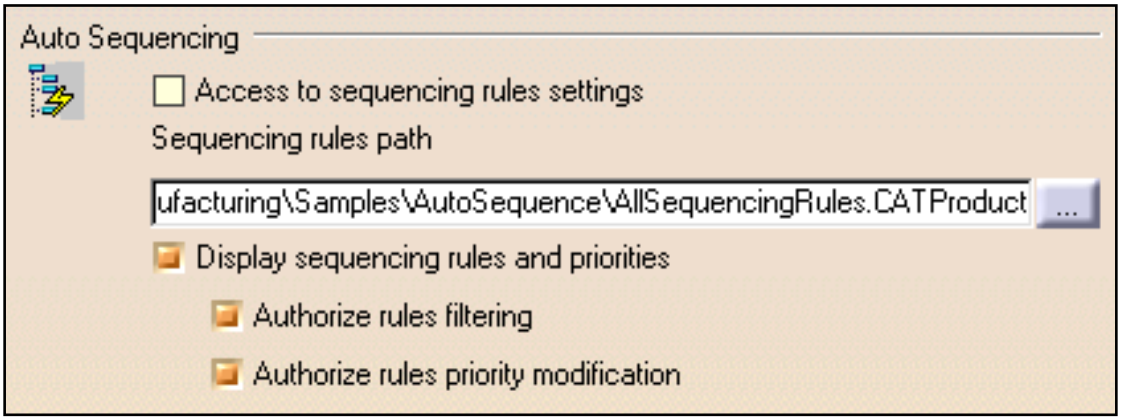
Program

This document explains how to customize manufacturing program settings for Machining products.

Select the **Program** tab to customize program auto-sequencing rules and priorities. These settings are mainly intended for the administrator.

Make sure that the document in the sequencing rules path (AllSequencingRules.CATProduct in the example below) is accessible in Read/Write.

Auto Sequencing



Access to sequencing rules settings

Select the **Access to sequencing rules settings** checkbox to authorize user access to sequencing rules.

You can then specify the path for the rules base
You can choose a rules base easily by clicking the **[...]** button.

By default, this checkbox is selected.

Display sequencing rules and priorities

Select the **Display sequencing rules and priorities** checkbox to authorize the display of sequencing rules and priorities in the user's view. In this case two more checkboxes can be selected in order to:

- allow the user to filter rules
- allow the user to modify rule priorities.

By default, these checkboxes are selected.

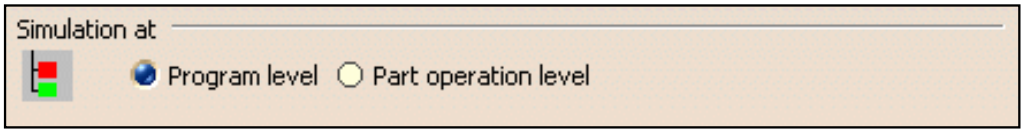
Photo/Video

This document explains how to customize material removal simulation settings for Machining products.

Select the **Photo/Video** tab, which is divided up into areas.

Parameters in this area...	Allow you to customize...
Simulation at	material removal simulation at program of Part Operation level
Video	Video material removal simulation options
Photo	Photo material removal simulation options
Performance	settings that influence performance
Color	color during material removal simulation
Positioning Move	allowed tool axis variation between two operations

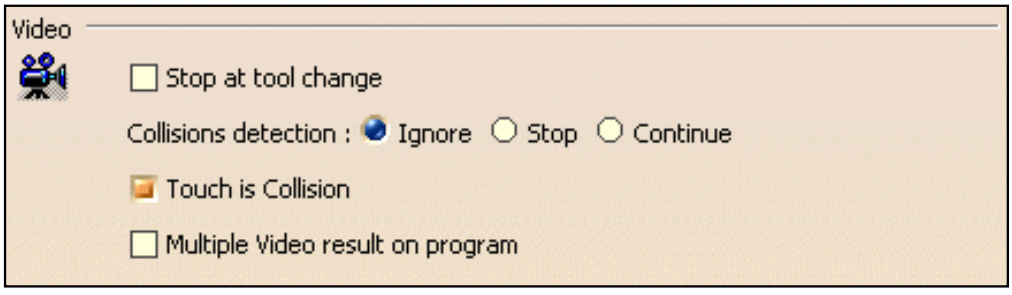
Simulation at



Select the desired option to perform material removal simulation at either Program or Part Operation level.
Depending on the selected level, simulation begins either from the start of the manufacturing program or from the start of the Part Operation.
Best performance is obtained with Program level.

By default, the **Program level** option is selected.

Video



Stop at tool change

Select the **Stop at tool change** checkbox if you want the Video simulation to stop each time a tool change is encountered in the program.

By default, this checkbox is not selected.

Collision detection

Select the desired **Collisions detection** option to:

- ignore collisions during the Video simulation

- stop the Video simulation at the first collision
- continue the Video simulation even when collisions are detected. In this case, you can consult the list of collisions at any time during the simulation.

Best performance is obtained when collisions are ignored.

By default, the **Ignore** option is selected.

Touch is collision

Select the **Touch is collision** checkbox if you want touch (or contact) type of collision to be detected.

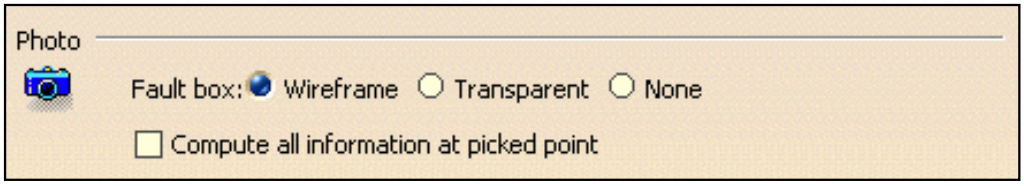
By default, this checkbox is selected.

Multiple Video result on program

Select the **Multiple Video result on program** checkbox if you want to store video results on more than one operation in the program.

By default, this checkbox is not selected.

Photo



Select the desired **Fault box** type for examining remaining material or gouges:

- Transparent: to display a transparent bounding box
- Wireframe: to display a wireframe bounding box
- None: if no bounding box is required.

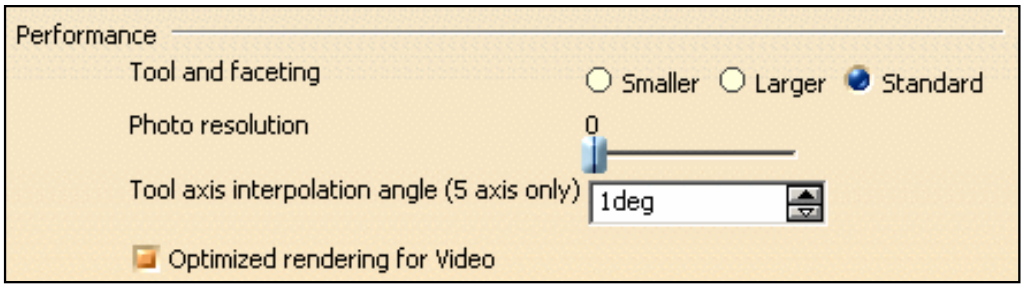
By default, the **Wireframe** option is selected.

Select the checkbox to compute all information at the picked point.

By default, this checkbox is not selected.

Best performance is obtained when **Fault box: None** is set and the **Compute all information at picked point** checkbox is not selected.

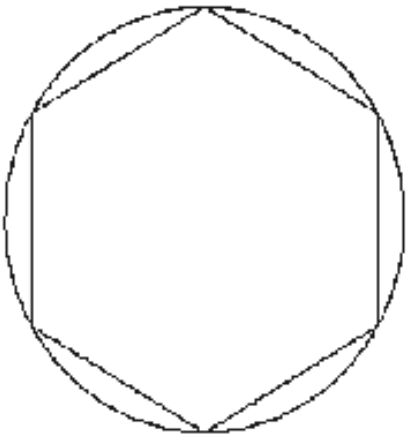
Performance



Tool and faceting

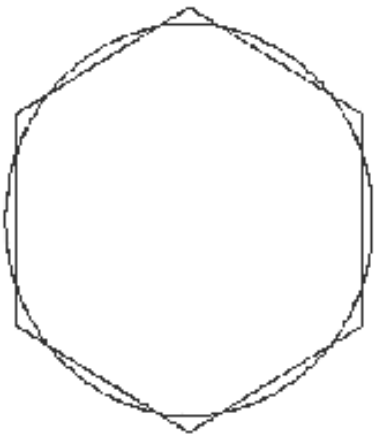
There are three methods of tool faceting used in Video simulation: Standard, Smaller and Larger.
The number of facets for a tool representation is determined by the chord deviation that is set for the tool diameter (0.005% of the tool diameter).

- **Smaller:** The picture shows a rough approximation of a tool with six facets. Note that the chord deviation is always inside the actual circle, and that the points are always on the circle (accurate).



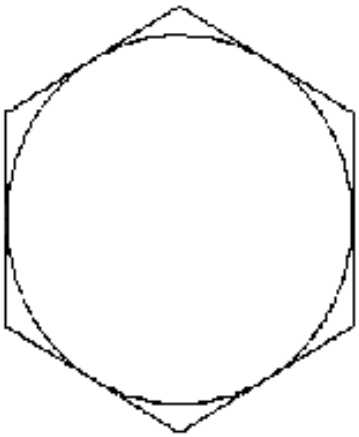
This is the most accurate method for the Arc through Three Points command.

- **Standard**: The picture shows a rough approximation of a tool with six facets. Note that the chord deviation is partly inside and partly outside the actual circle, and that the points are not always on the circle.



This is the best method for material removal simulation.
However, this is not suitable for the Arc through Three Points command.

- **Larger**: The picture shows a rough approximation of a tool with six facets. Note that the chord deviation is outside the actual circle, and that the points are not on the circle.



This is not suitable for the Arc through Three Points command.
However, it can be useful for gouge detection.

By default, the **Standard** option is selected.

Photo resolution

Best performance is obtained when the Photo resolution is set to 0. In this case, a detailed simulation of a portion of the part can be obtained using the **Closeup** command.
Increasing the resolution improves machining accuracy and gives a very detailed simulation. However, this requires increased memory and computation time.

By default, this resolution is set to 0.

Tool axis interpolation angle (5-axis only)

Specify the maximum angle that the tool axis is allowed to vary between two consecutive points.
Best performance is obtained for an angle of 10 degrees. Decreasing the angle improves the precision of the simulation. However, this requires increased memory and computation time.

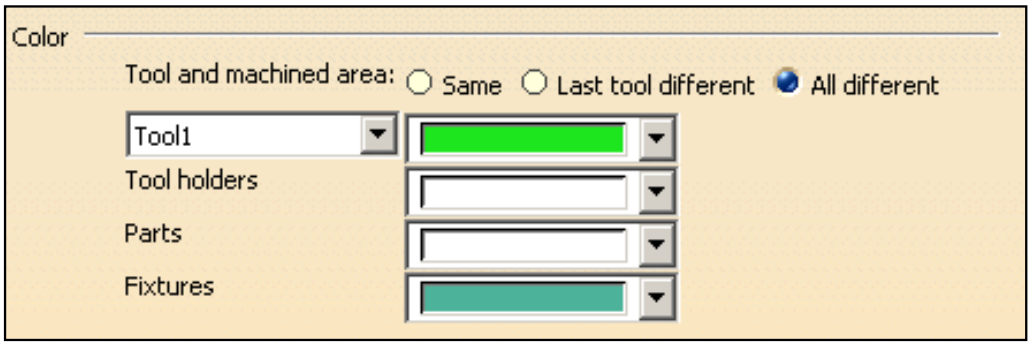
By default, this angle is set to 1degree.

Optimized rendering for Video

Set the **Optimized rendering for Video** checkbox to obtain an optimized rendering that improves Video simulation performance.
Otherwise, more realistic colors are obtained with a slightly degraded performance. Milling, drilling, and turning operations are supported .

By default, this checkbox is selected.

Color



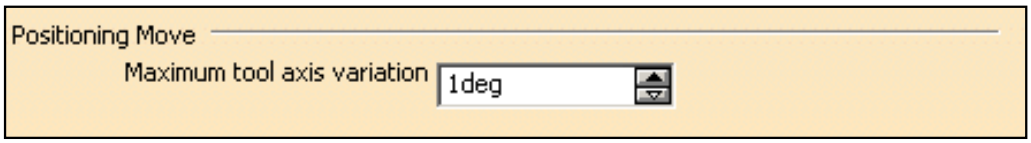
Set the tool (and associated machined area) color to be the same as or different from the last tool, or have different colors for all tools. Best performance is obtained with same colored tools.

By default, the **All different** option is selected.

Assign colors to the different tools using the associated color combo.

Assign colors to tool holders, parts, and fixtures using the associated color combos.

Positioning Move



Set the **Maximum tool axis variation** that is to be allowed between the end point of an operation and the start point of the next operation. If the tool axis varies by an amount greater than the specified value, then the tool is positioned at the start of the following operation.

By default, this angle is set to 1degree.

Reference Information

Reference information that is specific to the Lathe Machining product can be found in this section.

Rough Turning Operations
Recess Turning Operations
Groove Turning Operations
Profile Finish Turning Operations
Groove Finish Turning Operations
Ramp Rough Turning Operations
Ramp Recess Turning Operations
Thread Turning Operations
Sequential Turning Operations
Tool Assembly Conventions for Turning

Essential reference information on the following topics is provided in the *NC Manufacturing Infrastructure User's Guide*.

NC Manufacturing Resources
NC Macros
PP Tables and PP Word Syntaxes
Feeds and Speeds
APT Formats
CLfile Formats

Rough Turning Operations

The information in this section will help you create and edit Rough Turning operations in your manufacturing program.

The Rough Turning operation allows you to specify:

- Longitudinal, Face and Parallel Contour roughing modes
- external, internal or frontal machining according to the type of area to machine
- relimitation of the area to machine
- various approach and retract path types
- various lead-in and lift-off options with specific feedrates
- recess machining
- various contouring options with specific feedrates.

The following topics are dealt with in the paragraphs below:

- Tooling
- Geometry including Part and Stock Offsets
- Location and Limits
- Machining Strategy Parameters
- Feeds and Speeds
- Tool Compensation
- Macros.

Tooling for Rough Turning

The following tooling may be used:

- External and Internal insert-holders with all insert types except groove and thread.
- External and Internal Groove insert-holders with groove inserts.

Note that the following attributes may influence machining (they are located on the Insert-holder's Technology tab):

- Gouging angle
- Trailing angle
- Leading angle
- Max Recessing Depth
- Max Cutting Depth
- Max Boring Depth.

These attributes take tooling accessibility into account and may reduce the machined area.
However, you can use the **Insert-Holder Constraints** option on the operation editor to either ignore or apply these tooling attributes. You can replay the operation to verify the influence of these attributes on the generated

tool path.

Note that the **Insert-Holder Constraints** setting does not influence the Leading and Trailing Safety Angles defined in the operation editor.

Geometry for Rough Turning

Part and Stock profiles are required. They can be specified as follows:

- select edges either directly or after selecting the **By Curve** contextual command. In this case the Edge Selection toolbar appears to help you specify the guiding contour.
- select the **Sectioning** contextual command. Please refer to Sectioning for details of how to use this capability. Please note that the sectioning selection method is not associative.

The **End Limit** option allows you to specify a point, line, curve or face to relimit or extrapolate the selected part profile. If a face is specified, the end element is the intersection of the face and the working plane. The position of the end of machining is defined with respect to this element by one of the following settings: None / In / On / Out.

Relimiting the area to machine by means of limit elements

If you specify a point, it is projected onto the part profile.
A line through the projected point parallel to the radial axis delimits the area to machine.

If you specify a line, its intersection with the part profile is calculated (if necessary, the line is extrapolated).
A line through the intersection point parallel to the radial axis delimits the area to machine.

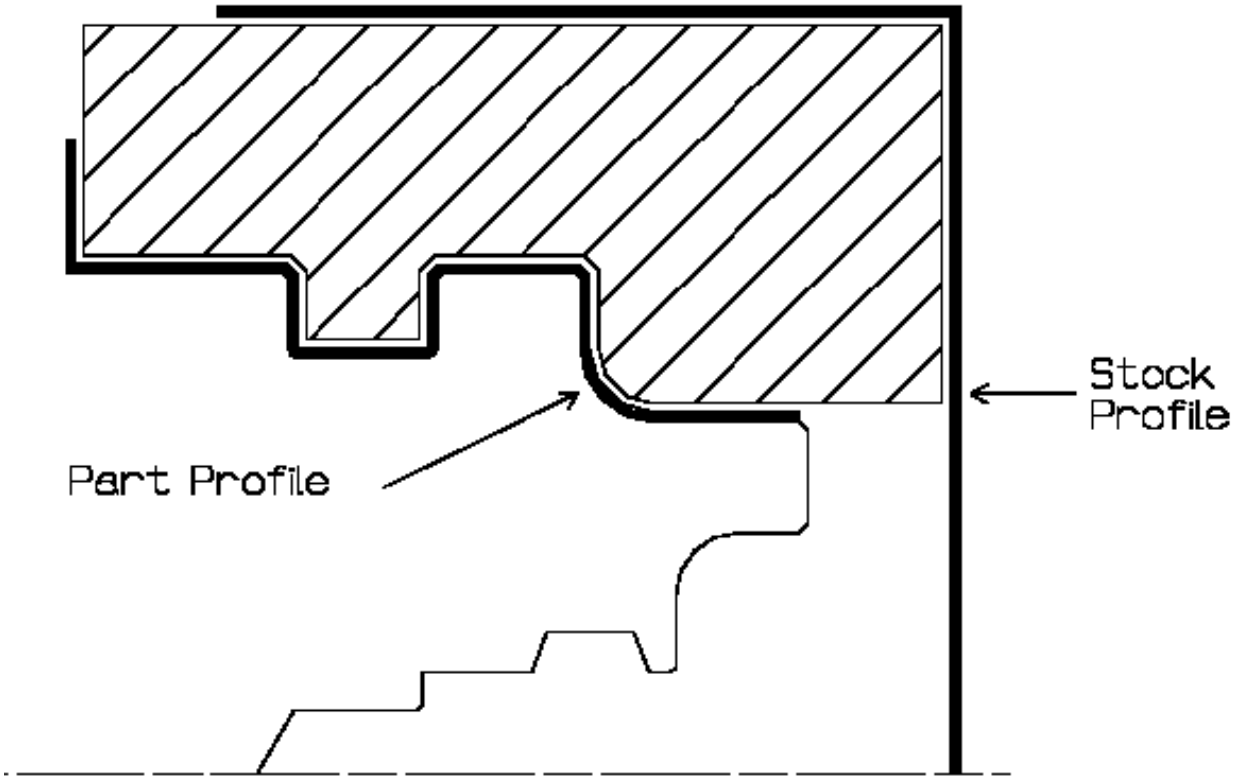
If you specify a curve, its intersection with the part profile is calculated (if necessary, the curve is extrapolated using the tangent at the curve extremity).
A line through the intersection point parallel to the radial axis delimits the area to machine.

Orientation for Rough Turning

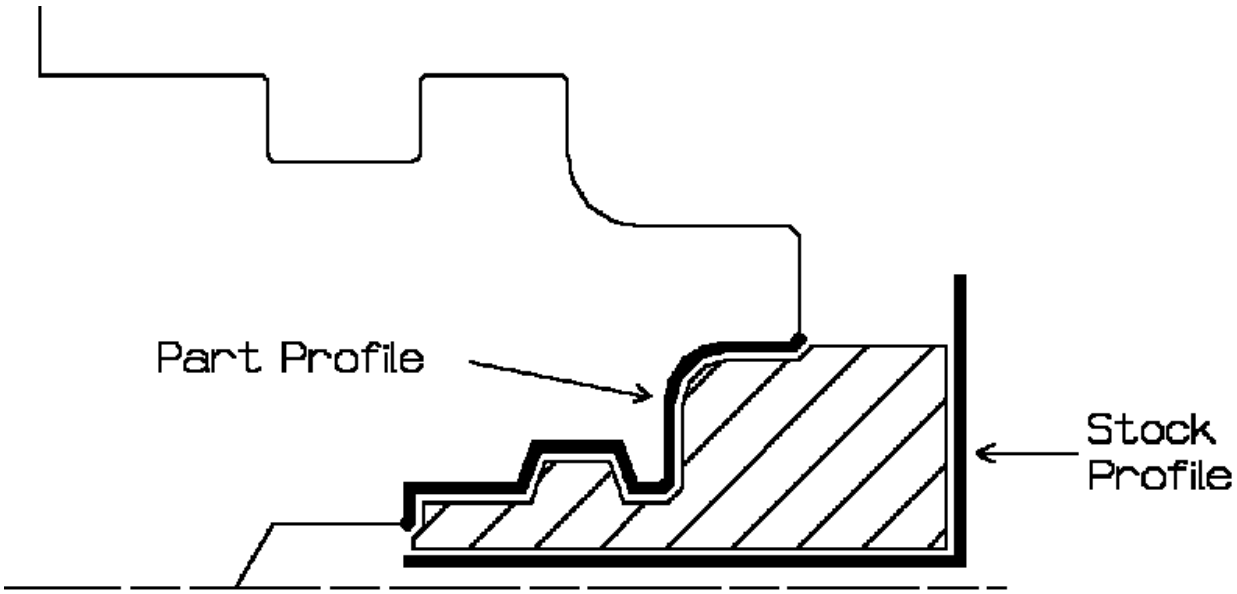
The following **Orientations** are proposed: Internal, External and Frontal (for Face and Parallel Contour Rough Turning only).

The selected Orientation defines the type of geometric relimitation to be done between the stock and part geometry in order to determine the area to machine. Selected part and stock profiles do not need to be joined (see the following figures).

External Rough Turning



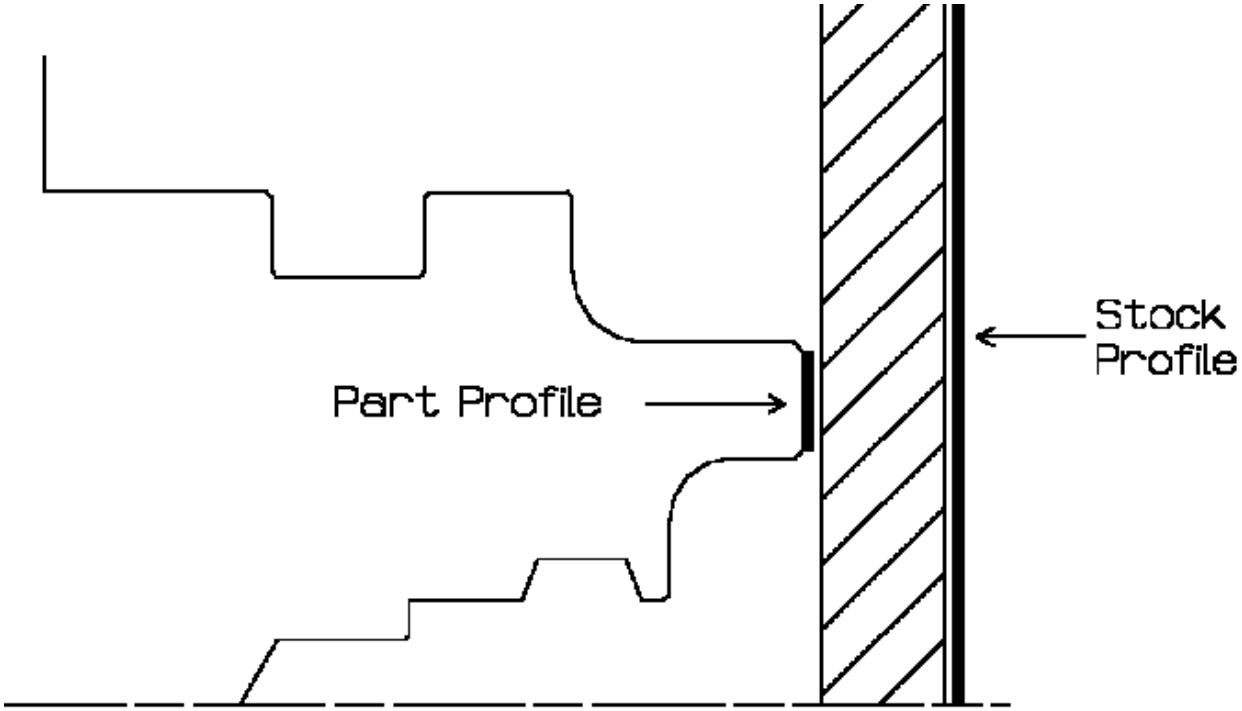
Internal Rough Turning



Frontal Rough Turning

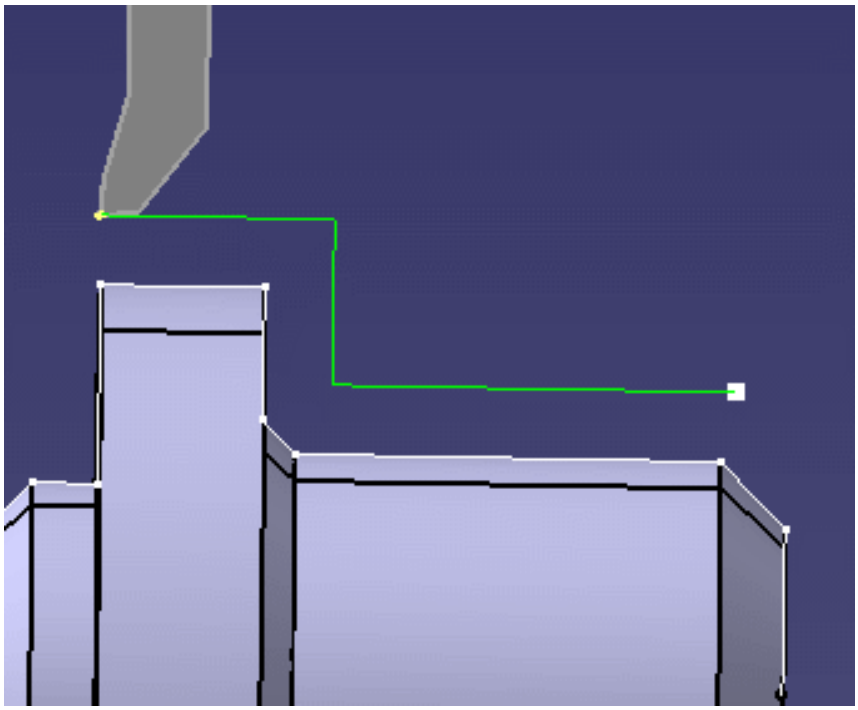
Frontal machining is proposed for face Rough Turning. In that case, the minimum and maximum diameters of the area to machine are determined by the stock profile dimensions.

For example, in the following figure the area to machine is relimited by the spindle axis because the stock profile is also relimited by the spindle axis.

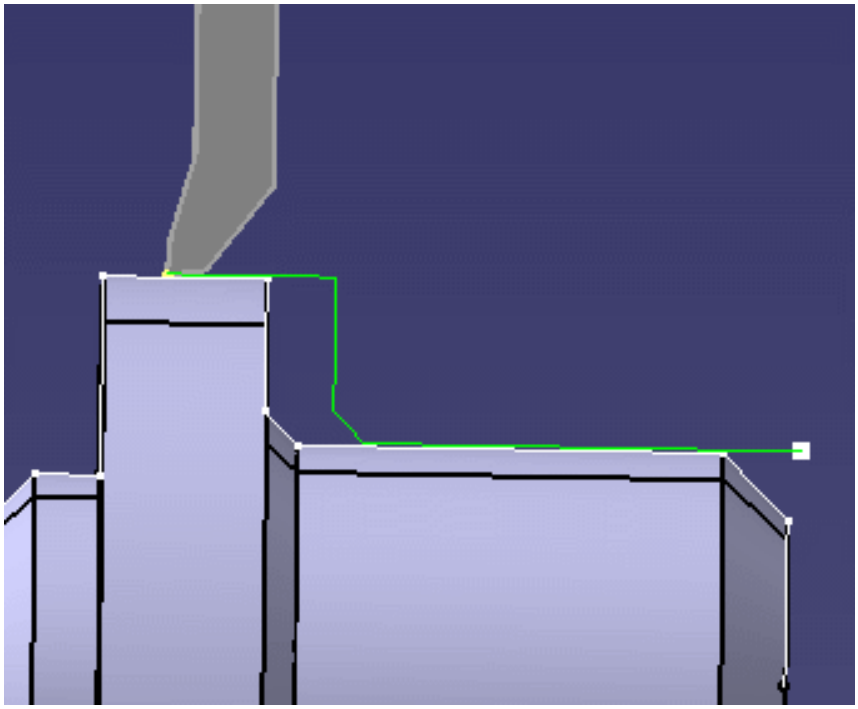


Part and Stock Offsets for Rough Turning

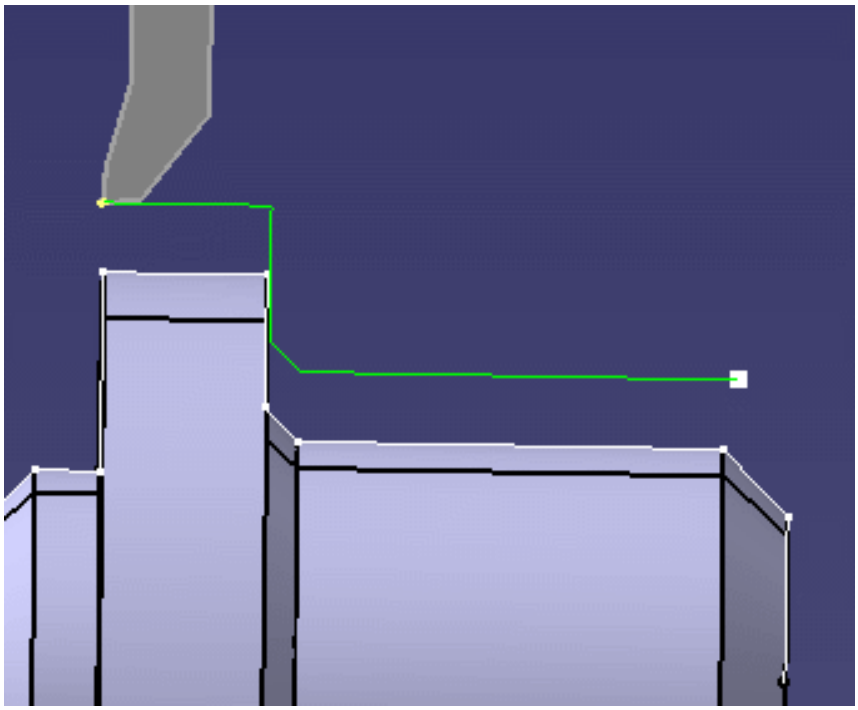
- **Stock offset**: specifies a virtual displacement of the stock profile.
- **Part offset**: specifies a virtual displacement of the part profile.



- **Axial part offset**: specifies a virtual displacement of the part profile along the spindle axis direction.



- **Radial part offset**: specifies a virtual displacement of the part profile in the radial axis direction.



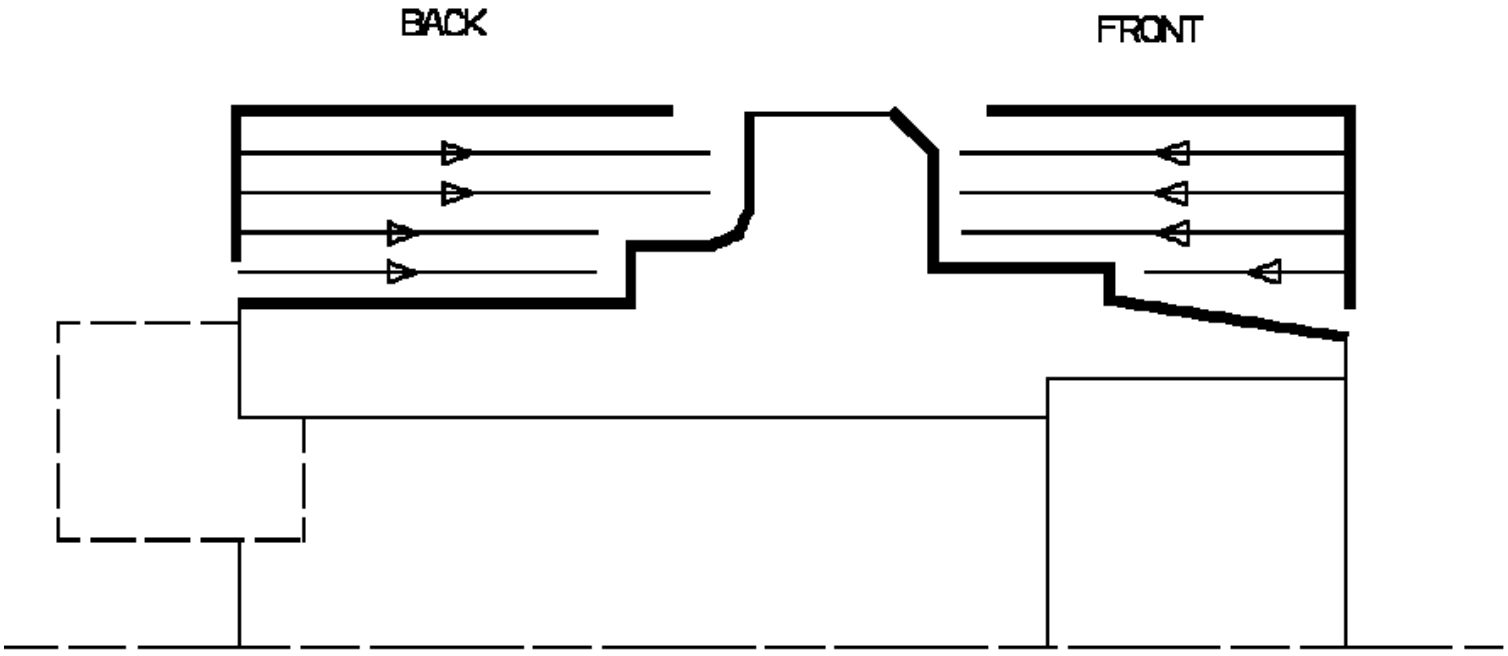
- **End limit offset**: distance with respect to the end element (only if end element is a line or a curve, and when In or Out is set for end element positioning).

Offsets can be positive or negative with any absolute value. The global offset applied to the part profile is the resulting value of the normal, axial and radial offsets.

Location and Limits for Rough Turning

The following machining **Locations** are proposed:

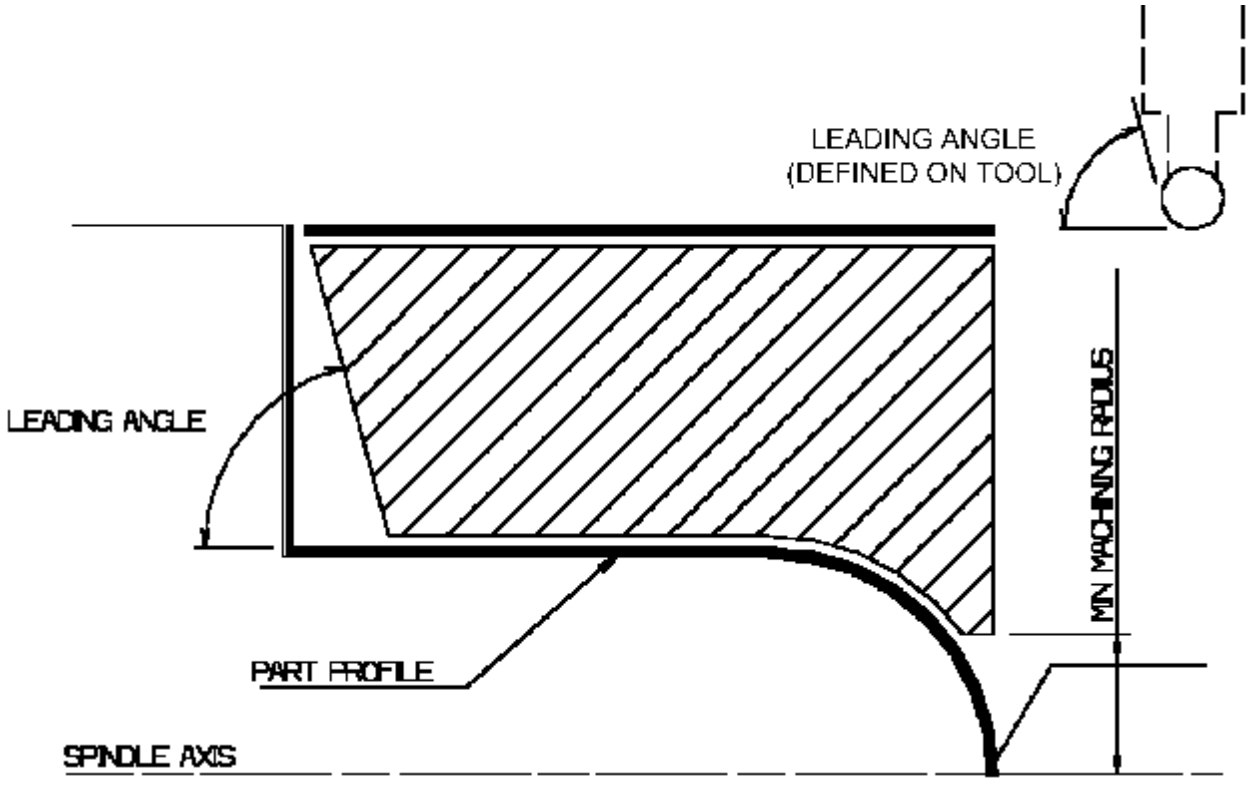
- Front, the part is machined toward the head stock
- Back, the part is machined from the head stock.



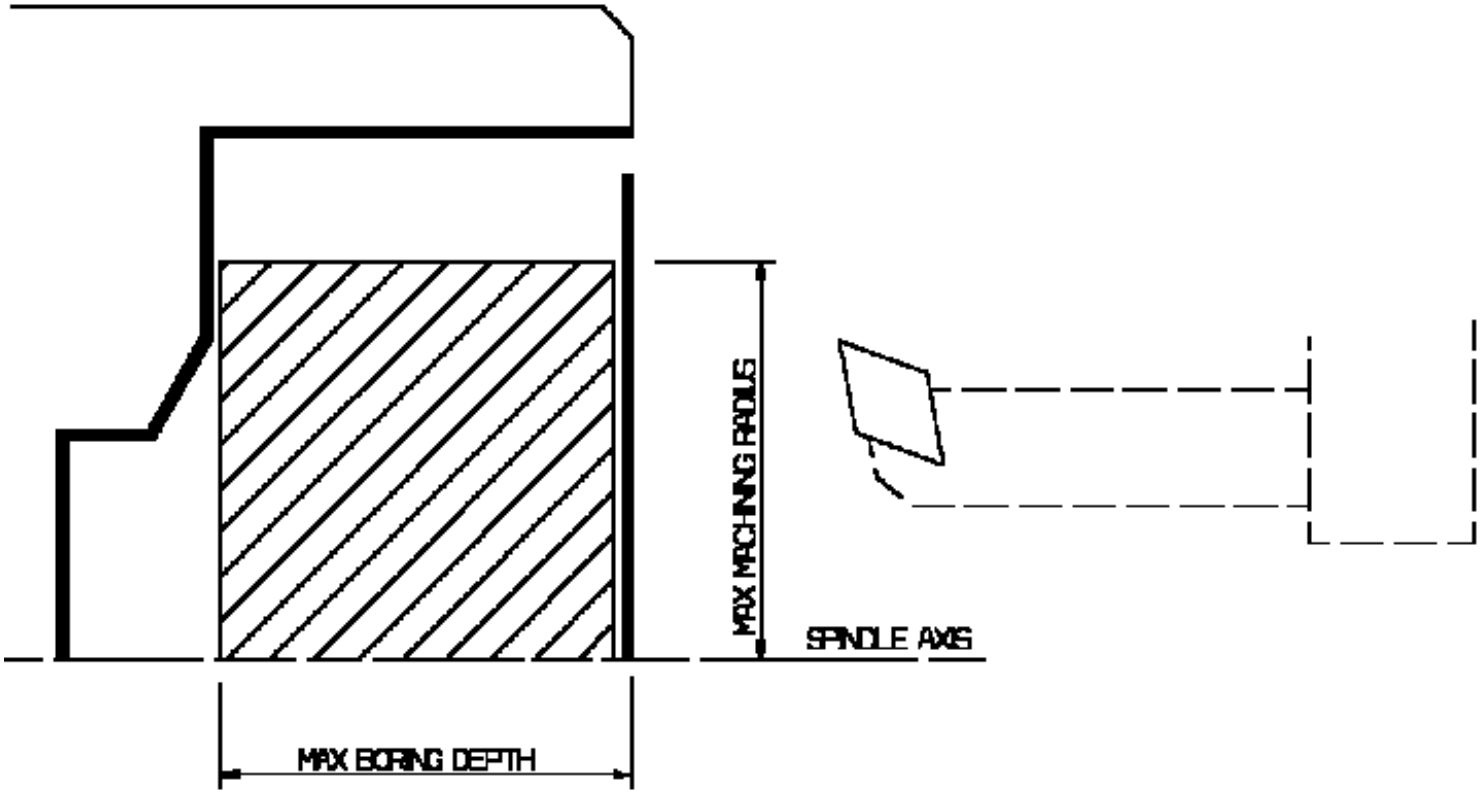
Orientation and Location settings determine the way the program closes the area to machine using radial, axial, axial-radial or radial-axial relimitation.

The following options allow you to restrict the area to machine that is pre-defined by the stock and part. You may want to restrict this area due to the physical characteristics of the tool and the type of machining to be done.

Minimum Machining Radius



Maximum Machining Radius (for internal machining)



Note that Max Boring Depth is defined on the tool.

Axial Limit for Chuck Jaws (for external or frontal machining): Offset defined from the machining axis system.

Machining Strategy Parameters for Rough Turning

Path Definition for Rough Turning

- **Machining tolerance**
- **Max Depth of Cut**
  This option is used to specify the maximum distance between passes.
  It is replaced by **Radial Depth of Cut** and **Axial Depth of Cut** for Parallel Contour Rough Turning.
- **Leading** and **Trailing Safety Angles**
  The insert geometry is taken into account to avoid collision by reducing the maximum slope on which the tool can machine. The Leading Safety Angle and Trailing Safety Angle allow you to further reduce the area to machine.
  Note that Trailing Angle can be used only when Recess Machining is set.

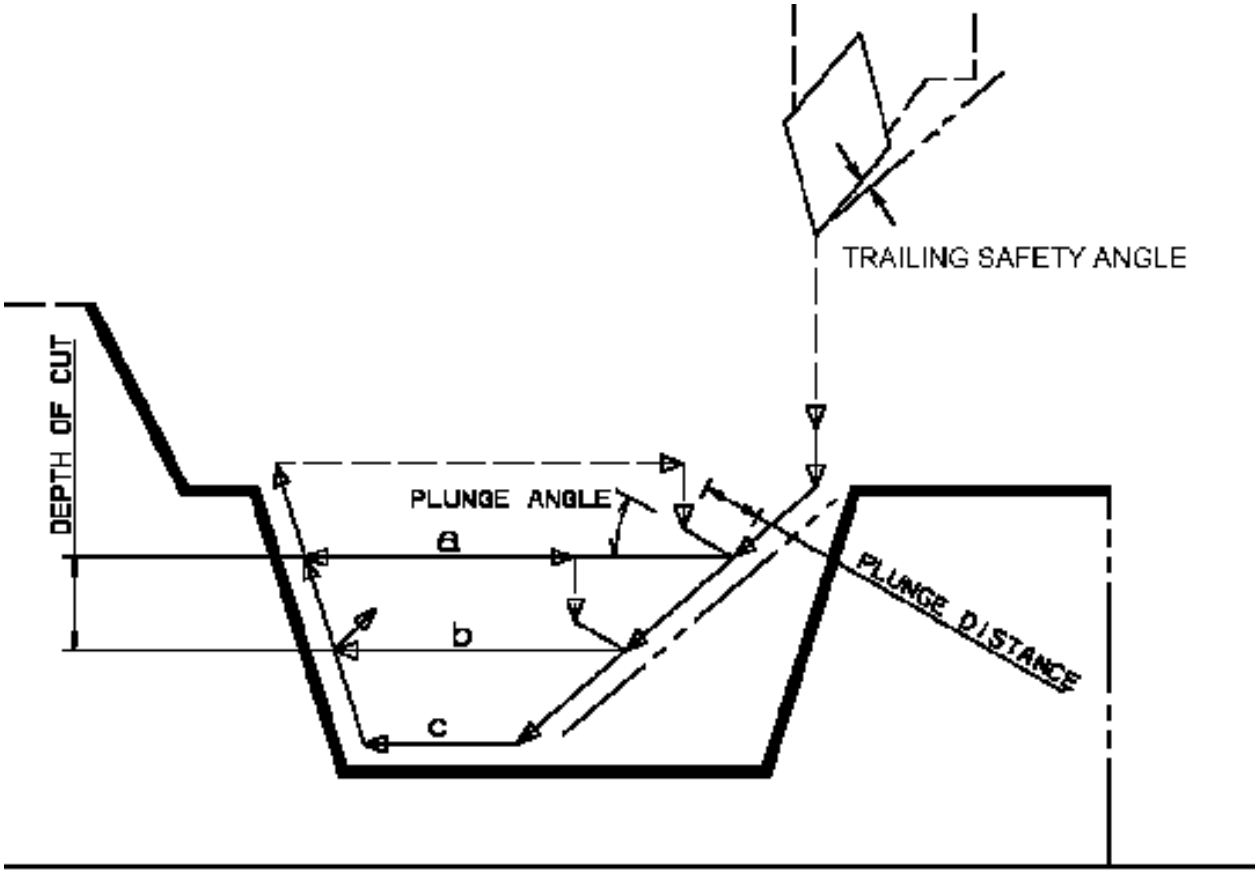
  Leading and trailing angles can also be defined on the insert-holder to define the maximum slope on which machining can be done. In this case and if the **Insert-Holder Constraints** setting is applied (see above), the angles that reduce the slope the most will be taken into account.
- **Part Contouring**
  You can specify a contouring type for longitudinal and face Rough Turning in order to clear the part profile by means of the following settings:
  - No: no contouring
  - Each path: profile following at each roughing pass
  - Last path only: profile following at last roughing pass only.
- **Under Spindle Axis Machining**
  For Face and Parallel Contour Rough Turning with Frontal machining, this option allows you to request machining under the spindle axis.
- **Machining Direction** (only for Parallel Contour and Face Rough Turning with Frontal machining)

You can specify the machining direction with respect to the spindle axis by means of the To/From Spindle Axis choice.

- **Recess Machining** (if Contouring Type is Each Pass or Last Path Only)
  If you require recess machining, activate this checkbox.
  When recess machining is active in Parallel Contour Rough Turning, Axial and Radial Depth of Cut must have suitable values to ensure a collision free toolpath. See Recommendations below.

The following options are proposed for recess machining:

- **Plunge Distance** and **Plunge Angle** (for longitudinal and face Rough Turning)
  Define the plunge vector before each new pass with respect to the cutting direction.



Example of Recess with External Longitudinal Rough Turning

In the figure above the tool motion is as follows:

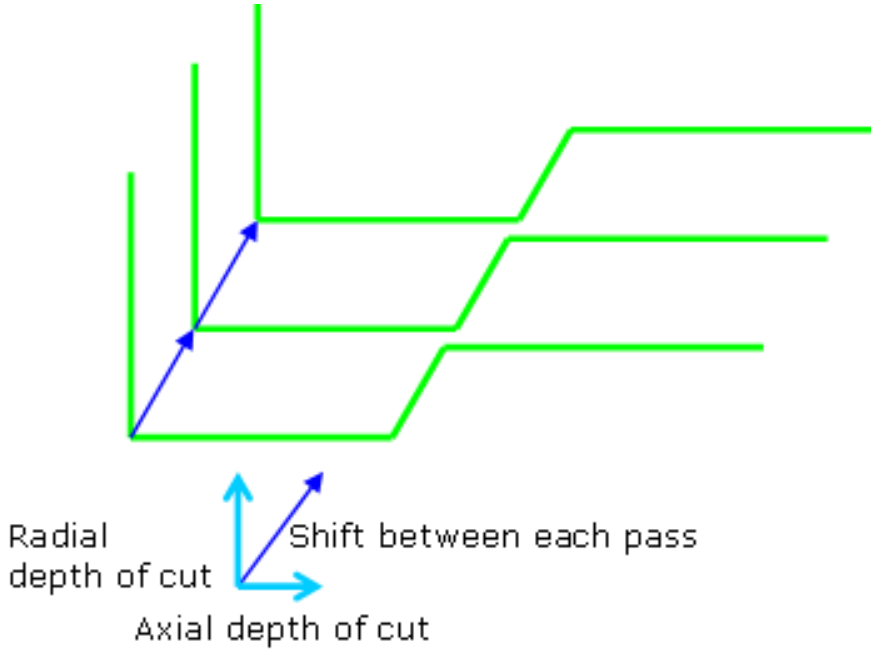
- approach in RAPID mode
- lead-in at the first recess pass and plunge approach for other passes
- plunge at plunge feedrate
- machine at machining feedrate
- contouring at contouring feedrate
- lift-off at last recess pass at lift-off feedrate.

Note that Trailing angle is defined on the tool.

Recommendations for Collision Free Recess Machining in Parallel Contour Rough Turning

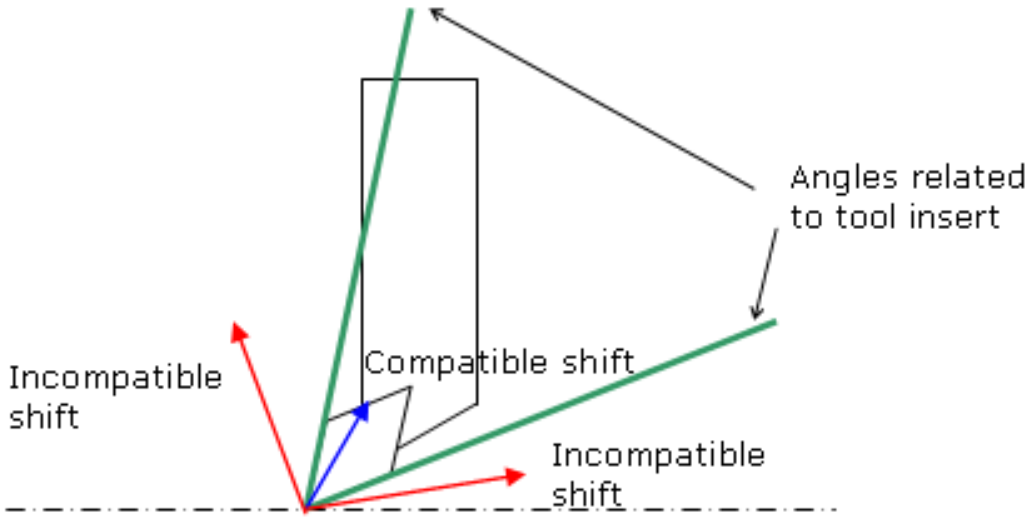
Machining in Parallel Contour Rough Turning is done by means of successive offsets of the toolpath: the offset depends on the axial and radial depth of cut values.
The following recommendations describe how to set the axial and radial depth of cuts to ensure a collision free toolpath.

Parallel contour principle:
The parallel contours of the toolpath are obtained by first shifting the last pass then shifting each successive pass depending on the axial and radial depth of cut values.



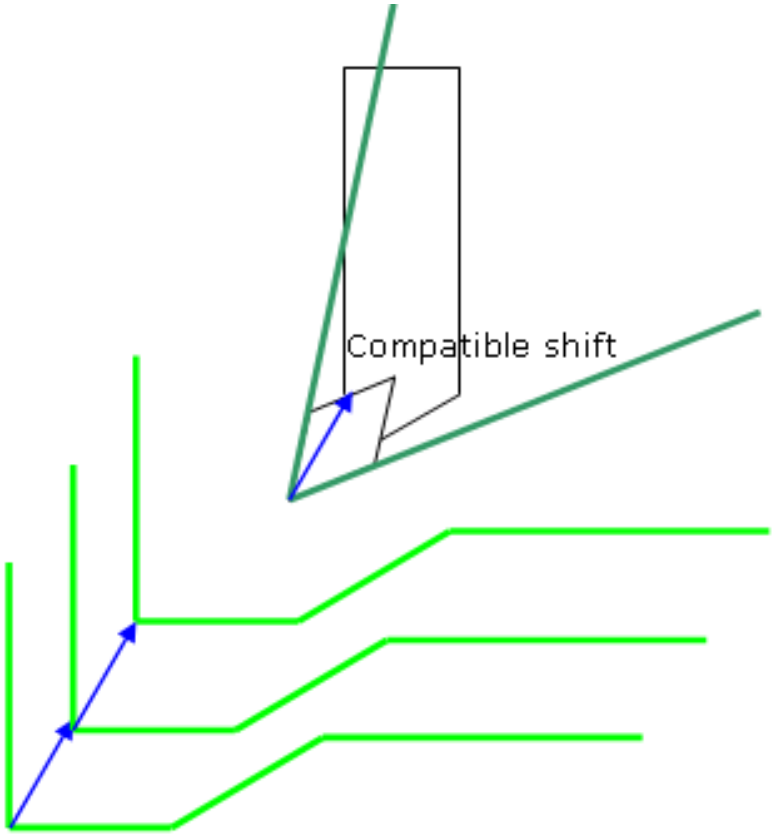
Then the passes are relimited taking into account the stock definition.

Correct combination of axial and radial depth of cuts
Compared with the left and right angles on the insert, the axial and radial depth of cut values define a shift direction that must be compatible with the insert.

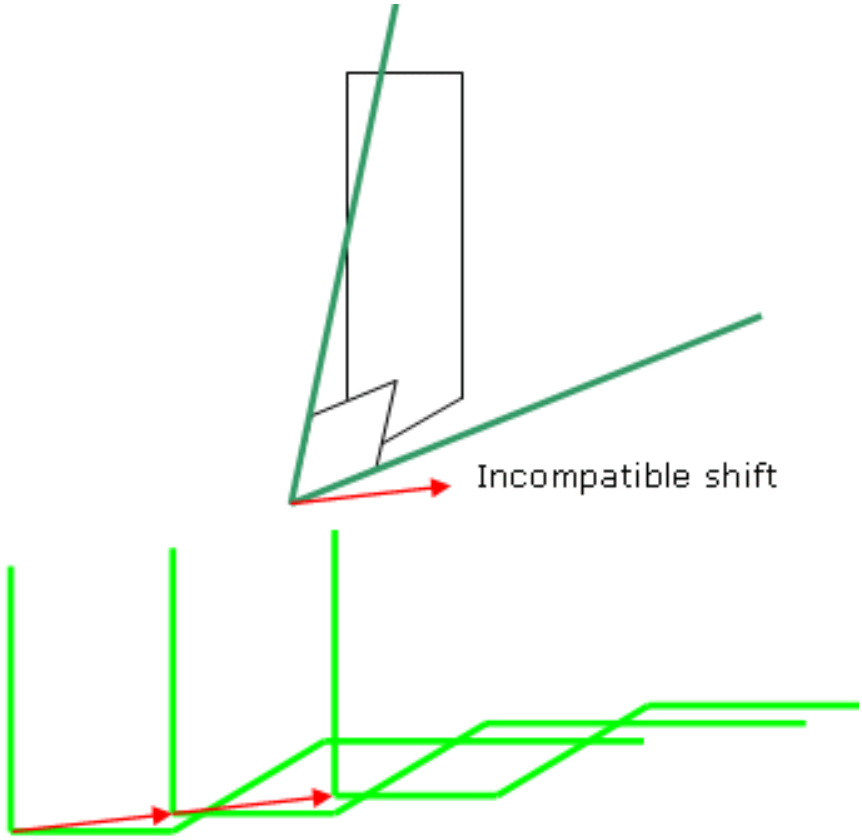


The shift direction must be inside the limit angles defined on the tool insert otherwise a collision may occur. In collision cases, the collision is on the first pass, not on the last passes of the toolpath.

Example of a correct combination



Example of an incorrect combination (right of tool)



The parallel contour toolpath is in collision because the shift does not respect the tool insert angle on the right side.

Lead-in, Lift-off and Attack for Rough Turning

These options allow penetration into the workpiece at a reduced feedrate in order to prevent tool damage. Once the attack distance has been run through, the tool moves at machining feedrate.

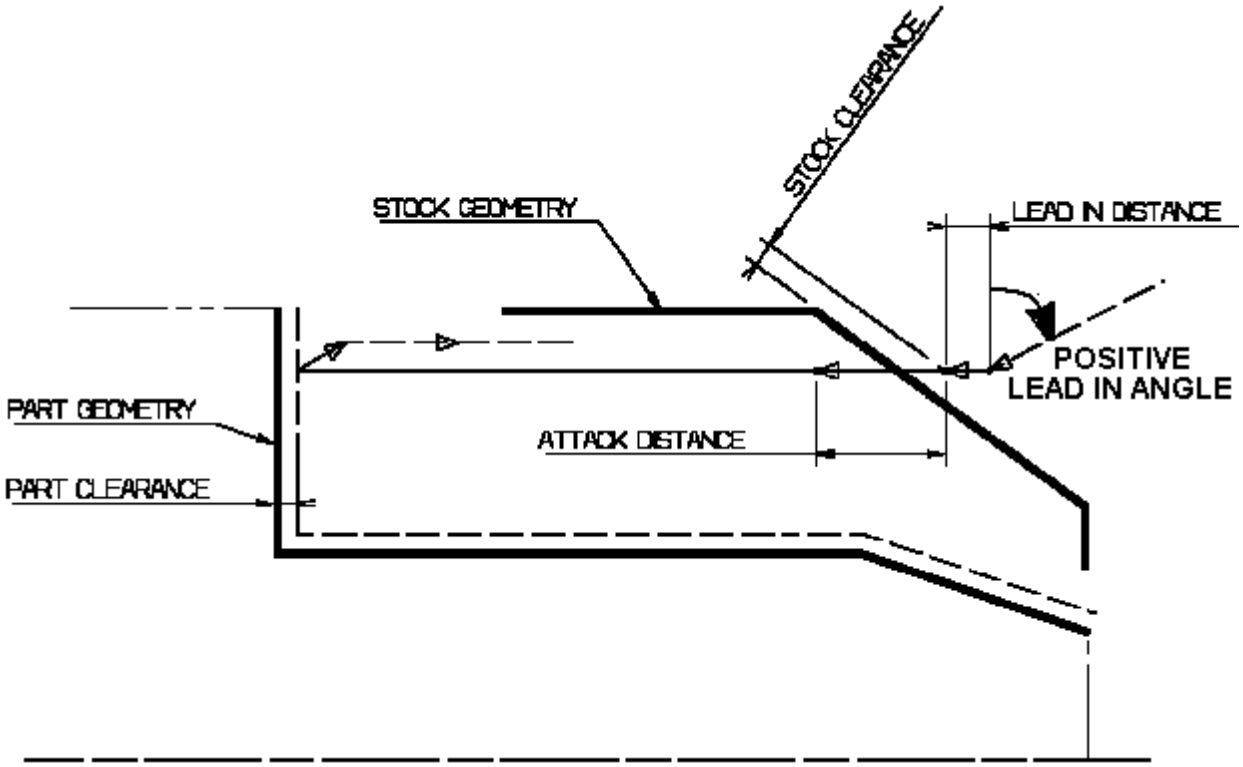
- **Lead-in Distance** and **Lead-in Angle**
  These parameters define the lead-in vector at the start of each pass with respect to the normal to the cutting direction.

  Lead-in distance takes the stock profile and stock clearance into account. The tool is in RAPID mode before this distance.

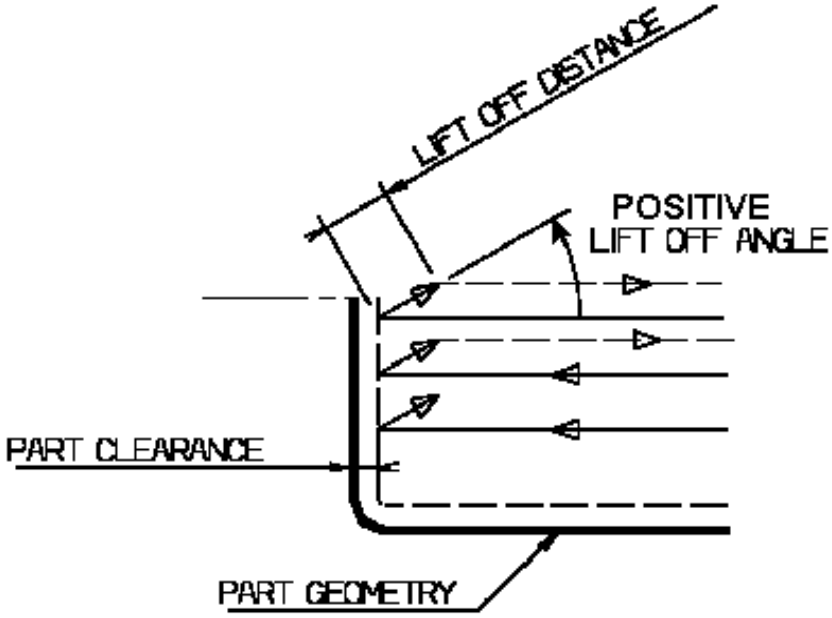
  If no lead-in angle is requested, the lead-in path is normal to the cutting direction.
  For Longitudinal and Face Rough Turning the lead-in angle can be applied as follows:

  - no angle applied to lead-in path
  - lead-in angle applied to each path
  - lead-in angle applied to last path only.

- **Attack Distance**
  Defined with respect to the cutting direction and takes the stock profile and stock clearance into account.



- **Lift-off Distance** and **Lift-off Angle**
  These parameters define the lift-off vector at the end of each pass with respect to the cutting direction.



For Longitudinal or Face Rough Turning, lift-off occurs:

- at the end of each pass when Contouring Type is set to None or Last Path Only.
- At the end of the last pass of the operation when the contouring type is set to Each Path. This prevents the tool from damaging the part when returning to the end point in RAPID mode.
- at the end of each pass that ends on the stock profile.

For Parallel Contour Rough Turning, lift-off occurs when the end of the pass has already been machined by a previous pass.

Feeds and Speeds for Rough Turning

Speed unit can be set to:

- Angular: spindle speed in revolutions per minute
- Linear: constant cutting speed in units per minute

then you can give a Machining Speed value.

Available feedrates in units per revolution are as follows:

- Machining Feedrate
- Lead-in Feedrate which is applied during the lead-in and attack distance
- Lift-off Feedrate
- Contouring Feedrate (if contouring type is Each Path or Last Path Only)
- Plunge Feedrate (for longitudinal and face Rough Turning).

Feedrates in units per minute are also available for air cutting such as macro motions and path transitions. Note that RAPID feedrate can be replaced by Air Cutting feedrate in tool trajectories (except in macros) by selecting the checkbox in the Feed and Speeds tab page  .

Dwell setting indicates whether the tool dwell at the end of each path is to be set in seconds or a number of spindle

revolutions.

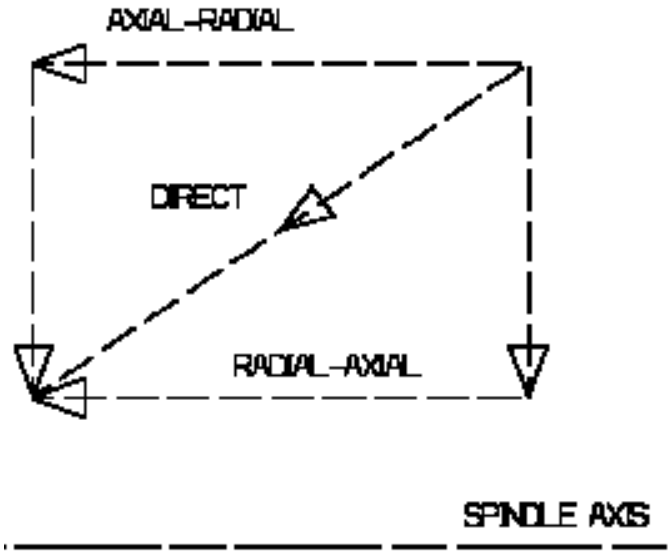
Please note that decimal values can be used for the number of revolutions. For example, when machining big parts that have a large volume, it can be useful to specify a dwell using a value of less than one revolution (0.25, for example).

Tool Compensation for Rough Turning

You can select a **tool compensation number** corresponding to the desired tool output point. Note that the usable compensation numbers are defined on the tool assembly linked to the machining operation. If you do not select a tool compensation number, the output point corresponding to type P9 will be used by default.

Approach, Retract and Linking Macros for Rough Turning

The following Approach and Retract macro modes are proposed: Direct, Axial-radial, Radial-axial, and Build by user.
The selected macro type (Approach or Retract) defines the tool motion before or after machining. Various feedrates are available for the approach and retract motions (RAPID, lead-in, lift-off, and so on).



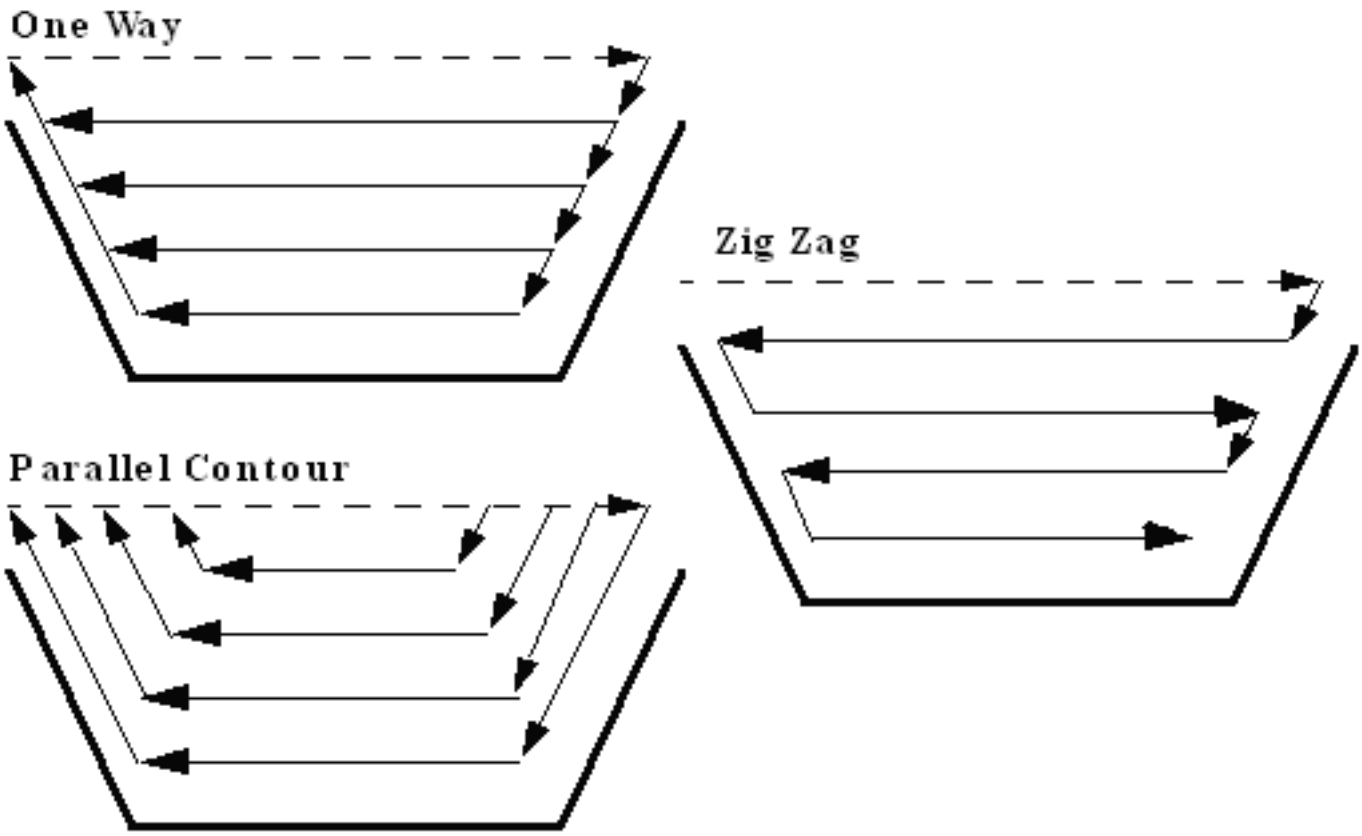
Linking macros, which comprise retract and approach motion can also be used on Rough Turning operations. Approach and retract motions of linking macros are interruptible. It can be useful to interrupt an operation when the foreseeable lifetime of the insert is not long enough to complete the machining.

See Define Macros on a Turning Operation for more information.

Recess Turning Operations

The information in this section will help you create and edit Recess Turning operations in your manufacturing program.

The Recess Turning operation allows you to machine a recess by means of a One Way, Zig Zag or Parallel Contour tool motion.



You can specify:

- external, internal, frontal or inclined machining according to the type of area to machine
- various approach and retract path types
- various lead-in and lift-off options with specific feedrates
- part contouring
- tool output point change.

The following topics are dealt with in the paragraphs below:

- Tooling
- Geometry
- Machining Strategy Parameters
- Feeds and Speeds
- Tool Compensation

- Macros.

Tooling for Recess Turning

The following tooling may be used:

- External and Internal insert-holders with all insert types except groove and thread.
- Internal, External and Frontal Groove insert-holders with groove inserts.

Note that the following attributes may influence machining (they are located on the Insert-holder's Technology tab):

- Gouging angle
- Trailing angle
- Leading angle
- Max Recessing Depth
- Max Cutting Depth
- Max Boring Depth.

These attributes take tooling accessibility into account and may reduce the machined area.
However, you can use the **Insert-Holder Constraints** option on the operation editor to either ignore or apply these tooling attributes. You can replay the operation to verify the influence of these attributes on the generated tool path.

Note that the **Insert-Holder Constraints** setting does not influence the Gouging Safety Angle or the Leading and Trailing Safety Angles defined in the operation editor.

Geometry for Recess Turning

Part and Stock profiles are required. They can be specified as follows:

- select edges either directly or after selecting the **By Curve** contextual command. In this case the Edge Selection toolbar appears to help you specify the guiding contour.
- select the **Sectioning** contextual command. Please refer to Sectioning for details of how to use this capability.
  Please note that the sectioning selection method is not associative.

Orientation for Recess Turning

The following **Orientations** are proposed: internal, external, frontal and inclined.

The selected Orientation defines the type of geometric relimitation to be done between the stock and part geometry in order to determine the area to machine. For an inclined orientation you must specify the **Angle of Incline**.

Part and Stock Offsets for Recess Turning

- **Stock offset**, which is defined perpendicular to the stock profile
- **Part offset**, which is defined perpendicular to the part profile.
- **Axial part offset**
- **Radial part offset**.

Offsets can be positive or negative with any absolute value. The global offset applied to the part profile is the resulting value of the normal, axial and radial offsets.

Machining Strategy Parameters for Recess Turning

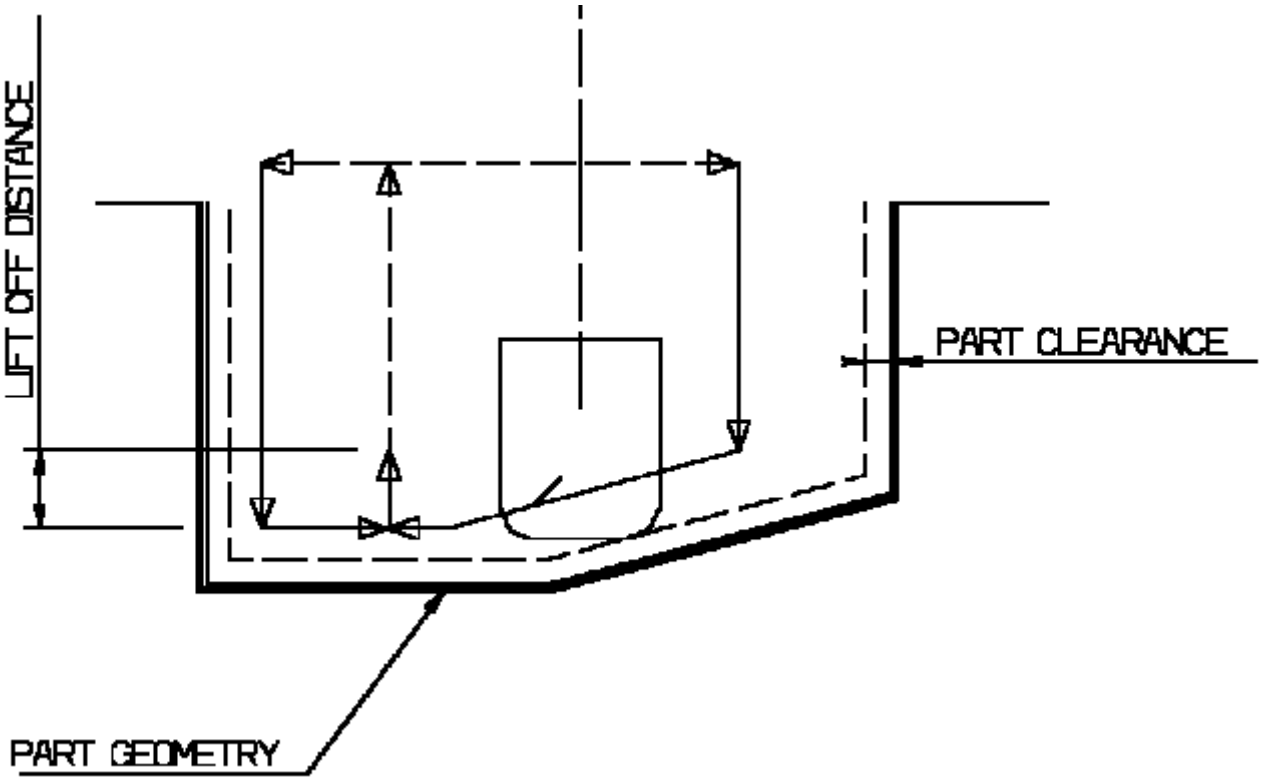
Path Definition for Recess Turning

- **Recess Turning Mode: One Way, Zig Zag or Parallel Contour**
- **Max Depth of Cut**
  This option is used to specify the maximum distance between passes.
  Axial and **Radial Depth of Cut**
  These options are used to specify the maximum axial and radial distances between passes for Parallel Contour mode.
- **Machining Tolerance**
- **Machining Direction**
  For Zig Zag tool motion, you must specify a first cutting direction as follows:
  - To or From Head Stock for Internal and External machining
  - To or From Spindle for Frontal machining
  - To Right or Left of Recess for Inclined machining

  When a part profile has multiple recesses (that is, a non-convex profile along the cutting direction), only the first recess along the specified direction is machined.
- **Leading** and **Trailing Safety Angles** for One way and Parallel Contour modes
  The insert geometry is taken into account to avoid collision by reducing the maximum slope on which the tool can machine. The Leading and Trailing Safety Angles allow you to further reduce this slope.

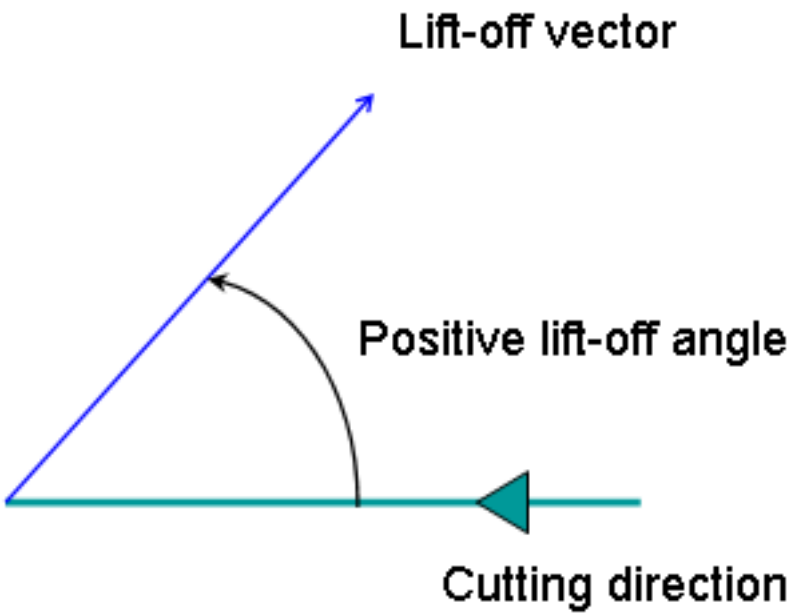
  Leading and trailing angles can also be defined on the insert-holder to define the maximum slope on which machining can be done. In this case and if the **Insert-Holder Constraints** setting is applied (see above), the angles that reduce the slope the most will be taken into account.
- **Gouging Safety Angle** (for Zig Zag mode only)
  Angles of the insert are taken into account to avoid collision by reducing the maximum slope on which the tool can machine. The Gouging Safety Angle allows you to further reduce this slope.

  Note that a gouging angle can also be defined on the insert-holder to define the maximum slope on which the tool can machine. In this case and if the **Insert-Holder Constraints** setting is applied (see above), the angle that reduces the slope the most will be taken into account.
- **Under Spindle Axis Machining**
  For Frontal or Inclined machining, this option allows you to request machining under the spindle axis.
- **Part Contouring**
  You can specify if contouring is required by means of the proposed checkbox.
  The part profile is followed at the end of Recess Turning. This is done by machining down the sides of the recess in order to clear the profile.



Lead-in, Lift-off and Attack for Recess Turning

- **Lead-in Distance**
  Defined with respect to the cutting direction. It takes the stock profile and stock clearance into account. The tool is in RAPID mode before this distance.
- **Attack Distance**
  Defined with respect to the cutting direction and the stock profile with a stock clearance.
- **Angle and Distance before Plunge**
  Define the plunge vector before each new pass with respect to the cutting direction.
- **Lift-off Distance** and **Lift-off Angle**
  Define the lift-off vector at the end of the last pass with respect to the cutting direction. The figure below shows the effect of a positive lift-off angle (external machining is assumed).



Feeds and Speeds for Recess Turning

Speed unit can be set to:

- Angular: spindle speed in revolutions per minute
- Linear: constant cutting speed in units per minute

then you can give a Machining Speed value.

Available feedrates in units per revolution are as follows:

- Machining Feedrate
- Lead-in Feedrate, which is applied during the lead-in and attack distance.
- Lift-off Feedrate
- Contouring Feedrate
- Plunge Feedrate.

Feedrates in units per minute are also available for air cutting such as macro motions and path transitions. Note that RAPID feedrate can be replaced by Air Cutting feedrate in tool trajectories (except in macros) by selecting the checkbox in the Feed and Speeds tab page  .

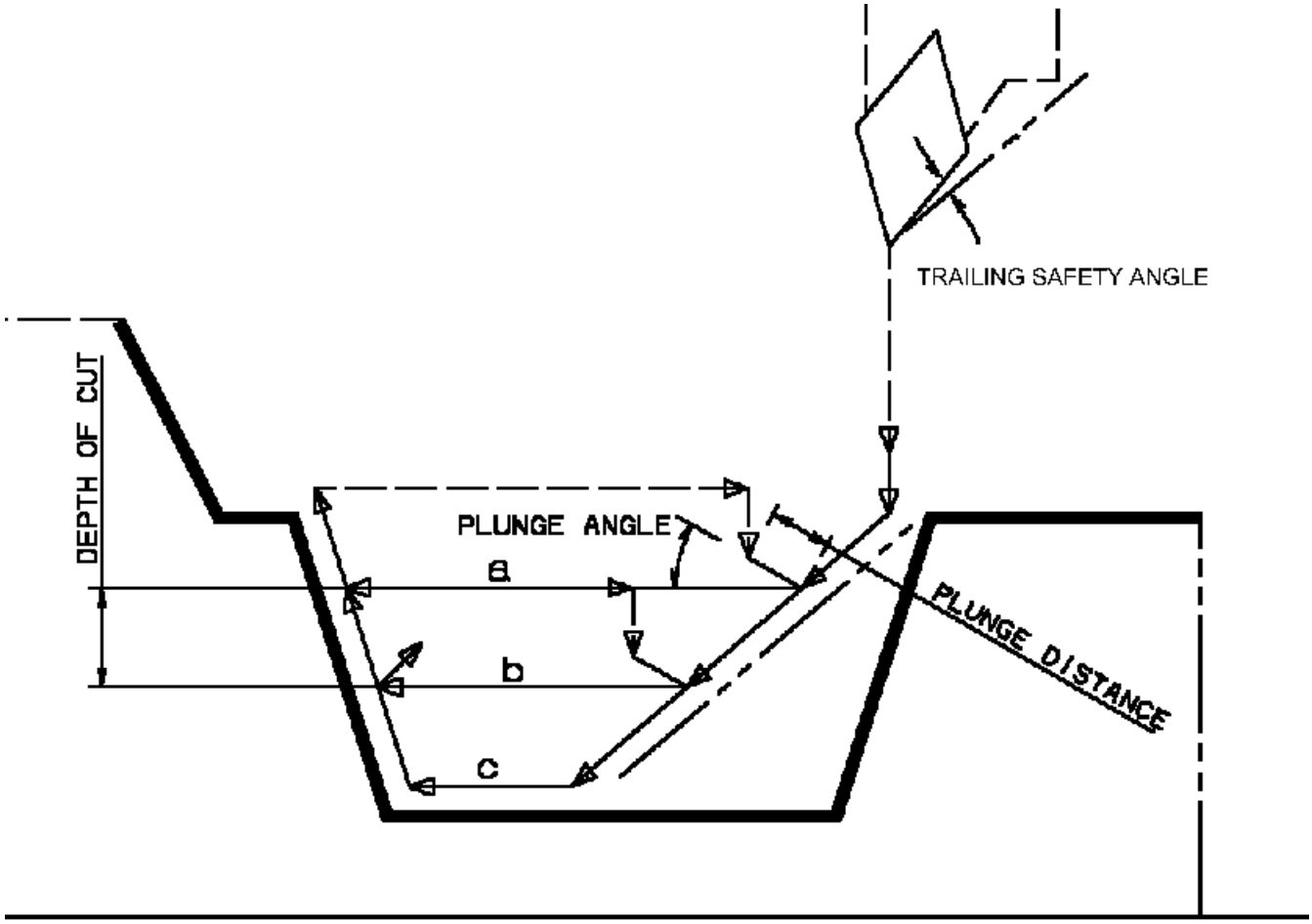
Dwell setting indicates whether the tool dwell at the end of each path is to be set in seconds or a number of spindle revolutions.

Please note that decimal values can be used for the number of revolutions. For example, when machining big parts that have a large volume, it can be useful to specify a dwell using a value of less than one revolution (0.25, for example).

Example of one-way Recess Turning. Note that Trailing angle is defined on the tool.

In the figure below the tool motion is as follows:

- approach in RAPID mode
- lead-in at the first recess pass and plunge approach for other passes
- plunge at plunge feedrate
- machine at machining feedrate
- contouring at contouring feedrate
- lift-off at last recess pass at lift-off feedrate.



Tool Compensation for Recess Turning

You can select a **tool compensation number** corresponding to the desired tool output point. Note that the usable compensation numbers are defined on the tool assembly linked to the machining operation. If you do not select a tool compensation number, the output point corresponding to type P9 will be used by default.

Note that the change of output point is managed automatically if you set the **Change Output Point** option.

If the output point is consistent with the flank of the recess to be machined, the output point is changed when the other flank of the recess is machined.

At the end of the operation, the output point is the same as it was at the start of the operation. See Changing the Output Point for more information.

Approach, Retract and Linking Macros for Recess Turning

The following Approach and Retract macro modes are proposed: Direct, Axial-radial, Radial-axial, and Build by user. The selected macro type (Approach or Retract) defines the tool motion before or after machining. Various feedrates are available for the approach and retract motions (RAPID, lead-in, lift-off, and so on).

Linking macros, which comprise retract and approach motion can also be used on Recess Turning operations.
Approach and retract motions of Linking macros are interruptible. It can be useful to interrupt an operation when the foreseeable lifetime of the insert is not long enough to complete the machining.

See Define Macros on a Turning Operation for more information.

Groove Turning Operations

The information in this section will help you create and edit Groove Turning operations in your manufacturing program.

The Groove Turning operation allows you to machine a groove by a series of plunging cuts. You can specify:

- external, internal, frontal or inclined machining according to the type of area to machine
- various approach and retract path types
- various lead-in and lift-off options with specific feedrates
- various plunge locations
- tool output point change.

The following topics are dealt with in the paragraphs below:

- Tooling
- Geometry
- Machining Strategy Parameters
- Feeds and Speeds
- Tool Compensation
- Macros.

Tooling for Groove Turning

The following tooling may be used:

- Internal, External and Frontal Groove insert-holders with groove inserts.
- External and Internal insert-holders with round inserts.

Note that the following attributes may influence machining (they are located on the Insert-holder's Technology tab):

- Gouging angle
- Trailing angle
- Leading angle
- Max Recessing Depth
- Max Cutting Depth
- Max Boring Depth.

These attributes take tooling accessibility into account and may reduce the machined area.
However, you can use the **Insert-Holder Constraints** option on the operation editor to either ignore or apply these tooling attributes. You can replay the operation to verify the influence of these attributes on the generated tool path.

Note that the **Insert-Holder Constraints** setting does not influence the Gouging Safety Angle or Max Depth of Cut defined in the operation editor.

Geometry for Groove Turning

Part and Stock profiles are required. They can be specified as follows:

- select edges either directly or after selecting the **By Curve** contextual command. In this case the Edge Selection toolbar appears to help you specify the guiding contour.
- select the **Sectioning** contextual command. Please refer to Sectioning for details of how to use this capability.
  Please note that the sectioning selection method is not associative.

Orientation for Groove Turning

The following Orientations are proposed: internal, external, frontal and inclined. The selected Orientation defines the type of geometric relimitation to be done between the stock and part geometry in order to determine the area to machine. For an inclined orientation you must specify the **Angle of Incline**.

Part and Stock Offsets for Groove Turning

- **Stock offset**, which is defined perpendicular to the stock profile
- **Part offset**, which is defined perpendicular to the part profile.
- **Axial part offset**
- **Radial part offset**.

Offsets can be positive or negative with any absolute value. The global offset applied to the part profile is the resulting value of the normal, axial and radial offsets.

Machining Strategy Parameters for Groove Turning

Path Definition for Groove Turning

- **Max Depth of Cut**
  This option is used to specify the maximum distance between plunges.
- **First Plunge**
  You must specify a first plunge position according to the groove orientation by means of the following choice: Left/Down - Center - Right/Up - Automatic.
  Automatic is only available for frontal machining. In this case, the position of the first plunge is deduced from the tool's minimum and maximum cut diameters. The position is at a distance:
  (Minimum cut radius + Maximum cut radius)/2.
- **Next Plunges** (if First Plunge is set to Center).
  You can specify the position of the plunges that follow the first plunge with respect to:
  - the spindle axis by means of the To or From Spindle for frontal machining
  - the head stock by means of the To or From Head Stock for internal or external machining
  - the groove by means of Left or Right of Groove for Inclined machining.

- **Part Contouring**
  You can specify if contouring is required by means of the proposed checkbox.
  The part profile is followed at the end of grooving. This is done by machining down the sides of the groove in order to clear the profile.
- **Grooving by Level** mode
  This option allows you to machine the groove in one or more level.
  Multiple-levels mode is particularly useful when the groove is too deep to machine in one level. In this case Maximum grooving depth defines the maximum depth of each level. If it is greater than the Maximum Depth of Cut defined on the tool, the value on the tool is taken into account.
- **Under Spindle Axis Machining**
  When Groove Turning in frontal mode, this option allows you to request machining under the spindle axis.
- **Chip Break**
  You can specify if chip clearing is to be done during machining by setting the check box. In this case you must specify Plunge, Retract and Clear distances.
- **Gouging Safety Angle**
  Angle attributes on the grooving insert are taken into account to avoid collision by reducing the maximum slope on which the tool can machine. The Gouging Safety Angle allows you to further reduce this slope.

  Note that a gouging angle can also be defined on the insert-holder to define the maximum slope on which the tool can machine. In this case and if the **Insert-Holder Constraints** setting is applied (see above), the angle that reduces the slope the most will be taken into account.
- **Machining Tolerance**.

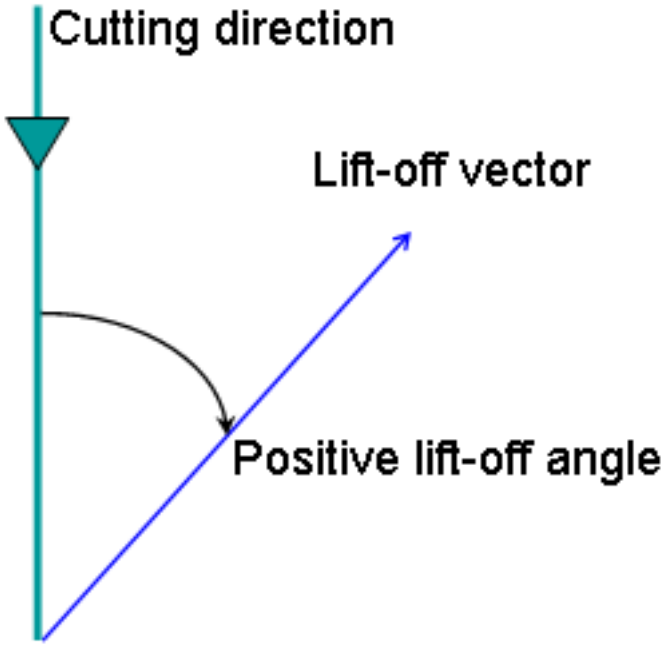
Lead-in, Lift-off and Attack for Groove Turning

- **Lead-in Distance**
  Defined with respect to the cutting direction. It takes the stock profile and stock clearance into account.
  The tool is in RAPID mode before this distance.
- **Attack Distance**
  Defined with respect to the cutting direction and the stock profile with a stock clearance.

These options allow penetration into the workpiece at a reduced feedrate in order to prevent tool damage. Once the attack distance has been run through, the tool moves at machining feedrate.

When tool motion between two passes is in contact with the part profile, in order to avoid collisions the corresponding feed is the lift-off feedrate and not RAPID.

- **Lift-off Distance** and **Lift-off Angle**
  These parameters define the lift-off vector at the end of each pass with respect to the cutting direction.
  The figure below shows the effect of a positive lift-off angle (external machining is assumed).



Feeds and Speeds for Groove Turning

Speed unit can be set to:

- Angular: spindle speed in revolutions per minute
- Linear: constant cutting speed in units per minute

then you can give a Machining Speed value.

Available feedrates in units per revolution are as follows:

- Contouring Feedrate
- Lead-in Feedrate, which is applied during lead-in and attack distances
- Lift-off Feedrate
- First Plunge Feedrate and Next Plunges Feedrate.
  Different feedrates can be assigned to the first plunge and the following plunges.

Feedrates in units per minute are also available for air cutting such as macro motions and path transitions. Note that RAPID feedrate can be replaced by Air Cutting feedrate in tool trajectories (except in macros) by selecting the checkbox in the Feed and Speeds tab page  .

Dwell setting indicates whether the tool dwell at the end of a path or a plunge is to be set in seconds or a number of spindle revolutions.

Please note that decimal values can be used for the number of revolutions. For example, when machining big parts that have a large volume, it can be useful to specify a dwell using a value of less than one revolution (0.25, for example).

Tool Compensation for Groove Turning

You can select a **tool compensation number** corresponding to the desired tool output point. Note that the usable compensation numbers are defined on the tool assembly linked to the machining operation. If you do not select a tool compensation number, the output point corresponding to type P9 will be used by default.

Note that the change of output point is managed automatically if you set the **Change Output Point** option. If the output point is consistent with the flank of the groove to be machined, the output point is changed when the other flank of the groove is machined.
At the end of the operation, the output point is the same as it was at the start of the operation. See Changing the Output Point for more information.

Approach, Retract and Linking Macros for Groove Turning

The following Approach and Retract macro modes are proposed: Direct, Axial-radial, Radial-axial, and Build by user.
The selected macro type (Approach or Retract) defines the tool motion before or after machining. Various feedrates are available for the approach and retract motoins (RAPID, lead-in, lift-off, and so on).

Linking macros, which comprise retract and approach motion can also be used on Groove Turning operations.
Approach and retract motoins of linking macros are interruptible. It can be useful to interrupt an operation when the foreseeable lifetime of the insert is not long enough to complete the machining.

See Define Macros on a Turning Operation for more information.

Profile Finish Turning Operations

The information in this section will help you create and edit Profile Finish Turning operations in your manufacturing program.

The Profile Finish Turning operation allows you to finish a part profile. You can specify:

- the type of machining according to the profile of the area to machine (external, internal or frontal)
- relimitation of the profile by start and end elements
- various approach and retract path types
- linear and circular lead-in and lift-off options with specific feedrates
- recess machining
- various corner processing options
- cutter compensation.

The following topics are dealt with in the paragraphs below:

- Tooling
- Geometry
- Machining Strategy Parameters
- Feeds and Speeds
- Tool Compensation
- Macros.

Tooling for Profile Finish Turning

The following tooling may be used:

- External and Internal insert-holders with all insert types except groove and thread.
- External and Internal Groove insert-holders with groove inserts.

Note that the following attributes may influence machining (they are located on the Insert-holder's Technology tab):

- Gouging angle
- Trailing angle
- Leading angle
- Max Recessing Depth
- Max Cutting Depth
- Max Boring Depth.

These attributes take tooling accessibility into account and may reduce the machined area.
However, you can use the **Insert-Holder Constraints** option on the operation editor to either ignore or apply these tooling attributes. You can replay the operation to verify the influence of these attributes on the generated tool path.

Note that the **Insert-Holder Constraints** setting does not influence the Leading and Trailing Safety Angles defined in the operation editor.

Geometry for Profile Finish Turning

A Part profile is required. It can be specified as follows:

- select edges either directly or after selecting the **By Curve** contextual command. In this case the Edge Selection toolbar appears to help you specify the guiding contour.
- select the **Sectioning** contextual command. Please refer to Sectioning for details of how to use this capability.
  Please note that the sectioning selection method is not associative.

Start Limit: None / In / On / Out
This option allows you to specify a point, line, curve or face as the start element of the part profile. If a face is specified, the start element is the intersection of the face and the working plane. The position of the start of machining is also defined with respect to this element.
In / On / Out: allows you to specify the Go-Go type positioning of the tool with respect to the start element. The On option is always used for a point type start element.
If needed, the profile may be extrapolated to the start element.

End Limit: None / In / On / Out
This option allows you to specify a point, line, curve or face as the end element of the part profile. If a face is specified, the end element is the intersection of the face and the working plane. The position of the end of machining is also defined with respect to this element.
In / On / Out: allows you to specify the Go-Go type positioning of the tool with respect to the end element. The On option is always used for a point type end element.

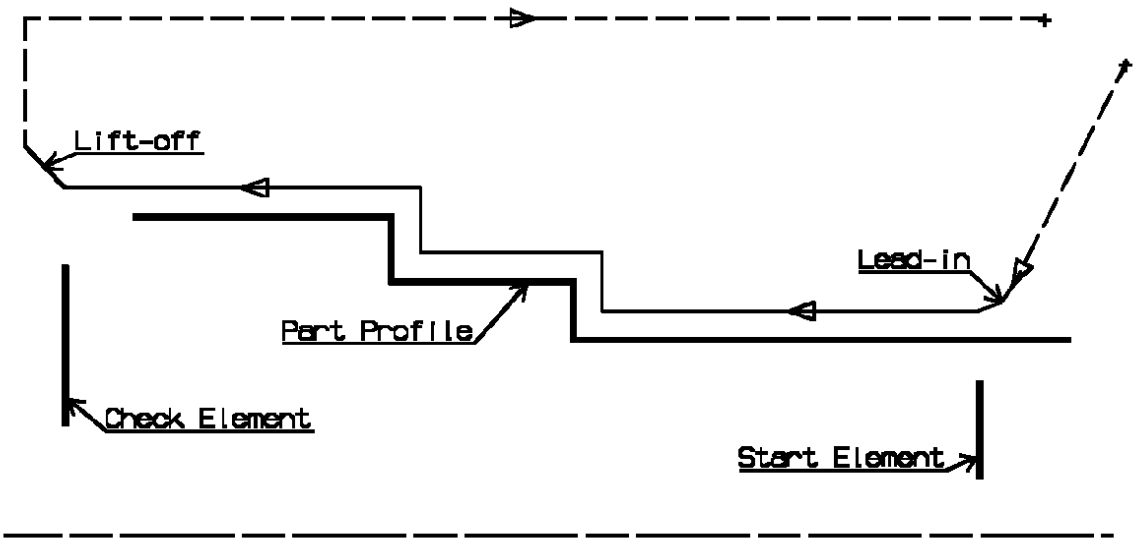
Relimiting the area to machine by means of limit elements

If you specify a point, it is projected onto the part profile.
A line through the projected point parallel to the radial axis delimits the area to machine.

If you specify a line, its intersection with the part profile is calculated (if necessary, the line is extrapolated).
A line through the intersection point parallel to the radial axis delimits the area to machine.

If you specify a curve, its intersection with the part profile is calculated (if necessary, the curve is extrapolated using the tangent at the curve extremity).
A line through the intersection point parallel to the radial axis delimits the area to machine.

If needed, the profile may be extrapolated to the end element.



The figure above illustrates the use of start and end elements for Profile Finish Turning. Profile is machined from start element. Profile is extrapolated up to end element. Direct approach and radial-axial retract.

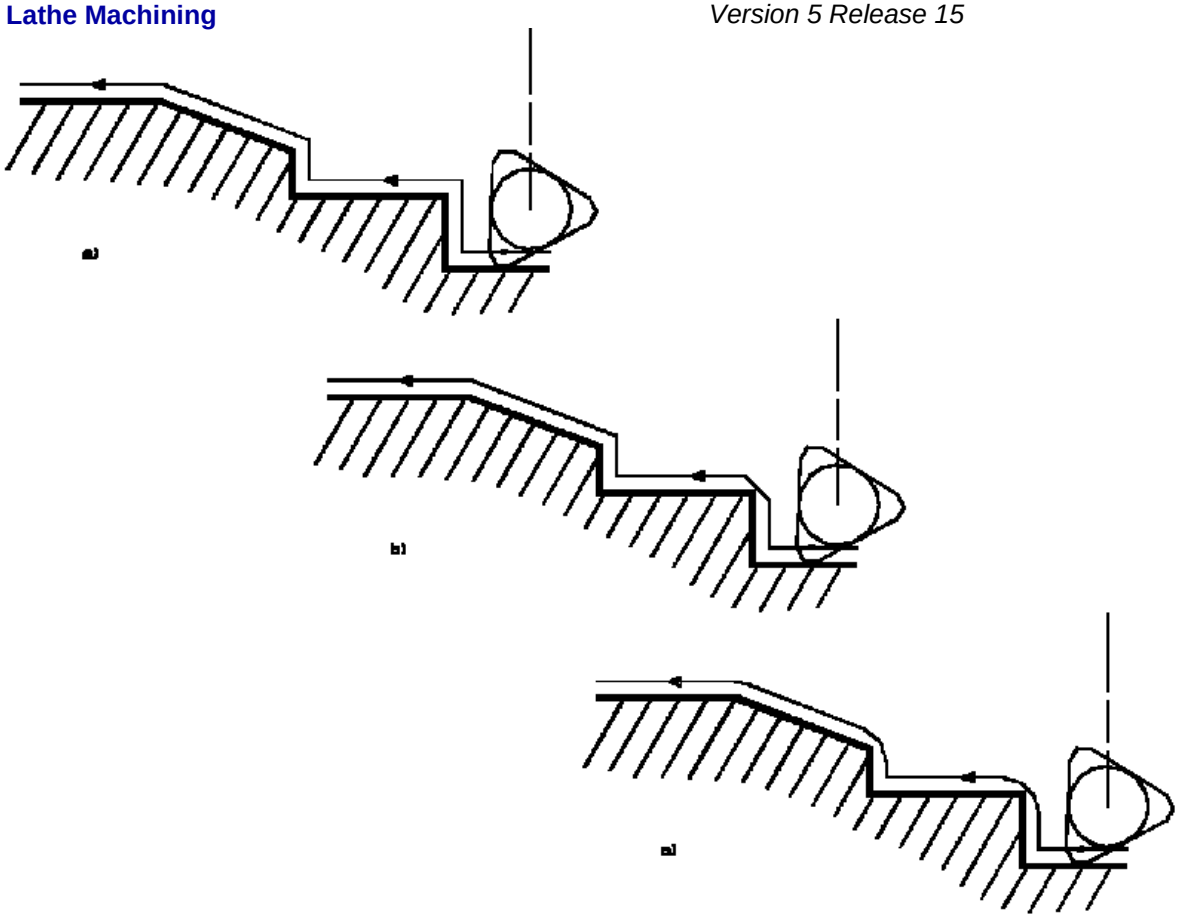
Orientation and Location for Profile Finish Turning

- **Orientation**: Internal / External / Frontal
  This option allows you to specify the type of machining according to the location of the area to machine on the part.
- **Location**:
  - Front, the profile is machined toward the head stock
  - Back, the profile is machined from the head stock.

Corner Processing for Profile Finish Turning

The following options allow you to define how corners of the profile are to be machined:

- None: no corners are to be machined along the profile
- Chamfer: only 90 degree corners of the profile are chamfered
- Rounded: all corners of the profile are rounded.



Corner processing options are also available to define how the entry and exit corners are to be machined. Entry and exit corners are defined by either a chamfer length, or a corner radius and corner angle.

Part Offsets for Profile Finish Turning

- **Part offset**, which is defined perpendicular to the part profile.
- **Axial part offset**.
- **Radial part offset**.
- **Start limit offset**: distance with respect to the start element (only if start element is a line or a curve, and when In or Out is set for start element positioning).
- **End limit offset**: distance with respect to the end element (only if end element is a line or a curve, and when In or Out is set for end element positioning).

Offsets can be positive or negative with any absolute value. The global offset applied to the part profile is the resulting value of the normal, axial and radial offsets.

In addition to these global values, local offsets can be applied to segments, curves and arcs of the part profile.

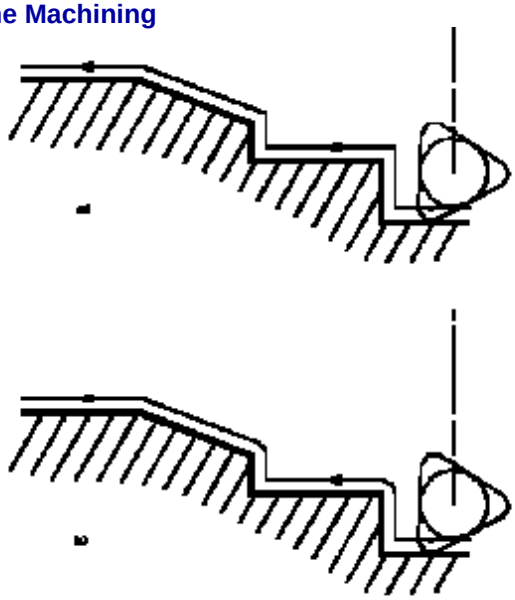
Machining Strategy Parameters for Profile Finish Turning

Path Definition for Profile Finish Turning

- **Machining Direction**: To or From Spindle
  This option is only available for frontal machining for specifying the machining direction with respect to the spindle axis.

  If start and end elements are defined that are in conflict with the machining direction, then these elements will be reversed automatically.

- **Contouring for Outside Corners**: Angular / Circular
  Allows you to define whether angular or circular contouring is to be applied to corners of the profile (only if corner processing is set to Rounded or Chamfer).



- **Under Spindle Axis Machining**
  When finishing in frontal mode, this option allows you to request machining under the spindle axis.
- **Recess Machining**
  When this option is set, a recess machining path is done after the profile finish path.
  The trailing safety angle option becomes available.
- **Leading** and **Trailing Safety Angles**
  The insert geometry is taken into account to avoid collision by reducing the maximum slope on which the tool can machine. The Leading Safety Angle and Trailing Safety Angle allow you to further reduce this slope.

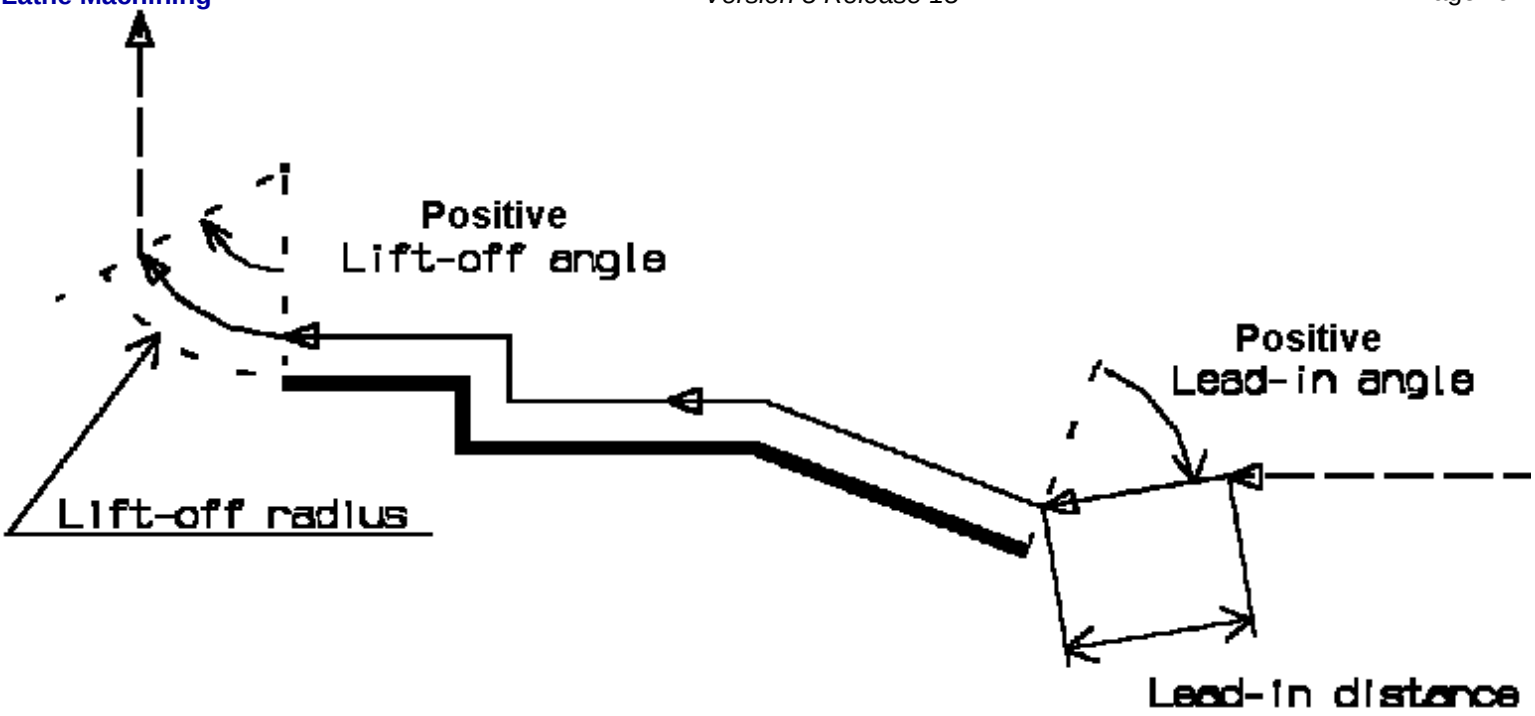
  Leading and trailing angles can also be defined on the insert-holder to define the maximum slope on which machining can be done. In this case and if the **Insert-Holder Constraints** setting is applied (see above), the angles that reduce the slope the most will be taken into account.
- **Machining Tolerance** for following the profile.

Lead-in and Lift-off for Profile Finish Turning

- **Lead-in type**: Linear / Circular
  Defines the type of lead-in onto the profile at lead-in feedrate
  - Linear: lead-in up to the point where profile machining starts is defined by means of the **lead-in distance** and **lead-in angle** parameters.
  - Circular: lead-in is circular and tangent to the point where profile machining starts. It is defined by means of the **lead-in radius** and **lead-in angle** parameters.

  Note that the lead-in angle is defined with respect to the normal to the cutting direction.

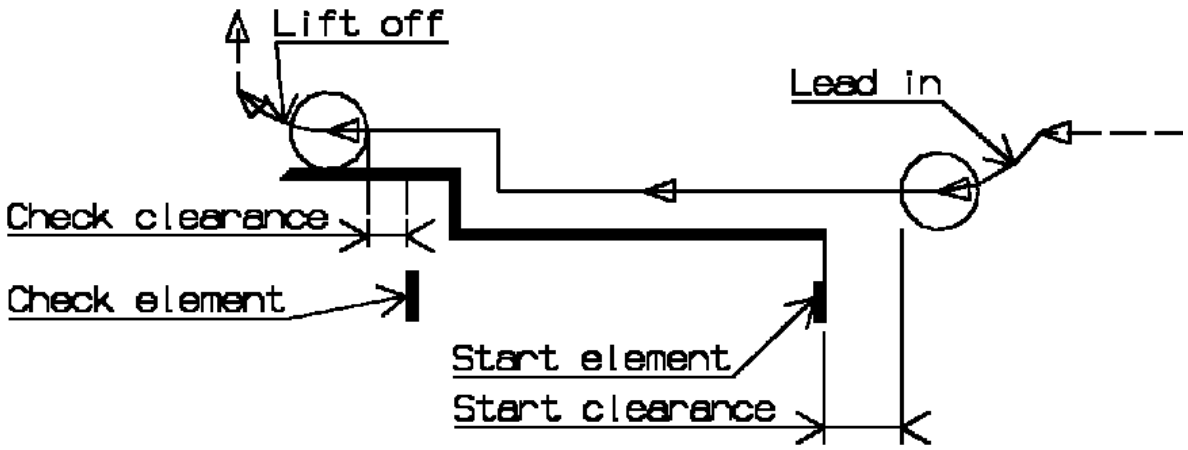
The figure below shows an example of linear lead-in and circular lift-off (external machining is assumed).



- **Lift-off type**: Linear / Circular.
  Defines the type of lift-off from the profile at lift-off feedrate
  - Linear: lift-off from the point where profile machining ends is defined by means of the **lift-off distance** and **lift-off angle** parameters.
  - Circular: lift-off is circular and tangent from the point where profile machining ends. It is defined by means of the **lift-off radius** and **lift-off angle** parameters.

  Note that the lift-off angle is defined with respect to the normal to the cutting direction.

In the example below, the round tool is tangent **In** start element plus clearance at start of profiling. Round tool is tangent **Out** end element plus clearance at end of profiling.



Local Invert

Invert strategy

- None: no Invert Strategy
- Overlap: defined by a clearance and a overlap parameter.
- Thickness: defined by a clearance and a thickness parameter.

Machine inverted elements first: you machine inverted elements the first path, and then the other elements.

Lead-in Angle and Distance or Radius for entry path.
Lift-off Angle and Distance or Radius for exit path.

Lift-off type allows to select the type of entry and exit motion between Linear and Circular.
Linear motion is defined by a distance and an angle.
Circular motion is defined by a radius and an angle.

Feeds and Speeds for Profile Finish Turning

Speed unit can be set to:

- Angular: spindle speed in revolutions per minute
- Linear: constant cutting speed in units per minute

then you can give a Machining Speed value.

Available feedrates in units per revolution are as follows:

- Machining Feedrate
- Chamfering Feedrate for machining chamfers or corners
- Lift-off Feedrate
- Lead-in Feedrate.

In addition to these global feedrates, local feedrates can be applied to segments, curves and arcs of the part profile.

Feedrates in units per minute are also available for air cutting such as macro motions and path transitions.
Note that RAPID feedrate can be replaced by Air Cutting feedrate in tool trajectories (except in macros) by selecting the checkbox in the Feed and Speeds tab page  .

Tool Compensation for Profile Finish Turning

You can select a **tool compensation number** corresponding to the desired tool output point. Note that the usable compensation numbers are defined on the tool assembly linked to the machining operation. If you do not select a tool compensation number, the output point corresponding to type P9 will be used by default.

CUTCOM (Cutter Compensation): None / On / Reverse.
If this option is set to On or Reverse, the NC output will include CUTCOM instructions in approach and retract paths for cutter compensation.

- On: CUTCOM/RIGHT instruction generated if tool is to the right of the toolpath and CUTCOM/LEFT if tool is to the left of the toolpath.
- Reverse: CUTCOM/RIGHT instruction generated if tool is to the left of the toolpath and CUTCOM/LEFT if tool is to the right of the toolpath.

See Cutter Compensation with Finish Operations for more information.

Approach and Retract Macros for Profile Finish Turning

The following Approach and Retract macros are proposed: Direct, Axial-radial, Radial-axial, and Build by user.
The selected macro type (Approach or Retract) defines the tool motion before or after machining: Various feedrates are available for the approach and retract motions (RAPID, lead-in, lift-off, and so on).

See Define Macros on a Turning Operation for more information.

Groove Finish Turning Operations

The information in this section will help you create and edit Groove Finish Turning operations in your manufacturing program.

The Groove Finish Turning operation allows you to finish a groove by means of downward profile following. You can specify:

- the type of machining according to the groove profile to be machined (external, internal, frontal or inclined)
- relimitation of the profile by start and end elements
- various approach and retract path types
- linear and circular lead-in and lift-off options with specific feedrates
- various corner processing options with specific feedrates
- local feedrates for individual elements of the machined profile
- tool output point change
- cutter compensation.

The following topics are dealt with in the paragraphs below:

- Tooling
- Geometry
- Machining Strategy Parameters
- Feeds and Speeds
- Tool Compensation
- Macros.

Tooling for Groove Finish Turning

The following tooling may be used:

- Internal, External and Frontal Groove insert-holders with groove inserts.
- External and Internal insert-holders with round inserts.

Note that the following attributes may influence machining (they are located on the Insert-holder's Technology tab):

- Gouging angle
- Trailing angle
- Leading angle
- Max Recessing Depth
- Max Cutting Depth
- Max Boring Depth.

These attributes take tooling accessibility into account and may reduce the machined area.

However, you can use the **Insert-Holder Constraints** option on the operation editor to either ignore or apply these attributes. You can replay the operation to verify the influence of these attributes on the generated tool path.

Geometry for Groove Finish Turning

The Part profile (that is, the groove finish profile) is required. It can be specified as follows:

- select edges either directly or after selecting the **By Curve** contextual command. In this case the Edge Selection toolbar appears to help you specify the guiding contour.
- select the **Sectioning** contextual command. Please refer to Sectioning for details of how to use this capability.
  Please note that the sectioning selection method is not associative.

Start Limit: None / In / On / Out
This option allows you to specify a point, line, curve or face as the start element of the groove finish profile. The position of the start of machining is also defined with respect to this element. If a face is specified, the start element is the intersection of the face and the working plane. If needed, the profile may be extrapolated to the start element.

In / On / Out allows you to specify the Go-Go type positioning of the tool with respect to the start element. The On option is always used for a point type start element.

End Limit: None / In / On / Out
This option allows you to specify a point, line, curve or face as the end element of the groove finish profile. If a face is specified, the end element is the intersection of the face and the working plane. The position of the end of machining is also defined with respect to this element. If needed, the profile may be extrapolated to the end element.

In / On / Out allows you to specify the Go-Go type positioning of the tool with respect to the end element. The On option is always used for a point type start element.

Relimiting the area to machine by means of limit elements

If you specify a point, it is projected onto the part profile.
A line through the projected point parallel to the radial axis delimits the area to machine.

If you specify a line, its intersection with the part profile is calculated (if necessary, the line is extrapolated).
A line through the intersection point parallel to the radial axis delimits the area to machine.

If you specify a curve, its intersection with the part profile is calculated (if necessary, the curve is extrapolated using the tangent at the curve extremity).
A line through the intersection point parallel to the radial axis delimits the area to machine.

Orientation for Groove Finish Turning

- **Orientation**: Internal / External / Frontal / Inclined
  This option allows you to define the orientation of the groove to be machined.

For an inclined orientation you must specify the **Angle of Incline**.

Corner Processing for Groove Finish Turning

The following options allow you to define how corners of the profile are to be machined:

- Follow profile: no corners are to be machined along the profile
- Chamfer: only 90 degree corners of the profile are chamfered
- Rounded: all corners of the profile are rounded.

Corner processing is proposed for **Entry, Exit** and **Other corners**.

- Chamfer Length if Other corner processing mode is Chamfer.
- Corner Radius if Other corner processing mode is Rounded.
- Entry Corner Chamfer Length on first flank of groove when Entry corner processing mode is Chamfer
- Entry Corner Radius on first flank of groove when Entry corner processing mode is Corner
- Entry Corner Angle on first flank of groove when Entry corner processing mode is Corner
- Exit Corner Chamfer Length on last flank of groove when Exit corner processing mode is Chamfer
- Exit Corner Radius on last flank of groove when Exit corner processing mode is Corner
- Exit Corner Angle on last flank of groove when Exit corner processing mode is Corner.

Part Offsets for Groove Finish Turning

- **Part offset**, which is defined perpendicular to the part profile.
- **Axial part offset.**
- **Radial part offset**.
- **Start limit offset**: distance with respect to the start element (only if start element is a line or a curve, and when In or Out is set for start element positioning).
- **End limit offset**: distance with respect to the end element (only if end element is a line or a curve, and when In or Out is set for end element positioning).

Offsets can be positive or negative with any absolute value. The global offset applied to the part profile is the resulting value of the normal, axial and radial offsets. In addition to these global offsets, local values can be applied to segments, curves and arcs of the part profile.

Machining Strategy Parameters for Groove Finish Turning

Path Definition for Groove Finish Turning

- **Machining Direction**
  You can specify the machining direction by means of:
  - To or From Head for Internal and External machining
  - To or From Spindle for Frontal machining
  - To Right or Left of Groove for Inclined machining

If start and end elements are defined that are in conflict with the machining direction, then these elements will be reversed automatically.

- **Clearance**: this value defines the clearance to be applied to the next flank after the first machined flank. The bottom of the groove will be machined up to the position defined by this clearance value.
- Tool **Overlap** Distance on Groove Bottom.

- **Under Spindle Axis Machining**
  When finishing in frontal mode, this option allows you to request machining under the spindle axis.
- **Contouring for Outside Corners**: Angular / Circular
  Allows you to define whether an angle or circle contouring mode is to be applied to corners of the groove profile (only if corner processing is set to None or Chamfer).
- **Machining Tolerance** for following the groove finish profile.

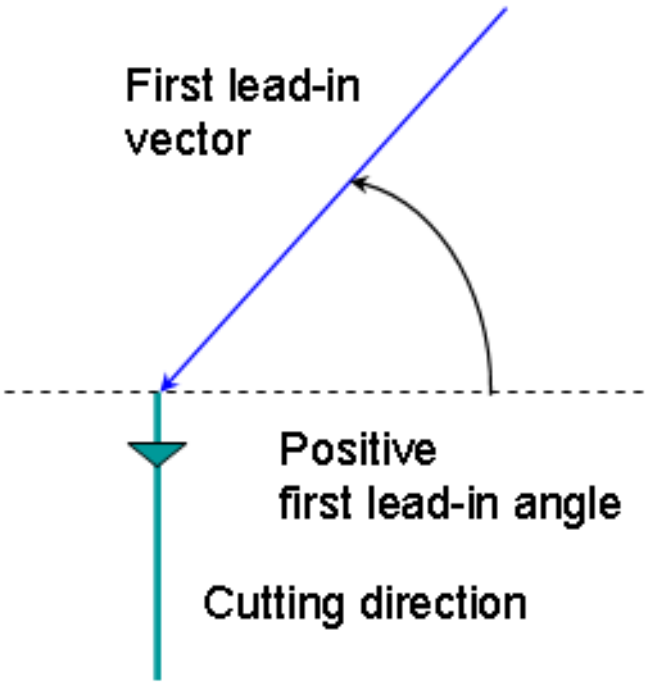
Lead-in for Groove Finish Turning

- **First Flank Lead-in: Linear / Circular**

Defines the type of lead-in at lead-in feedrate on the first flank of the groove.

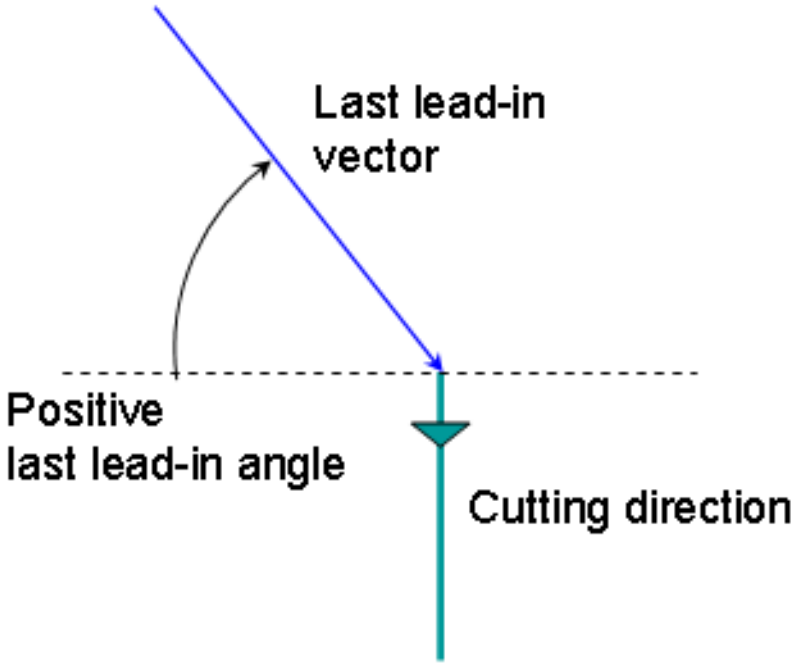
- Linear: lead-in up to the point where first flank machining starts is defined by means of the **first lead-in distance** and **first lead-in angle** parameters.
- Circular: lead-in is circular and tangent to the point where first flank machining starts. It is defined by means of the **first lead-in radius** and **first lead-in angle** parameters.

Note that the first lead-in angle is defined with respect to the normal to the cutting direction. The figure below shows the effect of a positive first lead-in angle (external machining is assumed).



- **Last Flank Lead-in: Linear / Circular**
  Defines the type of lead-in at lead-in feedrate on the last flank of the groove.
  - Linear: lead-in up to the point where last flank machining starts is defined by means of the **last lead-in distance** and **last lead-in angle** parameters.
  - Circular: lead-in is circular and tangent to the point where last flank machining starts. It is defined by means of the **last lead-in radius** and **last lead-in angle** parameters.

Note that the last lead-in angle is defined with respect to the normal to the cutting direction. The figure below shows the effect of a positive last lead-in angle (external machining is assumed).



- **Other Flank Lead-in: Linear / Circular**
  For a groove that has multiple recesses, this option defines the type of lead-in required to machine flanks other than the first and last flanks.

  - **Other Lead-in Distance** on other flanks of the groove when other flank lead-in type is Linear
  - **Other Lead-in Angle** on other flanks of the groove when other flank lead-in type is Linear or Circular
  - **Other Lead-in Radius** on other flanks of the groove when other flank lead-in type is Circular.

  Note that the other lead-in angle is defined with respect to the cutting direction.

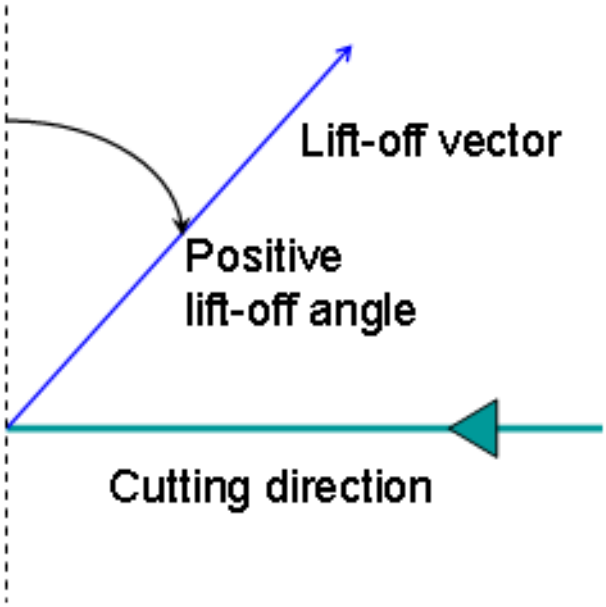
Lift-off for Groove Finish Turning

- **Lift-off Type: Linear / Circular**

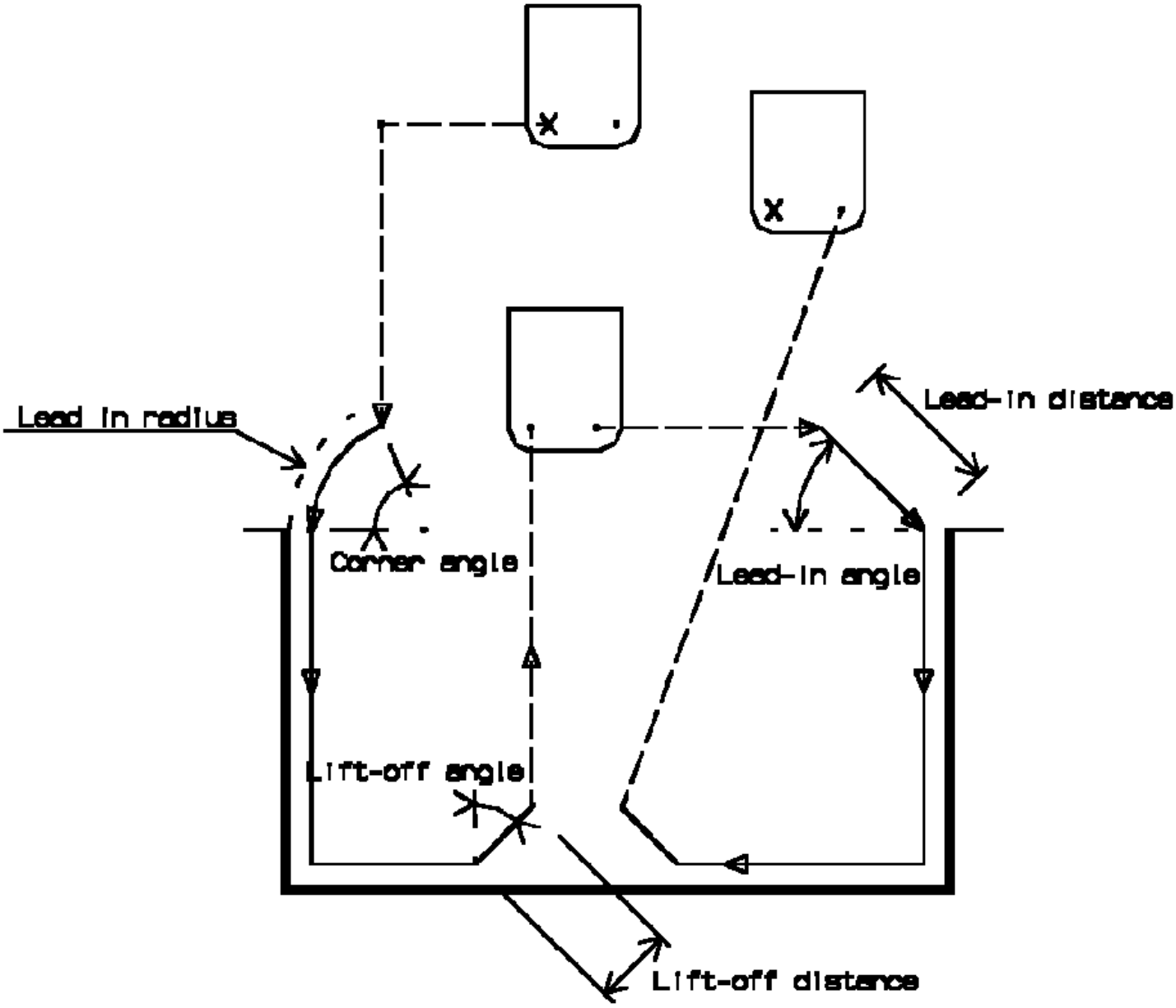
Defines the type of lift-off from the groove at lift-off feedrate.

- **Lift-off Distance** when lift-off type is Linear.
- **Lift-off Angle** when lift-off type is Linear or Circular.
- **Lift-off Radius** when lift-off type is Circular.

Note that the lift-off angle is defined with respect to the normal to the cutting direction. The figure below shows the effect of a positive lift-off angle (external machining is assumed).



The example below shows Linear lead-in and Circular lift-off for Groove Finish Turning.



Feeds and Speeds for Groove Finish Turning

Speed unit can be set to:

- Angular: spindle speed in revolutions per minute
- Linear: constant cutting speed in units per minute

then you can give a Machining Speed value.

Available feedrates in units per revolution are as follows:

- **Machining Feedrate**
- **Chamfering Feedrate** for machining chamfers or corners
- **Lift-off Feedrate**
- **Lead-in Feedrate**.

In addition to these global feedrates, local feedrates can be applied to segments, curves and arcs of the part profile.

Feedrates in units per minute are also available for air cutting such as macro motions and path transitions. Note that RAPID feedrate can be replaced by Air Cutting feedrate in tool trajectories (except in macros) by selecting the checkbox in the Feed and Speeds tab page  .

Tool Compensation for Groove Finish Turning

You can select a **tool compensation number** corresponding to the desired tool output point. Note that the usable compensation numbers are defined on the tool assembly linked to the machining operation. If you do not select a tool compensation number, the output point corresponding to type P9 will be used by default.

CUTCOM (Cutter Compensation): None / On / Reverse
If this option is set to On or Reverse, the NC output will include CUTCOM instructions in approach and retract paths for cutter compensation.

- On: CUTCOM/RIGHT instruction generated if tool is to the right of the toolpath and CUTCOM/LEFT if tool is to the left of the toolpath.
- Reverse: CUTCOM/RIGHT instruction generated if tool is to the left of the toolpath and CUTCOM/LEFT if tool is to the right of the toolpath.

See Cutter Compensation with Finish Operations for more information.

Note that the change of output point is managed automatically if you have set the **Change Output Point** option. If the output point is consistent with the flank of the groove to be machined, the output point is changed when the other flank of the groove is machined. At the end of the operation, the output point is the same as it was at the start of the operation. See Changing the Output Point for more information.

Approach and Retract Macros for Groove Finishing

The following Approach and Retract macros are proposed: re proposed: Direct, Axial-radial, Radial-axial, and Build by user.
The selected macro type (Approach or Retract) defines the tool motion before or after machining: Various

feedrates are available for the approach and retract motions (RAPID, lead-in, lift-off, and so on).

See Define Macros on a Turning Operation for more information.

Ramp Rough Turning Operations

The information in this section will help you create and edit Ramp Rough Turning operations in your manufacturing program. This type of operation is suitable for machining hard materials using round ceramic inserts, thereby minimizing wear and cutting stress.

The Ramp Rough Turning operation allows you to specify:

- Longitudinal and Face roughing modes
- external, internal or frontal machining according to the type of area to machine
- relimitation of the area to machine
- various approach and retract path types
- various lead-in and lift-off options with specific feedrates
- rework machining.

The following topics are dealt with in the paragraphs below:

- Tooling
- Geometry including Part and Stock Offsets
- Location and Limits
- Machining Strategy Parameters
- Feeds and Speeds
- Tool Compensation
- Macros.

Tooling for Ramp Rough Turning

The following tooling may be used:

- External and Internal insert-holders with round inserts.

Note that the following attributes may influence machining (they are located on the Insert-holder's Technology tab):

- Trailing angle
- Leading angle
- Max Recessing Depth
- Max Boring Depth.

These attributes take tooling accessibility into account and may reduce the machined area.
However, you can use the **Insert-Holder Constraints** option on the operation editor to either ignore or apply these tooling attributes. You can replay the operation to verify the influence of these attributes on the generated tool path.

Note that the **Insert-Holder Constraints** setting does not influence the Entry Flank Angle defined in the operation editor.

Geometry for Ramp Rough Turning

Part and Stock profiles are required. They can be specified as follows:

- select edges either directly or after selecting the **By Curve** contextual command. In this case the Edge Selection toolbar appears to help you specify the guiding contour.
- select the **Sectioning** contextual command. Please refer to Sectioning for details of how to use this capability. Please note that the sectioning selection method is not associative.

The **End Limit** option allows you to specify a point, line, curve or face to relimit or extrapolate the selected part profile. If a face is specified, the end element is the intersection of the face and the working plane. The position of the end of machining is defined with respect to this element by one of the following settings: None / In / On / Out.

Relimiting the area to machine by means of limit elements

If you specify a point, it is projected onto the part profile.
A line through the projected point parallel to the radial axis delimits the area to machine.

If you specify a line, its intersection with the part profile is calculated (if necessary, the line is extrapolated).
A line through the intersection point parallel to the radial axis delimits the area to machine.

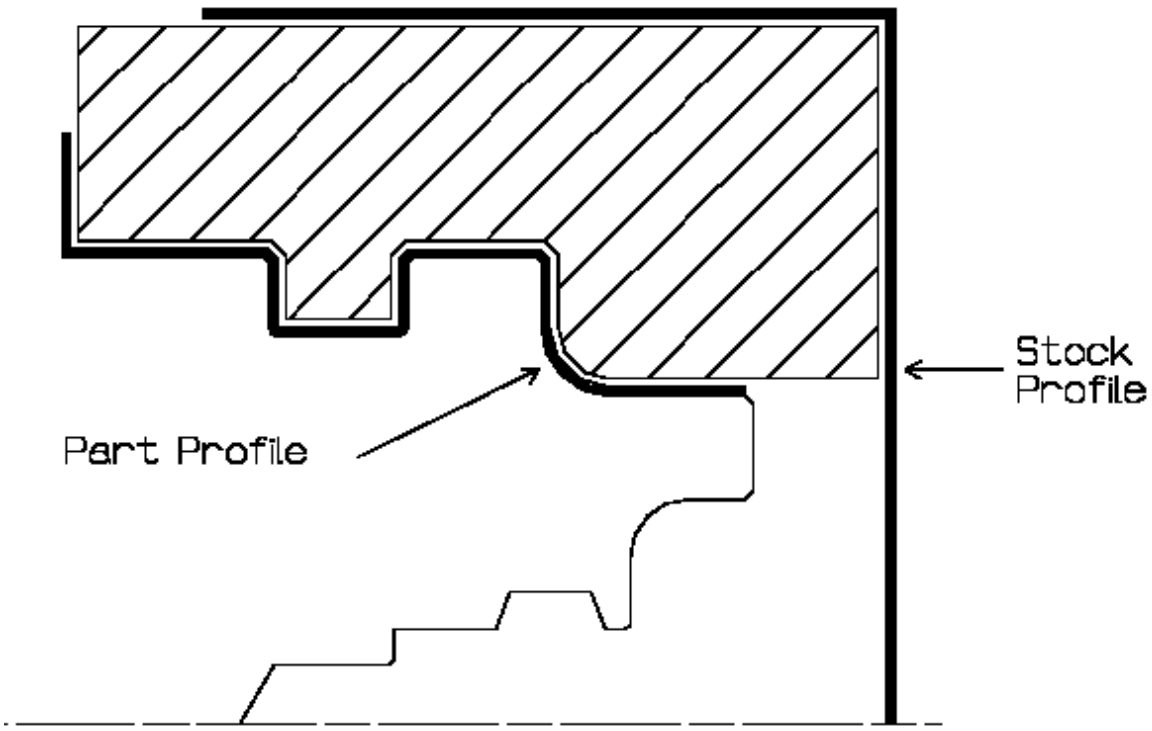
If you specify a curve, its intersection with the part profile is calculated (if necessary, the curve is extrapolated using the tangent at the curve extremity).
A line through the intersection point parallel to the radial axis delimits the area to machine.

Orientation for Ramp Rough Turning

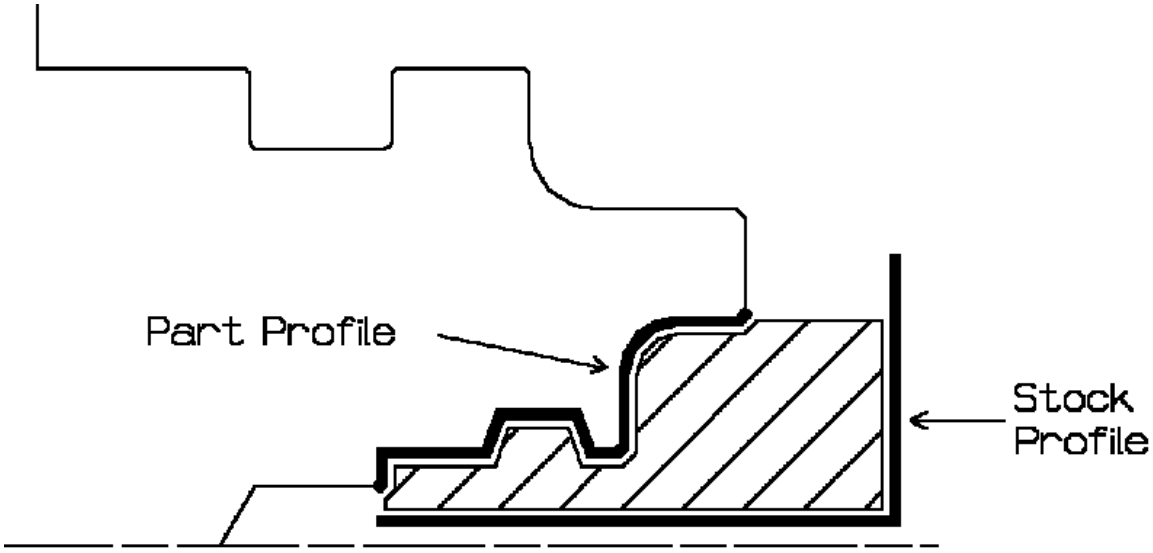
The following **Orientations** are proposed: Internal, External and Frontal (for Face Roughing only).

The selected Orientation defines the type of geometric relimitation to be done between the stock and part geometry in order to determine the area to machine. Selected part and stock profiles do not need to be joined (see the following figures).

External Ramp Rough Turning



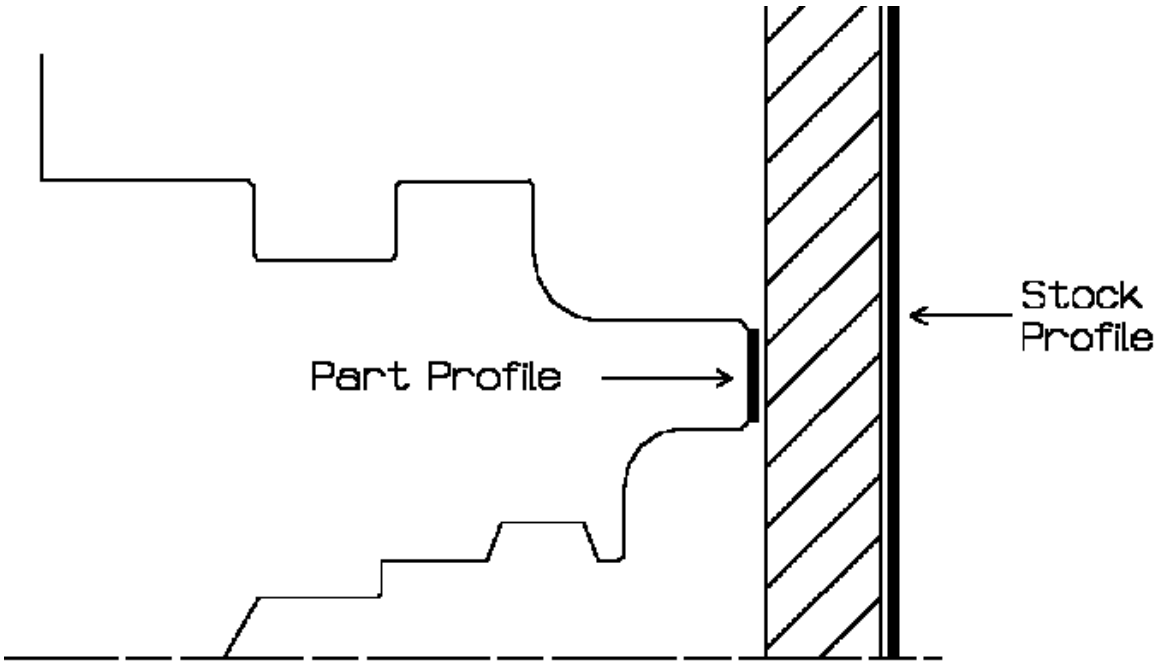
Internal Ramp Rough Turning



Frontal Ramp Rough Turning

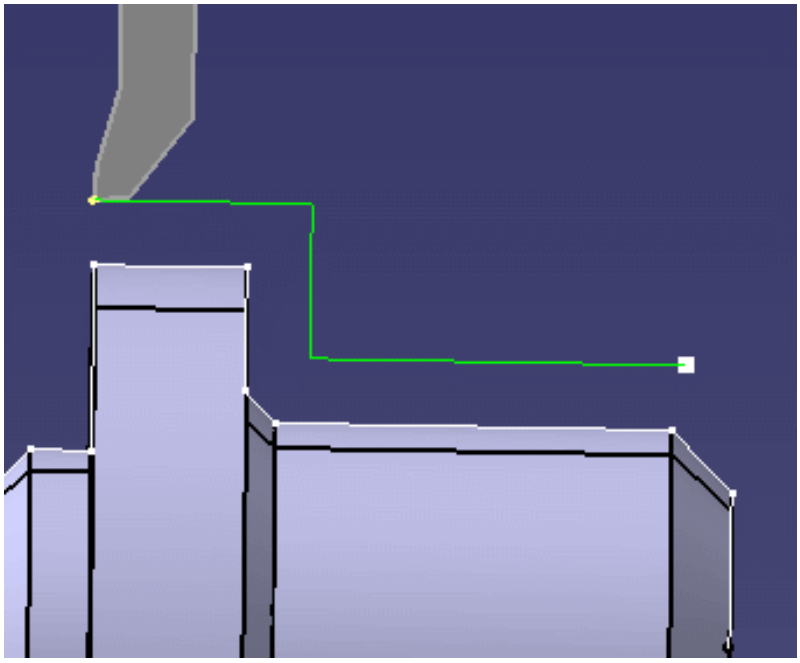
Frontal machining is proposed for face Ramp Rough Turning. In that case, the minimum and maximum diameters of the area to machine are determined by the stock profile dimensions.

For example, in the following figure the area to machine is relimited by the spindle axis because the stock profile is also relimited by the spindle axis.

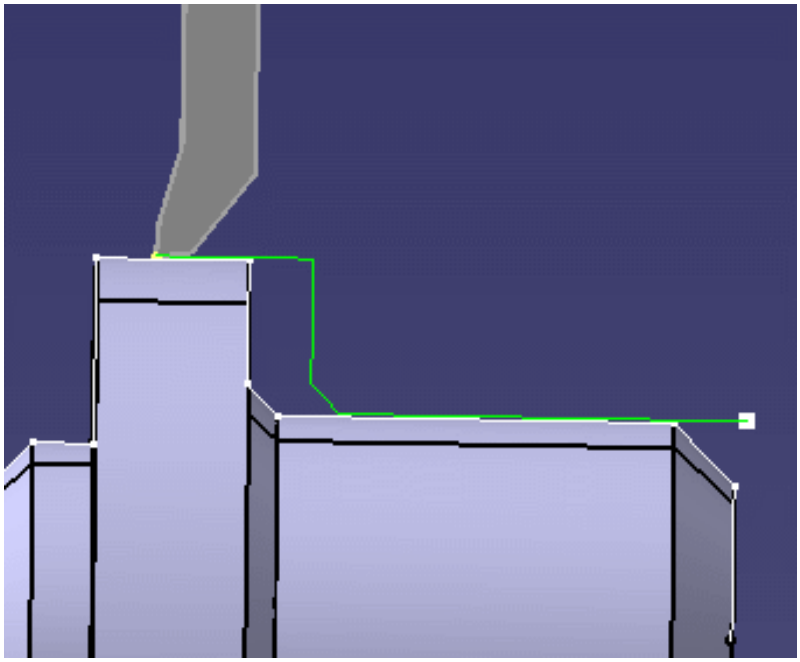


Part and Stock Offsets for Ramp Rough Turning

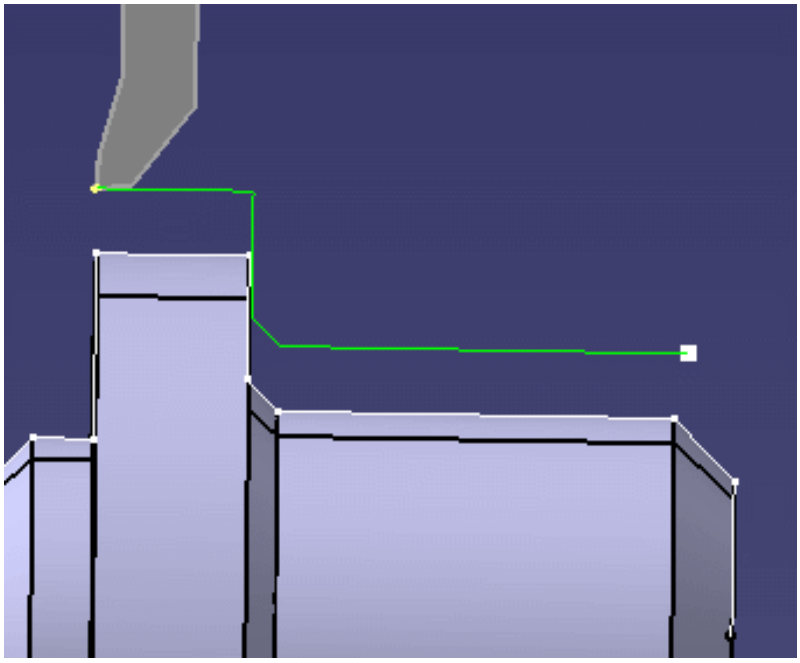
- **Stock offset**: specifies a virtual displacement of the stock profile.
- **Part offset**: specifies a virtual displacement of the part profile.



- **Axial part offset**: specifies a virtual displacement of the part profile along the spindle axis direction.



- **Radial part offset**: specifies a virtual displacement of the part profile in the radial axis direction.



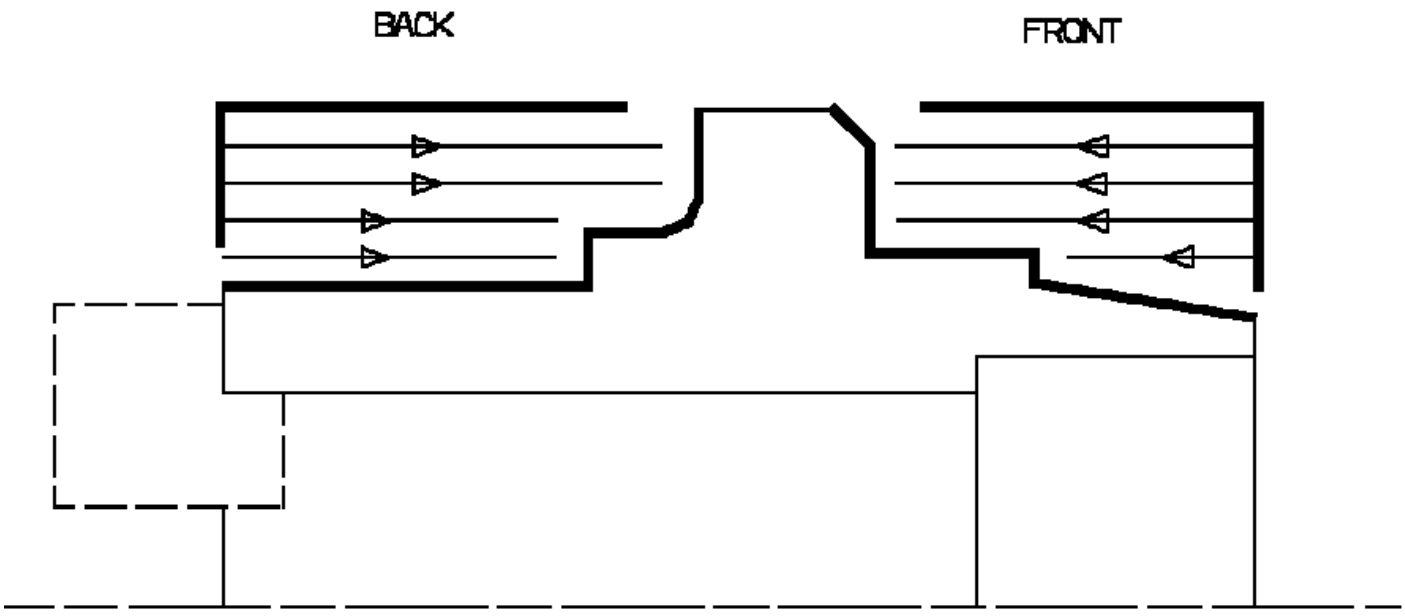
- **End limit offset**: distance with respect to the end element (only if end element is a line or a curve, and when In or Out is set for end element positioning).

Offsets can be positive or negative with any absolute value. The global offset applied to the part profile is the resulting value of the normal, axial and radial offsets.

Location and Limits for Ramp Rough Turning

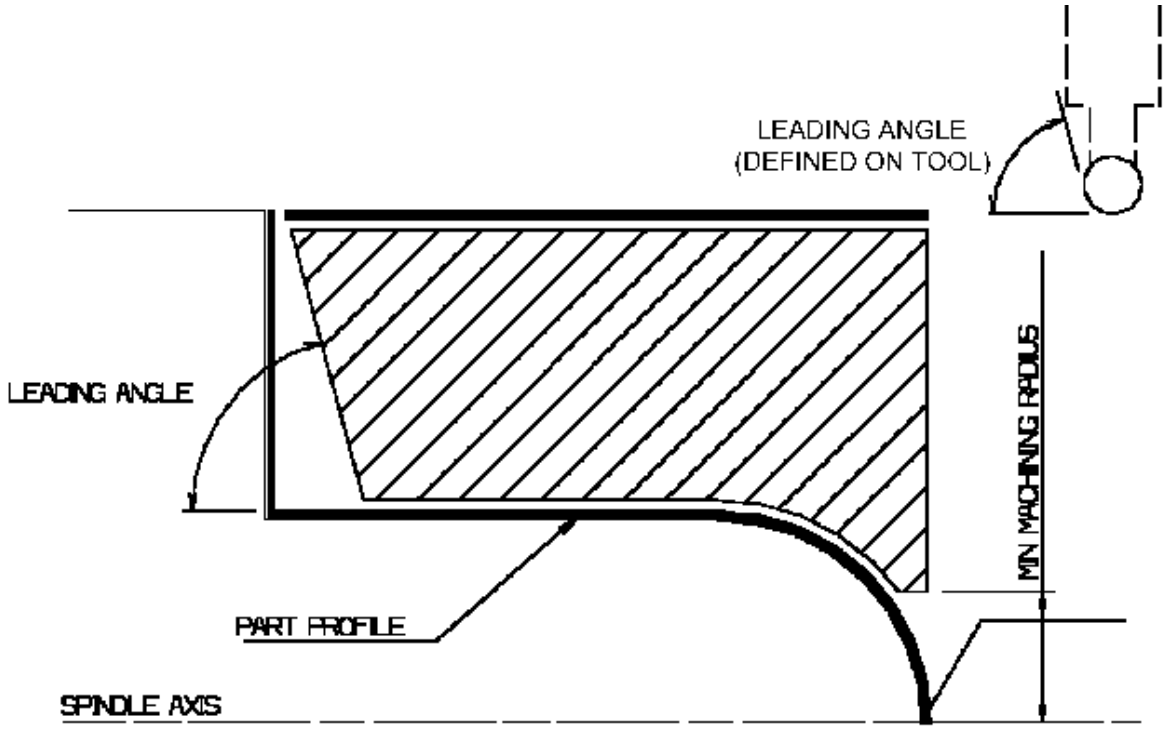
The following machining **Locations** are proposed:

- Front, the part is machined toward the head stock
- Back, the part is machined from the head stock.

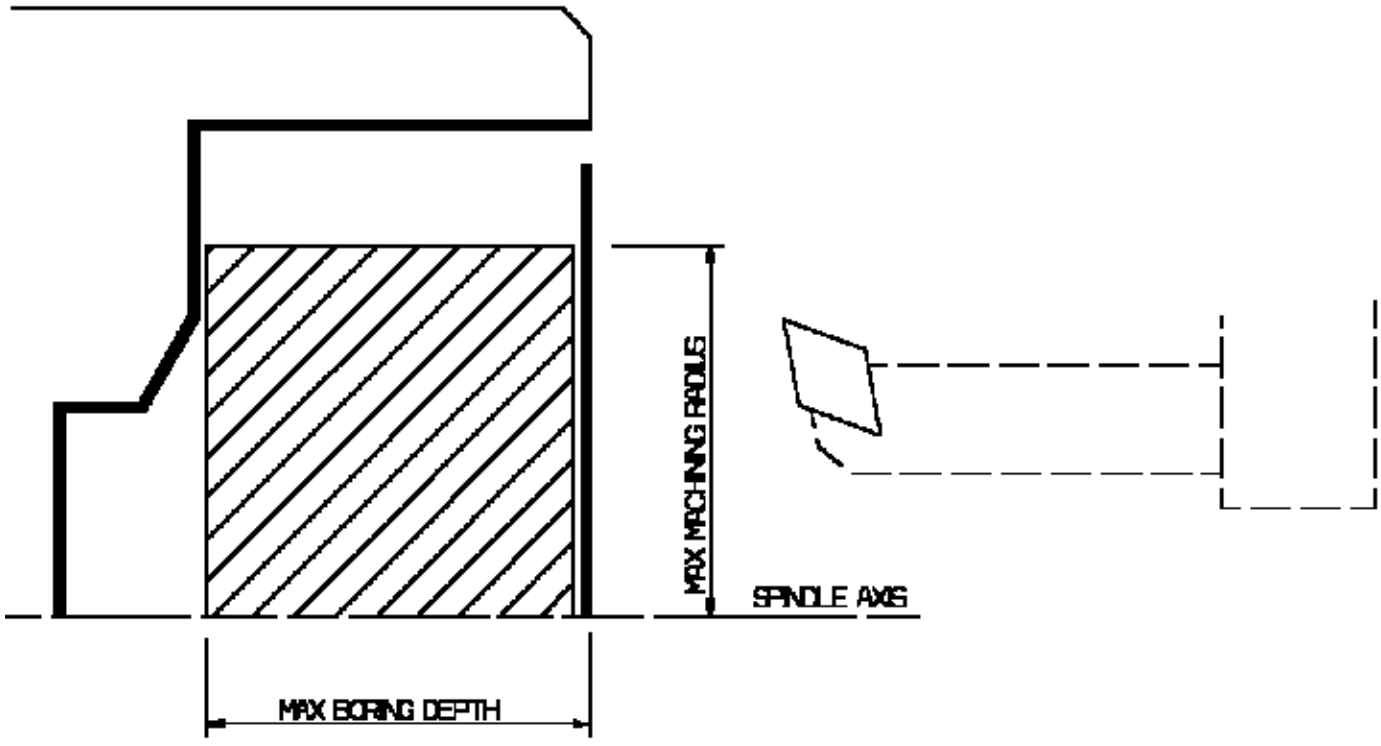


Orientation and Location settings determine the way the program closes the area to machine. The following options allow you to restrict the area to machine that is pre-defined by the stock and part. You may want to restrict this area due to the physical characteristics of the tool and the type of machining to be done.

Minimum Machining Radius



Maximum Machining Radius (for internal machining)



Note that Max Boring Depth is defined on the tool.

Axial Limit for Chuck Jaws (for external or frontal machining): Offset defined from the machining axis system.

Machining Strategy Parameters for Ramp Rough Turning

Path Definition for Ramp Rough Turning

- **Machining tolerance**
- **Max Depth of Cut**

This option is used to specify the maximum distance between passes.

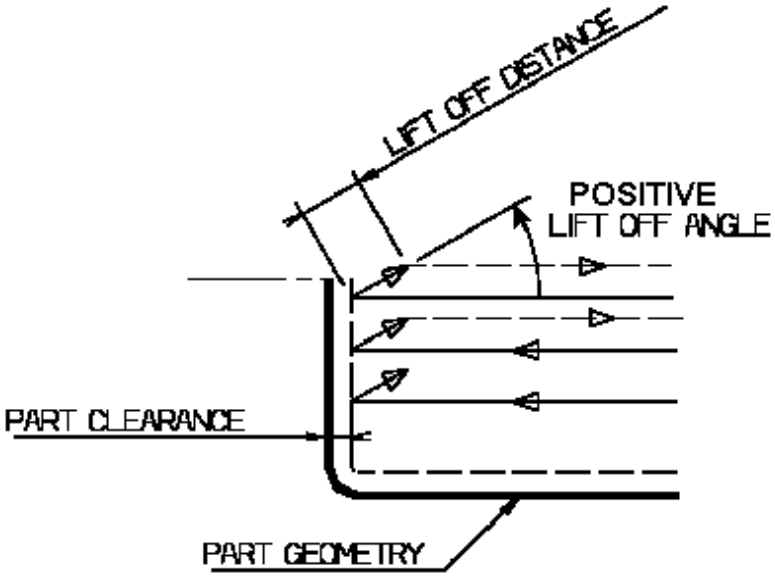
- **Under Spindle Axis Machining**
  For Face Ramp Rough Turning, this option allows you to request machining under the spindle axis.
- **Machining Direction** (only for Face Roughing with Frontal machining)
  You can specify the machining direction with respect to the spindle axis by means of the To/From Spindle Axis choice.
- **Entry Flank Angle**
  The insert geometry is taken into account to avoid collision by reducing the maximum slope on which machining can be done. Defining a Entry Flank Angle on the operation allows you to further reduce the area to machine.

  Note that a leading angle can also be defined on the insert-holder to define the maximum slope on which machining can be done. In this case and if the **Insert-Holder Constraints** setting is applied (see above), the angle that reduces the slope the most will be taken into account.
- **Start pass mode** is defined by one of the following settings: Chamfer, Rounded, None.
- If **Rework** mode is set, the following options are available:
  - Distance before Rework Plunge
  - Angle before Rework Plunge.

Lead-in and Lift-off for Ramp Rough Turning

These options allow penetration into the workpiece at a reduced feedrate in order to prevent tool damage.

- **Lead-in Distance**
  Defined with respect to the cutting direction. It takes the stock profile and stock clearance into account. The tool is in RAPID mode before this distance.
- **Lift-off Distance** and **Lift-off Angle**
  These parameters define the lift-off vector at the end of each pass with respect to the cutting direction.



Feeds and Speeds for Ramp Rough Turning

Speed unit can be set to:

- Angular: spindle speed in revolutions per minute
- Linear: constant cutting speed in units per minute

then you can give a Machining Speed value.

Available feedrates in units per revolution are as follows:

- Machining
- Lead-in
- Lift-off
- Plunge
- Light loading
- Air cutting.

Feedrates in units per minute are also available for air cutting (for example, during macro motions and path transitions). Note that RAPID feedrate can be replaced by Air Cutting feedrate in tool trajectories (except in macros) by selecting the checkbox in the Feed and Speeds tab page  .

Dwell setting indicates whether the tool dwell at the end of each path is to be set in seconds or a number of spindle revolutions.

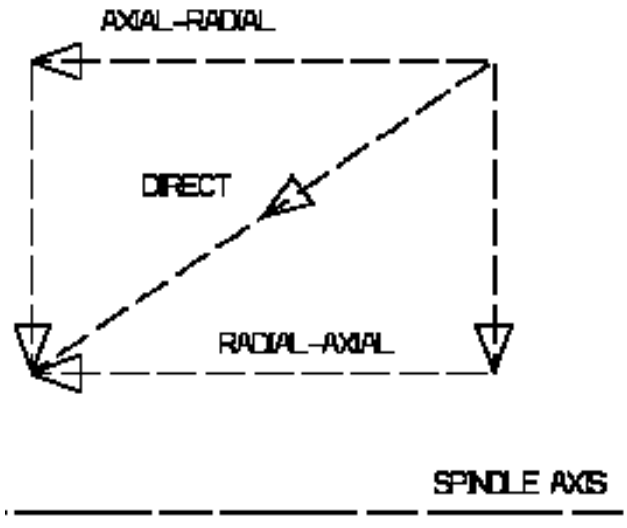
Please note that decimal values can be used for the number of revolutions. For example, when machining big parts that have a large volume, it can be useful to specify a dwell using a value of less than one revolution (0.25, for example).

Tool Compensation for Ramp Rough Turning

You can select a **tool compensation number** corresponding to the desired tool output point. Note that the usable compensation numbers are defined on the tool assembly linked to the machining operation. If you do not select a tool compensation number, the output point corresponding to type P9 will be used by default.

Approach, Retract and Linking Macros for Ramp Rough Turning

The following Approach and Retract macro modes are proposed: Direct, Axial-radial, Radial-axial, and Build by user. The selected macro type (Approach or Retract) defines the tool motion before or after machining. Various feedrates are available for the approach and retract motions (RAPID, lead-in, lift-off, and so on).



Linking macros, which comprise retract and approach motion can also be used on Ramp Rough Turning operations. Approach and retract motions of Linking macros are interruptible. It can be useful to interrupt an operation when the foreseeable lifetime of the insert is not long enough to complete the machining.

See Define Macros on a Turning Operation for more information.

Ramp Recess Turning Operations

The information in this section will help you create and edit Ramp Recess Turning operations in your manufacturing program. This type of operation is suitable for machining hard materials using round ceramic inserts, thereby minimizing wear and cutting stress.

The Ramp Recess Turning operation allows you to machine a recess by means of a One Way or Zig Zag tool motion.

You can specify:

- external, internal, frontal or inclined machining according to the type of area to machine
- various approach and retract path types
- various plunge and lift-off options with specific feedrates
- part contouring
- rework
- tool output point change.

The following topics are dealt with in the paragraphs below:

- Tools
- Geometry
- Machining Strategy Parameters
- Feeds and Speeds
- Tool Compensation
- Macros.

Tooling for Ramp Recess Turning

The following tooling may be used:

- External and Internal insert-holders with round inserts.

Note that the following attributes may influence machining (they are located on the Insert-holder's Technology tab):

- Trailing angle
- Leading angle
- Max Recessing Depth
- Max Boring Depth.

These attributes take tooling accessibility into account and may reduce the machined area.
However, you can use the **Insert-Holder Constraints** option on the operation editor to either ignore or apply these tooling attributes. You can replay the operation to verify the influence of these attributes on the generated tool path.

Note that the **Insert-Holder Constraints** setting does not influence the Flank Gouging Angle or the Entry and Exit Flank Angles defined in the operation editor.

Geometry for Ramp Recess Turning

Part and Stock profiles are required. They can be specified as follows:

- select edges either directly or after selecting the **By Curve** contextual command. In this case the Edge Selection toolbar appears to help you specify the guiding contour.
- select the **Sectioning** contextual command. Please refer to Sectioning for details of how to use this capability.
  Please note that the sectioning selection method is not associative.

Orientation for Ramp Recess Turning

The following **Orientations** are proposed: internal, external, frontal and inclined.

The selected Orientation defines the type of geometric relimitation to be done between the stock and part geometry in order to determine the area to machine. For an inclined orientation you must specify the **Angle of Incline**.

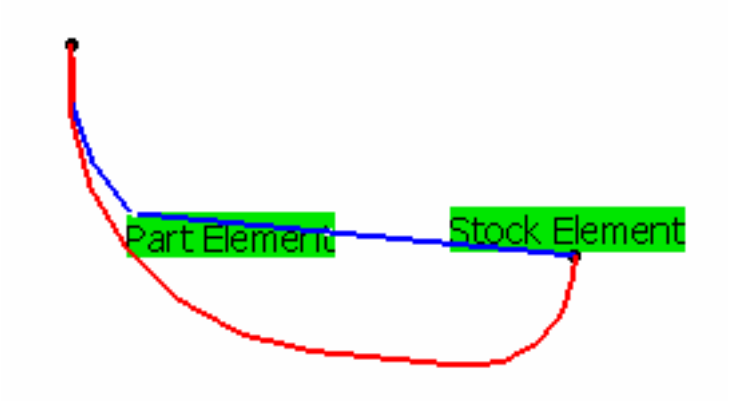
Part and Stock Offsets for Ramp Recess Turning

- **Stock offset**, which is defined perpendicular to the stock profile
- **Part offset**, which is defined perpendicular to the part profile.
- **Axial part offset.**
- **Radial part offset**.

Offsets can be positive or negative with any absolute value. The global offset applied to the part profile is the resulting value of the normal, axial and radial offsets.

Unsupported Geometry for Ramp Recess Turning

In a case like the one shown in the figure below, if the depth of cut is not sufficient, it is not possible to reach the righthand flank.



A warning message is issued recommending you to machine the profile in two operations.
Other possibilities to work around the problem are:

- add an offset to the profile
- increase the depth of cut.

Machining Strategy Parameters for Ramp Recess Turning

Path Definition for Ramp Recess Turning

- **Ramp Recess Turning Mode: One Way or Zig Zag**
- **Max Depth of Cut**
  This option is used to specify the maximum distance between passes.
- **Machining Tolerance**
- **Machining Direction**
  For Zig Zag tool motion, you must specify a first cutting direction as follows:
  - To or From Head Stock for Internal and External machining
  - To or From Spindle for Frontal machining
  - To Right or Left of Recess for Inclined machining.

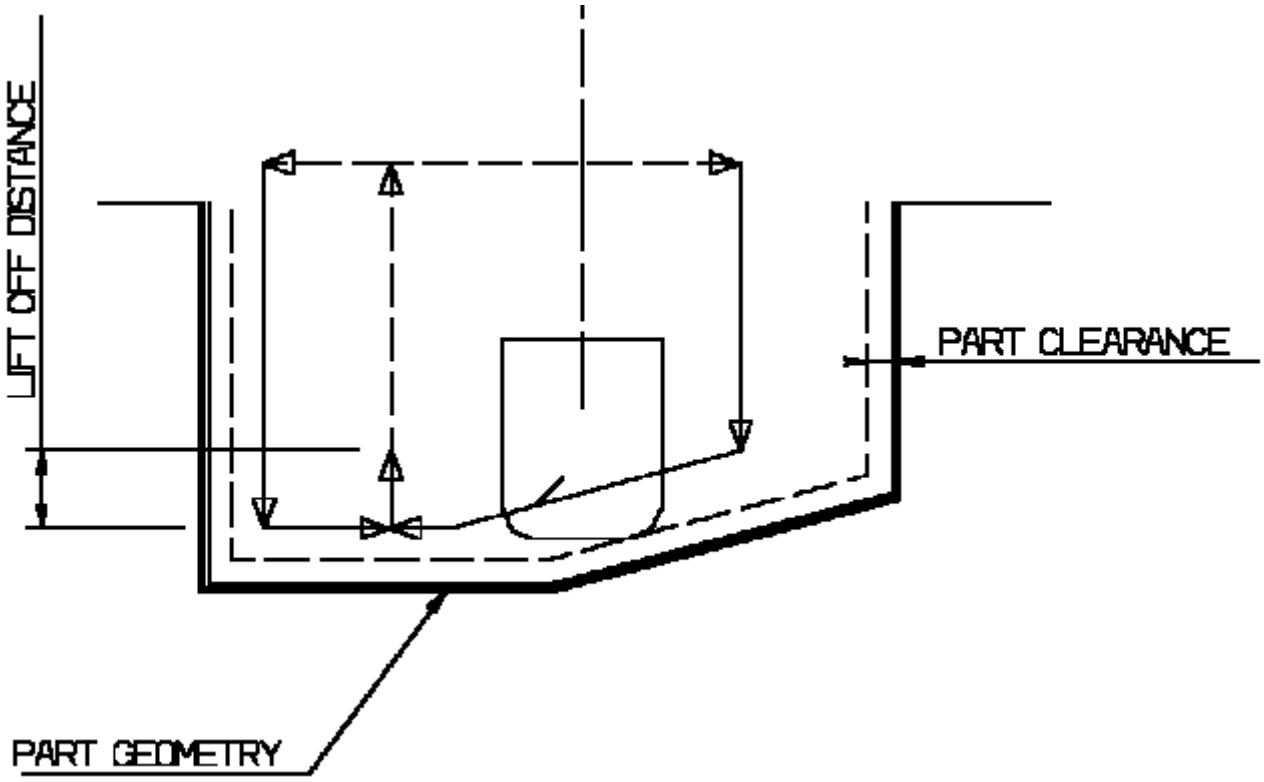
  When a part profile has multiple recesses (that is, a non-convex profile along the cutting direction), only the first recess along the specified direction is machined.
- **Entry** and **Exit Flank Angles** (for **One way** mode only).
  The insert geometry is taken into account to avoid collision by reducing the maximum slope on which machining can be done. Defining Entry and Exit Flank Angles on the operation allow you to further reduce the area to machine.

  Note that leading and trailing angles can also be defined on the insert-holder to define the maximum slope on which machining can be done. In this case and if the **Insert-Holder Constraints** setting is applied (see above),the angles that reduce the slope the most will be taken into account.
- **Flank Gouging Angle** (for **Zig Zag** mode only)
  Angles of the insert are taken into account to avoid collision by reducing the maximum slope on which machining can be done. Defining a Flank Gouging Angle on the operation allows you to further reduce the area to machine.

  Note that a gouging angle can also be defined on the insert-holder to define the maximum slope on which machining can be done. In this case and if the **Insert-Holder Constraints** setting is applied (see above),the angle that reduces the slope the most will be taken into account.
- **Under Spindle Axis Machining**

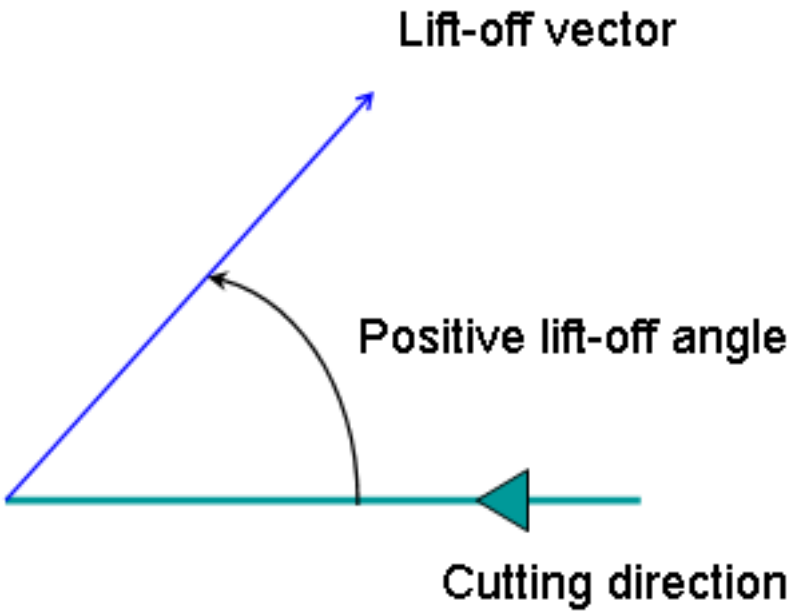
For Frontal or Inclined machining, this option allows you to request machining under the spindle axis.

- **Part Contouring**
  You can specify if contouring is required by means of the proposed checkbox.
  The part profile is followed at the end of recessing. This is done by machining down the sides of the recess in order to clear the profile.



Plunge and Lift-off for Ramp Recess Turning

- **Angle and Distance before Plunge**
  Define the plunge vector before each new pass with respect to the cutting direction.
- **Lift-off Distance** and **Lift-off Angle**
  Define the lift-off vector at the end of the last pass with respect to the cutting direction. The figure below shows the effect of a positive lift-off angle (external machining is assumed).



- If **Rework** mode is set, the following options are available:
  - Distance before Rework Plunge
  - Angle before Rework Plunge.

Feeds and Speeds for Ramp Recess Turning

Speed unit can be set to:

- Angular: spindle speed in revolutions per minute
- Linear: constant cutting speed in units per minute

then you can give a Machining Speed value.

Available feedrates in units per revolution are as follows:

- Machining
- Lift-off
- Contouring
- Plunge
- Light loading
- Air cutting.

Dwell setting indicates whether the tool dwell at the end of each path is to be set in seconds or a number of spindle revolutions.

Feedrates in units per minute are also available for air cutting such as macro motions and path transitions. Note that RAPID feedrate can be replaced by Air Cutting feedrate in tool trajectories (except in macros) by selecting the checkbox in the Feed and Speeds tab page  .

Please note that decimal values can be used for the number of revolutions. For example, when machining

big parts that have a large volume, it can be useful to specify a dwell using a value of less than one revolution (0.25, for example).

Tool Compensation for Ramp Recess Turning

You can select a **tool compensation number** corresponding to the desired tool output point. Note that the usable compensation numbers are defined on the tool assembly linked to the machining operation. If you do not select a tool compensation number, the output point corresponding to type P9 will be used by default.

Note that the change of output point is managed automatically if you set the **Change Output Point** option.

If the output point is consistent with the flank of the recess to be machined, the output point is changed when the other flank of the recess is machined.
At the end of the operation, the output point is the same as it was at the start of the operation. See Changing the Output Point for more information.

Approach, Retract and Linking Macros for Ramp Recess Turning

The following Approach and Retract macros are proposed: Direct, Axial-radial, Radial-axial, and Build by user.
The selected macro type (Approach or Retract) defines the tool motion before or after machining. Various feedrates are available for the approach and retract motions (RAPID, lead-in, lift-off, and so on).

Linking macros, which comprise retract and approach motion can also be used on Ramp Recess Turning operations.
Approach and retract motions of Linking macros are interruptible. It can be useful to interrupt an operation when the foreseeable lifetime of the insert is not long enough to complete the machining.

See Define Macros on a Turning Operation for more information.

Thread Turning Operations

The information in this section will help you create and edit Thread Turning operations in your manufacturing program.

The Thread Turning operation allows you to specify:

- the type of machining according to the required thread (external or internal)
- relimitation of the profile by start and end elements
- thread machining options
- various approach and retract types
- PP Word syntaxes.

The following topics are dealt with in the paragraphs below:

- Tooling
- Geometry
- Machining Strategy Parameters
- CYCLE or GOTO Output Syntaxes
- Editing CYCLE Syntaxes
- Tool Compensation
- Macros.

Tooling for Thread Turning

The following tooling may be used:

- Internal and External threading insert-holders with thread inserts.

Geometry for Thread Turning

A Part profile is required. It can be specified as follows:

- select edges either directly or after selecting the **By Curve** contextual command. In this case the Edge Selection toolbar appears to help you specify the guiding contour.
- select the **Sectioning** contextual command. Please refer to Sectioning for details of how to use this capability.
  Please note that the sectioning selection method is not associative.

End Limit: None / In / On / Out
This option allows you to specify a point, line, curve or face as the end element of the profile to be machined. If a face is specified, the end element is the intersection of the face and the working plane. The position of the end of machining is also defined with respect to this element.
In / On / Out allows you to specify the Go-Go type positioning of the tool with respect to the end element.

The On option is always used for a point type end element.
If needed, the profile may be extrapolated to the end element.

End Limit Offset: distance with respect to the end element (only if end element is a line, curve or face, and when In or Out is set for end element positioning).

Start Limit: None / In / On / Out
This option allows you to specify a point, line, curve or face as the start element of the profile to be machined. If a face is specified, the start element is the intersection of the face and the working plane. The position of the start of machining is also defined with respect to this element.
In / On / Out allows you to specify the Go-Go type positioning of the tool with respect to the start element.
The On option is always used for a point type end element.
If needed, the profile may be extrapolated to the start element.

Start Limit offset: distance with respect to the start element (only if start element is a line, curve or face, and when In or Out is set for start element positioning).

Relimiting the area to machine by means of limit elements

If you specify a point, it is projected onto the part profile.
A line through the projected point parallel to the radial axis delimits the area to machine.

If you specify a line, its intersection with the part profile is calculated (if necessary, the line is extrapolated).
A line through the intersection point parallel to the radial axis delimits the area to machine.

If you specify a curve, its intersection with the part profile is calculated (if necessary, the curve is extrapolated using the tangent at the curve extremity).
A line through the intersection point parallel to the radial axis delimits the area to machine.

Orientation and Location for Thread Turning

- **Orientation**: Internal / External
  This option allows you to specify the type of machining according to the location of the area to machine on the part.
- **Location**: Front / Back
  - Front, the profile is machined toward the head stock
  - Back, the profile is machined from the head stock.

Threads

- **Thread profile**: ISO / Trapezoidal / UNC / Gas / Other
  Other allows defining a specific thread profile.
- **Thread unit**: Pitch / Threads per Inch
  You must specify the thread type when the Thread profile is Other. Thread is automatically set to Pitch for the ISO and Trapezoidal types and set to Threads per Inch for UNC and Gas.
- **Nominal diameter**
  This value must be given when Thread type is internal and Thread profile is Other.
- **Thread length**
  This value must be given when the Start or End relimiting element is set to Profile End.
- **Thread pitch**
  This value must be given when the Thread type is set to Pitch or the Thread profile is ISO or

Trapezoidal.

- **Threads/inch**
  This value must be given when the Thread type is set to Thread per inch or when the Thread profile is UNC or Gas.
- **Thread depth**
  This value must be given when the Thread profile is Other.
- **Number of threads**
  When greater than 1, this value allows you to specify whether a multi-start thread is to be machined.

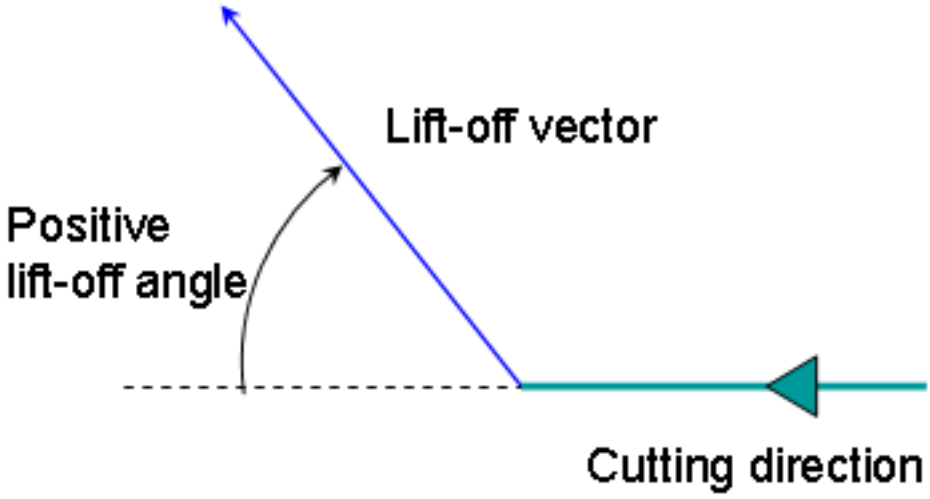
Machining Strategy Parameters for Thread Turning

Machining Options for Thread Turning

- **Threading type.** You must choose the desired threading type:
  - Constant depth of cut
  - Constant section of cut
- Maximum **Depth of cut** when Threading type is set to Constant depth of cut
- **Number of passes** when Threading type is set to Constant section of cut.
  When the number of passes is defined, the Section of cut value is automatically set.
- **Machining spindle speed** in revolutions per minute
- **Thread Penetration type**:
  - Straight
  - Oblique (penetration angle must be specified)
  - Alternate (penetration angle must be specified).

Path Computation options for Thread Turning

- **Clearance on crest diameter**
- **Lead-in Distance**
  Defined with respect to the cutting direction. The tool is in RAPID mode before this distance.
- **Lift-off Distance** and **Lift-off Angle**.
  These parameters define the lift-off vector at the end of each pass with respect to the cutting direction. The figure below shows the effect of a positive lift-off angle (external machining is assumed).



First and Last Passes options for Thread Turning

- Manage penetration on first passes by means of the **First passes** check box. This option is available when Threading type is set to Constant section of cut. When activated, you must specify values for:
  - **Number of first passes**
  - **First section rate**.

  When these two values are specified, the Section of cut for first passes value is automatically set.

- Manage penetration on the last passes by means of the **Last passes** check box. When activated, you must specify:
  - **Number of last passes**
  - **Depth of cut for last passes**.
- Manage the spring passes by means of the **Spring passes** check box. When activated, you must specify a **Number of spring passes**.

CYCLE or GOTO Output Syntaxes

If you want to generate CYCLE statements, you must select the **Output CYCLE syntax** checkbox in the Options tab and set the Syntax Used option to Yes in the NC Output generation dialog box. Otherwise, GOTO statements will be generated.

The parameters available for PP word syntaxes for this type of operation are described in the NC_LATHE_THREADING section of the Manufacturing Infrastructure User's Guide.

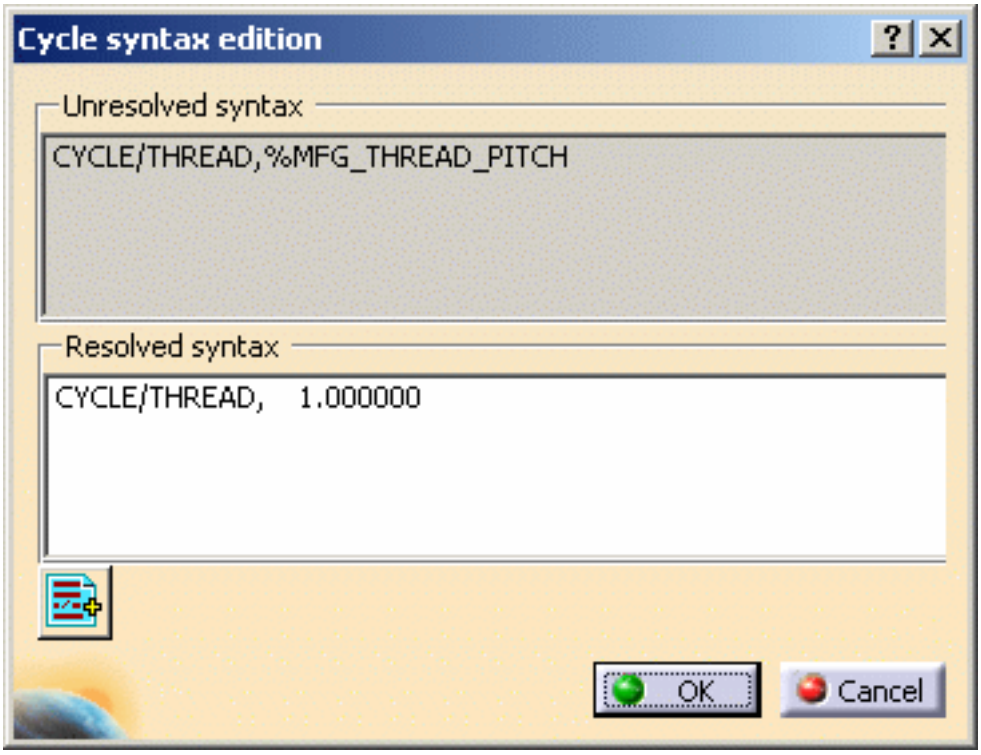
Editing CYCLE Syntaxes

The **Edit Cycle** command in the Thread Turning dialog box allows you to:

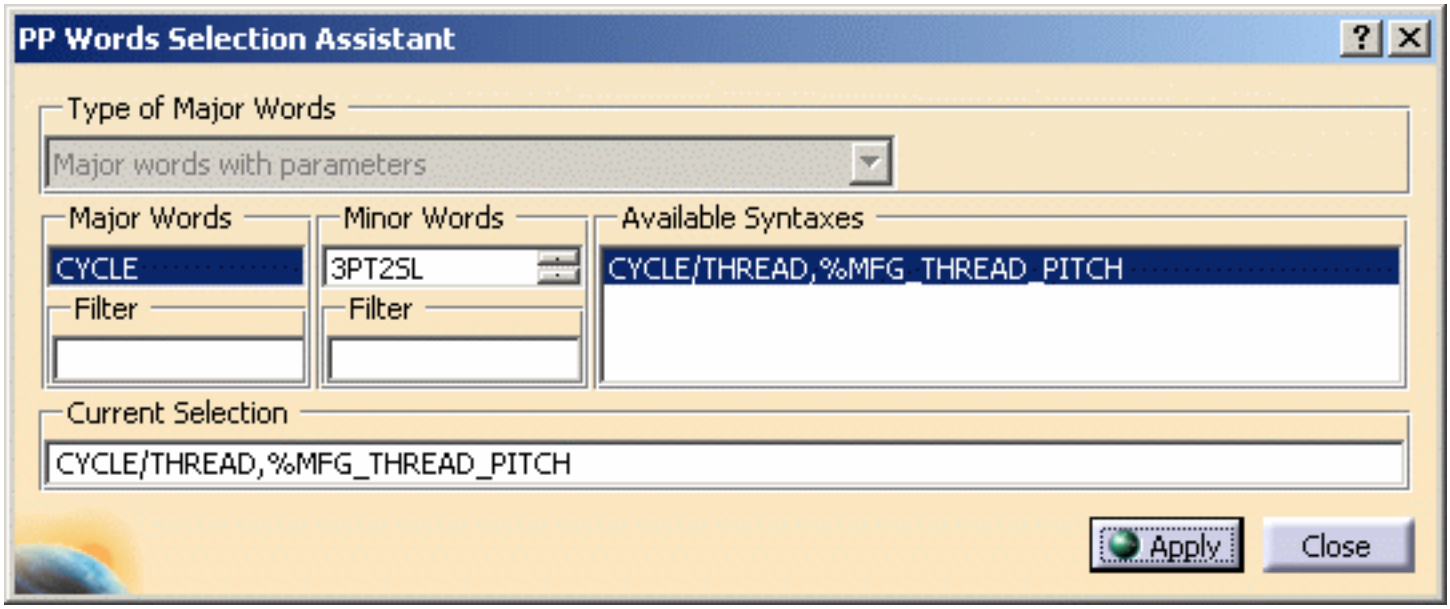
- display the unresolved syntax of the NC Instruction of the operation. This is the syntax as specified in the PP table referenced by the current Part Operation.

- display and, if needed, edit the syntax that is resolved either by geometric selection and user entries.

The Cycle Syntax Edition dialog box is displayed when you click the **Edit Cycle** command.



You can access all the CYCLE syntaxes contained in the current PP table by means of the PP instruction  icon. You can then select the desired syntax to be used by means of the procedure described in the Insert PP Instruction section.



Tool Compensation for Thread Turning

You can select a **tool compensation number** corresponding to the desired tool output point. Note that the usable compensation numbers are defined on the tool assembly linked to the machining operation. If you do not select a tool compensation number, the output point corresponding to type P9 will be used by default.

Note that the change of output point is managed automatically if you set the **Change Output Point** option for Trapezoidal or Other Thread profile.

Approach and Retract Macros for Thread Turning

The following Approach and Retract macros are proposed: Direct, Axial-radial, Radial-axial, and Build by user.
The selected macro type (Approach or Retract) defines the tool motion before or after machining. Various feedrates are available for the approach and retract motions (RAPID, air-cutting, and so on).

See Define Macros on a Turning Operation for more information.

Sequential Turning Operations

The information in this section will help you create and edit Sequential Turning operations in your manufacturing program.

You can create a sequence of basic Go, Go InDirv, Go Delta, and Follow tool motions in a single Sequential Turning operation. PP words can be inserted in the sequence of tool motions.

When you select the Tool motion management icon a panel is displayed listing any previously created tool motions and PP words. You can create or edit tool motions and PP words.

If there are no previously created tool motions listed in the Sequential Turning dialog box, the first one must be a Go Standard motion to a point. It specifies either the absolute start position from where the following position can be computed or the end point of the approach macro (if the operation has one).

When defining a tool motion, the corresponding new tool position is displayed according to all the options that were selected for creating this motion.

The following topics are dealt with in the paragraphs below:

- Tooling
- Geometry
- Go Motion
- Go InDirv Motion
- Go Delta Motion
- Follow Motion
- Feeds and Speeds.

Tooling for Sequential Turning

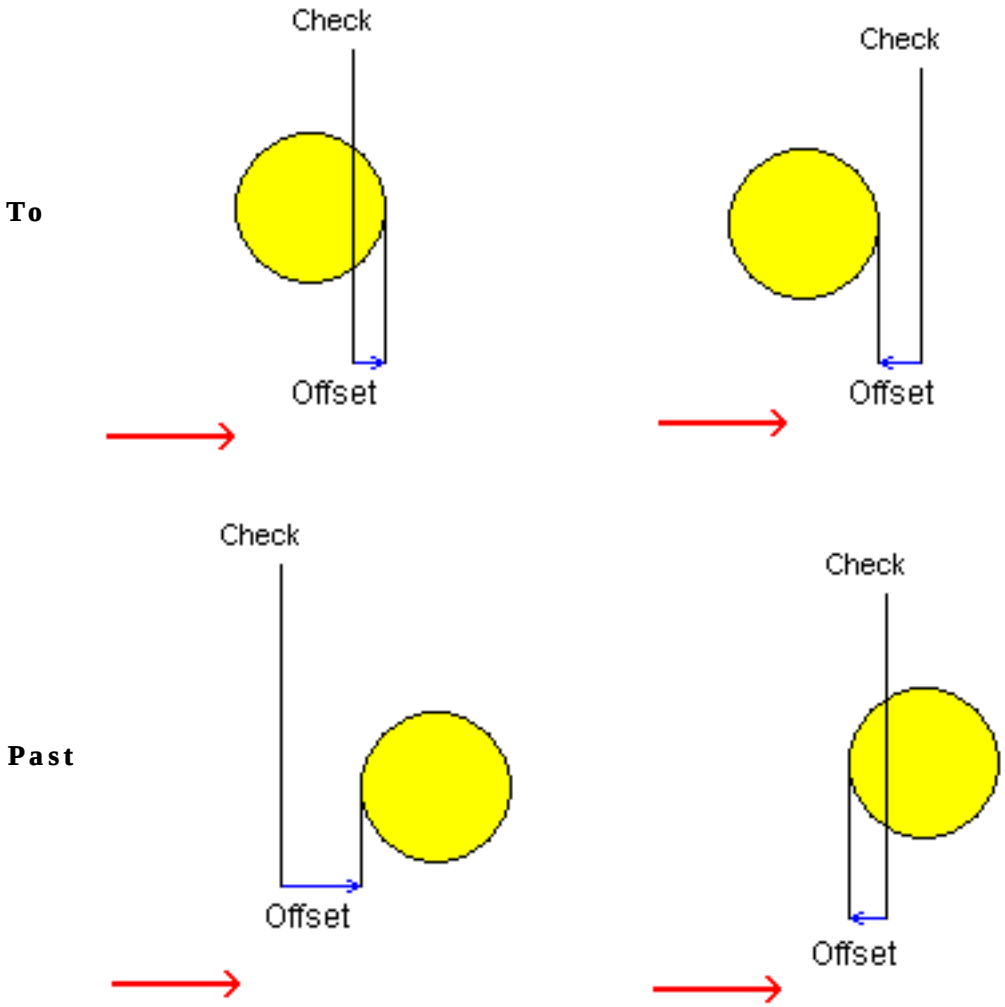
All tool types with compatible inserts are authorized.

Geometry Selection for Sequential Turning

You can select lines, curves and edge elements as check and drive elements. Points can also be selected as checks. Selected elements must be located in the plane defined by the machine spindle axis and radial axis.

The following table summarizes how the tool is positioned with respect to a check element depending on the TO/PAST modes and negative/positive offset values.

	Positive offset	**Negative offset**



Go Motion

In Go motion, the program positions the tool with respect to one or two check elements.

When only one check element is selected, the tool is projected onto the check element.

When two check elements are selected, the tool is positioned with respect to the two check elements.

The tool is positioned with respect to each check element depending on the TO-ON-PAST mode.

TO: the tool nose is positioned tangent to the near side of the selected element with a possible offset.
ON: the tool nose is positioned on the selected element, with a possible axial or radial offset (normal offset, collision avoidance and safety angles are not applicable).
PAST: the tool nose is positioned tangent to the far side of the selected element with a possible offset.

The first check element can be point, line, curve, or edge type elements.
If it is a point, no second check element can be selected. Otherwise, the second check element can be line, curve or edge type elements.

The following options are available when defining a Go motion:

- Control over geometric extrapolation of each check curve.
- Axial, radial and normal offsets with respect to each check element.

- Feedrate and spindle speed.
- Collision avoidance for each check element when the tool is positioned in TO or PAST mode.
- Left and right safety angles when collision avoidance is used.
- Machining tolerance
- Tool compensation
- Guiding Point, if a square or grooving insert is used.

Note that the positioning mode (TO, ON, PAST) and offsets are not applicable when the tool is positioned on a point.

Go InDirv Motion

In Go InDirv (Go in Direction Vector) motion, the tool moves in a given direction up to the selected check curve.
The tool is positioned with respect to the selected element depending on the TO-ON-PAST mode.

TO: the tool nose is positioned tangent to the near side of the selected element with a possible offset.
ON: the tool nose is positioned on the selected element, with a possible axial or radial offset (normal offset, collision avoidance and safety angles are not applicable).
PAST: the tool nose is positioned tangent to the far side of the selected element with a possible offset.

The selected check curve can be edge, line and curve type elements.

The following options are available when defining a Go InDirv motion:

- Drive element type: line or angle.
- Angle of drive, if drive element type is angle.
- Drive direction: Same or Inverted.
  This is useful in the particular case when two positions are reachable.
- Control over geometric extrapolation of check curve.
- Axial, radial and normal offsets with respect to the check element.
- Feedrate and spindle speed.
- Collision avoidance when the tool is positioned in TO or PAST mode.
- Left and right safety angles when collision avoidance is used.
- Machining tolerance
- Tool compensation
- Guiding Point, if a square or grooving insert is used.

Go Delta Motion

In Go Delta motion, the tool move is based on the current position of the tool. The TO-ON-PAST mode is not proposed.

The following options are available when defining a Go Delta motion:

- Delta check mode:

- Distance between two points (two points are to be selected)
- Line and distance (line and distance are to be selected)
- Angle and distance (angle and distance are to be given)
- Axial and radial (axial and radial offsets are to be given)

- Feedrate and spindle speed.
- Machining tolerance
- Tool compensation.

Follow Motion

In Follow motion, the tool follows a drive element up to a check element. The check element is to be selected in the Follow motion and can be an edge, line, a curve or a point.

The drive element is a curve on which the tool is positioned by a preceding motion. Therefore, the preceding motion must be a Go, Go InDirv or Follow motion. A Go to a point or a Go Delta cannot precede a Follow motion.

If the preceding motion is a Go motion, the first selected check curve is used as drive curve for the Follow motion.

If the preceding motion is a Go InDirv motion, the selected check curve is used as drive curve for the Follow motion.

If the preceding motion is a Follow motion, the check element of the preceding Follow motion is used as drive curve when this check element is not a point and the Change Drive option of the preceding Follow motion is set. Otherwise, both consecutive Follow motions share the same drive curve.

Note that:

- the offsets applied to follow the drive element are also defined on the same preceding motion. When the drive and check curves are tangent, any offset values on the check curve should be the same on the tangent drive curve.
- the collision avoidance and the control over geometry extrapolation on drive curve are also defined on the same preceding motion.

The tool will follow the drive element up to the check element. The final tool position will be determined by the TO-ON-PAST mode:
TO: tool nose tangent to the drive element and before the check element with a possible offset.
ON: tool nose tangent to the drive element and on the check element, with a possible axial or radial offset (normal offset is not applicable).
PAST: tool nose tangent to the drive element and after the check element with a possible offset.

The following options are available when defining a Follow motion:

- Drive direction: Same or Inverted.
  This is useful in the particular case when two positions are reachable.
- Control over geometric extrapolation of check curve.
- Axial, radial and normal offsets with respect to the check element.

- Feedrate and spindle speed.
- Collision avoidance when the tool is positioned in TO or PAST mode.
- Left and right safety angles when collision avoidance is used.
- Machining tolerance
- Tool compensation
- Guiding Point, if a square or grooving insert is used.

Control over Geometric Extrapolation of Check Curve

When check limit mode is set to Extended, a virtual tangent element is added to each extremity of the check element before it is actually taken into account as a check element.
Therefore, if there is no intersection between the drive and check elements, the final tool position may be located at the intersection of the drive curve and a tangent of the check element.

When check limit mode is set to Actual, no virtual tangent element is added.
Therefore, the final tool position is located on the check element.

Furthermore, please note that control over geometric extrapolation is not controlled in the Follow motion but on the preceding motion that positions the tool on the drive element.

Note that collision avoidance is taken into account for positioning in ON mode, and that left and right safety angles are only considered in collision avoidance mode.

Feeds and Speeds for Sequential Turning

Speed unit can be set to:

- Angular: spindle speed in revolutions per minute
- Linear: constant cutting speed in units per minute

then you can give a Machining Speed value.

Available feedrates in units per revolution are as follows:

- **Machining Feedrate**
- **Lift-off Feedrate**
- **Lead-in Feedrate**.

In addition to these global feedrates, local feedrates can be applied to each tool motion.

Tool Assembly Conventions for Turning

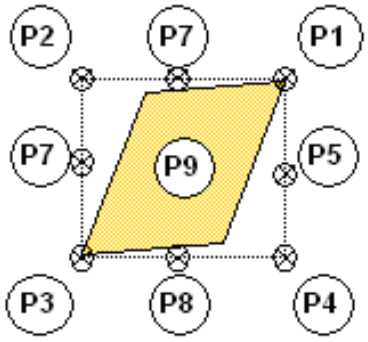
General Comments

The Machining Axis is always a direct axis system. For example, ZX defines a direct axis system with Z as the spindle axis and X as the radial axis.

Selected geometry (for example, Finish Profile Turning) can be selected either in X+ or X- :

- the system will figure out whether machining is in X+ or X- based on Insert-Holder orientation (as defined using Set-Up Angle)
- depending on the options of a Machining Operation, several possibilities might exist,
(for example, Frontal machining). User option is then available (X+/X-) to specify what the system should do.

The Output Point definition is considered according to the X+/Z+ quadrant.

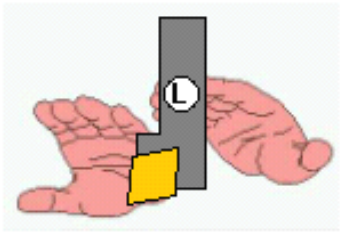


Examples below deal with Horizontal Lathe Machine Tool.

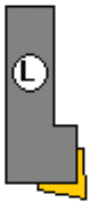
Insert Holder

Different settings of the Setup Angle, Invert Tool and Hand Style parameters result in different configurations for positioning the tool .

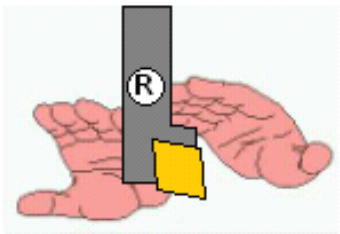
Left Tool: Insert is visible and points to left:



Inverted Left Tool :



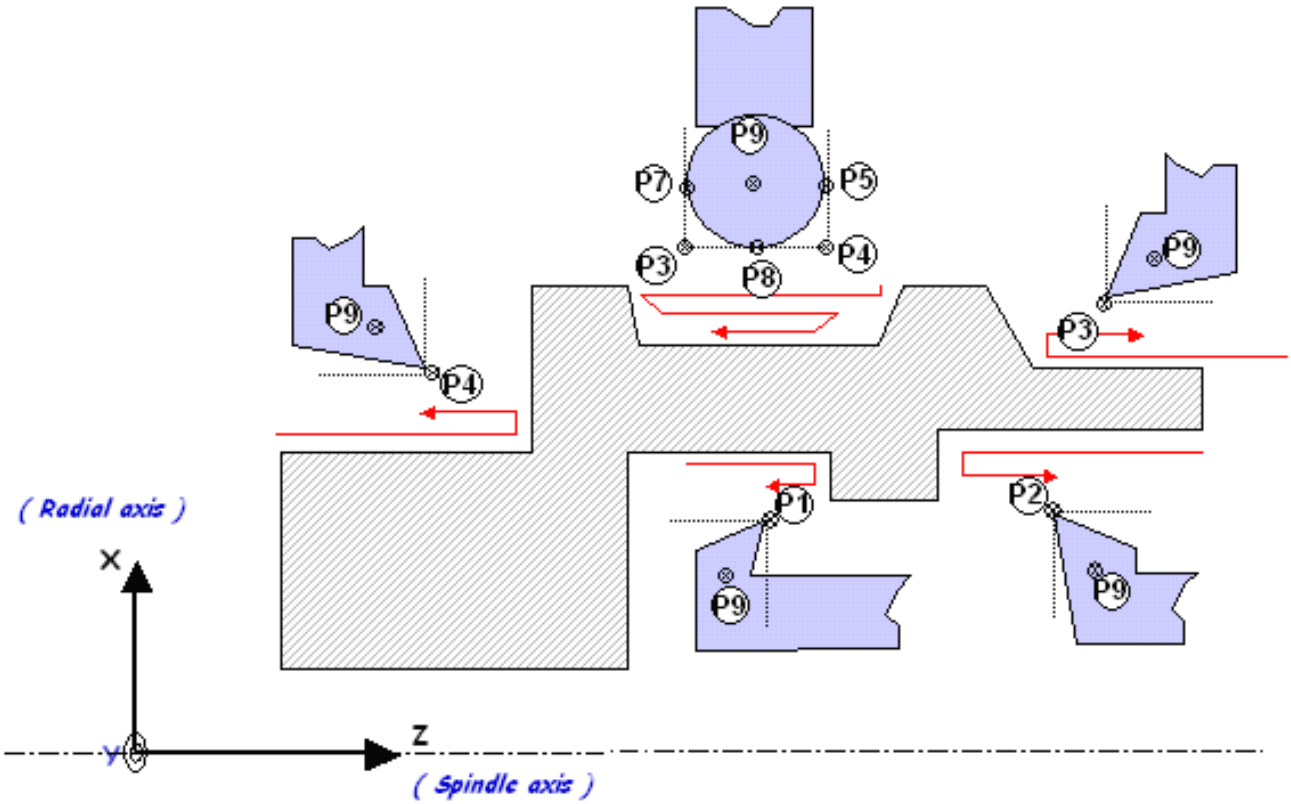
Right Tool: Insert is visible and points to right:



Inverted Right Tool:

Output Point

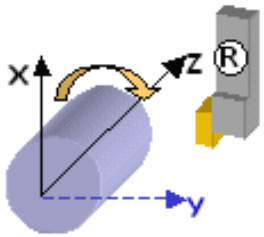
In the following figure reference is made according to the Spindle/Radial Axis (Z/X)



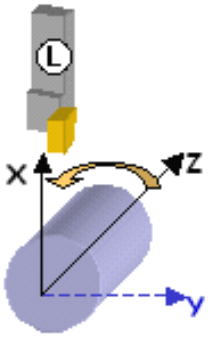
Spindle Rotary Direction

With a Horizontal Lathe Machine Tool the convention for spindle rotary direction defined in the Machine Editor dialog box is as follows:
Looking into Z direction, Chuck & Jaws at your back.

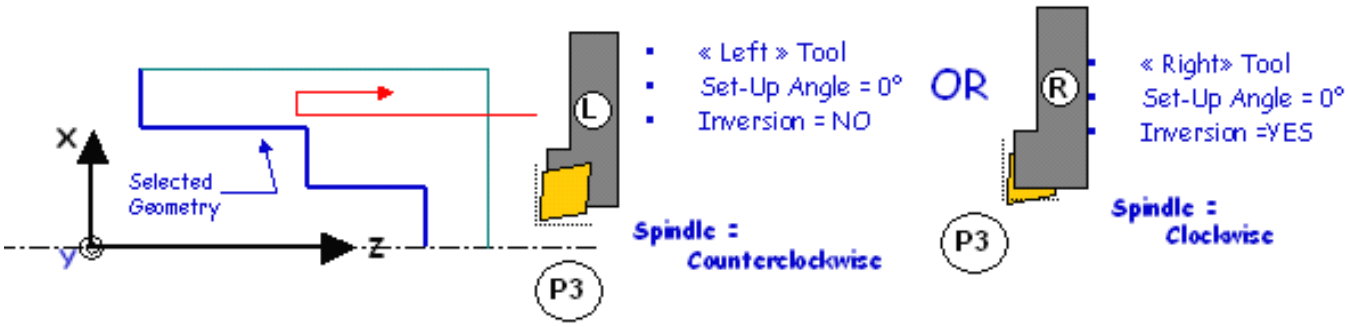
Clockwise



Counterclockwise

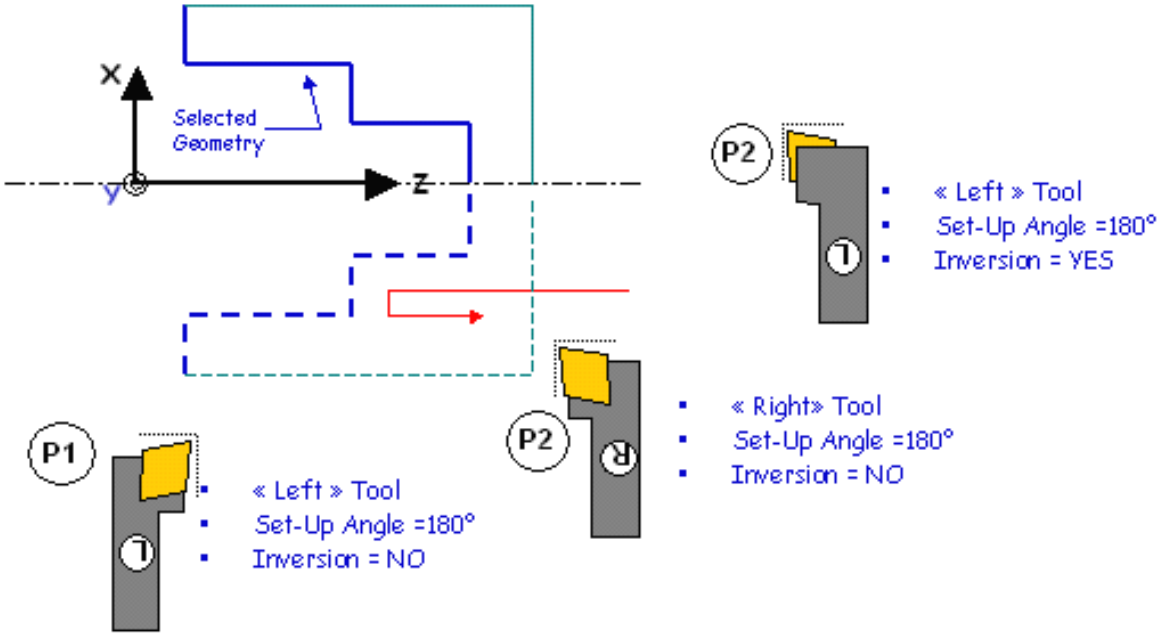


The following represent common settings:



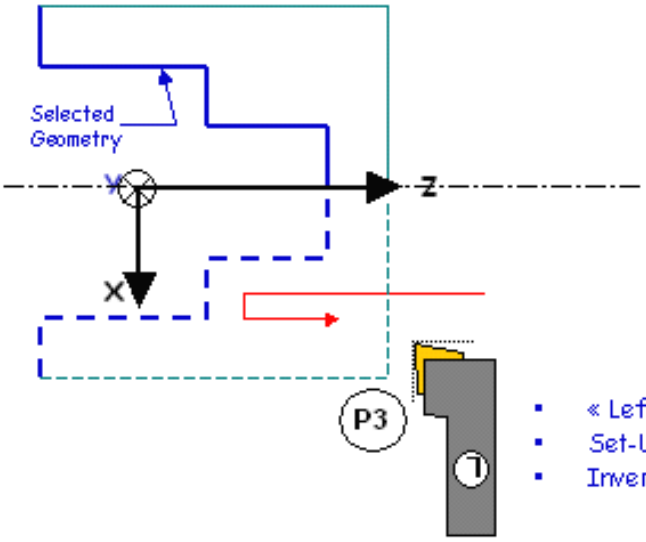
180deg Setup Angle

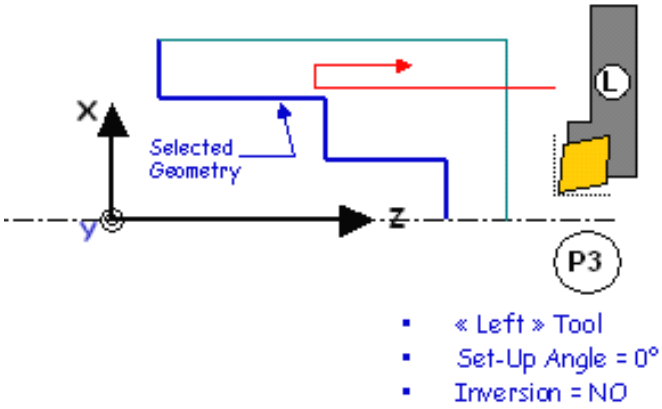
When Setup angle is set to 180deg, machining takes places in X- as system detects that this is what the tool can machine (geometry can be selected in X+ or X-).



Direct Machining Axis

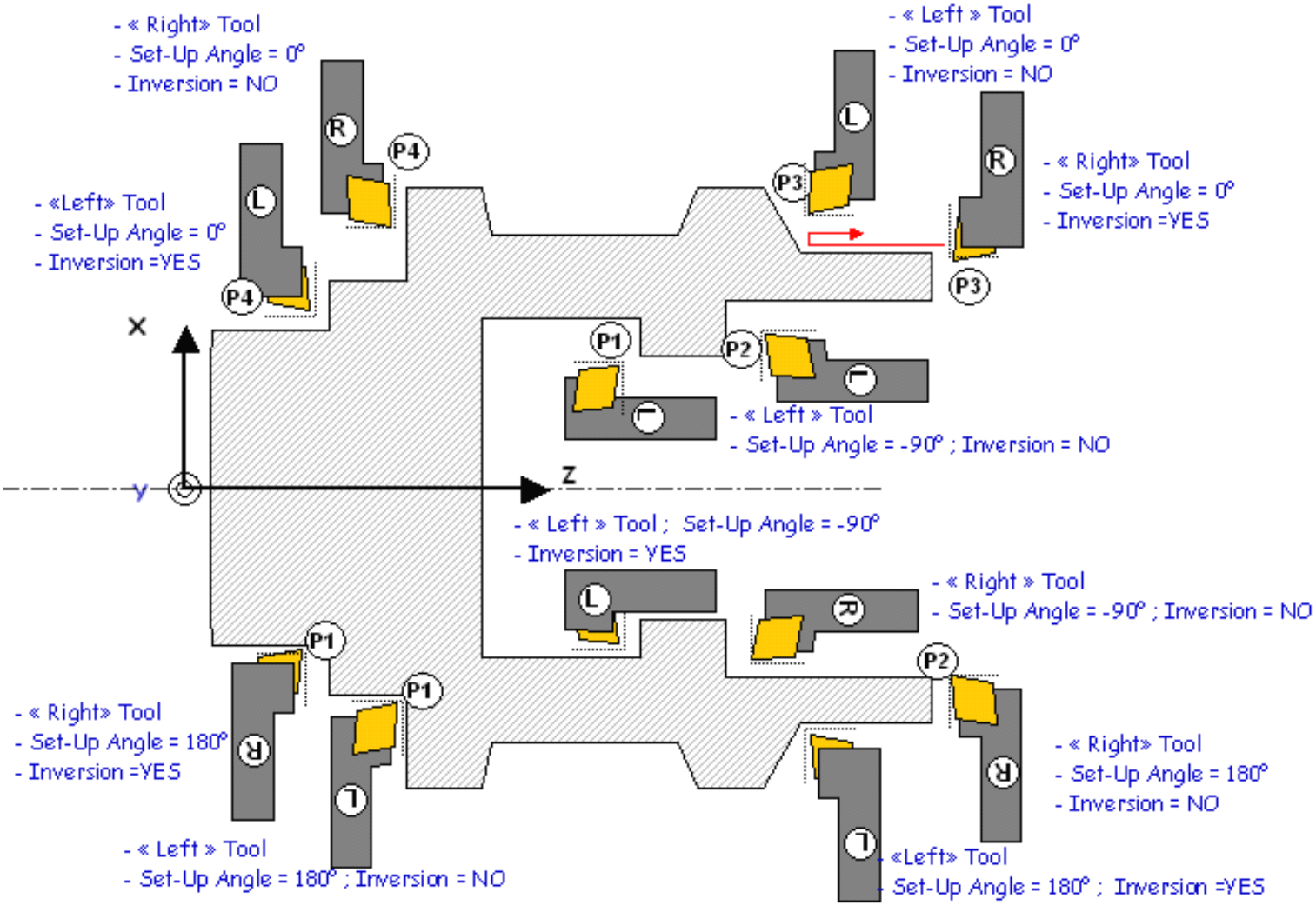
As the Machining Axis is always considered as being direct, the two situations below are the same (this corresponds to a rotation around the spindle axis).



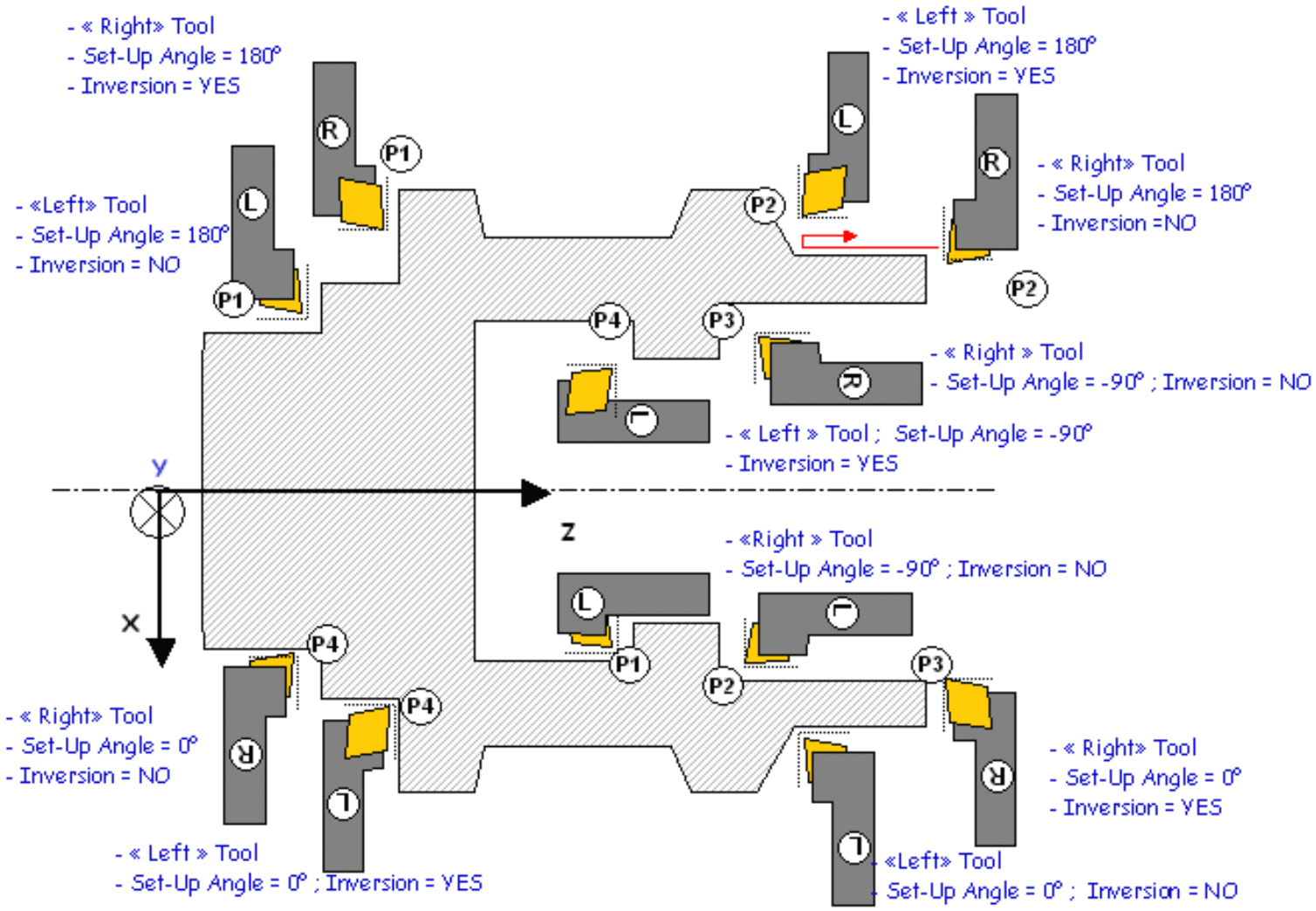


Summary

The following figures summarize conditions for lathe tool assemblies.



The figure below shows other settings for different output points.


- « Right» Tool
- Set-Up Angle = 180°
- Inversion = YES
- « Left » Tool
- Set-Up Angle = 180°
- Inversion = YES
- «Left» Tool
- Set-Up Angle = 180°
- Inversion = NO
- « Right» Tool
- Set-Up Angle = 180°
- Inversion =NO
- « Right » Tool
- Set-Up Angle = -90° ; Inversion = NO
- « Left » Tool ; Set-Up Angle = -90°
- Inversion = YES
- «Right » Tool
- Set-Up Angle = -90° ; Inversion = NO
- « Right» Tool
- Set-Up Angle = 0°
- Inversion = NO
- « Left » Tool
- Set-Up Angle = 0° ; Inversion = YES
- « Right» Tool
- Set-Up Angle = 0°
- Inversion = YES
- «Left» Tool
- Set-Up Angle = 0° ; Inversion = NO
Y
Z
X
P1
P2
P3
P4

Methodology

This section provides methodology and conceptual information on the following Lathe Machining topics.

Cutter Compensation and Finish Operations
How to Change the Output Point
How to Update Input Stock
Turning on a Milling Center with Facing Head

Methodology and conceptual information on the following topics is provided in the *NC Manufacturing Infrastructure User's Guide*.

Machining Processes
Knowledgeware in Machining Processes
CATProduct and CATProcess Document Management
Design Changes and Associativity Mechanisms
Part Operation and Set Up Documents
User Features for NC Manufacturing

Cutter Compensation with Finish Operations

The **Cutter Compensation: None / On / Reverse** option is proposed for finish operations. If this option is set to On or Reverse, the NC output will include CUTCOM instructions in the APT or clfile output for cutter compensation (CUTCOM/RIGHT, CUTCOM/LEFT, CUTCOM/OFF).

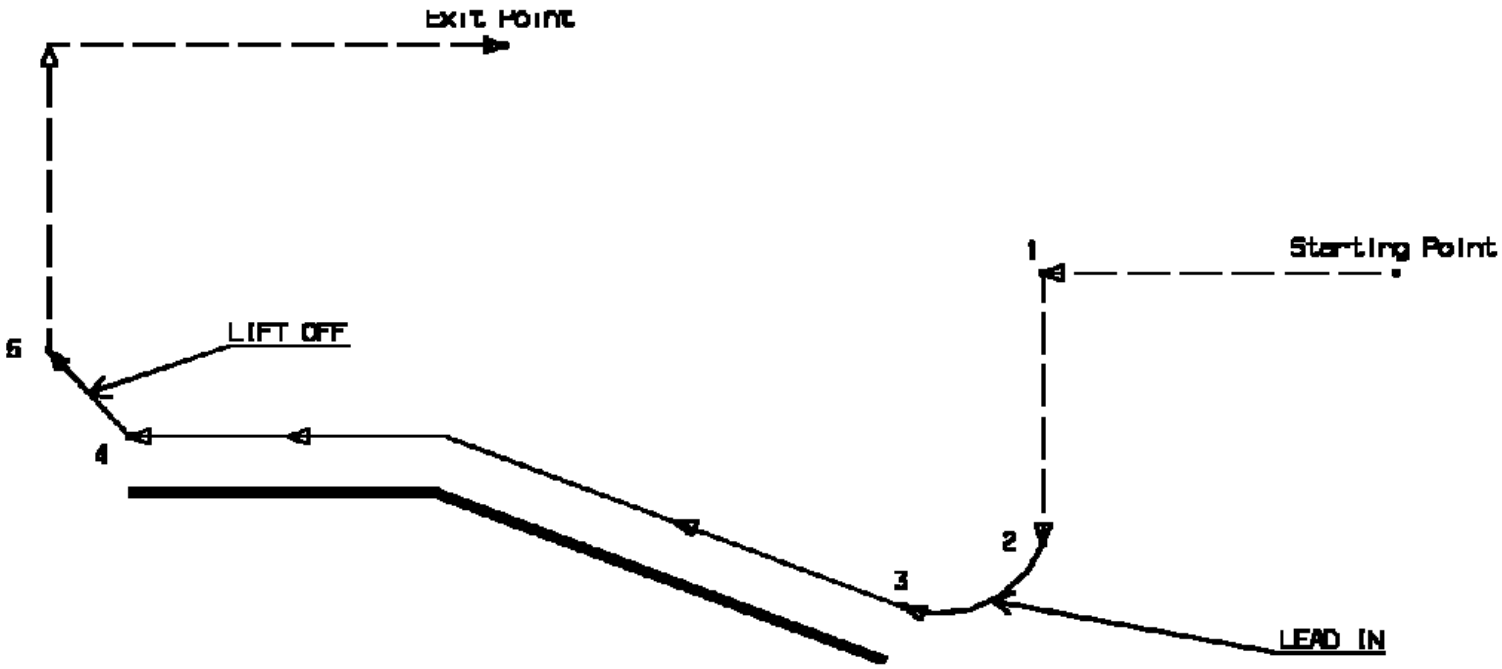
- **On**: CUTCOM/RIGHT instruction generated if tool is to the right of the toolpath and CUTCOM/LEFT if tool is to the left of the toolpath
- **Reverse**: CUTCOM/RIGHT instruction generated if tool is to the left of the toolpath and CUTCOM/LEFT if tool is to the right of the toolpath.

Otherwise, if the option is set to **None**, no CUTCOM instruction will be included in the NC data output.

A CUTCOM instruction is always generated before a linear trajectory in order to be active on that displacement:

- for a tool approach, the instruction CUTCOM/RIGHT or CUTCOM/LEFT is generated at latest in the approach phase of the trajectory, before the lead-in and on a linear trajectory
- for a tool retract, the instruction CUTCOM/OFF is generated at earliest in the retract phase of the trajectory, before a linear trajectory or the last point of the operation.

The figure below illustrates a Profile Finishing operation that has circular lead-in and linear lift-off.



If the cutter compensation is set to ON, the CUTCOM instructions are generated as follows:

- CUTCOM/RIGHT is generated at point 1, before the tool motion to point 2.
  Note that if lead-in was linear, CUTCOM/RIGHT would be generated at point 2, before the tool motion to point 3.
- CUTCOM/OFF is generated at point 4, before the tool motion to point 5.
  Note that if lift-off was circular CUTCOM/OFF would be generated at point 5, before the linear retract motion.

How to Use Cutter Compensation

The computed toolpath corresponds to the trajectory followed by the output point of the tool used in the Part Operation.

You should set Cutter Compensation to On in the following cases:

- the cutter radius of the actual tool used for machining is greater than the radius of the programmed tool and a positive compensation value is entered at the NC machine
- the cutter radius of the actual tool used for machining is less than the radius of the programmed tool and a negative compensation value is entered at the NC machine.

You should set Cutter Compensation to Reverse in the following cases:

- the cutter radius of the actual tool used for machining is less than the radius of the programmed tool and a positive compensation value is entered at the NC machine
- the cutter radius of the actual tool used for machining is greater than the radius of the programmed tool and a negative compensation value is entered at the NC machine.

Some Recommendations

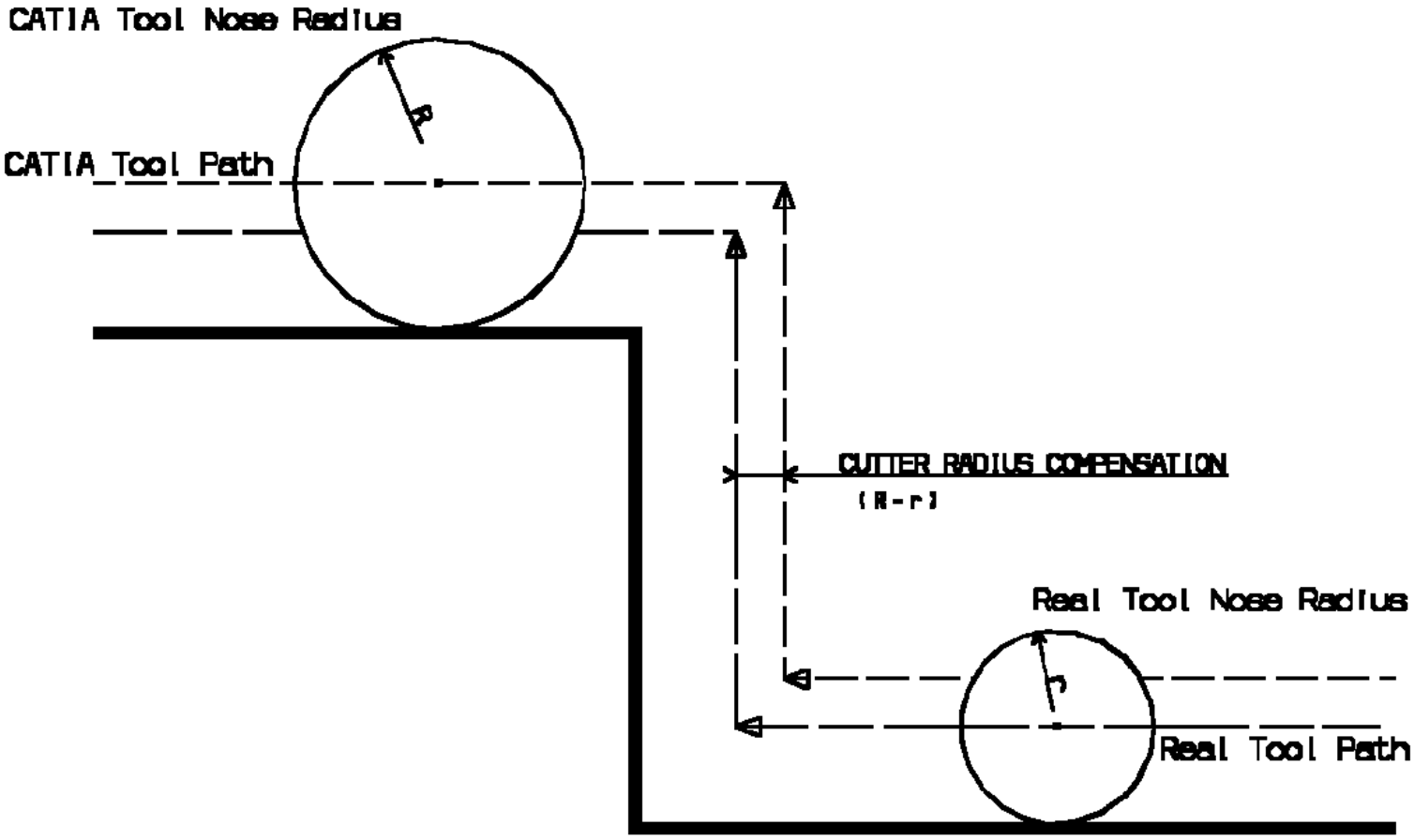
In general you should program with tools whose cutter radius is greater than those that will actually be used on the machine. This will help you anticipate tool/part collisions that may arise when cutter compensation is used.

If negative compensation values are allowed on the machine, set Cutter Compensation to On.

If negative compensation values are not allowed on the machine set Cutter Compensation to:

- On, if the tool actually used has a greater cutter radius than the programmed tool
- Reverse, if the tool actually used has a smaller cutter radius than the programmed tool.

The figure below illustrates cutter compensation for profile finishing.


CATIA Tool Nose Radius
R
CATIA Tool Path
CUTTER RADIUS COMPENSATION
(R-r)
Real Tool Nose Radius
r
Real Tool Path

Changing the Output Point

An option for changing the tool output point is available for:

- Recessing, Ramping Recessing, Grooving, and Groove Finishing operations using grooving tools or inserts
- certain Threading operations using threading tools or inserts.

When **Change Output Point** is set the tool output point is changed automatically during the operation according to the profile geometry to be machined.

For Grooving and Groove Finishing operations, tool output point changes are made out of the profile.

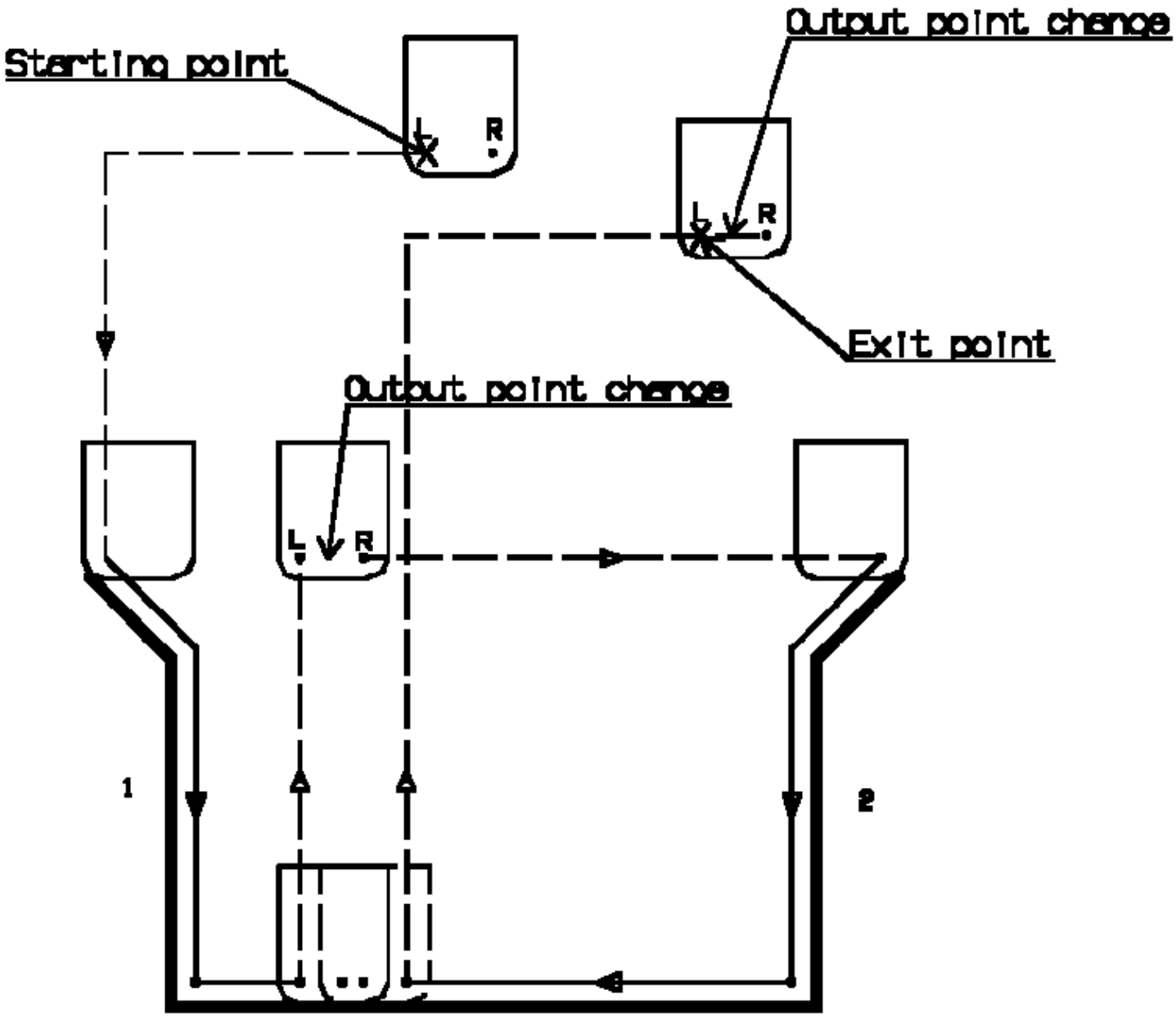
For Recessing operations, tool output point changes are made before each tool motion involving machining (that is, after each plunge). However, changes are only done:

- if machining is consistent with the selected tool output point
- if another output point is defined on the tool so that the tool output point change can be made.

Otherwise, the tool output point will not be changed.

Example of Groove Finishing

The following figure illustrates tool output point changes in a Groove Finishing operation that uses a grooving tool. In this example, at the start of operation the tool output point is P9.



If the first flank to machine is flank 1, the tool motion is as follows:

- approach and lead-in motion to flank 1
- machine down flank 1
- lift-off from part profile
- tool output point change: tool output point is P9R
- approach and lead-in motion to flank 2
- machine down flank 2
- lift-off to Exit Point
- tool output point change: tool output point is P9 (as at start of operation).

If the first flank to machine was flank 2, the tool motion would be as follows:

- tool output point change: tool output point is P9R

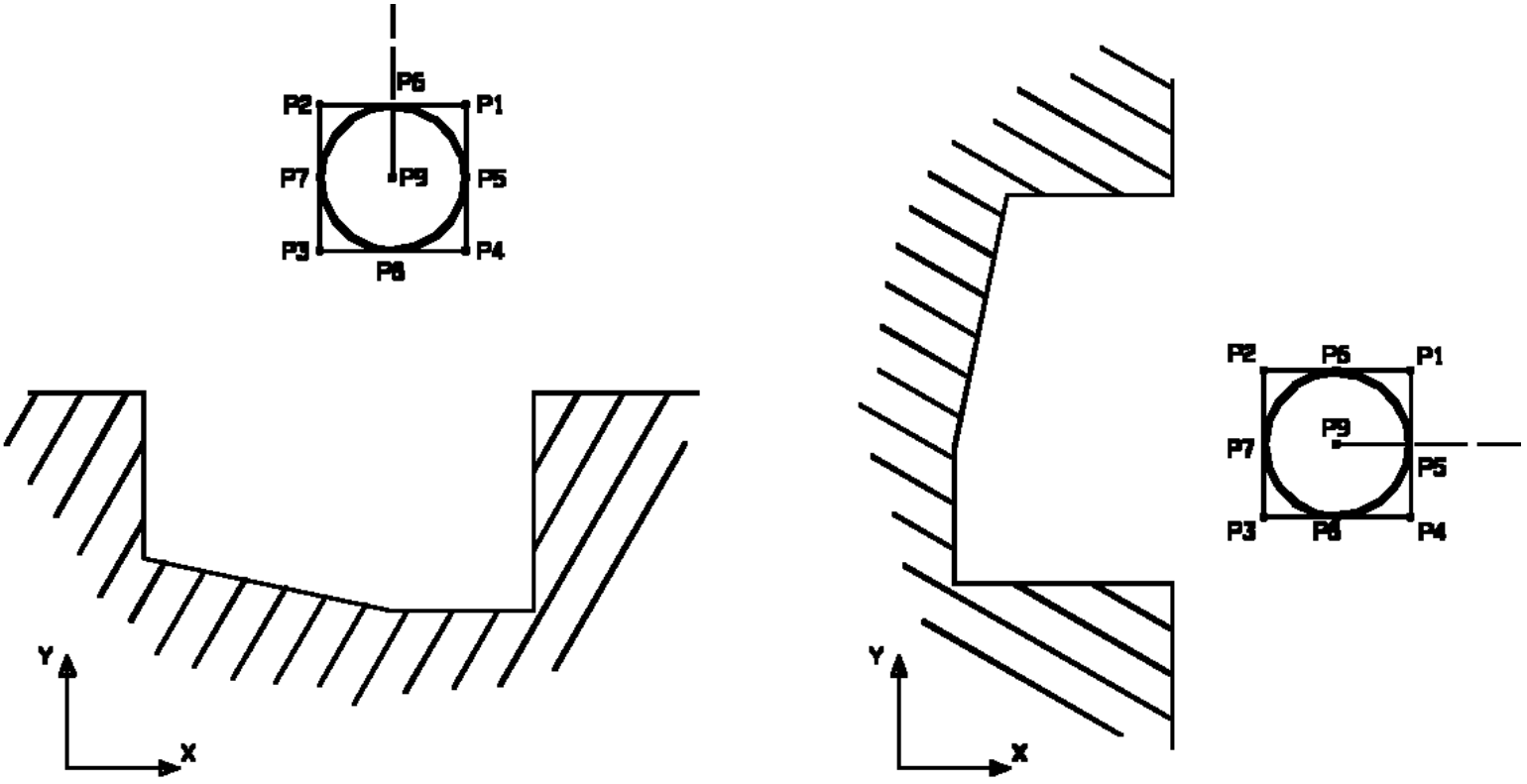
- approach and lead-in motion to flank 2
- machine down flank 2
- lift-off from part profile
- tool output point change: tool output point is P9 and the guiding point is LEFT
- approach and lead-in motion to flank 1
- machine down flank 1
- lift-off to Exit Point.

If P9 is the tool output point and if the output point P9R is defined on the tool, the output point change is only done for grooving tools.

The tool output point at the end of operation is the same as at the start of operation.

Example of Recessing

The figure below illustrates a Recessing operation when a round insert is used. The tool output point changes during an operation only if the output point at the start of operation is P2, P3 or P9 for a frontal recess or P3, P4 or P9 for an external recess.



The tool output point is dependent on the machine axis system.

How to Update Input Stock

When you create a new turning operation, you can generate geometry from the previous operations in your program then use that geometry as stock for the new operation.

This is done using the **Update Input Stock** command in the machining operation editor. The Stock computation is done before you select geometry (stock or part) in your new operation. It creates a sketch that highlights the remaining material on the part. The computation takes into account all complete operations in the program before the one being edited.

Stock computation is based on the stock solid specified in the Part Operation dialog box. The primitives of the material removed by each previous operation are subtracted from this stock. The removed area computation takes the parameters of the machining operations into account. Stored or locked tool paths are not taken into account.

The result of the computation is a sketch which contains the view of the remaining material in the working plane.

The operations taken into account for Stock computation are:

- Rough Turning
- Groove Turning
- Recess Turning
- Profile Finish Turning
- Groove Finish Turning
- Sequential Turning
- Ramp Rough Turning and Ramp Recess Turning
- All Drilling operations along the spindle axis.

Thread Turning operations are not taken into account.

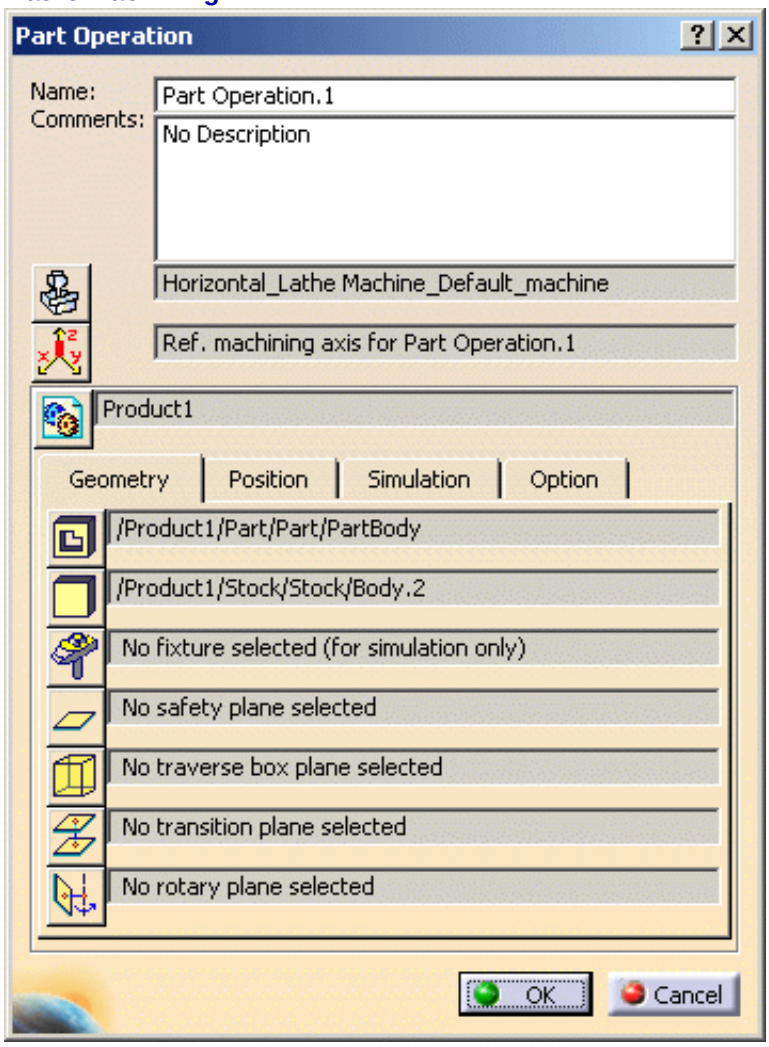
Defining the Machine and Stock

Firstly, you need to define a stock and a lathe machine on the Part Operation.

Double click the Part Operation in the tree to display the Part Operation dialog box.

Click the Machine icon in the dialog box the select a horizontal lathe machine. You must set up the spindle and radial axes correctly (that is, spindle to Z and radial axis to X in this example).
Click the Stock icon then select a solid representing the stock. Double click on the background to leave the geometry selection mode.

The dialog box is updated with the name of the selected machine and stock.



Note:

The generation of the stock is based on In Process Models (IPM).

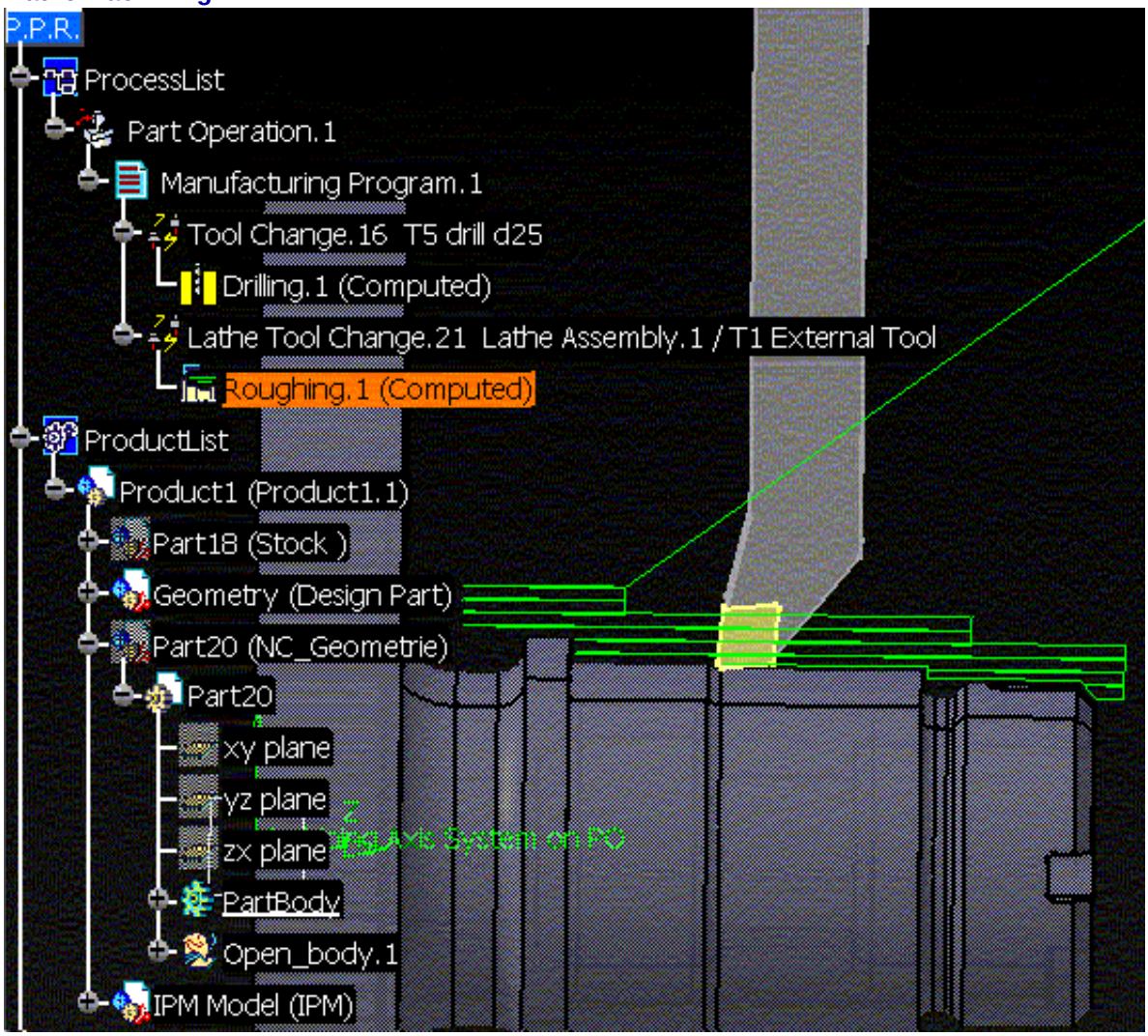
For each stock part specified in the Part Operation dialog box, an IPM will be generated. The name of the IPM part will be the same as the reference stock part name with the prefix "IPM_". The IPM part will be created in the same location as the corresponding design part (the design part is considered instead of the stock part because often the stock part is just a body inside the design part).

For file system users, this means that the CATProduct that contains the design part will be modified with the addition of the IPM CATPart (under the root node of the CATProduct).
The CATProcess must be linked to a CATProduct that include the design part. The input stock calculation cannot be done if the CATProcess is not linked to the CATProduct.
The IPM CATPart is dedicated to the input stock calculation. No other geometry must be created in this CATPart.

For PPR hub users, the IPM CATPart is inserted under the "Subassembly" node of the corresponding design part.

Computing Stock

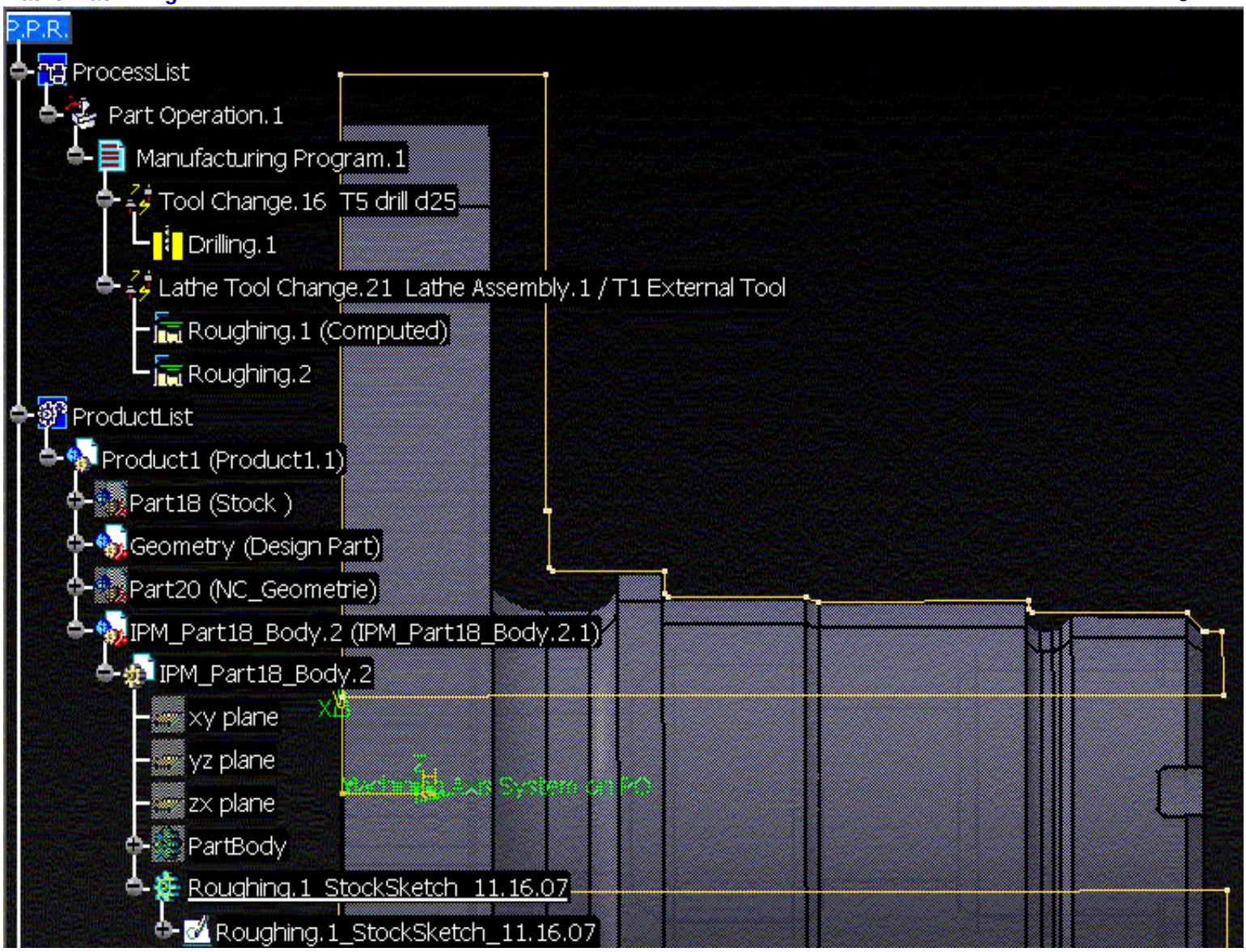
The following figure illustrates Stock computation after a Drilling (along the spindle axis) and a Roughing operation. Here is the tool path of the Roughing operation:



When creating a new Roughing operation after Roughing.1, click the **Update Input Stock** icon in the dialog box of the new Roughing operation.

The following figure shows the result of the computation. The Stock has been created in orange. It is located in a sketch whose identifier is the name of the last operation taken into account for the computation (Roughing.1) plus the item StockSketch and the time of the computation.

The Stock sketch has been created in a body with the same name, in a part named IPM_xxx (where xxx is the name of the selected stock solid). The part is located in a product with the same name.



You can now select elements of the Stock sketch as new Part or Stock of the Roughing.2 operation.

Stock Sketches, IPM Bodies and Primitives

The Stock computation automatically creates an IPM body (CATPart).

In order to create the Stock, some primitives are created in the IPM body. Once the stock sketch is created, the sketch is no longer linked to them.

The sketch of the input stock is linked to the operation. When editing the operation the sketch is in Show mode. When the operation editor is closed, the sketch is in Hide mode.

By right-clicking the **Input stock status** field in the Geometry tab page of the operation editor, it is possible to remove the stock sketch. In this case the sketch is deleted.

A primitive is created for each Turning operation. Each primitive takes the machining paths of the tool trajectory into account. Note that macro, lead-in and lift-off motions are not taken into account in the computation of the operation's primitive as these are not machining paths.

If you use the Stock computation again, the existing primitives will speed up the computation. They will be used if the operations to which they are linked have not been modified since the Stock computation. Otherwise they will be modified.

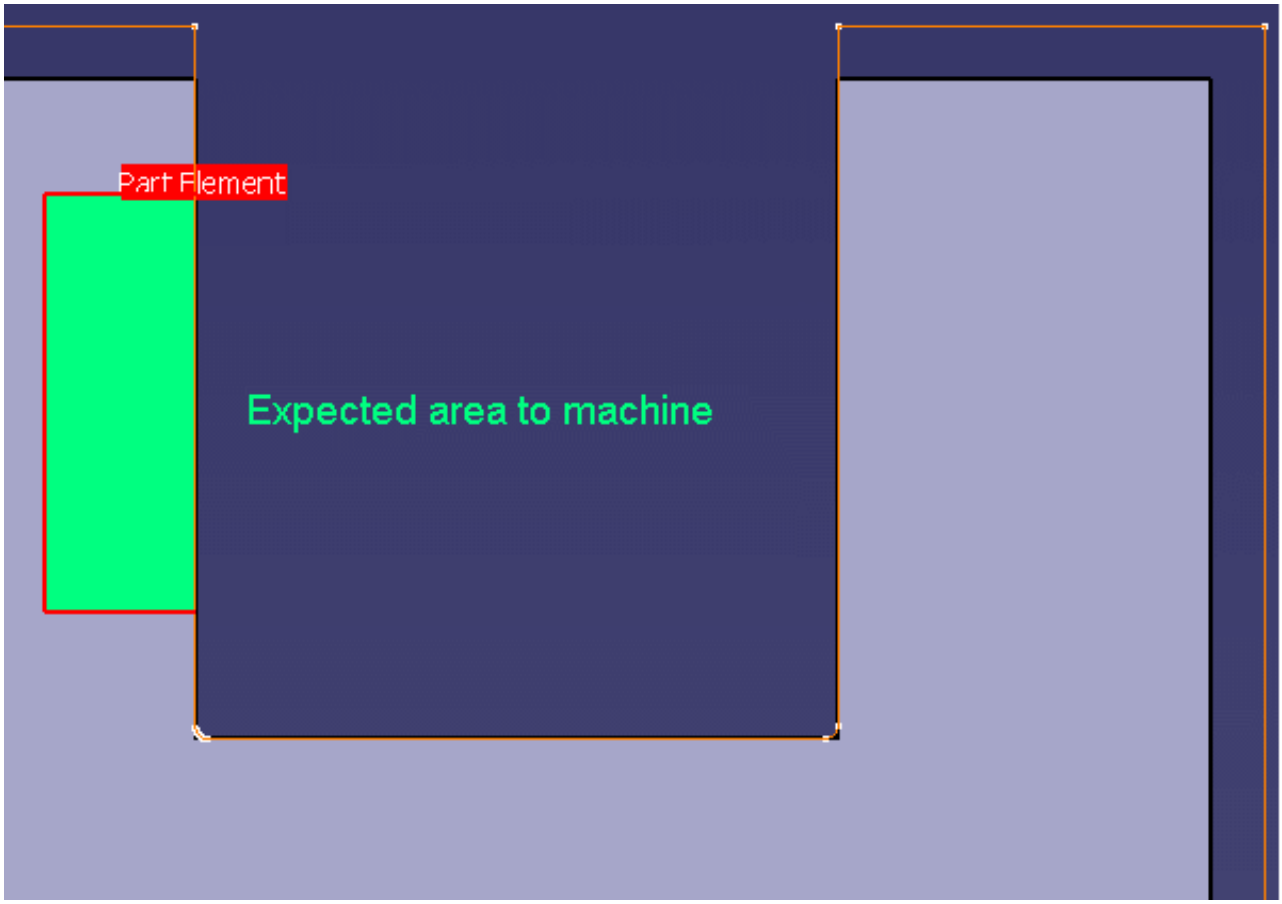
If you have finished your Part Operation, you can remove these primitives by using the **Clean Computed Stock Data** contextual command on the Part Operation. This command removes each primitive of the IPM body linked to the operations in the Part Operation. It does not remove the Stock sketches or the IPM body.

For a CATProcess created in a previous version, the previous IPM body will be migrated automatically. An IPM CATPart will be created for each solid selected as stock.

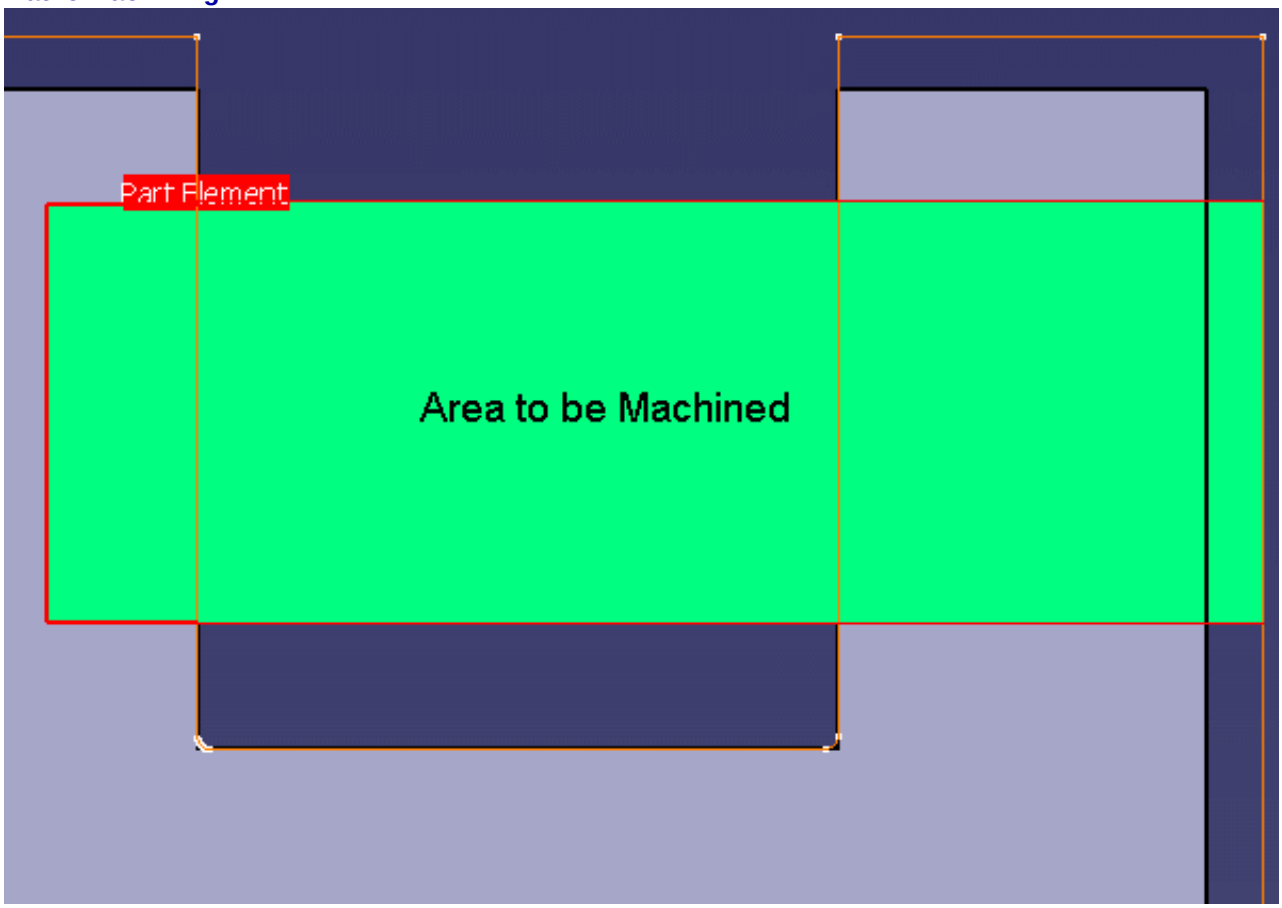
Considerations for Machining Local Recesses

The **Update Input Stock** functionality is best suited to operations in which the stock is to be updated globally.

For a local recesses or grooves like the one shown below, you may be advised to to update the input stock manually.



For example, the *expected area to machine* (in green) above may be interpreted by the program as the *area to be machined* (in green) below.



This is due to the ambiguity that arises due to the shape the selected stock profile.

Turning on a Milling Center with Facing Head

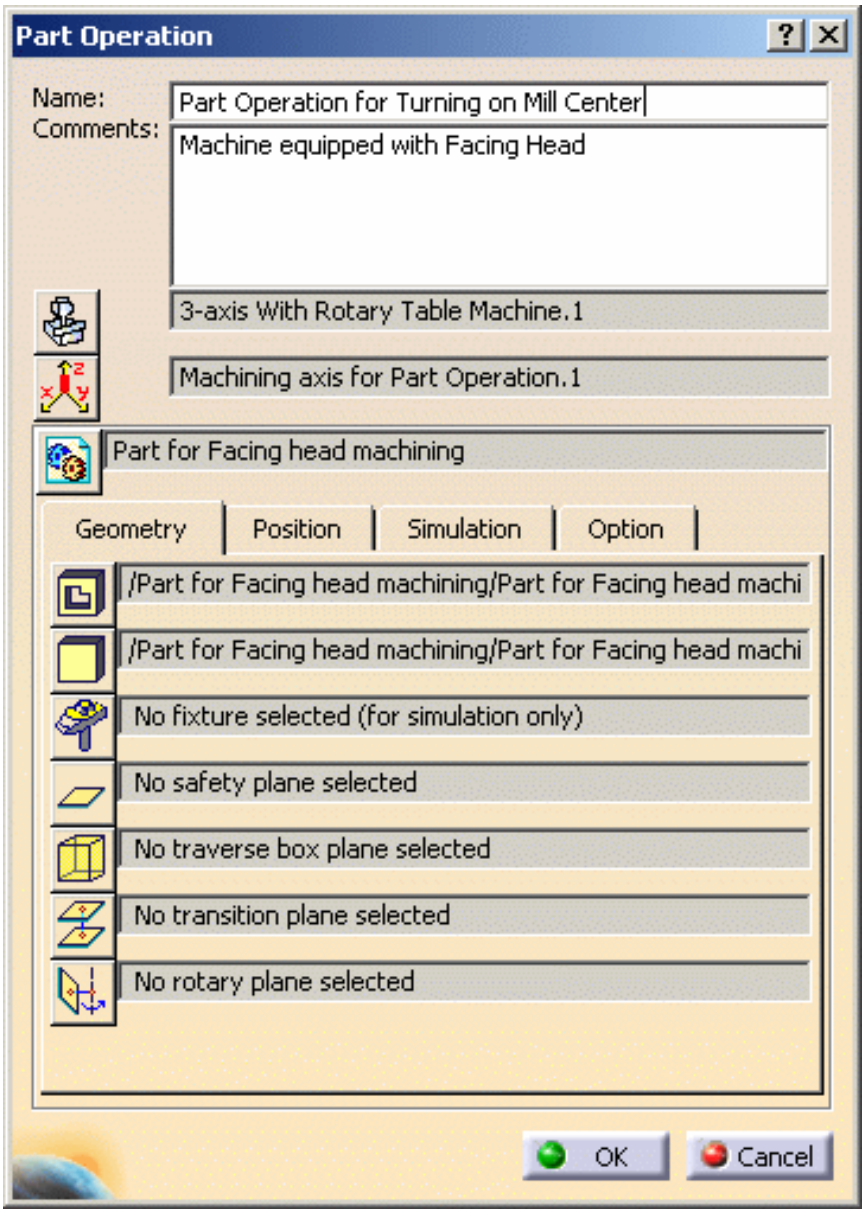
This section provides information about creating turning operations on a 3-axis milling machine that is equipped with a rotary table and facing head. The main objective is to machine large diameter holes using turning techniques, which will give better quality results than milling.

In particular, this section explains how to:

- set up your part and machining environment
- define local machining axis systems and turning planes
- create your program using either Machining Axis Changes or Table Rotations
- replay and simulation
- generate NC data from your program.

Set Up the Part and Machining Environment

The Part Operation editor allows you to set up the part and the machining environment.



Define the Reference Machining Axis System

Click the Reference Machining Axis System icon . The Machining Axis System dialog box appears for assigning a reference machining axis system to the part operation. This is similar to the procedure described in Insert Machining Axis Change.

The coordinates of the NC output data will be expressed in this axis system. However, when a local machining axis system is inserted in the program, coordinates will be expressed in the local axis system.

The reference machining axis system should be positioned such that it is Y-axis is collinear with the rotary axis of the machine (B).

Assign the Part to be Machined

Click the Product icon to associate the part to machine (CATProduct or CATPart) to the part operation.

Other important parameters to set in the Part Operation Editor are:

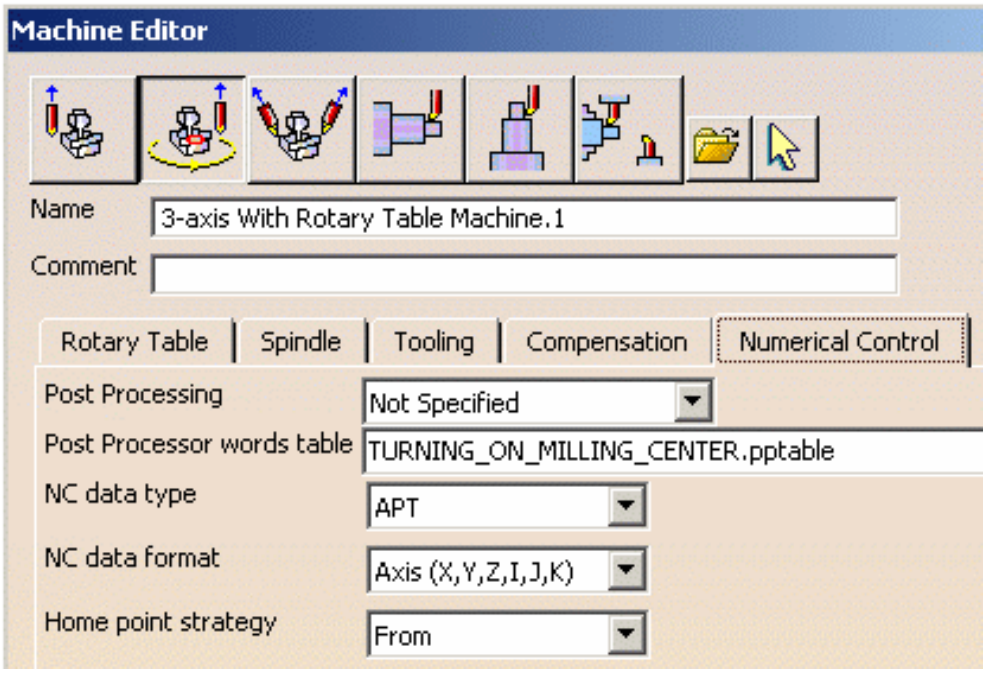
- Tool change point
- Home Point: From Machine.

Select the Machine and Set Parameters

Click the Machine icon the select the **3-axis with Rotary Table Machine** type in the Machine Editor.

In the case of an actual machine with facing head on the shop floor, the XYZIJK output could be post-processed to pilot the U-axis of the facing head.
The tool mounted on the facing head machines along the profile of the hole.
Note that this information is not specified in the Machine Editor dialog box.

In the Numerical Control tab, select the following sample PP word table is delivered with the product in \Startup\Manufacturing\PPTables\TURNING_ON_MILLING_CENTER.pptable



Other important parameters to set in the Machine Editor are:

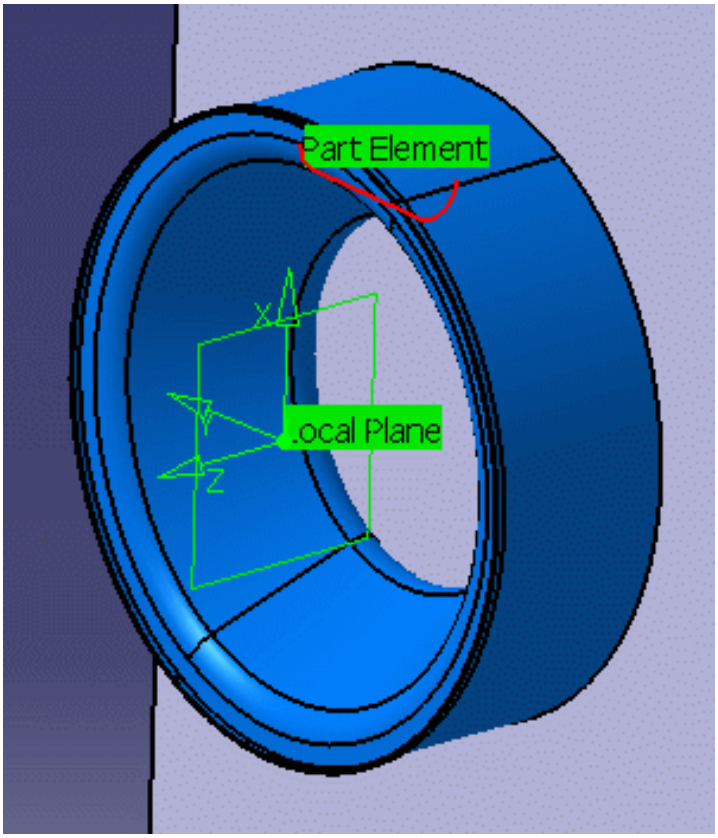
- Reference orientation of rotary table: set to 0, 0, 1 for example
- Rotary axis: set to B for example

- Home point, which is the start position when replaying a rotation if any motion is defined in the program.

Define Local Machining Axis Systems and Turning Planes

If you specify turning operations on a milling machine, they will be described in the ZX turning plane of the reference machining axis system.

To specify turning operations on a milling machine equipped with a facing head and rotary table, you must define local machining axis systems. Local turning planes will be derived from these Z and X axes of these axis systems. The geometry to machine must lie in the turning plane in order to create the turning operations.
The figure below shows a local machining axis system with the ZX turning plane and the selected geometry (Part Element in red) that lies in this plane.



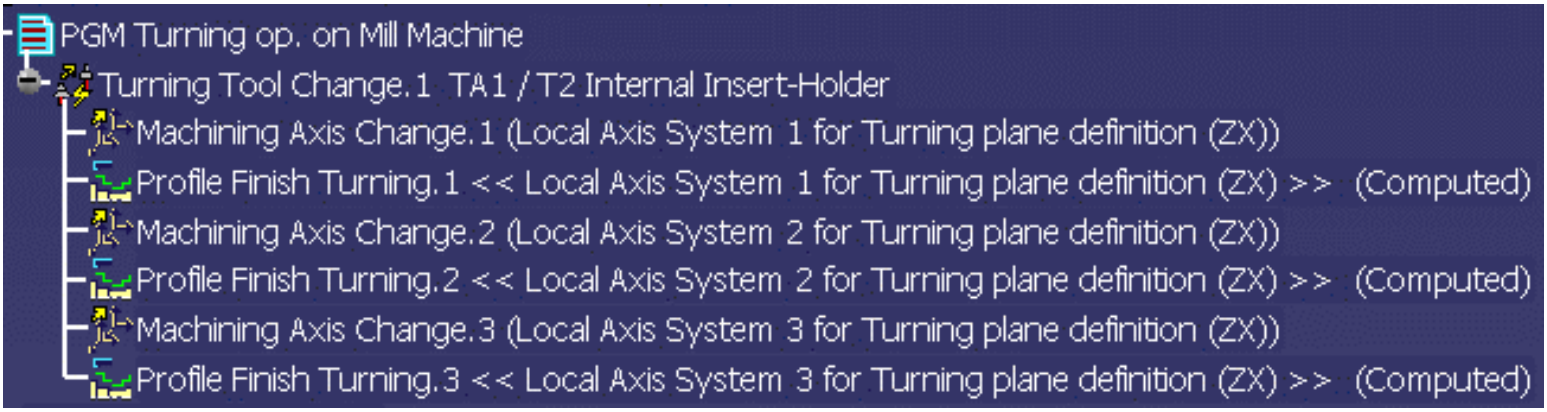
Therefore at each change of turning plane, you must define a local machining axis for turning operations. This is needed for processing geometry and visualizing the tool assembly.

Methodology: Machining Axis Changes or Table Rotations

Using Machining Axis Changes

To output NC data in the axis system defining the local plane, you must define a Machining Axis System Change before each turning plane.

A typical program is shown in the figure below:

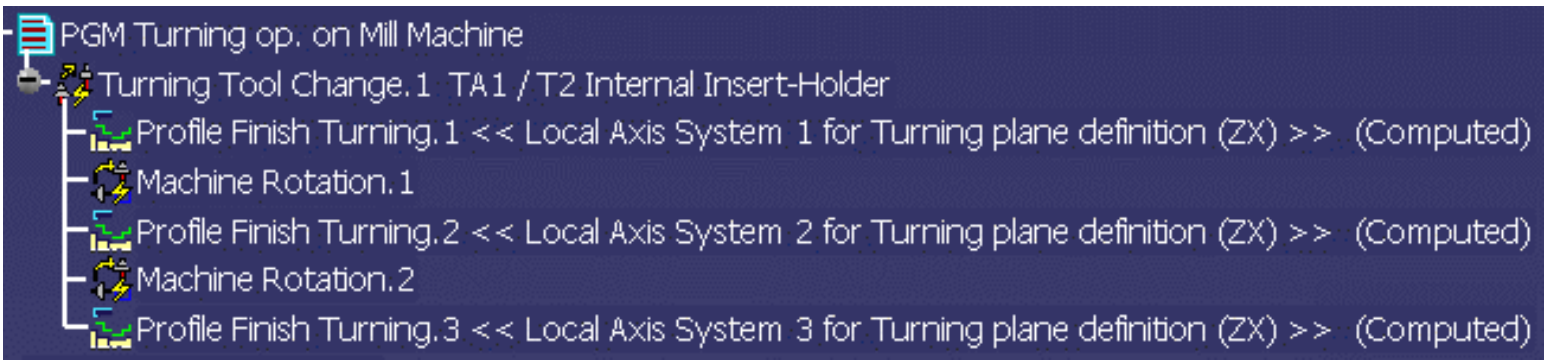


When NC data is generated in XYZIJK format, the IJK components will be the Z-axis of the local plane (see NC Data Output for more information).

Using Table Rotations

Define your turning operations in the corresponding local axis systems, then generate Machine Rotation instructions in the program using the command **Generate Machine Rotation** command.

A typical program is shown in the figure below:



When NC data is generated in XYZIJK format, the IJK components will be the Z-axis of the local plane (see NC Data Output for more information).

Replay and Simulation

You should replay the tool path to check each operation.

You should simulate the material removed by the program. You will need to specify design part and stock in the Part Operation editor.

NC Data Output

NC data output can be generated in XYZ or XYZIJK format.
For XYZ data, you must generate table rotations in your program.
For XYZIJK data, the value that is output for IJK is taken on the Z axis (spindle) of the local machining axis system.

The following NC data statements will be generated at the start of each turning operation:

- The coordinates of the origin of the local machining axis
- An order to switch from the X-axis to the U-axis.
  This is done through parameterized syntaxes in the PP table (NC_SPINDLE_LATHE or NC_LATHE_MO_START_COMMENT)

Using NC_SPINDLE_LATHE:

```
*START_NC_COMMAND NC_SPINDLE_LATHE
*START_LIST MFG_SPNDL_UNIT
RPM ,SFM
*END
LOCAL_ORIGIN,%MFG_NCAXIS_X_ORIG,%MFG_NCAXIS_Y_ORIG,%MFG_NCAXIS_Z_ORIG
$$ SWITCH_FROM_X_TO_U
SPINDL/%MFG_SPNDL_SPEED,&MFG_SPNDL_UNIT
*END
*END
```

Using NC_LATHE_MO_START_COMMENT:

```
*START_NC_INSTRUCTION NC_LATHE_MO_START_COMMENT
*START_SEQUENCE
LOCAL_ORIGIN,%MFG_NCAXIS_X_ORIG,%MFG_NCAXIS_Y_ORIG,%MFG_NCAXIS_Z_ORIG
$$ SWITCH_FROM_X_TO_U
*END
*END
```

A typical APT output would be:

```
$$ OPERATION NAME : Profile Finish Turning.1
$$ Start generation of : Profile Finish Turning.1
SWITCH/9
FEDRAT/ 0.3000,MMPR
LOCAL_ORIGIN, 0.00000, 130.00000, 0.000000
$$ SWITCH_FROM_X_TO_U
SPINDL/ 70.0000,RPM
GOTO / ...
```

Some Points to Note

The **Update Input Stock** capability is not available for turning operations on milling centers. The corresponding commands are not available in the Part Operation or Turning Operation editors.

Glossary

A D E F G M O P R S T U Z

A

approach macro Motion defined for approaching the operation start point

auxiliary command A control function such as tool change or machine table rotation. These commands may be interpreted by a specific post-processor.

axial machining operation Operation in which machining is done along a single axis and is mainly intended for hole making (drilling, counter boring, and so on).

D

DPM Digital Process for Manufacturing.

E

extension type Defines the end type of a hole as being through hole or blind.

F

feedrate Rate at which a cutter advances into a work piece.
Measured in linear or angular units (mm/min or mm/rev, for example).

fixture Elements used to secure or support the workpiece on a machine.

G

gouge Area where the tool has removed too much material from the workpiece.

M

machine rotation An auxiliary command in the program that corresponds to a rotation of the machine table.

machining axis system Reference axis system in which coordinates of points of the tool path are given.

machining operation Contains all the necessary information for machining a part of the workpiece using a single tool.

machining tolerance The maximum allowed difference between the theoretical and computed tool path.

manufacturing process Defines the sequence of part operations necessary for the complete manufacture of a part.

manufacturing program Describes the processing order of the NC entities that are taken into account for tool path computation: machining operations, auxiliary commands and PP instructions.

O

offset Specifies a virtual displacement of a reference geometric element in an operation (such as the offset on the bottom plane of a pocket, for example).
An offset value can be **greater than, less than, or equal to zero**. It is measured normal to the referenced geometry **or in a specific direction** such as axial or radial.
For example, a 5mm Offset on Contour means that a virtual displacement is applied normal to the contour geometry. A 5mm Axial Part Offset means that a virtual displacement is applied to the part geometry along the tool axis direction.
Compare with thickness.

one way Machining in which motion is always done in the same direction. Compare with zig zag.

P

part operation Links all the operations necessary for machining a part based on a unique part registration on a machine. The part operation links these operations with the associated fixture and set-up entities.

PP instruction Instructions that control certain functions that are auxiliary to the tool-part relationship. They may be interpreted by a specific post processor.

PPR Process Product Resources.

R

retract macro Motion defined for retracting from the operation end point

S

safety plane A plane normal to the tool axis in which the tool tip can move or remain a clearance distance away from the workpiece, fixture or machine.

set up Describes how the part, stock and fixture are positioned on the machine.

spindle speed The angular speed of the machine spindle.
Measured in linear or angular units (m/min or rev/min, for example).

stock Workpiece prior to machining by the operations of a part operation.

T

thickness Specifies a thickness of material to be removed by machining. A thickness value must be **greater than zero** and is measured **normal** to the machined geometry. For example, if a 5mm Finish Thickness is specified on an operation then 5mm of material will be removed during the finish pass.
Compare with offset.

tool axis Center line of the cutter.

tool change An auxiliary command in the program that corresponds to a change of tool.

tool clash Area where the tool collided with the workpiece during a rapid move.

tool path The trajectory that the tool follows during a machining operation.

total depth The total depth including breakthrough distance that is machined in a hole making operation.

turning plane Plane defined by two axes (called axial and radial) of a machining axis system. Selected geometry must lie on the turning plane.
In the Lathe Machining product, the tool assembly is positioned according to axial and radial axes (usually ZX) as well as the origin of the machining axis system. This is not the case in the Multi-Slide Lathe Machining product. In this case there is one plane per turret.

U

undercut Area where the tool has left material behind on the workpiece.

Z

zig zag Machining in which motion is done alternately in one direction then the other. Compare with one way.

Index

A B C D E F G I L M N O P R S T U Z

A

Add Local Information contextual command
Add User Representation contextual command
Analyze contextual command
Angle of Incline
- Groove Finish Turning parameter
- Groove Turning parameter
- Ramp Recess Turning parameter
- Recess Turning parameter

APT import
APT source generation
Attack Distance
- Groove Turning parameter
- Recess Turning parameter
- Rough Turning parameter

Automatic mode
- Update Input Stock

Automatic Stock Selection for Turning Operations contextual command
Auxiliary operation
- COPY Operator
- Copy Transformation
- Machine Rotation
- Machining Axis Change
- PP Instruction
- Tool Change
- TRACUT Operator

Axial Depth of Cut
- Recess Turning parameter
- Rough Turning parameter

Axial Limit for Chuck Jaws
- Ramp Rough Turning parameter
- Rough Turning parameter

Axial part offset
Groove Finish Turning parameter
Groove Turning parameter
Profile Finish Turning parameter
Recess Turning parameter
Rough Turning parameter

B

Browse Local Information contextual command
By Curve contextual command

C

CATProduct generation
Change Output Point
Groove Finish Turning parameter
Groove Turning parameter
Ramp Recess Turning parameter
Recess Turning parameter
Thread Turning parameter
Chip Break
Groove Turning parameter
Clean Computed Stock Data contextual command
Clearance on crest diameter
Thread Turning parameter
Clfile code generation command
Edit Cycle
Groove Turning
Lathe Groove Finish Turning
Profile Finish Turning
Ramp Recess Turning
Ramp Rough Turning
Recess Turning

Rough Turning

Sequential Turning

Thread Turning

Update Input Stock

Compute Stock and Tool Path contextual command

contextual command

Add Local Information

Add User Representation

Analyze

Automatic Stock Selection for Turning Operations

Browse Local Information

By Curve

Clean Computed Stock Data

Compute Stock and Tool Path

Copy Local Information

Edit Local Information

Edit NC Resources

Paste Local Information

Remove Input Stock

Remove Local Information

Save in Catalog

Sectioning

Update Computed Stock Status

Update Input Stock

Contouring for Outside Corners

Groove Finish Turning parameter

Profile Finish Turning parameter

Copy Local Information contextual command

COPY Operator

Copy-Transformation Instruction

Corner Processing

Groove Finish Turning parameter

Profile Finish Turning parameter

CUTCOM

Groove Finish Turning parameter

Profile Finish Turning parameter

D

Depth of cut
Thread Turning parameter
direct axis system
direct machining axis system
Documentation generation

E

Edge Selection toolbar
Edit Cycle command
Edit Local Information contextual command
Edit NC Resources contextual command
End Limit
Groove Finish Turning parameter
Profile Finish Turning parameter
Ramp Rough Turning parameter
Rough Turning parameter
Thread Turning parameter
End Limit Offset
Groove Finish Turning parameter
Profile Finish Turning parameter
Thread Turning parameter
End limit offset
Rough Turning parameter
Entry Flank Angle
Ramp Recess Turning parameter
Ramp Rough Turning parameter
Exit Flank Angle
Ramp Recess Turning parameter
External Groove Finish Turning
External Groove Turning

External Profile Finish Turning
External Ramp Recess Turning
External Ramp Rough Turning
External Recess Turning
External Rough Turning
External Threading

F

Face
- Ramp Rough Turning mode
- Rough Turning mode

Face Ramp Rough Turning
Face Rough Turning
Feeds and Speeds
- Groove Finish Turning parameter
- Groove Turning parameter
- Profile Finish Turning parameter
- Ramp Recess Turning parameter
- Ramp Rough Turning parameter
- Recess Turning parameter
- Rough Turning parameter
- Sequential Turning parameter

First passes
- Thread Turning parameter

First Plunge
- Groove Turning parameter

Flank Gouging Angle
- Ramp Recess Turning parameter

Follow
- Sequential Turning motion

Follow Tool Motion
Frontal Groove Finish Turning
Frontal Groove Turning
Frontal Profile Finish Turning

Frontal Ramp Recess Turning
Frontal Ramp Rough Turning
Frontal Recess Turning
Frontal Rough Turning

G

Go Delta
Sequential Turning motion
Go Delta Motion
Go InDirv
Sequential Turning motion
Go InDirv Tool Motion
Go Standard
Sequential Turning motion
Go Standard Tool Motion
Gouging Safety Angle
Groove Turning parameter
Recess Turning parameter
Groove Finish Turning operation
Groove Finish Turning parameter
Angle of Incline
Axial part offset
Change Output Point
Contouring for Outside Corners
Corner Processing
CUTCOM
End Limit
End Limit Offset
Feeds and Speeds
Lead-in
Lift-off
Machining Direction
Machining Tolerance
Offset

- Orientation
- Overlap
- Part offset
- Radial part offset
- Start Limit
- Start Limit Offset
- Tool Compensation
- Under Spindle Axis Machining

Groove Turning command

Groove Turning operation

Groove Turning parameter

- Angle of Incline
- Attack Distance
- Axial part offset
- Change Output Point
- Chip Break
- Feeds and Speeds
- First Plunge
- Gouging Safety Angle
- Grooving by Level
- Lead-in
- Lift-off
- Machining Tolerance
- Maximum Depth of Cut
- Next Plunges
- Orientation
- Part Contouring
- Part offset
- Radial part offset
- Stock offset
- Tool Compensation
- Under Spindle Axis Machining

Grooving by Level

- Groove Turning parameter

I

Inclined Groove Finish Turning
Inclined Groove Turning
Inclined Ramp Recess Turning
Inclined Recess Turning
insert holder
Insert Holder in operation
edit or select another
Insert Holder in Resource List
edit
Insert in operation
edit or select another
Insert in Resource List
edit
Internal Groove Finish Turning
Internal Groove Turning
Internal Profile Finish Turning
Internal Ramp Recess Turning
Internal Ramp Rough Turning
Internal Recess Turning
Internal Rough Turning
Internal Threading

L

Last passes
Thread Turning parameter
Lathe Groove Finish Turning command
Lead-in
Groove Finish Turning parameter
Groove Turning parameter
Profile Finish Turning parameter
Recess Turning parameter
Thread Turning parameter

Lead-in Angle
Rough Turning parameter
Lead-in Distance
Rough Turning parameter
Leading Safety Angle
Profile Finish Turning parameter
Recess Turning parameter
Rough Turning parameter
Lift-off
Groove Finish Turning parameter
Groove Turning parameter
Profile Finish Turning parameter
Recess Turning parameter
Thread Turning parameter
Lift-off Angle
Rough Turning parameter
Lift-off Distance
Rough Turning parameter
Local Information
Location
Profile Finish Turning parameter
Ramp Rough Turning parameter
Rough Turning parameter
Thread Turning parameter
Longitudinal
Ramp Rough Turning mode
Rough Turning mode
Longitudinal Ramp Rough Turning
Longitudinal Rough Turning

M

Machine Rotation
Machining Axis Change
Machining Direction
Groove Finish Turning parameter

Profile Finish Turning parameter
Recess Turning parameter
Rough Turning parameter
Machining Process command
Machining Process View command
Machining Process, Apply
Machining Process, Create
Machining spindle speed
Thread Turning parameter
Machining Tolerance
Groove Finish Turning parameter
Groove Turning parameter
Profile Finish Turning parameter
Recess Turning parameter
Rough Turning parameter
Manual mode
Update Input Stock
Manufacturing Program
Maximum Depth of Cut
Groove Turning parameter
Recess Turning parameter
Rough Turning parameter
Maximum Machining Radius
Ramp Rough Turning parameter
Rough Turning parameter
Minimum Machining Radius
Ramp Rough Turning parameter
Rough Turning parameter

N

NC code generation
Next Plunges
Groove Turning parameter
Nominal diameter
Thread Turning parameter

Number of passes
Thread Turning parameter
Number of threads
Thread Turning parameter

O

Offset
Groove Finish Turning parameter
One Way
Ramp Recess Turning mode
Recess Turning mode
One Way Ramp Recess Turning
One Way Recess Turning
Opposite Hand Machining
Orientation
Groove Finish Turning parameter
Groove Turning parameter
Profile Finish Turning parameter
Ramp Recess Turning parameter
Ramp Rough Turning parameter
Recess Turning parameter
Rough Turning parameter
Thread Turning parameter
output point, definition
output point, examples
Overlap
Groove Finish Turning parameter

P

Parallel Contour
Recess Turning mode
Rough Turning mode
Parallel Contour Recess Turning

Parallel Contour Rough Turning
Part Contouring
Groove Turning parameter
Recess Turning parameter
Rough Turning parameter
Part offset
Groove Finish Turning parameter
Groove Turning parameter
Profile Finish Turning parameter
Recess Turning parameter
Rough Turning parameter
Part Operation
Paste Local Information contextual command
Plunge
Recess Turning parameter
Plunge Angle
Rough Turning parameter
Plunge Distance
Rough Turning parameter
PP Instruction
Process List
Product List
Profile Finish Turning command
Profile Finish Turning operation
Profile Finish Turning parameter
Axial part offset
Contouring for Outside Corners
Corner Processing
CUTCOM
End Limit
End Limit Offset
Feeds and Speeds
Lead-in
Leading Safety Angle
Lift-off
Location

Machining Direction
Machining Tolerance
Orientation
Part offset
Radial part offset
Recess Machining
Start Limit
Start Limit Offset
Tool Compensation
Trailing Safety Angle
Under Spindle Axis Machining

R

Radial Depth of Cut
Recess Turning parameter
Rough Turning parameter
Radial part offset
Groove Finish Turning parameter
Groove Turning parameter
Profile Finish Turning parameter
Recess Turning parameter
Rough Turning parameter
Ramp Recess Turning command
Ramp Recess Turning mode
One Way
Zig Zag
Ramp Recess Turning operation
Ramp Recess Turning parameter
Angle of Incline
Change Output Point
Entry Flank Angle
Exit Flank Angle
Feeds and Speeds
Flank Gouging Angle

Orientation

Tool Compensation

Ramp Rough Turning command

Ramp Rough Turning mode

Face

Longitudinal

Ramp Rough Turning operation

Ramp Rough Turning parameter

Axial Limit for Chuck Jaws

End Limit

Entry Flank Angle

Feeds and Speeds

Location

Maximum Machining Radius

Minimum Machining Radius

Orientation

Tool Compensation

Recess Machining

Profile Finish Turning parameter

Rough Turning parameter

Recess Turning command

Recess Turning mode

One Way

Parallel Contour

Zig Zag

Recess Turning operation

Recess Turning parameter

Angle of Incline

Attack Distance

Axial Depth of Cut

Axial part offset

Change Output Point

Feeds and Speeds

Gouging Safety Angle

Lead-in

Leading Safety Angle

Lift-off

Machining Direction

Machining Tolerance

Maximum Depth of Cut

Orientation

Part Contouring

Part offset

Plunge

Radial Depth of Cut

Radial part offset

Stock offset

Tool Compensation

Trailing Safety Angle

Under Spindle Axis Machining

Remove Input Stock contextual command

Remove Local Information contextual command

Resources List

rotary direction

Rough Turning command

Rough Turning mode

Face

Longitudinal

Parallel Contour

Rough Turning operation

Rough Turning parameter

Attack Distance

Axial Depth of Cut

Axial Limit for Chuck Jaws

Axial part offset

End Limit

End limit offset

Feeds and Speeds

Lead-in Angle

Lead-in Distance
Leading Safety Angle
Lift-off Angle
Lift-off Distance
Location
Machining Direction
Machining Tolerance
Maximum Depth of Cut
Maximum Machining Radius
Minimum Machining Radius
Orientation
Part Contouring
Part offset
Plunge Angle
Plunge Distance
Radial Depth of Cut
Radial part offset
Recess Machining
Stock offset
Tool Compensation
Trailing Safety Angle
Under Spindle Axis Machining

S

Save in Catalog contextual command
Sectioning contextual command
Sequential Turning command
Sequential Turning motion
Follow
Go Delta
Go InDirv
Go Standard
Sequential Turning operation

Sequential Turning parameter
Feeds and Speeds
setup angle, 180deg
Simulate material removal
spindle rotary direction
Spring passes
Thread Turning parameter
Start Limit
Groove Finish Turning parameter
Profile Finish Turning parameter
Thread Turning parameter
Start Limit Offset
Groove Finish Turning parameter
Profile Finish Turning parameter
Thread Turning parameter
Stock offset
Groove Turning parameter
Recess Turning parameter
Rough Turning parameter

T

Thread depth
Thread Turning parameter
Thread length
Thread Turning parameter
Thread Penetration type
Thread Turning parameter
Thread pitch
Thread Turning parameter
Thread profile
Thread Turning parameter
Thread Turning command
Thread Turning operation
Thread Turning parameter
Change Output Point
Clearance on crest diameter

Depth of cut

End Limit

End Limit Offset

First passes

Last passes

Lead-in

Lift-off

Location

Machining spindle speed

Nominal diameter

Number of passes

Number of threads

Orientation

Spring passes

Start Limit

Start Limit Offset

Thread depth

Thread length

Thread Penetration type

Thread pitch

Thread profile

Thread unit

Threading type

Threads/inch

Tool Compensation

Thread unit

Thread Turning parameter

Threading type

Thread Turning parameter

Threads/inch

Thread Turning parameter

Tool Assembly in operation

edit or select another

Tool Assembly in Resource List

edit

Tool Change
Tool Compensation
- Groove Finish Turning parameter
- Groove Turning parameter
- Profile Finish Turning parameter
- Ramp Recess Turning parameter
- Ramp Rough Turning parameter
- Recess Turning parameter
- Rough Turning parameter
- Thread Turning parameter

Tool path replay
tool positioning
tooling query in machining process
Tools Options - Machining
- General
- Operation
- Output
- Photo/Video
- Program
- Resources

TRACUT Operator
Trailing Safety Angle
- Profile Finish Turning parameter
- Recess Turning parameter
- Rough Turning parameter

turning plane

U

Under Spindle Axis Machining
- Groove Finish Turning parameter
- Groove Turning parameter
- Profile Finish Turning parameter
- Recess Turning parameter
- Rough Turning parameter

Update Computed Stock Status contextual command
Update Input Stock
Automatic mode
Manual mode
Update Input Stock command
Update Input Stock contextual command
Update Input Stock restrictions

Z

Zig Zag
Ramp Recess Turning mode
Recess Turning mode
Zig Zag Ramp Recess Turning
Zig Zag Recess Turning