

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
Local Information
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

NC Manufacturing Entities

Edit Lathe Tool on an Operation
Edit Lathe Tool in Resource List
Edit Lathe Tool Assembly in Resource List
Edit Lathe Insert in 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
Lathe Tool Assembly Conventions

Methodology

Cutter Compensation and Finish Operations
How to Change the Output Point
How to Update Input Stock

Glossary

Index

Prasanth

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*.

Prasanth

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 the Advanced Tasks
	Workbench Description
	Customizing
	Reference
	Methodology
	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

Enhancements brought by the NC Manufacturing Infrastructure

This product benefits from enhancements to the infrastructure's general functions (NC Program Review, NC Resources, Program Management, Geometry Management, Replay and Simulation, NC Data Output, and so on).

Please refer to the *NC Manufacturing Infrastructure User's Guide* for more information.

Prasanth

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

Prasanth

Open the Part to Machine



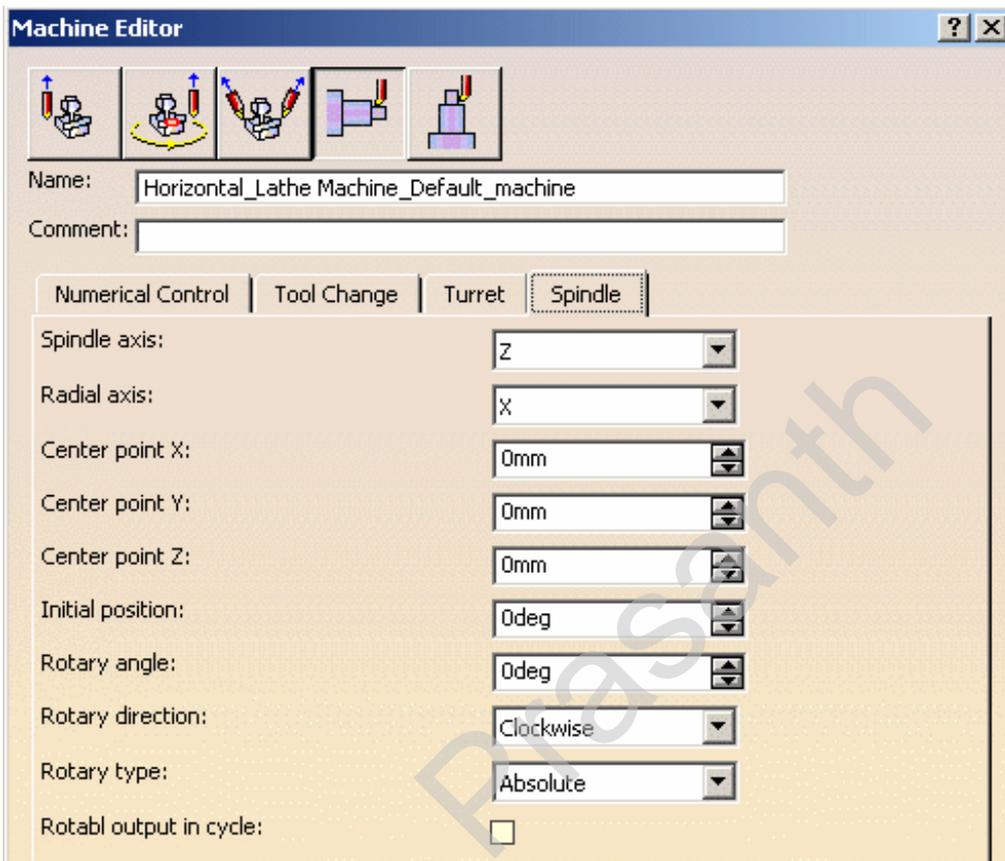
This first task shows you how to open a part, enter the Lathe Machining workbench and make basic modifications to the Part Operation.



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.

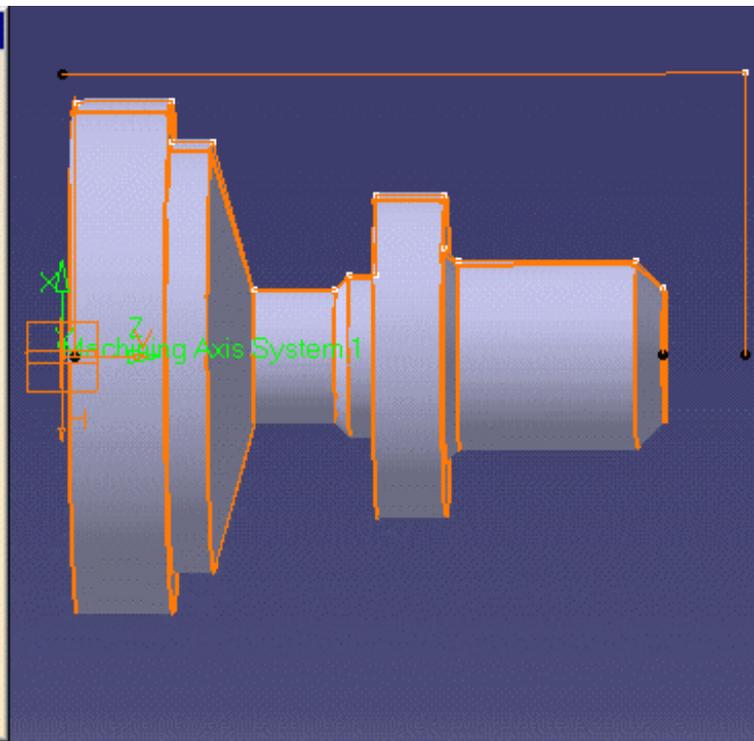
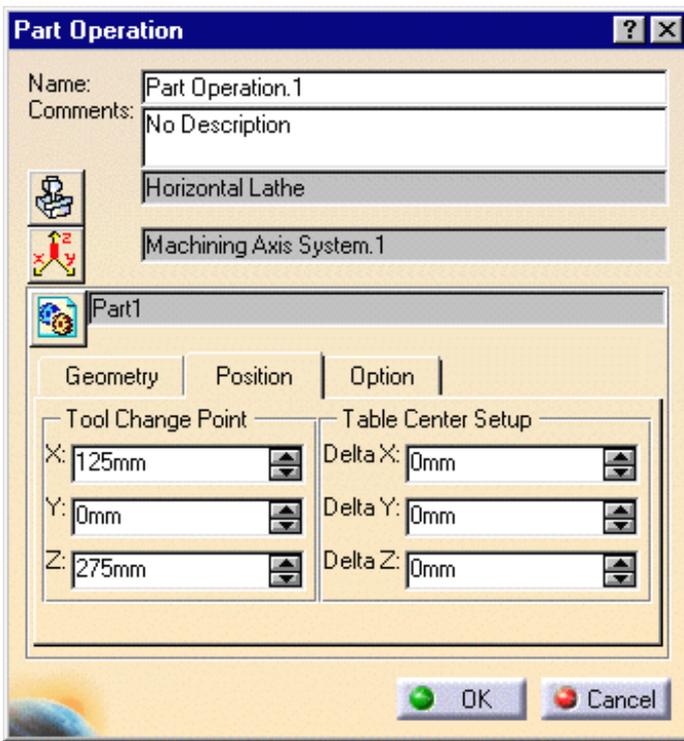


- Select the Horizontal Lathe Machine icon.
- Set the spindle axis to Z and the radial axis to X.
- Click OK.



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.



Prasanth

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.

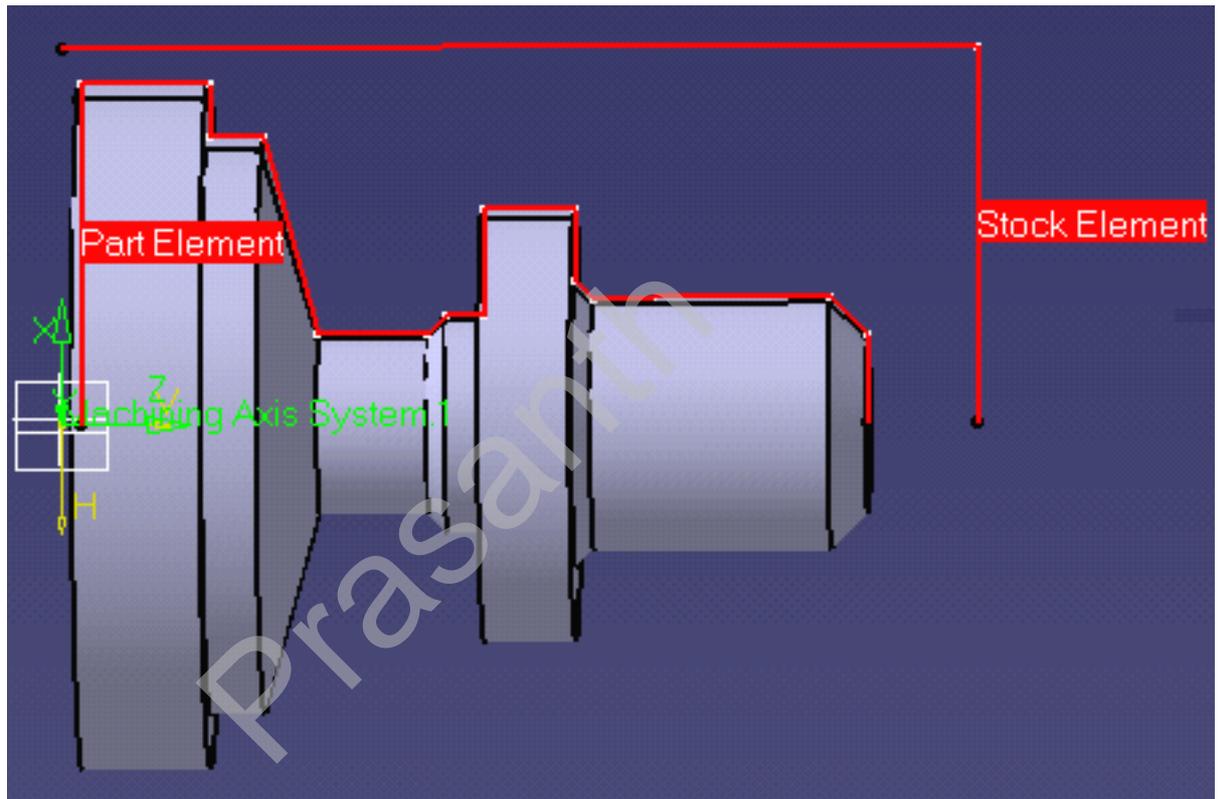
The Rough Turning dialog box appears directly at the Geometry tab page .

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

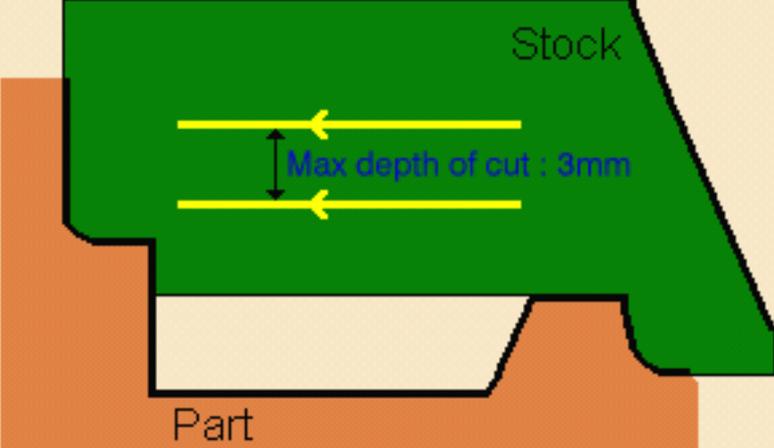
Click OK in the Edge Selection toolbar to end your selection.

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.



Stock

Part

Max depth of cut : 3mm

Roughing mode:

Orientation:

Location:

Machining direction:

Part contouring:

5. Click OK to create the operation.


Prasanth

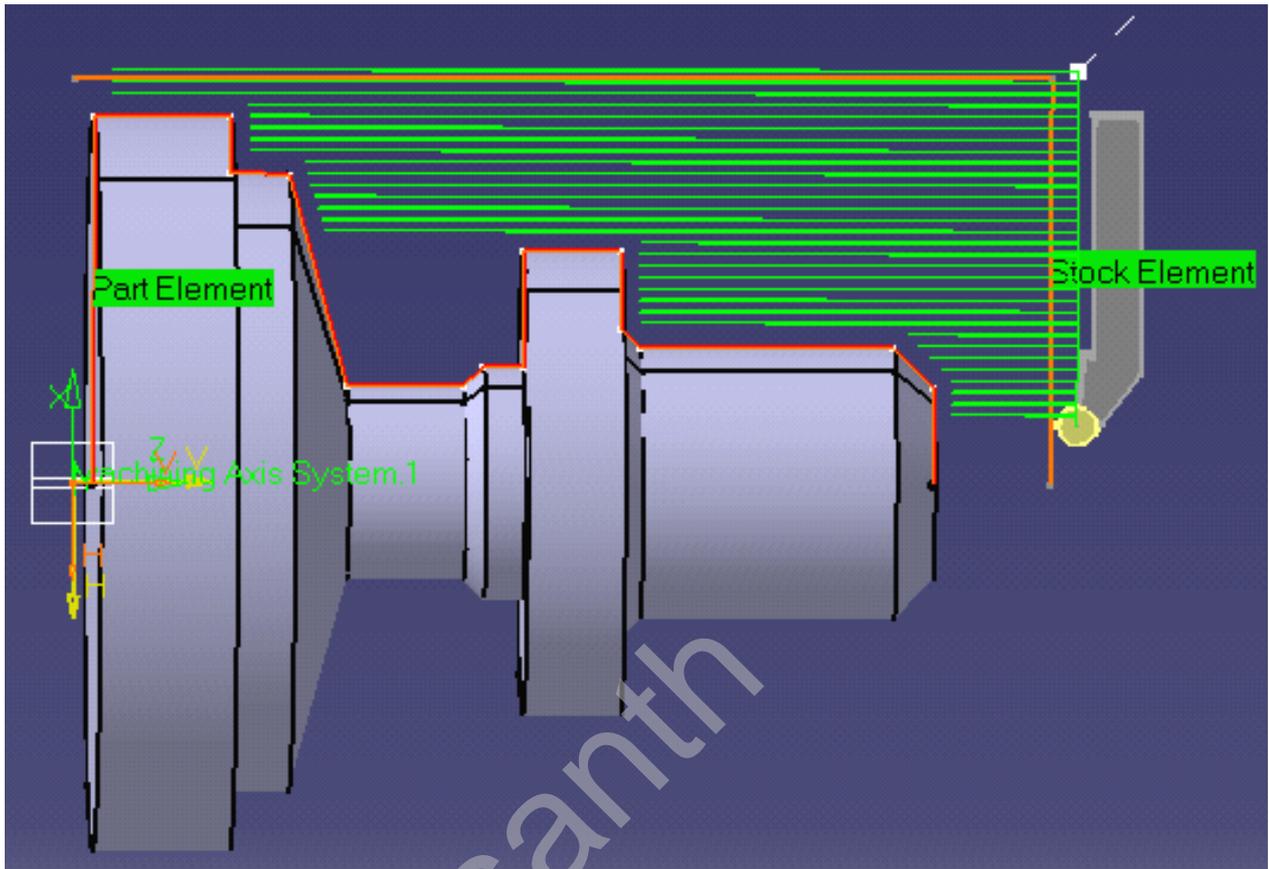
Replay the Tool Path

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.



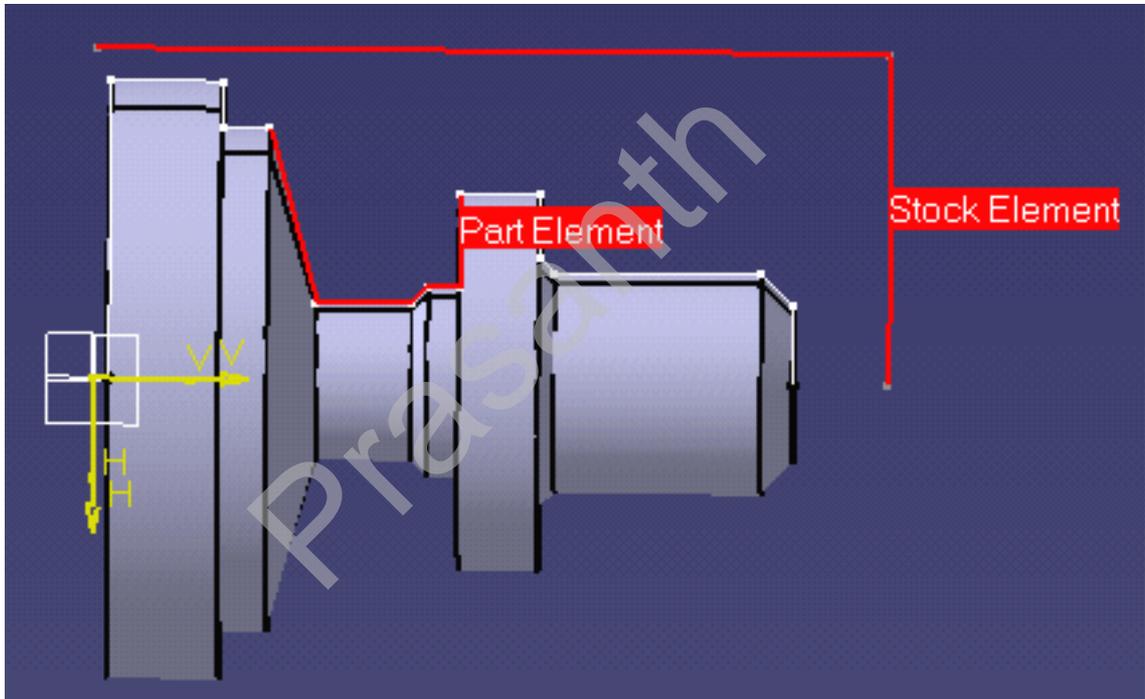
Make sure that the Rough Turning operation is the current entity in the program.



1. Select the Groove Turning icon .

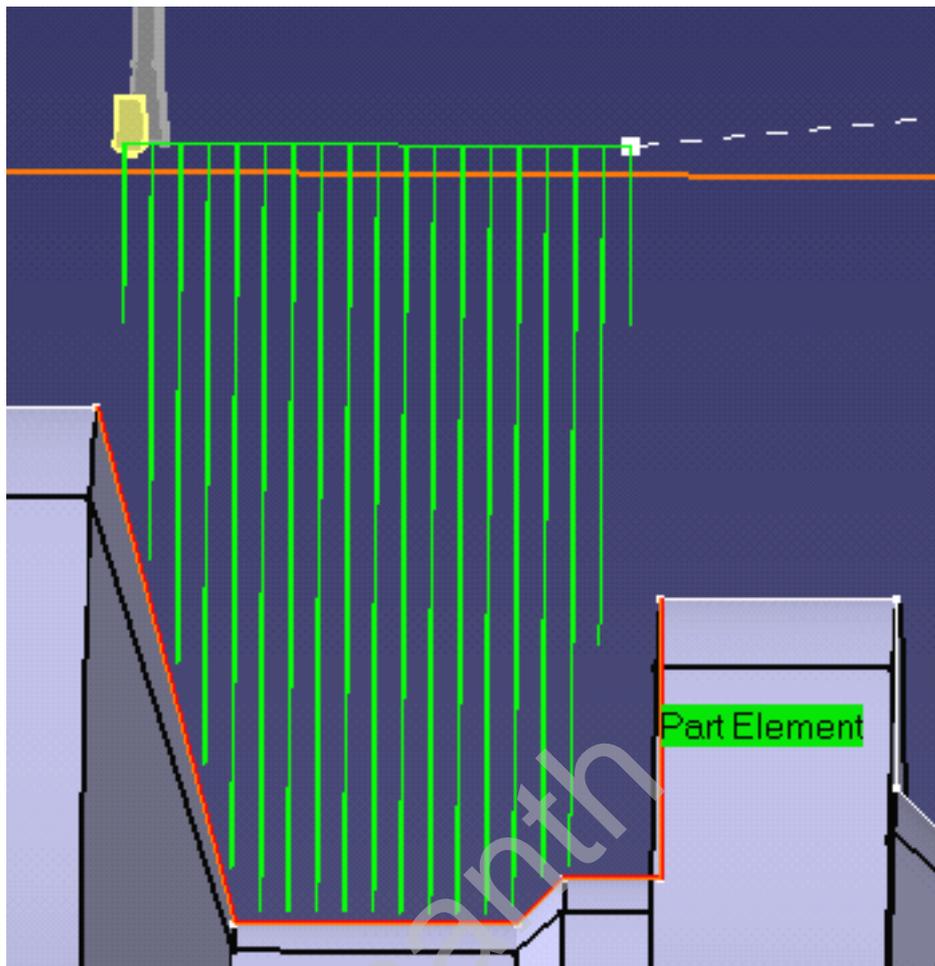
The Groove Turning dialog box appears directly at the Geometry page .

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 Tool tab  in the Tooling tab 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.



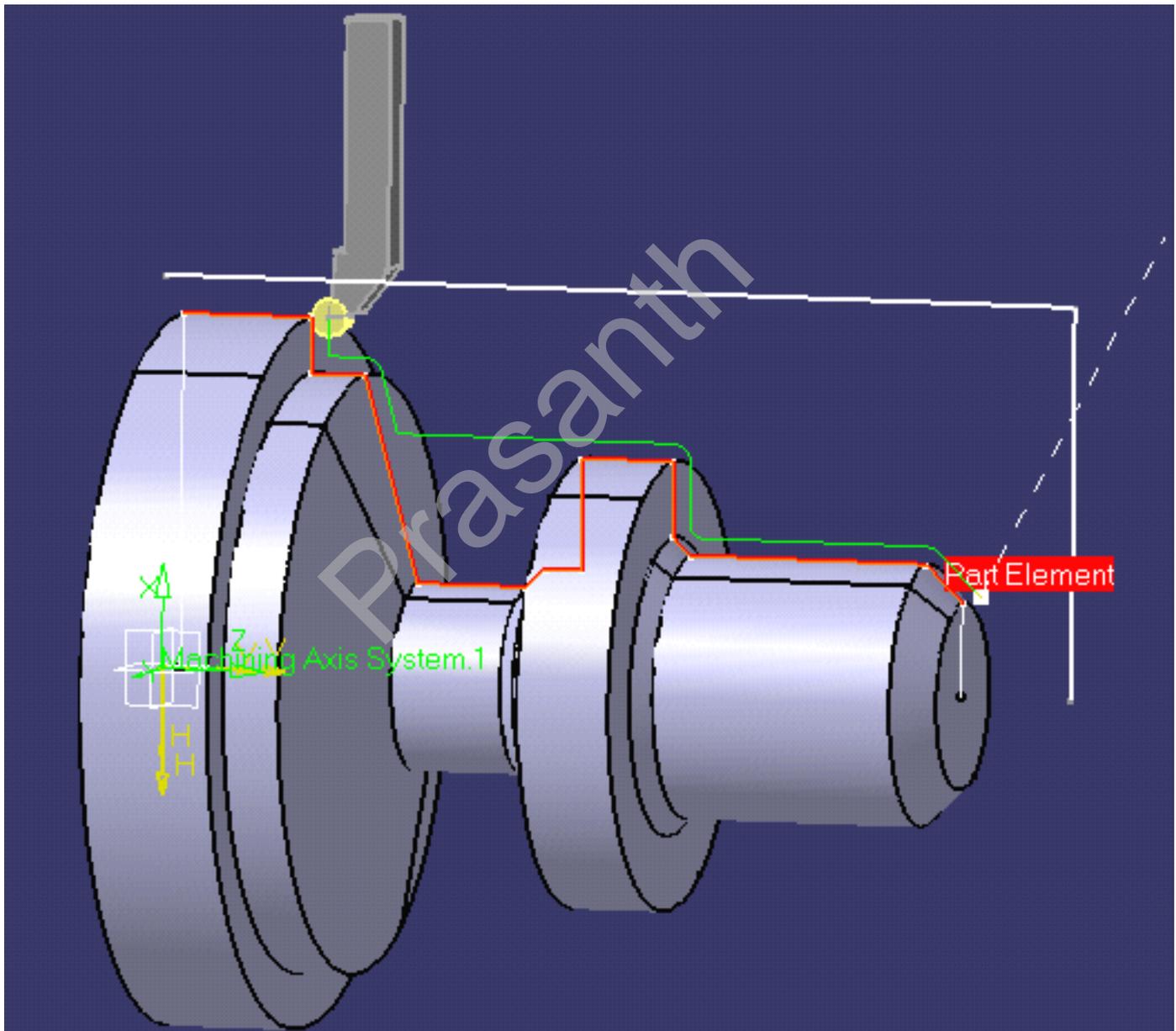
Create a Profile Finish Turning Operation



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.

Generate NC Output Interactively [?] [X]

In/Out | Tool motions | Formatting | NC Code

Input

Input CATProcess :
Process1.CATProcess

Selection Part Operations
 Programs

Manufacturing Program.1

Resulting NC Data

NC data type: APT

One file... for all selected programs
 by program
 by machining operation

Output File :
 Store at the same location as the CATProcess
E:\tmp\Process1.aptsource
 Replace like-named file

CATProcess after NC data generation

Save input CATProcess :
E:\tmp\Process1.CATProcess
 Replace like-named CATProcess
 Lock operations
 Associate output NC file to the program

Execute [Cancel]



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
$$*CATIAO
$$ 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 Tools
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

Prasanth

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

NC Manufacturing Entities

Verification, Simulation and NC Output

Prasanth

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.

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.

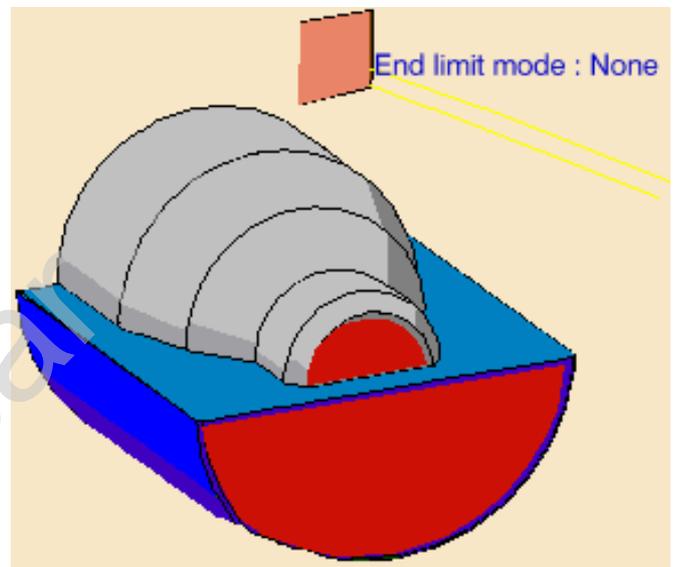


1. Select the Rough Turning icon .

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.

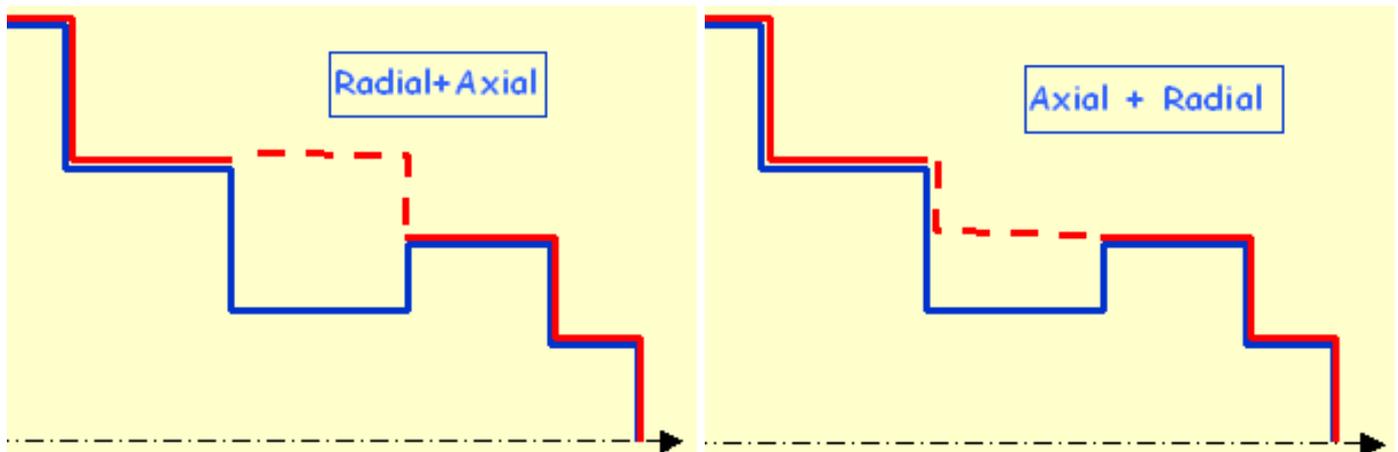


The part and stock of the icon are colored red indicating that this geometry is required. All other geometry is optional.

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.



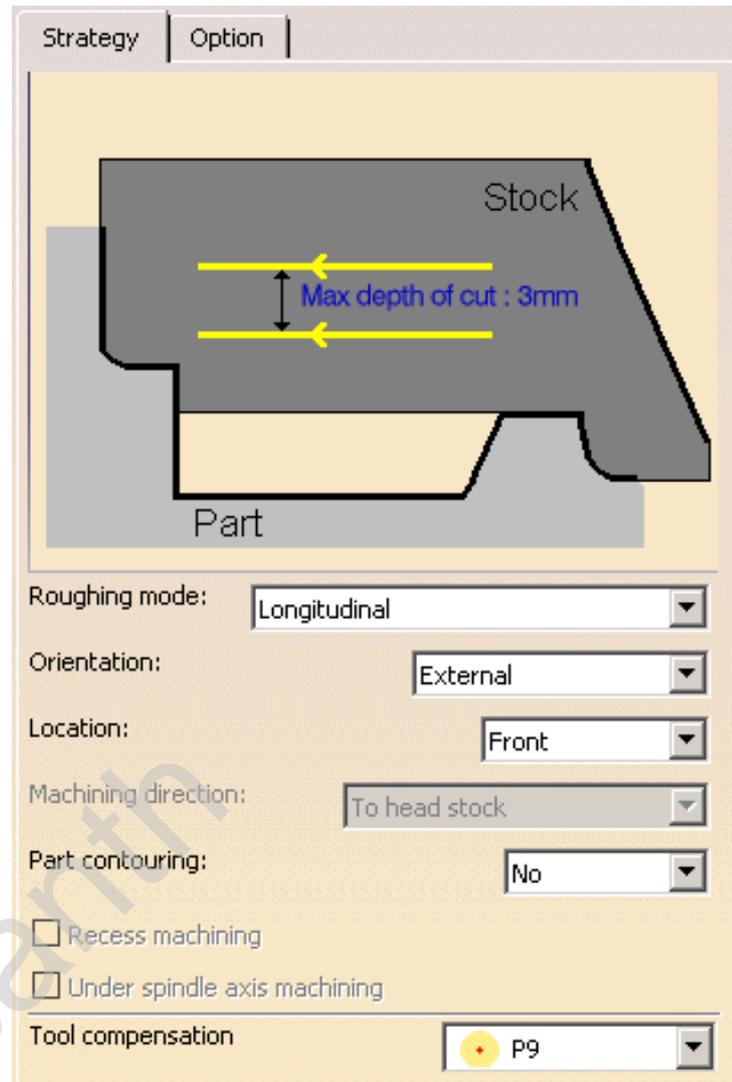
The part of the icon is now colored green indicating that this geometry is now defined.

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.

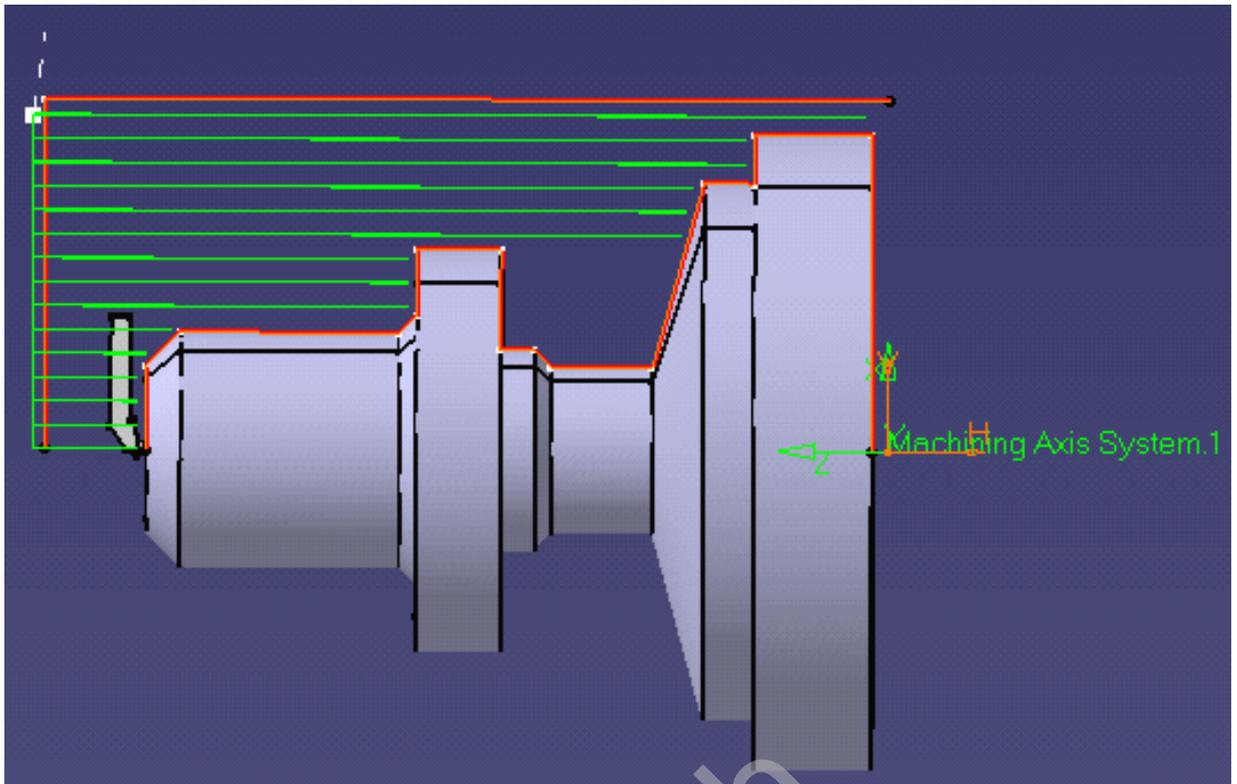
8. Select the Macros tab page  to specify the operation's transition paths (approach and retract motion, for example).

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.

Prasanth

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)** 

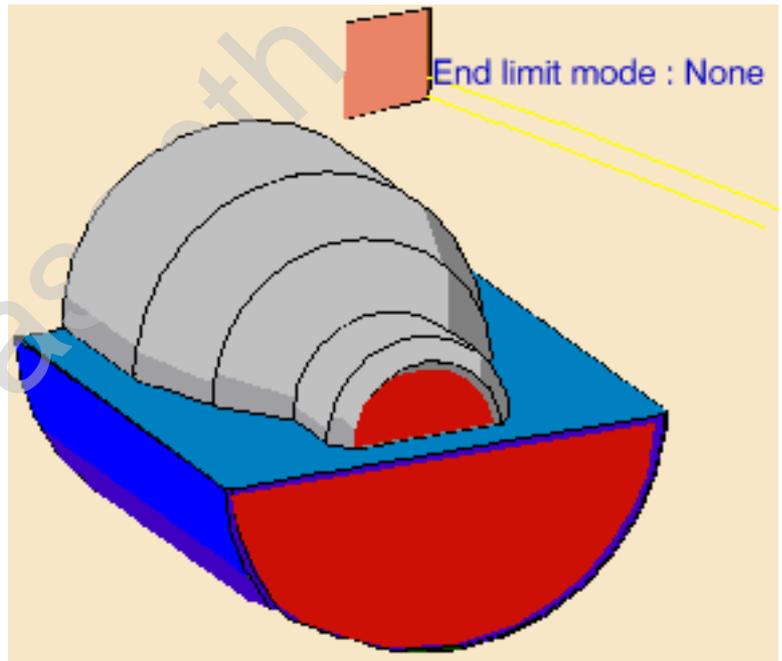
 Open the **Lathe01.CATPart** document, then select **Machining > Lathe Machining** from the **Start** menu. Make the Manufacturing Program current in the specification tree.

-  **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.

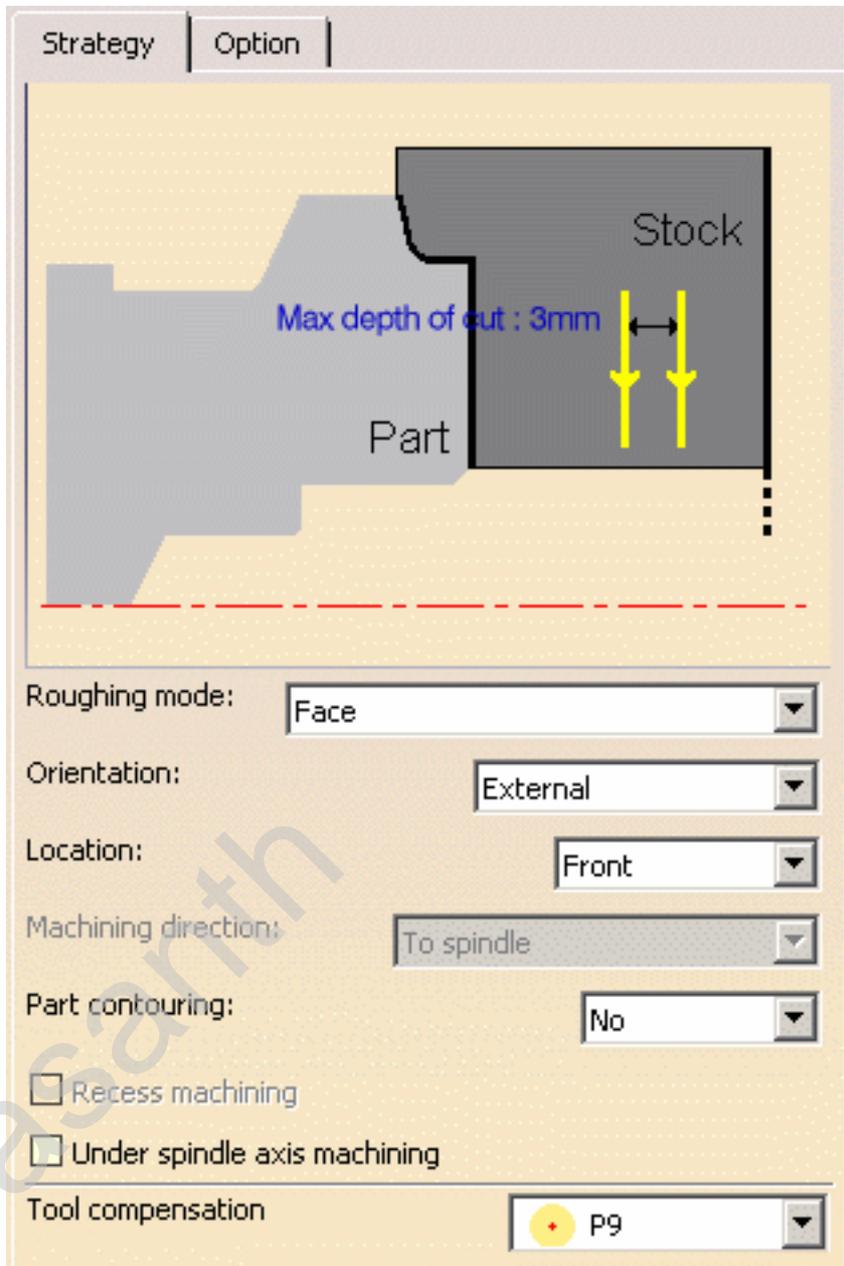
 The part and stock in the icon are colored red indicating that this geometry is required for defining the operation.



- 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.

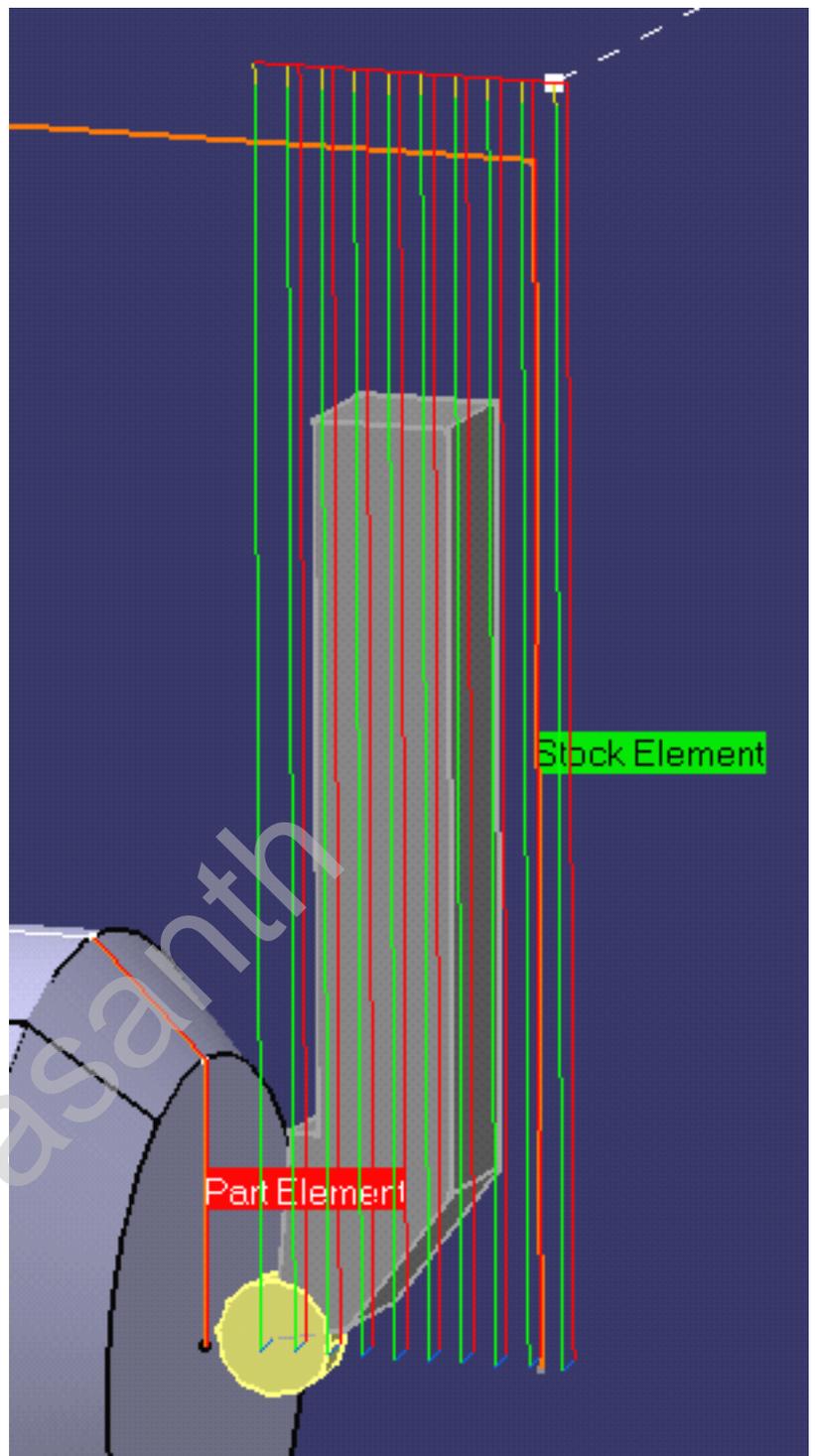


If you want to specify approach and retract motion for the operation, select the Macros tab page  to specify the desired transition paths.

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.



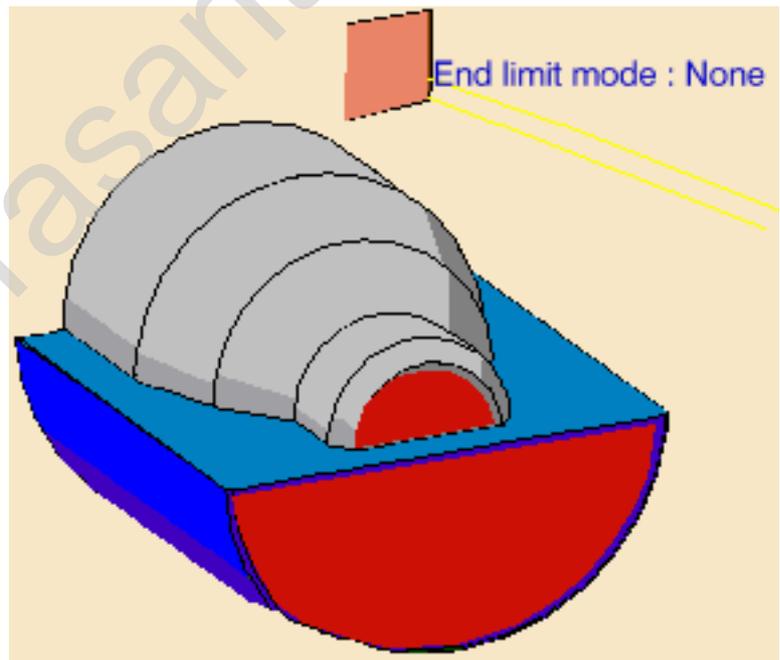
1. Select the Rough Turning icon



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.

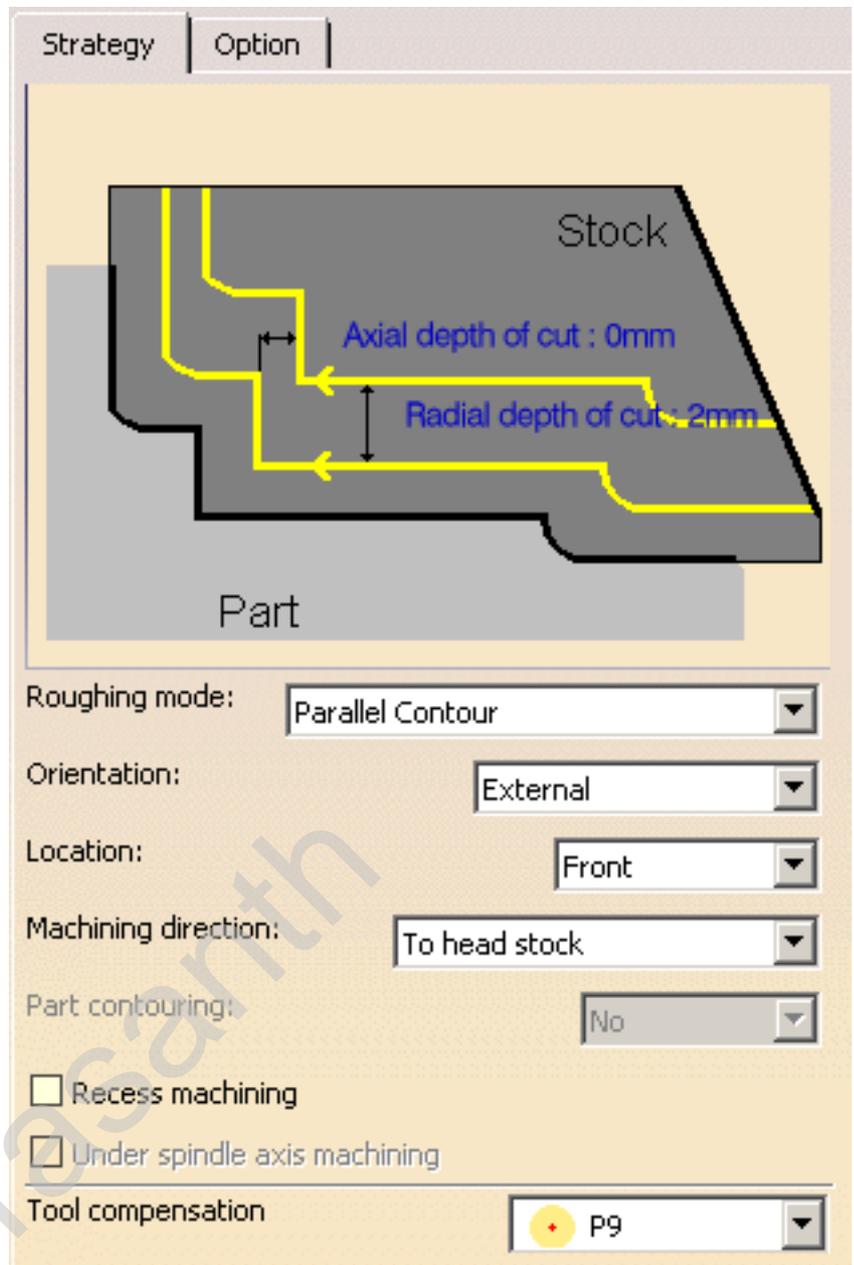


The part and stock of the icon are colored red indicating that this geometry is required.

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.

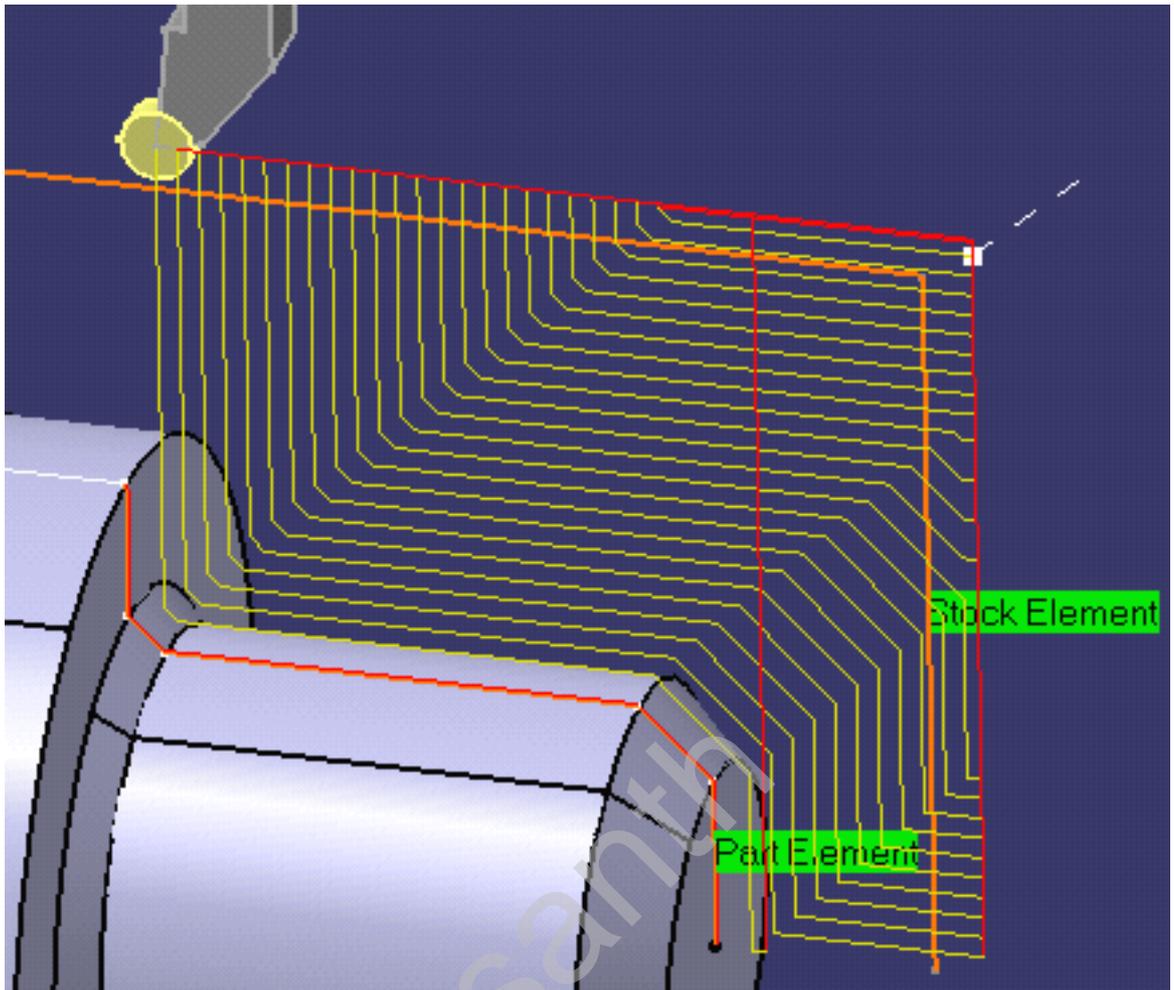
See [Feeds and Speeds for Rough Turning](#) for more information.

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.

Prasanjit

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)** 

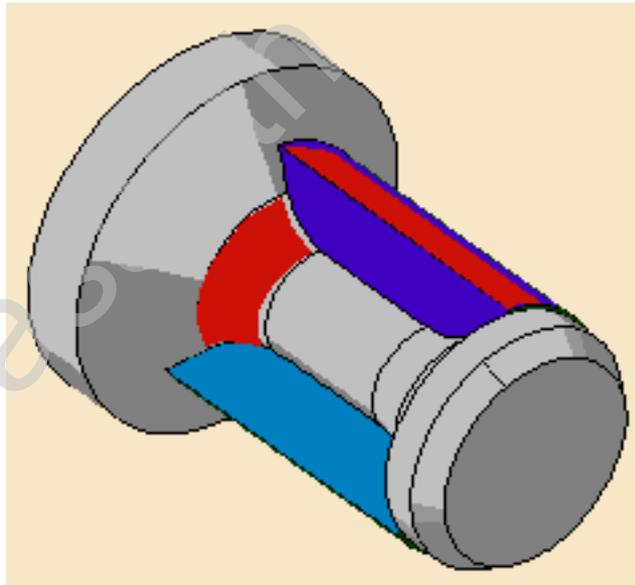
 Open the **Lathe01.CATPart** document, then select **Machining > Lathe Machining** from the Start menu. Make the Manufacturing Program current in the specification tree.

 **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.

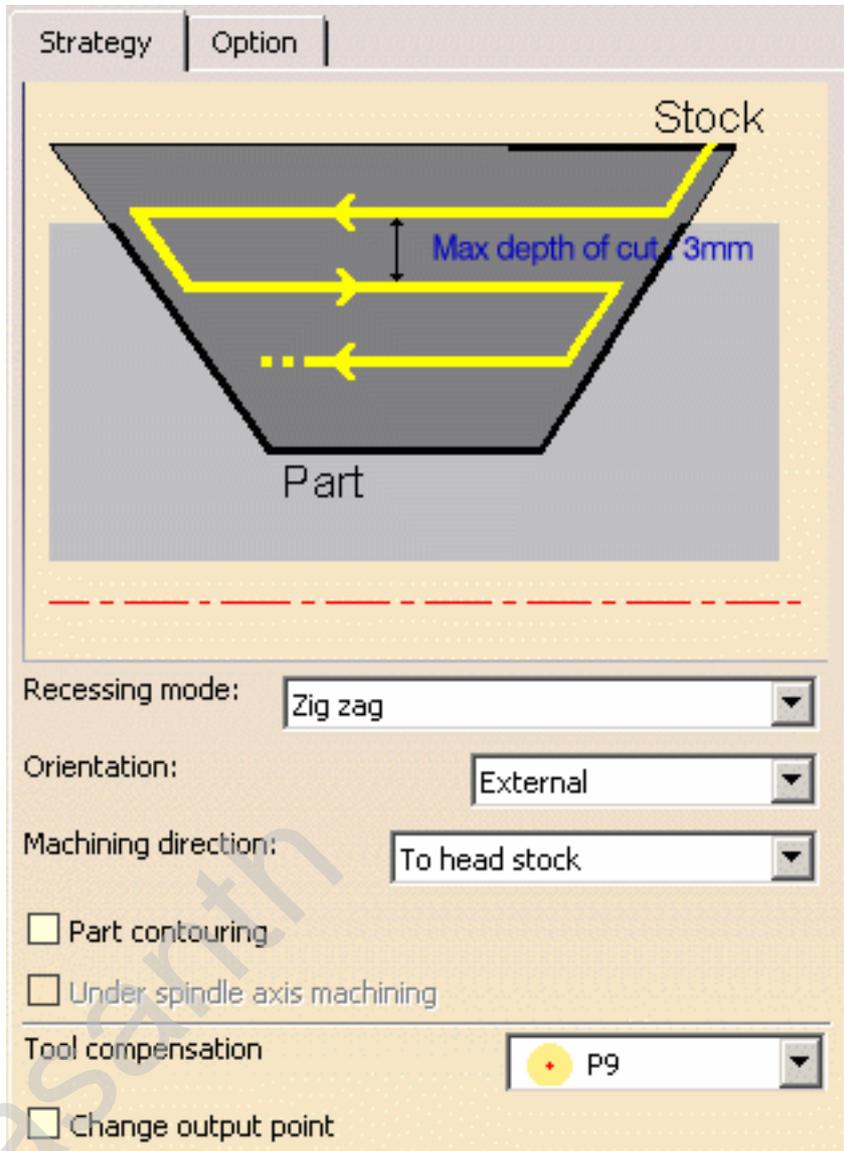


 The part and stock in the icon are colored red indicating that this geometry is required.

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.

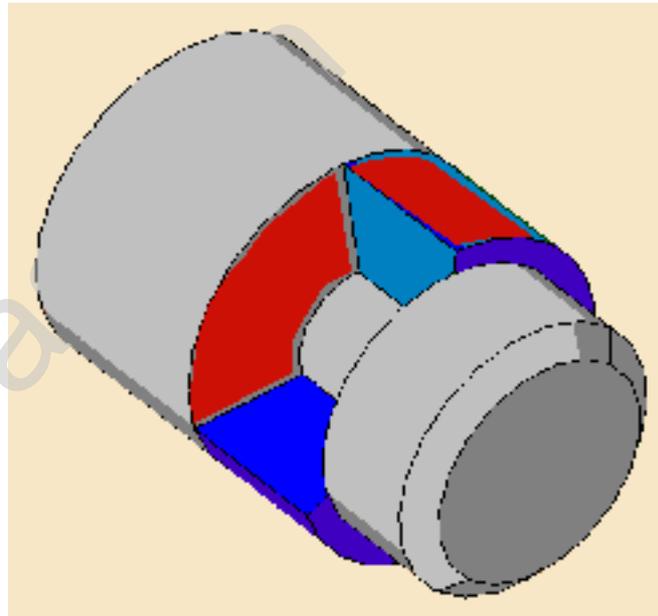
-  **1.** Select the Groove Turning icon 

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.

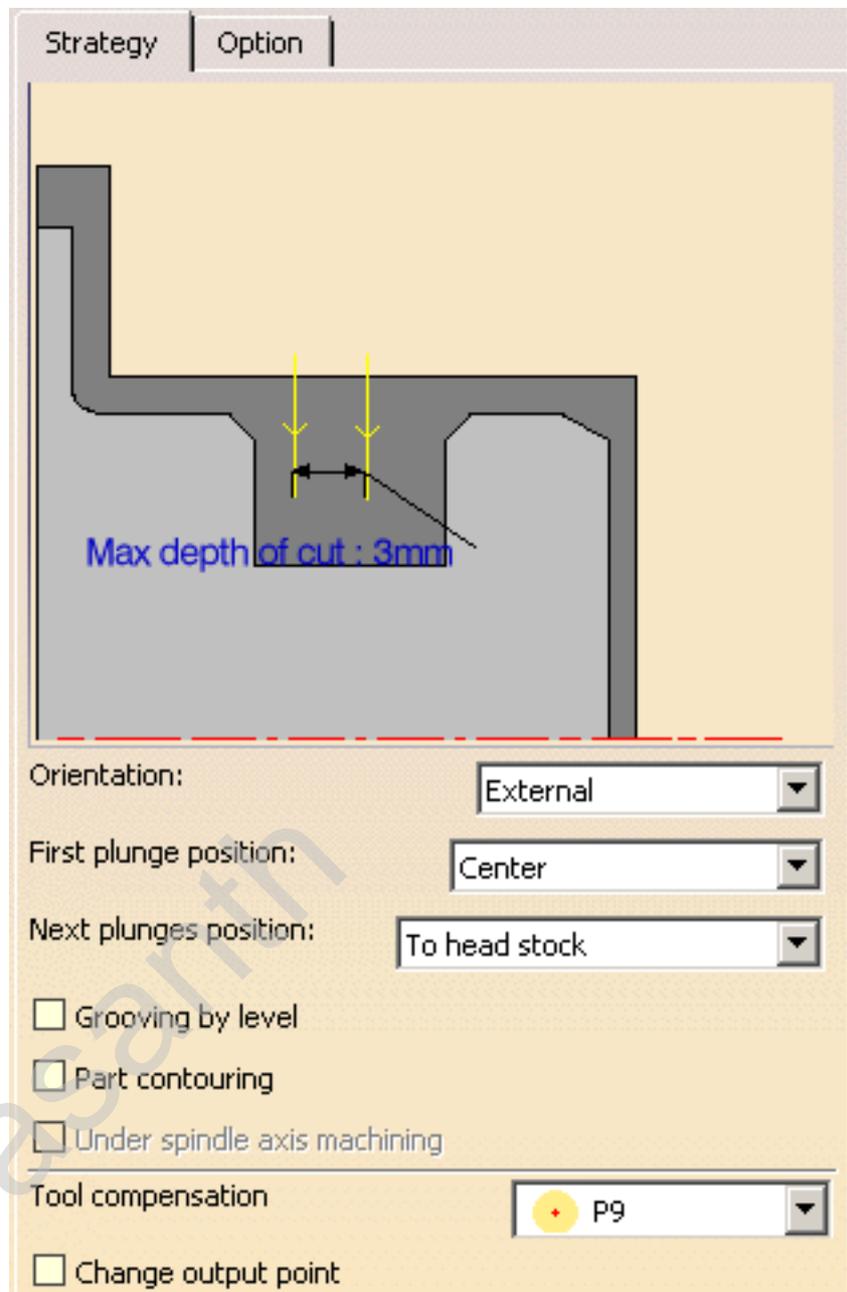
3. Select the Strategy tab page  to specify the main machining strategy parameters:

- 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.



Prasanth

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 

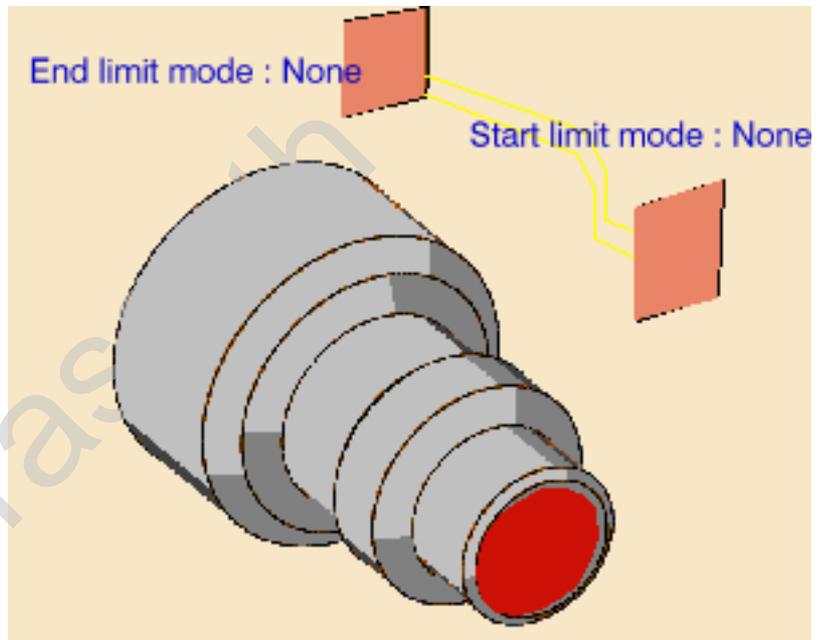
The Profile Finish Turning dialog box appears directly at the Geometry tab page 

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.

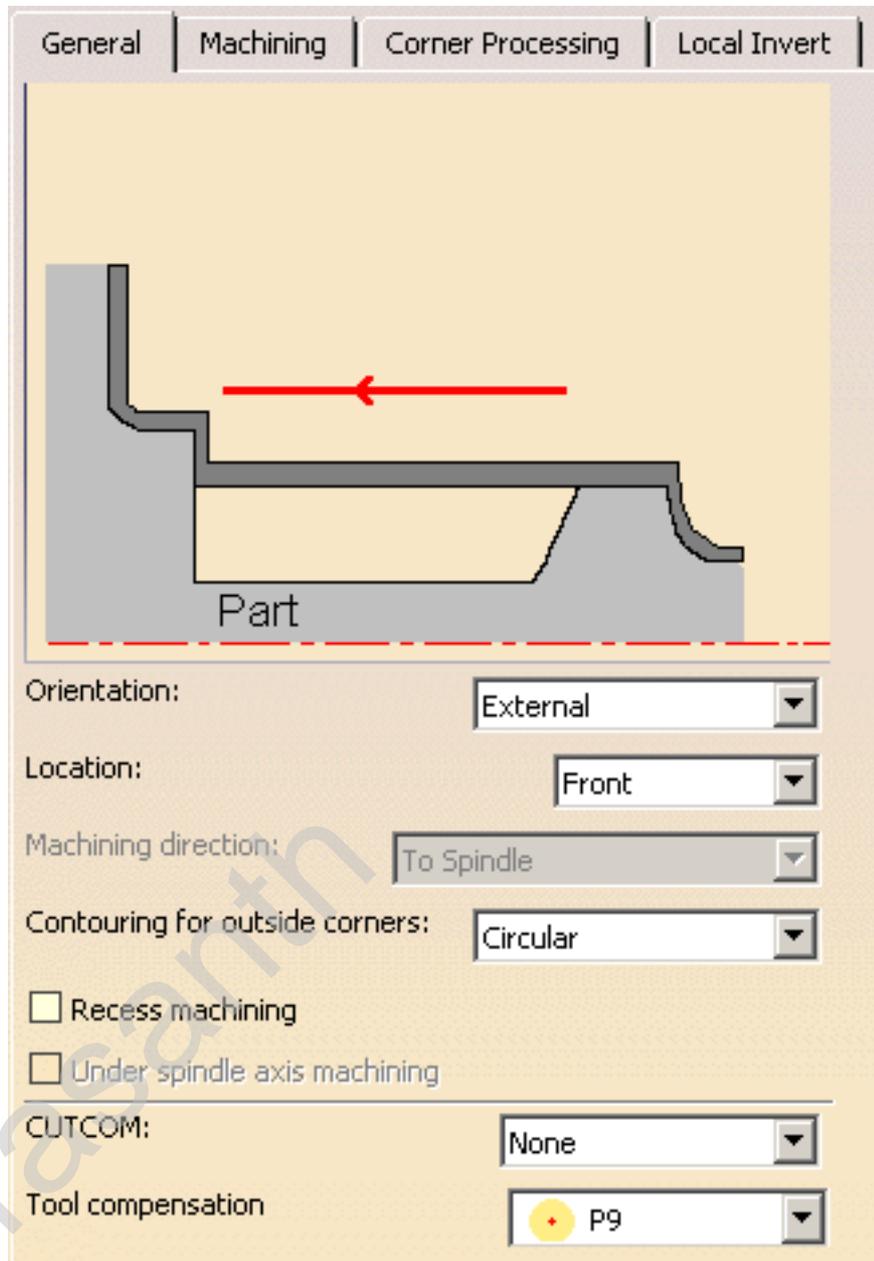
4. Select the Strategy tab page



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.



Prasanth

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.



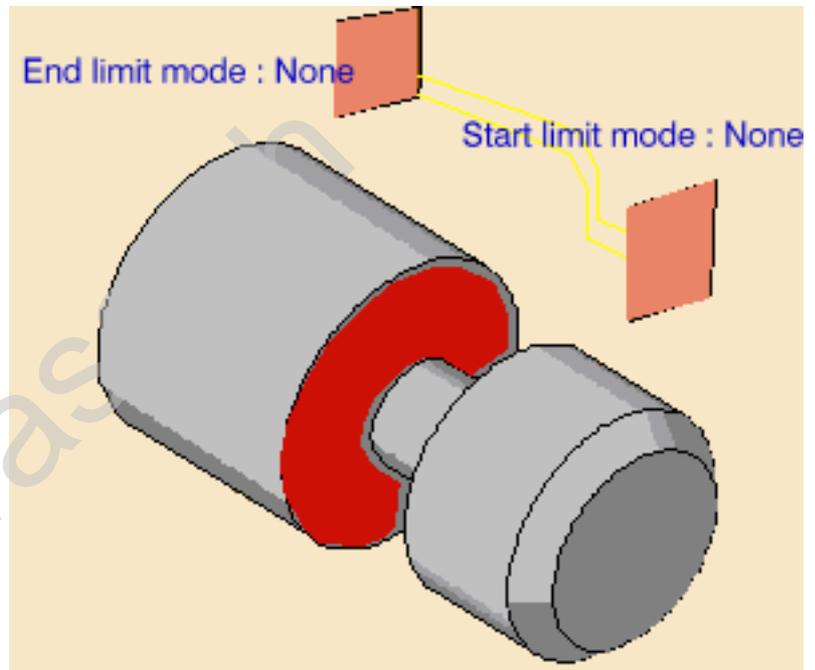
The part in the icon is colored red indicating that this geometry is required.

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.

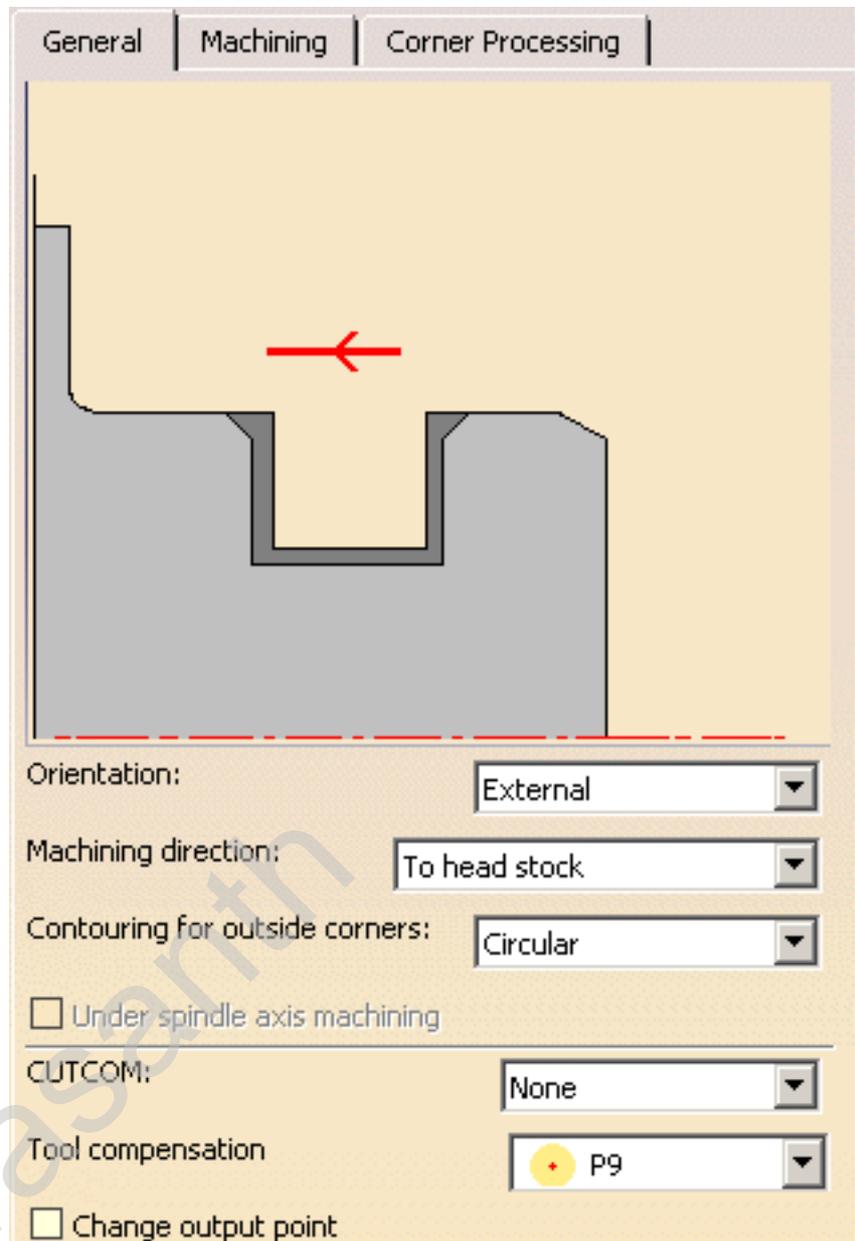


3. Select the Strategy tab page



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.



Prasanth

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)** 



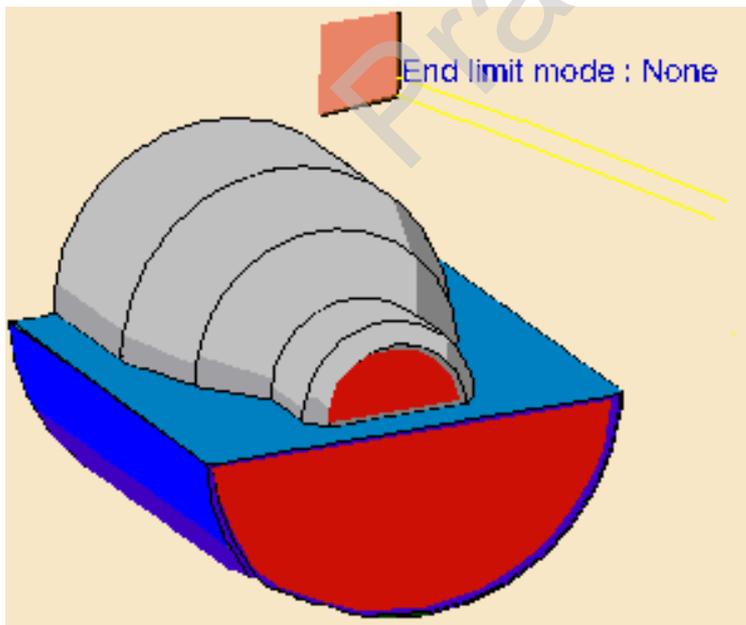
Open the **Lathe01.CATPart** document, then select **Machining > Lathe Machining** from the Start menu. Make the Manufacturing Program current in the specification tree.



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.



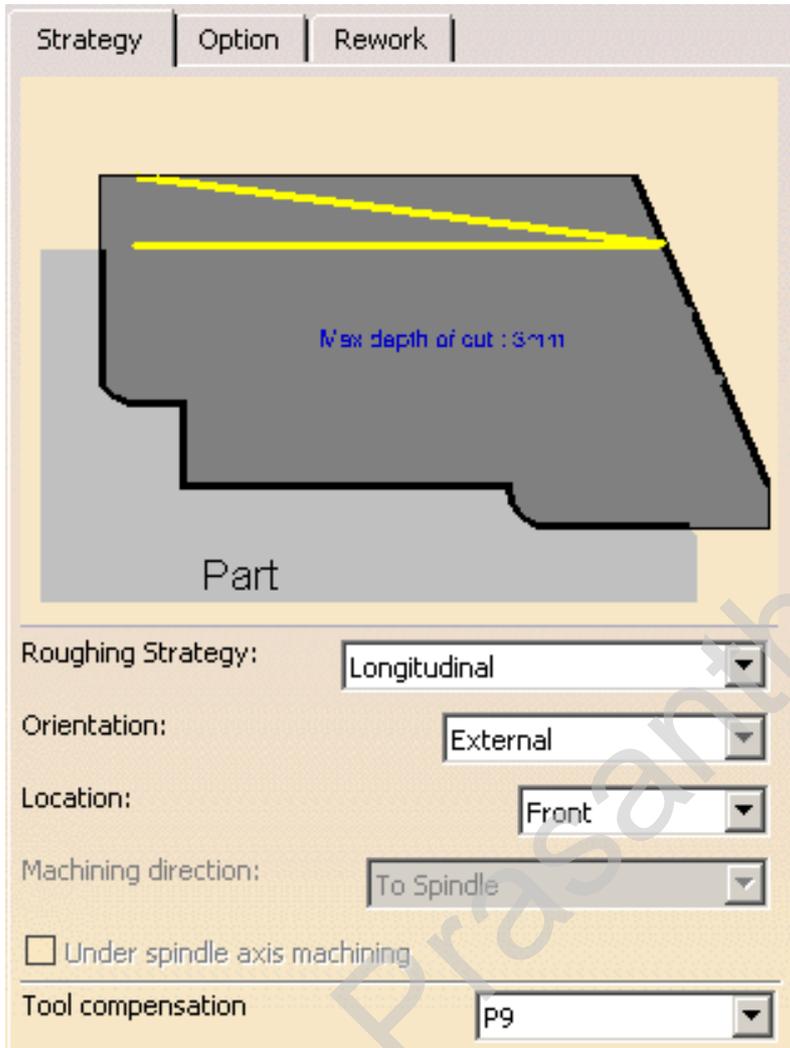
The Part and Stock areas of the icon are colored red indicating that this geometry is required. All other geometry is optional.

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.

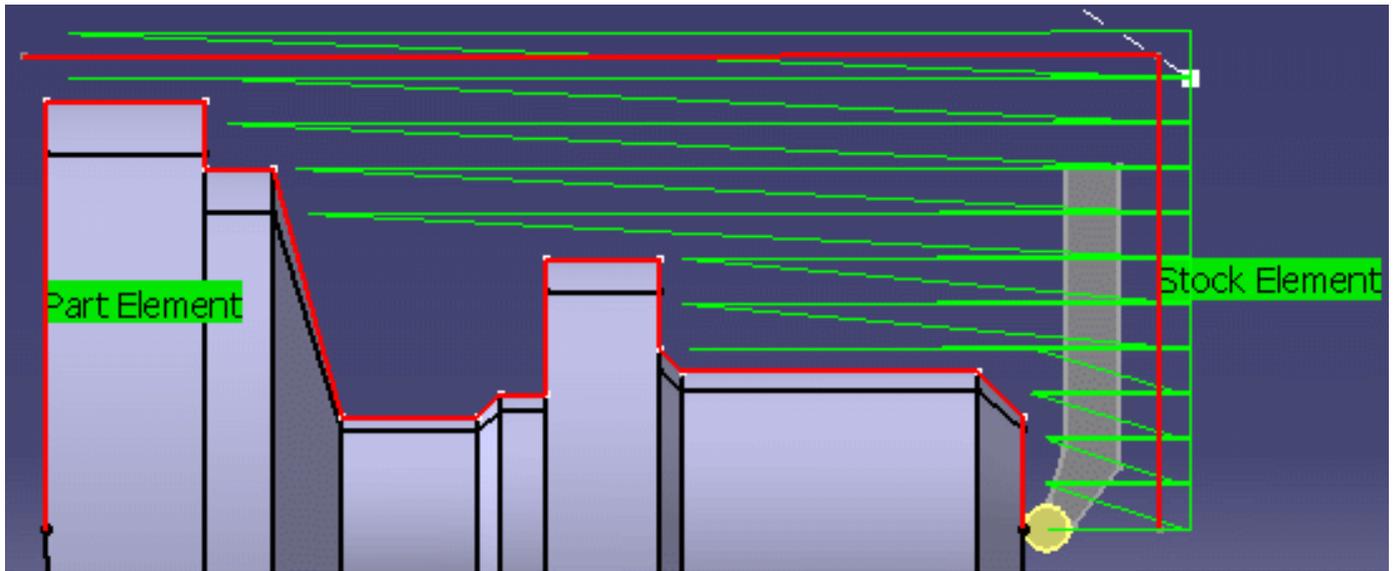
See [Feeds and Speeds for Ramp Rough Turning](#) for more information.

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.



Prasanth

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)** 



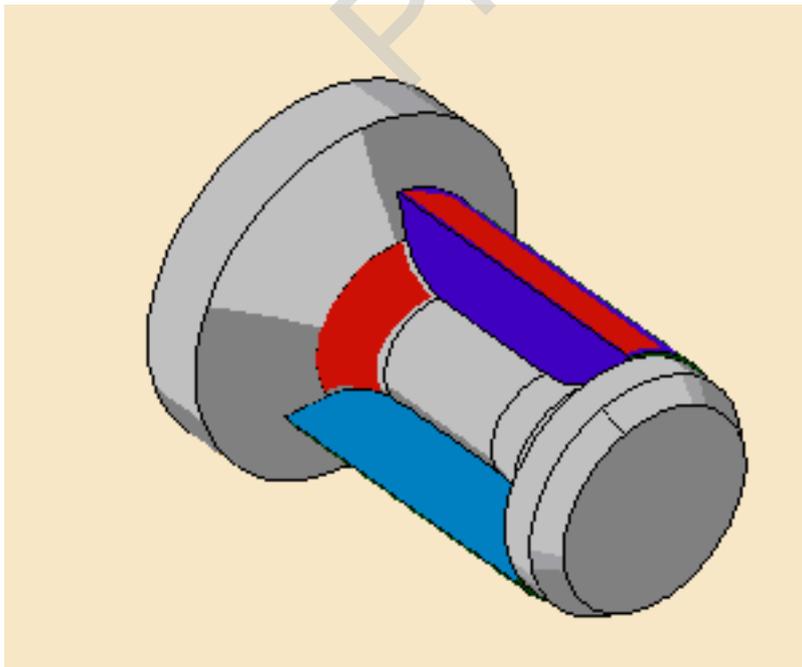
Open the **Lathe01.CATPart** document, then select **Machining > Lathe Machining** from the Start menu. Make the Manufacturing Program current in the specification tree.



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.

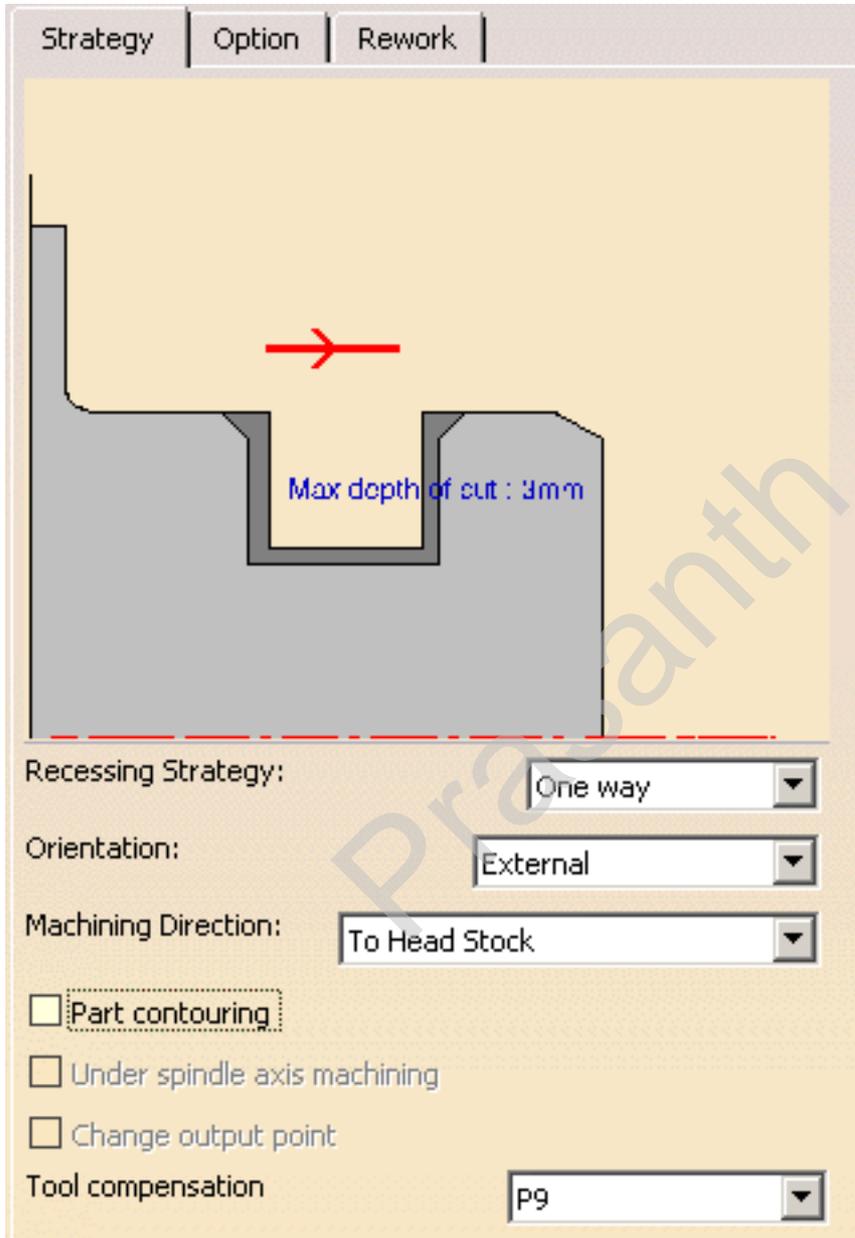
2. Click the red Part area in the icon then select the desired part profile in the 3D window.

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](#).

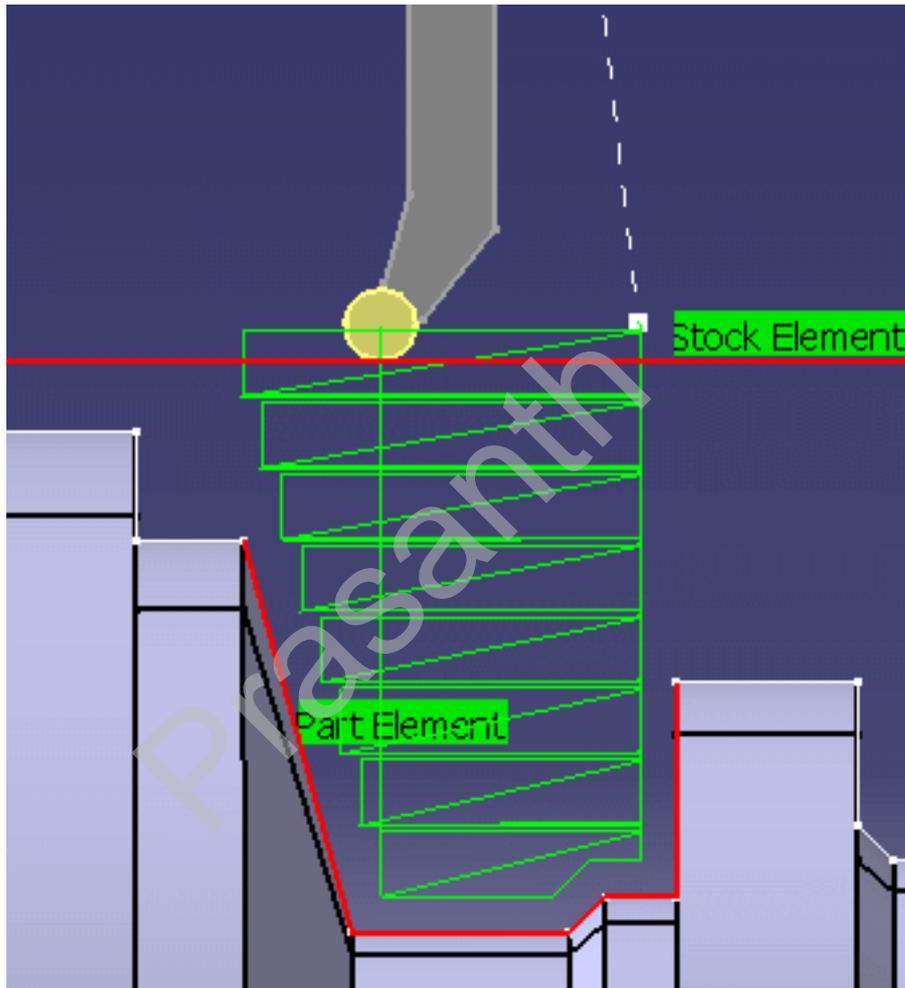
5. Select the Feeds and Speeds tab page  to specify the feedrates and spindle speeds for the operation.

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.

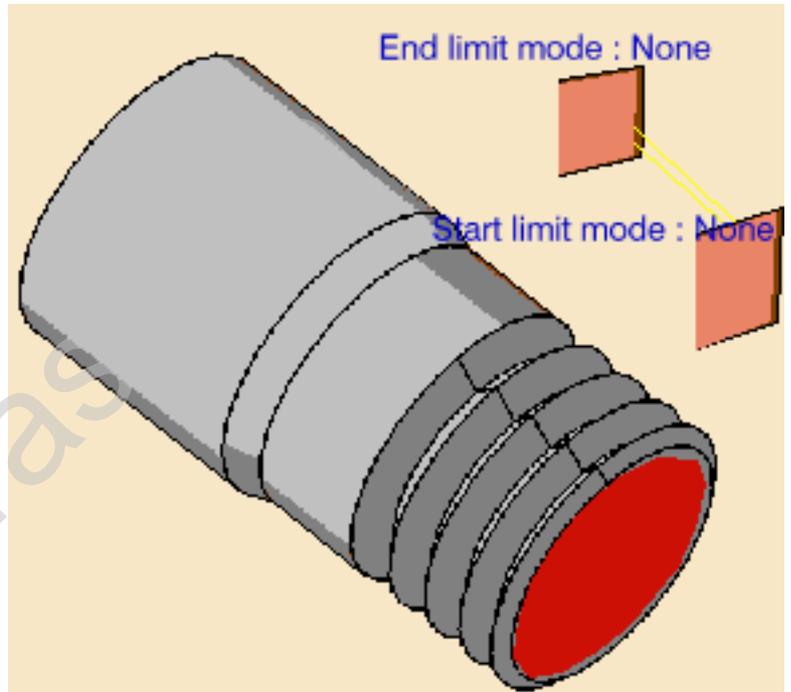
The Thread Turning dialog box appears directly at the Geometry tab page 

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.

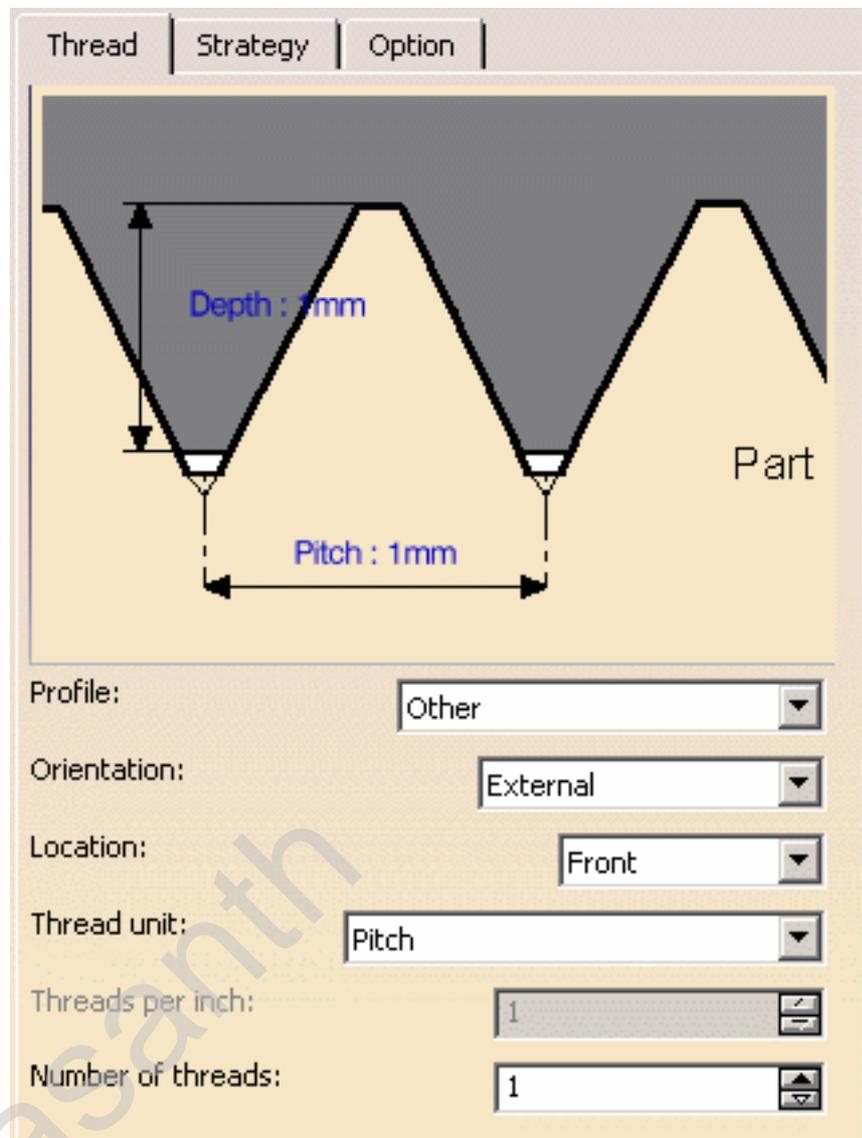


4. Select the Strategy tab page  to specify the main machining parameters that are organized in three tabs: Thread, Strategy and Options.

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](#).



Prasanth

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 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 .

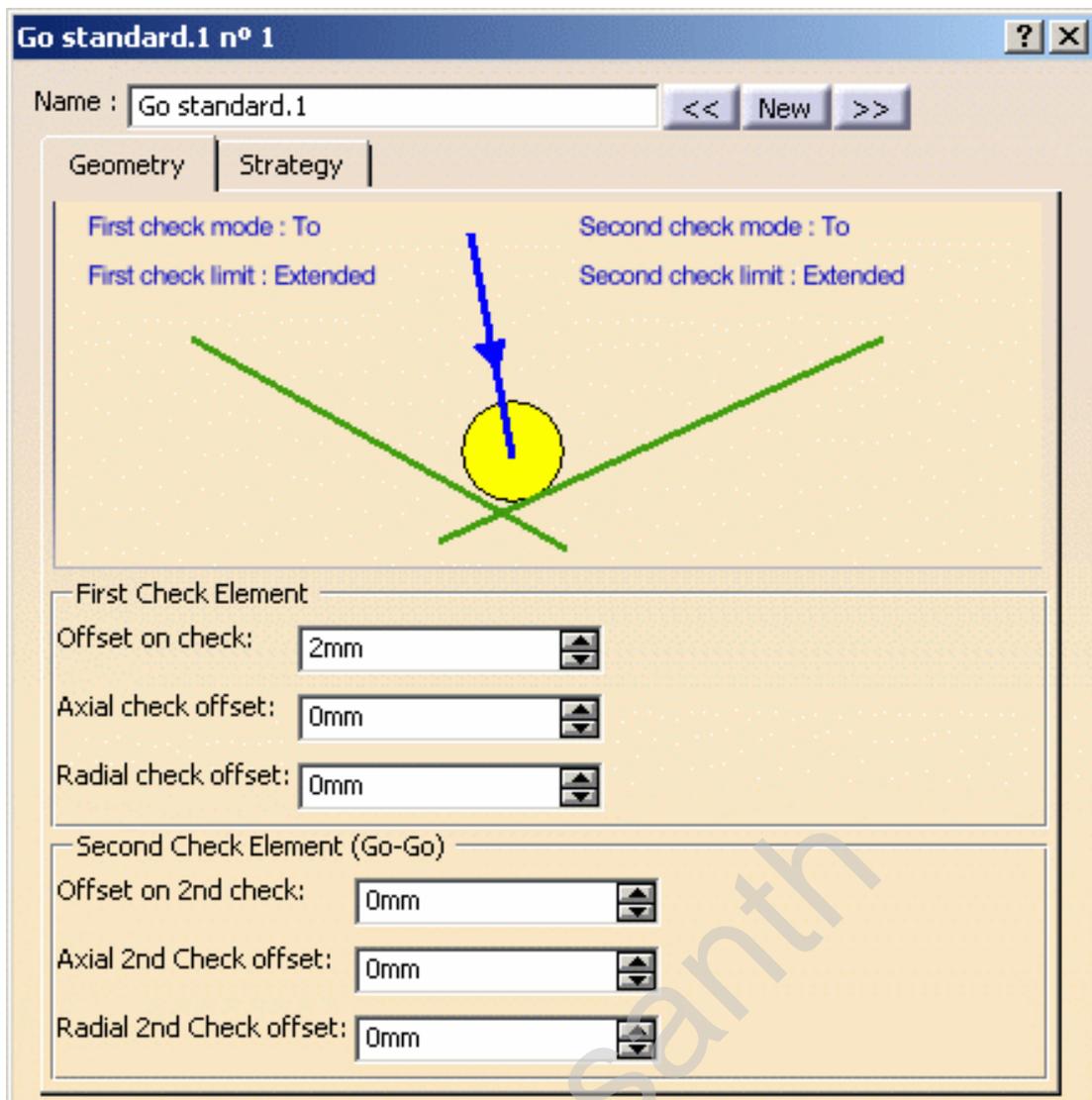


The first motion must be a Go motion to a point or a Go motion with two check elements.

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

In the Geometry tab:

- Successively select the two check elements 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.



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

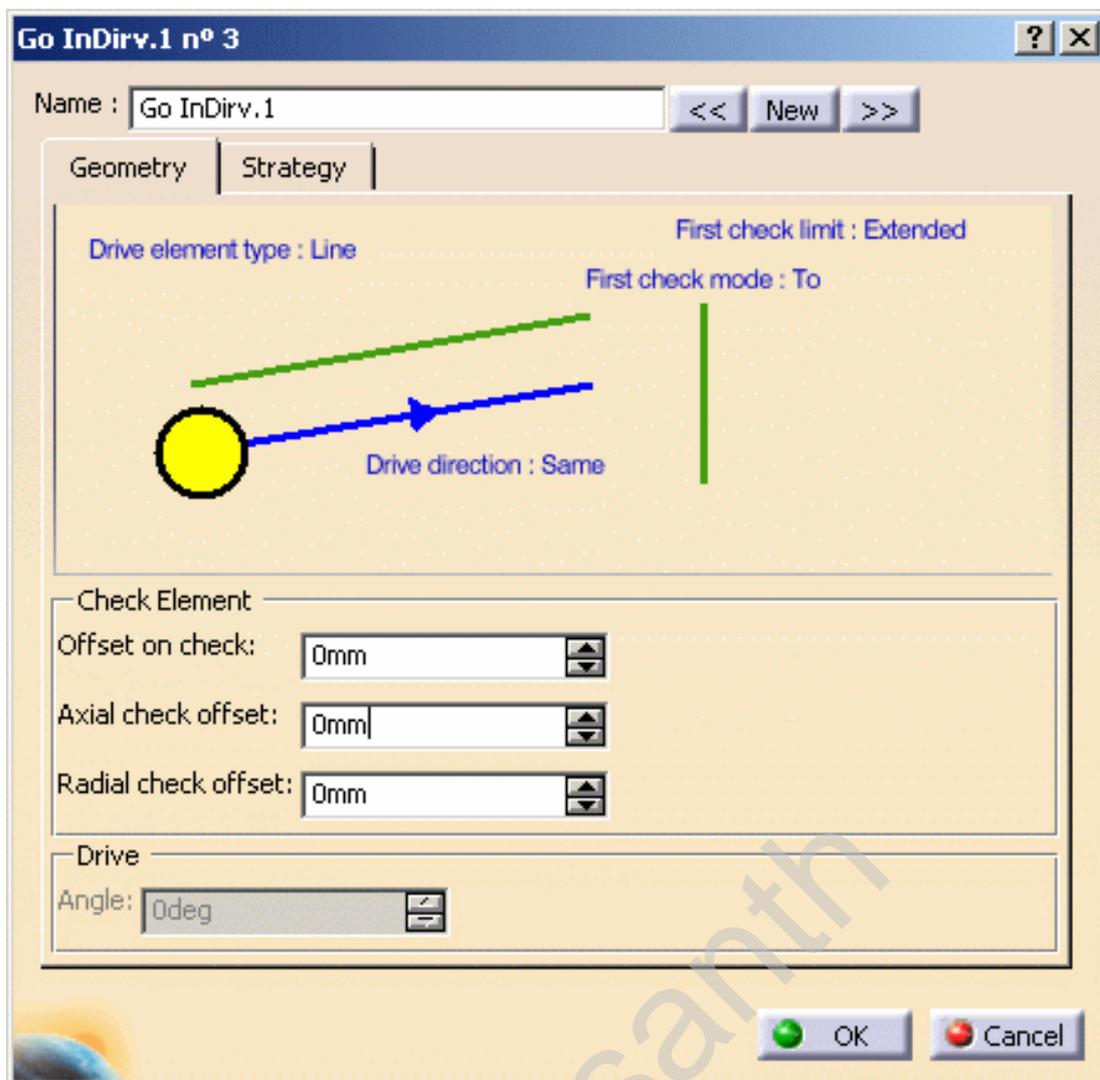
In the Geometry tab:

- Select a check element 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.

4. Select the Go InDirv icon .

In the Geometry tab:

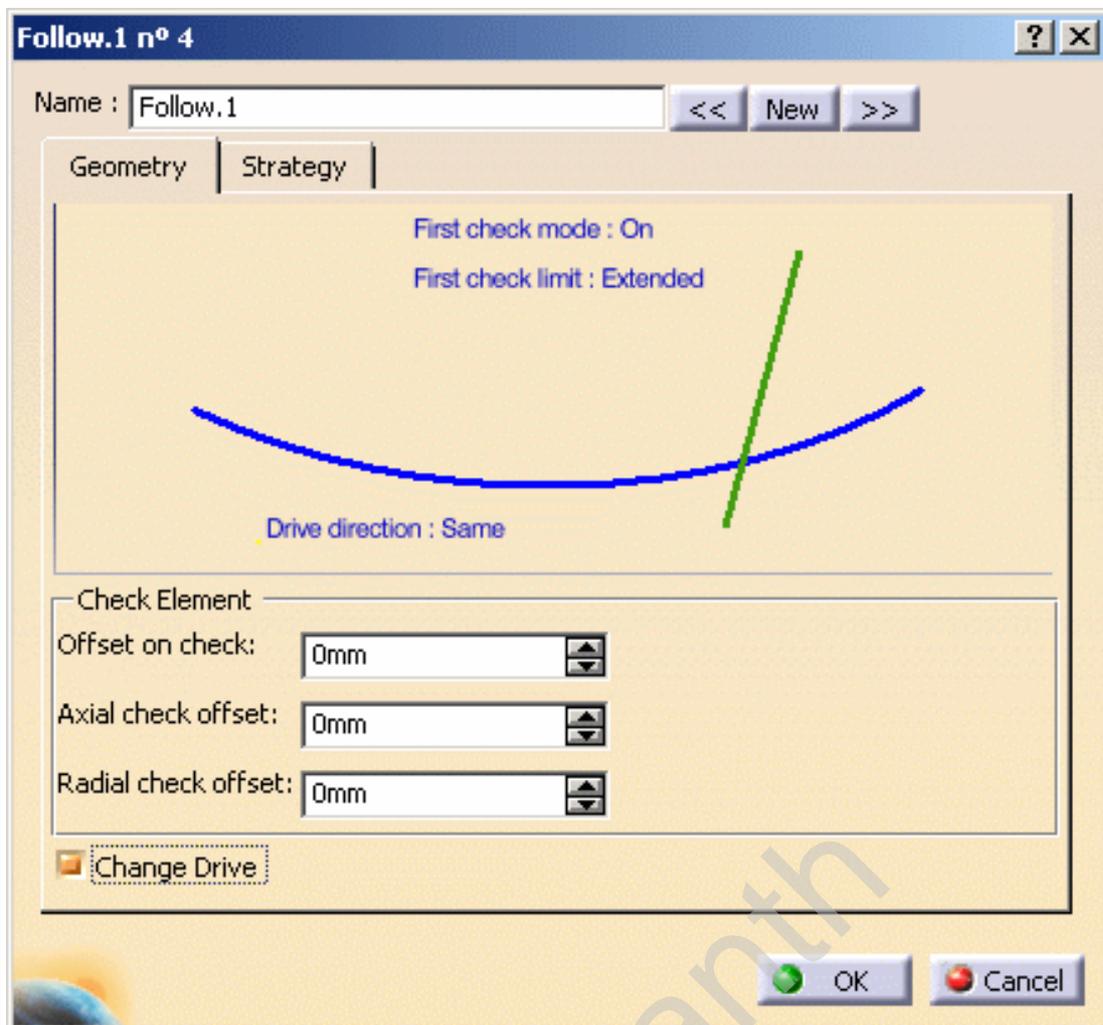
- Select the drive direction in the dialog box then the desired linear element in the 3D view.
- Select a check element in the dialog box then the desired check element in the 3D view.
- Set the check mode to **To**.
- Click OK to define the motion.



5. 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:

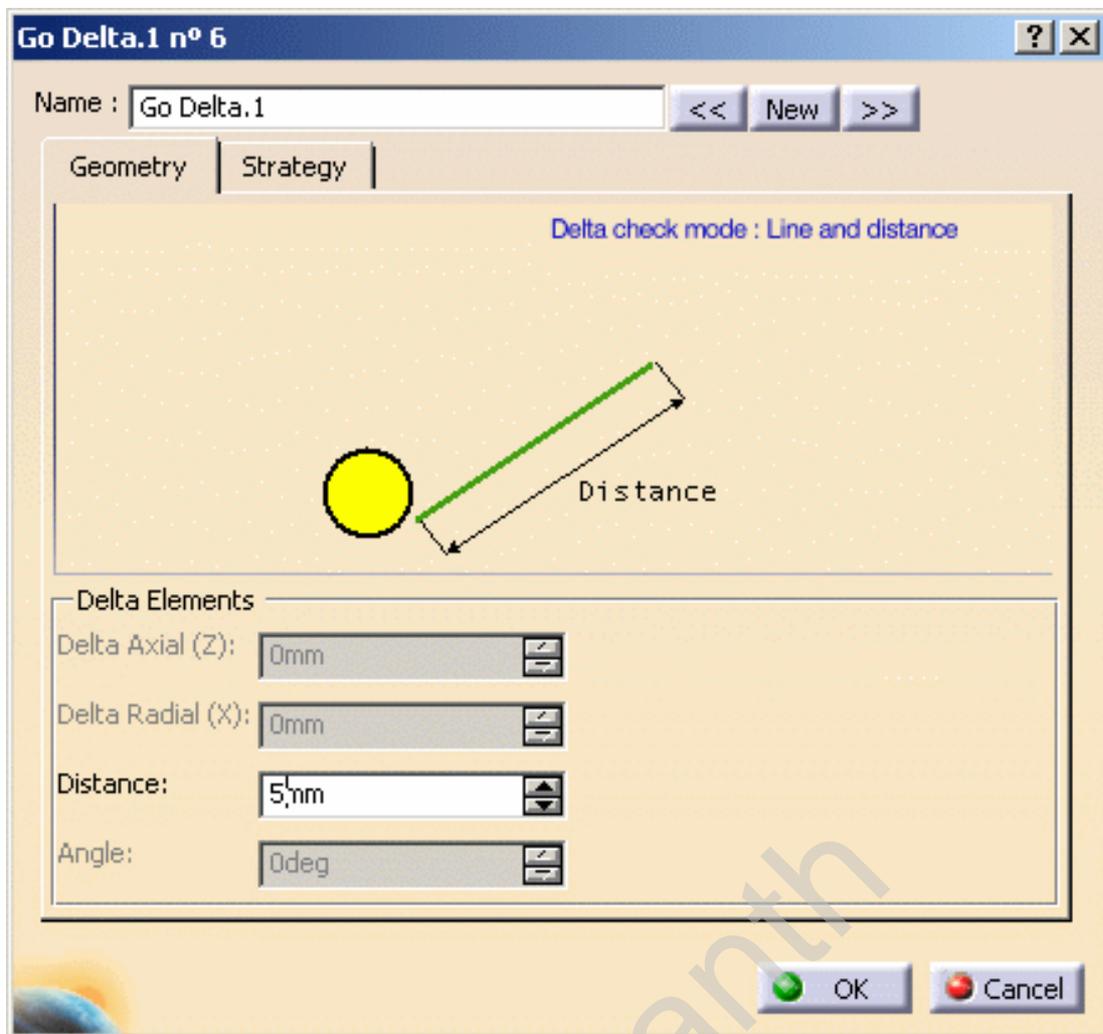
- Select a check element in the dialog box then the desired check element in the 3D view.
- Set the check mode to **On**.
- Click OK to define the motion.



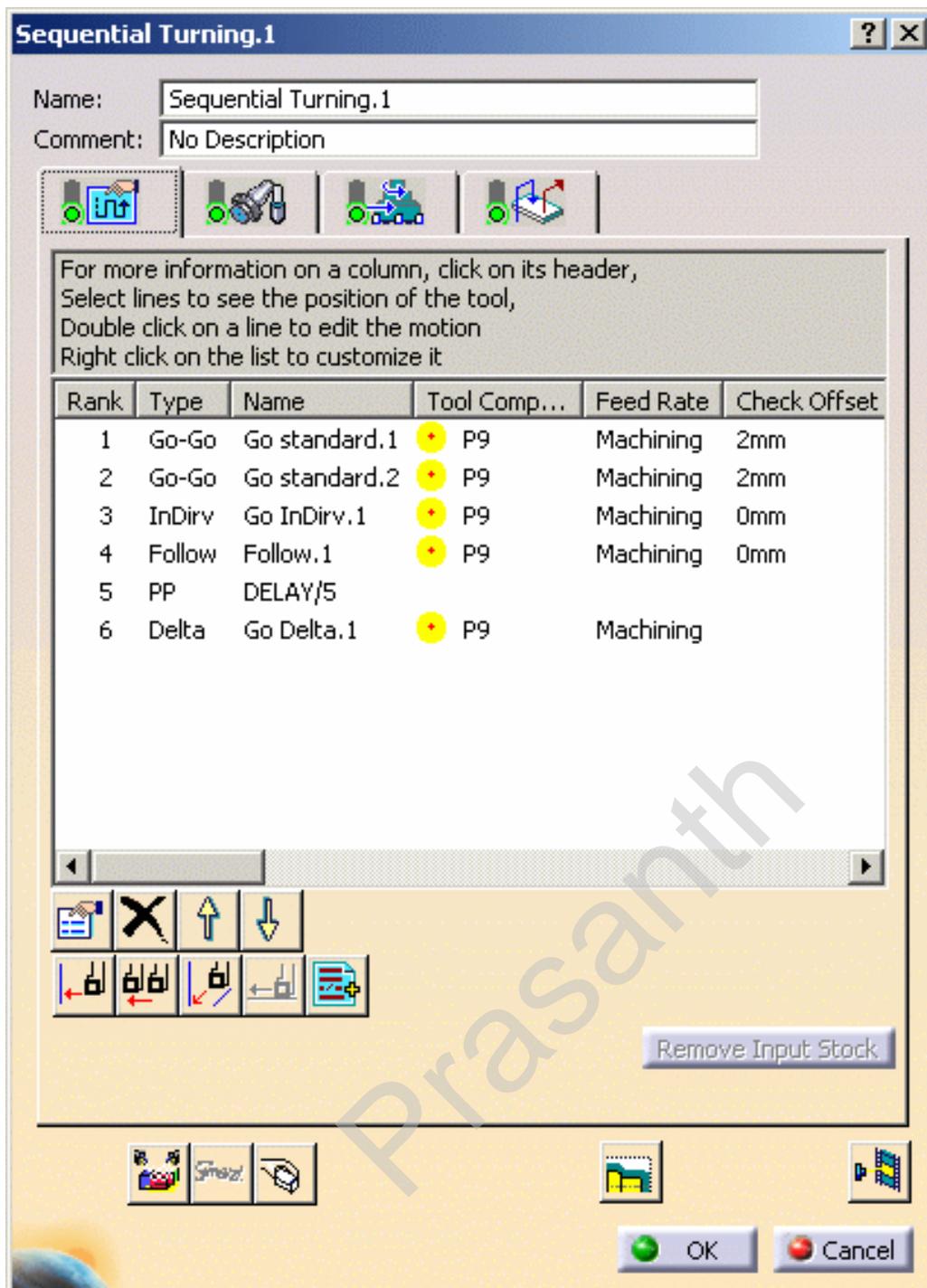
6. Select the PP word icon , then specify a PP word in the dialog box that appears (DELAY/5, for example).
7. Select the Go Delta icon .

In the Geometry tab:

- Set the check mode to **Line and distance**.
- Select the delta element 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.

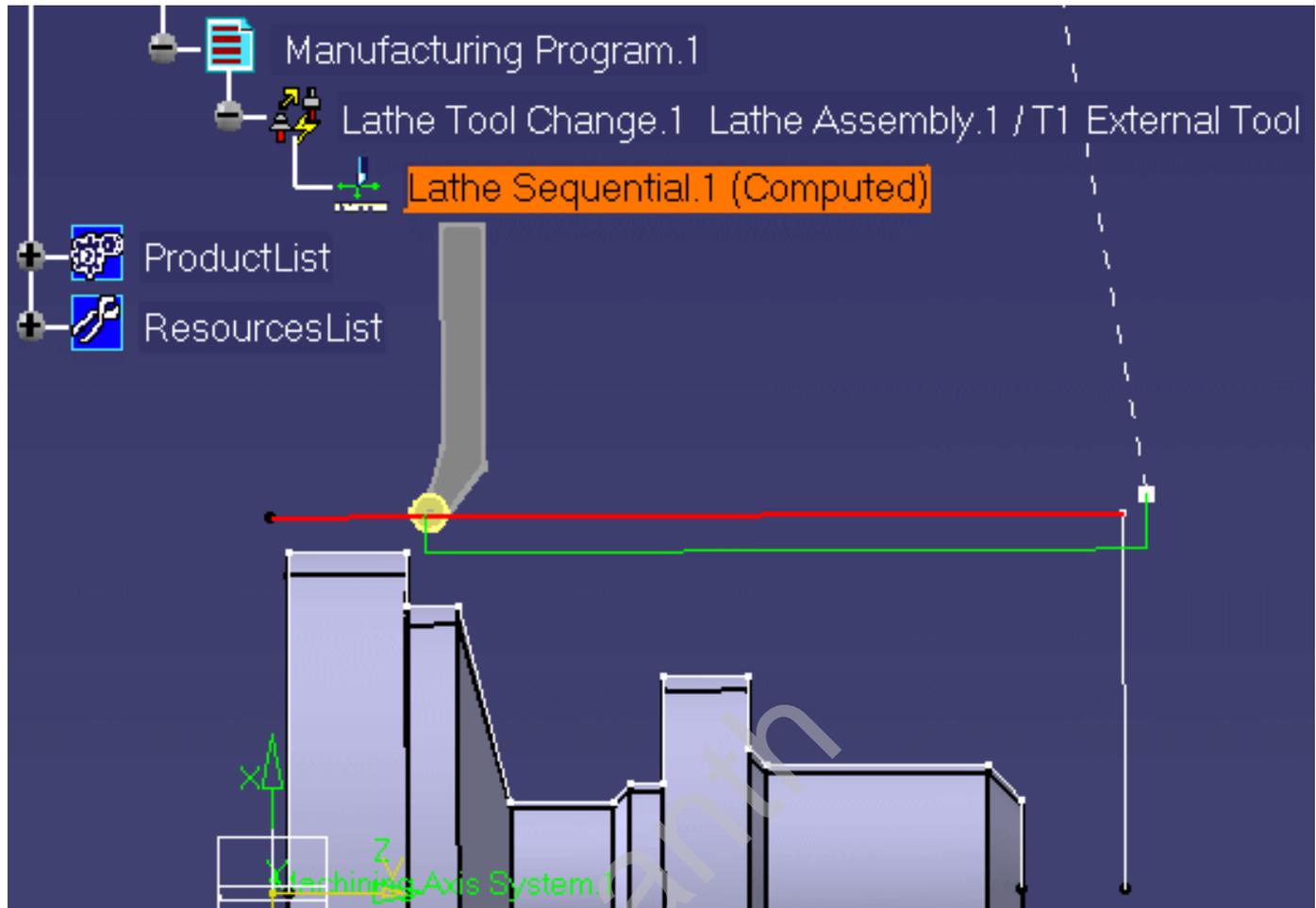


A tool is proposed by default when you want to create a machining operation. If the proposed tool is not suitable, just select the Tool tab page  to specify the tool you want to use. Please refer to [Edit the Tool of a Lathe Operation](#).

8. 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.
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 an example.



Before accepting the operation, you should check its validity by [replaying the tool path](#).

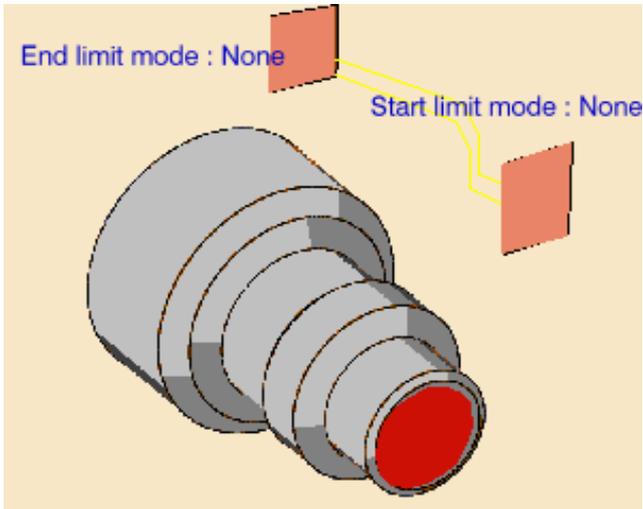


10. Click OK to create the operation.

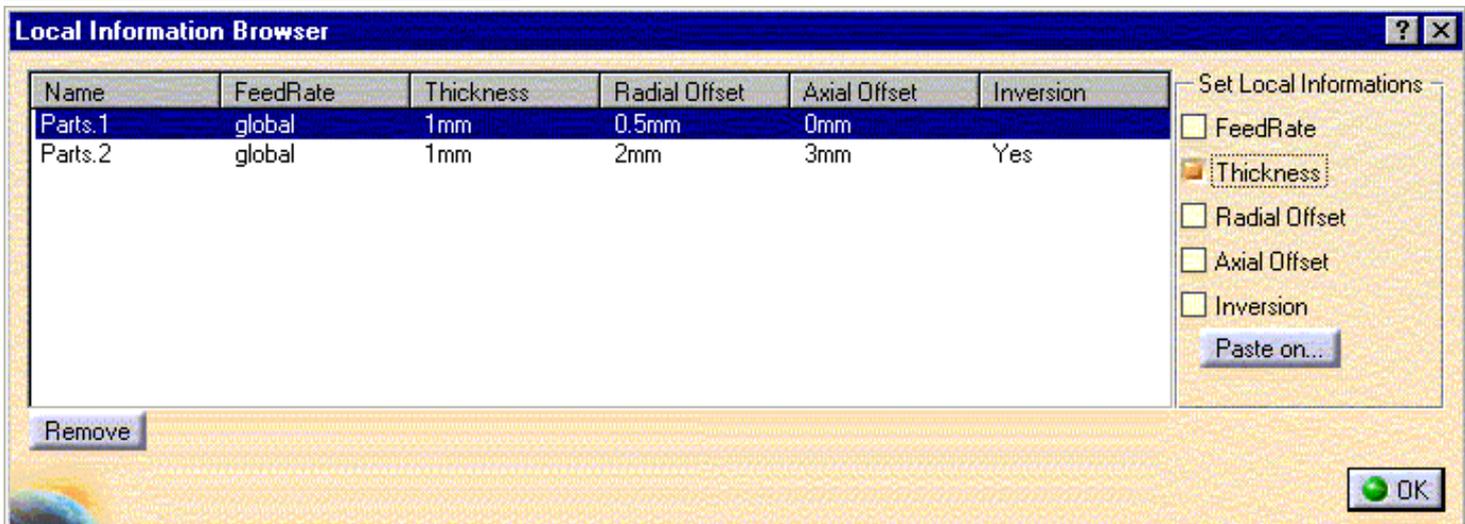
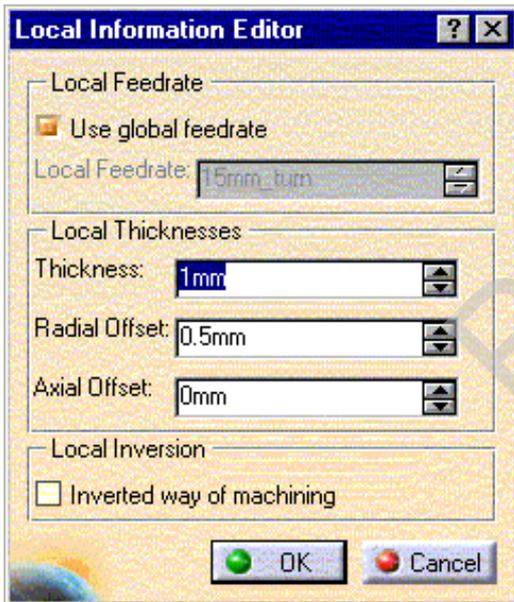
Manage Local Information

 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.

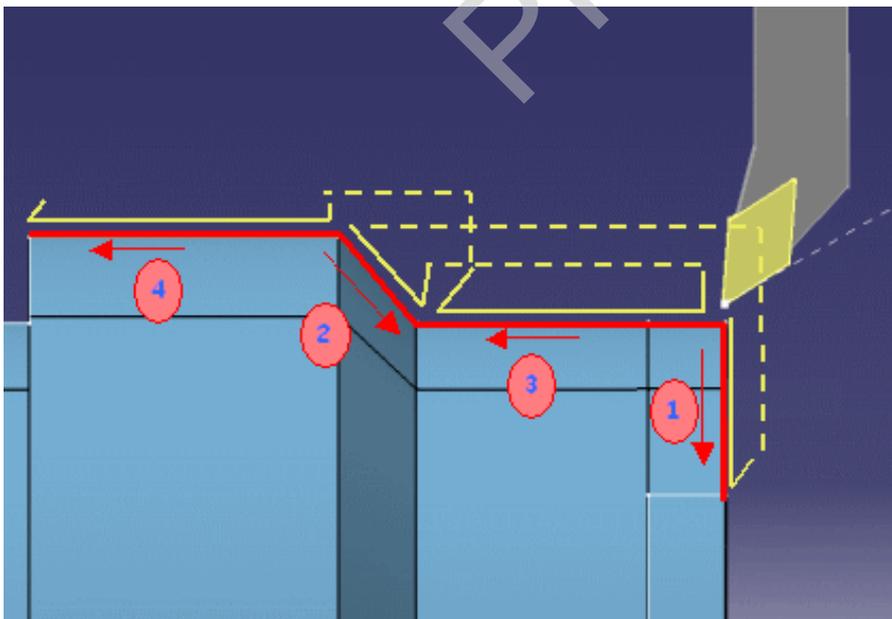
To remove local information

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.

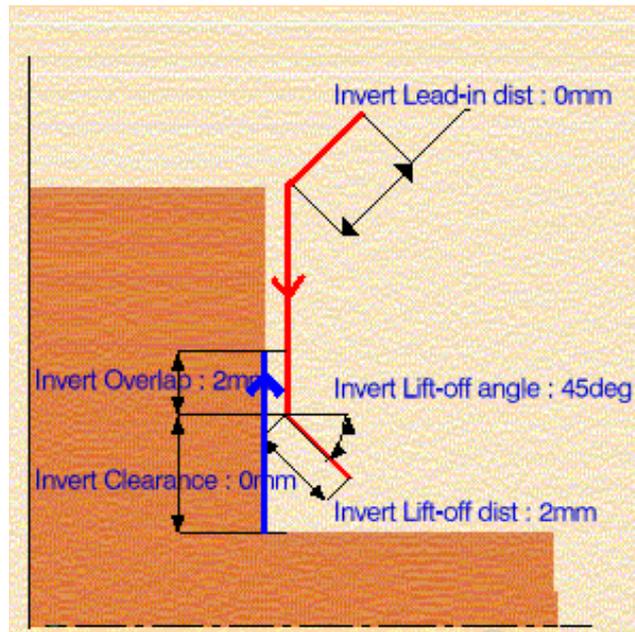
To machine part with inverted elements, several strategies are available:

- **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.



Prasanth

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.

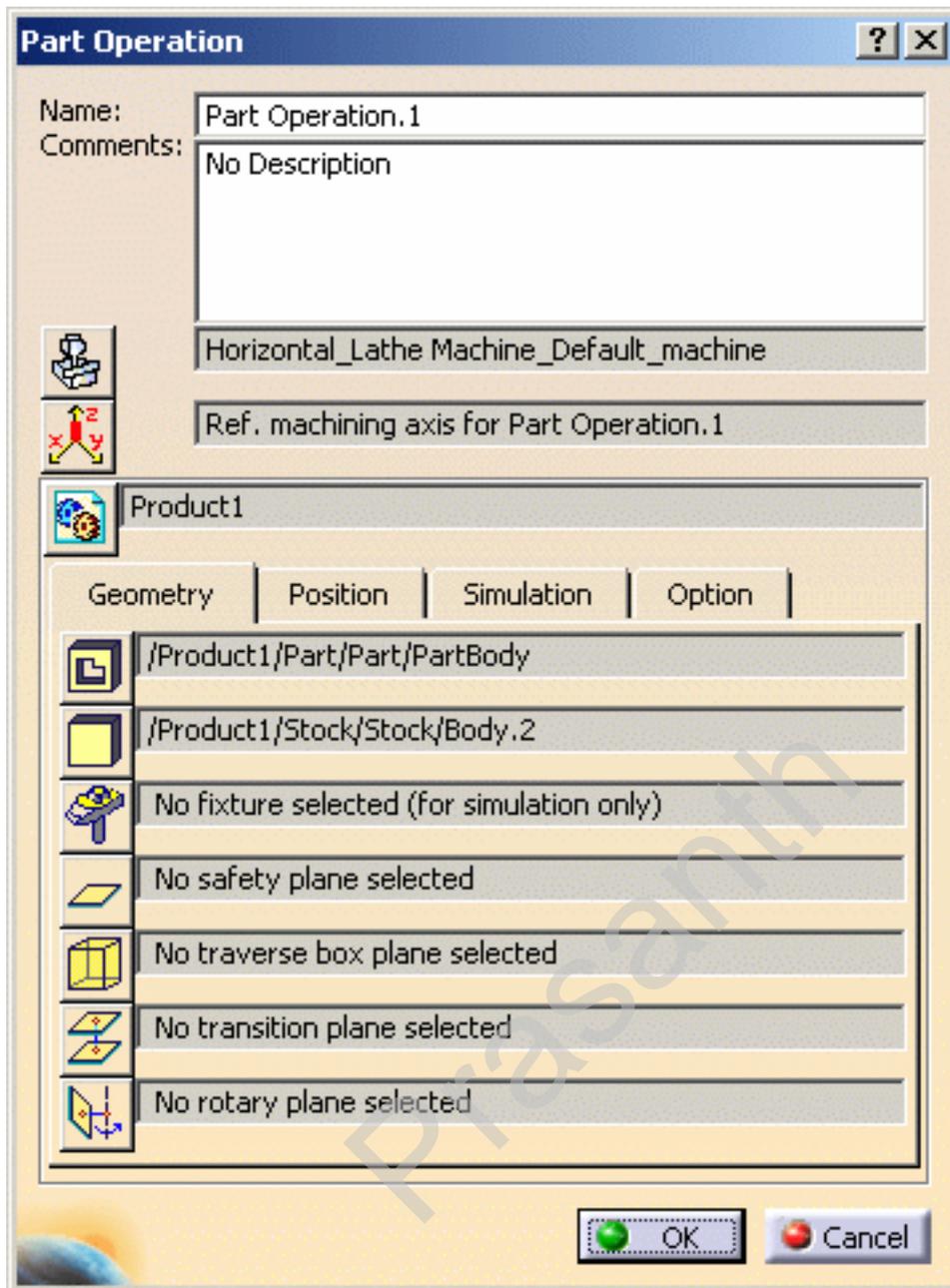


1. Select **File > Open** then select the [StartStockUpdateManualMode.CATProcess](#) document
2. Select **Machining > Lathe Machining** from the **Start** menu.

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.

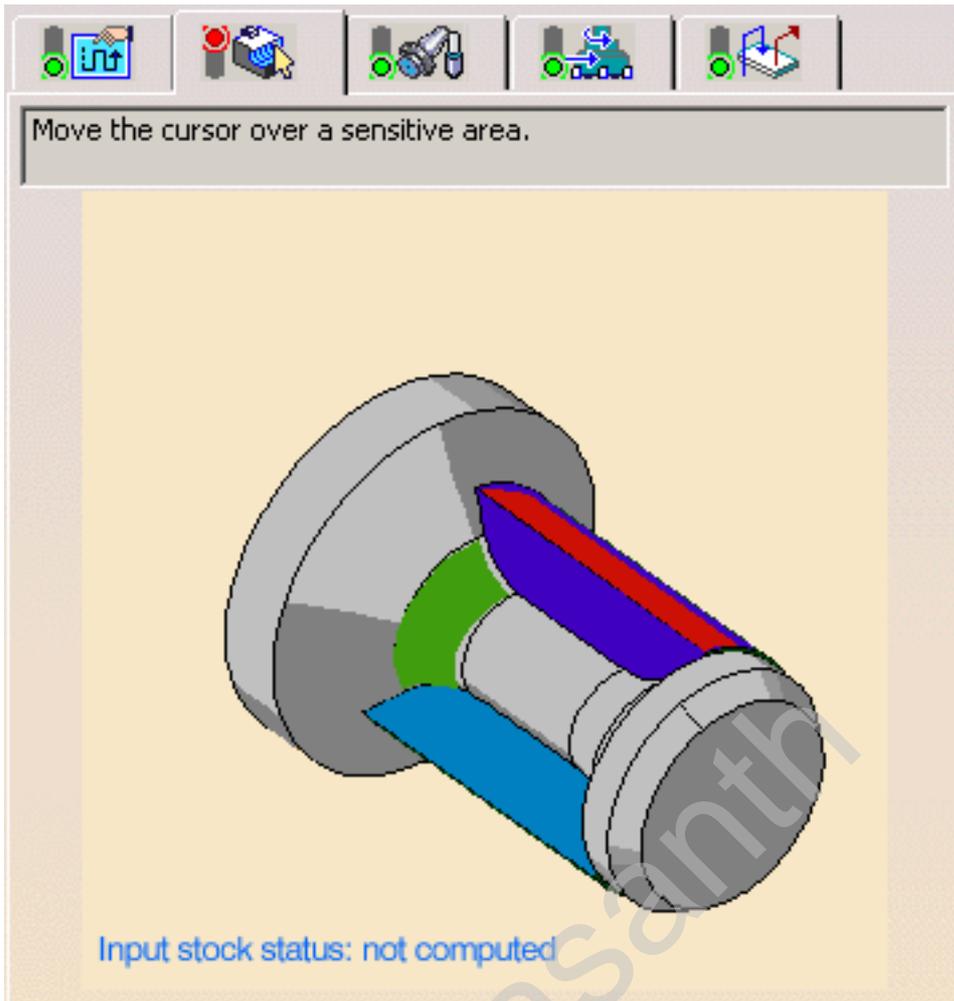


Right-click the Stock field to make sure that **Automatic Stock Selection for Turning Operations** is not activated in the contextual menu.

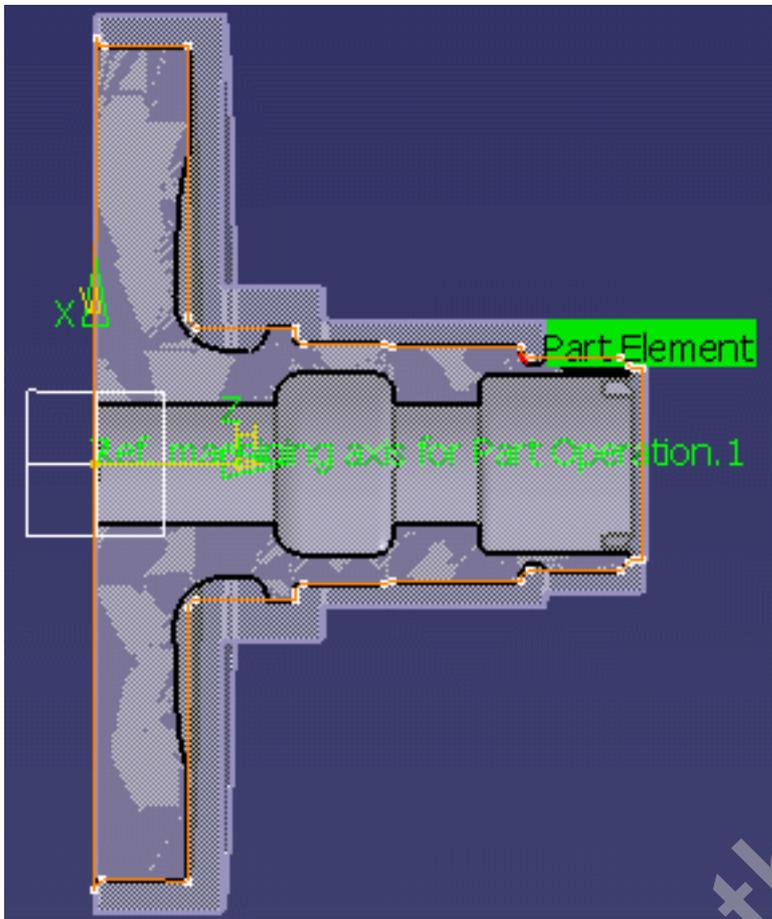


4. Double click the second lathe operation in the program: **External lathe recessing**.

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

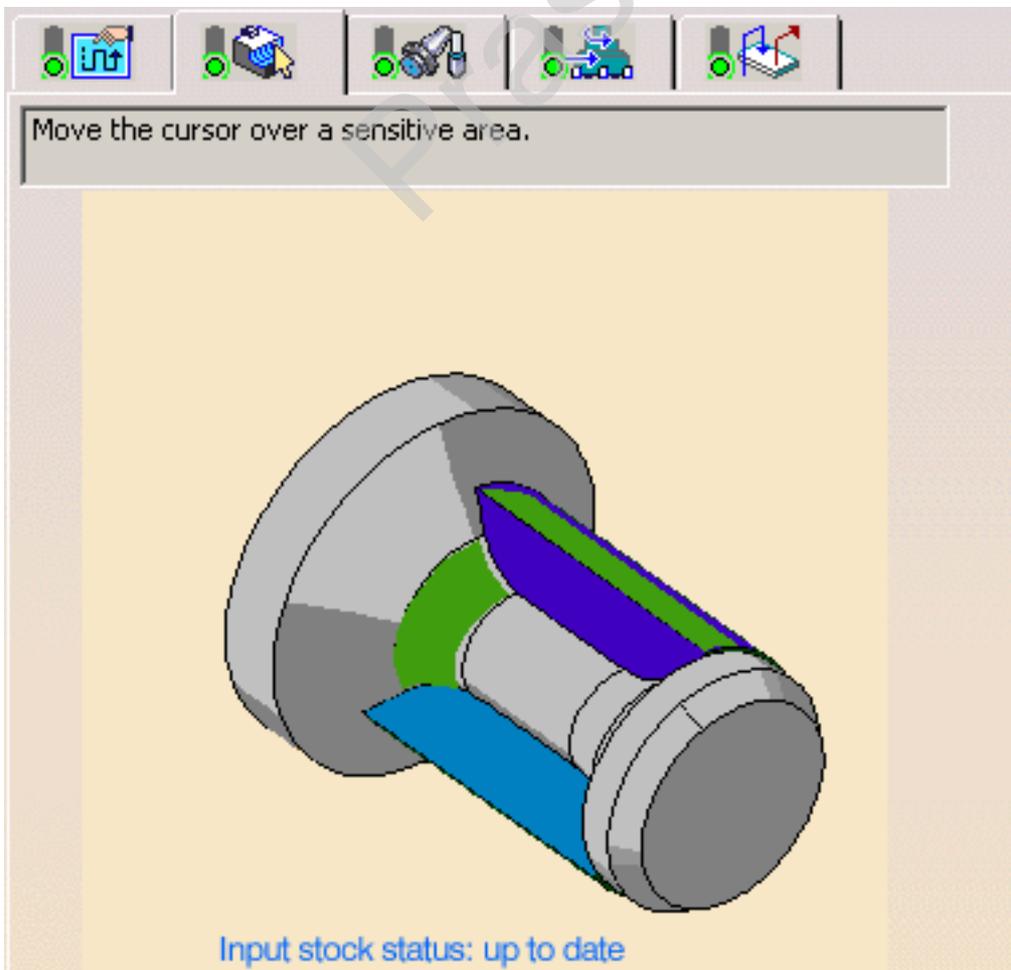


6. Select the **Update Input Stock** command by either clicking the icon  or right-clicking the **Input stock status** field in the sensitive icon. The stock is computed and the corresponding sketch profile is displayed in the viewer.



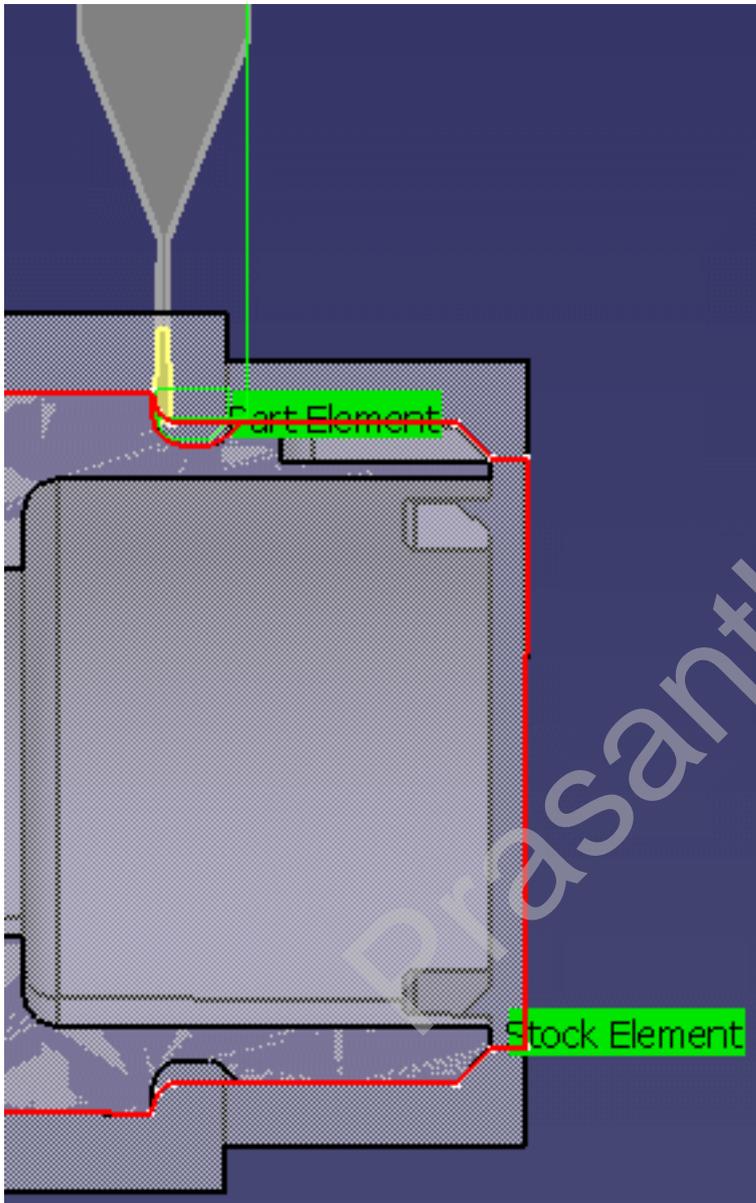
7. Select this profile as the input stock for the operation.

The Geometry page is updated with this stock



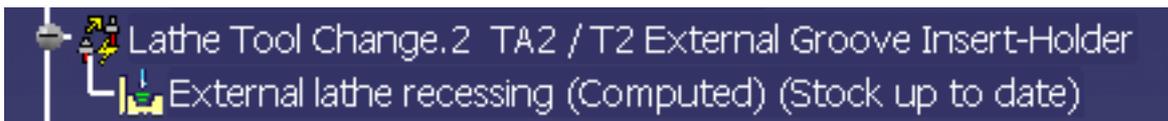
You can use the **Analyze** contextual command (right-click the stock area in the sensitive icon) to check that an input stock has been selected.

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



Click OK to accept the operation.

9. The status of the operation is as follows:

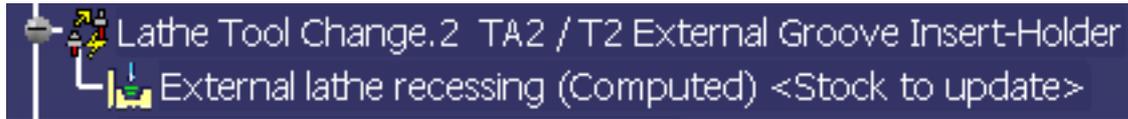


10. The input stock of the other Lathe operations in the program can be updated in the same way.



Updating Computed Stock Status

For modifications to the Part Operation, 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 Program in the tree and select the **Update Computed Stock Status** contextual command. All necessary updates will be done.

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.



Prasanth

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.

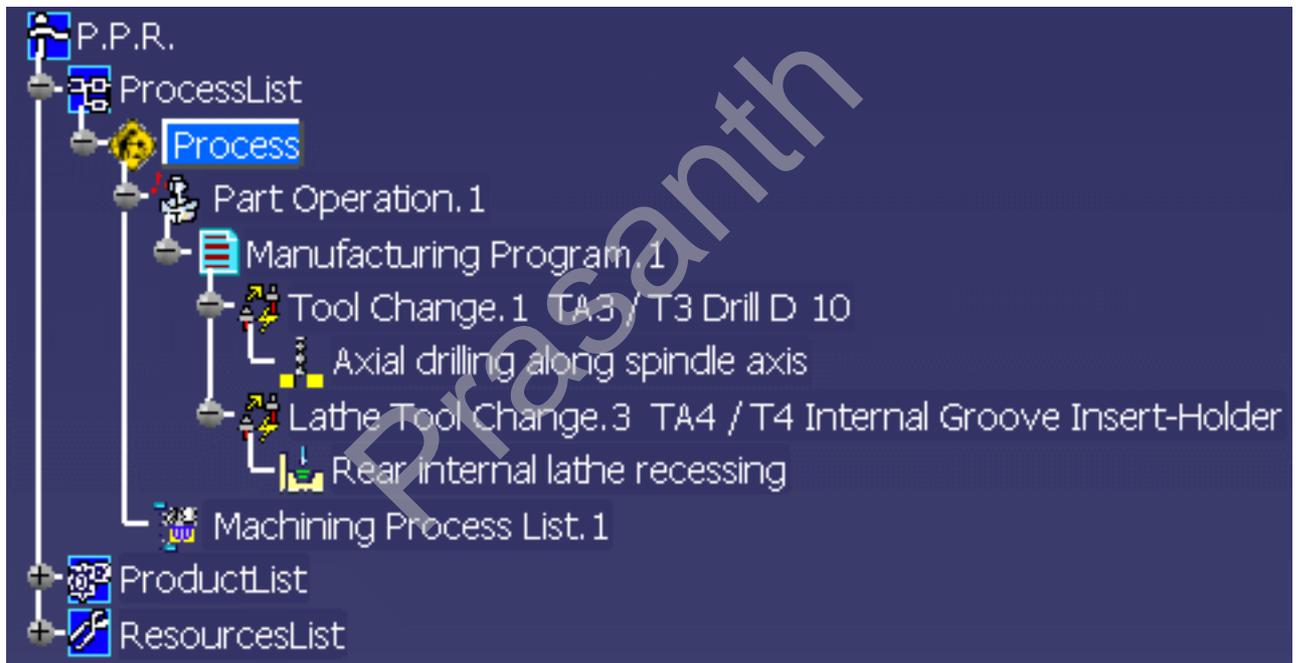
In Automatic mode, the input stock takes into account all the previous operations. The axial operations must be completed and the lathe operations that use a stock must have either a manually selected stock or a saved input stock.

Please refer to [How to Update Input Stock](#) for more information about this capability.

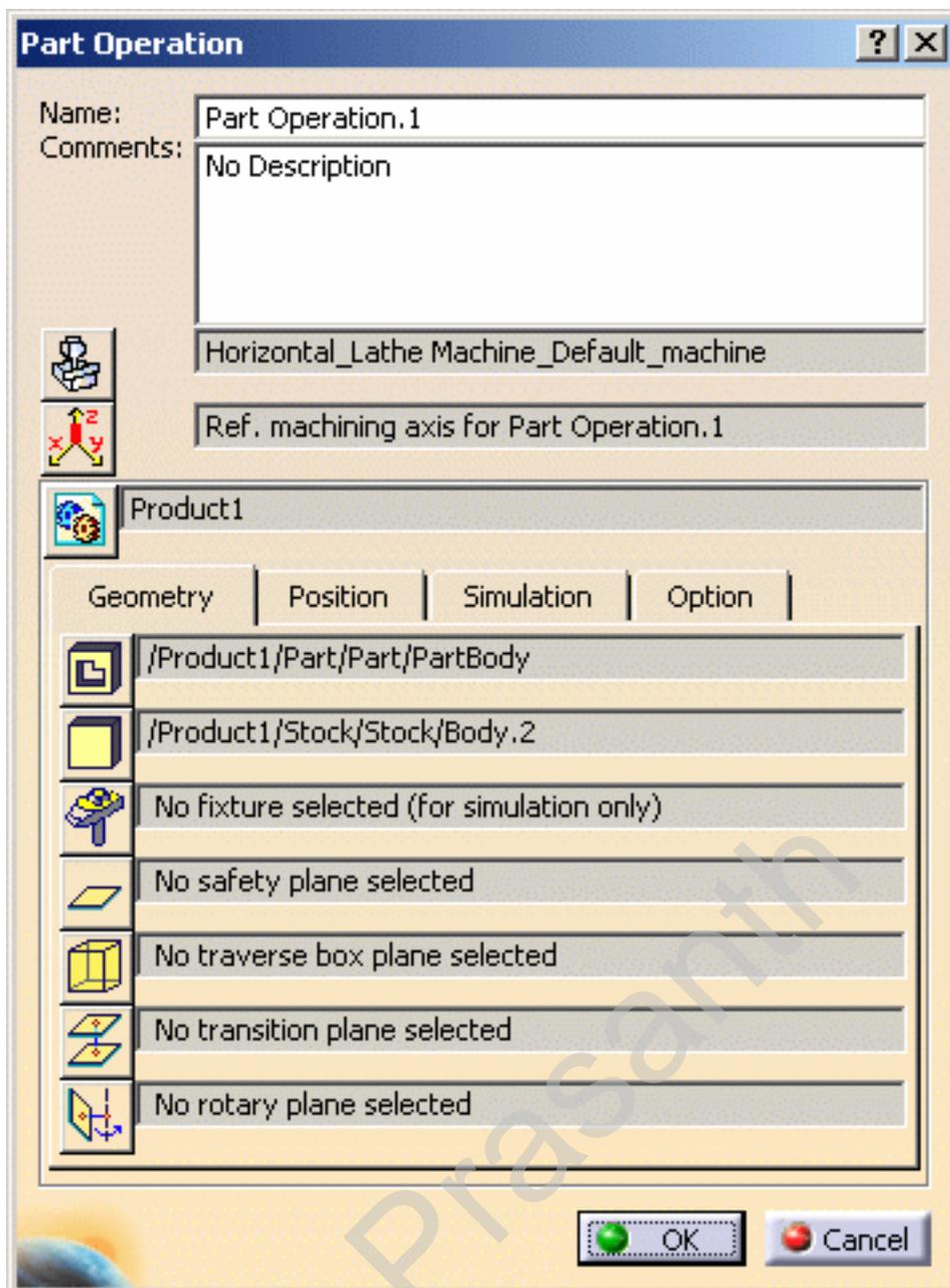


1. Select **File > Open** then select the [StartStockUpdateAutoMode.CATProcess](#) document
2. Select **Machining > Lathe Machining** from the **Start** menu.

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



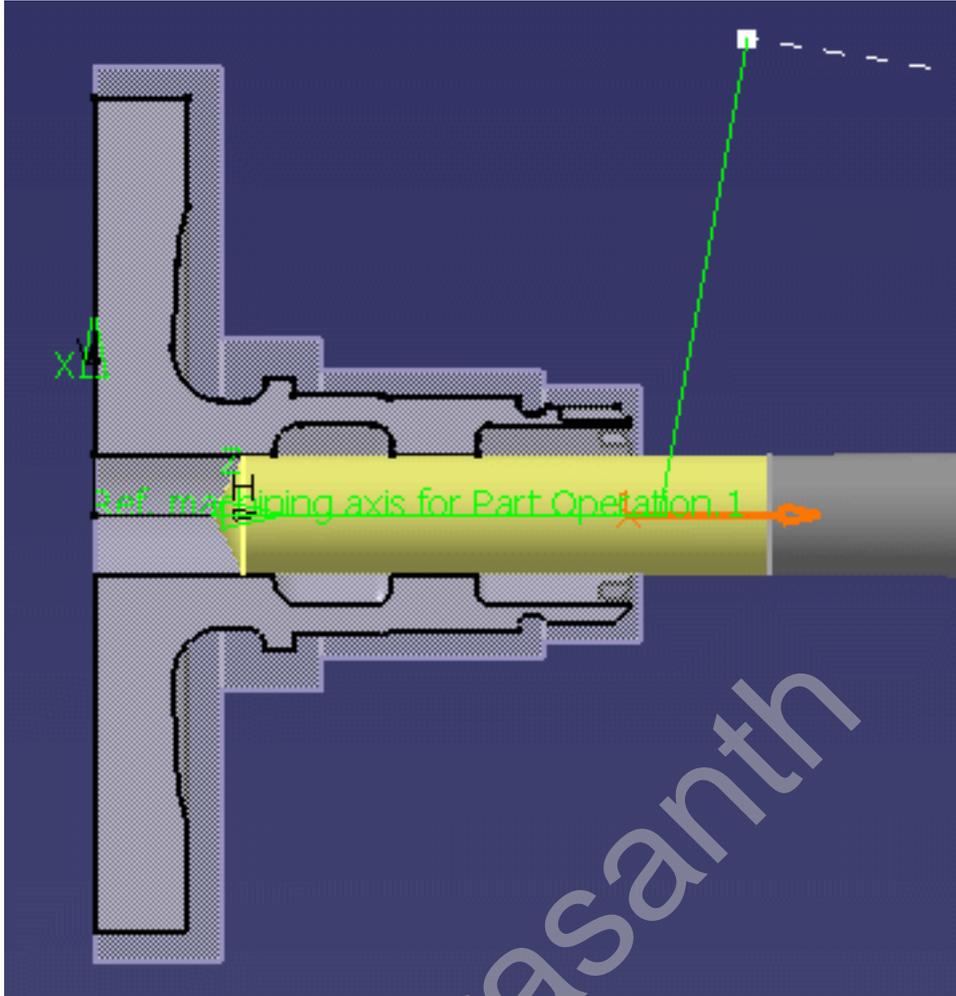
3. Double click Part Operation.1 in the tree to display the Part Operation dialog box.



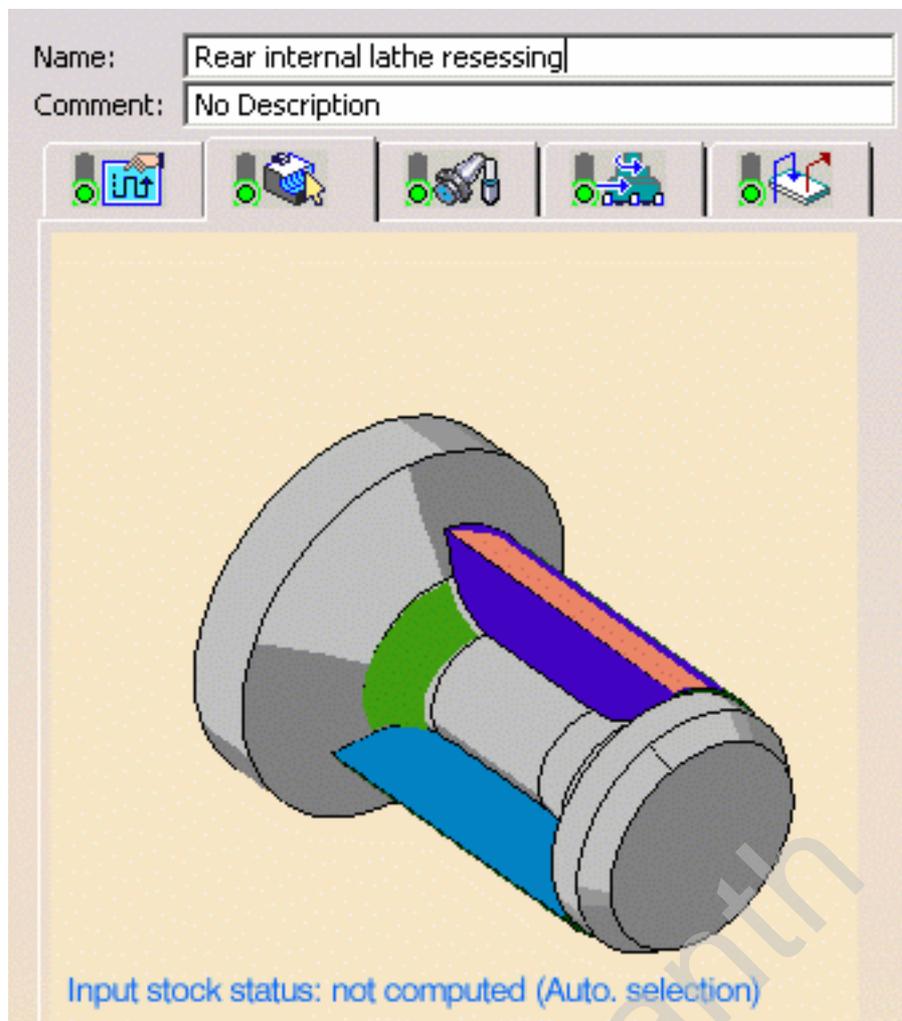
Right-click the Stock field to make sure that **Automatic Stock Selection for Turning Operations** is activated in the contextual menu.



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



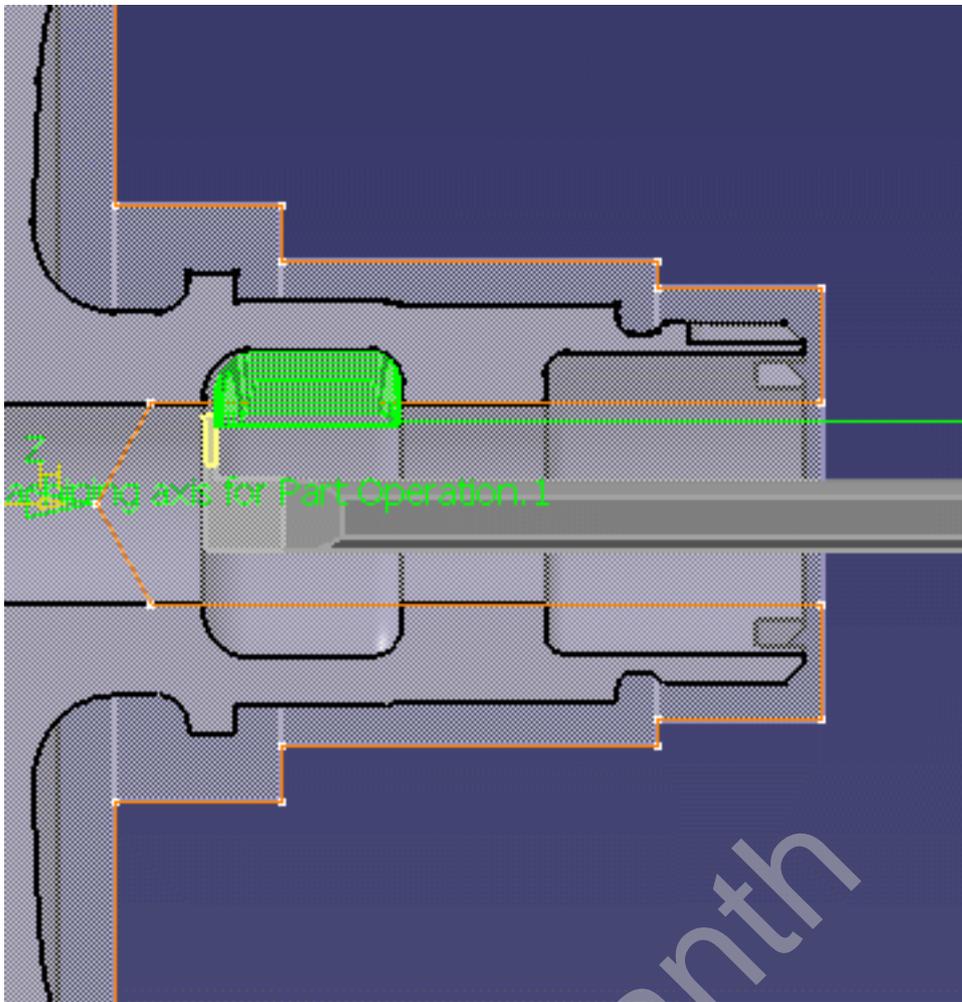
- Click OK in the Replay dialog box. The operation status in the tree is now **Computed**.
- Double click the lathe recessing operation in the tree. Select the Strategy tab  in the machining operation editor.



Note that:

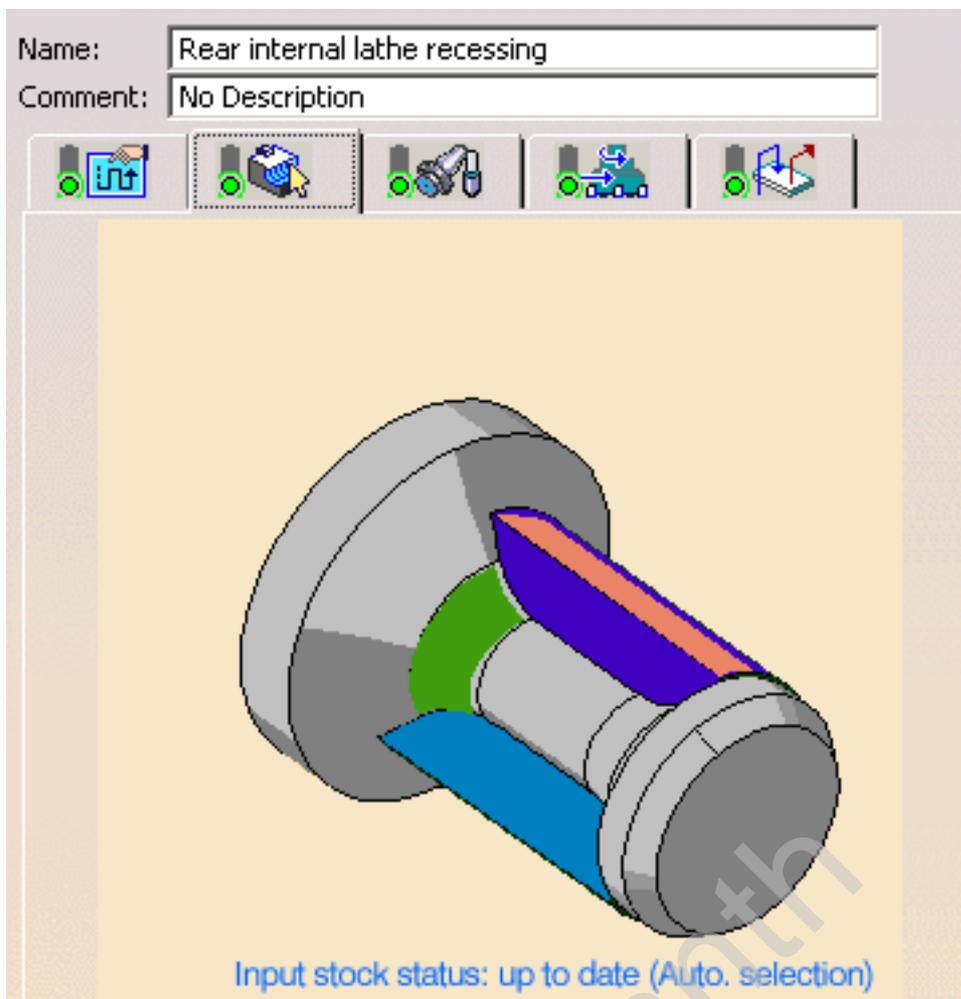
- the color of the stock area indicates that stock selection is optional
- the Input stock status is **Not computed** and that the automatic Stock selection is activated (**Auto. selection**).

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



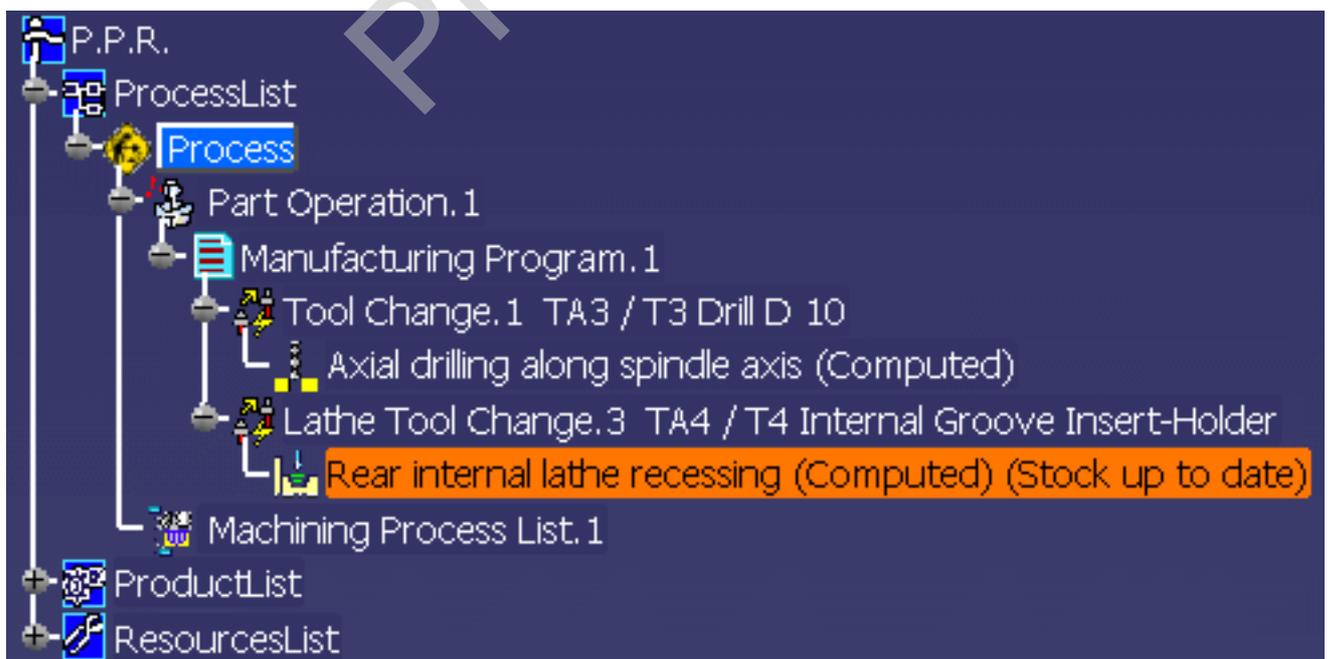
Click OK in the Replay dialog box.

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



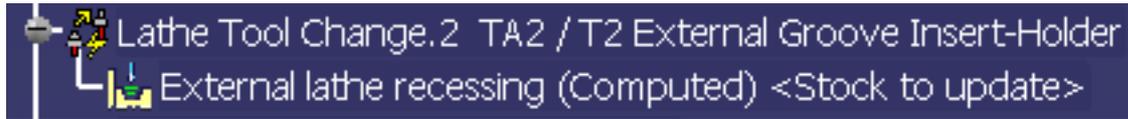
Click OK in the machining operation editor.

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

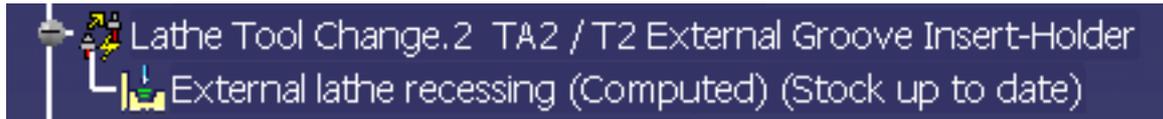


Computing Stock and Tool Path

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.

Prasarikh

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.

Prasanth

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.

Prasanth

Create a Machining Process for Lathe Machining

P2



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.

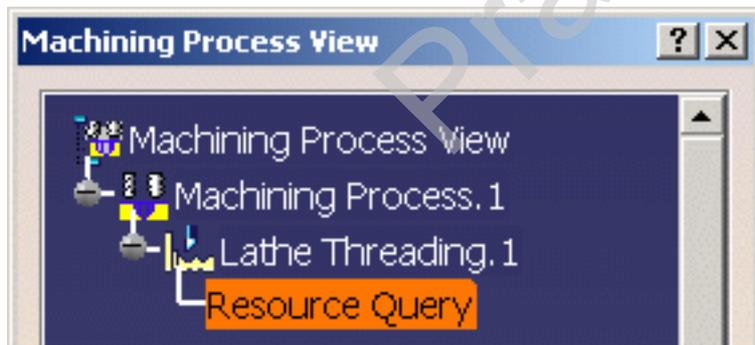
2. Select the Machining Process icon . The dialog box is updated with a new machining process as shown.

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.

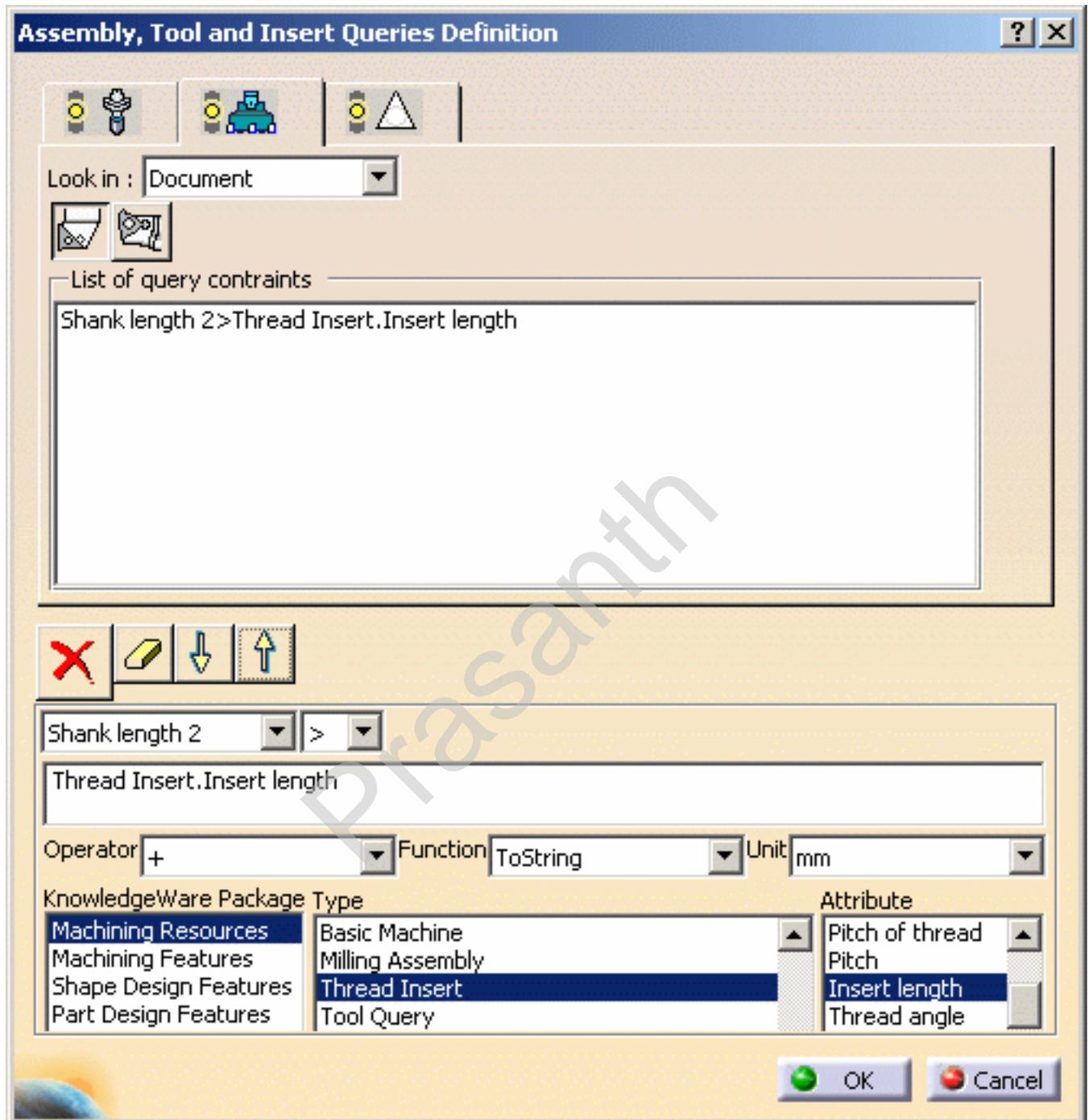


The reference operation has an associated Tooling Query.

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.



Prasanth

Machining Entities

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

- [Edit the Tool of a Lathe Machining Operations](#)
- [Edit a Lathe Tool in the Resource List](#)
- [Edit a Lathe Tool Assembly in the Resource List](#)
- [Edit a Lathe 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 Tool of a Turning Operation



This task shows you how to edit the tool of a turning operation. A machining operation always has a tool assigned to it (default tool, for example).

You can modify this tool in several ways:

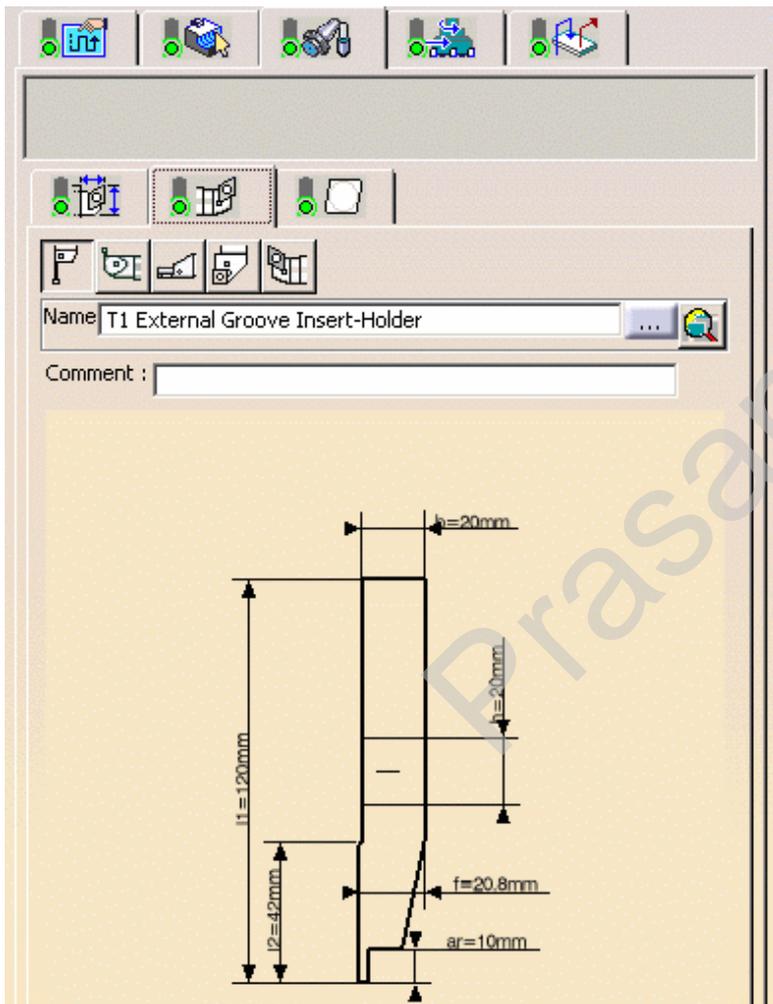
- edit its characteristics, thereby **creating a new tool**
- replace it by **selecting another tool that is already used in the document**
- replace it by **selecting another tool by means of a query**.



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

2. To create a new tool:

If you want to change tool type, select the icon corresponding to the desired tool type.



In this case the corresponding tool 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 tool representation is updated to take the new values into account.

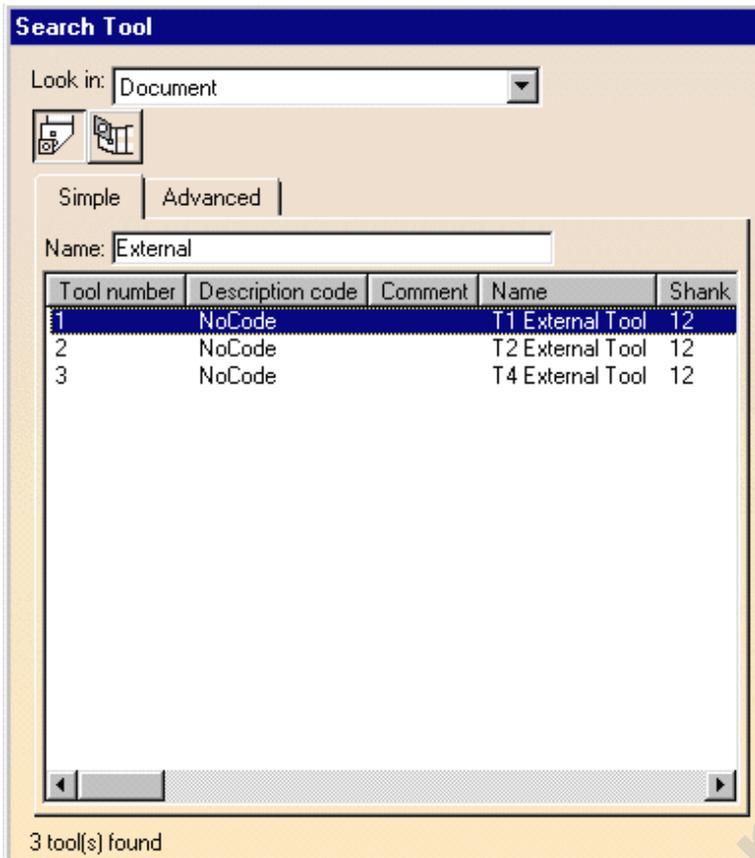
- Click More to expand the dialog box to access all the tool's parameters. Modify the values as desired.
- Enter a name for the new tool.

3. To select a tool that is already used in the document:

- Select the button opposite Name.
- Select the desired tool from the list of tools already used in your document.
- The tool representation is displayed in the 2D viewer. It can be edited as described above.

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

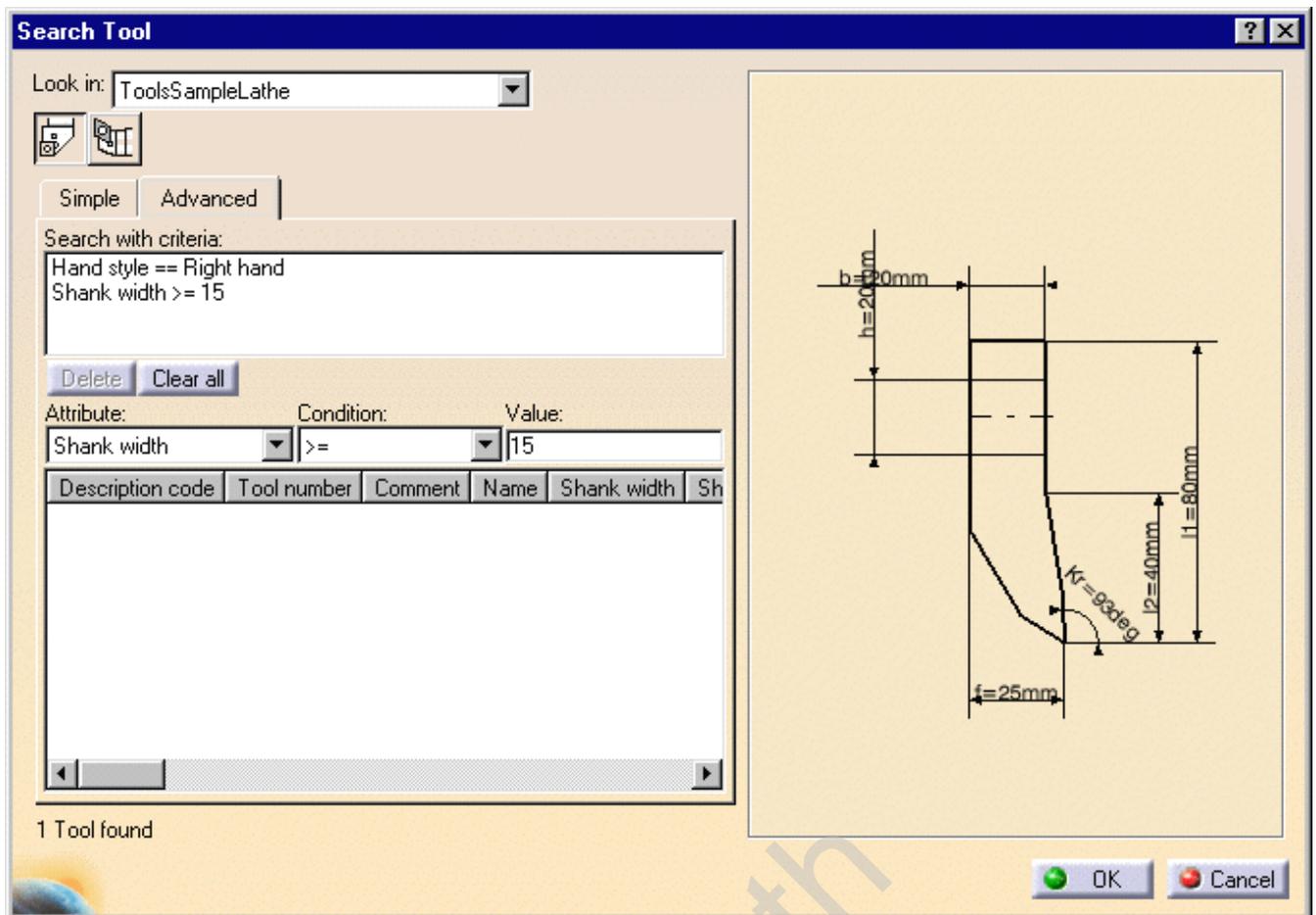
- Click the **Select a tool with query** icon opposite Name. The Search Tool dialog box appears.



- Use the **Look in** combo to specify where you want to search for the tool:
 - 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 tool type, select the icon corresponding to the desired tool.
- You can do a quick search in the **Simple** tab page by means of a character string on the tool name. The tools meeting the simple search criteria are listed.
- Select the desired tool from the list and click OK. The tool representation is displayed in the 2D viewer. It can be edited as described above.
- You can search a tool using finer constraints by selecting the **Advanced** tab page. The example below shows the result of a search for a right hand tool with shank width of 15mm or more in the catalog ToolsSampleLathe.



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

Edit a Lathe Tool in the Resource List

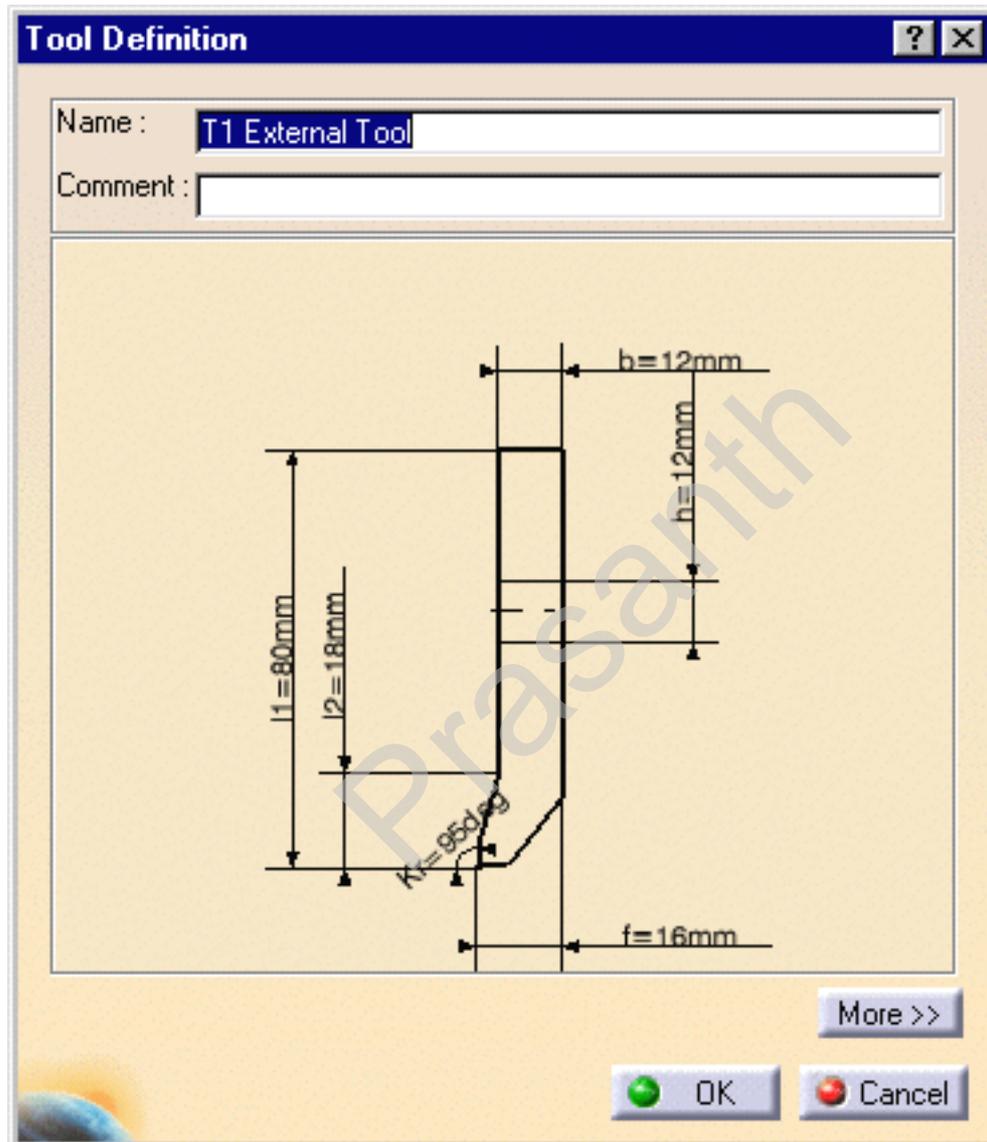


This task shows you how to edit a lathe tool that is already used in your document.



1. To edit a tool in the Resource List right click it and select the **Edit NC Resource** contextual command.

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



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



Please refer to [Lathe Tools](#) for a description of the available attributes for this resource.

4. You can specify the tool geometry in two ways:

- double click a parameter in the large tool icon and enter the desired value in the Edit Parameter dialog box that appears
- or enter the desired values in the **Geometry** tab page.

Geometry	Technology	Compensation
Hand style :	Left hand	
Holder capability :	Surface	
Kappa R (Kr):	95deg	
Insert angle (a):	80deg	
Insert length (l):	6.45mm	
Clearance angle :	7deg	
Shank cut width (f):	16mm	
Shank height (h):	12mm	
Shank length 1 (l1):	80mm	
Shank length 2 (l2):	18mm	
Shank width (b):	12mm	

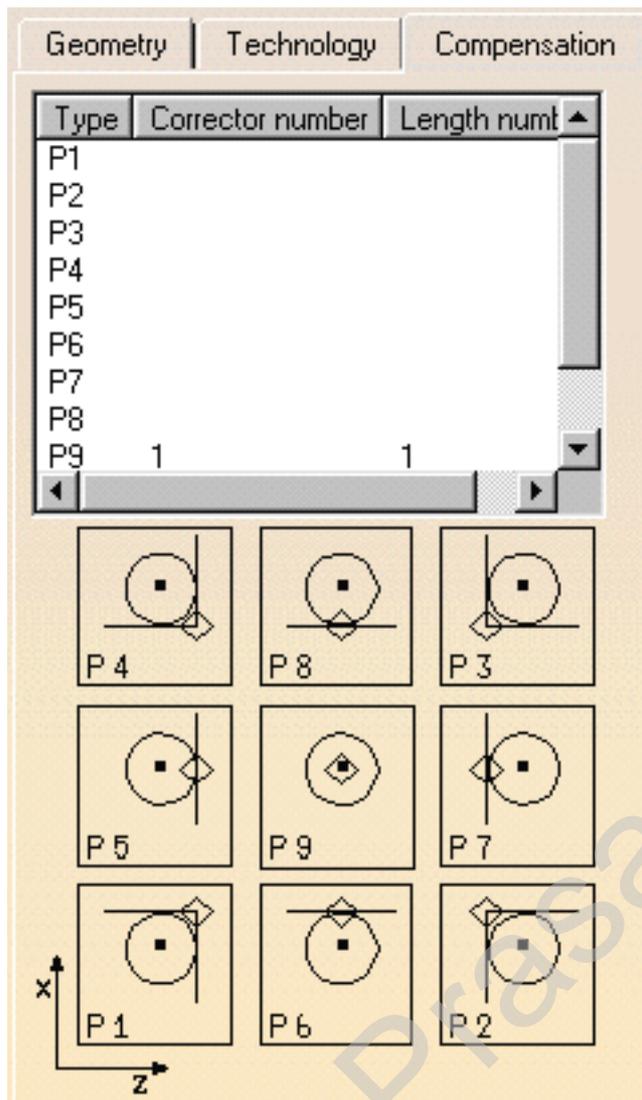
The icon representation of the tool is updated with these values.

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

Geometry	Technology	Compensation
Max machining length :	0mm	
Max life time :	0s	
Coolant syntax :		
Weight syntax :		
Trailing angle :	90deg	
Leading angle :	90deg	
Max rec. depth :	1000mm	

6. If tool 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 tool compensation data.

The Compensation Definition dialog box appears.

8. Enter the desired values for the tool's compensation sites.

See [Specify Tool Compensation](#) for more information.

9. Click OK to accept the modifications made to the tool.

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



Edit a Lathe Tool Assembly in the Resource List



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



1. To edit a tool assembly in the Resource List right click it and select the **Edit NC Resources** contextual command.

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

Tool Assembly Definition

Name : Lathe Assembly.1

Comment :

Tool number : 0

Setup angle : 0deg

More >>

OK Cancel



Please refer to [MfgLatheToolAssembly \(Lathe Assembly\)](#) for a description of the available attributes for this resource.

2. If needed, enter a new name and comment for the tool assembly.
3. If needed, use the spinner to change the Tool number.
4. Click More to expand the dialog box to access the Geometry and Technology tab pages.
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.

Geometry Technology

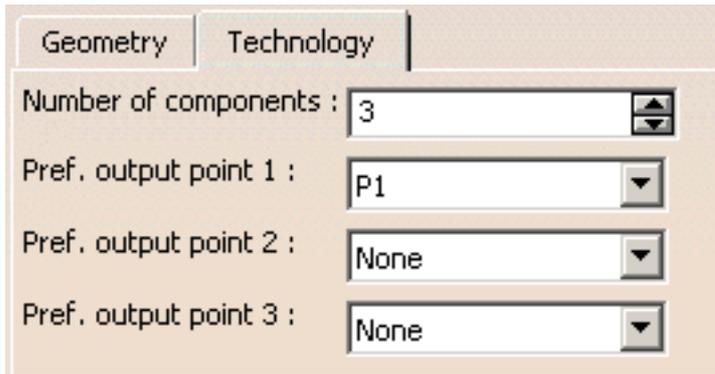
Set X : 0mm

Set Y : 0mm

Set Z : 0mm

Tool inverted

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



Geometry	Technology
Number of components :	3
Pref. output point 1 :	P1
Pref. output point 2 :	None
Pref. output point 3 :	None

- 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. In this case, if a component (tool or insert, for example) of the assembly has a user representation, it will not be taken into account.



Prasanth

Edit a Lathe Insert in the Resource List

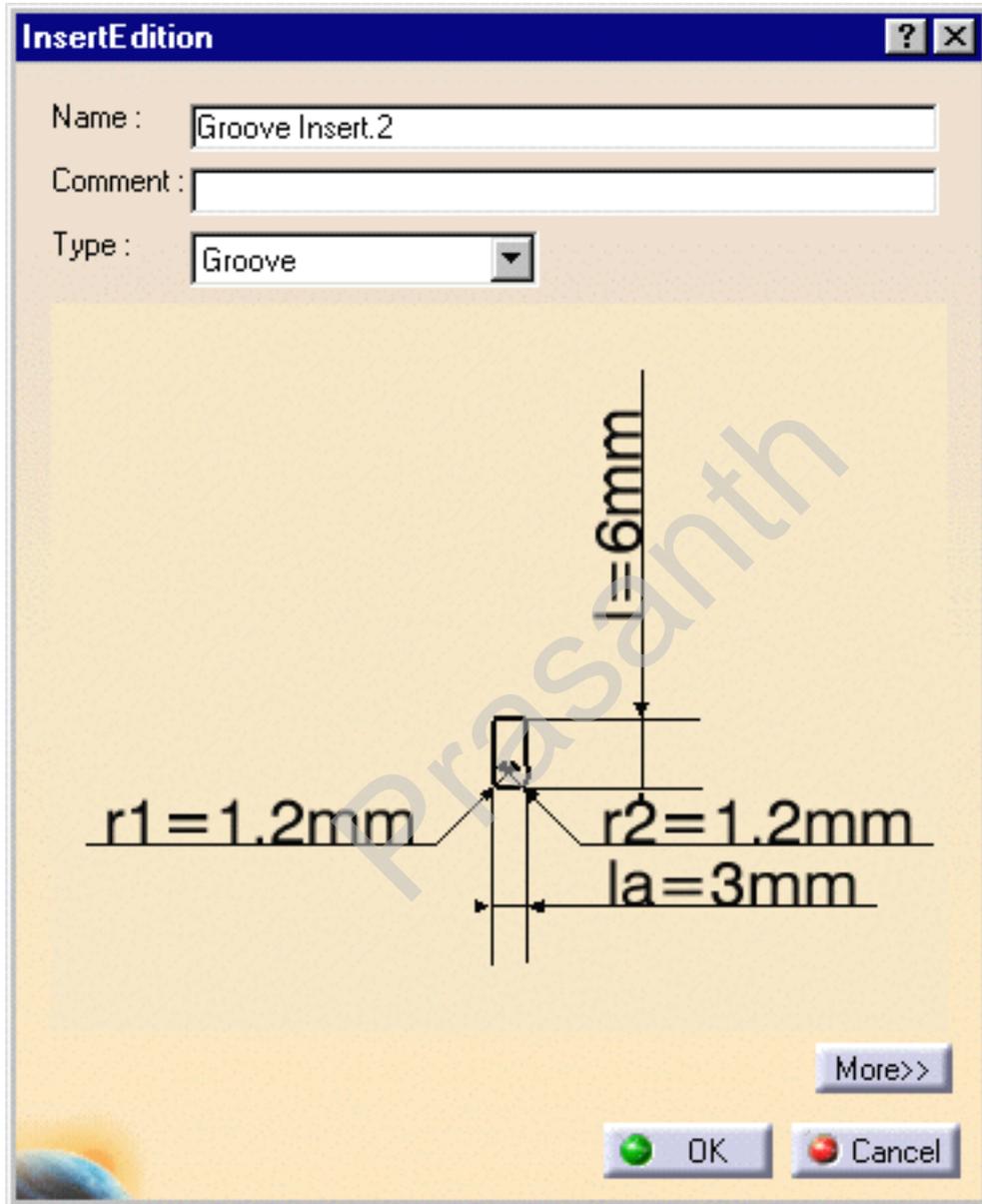


This task shows you how to edit a lathe insert that is already used in your document.



1. To edit a lathe insert in the resource list right click it and select the **Edit NC Resources** contextual command.

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



2. If needed, enter a new name for the lathe 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 and Technology tab pages.

Please refer to [Lathe Inserts](#) for a description of the available attributes for this resource.



5. You can specify the lathe insert geometry in two ways:

- double click a parameter in the large lathe insert icon and enter the desired value in the Edit Parameter dialog box that appears
- or enter the desired values in the **Geometry** tab page.

Geometry	Technology	Feeds & Speeds
Height (l):	6mm	
Insert width (la):	3mm	
Bottom angle (b):	0deg	
Left flank angle (a1):	2.5deg	
Right flank angle (a2):	2.5deg	
Left nose radius (r1):	1.2mm	
Right nose radius (r2):	1.2mm	
Thickness (s):	2.5mm	
Clearance angle :	6deg	

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

Geometry	Technology	Feeds & Speeds
Machining quality :	Either	
Material :	Other	
Cutting length (Lc):	0mm	

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

Geometry	Technology	Feeds & Speeds
Finishing cutting speed :	0mm_mn	
Finishing feedrate per tooth :	0mm_turn	
Roughing cutting speed :	0mm_mn	
Roughing feedrate per tooth :	0mm_turn	

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



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



Prasanth

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 CGR File in Batch Mode (P2 functionality):** Select the Generate NC Code in Batch Mode icon then select the manufacturing program to be processed and define the CGR file processing options.
- **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.



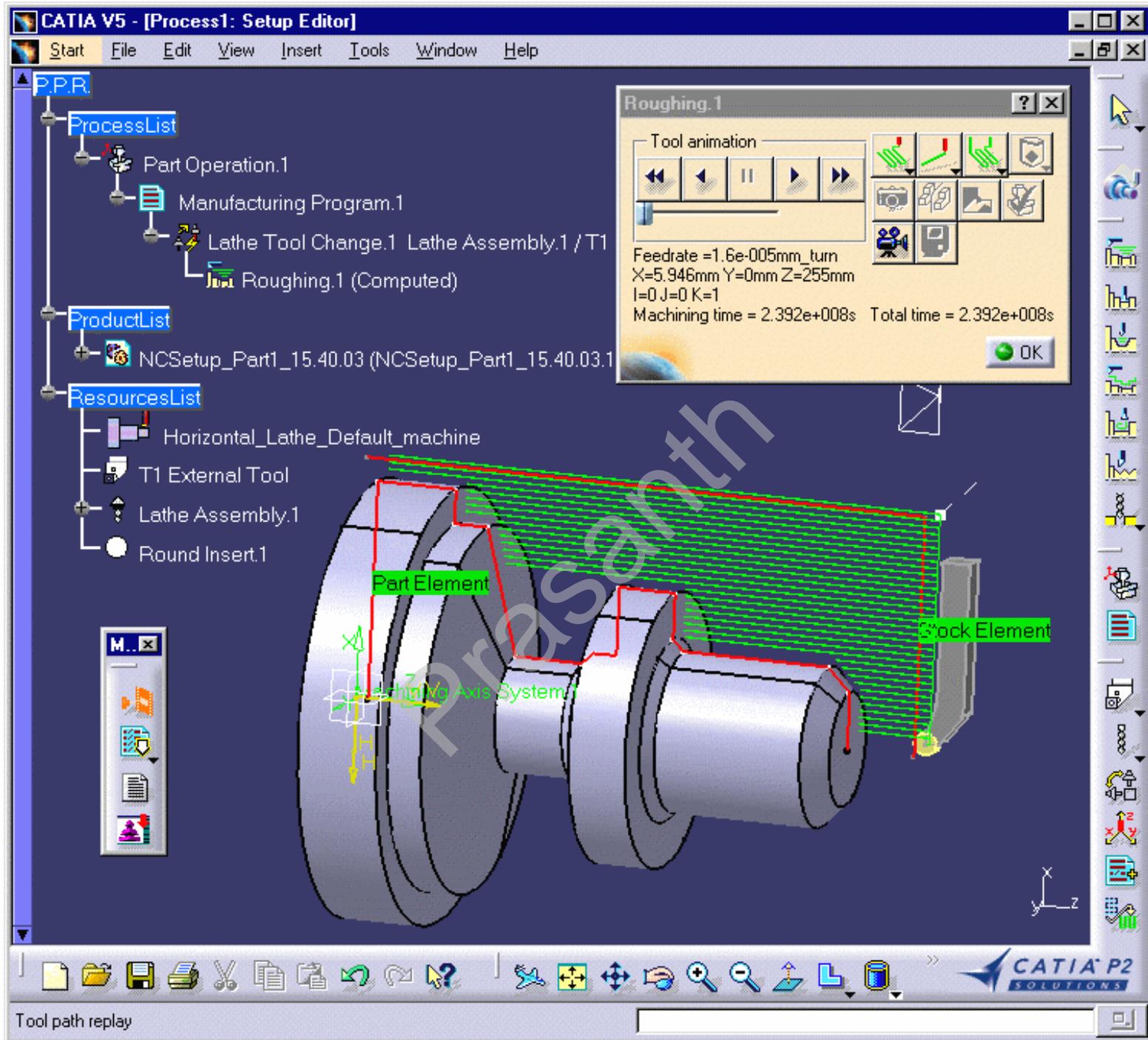
Generate Documentation: Select the Generate Documentation icon to produce shop floor documentation in HTML format.

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

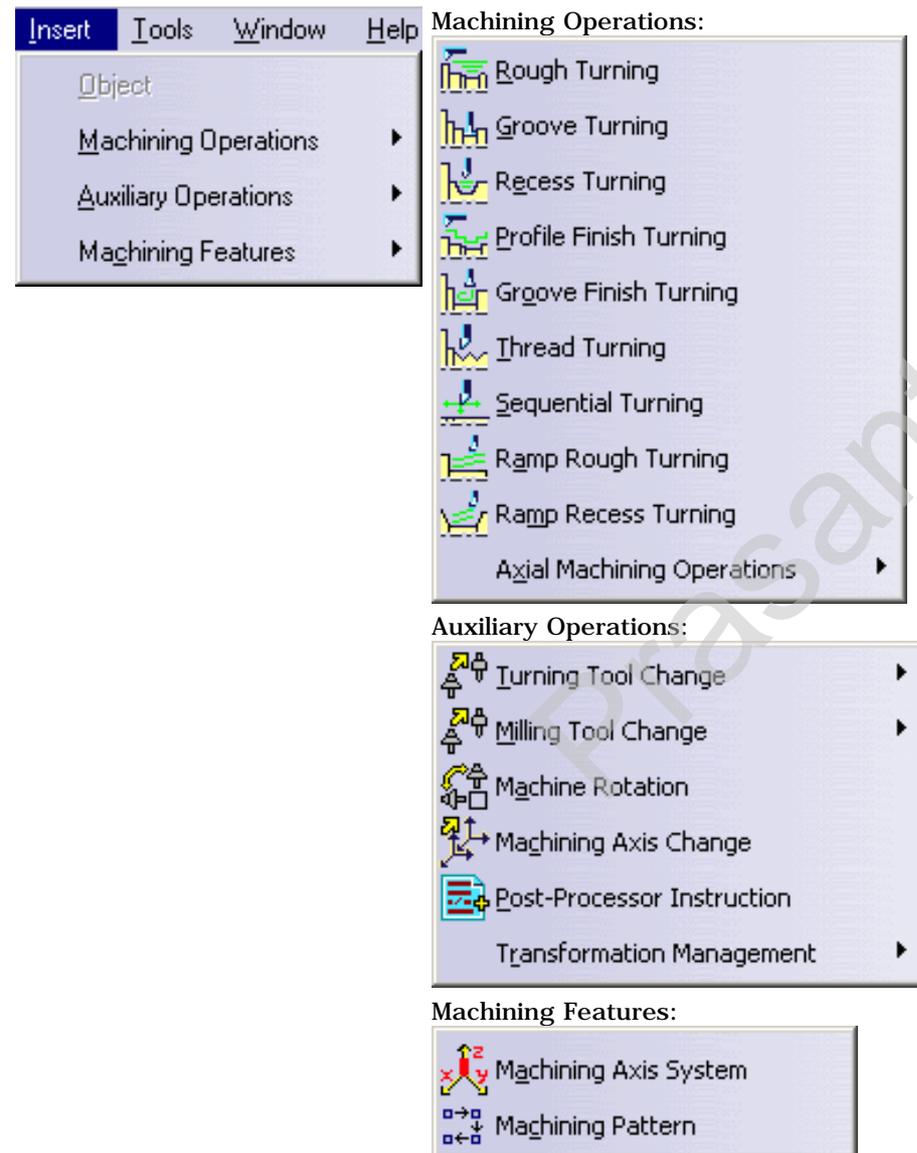
The menu commands that are specific to Lathe Machining are described below.



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



Insert > Machining Operations

Command...

Rough Turning

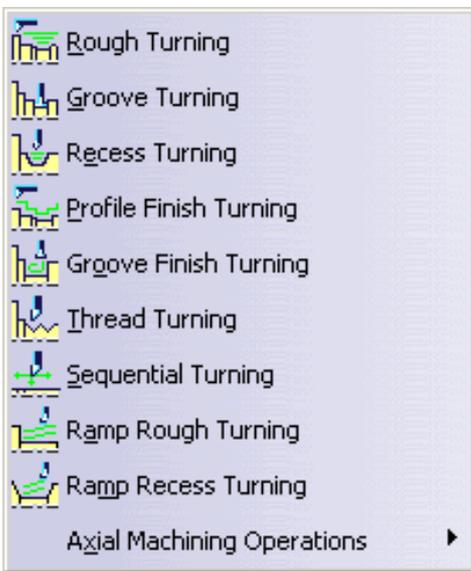
Description...

Create a Longitudinal Rough Turning Operation
 Create a Parallel Contour Rough Turning Operation
 Create a Face Rough Turning Operation

Groove Turning

Recess Turning

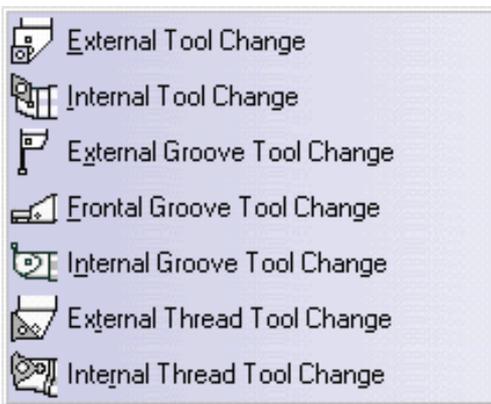
Create a Groove Turning Operation
 Create a Recess Turning Operation



Profile Finish Turning
Groove Finish Turning
Thread Turning
Sequential Turning
Ramp Rough Turning
Ramp Recess Turning
Axial Machining Operations

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 Ramp Recess Turning Operation
Create Axial Machining Operations

Insert > Auxiliary Operations > Turning Tool Change



Description...

Allows inserting turning tool changes in the program.

Prasanth

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.

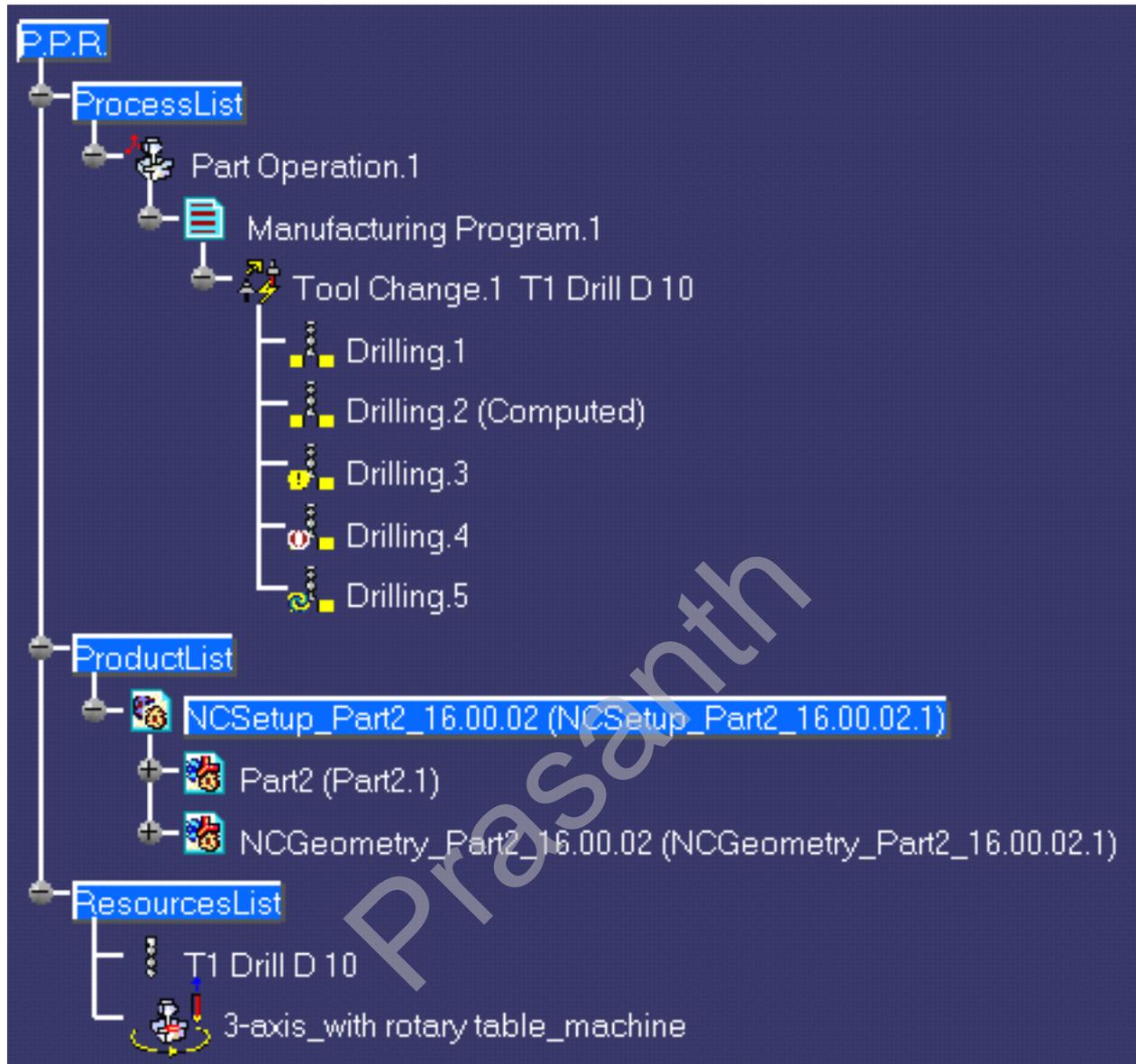


Please note that the icon representing a Tool Change operation in the PPR tree looks like this:

Prasanth

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.



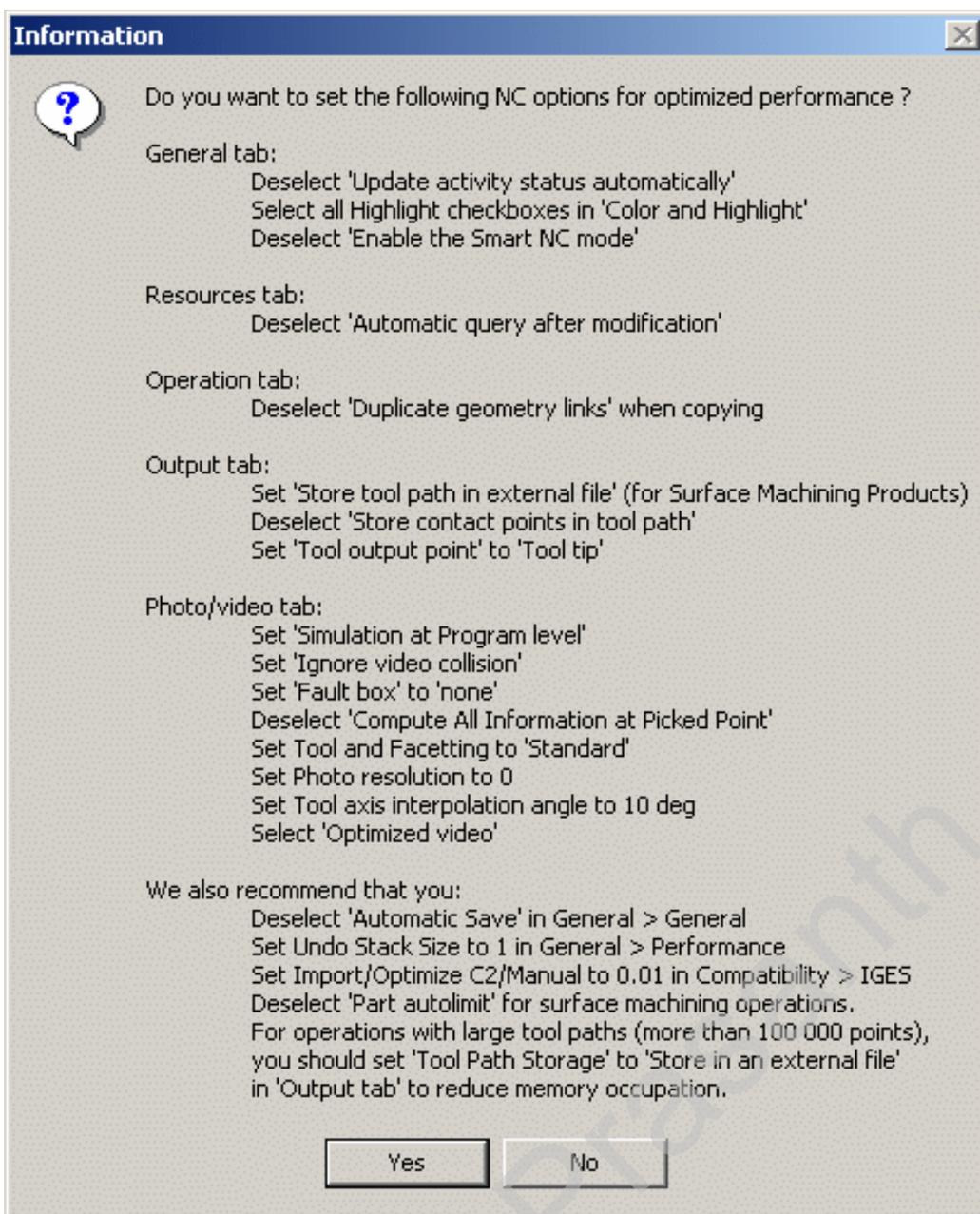
Select the **General** tab, which is divided up into areas.

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

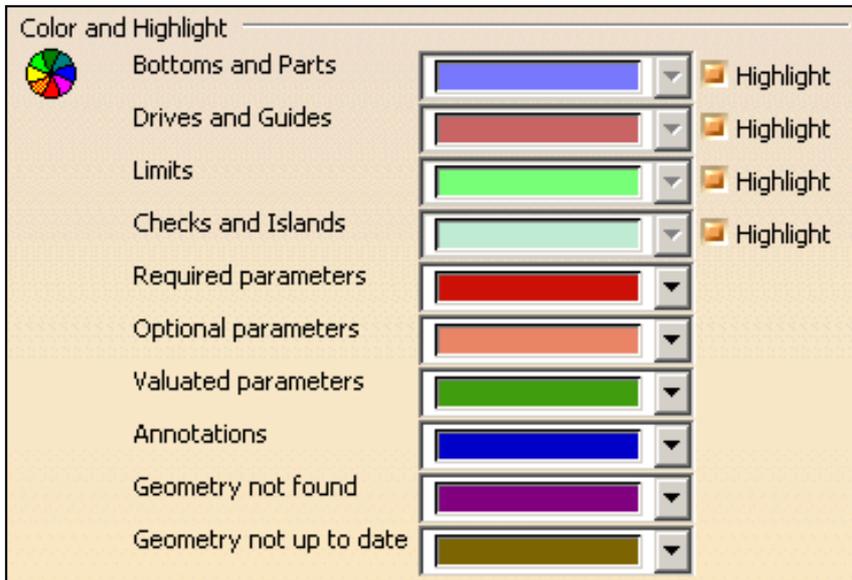


- Select the checkbox if you want the status of activities in the tree to be updated automatically.
- If this checkbox is not selected:
 - you can update activity status manually in your workbench using the Update Status icon  in the Auxiliary Commands toolbar.
 - the status of the activity after a manual 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

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.

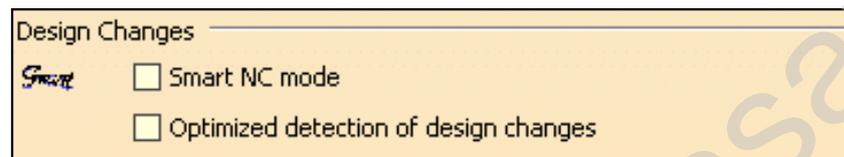
Complementary Geometry



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 check box 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:

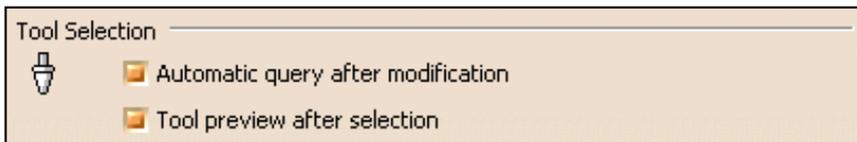
- a semi colon (;) character for Windows NT platforms.
- a colon (:) character for UNIX platform.

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

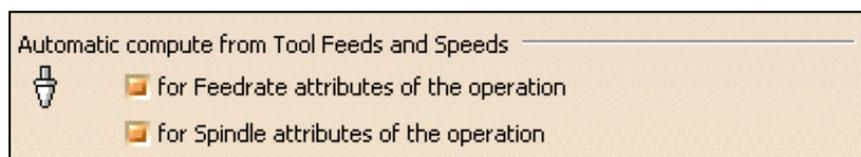
Tool preview after selection

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.

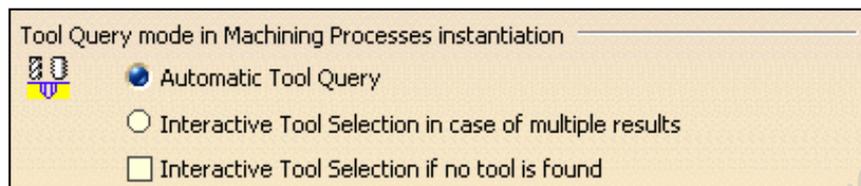
Spindle attributes of the operation

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:

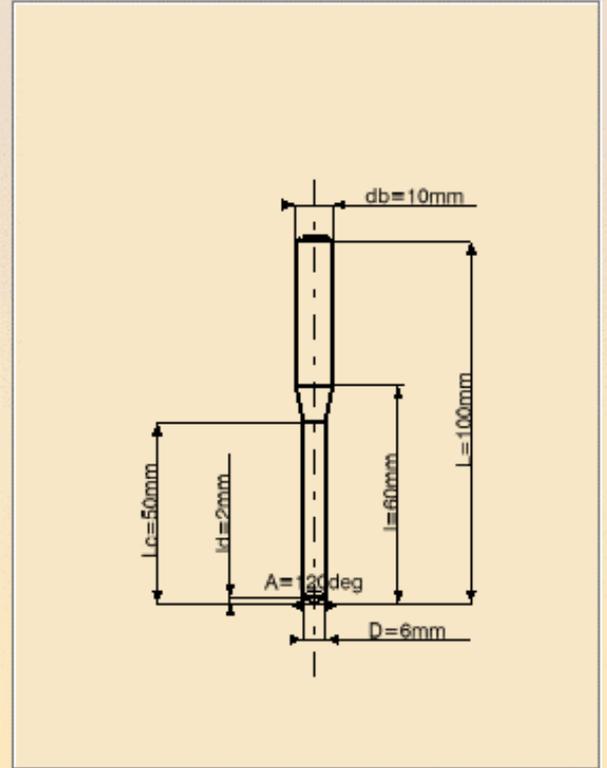
Nominal diameter ≤ 10mm
Nominal diameter >= 5mm

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

Prasanth

Operation

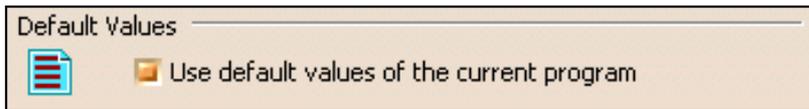
This document explains how to customize machining operation settings for Machining products.



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

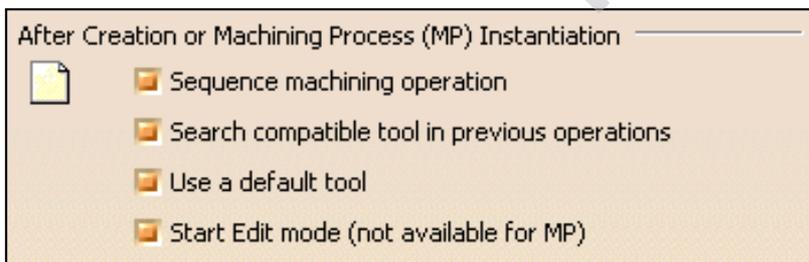


Select the checkbox if you want operations to be created with the values used in the current program. The values and units of attributes at the creation step of an operation are set to the values and units of the last edited and validated operation whatever its type (that is, exit the operation definition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

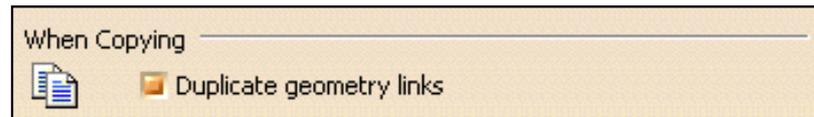
When creating an operation, a search is done in the document to find a compatible tool. If no compatible tool exists, a default one is created in the document and set in the created operation. If checkbox is not selected, no tool will be defined on the operation.

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.

 By default, these checkboxes are selected.

When Copying

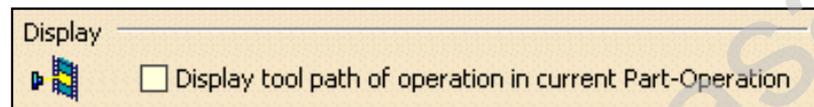


Select the checkbox if you want geometry links to be duplicated in a copied operation.

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

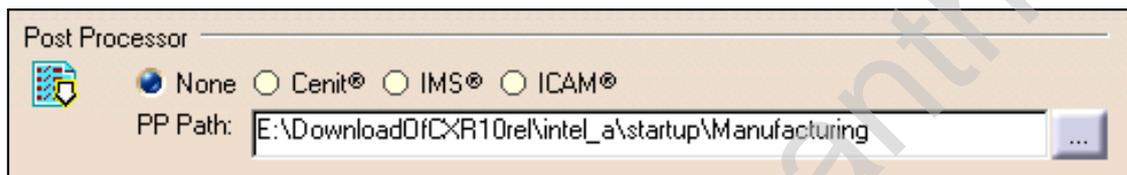
This document explains how to customize data output settings for Machining products.



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



Tool Path Storage



- Store tool path in the current document
- Store tool path in an external file

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



Tool Path Edition



- Edit Tool Path is available

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



During Tool Path Computation



- Store contact points in tool path

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



Tool Output Point



- Tool Tip
- Tool Center
- Tool Center for Ball End Tools

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

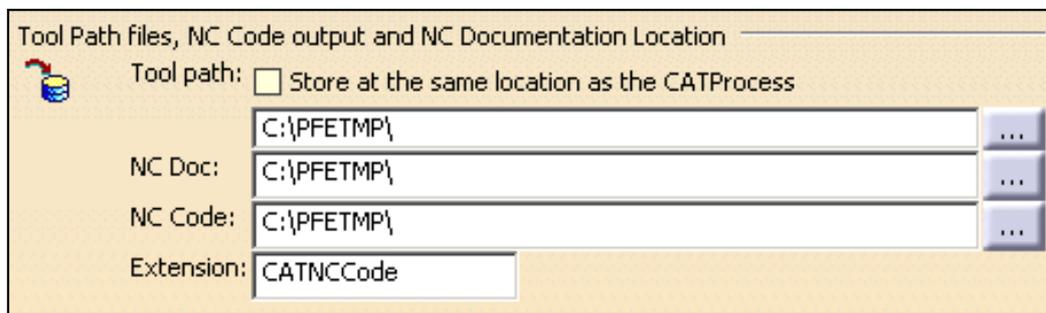
- tool tip
- tool center point
- tool center point for ball end tools (that is, any tool with the Ball-end tool attribute selected or an end mill whose nominal diameter is equal to twice the corner radius).

Performance is better when the Tool Tip option is selected.

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

Default File Locations





Tool Path files, NC Code output and NC Documentation Location

Tool path: Store at the same location as the CATProcess

NC Doc: C:\PFETMP\

NC Code: C:\PFETMP\

Extension: CATNCCode

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).

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

Prasanth

Program

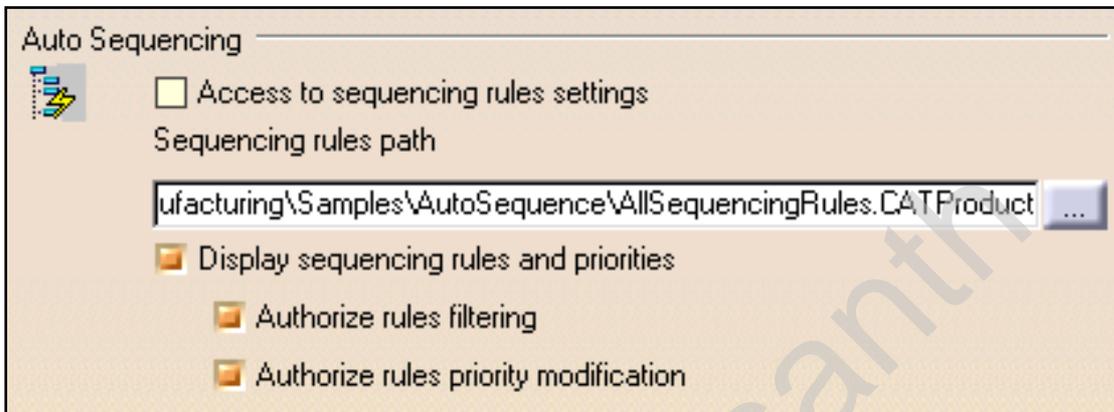
This document explains how to customize manufacturing program settings for Machining products.



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

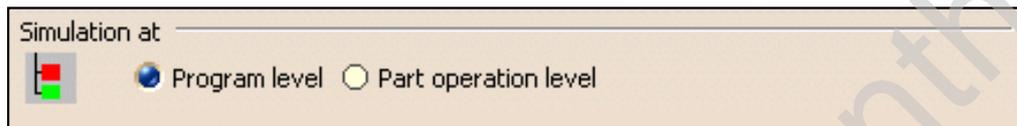
This document explains how to customize material removal simulation settings for NC Manufacturing products.



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

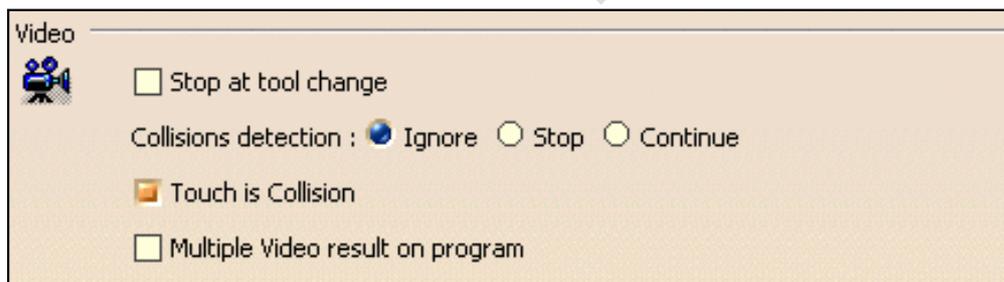
Simulation at



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.

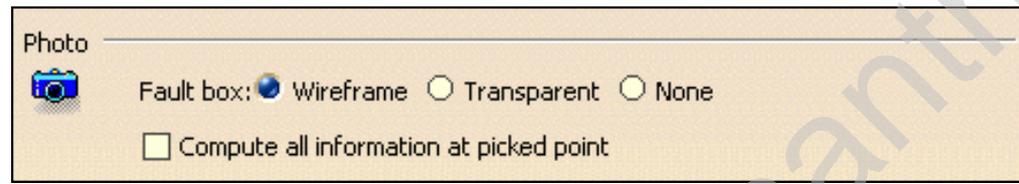
 By default, this checkbox is selected.

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.

Best performance is obtained when no bounding box is required and the checkbox is not selected.

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

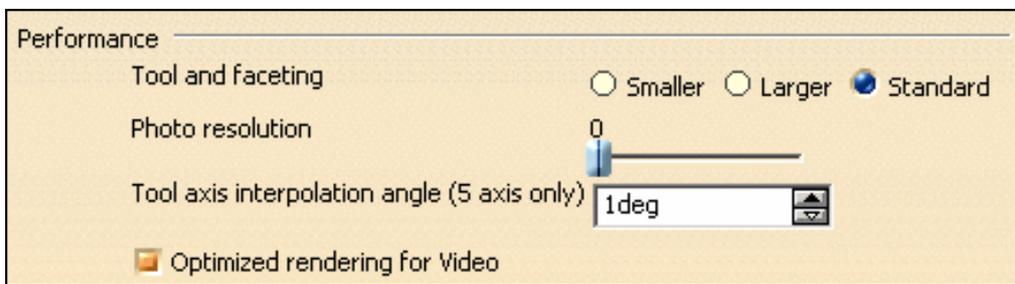
Select the checkbox to compute all information at picked point.

 By default, this checkbox is not selected.

Best performance is obtained when **Fault box: None** is selected and the checkbox is not selected.

Performance



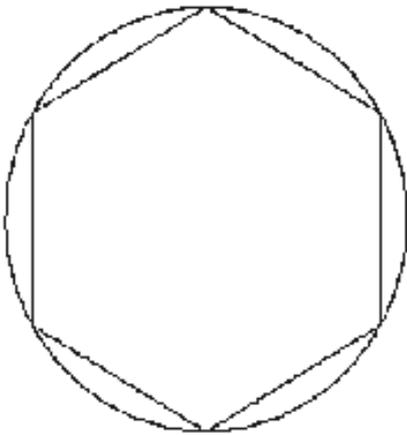


Tool and faceting

There are three methods of tool faceting used in Video simulation: Standard, Smaller and Larger.

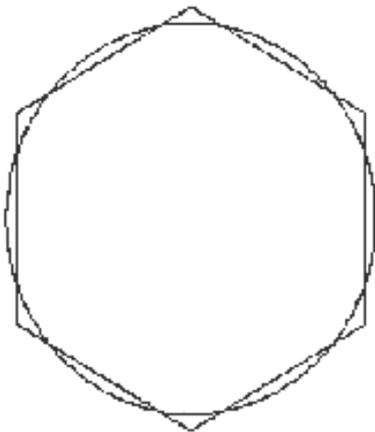
The number of facets for a tool representation is determined by the chord deviation that is set for the tool diameter (0.005% of the tool diameter).

- **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.

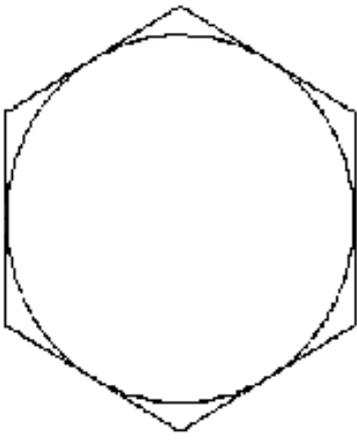
- **Standard:** The picture shows a rough approximation of a tool with six facets. Note that the chord deviation is partly inside and partly outside the actual circle, and that the points are not always on the circle.



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 1 degree.

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



Color

Tool and machined area: Same Last tool different All different

Tool1

Tool holders

Parts

Fixtures

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



Positioning Move

Maximum tool axis variation

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](#)
[Lathe Tool Assembly Conventions](#)

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:

- [Tools](#)
- [Geometry](#) including [Part and Stock Offsets](#)
- [Location and Limits](#)
- [Machining Strategy Parameters](#)
- [Feeds and Speeds](#)
- [Tool Compensation](#)
- [Macros](#).

Tools for Rough Turning

The following tools may be used:

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

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).

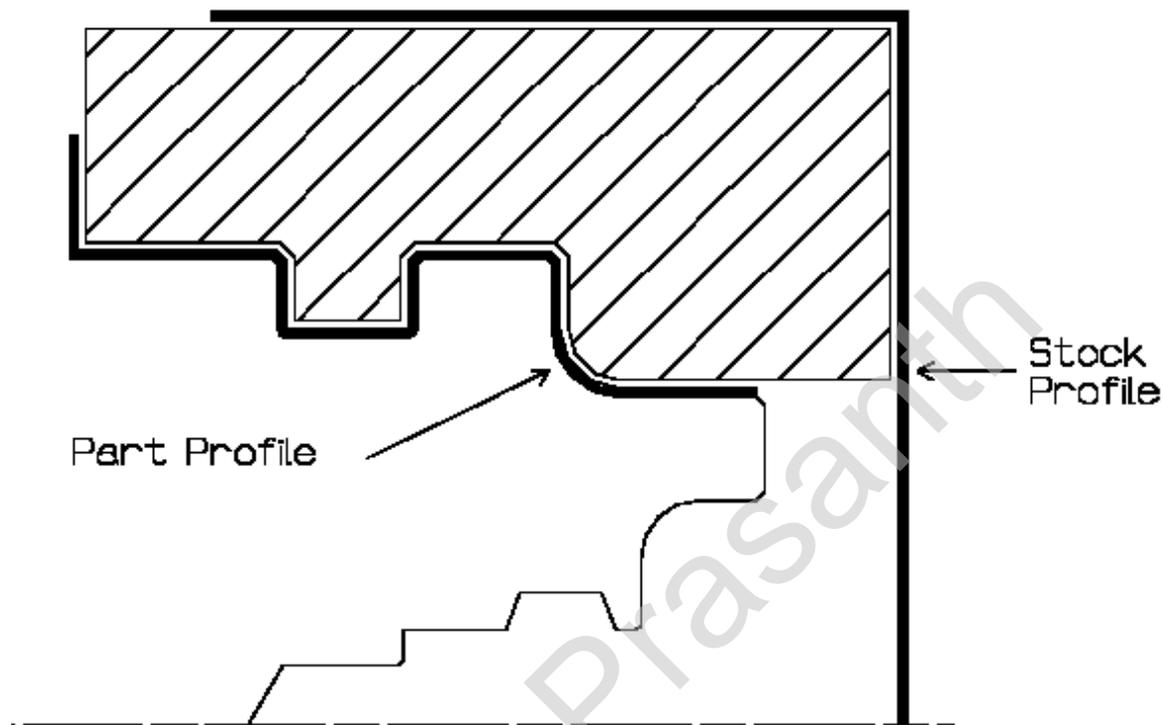
A line through the intersection point parallel to the radial axis delimits the area to machine.

Orientation for Rough Turning

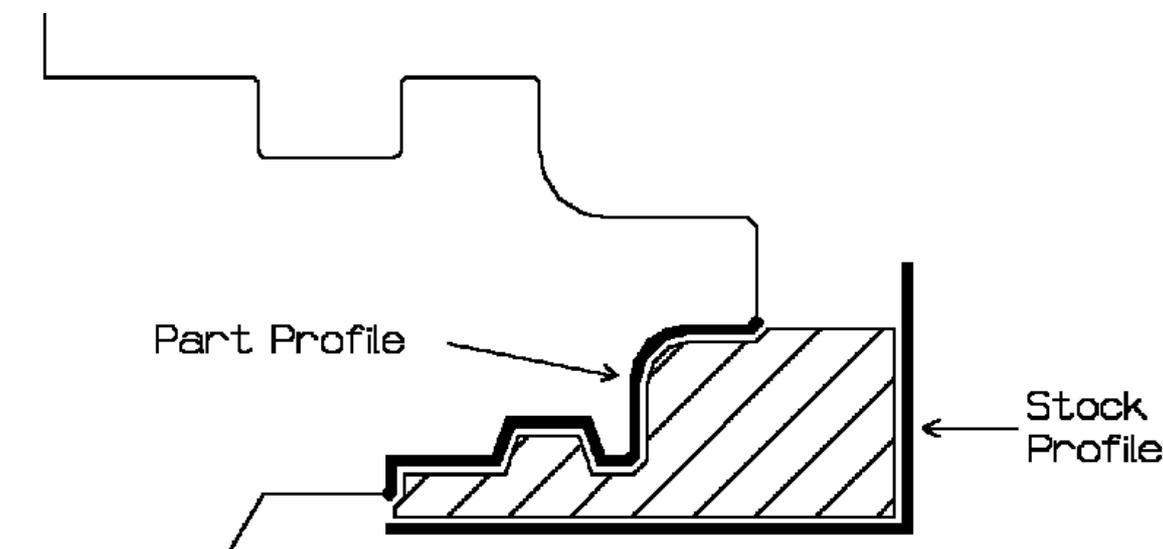
The following **Orientations** are proposed: Internal, External and Frontal (for Face and Parallel Contour Rough Turning only).

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



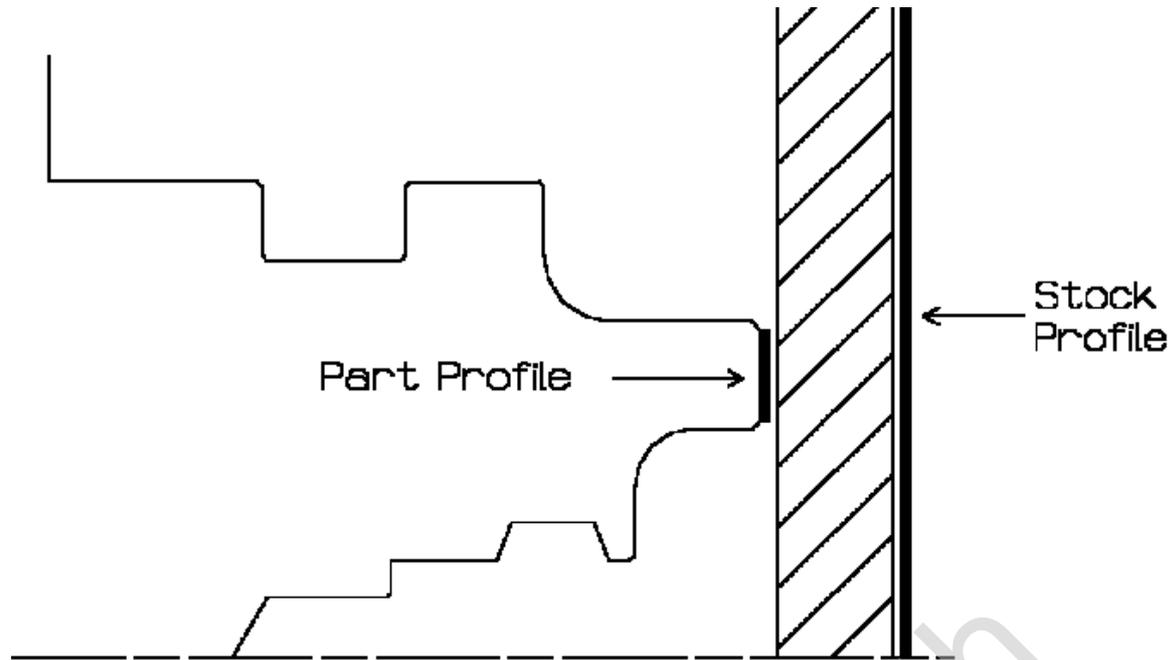
Internal Rough Turning



Frontal Rough Turning

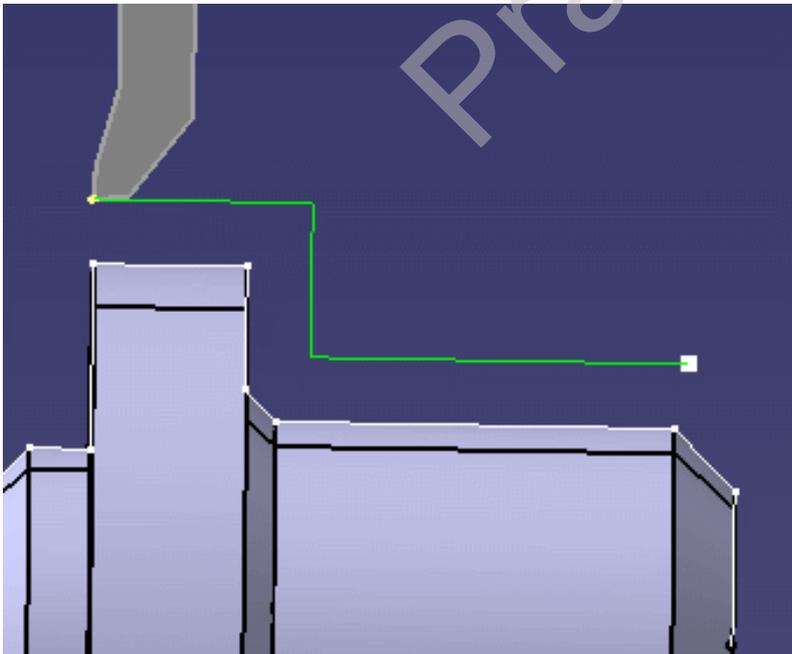
Frontal machining is proposed for face Rough Turning. In that case, the minimum and maximum diameters of the area to machine are determined by the stock profile dimensions.

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.

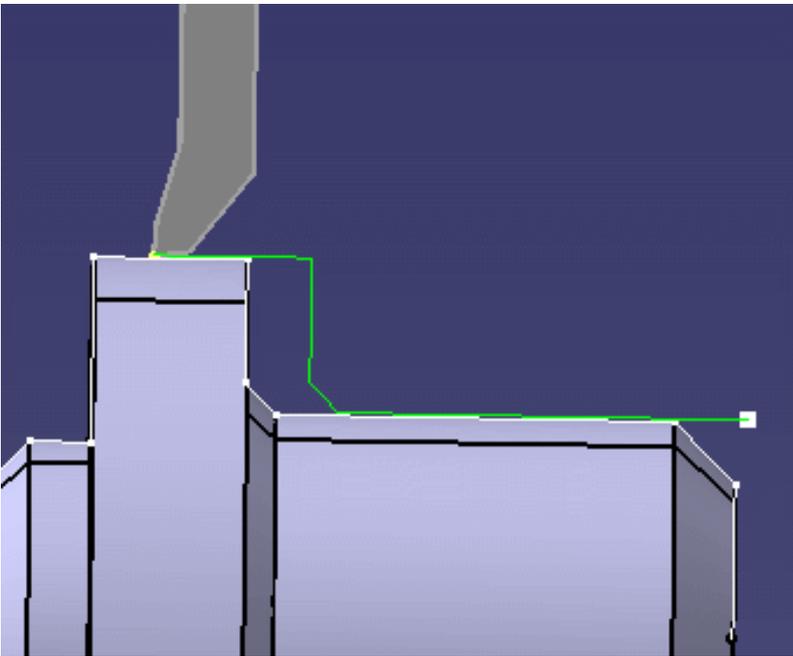


Part and Stock Offsets for Rough Turning

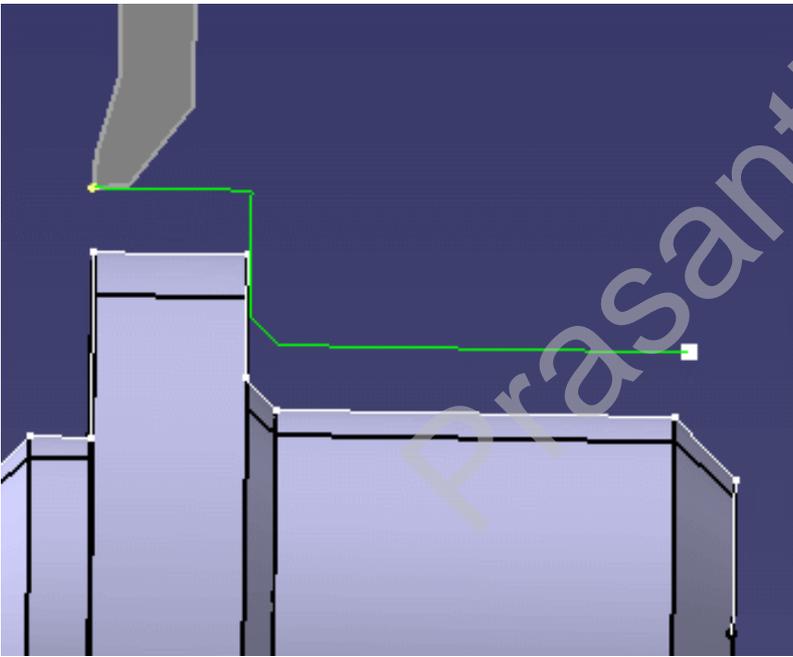
- **Stock offset:** specifies a virtual displacement of the stock profile.
- **Part offset:** specifies a virtual displacement of the part profile.



- **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.

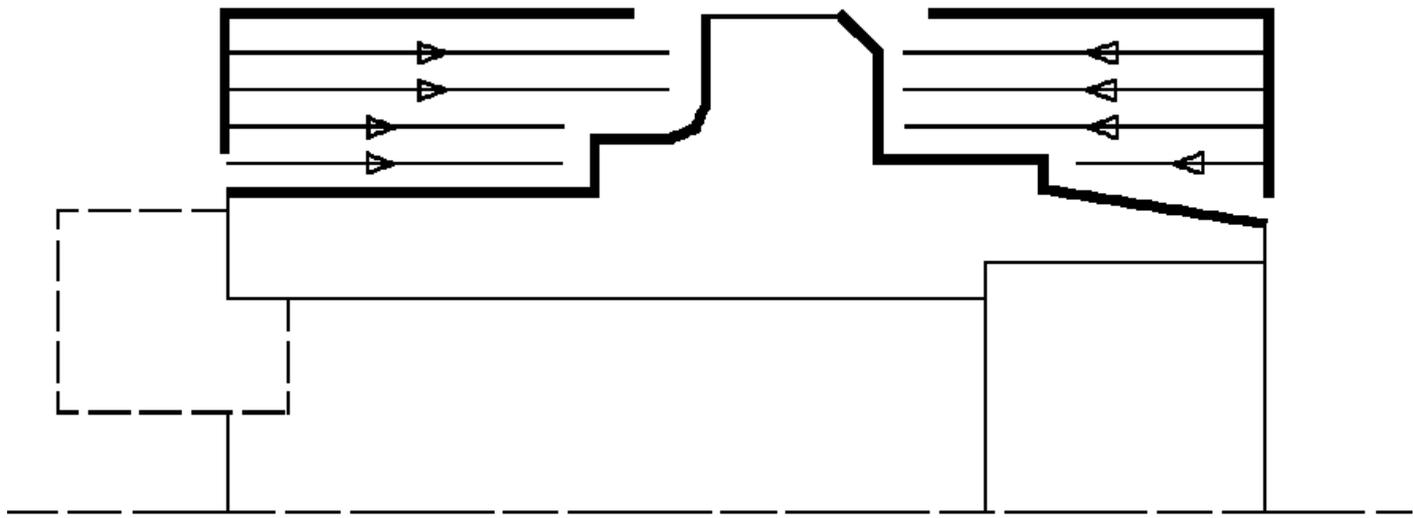
Location and Limits for Rough Turning

The following machining **Locations** are proposed:

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

BACK

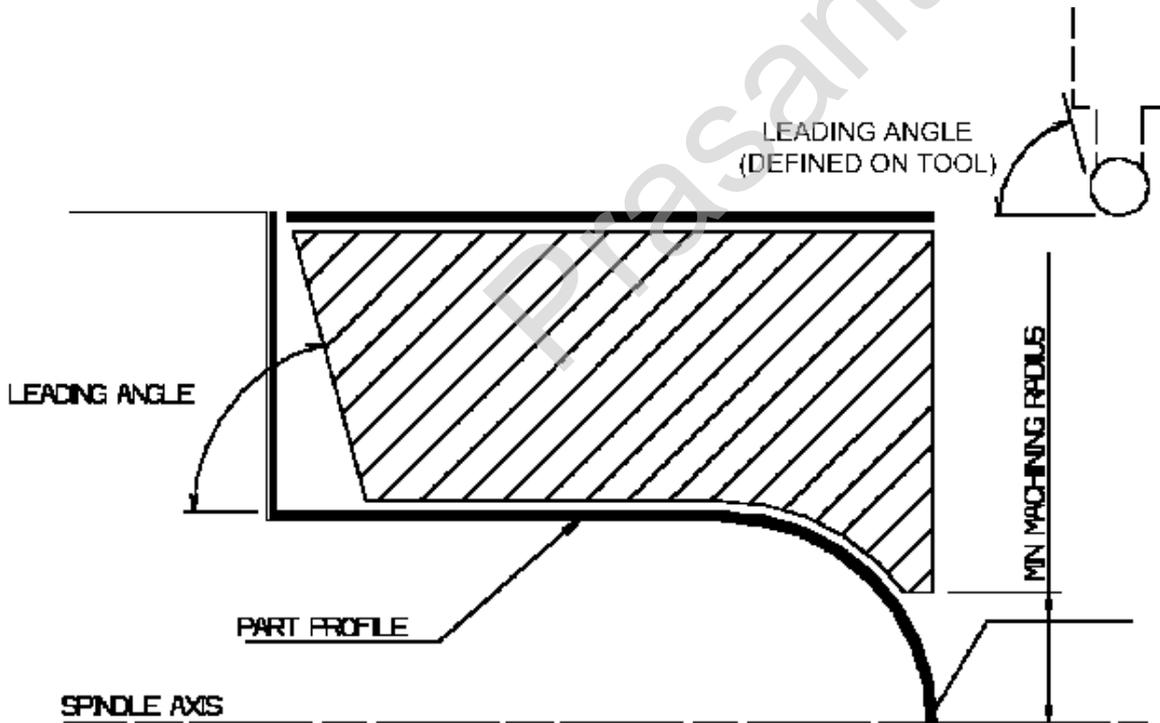
FRONT



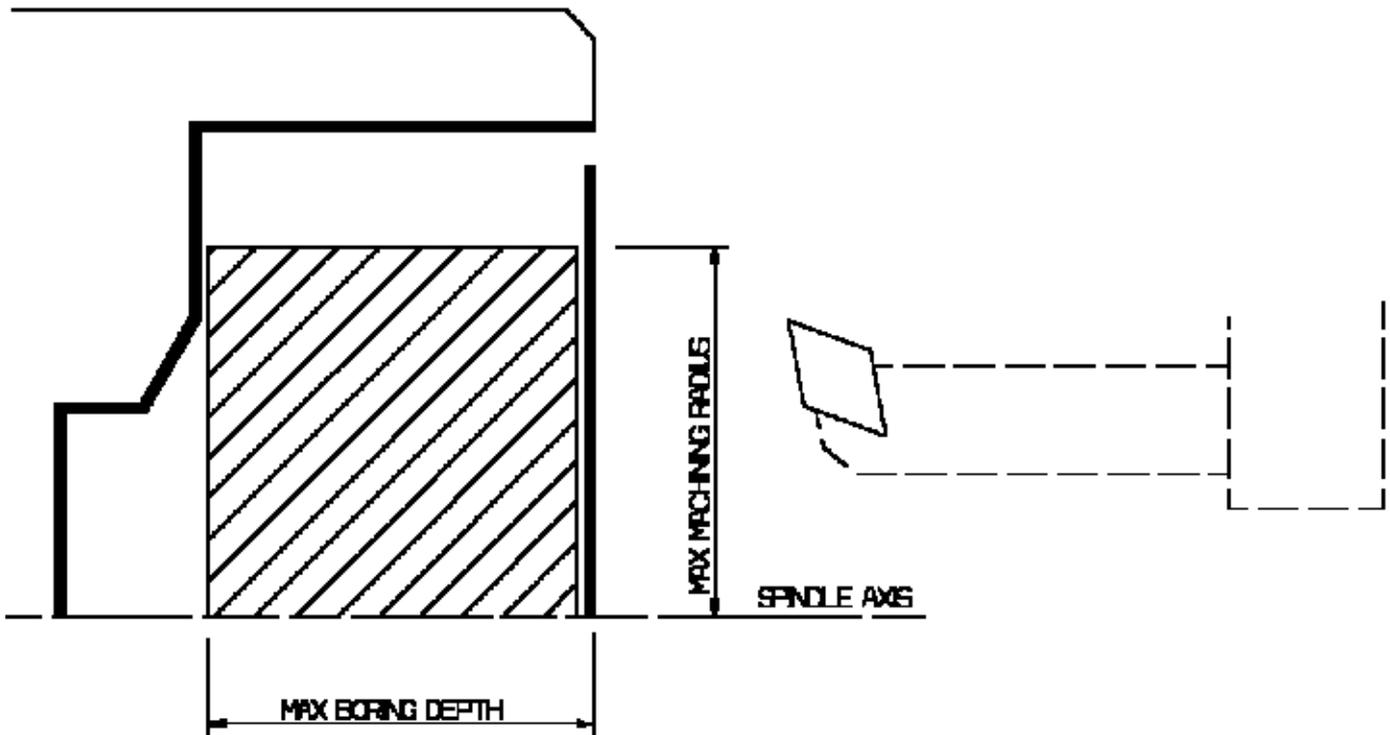
Orientation and Location settings determine the way the program closes the area to machine using radial, axial, axial-radial or radial-axial retraction.

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 this slope.

Note that Trailing Angle can be used only when Recess Machining is set.

Leading and trailing angles can also be defined on the tool to define the maximum slope on which the tool can machine. In this case, 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.

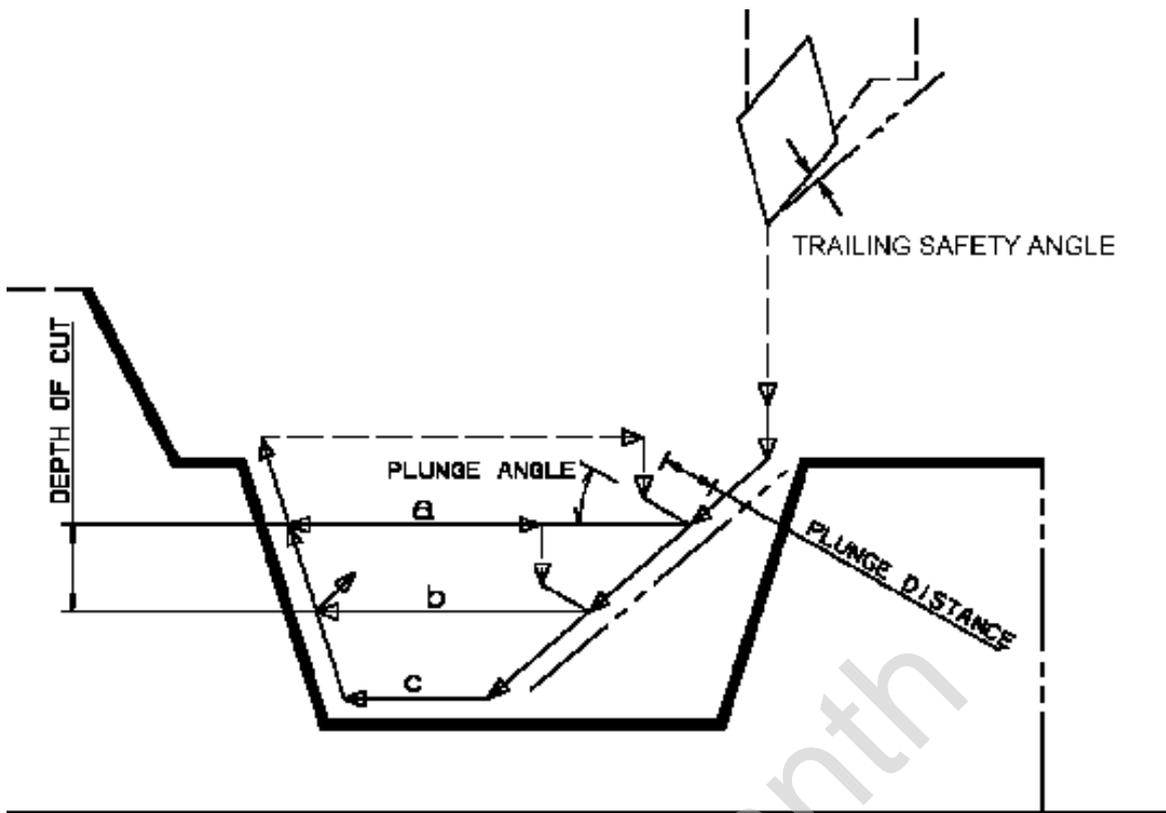
- **Recess Machining** (if Contouring Type is Each Pass or Last Path Only)

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.

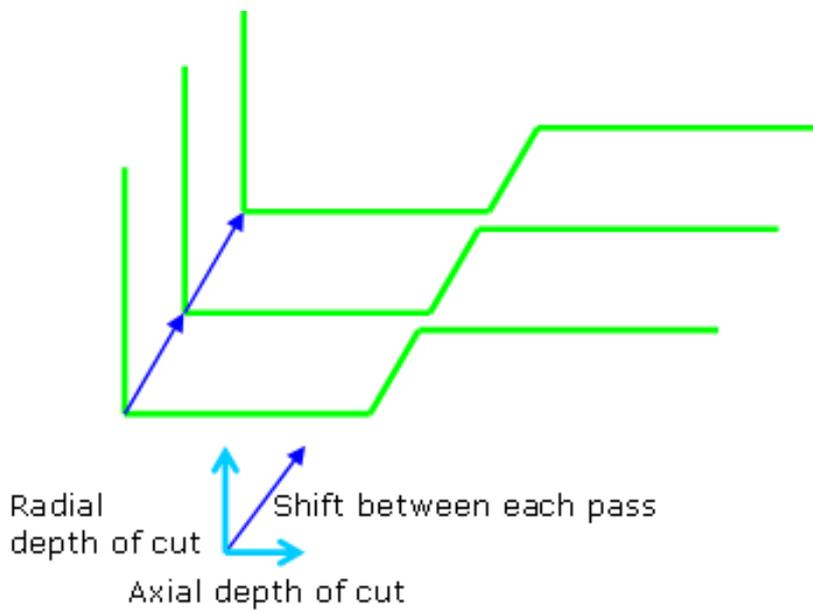
Recommendations for Collision Free Recess Machining in Parallel Contour Rough Turning

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:

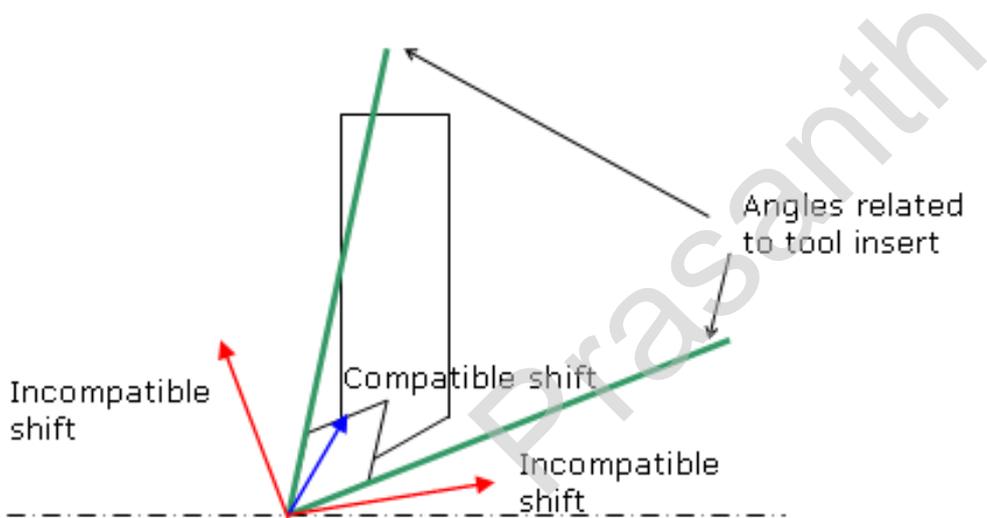
The parallel contours of the toolpath are obtained by first shifting the last pass then shifting each successive pass depending on the axial and radial depth of cut values.



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

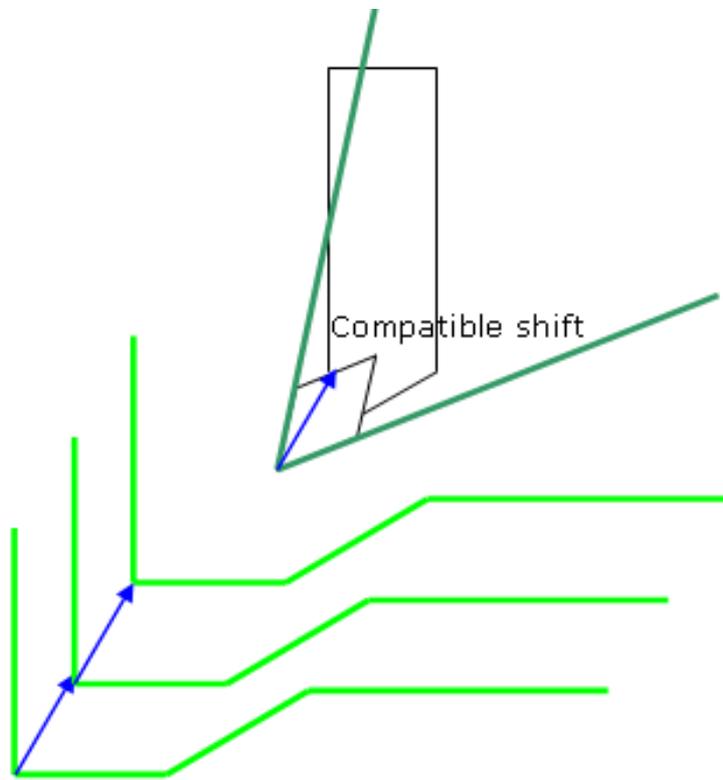
Correct combination of axial and radial depth of cuts

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.

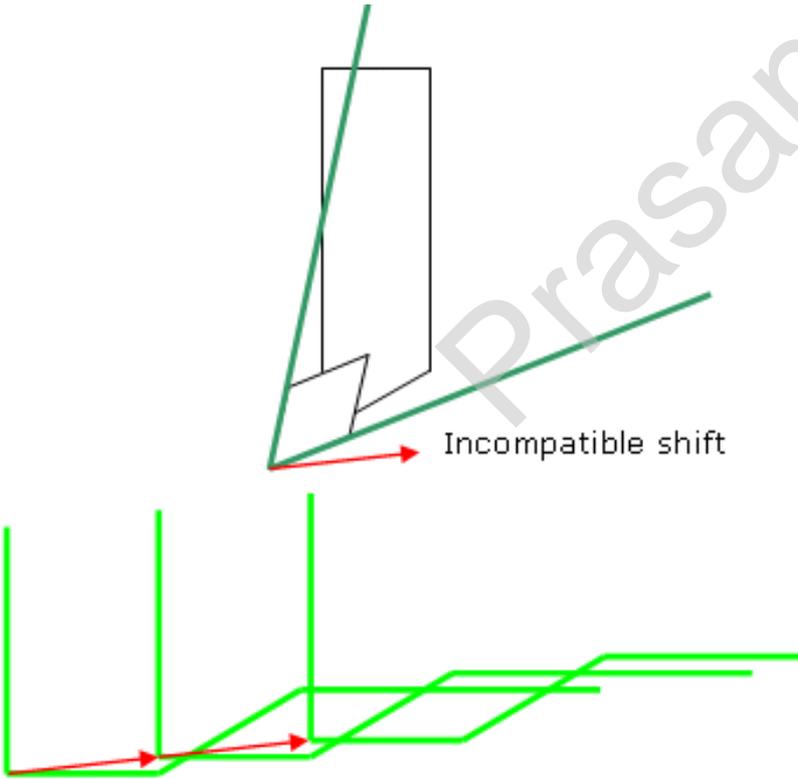


The shift direction must be inside the limit angles defined on the tool insert otherwise a collision may occur. In collision cases, the collision is on the first pass, not on the last passes of the toolpath.

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**
 Defined with respect to the cutting direction and the stock profile with a stock clearance. The tool is in RAPID mode before this distance.

- **Lead-in Angle.**

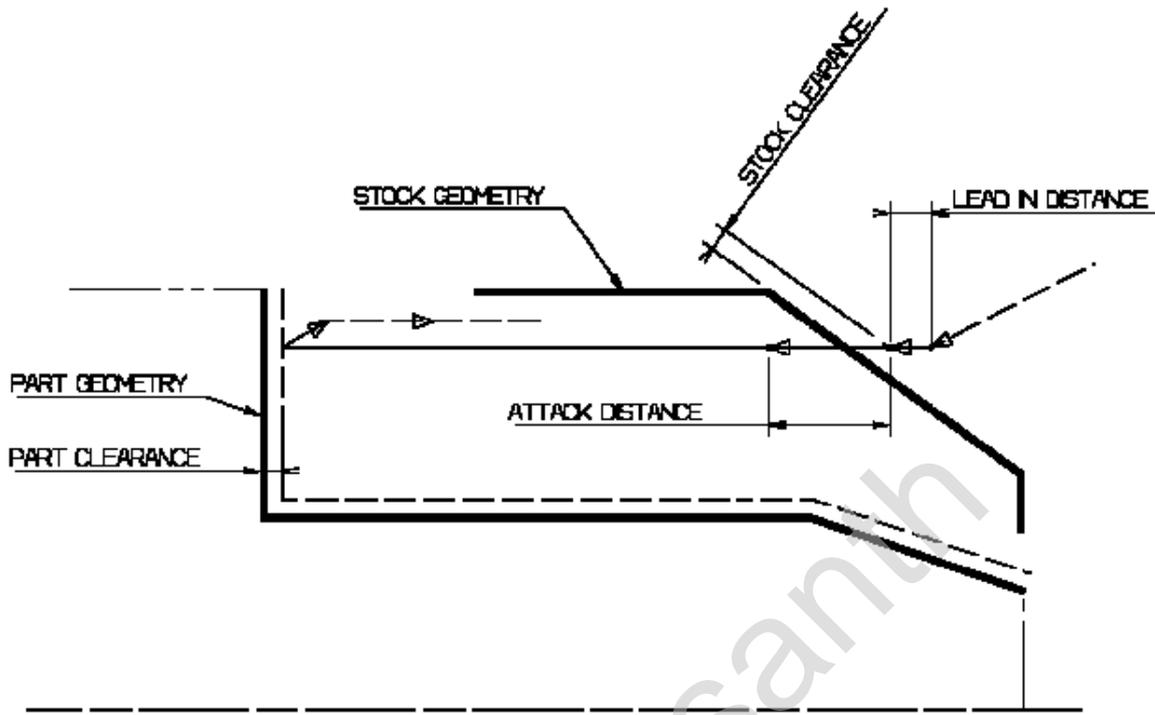
If no lead-in angle is requested, lead-in path is parallel to the machining direction.

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.

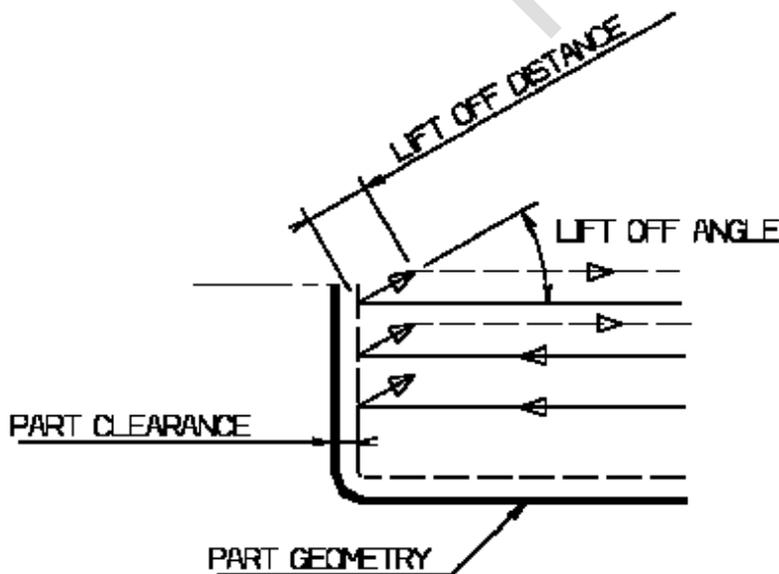
- **Attack Distance**

Defined with respect to the cutting direction and the stock profile with a stock clearance.



- **Lift-off Distance and Lift-off Angle**

Define the lift-off vector at the end of each new 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.

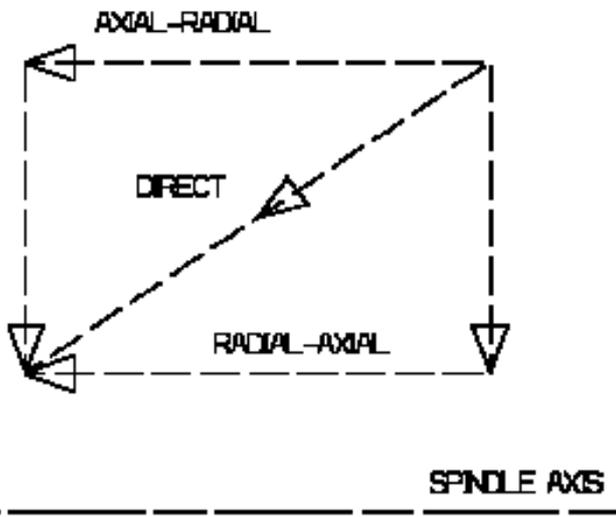
Please note that decimal values can be used for the number of revolutions. For example, when machining big parts that have a large volume, it can be useful to specify a dwell using a value of less than one revolution (0.25, for example).

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 and Retract Macros for Rough Turning

The following Approach and Retract macros are proposed: direct, axial-radial, radial-axial. The selected type (approach or retract) defines the tool motion before or after machining: the tool moves in RAPID mode during the approach or retract.



Linking macros, which comprise retract and approach motion can also be used on Rough Turning operations.

Approach and retract linking motions 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.

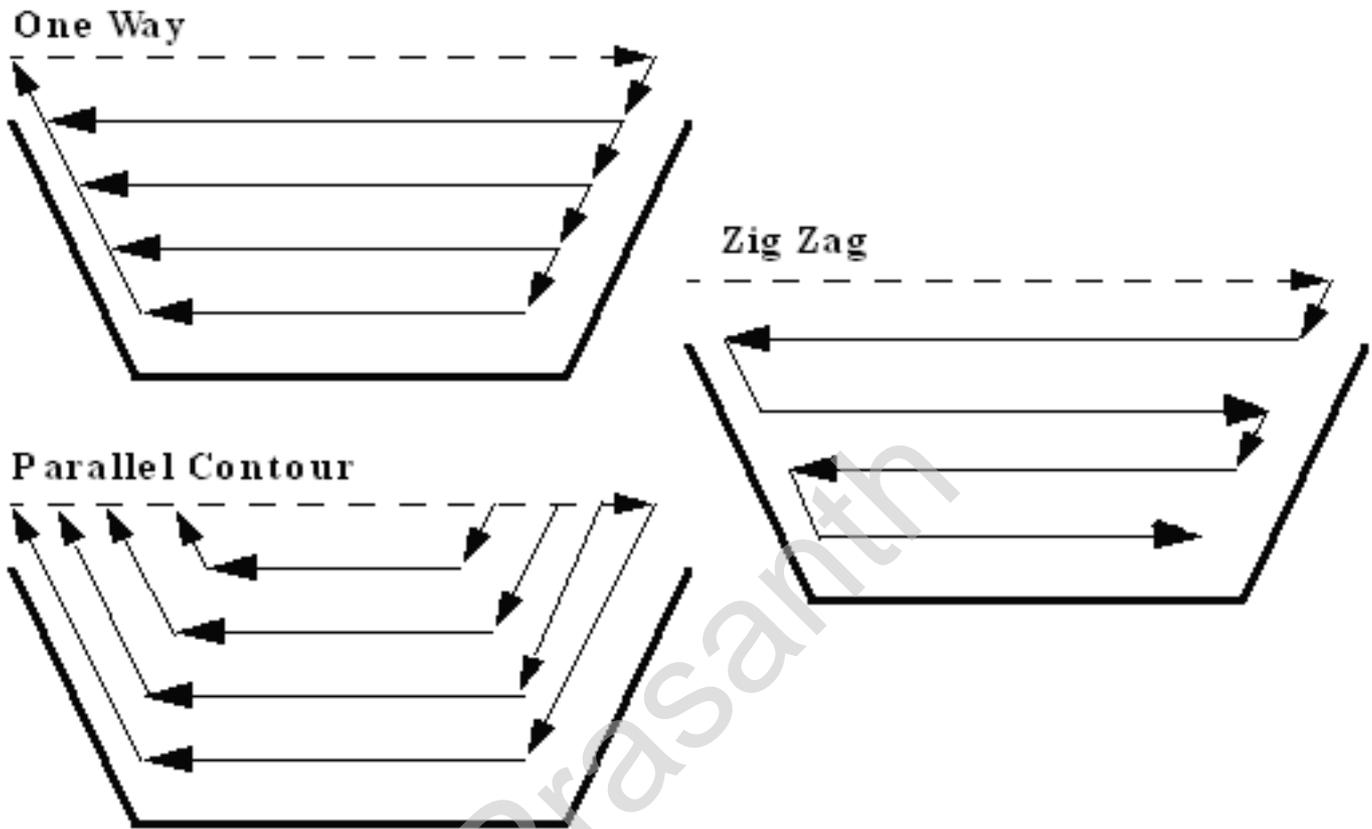
See [Define Macros on a Turning Operation](#) for more information.

Prasanth

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:

- [Tools](#)
- [Geometry](#)
- [Machining Strategy Parameters](#)
- [Feeds and Speeds](#)
- [Tool Compensation](#)

- [Macros](#).

Tools for Recess Turning

The following tools may be used:

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

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. Note that leading and trailing angles can also be defined on the tool to define the maximum slope on which the tool can machine. In this case, 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 tool to define the maximum slope on which the tool can machine. In this case, the angle that reduces the slope the most will be taken into account.

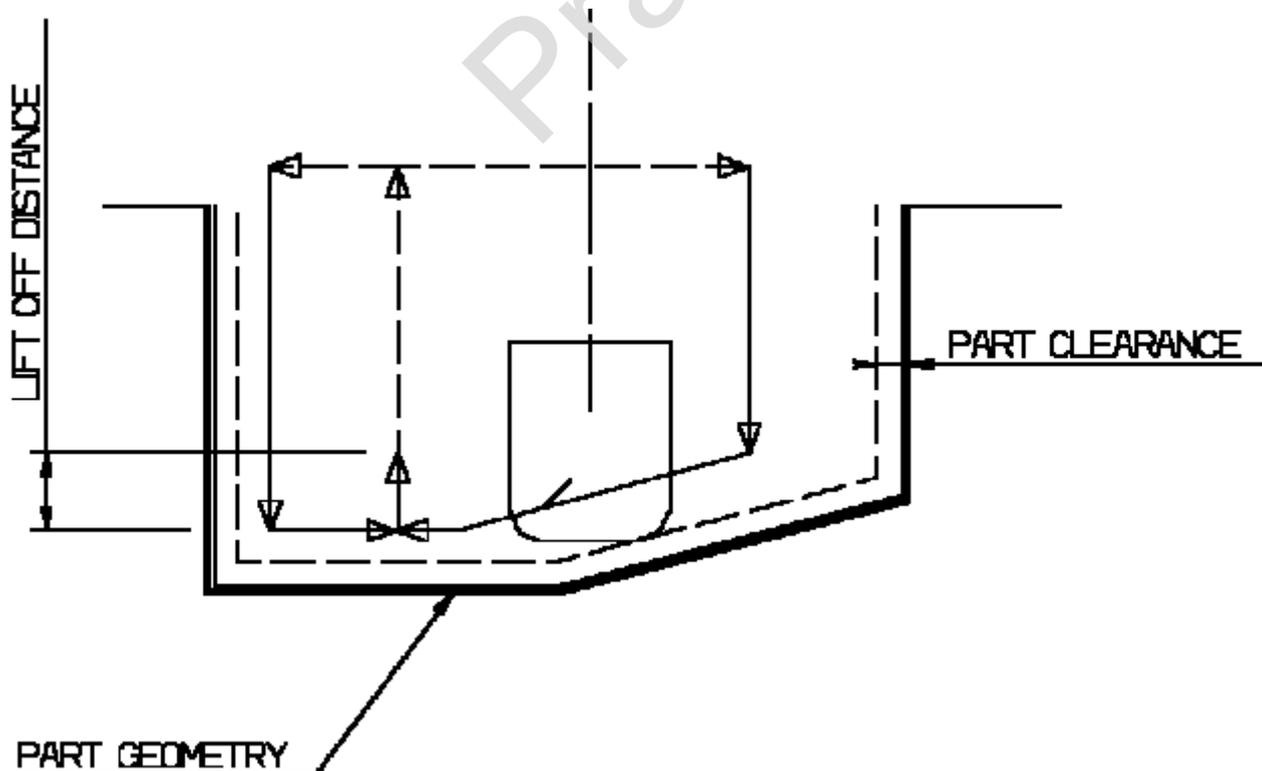
- **Under Spindle Axis Machining**

For Frontal or Inclined machining, this option allows you to request machining under the spindle axis.

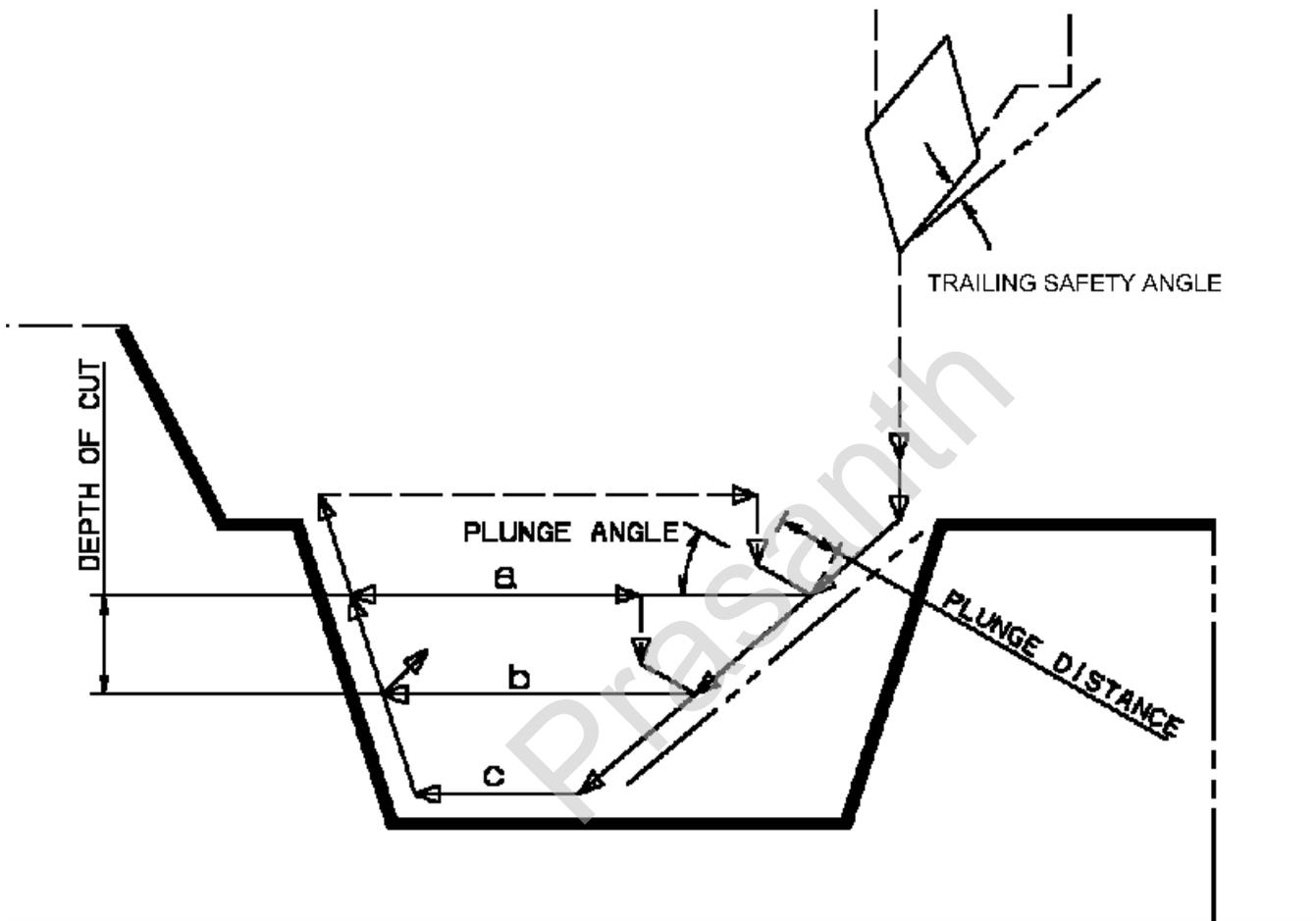
- **Part Contouring**

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 Distance**
Defined with respect to the cutting direction and the stock profile with a stock clearance. 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.



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

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.

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).

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 and Retract Macros for Recess Turning

The following Approach and Retract macros are proposed: direct, axial-radial, radial-axial. The selected type (approach or retract) defines the tool motion before or after machining: the tool moves in RAPID mode during the approach or retract.

Linking macros, which comprise retract and approach motion can also be used on roughing operations.

Approach and retract linking motions 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.

Prasanth

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:

- [Tools](#)
- [Geometry](#)
- [Machining Strategy Parameters](#)
- [Feeds and Speeds](#)
- [Tool Compensation](#)
- [Macros](#).

Tools for Groove Turning

The following tools may be used:

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

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: $(\text{Minimum cut radius} + \text{Maximum cut radius})/2$.
- **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**
Angles of 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 tool to define the maximum slope on which the tool can machine. In this case, 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 and the stock profile with a stock clearance. 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**
Define the lift-off vector at the end of each new pass with respect to the cutting direction.

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 and Retract Macros for Groove Turning

The following Approach and Retract macros are proposed: direct, axial-radial, radial-axial. The selected type (approach or retract) defines the tool motion before or after machining: the tool moves in RAPID mode during the approach or retract.

Linking macros, which comprise retract and approach motion can also be used on roughing operations.

Approach and retract linking motions 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:

- [Tools](#)
- [Geometry](#)
- [Machining Strategy Parameters](#)
- [Feeds and Speeds](#)
- [Tool Compensation](#)
- [Macros](#).

Tools for Profile Finish Turning

The following tools may be used:

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

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.

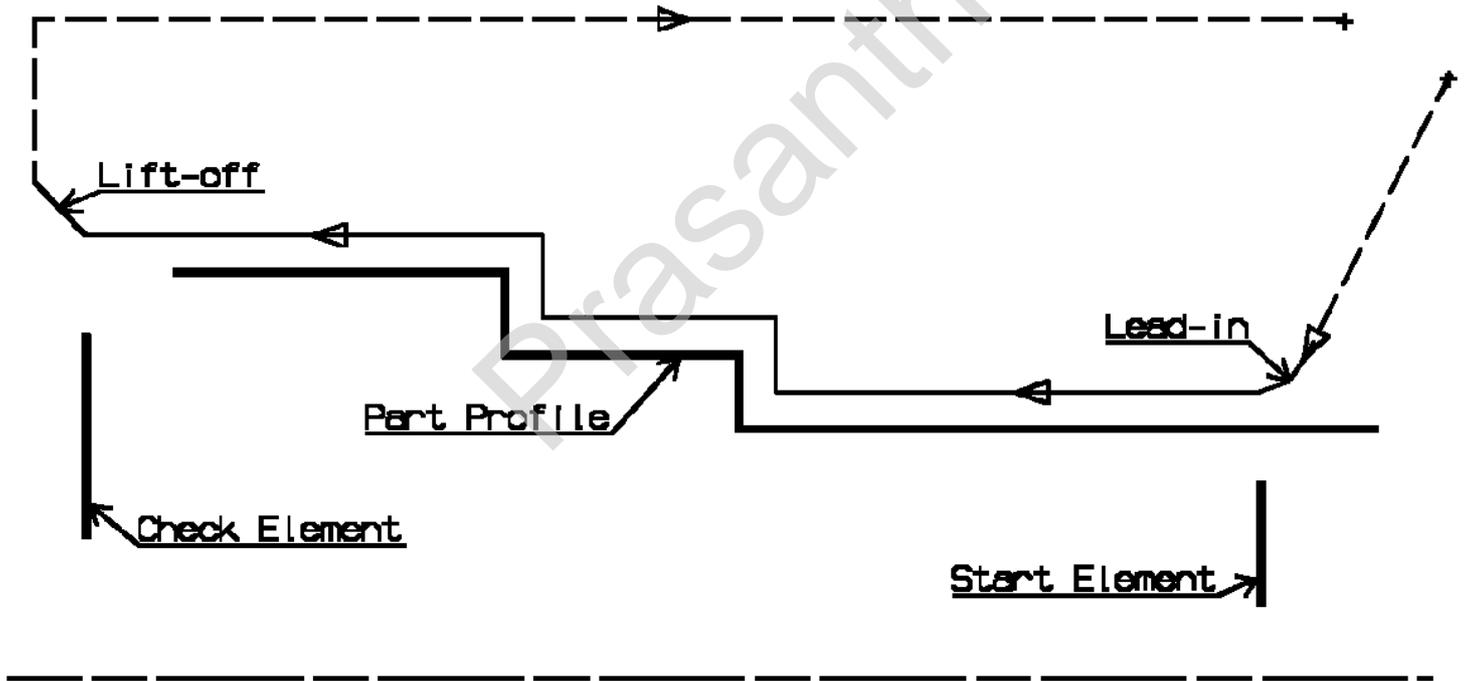
If you specify a line, its intersection with the part profile is calculated (if necessary, the line is extrapolated).

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.

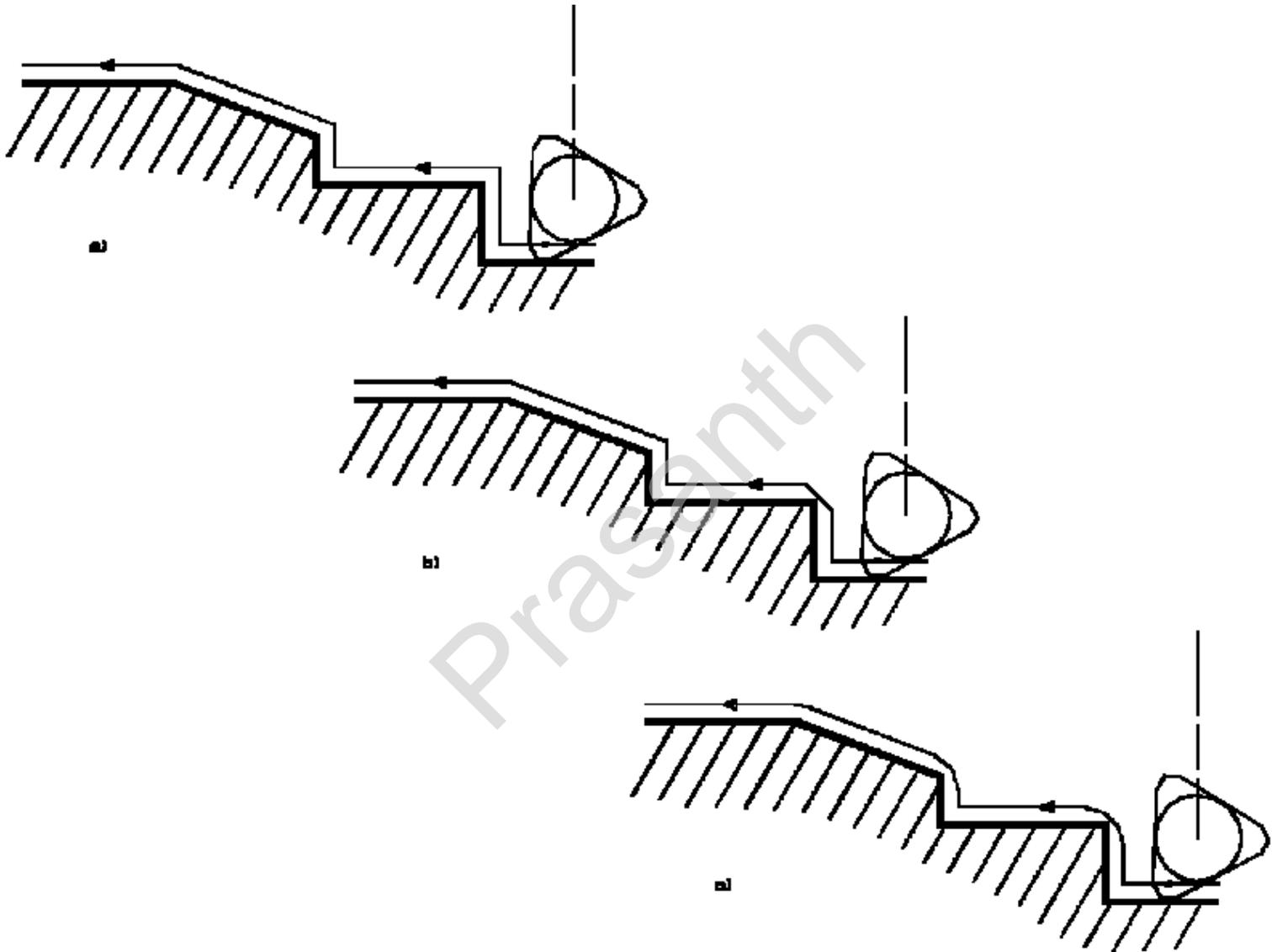
- **Location:**

- 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.

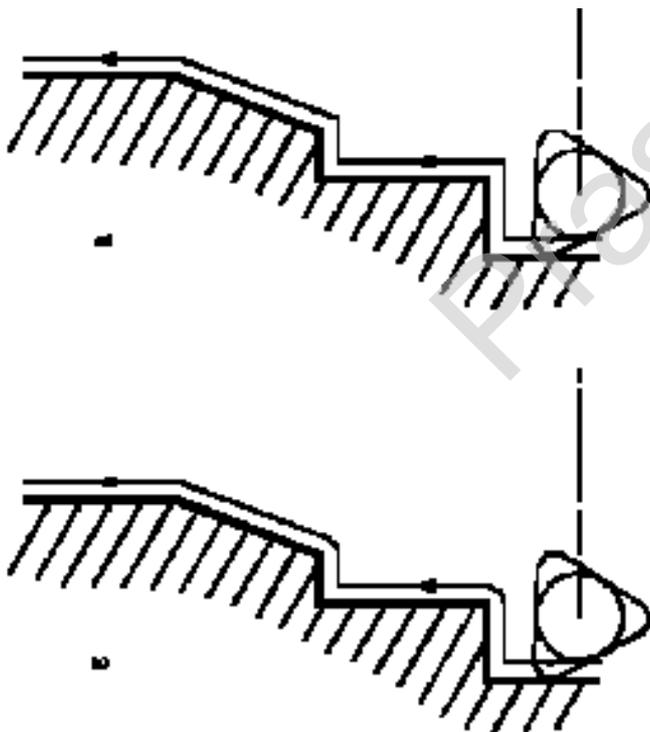
Machining Strategy Parameters for Profile Finish Turning

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. Note that leading and trailing angles can also be defined on the tool to define the maximum slope on which

the tool can machine. In this case, the angles that reduce the slope the most will be taken into account.

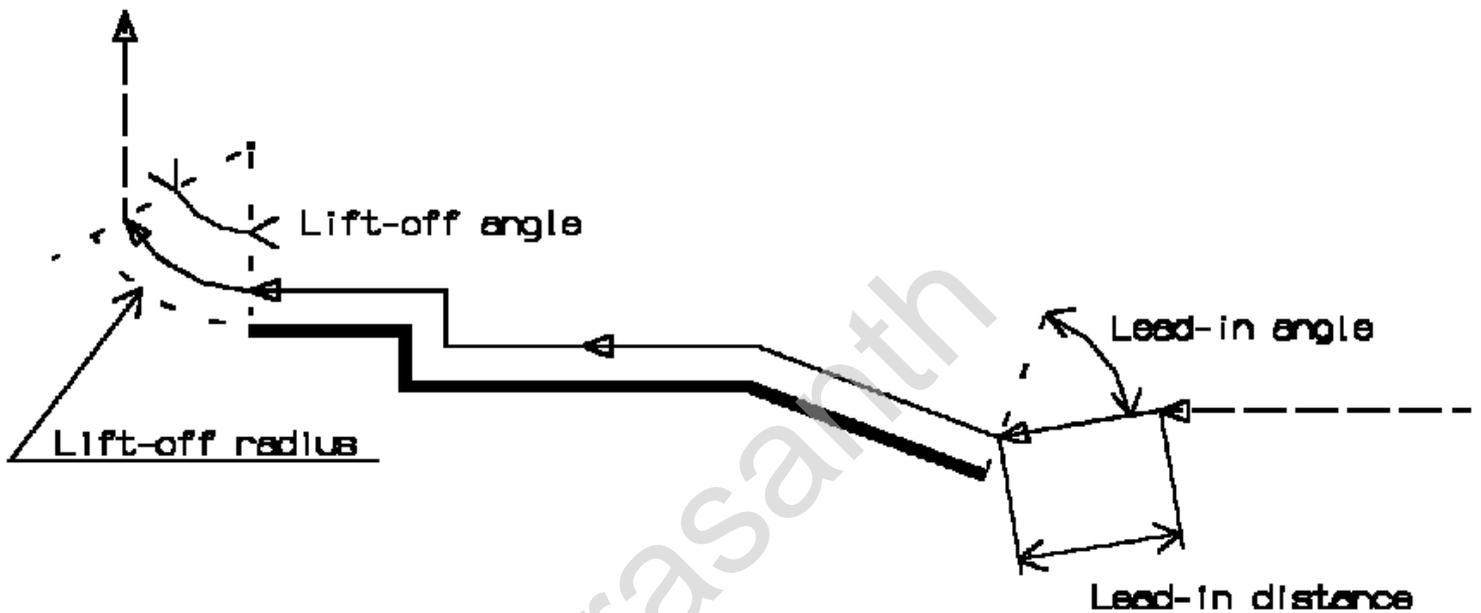
- **Machining Tolerance** for following the profile.

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** options.
- 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** options.



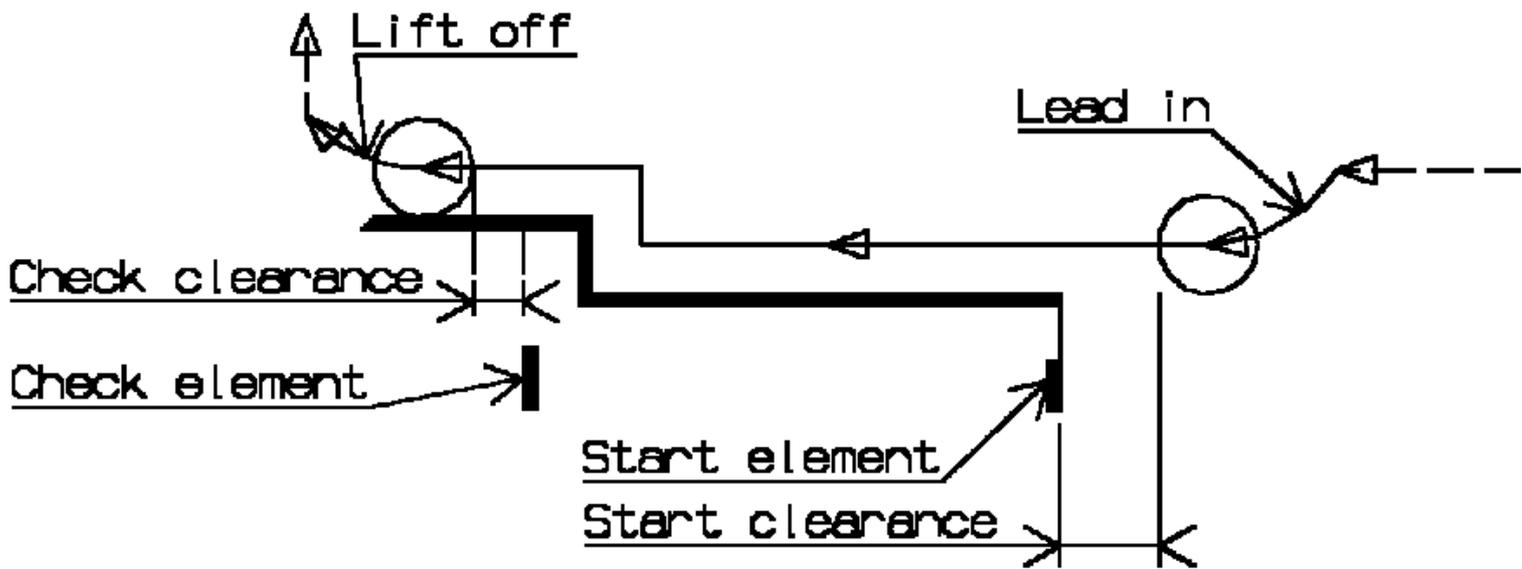
Linear lead-in and circular lift-off (Profile Finish Turning)

- **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** options.
- 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** options.

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. The selected type (approach or retract) defines the tool motion before or after machining: the tool moves in RAPID mode during the approach or retract.

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:

- [Tools](#)
- [Geometry](#)
- [Machining Strategy Parameters](#)
- [Feeds and Speeds](#)
- [Tool Compensation](#)
- [Macros](#).

Tools for Groove Finish Turning

The following tools may be used:

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

Geometry

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

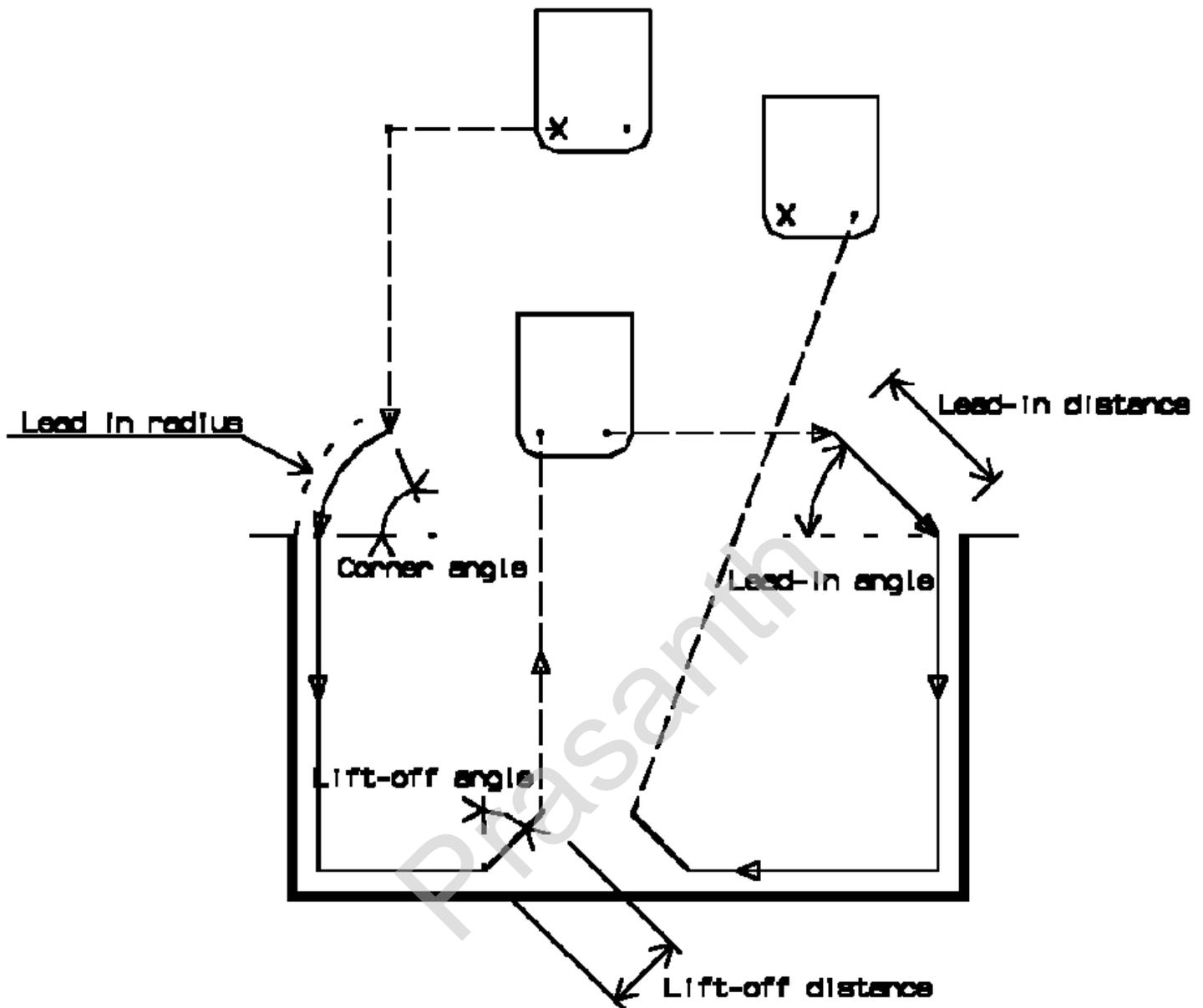
- **First Flank Lead-in: Linear / Circular**

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** options.
- **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** options.

The example below shows Linear lead-in and Circular lift-off for Groove Finish Turning.



• **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** options.
- 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** options.

• **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.

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.

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: direct, axial-radial, radial-axial. The selected type (approach or retract) defines the tool motion before or after machining: the tool moves in RAPID mode during the approach or retract.

Prasanth

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:

- [Tools](#)
- [Geometry](#) including [Part and Stock Offsets](#)
- [Location and Limits](#)
- [Machining Strategy Parameters](#)
- [Feeds and Speeds](#)
- [Tool Compensation](#)
- [Macros](#).

Tools for Ramp Rough Turning

The following tools may be used:

- External and Internal tools with round inserts.

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).

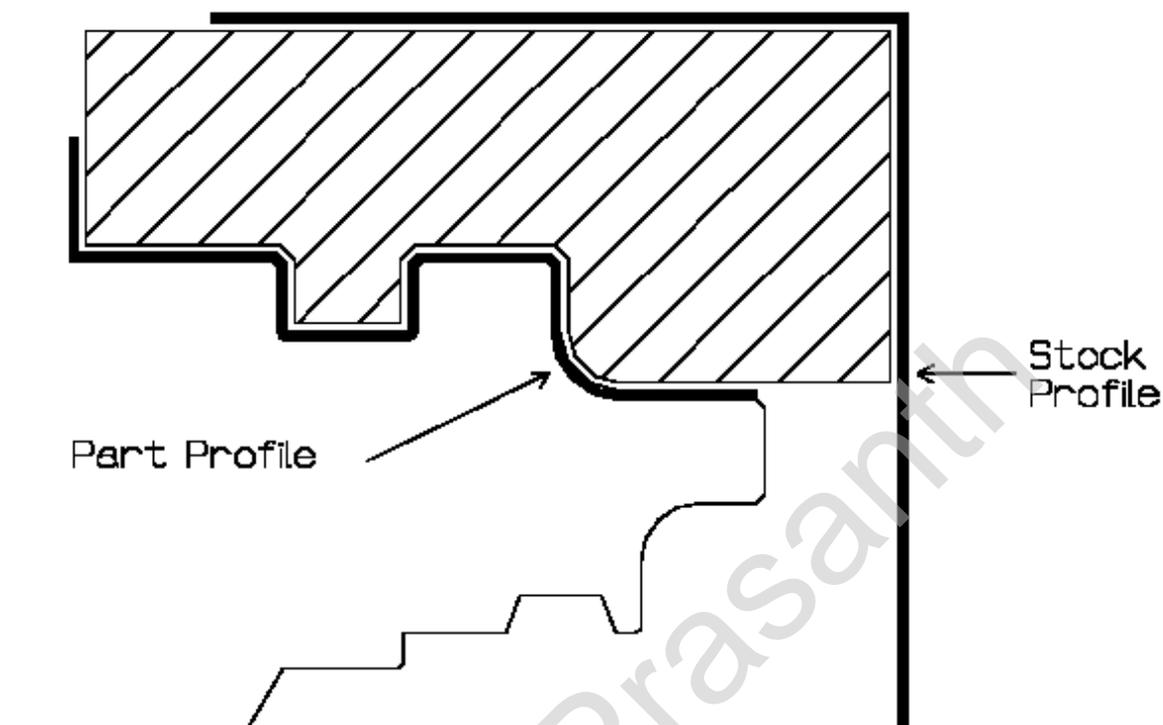
A line through the intersection point parallel to the radial axis delimits the area to machine.

Orientation for Ramp Rough Turning

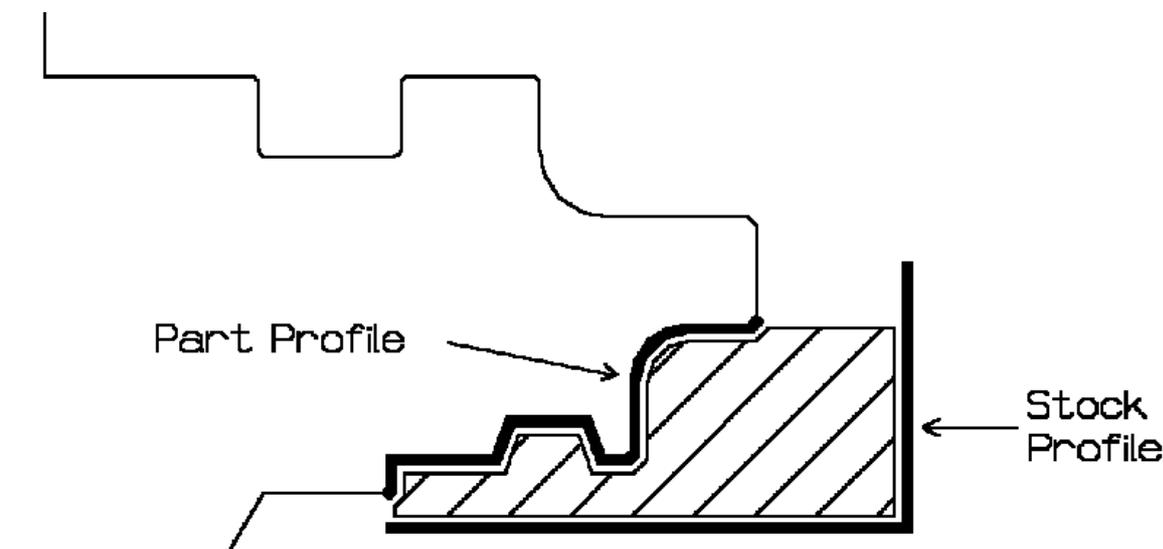
The following **Orientations** are proposed: Internal, External and Frontal (for Face Roughing only).

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

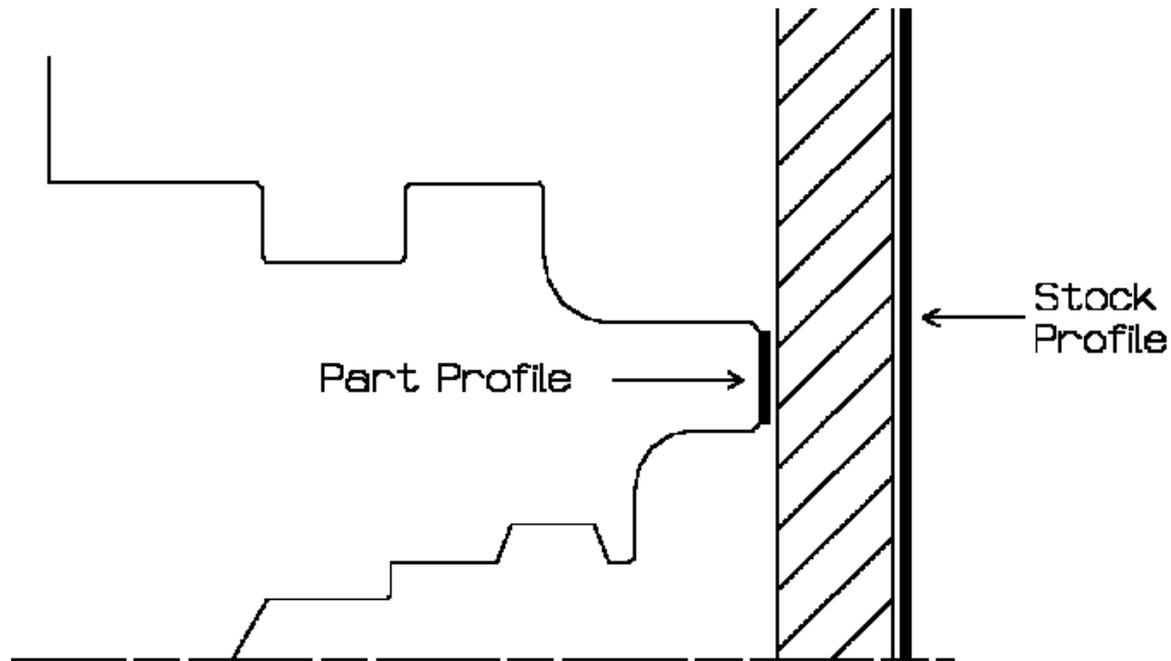


Frontal Ramp Rough Turning

Frontal machining is proposed for face Ramp Rough Turning. In that case, the minimum and maximum diameters of the

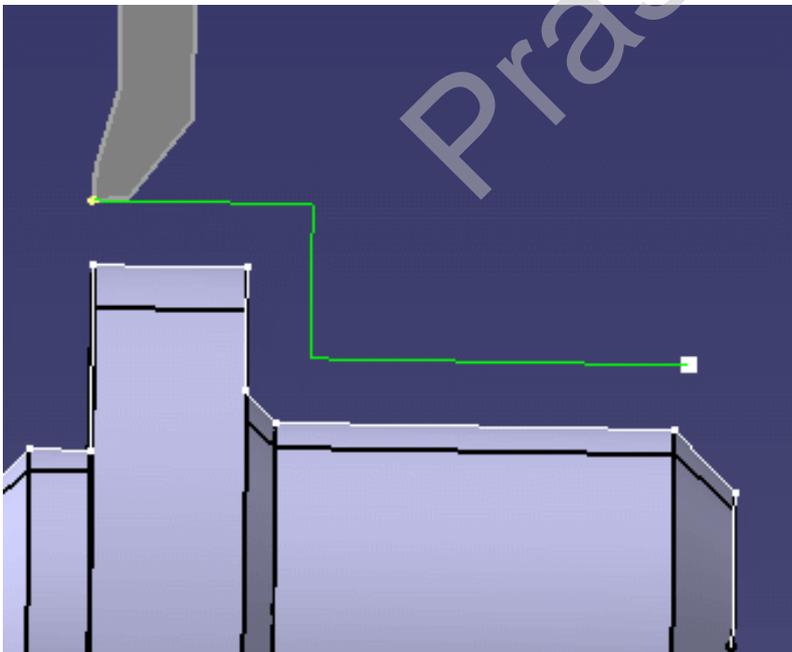
area to machine are determined by the stock profile dimensions.

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

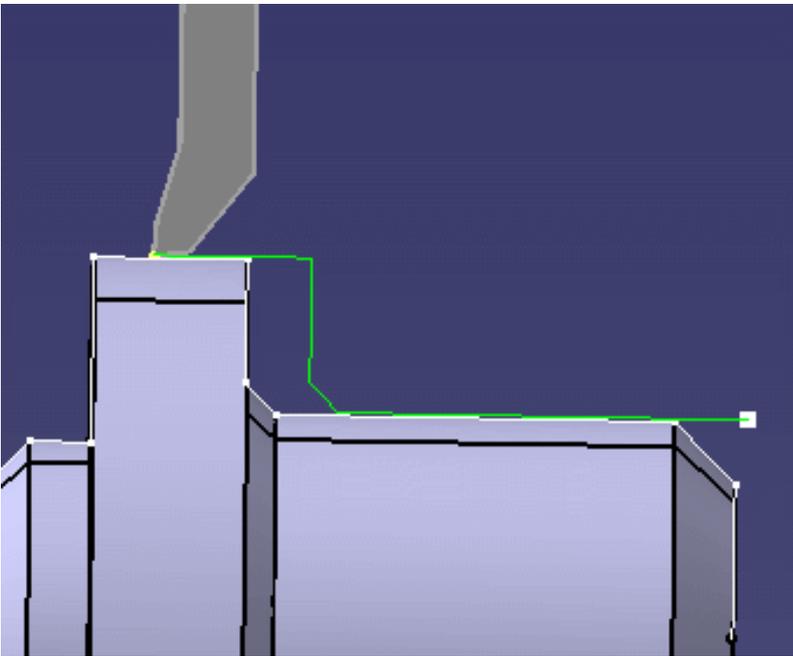


Part and Stock Offsets for Ramp Rough Turning

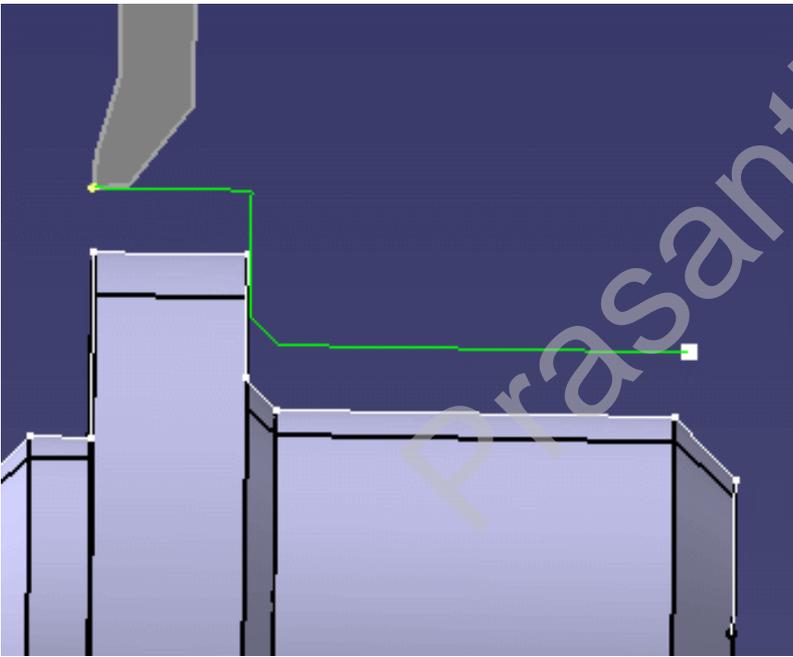
- **Stock offset:** specifies a virtual displacement of the stock profile.
- **Part offset:** specifies a virtual displacement of the part profile.



- **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.



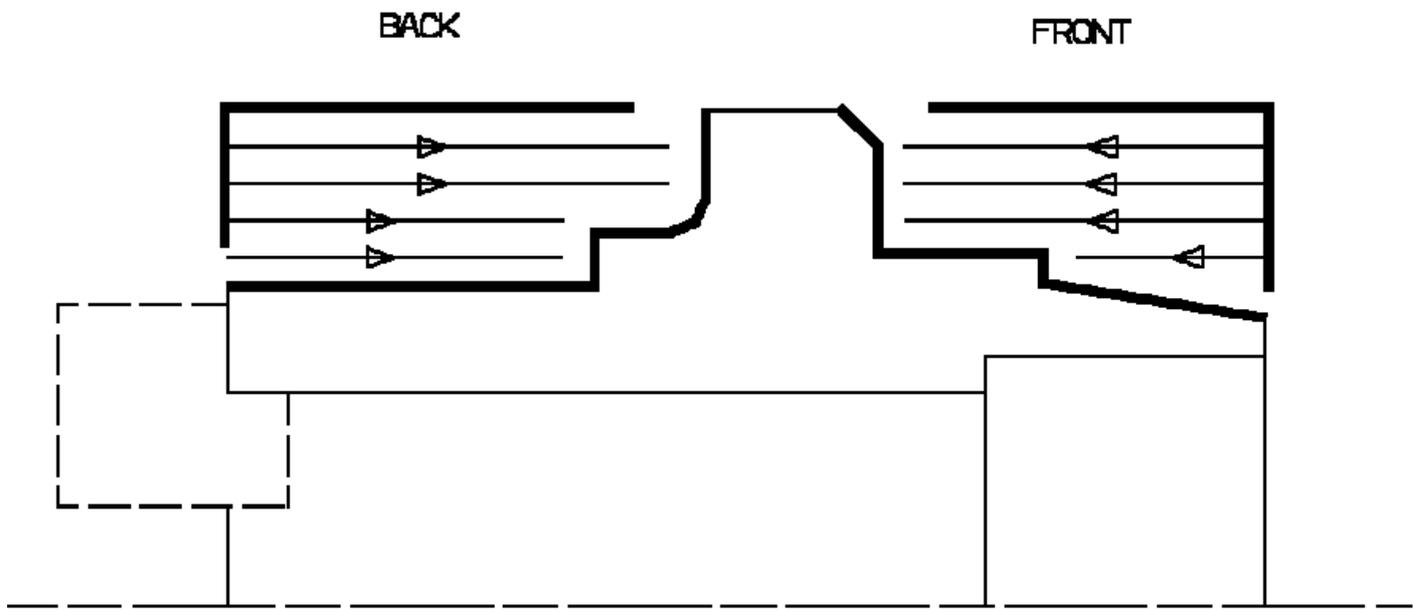
- **End limit offset:** distance with respect to the end element (only if end element is a line or a curve, and when In or Out is set for end element positioning).

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

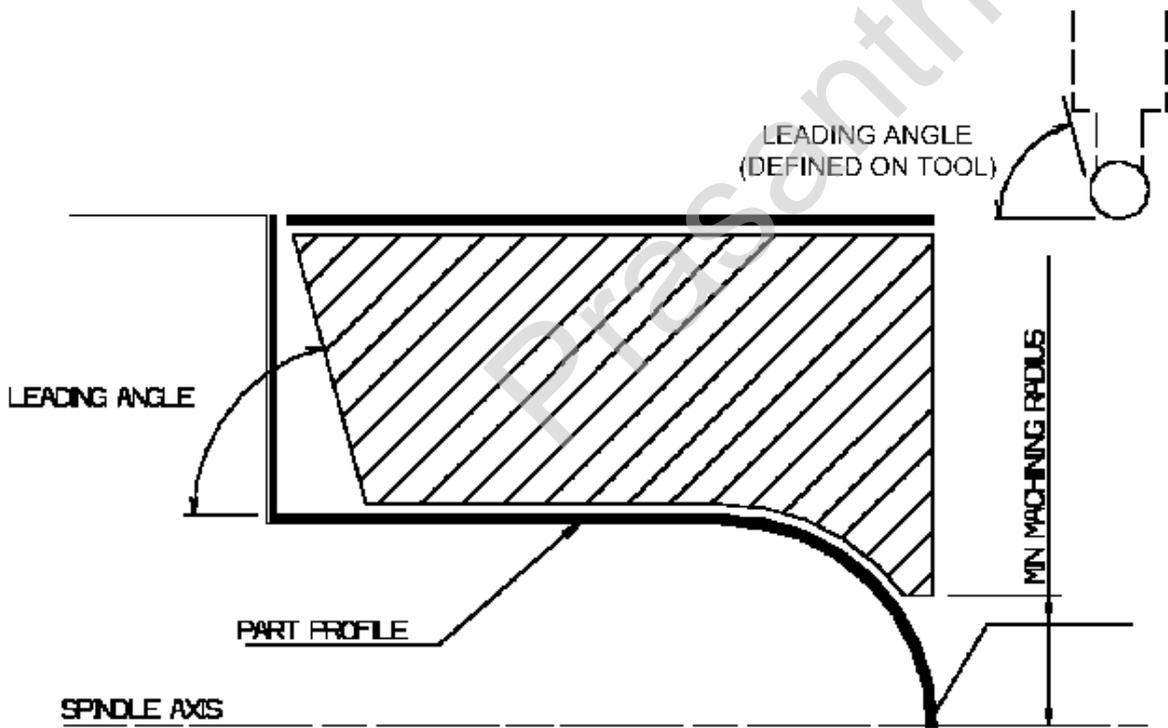
The following machining **Locations** are proposed:

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

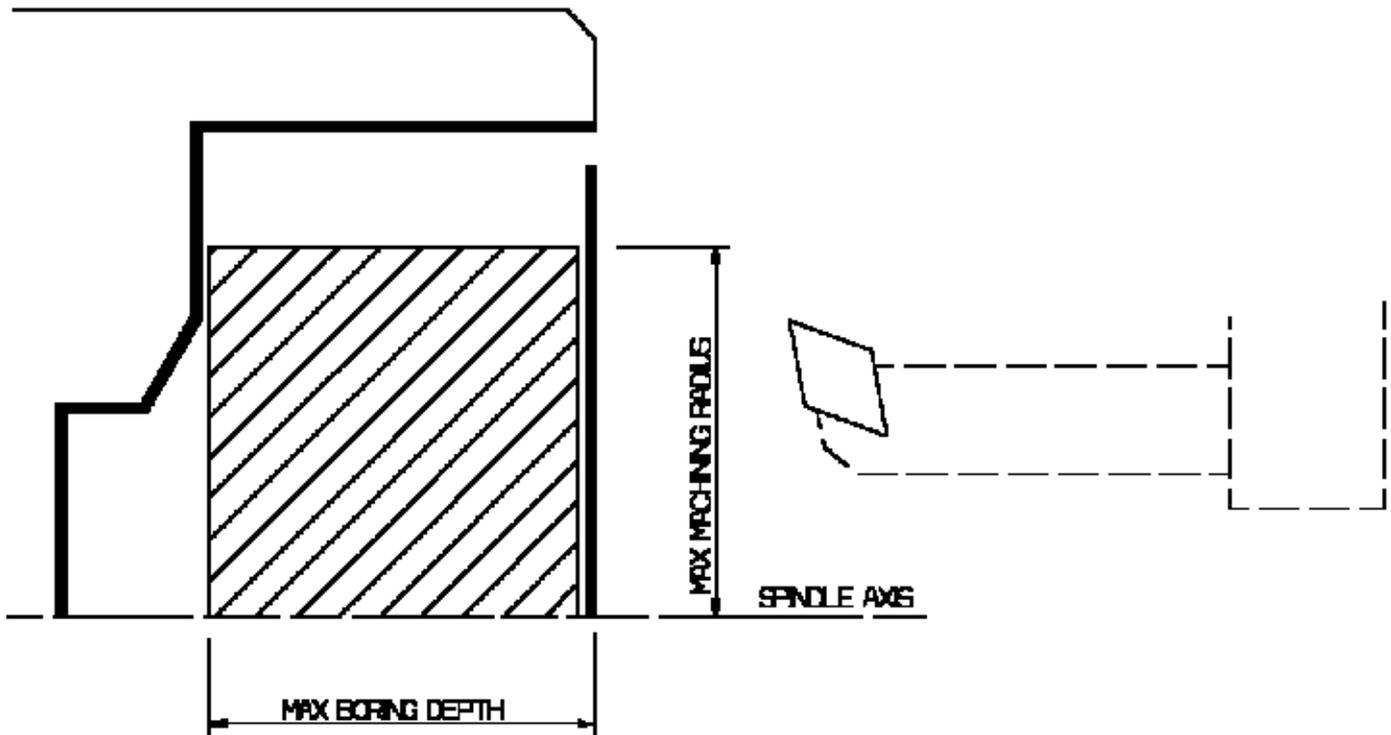


Orientation and Location settings determine the way the program closes the area to machine. The following options allow you to restrict the area to machine that is pre-defined by the stock and part. You may want to restrict this area due to the physical characteristics of the tool and the type of machining 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

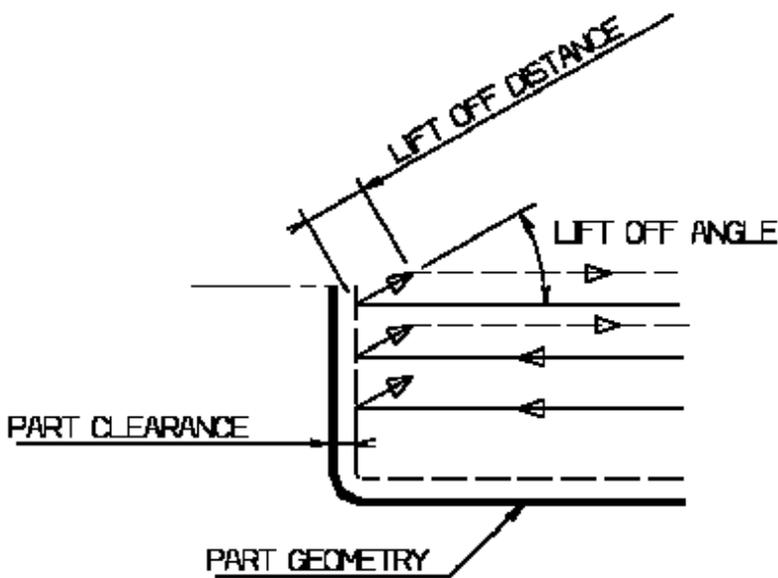
Path Definition for Ramp Rough Turning

- **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.
- **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 and the stock profile with a stock clearance. The tool is in RAPID mode before this distance.
- **Lift-off Distance** and **Lift-off Angle**
Define the lift-off vector at the end of each new 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).

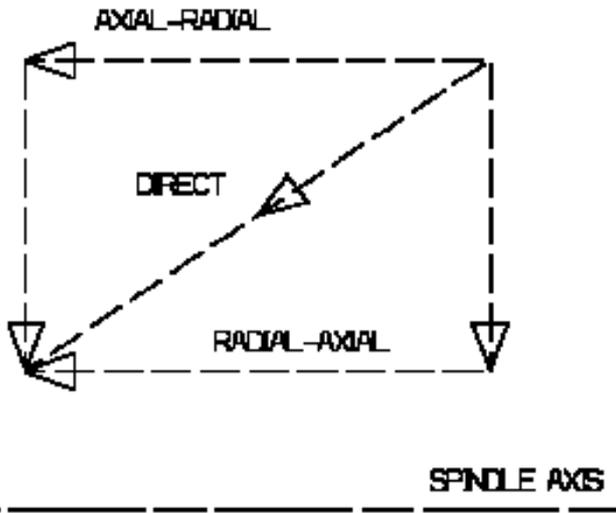
Tool Compensation for Ramp Rough Turning

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 and Retract Macro Motions for Ramp Rough Turning

The following predefined Approach and Retract motions are proposed: direct, axial-radial, radial-axial. The selected type

(approach or retract) defines the tool motion before or after machining.



Linking macros, which comprise retract and approach motion can also be used on roughing operations.

Approach and retract linking motions 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.

You can also build your own macros from elementary paths.

See [Define Macros on a Turning Operation](#) for more information.

Prasanth

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](#).

Tools for Ramp Recess Turning

The following tools may be used:

- External and Internal tools with round inserts.

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**.

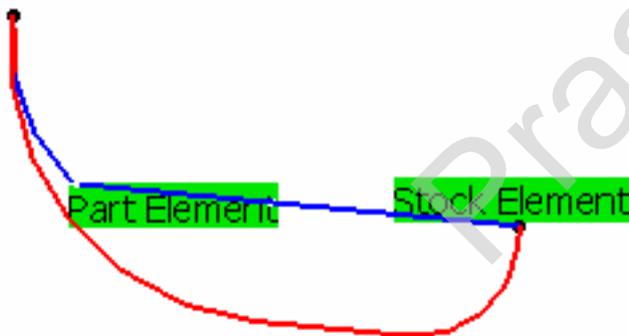
Part and Stock Offsets for Ramp Recess Turning

- **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.

- **Leading and Trailing Safety Angles** for One way mode.

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. Note that leading and trailing angles can also be defined on the tool to define the maximum slope on which the tool can machine. In this case, 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 tool to define the maximum slope on which the tool can machine. In this case, the angle that reduces the slope the most will be taken into account.

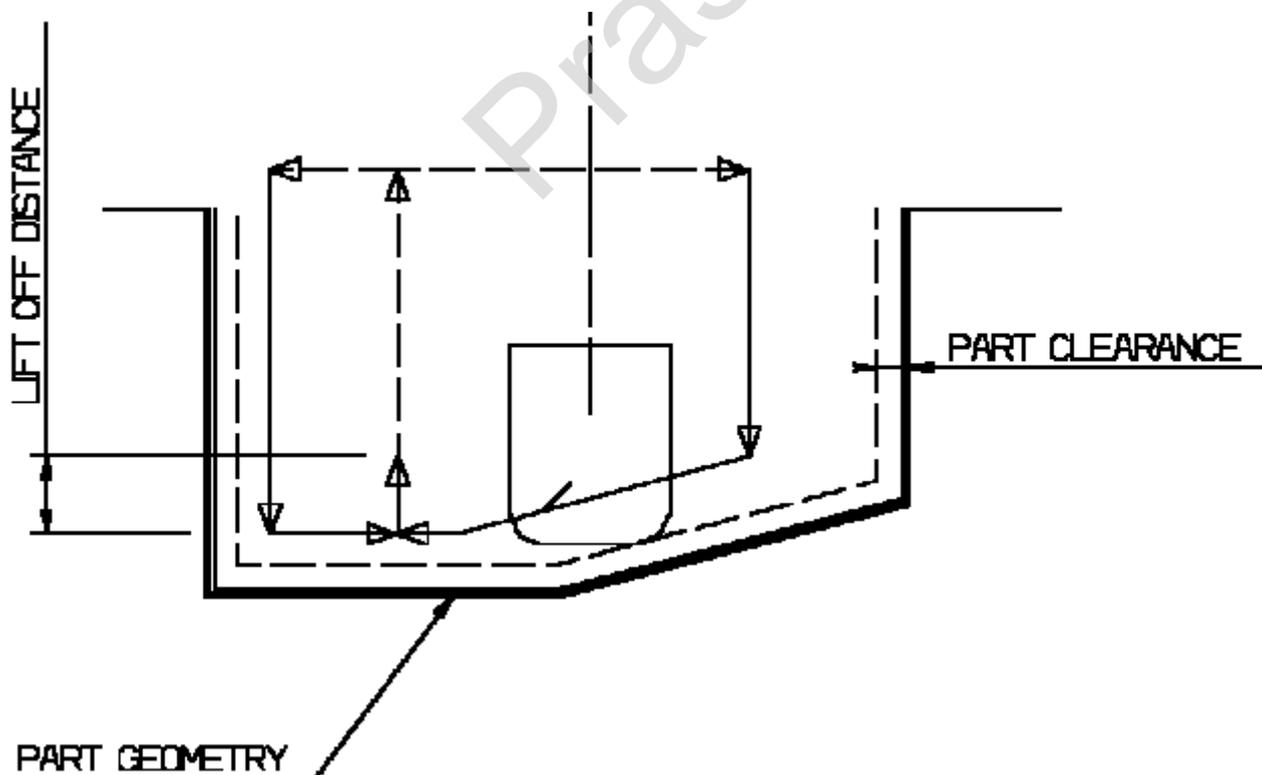
- **Under Spindle Axis Machining**

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.
- 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 and Retract Macro Motions for Ramp Recess Turning

The following predefined Approach and Retract macros are proposed: direct, axial-radial, radial-axial. The selected type (approach or retract) defines the tool motion before or after machining.

Linking macros, which comprise retract and approach motion can also be used on roughing operations. Approach and retract linking motions 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. You can also build your own macros from elementary paths.

See [Define Macros on a Turning Operation](#) for more information.

Prasanth

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:

- [Tools](#)
- [Geometry](#)
- [Machining Strategy Parameters](#)
- [CYCLE or GOTO Output Syntaxes](#)
- [Editing CYCLE Syntaxes](#)
- [Tool Compensation](#)
- [Macros](#).

Tools for Thread Turning

The following tools may be used:

- Internal and External threading tools 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**
- **Lift-off Distance** and **Lift-off Angle**.

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.

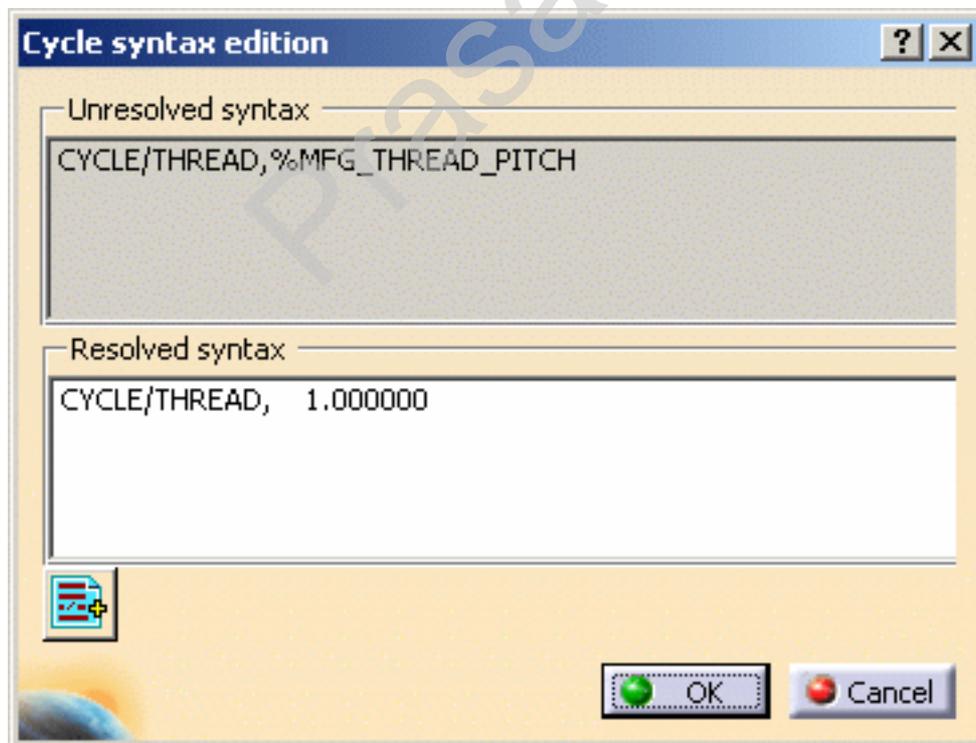
The parameters available for PP word syntaxes for this type of operation are described in the [NC_LATHE_THREADING section of the Manufacturing Infrastructure User's Guide](#).

Editing CYCLE Syntaxes

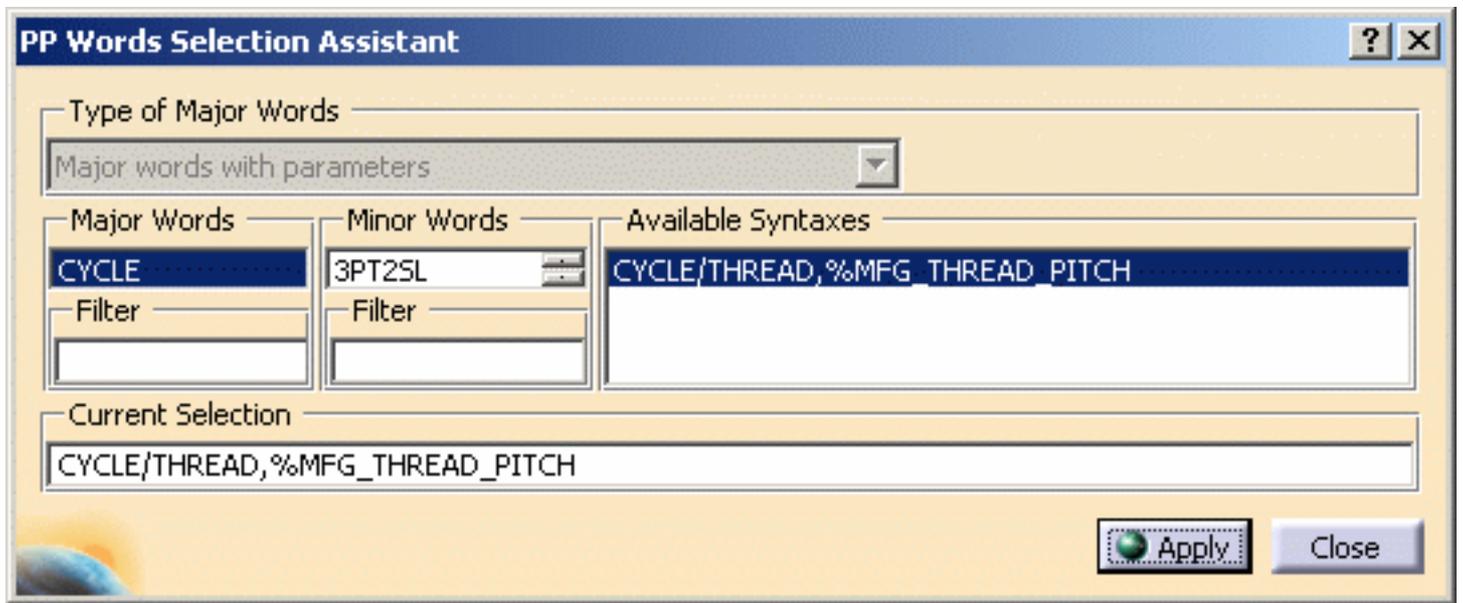
The **Edit Cycle** command in the Thread Turning dialog box allows you to:

- 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. The selected type (approach or retract) defines the tool motion before or after machining: the tool moves in RAPID mode during the approach or retract.

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 panel, you are prompted to create a start position. This start position must be a Go motion that uses either a point or two different check elements.

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:

- [Tools](#)
- [Geometry](#)
- [Go Motion](#)
- [Go InDirv Motion](#)
- [Go Delta Motion](#)
- [Follow Motion](#)
- [Feeds and Speeds.](#)

Tools for Sequential Turning

All tool types with compatible inserts are authorized.

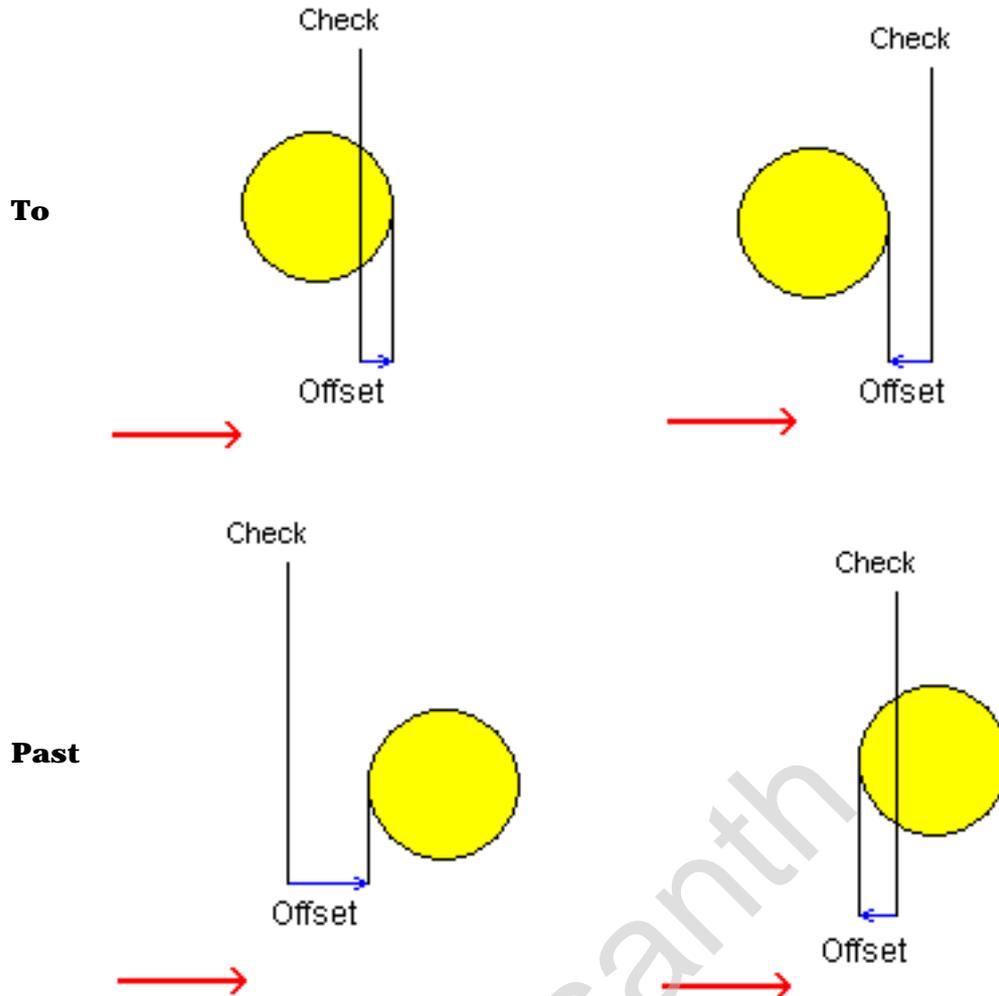
Geometry Selection for Sequential Turning

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.

Lathe Tool Assembly Conventions

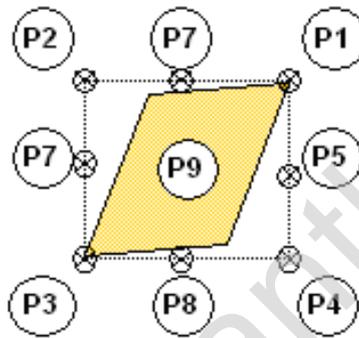
General Comments

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) 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.

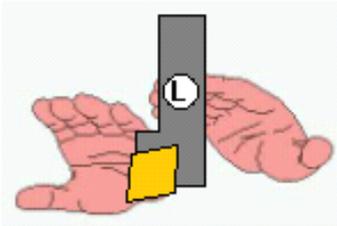


Examples below deal with Horizontal Lathe Machine Tool.

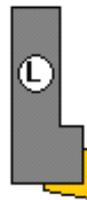
Insert Holder

Different settings of the Setup Angle, Invert Tool and Hand Style parameters result in different configurations for positioning the tool .

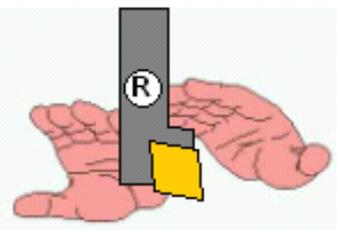
Left Tool: Insert is visible and points to left:



Inverted Left Tool :



Right Tool: Insert is visible and points to right:

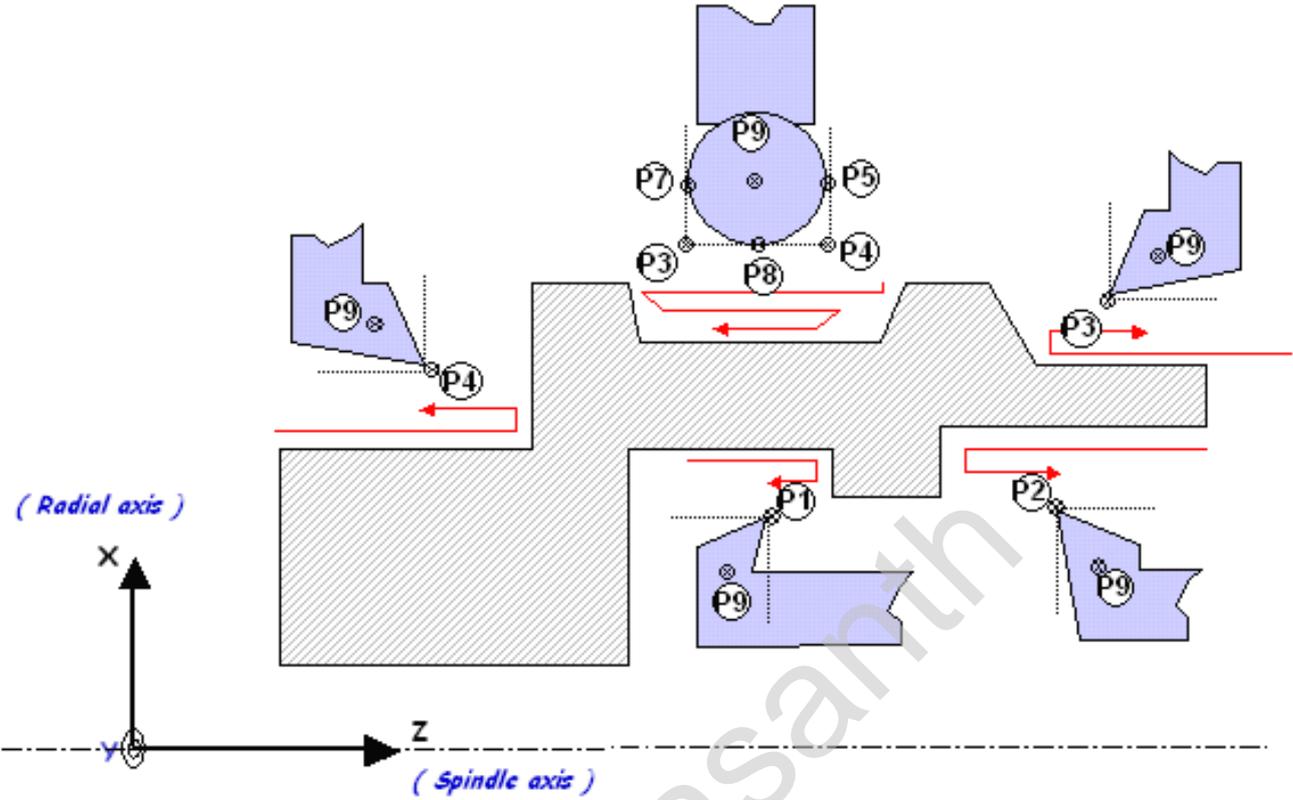


Inverted Right Tool:



Output Point

In the following figure reference is made according to the Spindle/Radial Axis (Z/X)

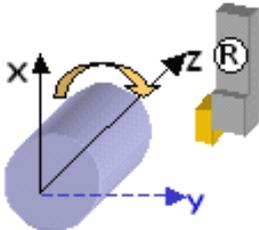


Spindle Rotary Direction

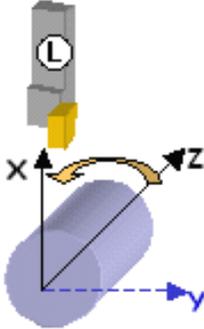
With a Horizontal Lathe Machine Tool the convention for spindle rotary direction defined in the Machine Editor dialog box is as follows:

Looking into Z direction, Chuck & Jaws at your back.

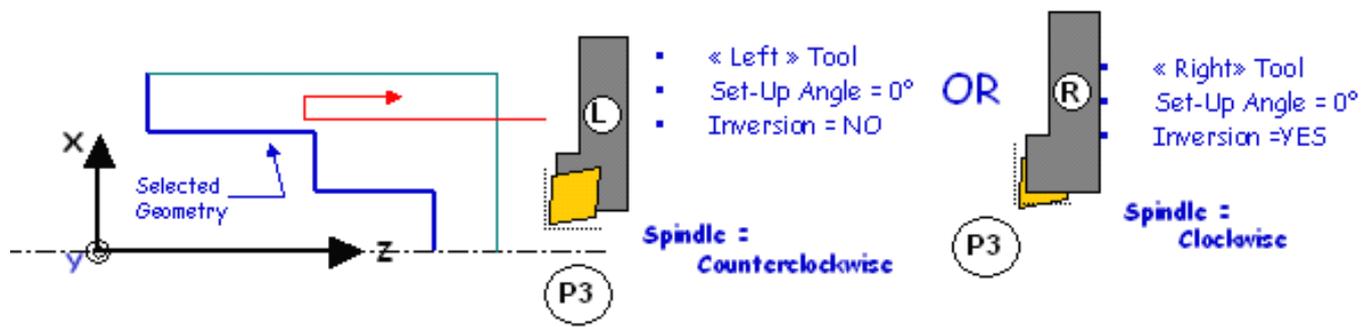
Clockwise



Counterclockwise

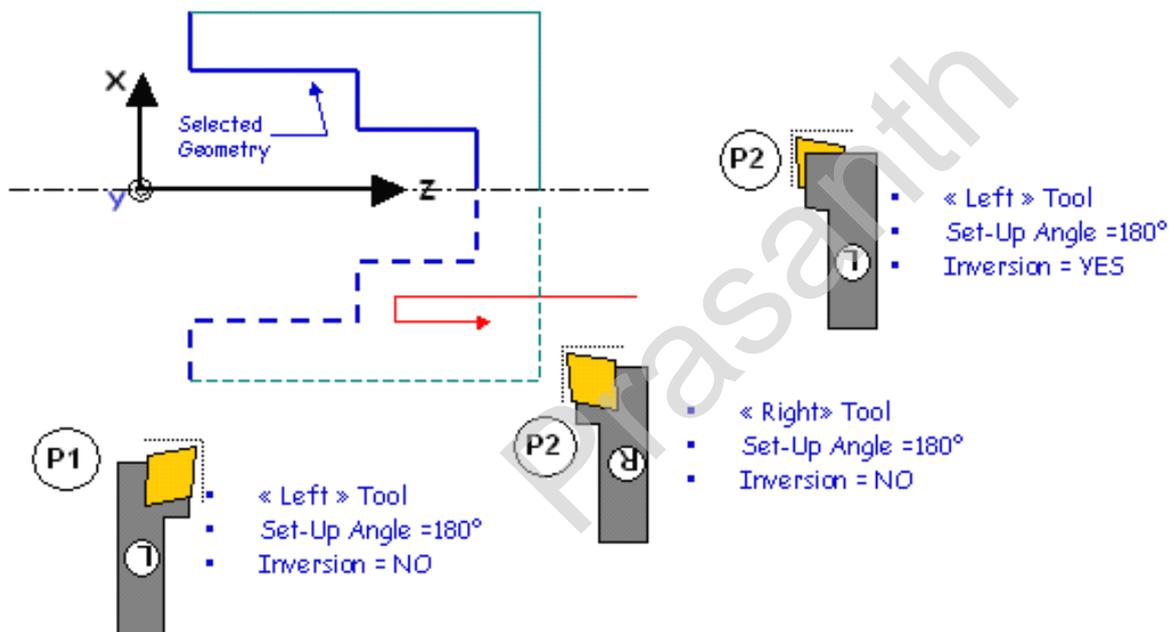


The following represent common settings:



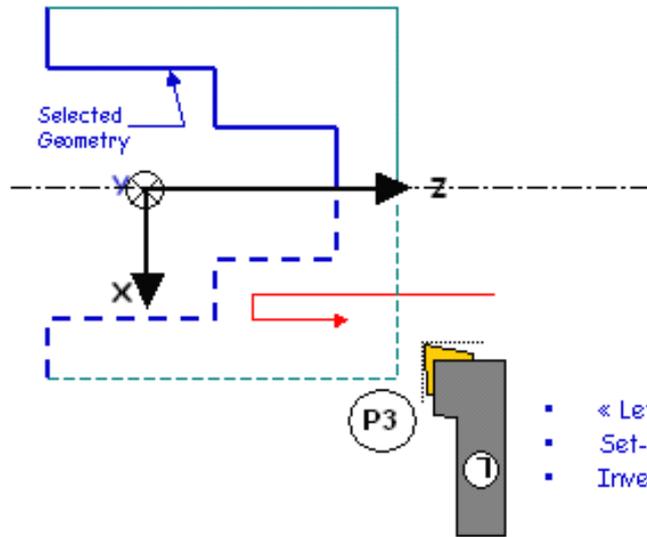
180deg Setup Angle

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-).

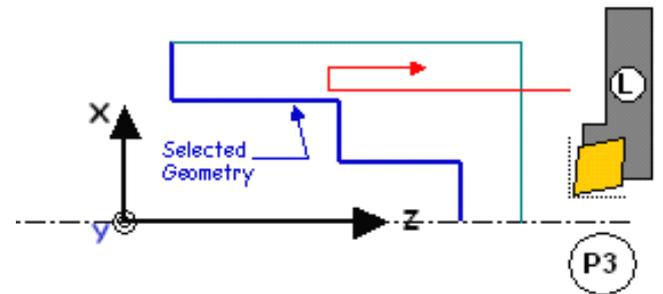


Direct Machining Axis

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).



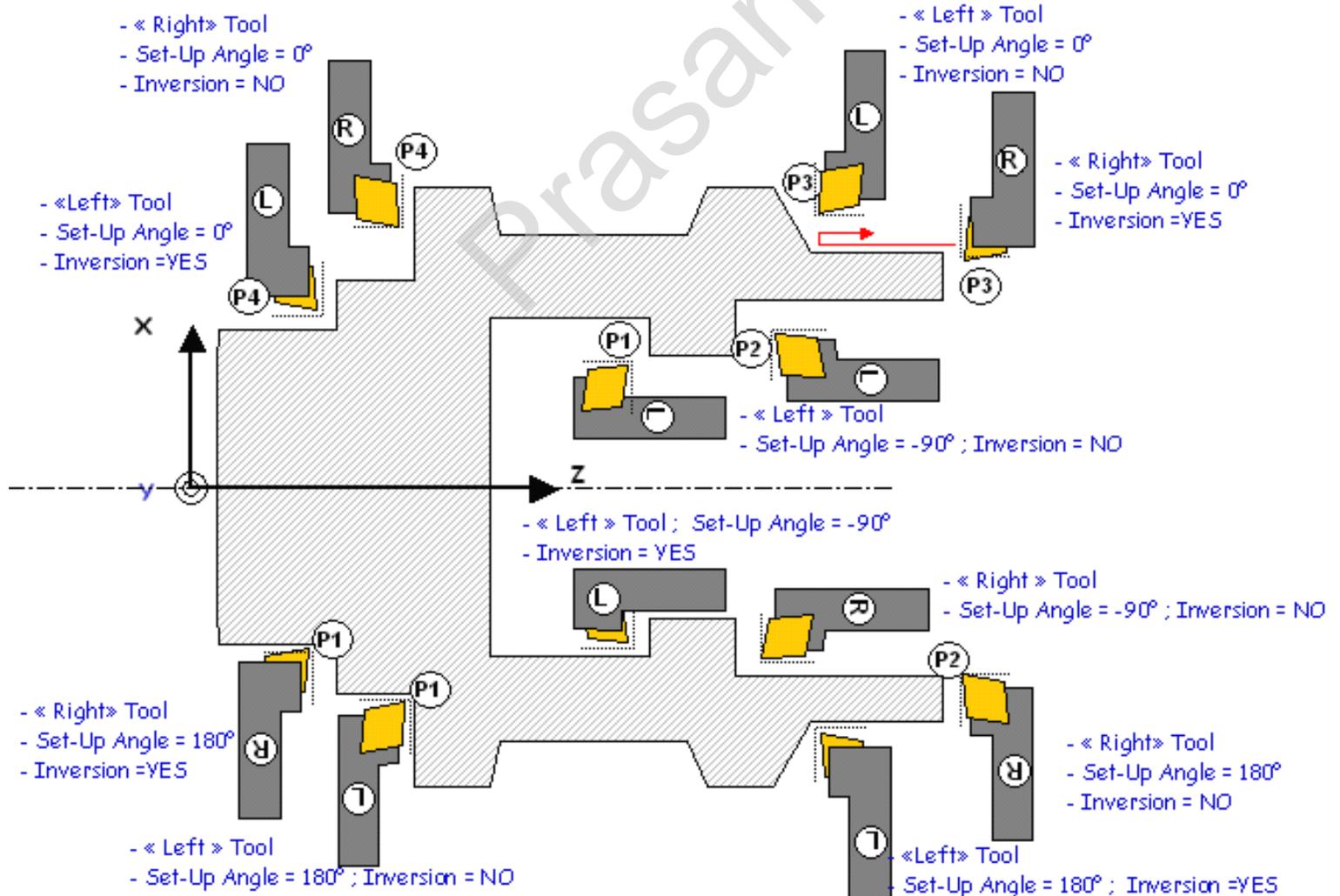
- « Left » Tool
- Set-Up Angle = 0°
- Inversion = NO



- « Left » Tool
- Set-Up Angle = 0°
- Inversion = NO

Summary

The following figures summarize conditions for lathe tool assemblies.



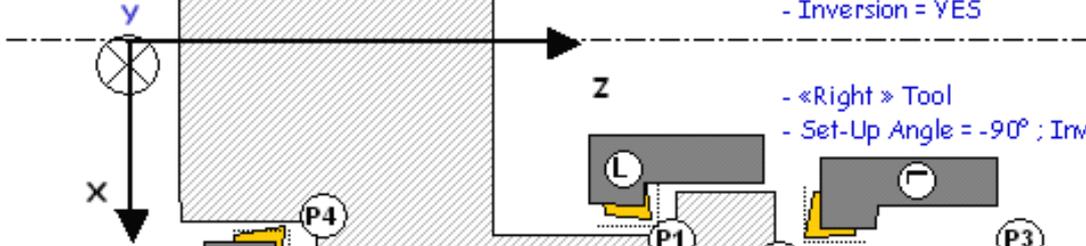
The figure below shows other settings for different output points.

- « 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

- « Right » Tool
- Set-Up Angle = 0°
- Inversion = YES

- « Left » Tool
- Set-Up Angle = 0° ; Inversion = YES

- « Left » Tool
- Set-Up Angle = 0° ; Inversion = NO

Prasant

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

Methodology and conceptual information on the following topics is provided in the *NC Manufacturing Infrastructure User's Guide*.

Machining Processes

Knowledgware in Machining Processes

CATProduct and CATProcess Document Management

Design Changes and Associativity Mechanisms

Part Operation and Set Up Documents

User Features for NC Manufacturing

Prasanth

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).

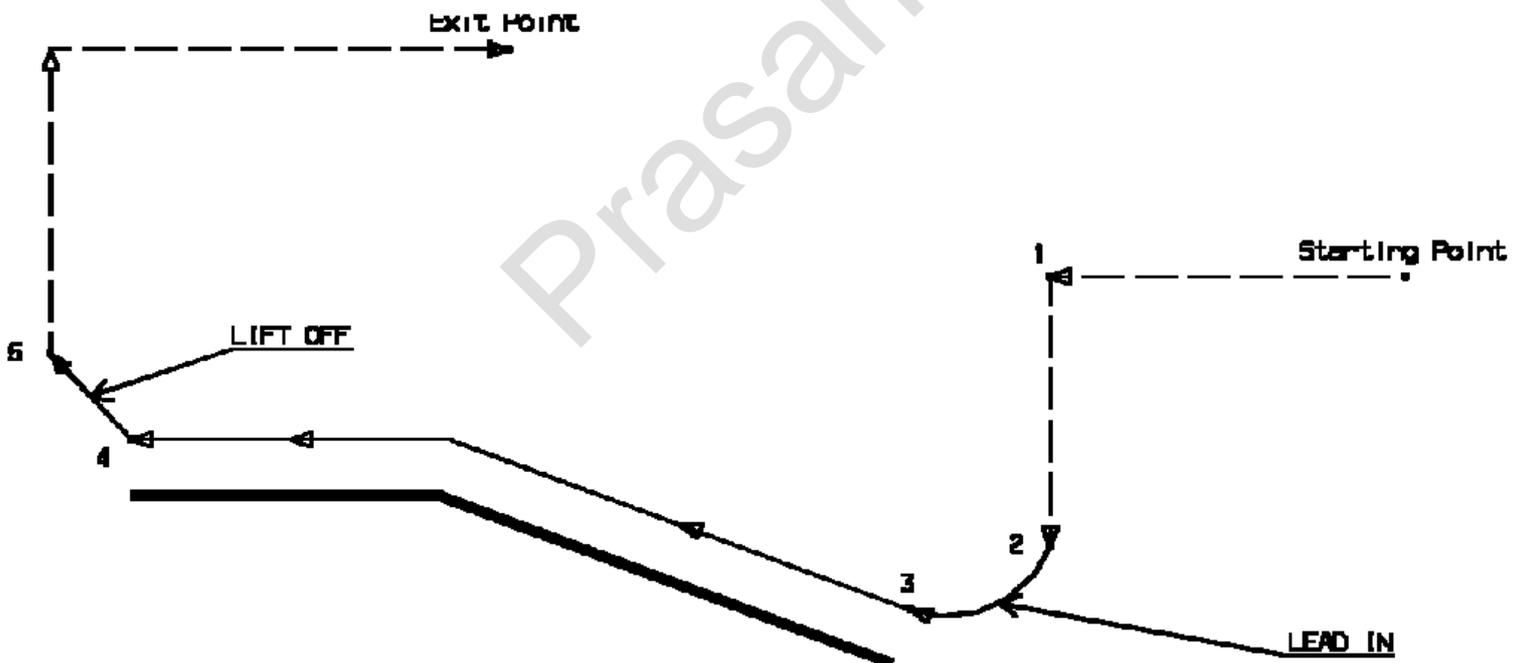
- **On:** CUTCOM/RIGHT instruction generated if tool is to the right of the toolpath and CUTCOM/LEFT if tool is to the left of the toolpath
- **Reverse:** CUTCOM/RIGHT instruction generated if tool is to the left of the toolpath and CUTCOM/LEFT if tool is to the right of the toolpath.

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.

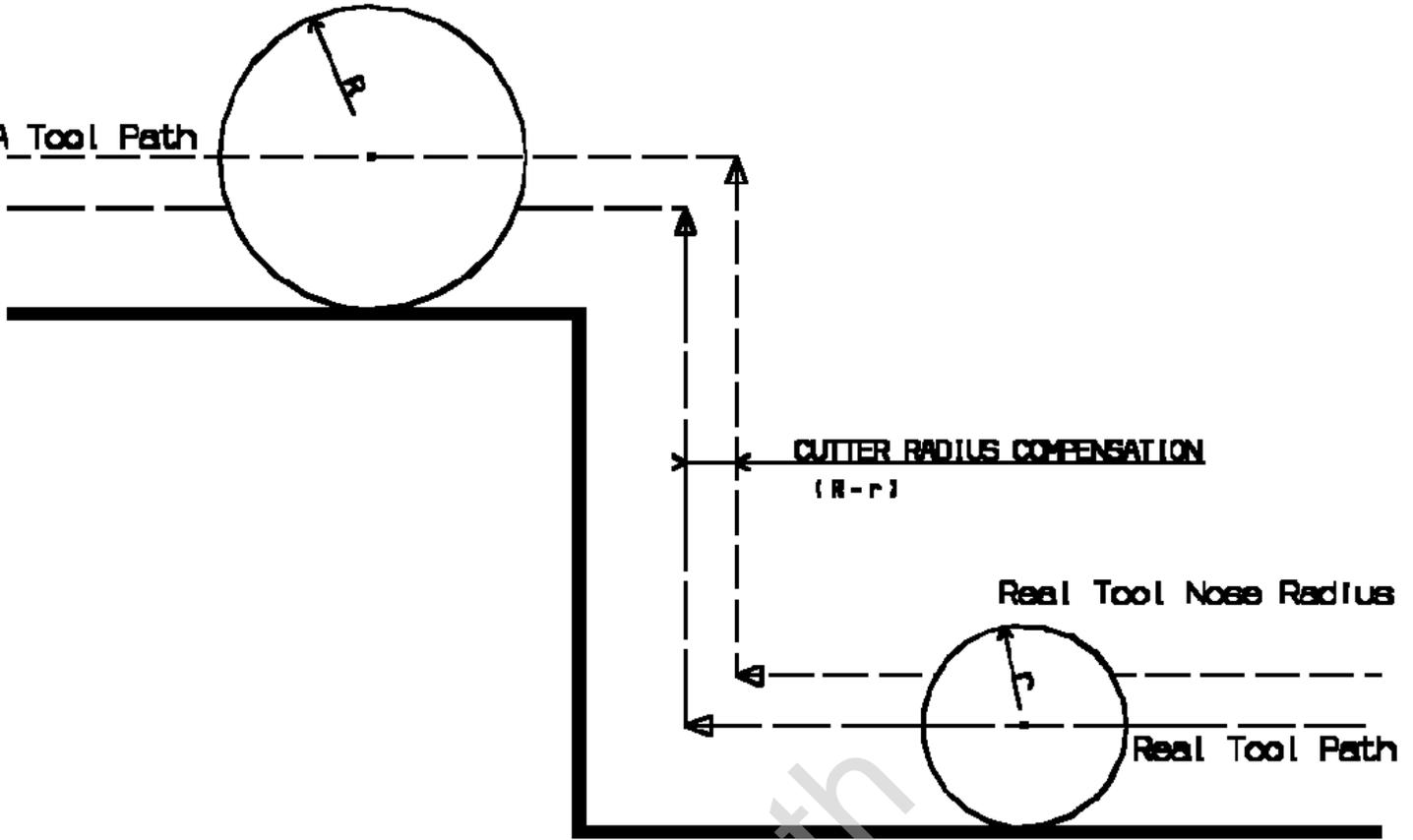
If negative compensation values are not allowed on the machine set Cutter Compensation to:

- 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

CATIA Tool Path



Prasanth

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.

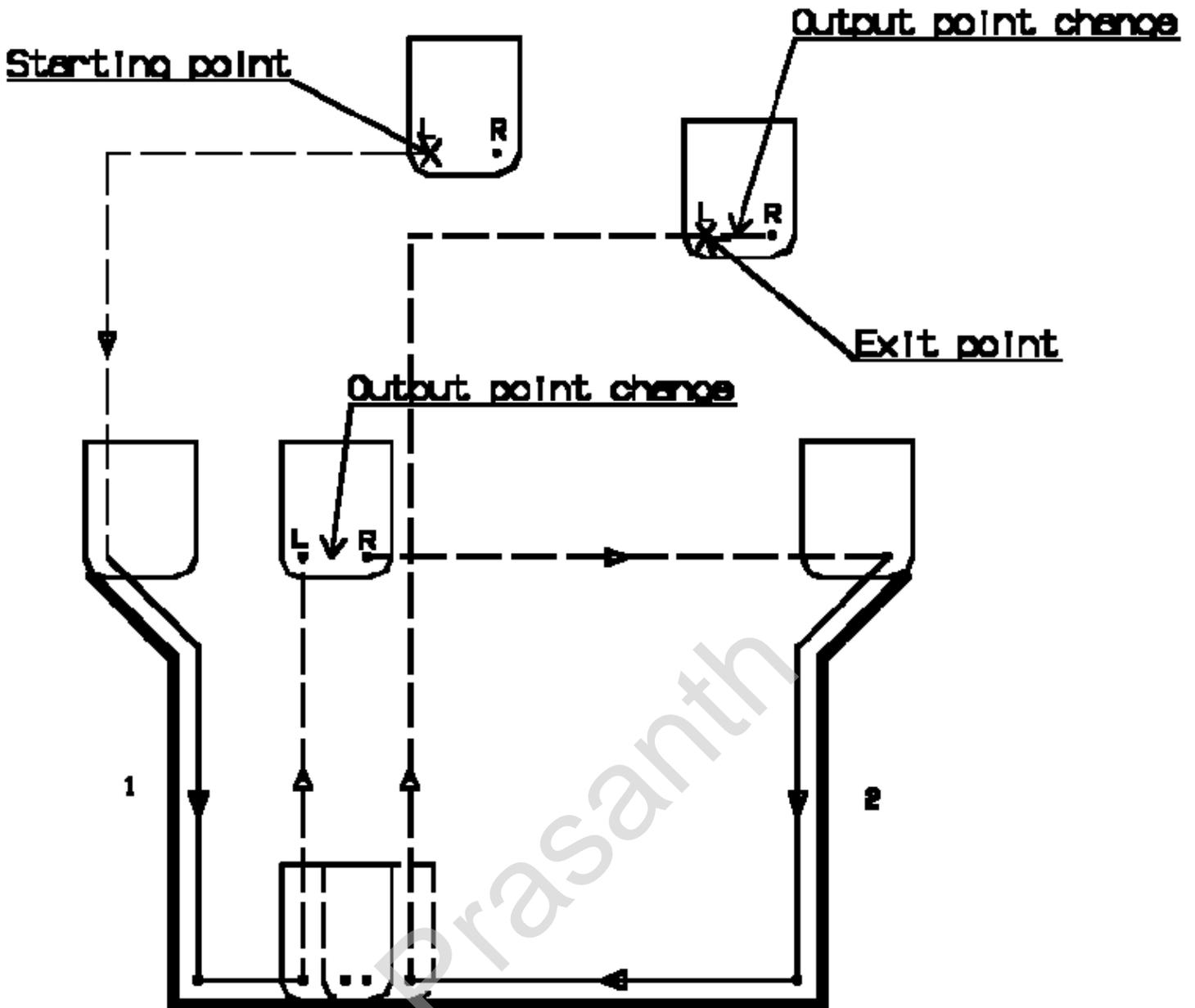
For Recessing operations, tool output point changes are made before each tool motion involving machining (that is, after each plunge). However, changes are only done:

- 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

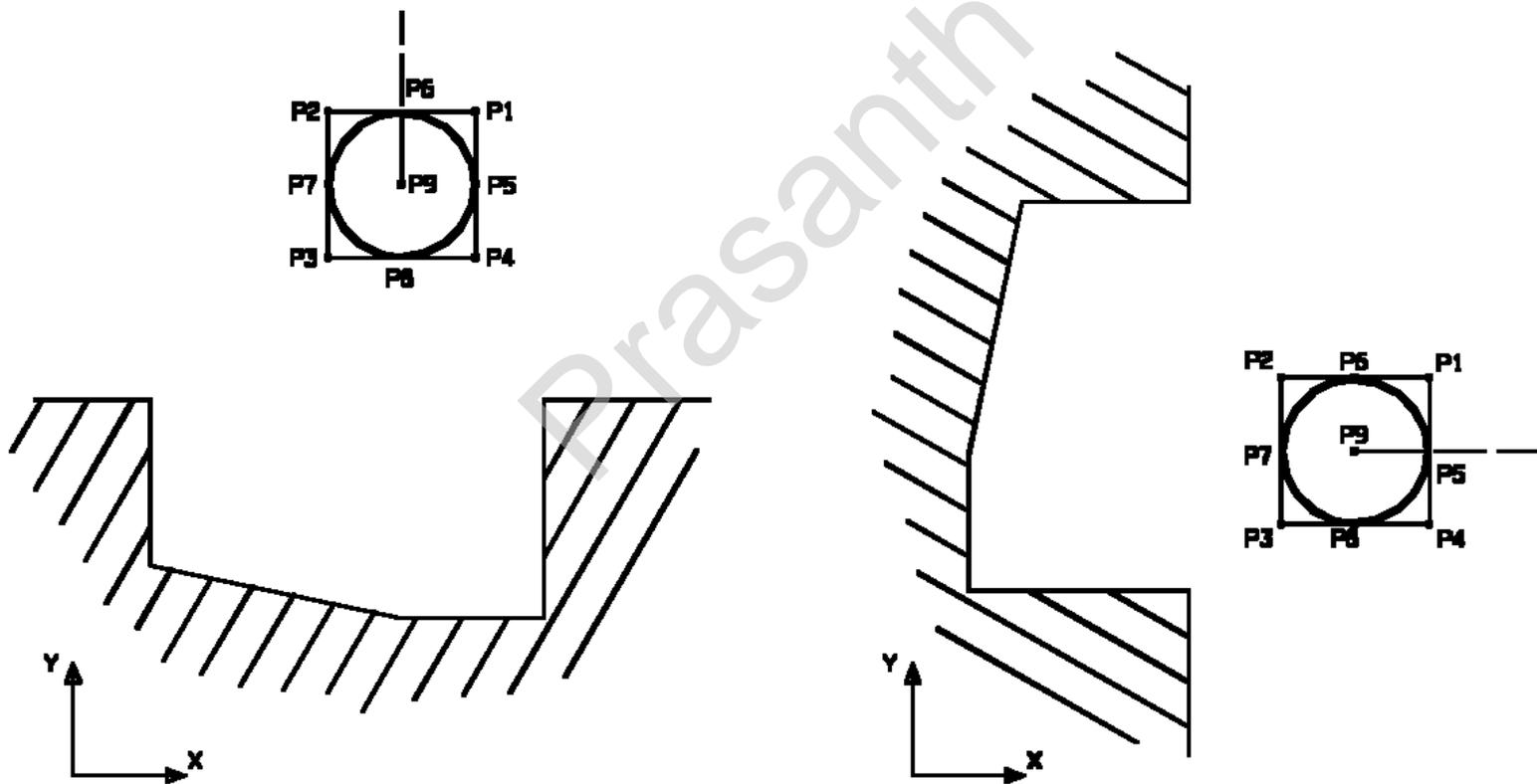
- approach and lead-in motion to flank 2
- machine down flank 2
- lift-off from part profile
- tool output point change: tool output point is P9 and the guiding point is LEFT
- approach and lead-in motion to flank 1
- machine down flank 1
- lift-off to Exit Point.

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.

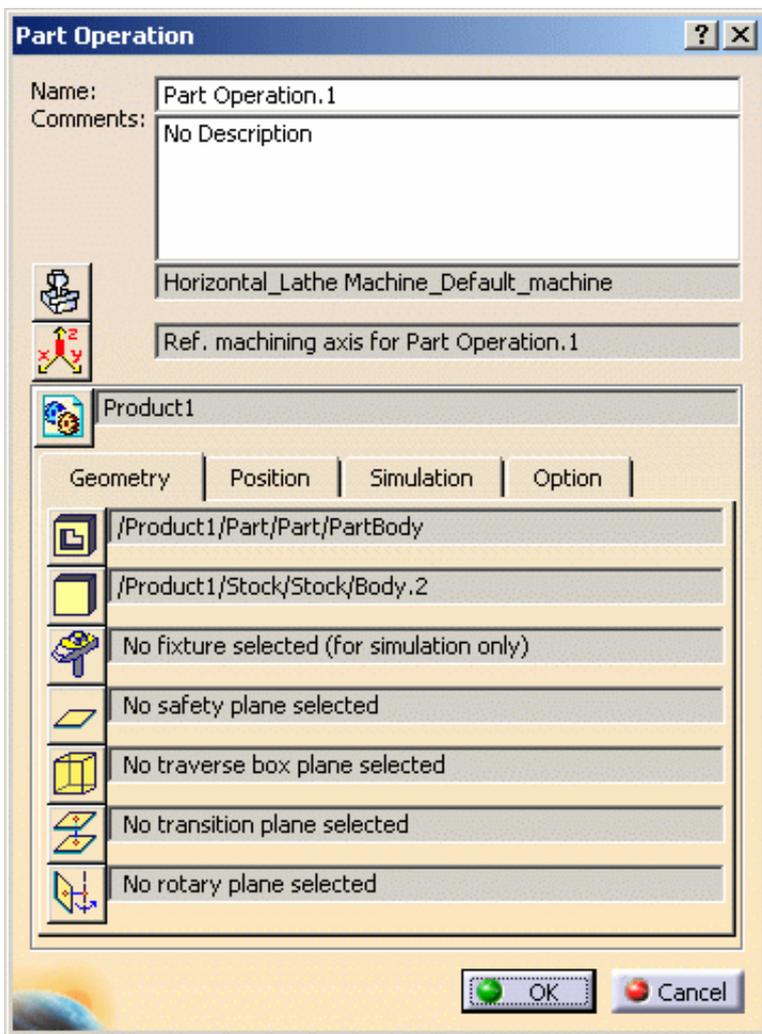
Defining the Machine and Stock

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).

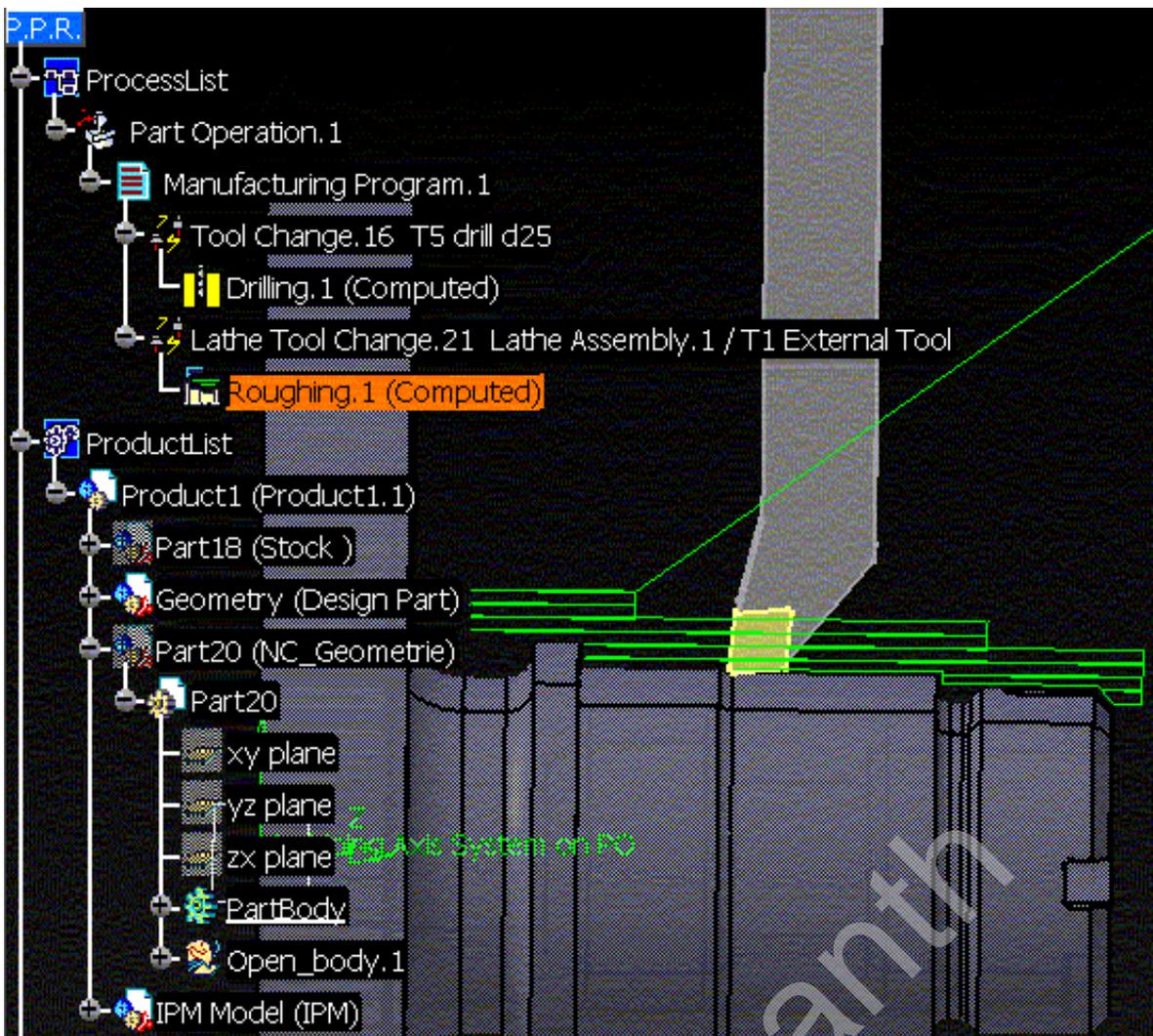
The CATProcess must be linked to a CATProduct that include the design part. The input stock calculation cannot be done if the CATProcess is not linked to the CATProduct.

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

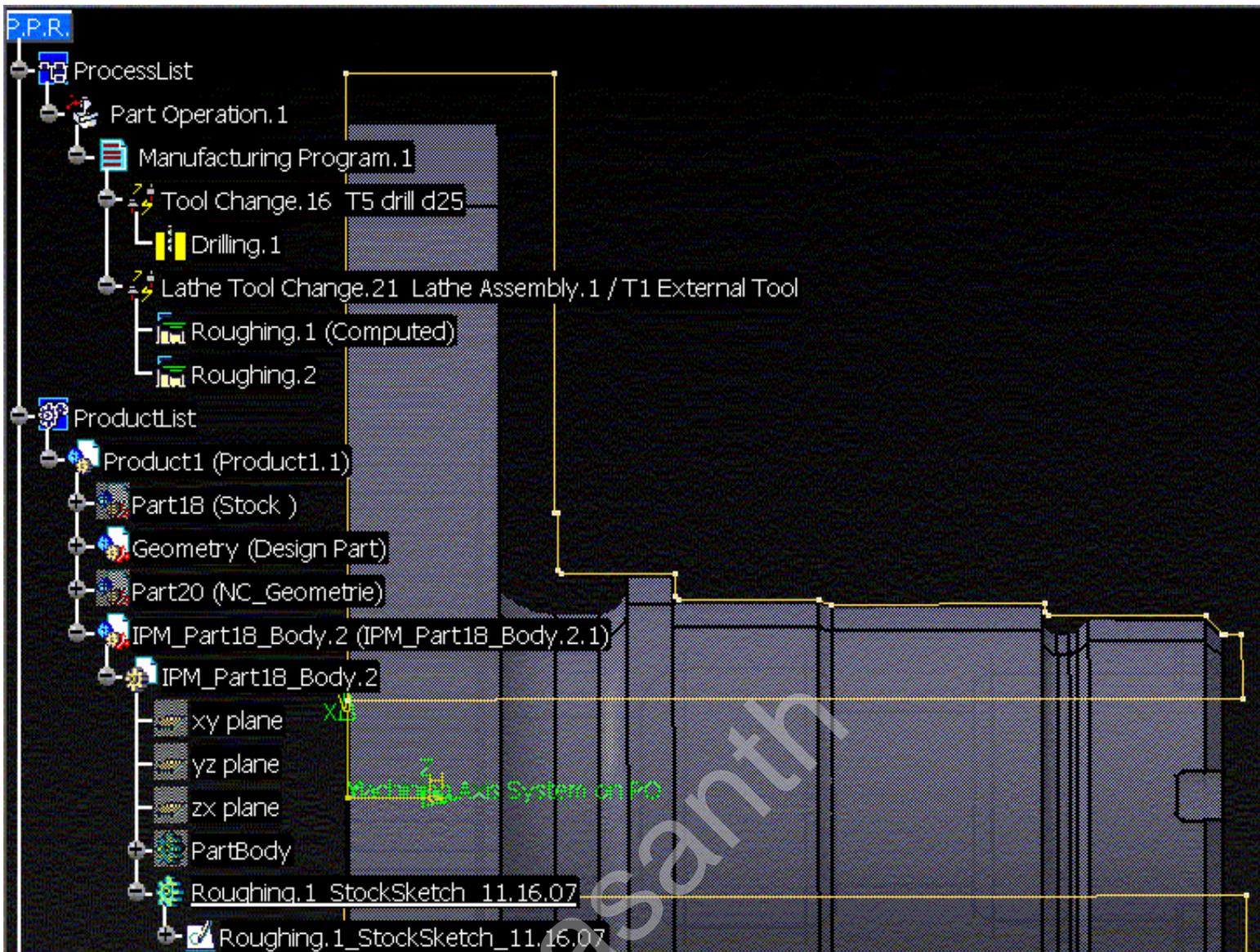
The following figure illustrates Stock computation after a Drilling (along the spindle axis) and a Roughing operation. Here is the tool path of the Roughing operation:



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.

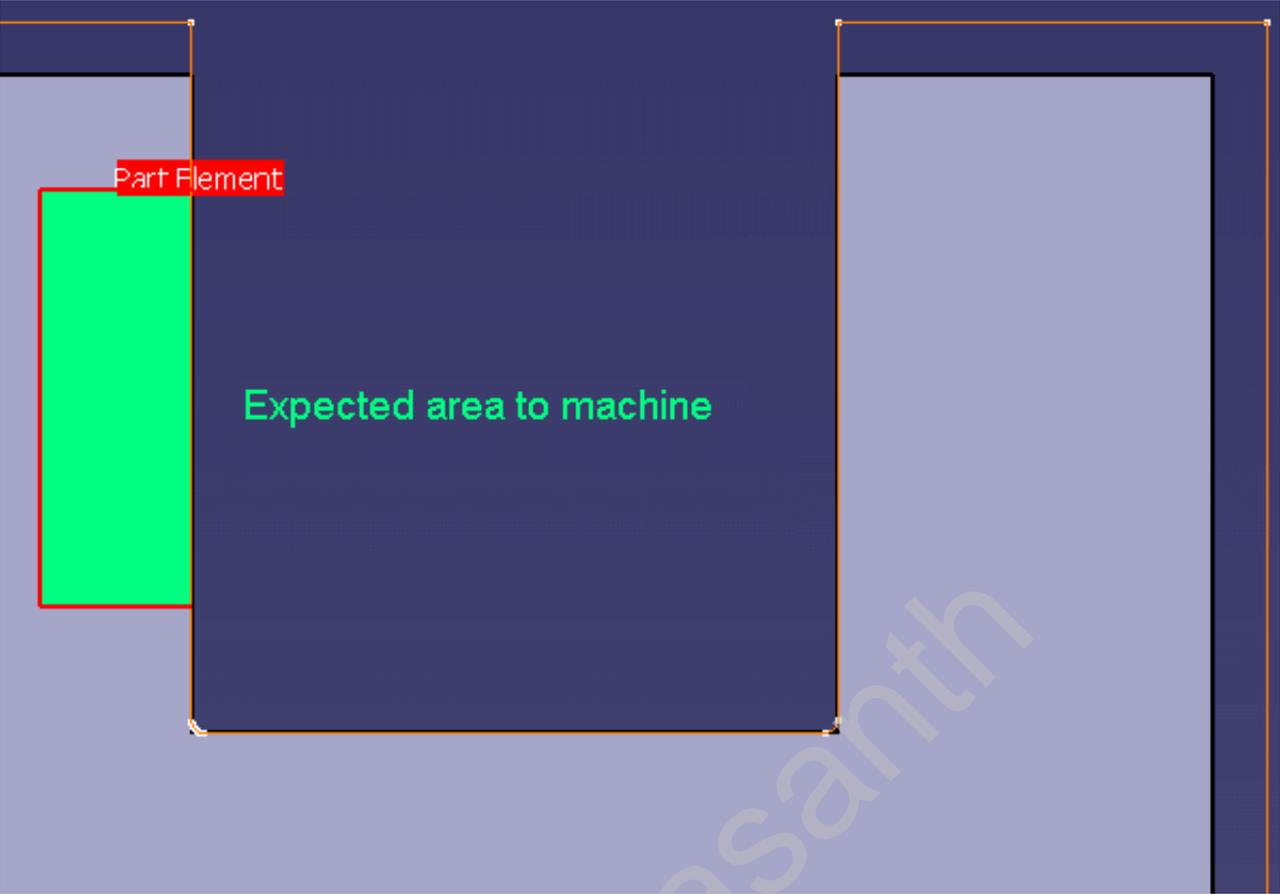
If you have finished your Part Operation, you can remove these primitives by using the **Clean Computed Stock Data** contextual command on the Part Operation. This command removes each primitive of the IPM body linked to the operations in the Part Operation. It does not remove the Stock sketches or the 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.

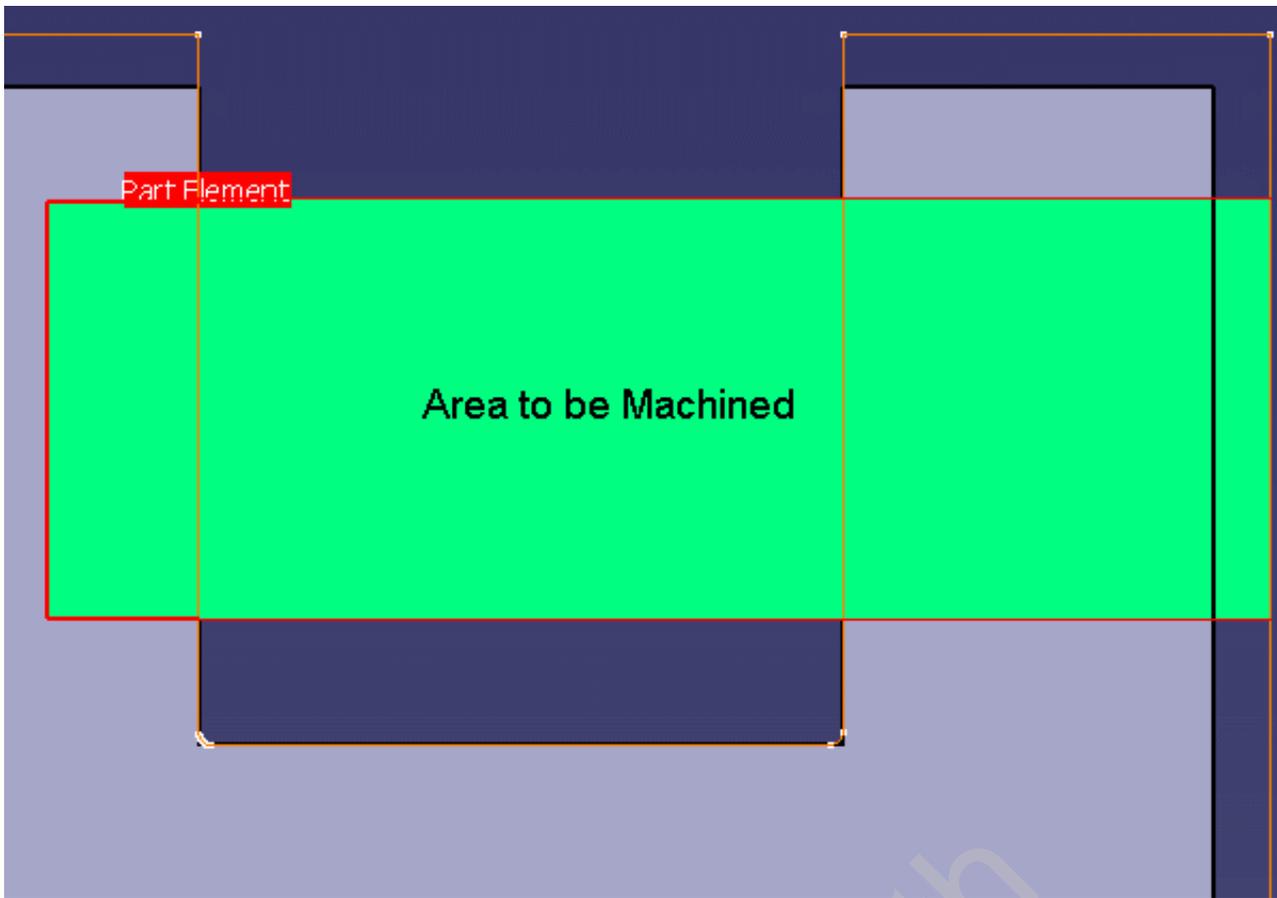
Considerations for Machining Local Recesses

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.

Prasarith

Glossary



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.

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](#).



Prasanth

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

- CGR file 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 
- Cfile 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



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 

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

Prasanth

- 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 
- 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 

Prasanth



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 

Feeds and Speeds 

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 

Feeds and Speeds 

Location 

Maximum Machining Radius 

Minimum Machining Radius 

Orientation 

Prasanth

- 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 

Prasanth

- 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 

Prasanth

- 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 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 in operation
- edit or select another 
- tool in resource list
- edit 
- 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 



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 



Prasanth