Lathe Machining

Preface

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.

Certain portions of this product contain elements subject to copyright owned by the following entities:

What's New?

Sequential Machining

New: You can now create and manage a sequence of basic Go Standard, Go-Go, Go InDirv, Go Delta, and Follow tool motions in a single operation called Sequential Machining.

General Functions

Enhanced: This product benefits from the enhancements brought to the NC Manufacturing Infrastructure.

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 Lathe Roughing Operation
Replay the Toolpath
Create a Lathe Grooving Operation
Create Lathe Profile Finishing Operation
Generate NC Code

Open the Part to Machine

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

1. Select File > Open then select the Lathe01.CATPart document.

2. Select NC Manufacturing > 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, then in the Machine Editor dialog box:
- select the Horizontal Lathe Machine icon
- make sure that the radial axis is set to X and the spindle axis is set to Z
- click OK.

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

6. Click OK to confirm your modifications to the Part Operation.

7. Select Manufacturing Program.1 in the tree to make it the current entity.
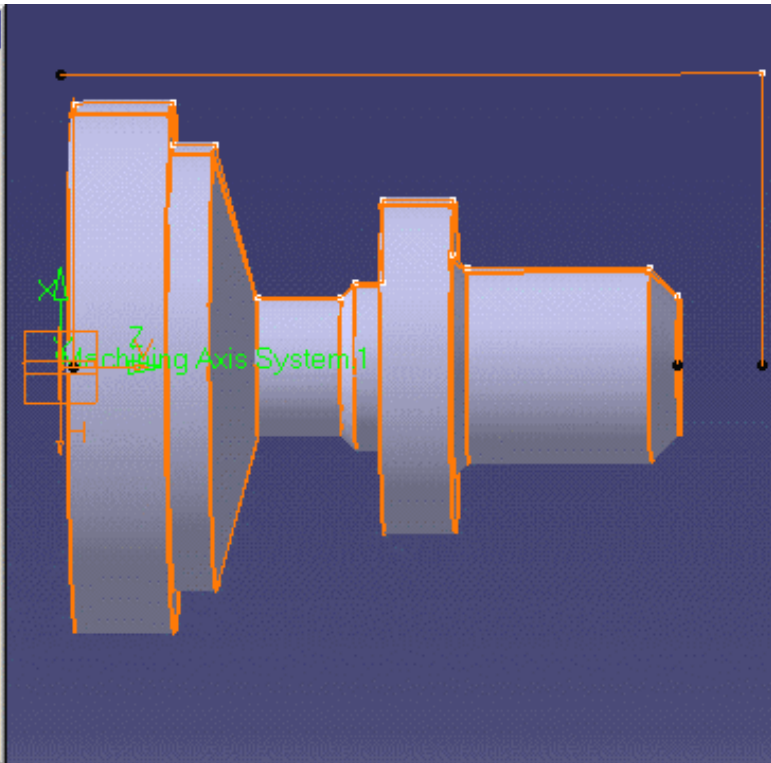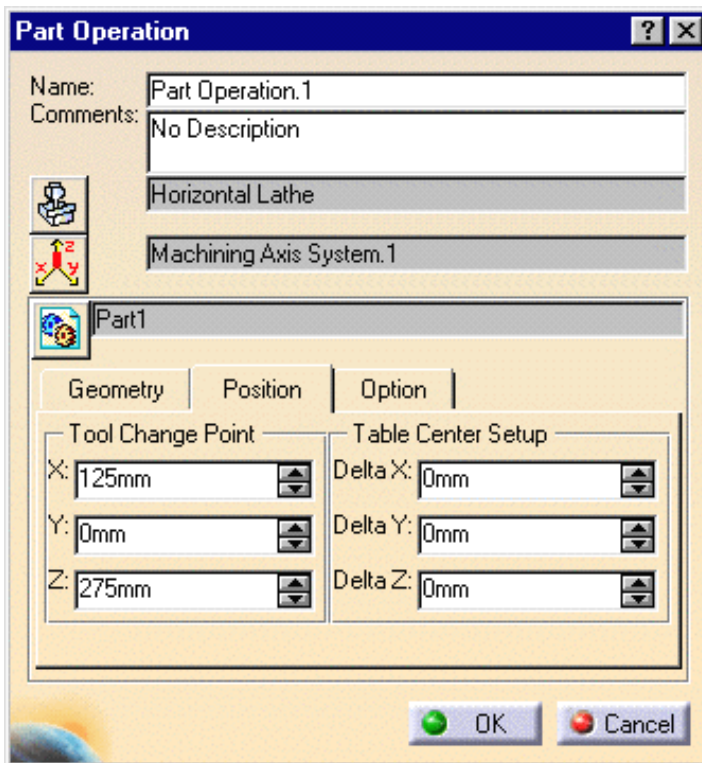
To insert program entities such as machining operations, tools and auxiliary commands you can either:
- make the program current before clicking the *insert program entity* command
- click the *insert program entity* command then make the program current.

Create a Lathe Roughing Operation

This task shows you how to create a Longitudinal Roughing 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 Roughing icon .

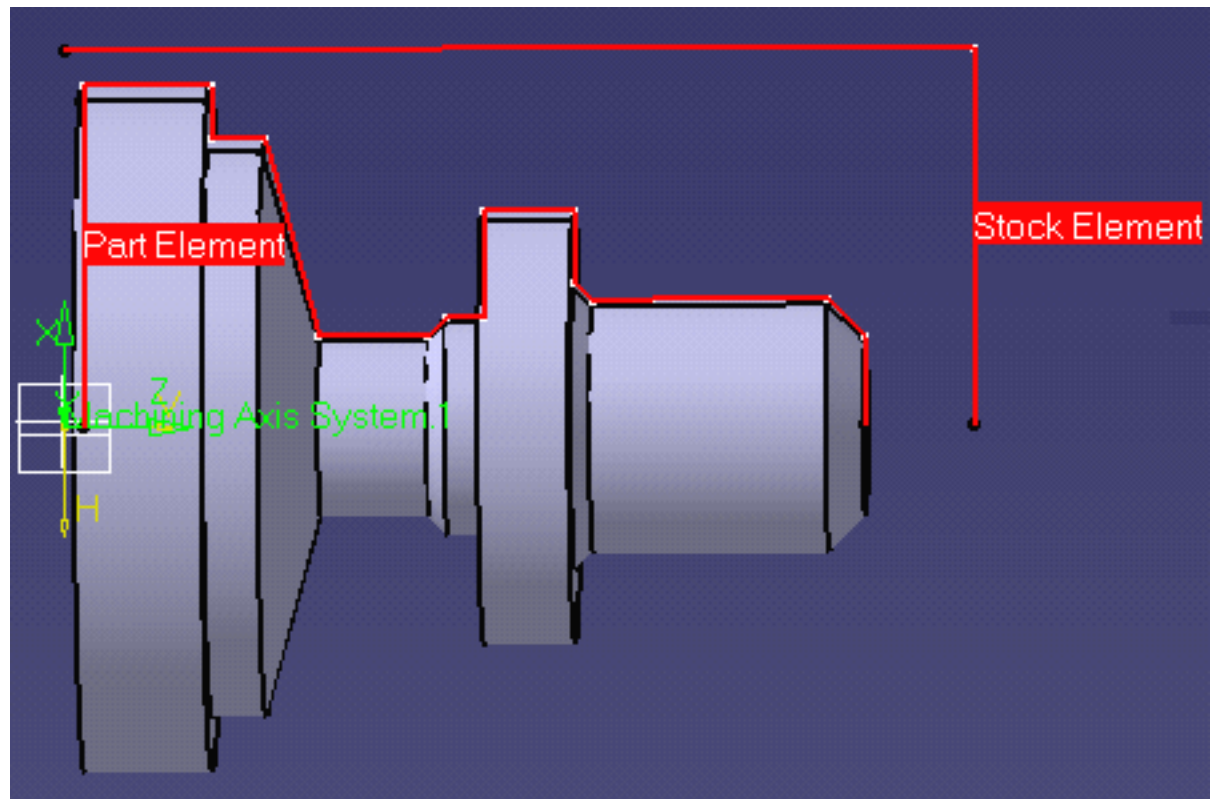A Roughing.1 entity along with a default tool is added to the program.

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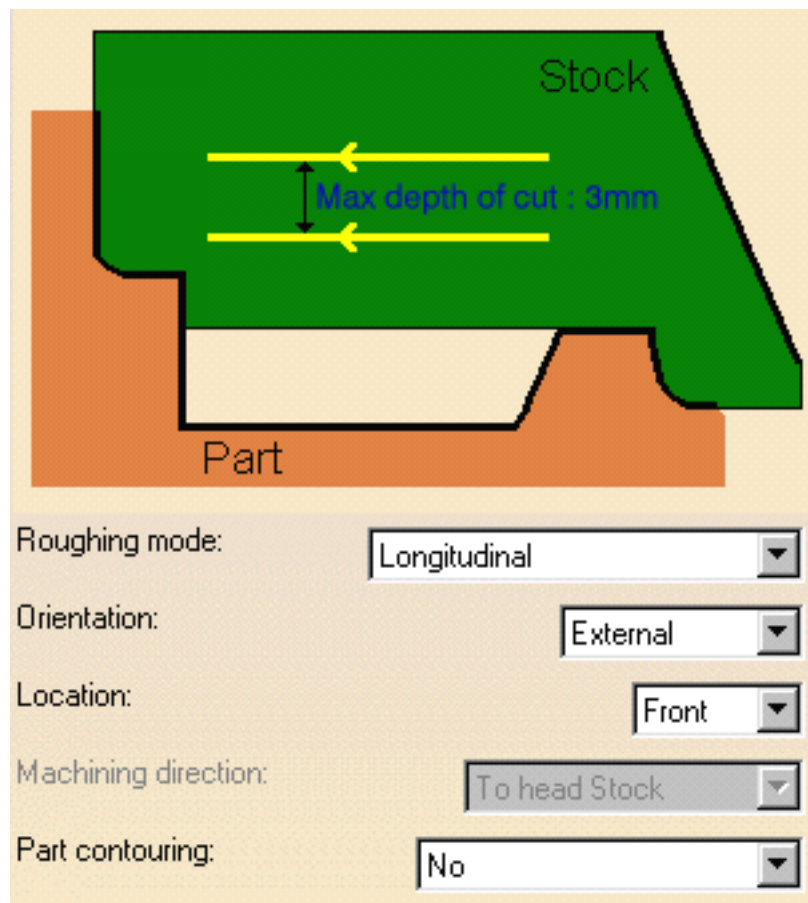
Click OK in the Edge Selection toolbar to end your selection.

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.

Roughing mode:	Longitudinal
Orientation:	External
Location:	Front
Machining direction:	To head Stock
Part contouring:	No

5. Click OK to create the operation.
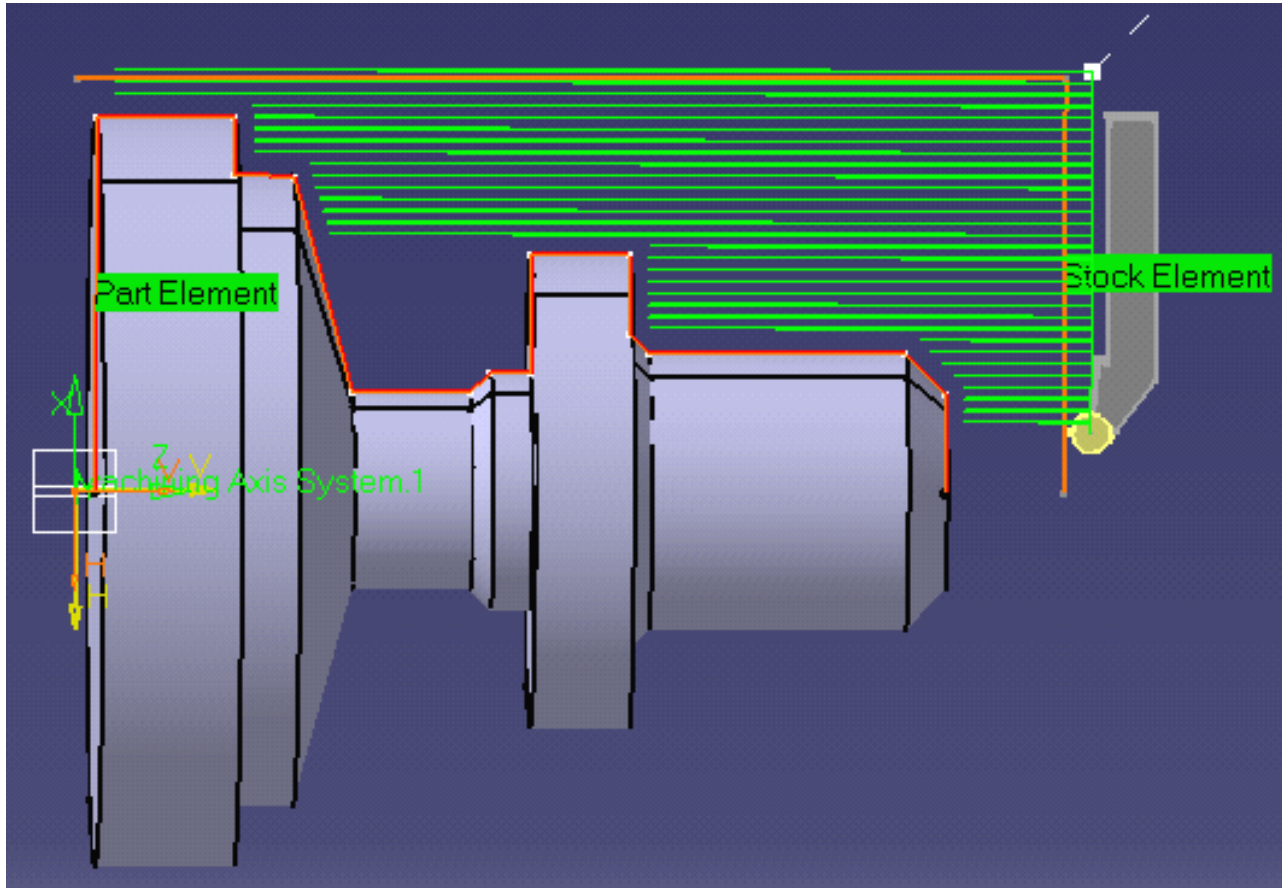
Replay the Tool Path

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

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

The Replay dialog box appears.

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

3. Click the ![icon] button to position the tool at the start point of the operation.

4. Click the ![icon] button to start the replay. The tool moves along the computed trajectory.

5. Click OK to quit the replay mode.

Create a Lathe Grooving Operation

This task shows you how to create a Grooving 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.

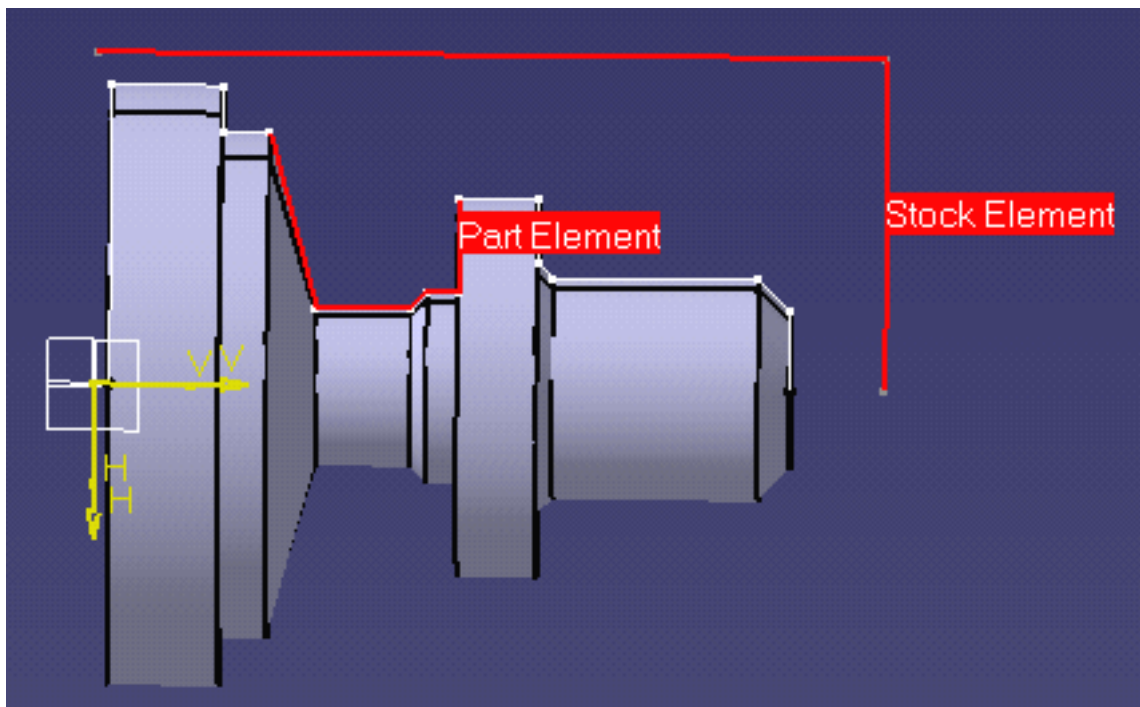Make sure that the Roughing operation is the current entity in the program.

1. Select the Grooving icon .

   The Grooving 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 Tool Assembly 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.
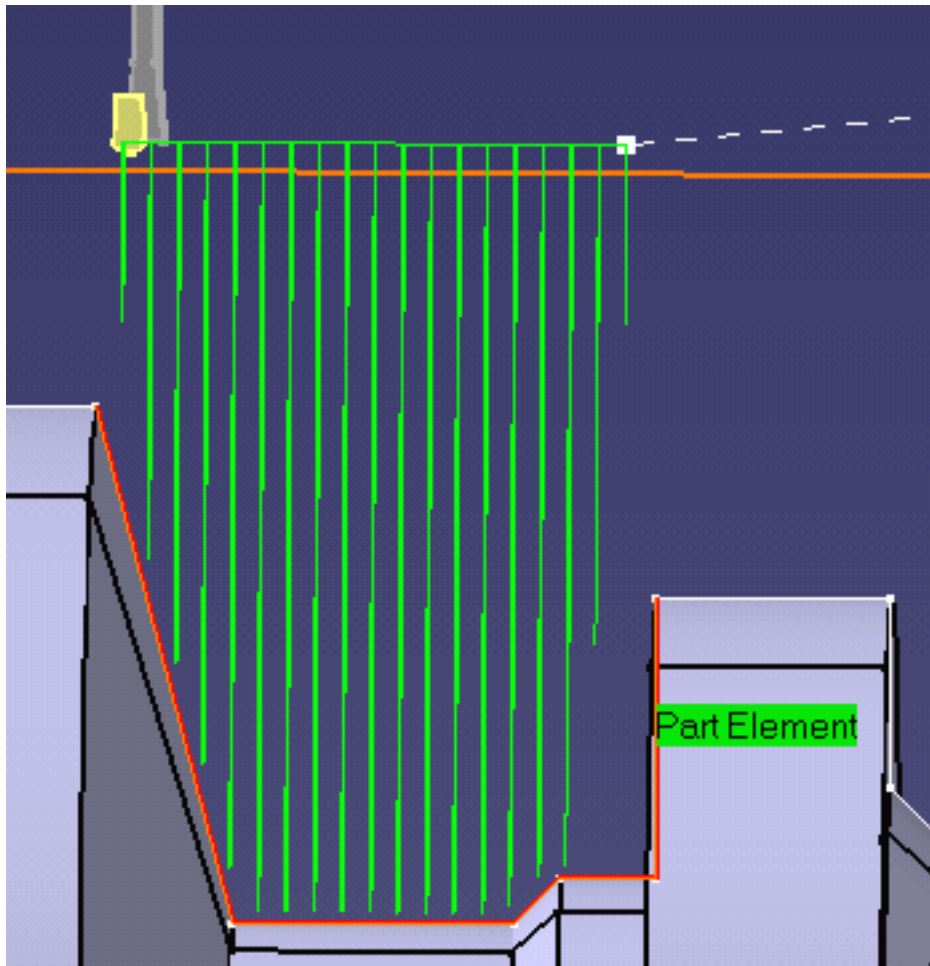
Click OK to exit the replay mode and return to the Grooving dialog box.

7. Click OK to create the operation.

Create a Lathe Profile Finishing Operation

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

1. Select the Profile Finishing icon .

The Profile Finishing 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.
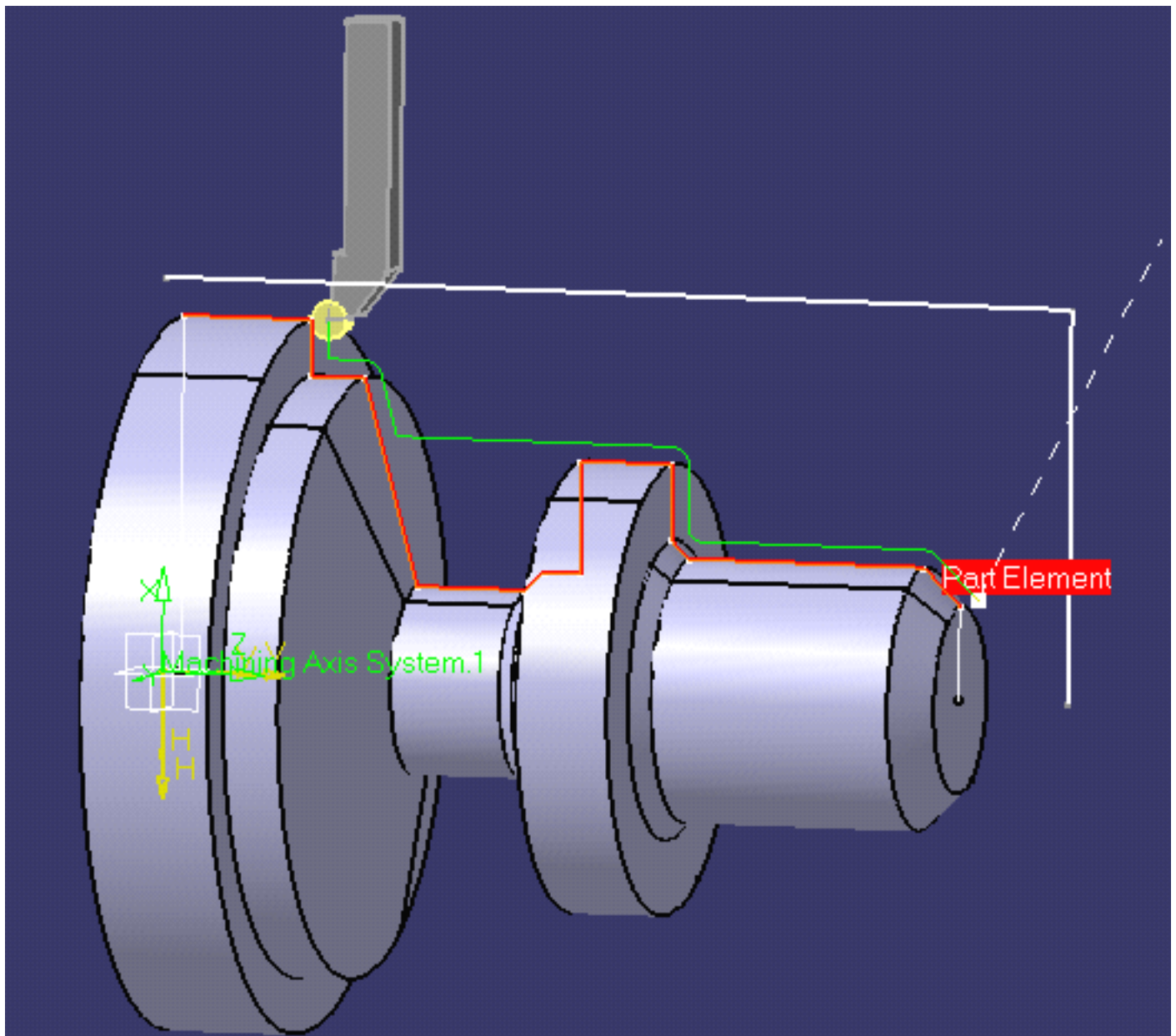
Click OK to exit the replay mode and return to the Profile Finishing dialog box.

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 Save NC File dialog box appears.

2. Select the folder where you want the file to be saved and specify the name of the file.
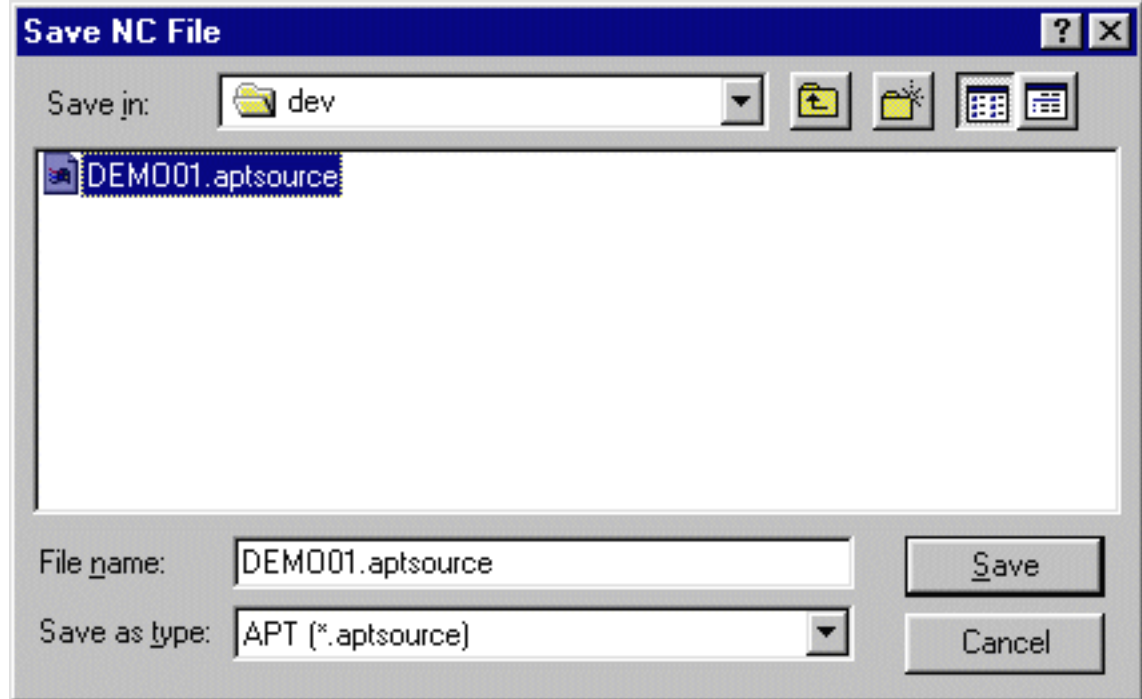
3. Click Save to create the APT source file.

Here is an extract from the Apt source file that could be generated:

```
$$ --------------------------------------------------------------
$$ Generated on Tuesday, May 15, 2001 05:14:58 PM
$$ --------------------------------------------------------------
$$ Manufacturing Program.1
$$ Part Operation.1
$$*CATIA0
$$ Manufacturing Program.1
$$ 1.00000 0.00000 0.00000 0.00000
$$ 0.00000 1.00000 0.00000 0.00000
$$ 0.00000 0.00000 1.00000 0.00000
PARTNO PART TO BE MACHINED
COOLNT/ON
CUTCOM/OFF
PPRINT OPERATION NAME : Lathe Tool Change.1
$$ Start generation of : Lathe Tool Change.1
TLAXIS/ 0.000000, 0.000000, 1.000000
$$ TOOLCHANGEBEGINNING
```

```
RAPID
GOTO / 125.00000, 0.00000, 275.00000
CUTTER/ 5.000000
TOOLNO/0,TURN
$$ End of generation of : Lathe Tool Change.1
PPRINT OPERATION NAME : Roughing.1
$$ Start generation of : Roughing.1
FEDRAT/ 0.3000,MMPR
SPINDL/ 70.0000,RPM
GOTO / 107.02703, 0.00000, 257.00000
GOTO / 107.02703, 0.00000, 255.00000
...
FEDRAT/ 0.8000,MMPR
GOTO / 0.21213, 0.00000, 225.21213
$$ End of generation of : Roughing.1
CUTCOM/OFF
$$ ------ CUTCOM OFF END OF LATHE ------
PPRINT OPERATION NAME : Lathe Tool Change.2
$$ Start generation of : Lathe Tool Change.2
$$ TOOLCHANGEBEGINNING
RAPID
GOTO / 125.00000, 0.00000, 275.00000
CUTTER/ 1.200000
TOOLNO/0,TURN
$$ End of generation of : Lathe Tool Change.2
PPRINT OPERATION NAME : Grooving.1
$$ Start generation of : Grooving.1
FEDRAT/ 0.3000,MMPR
SPINDL/ 70.0000,RPM
GOTO / 108.20000, 0.00000, 111.70000
GOTO / 106.20000, 0.00000, 111.70000
...
RAPID
GOTO / 108.20000, 0.00000, 57.91213
$$ End of generation of : Grooving.1
CUTCOM/OFF
$$ ------ CUTCOM OFF END OF LATHE ------
PPRINT OPERATION NAME : Lathe Tool Change.3
$$ Start generation of : Lathe Tool Change.3
$$ TOOLCHANGEBEGINNING
RAPID
GOTO / 125.00000, 0.00000, 275.00000
CUTTER/ 5.000000
TOOLNO/0,TURN
$$ End of generation of : Lathe Tool Change.3
PPRINT OPERATION NAME : Profile Finishing.1
$$ Start generation of : Profile Finishing.1
FEDRAT/ 0.3000,MMPR
SPINDL/ 70.0000,RPM
GOTO / 27.12132, 0.00000, 224.94975
GOTO / 28.53553, 0.00000, 223.53553
...
FEDRAT/ 0.8000,MMPR
GOTO / 100.21213, 0.00000, 4.78787
$$ End of generation of : Profile Finishing.1
SPINDL/OFF
```

Basic Tasks

The basic tasks you will perform with Lathe Machining involve creating, editing and managing machining operations and other entities of the NC manufacturing process.

Lathe Machining Operations
Axial Machining Operations
Manufacturing Entities
Auxiliary Operations
Verification, Simulation and Program Output
Part Operations, Programs and Processes

Lathe Machining Operations

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

Create a Roughing operation:
Select the Roughing 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 Recessing operation:
Select the Recessing 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 Grooving operation:
Select the Grooving 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 Finishing operation:
Select the Profile Finishing 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 Finishing operation:
Select the Groove Finishing 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 Threading operation:
Select the Threading 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 Machining 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 thicknesses to Profile Finishing and Groove Finishing operations.

Create a Longitudinal Roughing Operation

This task illustrates how to create a Longitudinal Roughing 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).

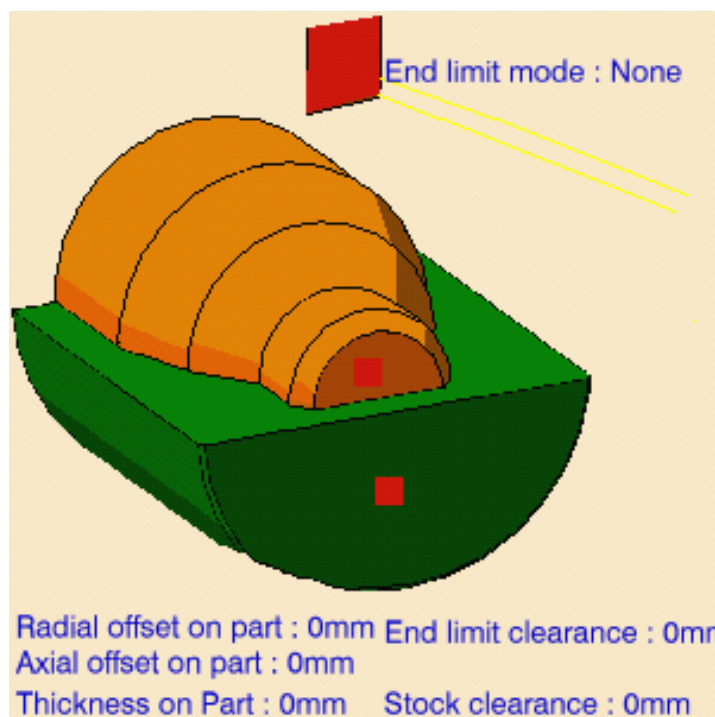Open the Lathe01.CATPart document, then select NC Manufacturing > Lathe Machining from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Roughing icon.

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

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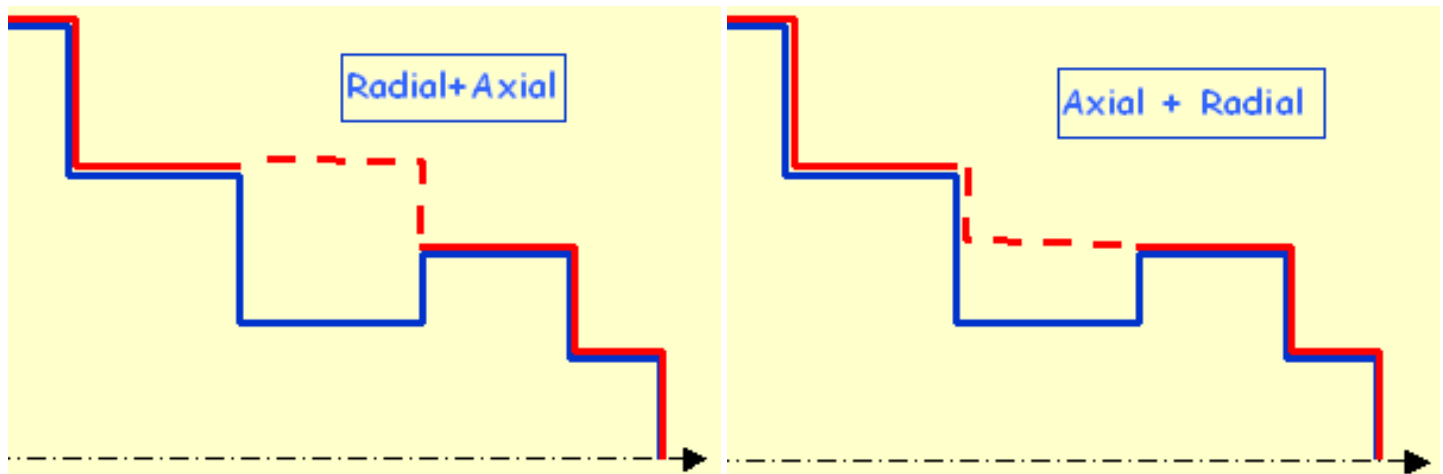
2. Click the red part in the icon then select the desired part profile in the 3D window.

The Edge Selection toolbar appears to help you with contour selection:
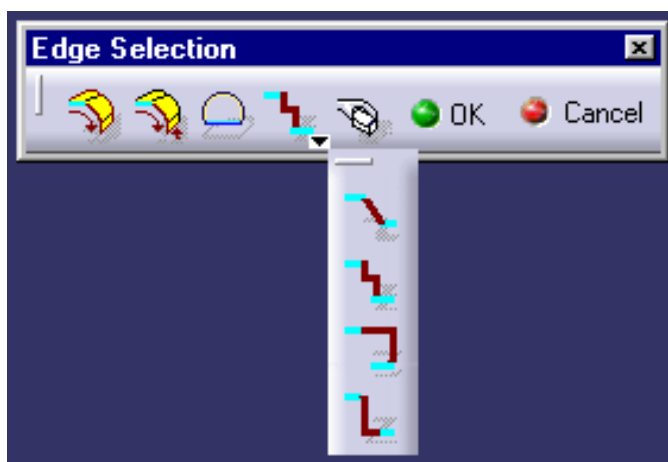
The Auto-link command allows you to select a first element then the element to Navigate to in

order to complete the profile selection.

The Axial/Radial, Radial/Axial, and Direct commands are also useful for profile selection.


i 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. Double click **Thickness on Part** in the icon.

Set this value to 5mm in the Edit Parameter dialog box and click OK.

5. Select the Strategy tab page [icon] to specify the main machining strategy parameters:

- Roughing mode: **Longitudinal**
- Orientation: **External**
- Location: **Front**.
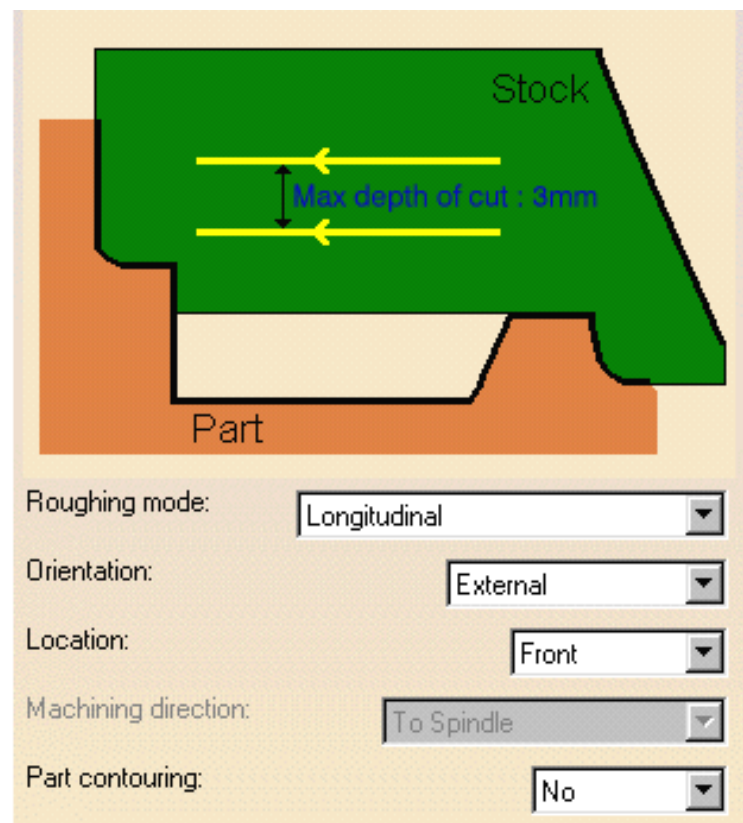
6. Double click **Max depth of cut** in the icon.

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

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

i 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 [icon] 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 [icon] to specify the feedrates and spindle speeds for the operation. See Feeds and Speeds for Roughing for more information.

8. Select the Macros tab page ![icon] 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.
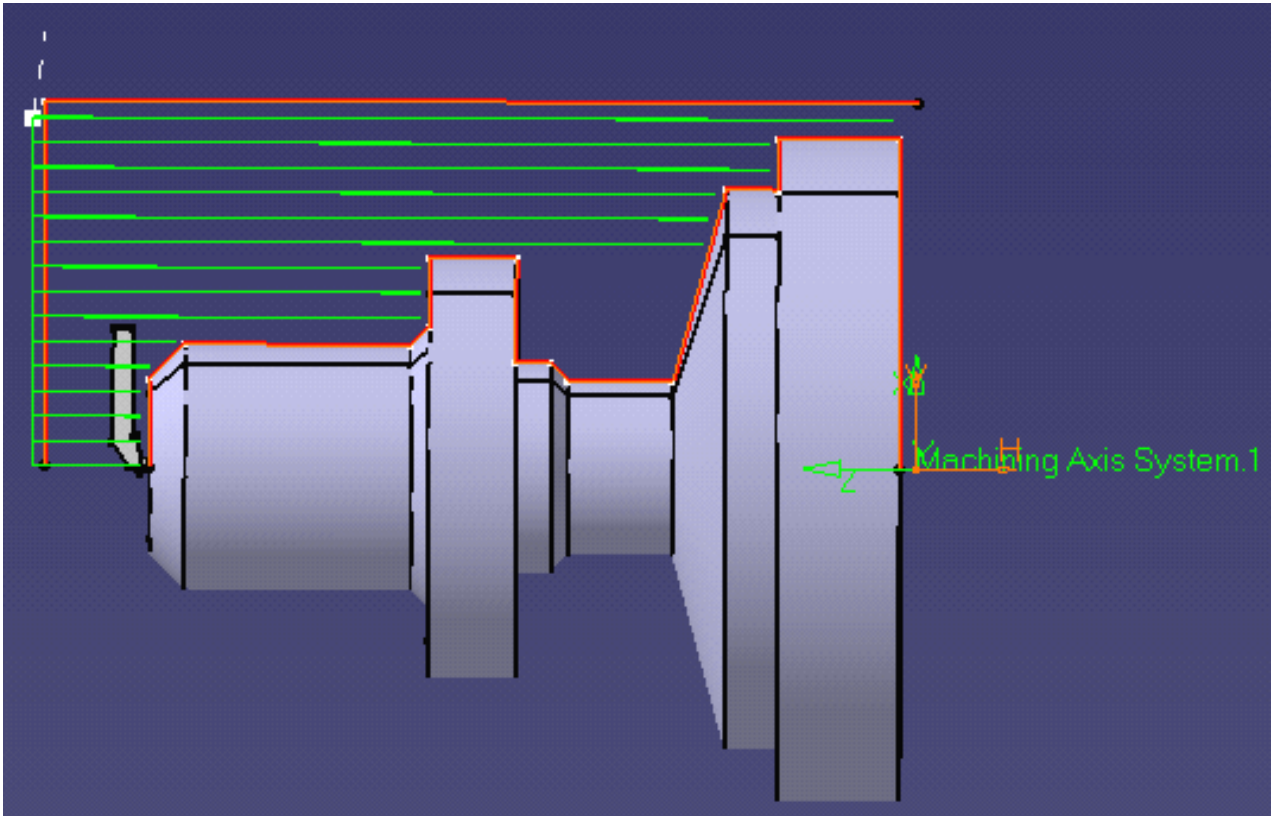
9. Click OK to create the operation.

Create a Face Roughing Operation

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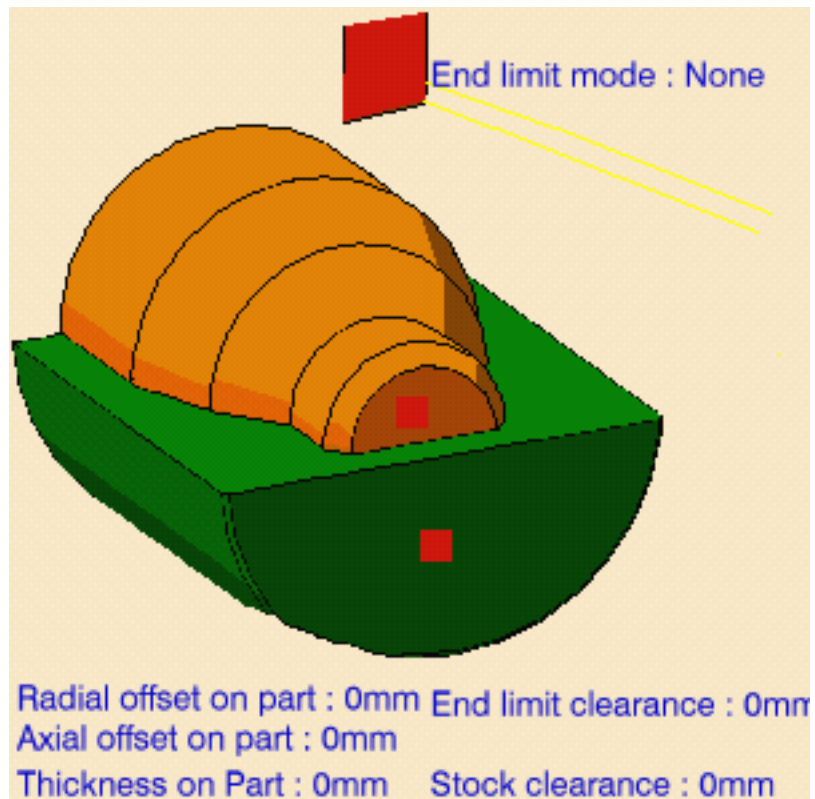
To create the operation you must define:

- 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 NC Manufacturing > Lathe Machining from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Roughing icon.

    The Roughing 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.

    End limit mode : None

    Radial offset on part : 0mm   End limit clearance : 0mm
    Axial offset on part : 0mm
    Thickness on Part : 0mm   Stock clearance : 0mm

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. Double click **Thickness on Part** in the icon.

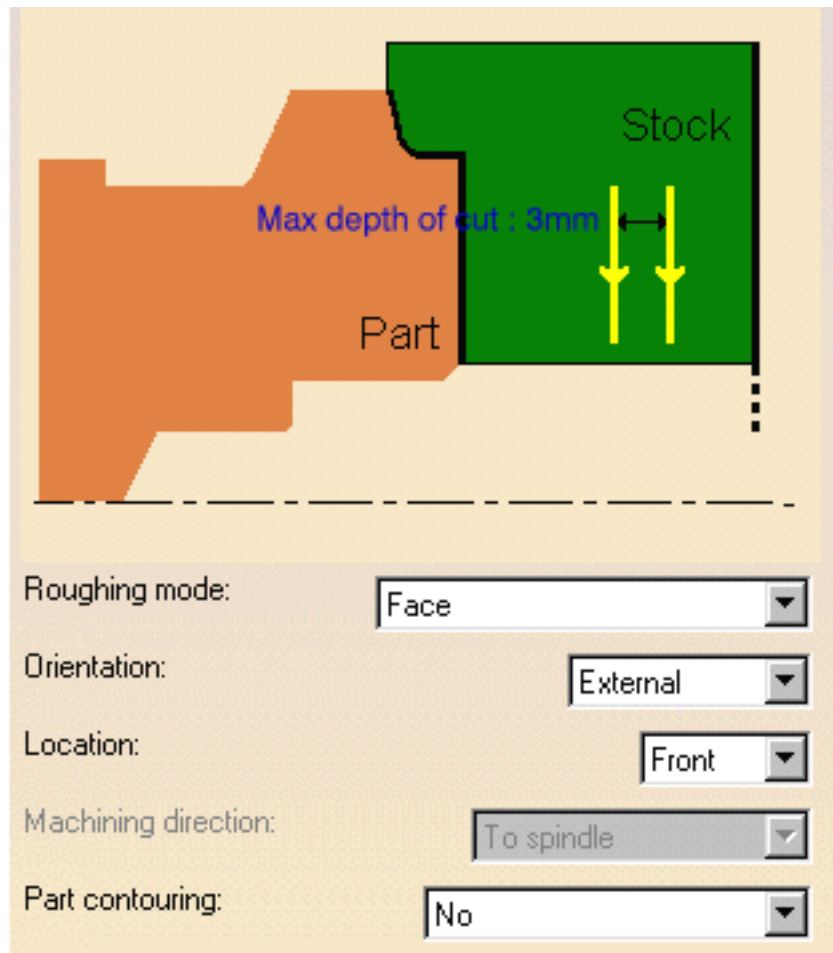   Set this value to 5mm in the Edit Parameter dialog box and click OK.

5. Select the Strategy tab page ![icon] 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 page, set the lift-off distance to 1.5mm.

|	
Roughing mode:	Face
Orientation:	External
Location:	Front
Machining direction:	To spindle
Part contouring:	No

Max depth of cut : 3mm

Stock

Part

ⓘ 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 ![icon] 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 ![icon] to specify the feedrates and spindle speeds for the operation. See Feeds and Speeds for Roughing for more information.

ⓘ If you want to specify approach and retract motion for the operation, select the Macros tab page ![icon] to specify the desired transition paths. See Define Macros on a Lathe Operation for an example.

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

10. Click OK to create the operation.
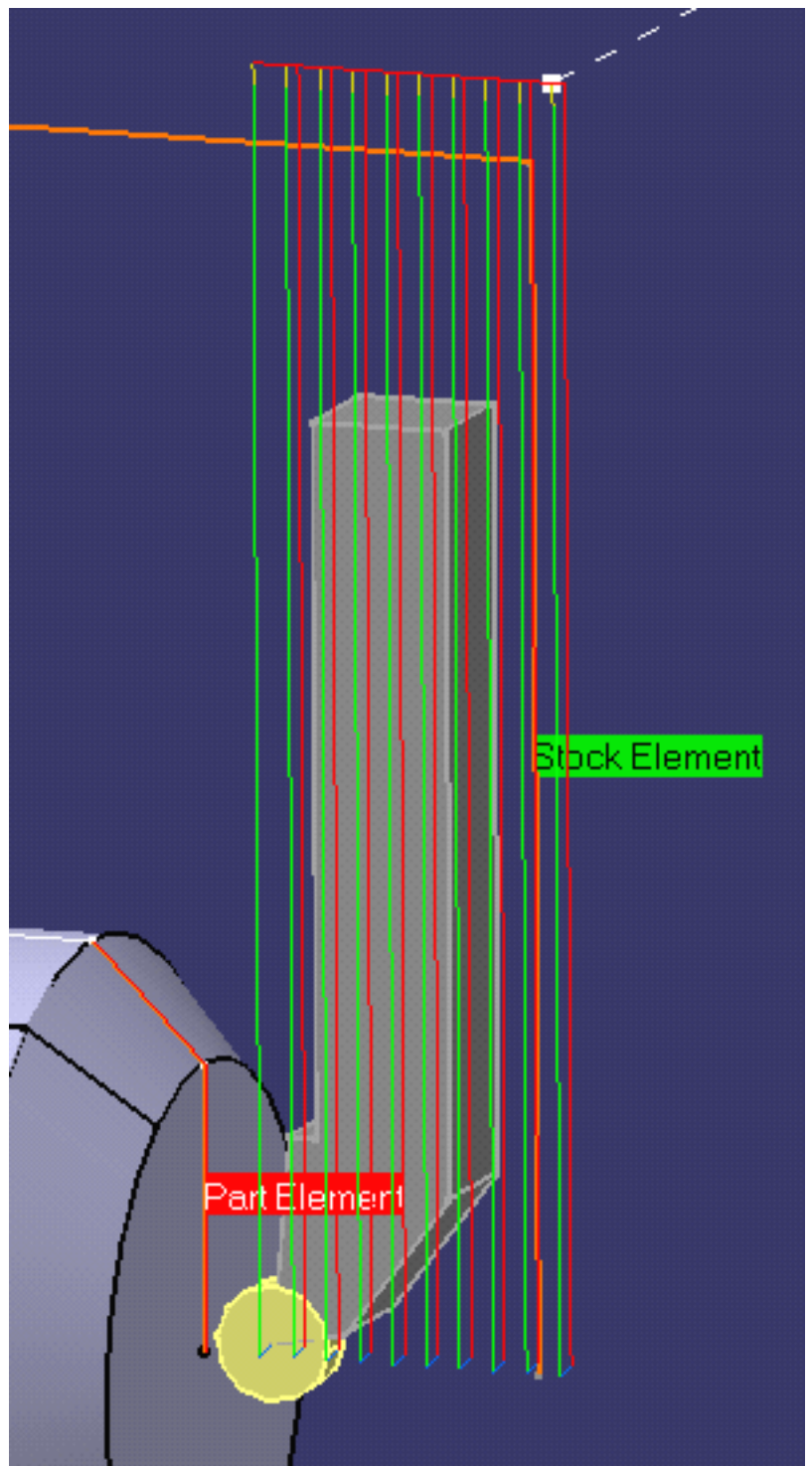
Create a Parallel Contour Roughing Operation

This task shows how to insert a Parallel Contour Roughing 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 NC Manufacturing > Lathe Machining from the Start menu. Make the Manufacturing Program current in the specification tree.
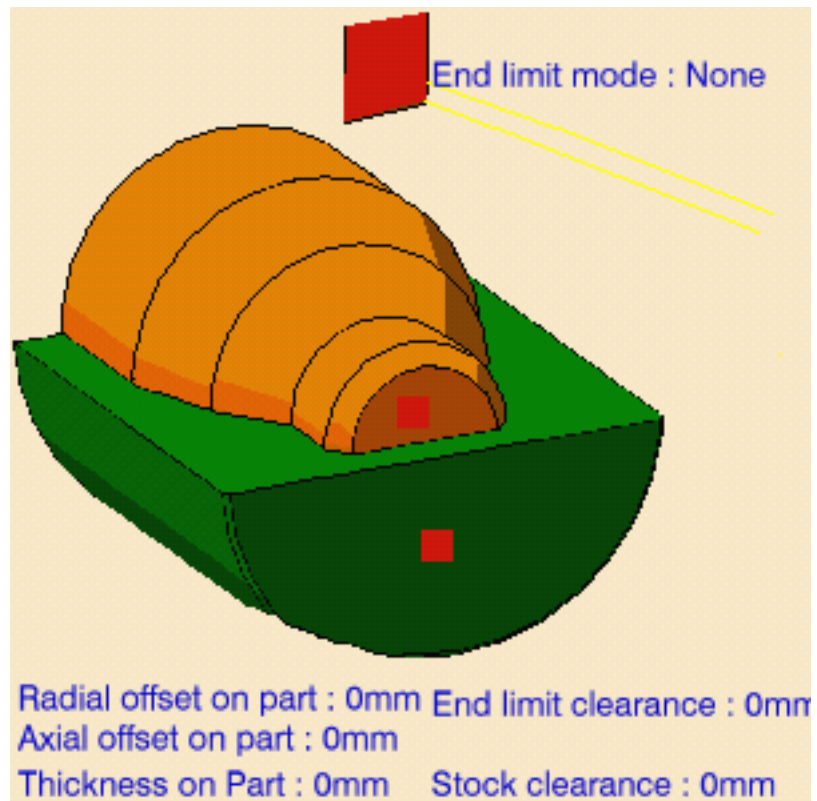
1. Select the Roughing icon.

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

   The Roughing 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.

   End limit mode : None

   Radial offset on part : 0mm   End limit clearance : 0mm
   Axial offset on part : 0mm
   Thickness on Part : 0mm   Stock clearance : 0mm

   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 ![icon] to specify the main machining strategy parameters:

    - Roughing mode: **Parallel Contour**
    - Orientation: **External**
    - Location: **Front**
    - Machining direction: **To head stock**.

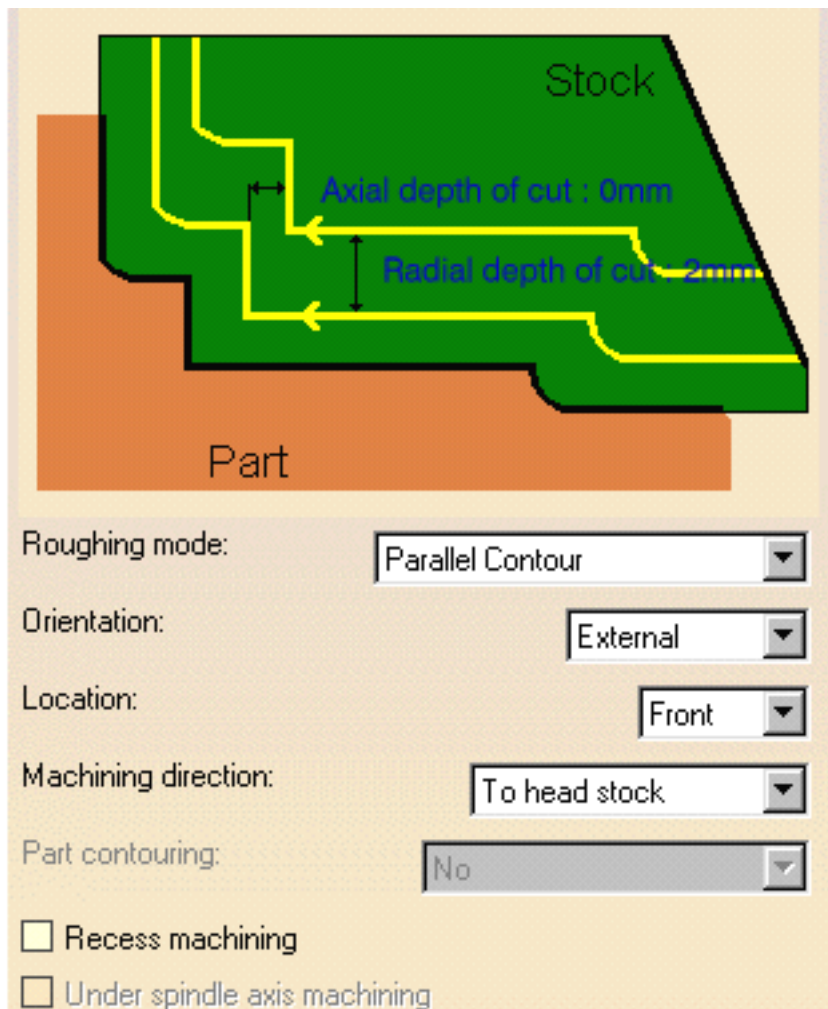4. Double click **Axial depth of cut** in the icon.

    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 page (lead-in and so on).
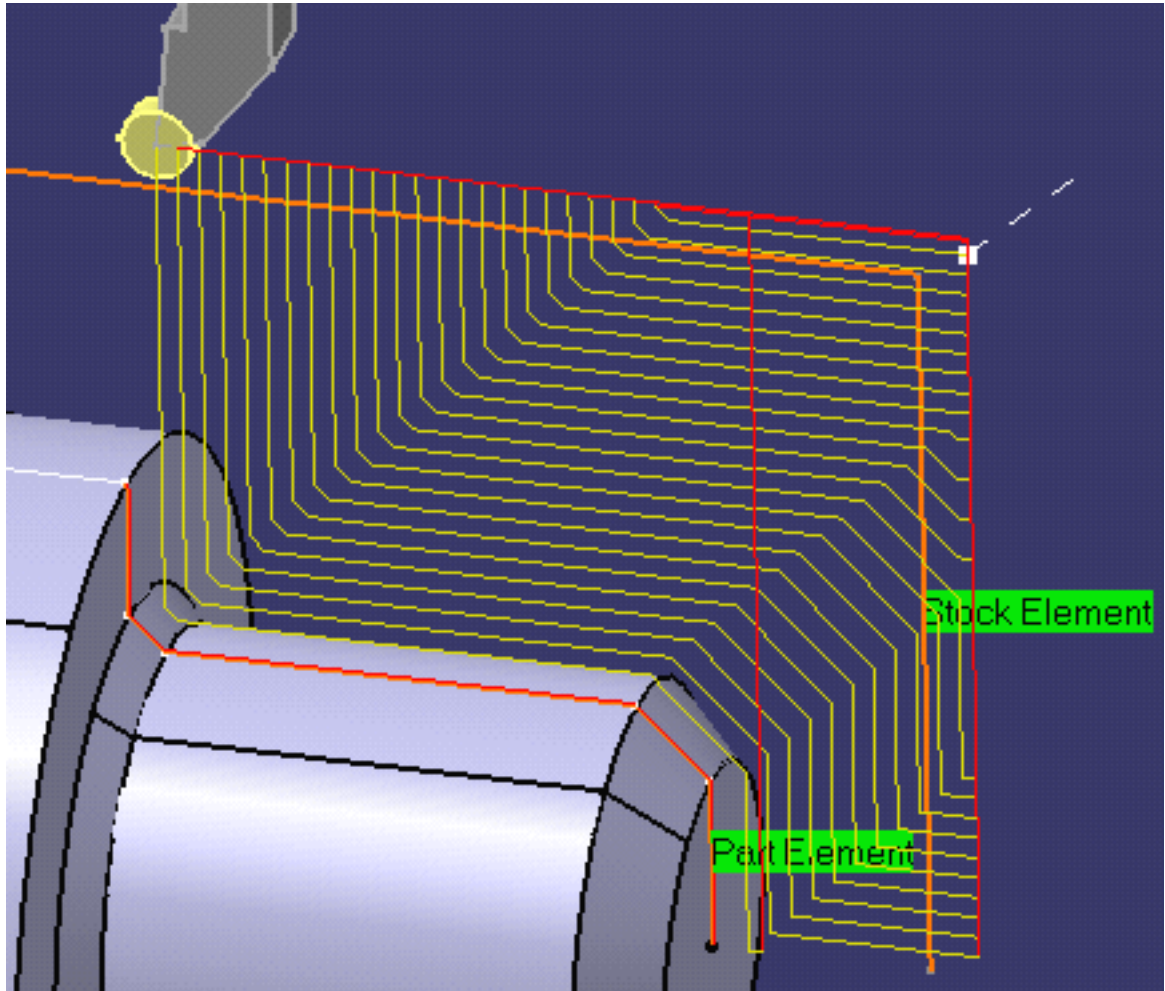
Roughing mode:	Parallel Contour
Orientation:	External
Location:	Front
Machining direction:	To head stock
Part contouring:	No
☐ Recess machining	
☐ Under spindle axis machining	

(Stock / Axial depth of cut : 0mm / Radial depth of cut : 2mm / Part shown in illustration)

ⓘ   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 ![icon] 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 ![icon] to specify the feedrates and spindle speeds for the operation. See Feeds and Speeds for Roughing for more information.

6. If you want to specify approach and retract motion for the operation, select the Macros tab page ![icon] to specify the desired transition paths. See Define Macros on a Lathe Operation for an example.

Stock Element

Part Element

7. Click OK to create the operation.

Create a Recessing Operation

This task shows how to insert a Recessing 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 NC Manufacturing > Lathe Machining from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Recessing icon.

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

   The Recessing 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.

   Radial offset on part : 0mm
   Axial offset on part : 0mm
   Thickness on part : 0mm   Clearance on stock : 0mm

   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 page (lead-in and so on).

Stock

Max depth of cut: 3mm

Part

Recessing mode:	Zig zag
Orientation:	External
Machining direction:	To head stock

☐ Part contouring

☐ Under spindle axis machining

ⓘ 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 Recessing 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 an example.

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

7. Click OK to create the operation.

Create a Grooving Operation

This task shows how to insert a Grooving 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 NC Manufacturing > Lathe Machining from the Start menu. Make the Manufacturing Program current in the specification tree.
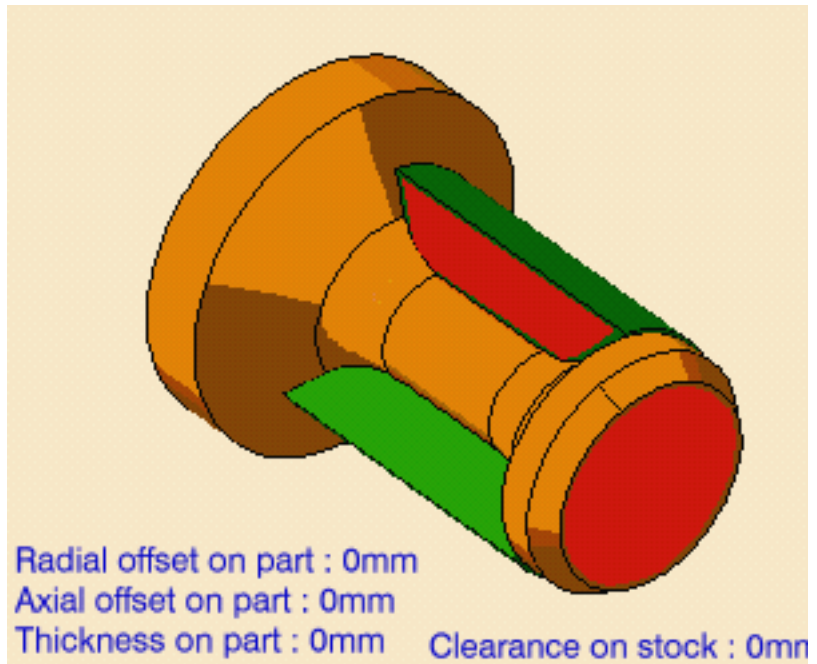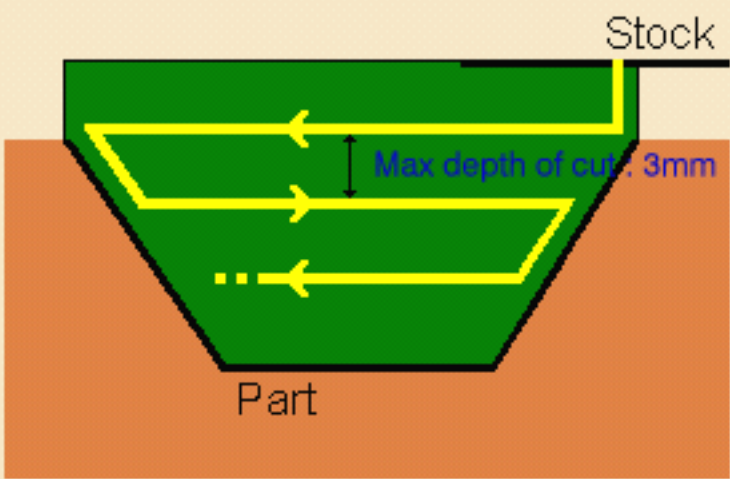
1. Select the Grooving icon .

   The Grooving 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.

Radial offset on part : 0mm Fixture clearance : 0mm
Axial offset on part : 0mm
Thickness on part : 0mm   Clearance on stock : 0mm

3. Select the Strategy tab page [icon] 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 page (lead-in and so on).

Max depth of cut : 3mm

Orientation:	External ▼
First plunge position:	Center ▼
Next plunges position:	To head stock ▼
☐ Grooving by level	
☐ Part contouring	
☐ Under spindle axis machining	
Tool compensation	P9 ▼
☐ Change output point	

(i) 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 [icon] 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 [icon] to specify the feedrates and spindle speeds for the operation. See Feeds and Speeds for Grooving for more information.

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

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

7. Click OK to create the operation.

Create a Profile Finishing Operation

This task shows how to insert a Profile Finishing 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 NC Manufacturing > Lathe Machining from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Profile Finishing icon .

   The Profile Finishing 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.

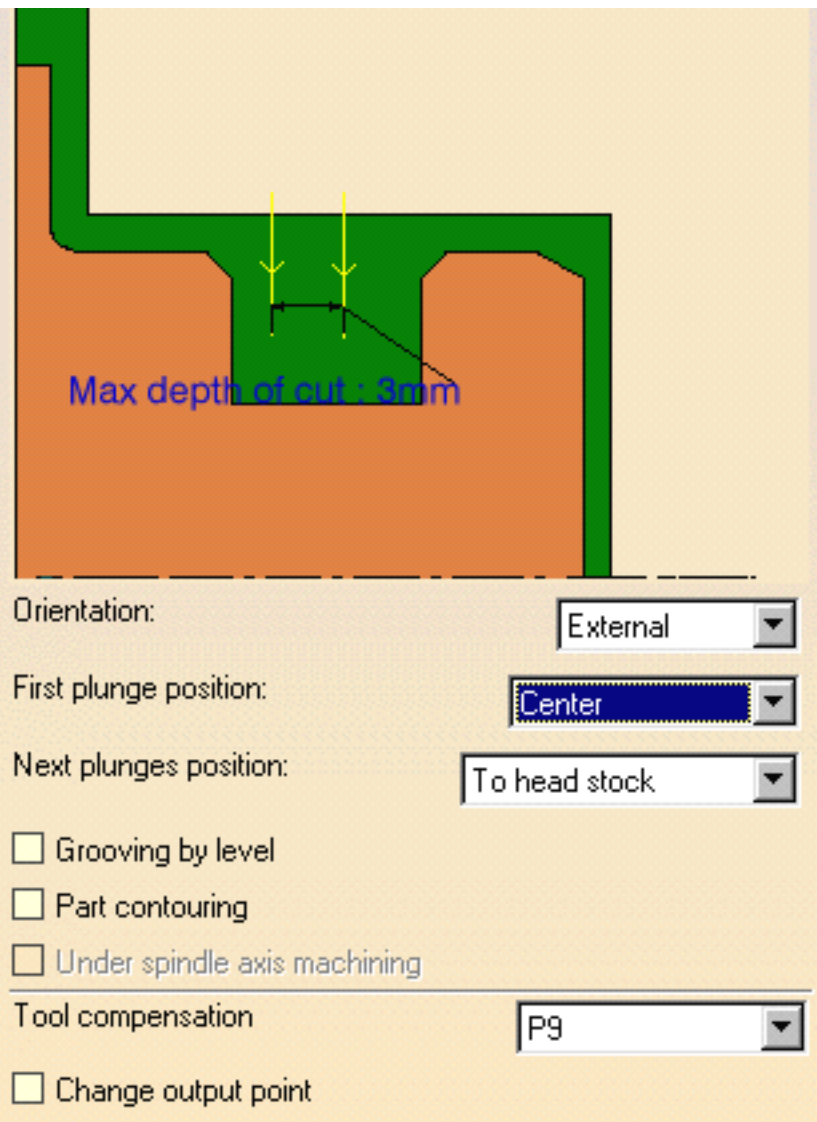2. Click the red part in the icon, then select the desired part profile in the 3D window.

End limit mode : None

Start limit mode : None

Radial offset on part : 0mm   End limit clearance : 0mm
Axial offset on part : 0mm
Thickness on part : 0mm   Start limit clearance : 0mm

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

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.

It is possible to locally invert the machining direction.

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 main 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 and Corner processing tab pages.

Orientation: External

Location: Front

Machining direction: To Spindle

Contouring for outside corners: Circular

☑ Recess machining
☐ Under spindle axis machining

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

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

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

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

Other contextual commands are available for analyzing and resetting local information.

Please refer to Local Information for more details.

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

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

Create a Groove Finishing Operation

This task shows how to insert a Groove Finishing 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 NC Manufacturing > Lathe Machining from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Groove Finishing icon.

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

   The Groove Finishing 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 and thicknesses that you can assign to the selected profile, you can also add local offsets and thicknesses.

   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.
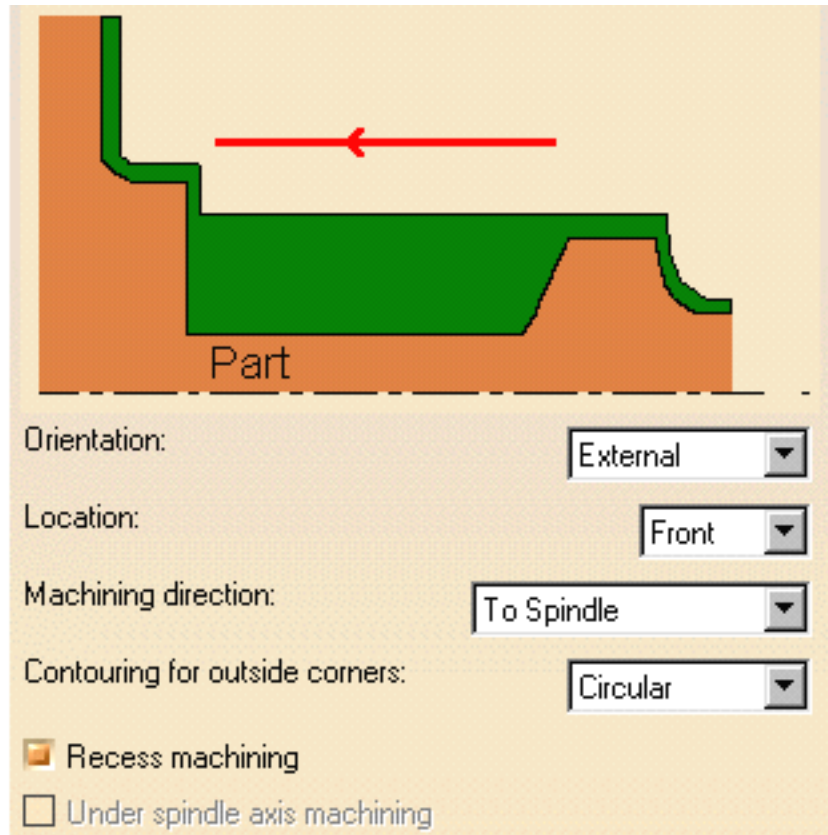
   Please refer to Local Information for more details.

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 tab pages.
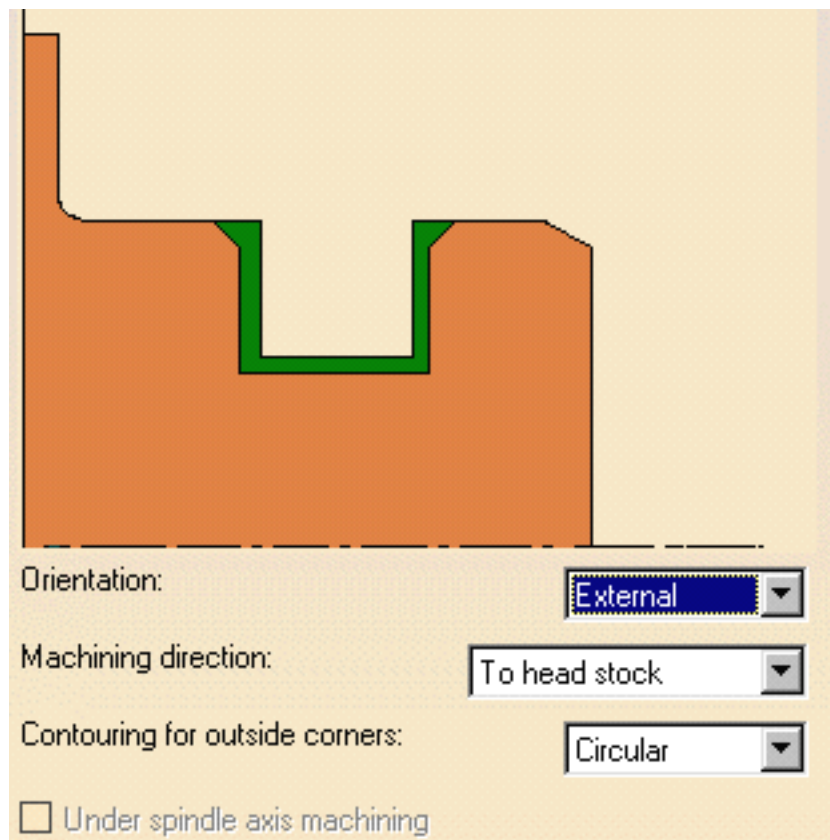
   Orientation: External

   Machining direction: To head stock

   Contouring for outside corners: Circular

   ☐ Under spindle axis machining

   i 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.

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

6. Click OK to create the operation.

Create a Threading Operation

This task shows how to insert a Threading 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 NC Manufacturing > Lathe Machining from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Threading icon.

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

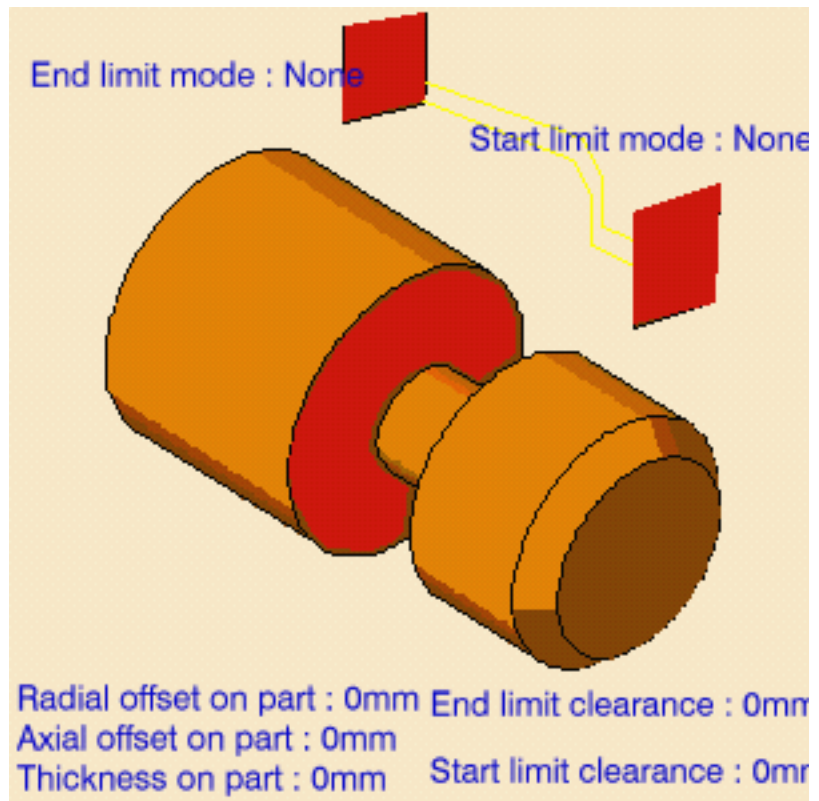   The Threading dialog box appears directly at the Geometry tab page.

   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. Double click Length in the icon then specify the desired length of threading in the Edit Parameters dialog box that appears.

End limit mode : None

Start limit mode : None

Length : 1mm

Start limit clearance : 0.8mm   End limit clearance : 0.8m
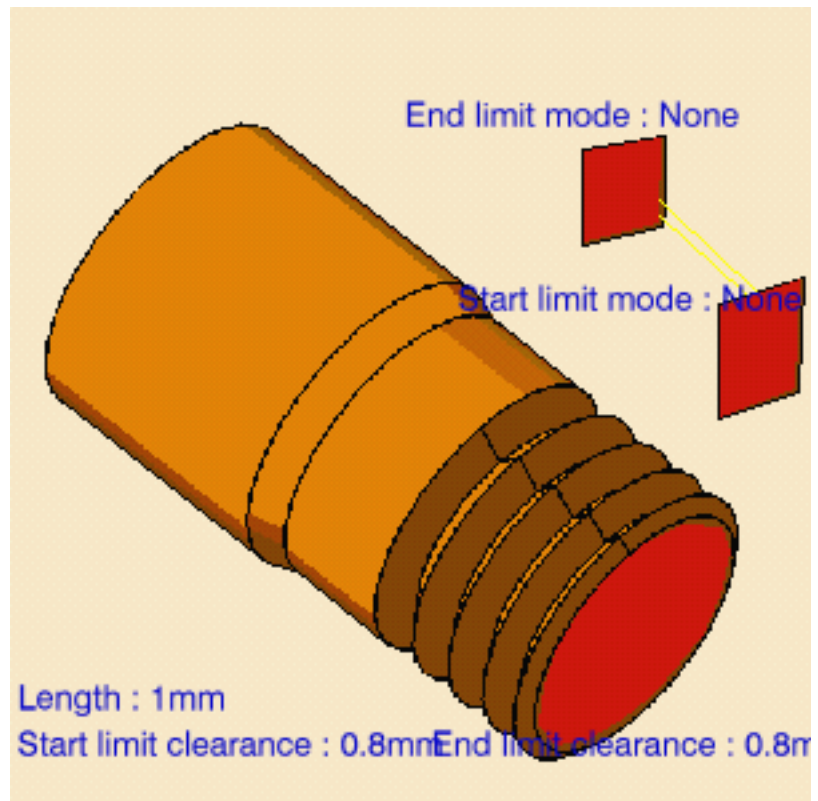
4. Select the Strategy tab page

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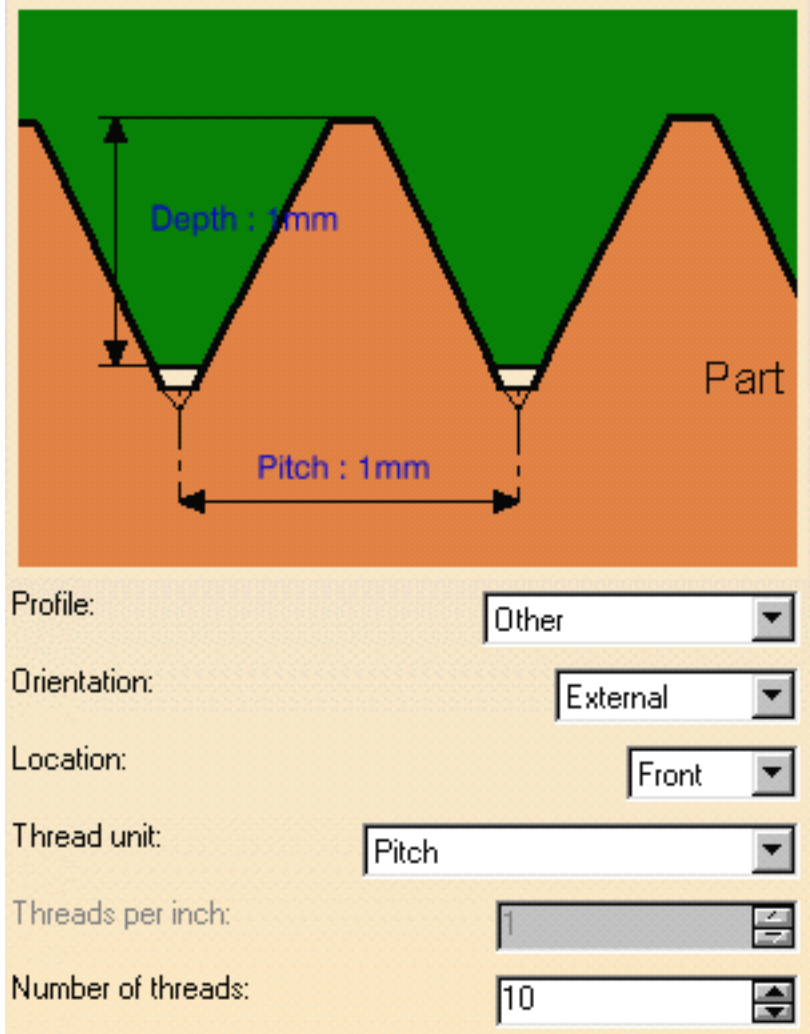
5. If you want to generate CYCLE statements, you must select the **Output CYCLE syntax** checkbox in the Options tab and set the Syntax Used option to Yes in the NC Output generation dialog box.
Otherwise, GOTO statements will be generated.

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 machining spindle speed

for threading.

7. 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.

8. Click OK to create the operation.

Example of output

If your PP table is customized with the following statement for Lathe Threading operations:

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

A typical NC data output is as follows:

```
CYCLE/THREAD, 10.000000
```

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

Create a Sequential Machining Operation

This task illustrates how to create a Sequential Machining 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 NC Manufacturing > Lathe Machining from the Start menu. Make the Manufacturing Program current in the specification tree.

1. Select the Sequential Machining icon.

   A Sequential Machining 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 a curve**.
   - Set a thickness on the first check element to 2mm.
   - Click OK to define the motion.

Go.1 n° 1

Name : Go.1   << | New | >>

Geometry | Strategy

First check mode : To a curve   Second check mode : To a curve

Check limit mode : Extended   Second Check limit mode : Extended

First Check Element

Thickness on check: 2mm

Axial check offset: 0mm

Radial check offset: 0mm

Second Check Element (Go-Go)

Thickness on 2nd check: 0mm

Axial 2nd Check offset: 0mm

Radial 2nd Check offset: 0mm

OK | Cancel
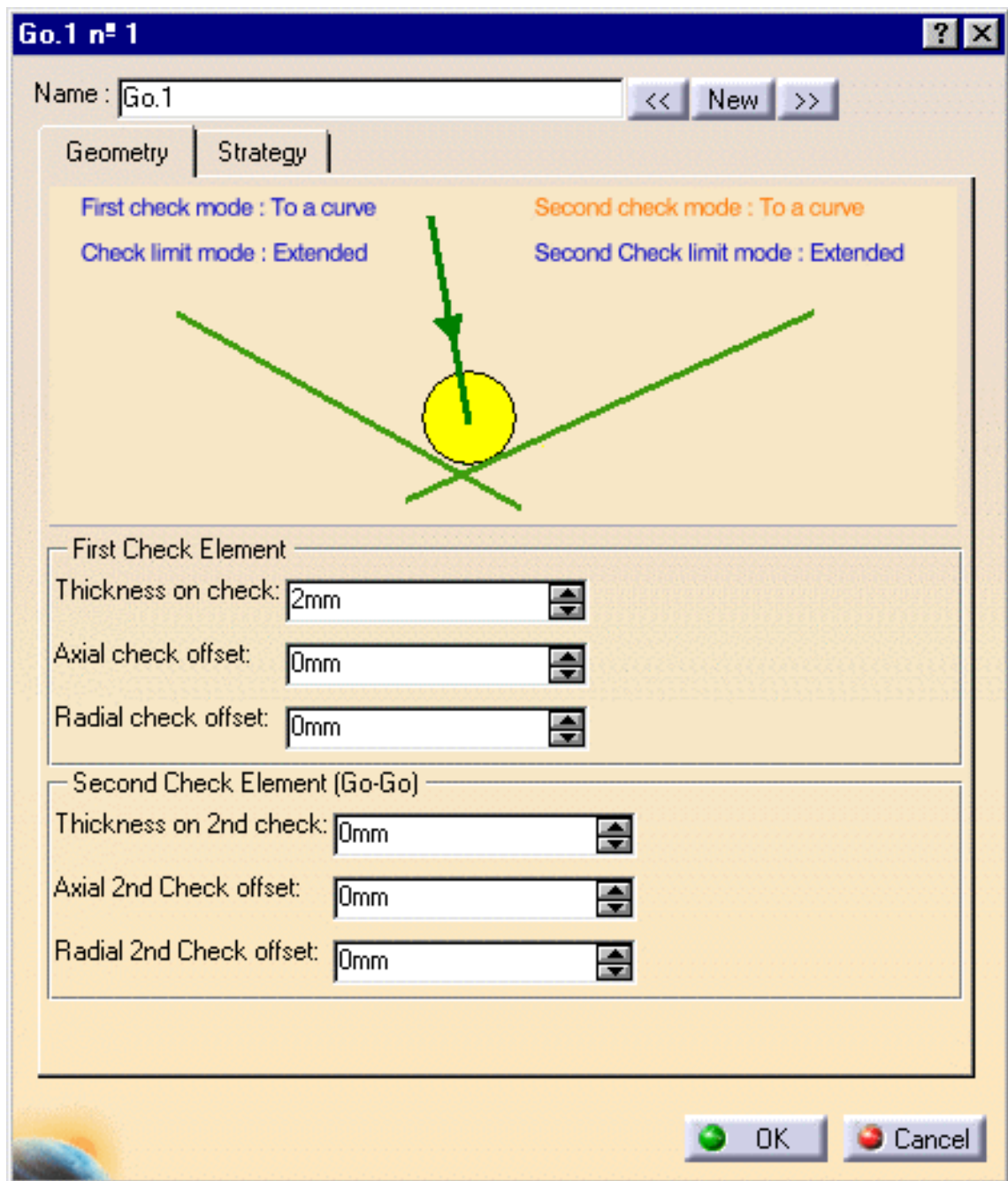
3. Select the Go icon again to define the second motion.

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 a curve**.
- Set a thickness 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.
- Set the check mode to **To a curve**.
- Click OK to define the motion.

GoIndirv.1 n⁼ 3   ? ✕

Name : GoIndirv.1   << | New | >>

| Geometry | Strategy |

Drive element type : Line

Check limit mode : Extended

First check mode : To a curve

Drive direction : Same

Check Element

Thickness on check: 0mm

Axial check offset: 0mm

Radial check offset: 0mm

Drive

Angle: 0deg
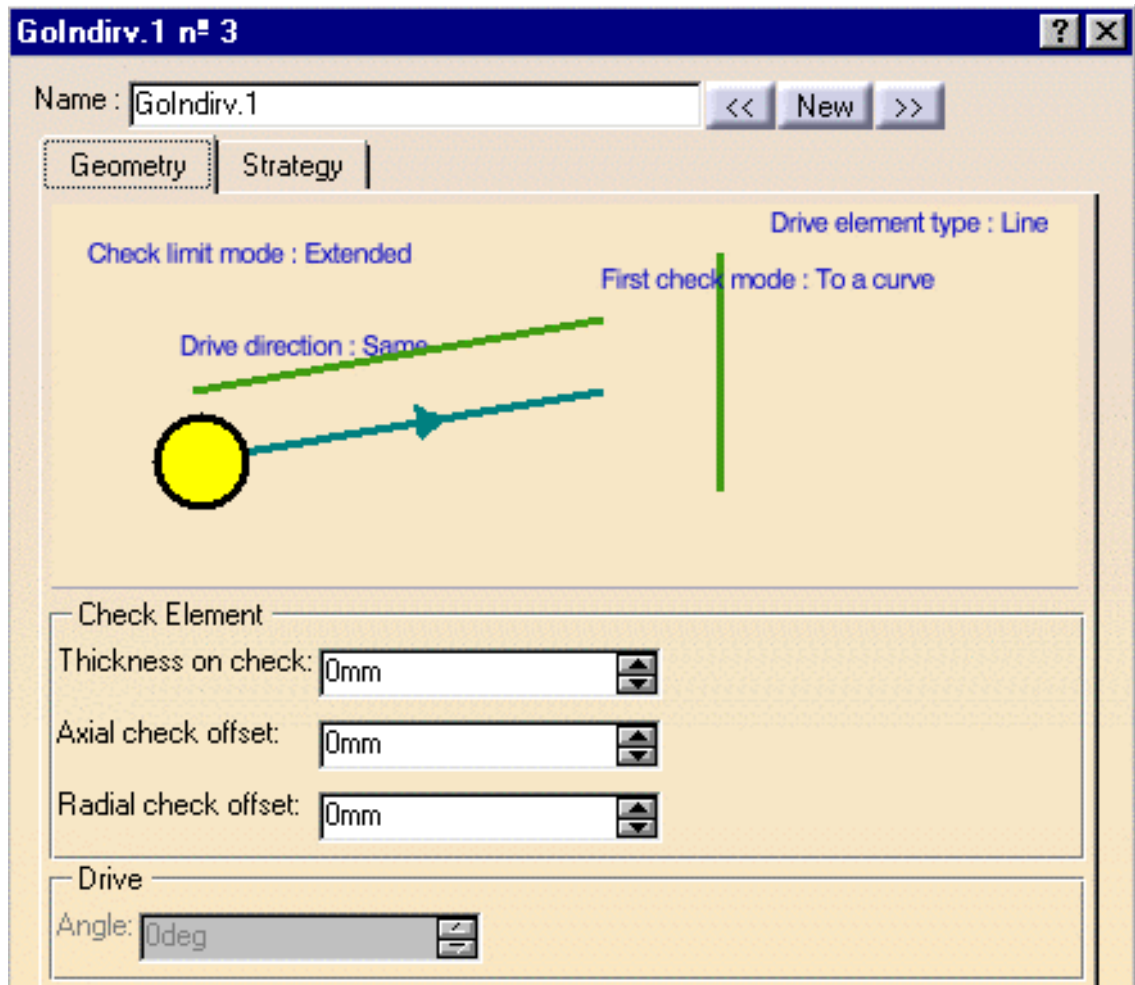
5.

Select the Follow icon .

In the Geometry tab:

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

Follow.1 nº 5

Name : Follow.1   << | New | >>

Geometry | Strategy

First check mode : On a curve

Check limit mode : Extended

Check Element

Thickness on check: 0mm

Axial check offset: 0mm

Radial check offset: 0mm

☑ Change Drive

Drive direction: Same
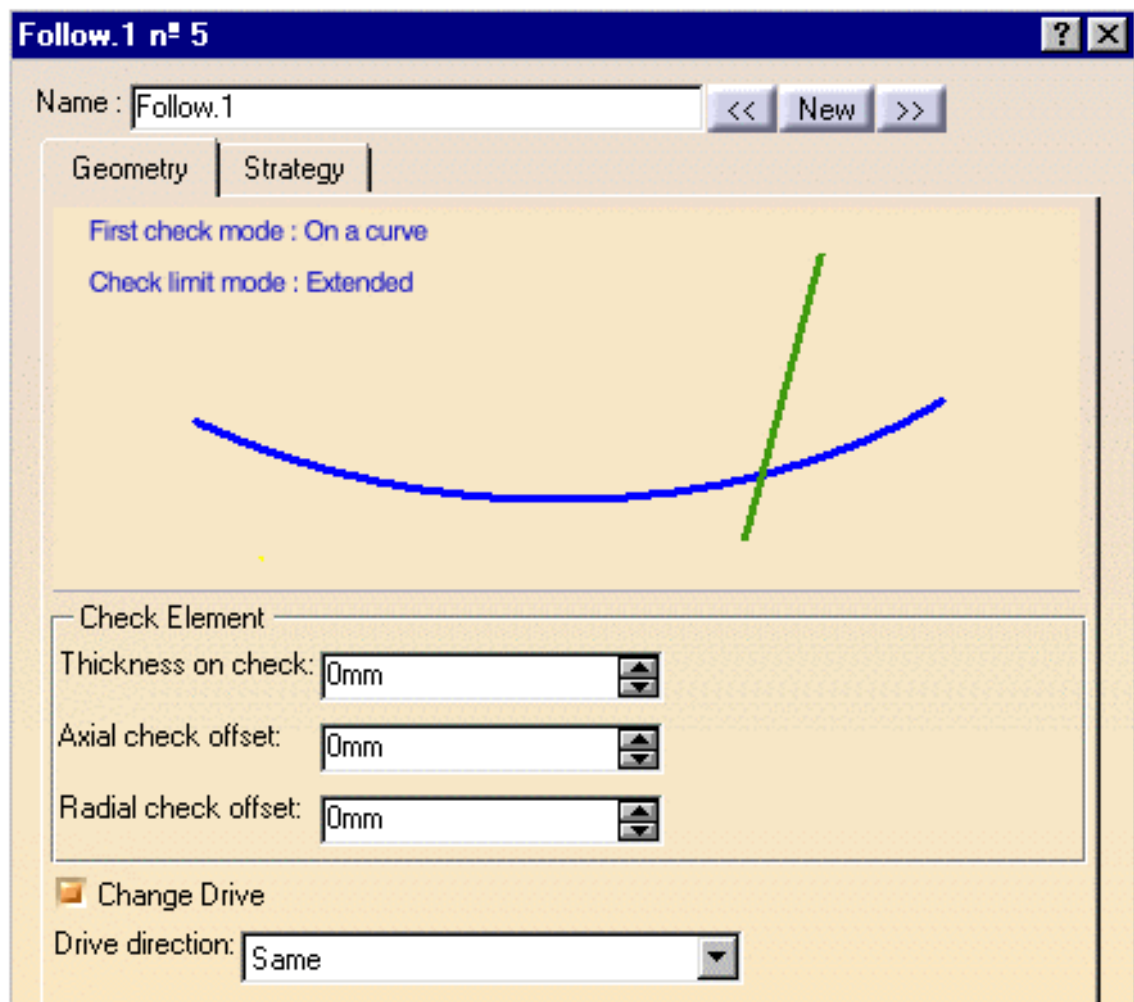
6. Select the PP word icon 📄, then specify a PP word in the dialog box that appears (DELAY/5, for example).

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.

GoDelta.1 n° 6

Name : GoDelta.1   << | New | >>

Geometry | Strategy

Delta check mode : Line and distance

Distance

Delta Elements

Delta Axial (Z):   3mm

Delta Radial (X):   3mm

Distance: 2mm

Angle: 0deg
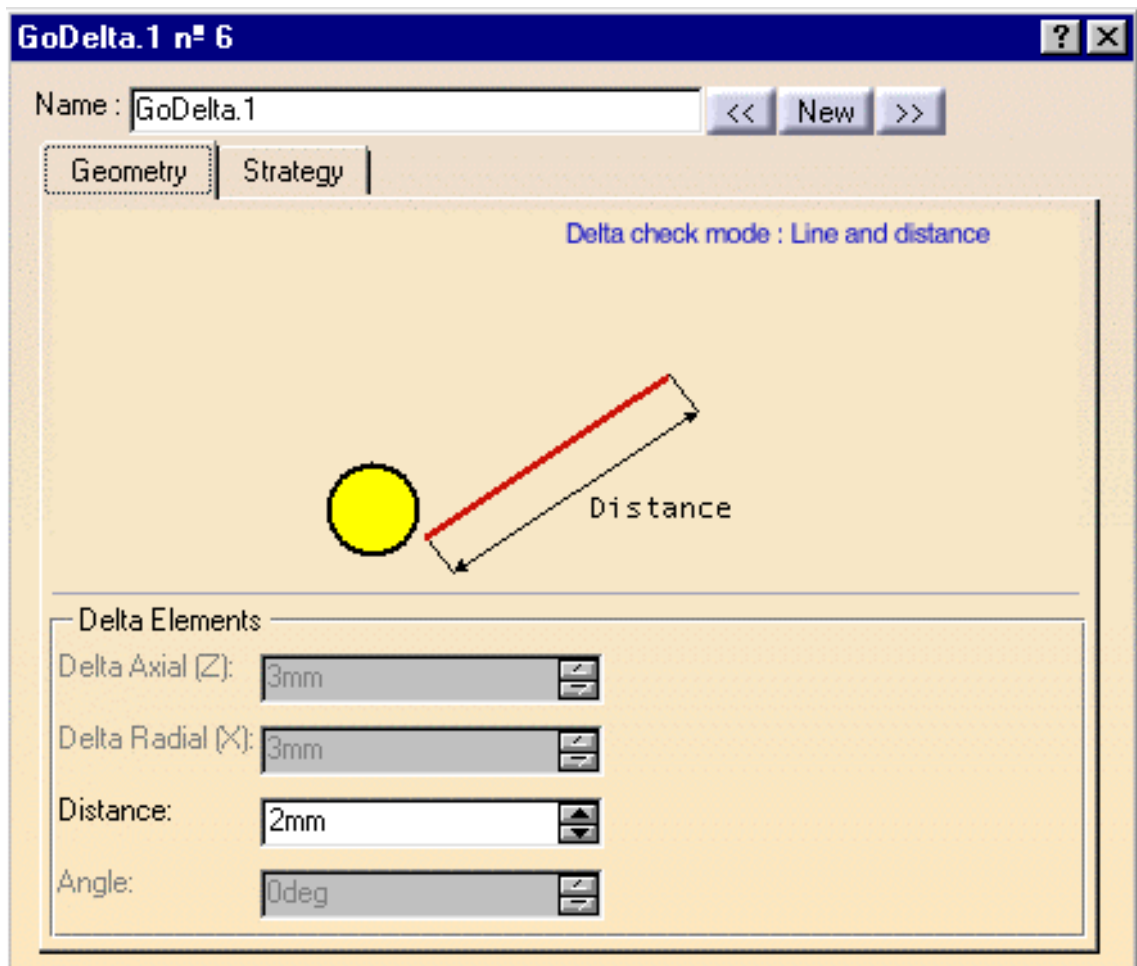
Other optional parameters can be set in the Strategy tab page for each of the motions.
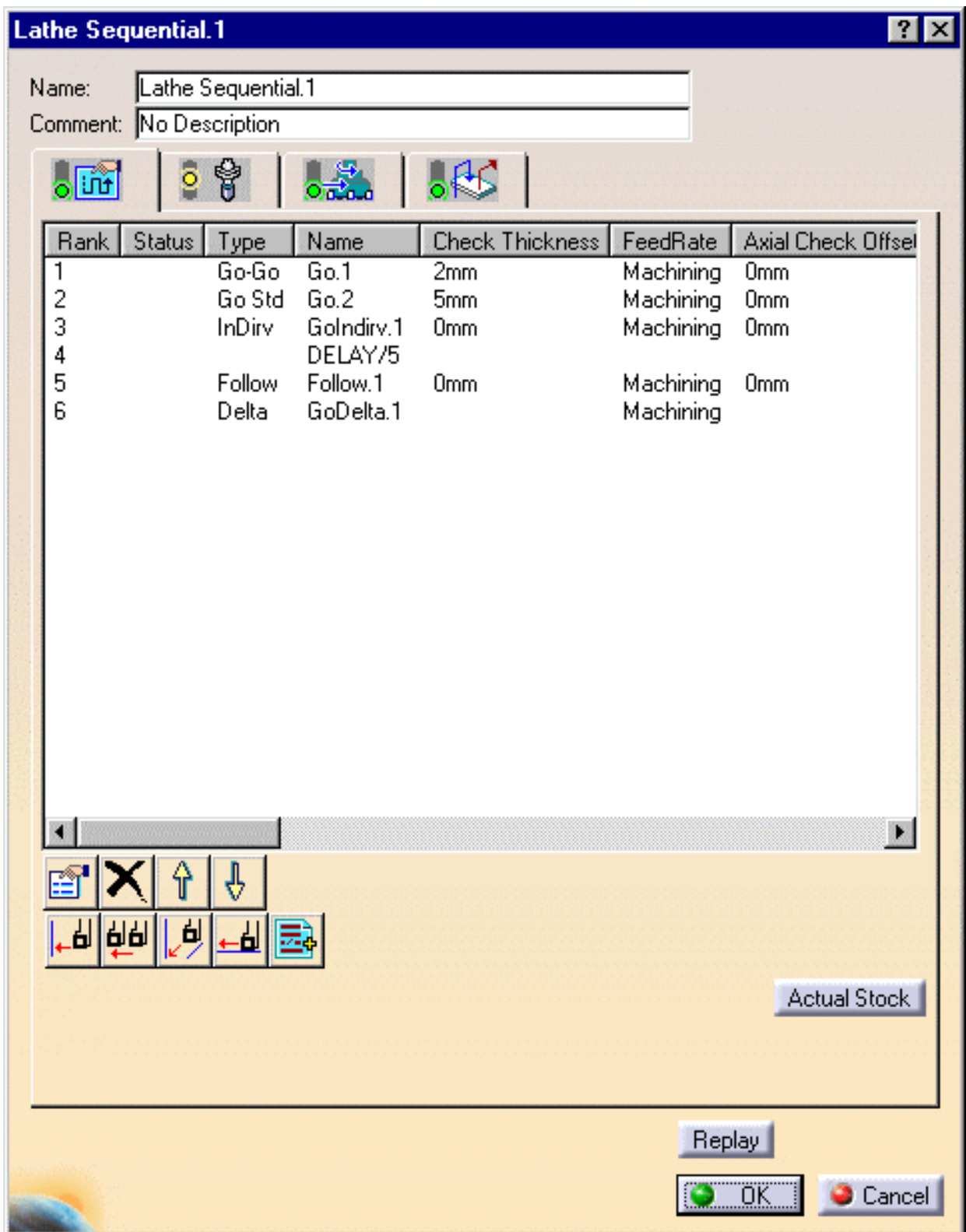
Lathe Sequential.1

Name: Lathe Sequential.1
Comment: No Description

Rank	Status	Type	Name	Check Thickness	FeedRate	Axial Check Offset
1		Go-Go	Go.1	2mm	Machining	0mm
2		Go Std	Go.2	5mm	Machining	0mm
3		InDirv	GoIndirv.1	0mm	Machining	0mm
4			DELAY/5			
5		Follow	Follow.1	0mm	Machining	0mm
6		Delta	GoDelta.1		Machining	

Actual Stock

Replay   OK   Cancel

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 Roughing for more information.

9. Select the Macros tab page ![icon] 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.
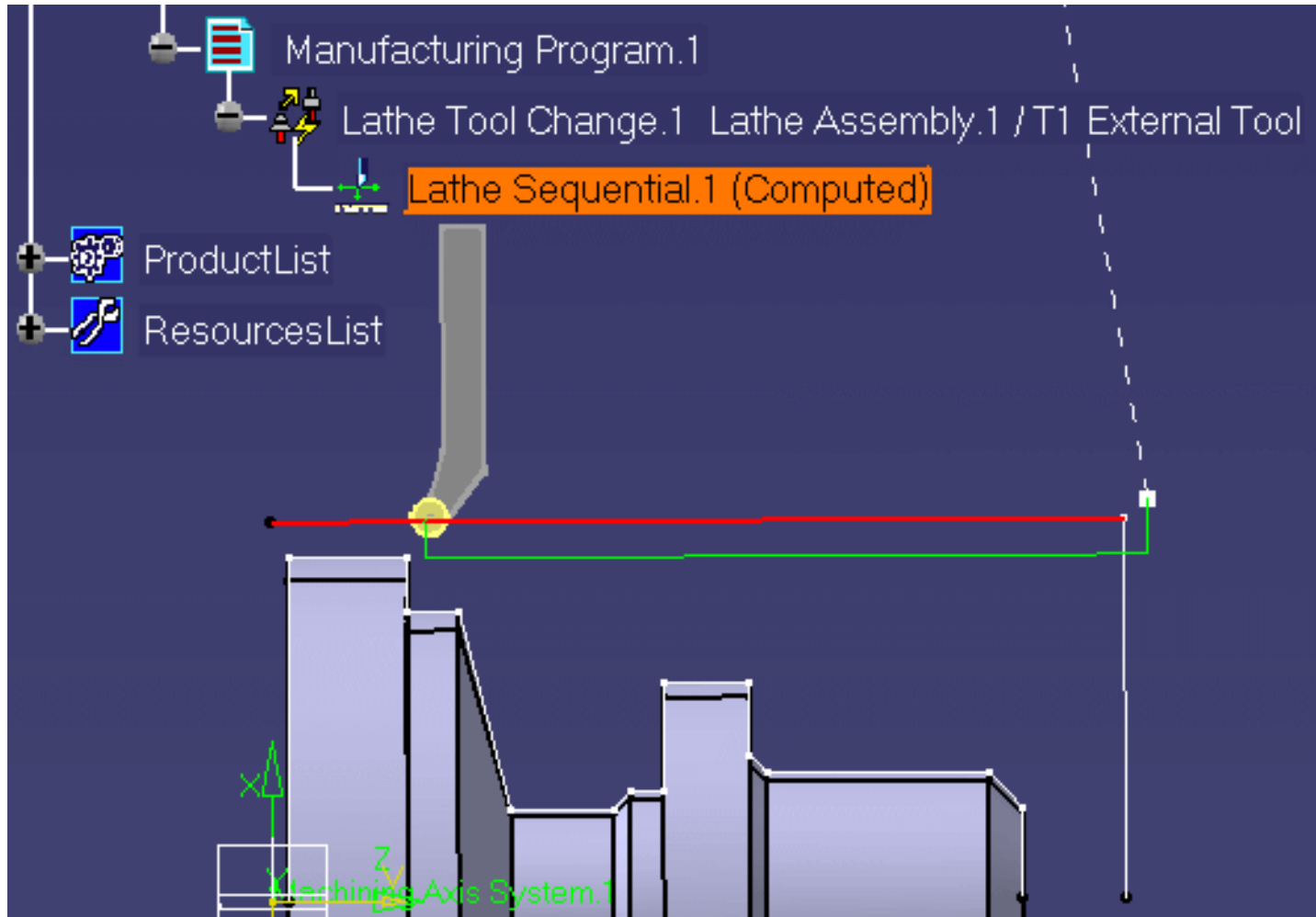
10. Click OK to create the operation.

Manage Local Information

This task illustrates how to manage Local Information (local feedrates and thicknesses) 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 .
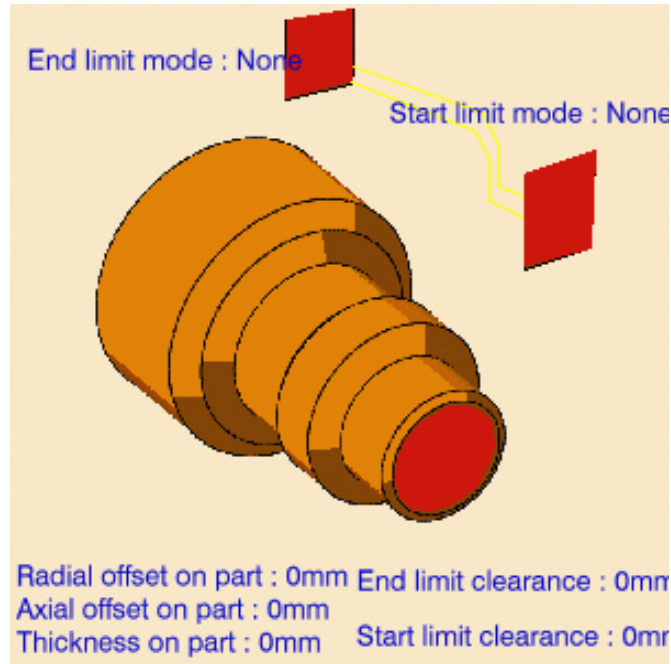
Local Information can be added on lines, arcs and curves if they are part of a finished profile.
Click the red part in the icon then select the desired finish profile in the 3D window.
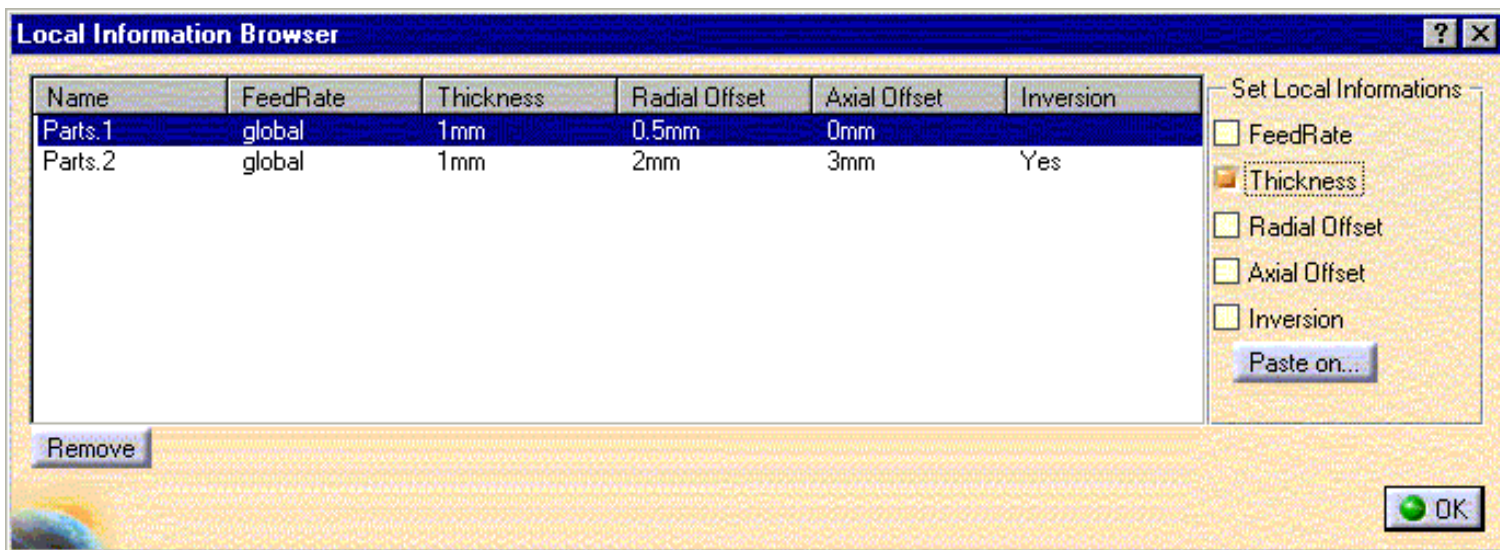
To create or edit local information

To add local information on an element, right click on it and select "Add Local Information" in the contextual menu.

To edit Local Information from an element, right click on it and select "Edit Local Information" in the contextual menu.

In both cases, set the information you want and validate with "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 "Copy Local Information" in the contextual menu, then right click on the target element and select "Paste Local Information".

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 "Browse Local Information".

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 "Remove Local Information" in the contextual menu.

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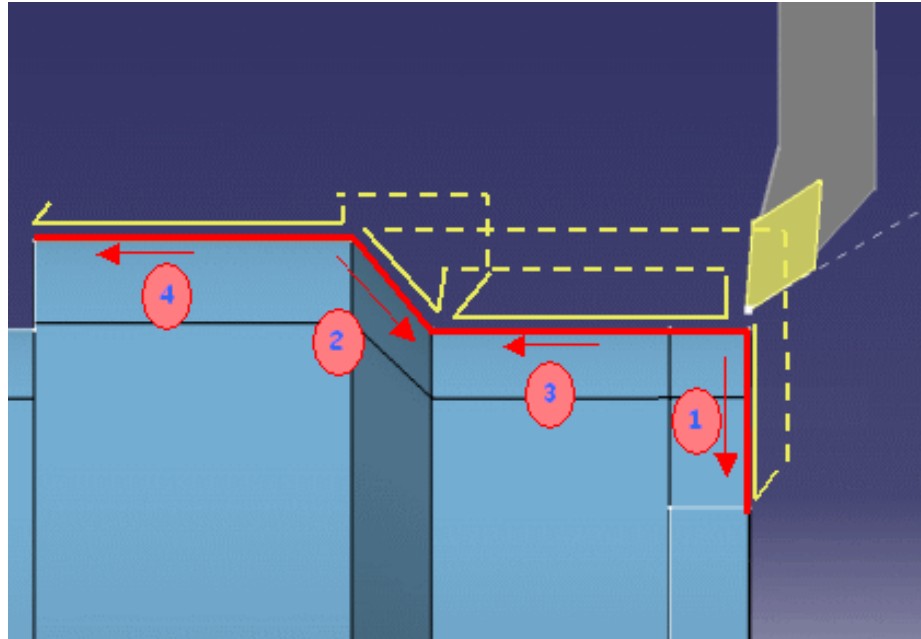
In the Strategy tab page, click on the Local Invert tab.

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.
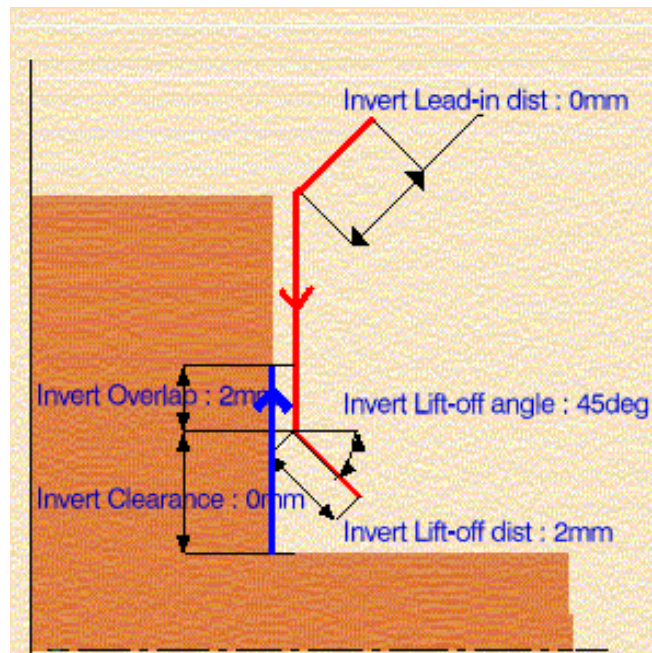
Axial Machining Operations

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.

Managing Manufacturing 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 NC manufacturing 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 Lathe Machining Operation: Select the Macros tab page when creating or editing a lathe machining 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 Lathe Machining Operation

This task shows you how to edit the tool of a lathe machining 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 Tool 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.
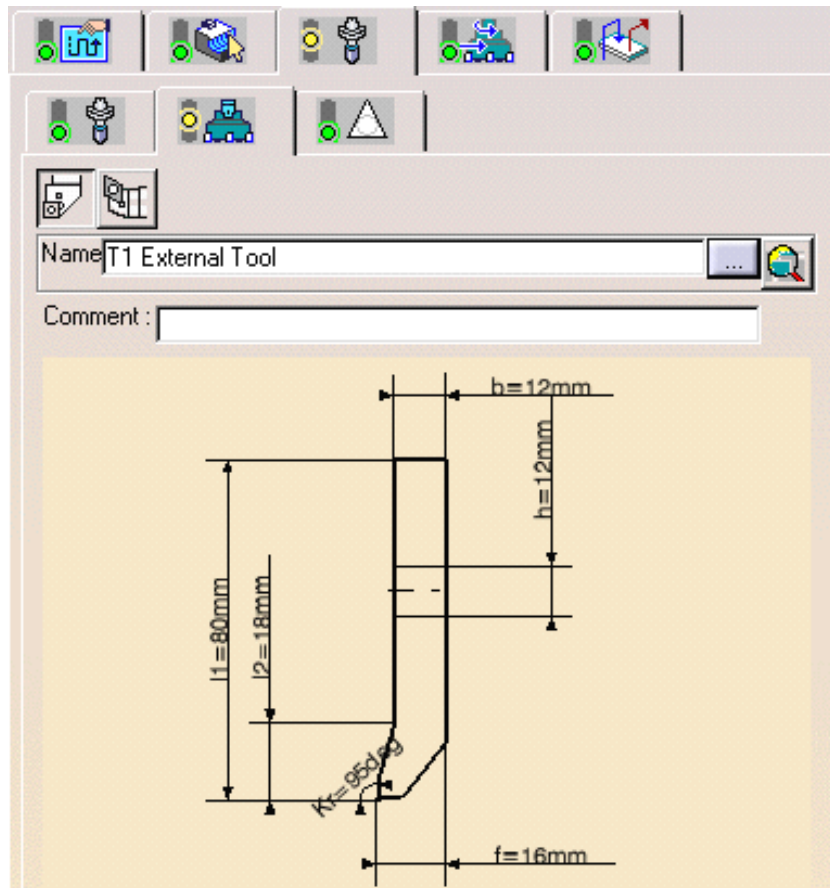
3. To select a tool that is already used in the document:

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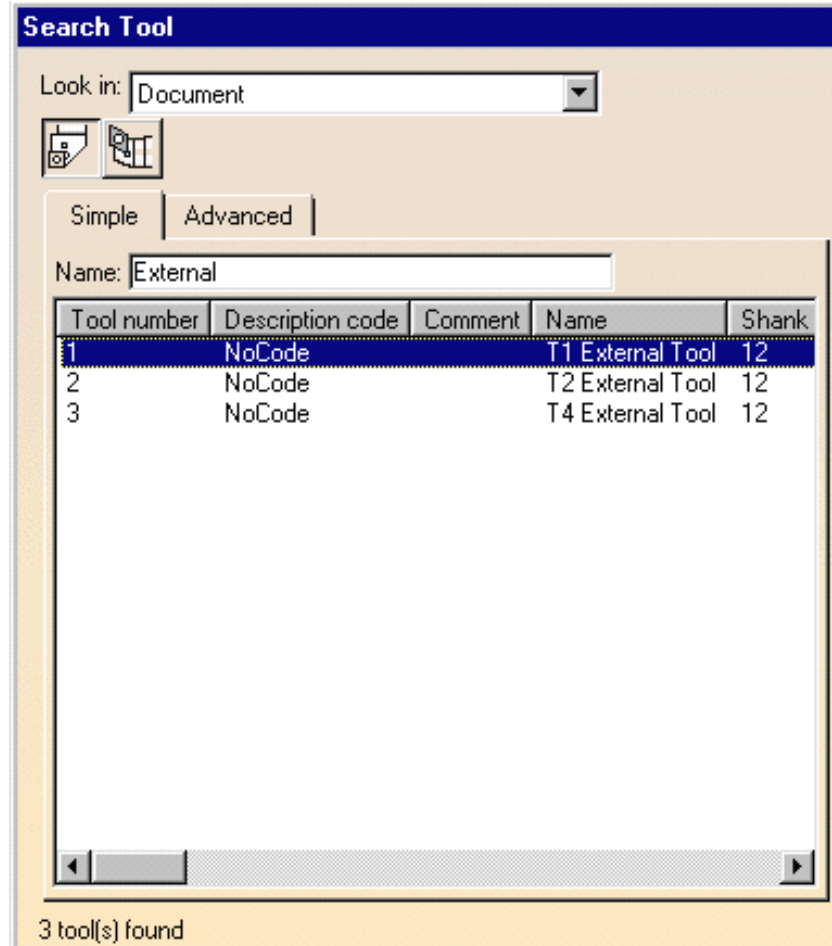
● Use the **Look in** combo to specify where you want to search for the tool:
   ● in the current document,
   ● in a tool catalog
   ● in an external tool database such as the Walter TDM (Tool Data Management) or CATIA Version 4 Manufacturing database
   ● in a Feeds and Speeds catalog.

● 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

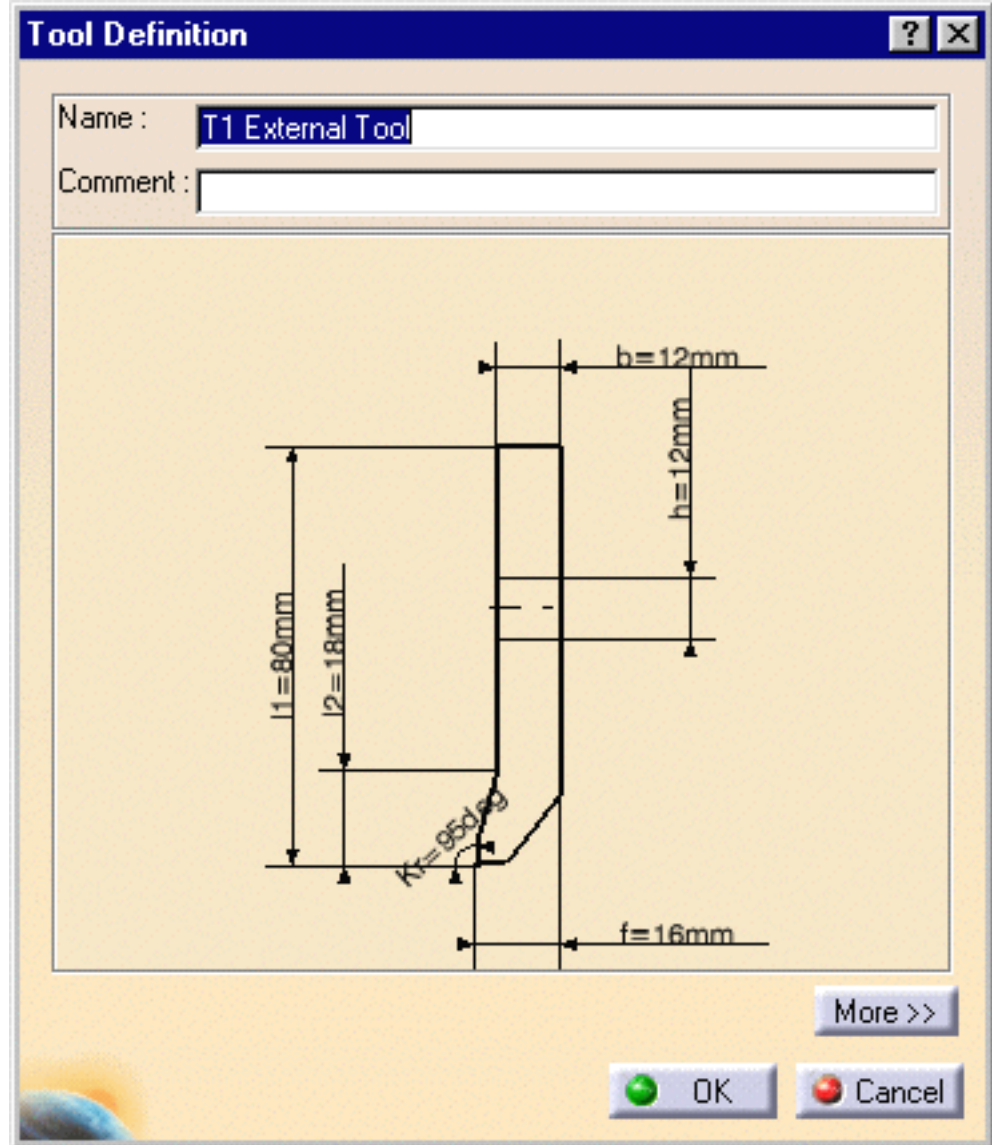This task shows you how to edit a lathe tool that is already used in your document.

1. To edit a tool in the resource list either double click it or right click it and select the **Definition** 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 for the tool.

   You can also assign a comment.

3. Click More to expand the dialog box to access the Geometry, Technology, and Compensation tab pages.

Tool Definition

Name : T1 External Tool

Comment :

b=12mm

h=12mm

l1=80mm

l2=18mm

Kr=95dg

f=16mm

More >>

OK   Cancel

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.

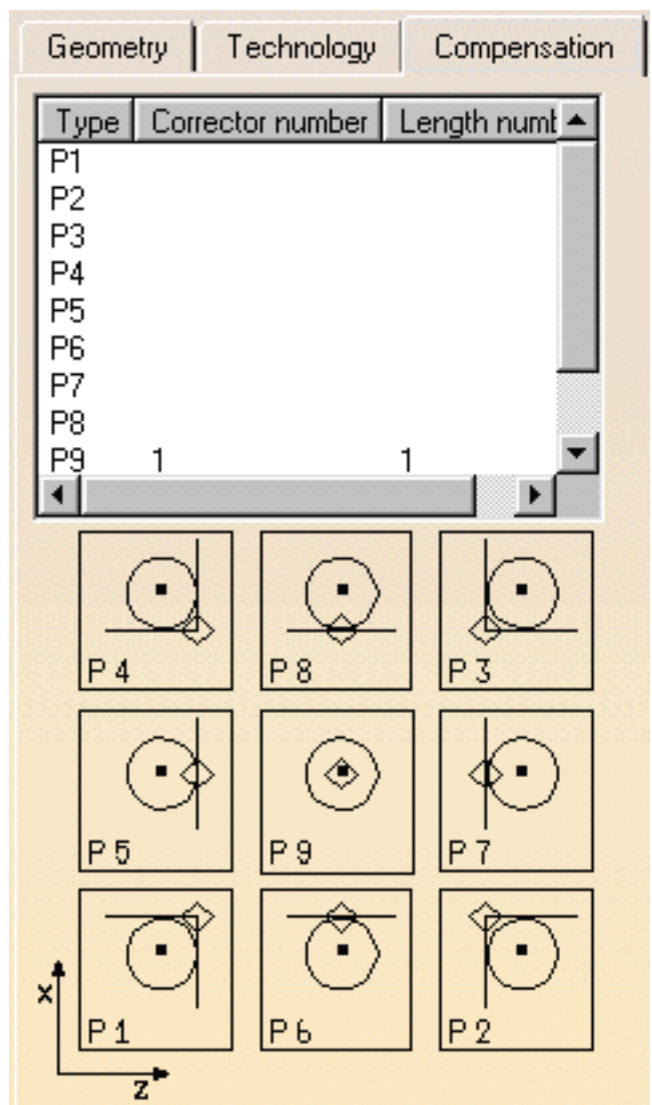The icon representation of the tool is updated with these values.

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	

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.

Type	Corrector number	Length numb ▲
P1		
P2		
P3		
P4		
P5		
P6		
P7		
P8		
P9	1	1

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 user representation (CATPart) can be assigned to the tool by the 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 either double click it or right click it and select the **Definition** contextual command.

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

2. If needed, enter a new name for the tool assembly.

   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.

Tool Assembly Definition

Name : Lathe Assembly.1

Comment :

Tool number : 0
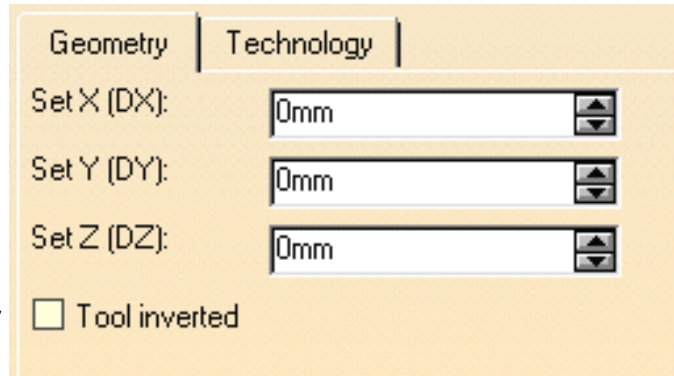
Setup angle : 0deg

More >>

OK   Cancel
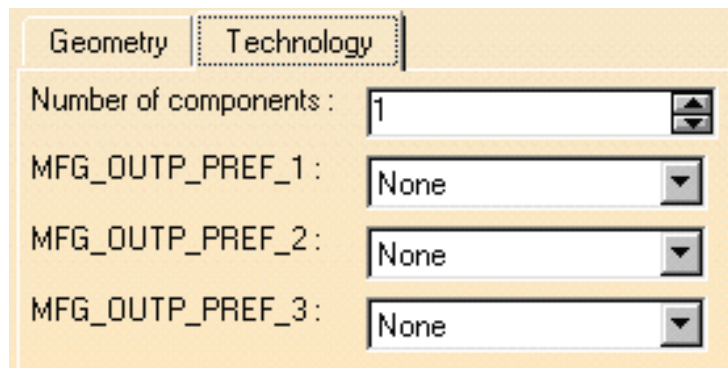
5. You can specify the tool assembly geometry in two ways:

- double click a parameter in the large tool assembly icon and enter the desired value in the Edit Parameter dialog box that appears

Geometry	Technology
Set X (DX):	0mm
Set Y (DY):	0mm
Set Z (DZ):	0mm
☐ Tool inverted	

- or enter the desired values in the **Geometry** tab page.

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

Geometry	Technology
Number of components :	1
MFG_OUTP_PREF_1 :	None
MFG_OUTP_PREF_2 :	None
MFG_OUTP_PREF_3 :	None

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

A user representation (CATPart) can be assigned to the tool assembly by the contextual command in the Resource List.

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 either double click it or right click it and select the **Definition** contextual command.

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

InsertEdition

Name : Groove Insert.2

Comment :

Type : Groove

$$l = 6mm$$

$$r1 = 1.2mm \qquad r2 = 1.2mm$$
$$la = 3mm$$

More>>

OK   Cancel

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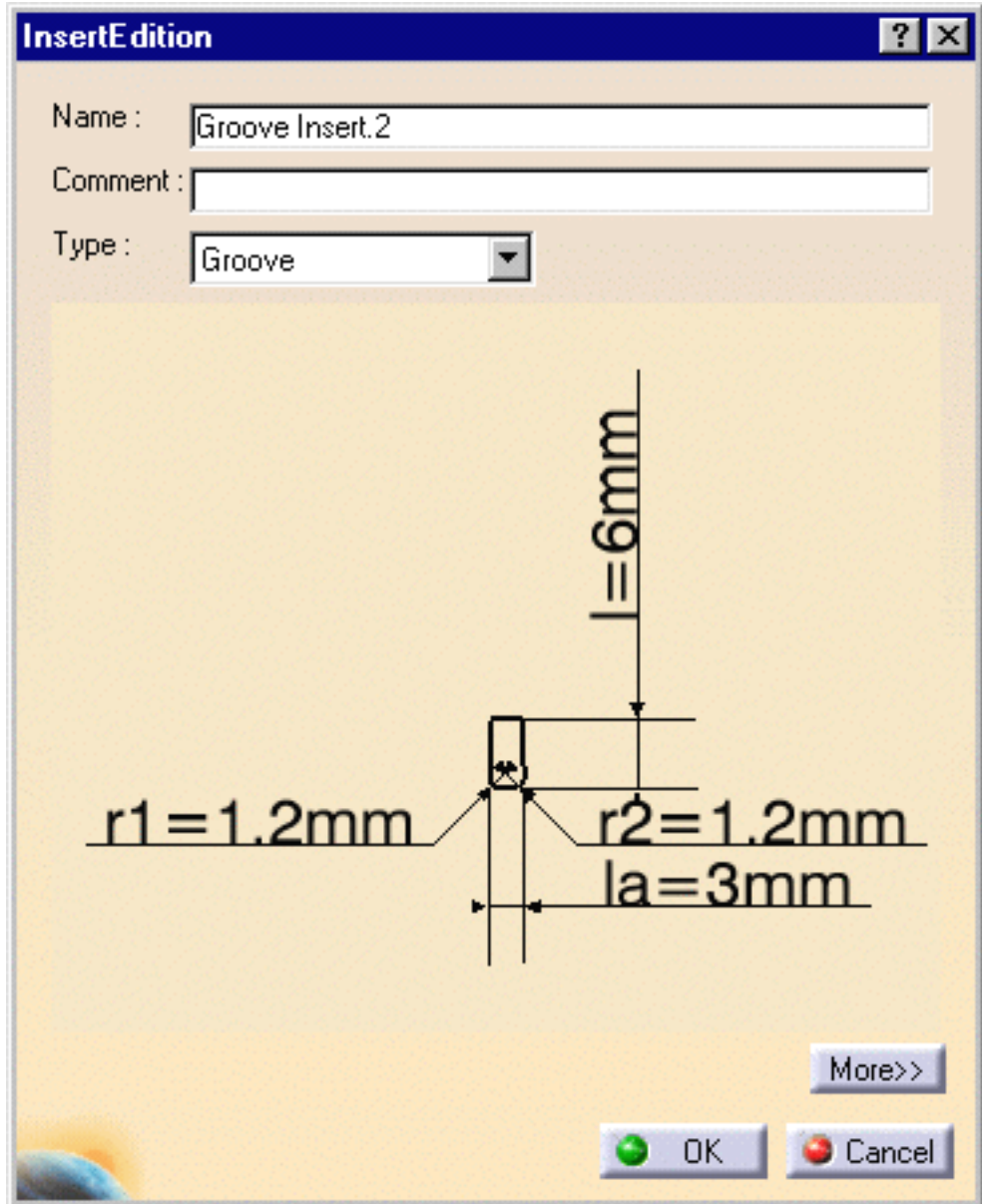
5. You can specify the lathe insert geometry in two ways:

- 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 feed/tooth : 0mm_turn

Roughing cutting speed : 0mm_mn

Roughing feed/tooth : 0mm_turn

MFG_PP_ROUGH : 0mm

MFG_VC_ECROUTAGE : 0mm_mn

MFG_SZ_ECROUTAGE : 0mm_turn

MFG_PP_ECROUTAGE : 0mm

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

A user representation (CATPart) can be assigned to the lathe insert by the contextual command in the Resource List.

Auxiliary Operations

The tasks for inserting auxiliary operations in the 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 Machining Axis System or Origin: Select the Machining Axis or Origin icon then specify the characteristics of the machining axis system or origin.

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

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

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 desired icon in the Tool Path Replay dialog box to run a material removal simulation either in Photo or 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.

Generate APT Source Code in Interactive Mode: Select the Generate NC Code Interactively icon to generate APT source code for the current manufacturing program.

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.

Part Operations, Programs and 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 will be stored in a CATProcess document and then as a catalog component.

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

Advanced Tasks

Tasks dealing with the following NC Manufacturing functionalities are described in the *NC Manufacturing Infrastructure User's Guide.*

Design Changes
Set Up and Part Positioning

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
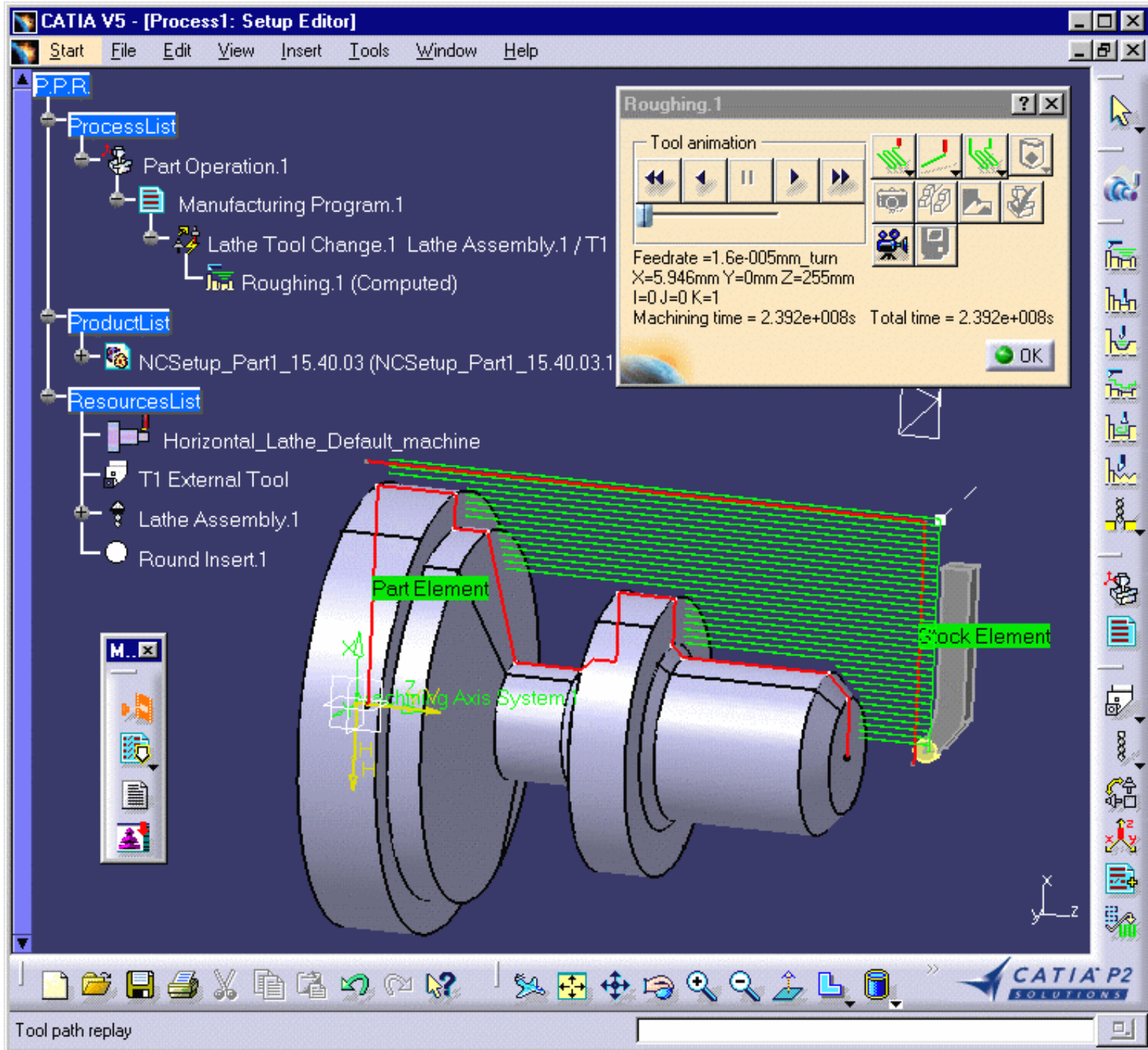
Lathe Machining Menu Bar

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 NC Manufacturing products are described in the *NC Manufacturing Infrastructure User's Guide*.

Insert Menu
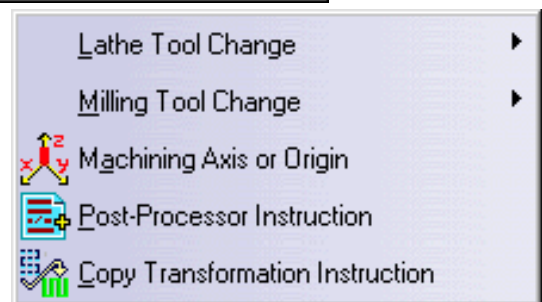
Insert > Machining Operations

Command...	Description...
Roughing	Create a Longitudinal Roughing Operation
	Create a Parallel Contour Roughing Operation
	Create a Face Roughing Operation
Grooving	Create Grooving Operation
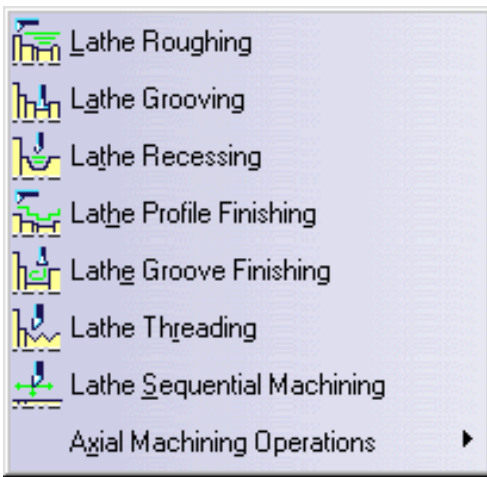
Recessing	
Profile Finishing	
Groove Finishing	
Threading	
Sequential Machining	
Axial Machining Operations	

Insert > Auxiliary Operations > Lathe Tool Change
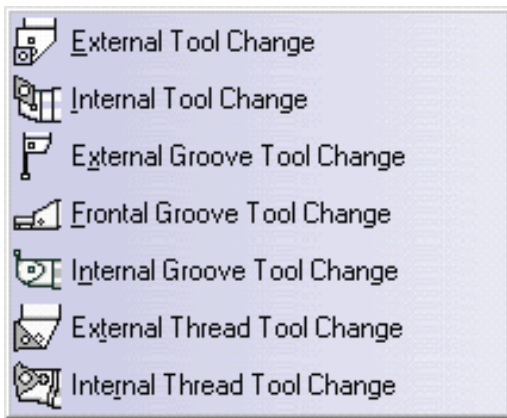
Description...

Allows inserting lathe tool changes in the program.

Lathe Machining Toolbars

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

The following toolbar is specific to the Lathe Machining workbench.

It contains commands to create and edit lathe machining operations as follows.

Create a Roughing operation.
Basic tasks illustrate the following roughing modes:
- Longitudinal
- Face
- Parallel Contours.

Create a Recessing operation.

Create a Grooving operation.

Create a Profile Finishing operation.

Create a Groove Finishing operation.

Create a Threading operation.

Create a Lathe Sequential 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 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 tree looks like this: .

Specification Tree

Here is an example of a Process Product Resources (PPR) specification tree for NC Manufacturing products.
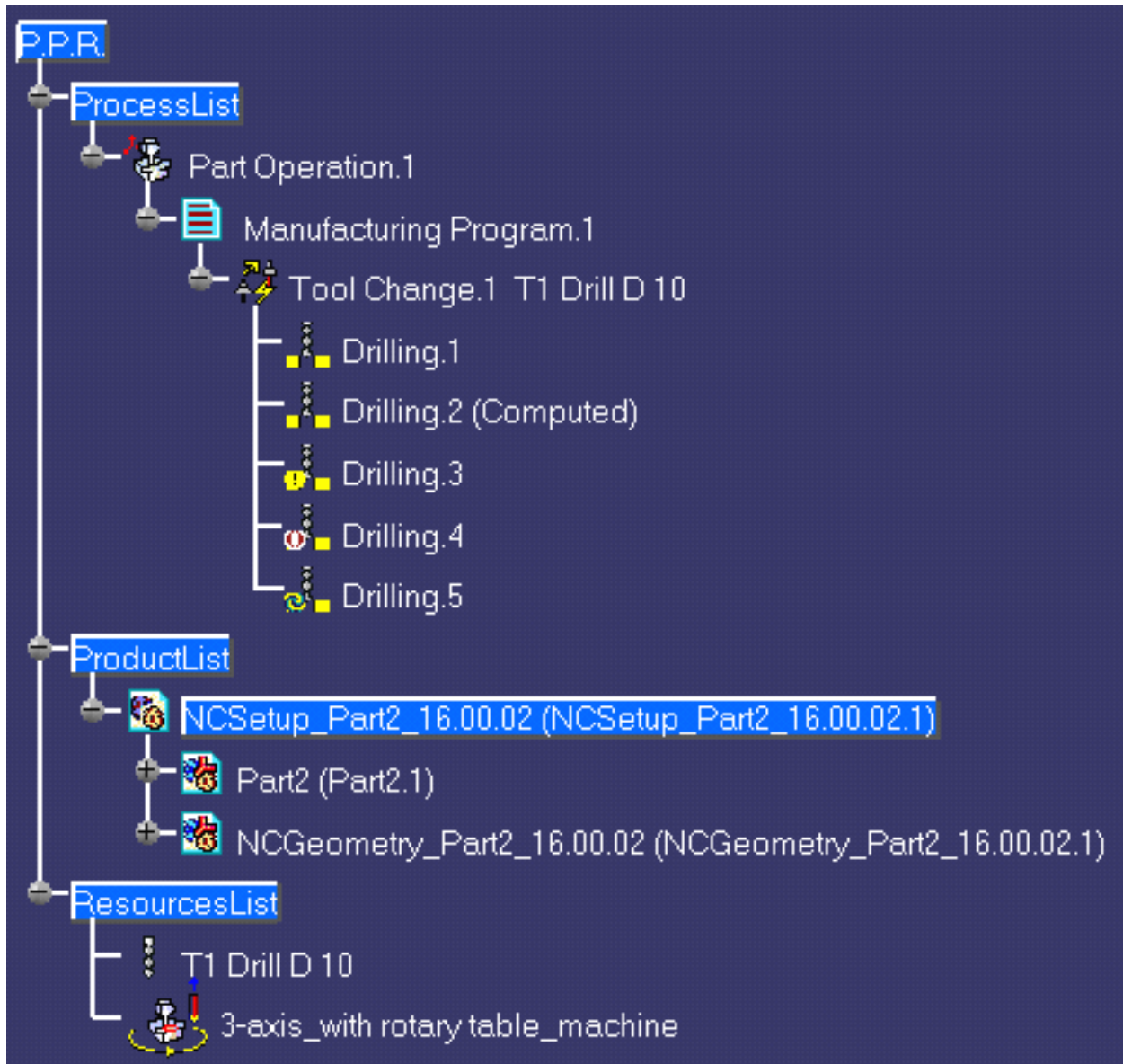
Process List is a plan that gives all the activities and machining operations required to transform a part from a rough to a finished state.

- **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

Tasks for customizing your NC Manufacturing environment are described in the *NC Manufacturing Infrastructure User's Guide*.

NC Manufacturing Settings
Build a Tools Catalog
Access External Tools Catalogs
PP Word Syntaxes
NC Documentation

Reference Information

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

Lathe Machining Operations
Cutter Compensation for Finish Operations
Changing the Output Point
Lathe Tool Assembly Considerations
Actual Stock Computation

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

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

Lathe Machining Operations

The information in this section will help you create and manage the machining operations supported by the Lathe Machining product. These operations are:

- Roughing
- Recessing
- Grooving
- Profile Finishing
- Groove Finishing
- Threading
- Sequential Machining:
    - Go motion
    - Go InDirv motion
    - Go Delta motion
    - Follow motion.

Roughing Operations

The Roughing operation allow 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.

External and Internal turning and grooving tools may be used.

Geometry for Roughing

Part and Stock profiles are required.

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

Orientation for Roughing

The following **Orientations** are proposed: Internal, External and Frontal (for Face and Parallel Contour 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 Roughing
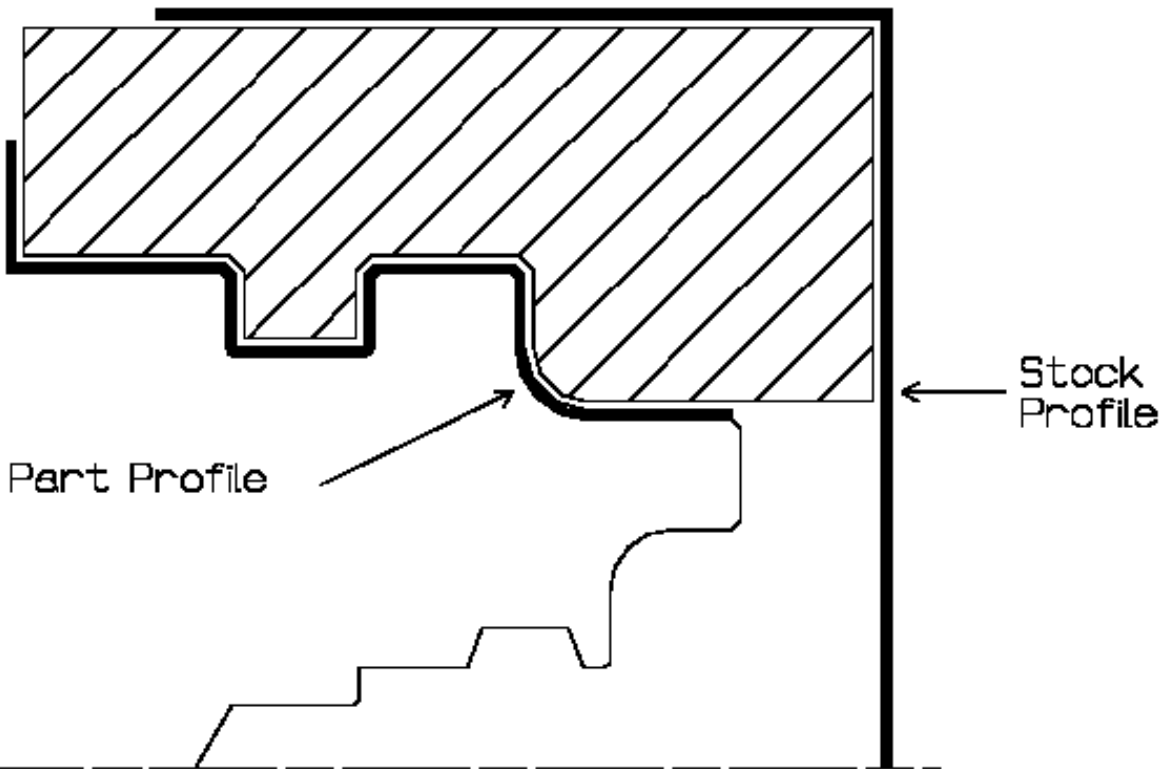
Internal Roughing
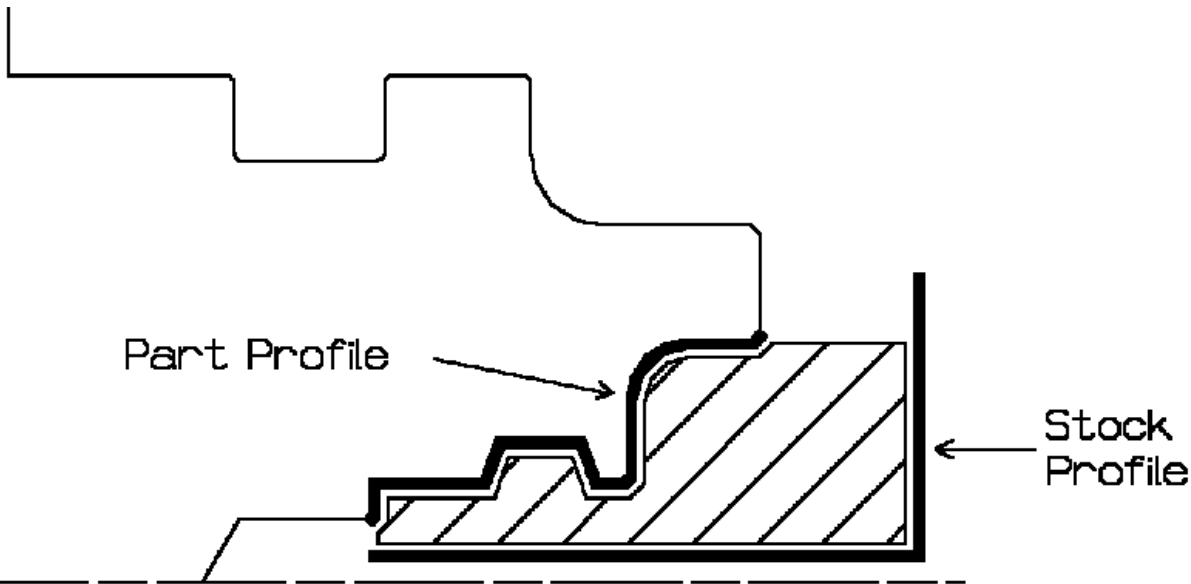
Frontal Roughing

Frontal machining is proposed for face roughing. 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 Profile

Stock Profile
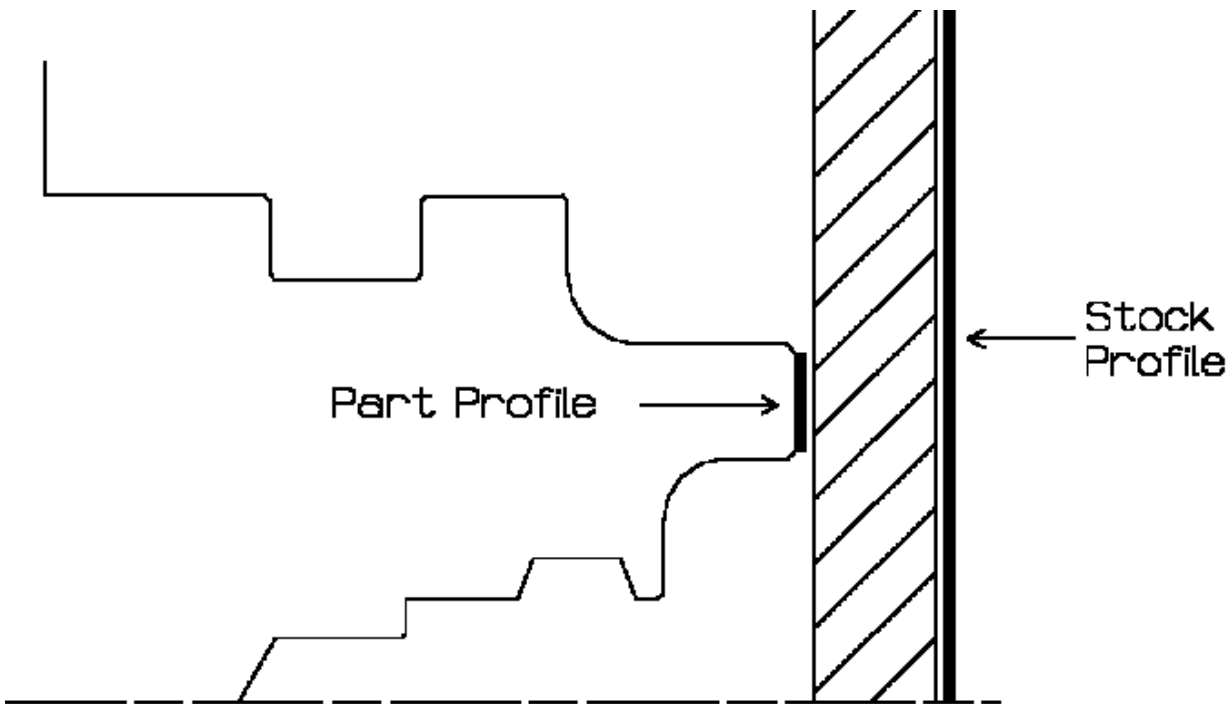
Location and Limits for Roughing

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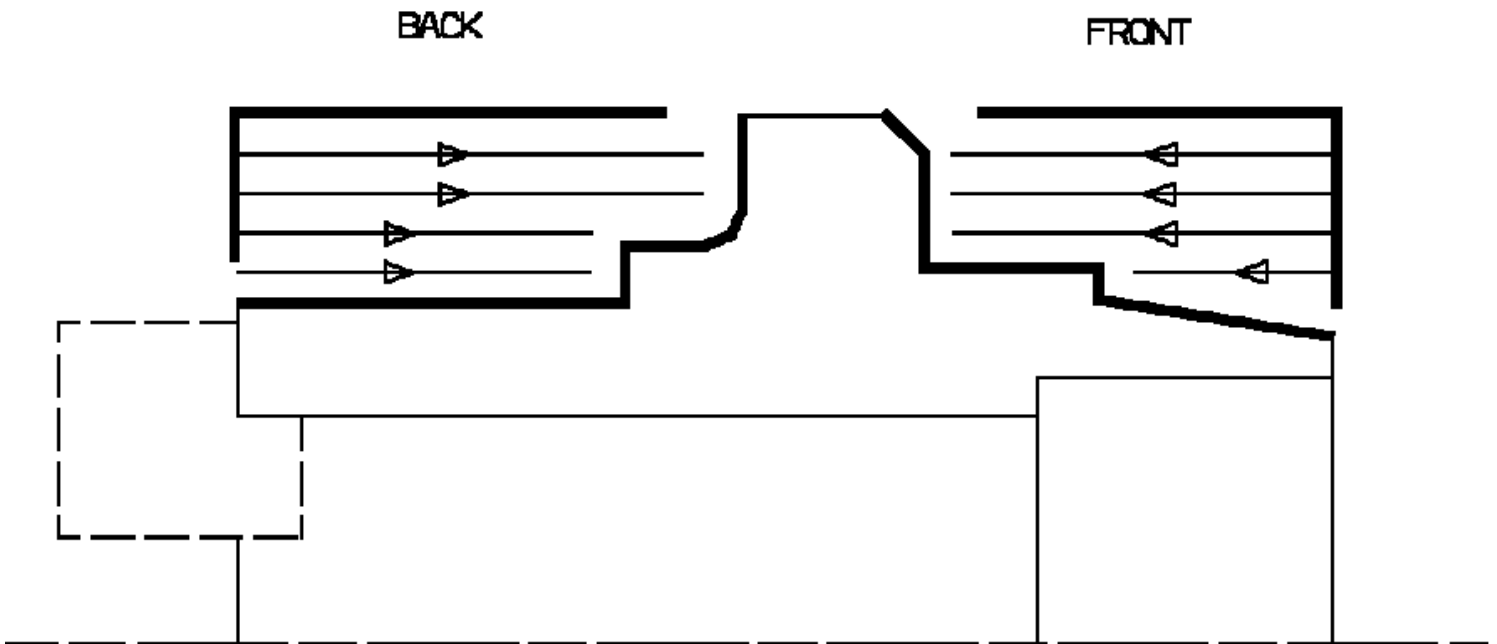
Orientation and Location settings determine the way the program closes the area to machine using radial, axial, axial-radial or radial-axial relimitation.

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

Minimum Machining Radius
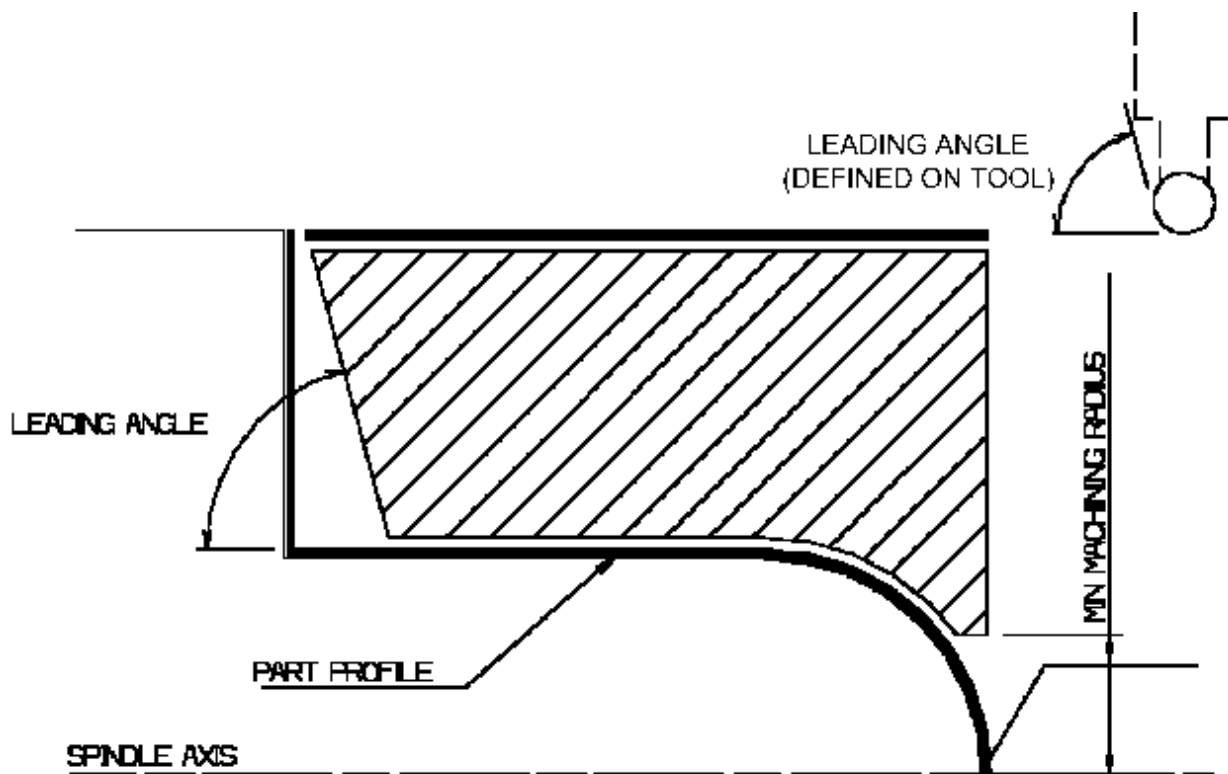
Maximum Machining Radius (for internal machining)
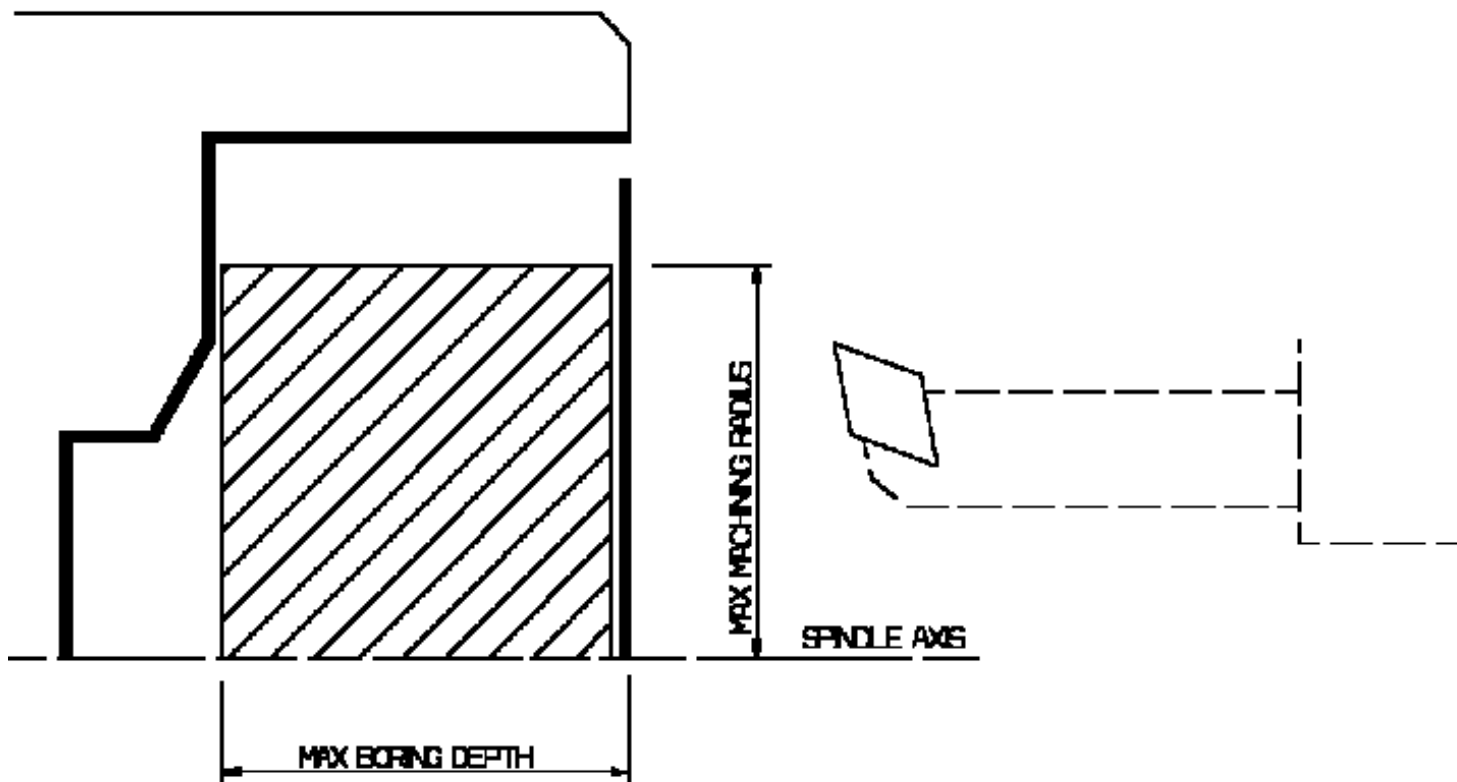
Note that Max Boring Depth is defined on the tool.

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

Part and Stock Thicknesses for Roughing

- **Clearance on stock**, which is defined perpendicular to the stock profile.
- **Thickness on part**, which is defined perpendicular to the part profile.
- **Axial offset on part**.
- **Radial offset on part**.
- **End limit clearance**: 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).

Radial and axial offsets can be positive or negative with any absolute value. The thickness applied to the part profile is the resulting value of the normal thickness and the two offsets.

Thickness on part

In this example, a "virtual nose radius" is obtained by adding the specified thickness value to the real nose radius.
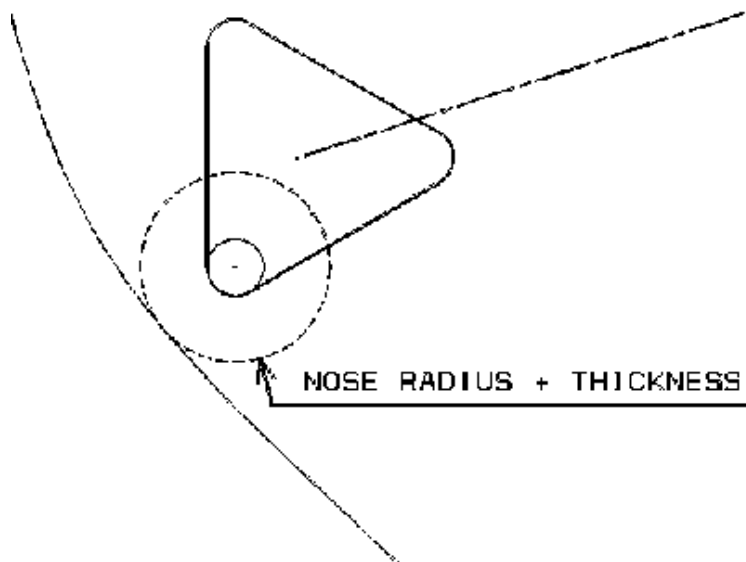
NOSE RADIUS + THICKNESS

Axial offset on part

A "virtual part profile" is obtained by translating the part profile the specified offset value along the spindle axis direction.
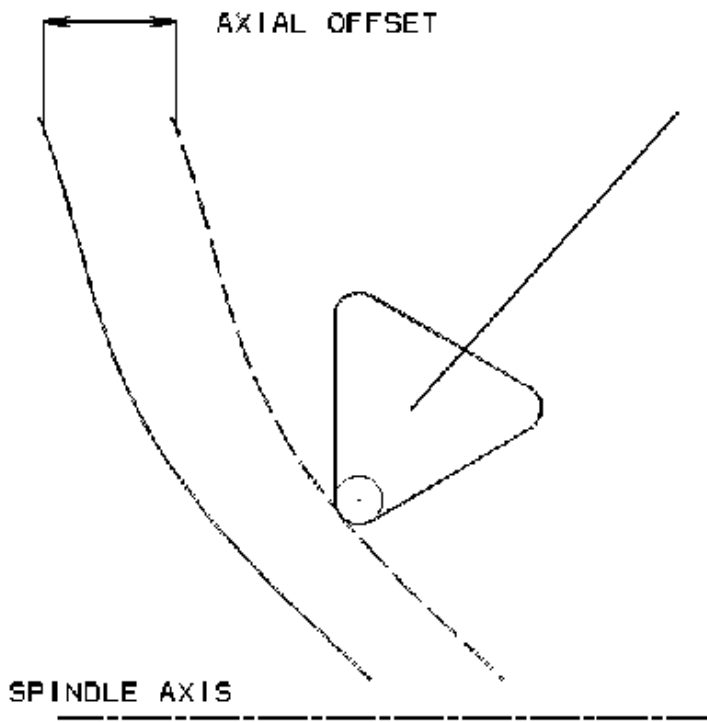
AXIAL OFFSET

SPINDLE AXIS

Radial offset on part

A "virtual part profile" is obtained by translating the part profile the specified offset value perpendicular to the spindle axis direction.

RADIAL OFFSET

SPINDLE AXIS
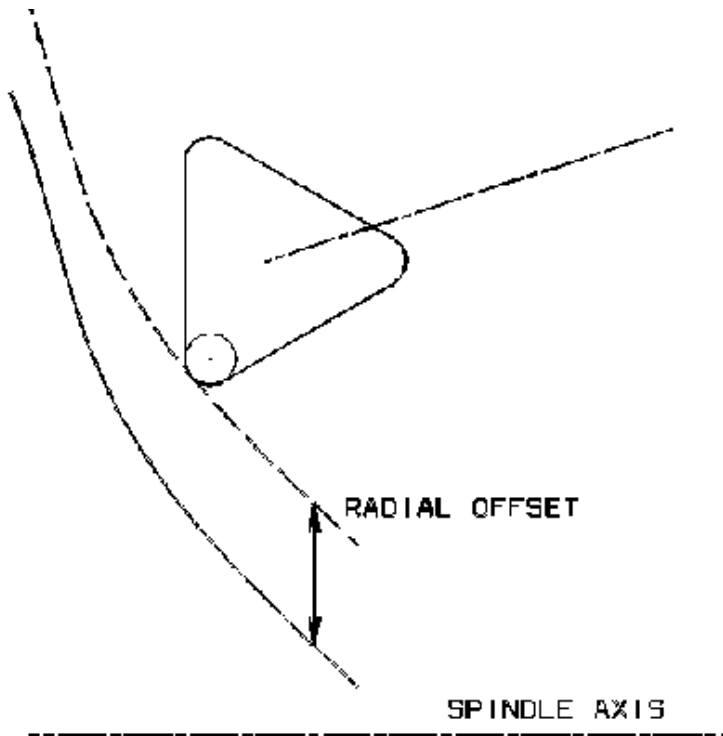
Machining Strategy Parameters for Roughing

Path Definition for Roughing

- **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 Roughing.
- **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 roughing 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 Roughing and Parallel Contour Roughing with Frontal machining, this option allows you to request machining under the spindle axis.
- **Machining Direction** (only for Parallel Contour and 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.
- **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 Roughing, the toolpath may not be collision free for some combinations of Axial and Radial Cut Depth.

  The following options are proposed for recess machining:
  - **Plunge Distance** and **Plunge Angle** (for longitudinal and face roughing)
    Define the plunge vector before each new pass with respect to the cutting direction.
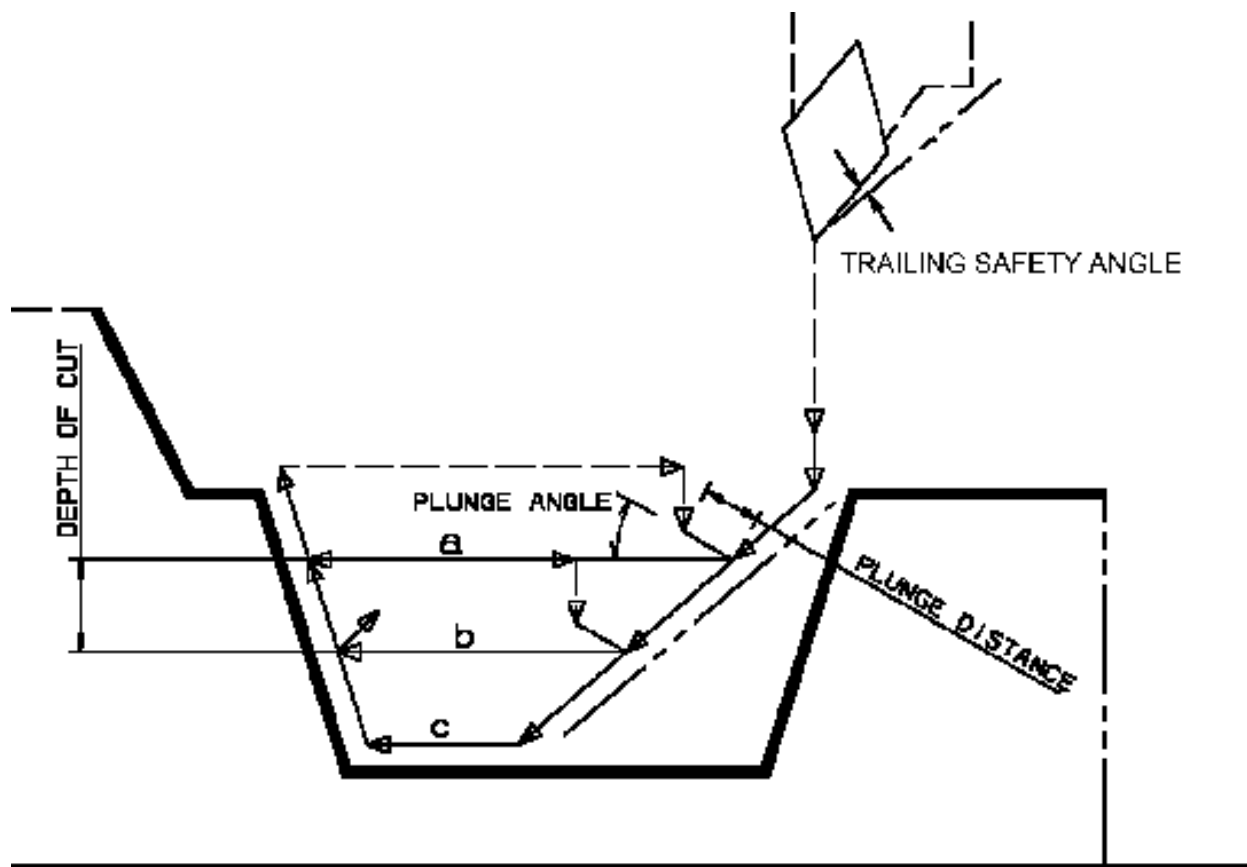
Example of recess with longitudinal external roughing. Note that Trailing angle is defined on the tool.

In the figure above the tool motion is as follows:

- approach in RAPID
- 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.

Lead-in, Lift-off and Attack for Roughing

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 Roughing 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.
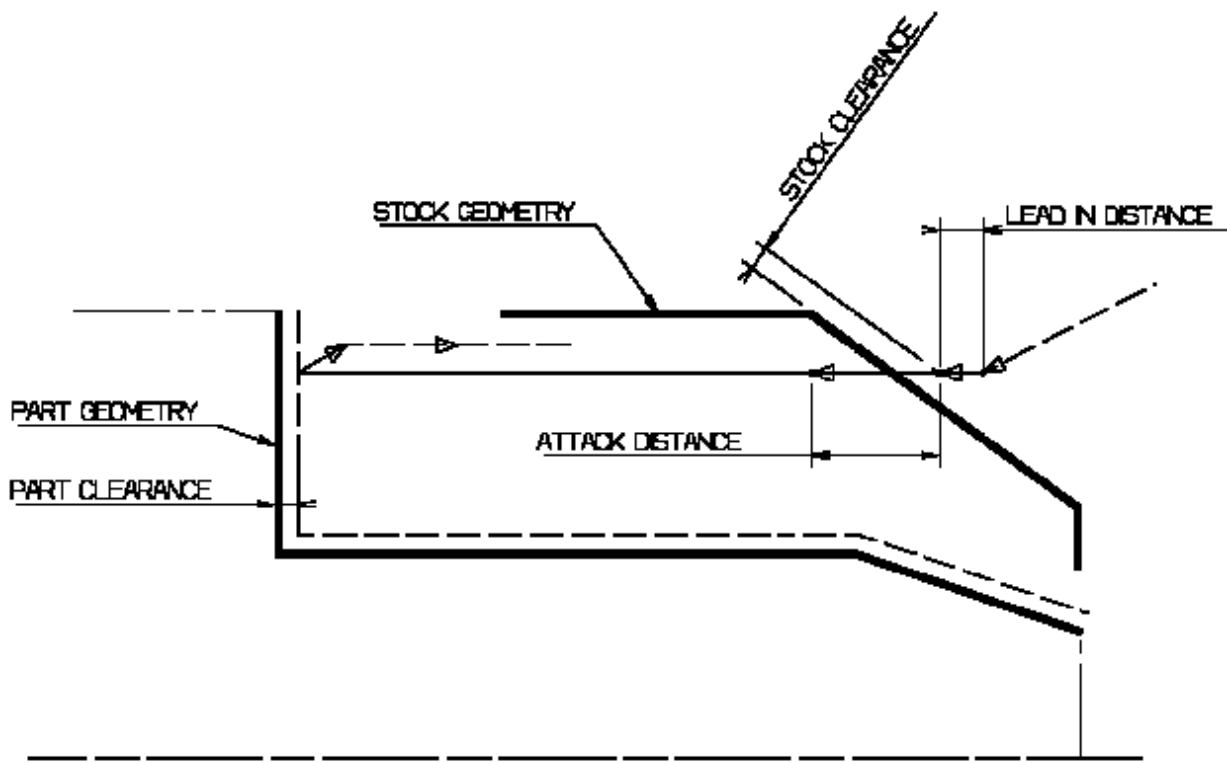
- **Lift-off Distance** and **Lift-off Angle**
  Define the lift-off vector at the end of each new pass with respect to the cutting direction.
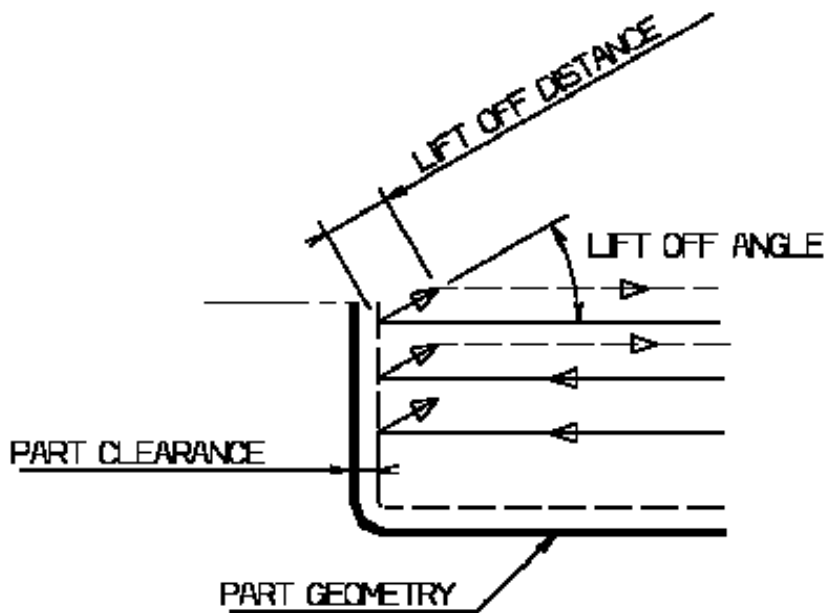
For Longitudinal or Face roughing, lift-off occurs:

- 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 Roughing, lift-off occurs when the end of the pass has already been machined by a previous pass.

Feeds and Speeds for Roughing

- 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.
- Machining Feedrate in units per revolution
- 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 roughing)
- 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.

Tool Compensation for Roughing

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 Roughing

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 motion during the approach or retract.
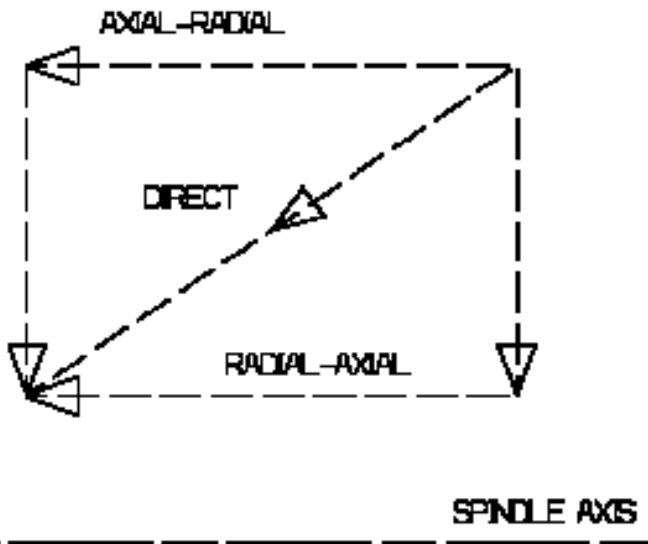
Recessing Operation

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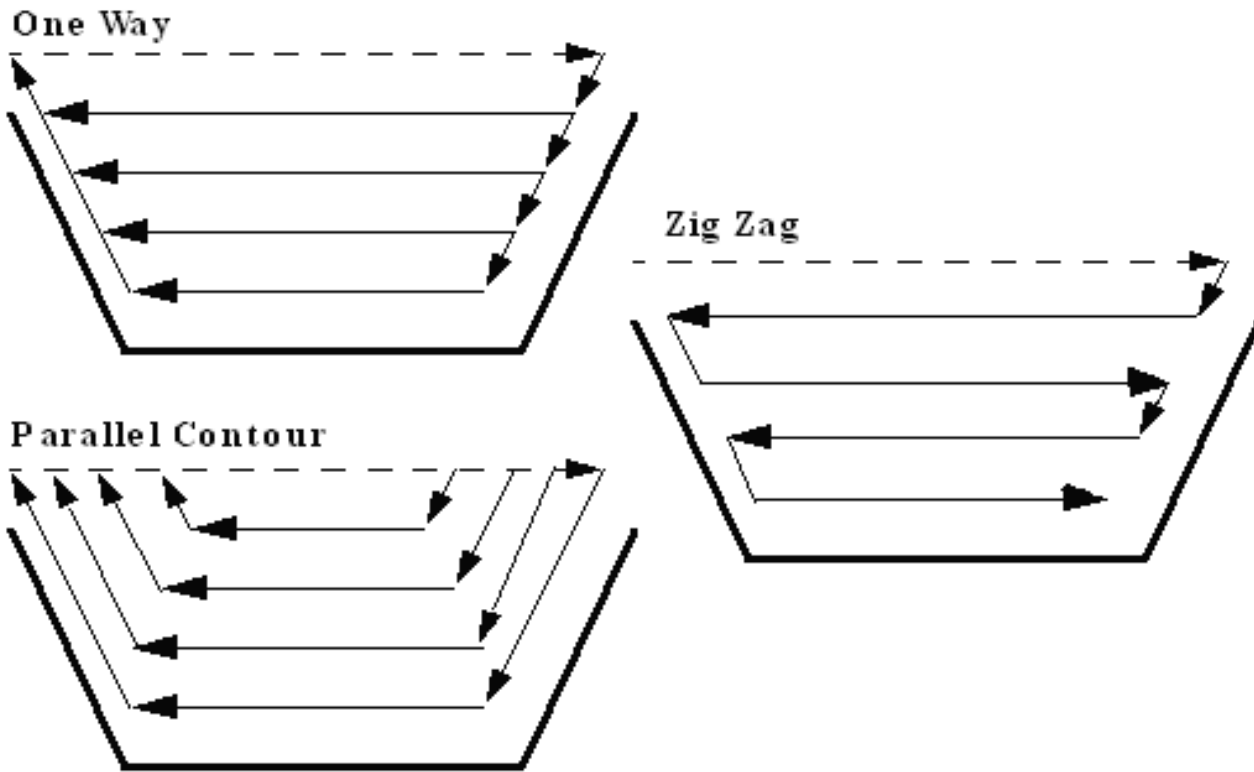
One Way

Zig Zag

Parallel Contour

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.

This operation supports all lathe tool types and inserts except threading tools.

Geometry for Recessing

Part and Stock profiles are required.

Orientation for Recessing

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 Thicknesses for Recessing

- **Clearance on stock**, which is defined perpendicular to the stock profile
- **Thickness on part**, which is defined perpendicular to the part profile.
- **Axial offset on part**
- **Radial offset on part**.

Radial and axial offsets can be positive or negative with any absolute value. The thickness applied to the part profile is the resulting value of the normal thickness and the two offsets.

Machining Strategy Parameters for Recessing

Path Definition for Recessing

- **Recessing 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 recessing. This is done by machining down the sides of the recess in order to clear the profile.

LIFT OFF DISTANCE

PART CLEARANCE

PART GEOMETRY
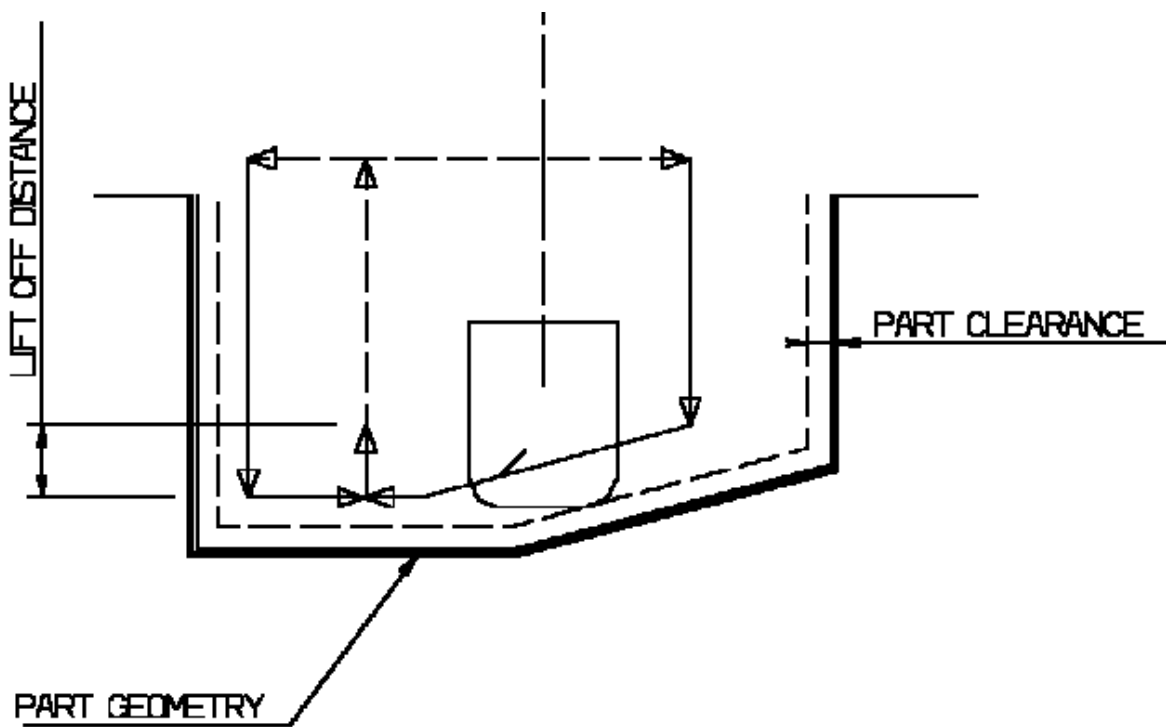
Lead-in, Lift-off and Attack for Recessing

- **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.
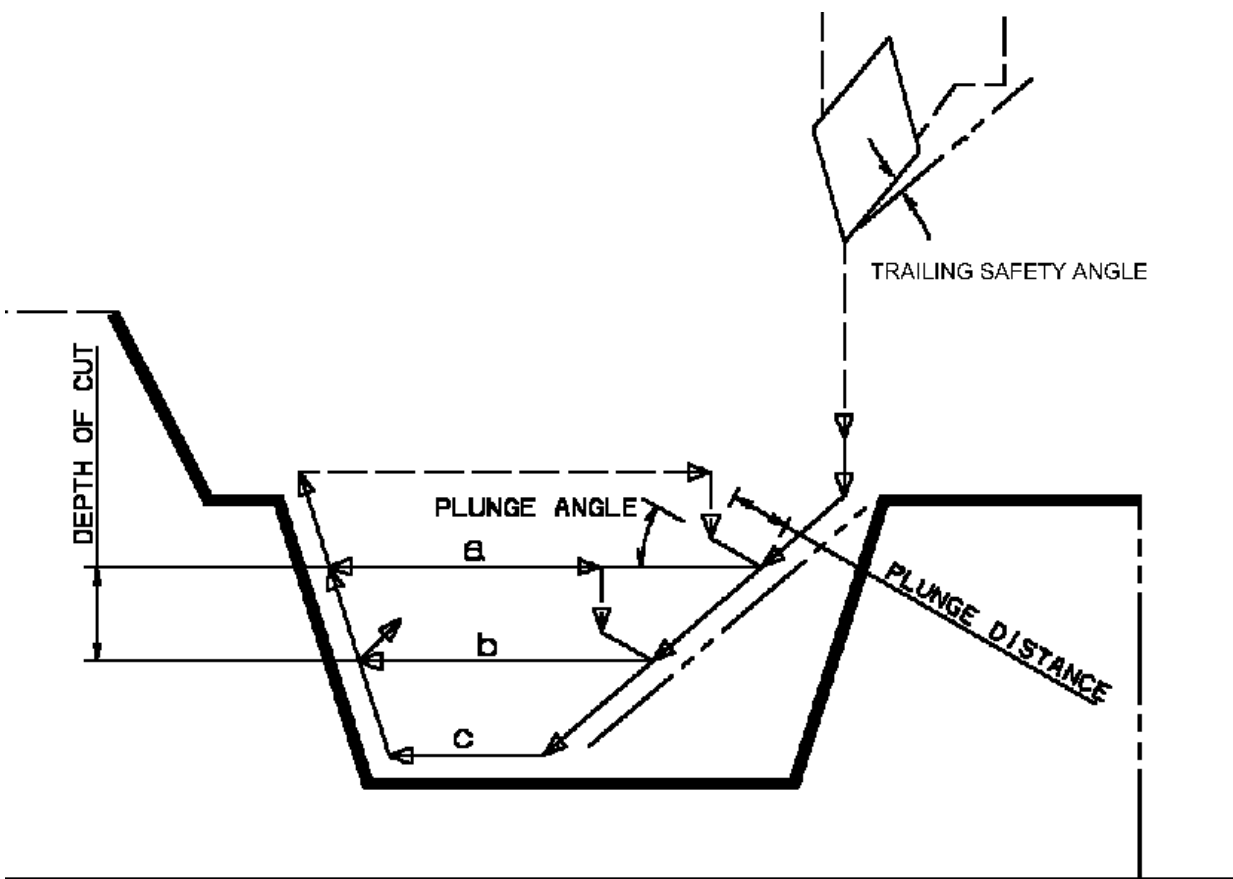
Example of one-way recessing. Note that Trailing angle is defined on the tool.

In the figure above the tool motion is as follows:

- approach in RAPID
- 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 Recessing

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

Tool Compensation for Recessing

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 Recessing

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 motion during the approach or retract.

Grooving Operation

The Grooving 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.

Grooving tools should be used.

Geometry for Grooving

Part and Stock profiles are required.

Orientation for Grooving

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 Thicknesses for Grooving

- **Clearance on stock**, which is defined perpendicular to the stock profile
- **Thickness on part**, which is defined perpendicular to the part profile.
- **Axial offset on part**
- **Radial offset on part**.

Radial and axial offsets can be positive or negative with any absolute value. The thickness applied to

the part profile is the resulting value of the normal thickness and the two offsets.

Machining Strategy Parameters for Grooving

Path Definition for Grooving

- **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 on the tool by reading the tool's minimum and maximum cutting diameters.
- **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 grooving 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 Grooving

- **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 feedrate.

- **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 Grooving

- 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.
- 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.
- 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.

Tool Compensation for Grooving

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 Grooving

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 motion during the approach or retract.

Profile Finishing Operation

The Profile Finishing 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.

External and Internal tools can be used.

Geometry for Profile Finishing

A Part profile is required.

- **Start Limit**: None / In / On / Out
  This option allows you to specify a point, line or curve as the start element of the part profile. 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 or curve as the end element of the part profile. 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.

Use of start and end elements for profile finishing. Profile is machined from start element. Profile is extrapolated up to end element. Direct approach and radial-axial retract.

Orientation and Location for Profile Finishing

- **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 Finishing

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.
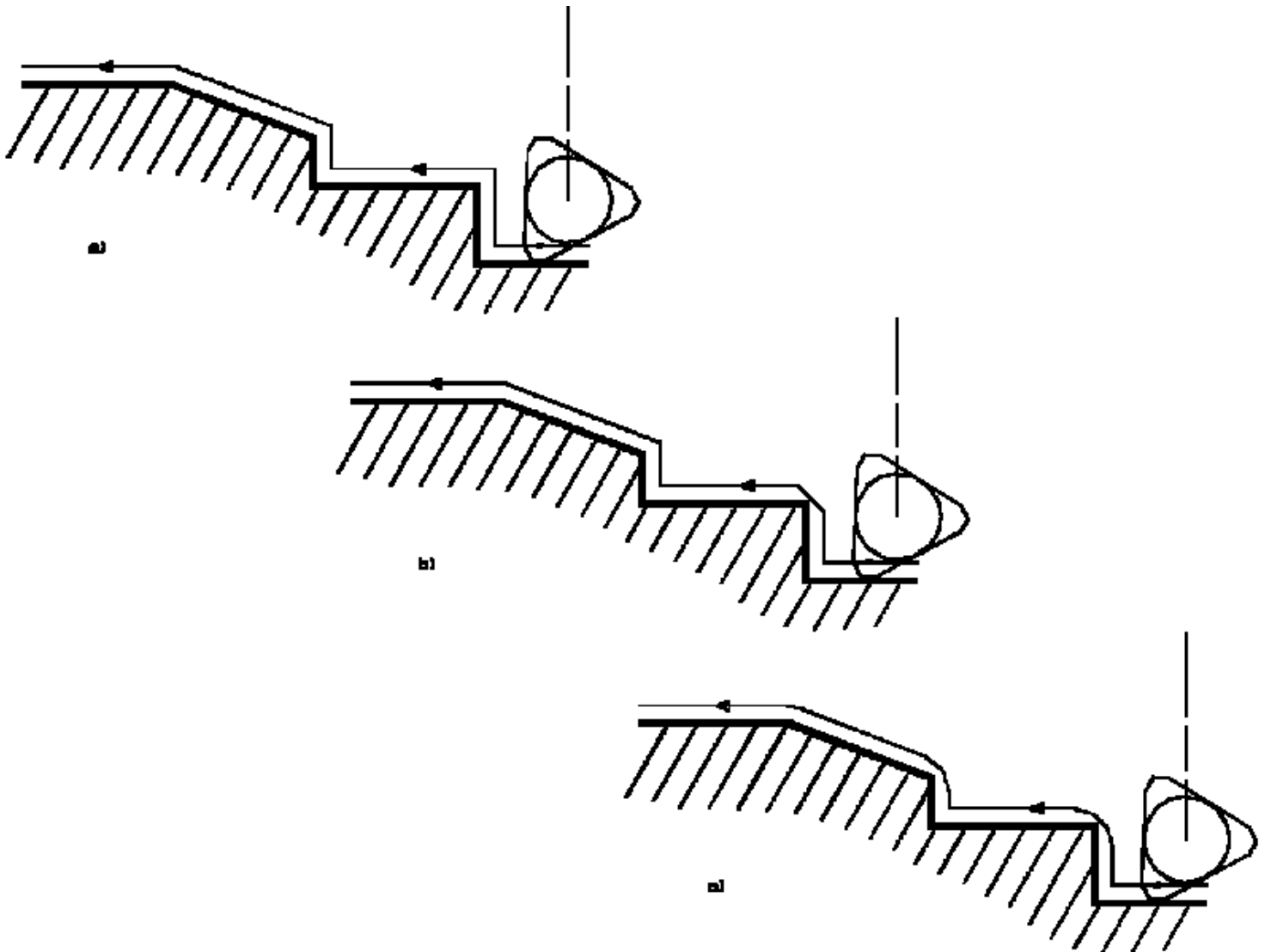
Corner processing options are also available to define how the entry and exit corners are to be machined. Entry and exit corners are defined by either a chamfer length, or a corner radius and corner angle.

Part Thicknesses for Profile Finishing

- **Thickness on part**, which is defined perpendicular to the part profile.

- **Axial offset on part**.
- **Radial offset on part**.
- **Start limit clearance**: 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 clearance**: 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).

Radial and axial offsets can be positive or negative with any absolute value. The thickness applied to the part profile is the resulting value of the normal thickness and the two offsets.

In addition to these global thickness and offsets, local thicknesses and offsets can be applied to segments, curves and arcs of the part profile.

Machining Strategy Parameters for Profile Finishing

Path Definition for Profile Finishing

- **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).
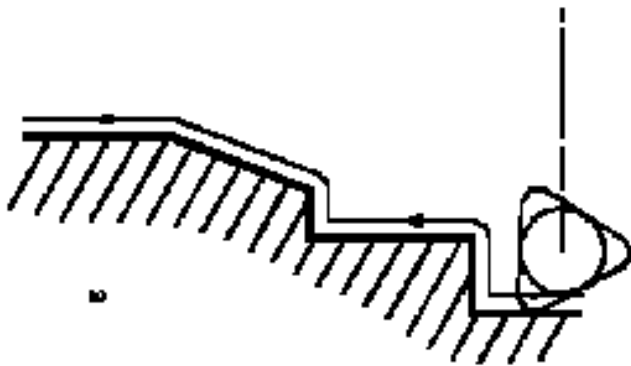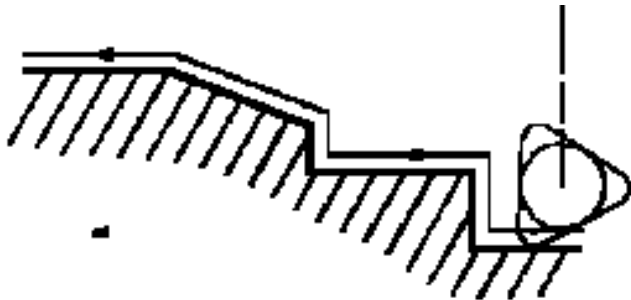
- **Under Spindle Axis Machining**
  When finishing in frontal mode, this option allows you to request machining under the spindle axis.

- **Recess Machining**
  When this option is set, the trailing safety angle option is 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 Finishing

- **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.
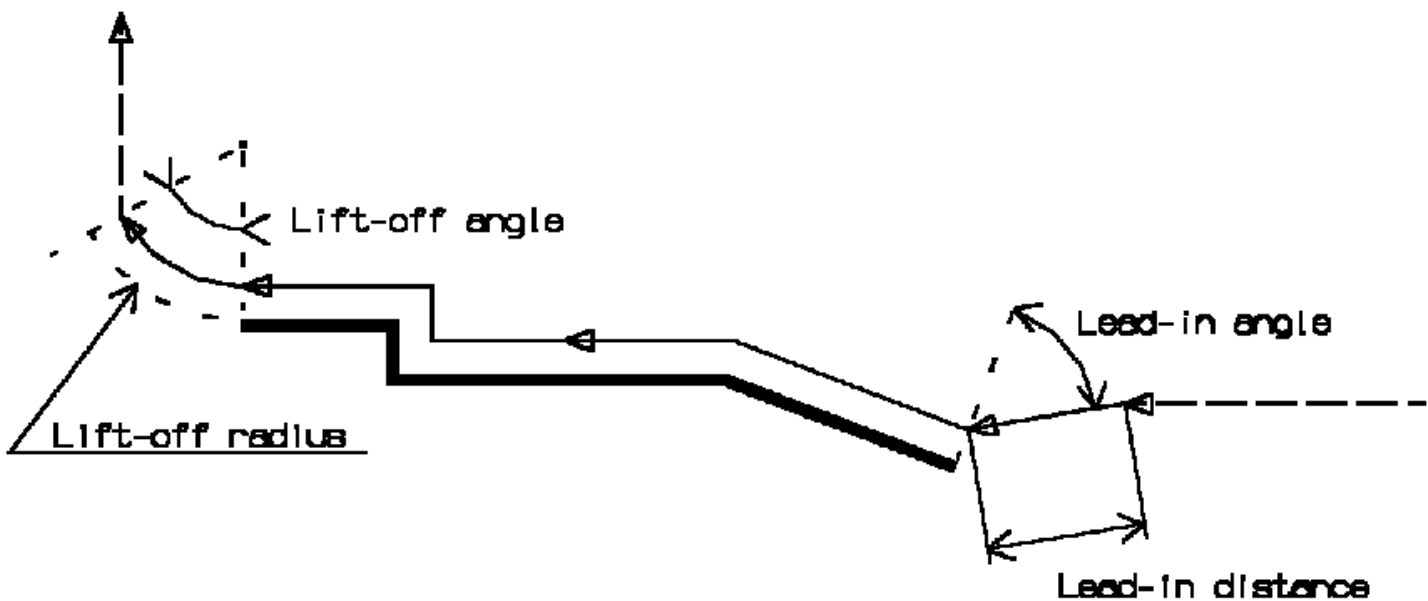
Linear lead-in and circular lift-off (profile finishing)

- **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.
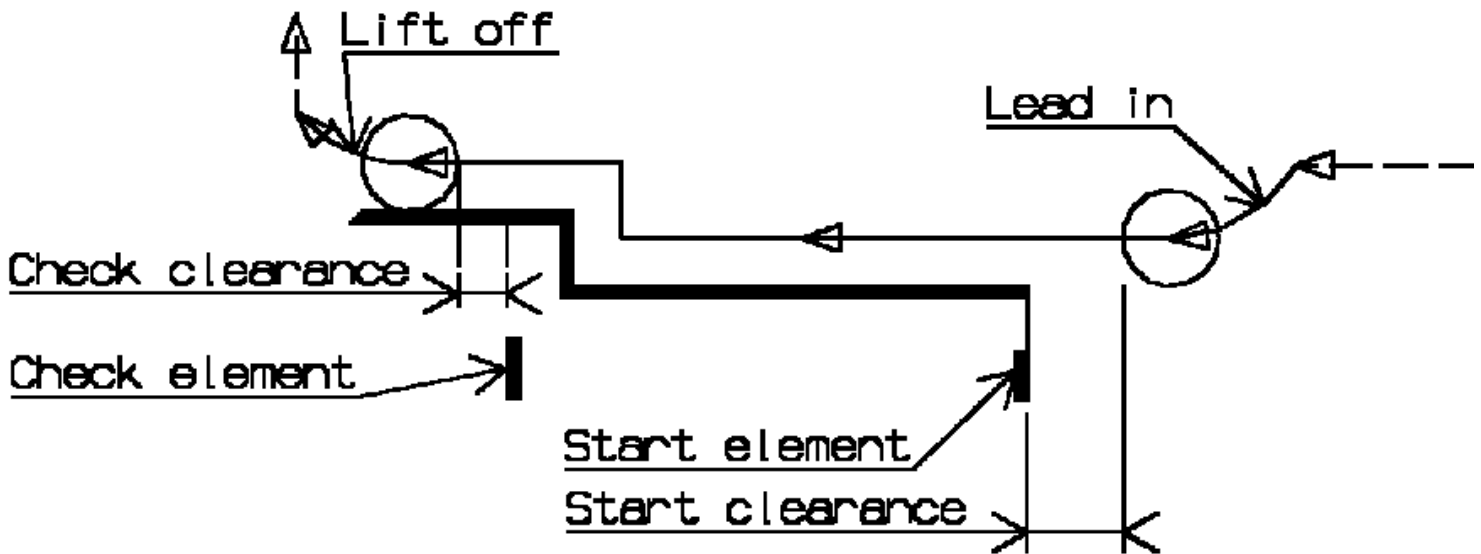
Feeds and Speeds for Profile Finishing

- 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.
- Machining Feedrate in units per revolution
- Chamfering Feedrate for machining chamfers or corners (in units per revolution)
- Lift-off Feedrate in units per revolution
- Lead-in Feedrate in units per revolution.

In addition to these global feedrates, local feedrates can be applied to segments, curves and arcs of the part profile.

Tool Compensation for Profile Finishing

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 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 motion during the approach or retract.

Groove Finishing Operation

The Groove Finishing 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.

Grooving tools or round inserts should be used.

Geometry

The Part profile is required.

- **Start Limit**: None / In / On / Out
  This option allows you to specify a point, line or curve as the start element of the groove finish profile. 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 groove finish profile may be extrapolated to the start element.

- **End Limit**: None / In / On / Out
  This option allows you to specify a point, line or curve as the end element of the groove finish profile. 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 start element.

    If needed, the groove finish profile may be extrapolated to the end element.

Orientation for Groove Finishing

- **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 Finishing

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 Thicknesses for Groove Finishing

- **Thickness on Part**, which is defined perpendicular to the part profile.
- **Axial Offset on Part**
- **Radial Offset on Part**.

- **Start Limit Clearance**: 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 Clearance**: 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).

Radial and axial offsets can be positive or negative with any absolute value. The thickness applied to the part profile is the resulting value of the normal thickness and the two offsets. In addition to these global thickness and offsets, local thicknesses and offsets can be applied to segments, curves and arcs of the part profile.

Machining Strategy Parameters for Groove Finishing

Path Definition for Groove Finishing

- **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 Finishing

- **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 finishing.
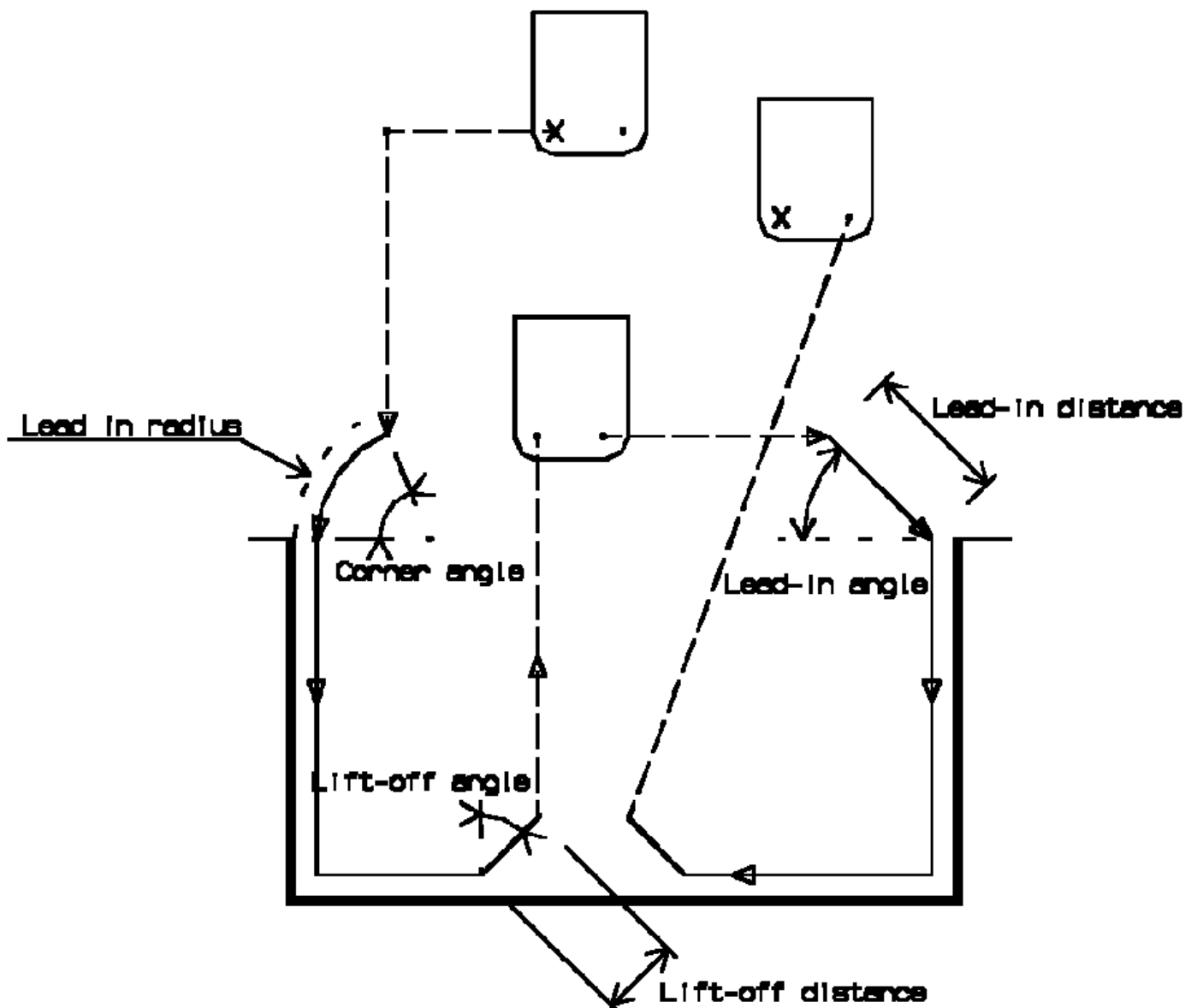
- **Last Flank Lead-in: Linear / Circular**

  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 Finishing

- **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 Finishing

- 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.
- **Machining Feedrate** in units per revolution
- **Chamfering Feedrate** for machining chamfers or corners (in units per revolution)
- **Lift-off Feedrate** in units per revolution
- **Lead-in Feedrate** in units per revolution.

In addition to these global feedrates, local feedrates can be applied to segments, curves and arcs of the part profile.

Tool Compensation for Groove Finishing

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 motion during the approach or retract.

Threading

The Threading 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.

External and Internal threading tools can be used.

Geometry for Threading

A Part profile is required.

- **End Limit**: None / In / On / Out
  This option allows you to specify a point, line or curve as the end element of the profile to be machined. 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 Clearance**: 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).
- **Start Limit**: None / In / On / Out
  This option allows you to specify a point, line or curve as the start element of the profile to be machined. 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 Clearance**: 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).

Orientation and Location for Threading

- **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 Threading

Machining Options for Threading

- **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**

- **Thread Penetration type**:
  - Straight
  - Oblique
  - Alternate.

  Penetration angle must be specified for Oblique or Alternate entry types.

Path Computation options for Threading

- **Clearance on crest diameter**
- **Lead-in Distance**
- **Lift-off Distance** and **Lift-off Angle**.

First and Last Passes options for Threading

- 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.

Tool Compensation for Threading

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 Threading

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 motion during the approach or retract.

Sequential Machining

You can create a sequence of basic Go, Go InDirv, Go Delta, and Follow tool motions in a single Sequential Machining 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.

Geometry Selection

You can select point / line / curve / edge elements as follows:

Point / line and curve elements must belong to the plane that is defined by the machine spindle axis and radial axis.

Edge elements must be coaxial with the machine spindle axis.

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 thickness or offset.
ON: the tool nose is positioned on the selected element, with a possible axial or radial offset (normal thickness, 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 thickness or 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.
- Normal thickness, and axial and radial offset 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), clearance, thickness and offset 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 thickness or offset.
ON: the tool nose is positioned on the selected element, with a possible axial or radial offset (normal thickness, 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 thickness or 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.
- Normal thickness, and axial and radial offset 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 thickness and offsets applied to follow the drive element are also defined on the same preceding motion. When the drive and check curves are tangent, any thickness and 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 thickness or offset.
ON: tool nose tangent to the drive element and on the check element, with a possible axial or radial offset (normal thickness is not applicable).
PAST: tool nose tangent to the drive element and after the check element with a possible thickness or 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.
- Normal thickness, and axial and radial offset 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.

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.
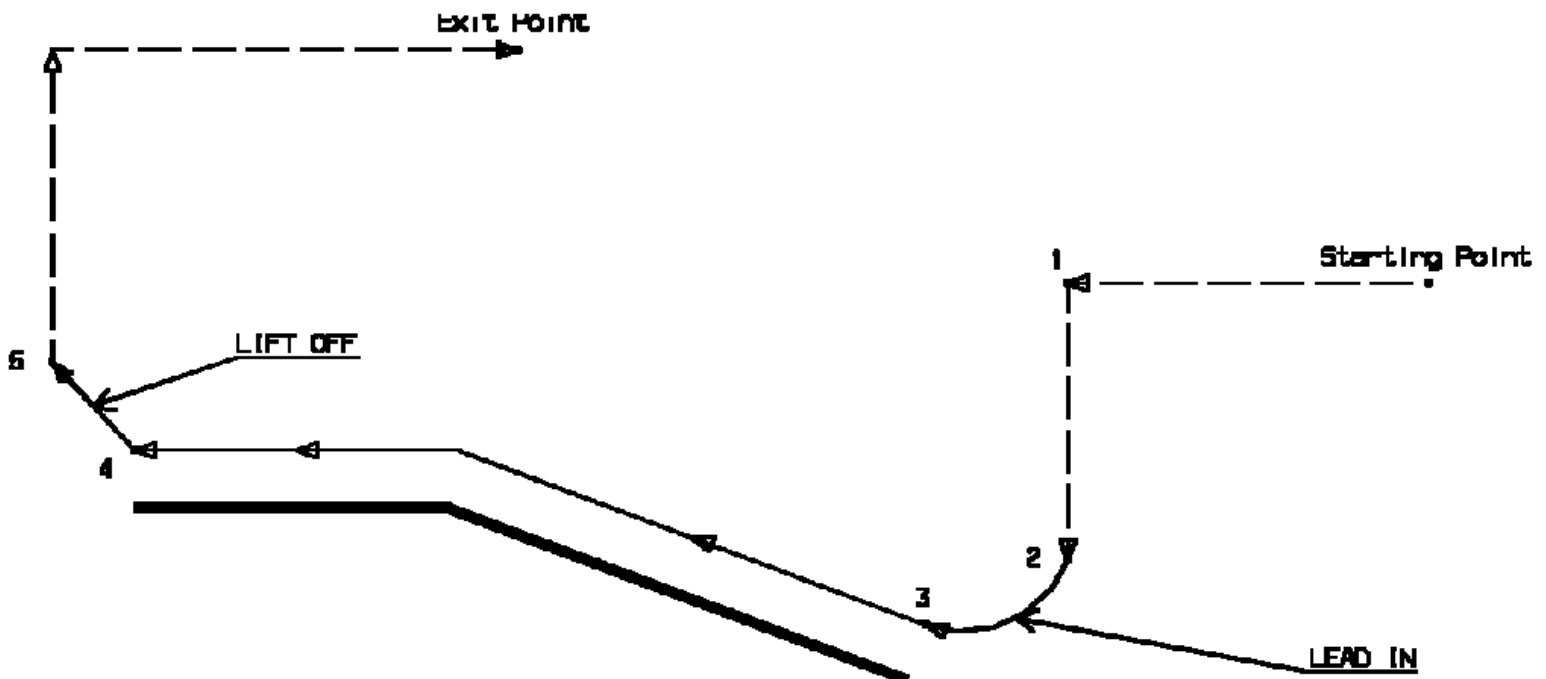
If the cutter compensation is set to ON, the CUTCOM instructions are generated as follows:

- 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

R

CUTTER RADIUS COMPENSATION
$(R-r)$

Real Tool Nose Radius

r

Real Tool Path
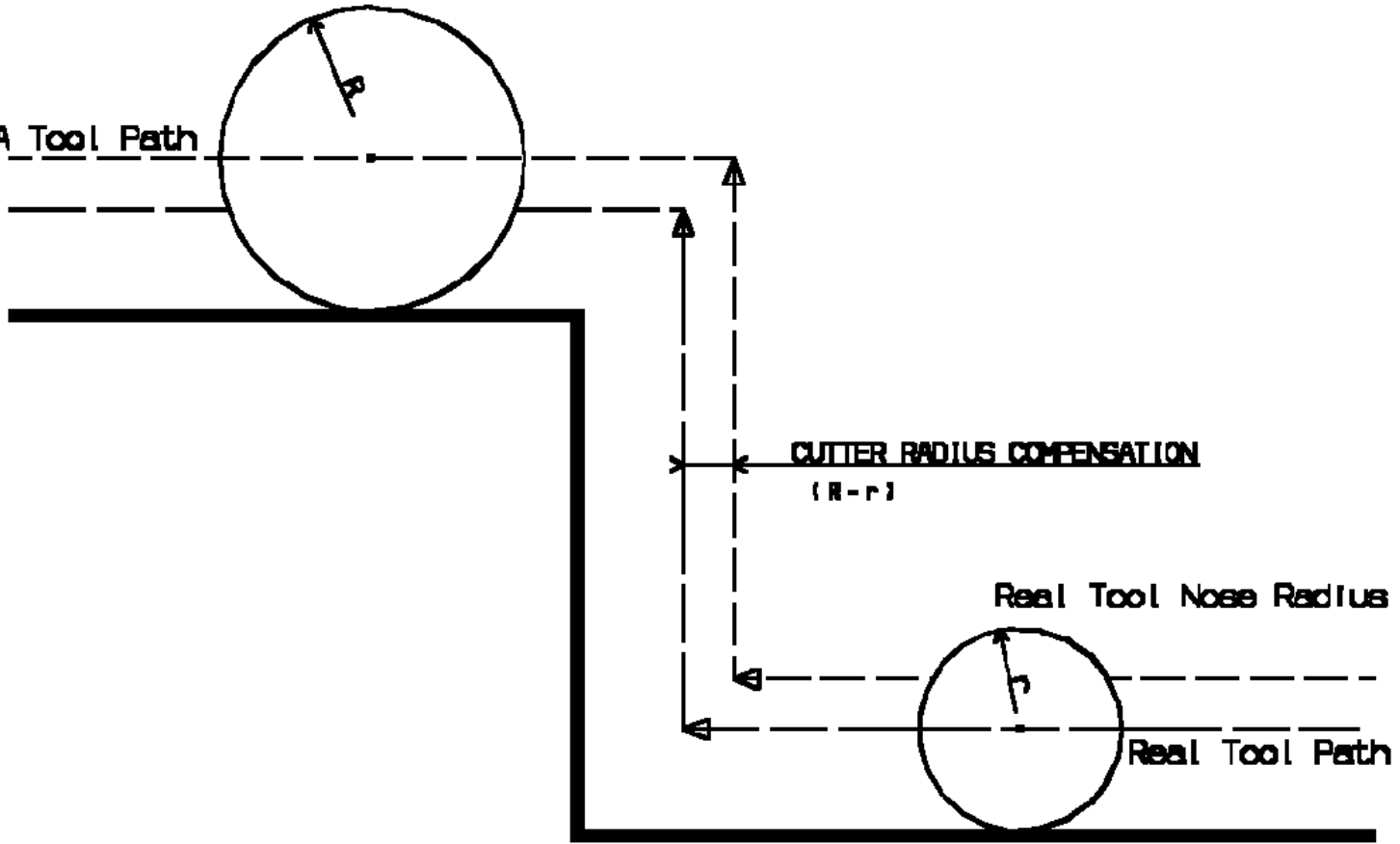
Changing the Output Point

An option for changing the tool output point is available for:

- Recessing, Grooving and Groove Finishing operations using grooving tools or inserts
- certain Threading operations using threading tools or inserts.

When the **Change Output Point** option is set the tool output point will be 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.

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.

Starting point

Output point change

Output point change

Exit point
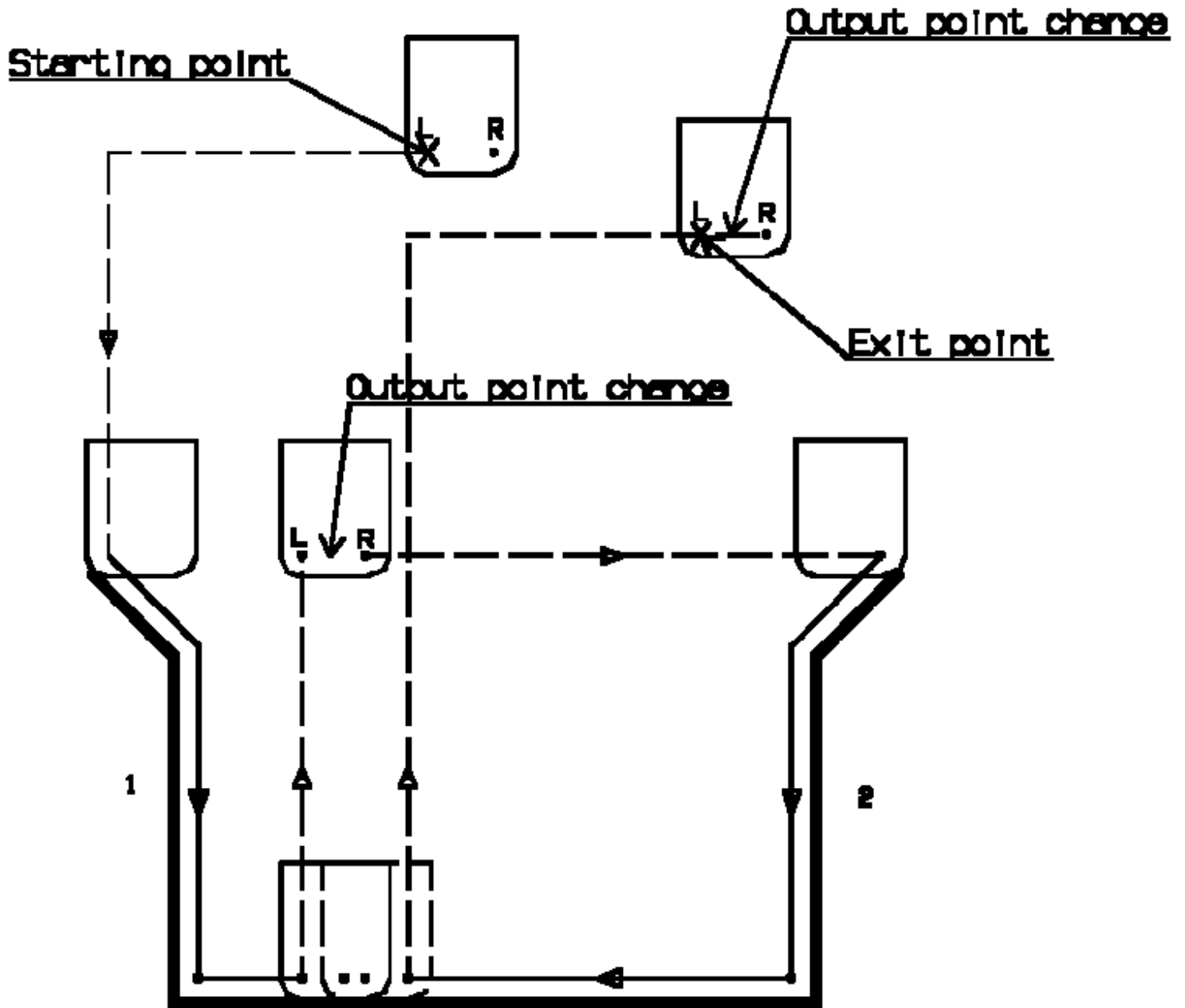
If the first flank to machine is flank 1, the tool motion is as follows:

- 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.

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.
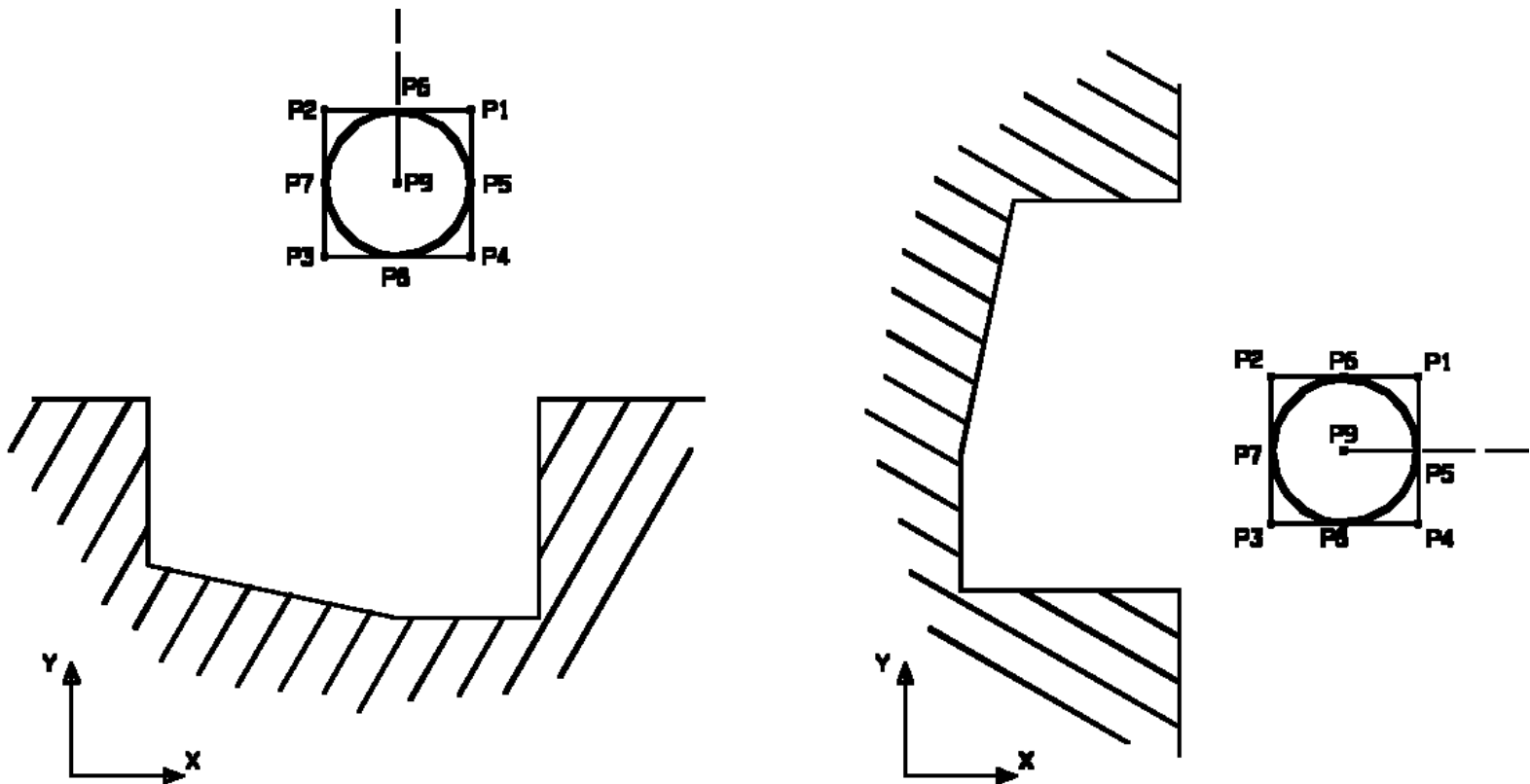
The tool output point is dependent on the machine axis system.

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.
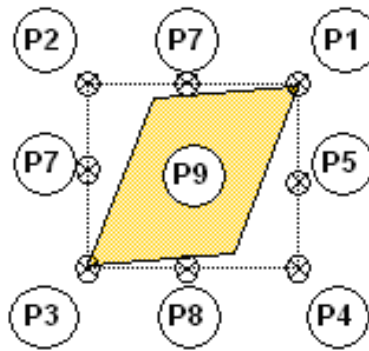
Examples below deal with Horizontal Lathe Machine Tool.

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 :
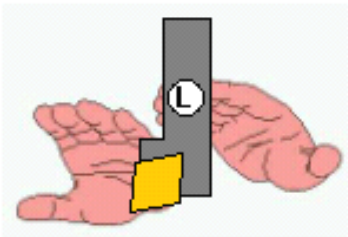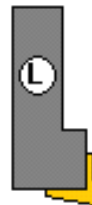
Right Tool: Insert is visible and points to right: Inverted Right Tool:
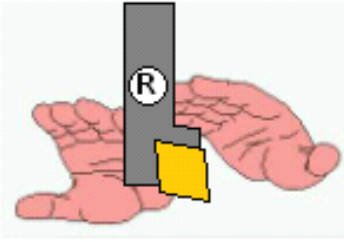
Output Point

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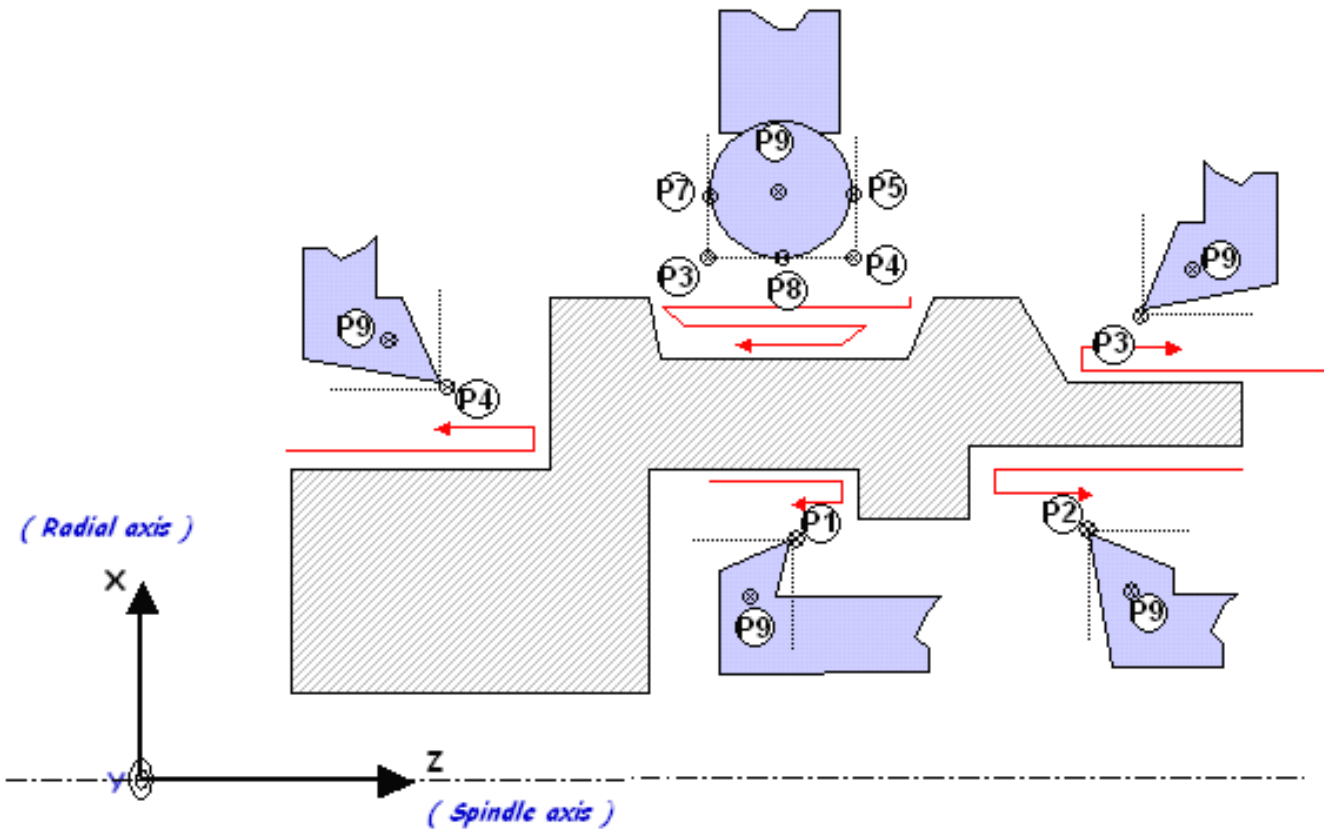
Spindle Rotary Direction

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.
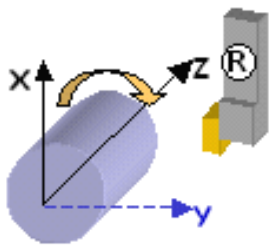
Clockwise   **Counterclockwise**
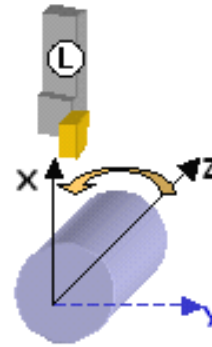
The following represent common settings:

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

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

Spindle :
Counterclockwise

Spindle :
Clockwise
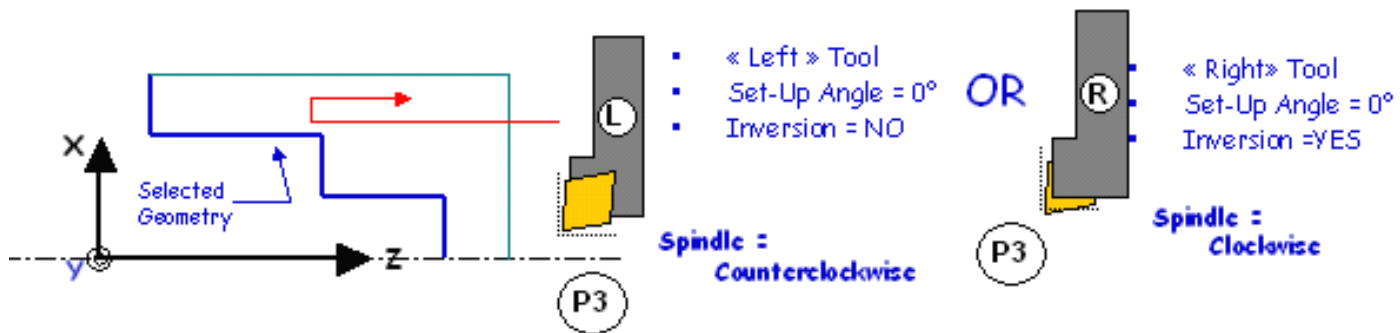
180° Setup Angle

When Setup angle is set to180°, machining takes places in X- as system detects that this is what the tool can machine (geometry can be selected in X+ or X-).

- « Left » Tool
- Set-Up Angle =180°
- Inversion = YES

- « Right» Tool
- Set-Up Angle =180°
- Inversion = NO

- « Left » Tool
- Set-Up Angle =180°
- Inversion = NO
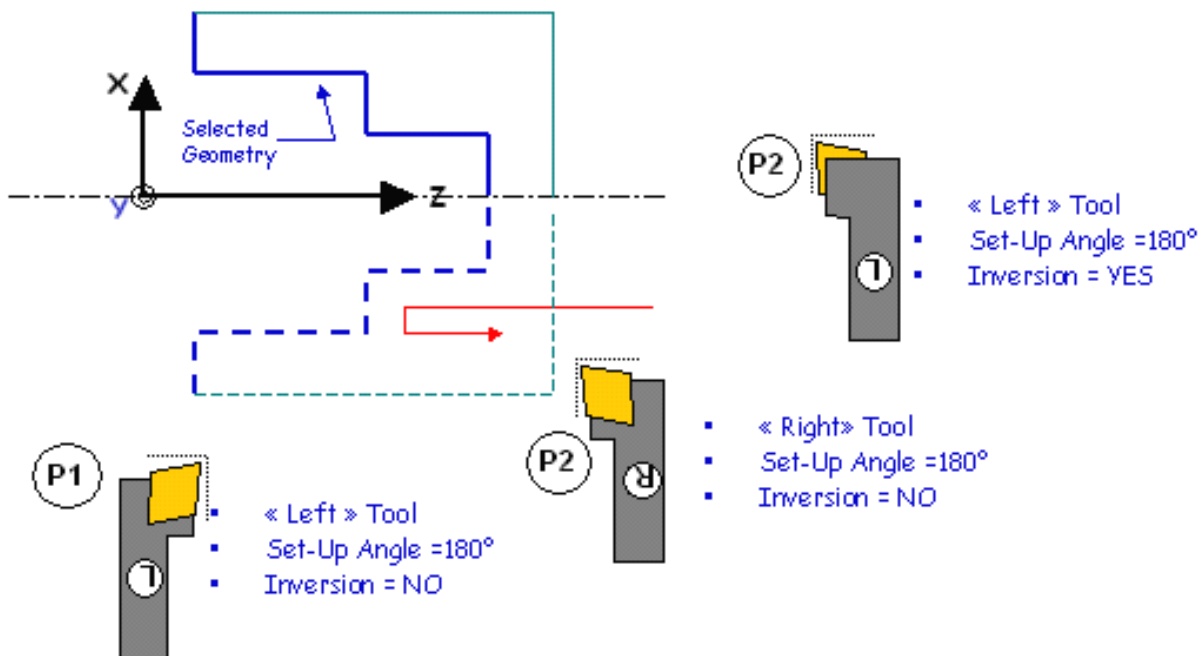
Direct Machining Axis

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).

Selected Geometry

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

Selected Geometry

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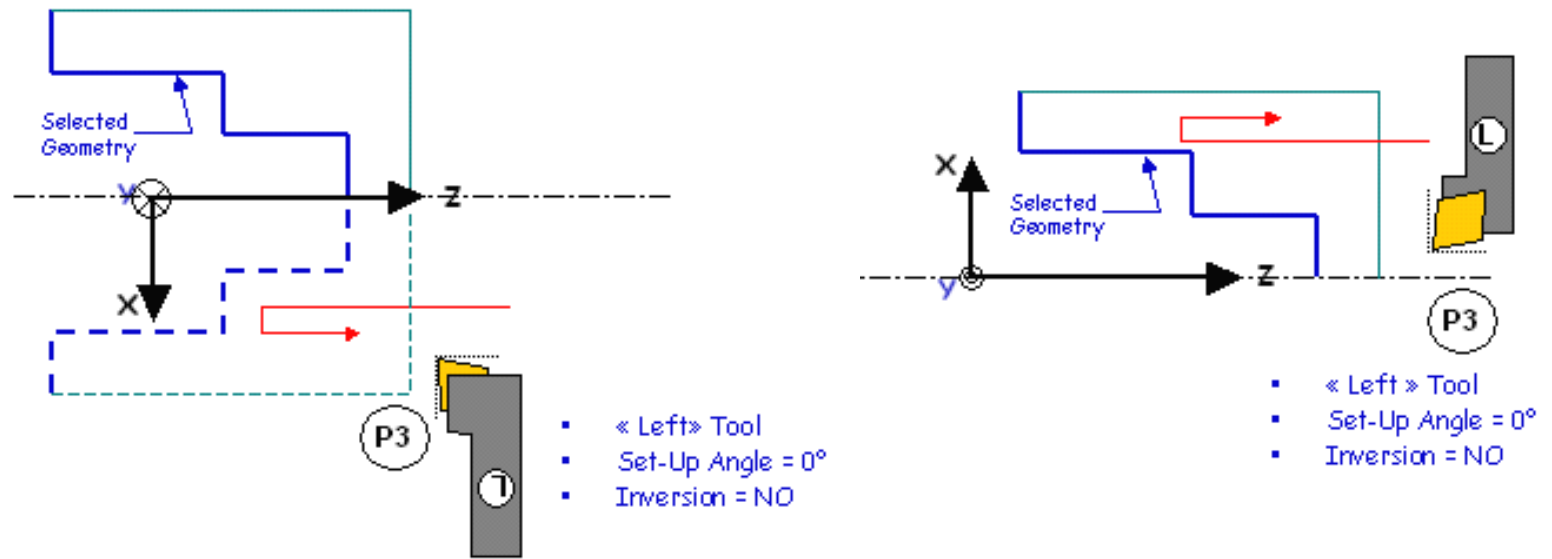
Summary

The following figures summarize conditions for lathe tool assemblies.
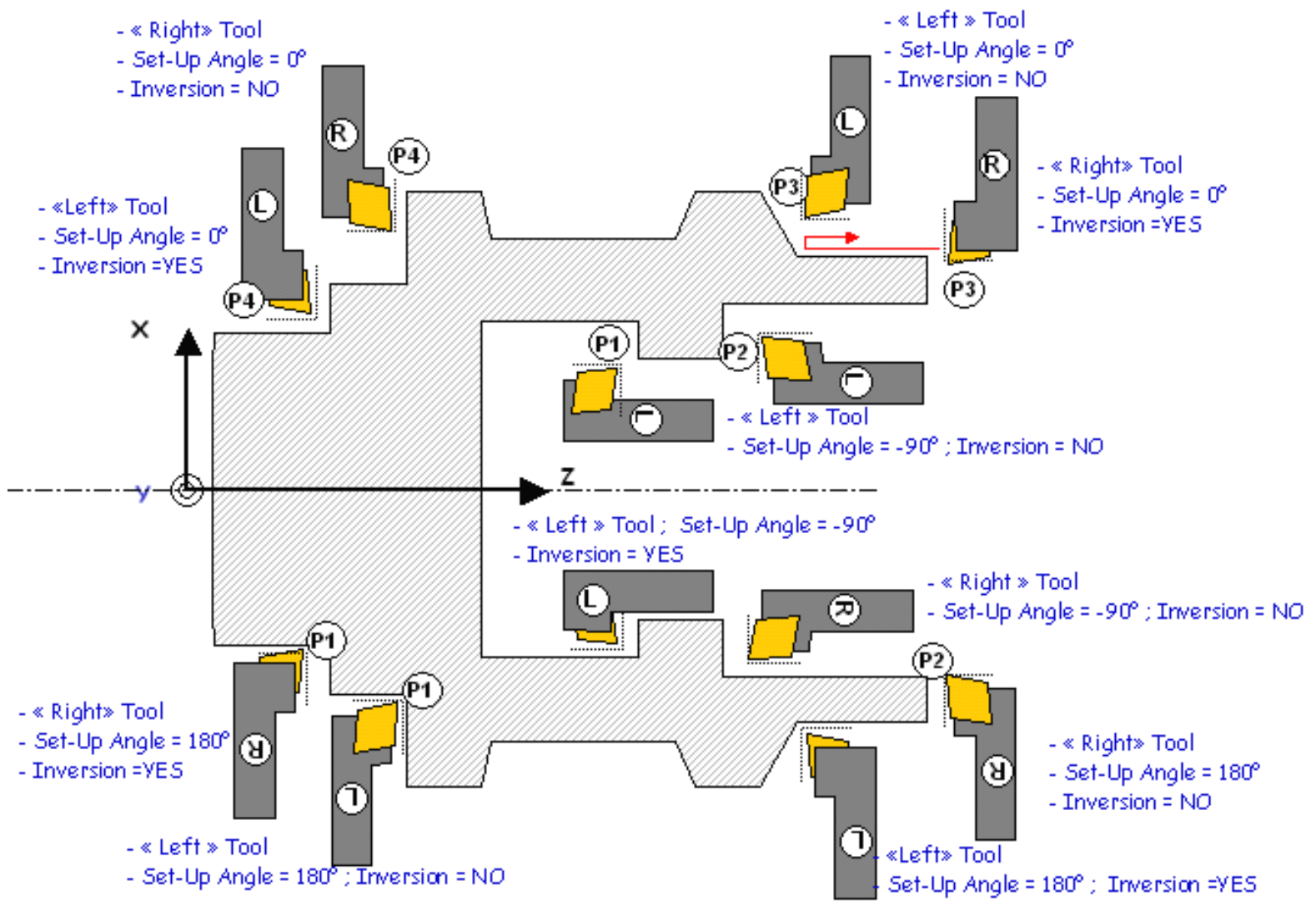
The figure below shows other settings for different output points.

- « 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
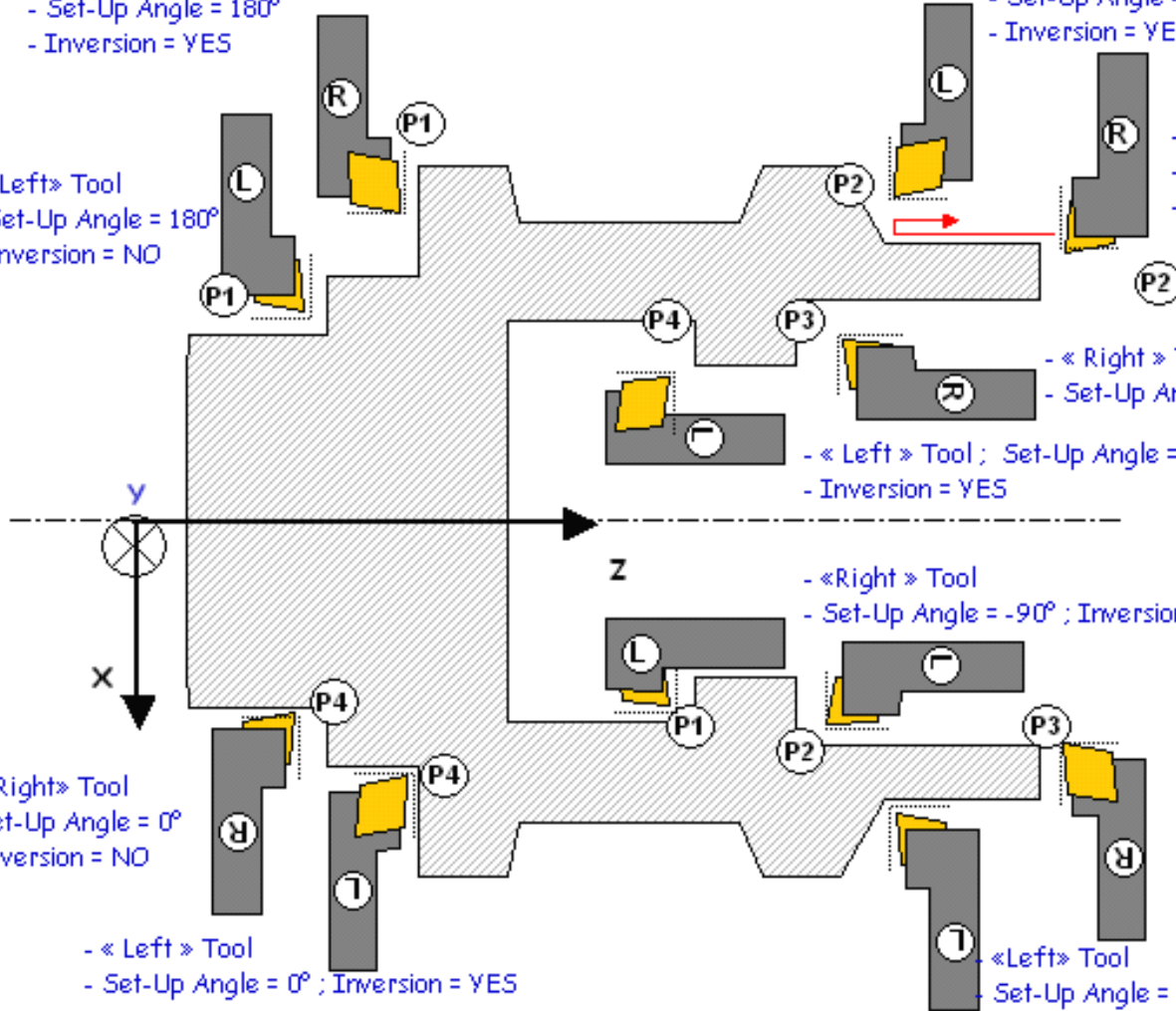
- « Right » Tool
- Set-Up Angle = -90° ; Inversion = NO

- « Left » Tool ; Set-Up Angle = -90°
- Inversion = YES

- «Right » Tool
- Set-Up Angle = -90° ; Inversion = NO

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

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

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

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

How to Use Actual Stock Computation

When you want to select the stock and part of a new machining operation, you can create the Actual Stock from the current operation.

This is done using the **Actual Stock Computation** command in the geometry tab page. The Actual Stock computation can be launched before the part or stock selection to create a sketch putting in emphasis the remaining material on the part. The computation takes into account all complete operations before the one which is edited.

The computation of the Actual Stock uses the "Stock for simulation", that you have to select in the Part Operation panel. The primitives of the material removed by each lathe operation will be subtracted from this stock.

The result of the computation is a sketch which contains the view of the remaining material in the working plane.

The first step before using Actual Stock Computation is to select a stock for simulation.

Double click on the Part Operation icon and select the Stock icon in the Part Operation dialog box.
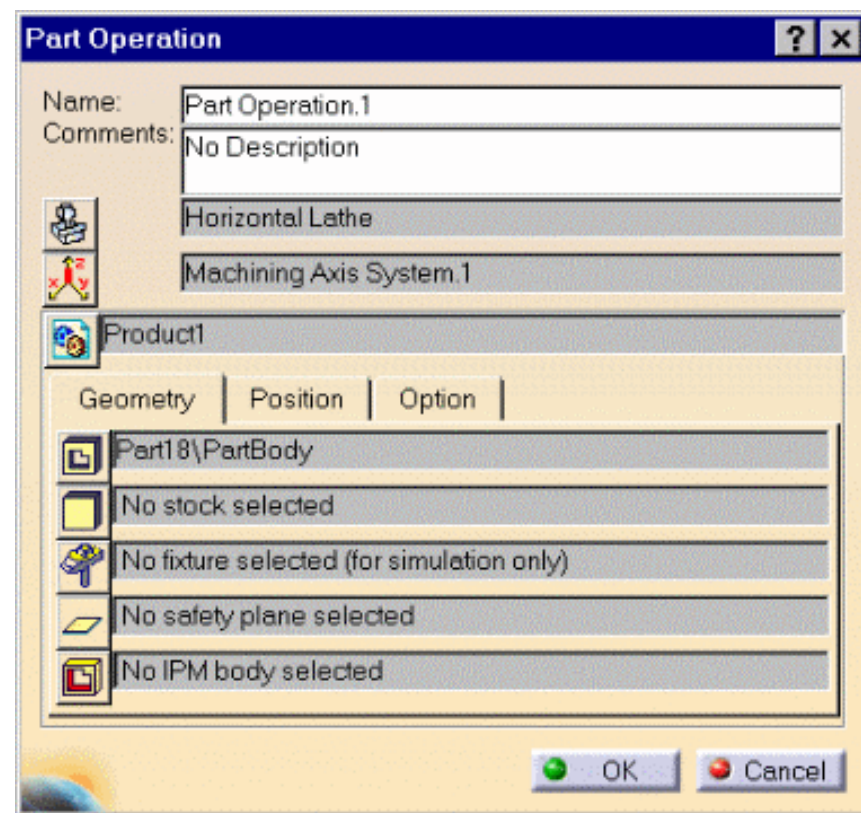
Select the stock, then double click on the background to leave selection mode.
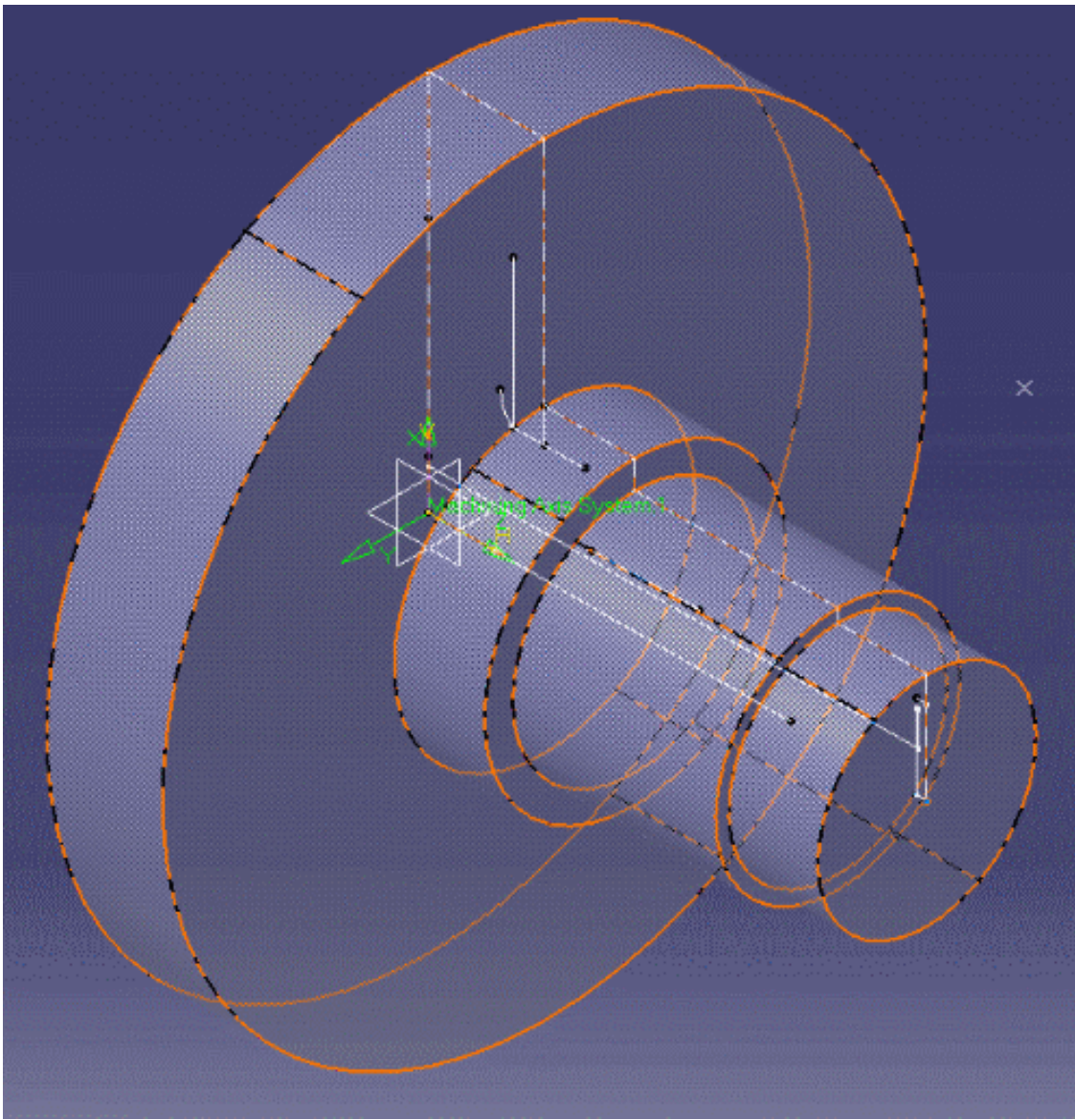
The Part Operation panel shows the name of the selected stock. In order to use Actual Stock Computation, you must select a lathe machine. The spindle and radial machine must be initialized correctly.
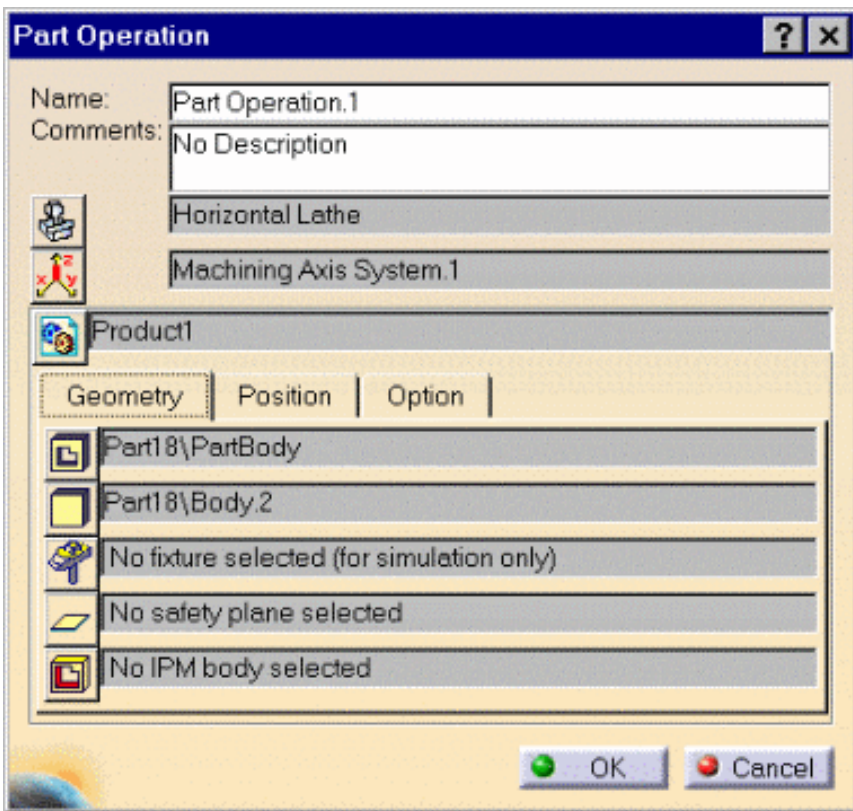
The second step is to initialize the In-Process Model body (IPM body) in the Part Operation panel.

The actual stock contour will be computed inside this body. Select a Part linked to your Product (or use "Insert a New Part" in the Assembly workbench on your existing product).

Double click on the part to enter the Part Design workbench and use the Insert > Body command to create a new body in your part. In this sample, call it "IPM_BODY".
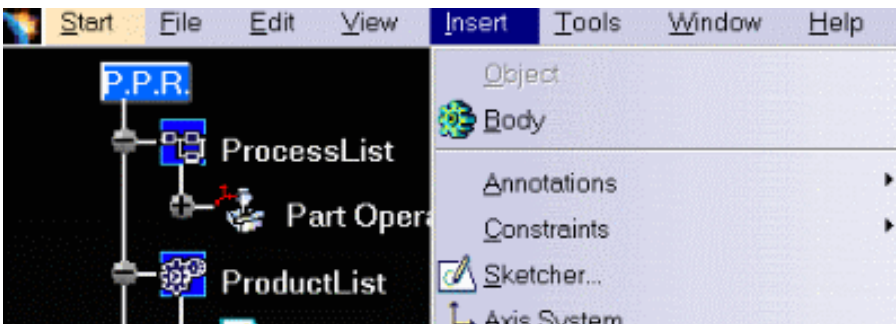
The created "IPM_BODY" body will be put in "No Show" mode after the Actual Stock computation. So do not use an existing body otherwise your existing geometry will be in the No Show layer after the computation.
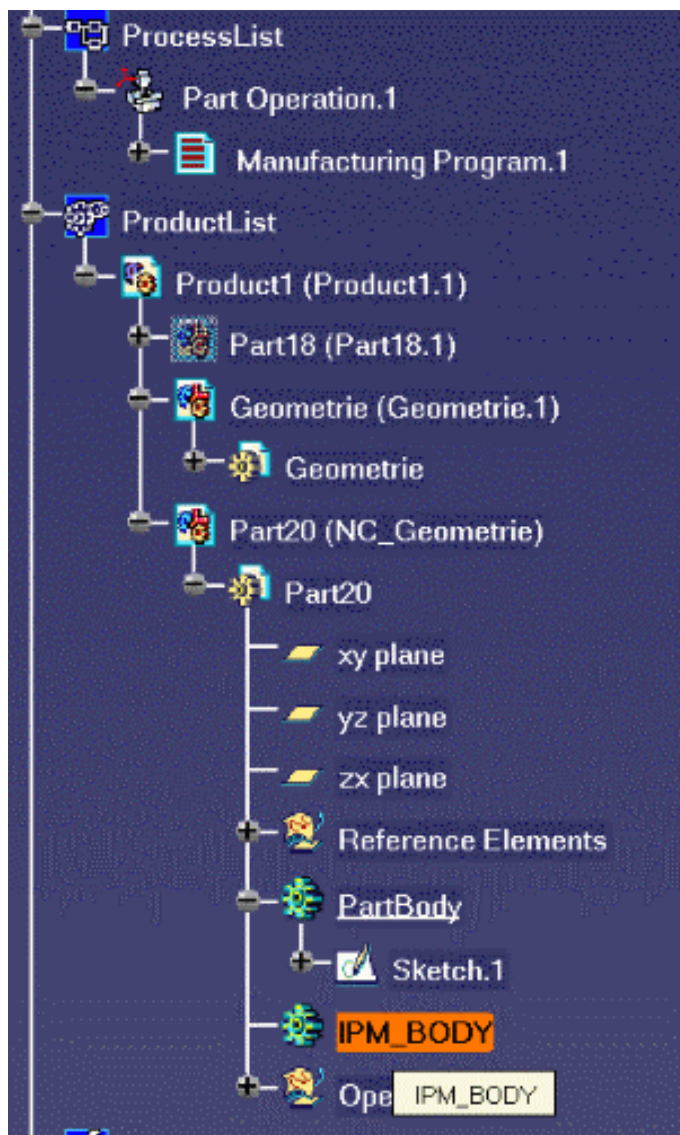
To summarize:

The setup for the computation, we need to define in Part Operation panel:

- a Lathe Machine with its spindle and radial axis initialized
- a Stock for simulation (solid) selected in the Part Operation panel
- an In Process Model body created inside an existing or newly created Part.

The operations taken into account by the Actual Stock computation are:

- Lathe Operations:
  - Roughing
  - Grooving
  - Recessing
  - Profile Finishing
  - Groove Finishing.
- All Drilling operations using a "fixed" Tool Assembly are taken into account. For this, select the Assembly tab page and click on More. In the Technology tab page the parameter "Power" must be set to "Fixed".

Lathe Threading operations are not taken into account for Actual Stock computation.

Example of Actual Stock Computation

The following figure illustrates the launching of the Actual Stock computation after a Drilling (with fixed Tool

assembly) and a Roughing operation. Here is the tool path of the roughing operation:
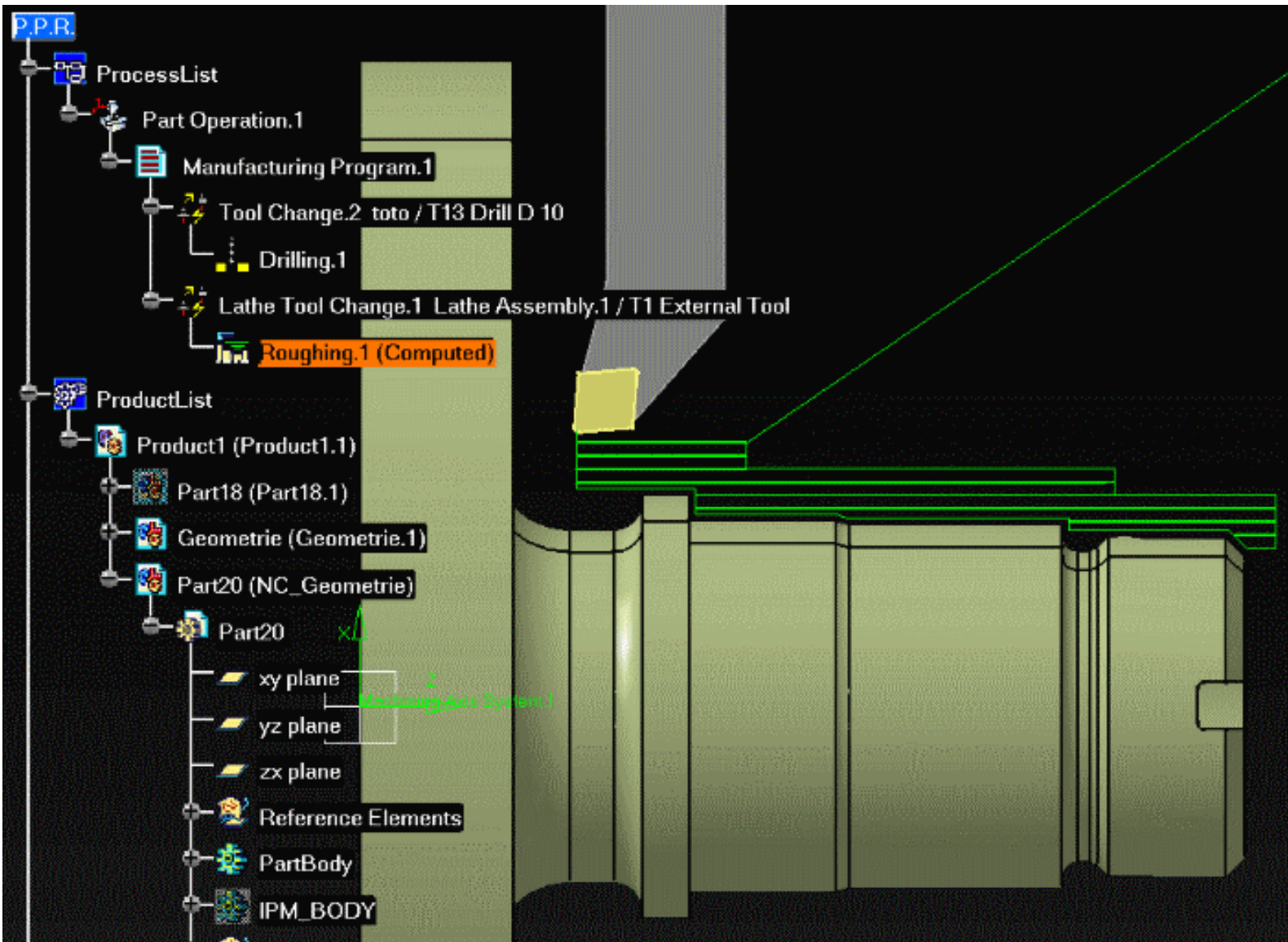
If you want to create another Roughing operation after the first one, click "Actual Stock computation" in the new Roughing operation's Geometry tab page.

The following figure shows the result of the computation. The Actual Stock has been created in blue. It is inside a sketch whose name is the name of the last machining operation taken into account for the computation: Roughing.1 plus the item StockSketch and the time of the computation. You can see that the Actual Stock sketch has been created in a body with the same name, just under the "IPM body". The "IPM body" is now in No Show mode.
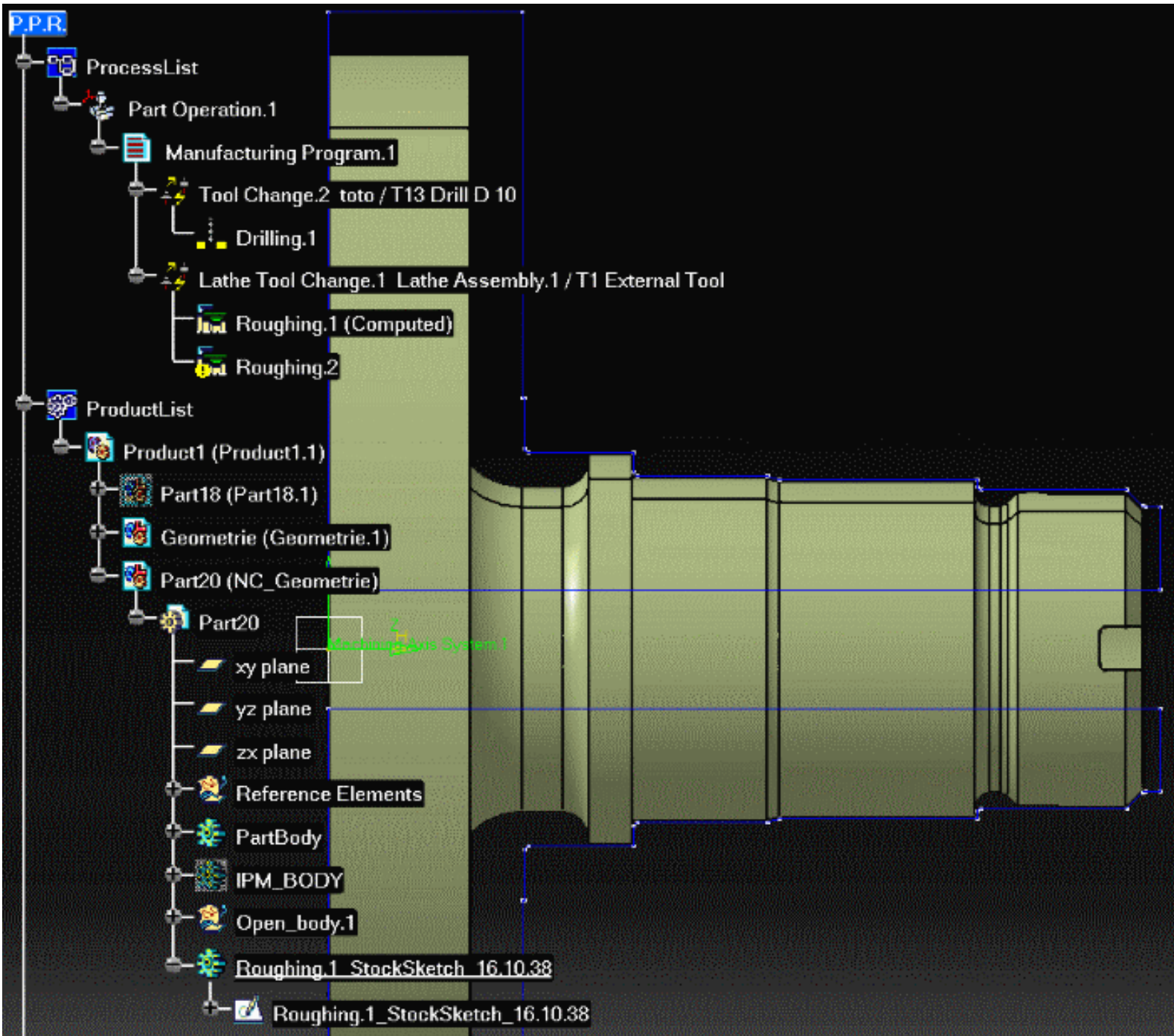
You can now select elements of the Actual Stock sketch as new Part or new Stock item of the Roughing.2 operation.

To create the Actual Stock, some primitives are created inside the IPM body. Once the Actual Stock sketch is created, the Actual Stock sketch is no longer linked to them. If you use the Actual Stock computation again, the existing primitives will speed up the computation (they will be used if the operation they are linked with have not been been modified since the first call to Actual Stock, otherwise they will be modified). This command will not remove the Actual Stock sketches or the IPM body.

To manage different Actual Stock sketches, you can change the color of each sketch by modifying the graphic property of the sketch.

If you have finished your Part Operation, you can remove these primitives by using the Remove IPM primitives contextual command on the Manufacturing Program. This command will remove each primitive of the IPM body linked to the operation inside the program.

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). 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. 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.

Index

External Profile Finishing

External Recessing

External Roughing

External Threading

F

Face

    Roughing mode

Face Roughing

Feeds and Speeds

    Groove Finishing parameter

    Grooving parameter

    Profile Finishing parameter

    Recessing parameter

    Roughing parameter

First passes

    Threading parameter

First Plunge

    Grooving parameter

Follow

    Sequential Machining motion

Follow Tool Motion

Frontal Groove Finishing

Frontal Grooving

Frontal Profile Finishing

Frontal Recessing

Frontal Roughing

G

Go Delta

    Sequential Machining motion

Go Delta Motion

Go InDirv

    Sequential Machining motion

Go InDirv Tool Motion

Go Standard

    Sequential Machining motion

Go Standard Tool Motion

Go-Go

    Sequential Machining motion

Go-Go Tool Motion

Gouging Safety Angle

    Grooving parameter

    Recessing parameter

Groove Finishing operation

Groove Finishing parameter

    Angle of Incline

    Axial offset on part

    Change Output Point

    Clearance

    Contouring for Outside Corners

    Corner Processing

    CUTCOM

    End Limit

    End Limit Clearance

    Feeds and Speeds

Lead-in ⊞

Lift-off ⊞

Machining Direction ⊞

Machining Tolerance ⊞

Orientation ⊞

Overlap ⊞

Radial offset on part ⊞

Start Limit ⊞

Start Limit Clearance ⊞

Thickness on part ⊞

Tool Compensation ⊞

Under Spindle Axis Machining ⊞

Grooving by Level

Grooving parameter ⊞

Grooving operation ⊞ ⊞

Grooving parameter

Angle of Incline ⊞

Attack ⊞

Axial offset on part ⊞

Change Output Point ⊞

Chip Break ⊞

Clearance on stock ⊞

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

Radial offset on part

Thickness on part

Tool Compensation

Under Spindle Axis Machining

I

Inclined Groove Finishing

Inclined Grooving

Inclined Recessing

insert holder

Internal Groove Finishing

Internal Grooving

Internal Profile Finishing

Internal Recessing

Internal Roughing

Internal Threading

L

Last passes

Threading parameter

Lead-in

Groove Finishing parameter

Grooving parameter

Profile Finishing parameter

Recessing parameter

Roughing parameter

Threading parameter

Leading Safety Angle

Profile Finishing parameter

Recessing parameter

Roughing parameter

Lift-off

Groove Finishing parameter

Grooving parameter

Profile Finishing parameter

Recessing parameter

Roughing parameter

Threading parameter

Local Information

Location

Profile Finishing parameter

Roughing parameter

Threading parameter

Longitudinal

Roughing mode

Longitudinal Roughing

M

Machine Rotation 🔲

Machining Axis or Origin 🔲

Machining Direction

      Groove Finishing parameter 🔲

      Profile Finishing parameter 🔲

      Recessing parameter 🔲

      Roughing parameter 🔲

Machining Process, Apply 🔲

Machining Process, Create 🔲

Machining spindle speed

      Threading parameter 🔲

Machining Tolerance

      Groove Finishing parameter 🔲

      Grooving parameter 🔲

      Profile Finishing parameter 🔲

      Recessing parameter 🔲

      Roughing parameter 🔲

Manufacturing Program 🔲

Maximum Depth of Cut

      Grooving parameter 🔲

      Recessing parameter 🔲

      Roughing parameter 🔲

Maximum Machining Radius

      Roughing parameter 🔲

Minimum Machining Radius

      Roughing parameter 🔲

🔲

N

O

P

Parallel Contour

    Recessing mode

    Roughing mode

Parallel Contour Recessing

Parallel Contour Roughing

Part Contouring

    Grooving parameter

    Recessing parameter

    Roughing parameter

Part Operation

Plunge

    Recessing parameter

Plunge Angle

    Roughing parameter

Plunge Distance

    Roughing parameter

PP Instruction

Process List

Product List

Profile Finishing operation

Profile Finishing parameter

    Axial offset on part

    Contouring for Outside Corners

    Corner Processing

    CUTCOM

    End Limit

    End Limit Clearance

Feeds and Speeds

Lead-in

Leading Safety Angle

Lift-off

Location

Machining Direction

Machining Tolerance

Orientation

Radial offset on part

Recess Machining

Start Limit

Start Limit Clearance

Thickness on part

Tool Compensation

Trailing Safety Angle

Under Spindle Axis Machining

R

Radial Depth of Cut

Recessing parameter

Roughing parameter

Radial offset on part

Groove Finishing parameter

Grooving parameter

Profile Finishing parameter

Recessing parameter

Roughing parameter

Recess Machining

    Profile Finishing parameter

    Roughing parameter

Recessing mode

    One Way

    Parallel Contour

    Zig Zag

Recessing operation

Recessing parameter

    Angle of Incline

    Attack

    Axial Depth of Cut

    Axial offset on part

    Change Output Point

    Clearance on stock

    Feeds and Speeds

    Gouging Safety Angle

    Lead-in

    Leading Safety Angle

    Lift-off

    Machining Direction

    Machining Tolerance

    Maximum Depth of Cut

    Orientation

    Part Contouring

    Plunge

    Radial Depth of Cut

    Radial offset on part

S

Groove Finishing parameter

Profile Finishing parameter

Threading parameter

Start Limit Clearance

Groove Finishing parameter

Profile Finishing parameter

Threading parameter

T

Thickness on part

Groove Finishing parameter

Grooving parameter

Profile Finishing parameter

Recessing parameter

Roughing parameter

Thread depth

Threading parameter

Thread length

Threading parameter

Thread Penetration type

Threading parameter

Thread pitch

Threading parameter

Thread profile

Threading parameter

Thread unit

Threading parameter

Threading operation

Threading parameter

Change Output Point

Clearance on crest diameter

Depth of cut

End Limit

End Limit Clearance

First passes

Last passes

Lead-in

Lift-off

Location

Machining spindle speed

Nominal diameter

Number of passes

Number of threads

Orientation

Spring passes

Start Limit

Start Limit Clearance

Thread depth

Thread length

Thread Penetration type

Thread pitch

Thread profile

Thread unit

Threading type

Threads/inch

Tool Compensation

Threading type

U

Profile Finishing parameter ⬚

Recessing parameter ⬚

Roughing parameter ⬚

⬚

Z

Zig Zag

Recessing mode ⬚

Zig Zag Recessing ⬚

⬚