# NC Manufacturing Review

# Overview

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

This overview provides the following information:



## NC Manufacturing Review in a Nutshell

NC Manufacturing Review provides fast and easy visualization of 3D NC Manufacturing data. It provides easy access, visualization, and browsing of CATIA V4 and V5 manufacturing data but also any CAM systems pre-processing specifications and results. This product delivers infrastructure basics to create and integrate machining workbenches and additional machining strategies with a full integration in V5 NC Manufacturing platform.

NC Manufacturing Review offers the following main functions:

- Multi-CAM reviewer
- Import and review NC Data (APTfile, CLfile or NC code) which may be CATIA or other files
- Import and review CATIA V4 NCMILL and NCLATHE programs
- Import and review CATIA V5 CATProcess part programs
- Infrastructure for other V5 NC Manufacturing programming products
- Infrastructure for partner products.

## Before Reading this Guide

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

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

- *NC Manufacturing Infrastructure User's Guide*: explains how to use common NC Manufacturing functionalities.

## Getting the Most Out of this Guide

To get the most out of this guide, we suggest that you start reading and performing the step-by-step Getting Started tutorial. This tutorial will show you how to how to review NC manufacturing data.

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

The Workbench Description section, which describes the commands that are specific to NC Manufacturing Review, and the Customizing section, which explains how to customize settings, will also certainly prove useful.

## Accessing Sample Documents

To perform the scenarios, you will be using sample documents contained in the doc/online/ncgug_C2/samples or doc/online/ncgug_D2/samples folder. For more information about this, refer to Accessing Sample Documents in the *Infrastructure User's Guide*.

# Conventions

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

## Graphic Conventions

The three categories of graphic conventions used are as follows:

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

### Graphic Conventions Structuring the Tasks

Graphic conventions structuring the tasks are denoted as follows:

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

## Graphic Conventions Indicating the Configuration Required

Graphic conventions indicating the configuration required are denoted as follows:

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

## Graphic Conventions Used in the Table of Contents

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

| This icon... | Gives access to... |
|---|---|
|  | Site Map |
| | Split View mode |
| | What's New? |
| | Overview |
| | Getting Started |
| | Basic Tasks |
| | User Tasks or the Advanced Tasks |
| | Workbench Description |
| | Customizing |
| | Reference |
| | Methodology |
| | Glossary |
| | Index |

# Text Conventions

The following text conventions are used:

- The titles of CATIA, ENOVIA and DELMIA documents *appear in this manner* throughout the text.
- **File** -> **New** identifies the commands to be used.

- Enhancements are identified by a blue-colored background on the text.

# How to Use the Mouse

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

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

# What's New?

No enhancements in this release.

# Getting Started

Before getting into the detailed instructions for using NC Manufacturing Review, 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.

Reviewing NC Manufacturing Data

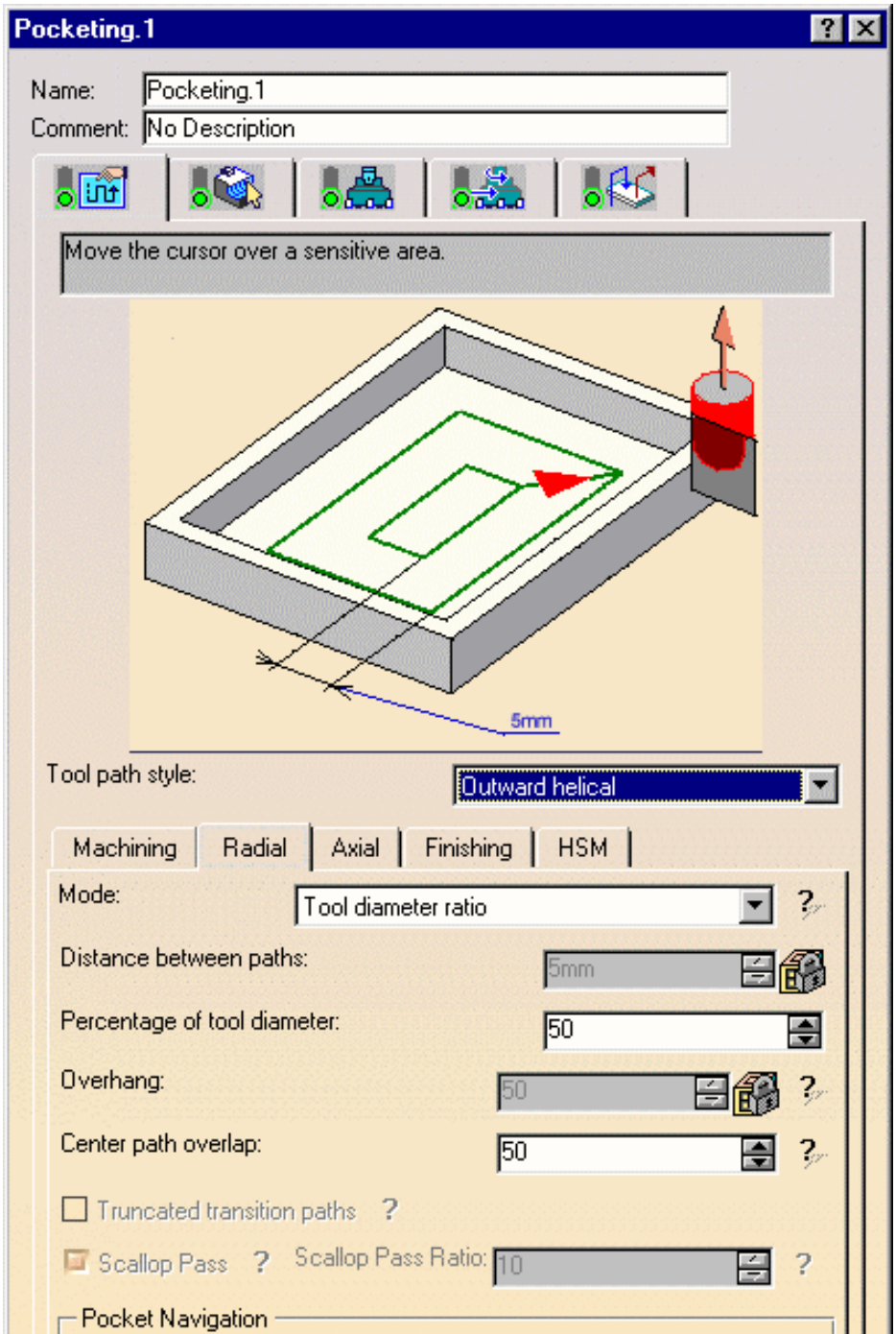
# Reviewing NC Manufacturing Data

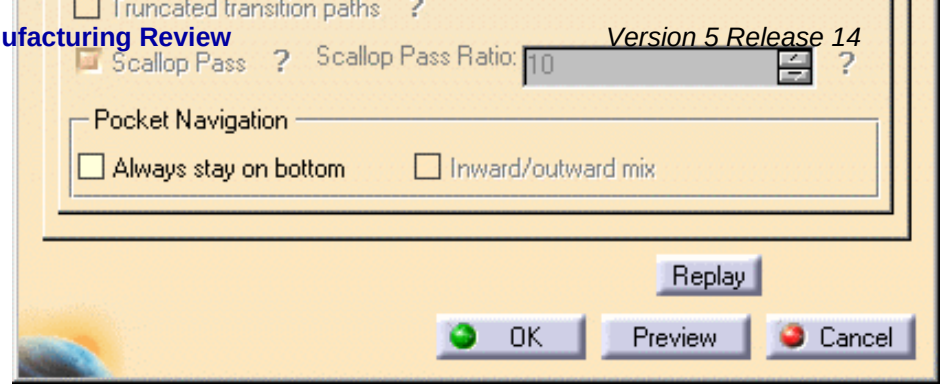
This task shows you how to review NC manufacturing data.

1. Enter the NC Manufacturing Review workbench and click File > Open to select a CATProcess.

   The PPR tree appears.


2. Double click the Pocketing operation in the tree. The Pocketing dialog box appears.

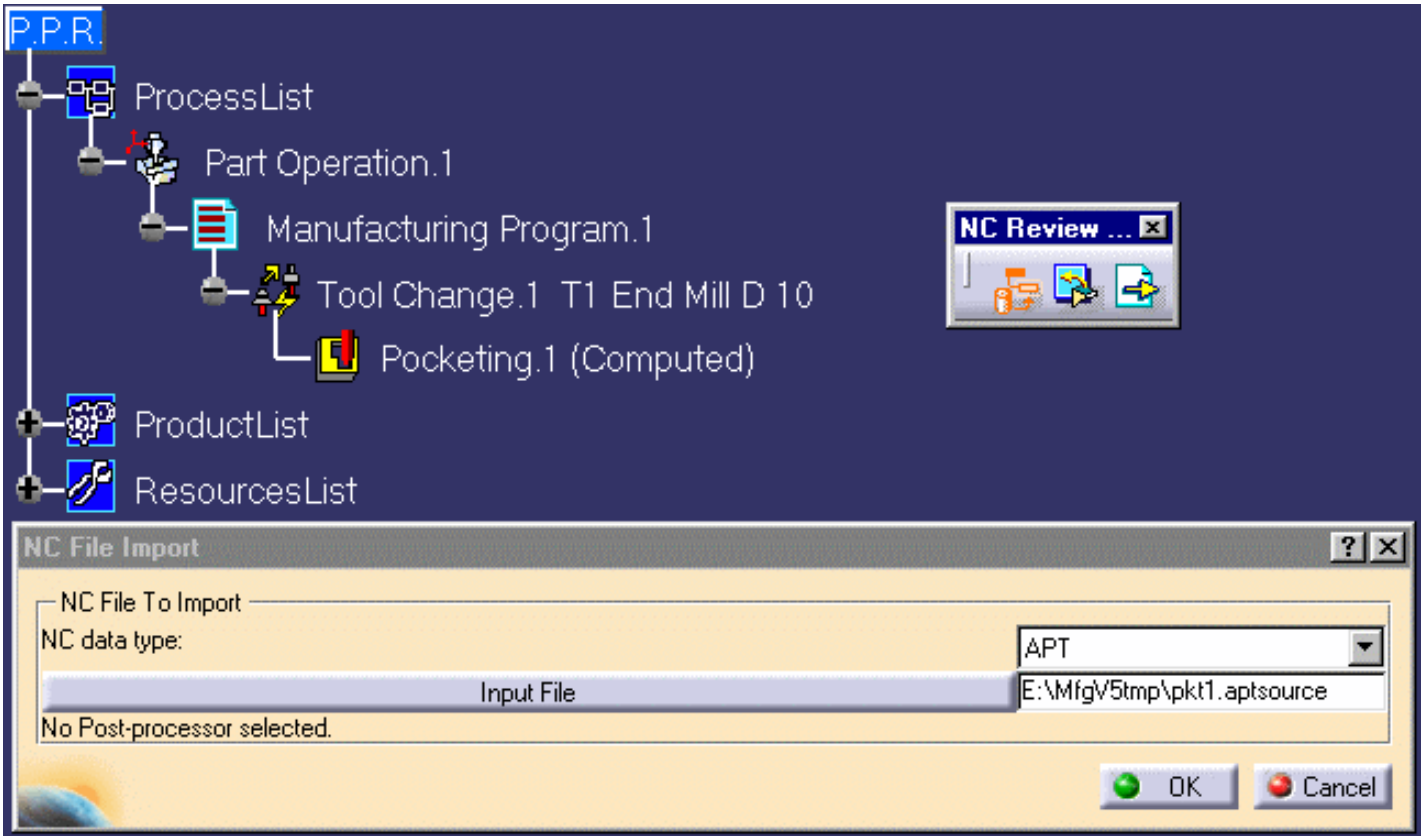





3. The properties of the operation are organized in tab pages. You can browse and, under certain conditions, edit these properties.
To end the consultation of the pocketing data, just click Cancel to close the machining operation dialog box.

4. Click the Import APT, Clfile or NC code icon or right-click the program and select **Import APT, Clfile or NC code** from the contextual menu.

The NC File Import dialog box appears.

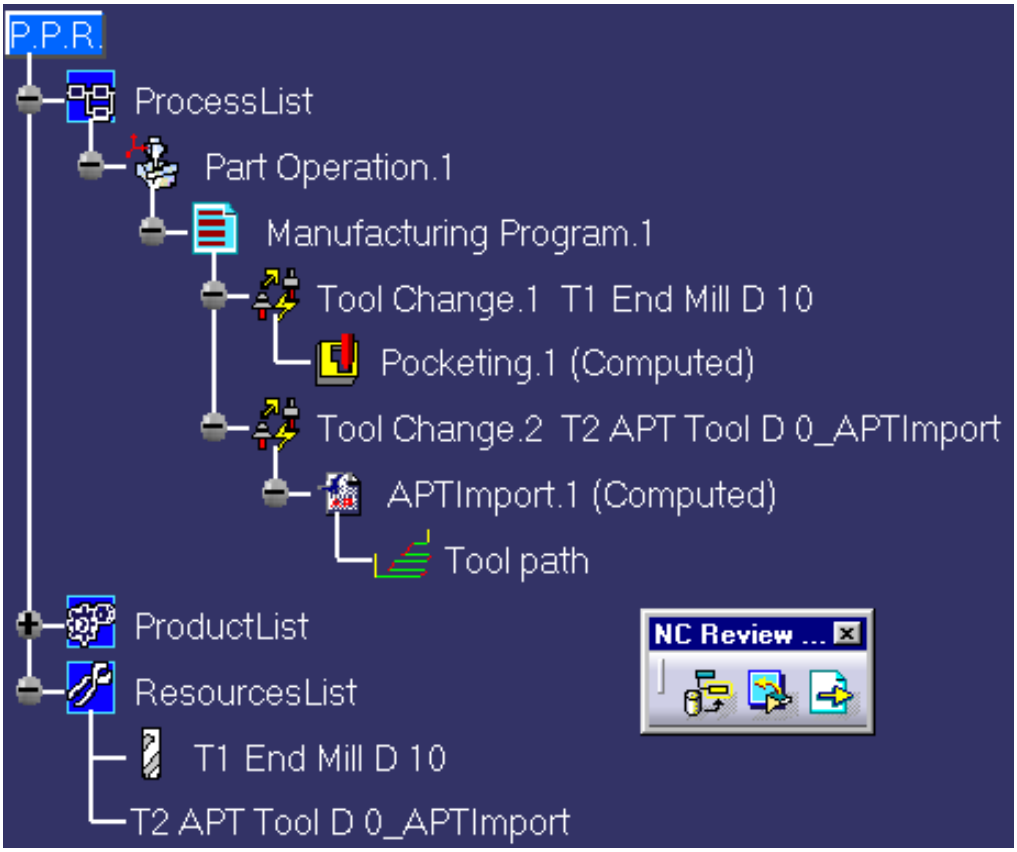



5. Select the type of NC data file to import using the combo and specify the file to import.

Click OK to import the specified file.
The PPR tree is updated with the manufacturing information of the imported file.

NC data objects appear in the Manufacturing Program for each tool change in the imported file.
A Tool path object is associated to each NC data object.

All tools referenced in the imported NC file appear in the Resources List.




Note that you can replay the imported APT file using the contextual menu command.

There is also the possibility of editing the tool path using contextual menu commands.

# User Tasks

The user tasks you will perform in the NC Manufacturing Review workbench involve reviewing manufacturing data in a Version 5 environment. This manufacturing data includes Version 5 CATProcess documents, Version 4 NC Mill and NC Lathe models, and NC data such as APT source files.

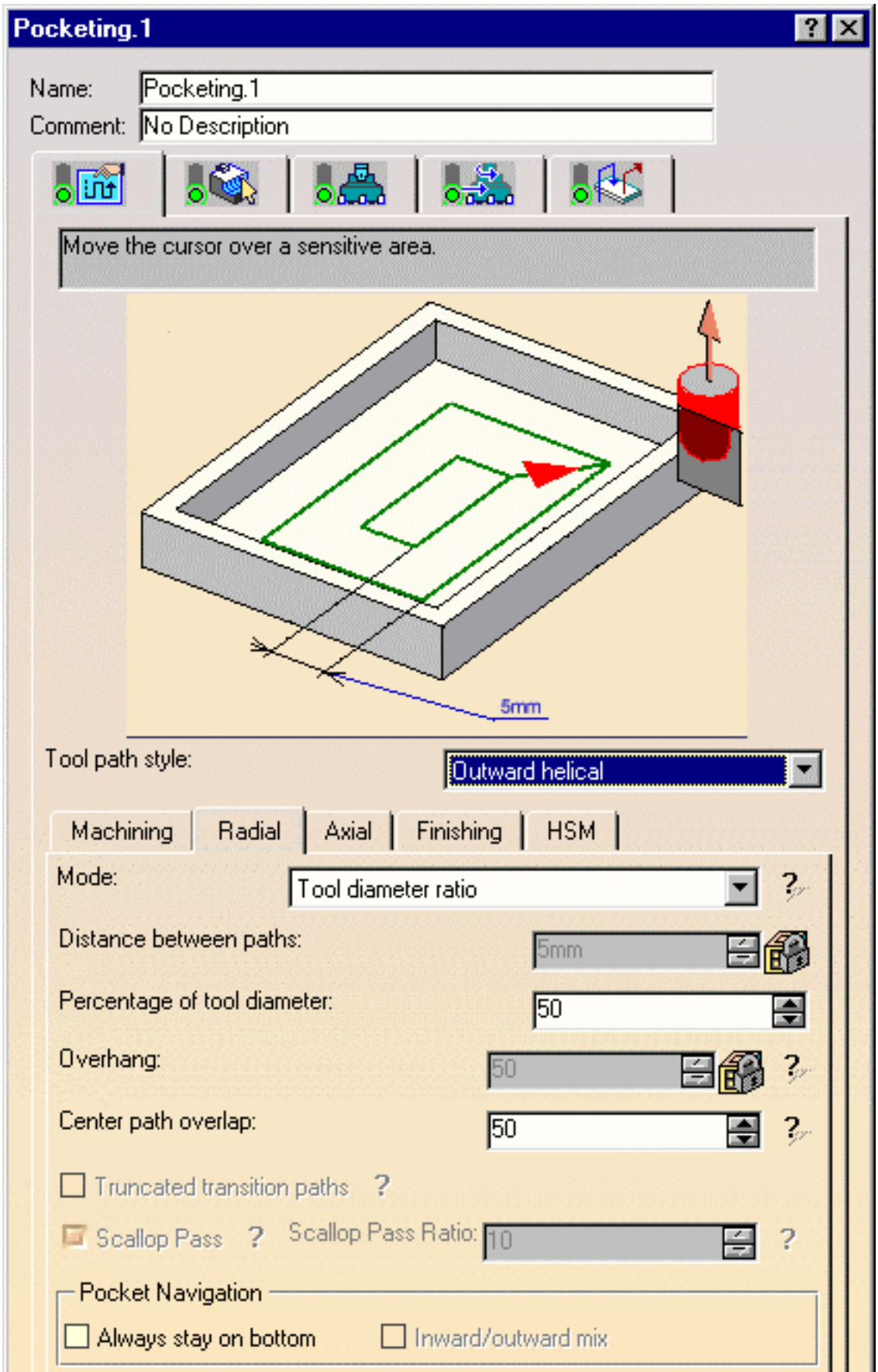
Browse V5 Manufacturing CATProcess
Browse V4 NC Mill Model
Browse V4 NC Mill Operation
Associate APT File to NC Mill Program
Browse V4 NC Lathe Model
Browse V4 NC Lathe Operation
Associate APT File to NC Lathe Program
Import NC Data
Replay and Simulate Tool Path
Output NC Data
Generate Documentation
Edit Tool Path

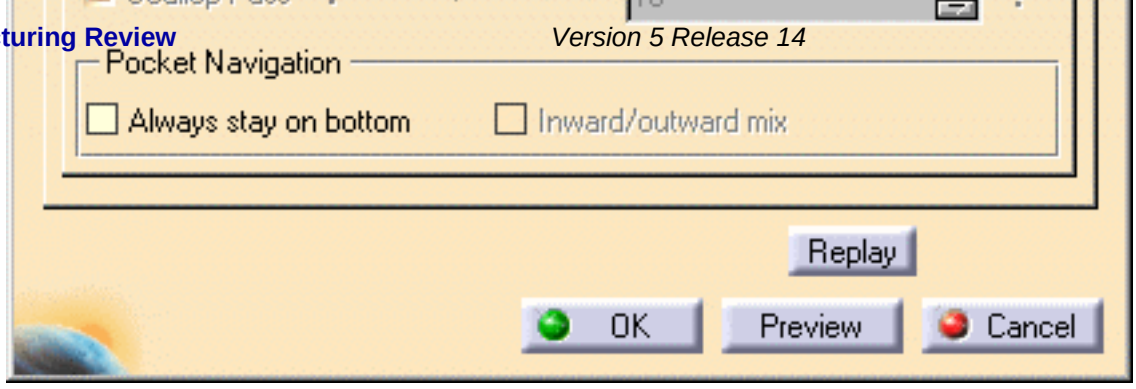
# Browse a V5 Manufacturing CATProcess

This task shows you how to browse a V5 Manufacturing CATProcess in the NC Manufacturing Review workbench.

1. Enter the NC Manufacturing Review workbench and click File > Open to select a CATProcess.

2. Double click the Pocketing operation in the tree. The Pocketing dialog box appears.






3. The properties of the operation are organized in tab pages as follows:

the parameters of the machining strategy, which are divided up into different tabs

the geometry to be machined

the tool that is used in the operation

the feedrates and spindle speeds

the macros (transition paths).

the PP word syntaxes (this tab is displayed for some operation types only).

You can browse and, under certain conditions, edit these properties.

If the license of the application allowing creation /edition of the machining operation is available, the OK button is accessible: the machining operation can be modified.

Otherwise, the OK button is inactive: the machining operation cannot be modified.

The Tool Path Replay functionality is accessible:

- if a tool path is present on the machining operation (computed status)
- if no tool path but the license of the application allowing creation/edition of the machining operation is available.

Otherwise the Replay button is inactive (not accessible).

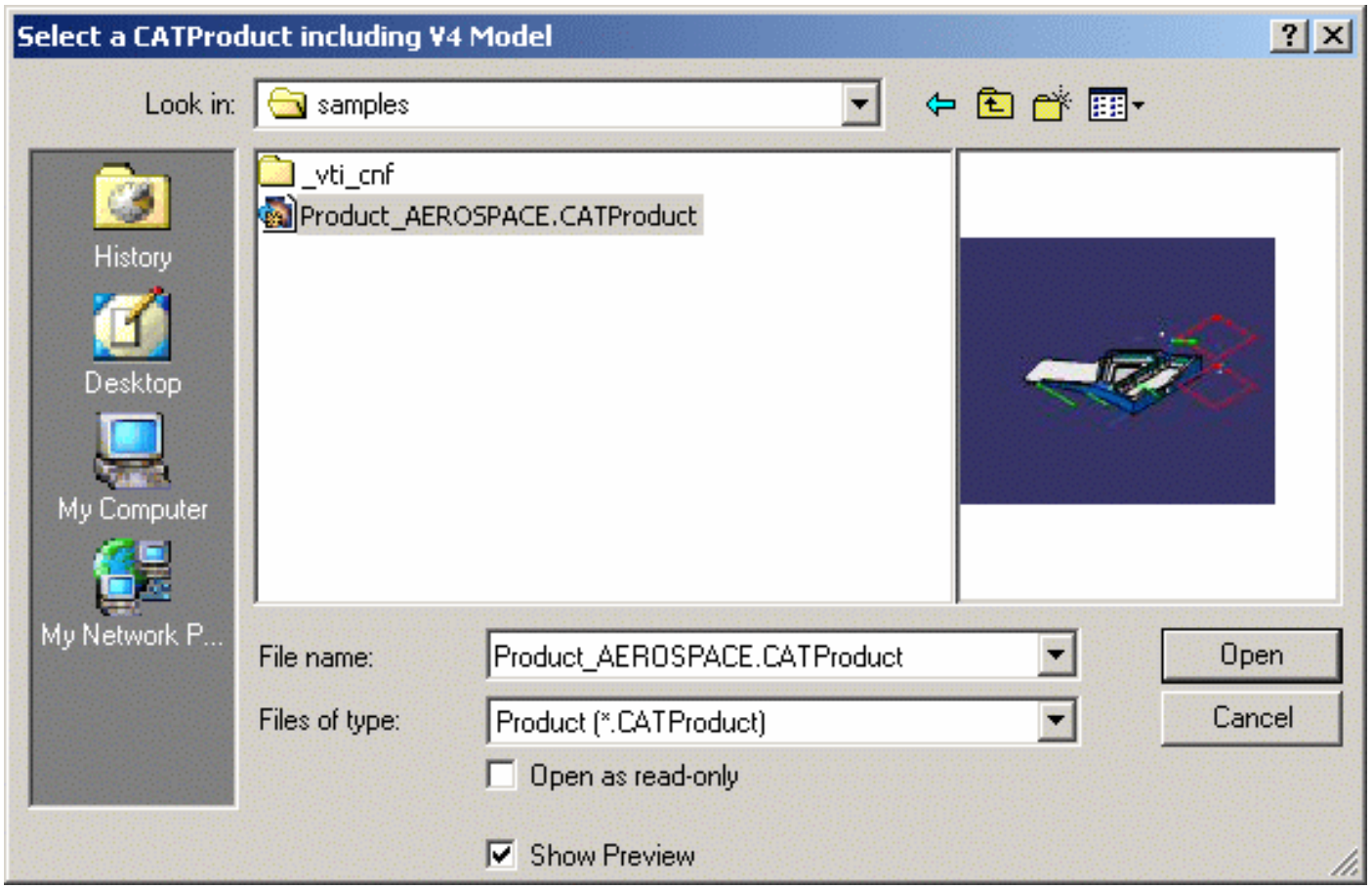
4. Just click Cancel to close the machining operation dialog box.

# Browse a V4 NC Mill Model

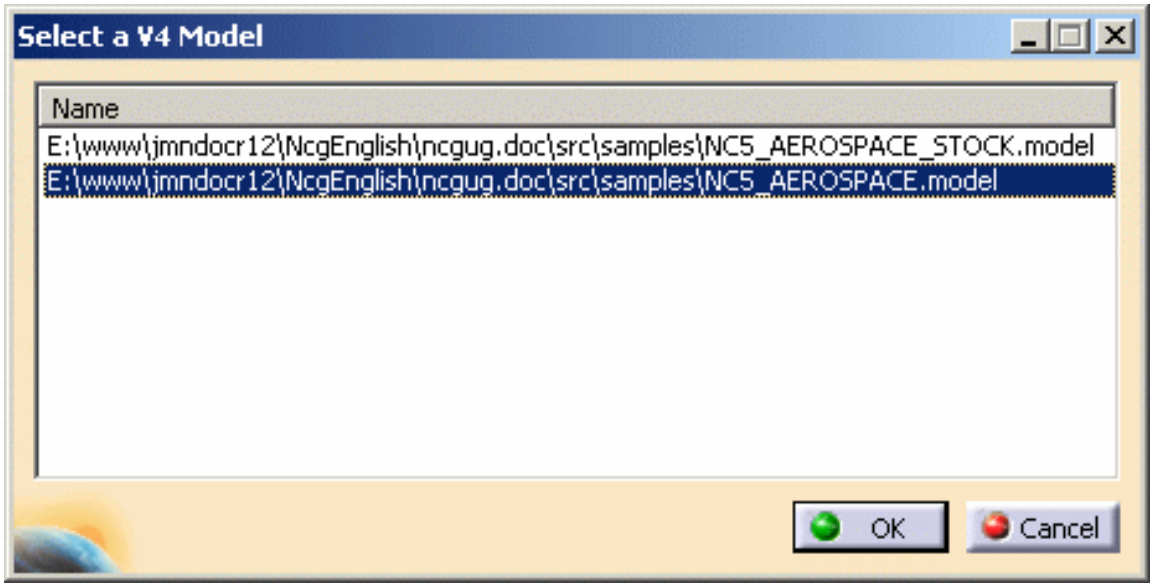
This task shows you how to browse a V4 NC Mill Model in the NC Manufacturing Review workbench.

For this task you should use a CATProduct that includes one or more V4 models.

1. Enter the NC Manufacturing Review workbench and click the Read Manufacturing Data from a V4 Model icon . The following dialog box appears.



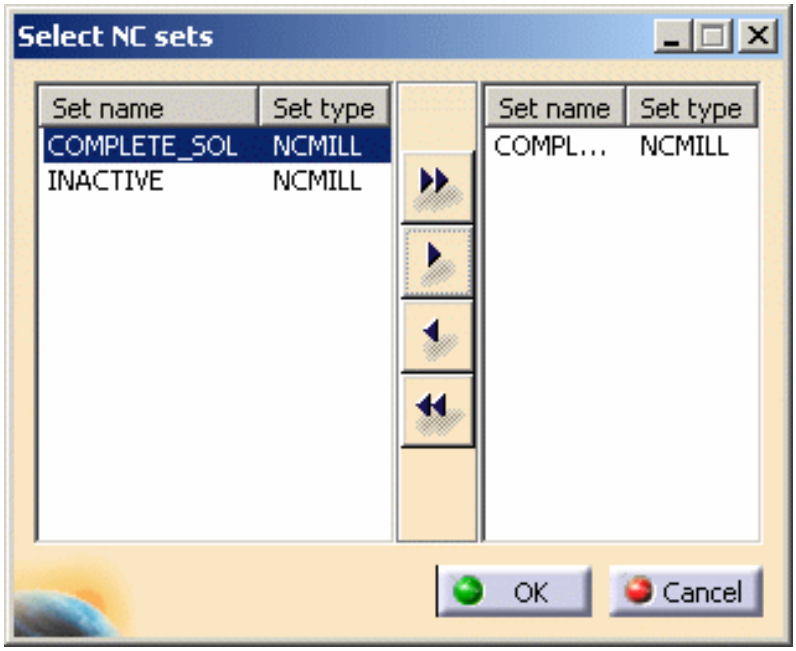

2. Navigate to the desired CATproduct and click Open.
3. If the CATProduct includes more than one V4 model, the following dialog box appears.




Select the desired V4 model and click OK.

4. The Select NC Mill Sets dialog box appears showing the list of NC Mill sets in the selected model in the left-hand column.

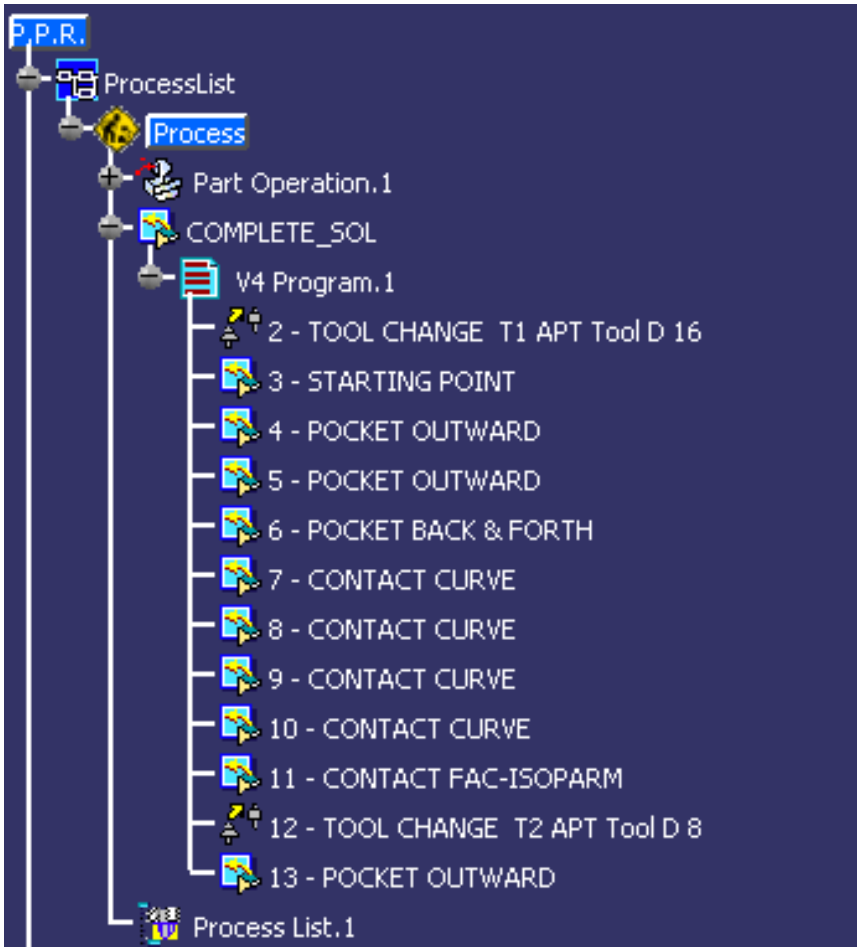
Use the arrow buttons to move the NC Mill sets to be processed to the right-hand column. One or more sets can be selected and ordered using the arrow buttons. Click OK.




A progression bar appears giving the status of the NC Mill set and machining operation being processed.

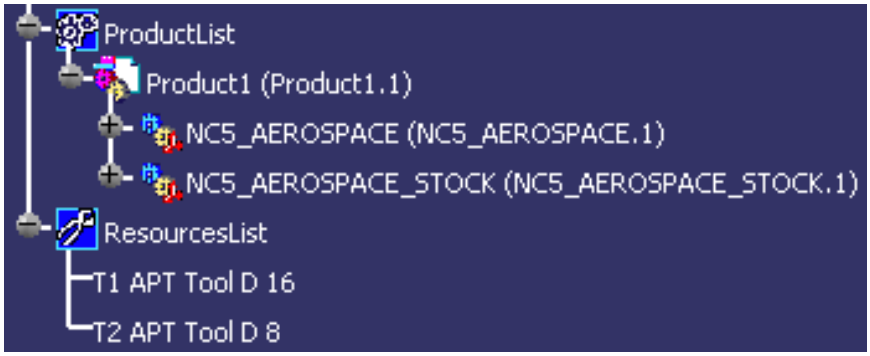
The selected NC Mill set is added to the Process List at Part Operation level. The PPR tree is updated with the manufacturing information of the selected NC Mill set.

A program **V4 Program.1** containing all the machining operations and tool changes of the NC Mill set is added at Manufacturing Program level.




A product Product1 is created and associated to the selected NC Mill set.

All tools referenced in the NC Mill set are added to the Resources List.




5. Double click the Part Operation entity in the PPR tree to access the Part Operation editor. Use the editor:
   - to associate a machine to the Part Operation
   - to associate a stock to the Part Operation. This is useful if you want to do material removal simulation at a later stage (that is, once NC data has been associated to the V4 program).
6. Double click a machining operation in the program to browse its properties. Please refer to Browse a V4 NC Mill Operation for more information.

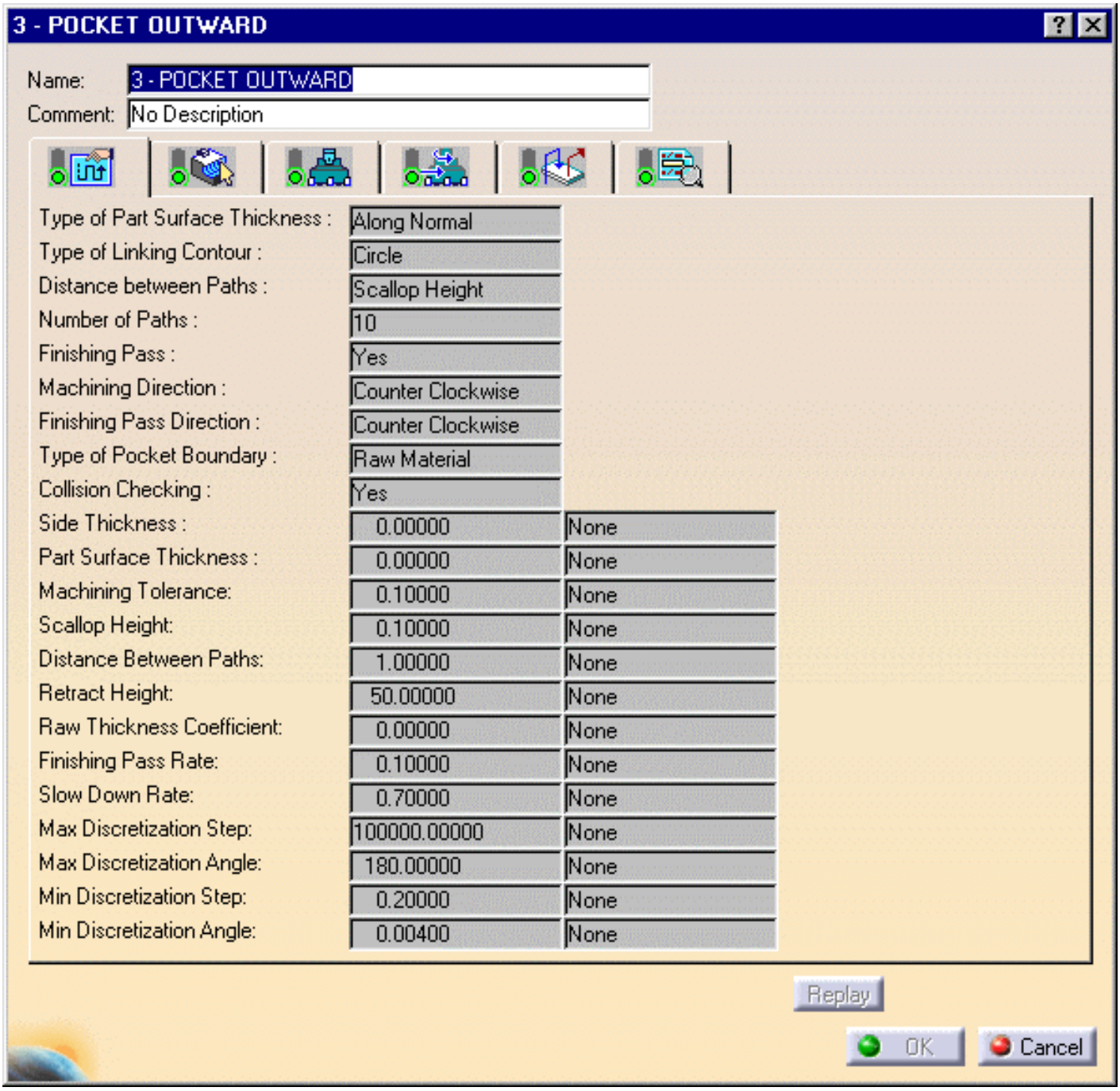
- Only geometry with SHOW status in the V4 model is displayed.
- You can now associate NC data to your V4 program. Please refer to Associate an APT File to an NC Mill Program for more information.

# Browse a V4 NC Mill Operation

This task shows you how to browse a V4 NC Mill Machining Operation in the NC Manufacturing Review workbench.

1. In the specification tree containing a Part Operation originating from V4, double click the machining operation to be browsed (see Browse a V4 NC Mill Model).

   The machining operation browse panel appears.




The properties are organized in tab pages as follows:

the parameters of the machining strategy

the geometry to be machined

the tool that is used in the operation

the feedrates

the macros (transition paths)

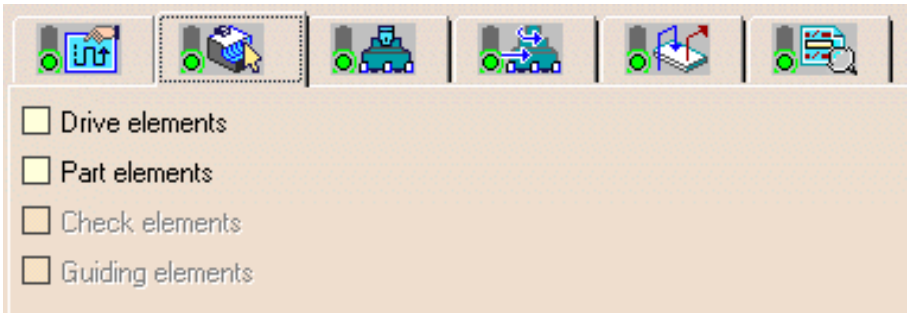
the PP word syntaxes.

**Parameters of the machining strategy page** 

The set of machining options of the NC Mill machining operation are displayed along with their values and any associated parameters names. The right field is set to None if the option is not parameterized.

Note that all possible option values are listed for each operation.
However, some options may not be used by the operation. For example, the Distance between Paths option may be set to Scallop Height, Number of Paths or Distance between Paths. In the panel above it is set to Scallop Height. In this case only the Scallop Height value is useful: the Number of Paths and Distance between Paths values should be ignored.

**Geometry to be machined** 




The set of possible geometry entity types are listed for the operation. Entity types that are not referenced by the selected operation are dimmed. Entity types that are referenced by the selected operation can be highlighted by clicking the corresponding checkbox.

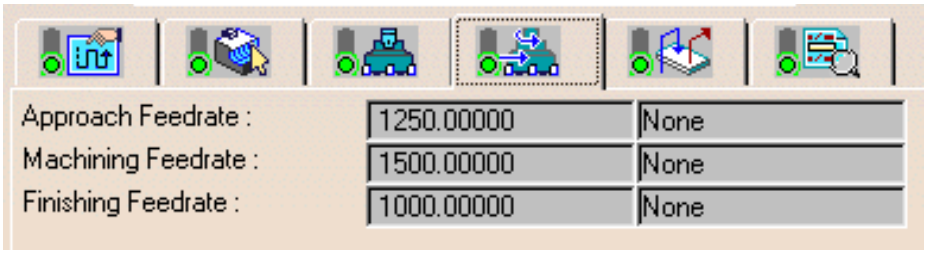
Only geometry with SHOW status in the V4 model is considered to be linked to the operation.

**Tool used in the operation** 


The V4 tool information is listed along with the corresponding CUTTER/ statement. If a Detailed tool was used no information is displayed.

This is the information of the previous Tool Change operation of the NC Mill set.

**Feedrates**

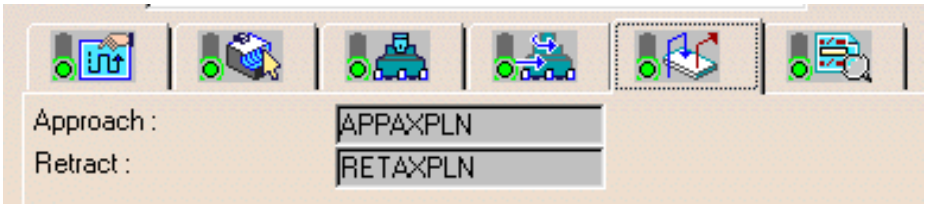



The set of feedrates of the NC Mill machining operation are displayed along with their values and any associated parameters names. This is set to None if the feedrate is not parameterized.

Note that all possible feedrates are listed for each operation.

Note that no spindle speeds are displayed as these were not managed for NC mill operations.

**Macros (transition paths)**




All possible macro types of the operation are listed along with the corresponding macro names. If no macro of a given type is linked to the operation, the name field is left blank.

**PP word syntaxes**

This tab page contains information for CYCLE type operations only.

2. Just click Cancel to close the machining operation panel.

# Associate an APT File to an NC Mill Program

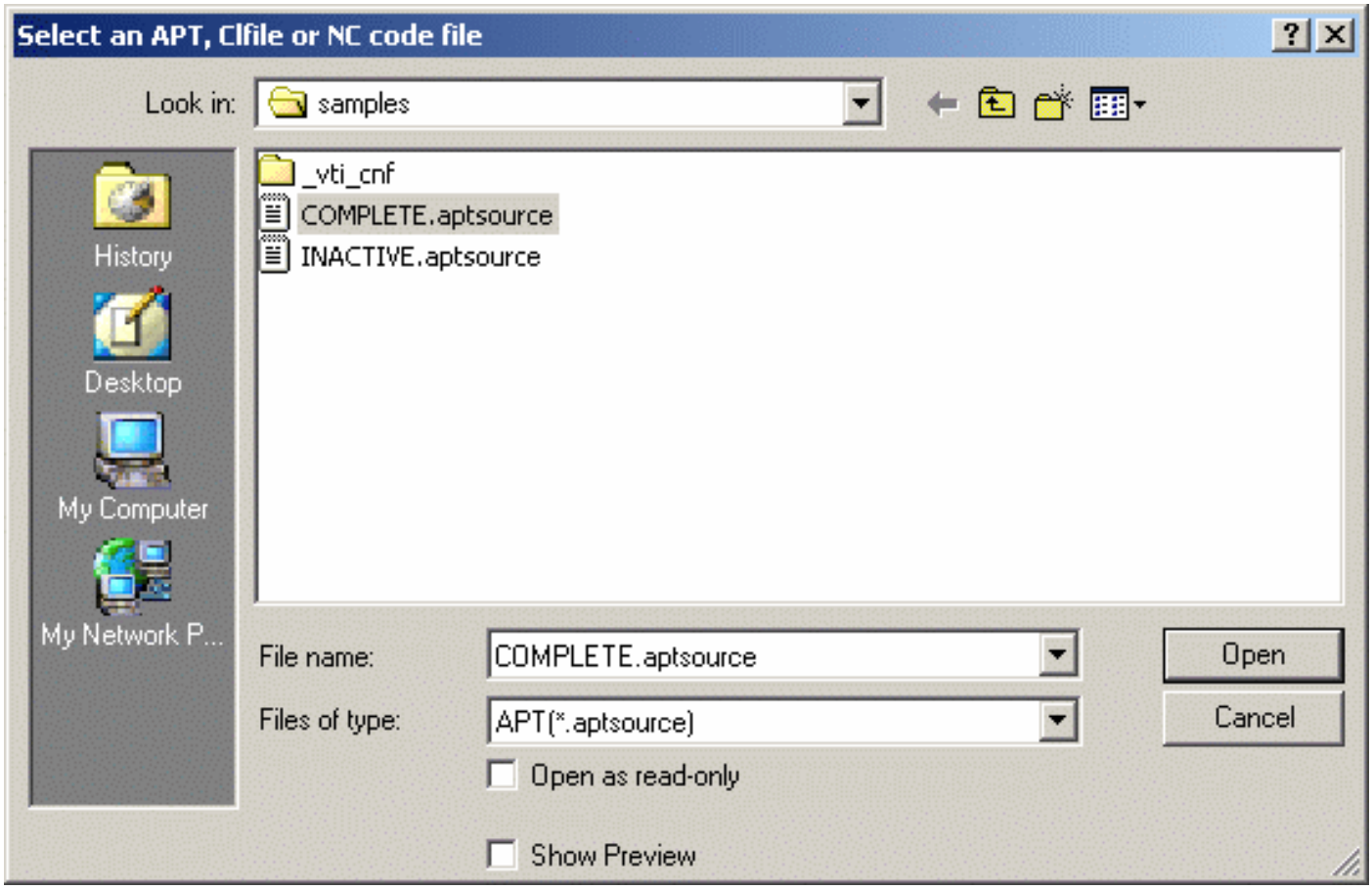
This task shows you how to associate an APT file to an NC Mill program using the NC Manufacturing Review workbench.

This association consists in creating a tool path for each operation by extracting the necessary information from the APT file. This allows verification and simulation of these operations.

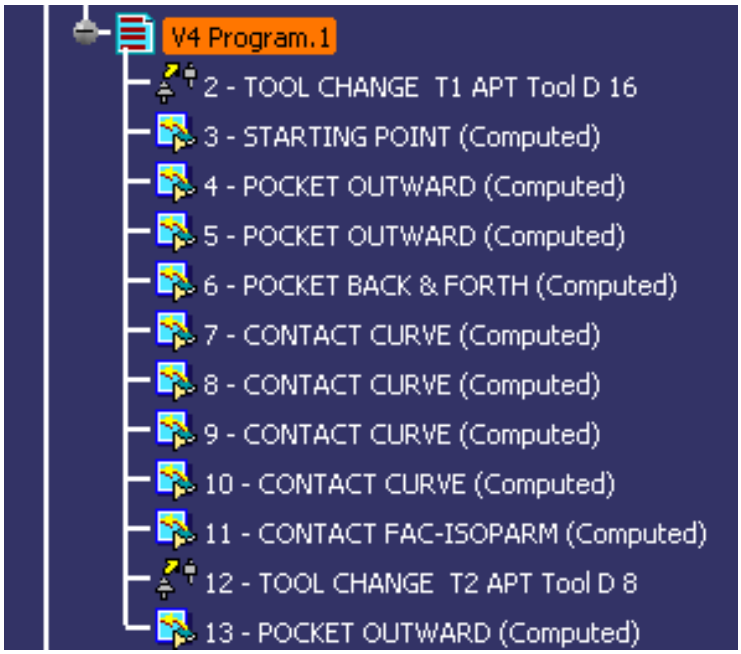
You must have a APT source file generated by the CATNC utility for the imported NC Mill program.

Note that CATNC must be executed with the PPRINT processing option set to PPRINT (that is, *PPR or *PPRINT keyword set to 1).
This is to allow identifying the part of the tool path corresponding to each operation and so to be able to define the tool path of each operation. Identification is processed by means of the PPRINT OPERATION NUMBER statements.

1. Select the V4 program in the specification tree.
2. Select the Associate Output NC File to a Program icon . The dialog box for selecting the NC file appears.
3. Select the NC file to associate to the program.




The tree is updated with tool paths defined on the machining operations.




- The machining operations with **Computed** status have their own tool path which can be replayed and, if a stock geometry is linked to the Part Operation, simulated.
- APT association must be done in a CATIA session working in the same units as those in which APT was generated.
- The APT association can only be done on V4 type programs.
- The APT association can only be done whenever there is a PPRINT OPERATION NUMBER statement and there are as many entities in the program as PPRINT OPERATION NUMBER statements in the APT (except the first one, which defines CATIA0).
- The APT association may be done if there are COPY and TRACUT type entities present, and Tool Path Replay is available for these entities.
- A 'continue' statement followed by a PPRINT OPERATION NUMBER statement may present a problem. Example :

```
PPRINT OPERATION NUMBER : 6
PPRINT *** TOOL NO SF8-1093-10 6.00DIA. .125CRAD. .625LOC. 4.500SL$
PPRINT OPERATION NUMBER : 7
PPRINT ************************************************************
```

You should remove all unnecessary 'continue' statements before APT association.

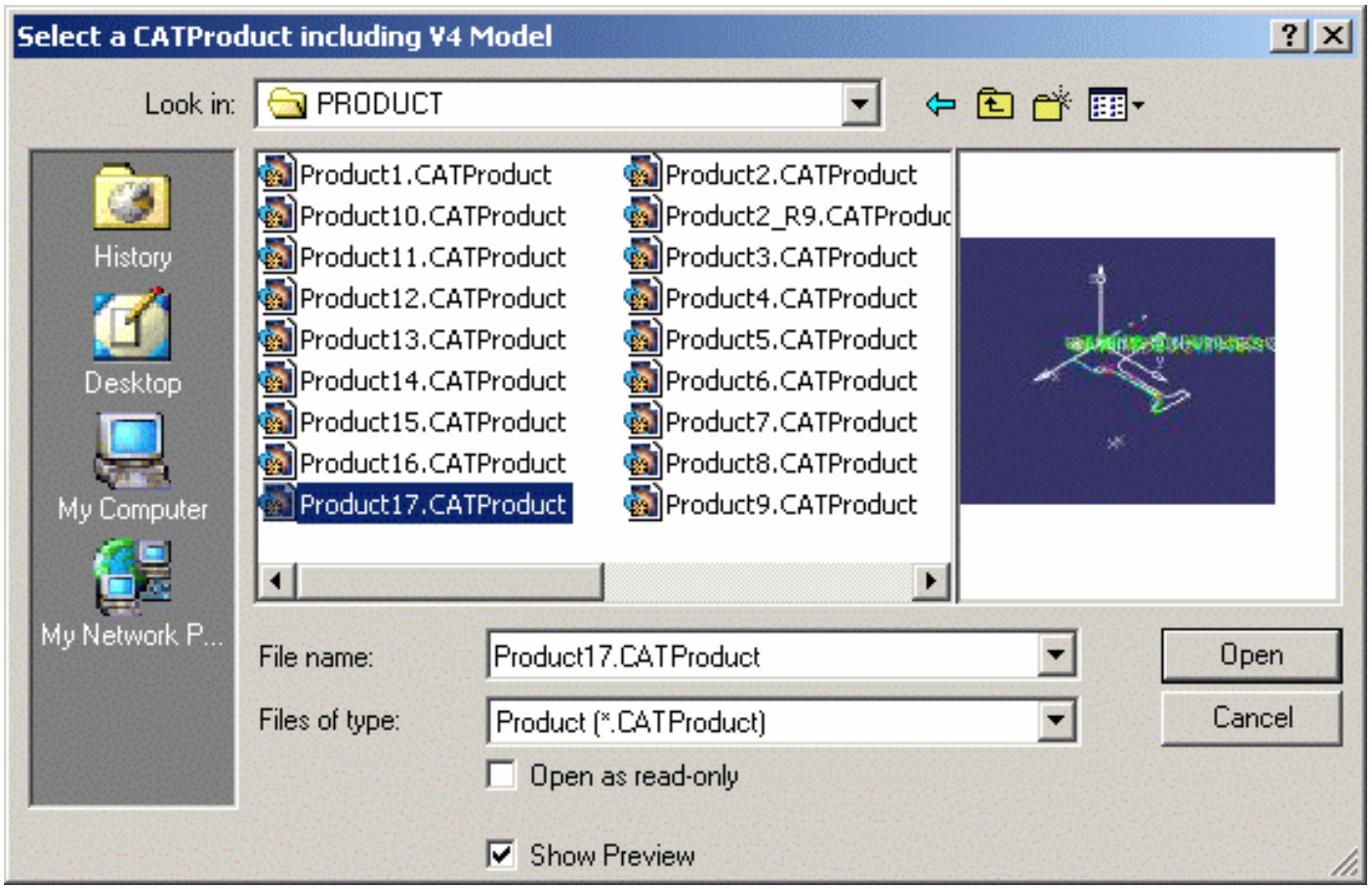
- APT association is not possible for files containing OUTPUT PROFILE, TOOL FLANK OUTPUT, TOOL CENTER OUTPUT, PQR OUTPUT, or NURBS statements.

# Browse a V4 NC Lathe Model

This task shows you how to browse a V4 NC Lathe Model in the NC Manufacturing Review workbench.

For this task you should use a CATProduct that includes one or more V4 models.

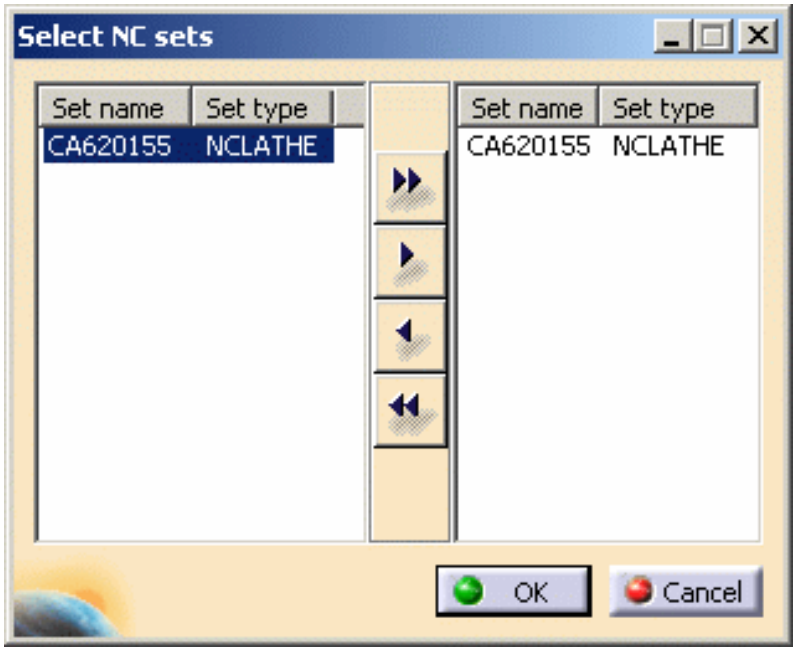
1. Enter the NC Manufacturing Review workbench and click the Read Manufacturing Data from a V4 Model icon . A dialog box like the one shown below appears.




2. Navigate to the desired CATproduct and click Open.
3. If the CATProduct includes more than one V4 model, the following dialog box appears allowing you to select the desired V4 model.

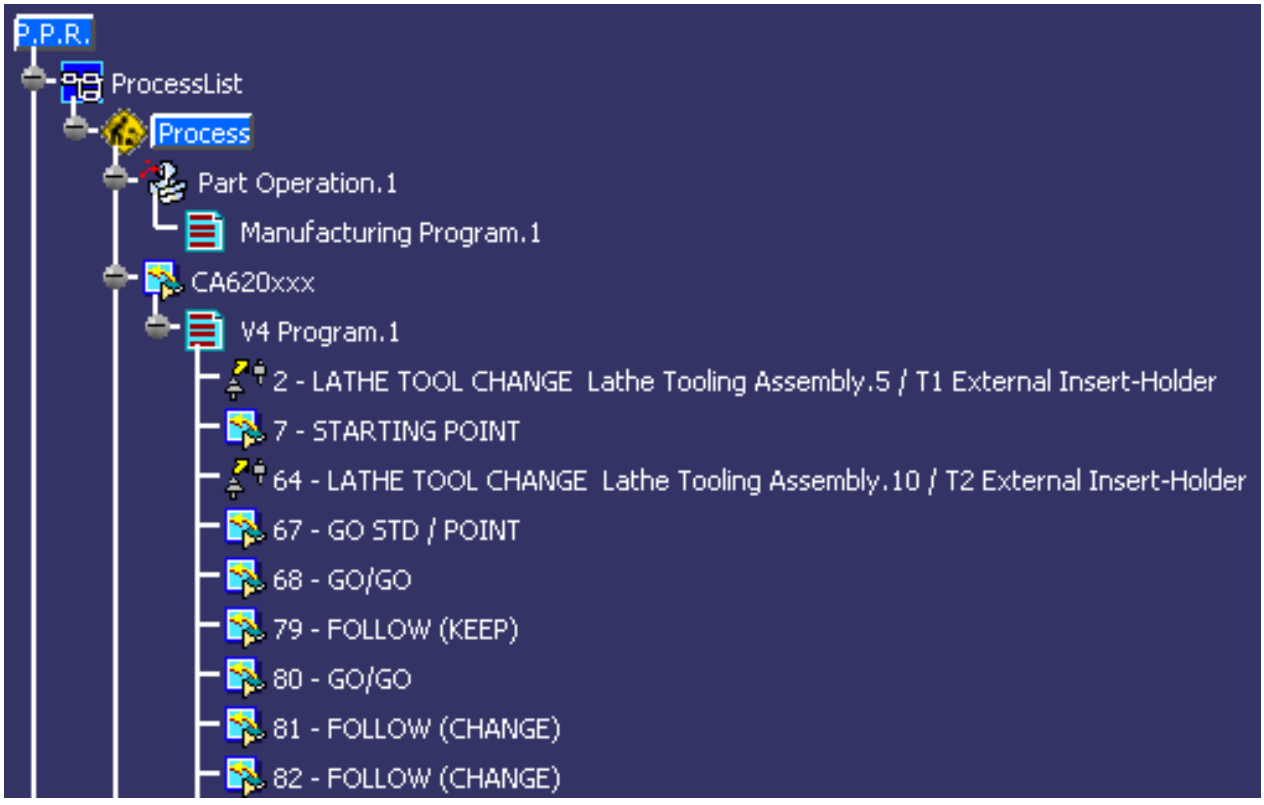
   The Select NC Lathe Sets dialog box appears.

4. Select one or more NC Lathe sets from the list and click OK.




A progression bar appears giving the status of the NC Lathe set and machining operation being processed.

The selected NC Lathe set is added to the Process List at Part Operation level. The PPR tree is updated with the manufacturing information of the selected NC Lathe set.




A program **V4 Program.1** containing all the machining operations and tool changes of the NC Lathe set is added at Manufacturing Program level.

A product **Product1** is created and associated to the selected NC Lathe set.
All tools referenced in the NC Lathe set are added to the Resources List.

5. Double click the Part Operation entity in the PPR tree to access the Part Operation editor. Use this editor:
   - to associate a lathe machine to the Part Operation. In the Machine Editor remember to set the Spindle axis and Radial axis to appropriate values (XY or ZX, for example)
   - to associate a stock to the Part Operation. This is useful if you want to do a Video material removal simulation at a later stage (that is, once NC data has been associated to the V4 program).

6. Double click a machining operation in the program to browse its properties. Please refer to Browse a V4 NC Lathe Operation for more information.

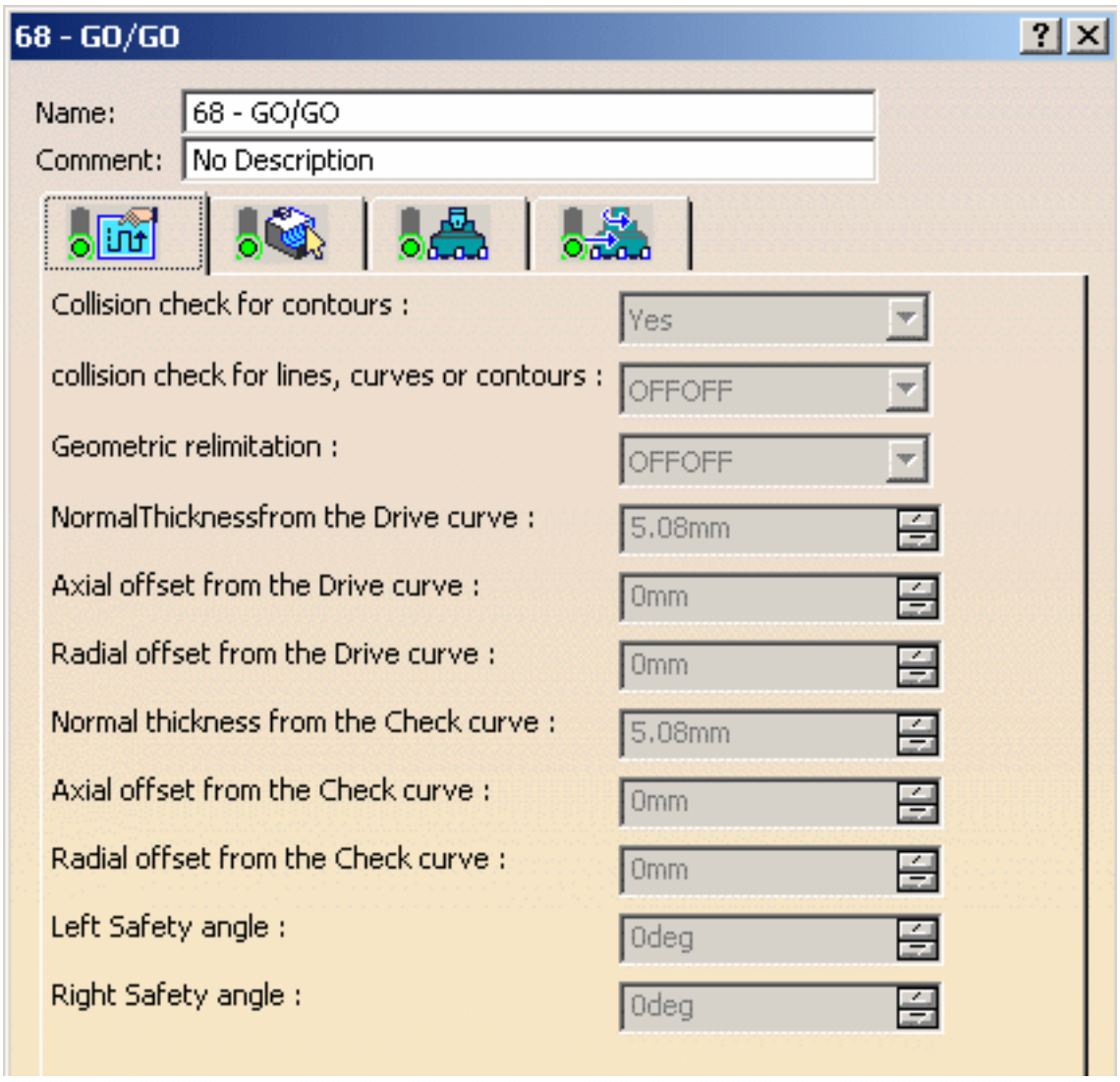
- Only geometry with SHOW status in the V4 model is displayed.
- You can now associate NC data to your V4 program. Please refer to Associate an APT File to an NC Lathe Program for more information.

# Browse a V4 NC Lathe Operation

This task shows you how to browse a V4 NC Lathe Machining Operation in the NC Manufacturing Review workbench.

1. In the specification tree containing a Part Operation originating from V4, double click the machining operation to be browsed (see Browse a V4 NC Lathe Model).

The machining operation browse panel appears.




The properties are organized in tab pages as follows:

the parameters of the machining strategy

the geometry to be machined

the tool that is used in the operation

the feedrates and spindle speeds.

**Parameters of the machining strategy page** 

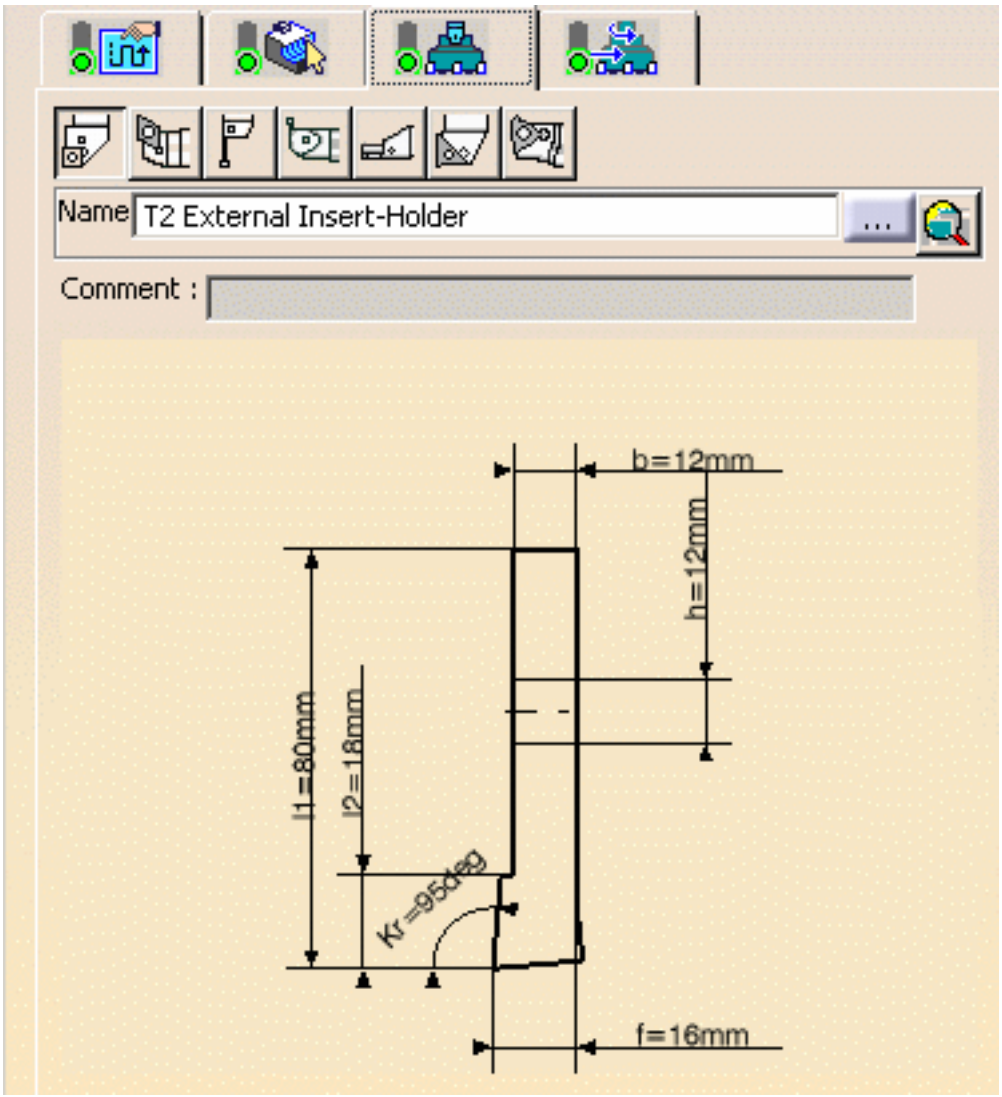
The set of machining options of the NC Lathe machining operation are displayed along with their values.

**Geometry to be machined** 


The set of possible geometry entity types are listed for the operation. Entity types that are not referenced by the selected operation are dimmed. Entity types that are referenced by the selected operation can be highlighted by clicking the corresponding checkbox.

Only geometry with SHOW status in the V4 model is considered to be linked to the operation.

**Tool used in the operation** 




The displayed tool corresponds to the previous Tool Change operation of the NC Lathe set.

If a Detailed tool was used no information is displayed.

**Feedrates**


The set of feedrates and spindle speeds of the NC Lathe machining operation are displayed along with their values.

2. Just click Cancel to close the machining operation panel.

# Associate an APT File to an NC Lathe Program

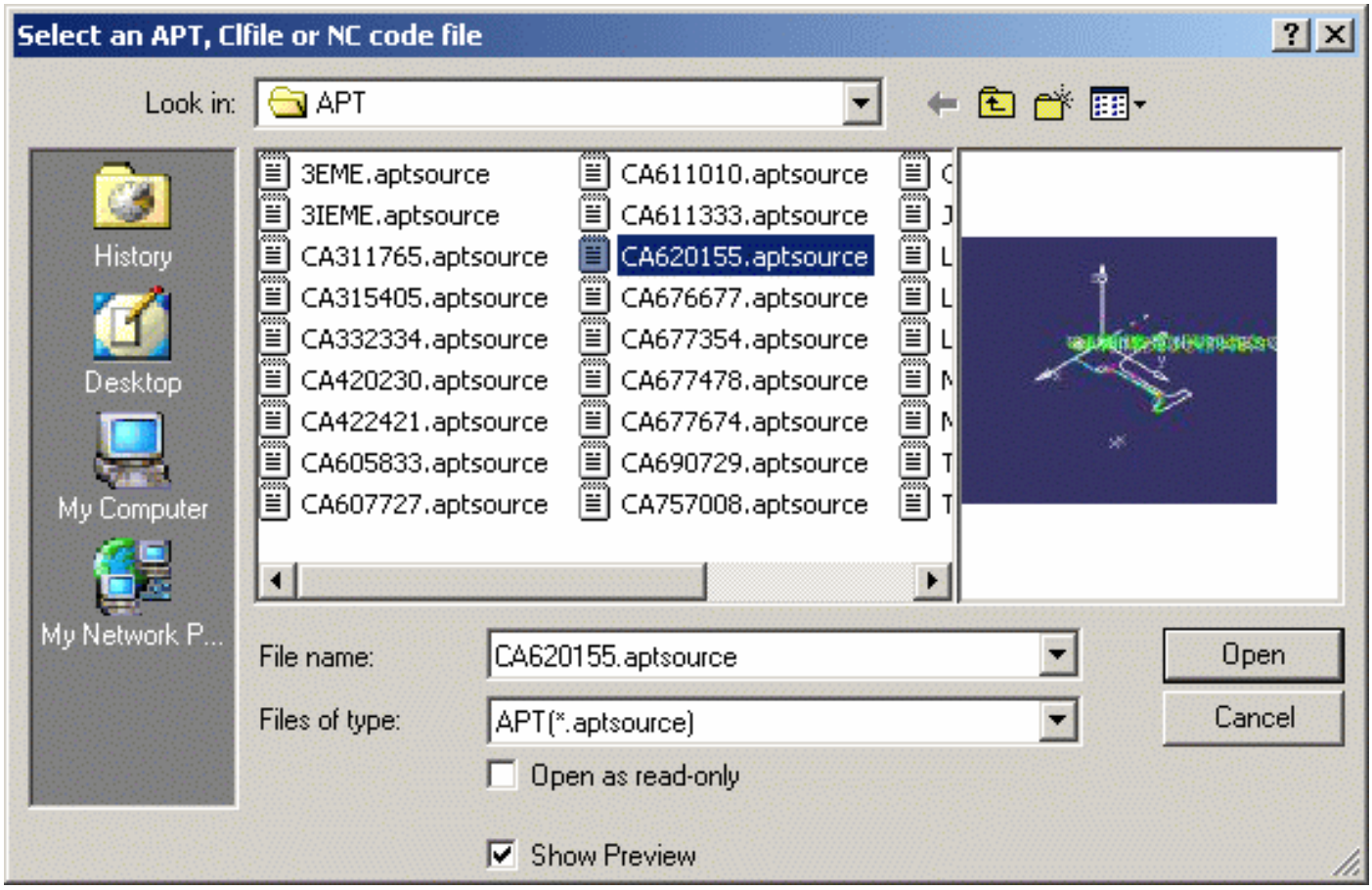
This task shows you how to associate an APT file to an NC Lathe program using the NC Manufacturing Review workbench.

This association consists in creating a tool path for each operation by extracting the necessary information from the APT file. This allows verification and simulation of these operations.

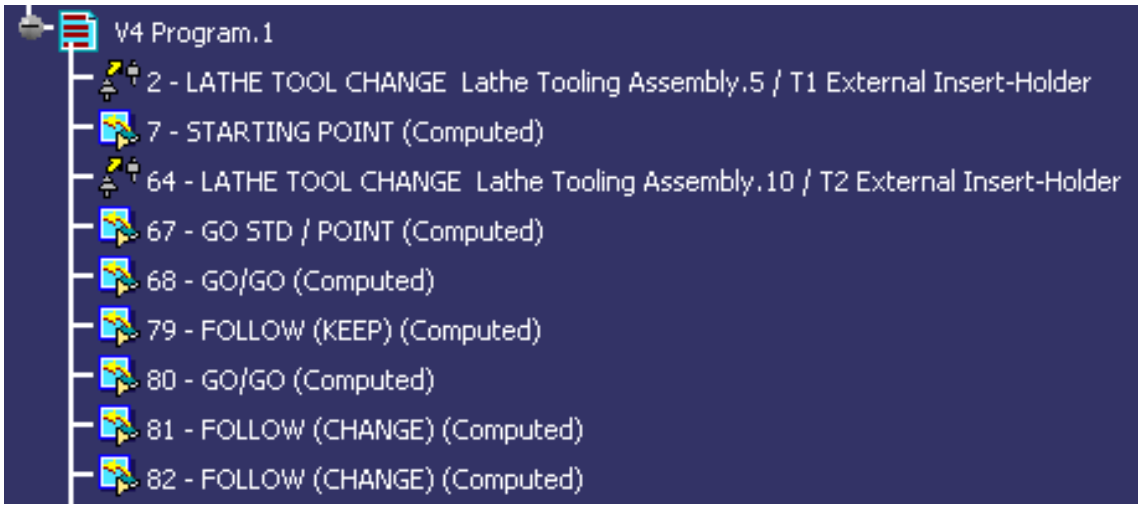
You must have a APT source file generated by the CATLATH utility for the imported NC Lathe program.

Note that CATLATH must be executed with the PPRINT processing option set to PPRINT (that is, *PPR or *PPRINT keyword set to 1).
This is to allow identifying the part of the tool path corresponding to each operation and so to be able to define the tool path of each operation. Identification is processed by means of the PPRINT OPERATION NUMBER statements.

1. Select the V4 program in the specification tree.
2. Select the Associate Output NC File to a Program icon . The dialog box for selecting the NC file appears.
3. Select the NC file to associate to the program.




The tree is updated with tool paths defined on the machining operations.




- The machining operations with **Computed** status have their own tool path which can be replayed and, if a stock geometry is linked to the Part Operation, simulated.
- APT association must be done in a CATIA session working in the same units as those in which APT was generated.
- The APT association can only be done on V4 type programs.
- The APT association can only be done whenever there is a PPRINT OPERATION NUMBER statement and there are as many entities in the program as PPRINT OPERATION NUMBER statements in the APT (except the first one, which defines CATIA0).
- A 'continue' statement followed by a PPRINT OPERATION NUMBER statement may present a problem. Example :

```
PPRINT OPERATION NUMBER : 6
PPRINT *** TOOL NO SF8-1093-10 6.00DIA. .125CRAD. .625LOC. 4.500SL$
PPRINT OPERATION NUMBER : 7
PPRINT ************************************************************
```

You should remove all unnecessary 'continue' statements before APT association.

- APT association is not possible for files containing OUTPUT PROFILE, TOOL FLANK OUTPUT, TOOL CENTER OUTPUT, PQR OUTPUT, or NURBS statements.

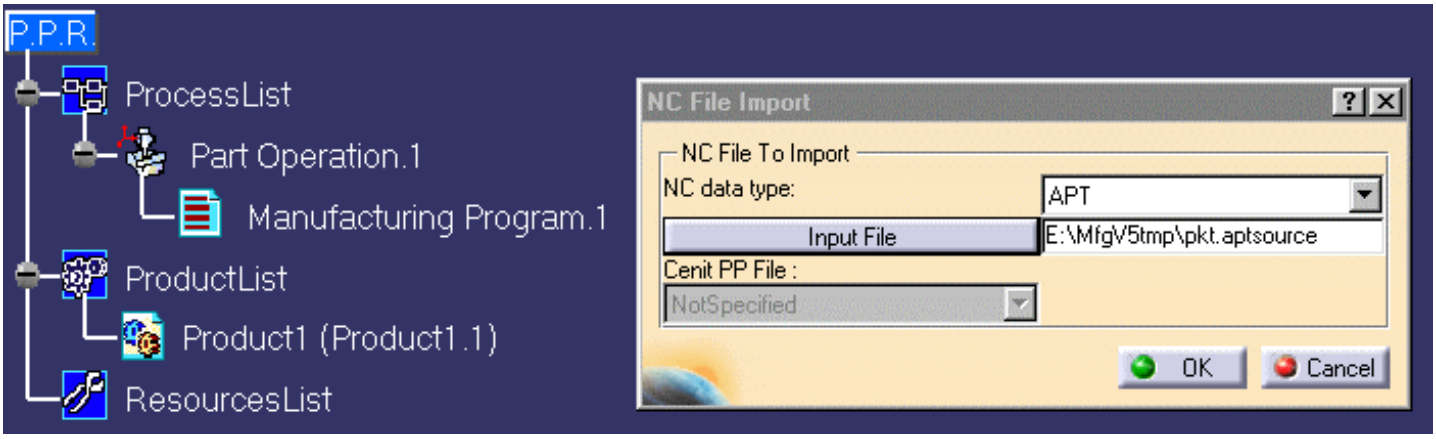
# Import NC Data

This task shows you how to import NC data in the NC Manufacturing Review workbench.

1. Enter the NC Manufacturing Review workbench.

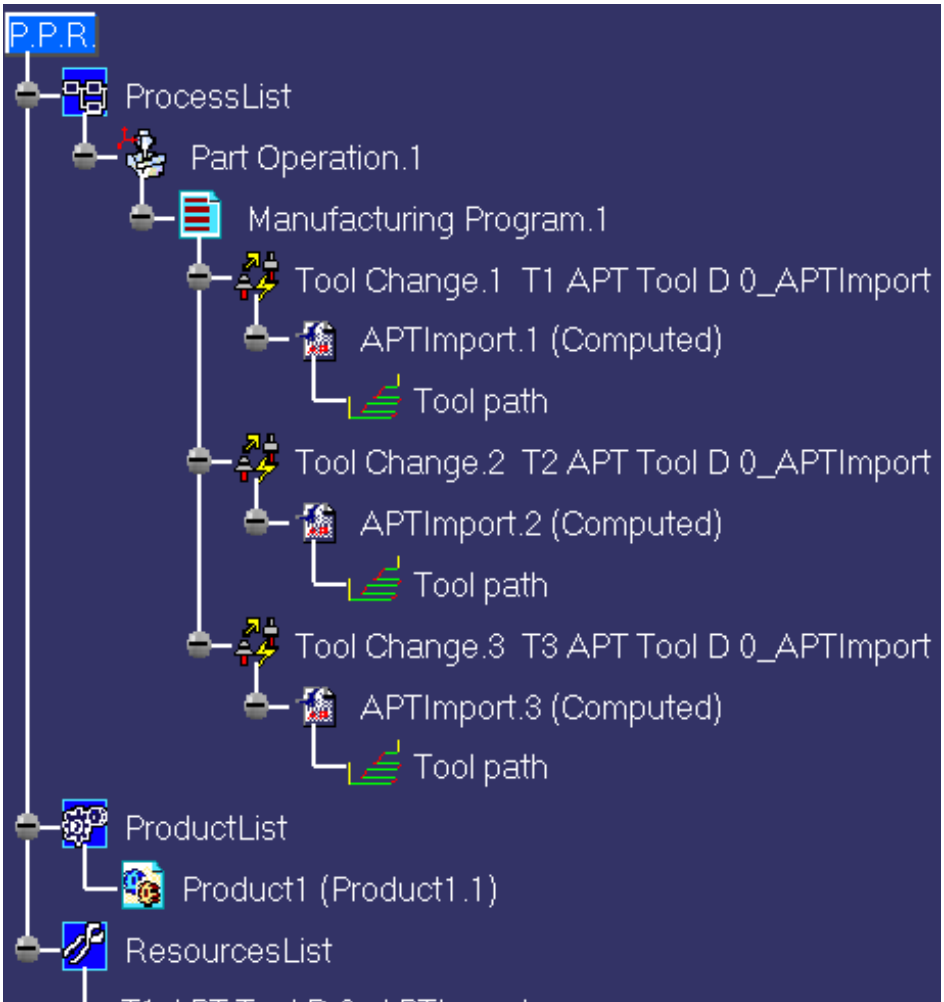
2. Select the Manufacturing Program in the tree.

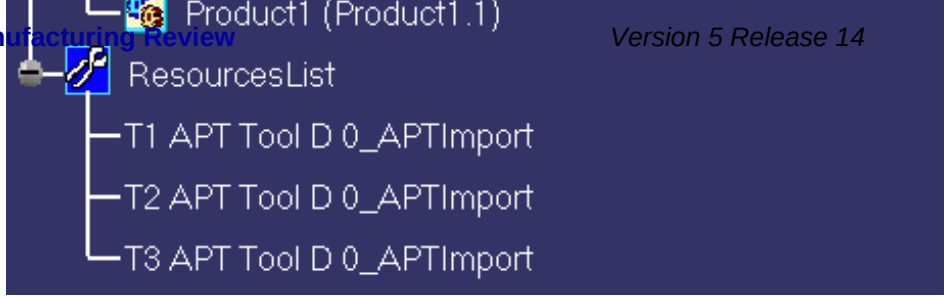
   Click the icon or right-click the program and select **Import APT, Clfile or NC code** from the contextual menu. The NC File Import dialog box appears.




3. Select the type of NC data file to import using the combo:
   - APT
   - Clfile
   - NC code.

4. Click **Input File**.
   In the dialog box that appears navigate to the desired NC data file and click Open.
5. Click OK to import the selected file.
   The PPR tree is updated with the manufacturing information of the imported file.






NC data objects appear in the Manufacturing Program for each tool change in the imported file.
A Tool path object is associated to each NC data object.

All tools referenced in the imported NC file appear in the Resources List.

- If selected NC data type is NC Code, you must specify the Post Processor in the proposed list.

  - Sample Post Processor parameter files are delivered with the product in the folder **\Startup\Manufacturing\PPPar** which provides NC output for various machine types.
    Post Processors are provided by Cenit, Intelligent Manufacturing Software (IMS), and ICAM Technologies Corporation.
  - Select the type of Post Processor parameter files to be made available for selection in the NC Code tab page using Tools > Options > Machining > Output tab. If the output option is set to None, no parameter files will be available for selection.
  - For information about how to acquire Post Processor parameters files that provide machine specific NC code output, please contact your IBM representative.

- If no PP provider is defined in the Tools > Options > Machining > Output tab, the PP File selection combo will not appear in the NC File Import dialog box.
- It is not possible to import APT files containing OUTPUT PROFILE, TOOL FLANK OUTPUT, TOOL CENTER OUTPUT, PQR OUTPUT, or NURBS statements.

# Tool Path Replay and Simulation

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

For a Part Operation originating from V4 NC Mill or NC Lathe, Replay and Simulation is only available whenever a successful APT association has been done.

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 NC Data

The tasks for using capabilities such as production of NC output data are documented in the *NC Manufacturing Infrastructure User's Guide*.

For Version 5 programs, you can generate NC output data when all machining operations have their associated tool path. For operations which have no associated tool path, the applications that are used to create and compute tool paths must be available.

For Version 4 programs issued from NC Mill or NC Lathe, you can generate NC output data when the APT association is done on the program.

- Generate APT Source Code in Batch Mode: Select the Generate NC Output in Batch Mode icon then select the manufacturing program or part operation to be processed and define the APT source processing options.
- Generate Clfile Code in Batch Mode: Select the Generate NC Output in Batch Mode icon then select the manufacturing program or part operation to be processed and define the Clfile processing options.
- Generate NC Code in Batch Mode: Select the Generate NC Output in Batch Mode icon then select the manufacturing program or part operation to be processed and define the NC code processing options.
- Generate a CGR File in Batch Mode: Select the Generate NC Output in Batch Mode icon then select the manufacturing program or part operation to be processed and define the CGR file processing options.

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

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

# Documentation Generation

The tasks for using capabilities such as production of NC documentation are documented in the *NC Manufacturing Infrastructure User's Guide*.

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

Screen capture for associating a JPEG image to an activity (that is, part operation, manufacturing program, machining operation, and so on).

# Edit Tool Path

Before using any of the functions below, the tool path must be computed. All of these functions are accessed via the tool path contextual menu.

**Point Modification**: either move or remove a point on a tool path.

**Area Modification**: edit an area on a tool path.
You can select areas of the tool path by using:

- one point on the tool path and deciding whether you want to use the portion before or after it
- two points and deciding whether you want to use the part of the tool path that is between the two points or outside of the two points
- a contour and deciding whether you want to use the part of the tool path that is inside or outside of the contour
- a polyline and deciding whether you want to use the part of the tool path that is inside or outside of the contour.

**Translation**: apply a Translation transformation to a tool path.

**Rotation**: apply a Rotation transformation to a tool path.

**Mirror**: apply a Mirror transformation to a tool path.

**Reverse**: reverse a tool path.

**Connection**: connect tool paths.

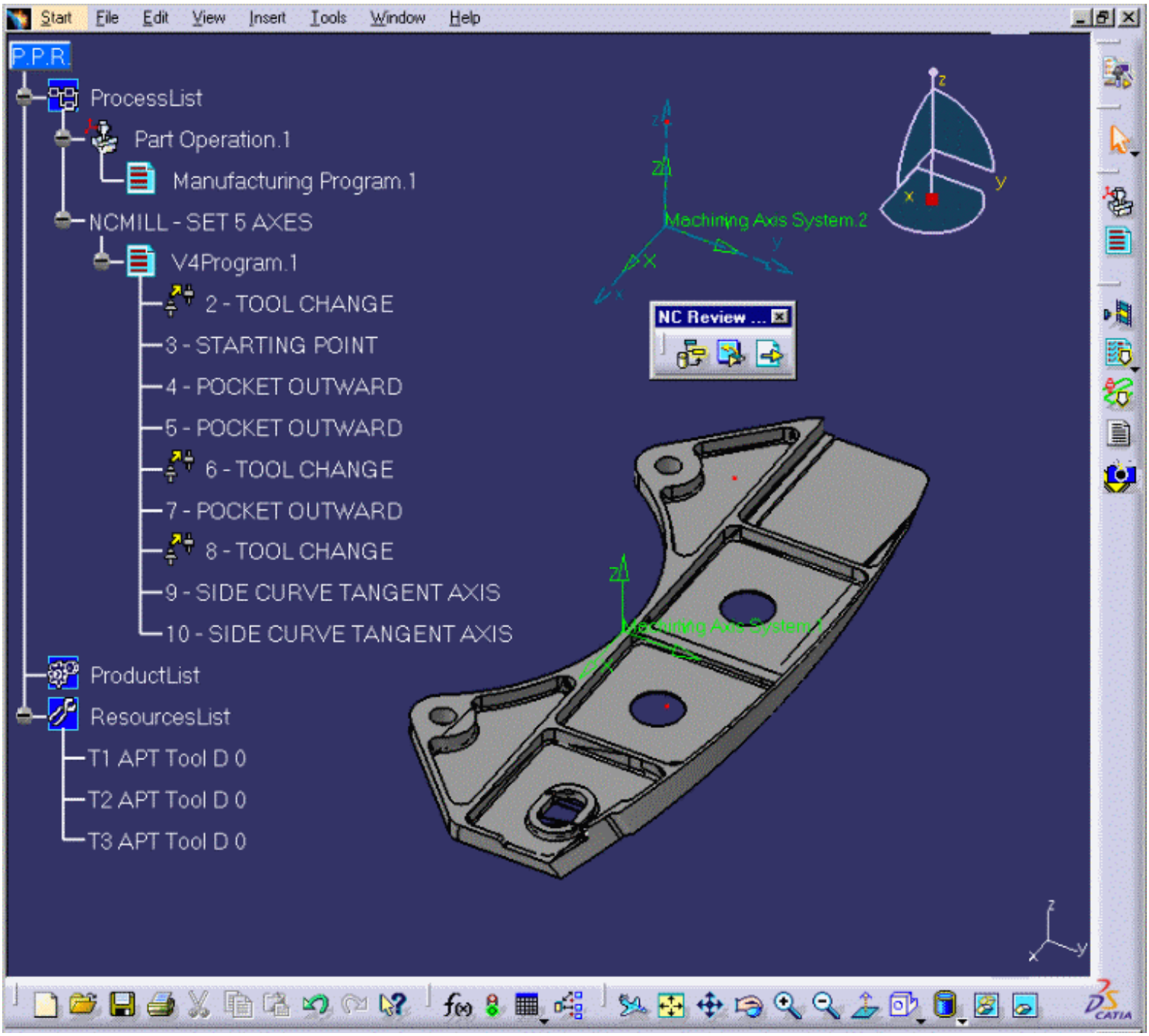
**Change Approach and Retract**: add or remove approaches and retracts in a tool path.

**Check Tool Length**: check a tool path to identify all the points where the tool holder collides with the part.

# Workbench Description

This section contains the description of the commands that are specific to the NC Manufacturing Review workbench, which is illustrated below.

Toolbars
Specification Tree



# NC Manufacturing Review Toolbars

The NC Manufacturing Review workbench includes several icon toolbars.

The **NC Review Management** toolbar is specific to this workbench and contains the following commands for managing the review.



See Import NC Data.


See Browse Version 4 NC Mill model
or Browse Version 4 NC Lathe model.


See Associate APT to an NC Mill program
or Associate APT to an NC Lathe program.

The other toolbars are common to all NC workbenches. The commands of these toolbars are documented in the *NC Manufacturing Infrastructure User's Guide*.

The **Manufacturing Program** toolbar contains the following commands for creating part operation and manufacturing program entities.



See Part Operation


See Manufacturing Program

The **NC Output Management** toolbar contains the following commands for reviewing different types of NC output.



See Tool Path Replay and Simulation


See Tool Path Computation in Batch Mode


See Tool Path Computation in Interactive Mode


See Batch Queue Management

See Documentation Generation

Screen capture for associating a JPEG image to an activity (that is, part operation, manufacturing program, machining operation, and so on).

During NC Documentation generation, an IDL interface can be called in order to read the image associated to the activity. A URL link must be made from the HTML documentation to the image in order to retrieve it in the generated HTML documentation.

The **Auxiliary Commands** toolbar contains a number of commands for various tasks.


Import Tools.
See procedure for searching tools described in Select or Create a Tool.

See Replace Tools

Choice between No Display, Displaying Tool Path or Display Geometry of selected operation.

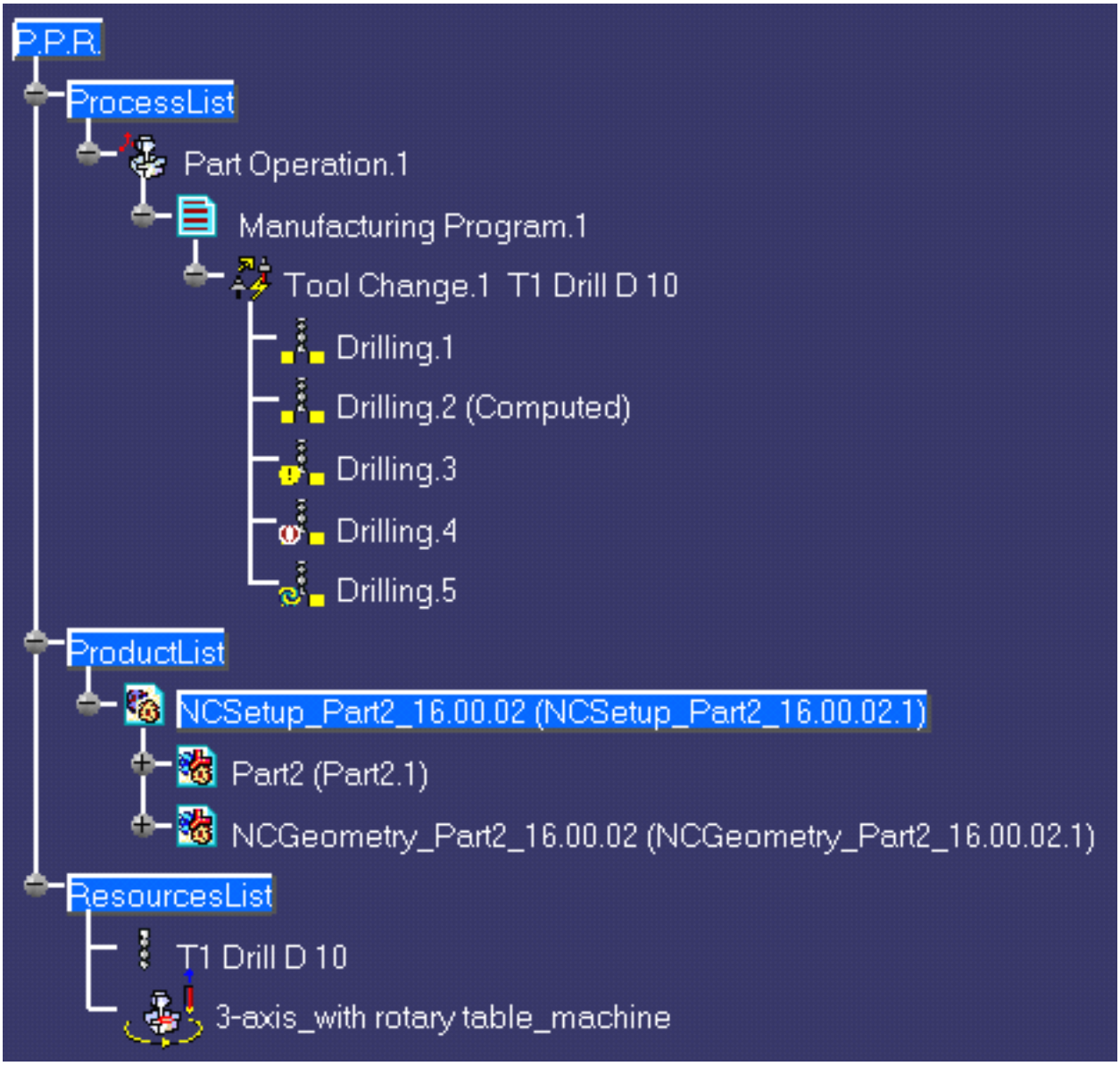
Allows manual update of activity status in tree.

The **Machining Features** toolbar gives access to the Manufacturing View.

See Manufacturing View.

# Specification Tree

Here is an example of a Process Product Resources (PPR) specification tree for NC Manufacturing Review.




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

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

- Drilling.4 has been deactivated by the user (indicated by the brackets symbol)

- Drilling.5 has been modified and needs to be recomputed (indicated by the update symbol).

**Product List** gives all of the parts to machine as well as CATPart documents containing complementary geometry.
**Resources List** gives all of the resources such as machine or tools that can be used in the program.

# Customizing

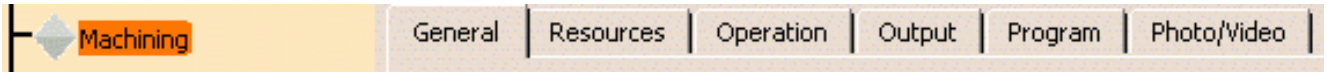
This section describes how to customize settings for Machining.

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

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

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

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



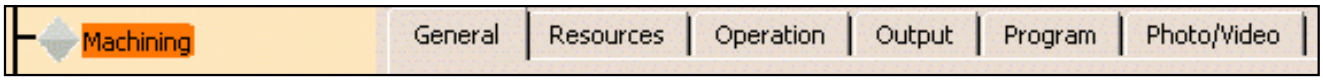

3. Select the tab corresponding to the parameters to be customized.

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

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

# General

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

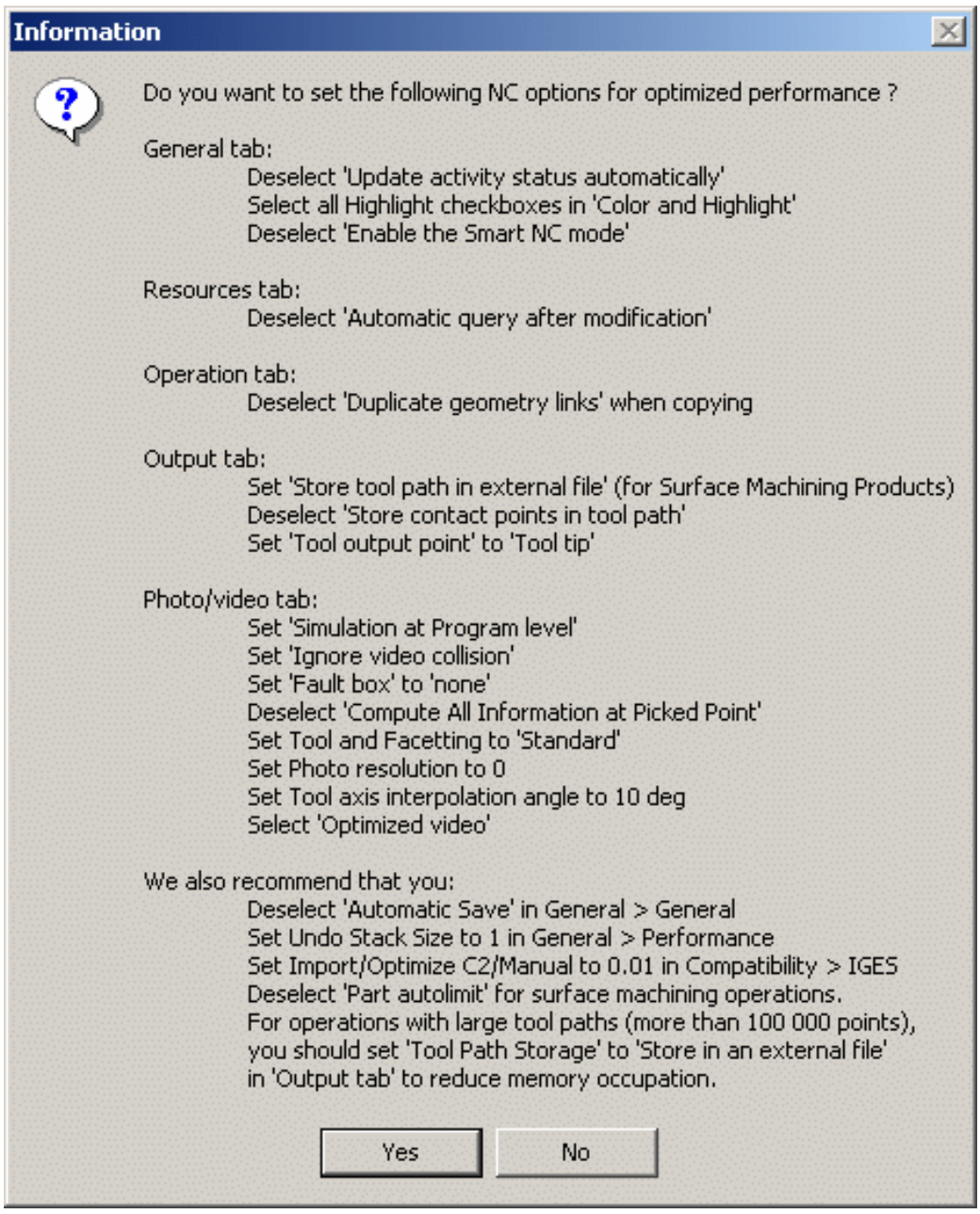



Select the **General** tab, which is divided up into areas.

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

## Performance

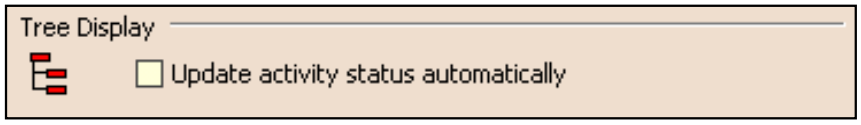

Click the **Optimize** button in order to automatically set a number of the Machining options for optimized performance. These options are listed in the Information dialog box that appears:




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

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

## Tree Display

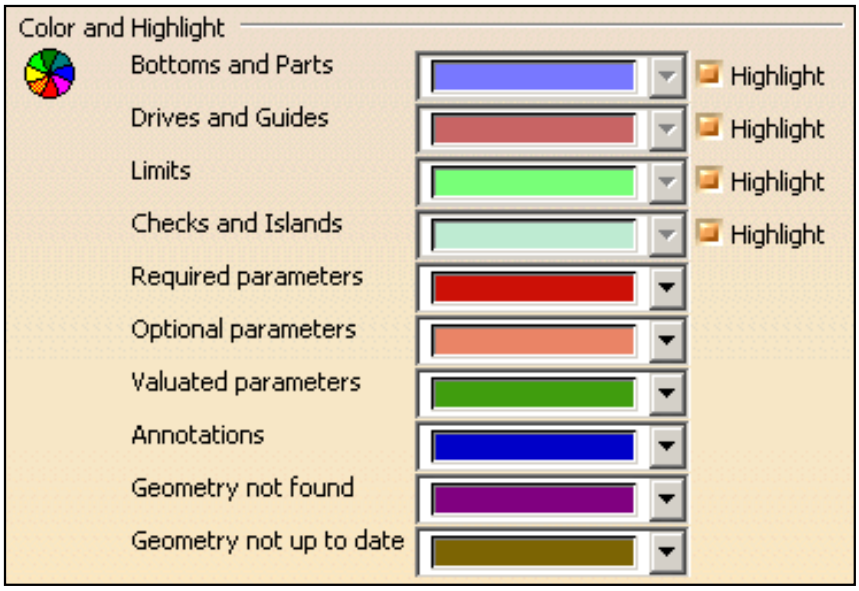



- Select the checkbox if you want the status of activities in the tree to be updated automatically.
- If this checkbox is not selected:
  - you can update activity status manually in your workbench using the Update Status icon  in the Auxiliary Commands toolbar.
  - the status of the activity after a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

If this checkbox is not selected, performance is improved.

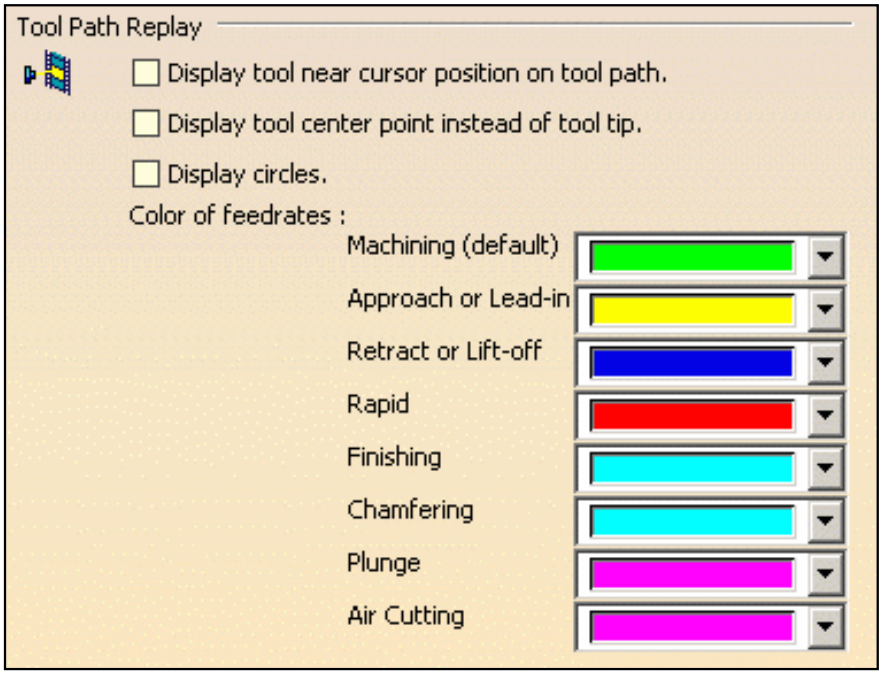
By default, the checkbox is not selected.

## Color and Highlight




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

## Tool Path Replay




### Display tool near cursor position on tool path

Select this checkbox if you want to display the tool near your cursor position on the trajectory during a tool path replay

### Display tool center instead of tool tip

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

### Display circles

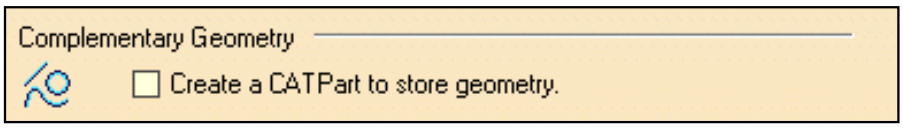
Select this checkbox if you want to display each circular trajectory as a circular arc instead of a set of discretization points. The extremities of the circular arc are indicated by means of 'O' symbols.
This allows better control of the Point by Point replay mode, where it is necessary to make several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

By default, these checkboxes are not selected.

### Color of feedrates

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

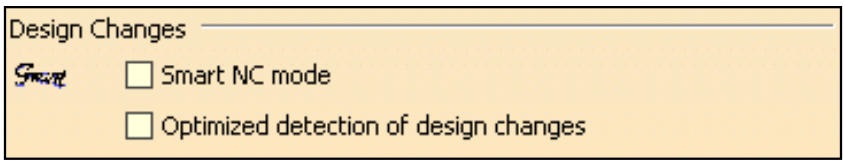
## Complementary Geometry




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

By default, the checkbox is not selected.

## Design Changes




### Smart NC mode

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

### Optimized detection of design changes

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

By default, these checkboxes are not selected.

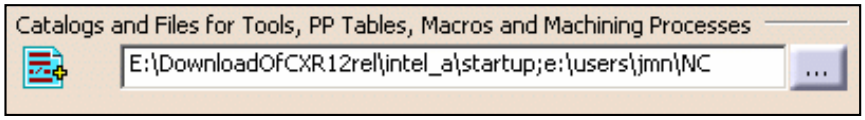
# Resources

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


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

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

## Catalogs and Files




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

You can concatenate paths using:

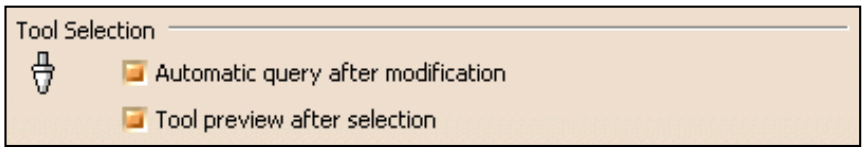
- a semi colon (;) character for Windows NT platforms.
- a colon (:) character for UNIX platform.

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

Please note that:

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

## Tool Selection




### Automatic query after modification

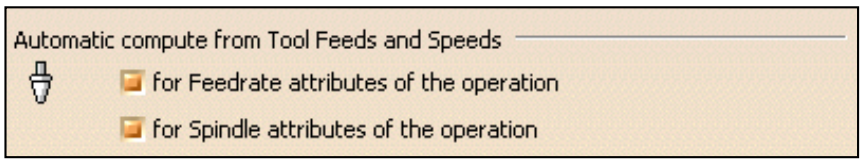
Select this checkbox if you want to to activate an automatic query after each modification of a tool parameter. Performance is improved when this checkbox is not selected.

### Tool preview after selection

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

By default, these checkboxes are selected.

# Automatic Compute from Tool Feeds and Speeds




## Feedrate attributes of the operation

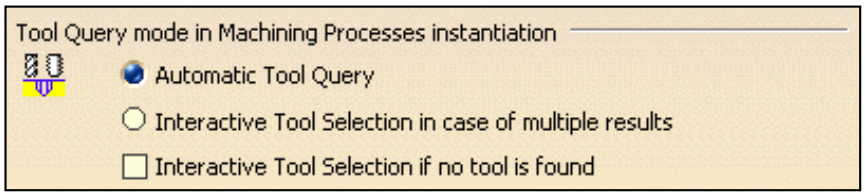
Select this checkbox if you want the Automatic Update of Feedrates option to be set by default in the Feeds and Speeds tab page of machining operations.
This option allows feedrates of operations to be automatically updated whenever feedrate information on the tool is modified.

## Spindle attributes of the operation

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

By default, these checkboxes are selected.

# Tool Query mode in Machining Processes Instantiation

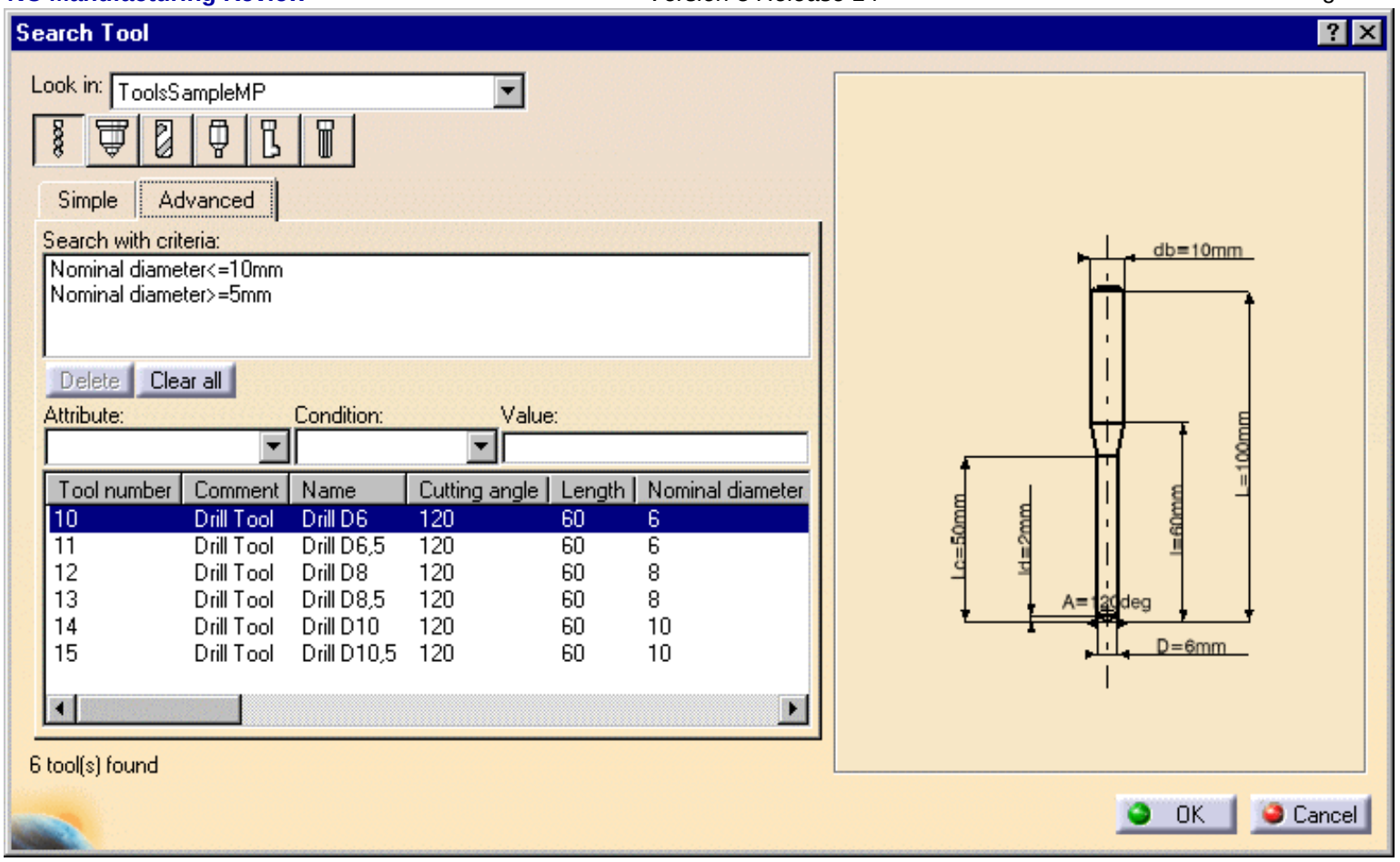



Select the type of Tool Query to be executed when a Machining Process is instantiated:

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

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

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


Search Tool
Look in:
ToolsSampleMP
Simple
Advanced
Search with criteria:
Nominal diameter<=10mm
Nominal diameter>=5mm
Delete
Clear all
Attribute:
Condition:
Value:
Tool number
Comment
Name
Cutting angle
Length
Nominal diameter
10 Drill Tool Drill D6 120 60 6
11 Drill Tool Drill D6,5 120 60 6
12 Drill Tool Drill D8 120 60 8
13 Drill Tool Drill D8,5 120 60 8
14 Drill Tool Drill D10 120 60 10
15 Drill Tool Drill D10,5 120 60 10
6 tool(s) found
db=10mm
L=100mm
l=60mm
Lc=50mm
ld=2mm
A=120deg
D=6mm
OK
Cancel

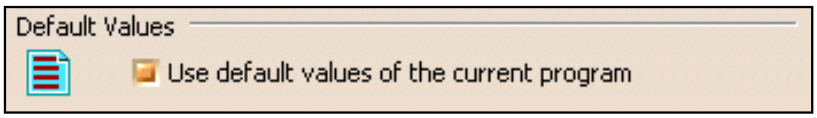
# Operation

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


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

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

## Default Values

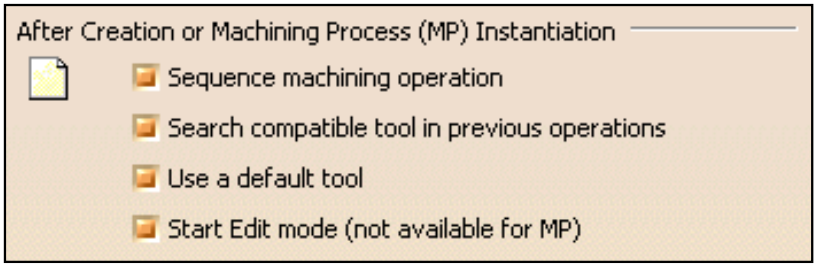



Select the checkbox if you want operations to be created with the values used in the current program.
The values and units of attributes at the creation step of an operation are set to the values and units of the last edited and validated operation whatever its type (that is, exit the operation definition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

Otherwise the default settings delivered with the application are used.

By default, this checkbox is selected.

## After Creation or Machining Process (MP) Instantiation




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

### Sequence machining operation

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

### Search compatible tool in previous operations

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

### Use a default tool

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

### Start edit mode (not available for machining processes)

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

By default, these checkboxes are selected.

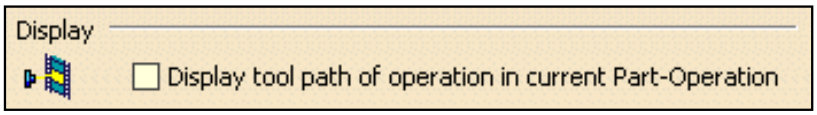
## When Copying


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

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

By default, this checkbox is selected.

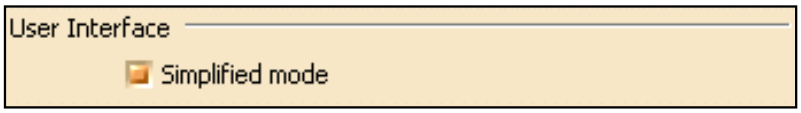
## Display




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

By default, this checkbox is not selected.

## User Interface




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

By default, this checkbox is not selected.

# Output

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


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

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

## Post Processor

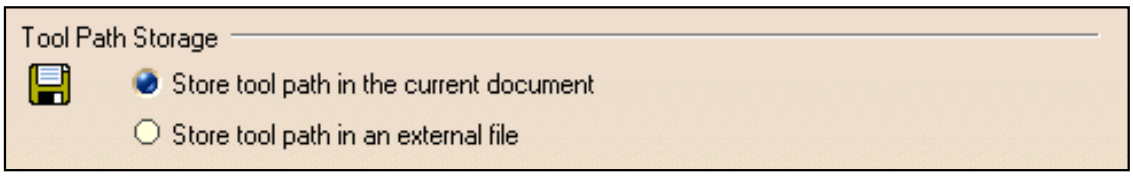

Select the desired Processor option:

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

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

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

## Tool Path Storage

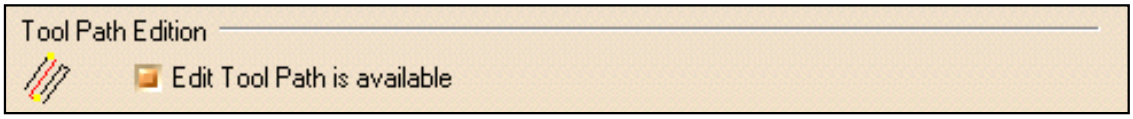



Select the desired option to store tool path data either in the current document or in an external file (as a tpl file).

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

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

## Tool Path Edition

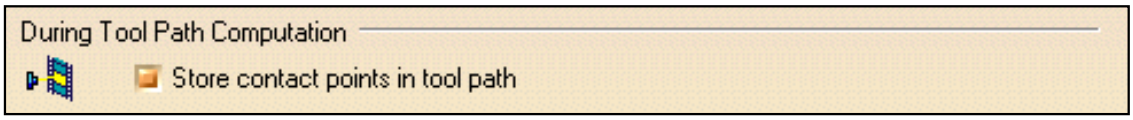



Select the checkbox if you want to be able to edit tool paths even when the operation is locked.

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

By default, this checkbox is selected.

## During Tool Path Computation

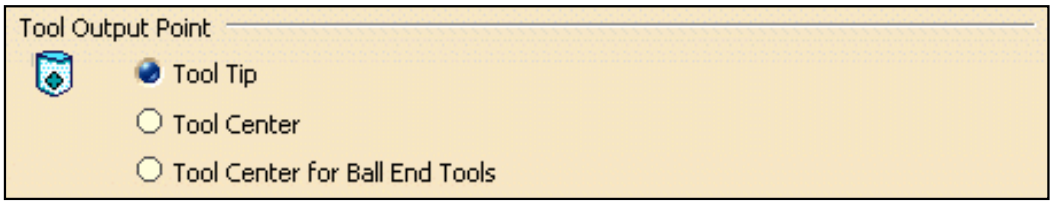



Select the checkbox if you want to store contact points in the tool path.

Performance is improved when this checkbox is not selected.

By default, this checkbox is selected.

## Tool Output Point



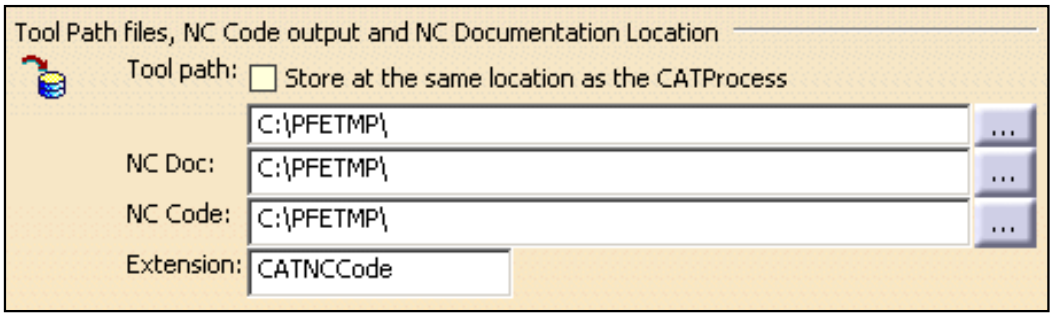

Select the desired option to select one of the following as output point:

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

Performance is better when the Tool Tip option is selected.

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

## Default File Locations




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

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

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

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

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

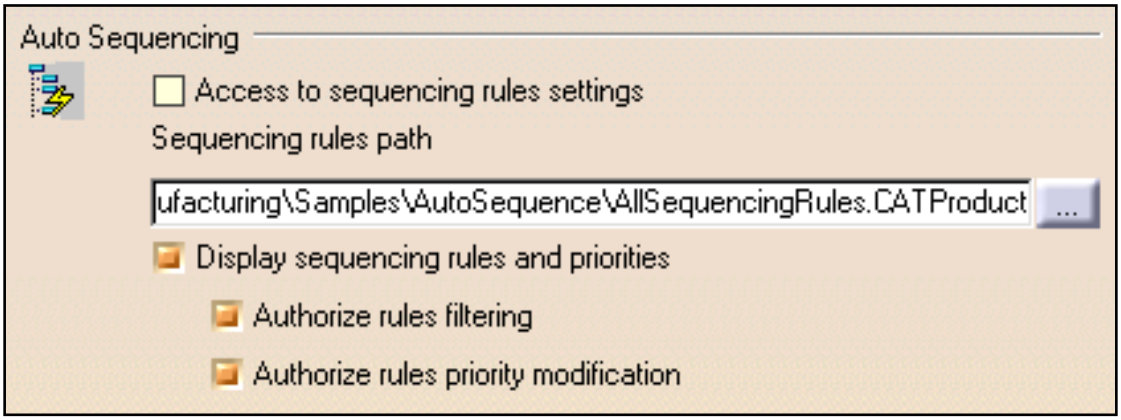
# Program

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


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

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

## Auto Sequencing




### Access to sequencing rules settings

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

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

By default, this checkbox is selected.

### Display sequencing rules and priorities

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

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

By default, these checkboxes are selected.

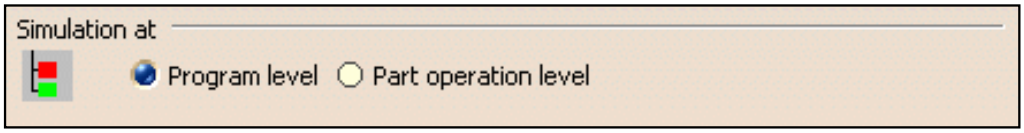
# Photo/Video

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


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

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

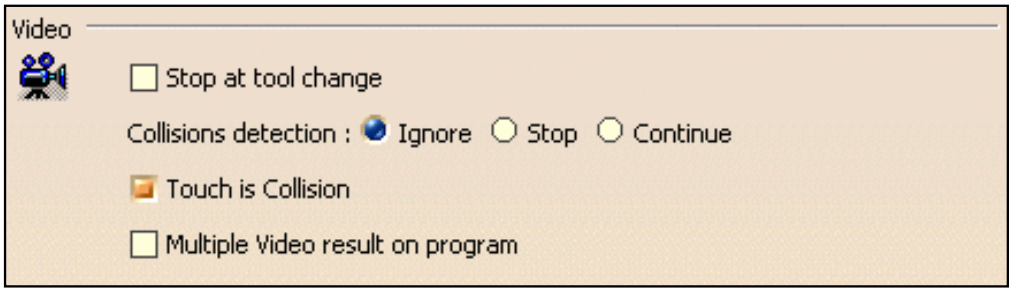
## Simulation at




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

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

## Video




### Stop at tool change

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

By default, this checkbox is not selected.

### Collision detection

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

- ignore collisions during the Video simulation

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

Best performance is obtained when collisions are ignored.

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

### Touch is collision

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

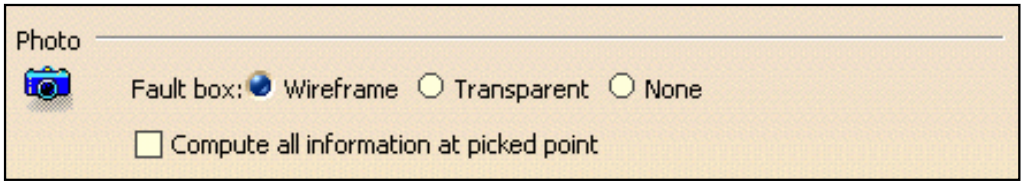
By default, this checkbox is selected.

### Multiple Video result on program

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

By default, this checkbox is not selected.

## Photo




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

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

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

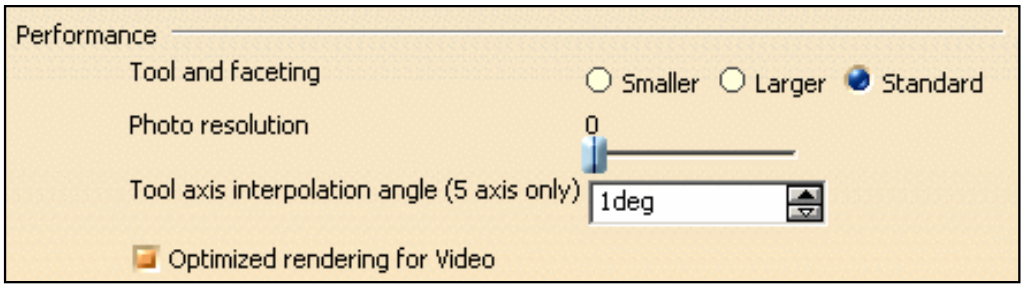
By default, the **Wireframe** option is selected.

Select the checkbox to compute all information at picked point.

By default, this checkbox is not selected.

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

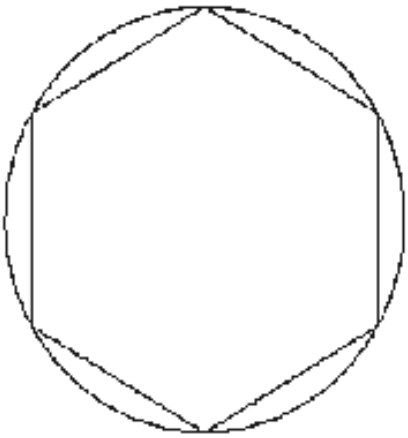
## Performance




## Tool and faceting

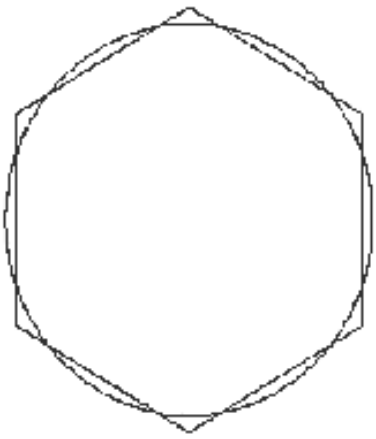
There are three methods of tool faceting used in Video simulation: Standard, Smaller and Larger.
The number of facets for a tool representation is determined by the chord deviation that is set for the tool diameter (0.005% of the tool diameter).

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



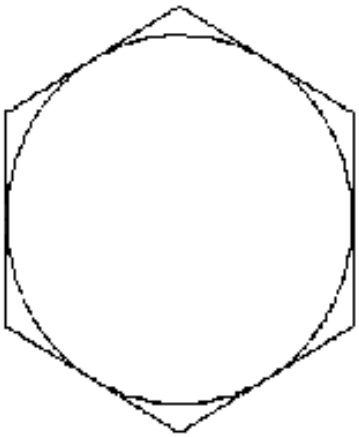
This is the most accurate method for the Arc through Three Points command.

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



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

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



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

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

## Photo resolution

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

By default, this resolution is set to 0.

## Tool axis interpolation angle (5-axis only)

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

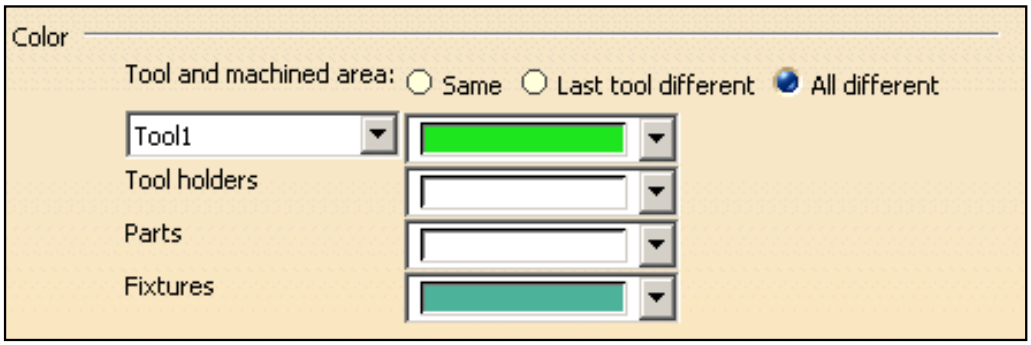
By default, this angle is set to 1degree.

## Optimized rendering for Video

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

By default, this checkbox is selected.

# Color



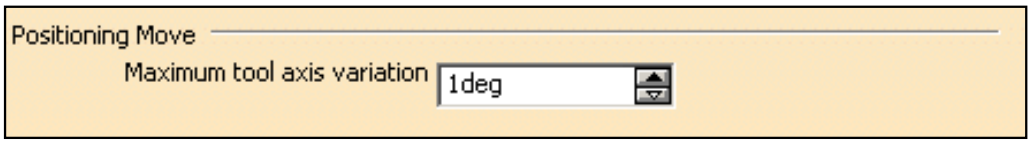

Set the tool (and associated machined area) color to be the same as or different from the last tool, or have different colors for all tools. Best performance is obtained with same colored tools.

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

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

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

## Positioning Move




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

By default, this angle is set to 1degree.

# Reference Information

Reference information on the following topic is provided in this guide.

NC Review of Copy/Index Operations

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

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

# NC Review of Copy/Index/Tracut Operations

In general the V4 customer's process is to use the CATIA V4 utility (CATNC) to obtain the APT source file, which is then processed by the APT processor. The INDEX, COPY, TRACUT statements as well as the APT macros are translated by the APT processor. The result is a clfile which is then processed by the Post Processor to get machine code.

This process is not modified when using CATIA V5 to convert CATIA V4 NC programs.

The methodology to get an APT source file allowing a successful association to the program read in NC Review, requires that COPY and TRACUT operations are not treated by the V4 CATNC utility, so that the corresponding statements remain in the APT source file.

In V5 the customer has the ability to perform replay of V4 Index/Copy operations, and to generate INDEX and COPY statements in the APT obtained using the V5 MfgBatch utility.

Note that many V4 NC MILL programs use Index/Copy features and, in some cases, multi-level indexes.

## Translation of V4 Copy Entities into V5 Entities

When reading V4 programs in NC Review, each time a Copy operation is treated, it is replaced by a COPY operator entity. This COPY operator entity is automatically created.

The possible cases for COPY statements in V4 are as follows:

```
COPY/n,SAME,nb
COPY/n,TRANSL,dx,dy,dz,nb
COPY/n,XYROT,angle,nb
COPY/n,YZROT,angle,nb
COPY/n,ZXROT,angle,nb
COPY/n,SCALE,factor,nb
COPY/n,MODIFY, $
(MATRIX/ ...),nb
```

In all cases, the Copy number (n), which is a kind of identifier, and the number of copies (nb) are stored on the Copy operator entity.

A specific conversion rule is defined for each of the COPY cases and is described below.

**Copy operation type: SAME**

This type of Copy is transformed into COPY operator with:
Transformation type: Matrix (identity)
In Absolute Machining Axis System

**Copy operation type: TRANSL**

This type of Copy is transformed into COPY operator with:
Transformation type: Translation
Translation type: Absolute coordinates
Distance along X, Y, Z: values of the components along each axis.

**Copy operation type: XYROT, YZROT, ZXROT**

This type of Copy is transformed into COPY operator with:

Transformation type: Rotation
Rotation type: Machining axis (current machining axis to be selected)
Angle: rotation angle in degrees.

**Copy operation type: SCALE**

This type of Copy is transformed into COPY operator with:

Transformation type: Scale
Machining Axis System: Current Machining Axis System to be selected
Scale factor: scale factor to be applied along the 3 vectors.

**Copy operation type: MODIFY MATRIX transformation**

This type of Copy is mapped into a Matrix type of COPY operator in Absolute Machining Axis System.

# INDEX and INDEX/NOMORE Entities

V4 INDEX and INDEX/NOMORE entities are converted into V5 INDEX and INDEX/NOMORE activities, which are considered as markers in the program: they delimit the copied part of the program in the tree.

# Translation of V4 Tracut Entities into V5 Entities

When reading V4 programs in NC Review, each time a Tracut operation is treated, it is replaced by a TRACUT operator entity. This TRACUT operator entity is automatically created.

The possible cases for TRACUT statements in V4 are as follows:

TRACUT/(MATRIX,TRANSL,dx,dy,dz)
TRACUT/(MATRIX,XYROT,angle)
TRACUT/(MATRIX,YZROT,angle)
TRACUT/(MATRIX,ZXROT,angle)
TRACUT/(MATRIX,SCALE,factor)
TRACUT/(MATRIX/ matrix transformation components)

A specific conversion rule is defined for each of the TRACUT cases and is described below.

**Tracut operation type: TRANSL**

This type of Tracut is transformed into TRACUT operator with:

Transformation type: Translation
Translation type: Absolute coordinates
Distance along X, Y, Z: values of the components along each axis.

**Tracut operation type: XYROT, YZROT, ZXROT**

This type of Tracut is transformed into TRACUT operator with:

Transformation type: Rotation
Rotation type: Machining axis (current machining axis to be selected)
Angle: rotation angle in degrees.

**Tracut operation type: SCALE**

This type of Tracut is transformed into TRACUT operator with:

Transformation type: Scale
Machining Axis System: Current Machining Axis System to be selected
Scale factor: scale factor to be applied along the 3 vectors.

**Tracut operation type: MATRIX transformation**

This type of Tracut is mapped into a Matrix type of TRACUT operator in Absolute Machining Axis System.

# TRACUT/NOMORE Entity

V4 TRACUT/NOMORE entities are converted into V5 TRACUT/NOMORE activities, which are considered as markers in the program: they delimit the transformed part of the program in the tree.

# Replay and APT Generation

V4 INDEX, INDEX/NOMORE and TRACUT/NOMORE entities have no replay. During APT generation, they allow INDEX, INDEX/NOMORE and TRACUT/NOMORE statements to be generated.

As V4 COPY and TRACUT entities are converted into COPY and TRACUT operators, their replay can be done.

During APT generation, COPY and TRACUT entity processing defines a COPY or TRACUT statement, or the tool path which has been displayed during Replay, depending on the MfgBatch utility settings.

# Methodology

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

Machining Processes
Knowledgeware in Machining Processes
CATProduct and CATProcess Document Management
Design Changes and Associativity Mechanisms
Part Operation and Set Up Documents
Opposite Hand Machining
User Features for NC Manufacturing

# Glossary

A B C D E F G H I L M O P R S T U Z

## A

**approach macro** Motion defined for approaching the operation start point

**auxiliary command** A control function such as tool change or machine table rotation. These commands may be interpreted by a specific post-processor.

**axial machining operation** Operation in which machining is done along a single axis and is mainly intended for hole making (drilling, counter boring, and so on).

## B

**back and forth** Machining in which motion is done alternately in one direction then the other. Compare with one way.

**bottom plane** A planar geometric element that represents the bottom surface of an area to machine. It is normal to the tool axis.

## C

**clearance macro** Motion that involves retracting to a safety plane, a linear trajectory in that plane and then plunging from that plane.

**climb milling** Milling in which the advancing tool rotates down into the material. Chips of cut material tend to be thrown behind the tool, which results to give good surface finish. Compare with conventional milling.

**conventional milling** Milling in which the advancing tool rotates up into the material. Chips of cut material tend to be carried around with the tool, which often impairs good surface finish. Compare with climb milling.

## D

**DPM** Digital Process for Manufacturing.

## E

**extension type** Defines the end type of a hole as being through hole or blind.

## F

**Facing operation** A surfacing operation in which material is removed in one cut or several axial cuts of equal depth according to a pre-defined machining strategy. Boundaries of the planar area to be machined are *soft*.

**Fault** Types of faults in material removal simulation are gouge, undercut, and tool clash.

**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.

## H

**hard** A geometric element (such as a boundary or a bottom face) that the tool cannot pass beyond.

**high speed milling (HSM)** Functionality that is supported for Pocketing and Facing operations in which corners and transitions in the tool path are rounded to ensure a smooth and continuous cutting effort.

## I

**inward helical** Machining in which motion starts from a point inside the domain to machine and follows paths parallel to the domain boundary towards the center of the domain. Compare with outward helical.

**island** Inner domain of a pocket that is to be avoided during machining. It has a closed hard boundary.

## L

**linking motion** Motion that involves retracting to a safety plane, a linear trajectory in that plane and then plunging from that plane.

## 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 feature** A feature instance representing a volume of material to be removed, a machining axis, tolerances, and other technological attributes. These features may be hole type or milling type.

**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.

**manufacturing view** The set of machining features defined in the part operation.

**multi-level operation** Milling operation (such as Pocketing or Profile Contouring) that is done in a series of axial cuts.

## 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 or back and forth.

**outward helical** Machining in which motion starts from a point inside the domain to machine and follows paths parallel to the domain boundary away from the center of the domain. Compare with inward helical.

## 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.

**pocket** An area to be machined that is defined by a closed boundary and a bottom plane. The pocket boundary may be either open or closed. The pocket definition may also include a top plane and one or more islands.

**Pocketing operation** A machining operation in which material is removed from a pocket in one or several axial cuts of equal depth according to a pre-defined machining strategy. The tool path style is either Inward helical, Outward helical or Back and forth.

**Point to Point operation** A milling operation in which the tool moves in straight line segments between user-defined points.

**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.

**Profile Contouring operation** A milling operation in which the tool follows a guide curve and possibly other guide elements while respecting user-defined geometric limitations and machining strategy parameters.

## R

**retract macro** Motion defined for retracting from the operation end point

**return macro** Motion for linking between paths or between levels. It involves retracting to a safety plane, a linear trajectory in that plane and then plunging from that plane.

# 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.

**soft** A geometric element (such as a boundary or a bottom face) that the tool can pass beyond.

**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.

**top plane** A planar geometric element that represents the top surface of an area to machine. It is always normal to the associated tool's rotational axis.

**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 path that the center of the tool tip 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

# E



# F



# G



# H



# I



# M

# O



# P



# R



# S

# T



# U



# V



# Z

Zig zag